1 Introduction

1.1 Introduction

The RAPT computer package is a design and analysis tool for reinforced and post-tensioned concrete slab and beam systems, developed by Prestressed Concrete Design Consultants.

RAPT is a computer package for computers running Microsoft Windows Operating System which, given the basic structural layout of a one or two-way reinforced or partially post-tensioned beam / slab system, will perform the analysis and design to

1. The Australian Concrete Structures Code AS3600
2. The British Concrete Structures Code BS 8110
3. The American Building Code ACI 318 for Reinforced Concrete
4. Eurocode2
5. South African Concrete Code SABS0100-1
6. Singapore Standard CP 65
7. Hong Kong Code of Practice for Structural Use of Concrete - CP2004
8. Indian Codes of Practice for Reinforced and Prestressed Concrete - IS456/IS1343

These seven base codes can then be further modified to simulate other codes. Users have the ability to adjust the following parameters

1. Code Specific Design Information
2. Load / Safety Factors
3. Concrete Properties
4. Reinforcement Properties
5. Prestressing Properties

The package has been written by practising structural engineers to be used as an everyday design tool. The following is a summary of design using RAPT

- The input is fully interactive using spreadsheet style entries with automatic updating of graphics. Common parameters are defaulted wherever possible. Interactive Graphic views of all input information are available for checking values input.

- The input briefly consists of defining the overall concrete dimensions, applied dead and live loads, the prestress strand data, tendon start and end locations and loads which the designer wishes to balance or tendon profiles and number of tendons.

- RAPT determines the number of tendons required and the profiles they should have to balance the requested loads. Designers are at liberty to alter any decision that RAPT makes on their behalf regarding numbers of tendons and tendon profiles. For post-tensioned systems RAPT recognises that there are many solutions for a given set of data. For example more tendons may be added and the drapes modified to achieve the same balanced loads. In fact, if during a run, the designer wishes to totally revise the prestressing layout he is free to do so, again and again until a solution is reached that accomplishes his requirements. In this way the most economical solution may be found.

- RAPT calculates all short-term and long-term losses of prestress using parabolic or harped profiles with automatic calculation of reverse curves.

- Tendon offsets from the concrete soffit to the underside of the ducts are given at approximately one metre centres. A graphics plot may be requested which draws the tendon profiles within the concrete cross-section, noting the offsets and spaces thereof. Strand extension figures are also calculated.

- An unlimited number of separate tendon profiles may be simultaneously incorporated in any single design. These may extend full length or stop short at any point within any span. Tendons may be jacked from either or both ends. For two-way systems separate profiling is done for column and middle strips. The tendon size, profile, start end locations etc can vary between column and middle strips.

- RAPT calculates all forces imposed on the system by the action of the prestress - distributed loads, moments, forces at changes in cross-section, eccentric anchorages and vertical forces at ends of drapes. The forces are calculated at long-term and at transfer forming two separate load cases that are automatically used in the ANALYSIS option.

- Secondary reactions and bending moments are automatically calculated by RAPT.

- Equivalent frame properties are calculated using specific inertias within each span and four inertias in each column.

- Loads can be entered as uniform distributed loads, linear distributed loads, point loads, point moments or as a moment diagram.

- RAPT will also calculate (if chosen by designer) moments and forces in the structure due to pattern loading. The pattern loading rules used vary depending on the design code selected and comply completely with the specific rules for loading pattern and load factors defined in each design code.
- RAPT performs a frame analysis for all the loads on the structure and combines them for transfer, service (moment controlled design envelopes are calculated, ultimate (both moment controlled and shear controlled design envelopes are calculated) and deflection cases / conditions using the combinations given in the code selected for the design (or as modified by the user in input). All the separate load cases or combinations may be plotted to the screen or printer. The plots and printouts show maximum moments and shears at the critical sections.
- For two-way systems the lateral distribution of bending moments across the panel width is automatically carried out. Again the designer is free to over-ride the factors determined by the program.
- The design of each span involves ultimate service, and transfer calculations. These are carried out at a series of evenly spaced points in each span with specific points included to cover all possible critical sections. These include critical moment regions, changes in cross-section and ends of internally stressed tendons. (column and middle strip). The locations of the critical sections for bending at the columns are automatically calculated by RAPT. Detailed output may be requested giving all steel stresses and concrete stresses. A cracked section analysis is performed if a section is in tension under working or transfer loads. A separate stand-alone cross-section utility is included in the package which enables a cross-section of any shape, reinforced/partially prestressed to be fully detailed or investigated for all of the above and for beam shear.
- Deflections are calculated using moment-area principles, using curvatures calculated based on the level of cracking at each of the design points and allowing for tension stiffening and long term creep and shrinkage effects based on the concrete properties and stresses and the reinforcement pattern in the member. Transfer (self weight for RC members), short-term, incremental and long-term deflections are calculated along with span/deflection ratios for comparison purposes . These may be plotted to the screen or printer if requested. RAPT also calculates deflections based on odd span and even span patterning of loads if requested.
- Punching shear and beam shear are investigated for each column and span. The required spacing of stirrups is printed for points along each span.
- Design results may be viewed interactively on the screen for each section of data in tabular text or graphical format with text information data available interactively. Output reports are created automatically to the requirements of the designer and may be viewed on the screen or previewed and printed to any windows printer including file utilities such as Adobe Acrobat. Each segment of the results may be individually selected. Thus reams of superfluous output may be controlled.

2 Installation

2.1 Program Installation

The RAPT installation program is easy to use. Double click the setup.exe program or when inserting the RAPT installation CD into your CD-ROM drive, the installation program will start.

Step1: The Welcome screen. Click *Next* button to go to next step.

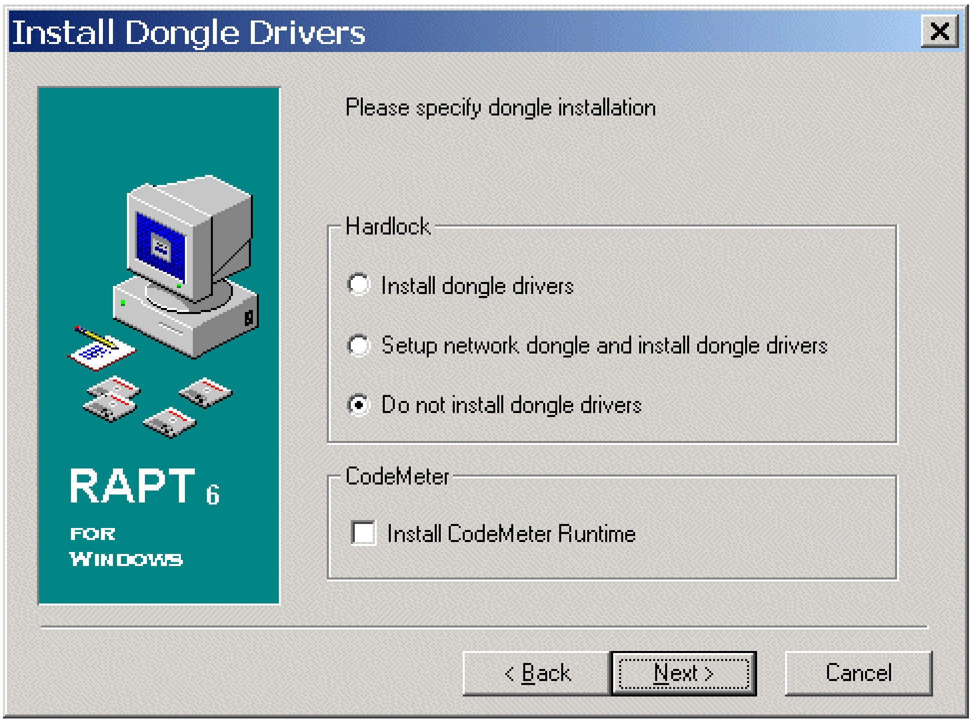
Step 2: The Licence Agreement screen. Please read carefully. If you don't agree with the licence agreement, please click Cancel to exit installation. Otherwise, click *Next* button.

Step 3: Choose the destination folder screen. The default folder to install RAPT is C:\RAPT6. To specify a different folder, click *Browse...* button. After the destination folder has been selected, click *Next* button to continue.

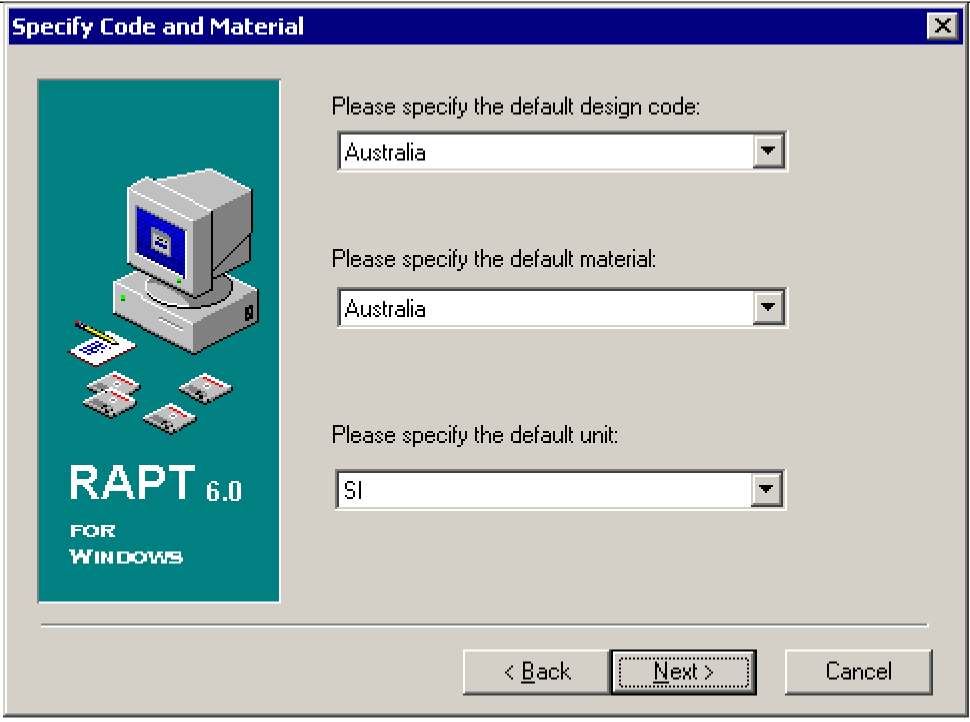
Step 4: Install Dongle driver screen. If you already have an up-to-date dongle driver, choose "Do not install dongle drivers" option. Otherwise, if you are going to access the dongle locally, choose "Install dongle drivers" option. If you are going to have a network dongle on this computer and allow other computers to access this network dongle via network, choose "Setup network dongle and install network drivers" option. Click *Next* button to go to next step. RAPT now has the option of 2 different dongle types

- Wibu CodeMeter dongles - USB standalone or network dongles.
- Aladdin Hardlock Dongle - either Parallel Port or USB versions of stand alone or network dongles.

Drivers for both dongle types can be installed on the same computer if a company has a mix of dongles. The same version of the RAPT executable will access both dongle types.



Step 5: Specify code and material screen. In this screen, choose the default design code and default material you want to use. Also choose your preferred default unit.



Step 6: Select program manager group. This screen will decide where to put the RAPT icon in the computer start menu. A RAPT short cut icon will also be put onto user's desktop.

Step 7: Now the setup program has gathered all the information it needs to start install RAPT. Click *Next* button to start installation or click *Back* button to go back and change installation settings.

Step 8: Setup is installing all the RAPT files to the computer. This includes the HTML help system. If your computer already has an up-to-date help system, you will see this dialog

Just click OK to continue.

Step 9: If you chose "Install dongle drivers" option or "Setup network dongle and install network drivers" option in Step 4, the installation program will start to install one of the Hardlock drivers.

Step 10: The Installation is complete screen. Click *Finish* button to finish installation. In some systems, installation will prompt you to restart your computer to complete the installation. You should close all the other applications, make sure all the current work has been saved, then click OK to restart your computer.

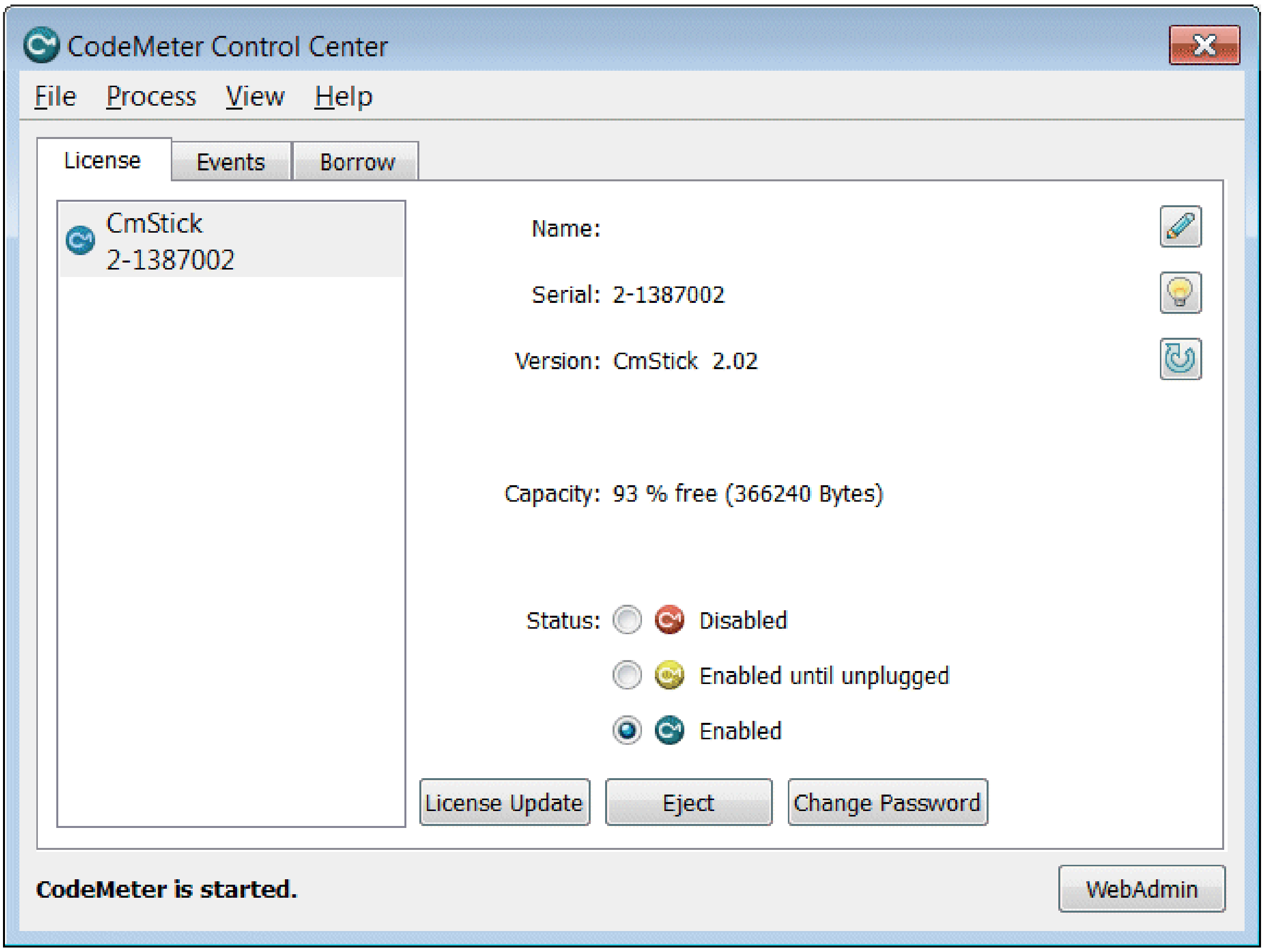
2.2 Dongle Setup

RAPT now has the option of 2 different dongle types

- Wibu CodeMeter dongles - USB standalone or network dongles.
- Aladdin Hardlock Dongle - either Parallel Port or USB versions of stand alone or network dongles.

Wibu CodeMeter Dongle Setup

The Wibu CodeMeter dongle does not have proprietary drivers, it uses standard Windows drivers. It does however have a comprehensive control program called CodeMeter Control Centre as shown below. We have only described the use of the functionality required for RAPT use below. The CodeMeter Control Centre has its own Help screens that describe all other functionality.



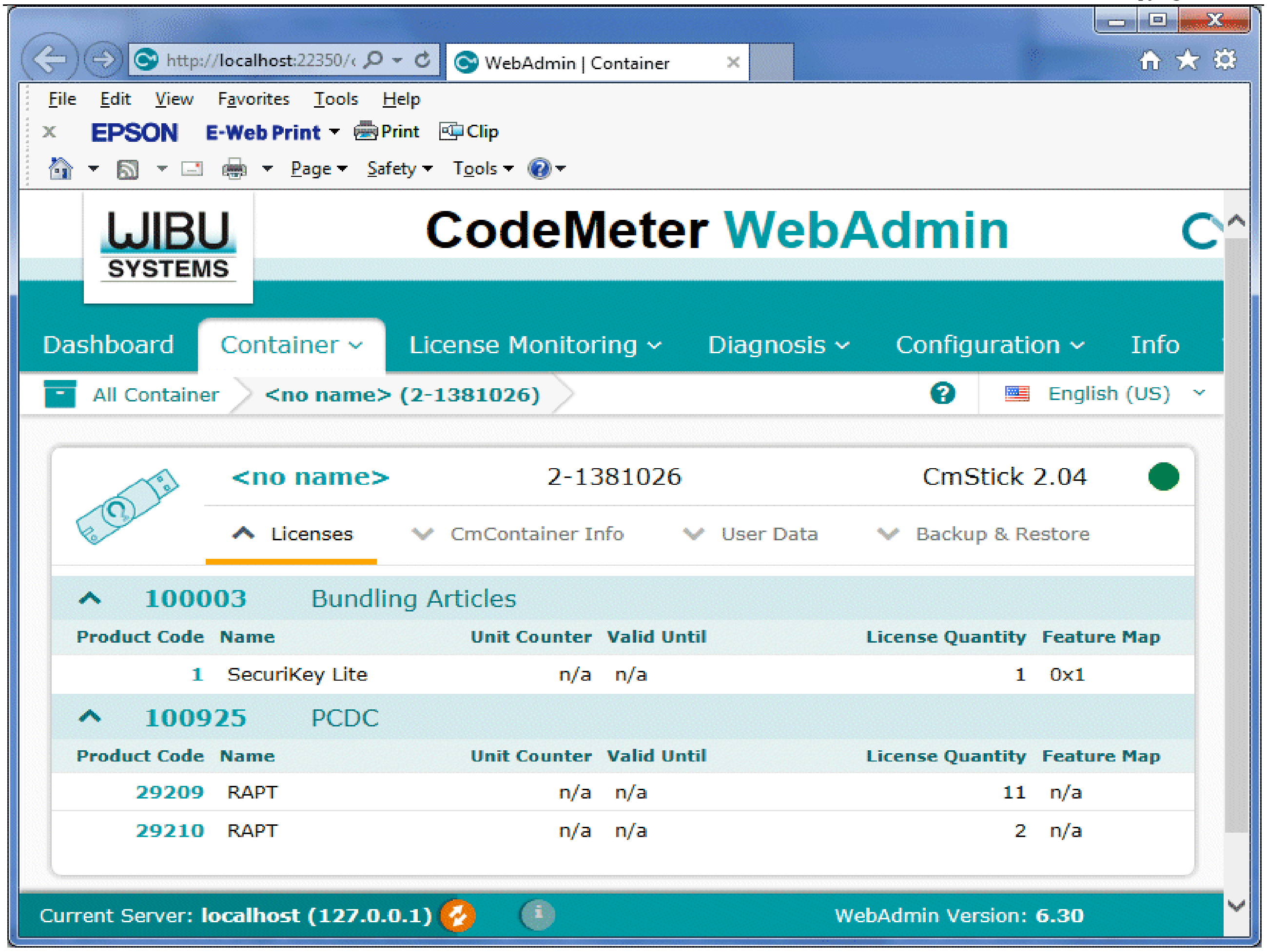
Licensing

The Licensing Update tab allows the user to create a Context file which must be sent to PCDC to create a remote update file in cases where the licensing information needs to be updated. Follow through the default options and create License Update Request. Email this file to PCDC for updating. We will modify the file with the updated information and return a License Update File which can be activated by going through the same process using the License Update Tab selecting Import License Update. Changes can be made to

- name and address
- number of licenses

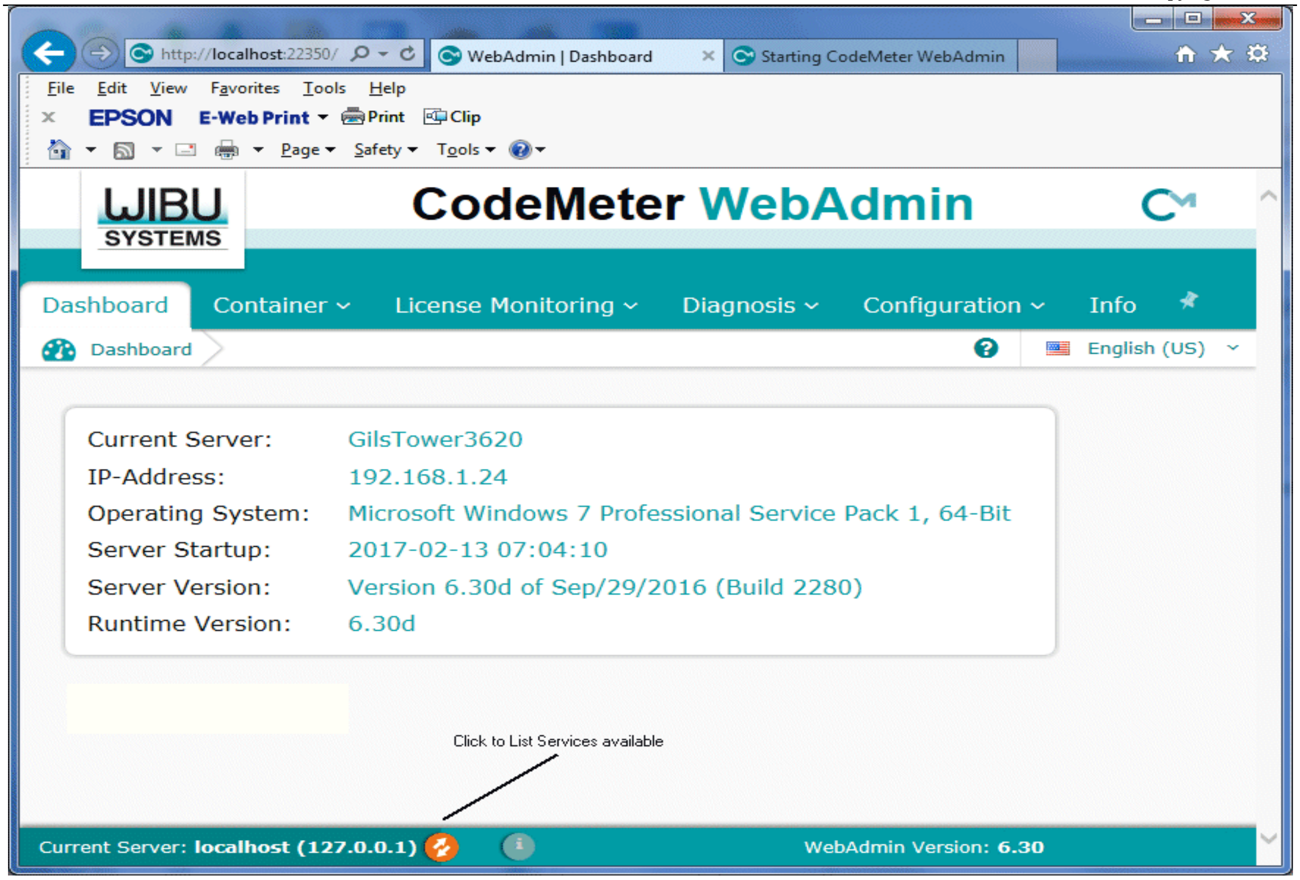
Administration

The WebAdmin button at the bottom of the dialog will take the user to the main administration pages in your web browser. The following page will be opened on clicking this button.



Dashboard - Server List

To obtain the list of services available on the computer/network, open the Dashboard and click on the Current Server at the bottom left of the page (shown below). CodeMeter will then search for all of the services available and list them. This will normally need to be done when the dongle is first installed on the computer. To view the properties of any service from any of the listed services, select it here. You will be able to view but not modify the service data on other computers. You will be able to view and modify the data on the service on your computer. To run to the server on your computer, select that server here.



Network Access
The basic network setup is carried out in the Configuration - Network pages shown below.

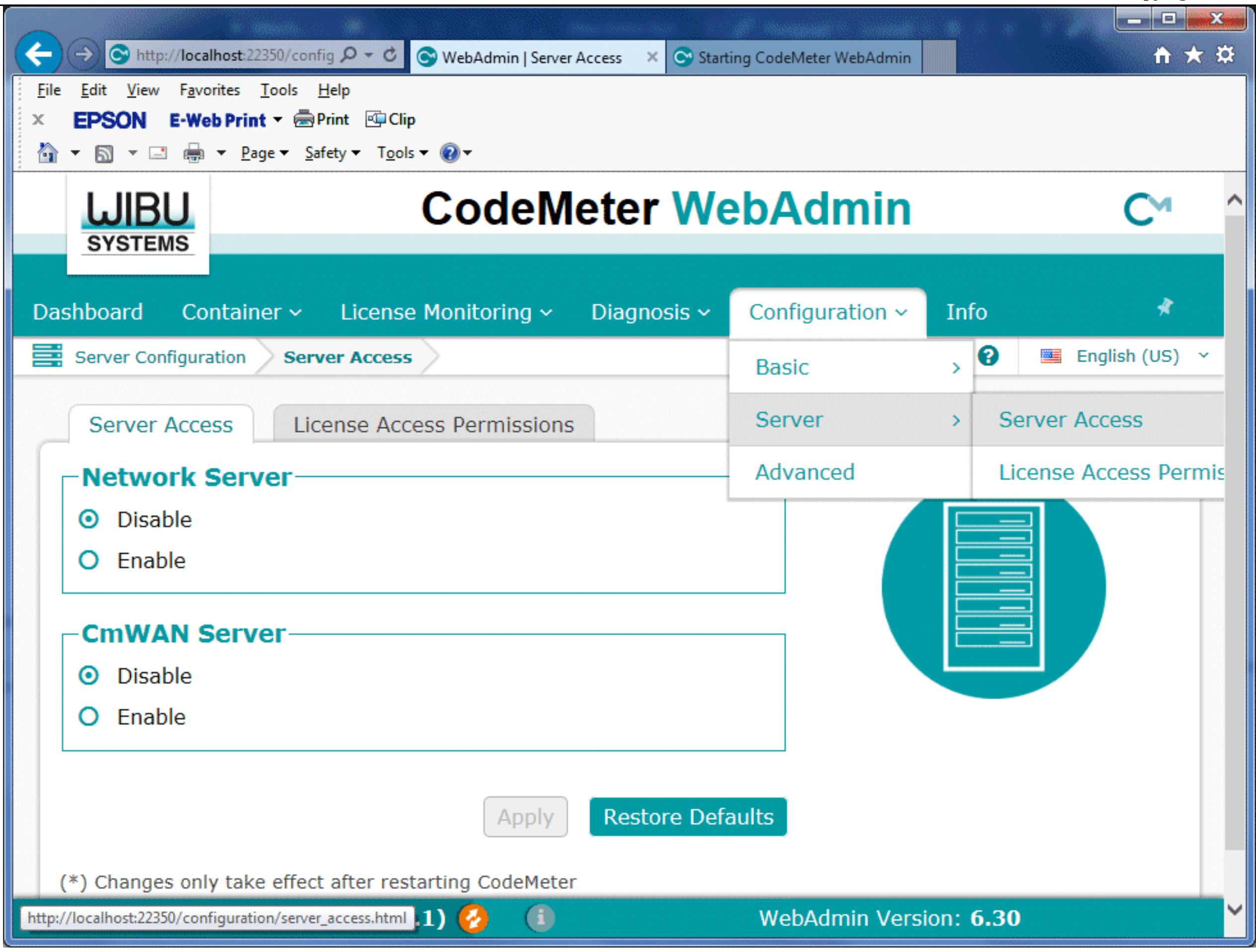
To set up for network access, you need to

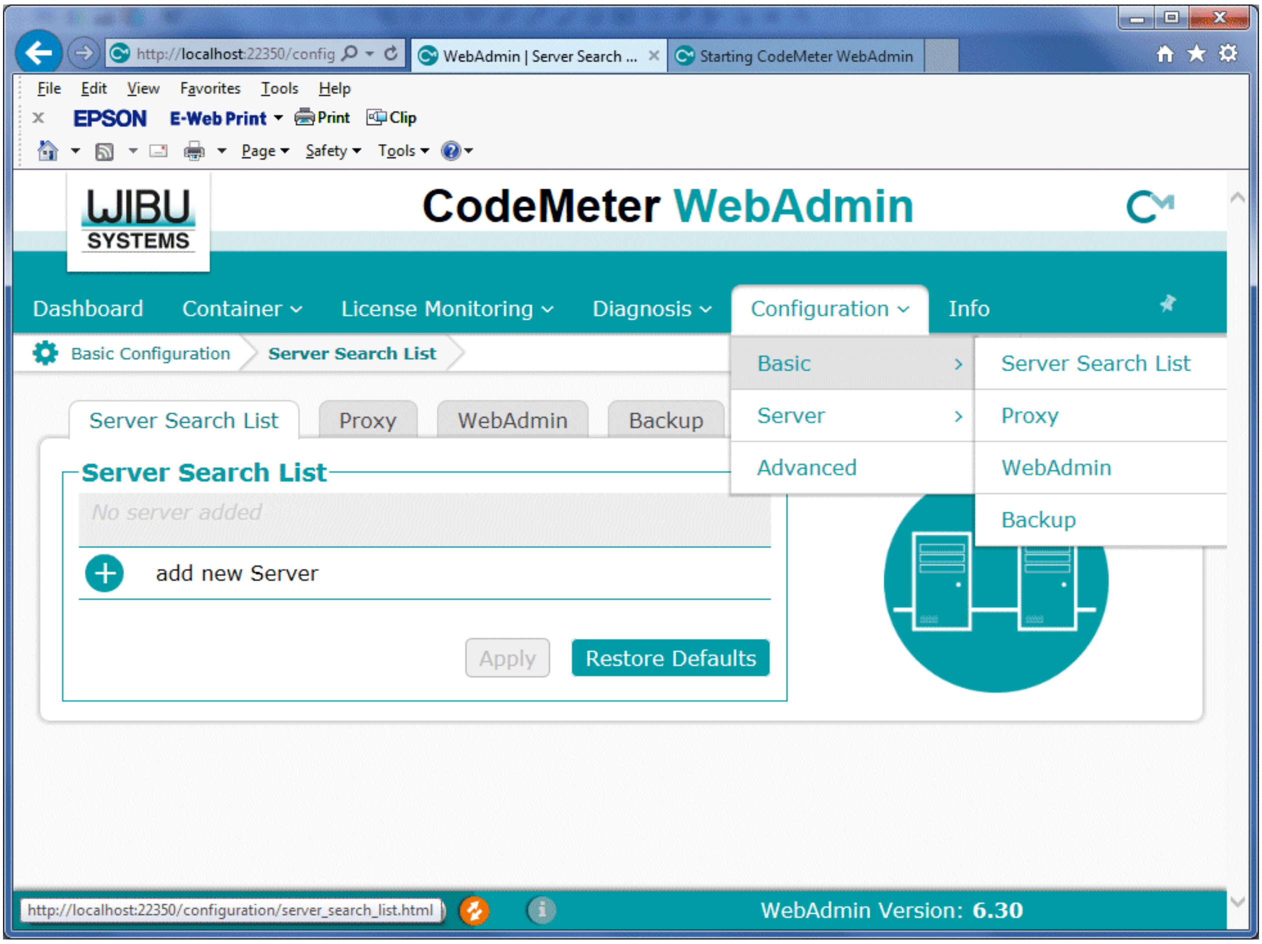
1. On the Dongle Server computer, tick the Run Network Server checkbox in Configuration - Server - Server Access. CmWan Server is not available for RAPT dongles.
2. On the Client computers running RAPT on a different subnet or on a WAN, nominate the IP address (preferred for different subnet) or the DNS name of the CodeMeter Server in the Server Search List in Configuration - Basic - Server Search List. Multiple Dongle servers can be nominated. If servers are listed here, they are the only servers that will be searched for licenses, so if it is necessary to limit access of certain computers to specific dongles, the dongle server should be nominated here.

When either of the 2 operations above are carried out, CodeMeter may need to be Stopped and Restarted in the CodeMeter Control Centre - Process Menu.

You will also need to ensure that the Network Port 22350 is open for access in your Firewall program (done automatically in Windows Firewall) and that both the CodeMeter Executable and the RAPTw executable are listed in allowed programs in the Firewall.

Where there are multiple CodeMeter dongles on a network, you can list the one/s you want to access in the Server Search List. CodeMeter will search through the list from top to bottom until it finds an available dongle.

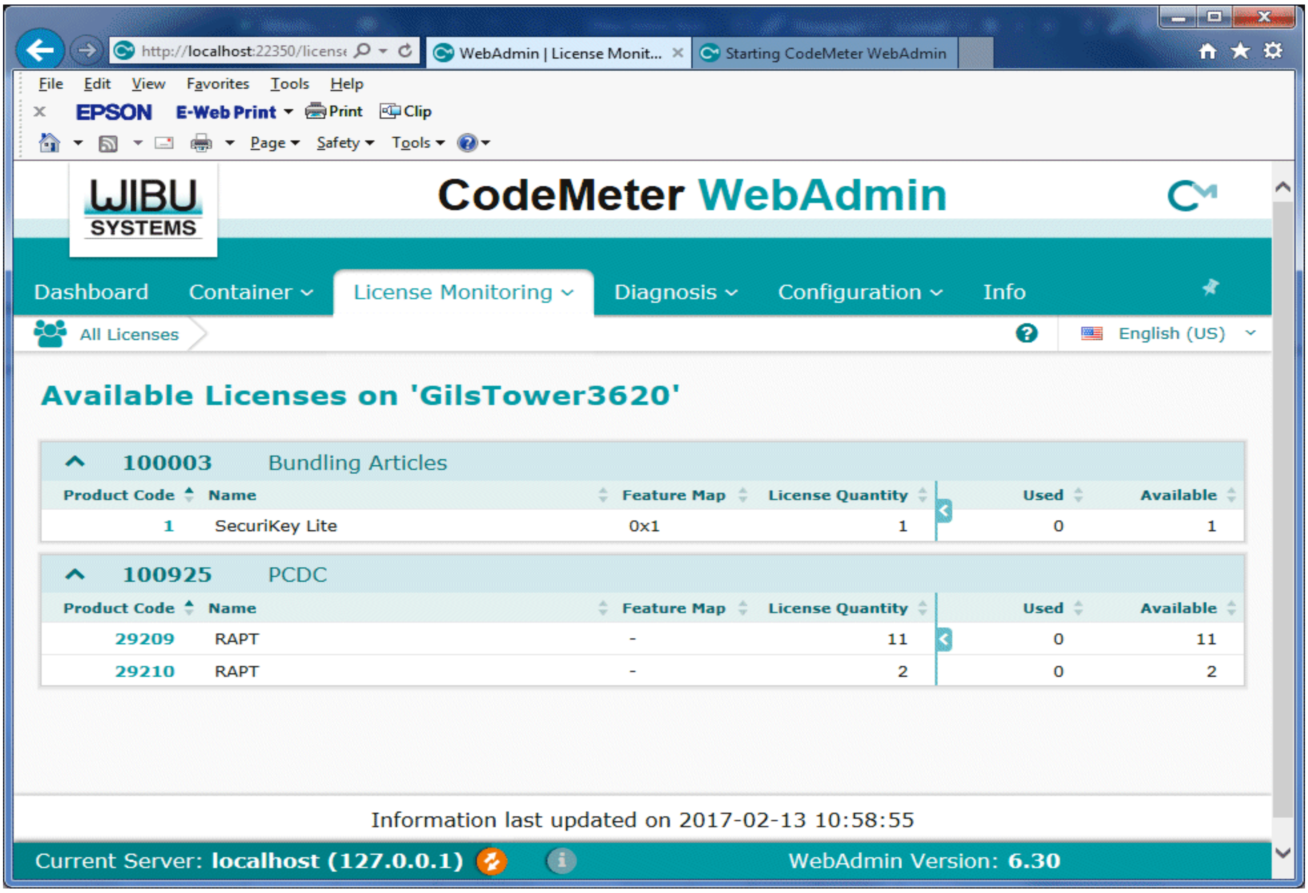




Network Monitoring

To monitor RAPT users logged into a CodeMeter server, select the server name from Dashboard - Current Server and go to the License Monitoring Page shown below.

This lists the dongle information summary including the number of licenses, number used (shared) and number free. Click on the Details Button for details of the users connected to this server and you can remove any user from the server from the details page (log them out).



Aladdin Hardlock Dongle Setup

The Hardlock comes in several forms

1. Parallel Port Standalone or Network
2. USB Standalone or Network
3. Internal Standalone or Network (ISA Bus or PCI Slot)

Drivers for all dongle types can be installed as described above when RAPT is being installed. The driver install programs are also separately available on the CD. The latest drivers available can be downloaded from our website www.raptsoftware.com. A manual, hlendusersmanual_en.pdf, which describes the dongle requirements and setup is available on the RAPT CD and in the \RAPT6\Drivers directory.

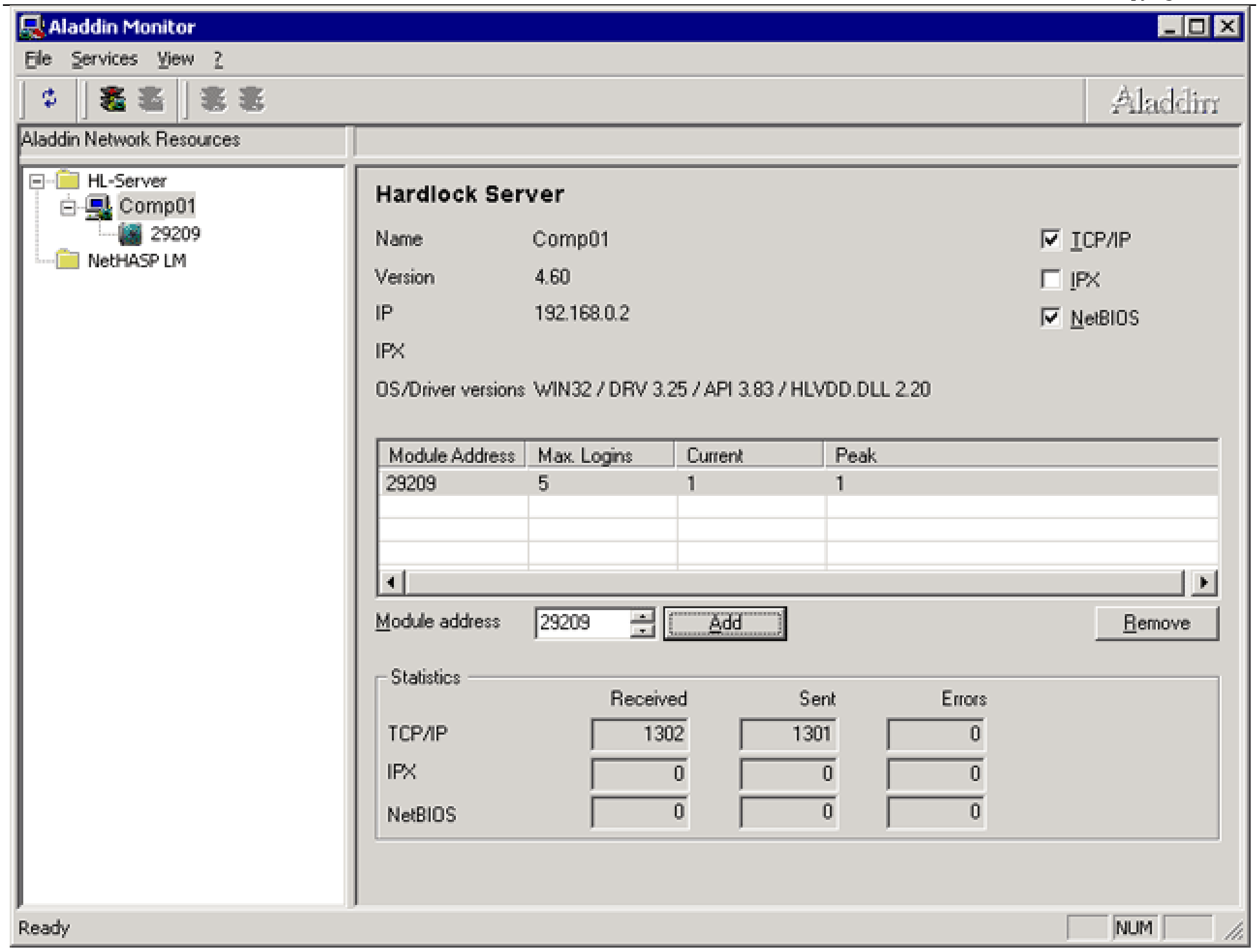
If RAPT does not recognise your dongle, first check that it is seated correctly and that it is closest to the parallel/USB port. Some other parallel port dongles do not allow full data flow on to following devices and may block some data required by the RAPT dongle.

Network dongle

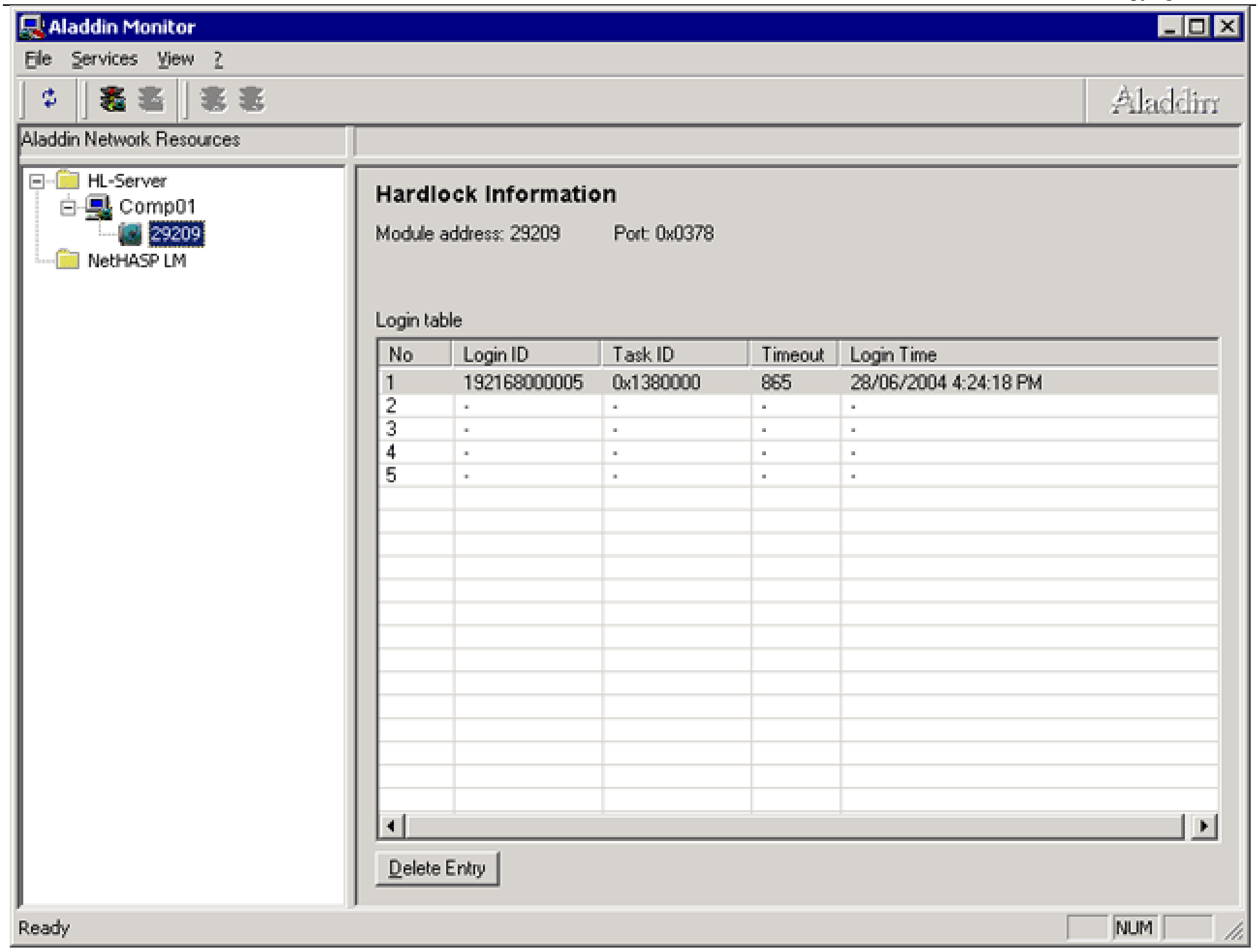
A network dongle may be installed on any computer on the network. It will normally be installed on a computer that is always on. The dongle drivers and service should be installed on this computer. This can be done during the RAPT installation or by running \RAPT6\drivers\hlsw32.exe. The computer user must have administration rights to install this program. This will install a service under WinNT/2000/XP or a server program under Win95/98/ME.

A dongle monitoring program has also been supplied in the drivers sub-directory. The file supplied, aksmon32.exe, is an installation program for the monitor. Install this program into the drivers sub-directory.

When the aksmon program runs, it will check for the network protocols and attached dongles and provide a screen similar to the one shown below.



To add a RAPT dongle to the service, click on the computer on which the dongle is to reside in the tree under the HL-Server branch. Then enter the Module Address 29209 in the space provided in the view on the right and click the Add button. The table of data will nominate a maximum number of logins allowed. This is the maximum number the dongle can accept and is not the number of RAPT licenses available. When RAPT runs on a network dongle, the maximum number of logins is listed in About RAPT in the Help menu along with the login Access Mode and the licensee details.



If you click on the dongle number in the tree on the left, the screen above will be shown. It lists the current logins to the RAPT dongle along with details on the login times for each user.

To logout a user from the dongle, use the delete Entry button at the bottom of this view when the entry to be deleted is selected.

When a user closes down a RAPT session, the entry will automatically be deleted. If, for some reason, RAPT crashes, the login entry will remain until the timeout period is finished. This view can be used to delete the entry.

Further information on use of network dongles is available in the \RAPT6\drivers\hlendusersmanual_en.pdf manual provided by the manufacturers of the dongles.

2.3 Configure RAPT for Shared Network Use

This option is available from File -> Config for Network Use menu when no file is opened. It enables multiple users to use one copy of RAPT in a networked environment while maintaining their own individual set of configuration files. It means one installation for multiple RAPT users instead of multiple installations for multiple users.

Despite of this advantage, it is recommended NOT to use it except in cases where users wish to be able to run RAPT from any computer in the network using their own configuration settings. It is much more efficient and easy to maintain by installing it onto each individual user's computer in other cases. For those who have to use single copy of RAPT for multiple users across the network because of hard disk limitations, network administration policies or just to save the installation time, here is how it works.

First, multiple users can share a RAPT network dongle. The number of concurrent RAPT users is limited to the number of allowed users in the network dongle. Or each user can have a standalone dongle on their own workstation. The following example demonstrates how to set up and configure RAPT for single installation multi user network use.

RAPT is installed to the \\AppServer in c:\rapt folder. This folder must be shared and should be set with read-only permission for normal users. Suppose the shared name is also rapt, then we can access \\AppServer\rapt from anywhere in the network. Create a network share for user John at any network station that is connected with \\AppServer. It should be fully accessible by John and not accessible by other peer users. In this example, we create \\WorkStation1\JohnsRAPT. We also create similar network share for Mary at \\WorkStation2\MarysRAPT and Peter at \\WorkStation3\PetersRAPT.

There are 3 configuration files used for RAPT. They are:

File Name	File Description
Default.rnc	Remember each user's name and that user's configuration directory. This is a text file. (Network users file, Server only)
Default.ruc	Remember the default design code and material, as well as last opened directory. This is a text file. (User Configuration)
Default.rup	Remember user's preferred unit format, calculation options and output options, etc. This is a binary file. (User Preferences)

The Default.rnc contains the data relating to the users on the network. The .ruc and .rup files are personal files for each user and a copy of each will be stored in each users private directory space in the User Preference File Path.

First we need to modify the default.ruc file to reflect the network share path for its path rather than it's default local computer path. To do this, open default.ruc in \\AppServer\rapt in a text editor. It should look like this:

```
[RAPT User Configuration]
Design Code Template Directory=C:\RAPT6\DefaultCodes\
Design Code File=C:\RAPT6\DefaultCodes\Australia.rdc
Materials Template Directory=C:\RAPT6\DefaultMaterials\
Materials File=C:\RAPT6\DefaultMaterials\Australia.rmc
Runtime Directory=C:\RAPT6\Raptrun\
```

Replace the string "C:\RAPT6" with "\\AppServer\rapt" and delete the content on the right hand side of the equal symbol in the last line. The modified file should look like this:

```
[RAPT User Configuration]
Design Code Template Directory=\\APPSERVER\RAPT\DefaultCodes\
Design Code File=\\APPSERVER\RAPT\DefaultCodes\Australia.rdc
Materials Template Directory=\\APPSERVER\RAPT\DefaultMaterials\
MaterialsFile=\\APPSERVER\RAPT\DefaultMaterials\Australia.rmc
Runtime Directory=
```

Save the modified default.ruc file.

The person who configures RAPT for shared network use can do it on \\AppServer computer or any other network computer as long as the person has access privilege to \\AppServer\rapt. After RAPT is started, click File -> Config for Network Use from the menu. Then, in the Config for Network Use dialog, click the Add button to add an entry and type in the information to get result a like the following:

You can either type the path name into User Preference File Path or just click the "..." browse button to specify the path from a dialog box. The network share path must be used instead of path name such as c:\Users\JohnsRAPT. This is because John may sometimes use \\WorkStation2 and RAPT won't be able to find the path in c:\Users\JohnsRAPT on \\WorkStation2. This is easiest done from the Browse Button and selected from the computer and path tree My Network Places. Now RAPT knows where to store John's configuration files. RAPT will prompt for both the default.ruc and default.rup files to be copied to \\WorkStation1\JohnsRAPT if there are no copies there already with a dialog like the following

Click Yes button to allow RAPT to copy the default.ruc and then the default.rup file from \\AppServer\rapt to \\JohnsRAPT. Repeat the same procedure to add Mary's entry and Peter's entry. To delete entries, click the row headers to highlight the rows to delete and click the Remove button. The "Change..." button on each row can be used

to change the data in the default.rup file in each users' user preference file path (for example: \\WorkStation1\JohnsRAPT\default.rup) directory. This will launch the User Preference Dialog and allow all user preference settings to be changed for that user.

The configuration procedures can be summarized using the diagrams:

When John is using \\WorkStation2 to access RAPT on \\AppServer, he logs into the network as John. RAPT looks at the record in Default.rnc and find John's configuration directory is at \\WorkStation1\JohnsRAPT. It will read Default.ruc and Default.rup in \\WorkStation1\JohnsRAPT to configure itself for information such as default design code, default material, John's preferred display unit and output format, colours etc. It means no matter which network computer John is using, RAPT will always find the correct configuration file for John.

For a network that contains Window98 computers, the shared folder name cannot be longer than 8 characters. Otherwise, Windows98 cannot recognize that share.

3 File Management

3.1 RAPT Data File Types

Data Files:

File Type	Description	Access
RDC file	RAPT design code file.	Editable in RAPT
RMC file	RAPT material file.	Editable in RAPT
RDS file	RAPT design code template file. Any newly created RDC file is based on one of these RDS files.	Not Editable by User
RSM file	RAPT material template file. Any newly created RMC file is based on one of these RSM files.	Not Editable by User
RPF file	RAPT frame file.	Editable in RAPT

Configuration Files:

File Name	Description
Default.rnc	Network Configuration file. Lists each user's name and that user's configuration directory. This is a text file.
Default.ruc	User Configuration File. It specifies where the template RDS and RMS files are, which RDC and RMC file should be used as default file, and the last accessed directory. This is a text file.
Default.rup	User Preference file. Saves a user's preferred unit format, calculation options, output options, etc. This is a binary file.

3.2 RAPT Frame File

The RAPT Frame file, .rpf, is a binary file which contains

1. The input data for a RAPT frame run
2. The output results data (optional)
3. A copy of the Design Code Data for the selected design code
4. A copy of the Materials Data for the selected country materials data
5. A copy of the output tree selections.

This file is only editable and the results can only be viewed/printed from RAPT.

3.3 Column

This section is not available yet.

3.4 Cross-Section

This section is not available yet.

3.5 Profile

This section is not available yet.

3.6 RAPT Design Standards File

RAPT allows designers to design to several national design standards and also allows the designer to customise the data in these files to suit different adaptations of these national standards by other countries.

To facilitate this, RAPT uses 2 types of file for design standards. These are

1 .rds file - These files are not editable. They are the template files which are used as the basis of new Country Design Standards created by designers. They have a name related to a Design Standard rather than a country, eg ACI318.rds.

2 .rdc file - These files are editable by the designer. Each is the Design Standard file for a country. They are based on a specific country Design Standard and may be modified to suit another countries adaptation of that design Standard. The designer has control over some factors used in Design Standard rules. The basic Design Standard logic, formulae and rules are used in RAPT with these factors. Designers can create their own versions of these files from the Template files or edit the files supplied with RAPT. Any changes made to these files are the responsibility of the designer.

3.7 RAPT Materials Data File

RAPT is used by designers in many countries to design to a variety of design standards, often developed by different countries. For this reason the materials data has been kept independent of the design standard and is defined for each country.

Again there are 2 file types. These are

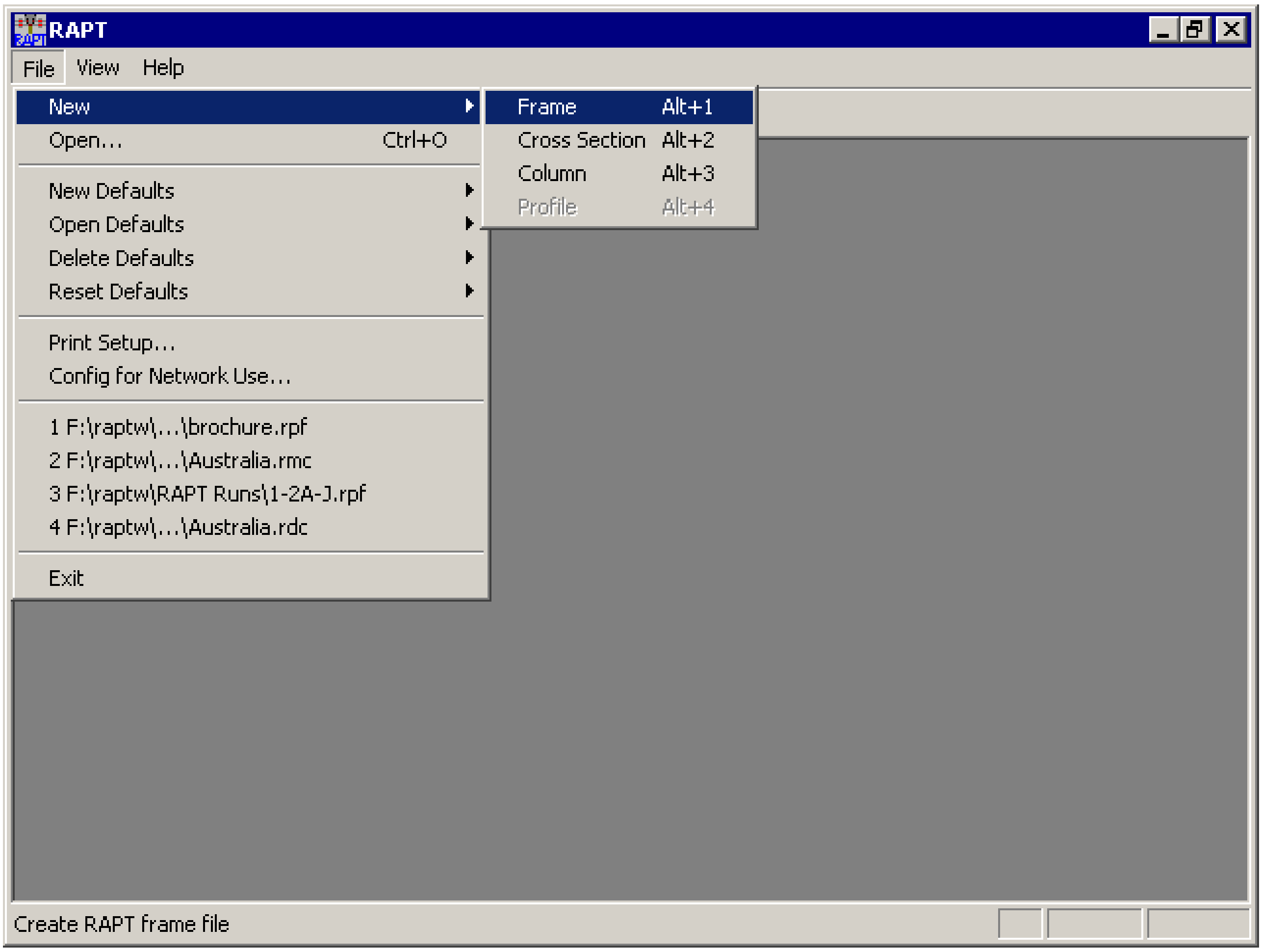
1 .rsm file - These files are not editable. They are the template files which are used as the basis of new Country Material Sets created by designers. They have a name related to the country for which they were compiled by the RAPT developers .

2 .rmc file - These files are editable by the designer. Each is the materials Data file for a country. They are based on a specific set of materials for a country and may be modified to suit the materials available in another country. The designer has complete control over the materials definition.

4 User Interface

4.1 Start Screen

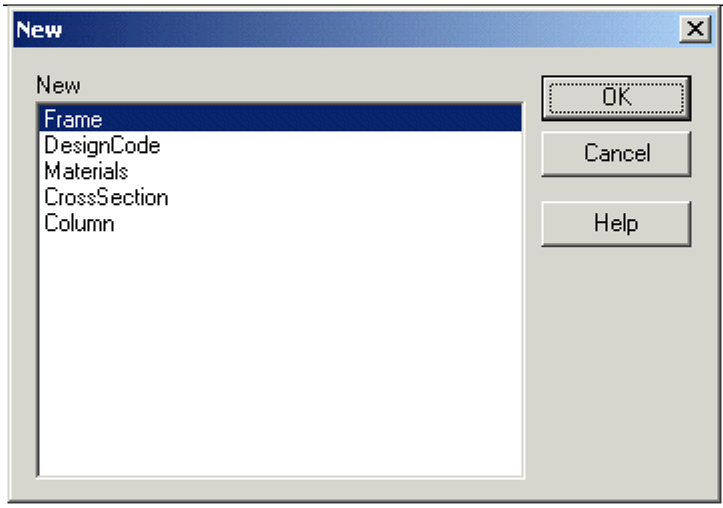
4.1.1 Start Screen - File Menu



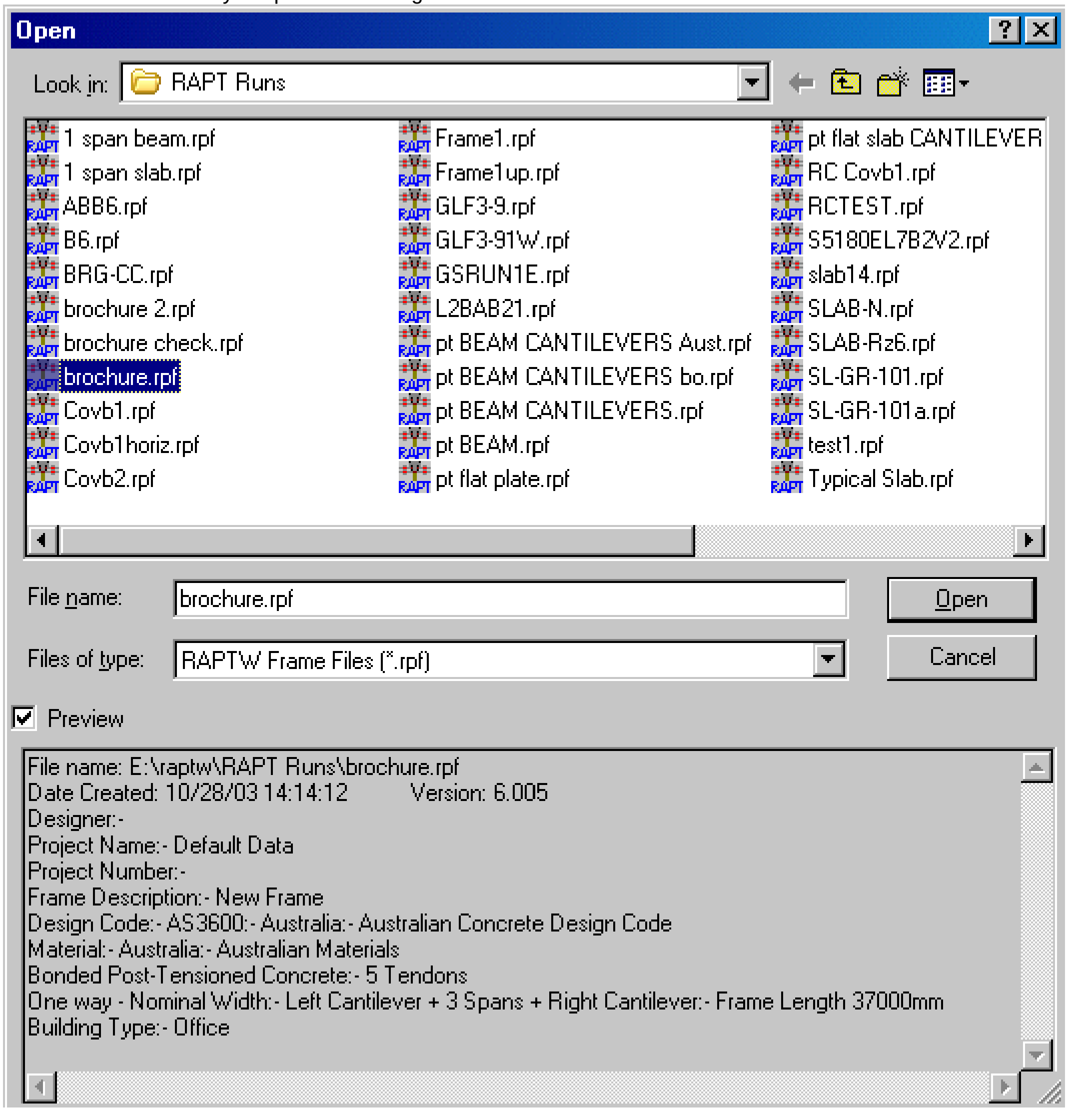
New (Ctrl + N)
Allows the user to create a new design file. Four types of design file can be created in RAPT

1. Frame - The user can define a 2D concrete sub-frame for automatic analysis, design and detailing as a reinforced or a prestressed concrete member.
2. Cross-Section - The user can define a single complex cross-section of a member with reinforcement and prestressing tendons for detailed design.
3. Column - The user can define a column shape with reinforcement and prestressing tendons and produce various interaction diagrams for it.
4. Tendon Profile - The user can define a prestress tendon geometry in terms of high and low points and produce a tendon profile drawing with calculated extensions

If the user clicks the  button from the toolbar, a dialog box will be prompted to ask which type of the RAPT file you want to create.

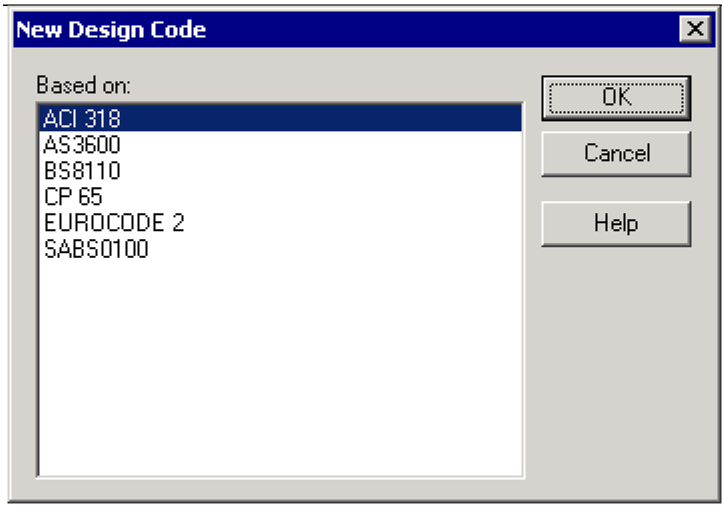


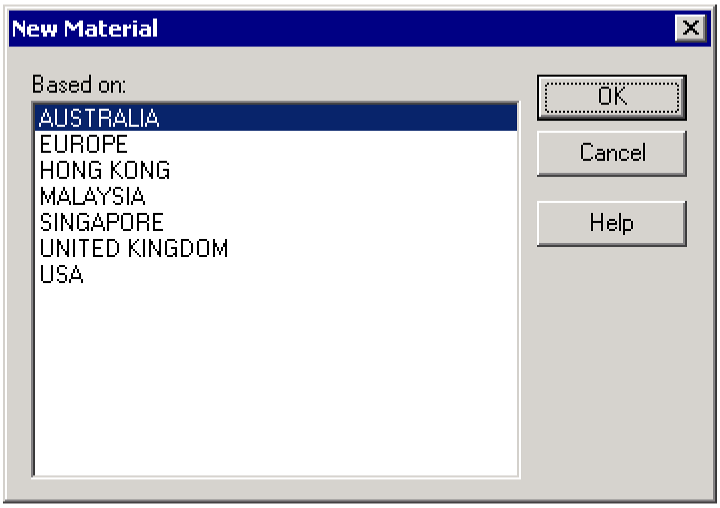
Open (Ctrl + O)
The user can open one of the design files mentioned above or a Design Code file or a Materials file for editing using the standard windows style open file dialog below..



For Design files, a preview pane has been provided at the bottom which lists some relevant data from the selected file to help with file selection.

New Defaults
Allows the user to create personalized Design Code and Materials files. These files are based on the Template files supplied with RAPT for various Design codes and Countries. The user selects a default file from the dialogs below and is then able to modify the data to suit his own design needs. Any changes made to Design Code factors should be in accordance with local design rules and the logic of the formulas in which they will be used for that design code.

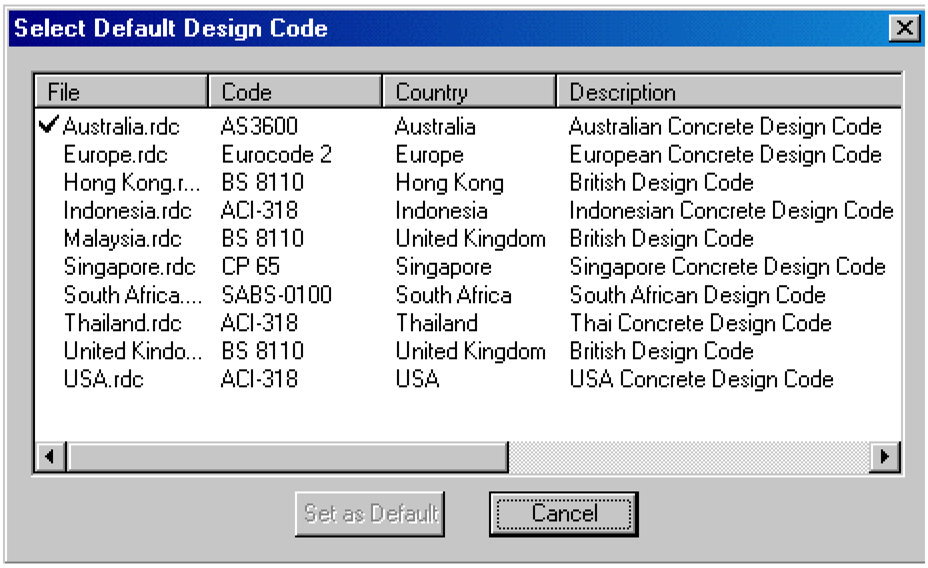




Open Defaults
Allows the user to open an existing Design Code or Materials file for viewing or editing.

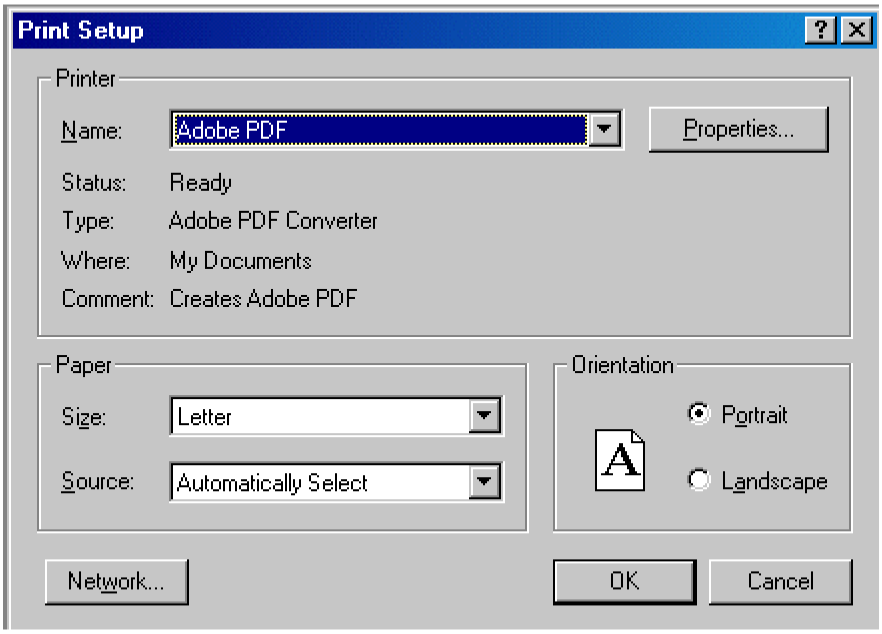
Delete Defaults
Allows the user to delete an existing Design Code or Materials file.

Set Defaults
Allows the user to select which Design Code or Materials file is to be used as the default file when creating a new design file. Simply select the desired file from the dialog (Design Code shown below) and press the Set As Default button. The current default file is indicated by a tick.



Page Setup
Allows the user to define the printer settings for the current RAPT session using the standard printer settings dialog shown below. The default settings are the default Windows settings. Changes made to printer settings in this dialog will be lost when another printer is selected or the current RAPT session is closed. To make permanent changes, right

click on the desired printer from the Windows Start->Settings->Printers menu and select Properties. Changes made there will be permanent.



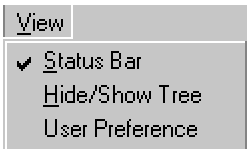
Configure for network use

Allows the user/administrator to update the configuration for network use of RAPT. See 2.3 Network Management for an explanation of the options and process.

Recent Files

Lists the last 9 files opened in RAPT for quick access.

4.1.2 Start Screen - View Menu



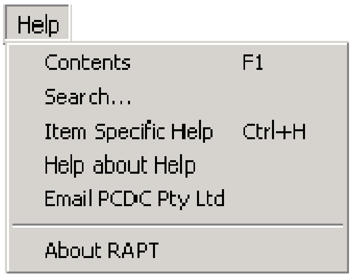
Status Bar
Allows the user to select to have the status bar visible or hidden. Clicking with the left mouse button will toggle between visible and hidden.

Hide/Show Tree
This item is only shown when a Control Tree is available. It allows user to hide the program control tree. For computers with small or low resolution screens, this allows the data views to use the full screen width while being viewed/edited.

User Preference
Allows the user to control a set of preferences for units, fonts and data formatting. See 4.2 User Preferences.

4.1.3 Start Screen - Help Menu

RAPT uses a standard Microsoft HTML Help system for viewing help information, If the files needed for viewing HTML Help are not loaded on your system RAPT will load them during installation.



Contents (F1)
Pressing F1 or selecting Contents from the Help Menu will open the Help Viewer at the Table of Contents. The user can then select the area of help to be viewed from the lists in the contents tree.

Search
Allows the user to search the complete help document for a keyword, phrase or group of works. A list of the topics containing the search request will be presented and the search request text will be highlighted wherever it occurs within each topic.

Item Specific Help (Ctrl + H)
When a data cell that has specific help is in focus, pressing Ctrl + H or selecting this item from the Help menu will open the Help document at the relevant item.

Help About Help
Selecting this item will give a help screen detailing the use of the Help viewer.

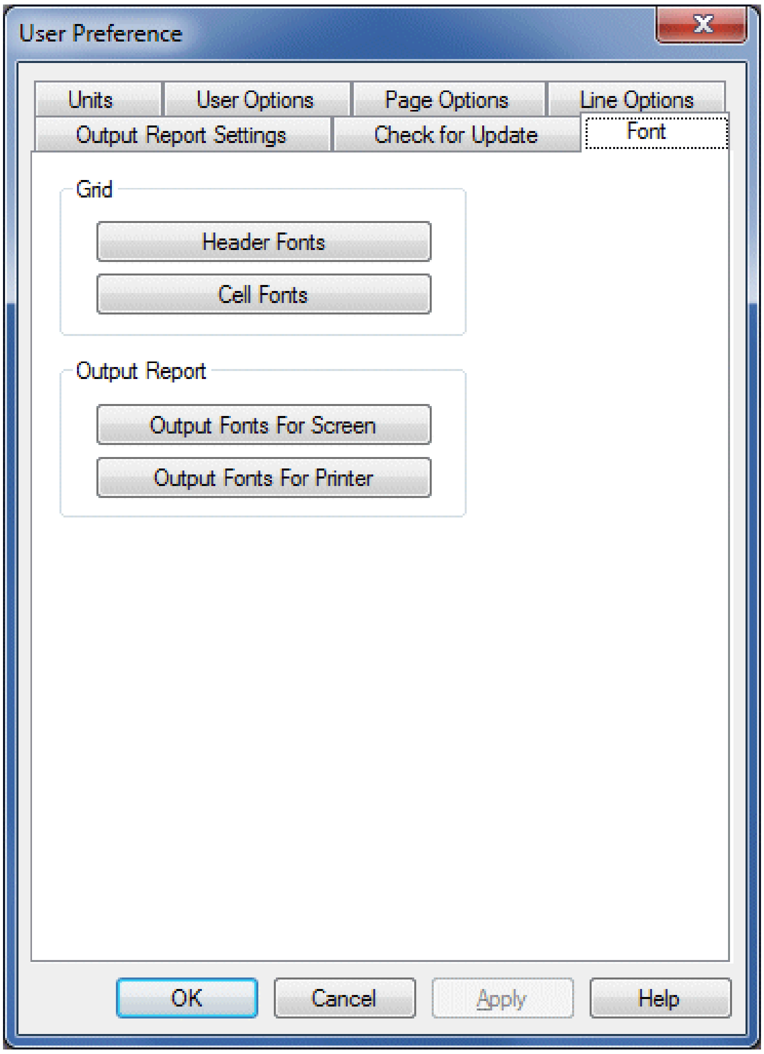
Email PCDC Pty Ltd
Automatically launches your computers default Email program with an email to PCDC support.

PCDC Website
Connect automatically to the PCDC website.

About RAPT
Opens a dialog which lists

1. RAPT Version number and copyright.
2. RAPT Licensee details and dongle number
3. Number of users logged onto a network dongle and maximum number of users allowed.
4. Current Usage for demonstration and usage licenses.

4.2 User Preferences



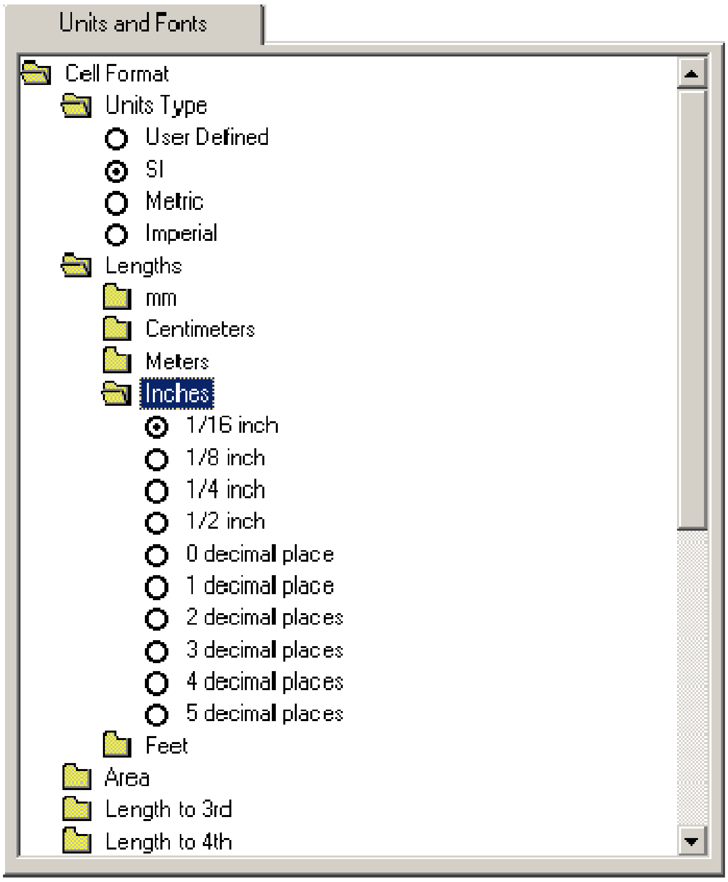
RAPT users can configure a series of options used by RAPT in the input and display of data and results. These can be accessed and modified through View->User Preference menu. This menu will bring up a dialog which allows the user to configure

- 4.2.1 Units and Fonts - The user can select the base unit type to show in all data and graphics views and to set the format to be used in showing each if the individual unit types as well as the font characteristics to be used in displaying the information.
- 4.2.2 User Options - The user can select various options used in controlling the data and report screens
- 4.2.3 Page Options - The user can configure the gape layout for output reports.
- 4.2.4 Line Options - The user can modify the default line colour and styles set in RAPT.
- 4.2.5 Output Report Settings - The user can select the sections of the Output Tree and Output Report that will be open by default for any new data file.
- Check For Update – The User can request that RAPT notifies via a dialog that an Update is available for download from www.RAPTsoftware.com Downloads page. This check is made whenever RAPT is started. The download is not automatic. The user still has to go to the website and download manually.

4.2.1 Unit and Fonts

The Unit and Fonts tab in User Preference dialog allows user to change their unit settings and font settings.

Units



RAPT supports 3 standard unit formats, SI unit, metric unit and imperial.

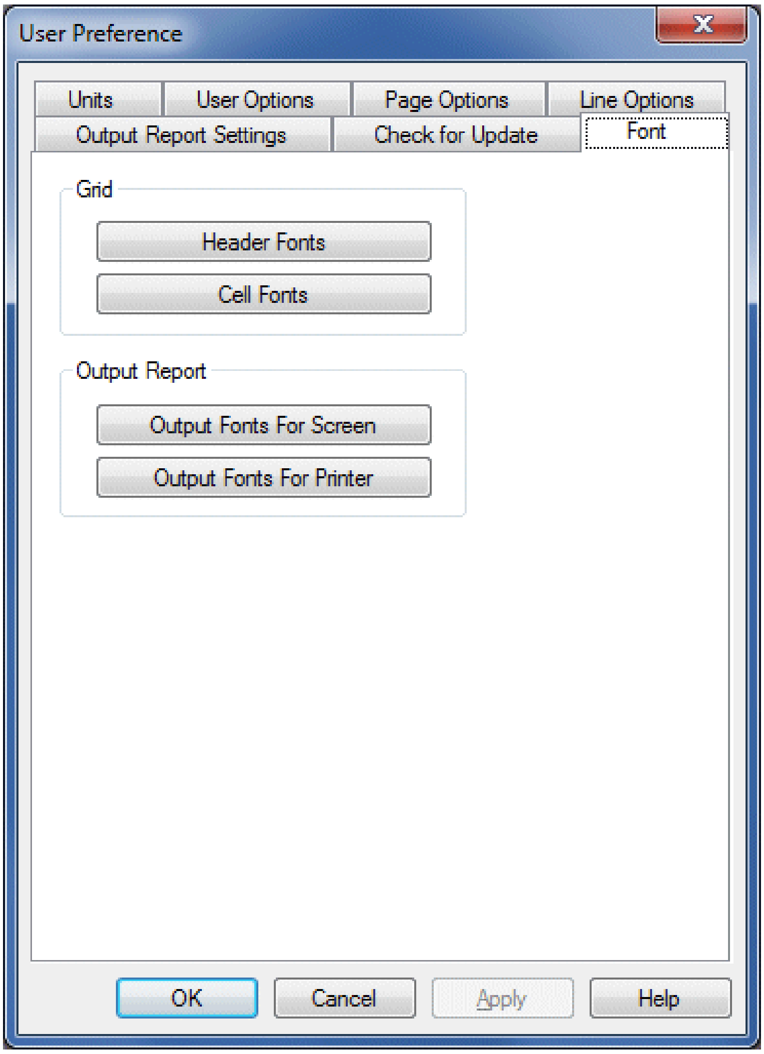
In addition, it also allows the user to define a personalised format. To do this, the user must select a base format from one of the standard formats. In every data grid, for each column/row of data that has a unit type the user can then select a unit type from the list for that data type. To access the list of unit types, click the right mouse button on the unit type data cell for that column/row of data. A list of unit types will be offered and the user can select any of those types for that column/row of data. Once all of the columns have been set as desired the unit format can be saved using the Tools->Save As User-defined Unit Format menu option. This unit format can then be selected as the default unit format for this user in the View->User Preference->Unit Type menu item.

For each unit, the user can further define the precisions that they want RAPT to use to display that unit type. The selected precisions will then be used consistently throughout the whole program.

The precisions available for SI and Metric and Imperial units are from 0 to 5 decimal places.

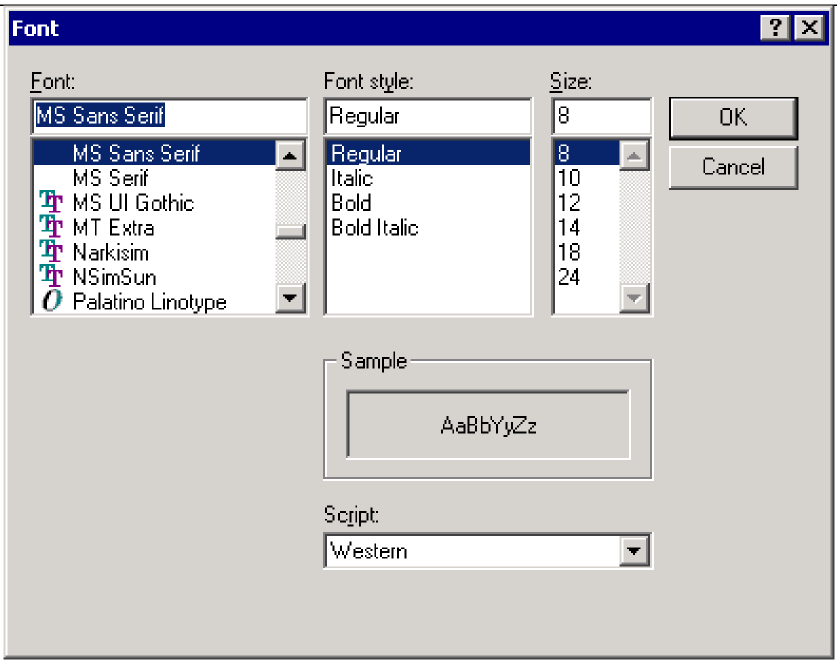
For Imperial Unit length measurements, fractions of an inch are accepted from 1/2" to 1/16" and combinations of feet and inches can be input, eg 12' 7 1/2" or 12.625' are both acceptable. The ' and " symbols must be used where a combination of feet and inches is used, otherwise in "ft-in" cells, the input will be assumed to be feet and in "in" cells it will be assumed to be inches. In cases where fractions of an inch have been selected as the precision, the fraction will be rounded to the lowest denominator, eg if 10/16" is entered, it will be shown as 5/8" once the edit operation on the cell is completed.

Fonts



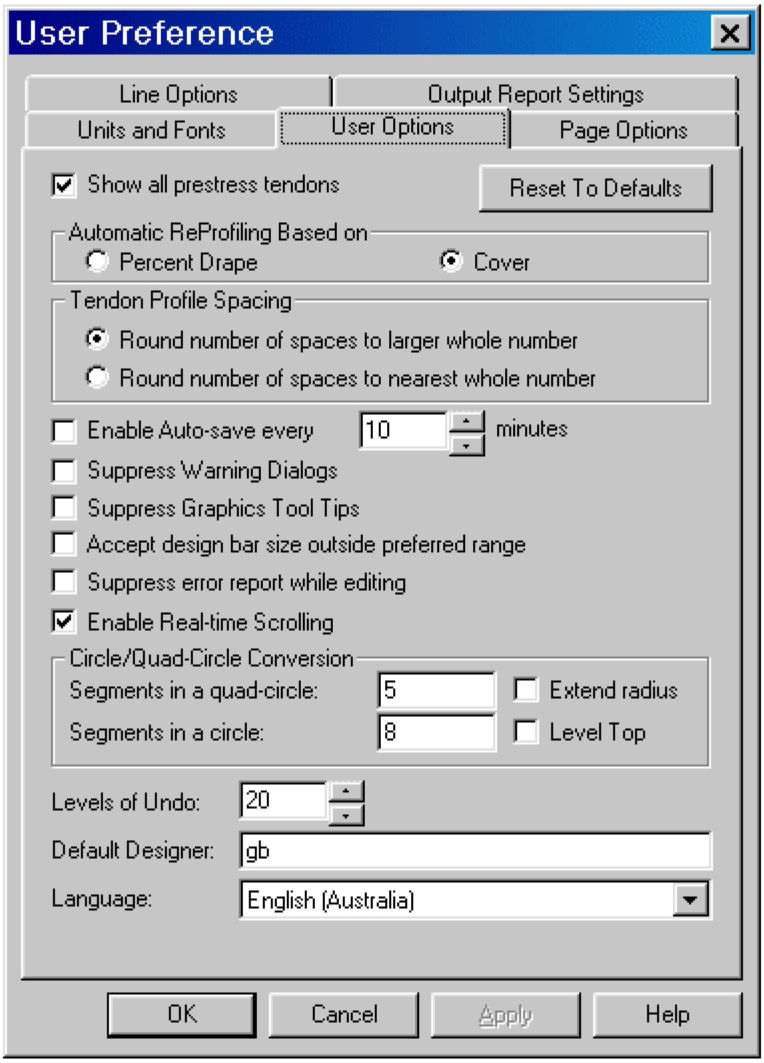
The user can also change the following fonts to suit their viewing and printing needs using standard Windows font selection dialogs (see below).

1. Input header cell font
2. Input data cell font
3. Output font for screen
4. Output font for printing (used when doing a Print Preview or printing to the printer).



As soon as a change is made to a single data item in the Units/Fonts page, the change will be made to the currently open RAPT views, allowing the user to see the effects of the change. The Cancel button at the bottom of the dialog will NOT over-ride changes made in the Units/Fonts page of this dialog.

4.2.2 User Options



Show all prestress tendons: For a RAPT frame data file having multiple prestress tendons, for extremely complex runs or on slow computers, the regeneration of the prestress graphics may be too slow if all the tendons are shown in the background. By turning this option off, only the current tendon is drawn and the redrawing would be much faster.

Automatic Re-Profiling Based on: When concrete sections are modified, RAPT will check the tendon profiling to see if any of the tendon profiles need to be modified to suit. The designer can select to have the re-profiling based on maintaining either

1. Percentage Drape:- the same percentage drape in each span as previously or
2. Cover:- the same cover at each nominated profile point as previously

Tendon Profile Spacing: When calculating the profile support points, RAPT will round spacings to either

1. the next whole number of profile spaces or
2. the numerically closest whole number of profile spaces

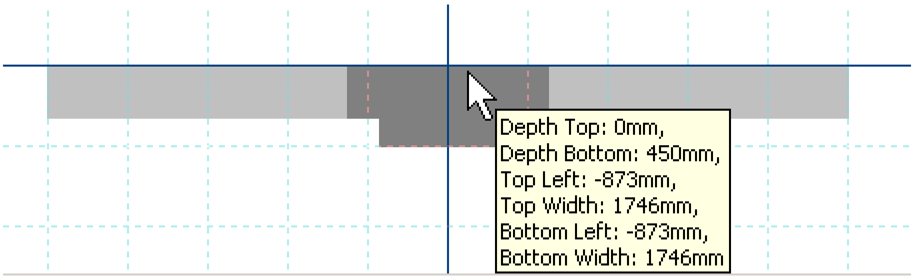
depending on the users selection.

Enable Auto-save every ** minutes: When selected, RAPT will automatically backup the current data to a temporary file at the selected time interval. This file will be the existing file with ~ at the start of the file extension, eg .~rpf. This file will be stored in the current directory for saving data files. If the file is closed normally, RAPT will automatically delete the backup file. A backup file will only exist in a directory if RAPT has crashed this not deleting the file. When RAPT is started it will automatically open an existing backup file or, if there are more than one backup files in a directory, will offer the option to select the backup file to be opened.

Suppress Warning Dialogs: When the user is about to make a major changes which could cause the loss of a large amount of data if proceeded with, a warning dialog will come out to ask for confirmation of the change, eg

If you find such dialog boxes is annoying, turn on this option to suppress warning dialogs and the changes will be made without warning.

Suppress Graphics Tool tips: Graphic tool tips can provide useful information about the graphics underneath the mouse cursor in the cross-section views. The same information is also available on the status bar but not as easy to read.



Turn it off if you don't like the tool tips. The graphic information is still available on the status bar even if the tool tips have been turned off.

Accept design bar size outside preferred range: When calculating the Detailed Reinforcement and the Reinforcement Layout from the design areas of reinforcement, RAPT will search within the range of bar sizes nominated at that location by the designer in the input to try to achieve a reinforcement spacing within the code and design limits. If you are willing to accept bar sizes outside this range, select this option. RAPT will check within the nominated range first. If there is no solution within this bar size range, RAPT will then check all other bar sizes for this bar type.

Suppress error report while editing: Basic Error Checking necessary to ensure that the graphics can be drawn properly will be conducted every time an input value has been changed. An error report window will automatically come out to replace the concrete shape graphic view if there are any errors. If you prefer to see the concrete shape graphics view by default even if there are errors to report, this option should be turned on. You can then switch to the error window using the  window toolbar button. This will change the setting of this flag in both the user preferences which are used to control all future RAPT sessions and also the setting for the current RAPT session which is controlled from 4.5.3.4 General Menu->Tools.

Enable Real-time Scrolling: Real-time scrolling means that, when the user clicks and drags scroll bars of a view without releasing the mouse, the view changes its current position continuously giving the user a clear idea where they are. For some slow computer systems, it may take a long time for a view to respond to the real-time scrolling. Turning this option off means the view will only change its current position when the user releases the mouse. So turn off this option if using a slow system.

Circle/Quad Circle Conversion: Allows the user to specify default parameters for the conversion of circular shapes to a series of trapezoidal shapes. This allows the designer to easily define multi-sided shapes such as hexagons and octagons and also to reduce circular shapes to shapes that are more easily manipulated to create partial circular shapes. This will be done automatically if the designer requests a cut across a circular shape. The parameters required are-

- Segments in a quadrant of a circle: Number of segments a quadrant circle shape is to be divided into.
- Segments in a circle: Number of segments a full circle shape is to be divided into.
- Extend Radius: Increase the radius of the circle shape so that the total area of the resulting trapezoidal shapes is the same as the area of the original circular shape. If this is not selected, the corners of the trapezoidal shapes will be placed on the original circle shape surface.
- Flat Top: If not selected, the top and bottom shapes will be triangles with a peak point at the centreline of the circle shape. If selected, the top and bottom shapes will be trapezoids with the top surface flat and the top points equally spaced either side of the centreline of the circular shape.

Levels of Undo: Sets the maximum levels of undo that RAPT will allow. Levels of undo can range between 0 to 20 inclusive. When this value is set to zero, the undo function is disabled. Setting this value to 5 for example means RAPT will remember the last 5 changes the user makes in a data file so that they can rollback at most 5 steps. If RAPT is running on a computer that has limited memory, it is recommended to set Levels of undo to a small value or even zero to accommodate memory problems. The default setting is 5 levels.

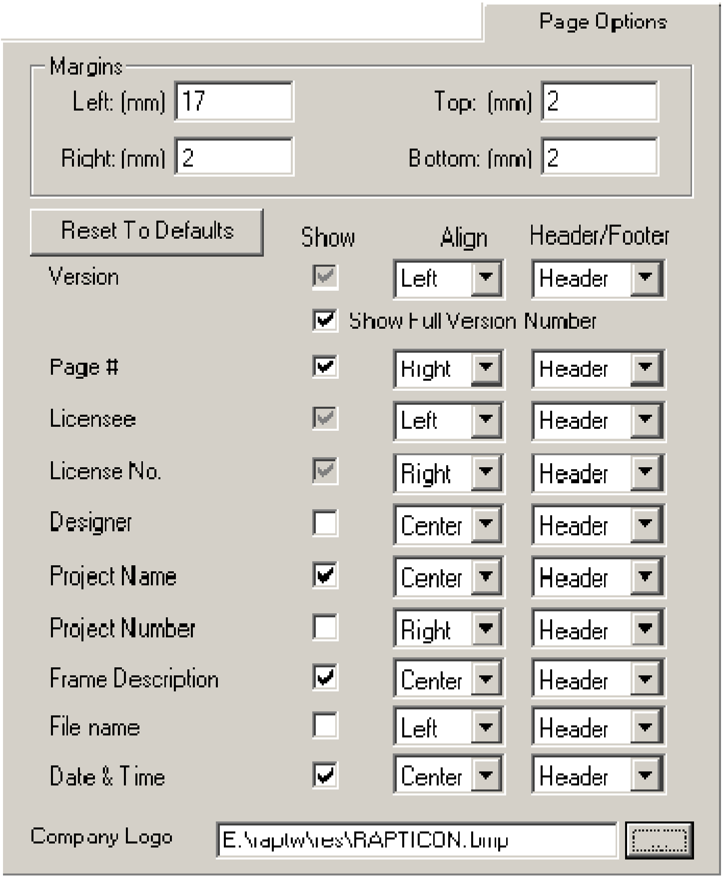
Default Designer: The default designer name will be automatically added to the RAPT frame data file in the General Data View whenever a new RAPT frame file is created.

The *Reset To Defaults* button resets the default settings of the User Options tab.

To cancel changes made to this page click the Cancel button at the bottom of the dialog.

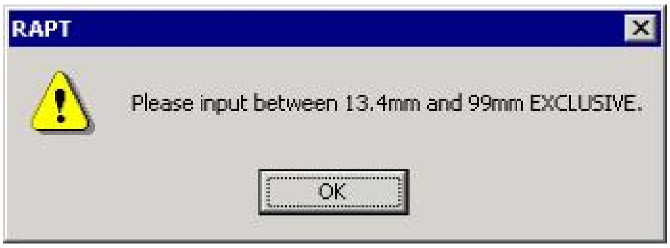
4.2.3 Page Options

Page options tab controls the page layout of a RAPT printed report.



Margins - The left, top, right, bottom margins are the printer margin settings for the page size selected. Different printers have different minimum print margin settings. The correct print margins for one printer sometimes are smaller than the minimum margin allowed for another printer. RAPT allows the user to set the margins between the current printer's minimum allowed margin and 1/3 of the paper width/height. *Tips*: The user can set larger margins on left or right if they need to put punch holes to bind the printed report.

If the values input for the margins by the user violate the printers limits, RAPT will check the values when this page goes out of focus and post the following warning dialog. The user will be forced to modify the settings to conform to the printer minimums. The limits shown in the warning dialog are the mutually exclusive extreme values between which the settings are acceptable.



If RAPT is started with a default printer which has one or more page margin minimums larger than the settings in User Preferences, the offending settings will be modified to the minimums for that printer. This check is also carried out when the user attempts to print to a printer. The changed values will be reflected in this dialog and can be viewed / modified by the user.

Header/Footer - The user may select any of the following elements to be included in the report header or footer.

1 version
2 page number
3 licensee
4 license number
5 designer name
6 project name
7 project number
8 frame description
9 file name
10 date & time

The user can specify whether an element should be

1 left, right or centre aligned.
2 in the header or the footer

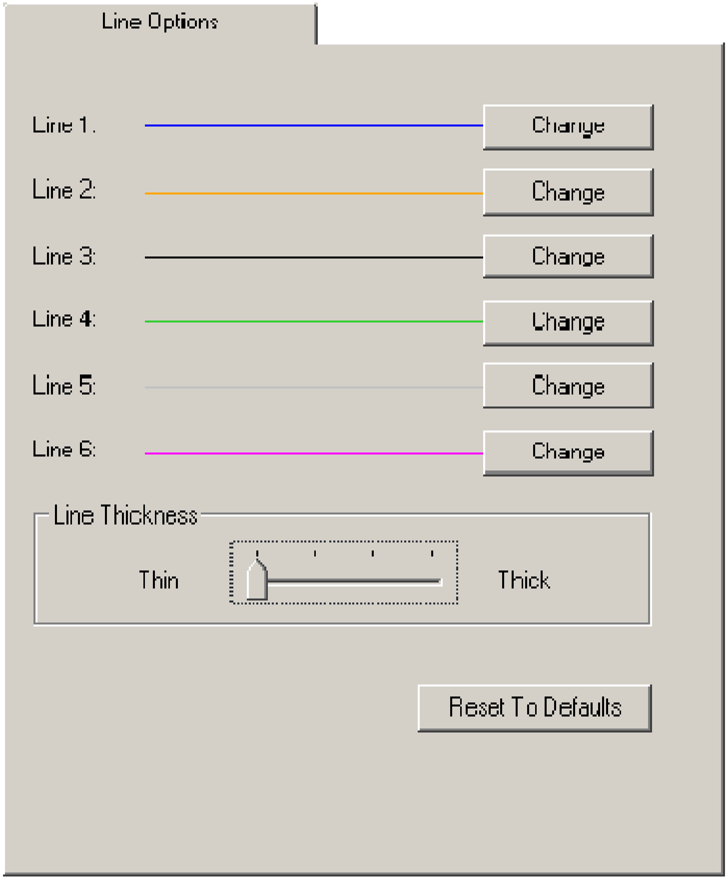
Show Full Version Number option enables user to show the full version number of RAPT (such as 6.0.0.2) or just a simplified version number (such as 6.0) in the report header or footer.

Company Logo field allows the user to specify a company logo bitmap (BMP file) to be printed on the first page of the report. There is no size restriction on the BMP file. If a bitmap image is too large, it will be reduced to a more usable size (50% of screen width for viewing on screen or page width inside margins for printing to a printer). The program will only accept BMP file format, other image formats such as JPEG or GIF will not be accepted. Contact PCDC if you don't know how to convert a JPEG or GIF file to a BMP file.

The *Reset To Defaults* button resets the setting of the Page Options tab to the RAPT default settings.

To cancel changes made to this page click the Cancel button at the bottom of the dialog.

4.2.4 Line Options



RAPT provides 6 defaults lines for drawing graphics output. Each line colour and line style can be changed to suit the user's viewing or printing needs. The default colours have been selected to attempt to provide a good contrast between different lines. To change line style or colour, select the Change button which will open the dialog below.

The line styles that user can choose from are

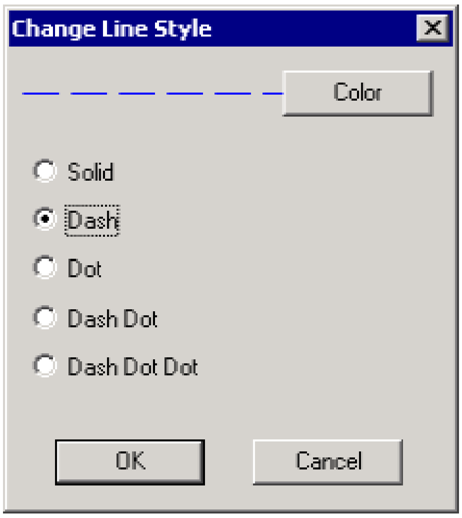
solid:

dash:

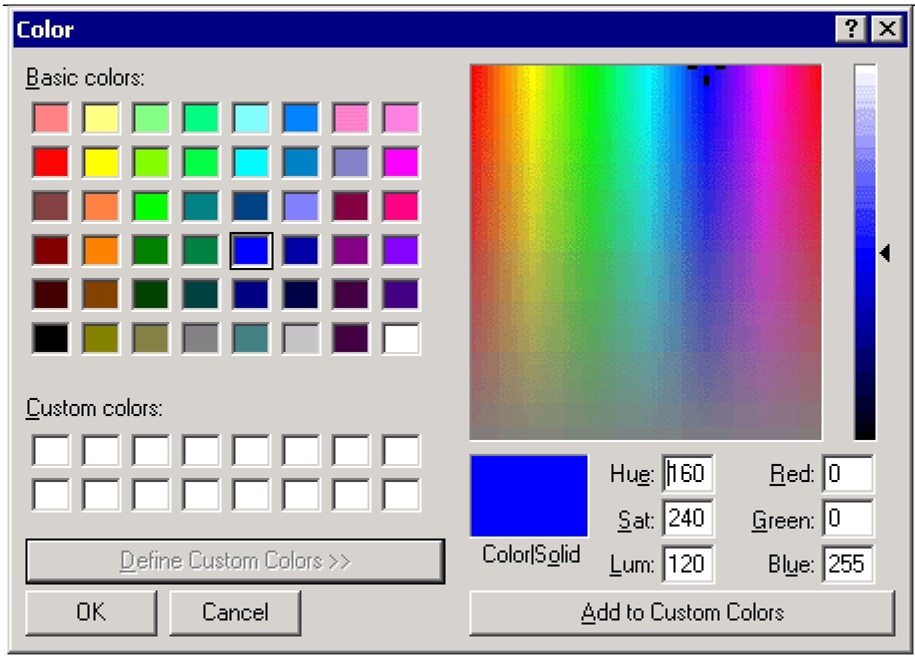
dot:

dash dot:

dash dot dot:



The colours for each line can be modified by selecting the Colour button in the Change Dialog which will present the user with the following colour selection palette from which colours can be selected. RAPT will not carry out any checks on relative colour difference between lines. This is completely up to the user.



These six lines will be used in order by RAPT in drawing graphics plots. If only one line is needed Line 1 will be used, if 2 are needed then Line 1 will be used for the first line and Line 2 for the second and so on.

Line Thickness

This option allows the designer to control the thickness of the main lines on graphics output to the printer. RAPT offers four settings from a minimum thickness to a maximum approximately four times as thick as the minimum. It will not have any effect on viewing a RAPT report on the screen.

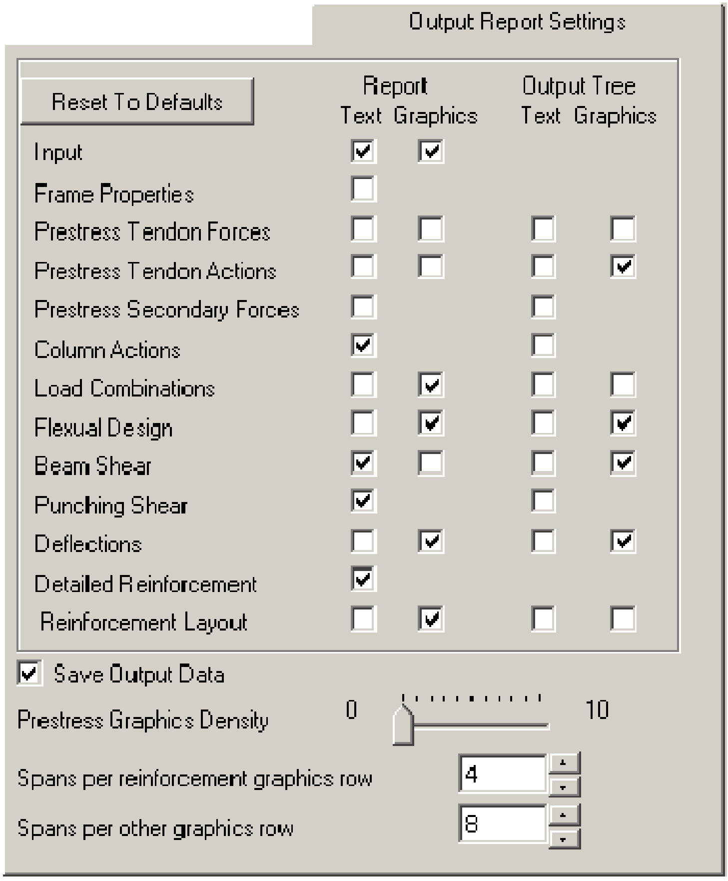
Reset To Defaults

The *Reset To Defaults* button resets the settings of the Line Options tab to the default settings. This will reset the line colours and styles and the line thickness.

Cancel

To cancel changes made to this page click the Cancel button at the bottom of the dialog.

4.2.5 Output Report Settings



The *Report Text and Graphics* columns preset which report elements should be included into the final report as default. When the user resets the Report settings, these options will be selected. The user can then modify the settings if desired before printing the report. In Load Combinations, only Ultimate Flexure within this folder will be selected by default . For Column Actions, both the Load Cases Column Actions and the Load Combinations Column Actions will be selected. In all other cases, all of the text and/or graphics options within that folder will be selected as nominated.

The Output Tree Text and Graphics columns determine which report elements in the report tree should be expanded to be visible when a new RAPT frame file is created. Any other elements in the tree can be opened manually by the user and all can be viewed on the screen. The logic is the same as for Report settings.

Save Output Data When this option is on, all the output data is saved along with the input and materials data so that the next time this RAPT frame file is opened, no calculation is needed. This option will result a larger RAPT frame file (RPF file). When this option is off, the saved RAPT frame file size is smaller containing only the input and materials data, but the next time this RAPT frame is opened, recalculation is needed to produce the RAPT Report.

Set Prestress Graphics Density The density is determined by the profile height spacings and the depth of a character. A setting of 0 will produce the narrowest prestress graphics with the closest pair of profile heights spaced by exactly 1 character height .This includes the standard "border" around a character but does not add any extra spaces between the characters. A setting of 10 will result in a full character height spacing between the closest pair of profile heights. Intermediate selections will result in that number of 1/10ths of a character height spacing between the profile heights. This density is used to determine how many rows the tendon profile must be spread over in the printed output. The final result will normally be a slightly larger spacing than selected as, for example, 1.5 rows may be needed by calculation from the density, so 2 rows will be used and the graphics will be evenly spread over the whole 2 rows.

Spans per reinforcement graphics row controls number of spans that will be printed to a paper width in a reinforcement graphics plot for either the user defined input or Final Reinforcement Layout in the calculated results . Any remaining spans will be wrapped around and printed to the next row/s.

Spans per other graphics row controls number of spans that will be printed to a paper width in graphics other than prestress and reinforcement. Any remaining spans will be wrapped around and printed to the next row/s.

4.3 Keyboard Map

For users who prefer to use keyboard instead of mouse, RAPT provides keyboard commands to do most of the operations. Please note that the keyboard commands work only on the window that has input focus.

Grid keyboard commands

Command	Keys	Description
Repeat	Ctrl + D	Repeat current cell value. A confirmation box will popup if the repeat operation is going to overwrite existing data.
Repeat without confirm	Ctrl + R	Repeat current cell value without user confirmation.
Show right-click menu	Right-click key or Context Menu Key (located close to the right Ctrl key in a Windows compatible keyboard)	Show the right-click menu (context menu) in a grid. The key looks like this: 
Change cell to edit mode	F2	Change the current grid cell mode to edit and select the contents of the current cell.
Exit edit mode	Esc	Exit edit mode, retaining previous data
Deselect cells	Esc	Deselect highlighted cells
Rollback error cell	Esc	Undo user input when data error occurs in cell
Move to next cell	Tab / Enter	Accept cell value change and move to the next available cell.
Highlight cells	Shift + (any) Arrow	Highlight a group of cells
Move to left	Left Arrow	Accept cell value change and move to the next cell to the left. In Edit mode, move one character to left. If at left end of data in cell, accept cell value change and move to the next cell to the left.
Move to right	Right Arrow	Accept cell value change and move to the next cell to the right. In Edit mode, move one character to right. If at right end of data in cell, accept cell value change and move to the next cell to the right.
Move up	Up Arrow	Accept cell value change and move to cell above.
Move down	Down Arrow	Accept cell value change and move to cell below.
Move to left most	Home	Accept cell value change and move to the left most cell
Move to right most	End	Accept cell value change and move to the right most cell
Dropdown dropdown list	Space	Dropdown the list of current cell in a dropdown list cell
Select next control row	Ctrl + Down Arrow	Select next control row while in child grid
Select previous control row	Ctrl + Up Arrow	Select previous control row while in child grid
Next View	Ctrl + Page Down	Select the next tree node when the grid view is in focus and open up the next grid

		view.
Previous View	Ctrl + Page Up	Select the previous tree node when the grid view is in focus and open up the previous grid view.

Output view keyboard commands

Command	Keys	Description
Next Checked Output Item	Ctrl + Page Down	If the current view is an individual text or graphics output view, this command will open up the view for the next checked output item. If the current view is the report view, this command will jump to the next output item in the report view.
Previous Checked Report Item	Ctrl + Page Up	If the current view is an individual text or graphics output view, this command will open up the view for the previous checked output item. If the current view is the report view, this command will jump to the previous output item in the report view.

Tree keyboard commands

Command	Keys	Description
Gain focus	Ctrl + Tab	Switch focus from grid to tree
Next Node	Down Arrow	Select the next tree node when the tree is in focus.
Previous Node	Up Arrow	Select the previous tree node when the tree is in focus.
Expand Sub-tree	Right Arrow	Expand the current node's sub-tree when the tree is in focus.
Collapse Sub-tree	Left Arrow	Collapse the current node's sub-tree when the tree is in focus.
Toggle check box	Space	Check/Uncheck a tree node's check box.
Show/Hide tree	Ctrl + T	Show/Hide all the trees.

Graphic view keyboard commands

Command	Keys	Description
Show Next Item	Ctrl + Right Arrow	Move to next item (span)
Show Previous Item	Ctrl + Left Arrow	Move to previous item (span)
Show Next Point	Shift + Right Arrow	Move to next point
Show Previous Point	Shift + Left Arrow	Move to previous point
Show/Hide Info Dialog	Ctrl + I	Show/Hide information dialog
Toggle Zoom	Ctrl + M	Zoom in/Zoom out of the graphic view

Menu / Toolbar keyboard commands

Command	Keys	Description
New file	Ctrl + N	Create a new RAPT file
Open file	Ctrl + O	Open a RAPT file
Save file	Ctrl + S	Save the current RAPT file

Cut	Ctrl + X	Cut the selected content and put it into clipboard
Copy	Ctrl + C	Copy the selected content and put it into clipboard
Paste	Ctrl + V	Paste clipboard content to current window
Undo	Ctrl + Z	Undo the last change
Redo	Ctrl + Y	Redo the last Undo
Print	Ctrl + P	Print current window
Help	F1	Get RAPT online help
Item Specific Help	Ctrl + H	Get specific help on the current item that has input focus
Create New Frame	Alt + 1	Create new frame file
Create New Cross-Section	Alt + 2	Create new cross-section file
Create New Column	Alt + 3	Create new column file
Create New Profile	Alt + 4	Create new profile file

4.4 Data Entry

4.4.1 Cell Data Types

String (Aa Aa)
Any combination of keyboard characters allowed to a maximum of 127 characters. When editing a string, the editor will place a ' character at the start. This will be removed when the cell returns to Overtype mode or loses focus. Users should add two ' characters if they want a string to start with one as the first will be stripped when leaving the cell.

Integer Number cells (# #)
Integer Numbers are whole numbers. There is no decimal or fraction portion to the number. All numeric characters can be used. If the user attempts to input a decimal place or a fraction or an equation in this type of cell it will result in an error that must be corrected before leaving the cell.

Decimal Number Cells (#.# #.# or unit name such as mm etc.)
Decimal numbers or Floating numbers have a decimal portion whose viewing length in Overtype Mode or when out of focus is controlled by the accuracy defined in the user preferences All numeric characters can be used as well as the formula characters defined in 4.4.6 formulae.

Percentage cells (% %)
Percentage cells are a special type of decimal number cell which show a maximum of 1 decimal places (accuracy level 1) when in Overtype Mode or are out of focus and a full decimal number when in Edit Mode. All numeric characters can be used as well as the formula characters defined in 4.4.6 formulae.

Column Number Cell (# #)
The column number cell is a special format cell which accepts both the reference column number and the distance from the column (the data required in the next cell to position the item). This allows the final position of the item being defined both to be entered in the one cell meaning that the distance from the column can be made consistent with the column number in one operation if they both need to change and there is only one set of error checking and also one set of background calculations based on the modified position of the item. The format of the input is
; ## (for example: 2;1000)
The first Integer Number (#) accepts the reference column number and is followed by a semi-colon delimiter and then the second Decimal Number (#.#) is the distance from the column to the point in the same units as the distance from column data.
The minimum input is the first Integer Number for the column number. A distance dimension is not required. A delimiter must be used if the distance is also to be defined in this cell. If no distance is defined, the column number will be accepted and used in conjunction with the value already defined in the distance from column cell.

Boolean Cell (y / n y/n)
Boolean cells accept only a Yes/No answer. No characters are allowed to be input. Values can be changed by clicking with the left mouse button or by pressing the Space Bar.

Dropdown List Cell (List)
Drop Down lists give a list of the available options to choose from. In some cases this is a multiple column list which provides more information on which the user can base his decision. Values can be changed by clicking left mouse button and choose an item from the dropped list. Values can also be changed by pressing Space Bar to dropdown the list, then Up or Down Arrow Key to choose an item and Enter key to accept the chosen item.

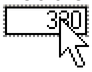
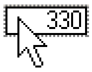
4.4.2 Cell Editing and Navigation

When the data in a cell is modified, RAPT will check to see whether this modification will affect other data in this data file and will make modifications as it thinks appropriate these changes will be 4.4.7 coloured blue in the data to indicate to the user which data has been changed automatically. If the same value is entered into a cell as was there before the edit was commenced, RAPT will not accept this as a data modification and no checking or modification of the data will be done.

String and Numeric Cells
Data cells in RAPT have 2 specific modes

1. Overtype Mode - The whole cell is selected.
   - Character Keys - Pressing any character key will delete the previous data and insert the new character.
   - Movement keys - Pressing and Movement keys will simply move focus to the next cell in that direction.
   - Del Key - The Del key will delete the previous data and leave the cell blank. In both cases, the cell will then be moved to Edit Mode.
   - F2 Key or Space Bar - Pressing the F2 key/Space Bar will move into Edit Mode at the left end of the character string.
2. Edit mode - The cursor is positioned within the character string within the cell.
   - Character Keys - Pressing any character key will insert that character at the cursor position.
   - Del Key - The Del key will delete the character to the right of the cursor.
   - BackSpace Key - The BackSpace Key will delete the character immediately to the left of the cursor.
   - Shift + Left Arrow or Right Arrow - Text within the string can be selected using Shift + Left Arrow or Right Arrow. Pressing character keys will overtype this selection and the Del key will delete it.
   - Esc Key - Pressing the Esc key will undo the current edit operation and move the cell to Overtype Mode.
   - F2 Key - Pressing F2 will accept the modifications to the data and move the cell into Overtype Mode.
   - Horizontal Movement Keys - Pressing Horizontal Movement Keys will move the cursor along the character string one character at a time. When the end of the character string is reached, the focus will move to the next cell and place it in Overtype Mode, accepting the data in the previous cell.
   - Vertical Movement Keys - Pressing Vertical Movement Keys will move the focus to the next cell and place it in Overtype Mode, accepting the data in the previous cell.

To move program focus to a single cell, the user can use

1. Left Mouse Click on the cell. The cell will go into Overtype Mode.
2. Double Left Mouse Click on the cell. If the mouse pointer is within the length of the data in the cell, eg , the cell will immediately go into Edit Mode and place the cursor at that point in the data string. If the mouse click is to the left of the data by more than 1 character, eg 

, the cell will go into Overtype Mode.
3. the Arrow keys - move in the arrow direction and wrap around on same row/column of data at ends of grid. The cell is selected in Overtype Mode.
4. the Tab key - accept modifications to the data in the current cell and move focus to the next cell to right and wrap around at right end to first column in next row. The cell is selected in Overtype Mode.
5. the Enter/Return key - accept modifications to the data in the current cell and move focus to the next to right and wrap around at right end to first column in next row. The cell is selected in Overtype Mode.

Note:-
1 When using the keyboard, the user has to make sure the grid has input focus first before keystrokes will be effective in that grid. If the grid is not in focus, click the grid window area once to gain input focus.
2 A range of cell content can be deleted by pressing the Del key after the range has been 4.4.3 selected. Selecting a whole row or column by selecting the header cell and pressing Del won't delete the whole row or column, nor will it delete the row or column content. Nothing will happen to the selected data. The user has to use the relevant toolbar buttons or right-click pop-up menus to delete rows/columns of data that have been added. Sometimes, row/column insertion or deletion cannot be done until the user specifically selects the row/column (see specific Input Screen chapters).

Dropdown List Cell
Clicking anywhere in a Dropdown List cell will automatically drop down a list of options for the user to choose. Clicking again in the currently selected option will leave the cell in focus but retract the list of options. Pressing the Space Bar or left mouse button when focus is in a dropdown list cell will drop down the list of options for user to choose.
To select an option, click on the desired option with the left mouse button. This will automatically select and accept this option. Alternatively, the user can press Up the (or Left) or Down (or Right) Arrow keys to choose one of the options. Pressing the first character of an option in the list will move the selection to that option. When the desired option is selected, press Return (enter) to accept the selection. If the user does not want to change anything, just want to retract the list that has been dropped down, press Esc key.

If the user only wants to move the current cell to a Dropdown list cell without causing the list to dropdown, use the keyboard movement keys to move the current cell.

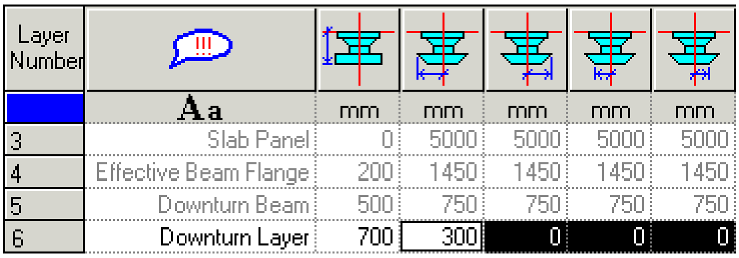
Boolean Cell (Yes/No)
Clicking a Boolean cell will invert the cell value from True to False or from False to True. Pressing the Space Bar on a Boolean cell will invert the cell value from True to False or from False to True.

If the user only wants to move the current cell to a Boolean cell without modifying the cell value, use keyboard movement keys instead of mouse click.

4.4.3 Cell Selection

Any group of data cells can be selected using the mouse and keyboard quick keys.

To select a continuous area, the user can left click a cell and move to another cell while holding the left mouse button. Using the keyboard, this can be done by holding the Shift key and pressing one of the arrow keys. The Cell Selection will not work if the first cell is in Edit Mode (press Esc to change the cell mode from Edit mode to Select mode). To select non-continuous areas, hold the Ctrl key while clicking different cells or dragging through cells.



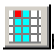
Clicking a row or column header will select a whole row or column. A continuous group of rows or columns can be selected by left clicking a cell and moving to another cell while holding the left mouse button down. Holding the Ctrl key to a click row or column header will select different rows or columns that are not next to each other.

The user can change the current cell within the selected range by pressing one of the arrow keys. Just make sure the current cell does not go beyond the current selection, otherwise, the selection will be cancelled. Using a movement key at the end of its row/column forcing a wrap-around to the other end of the row/column is equivalent to going outside the selected area and the cell Selection will be lost even if all cells in the row/column are selected.

To select Dropdown list cells, it is recommended to use the keyboard as this will not drop down the list of options after selection (press Esc to hide the list). It is also recommended to use the keyboard to select Boolean cells (y/n) as clicking a Boolean cell will immediately inverse its value.

4.4.4 Cell Repeating

Users can repeat the current cell value to other cells by using cell the repeating function. This is invoked by

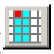
1. pressing button in the toolbar
2. clicking Tools->Repeat menu or press Ctrl + D key
3. clicking Tools->Repeat without confirm menu or press Ctrl + R key
4. clicking the right mouse button in the data cell containing the source data to repeat and select Repeat Cell Data

Repeating can only be done with cells of the same type (for example: Length Cells).

Cell repeating works in different ways depending on the user requirements. These are

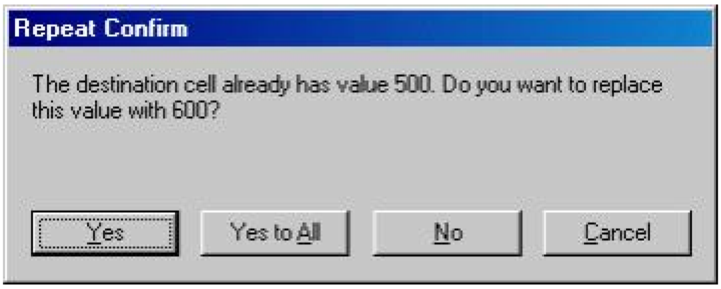
No User Selection of Repeat Cells
Without selection of a range of cells, all other cells in the current column of data will be replaced by the current cell value

1. If a cell that is to be replaced contains a value,

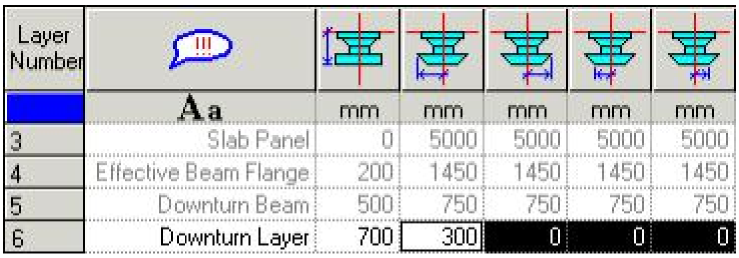
   If method 1 (), 2 (Ctrl + D) or 4 above is used RAPT will ask for confirmation whether to replace the value in that cell or not (see below). The user can select to skip the over-write of data in that cell (Cancel), over-write the data in the cell with the new value (Yes) or over-write the data in all further cells with the new value (Yes to All).

2. If method 3 above (Ctrl + R) is used, no confirmation will be asked for.

If a cell that is to be replaced does not contain a value (zero), in some special cases in various input screens a logical copy is performed from associated data already input e.g. column height below would be set to column height above. See the description in the help for each individual input screen for the special cases where this applies.



User Selection of Repeat Cells
If a range of cells is 4.4.3 selected by the user for repeating the data from a cell, all of the other cells in the selected range will be replaced by value in the current cell without a request for confirmation (same as Ctrl + R) no matter which of the three repeat methods is used. Any non-contiguous group of cells can be selected as long as they are of the same data type as the source cell. When cells are selected, the source cell is the editable cell.



Dropdown list cells and Boolean cells can also be repeated. The tips of selecting a range of Dropdown list cells or Booleans cells can be found in 4.4.3 Cell Selection.

If a validation error occurs at a cell, the repeat process will stop at that cell with an Error Coloured Cell (Red background) and all selected cells after this cell will not be modified. It is then up to the user to fix the data in the error cell until it is acceptable and then select any other cells that are to be changed and repeat the data into them as above.

4.4.5 Imperial Length

RAPT is capable of handling imperial length units as either
1 decimal feet
2 feet, inches and fractions of an inch
3 or inches and fractions of an inch.

The imperial length format must follow several rules:
1 Foot Part, defined as a whole number with foot symbol ('), immediately followed by the first inch character or by a dash (-) or space character as delimiter. Fractions of a foot are not accepted, decimals may be used but not in conjunction with inches.
2 Inches Part, defined as a whole number representing whole inches (may be omitted if there are no whole inches in the number), followed by a space character, followed by fraction of an inch, followed by inch symbol (").
3 If no symbols are defined and the number format is in the form # # #/#, the input will be assumed to be feet, inches and fractions of an inch.
4 If a fraction is defined, the number before it is assumed to be inches no matter what type of cell it is in unless feet and inch symbols are provided. Fractions of a foot must be input as a decimal (if feet only are input) or as inches.
5 If either the foot symbol or the inch symbol is used and there are feet and inches combined, both must be used.

The foot/inch imperial format can be used in length cells of any unit type. The input string will be converted to the input type for the data cell. This is shown in the table below.

When converting to fractions of an inch internally, RAPT will always convert the fraction to the lowest common denominator, so 8/16" will be represented as 1/2" even if the accuracy defined for the cell is 1/16".

When used in equations, it is always best to include the foot and inch symbols to avoid misinterpretation of the number.

The following table gives examples of what can or can not be accepted by RAPT.

Input	In feet cell with 1/16" precision	In inch cell with 1/16" precision	In millimetre cell with 0.1mm precision
2	OK, 2 feet	OK, 2 inches	OK, 2 mm
2.3	OK, 2.3 feet	OK, 2.3 inches	OK, 2.3 mm
1 1/2	OK, 1 1/2 inches	OK, 1 1/2 inches	OK, 38.1 mm (which is 1 1/2 inches)
1 1/2"	OK, 1 1/2 inches	OK, 1 1/2 inches	OK, 38.1 mm (which is 1 1/2 inches)
1-1/2"	Wrong, missing foot symbol	Wrong, missing foot symbol	Wrong, missing foot symbol
1'-1/2"	OK, 1 foot and half inch	OK, 12 and a half inches	OK, 317.5 mm (which is 1'-1/2")
1' 1 1/2"	OK, 1'-1 1/2"	OK, 13 inches and a half	OK, 342.9 mm
1 1 1/2	OK, 1'-1 1/2"	OK, 13 inches and a half	OK, 342.9 mm

4.4.6 Cell Formula Input

The Decimal Number cells can accept formula input. The user can input a formula and the program will calculate the result and put it into the cell. If the formula is copied before calculation, it can be pasted into another cell if it is in edit mode and edited before being calculated in the new cells. If it is pasted into a cell in Overtype Mode, it will paste as the result. After the result is calculated in a cell the formula is lost and only the result is retained for that cell.

For example: Entering 1 + 2 * 3 will put the result 7 in the cell.

For length cells, imperial length format can also be used in the formula along with decimal numbers. The user has to make sure the correct 4.4.5 imperial length format is used.

Helper math functions that can be used in formula:

Function	Description
+	Plus
-	Minus
*	Multiply
/	Divide
( )	Brackets
ABS(X)	Returns the absolute (positive) value of X.
ACOS(X)	Returns the arc cosine of X in radians. X is a numeric value between -1 and 1.
ASIN(X)	Returns the arc sine of X in radians. X is a numeric value between -1 and 1.
ATAN(X)	Returns the arc tangent of X in radians.
COS(X)	This function returns the cosine of X in radians.
DEGREES(X)	Returns the value in degrees. X: radian value
EXP(X)	Returns e raised to the power specified by the argument, where e is the base of the natural log 2.71828183.
LN(X)	Returns the log base e of X, where e is the base of the natural log 2.71828183.
LOG(X)	Returns the log base 10 of X.
LOG2(X)	Returns the log base 2 of X.
RADIANS(X)	Returns the value in radians where X is the degree value.
ROUND(X, N)	Returns X rounded to the number of decimal places specified by N (when N is positive); it returns X rounded to a whole number when N is negative.
SIN(X)	This function returns the sine X in radians.
SQRT(X)	Returns the positive square root of X.
TAN(X)	This function returns the tangent of X in radians.
X^Y	Returns X to the power of Y
X**Y	Returns X to the power of Y

The help function names are not case sensitive.

Examples:

Formula Entered	Result
ROUND(COS(60), 2)	-0.95
DEGREES(ASIN(0.5))	30
sin(radians(30))	0.5
2^3 + log(100) + 16 ** 0.25	12

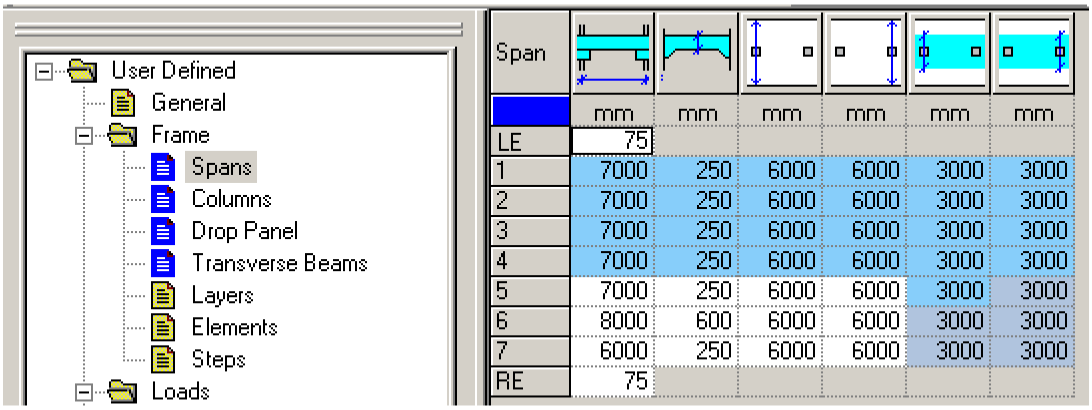
Formula Restrictions:

1. Formulae do not accept reference cell values. For example, adding cell value from row1, column2 and row2, column2 together into row3, column 2 is not possible.
2. Formulae are not remembered by each cell. Only the calculated result is remembered.
3. Although the user can use foot-inch format in a formula in any Length cell, sometimes they cannot be distinguished from the other numbers and operators. It is recommended not to omit the foot and inch symbols in formulae and to put brackets around feet-inches values.

4.4.7 Cell Colours

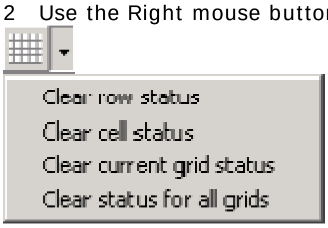
The grid cells are sometimes given different background colours to indicate their different states.

Data changed automatically by RAPT:



Sometimes editing one cell value or inserting data will cause RAPT to automatically change other dependant values in other cells in the grid or in other grids. The values changed by RAPT will be shown in light blue background like this . Any previous blue marked cells will be shown in a darker blue background like this . The corresponding tree node will change to blue as well. The blue cell colours are to indicate to the user that automatic changes have been made in the background. Sometimes the calculation of these changes will have required interpretation by RAPT and the solution it chooses may not be the one preferred by the user. The user should check all changes made by RAPT to make sure they are acceptable.

To remove the blue colour from a cell (the tree colours will automatically clear when the associated grids are clear),
1 Modify the data in a cell. This indicates to RAPT that the user has looked at the cell and that cell colour is cleared.
2 Use the Right mouse button in the coloured cell or select the Clear Cell Colours Toolbar Button

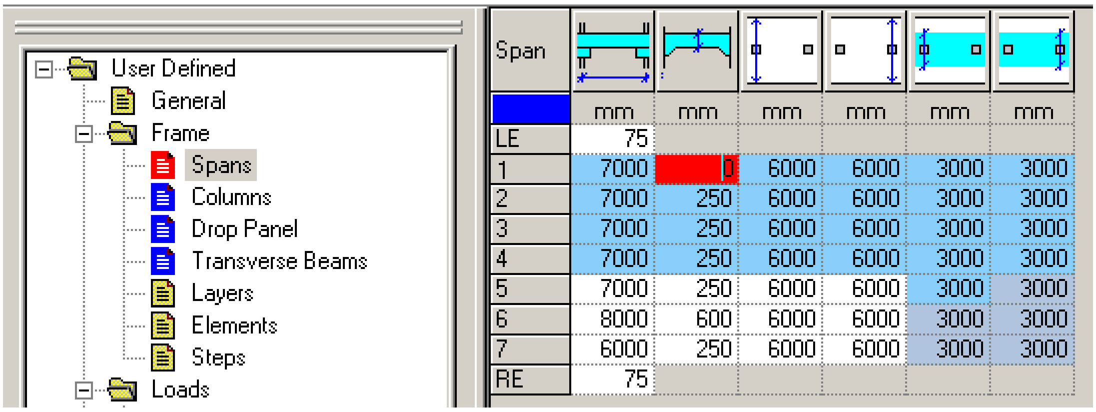


. Various options will be presented for clearing cell colours in both cases. These are

1. Clear all cells in the current data row.
2. Clear the current cell.
3. Clear all cells in the current grid
4. Clear all cells in all grids.

Remember that these colours have been added to show you what has been changed in the background in your data. Make sure you check the changes before clearing all of the cell colours.

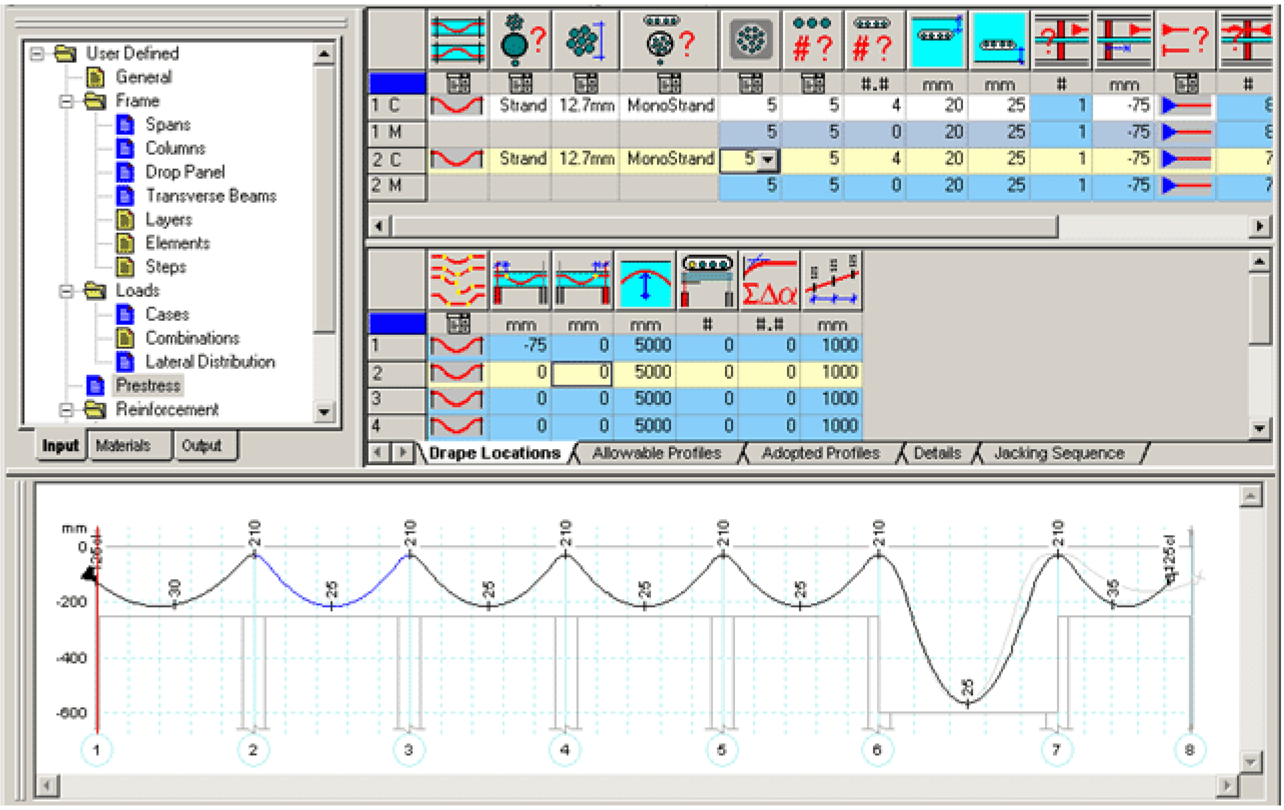
Error value:



When a cell input is invalid, the cell background will be changed to red like this . The corresponding tree node will change to red as well. To clear the colour, the user must input an acceptable value, or press Esc to return to the previous value in the cell. No other cells can be edited until this error is cleared. This type of error checking is only done in cases where an incorrect value will cause problems in the background input calculations or in the drawing of the graphics. All other checking is only done on user request or when the calculations are requested.

Current Control Row:

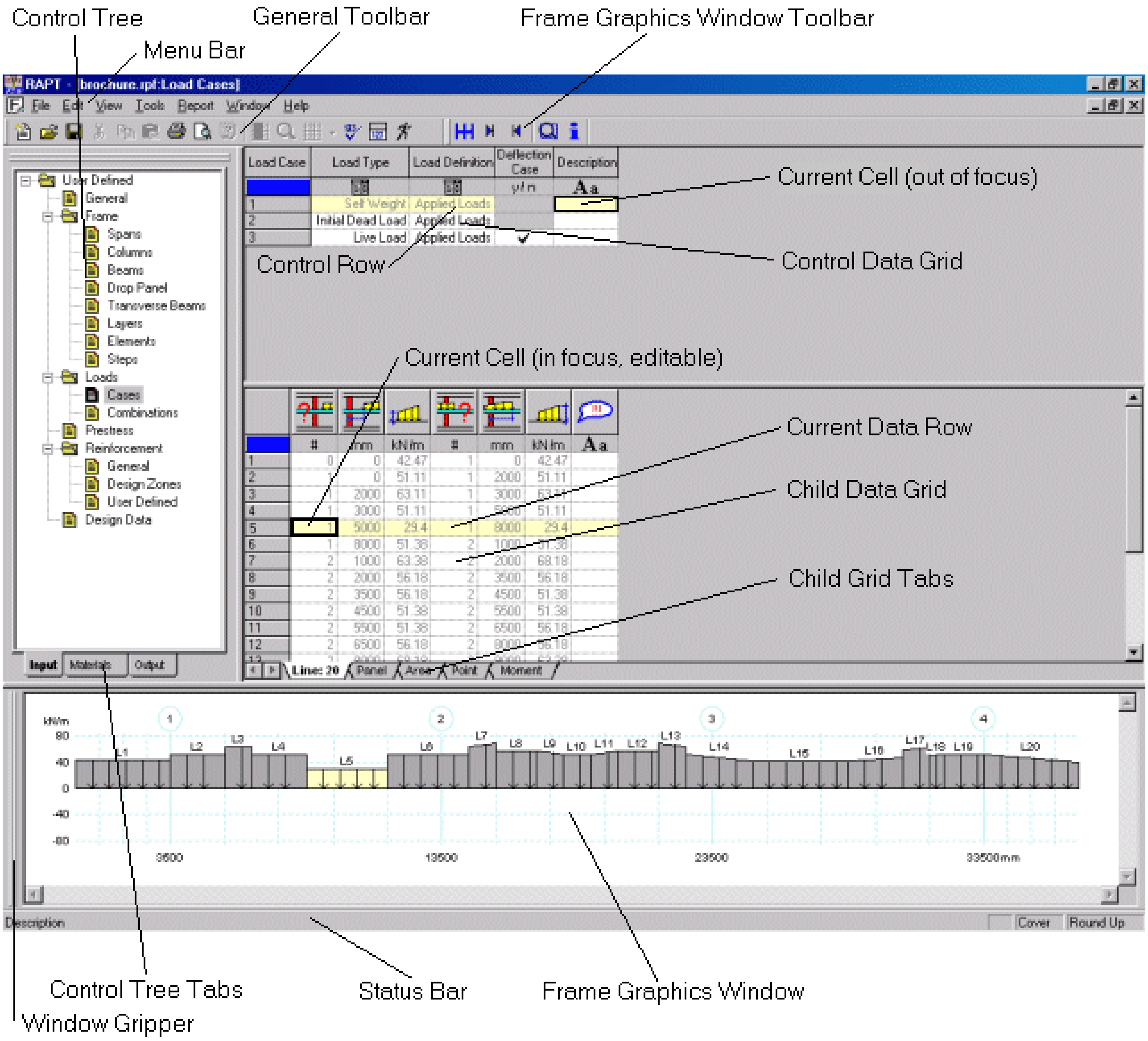
In situations where two views have parent-child relationship, the content of the child view depends on the current control row of the parent view. A high-light colour  is used to indicate the current control row. The current control row in a grid automatically moves to the row containing a selected cell so changing rows with the mouse or arrow keys will make the new row the current control row. The only time a new control row will not be selected is when a group of rows is selected with the mouse.



In the above case, Prestress row number 3 (with row header 2 C) in the top data grid is the control row for Drape Locations in the centre grid: so the drape locations data shown is for tendon 2 column strip (row 3). Drape Locations row number 2 is the control row for prestress graphics so the coloured span in the graphics is for span 2 (row 2) of tendon 2 column strip.

4.5 Screen Layout

4.5.1 General Screen Layout Principles



We will use the Load Cases screen to give an overview of the general screen layout principles used in RAPT. Most of the concepts used are standard Windows interface concepts and we will not go into detailed explanation on how these work.

The overall RAPT screen is divided into several windows with different functionality as explained below. The size and shape of the various windows can be adjusted by moving the mouse cursor over the boundary between two windows until the cursor changes to one of the following symbols ⟺ ⇕ . Then depress the left mouse button and drag the window border in the direction of the arrows in the cursor to the location desired on the screen. In some cases e.g. frame shape graphics with section views, the section view windows will automatically adjust in size when the frame shape window is adjusted.

A window with a gripper at one edge can be moved to different positions on the screen to suit the user and can be undocked from the current position either by double clicking on the gripper or by selecting and dragging the gripper to another location. Double clicking on an undocked window will re-dock it in its previous position. Moving it by selecting and dragging the gripper will also re-dock the window at an edge when the mouse reaches the edge of the RAPT screen space.

Menu Bar

RAPT uses a standard Windows style menu bar that users can use to access specific functionality of RAPT. This can be accessed with the left mouse button of Alt + Letter from the keyboard.

Status Bar

RAPT uses a standard Status bar. Tool Tip information is repeated on the left of the status bar and is often more detailed that the Tool Tip information.

Toolbars

RAPT does not use a single toolbar which is available at all times. Instead, each window has it's own toolbar for operations within that window only. This saves users from the situation of having enormous toolbars which waste a large amount of screen area and are difficult to navigate. A General Toolbar is provided at the left of the toolbar row which controls the standard functionality common to all windows such as edit, view, printing etc. As well, where needed, extra toolbars will be added to the right of the general toolbar. In the case shown above, with computer focus in the Frame Graphics Window, the Frame Graphics Window Toolbar is available for operations in the graphics window. If focus was in the Control Grid Window, the Control Grid Toolbar would be available, etc.
To make the toolbar for a window available, move focus to that window by clicking the left or right mouse button somewhere within that window area.
See discussion on each area of data for information on specific toolbars.

Control Tree

Access to different areas of data is controlled from the Control Tree. In the case shown above, the tree has 3 Control Tree Tabs which give access to different types of data,

1. Input data,
2. Materials data
3. Output Results data.

Within each Tab, the user can select different areas (pages) of data to view.

The Tree is a branch structure of data contained in Folders, Sub-Folders etc to the bottom level which are Pages. Folders are represented by when they are closed. When clicked on, they open, shown thus to admit access to Sub-Folders and eventually to Pages represented by which simply contain data (or may perform an operation e.g. Report in Output). A folder normally will simply allow access to sub-folders or pages within it but, in special cases, may open a window with it's own data which is related to the data in the Sub-Folders and pages it contains, e.g. Materials->Prestress Strand contains the strand size data and the sub-folders and pages have tendon and anchorage data for each strand size.

A folder may be opened or closed by left mouse click on the + or - to the left of the folder or by double clicking on the folder name or by using the Right Arrow key (open) or Left Arrow key (close).

A page may be opened by a single left mouse click on the name or by using the Enter key. Program focus will immediately move from the Tree to the Data Window that has been opened.

The tree can be traversed using the Up Arrow, Down Arrow, Page Up and Page Down keys. Also, the first character of a Folder or Page name can be used. Pressing the same character a second time will move to the next instance of that character starting a name.

If program focus is in a Data or Graphics Window, Ctrl + Tab can be used to move focus to the tree where the normal key strokes can then be used to move to and select a new page of data to view.

Data Windows

Data is presented in Grids which are in separate windows on the screen.

In the simplest form, a tree page will open one data window with a single grid of data in it. In this form there is only one set of the data, e.g. 7.2.3.1 Spans Data where there is a single grid of data for all of the spans in the frame.

As the data becomes more complex, there is often a requirement for multiple sets of the same type of data which cannot be placed in a single grid of data. The Load Cases Data shown above is a good example of this. For each load case, the user can define the loading in different ways e.g. as a series of applied loads or as a set of bending moment and shear diagrams. Also, for each load case defined as a series of applied loads, different types of loads can be defined e.g. line loads, panel loads, point loads etc.

To handle this type of situation, RAPT has introduced the concept of Control (or Parent) Data Windows and Child Data Windows. The Control Data nominates the number of instances of the Child Data that exist and also any general data that applies for that instance of the Child Data. In the Load Cases data shown above, the Control Data contains a grid of data that provides an individual row for each load case, Self Weight, Live load etc. This includes data about the load case including the load case type, load definition type and a description of the load case. For each row in the control grid (for each load case) there is a set of data defining the loads in that load case. This data is shown in the Child Data Window. Only one Child Data Window can be viewed at one time, for the currently selected Load Case in the Control Grid. This Child Window can have as many loads as the user wishes to define to make up the complete load case.

To change focus to the other window and it's current cell, simply click the left or right mouse button inside the boundary of the that window. The current cell for editing will immediately change and the toolbar for that window will replace the previous toolbar.

One row in the Control Grid must always be selected and this is the Control Row. The background of the Control Row for which the data is being shown in the Child Window is coloured yellow. Different control rows can be selected using the mouse of the cursor keys when the control grid is in focus.

Grid Tabs

In cases where multiple data types are associated with a single Control Row, RAPT uses Grid Tabs to provide several screens of data in the one window. This is shown in the Child Data above where there are separate sets of data for each load type of applied load, Line Load, Panel Load, Area Load, Point Load, Point Moment. Select a tab with the mouse to view the data in the grid for that load type and also to gain access to the toolbar associated with that data. In this case, the number of applied loads defined in load a type is also printed in the Tab associated with that load type along with the name.

Current Cell

The current data cell for editing is shown with a dark border. In cases where there are two grids of data on the screen, as shown above, there will be a current cell in each grid. However, only the current cell in the window that has focus is immediately editable. This cell will have a double line thickness border as shown above in the Child grid. This is also a quick way to determine which window has input focus. To change focus to the other window and it's current cell, simply click the left mouse button inside the boundary of the that window.

Frame Graphics Window

The Frame Graphics Window displays the data for the data view currently in focus. In the case of a complex set of data as shown above, it shows the loading diagram for the Current Control Row.

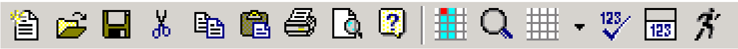
Clicking on a data item in the graphics will highlight that item and also the data that controls that item. In this case the Current Data Row in the child grid will adjust automatically and if the load type selected is different to the load type that is already showing, RAPT will change the load type showing in the Child Grid. The Current Data Row is shown highlighted with a yellow background. The item selected in the graphics window is shown highlighted with a yellow background in the case of loads. In other cases where only a line is being shown, e.g. prestress tendons, concrete reinforcing bars, it is shown as a blue line.

See the discussions on the various data screens for information on what is selectable in each case and how it will be selected in the data.

Message Window

The Frame Graphics Window also contains the 4.5.4 Message Window which is used to list the warnings and errors generated during the data check or when calculating the results data.

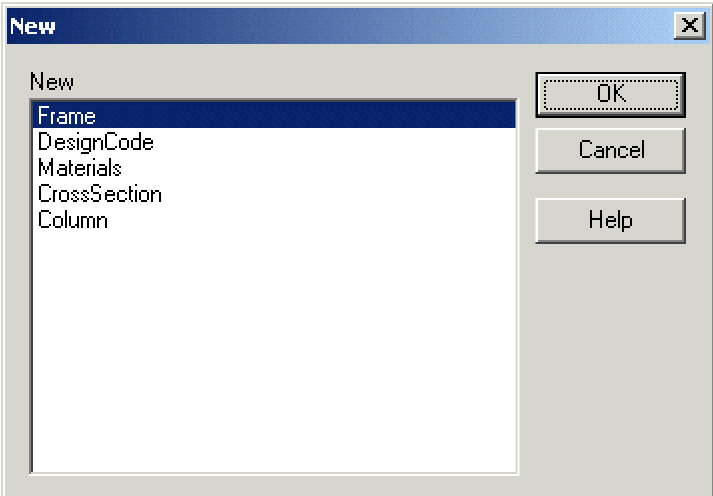
4.5.2 General Toolbar Layout



This toolbar will be available once a data file has been loaded or initiated. It allows user control over general functions within a run. The functions available are

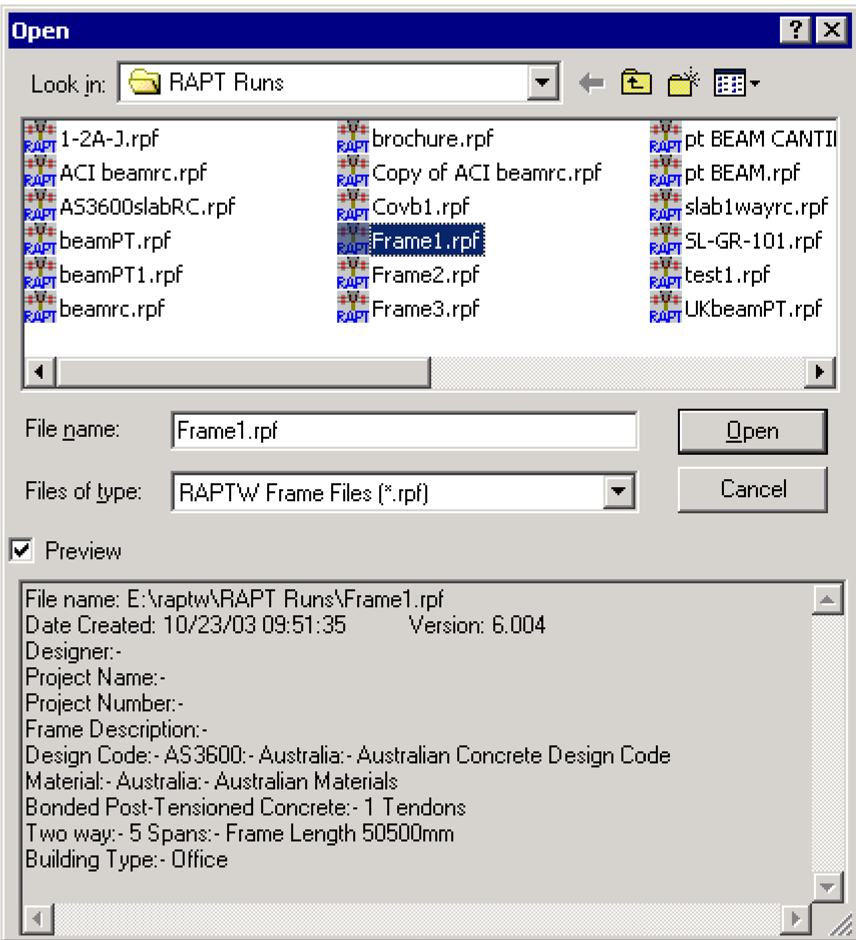

New (Ctrl + N)

Allows the designer to launch a new data input for any of the types of data files handled by RAPT. Any open data file will be closed automatically allowing the designer to save or not if necessary and the following dialog will be presented from which the designer can choose the new data file type.




Open (Ctrl + O)

Allows the designer to open an existing data file for any of the types of data files handled by RAPT. The default type presented will be the same as the last file loaded. The following dialog will be presented to the designer to choose the file to open.



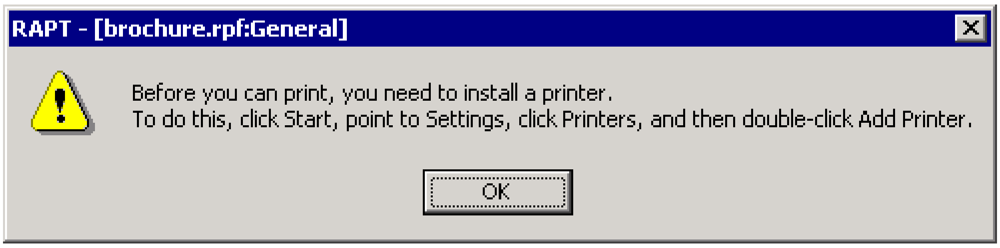
Save (Ctrl + S)
Save the current data. If the data has not been saved previously, a dialog will be presented for the definition of a file name and path.

Cut (Ctrl + X)
Delete the currently selected data and copy the data to the Windows Clipboard.

Copy (Ctrl + C)
Copy the currently selected data to the Windows Clipboard.

Paste (Ctrl + V)
Paste the contents of the Windows Clipboard to the current focused cell in the Data view. If a range of data items has been selected, the same range of cells must be available for pasting. RAPT will not create more rows of data automatically to fit the data being written to it. The data cells into which the data is to be written must be of the same type as the copied data.

Print (Ctrl + P)
Print the contents of the window currently in focus to the printer. This will print the contents of any window in RAPT. The standard Windows Printer Dialog will be presented. The default printer will be the current Windows Default printer the first time this RAPT session is printed from. Within a RAPT session, the last printer selected and printed to will be remembered for future printing operations. RAPT will check for the default printer when starting up. If there is no default printer there will be a reasonable delay before RAPT starts up. When printing, if there are no printers defined on the computer RAPT will present the following message dialog.

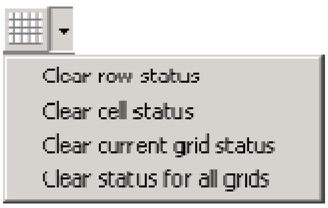


Print Preview
Same as Print except that RAPT will print the current window contents to the Windows Print Preview Window. This can be used for viewing the print format and direct printing to the printer.

Item Specific Help (Ctrl + H)
Will take you to the specific help for the data item selected with the cursor.

Repeat Cell Data
Users can 4.4.4 repeat the current cell value to other cells by using cell the repeating function. This button also works with the special repeat functions in some data grids where there is a logical column of data to copy into the current column.

Simple / Detail
In some data screens, the user is offered a choice as to whether a simple shape is to be defined or a complex one. Two examples are drop panels and transverse beams. This button will only be available in screens where it can be used.



Clear Cell Status
When RAPT automatically changes data in response to a user modification to associated data, the data cells modified are coloured blue. This button or the Right Mouse button clicked in a coloured cell will give you a method of 4.4.7 clearing these coloured cells. Make sure you have checked the changes before clearing all of the cells.

Check Logic
RAPT checks all data for inconsistencies before calculating results. This button allows the designer to force a check on the data at any time. Any warning or error messages will be printed in the 4.5.4 Message Window.

Show Message Window
The 4.5.4 Message Window and the 4.5.1 Frame Graphics Window share the same screen space in the RAPT interface. Use this button to switch between these windows. When the button is expressed, the frame graphics window will be in

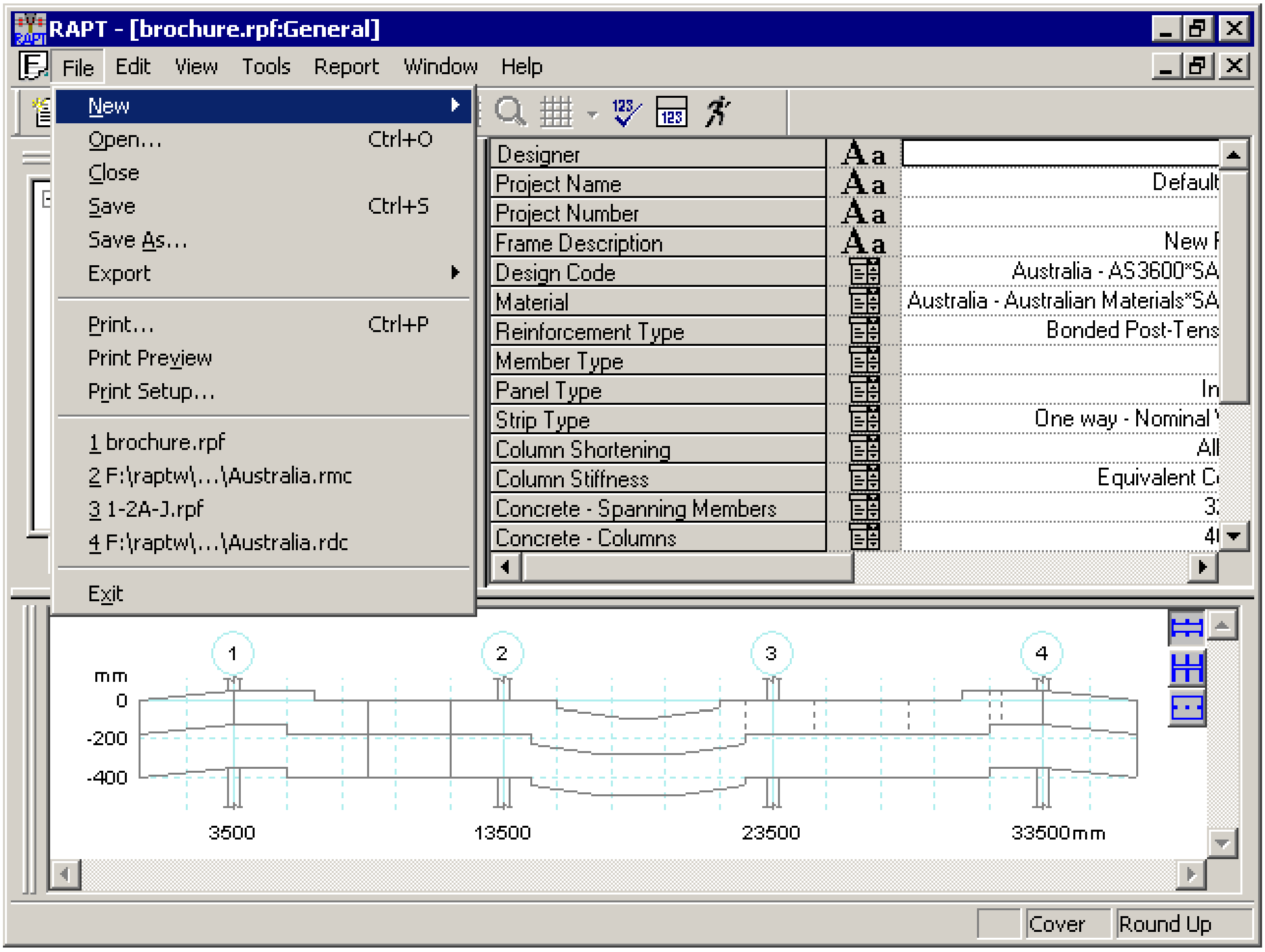
view. When it is depressed the message window will be in view. RAPT will automatically switch to the Message Window when  is pressed.

Calculate Results

Will carry out a full data check and calculate the results for the defined data. These can then be viewed from the Output Tab on the 4.5.1 Control Tree. See the Results section of each data type to see the results data that is available.

4.5.3 General Menu

4.5.3.1 General File Menu

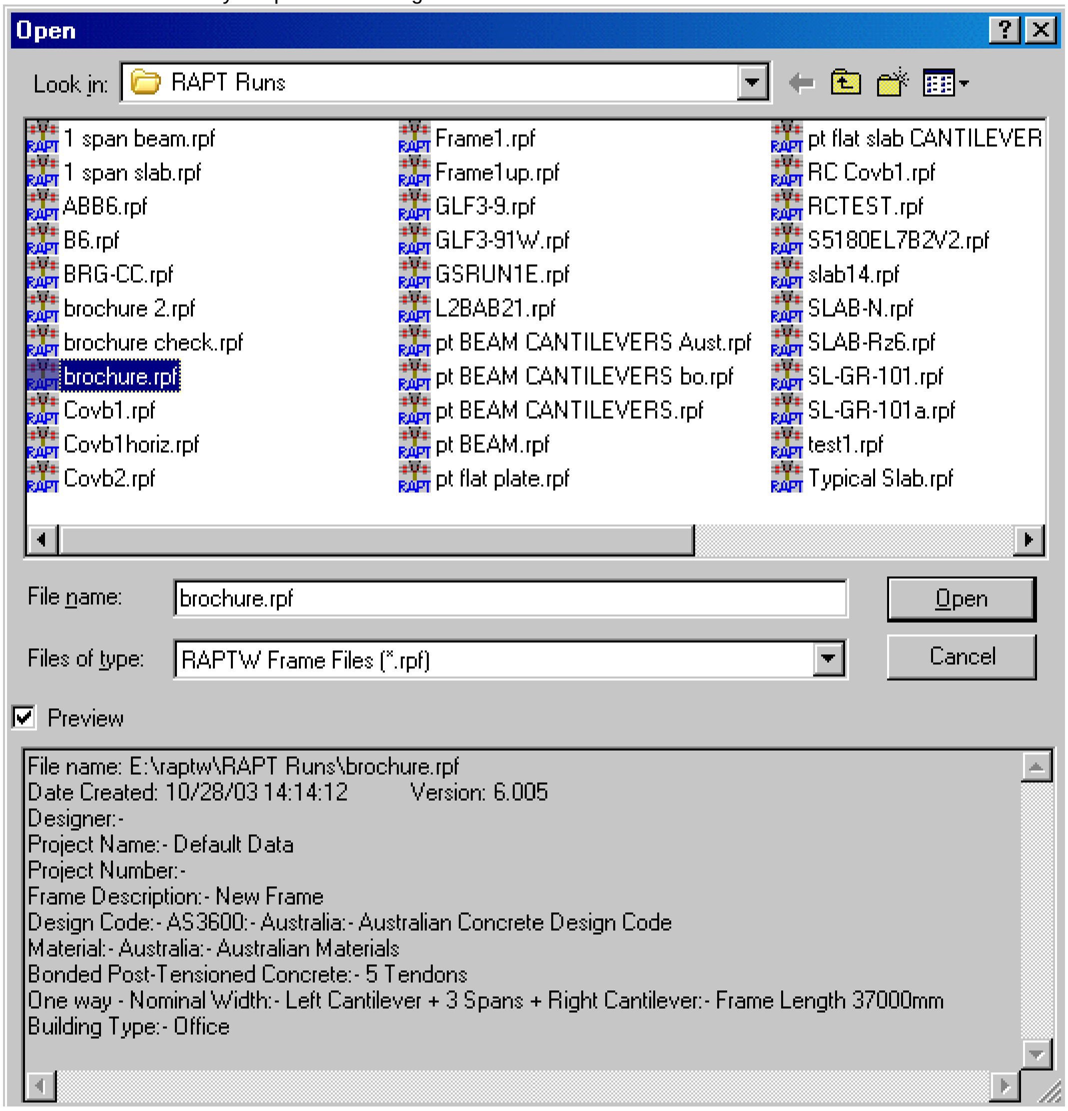


New
Allows the user to create a new design file. Four types of design file can be created in RAPT

1. Frame (Alt + 1) - The user can define a 2D concrete sub-frame for automatic analysis, design and detailing as a reinforced or a prestressed concrete member.
2. Cross-Section (Alt + 2) - The user can define a single complex cross-section of a member with reinforcement and prestressing tendons for detailed design.
3. Column (Alt + 3) - The user can define a column shape with reinforcement and prestressing tendons and produce various interaction diagrams for it.
4. Tendon Profile (Alt + 4) - The user can define a prestress tendon geometry in terms of high and low points and produce a tendon profile drawing with calculated extensions

Open (Ctrl + O)
The user can open one of the design files mentioned above or a Design Code file or a Materials file for editing using the

standard windows style open file dialog below..



For Design files, a preview pane has been provided at the bottom which lists some relevant data from the selected file to help with file selection.

Close

Close the current file. If the data, either defined or results, has been changed in the file the user will be asked whether or not to save the data with the following dialog.

Save (Ctrl + S)

Save the data for the current file. If a file name has not been defined yet, RAPT will prompt for a file name and path to save to. The default name offered will be the text in the 7.2.2 Frame Description. If no text is entered here Frame1 will be used, If the name given at this stage already exists, RAPT will prompt whether or not to proceed and overwrite the existing data.

Save As

Save the current data to a new file and path. The existing file will still exist with the data already saved to it previously. Any changes that have been made to the file since it was last saved will only be made in the "save as" file and will not be made in the existing file.

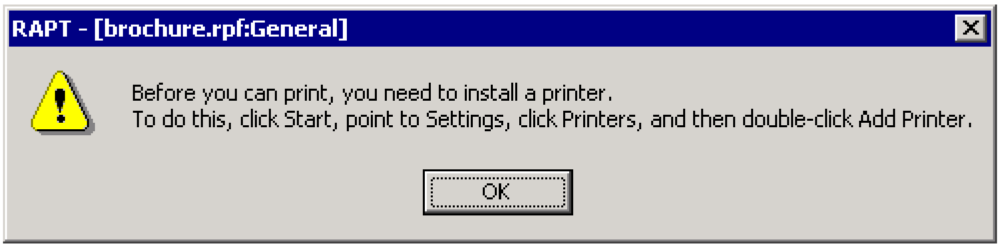
Export

Not implemented yet.

Print (CTRL + P)

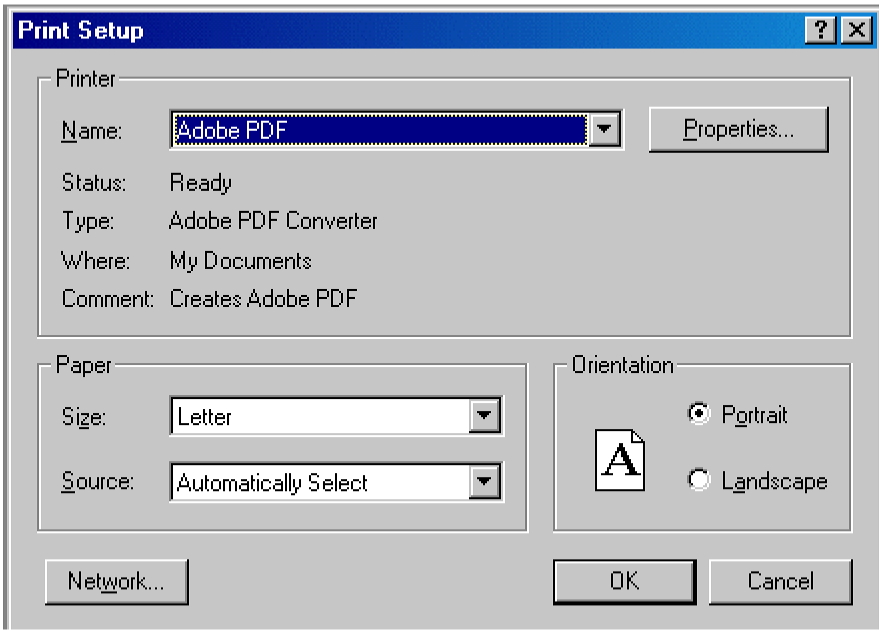
Print the contents of the window currently in focus to the printer. This will print the contents of any window in RAPT. The standard Windows Printer Dialog will be presented. The default printer will be the current Windows Default printer

the first time this RAPT session is printed from. Within a RAPT session, the last printer selected and printed to will be remembered for future printing operations. RAPT will check for the default printer when starting up. If there is no default printer there will be a reasonable delay before RAPT starts up. When printing, if there are no printers defined on the computer RAPT will present the following message dialog.



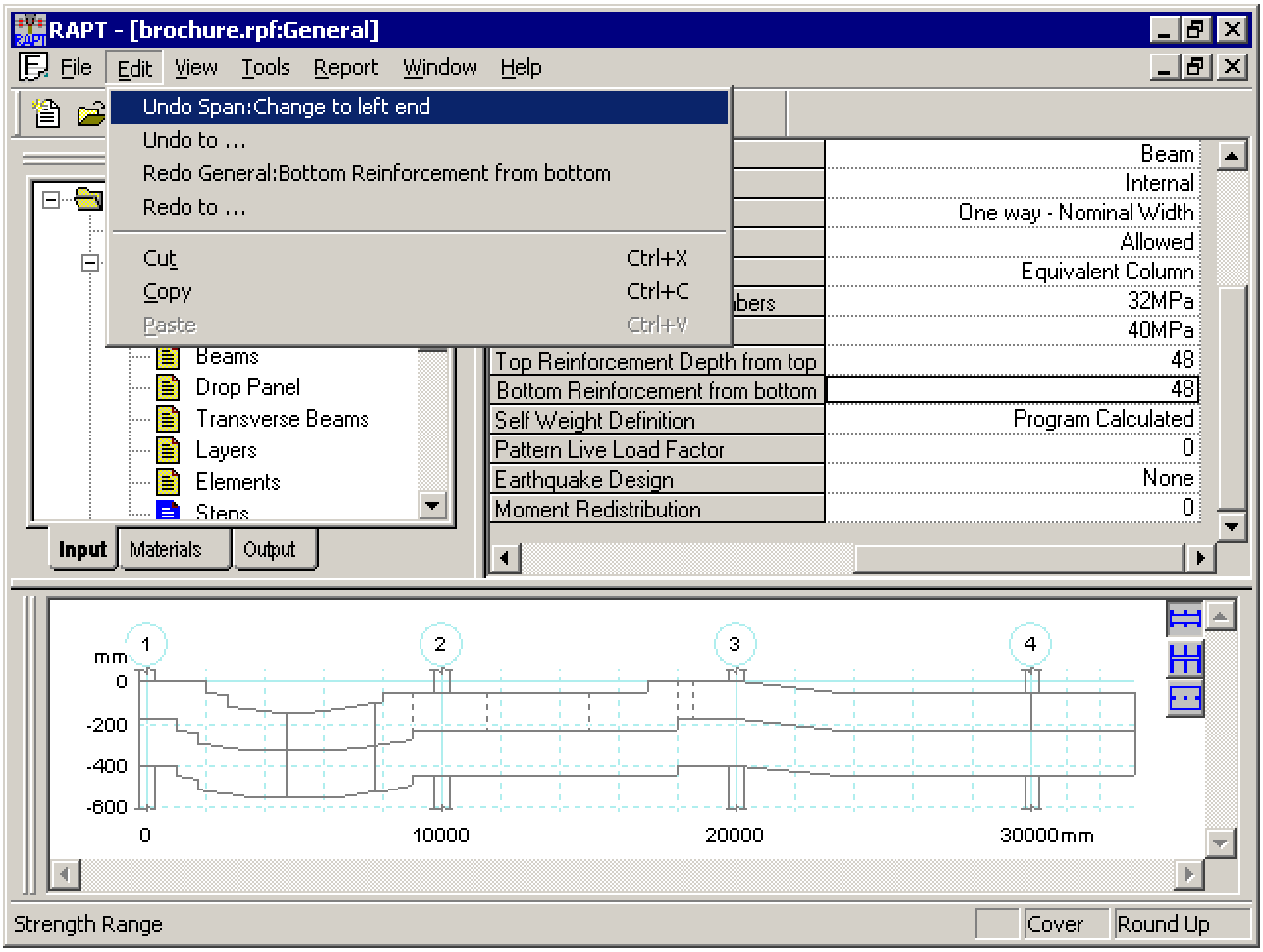
Print Preview
Same as Print except that RAPT will print the current window contents to the Windows Print Preview Window. This can be used for viewing the print format and direct printing to the printer.

Page Setup
Allows the user to define the printer settings for the current RAPT session using the standard printer settings dialog shown below. The default settings are the default Windows settings. Changes made to printer settings in this dialog will be lost when another printer is selected or the current RAPT session is closed. To make permanent changes, right click on the desired printer from the Windows Start->Settings->Printers menu and select Properties. Changes made there will be permanent.



Recent Files
Lists the last 9 files opened in RAPT for quick access.

4.5.3.2 General Edit Menu



Undo/Redo commands

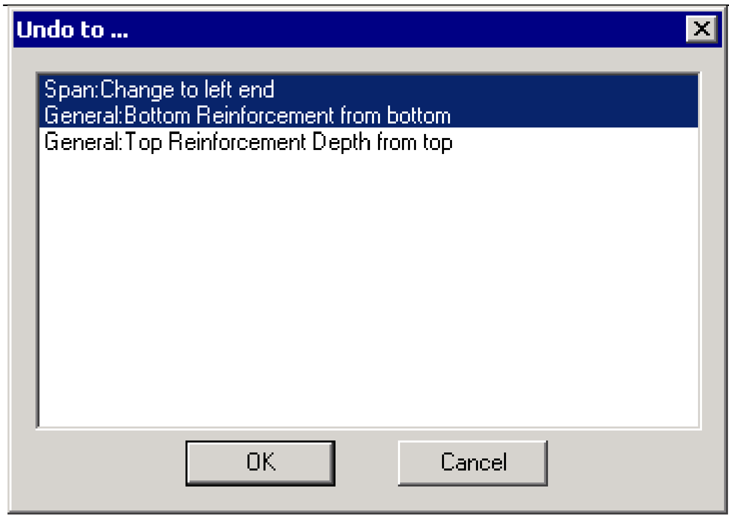
If the Undo/Redo function is turned on, the data state existing after each data change is made will be remembered so that the user can rollback the data to a previous state if desired. If the user rolls the data back too far, the state can be rolled forward using redo. Refer to 4.2.2 User Options to turn on or off Undo/Redo function or to change the number of levels of undo available. RAPT remembers up to the number of data states nominated in 4.2.2 User Options and the data change that caused each new state to be created. The designer can then use the Undo/Redo functions described below to revert to any one of the saved data states thus Undoing the effect of data changes made after that data state.

After a new data state is selected using any of the Undo and Redo functions below, the selected data state is the current data state. If another change is then made to the data, the data states after the current state in the list will be lost and the new changed data state will be added to the end of the list as the current data state.

Undo
Roll Back the Data State one level at a time. This will undo the last change to the data. Continuing to select Undo will roll the data back through the data states one at a time until the oldest remembered data state that exists is reached. In this menu, RAPT will give descriptions of what kind of change it is going to Undo in the format of

Undo Screen Name : Data/Action Name.

Undo to ...
Allows the user to roll back to a nominated Data State directly. This will allow the user to Undo several changes in one step. All of the data changes made from the selected Undo level to the latest change will be Undone. A dialog box will popup listing all the changes that have been remembered from the currently selected data state to the first data state remembered as shown below (all of the items left to undo). The first item in the list is the next available change to be undone to the earliest remembered change. Click the list item to indicate to which stage the user wants to roll back the data to. Then click OK button to perform the rollback. The data will roll back to the data state that existed before the data was changed for the selected Undo Level.



For example, the above dialog indicates that user first changed cell value in *Top Reinforcement Depth from top* in *General* screen, then changed cell value in *Bottom Reinforcement from bottom* in *General* screen, and finally changed left cantilever to left end in *Span* screen.
Clicking the 2nd item will highlight the first two items. Clicking OK will restore the data state to the state where the *Reinforcement Depth from top* has been modified (the data state before the selected items) and the left cantilever existed and the value for *Bottom Reinforcement from bottom* has not been modified.

Redo
Roll Forward the Data State one level. This option will only be available if an Undo operation has been performed. This will Redo the next change to the data. Continuing to select Redo will roll the data forward through the data states one at a time until the data state that existed after the last data change is reached. In this menu, RAPT will give descriptions of what kind of change it is going to Redo in the format of

Redo Screen Name : Action Name.

Redo to ...
Redo several undo actions that user made in one step. This option will only be available if an Undo operation has been performed. A dialog box will popup listing all the data states that have been undone. The first item in the list is the data state for the next change that was made after the currently selected data state and the last item on the list is the last data change made i.e. in the reverse order to that shown in the dialog above. Click the list item to indicate which stage you want to roll-forward to. Then click OK button to commit the roll-forward. The current data state will then be the state after the change nominated in the selected item on the list.

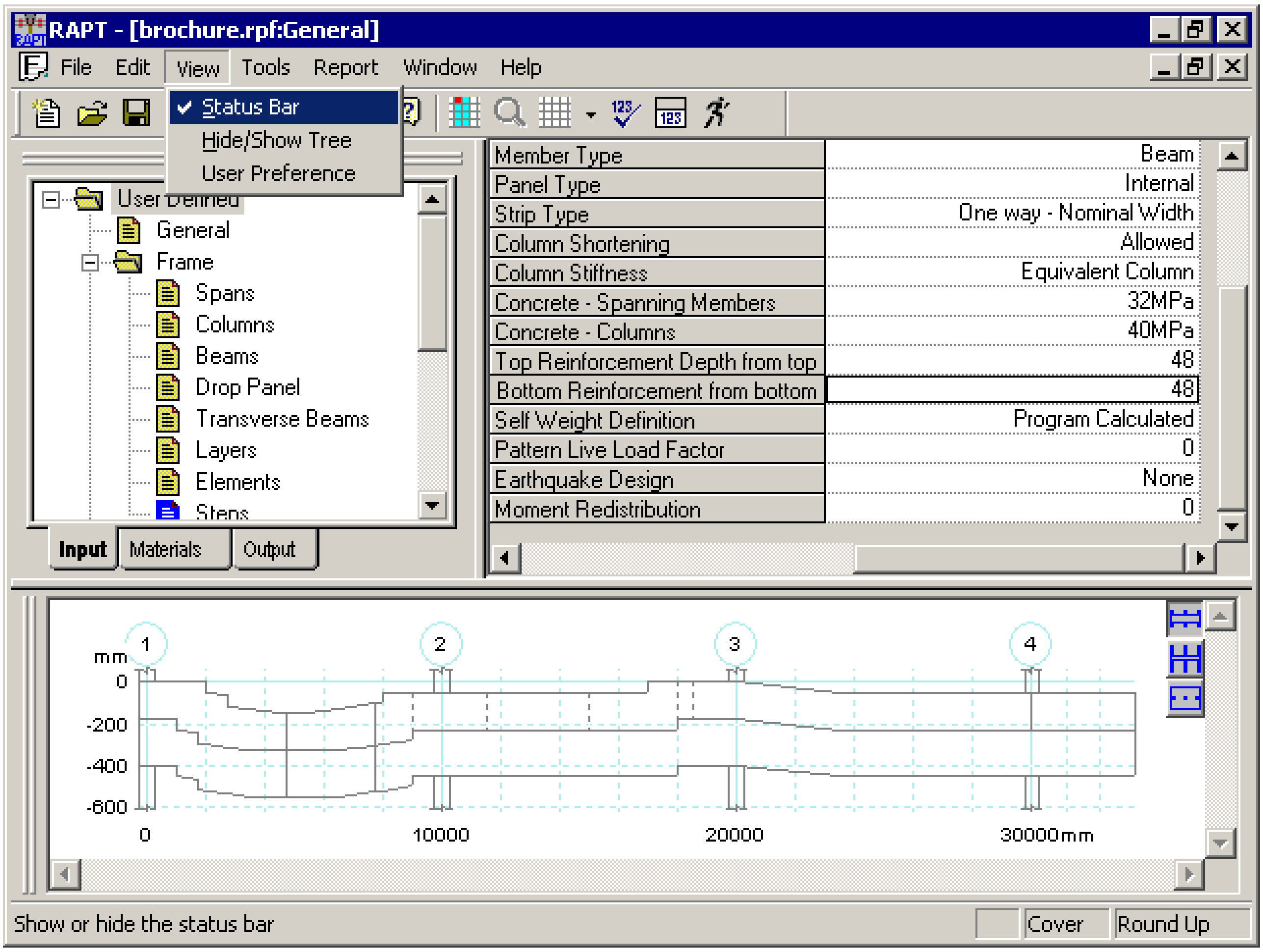
Clipboard commands

Cut (Ctrl + X) 
Delete the currently selected data and copy the data to the Windows Clipboard.

Copy (Ctrl + C) 
Copy the currently selected data to the Windows Clipboard.

Paste (Ctrl + V) 
Paste the contents of the Windows Clipboard to the current focused cell in the Data view. If a range of data items has been selected, the same range of cells must be available for pasting. RAPT will not create more rows of data automatically to fit the data being written to it. The data cells into which the data is to be written must be of the same type as the copied data.

4.5.3.3 General View Menu

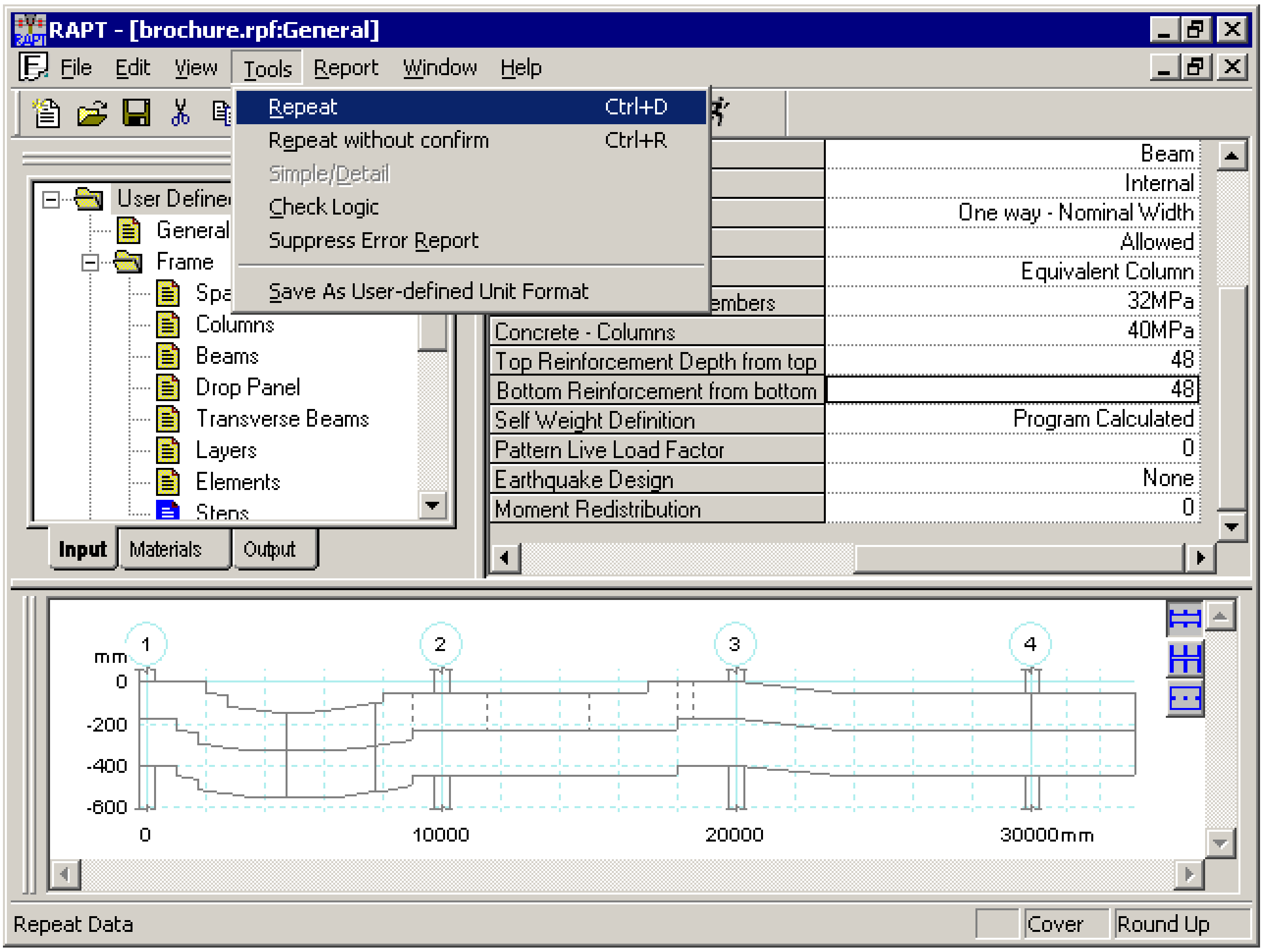


Status Bar
Allows the user to select to have the status bar visible or hidden. Clicking with the left mouse button will toggle between visible and hidden.

Hide/Show Tree
This item is only shown when a Control Tree is available. It allows user to hide the program control tree. For computers with small or low resolution screens, this allows the data views to use the full screen width while being viewed/edited.

User Preference
Allows the user to control a set of preferences for units, fonts and data formatting. See 4.2 User Preferences.

4.5.3.4 General Tools Menu



Repeat (Ctrl + D) 

Users can 4.4.4 repeat the current cell value to other cells by using cell the repeating function and asks for confirmation if data already exists in a repeat to cell. This button also works with the special repeat functions in some data grids where there is a logical column of data to copy into the current column.

Repeat Without Confirm (Ctrl + R) 

Users can 4.4.4 repeat the current cell value to other cells by using cell the repeating function and automatically overwrites any existing data in the repeat to cells. This button also works with the special repeat functions in some data grids where there is a logical column of data to copy into the current column.

Simple/Detail 

In some data screens, the user is offered a choice as to whether a simple shape is to be defined or a complex one. Two examples are drop panels and transverse beams. This button will only be available in screens where it can be used.

Check Logic 

RAPT checks all data for inconsistencies before calculating results. This button allows the designer to force a check on the data at any time. Any warning or error messages will be printed in the 4.5.4 Message Window.

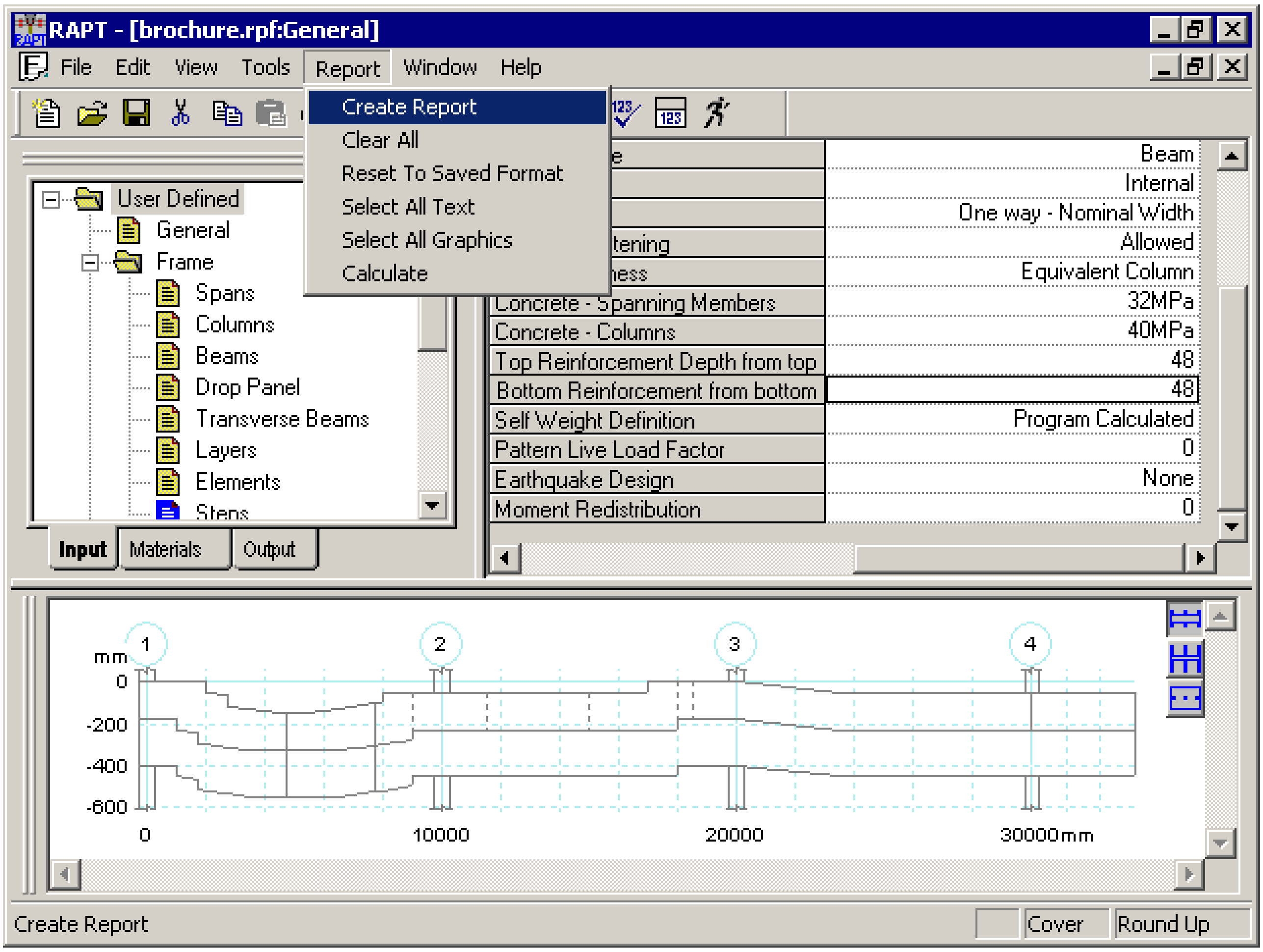
Suppress Error Report

When data errors are detected during background data checks, e.g. prestress profile calculations, the Message window will automatically replace the Frame Graphics window and show the error messages. Where multiple data changes are being made this can become annoying. Selecting this option will stop the Message window from being shown except when the user specifically asks for a data check or the data calculations are done and errors are found. This option only affects the current RAPT session. If you want to turn this off permanently for all RAPT sessions, select the option for suppressing error report in 4.2.2 View->User Preferences->User Options->suppress error report while editing

Save as User Defined Unit Format

The current unit settings for all data can be saved as a user defined format. This format is then available to be selected as the default units format in 4.2.1 View-User Preferences-Units and Fonts.

4.5.3.5 General Report Menu



Create Report
Creates an output report using the current settings. If the data has been modified and the results have not been recalculated, RAPT will automatically recalculate before creating the report.

Clear All
Clears all report settings in the Output Tree..

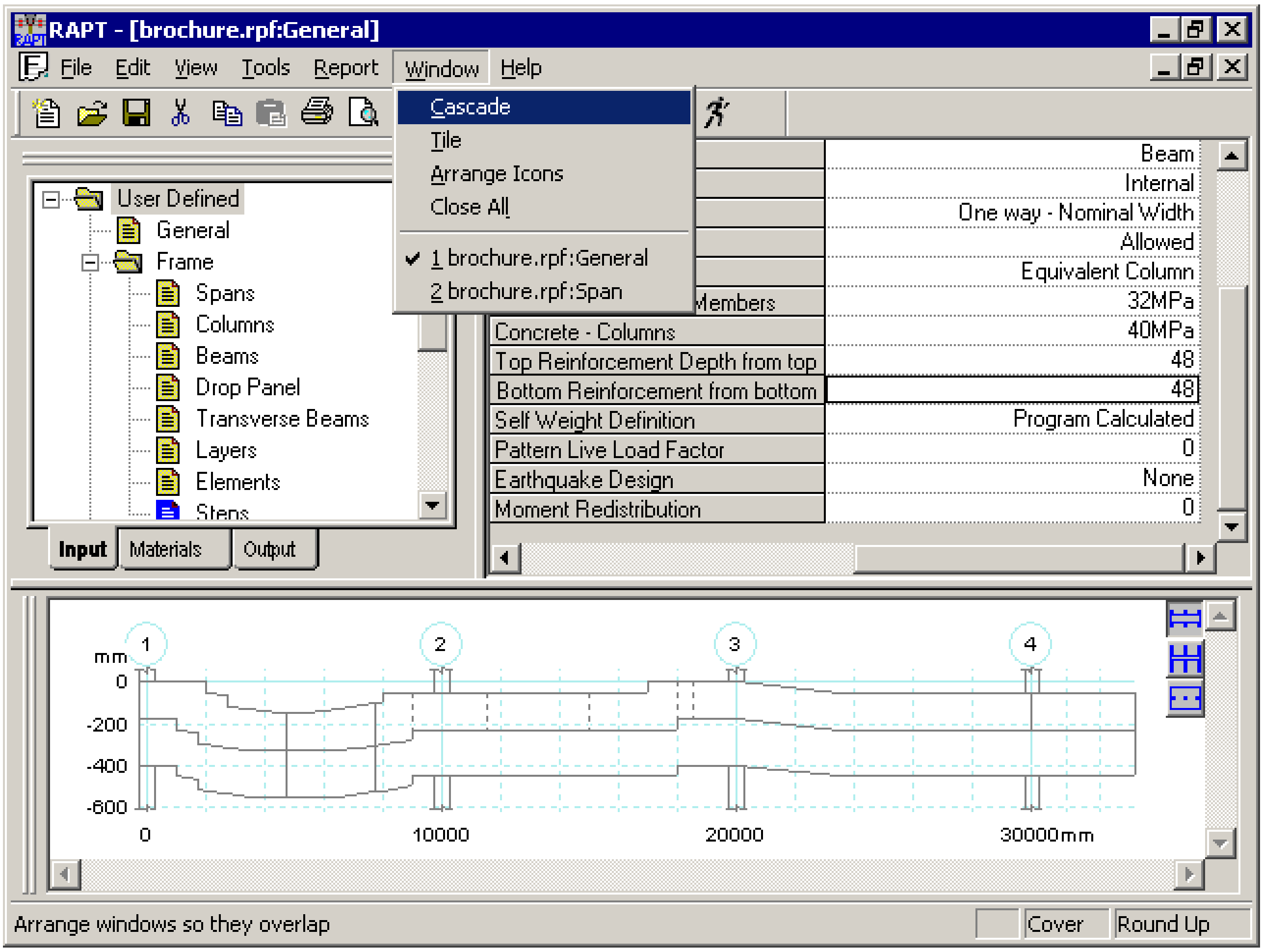
Reset To Saved Format
Resets all report settings in the Output Tree to the default settings as defined in 4.2.5 View->User Preferences->Output Report Settings.

Select All Text
Selects ALL text report settings in the Output Tree. Any graphics settings that are already selected will be maintained.

Select All Graphics
Selects ALL graphics report settings in the Output Tree. Any text settings that are already selected will be maintained.

Calculate
Forces the recalculation of the results.

4.5.3.6 General Window Menu



Cascade
Arranges all of the open windows in cascade style from top left of the viewing window to bottom right of the viewing window in the order in which they were opened. If there are too many windows open to fit into the viewing window a second set of cascaded windows will be placed over the first set.

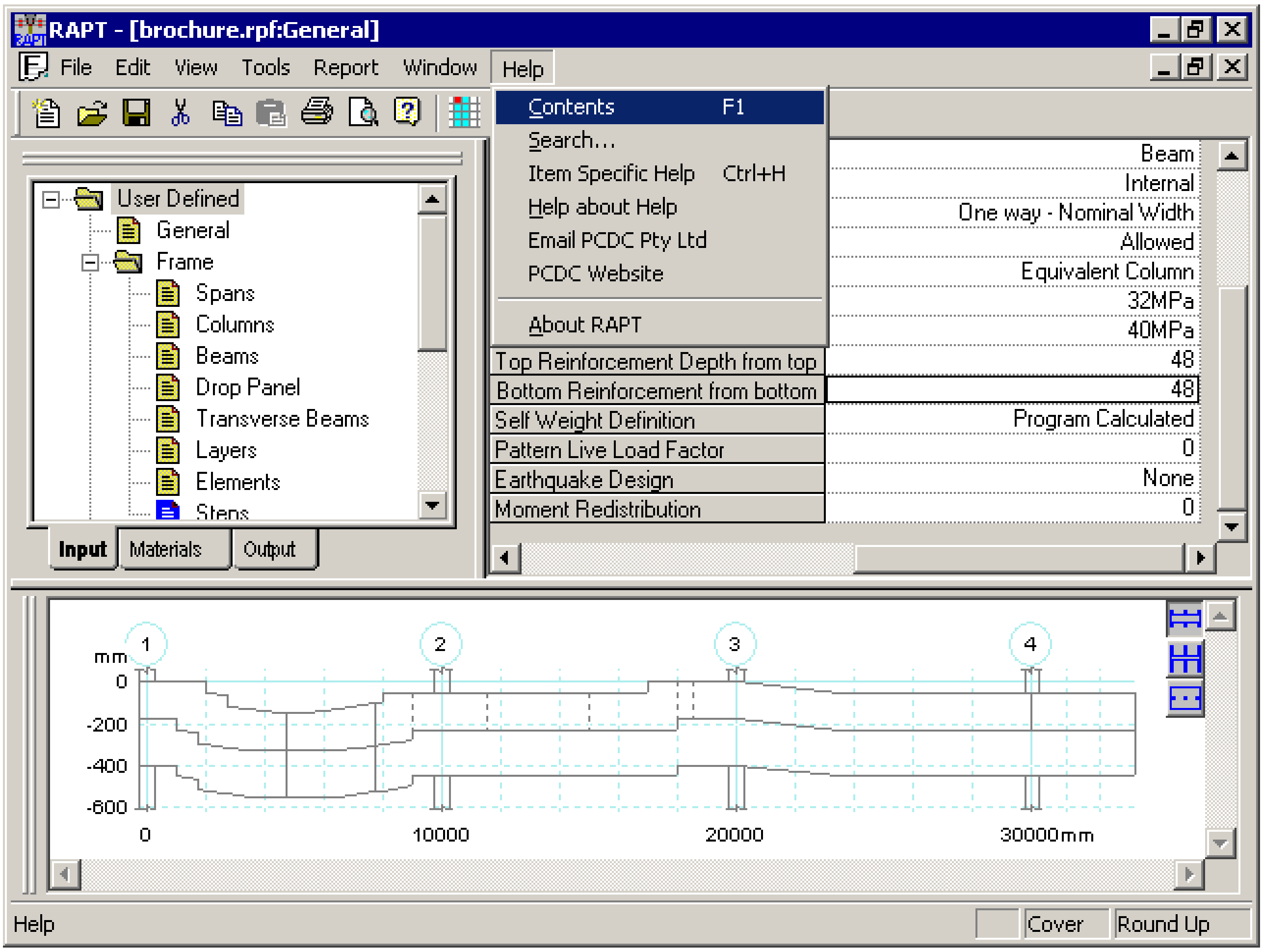
Tile
Arranges all of the open windows in tile style within the viewing window. The windows will all be of the same size and will fill the entire area of the viewing window.

Arrange Icons
Arranges all open minimized windows in order along the bottom of the viewing window in the order in which they were opened in rows from left to right..

Close All
Closes all open windows.

Windows List
Lists the windows that are currently open. To bring any window into focus, the designer can select a window using the mouse or up/down arrow keys.

4.5.3.7 General Help Menu



RAPT uses a standard Microsoft HTML Help system for viewing help information, If the files needed for viewing HTML Help are not loaded on your system RAPT will load them during installation.

Contents (F1)
Pressing F1 or selecting Contents from the Help Menu will open the Help Viewer at the Table of Contents. The user can then select the area of help to be viewed from the lists in the contents tree.

Search
Allows the user to search the complete help document for a keyword, phrase or group of works. A list of the topics containing the search request will be presented and the search request text will be highlighted wherever it occurs within each topic.

Item Specific Help (Ctrl + H or )
When a data cell that has specific help is in focus, pressing Ctrl + H or selecting this item from the Help menu will open the Help document at the relevant item.

Help About Help
Selecting this item will give a help screen detailing the use of the Help viewer.

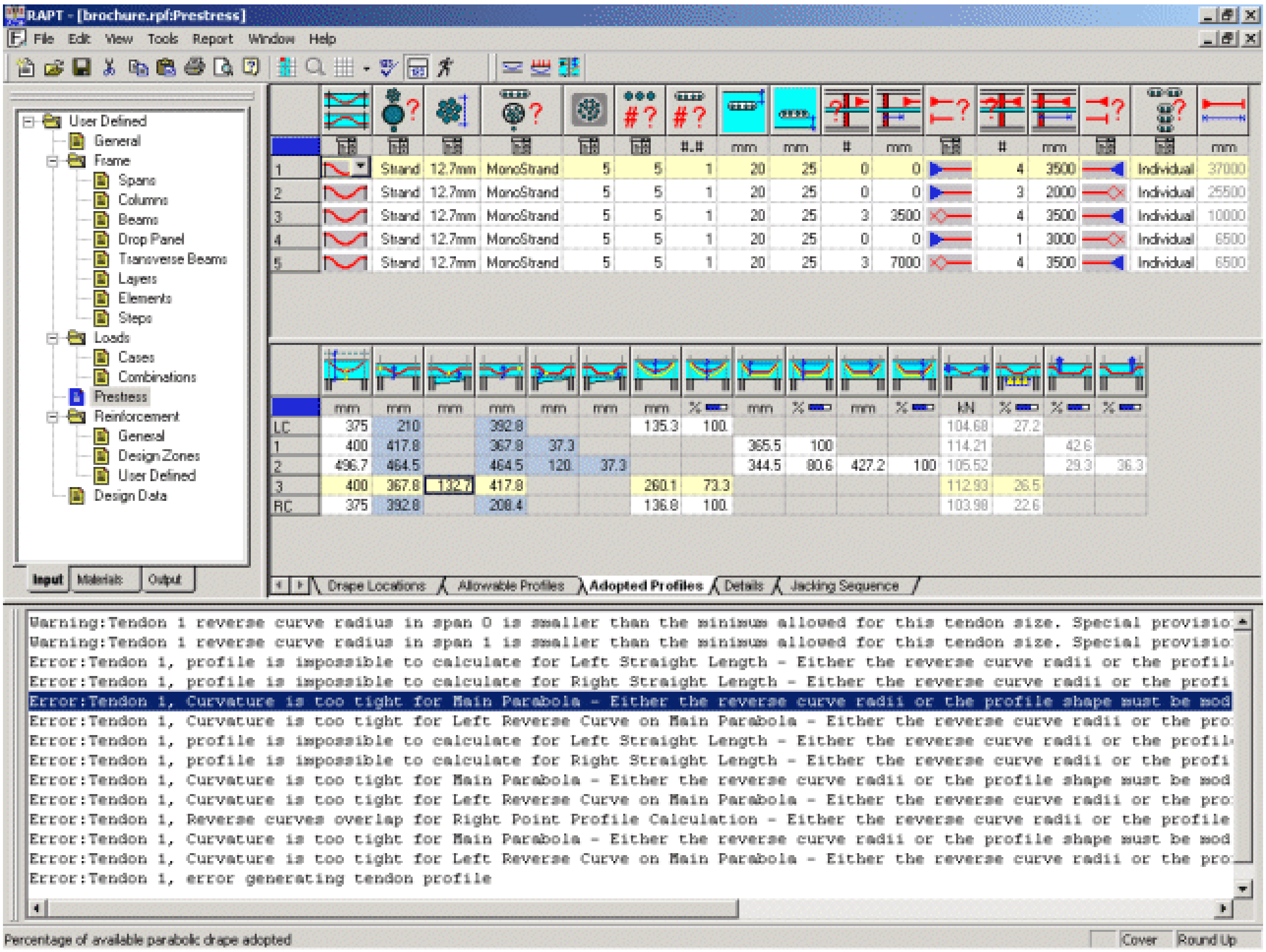
Email PCDC Pty Ltd
Automatically launches your computers default Email program with an email to PCDC support.

PCDC Website
Connect automatically to the PCDC website.

About RAPT
Opens a dialog which lists

1. RAPT Version number and copyright.
2. RAPT Licensee details and dongle number
3. Number of users logged onto a network dongle and maximum number of users allowed.
4. Current Usage for demonstration and usage licenses.

4.5.4 General Message Window



When RAPT encounters inconsistencies in the data or experiences errors in calculations which it can trap, it will list the problems in the message Window. This window will replace the Frame Graphics Window as shown above. The designer can toggle between these two windows using the  button from the toolbar. The messages may be of five types

1. Warning:- RAPT thinks something looks illogical but it will not interfere with RAPT carrying out the calculations. Calculation of results is still allowed but the designer should make sure that he is happy with the data being questioned.
   For example, RAPT will warn if the reverse curve radius for prestress tendons is less than the defined minimum or zero but will allow the calculations to proceed. It is up to the designer to justify this data. A zero radius is not able to be built so should not be accepted by the designer. A radius less than the defined minimum but greater than the absolute minimum bending radius is able to be built but requires special checks on bursting, bearing stresses and wear of the duct during stressing.
2. Error:- The data is not acceptable and calculation or results cannot be completed. RAPT will not allow the calculation of the data until this error is resolved.
3. Calculation Error:- There has been an error in the calculation of the results that causes RAPT to terminate the calculations.
4. Output Warning:- Design Warnings generated during the calculation of the results.
5. Output Error:- Design Errors generated during the calculation of the results

Relating Messages to Data

The first two message types above are related to the input data. If the designer double clicks on a data error message, RAPT will open the data window in which the data causing the error is defined and move data focus to the cell that it thinks is the main cause of the problem.

In the case above, the 5th message has been selected using a mouse double click. The message line is highlighted and the data cell focus has moved to

1. Prestress Data
2. Adopted Profiles data
3. for tendon 1,
4. span 3,
5. low point profile height for a parabolic profile.

This error occurred because the calculation of the main parabola could not be completed because the curvature of the main parabola was smaller than the minimum curvature set in Drape Locations for that span. RAPT has indicated the problem is with the low point profile and has nominated the span number and the tendon number but there are actually several data items that could possibly be modified to fix the problem. Selecting any of the Window Tabs to

view other data will also show the span number causing the problem. The other data items that can be modified to affect tendon curvature problems are

1. Heights at each end of the parabola
2. Locations of each end of the parabola
3. Reverse Curve Radius. This should not be reduced to less than the minimum unless specific action is taken to allow for high bearing stresses, design of bursting reinforcement and investigation of the wear effects on the this duct. It should never be reduced to zero for a final design.
4. For end spans of a tendon, the reverse curve at the anchorages.

Clicking on the  button will toggle between the Message View on Frame Graphics View so that the designer can also see the graphical representation of the data to help identify the problem.

Often more than one message will apply to the same problem so clearing up one problem may remove more errors.

5 Design Standards

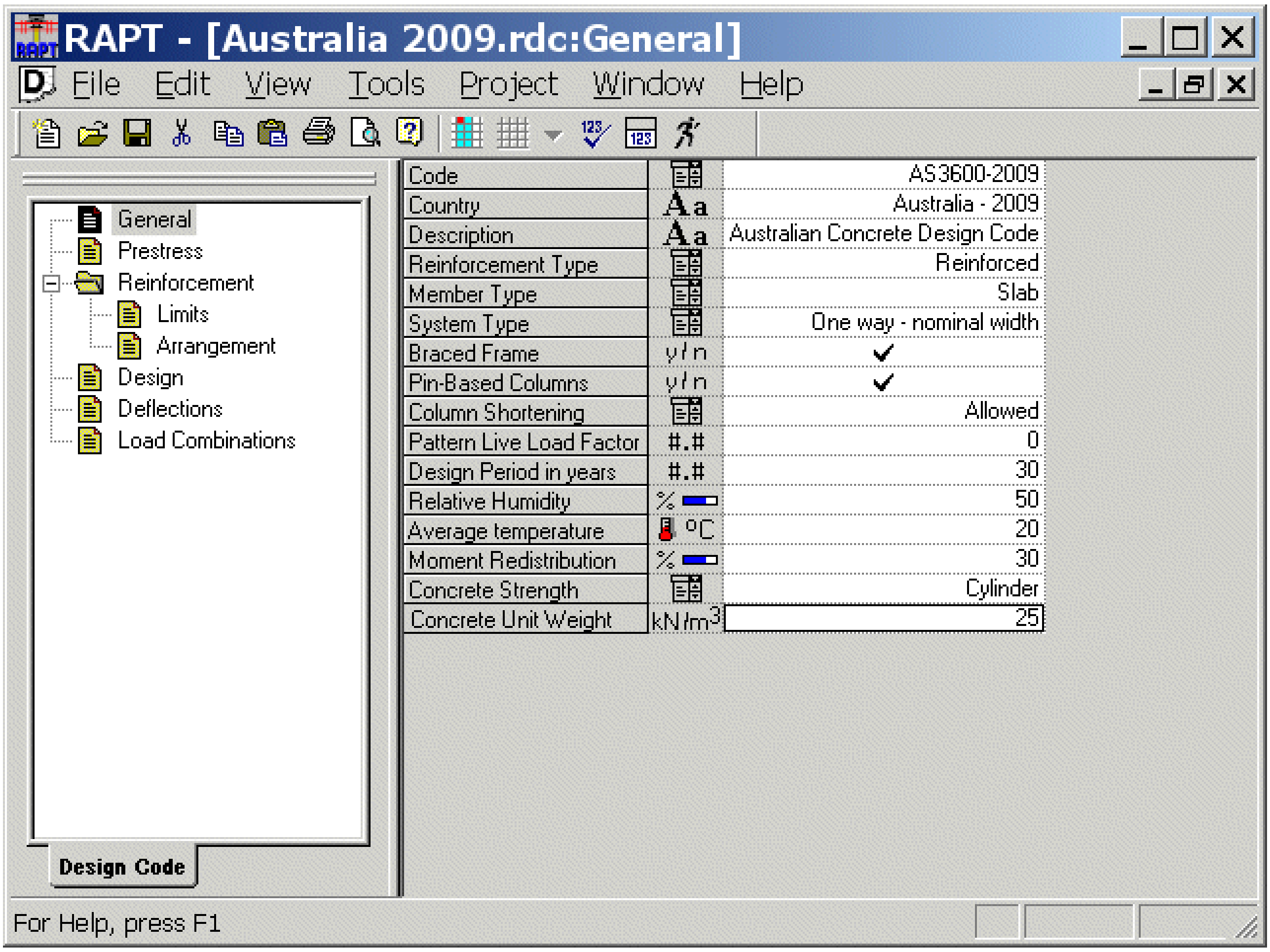
5.1 Introduction

The design routines for several design codes have been included in RAPT. A set of default data has been defined to allow designers to create modified versions of these design codes in which certain design parameters are able to be controlled by the designer. When these modified versions are used, the standard code formulae are used with the parameters defined in the modified version.

When a new run is commenced, the design parameters defined in the default Design Standard are inserted into the data for the run and the Design Standard set in the file in inserted as the Design Standard on which the calculations will be based. In the run data, the designer can select different Design Standards if desired. The basic code design data will be inserted into the run data from the newly selected design Standard.

Several Design Standard files are provided with RAPT and the designer can define more variations if desired. RAPT also provides a base Design Standard for each design code included in the program. These are used as the basis of user defined Design Standards when creating a new design standard variation.

5.2 General



Code
The design Standard that will be used as the basis for the design. The formulae from this standard will be used and design checks will be in accordance with this design standard. The current list of standards is

1. Australia - AS3600
2. USA - ACI318
3. Europe - Eurocode 2
4. United Kingdom - BS8110
5. South Africa - SABS0100
6. Singapore - CP65
7. Hong Kong - CP2004
8. India - IS456/IS1343

Other design standards will be added in time.

Country
The country for which the design standard is being set up. Various factors can be modified to suit the use of a Design Standard in a country other than the one it was written for. This is a text field and is for the use of the designer only. It is meaningless to RAPT.

Description
A brief description of this version of a design standard. This is a text field and is for the use of the designer only. It is meaningless to RAPT.

Reinforcement Type
The default design type for new RAPT runs. The options are

1. Reinforced Concrete
2. Bonded Prestress
3. Unbonded Prestress

Member Type
The default member type for new RAPT runs may be defined as

1. Slab

2. Beam

System Type

The default structural system for new RAPT runs. Options are

1. One way - Nominal Width:-
    1. Slab systems:- Designs only a nominal width of the defined slab e.g. a 1000mm strip of a 8000mm panel of slab as a one way spaning slab. There is no transverse distribution of moments. Used typically for slabs spanning between beams and slabs spanning across walls. Self weight and panel loads are calculated for the strip width. Point loads are applied to the effective (design strip) width. RAPT adjusts the frame properties to be consistent with the design strip width. Column moments and reactions are calculated from the design strip results by factoring them by the ratio of the transverse column spacing to the effective width. No horizontal steps or tapers are allowed in the side of the slab.
    2. Beam Systems:- Will design the effective beam to support the full panel load. There is no transverse distribution of moments. Self weight and panel loads are calculated for the strip width. Point loads are applied to the effective beam.
2. One Way - Full Width:-
    1. Slab systems:- Designs the full panel width of the defined slab. The design width is the panel width at all locations. There is no transverse distribution of moments. Used typically for slabs spanning between beams and slabs spanning across walls. Self weight and panel loads are calculated for the full slab width. Point loads are applied to the full slab width. There is no limitation on the use of steps and tapers to any surface.
    2. Beam Systems:- Not applicable.
3. Two way:- The slab column strips and middle strips. The moments and shears are distributed to these strips using standard distribution factors defined in most Design Standards which are further controllable by the designer in 7.2.4.3 Loads->Lateral Distribution Factors. The designer can control the distribution of the effects at the supports and the maximum span moment points in each span. RAPT uses a parabolic distribution of the factors between these points to more closely match the distribution in a finite element analysis. Flexural Design, Reinforcement Layouts and Deflection calculations are provided for each design strip. Beam Shear calculations are only provided for the column strip.
    1. Slab systems:- Can be used for any reinforcement type.
    2. Beam Systems:- Only allowed for Reinforced Concrete systems. The effective beam is apportioned a fraction of the column strip moments and shears in each span. The effective beam is designed to carry these moments and the middle strip is designed to carry its share of the moments. Normally it is sufficient to extend the middle strip reinforcement to the face of the beam to allow the remainder of the slab in the column strip to carry the left over column strip moment. The designer should check to see if this will be adequate especially in situations where the portion of moment assigned to the effective beam is relatively small.
4. Two way - Average:- This is the ACI318 method of designing two way prestressed slabs. Even though moments and shears are distributed unevenly over the width of a slab panel, the "average" effect of this is used in the design. The method is not allowed by AS3600 and a modified version is allowed under British Code design rules where stricter limits are placed on the allowable tension stresses in the concrete in recognition of the effects of averaging of the moments compared to the real moment concentrations especially at supports. This method should not be used for partially prestressed slabs.

Braced Frame

Whether or not the frame is braced against sidesway.

Pin-Based Columns

The condition at the base of the columns below the member. Options are

1. Fixed:- columns below are supported from another member of similar type with a moment connection.
2. Pinned:- Far end of the column below does not have a moment connection e.g. footing.

Column Shortening

RAPT gives the user the option to consider the effects of Column shortening. If Column Shortening is to be restrained, RAPT places vertical restraints at each column beam node in the Frame Analysis. Thus no axial shortening can occur. The column shortening can, at times, have a large effect on moment distribution especially if high loads and columns of varying lengths or axial stiffness are input into a run. Always be careful when mixing knife edge supports and columns at different support locations as knife edge supports do not allow vertical movement while columns do.

Pattern Live Load Factor

The Pattern Load factor is entered as a decimal (fraction of 1). If no Patterning of Live Loads is required, the user should input 0 and if full Live Load patterning is required, 1 should be input. The decimal refers to the amount of Live Load used in the pattern combinations i.e. 0.75 indicates that 75% of the Live Load is used in pattern combinations.

Design Standards define the percentage of load that should be applied for patterning of the live load. Also, the method used in patterning live loads varies from Standard to Standard. For more information on patterning of loads for various Design Standards see Theory Section T4.1.

Only the load case named Live Load will be patterned. The combination factors used for this case are used for the envelope of actions created for the patterned live load. RAPT will create two live load pattern envelopes, one based on the envelope of moments at each design location with co-existing shears and one based on the shear envelope at each point with co-existing moments. Pattern load cases are not used for deflection calculations. The full live load case is used.

Design Period in years

The time over which the structure is expected to be loaded in years. This figure is defaulted to 30 years as creep and shrinkage effects are minimal after this period. The long term losses are calculated at this time. The long term and incremental deflections are also calculated at this time. Estimates of effects at shorter times may be calculated by nominating the time at which calculations are required e.g. the expected deflection at 6 months can be calculated by substituting 6 months as the loaded period (.5 years).

Relative Humidity

This dictates the shrinkage and creep characteristics. The drier the air, the greater the tendency to lose water from the concrete and the greater will be the creep and shrinkage.

To simulate AS3600 Environment Variables, the following values can be used

1. tropical near coastal (65% Humidity)
2. temperate (55% Humidity)
3. interior (50% Humidity)
4. arid (40% Humidity)

Average temperature

The average temperature for the design area. RAPT modifies creep and shrinkage for temperature affects.

Moment Redistribution

The designer can define a % of moment to be redistributed. RAPT will then redistribute the ultimate moment envelope and the associated shear envelope for Ultimate strength calculations. Serviceability load combinations remain unaffected.

Where the Ultimate moment at a design point is of a different sign to the service moment, RAPT will apply a factored Ultimate Moment of the same sign as the service moment equal to 1.2 x $M_{service}$ to ensure that there is sufficient strength on any face that will be tensile through any of the loading stages of the member.

If Moment redistribution is requested, RAPT will automatically apply all limitations to the design for ductility as defined in the different design codes. It will over-ride any user defined setting for depth of neutral axis limit in the 7.2.7.1 Design Data in doing this unless the defined value is less than that calculated from the Design Standard rules.

Designers should be aware that the use of large amounts of redistribution of the moments in a member introduces severe limitations to the ductility requirements of a member due to the reliance in the strength calculations on increased rotations at plastic hinges and also could introduce a requirement for extra reinforcement under service load conditions to control crack widths. The resulting design could be less economical than a design without or with lesser redistribution of moments.

RAPT does not redistribute moments at end columns. To redistribute moment from an end column, a user should use the option to modify the I_{zz} of a column in the F3 Columns screen.

RAPT will only allow redistribution of moments for 7.2.3.2 Braced Frames. RAPT will limit the % redistribution defined in the input to the maximum allowed by the relevant design standard. It will allow both positive (reducing support moments) and negative (increasing support moments) values of the redistribution %.

The maximum limits are

1. AS3600:- 30% for normal ductility reinforcement and 0% for low ductility reinforcement
2. ACI-318:- 15%
3. Eurocode 2:- 30% for class B or C reinforcement and 20% for class A reinforcement (Low Ductility).
4. BS8110, CP 65, IS456/IS1343 and SABS0100: 30% for reinforced concrete members and 20% for prestressed members.
5. Hong Kong CP2004: If concrete strength less than 70MPa, 30% for reinforced concrete members and 20% for prestressed members. If concrete strength is greater than 70MPa, no redistribution is allowed.

Concrete Strength

This is the concrete strength basis on which the formulae in the code are based. The options are

1. Cylinder Strength.
2. Cube Strength.

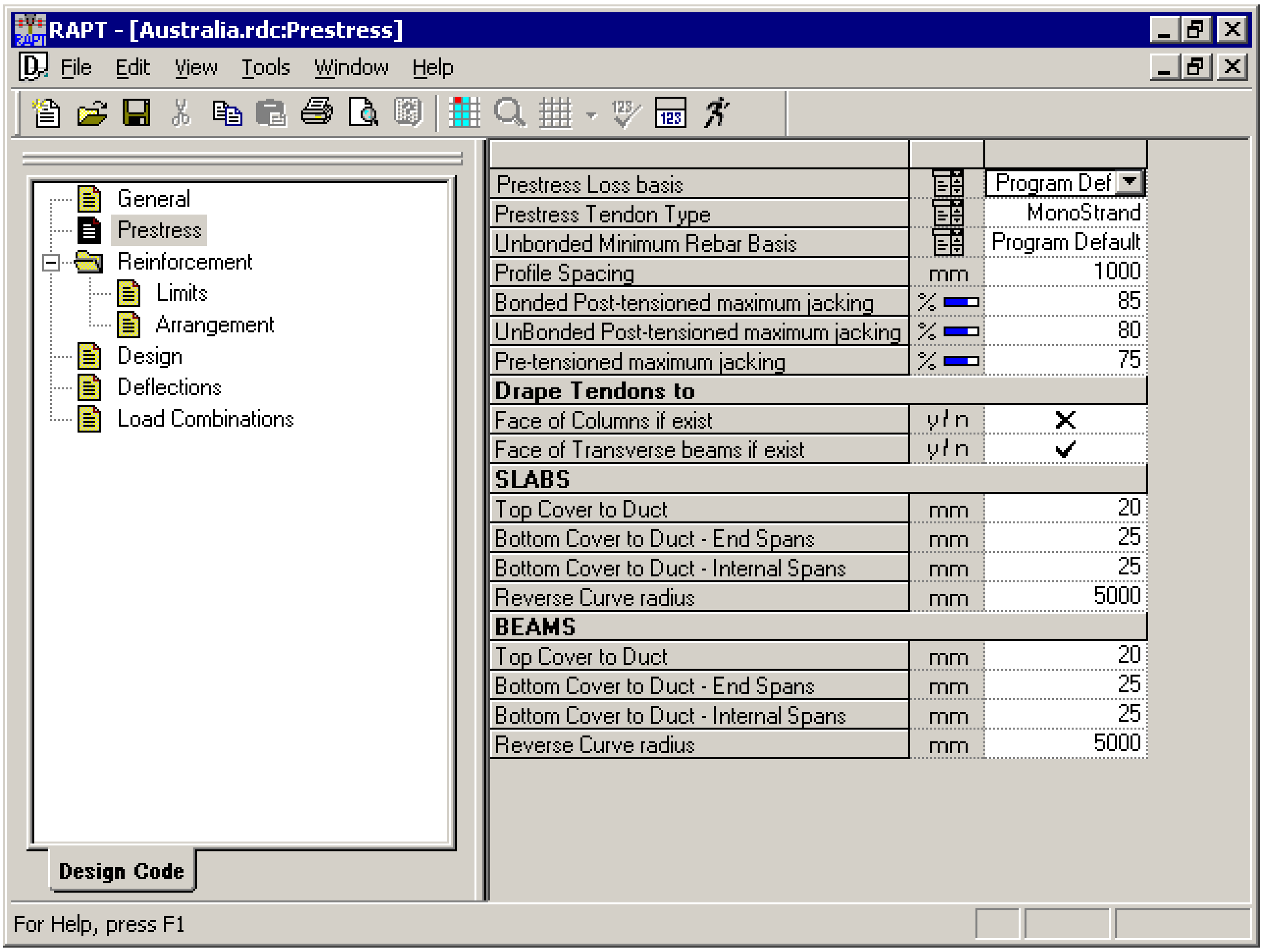
If the materials type selected is defined as the opposite type to that required by the design code RAPT will convert the materials data concrete strengths to the code strength type using

cylinder strength = .8 * cube strength

Concrete Unit Weight

The unit weight of concrete to be used in self weight calculations for concrete members. Un-reinforced concrete has a weight of approximately 24KN/m3. For every 1% by volume of steel reinforcement, the unit weight of structural concrete increases by about .6KN/m3.

5.3 Prestress



Prestress Loss basis

This field allows the user to define the calculation method to be used for prestress losses calculations. The options available are

1. Program Default
2. AS3600
3. Eurocode2
4. ACI-209
5. Zia et al
6. CEB-FIP 1970
7. CEB-FIP 1978

The default value in the field is Program Default which will cause RAPT to use the internal default method for each design code. This is

1. AS3600:- AS3600 method
2. ACI318:- ACI-209 method
3. Eurocode2:- Eurocode2 method
4. BS8110:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
5. SABS0100:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
6. CP 65:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
7. CP2004:- CEB-FIP 1970 method. The Hong Kong code requires an extra multiplying factor cs = 4 to allow for shrinkage values in local concretes. This is included only for the Hong Kong Code and is used no matter which shrinkage/creep model is selected for designs to the Hong Kong Code.
8. IS456/IS1343:- CEB-FIP 1970 method

A method which is acceptable in a designer's local design community and which is consistent with the concrete manufactured in that area should be used.

Prestress Tendon Type

Three prestressing types options are available for bonded prestressing and 2 for unbonded prestressing. These are

1. Monostrand:- Flat duct post-tensioning tendons used in both slabs and beams. Strands are normally stressed individually rather than as a tendon.
2. Multistrand:- Round duct post-tensioning tendons used mainly in larger beams. Strands are normally stressed as a group.

3. Pretensioned (not available for unbonded prestressing). Strands stressed between buttresses and released after the concrete has hardened.

These names are also used in the 6.1 Materials Data where the different tendon sizes available are defined.

Minimum Rebar Basis

This field allows the user to define the calculation method to be used for minimum reinforcement calculations for prestressed members. In general the selected code rules will be used unless one of the options below is selected. The options available are

1. Program default (UBC for unbonded prestressed members)
2. ACI-318
3. UBC
4. TR43 (This is not considered by RAPT to be the default method for BS8110 design or Eurocode 2 design or for any derivatives of BS8110. The designer must select this option if it is to be used for a design).
5. None

Profile Spacing

The default tendon profile spacing to use for in this member. RAPT will base the number of profile spaces in each profile section on this value and will round the number of spaces in each profile section to a whole number based on this value.

Drape Tendons to

Allows the designer to include these drape to locations in the default settings for RAPT. RAPT will drape tendons to column centreline or face of band beam or drop panels by default. If either of these are selected they will be added to these default drape to locations.

1. Face of Columns if exist
2. Face of Transverse beams if exist

Slabs

Allows the designer to specify values for the following for new slab designs

1. Top Cover to Duct:- cover to the top of the duct from top surface of the concrete member
2. Bottom Cover to Duct - End Spans:- cover to the bottom of the duct from bottom surface of the concrete member in end spans.
3. Bottom Cover to Duct - Internal Spans:- cover to the bottom of the duct from bottom surface of the concrete member in internal spans.
4. Reverse Curve radius:- Default Reverse Curve radius to use for all transition curves for all tendons in the member.

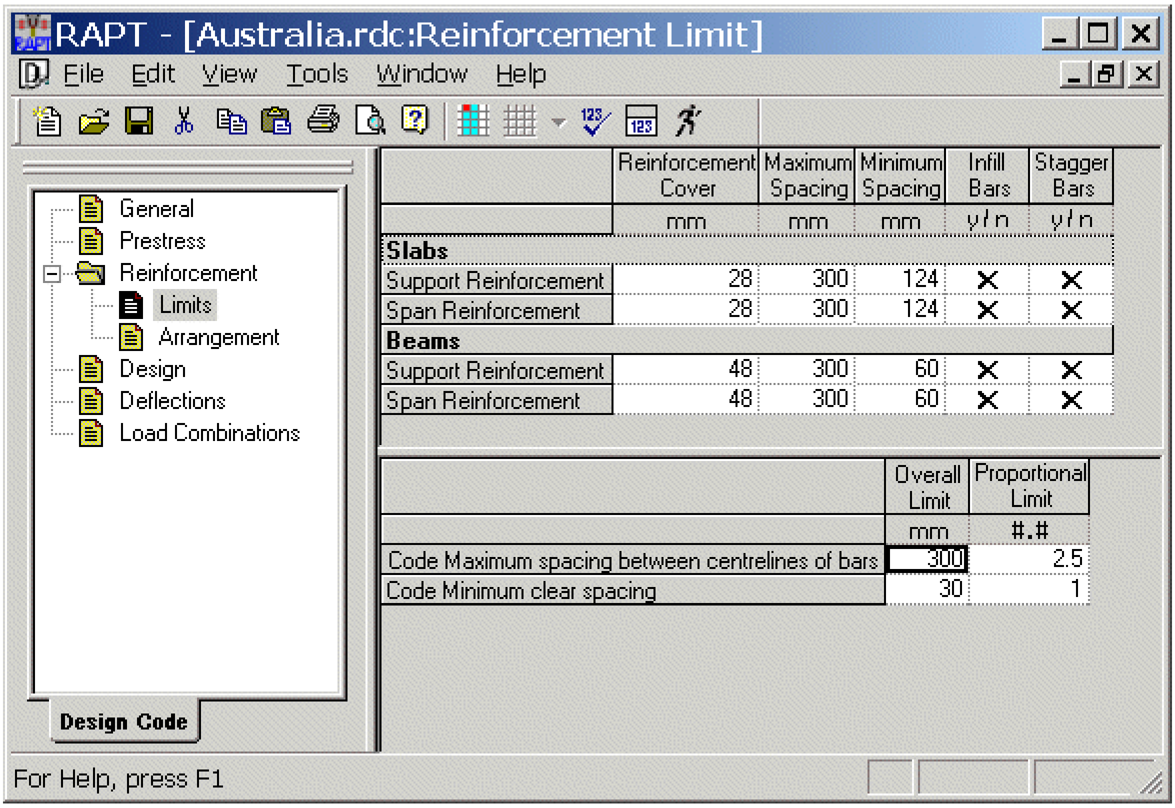
Beams

Allows the designer to specify values for the following for new beam designs

1. Top Cover to Duct:- cover to the top of the duct from top surface of the concrete member
2. Bottom Cover to Duct - End Spans:- cover to the bottom of the duct from bottom surface of the concrete member in end spans.
3. Bottom Cover to Duct - Internal Spans:- cover to the bottom of the duct from bottom surface of the concrete member in internal spans.
4. Reverse Curve radius:- Default Reverse Curve radius to use for all transition curves for all tendons in the member.

5.4 Reinforcement

5.4.1 Limits



Reinforcement Limits

The data defined below can be input for the following conditions

1. Slabs
   1. Support Reinforcement:- The face of the slab on which Reinforcement is concentrated at the support and extends either side of the support (normally top over the column).
   2. Span Reinforcement:- The face of the slab where the reinforcement is spread over the length of the span with the controlling requirement at midspan.
2. Beams
   1. Support Reinforcement:- The face of the beam on which Reinforcement is concentrated at the support and extends either side of the support (normally top over the column).
   2. Span Reinforcement:- The face of the beam where the reinforcement is spread over the length of the span with the controlling requirement at midspan.

Data Definition

Reinforcement Cover
The depth to nearest surface of the main reinforcement bar from the relevant concrete face.

Maximum Spacing
The maximum reinforcing bar spacing to use in determining the reinforcing bar size to use and in detailing the reinforcement on the relevant concrete face.

Minimum Spacing
The minimum reinforcing bar spacing to use in determining the reinforcing bar size to use and in detailing the reinforcement on the relevant concrete face.

Infill Bars
When detailing reinforcement, normally, the support face reinforcement is not continuous over the full length of the span. In beams it is often practical to add extra nominal bars to fill this area and provide supports for shear ties and cross reinforcement. If Yes is selected here, the default settings in a new run will be set to yes for this option and RAPT will fill in blank areas in the reinforcing pattern with nominal bars (nominally 12mm bars at 500 centres for slabs and 16mm bars at 500mm centres for beams).

Stagger Bars
Away from the peak bending moment location in a design zone, the area of reinforcement required is normally less than is required at the maximum point. It is often more economical to curtail some of the reinforcing bars at a point within the design zone and only continue half of the bars to the end. Selecting Yes for this option will tell RAPT to

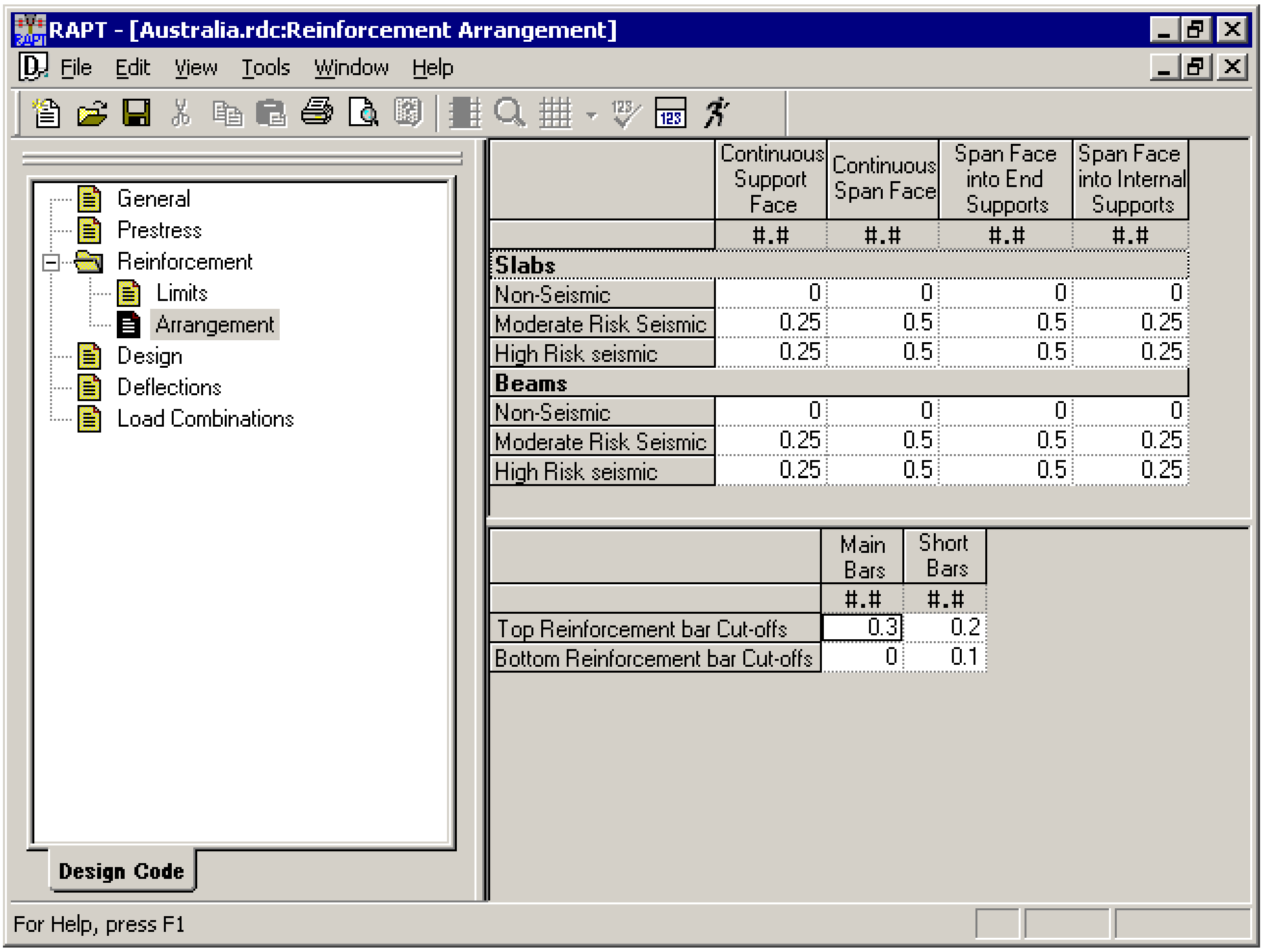
stagger the curtailment point so that half (rounded down for odd numbers of bars) of the bars are curtailed when they are no longer needed.

Reinforcement Arrangement

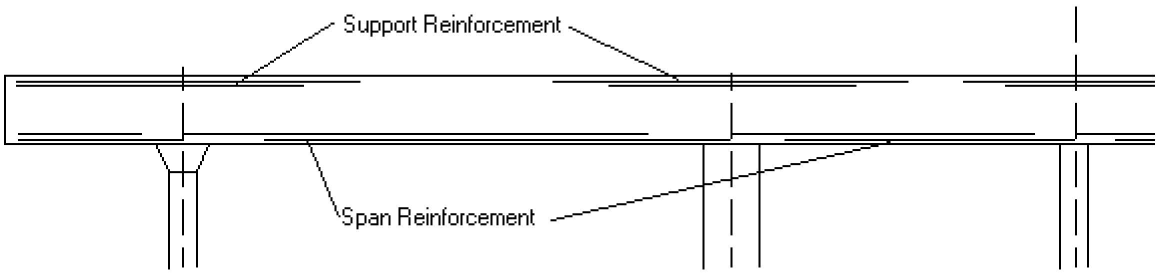
RAPT allows the designer to specify limits on the spacing of reinforcing bars. Two limits are provided for each of the following

1. Code Maximum spacing between centrelines of bars
    1. Overall Limit:- a spacing limit in length units
    2. Proportional Limit: a spacing limit based on a factor times the depth of the concrete.
2. Code Minimum clear spacing
    1. Overall Limit:- a clear spacing limit in length units
    2. Proportional Limit: a clear spacing limit based on a factor times the bar diameter.

5.4.2 Arrangement



Separate reinforcing detailing parameters can be set by the designer for both top and bottom reinforcement. RAPT asks for the information for Support Reinforcement and Span Reinforcement as described below.



Support Face Reinforcement:- The reinforcement on the face of the member which has a peak moment concentrated at a column and reducing away from the column. The main reinforcing bars normally are centred on the column and extend into the spans on either side of the column. For a member with downward loading, it would be the top face.

Span Face Reinforcement:- The reinforcement on the face of the member which has a maximum moment somewhere between the columns in a span and reducing towards the columns. The main reinforcing bars normally are centred on the middle of the span and confined to that span. For a member with downward loading, it would be the bottom face.

Reinforcement Detailing Limits

Default Detailing Limits can be defined separately for two structural types,

1. Slabs
2. Beams

For each structural type the default limits can be set for three alternate design conditions

1. Non-Seismic
2. Moderate Risk Seismic

3. High Risk seismic

Continuous Top
Continuous Bottom
The minimum area of continuous reinforcement in the face of the member in a span as a fraction of the maximum area of reinforcement in that face in that span.

Bottom into End Supports
The minimum area of span face reinforcement extending into the support at an end support as a fraction of the maximum area of span face reinforcement in that span. This option is only used at end columns where no cantilever exists.

Bottom into Internal Supports
The minimum area of span face reinforcement extending into the support at internal supports as a fraction of the maximum area of span face reinforcement in that span.

Default Bar Cut-off Locations

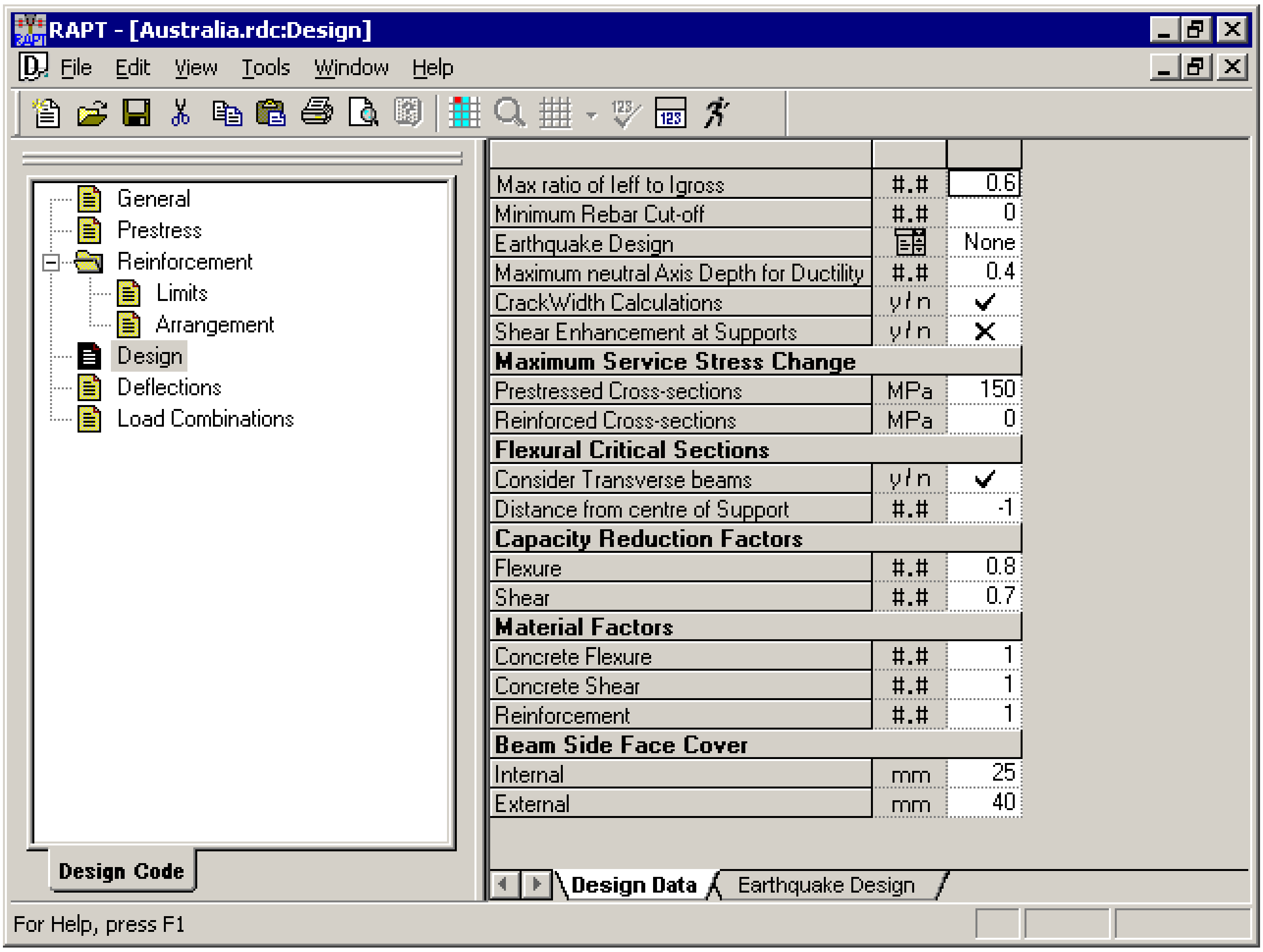
The default locations for bar cut-off positions for 7.2.6.2 staggered bars as shown below can be set by the user using for the two conditions below

1. Support Reinforcement bar Cut-offs
2. Span Reinforcement bar Cut-offs

Two cut-off locations are able to be defined for each case. These are

1. Main Bars:- The end of the reinforcing bar type.
2. Short Bars:- The stagger end location if staggering of the ends of the bars is required.

5.5 Design



General Design

Capacity reduction factors are used by many Design Standards, eg AS3600 and ACI318, to account for the variability of the material properties controlling strength and the likelihood of under-performance of a member. This factor also takes into account inaccuracies in design procedures and small deficiencies in workmanship on site (i.e. concrete dimensions etc). This factor should be set to 1 for other code types which use material factors.

Material factors are used by many Design Standards, eg BS8110, SABS0100, CP2004, IS456/IS1343 and Eurocode2, to account for the variability of the material properties controlling strength and the likelihood of under-performance of a member. This factor also takes into account inaccuracies in design procedures and small deficiencies in workmanship on site (i.e. concrete dimensions etc). This factor should be set to 1 for other code types which use capacity reduction factors.

Data Definition

Max ratio of Ieff to Igross

RAPT allows the user to specify a maximum $I_{effective}$ value. This value is specified as a fraction of I_{gross}. $I_{effective}$ is used by RAPT while calculating deflections. If a section is uncracked, RAPT will set $I_{effective} = I_{gross}$ unless specified in this input.

It is recommended by the authors that I_{gross} is not used by designers for deflection calculations unless a concrete member is completely unrestrained. The moments applied during the calculation of $I_{effective}$ normally only consider the vertically applied forces. Other forces such as restraint effects due to temperature, shrinkage and creep are not allowed for in the design moments. To make allowance for the effect of these additional forces on the cracked and effective inertia's in lightly loaded members, the authors recommend that designers restrict the $I_{effective}$ / I_{gross} ratio to 0.6 - 0.7.

Minimum Rebar Cut-off

AS3600 BS8110, CP65, SABS0100, CP2004, IS456/ IS1343 and ACI318 state that the minimum applied ultimate moment, M*, to be designed for is 1.2*Mcr. Eurocode 2 uses 1.15*Mcr.

The codes require designers to include sufficient reinforcement to satisfy a moment which causes the member to crack. The ultimate strength in bending is calculated assuming a fully cracked section. For small percentages of steel, this moment could be less than the moment Mcr to cause cracking. Failure of such a member would be quite sudden. To prevent such a failure, the ultimate strength in bending must be greater than Mcr.

However there is no concession for the situation when the section's design strength in bending, *Mu is much greater than the applied ultimate moment, M* (ie ØMu >> M*).

In AS1480 designers were permitted to relax this rule if ØMu was greater than 1.33M* (ie ØMu > 1.33 M*). [Also see ACI318 Clause 10.5.2] This, in effect, gave an extra factor of safety on the failure condition but did not ensure that cracking occurred prior to failure. Some other codes allow similar concessions.

RAPT has given the user the option of accepting the code as read by leaving the value for this parameter as 0 and then minimum M* is taken as defined in each code.

If this rule is to be supplanted with something similar to AS1480 then a number may be entered in the table. eg. 1.33. This will instruct the program to accept the section as being of adequate strength, i.e. no need to provide extra reinforcement to satisfy minimum flexural reinforcement rules, as long as ØMu > 1.33M*. If this is not satisfied then reinforcement is added until ØMu > 1.33M* or Ø1.2Mcr whichever is the lesser. 1.33 is the minimum value allowed by RAPT.

Earthquake Design

The designer can request that earthquake design rules be applied in designing and detailing for flexure and shear. The options are

1. None
2. Moderate Risk
3. High Risk

Maximum neutral Axis Depth for Ductility

Ductility is an important aspect of the design of a section / member. A ductile member will undergo large deflections before failing. This acts as a safety measure in that members can be seen to be dangerous before actually failing. For appropriate code ductility limits without redistribution of moments, see

1. AS3600 Clause 8.1.3, ku = 0.4
2. ACI318 Clause 18.8.1, ku = 0.428 (when rearranged from a tension strain limit of .004 and a compression strain limit of .003)
3. BS8110 and CP65 Clause 3.2.2.1, 3.4.4.4 and 4.2.3.1 ku = 0.5
4. Eurocode2 Clauses 5.4 and 5.5, ku = 0.448 Grades <= C50/60 ku = 0.368 Grade > C50/60
5. SABS0100 Clause 4.3.3.4, ku = 0.5
6. CP2004 Clause 6.1.2.4b, ku = .5 for fcu <= 40MPa, ku = .4 for fcu <= 70MPa and ku = .33 for fcu <= 100MPa
7. IS456 clause 37.1d and IS1343 clause 21.1.1d

RAPT allows the user to over-ride our interpretation of each code if they feel it is warranted.

If the designer nominates that 7.2.2 moment redistribution is to be allowed for in the design, RAPT use the formulae in each design standard to set a ductility limit based on the amount of redistribution used. RAPT will never use a limit higher than the one defined here no matter what the calculated value from the redistribution.

Crack Width Calculations

This field allows a designer to nominate whether or not crack width calculations are to be performed for a member and allows a user-defined crack width limitation to be defined for designs to BS8110, SABS0100, CP2004, IS456/ IS1343 and CP 65. If the value is left at DEFAULT, RAPT will use the default values of

1. Reinforced Concrete members - .3mm
2. Prestressed concrete members - .2mm

for designs to BS8110, SABS0100, CP2004, IS456/ IS1343 and CP 65.

For all other design standards, crack width calculations will be done based on limiting bar sizes and spacing as appropriate.

Selection of actual crack widths for Design Codes other than BS8110, SABS0100, CP2004, IS456/ IS1343 and CP 65 will have the same effect as selecting DEFAULT, as the other codes base their crack width calculations on stresses in the reinforcement rather than on actual crack widths.

Some codes allow the designer to ignore crack width calculations in some circumstances, eg AS3600 clause 8.6.1 and 9.4.1 and Eurocode 2. RAPT will always default to carrying out crack width calculations. If the designer considers that they are not necessary for a member, then this option can be used to force RAPT to ignore crack width calculations.

Shear Enhancement at Supports

This field allows users to nominate whether shear enhancement near supports (3 term in the beam shear equation) is to be used for AS3600, BS8110, SABS-0100, CP2004, IS456/ IS1343 and CP 65 designs (2d/a term in shear design). The default is No for AS3600 and Yes for BS8110, SABS-0100, CP2004, IS456/ IS1343 and CP 65. For AS3600 it should only be used under certain conditions defined in the standard and it's commentary. Other design standards are not affected by selection of this option.

When Shear Enhancement is selected, RAPT will calculate shear capacities at design locations from the face of the support rather than starting at the critical shear cross-section near the support.

Maximum Service Stress Change

Prestressed Cross-sections
Reinforced Cross-sections

Designers can limit the change in stress in the tension reinforcement under service load conditions to the input value. Separate limits can be applied for prestressed and reinforced cross sections. In a prestressed design where a member is stressed along only part of its length, RAPT will use the reinforced sections figure where no tendons exist. If the stress limit is exceeded, RAPT will add extra reinforcement to bring the change in stress back to the specified value. The various codes give limits depending on the type of member. Limiting the change in stress in tendons is a serviceability crack control check and more information is provided in 7.3.7.3 Flexural Design Service.

If the values input for either stress limit is < 10MPa, RAPT will ignore the stress check for that type, accepting the stress level in the reinforcement required for ultimate conditions. RAPT will initially set these values to the values set in the default file. If a value greater than 80% of the tensile strength of the reinforcement is defined, RAPT will reset it to 80% of the tensile strength of the reinforcement during the design.

Some codes refer to crack widths instead of a stress in the reinforcement. Users will need to calculate a reinforcement stress limit from the allowable crack width if they want to use this option. For Code limits see

1. AS3600 Clause 8.6.2 (Beams 200 MPa), 9.4.2 (Slabs 150 MPa)
2. ACI318 Clause 10.6.4, 18.10.2
3. BS8110 and CP65 Clause 2.2.3.4, 4.3.4.3 (Users need to calculate from crack widths)
4. Eurocode2 Clause 4.4.2, Table 4.11
5. SABS0100 Clause 3.2.3.3 (Users need to calculate an appropriate value)
6. CP2004 Clause 7.2.1 and 7.2.3
7. IS456/ IS1343 clause 43.1 and Annex F

For design codes which limit reinforcing bar spacing and/or bar size based on the stress in the reinforcement to achieve crack control for reinforced concrete members, RAPT will assume by default that the designer does not want to add extra reinforcement to provide crack control so the value of the reinforced Sections will default to 0. Crack control must then be achieved by limiting the bar spacing or bar size as appropriate.

If the designer wants to add extra reinforcement at service to reduce the reinforcement stress and thus provide a wider range of bar spacings or bar sizes from which to choose for crack control, then the stress limit can be applied here.

Flexural Critical Sections

Consider Transverse beams

Theory Section T5 defines how RAPT calculates the location of Flexural critical section near the face of the supports for different design standards. Users can have RAPT include or exclude any transverse beams entered in Input Screen in this determination. The YES option will include transverse beam dimensions in calculating the critical section for flexure.

Distance from centre of Support

RAPT allows users to pre-define the position of the flexural critical section. The critical section is defined as

Critical section = Factor x length of column to the face of the support.

where the factor is the value entered here. The input range allowed is 0 to 1.00. If users input -1 then RAPT defaults the critical sections to the code defaults. This factor is applied to all critical sections. The critical section is always measured from the column centre line. See Theory Section T5 for theory on calculation of Flexural Critical Sections for different design standards.

Capacity Reduction Factors

Flexure

The capacity reduction for flexural strength calculations.

Shear

The capacity reduction factor for beam and punching shear strength calculations.

Material Factors

Concrete Flexure

The material factor for concrete for flexural strength calculations.

Concrete Shear

The material factor for concrete for beam and punching shear strength calculations.

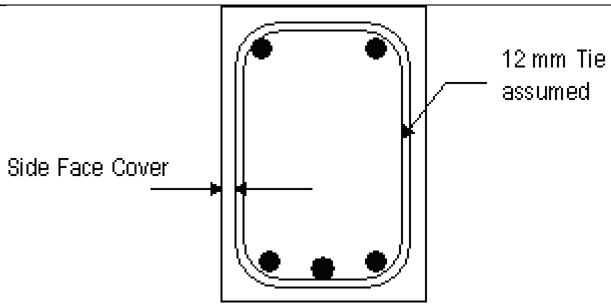
Reinforcement

The material factor for all reinforcement types for flexural strength and beam and punching shear strength calculations.

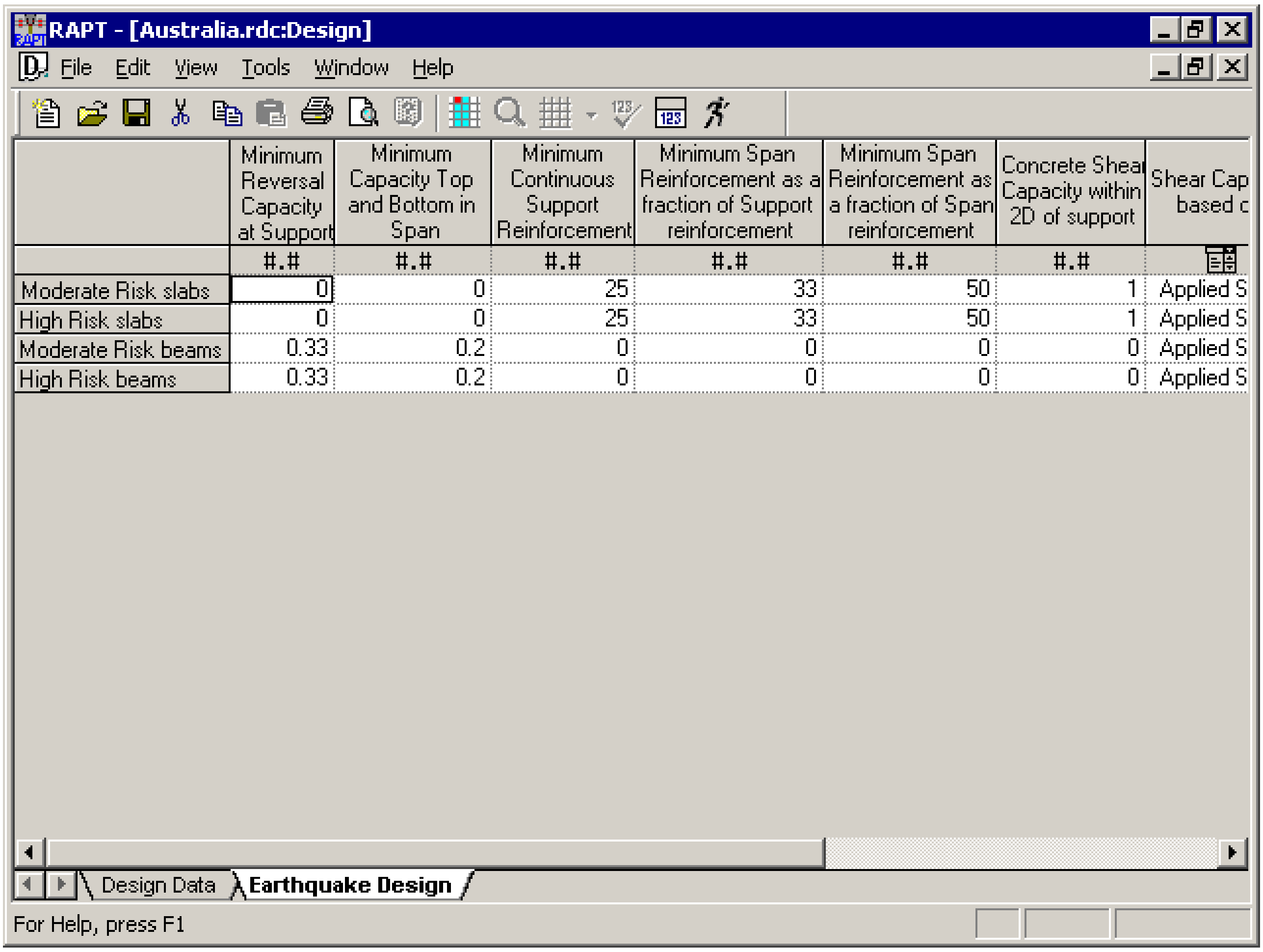
Beam Side Face Cover

Internal
External

Specified as the distance between the concrete surface and the outside of the shear tie. RAPT assumes that a 12 mm tie exists and then uses this data to decide what bar size and spacings are appropriate.



Earthquake Design



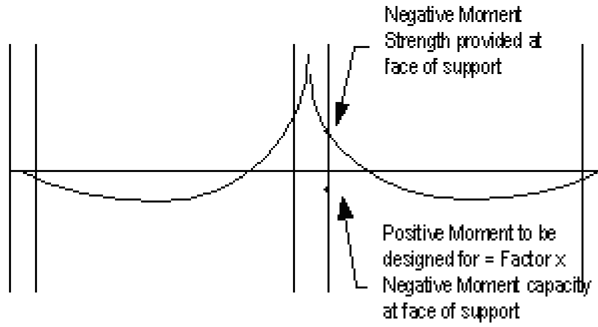
	Minimum Reversal Capacity at Support	Minimum Capacity Top and Bottom in Span	Minimum Continuous Support Reinforcement	Minimum Span Reinforcement as a fraction of Support reinforcement	Minimum Span Reinforcement as a fraction of Span reinforcement	Concrete Shear Capacity within 2D of support	Shear Cap based c
	#.#	#.#	#.#	#.#	#.#	#.#	
Moderate Risk slabs	0	0	25	33	50	1	Applied S
High Risk slabs	0	0	25	33	50	1	Applied S
Moderate Risk beams	0.33	0.2	0	0	0	0	Applied S
High Risk beams	0.33	0.2	0	0	0	0	Applied S

The data defined below can be input for the following conditions

1. Moderate Risk Slabs
2. High Risk slabs
3. Moderate Risk Beams
4. High Risk Beams

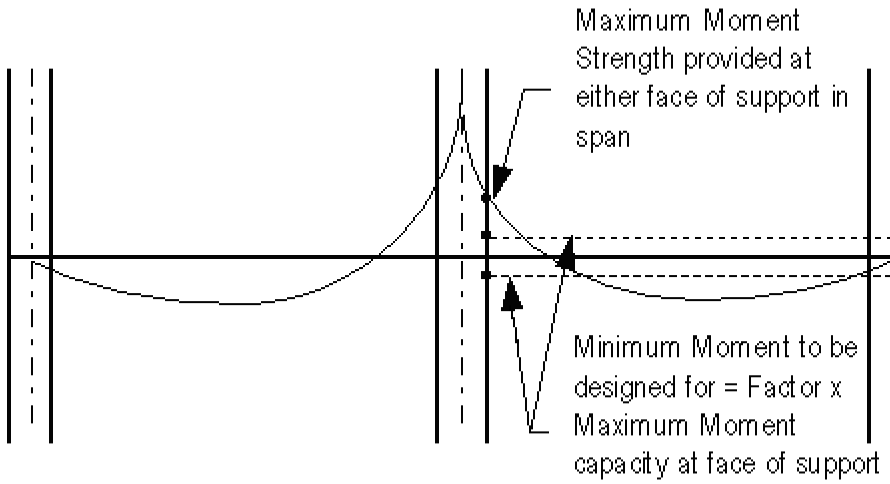
Data Definition

Min Reversal Capacity at Support
Some design codes require that the a fraction of the ultimate capacity at the critical section at the support be provided on the reverse face of the member for certain member types. At the critical sections at each support, this factor defines the minimum design moment for each face of the member. RAPT will look at the envelope of moments at the critical section and design for this fraction of the higher of the two moments as the minimum moment causing tension on each faces of the member. Even if the bending moment diagram indicates there is no tension stress on a face, this minimum moment will be designed for.



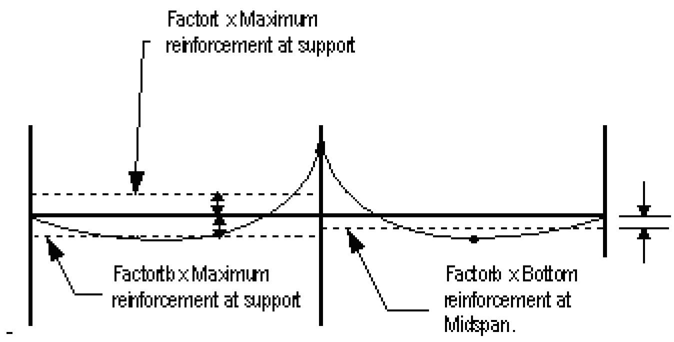
Min Capacity Top and Bottom in Span

Some design codes require that the a fraction of the ultimate capacity at the critical section at the support be provided at all points in both faces of the member for certain member types. At both faces of every design point in the member, the minimum design moment will be set to this factor multiplied by the largest support critical section moment. This will provide a minimum amount of continuous reinforcement in both faces of the member.



Min Continuous Top Reinforcement

Some design codes require that the a fraction of the reinforcement required at the critical sections at the supports in a span be provided as continuous reinforcement on that face of the member for certain member types (Factort below). RAPT will ensure that the reinforcement at each design section in a span on the support face is at least equal to this factor times the larger area of reinforcement at each end of the span.



Reinforcement Diagram

Min Bottom Reinforcement as a fraction of top reinforcement

Some design codes require that the a fraction of the reinforcement required at the critical sections at the supports in a span be provided as continuous reinforcement on the other face of the member for certain member types (Factortb above). RAPT will ensure that the reinforcement at each design section in a span on the span face is at least equal to this factor times the larger area of support reinforcement at each end of the span.

Min Bottom Reinforcement as a fraction of bottom reinforcement

Some design codes require that the a fraction of the maximum reinforcement required in a span be provided as continuous reinforcement on that face of the member for certain member types (Factorb above). RAPT will ensure that the reinforcement at each design section in a span on the span face is at least equal to this factor times the largest area of span reinforcement required in the span.

Concrete Shear Capacity within 2D of support
Some design codes specify that the concrete shear capacity should be reduced or ignored within 2D of the support. Input the factor to multiple the concrete shear capacity by here.

Shear Capacity based on
Some design codes specify that shear design be based on the capacity shear design in some earthquake members. This is normally required in areas of high seismic risk. RAPT gives the user the option of selecting

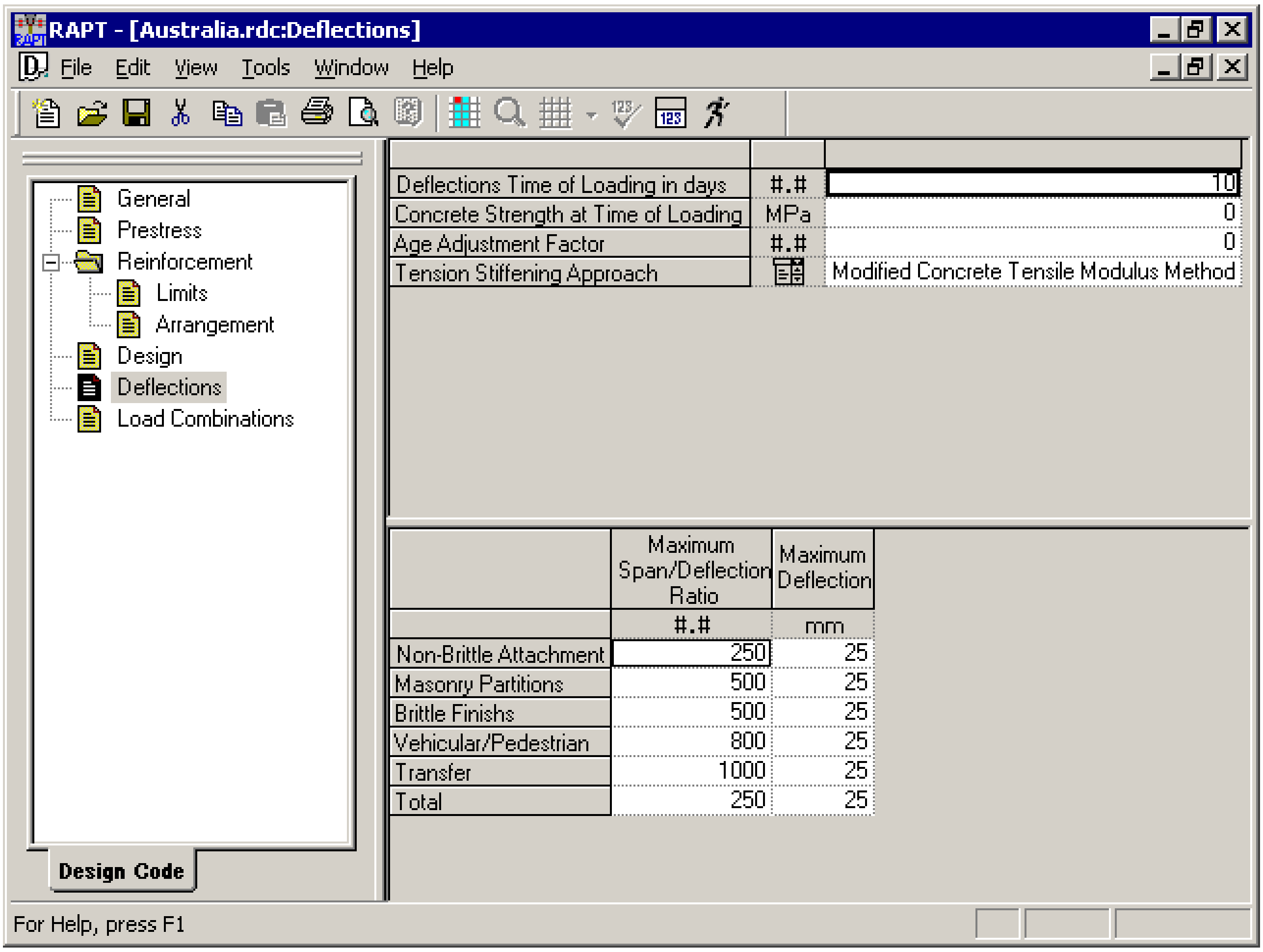
1. Applied Shear - Shear design is based on the elastic shear values calculated in the analysis.
2. Capacity Shear - Shear design is based on plastic shear values calculated from the Capacity Ultimate Moment Strength and the applied shear diagrams. This design option has not been added to RAPT at this stage.

If capacity design for shear is selected, the next two data fields are available.

Capacity Reduction Factor for Capacity Shear
When doing a capacity shear design, RAPT will use this factor as the capacity reduction factor in place of the normal ultimate strength capacity reduction factor defined in General Design above.

Steel Capacity Multiplier for Capacity Shear
When doing a capacity shear design, RAPT will use this factor to increase the reinforcing steel strength to a capacity strength rather than the normal yield strength defined in material properties. The Yield Strength will be multiplied by this factor.

5.6 Deflections



General Data

Deflections Time of Loading in days

Time at which the structure is first loaded. This is used for deflection calculations to determine long term creep and shrinkage coefficients. For Reinforced Concrete members, it is also the time at which the Transfer Deflection is calculated.

Concrete Strength at Time of Loading

RAPT calculates this value based on the strength gain formula using the strength gain constants defined in the 6.3 material properties. Users can modify the calculated value if required.

Age Adjustment Factor

This factor is used to adjust the effective modulus to allow for the time of loading and rate of application of load/stress. RAPT calculates this factor based on

$$\chi(t, \tau_0) = \frac{\tau_0^{\,0.5}}{\left(1 + \tau_0^{\,0.5}\right)} \qquad \text{Ageing coefficient}$$

$$\tau_0 = \text{Initial time of loading}$$

See Section T7.7.5 for more details.

Tension Stiffening Approach

RAPT can estimate the tension stiffening effect in concrete in three ways. Users can specify which method they would like RAPT to use. The three approaches are

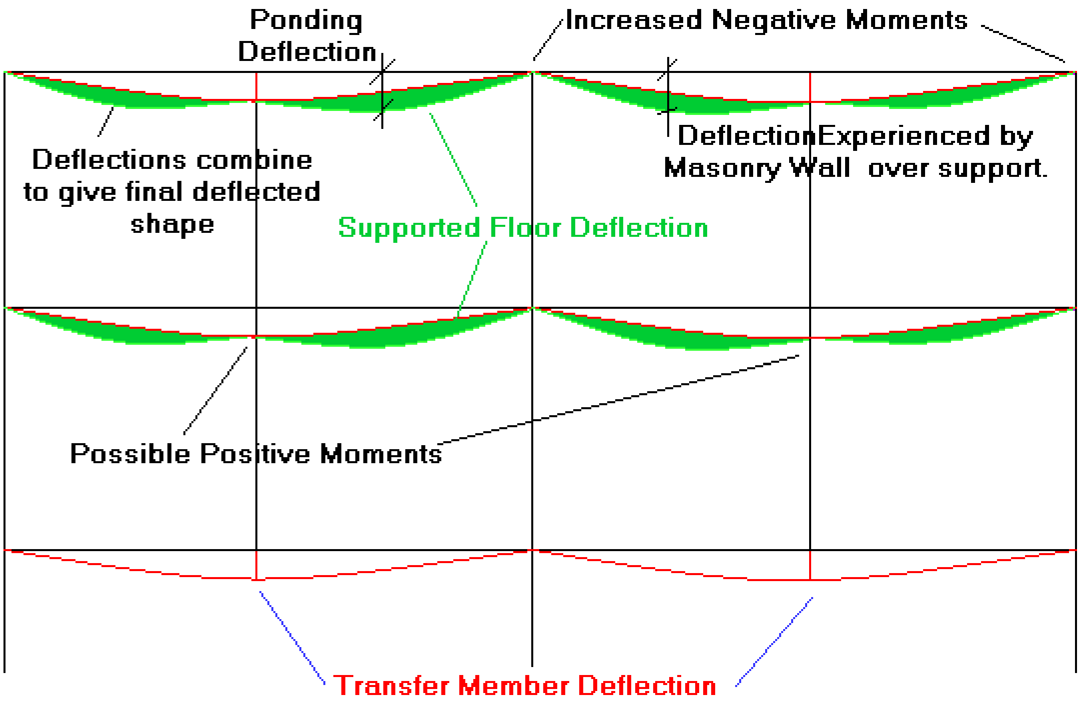
1. Branson's Formula
2. Eurocode 2 Formula
3. Modified Concrete Tensile Modulus Method. RAPT's default and preferred method. We prefer this method as it actually tries to estimate a tension stiffening effect in the actual cracked section calculations rather than fudging a guess at the effect based on an averaging formula.

See Theory Section T7.7.7. for more detailed information on these methods.

Deflection Warnings

The data defined below can be input for the following conditions

1. Non Brittle Attachments:- All attachments to the member are non-brittle and will not be damaged by deflections.
2. Masonry Partitions:- Masonry Partitions that will be affected by deflection of the member are being supported by the member. Some design standards, e.g. AS3600, have multiple limits for this limit depending on the degree of jointing in the wall. The limit chosen here is the lesser limit assuming walls are jointed as per the code requirements. Designers should consider increasing the limit if walls are not to be jointed sufficiently. This limit is normally applied to the incremental deflection.
3. Brittle Finishes:- Other brittle finishes are being attached to the member, e.g. glass curtain walls, Storage racking and Compactus units which may be deflection sensitive. This limit is normally applied to the incremental deflection.
4. Vehicle/Pedestrian Traffic:- This limit is often applied to limit vibration effects in structures. The limit is not normally applied to parking garages unless the vibrations will affect attached residential of office accommodation. This limit is normally applied to the live load deflection. If this limit is to be used in RAPT, the 7.2.4.2.4 Initial Load Combination should be modified to include all loading other than the live load. The resulting incremental deflection will then be the live load deflection.
5. Transfer Members:- Transfer members are a special case because they support other structural elements. The deflection of the transfer member will affect



1. The strength design of the supported structural elements due to support settlement
2. The detailing of the supported structural elements due to support settlement
3. The deflection of the supported structural elements is increased due to support settlement. This needs to be considered in the determination of the total deflection experienced by attachments to the member and also in the total deflection calculations and the effect on problems such as ponding on rooves.

If these effects are not fully allowed for in the design of the supported elements, the deflection of the transfer member should be limited severely. Our recommendation is a minimum limit of span/1000.

6. Total:- The limit to apply to the total deflection.

Data Definition

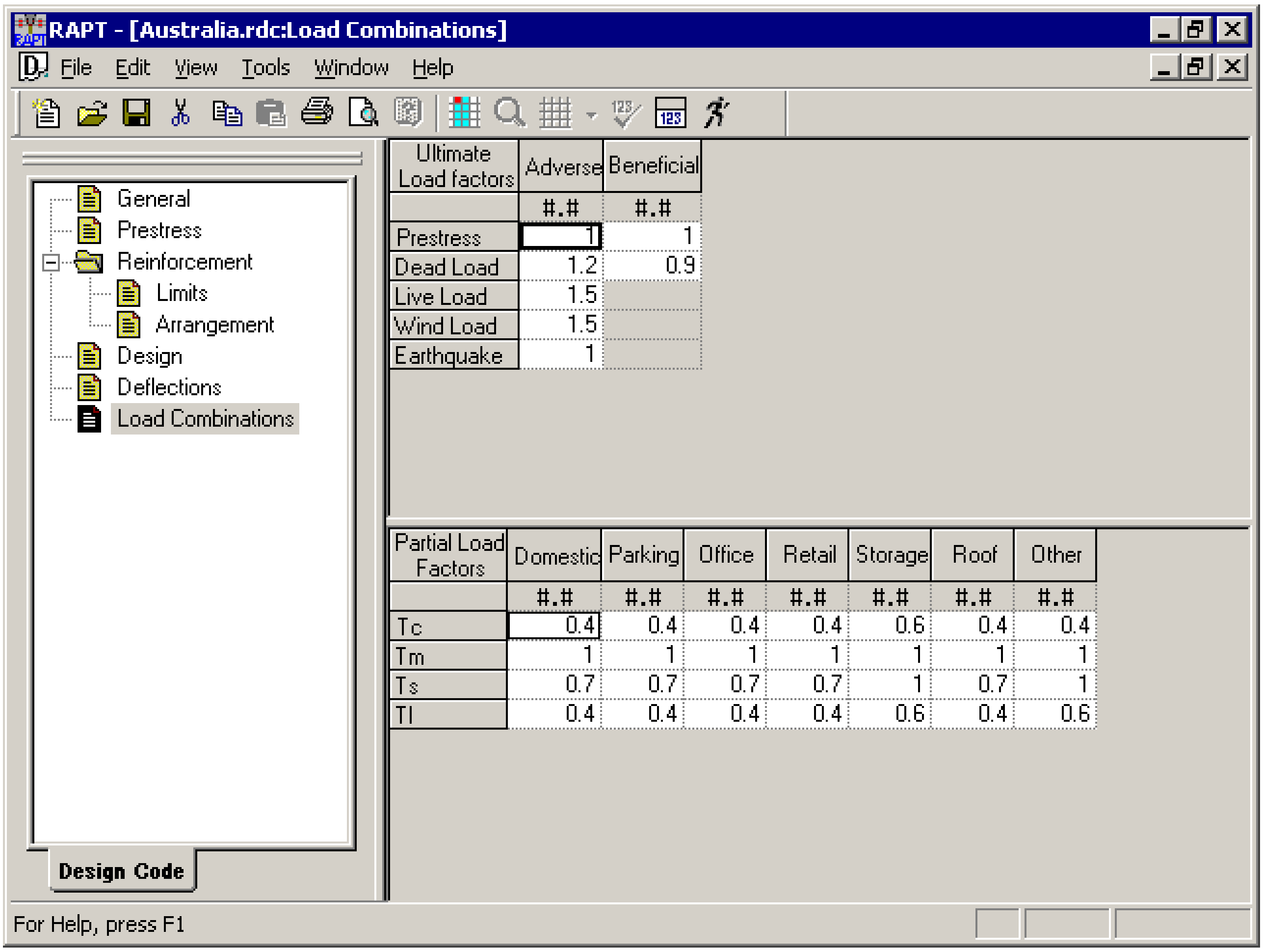
Maximum Span / Deflection Ratio

A limit at which RAPT will print warnings to the warning file telling the designer that the deflections have exceeded this value. RAPT will halve this figure when checking span / deflection ratios in cantilevers. If the value is set to zero RAPT will ignore the check.

Maximum Deflection

RAPT will write a warning message in the warnings report if this value is exceeded. RAPT will use the same maximum deflection for supported spans and cantilevers. If the value is set to zero RAPT will ignore the check.

5.7 Load Combinations



Ultimate Load factors

Ultimate Load Combination Factors are provided for two conditions

1. Adverse:- This load will increase the value of the combination
2. Beneficial:- This load will reduce the value of the combination

The load types for which default factors are provided are

1. Prestress
2. Dead Load
3. Live Load
4. Wind Load
5. Earthquake Load.

Partial Load Factors

The way in which RAPT uses partial factors is explained below for different Design Standards.

1. (i) AS3600
    1. Tc Combination live load factor used in assessing the design load for strength
    2. Ts Short term live load factor used in assessing the design load for serviceability
    3. Tl Long term live load factor used in assessing the design load for serviceability
    4. Tm Multiplying factor = 1
2. (ii) Eurocode2
    1. Tc (To in Eurocode2) Combination coefficient for representative value of a variable action. [See Eurocode1 8.4 & Table 3, Notation);
    2. Ts (T1 in Eurocode2) Combination coefficient for frequent value of a variable action.
    3. Tl (T2 in Eurocode2) Combination coefficient for quasi-permanent value of a variable action.
    4. Tm Multiplying factor = 1
3. (iii) ACI318
    1. Tc Combination factor = 1
    2. Ts Short term live load factor used in assessing the design load for serviceability
    3. Tl Long term live load factor used in assessing the design load for serviceability.

   4. Multiplying factor used in assessing the design load for strength. [See ACI318 Clause 9.2.2 & 9.2.3 - *m = 0.75]
4. (iv) BS8110, CP65, CP2004, IS456/ IS1343 and SABS0100
   1. Tc Combination factor = 1
   2. Short term live load factor used in assessing the design load for serviceability
   3. Tl Long term live load factor used in assessing the design load for serviceability.
   4. Multiplying factor used in assessing the design load for strength. [See BS8110 Table 2.1 Case 3 - *m = 1.2]
5. (v) AS1480/81
   1. Tc Combination factor = 1
   2. Ts Short term live load factor used in assessing the design load for serviceability
   3. Tl Long term live load factor used in assessing the design load for serviceability.
   4. Multiplying factor used in assessing the design load for strength. [See AS1480 Clause 13.1.1 - *m = 1.25]

6 Materials

6.1 Introduction

When a new run is commenced, the materials parameters defined in the default Materials are inserted into the data for the run. In the run data, the designer can select different Materials Data if desired. The basic Materials data will be inserted into the run data from the newly selected Materials Data set and RAPT will attempt to select materials as close as possible to those previously (e.g. concrete strengths, prestress steel and reinforcing steel types and sizes) selected from the new materials data file where applicable.

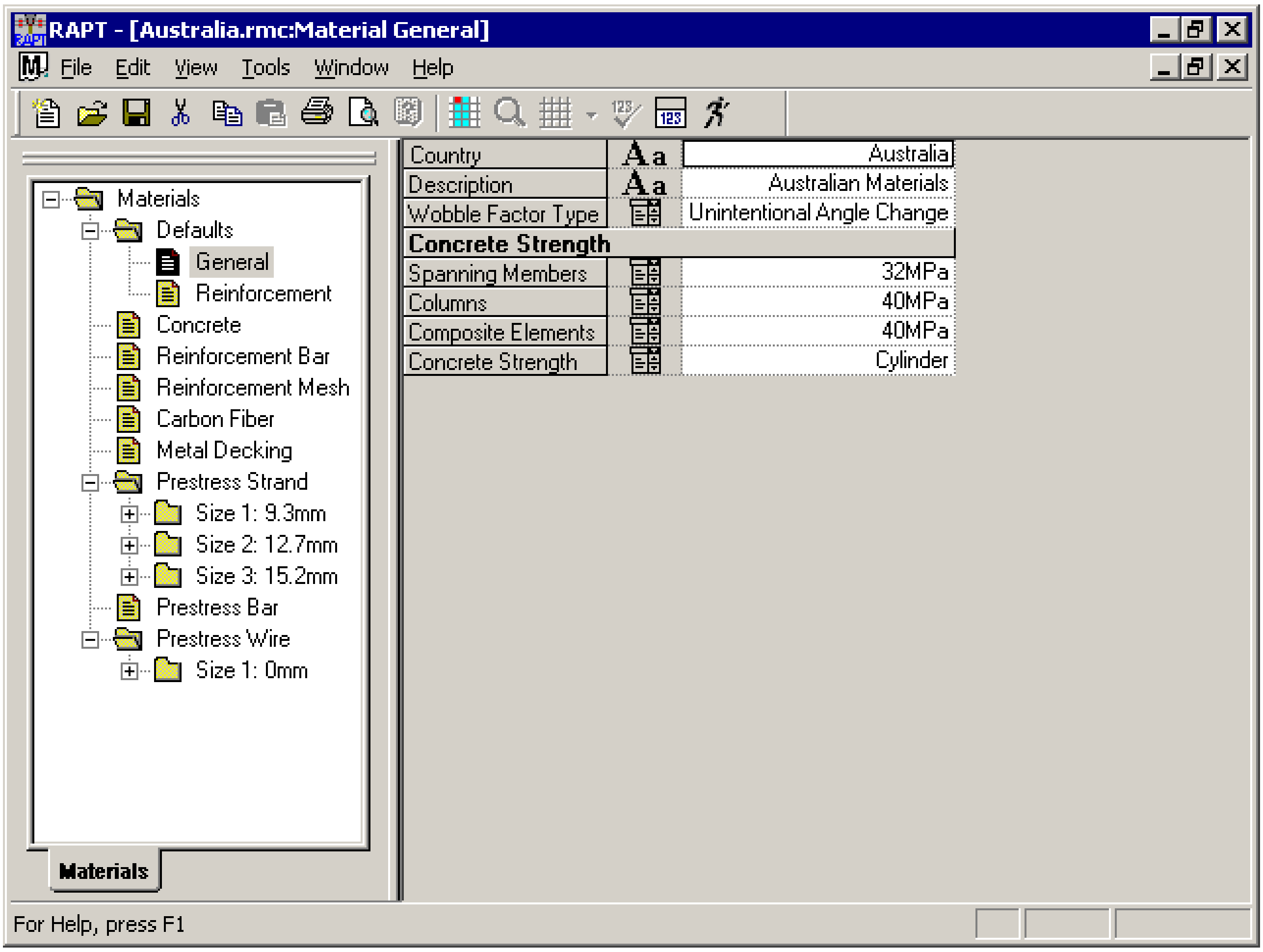
Several Materials Data files are provided with RAPT and the designer can define more variations if desired. RAPT also provides a base Materials Data set for each Country included in the program. These are used as the basis of user defined Materials Data sets when creating a Materials Data variation.

If the materials file has been opened directly for editing, changes made to the file will only affect new runs or existing runs if the materials data is reset be selecting it again from the list of available materials data files in the run.

If the materials data being edited is part of a run, changes made to it will only affect the current run.

6.2 Defaults

6.2.1 General



Country
The country for which the materials data in this file is to be used.

Description
A description of up to 127 characters of the materials data in this file.

Wobble Factor Type
The prestress wobble factor type used in defining the wobble factors for losses calculations in this file. The options are

1. Unintentional Friction *K*:- The unintentional friction per unit length for the tendons. $K = u * ß$
2. Unintentional Angle Change *ß*:- The unintentional angle change per unit length for the tendons. It must be multiplied by the friction factor to calculate the friction per unit length for friction calculations.

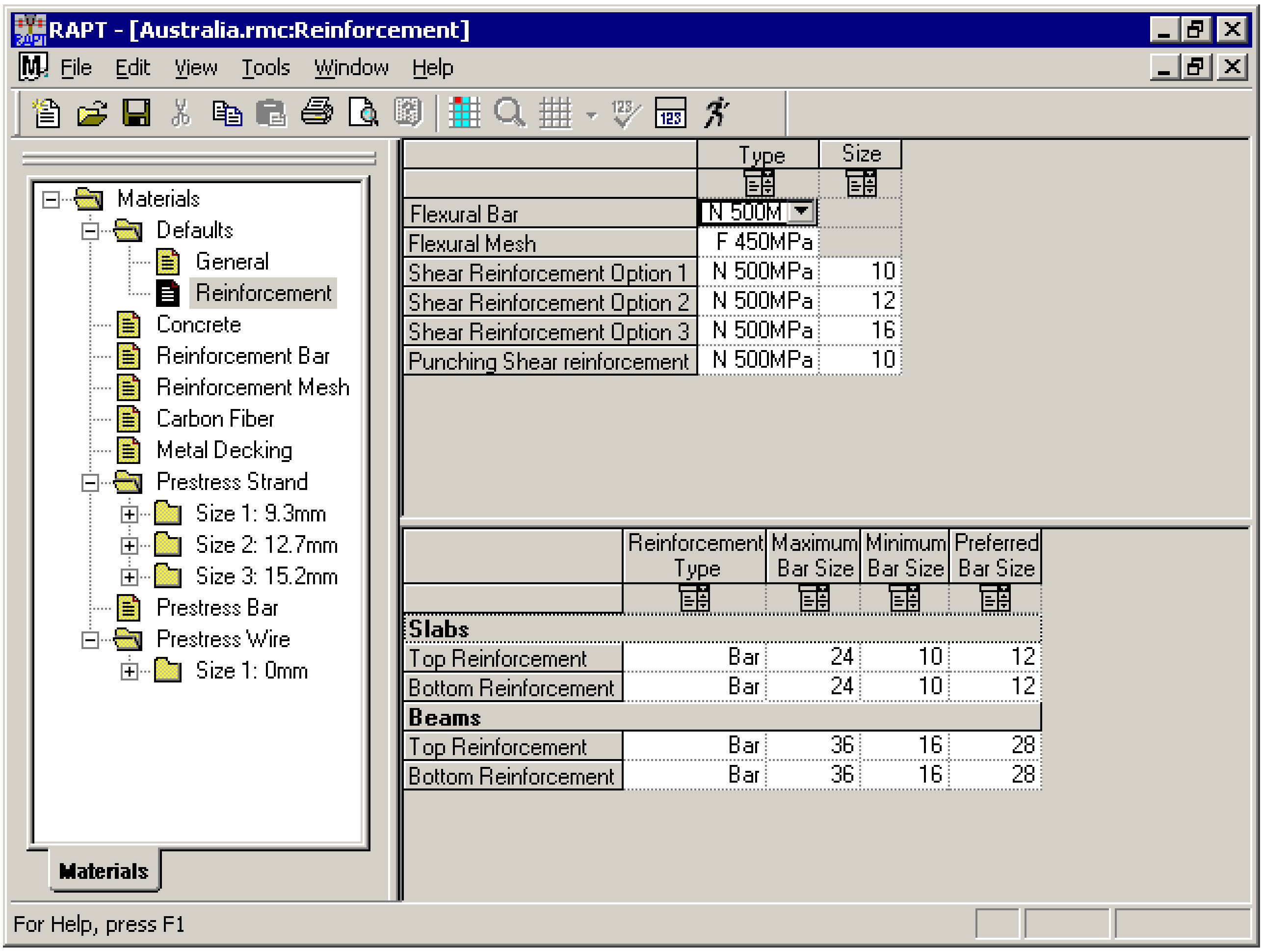
Concrete Strength

The default concrete strengths to use for different types of members. Values can be entered for

1. Spanning Members:- Slabs or beams
2. Columns:- Supporting columns
3. Composite Elements:- Precast members in composite construction.

.

6.2.2 Reinforcement



Reinforcement Types

For each of the following reinforcement requirements the designer can nominate the default data shown below.

1. Flexural Bar:- The type of reinforcing bar to be used as the default bar type for flexure calculations.
2. Flexural Mesh:- The type of reinforcing mesh to be used as the default mesh type for flexure calculations.
3. Shear Reinforcement Option 1 :- The type of reinforcing steel to be used as the 1st default steel type for beam shear calculations.
4. Shear Reinforcement Option 2:- The type of reinforcing steel to be used as the 2nd default steel type for beam shear calculations.
5. Shear Reinforcement Option 3:- The type of reinforcing steel to be used as the 3rd default steel type for beam shear calculations.
6. Punching Shear reinforcement:- The type of reinforcing steel to be used as the default steel type for punching shear calculations.

Data Definition

Type
The reinforcement type of this steel type selected from the list of available reinforcement types defined in this materials data file.

Size
The size of the reinforcement type selected above from the list of available sizes for this reinforcement type defined in this materials data file.

Reinforcement Sizes

For the following cases, the designer can specify the default data defined below

1. Slabs
    1. Top Reinforcement
    2. Bottom Reinforcement
2. Beams
    1. Top Reinforcement
    2. Bottom Reinforcement

Data Definition

Reinforcement Type

The type of flexural reinforcement to use by default for this member type. The options are

1. Bar
2. Mesh

The type of this flexural reinforcement type selected in Reinforcement Types above will be set as the default in any new runs for this member type.

Maximum Bar Size

The steel reinforcing size to use as the default maximum size in new runs for this member type.

Minimum Bar Size

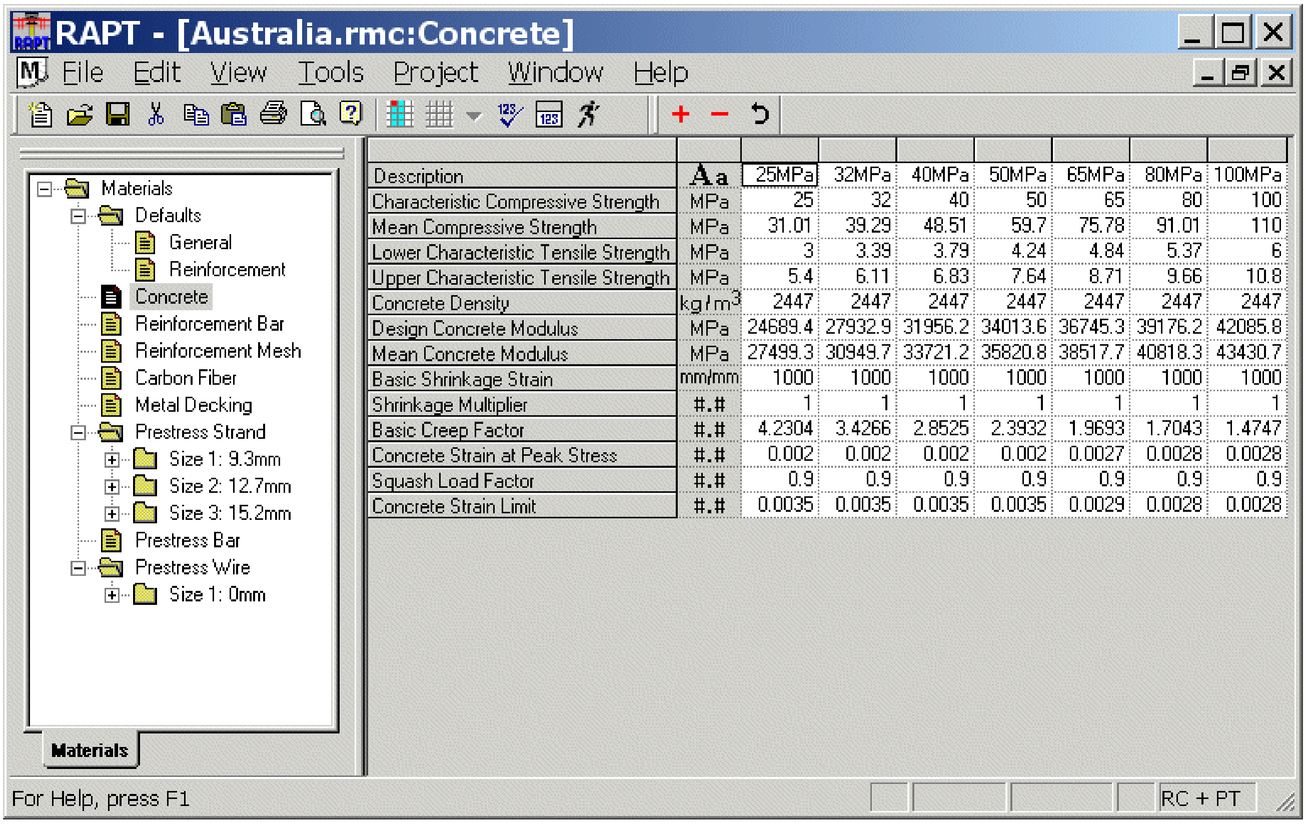
The steel reinforcing size to use as the default minimum size in new runs for this member type.

Preferred Bar Size

The steel reinforcing size to use as the default preferred size in new runs for this member type. This size must lie between the maximum and minimum sizes.

6.3 Concrete Properties

Any concrete strength that is to be used in a RAPT design must be defined here.



Description	Aa	25MPa	32MPa	40MPa	50MPa	65MPa	80MPa	100MPa
Characteristic Compressive Strength	MPa	25	32	40	50	65	80	100
Mean Compressive Strength	MPa	31.01	39.29	48.51	59.7	75.78	91.01	110
Lower Characteristic Tensile Strength	MPa	3	3.39	3.79	4.24	4.84	5.37	6
Upper Characteristic Tensile Strength	MPa	5.4	6.11	6.83	7.64	8.71	9.66	10.8
Concrete Density	kg/m^3	2447	2447	2447	2447	2447	2447	2447
Design Concrete Modulus	MPa	24689.4	27932.9	31956.2	34013.6	36745.3	39176.2	42085.8
Mean Concrete Modulus	MPa	27499.3	30949.7	33721.2	35820.8	38517.7	40818.3	43430.7
Basic Shrinkage Strain	mm/mm	1000	1000	1000	1000	1000	1000	1000
Shrinkage Multiplier	#.#	1	1	1	1	1	1	1
Basic Creep Factor	#.#	4.2304	3.4266	2.8525	2.3932	1.9693	1.7043	1.4747
Concrete Strain at Peak Stress	#.#	0.002	0.002	0.002	0.002	0.0027	0.0028	0.0028
Squash Load Factor	#.#	0.9	0.9	0.9	0.9	0.9	0.9	0.9
Concrete Strain Limit	#.#	0.0035	0.0035	0.0035	0.0035	0.0029	0.0028	0.0028

Toolbar

+ −

+ Add a new concrete strength. If a column is 4.4.3 selected by clicking the column header, the new concrete strength will be inserted at that location, otherwise it will be placed at the right end of the data grid.

− Delete the 4.4.3 selected column strengths.

Data Definition

Description

The name of the concrete strength. This is a text field so unit conversions do not work on it. It will be displayed in the strength selection lists.

Characteristic Compressive Strength

The characteristic compressive strength of the concrete as defined by various design codes. This filed will also be displayed in the strength selection lists. This strength is used for calculations based on the effects at an isolated location in the member e.g. ultimate flexure and shear calculations and crack width calculations.

When a value is entered in this field and accepted, RAPT will calculate a new set of data for all of the data below based on the formulae in each design code.

If the designer is editing the concrete strengths from within a run file, RAPT will use the design code selected in that file for the calculation of the values below. If the designer is editing a standalone Materials Data file, RAPT will prompt the designer for a design standard on which it should base these calculations.

Mean Compressive Strength

The mean compressive strength of the concrete. This strength is used for calculations based on the overall performance of the member e.g. analysis and deflection calculations. It is the characteristic mean, not the insitu mean as defined in AS3600.

Lower Characteristic Tensile Strength

The lower bound tensile strength of the concrete. This is often assumed to be 70% of the mean tensile strength of the concrete. This is the tensile strength defined in most design codes.

Upper Characteristic Tensile Strength

The upper bound tensile strength of the concrete. This is often assumed to be 130% of the mean tensile strength of the concrete. RAPT will assume the mean tensile strength of the concrete to be the average of the upper and lower bound values if it ever needs this value.

Concrete Density

The concrete density used for elastic modulus calculations for some design standards, eg AS3600 - 2400kg/m3.

Design Concrete Modulus

The characteristic elastic modulus of the concrete based on the Characteristic Concrete strength as defined in different design codes.

Mean Concrete Modulus

The mean elastic modulus of the concrete based on the Mean Concrete strength.

Basic Shrinkage Strain

The basic shrinkage strain is the shrinkage determined from tests on samples of concrete. The final shrinkage is determined from this by taking into account section shape and size, humidity conditions etc. This data is only used in cases where the ACI209 and AS3600 and Zia methods of calculation of long term shrinkage are used. The Eurocode 2, CEBFIP 1970 and CEBFIP 1978 methods all use formulae to calculate the basic shrinkage strain value. Any value nominated here will be ignored unless the ACI209, AS3600 or Zia calculation methods are selected either by default or as the method to use for that design.

Shrinkage Multiplier

Special Shrinkage Factor to allow the user to define modifications to the standard shrinkage of concrete.

Some shrinkage calculation methods do not define a fixed value for Basic Shrinkage Strain as mentioned above. IN these cases, designers can still allow for variations in shrinkage from the value calculated in the method by nominating a multiplier here to allow for local variations in shrinkage from the calculation model used or to allow for the use of shrinkage reducing agents. RAPT will allow this factor to be set to any value between .1 and 10. We are not suggesting that values at the extremes of this range be used. The actual value used should represent the actual shrinkage characteristics of the concrete being used in the design.

For Hong Kong CP2004, the multiplying factor c_s = 3 is nominated here. The shrinkage strain is calculated from the CEBFIP 1970 method on which the Kong Kong code shrinkage calculations are based and multiplied by the c_s factor nominated here. If a different value is required for this c_s factor, it can be substituted in each individual run or in the material defaults file for all future runs. The c_s value of 3 results in large estimated shrinkage values compared to other codes and countries and is inconsistent with the span/depth ratio logic of CP2004 which is based on a c_s factor of 1.

Basic Creep Factor

The basic creep strain is the reference creep factor determined from tests on samples of concrete. The final creep coefficient is determined from this by taking into account section shape and size, humidity conditions etc. This data is only used in cases where the ACI209 and AS3600 methods of calculation of long term creep are used. The Zia method does not calculate a logical creep effect at all and the Eurocode 2, CEBFIP 1970 and CEBFIP 1978 methods all use formulae to calculate the creep value. Any value nominated here will be ignored unless the ACI209 or AS3600 calculation methods are selected either by default or as the method to use for that design.

Concrete Strain at Peak Stress

The strain in the concrete at the peak compression stress. Normally in the range of .002 for low strength concretes to .0028 for higher strength concretes. This should never be larger than the Concrete Strain Limit and for very high strength concretes is normally equal to the Concrete Strain Limit.

Refer to Theory section T.7.3.1 T.7.3.1 for information on the concrete stress/strain curve to which these variables apply.

Squash Load Factor

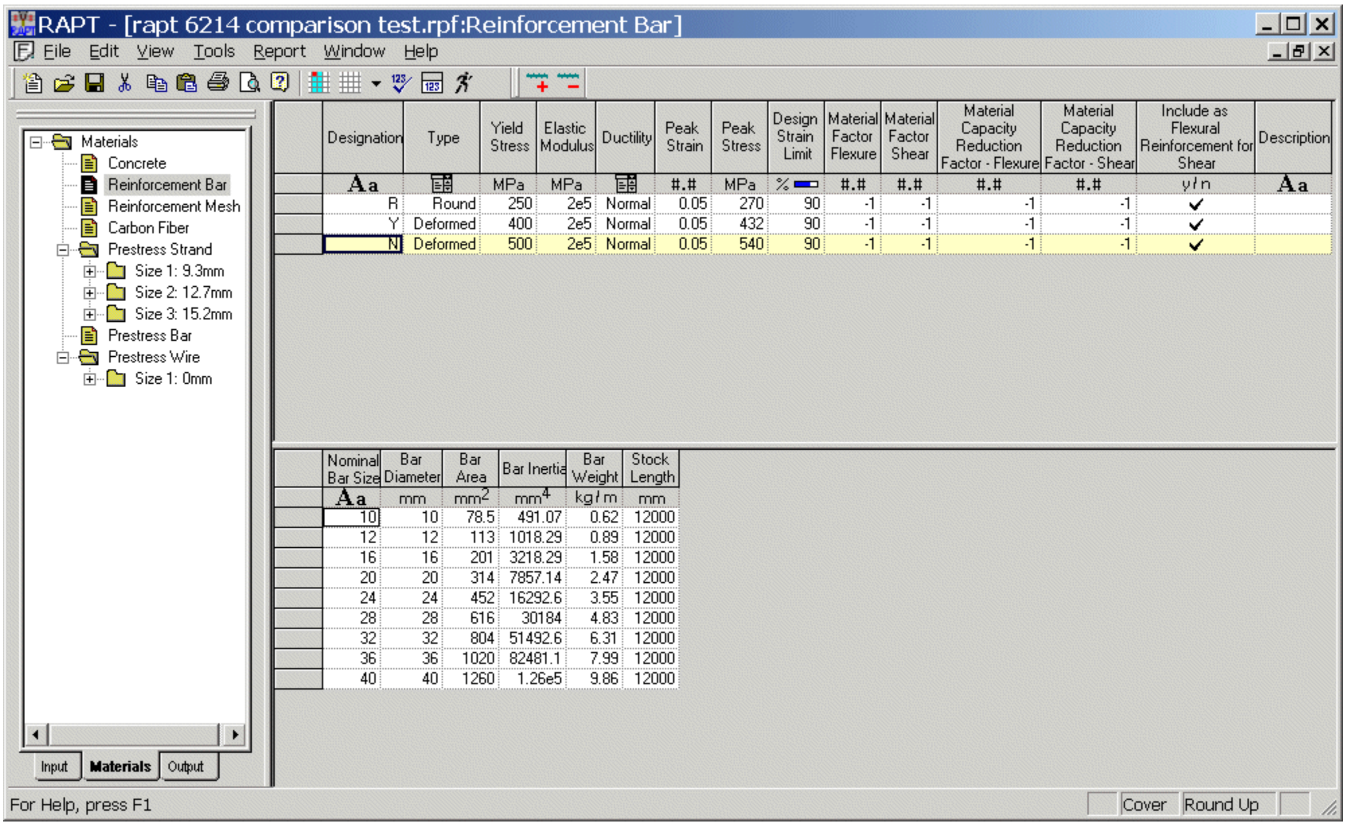
Fraction of Characteristic Concrete Compression Strength at the Peak Stress point on the concrete stress/strain curve.

AS3600 - 2009	.9
AS3600	.85
BS8110	.85 (RAPT applies a separate cube to cylinder factor to the concrete strength) to give a final factor of .67
ACI318	.85
Eurocode2	.85
SABS 0100	.85
Indian Code	.85
Canadian Code	.9
NZ Code	.85
Singapore Code	.85
Hong Kong Code	.85

Concrete Strain Limit

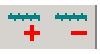
The limiting concrete strain at the extreme compressive fibre. Normally in the range of .0028 for low strength concretes to .0035 for higher strength concretes. For design codes based on ACI318 code, this is limited to .003. For design codes based on BS8110, Eurocode 2 and AS3600, this is limited to .0035.

6.4 Reinforcement Bar



In this data screen, the designer defines the types of reinforcement available and the basic properties for each reinforcement type in the control grid. Each reinforcement type has a child grid defining the Reinforcement Sizes associated with that type.

Toolbar Functions



 Add an new reinforcement type. If an existing reinforcement type is selected, the new type will be inserted at that location and the type at this location will me shifted down, otherwise the new type will be added after of the existing types.

 Delete the selected reinforcement types.

Reinforcement Types

Designation
The abbreviation to be used to describe this reinforcement type.

Type
The surface treatment of the bar. Options are

1. Round (plain)
2. Deformed

Yield Stress
The nominated yield strength of the bar.

Elastic Modulus
The elastic modulus of the bar.

Ductility
The ductility type of the bar. Options are

1. Normal
2. Low

Peak Strain
The strain at which the bar reaches it's peak stress level on the stress strain curve for the bar.

Peak Stress
The stress at the peak strain

Design Strain Limit
Set a limit on the maximum strain allowed in the bar as a percentage of the Peak Strain. This is only activated when the designer requests that limits on steel strain be applied in Design Data.

Material Factor Flexure
Material Factor for this type of reinforcement only for flexure design. This value will be used in ultimate flexure design using this type of reinforcement if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Factor Shear
Material Factor for this type of reinforcement only for shear design. This value will be used in shear design using this type of reinforcement if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Capacity Reduction Factor Flexure
Allows the designer to specify a material specific capacity reduction factor for flexural design for this type of reinforcement. If this value is less than the general capacity reduction factor for flexure, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Material Capacity Reduction Factor Shear
Allows the designer to specify a material specific capacity reduction factor for shear design for this type of reinforcement. If this value is less than the general capacity reduction factor for shear, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Include as Flexural Reinforcement for shear
This reinforcement is included in the calculations for the total amount of flexural reinforcement to be used in calculating the concrete contribution to shear capacity.

Description
A text description of the bar type.

Reinforcement Sizes

Nominal Bar Size
The nominal bar diameter used in defining and selecting this bar size. This is a text field and is unitless. The nominal size can include any text character.

Bar Diameter
The actual bar diameter.

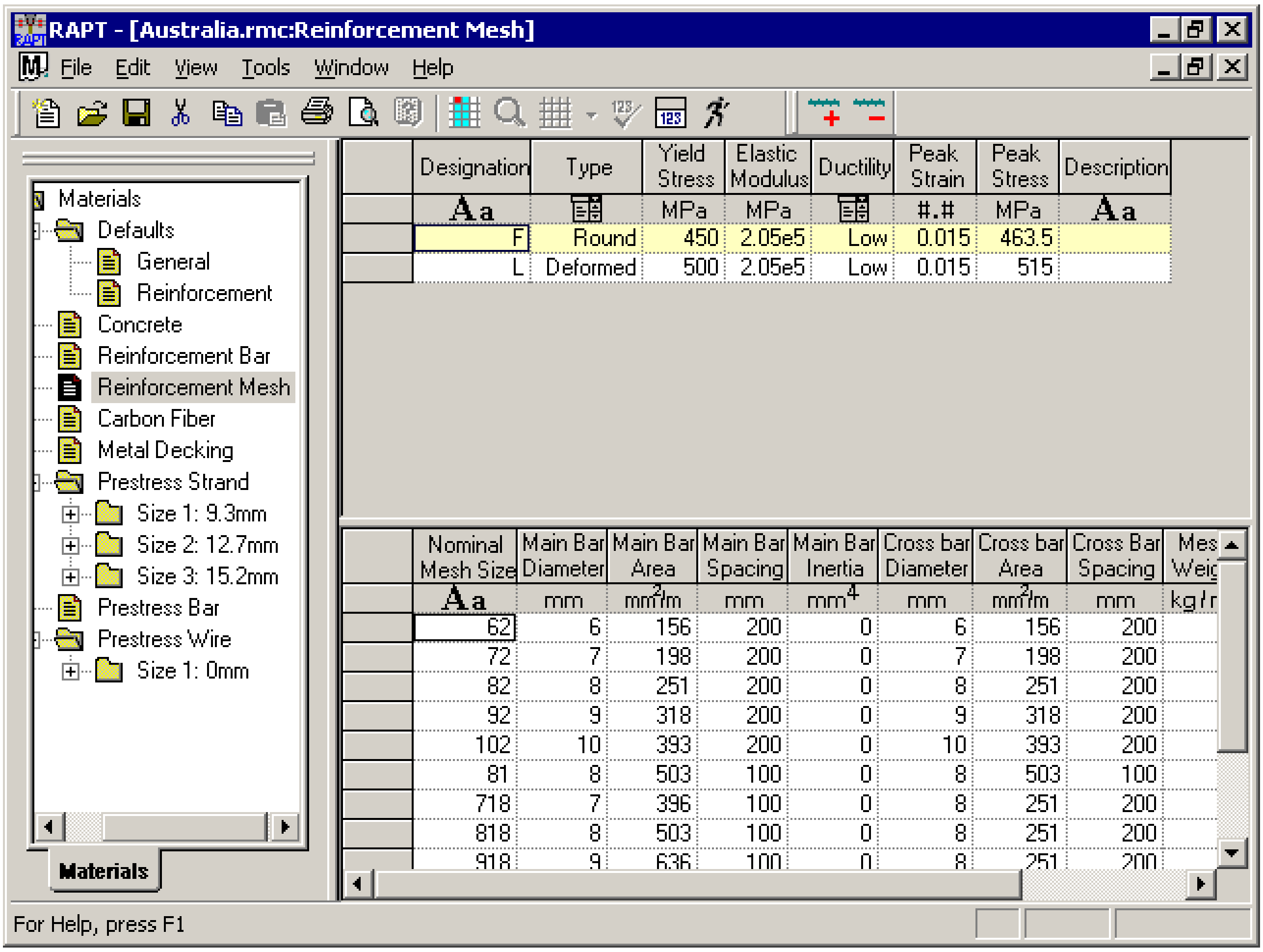
Bar Area
The area of the bar.

Bar Inertia
The moment of inertia of the bar about it's centroid.

Bar Weight
The weight of the bar per unit of length.

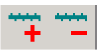
Stock Length
The maximum length of the bar that can be supplied.

6.5 Reinforcement Mesh



In this data screen, the designer defines the types of reinforcing mesh available and the basic properties for each reinforcing mesh type in the control grid. Each reinforcing mesh type has a child grid defining the Mesh Sizes associated with that type.

Toolbar Functions



 Add an new Reinforcing Mesh type. If an existing Reinforcing Mesh type is selected, the new type will be inserted at that location and the type at this location will me shifted down, otherwise the new type will be added after of the existing types.

 Delete the selected Reinforcing Mesh types.

Reinforcing Mesh Types

Designation
The abbreviation to be used to describe this reinforcing mesh type.

Type
The surface treatment of the bars used to make this reinforcing mesh. Options are

1. Round (plain)
2. Deformed

Yield Stress
The nominated yield strength of the bars used to make this reinforcing mesh.

Elastic Modulus
The elastic modulus of the bars used to make this reinforcing mesh.

Ductility
The ductility type of the bars used to make this reinforcing mesh. Options are

1. Normal
2. Low

Peak Strain
The strain at which the bar reaches it's peak stress level on the stress strain curve for the bars used to make this reinforcing mesh.

Peak Stress
The stress at the peak strain for the bars used to make this reinforcing mesh.

Design Strain Limit
Set a limit on the maximum strain allowed in the mesh as a percentage of the Peak Strain. This is only activated when the designer requests that limits on steel strain be applied in Design Data.

Material Factor Flexure
Material Factor for this type of mesh only for flexure design. This value will be used in shear design using this type of mesh if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Factor Shear
Material Factor for this type of mesh only for shear design. This value will be used in shear design using this type of mesh if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Capacity Reduction Factor Flexure
Allows the designer to specify a material specific capacity reduction factor for flexural design for this type of mesh. If this value is less than the general capacity reduction factor for flexure, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Material Capacity Reduction Factor Shear
Allows the designer to specify a material specific capacity reduction factor for shear design for this type of mesh. If this value is less than the general capacity reduction factor for shear, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Include as Flexural Reinforcement for shear
This mesh is included in the calculations for the total amount of flexural reinforcement to be used in calculating the concrete contribution to shear capacity.

Description
A text description of the bar type.

Reinforcing Mesh Sizes

Nominal Mesh Size
The nominal mesh size used in defining and selecting this mesh size. This is a text field and is unitless. The nominal size can include any text character. It is normally related to the nominal main bar size and spacing.

Main Bar Diameter
The actual diameter of the main bar in the this mesh.

Main Bar Area
The area of the main bars in the this mesh per unit of width of mesh.

Main Bar Spacing
The transverse spacing of the main bars in the this mesh.

Main Bar Inertia
Moment of Inertia of the main bars about the centroid of the bars in this mesh. This is the value for a single bar, not per unit width.

Cross bar Diameter
The actual diameter of the cross bars in the this mesh.

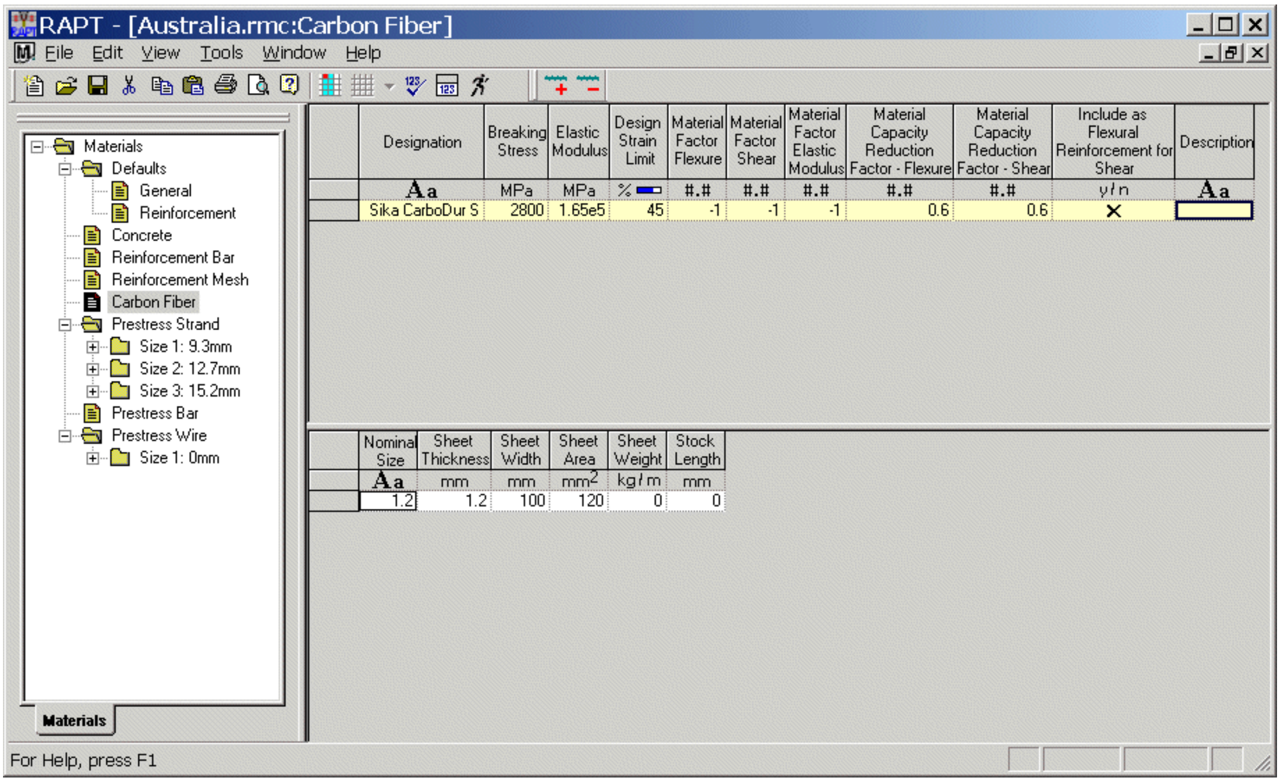
Cross bar Area
The area of the cross bars in the this mesh per unit of width of mesh.

Cross Bar Spacing
The transverse spacing of the cross bars in the this mesh.

Mesh Weight
The weight of this mesh per unit width square.

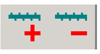
Stock Length
The standard length that this mesh is made to.

6.6 Carbon Fibre



In this data screen, the designer defines the types of Carbon Fibre or Fibre Reinforced Polymers available and the basic properties for each type in the control grid. Each Fibre Reinforced Polymer type has a child grid defining the sizes associated with that type.

Toolbar Functions



 Add an new Carbon Fibre type. If an existing Carbon Fibre type is selected, the new type will be inserted at that location and the type at this location will me shifted down, otherwise the new type will be added after of the existing types.

 Delete the selected Carbon Fibre types.

Carbon Fibre Types

Designation
The abbreviation to be used to describe this Carbon Fibre type.

Yield Stress
The nominated yield strength of the bars used to make this Carbon Fibre type.

Elastic Modulus
The elastic modulus of the bars used to make this Carbon Fibre type.

Design Strain Limit
Set a limit on the maximum strain allowed in the Carbon Fibre as a percentage of the Peak Strain. This is only activated when the designer requests that limits on steel strain be applied in Design Data.

Material Factor Flexure
Material Factor for this type of Carbon Fibre only for flexure design. This value will be used in flexure design using this type of Carbon Fibre if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Factor Shear
Material Factor for this type of Carbon Fibre only for shear design. This value will be used in shear design using this type of Carbon Fibre if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Factor Elastic Modulus
Material Factor for the Elastic Modulus for this type of Carbon Fibre. This value will be used in shear design using this type of Carbon Fibre in combination with the factored design strength.

Material Capacity Reduction Factor Flexure
Allows the designer to specify a material specific capacity reduction factor for flexural design for this type of Carbon Fibre. If this value is less than the general capacity reduction factor for flexure, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Material Capacity Reduction Factor Shear
Allows the designer to specify a material specific capacity reduction factor for shear design for this type of Carbon Fibre. If this value is less than the general capacity reduction factor for shear, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Include as Flexural Reinforcement for shear
This Carbon Fibre is included in the calculations for the total amount of flexural reinforcement to be used in calculating the concrete contribution to shear capacity.

Description
A text description of the bar type.

Carbon Fibre Sizes

Nominal Carbon Fibre Size
The nominal Carbon Fibre size used in defining and selecting this Carbon Fibre size. This is a text field and is unitless. The nominal size can include any text character. It is normally related to the nominal main bar size and spacing.

Sheet Thickness
The actual diameter of the main bar in the this mesh.

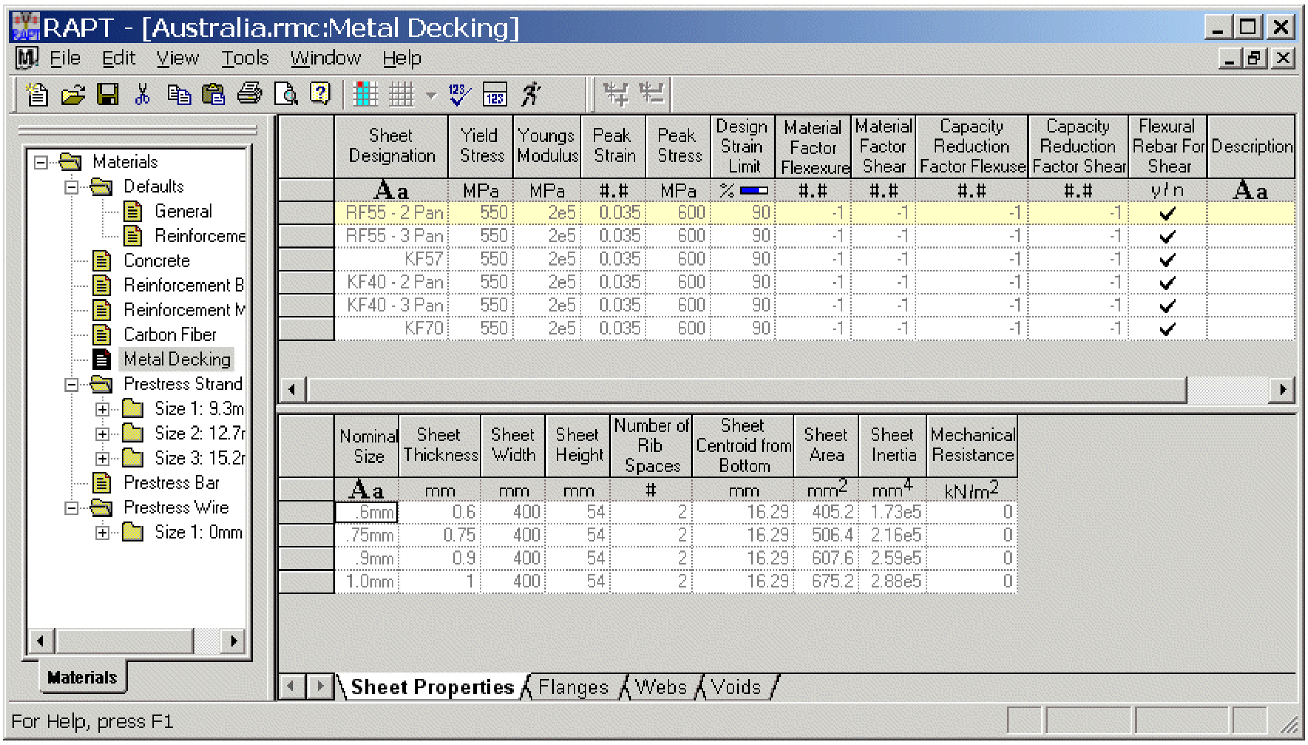
Sheet Width
The actual diameter of the main bar in the this mesh.

Sheet Area
The area of the main bars in the this mesh per unit of width of mesh.

Sheet Weight
The weight of this mesh per unit width square.

Stock Length
The standard length that this mesh is made to.

6.7 Structural Decking Types



Structural Decking types in RAPT define a sheet type of specific profile, metal properties, capacity factors and friction characteristics. The design method used does not rely on bond, it relies on friction between the concrete and the decking to mobilize stresses in the decking from deflection of the concrete member as discussed in the theory section on Metal Decking T.18.

RAPT requires that a metal decking sheet be defined as a series of horizontal flanges and vertical webs. This is an idealised modelling of the actual shape of the decking. There are often sloping elements, curves and lips that need to be accounted for. RAPT's logic is to define a series of horizontal flanges of the same thickness as the sheet thickness to model the flat surfaces, and then for the remainder of the sheeting to be included in a series of vertical webs. The thickness of these webs is defined as the Sheet Thickness * Thickness Ratio. The thickness Ratios are calculated to ensure that the total area of the combined flanges and webs is equal to the actual sheet area. As many webs as desired may be defined to ensure that the centroid of the area of the flanges and webs is equal to the centroid of area of the sheet. RAPT will give a warning if the calculated area is not within .1% of the nominated sheet area and if the calculated centroid is not within 5% of the nominated centroid. This data check will be carried out and warning given when an attempt is made to run a RAPT data file with a specific sheet type nominated.

As well, for each structural decking type in the Control data (each row of the grid), there are four further screens of data, accessed via the View Tabs at the bottom of the Child Views, to define

1. 6.7.1 Sheet Properties
2. 6.7.2 Sheet Flanges
3. 6.7.3 Sheet Webs
4. 6.7.4 Concrete Voids

Toolbar

 Add a structural decking type. If a structural decking type is selected before adding, the new type will be inserted at the location of the selected type, otherwise it will be added at the end of the list.

 Delete selected structural decking types. All data associated with a deleted structural decking type will be lost.

Data Definition

Sheet Designation
The abbreviation to be used to describe this structural decking type.

Yield Stress
The nominated yield strength of the structural decking type.

Young's Modulus
The elastic modulus of the structural decking type.

Peak Strain
The strain at which the metal decking reaches it's peak stress level on the stress strain curve for the structural decking type.

Peak Stress
The stress at the peak strain

Design Strain Limit
Set a limit on the maximum strain allowed in the structural decking type as a percentage of the Peak Strain. This is only activated when the designer requests that limits on steel strain be applied in Design Data.

Material Factor Flexure
Material Factor for this structural decking type only for flexure design. This value will be used in ultimate flexural design using this structural decking type if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Factor Shear
Material Factor for this structural decking type only for shear design. This value will be used in shear design using this structural decking type if it is greater than the general material factor for reinforcement defined in the Design Code or the input data for a run.

Material Capacity Reduction Factor Flexure
Allows the designer to specify a material specific capacity reduction factor for flexural design for this type of reinforcement. If this value is less than the general capacity reduction factor for flexure, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Material Capacity Reduction Factor Shear
Allows the designer to specify a material specific capacity reduction factor for shear design for this structural decking type. If this value is less than the general capacity reduction factor for shear, RAPT will use this factor in combination with the general capacity reduction factor by introducing a material factor equivalent to the general capacity reduction factor / the material capacity reduction factor.

Include as Flexural Reinforcement for shear
This structural decking type is included in the calculations for the total amount of flexural reinforcement to be used in calculating the concrete contribution to shear capacity.

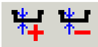
Description
A text description of the structural decking type.

6.7.1 Structural Decking Sheet Properties

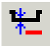
	Nominal Size	Sheet Thickness	Sheet Width	Sheet Height	Number of Rib Spaces	Sheet Centroid from Bottom	Sheet Area	Sheet Inertia	Mechanical Resistance	Sheet Weight
	Aa	mm	mm	mm	#	mm	mm^2	mm^4	kN/m^2	kg/m
	.6mm	0.6	400	54	2	16.29	405.2	1.73e5	0	0
	.75mm	0.75	400	54	2	16.29	506.4	2.16e5	0	0
	.9mm	0.9	400	54	2	16.29	607.6	2.59e5	0	0
	1.0mm	1	400	54	2	16.29	675.2	2.88e5	0	0

In this data screen, the designer defines the thickness of sheet available for this sheeting profile and the basic properties for each sheet thickness in the control grid.

Toolbar Functions



Add an new sheet thickness. If an existing sheet thickness row is selected, the new thickness will be inserted at that location and the type at this location will me shifted down, otherwise the new thickness will be added after of the existing rows.



Delete the selected sheet thicknesses.

Sheet Properties

Nominal Size
The nominal sheet thickness used in defining and selecting this sheet. This is a text field and is unitless. The nominal size can include any text character.

Sheet Thickness
The actual sheet thickness.

Sheet Width
The nominal coverage width of the profiled sheet.

Sheet Height
The overall height of the sheet, ignoring embossments. In later screens, the heights to the flanges and web ends must be within this height.

Number of Rib Spaces
The number of panels in the width of a sheet.

Sheet Centroid from Bottom
The height to the centroid of the sheet from the bottom surface.

Sheet Area
Total cross-section area of steel of one sheet.

Sheet Inertia
Moment of inertia of the sheet about its centroid.

Mechanical Resistance

The sum of any mechanical interlock, Hr, from embossments and frictional resistance from re-entrant profiles that can be developed between the concrete and the metal decking sheet after bond with the concrete is broken.

For Fielders Kingflor decking, the following table defines the Hr values used in RAPT by default.

Profile	Hr values
KF57	235kPa
RF55	$= \varphi\ (1.2\text{-}0.1np)\ \alpha t_{bm}\ \beta$
KF70	114kPa 206kPa 292kPa
KF40	$= \varphi\ (1.2\text{-}0.1np)\ \alpha t_{bm}\ \beta$

φ = 0.85
np = number of pans per sheeting panel
t_{bm} = base metal thickness
For RF55 decking:
  α = 380
  β = .9

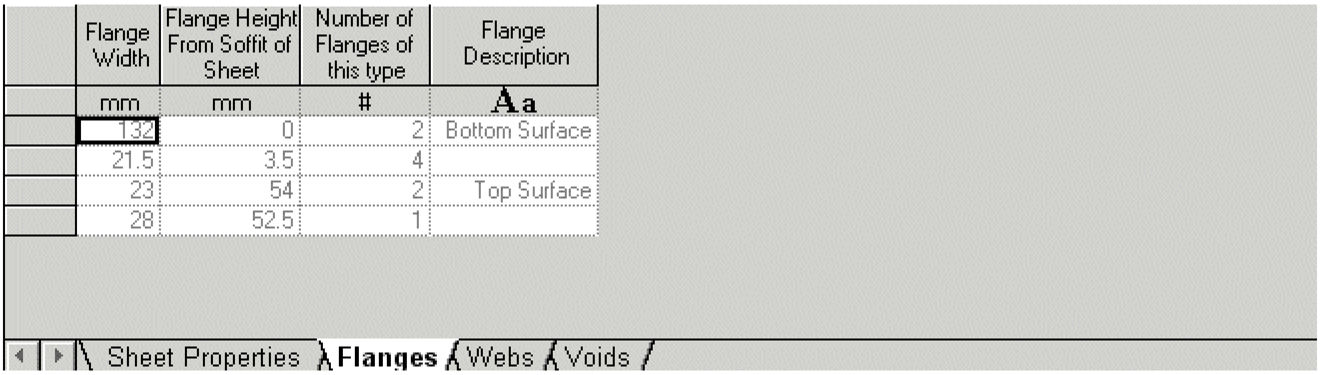
For KF40 decking:
  α = 345 + 3.8f'c
  β = 0.35 + 0.015 f'c

If the value defined in the RAPT Materials Data is set to a number greater than 10kPa, RAPT will use that value.
If the value is set to -1, RAPT will use the formula and parameters above for the for RF55 decking.
If the value is set to -2, RAPT will use the formula and parameters above for the for KF40 decking.

Sheet Weight

Weight of sheet per linear unit of length.

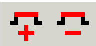
6.7.2 Structural Decking Flange Properties



To enable RAPT to analyse the strain a stress distribution in a sheet of metal decking, the sheet must be defined in terms of flanges (horizontal sections) and webs (vertical or sloping sections). The sum of the area these flanges and webs should be the same as the area of the sheet defined in Sheet Properties and the centroid should correspond as closely as possible. In this screen, the designer defines the different flanges that make up the overall sheet. The order in which the flanges are defined is immaterial to RAPT.

A flange's thickness is assumed to be the sheet thickness in all calculations.

Toolbar Functions



Add an new flange. If an existing flange row is selected, the new flange will be inserted at that location and the flange at this location will me shifted down, otherwise the new flange will be added after of the existing rows.

Delete the selected flanges.

Sheet Properties

Flange Width

The The width of this flange.

Flange Height From Soffit Of Sheet

The height of this flange from the soffit of the sheet. This height must be within the depth of the sheet defined in Sheet Properties.

For bottom flanges (at the extreme bottom of the sheet), the depth to the flange is measured to the underside of the flange and the flange depth is 0.

For all other flanges, the depth is measured to the top of the flange from the bottom of the sheet.

RAPT will use this logic for Bottom/Other flanges to determine the centroid of the flange for calculations.

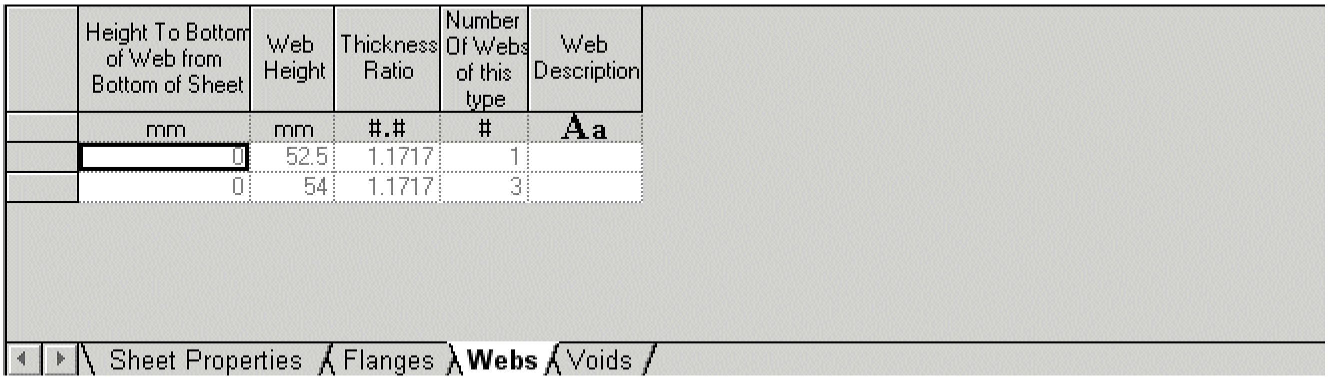
Number Of Flanges Of This Type

The number of flanges in this sheet with the exact same width and depth values.

Flange Description

A descriptive name for this flange type if desired.

6.7.3 Structural Decking Web Properties



To enable RAPT to analyse the strain a stress distribution in a sheet of metal decking, the sheet must be defined in terms of flanges (horizontal sections) and webs (vertical or sloping sections). The sum of the area these flanges and webs should be the same as the area of the sheet defined in Sheet Properties and the centroid should correspond as closely as possible. In this screen, the designer defines the different webs that make up the overall sheet. The order in which the webs are defined is immaterial to RAPT.

A web's thickness for design is calculated as the sheet thickness multiplied by the Thickness Ratio defined below.

Toolbar Functions

 Add an new web. If an existing web row is selected, the new web will be inserted at that location and the web at this location will me shifted down, otherwise the new web will be added after of the existing rows.

 Delete the selected webs.

Sheet Properties

Height To Bottom Of Web From Bottom Of Sheet
The vertical dimension from the bottom of the sheet (0) to the bottom of this web.

Web Height
The vertical height of this web type.

Thickness Ratio
Ratio of the calculation thickness of this web divided by the sheet thickness.

RAPT requires that a metal decking sheet be defined as a series of horizontal flanges and vertical webs. This is an idealised modelling of the actual shape of the decking. There are often sloping elements, curves and lips that need to be accounted for. RAPT's logic is to define a series of horizontal flanges of the same thickness as the sheet thickness to model the flat surfaces, and then for the remainder of the sheeting to be included in a series of vertical webs. The thickness of these webs is defined as the Sheet Thickness * Thickness Ratio. The thickness Ratios are calculated to ensure that the total area of the combined flanges and webs is equal to the actual sheet area. As many webs as desired may be defined to ensure that the centroid of the area of the flanges and webs is equal to the centroid of area of the sheet. RAPT will give a warning if the calculated area is not within .1% of the nominated sheet area and if the calculated centroid is not within 5% of the nominated centroid. This data check will be carried out and warning given when an attempt is made to run a RAPT data file with a specific sheet type nominated.

One way to calculate this, assuming the same thickness ratio for all webs, is to calculate the total area assigned to webs (sheet area - total flange area) and divide this by the total vertical height of all of the webs in the sheet. A more accurate estimate of sheet centroid may be achieved by assigning different Thickness Ratios to different webs.

Number Of Webs Of This Type
The number of webs in this sheet with the exact same properties.

Web Description
A descriptive name for this web type if desired.

6.7.4 Structural Decking Voids

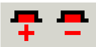
Height To Bottom of Void from Soffit of Sheet	Void Height	Left Bottom Corner Location from Left of Sheet	Left Top Corner Distance from bottom Left Corner	Top Width of Void	Right Bottom Corner Distance from Top Right Corner	Number Of This type of Void	Void Centre to Centre Spacing	Description
mm	mm	mm	mm	mm	mm	#	mm	Aa
0	51.5	0	0	18	-11.5	1	0	Right half void in left Rib
0	53	193.5	-11.5	36	-11.5	1	0	Centre Rib
0	51.5	393.5	-11.5	18	0	1	0	Left half void in right Rib

Sheet Properties / Flanges / Webs / **Voids**

To enable to allow RAPT to correctly account for the effect of metal decking on the design of a slab, RAPT needs to allow for the loss of some of the concrete section due to the void shapes created by the shape of the metal decking sheet. This will affect concrete weight in loads and the concrete section properties in the analysis and flexure, shear and deflection design sections of the RAPT calculations.

These void shapes can be defined by the user as a series of trapezoidal voids in the same form as Elements in the main concrete definition in RAPT Frame input. For each sheet of metal decking included in the input data for a frame, RAPT will automatically include it's associated voids as Void Trapezoidal Elements over the full length of each sheet defined. The designer gets no control over these elements in the Frame input. They will not show up in the Elements Data screen. They are inserted into the data automatically by RAPT based on the metal decking input in User Defined Reinforcement. Their positions will be modified as the metal decking data is modified. The only way to adjust their dimensions is in the Materials Data Input as described below.

Toolbar Functions



Add an new void shape. If an existing void row is selected, the new void shape will be inserted at that location and the void shape at this location will me shifted down, otherwise the new void shape will be added after of the existing rows.

Delete the selected void shapes.

Sheet Properties

Height To Bottom Of Void From Soffit Of Sheet
The vertical dimension from the bottom of the sheet (0) to the bottom of this void shape.

Void Height
The vertical height of this void shape. The top and bottom surfaces must be horizontal so there is only 1 height.

Left Bottom Corner Location From Left Of Sheet
The horizontal distance from the left edge of this sheet to the left bottom corner of this void shape. Positive dimensions to the right.

Left Top Corner Distance From Bottom Left Corner
The horizontal distance from the left bottom corner of the void to the left top corner of the void shape. Positive dimensions to the right.

Top Width Of Void
The width of the flat top surface of this void shape. Zero for a triangular shape.

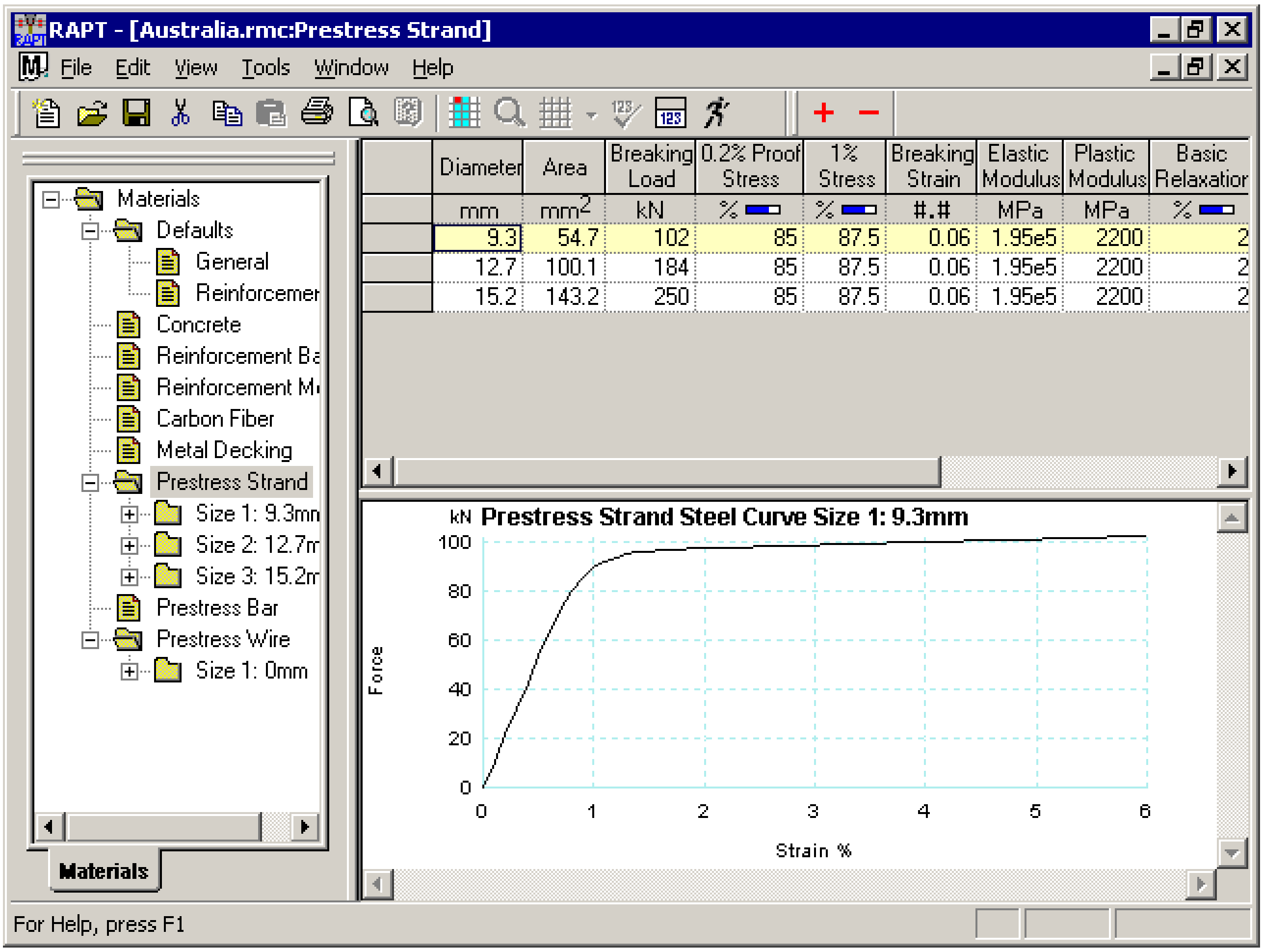
Right Bottom Corner Distance From Top Right Corner
The horizontal distance from the top right corner of the void to the right bottom corner of the void shape. Positive dimensions to the right.

Number Of This Type Of Void
The number of voids of this shape in this decking sheet.

Void centre to Centre Spacing
If the Number of This Type of Void is greater than 1, this defines the centre to centre spacing of the voids in this void shape

Description
A descriptive name for this void shape if desired.

6.8 Prestressing Strand



Different strand sizes can be defined in this view along with the basic data need to define strand properties. When the tree folder associated with strand is opened, a branch is available for each of the strand sizes defined in this table. Each branch then allows the designer sub-branches to define different types of tendons. These are

1. 6.8.1 Bonded Monostrand
2. 6.8.2 Bonded Multistrand
3. 6.8.3 Unbonded Monostrand
4. 6.8.4 Unbonded Multistrand
5. 6.8.5 Pretensioned

Toolbar

+ −

+ Add an new prestress strand size. If an existing size is selected, the new size will be inserted at that location and the size at this location will me shifted down, otherwise the new size will be added after of the existing types.

− Delete the selected strand sizes.

Data Definition

Diameter
The diameter of this strand size.

Area
The area of this strand size.

Breaking Load
The breaking load of this strand size.

Proof Stress
The percentage of the breaking stress at the proof strain. Depending on the design code being used, this could be either .1% or .2% proof stress.

1% Stress
The percentage of the breaking stress at 1% strain.

Breaking Strain
The breaking strain for this strand size.

Elastic Modulus
The elastic modulus of the strand. This defines the slope of the straight portion of the stress/strain curve shown below from the origin.

Plastic Modulus
The plastic modulus of the strand. This defines the slope of the near horizontal straight portion of the stress/strain curve to the breaking strain shown below.

Basic Relaxation
The basic relaxation is normally defined as the strand relaxation after 1000 hours and at a load of 70% or breaking load at a standard temperature of 20 degrees Celsius

Weight
The weight of the strand per unit length.

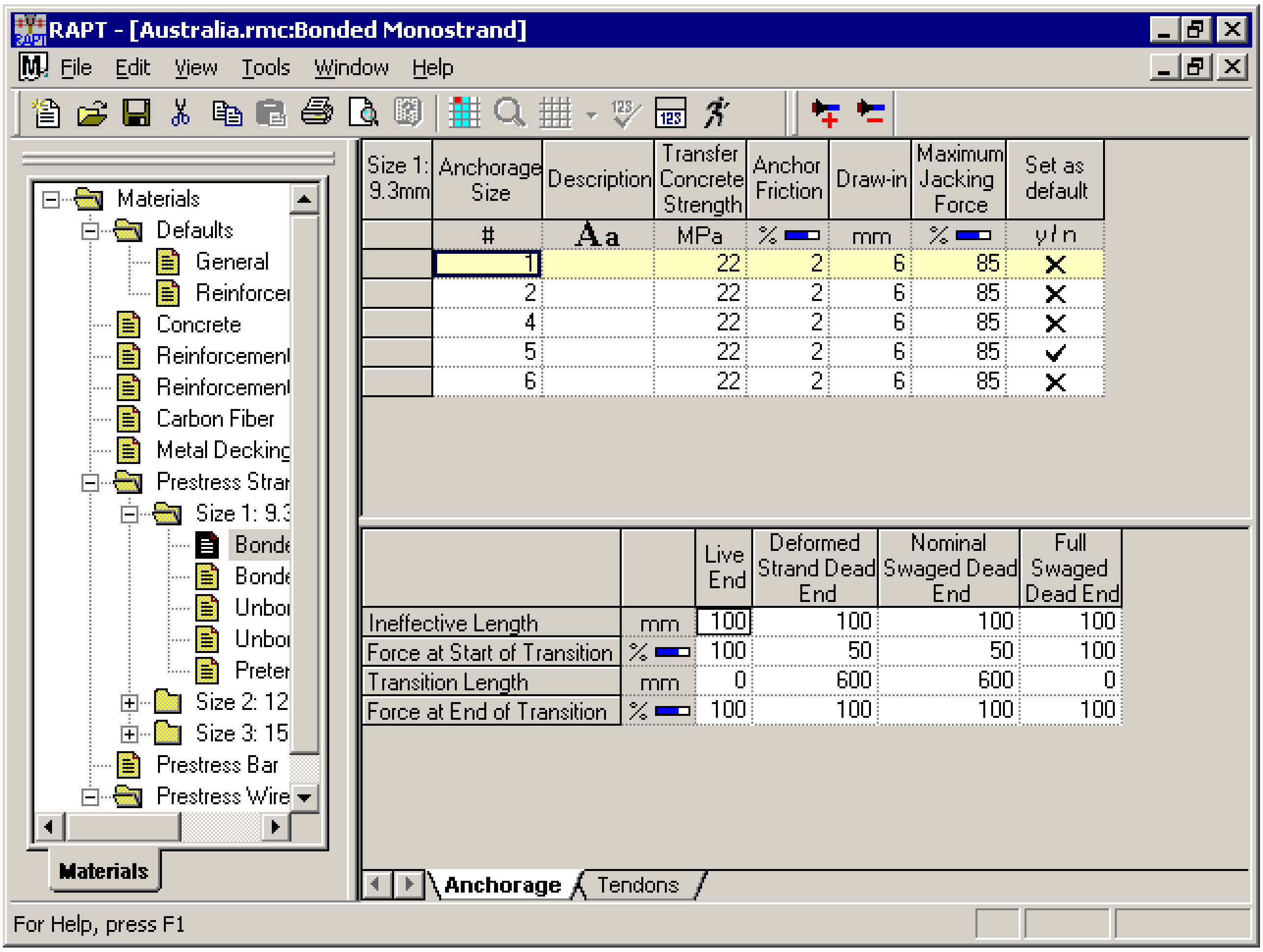
Set as default
The strand size to be used as the default size in setting up new runs.

Description
A text description of the strand size.

Graphic View

The graphic view shows the curvilinear force/strain diagram developed from the data defined above for the current wire size. This curve will be used for the determination of the prestress force in the strain compatibility calculations for design codes that allow its use. For BS8110, SABS0100, CP65 and Eurocode 2, the stress/strain curves defined in those codes are used.

6.8.1 Bonded Monostrand Anchorage Sizes

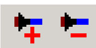


Monostrand anchorages are anchorages where the strands are stressed individually, are rectangular in shape and the strands lie side by side in a rectangular duct. Sometimes the strands in round duct tendons are stressed individually also but they are still referred to as multistrand tendons in RAPT.

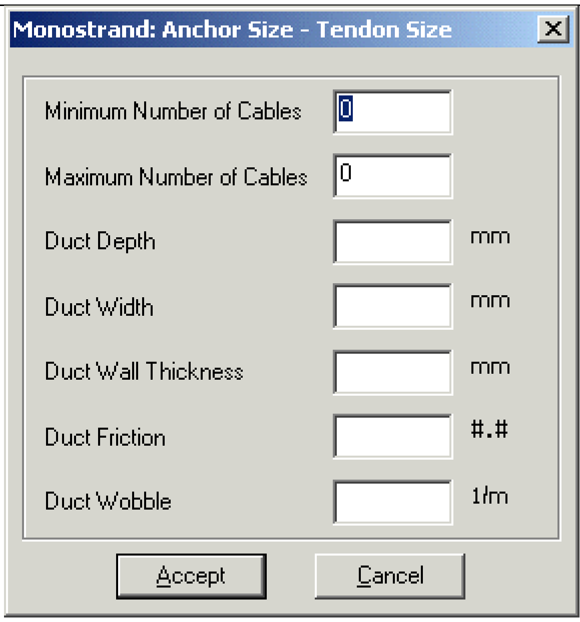
For each strand size the designer can define a range of anchorages to accommodate different numbers of strands. For each anchorage, the basic information to define the anchorage can be edited in this Control data screen. As well, for each anchorage size in the Control data (each row of the grid), there are two further screens of data, accessed via the View Tabs at the bottom of the Child Views, to define

1. 6.8.1.1 Anchorage properties
2. 6.8.1.2 Tendon properties

Toolbar



 Add a monostrand prestress anchor size for this strand size. If an anchor size is selected before adding, the new size will be inserted at the location of the selected size, otherwise it will be added at the end of the list. The following dialog will be presented to allow RAPT to automatically create the tendon data for this anchor size.



The data required in the dialog is

Minimum number of cables
The minimum number of strands that would be placed in this duct.

Maximum number of cables
The maximum number of strands that would be placed in this duct. In doing the calculations for strand centroid etc, if the number of strands nominated will not fit into the duct, RAPT will not create tendon data for the tendon sizes with more strands than can be physically fitted into the duct.

Duct Depth
The depth of the duct (external dimension)

Duct Width
The width of the duct (external dimension)

Duct Wall thickness
The thickness of the duct wall material.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view.

Using this information RAPT will create the anchorage and tendon data for this anchorage size. This data can then be edited by the designer.

 Delete selected anchorage sizes. All data associated with a deleted anchorage size will be lost.

Data Definition

Anchorage Size
The size of this anchorage in terms of the maximum number of strands that can be accommodated.

Description
A text description of this anchorage.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

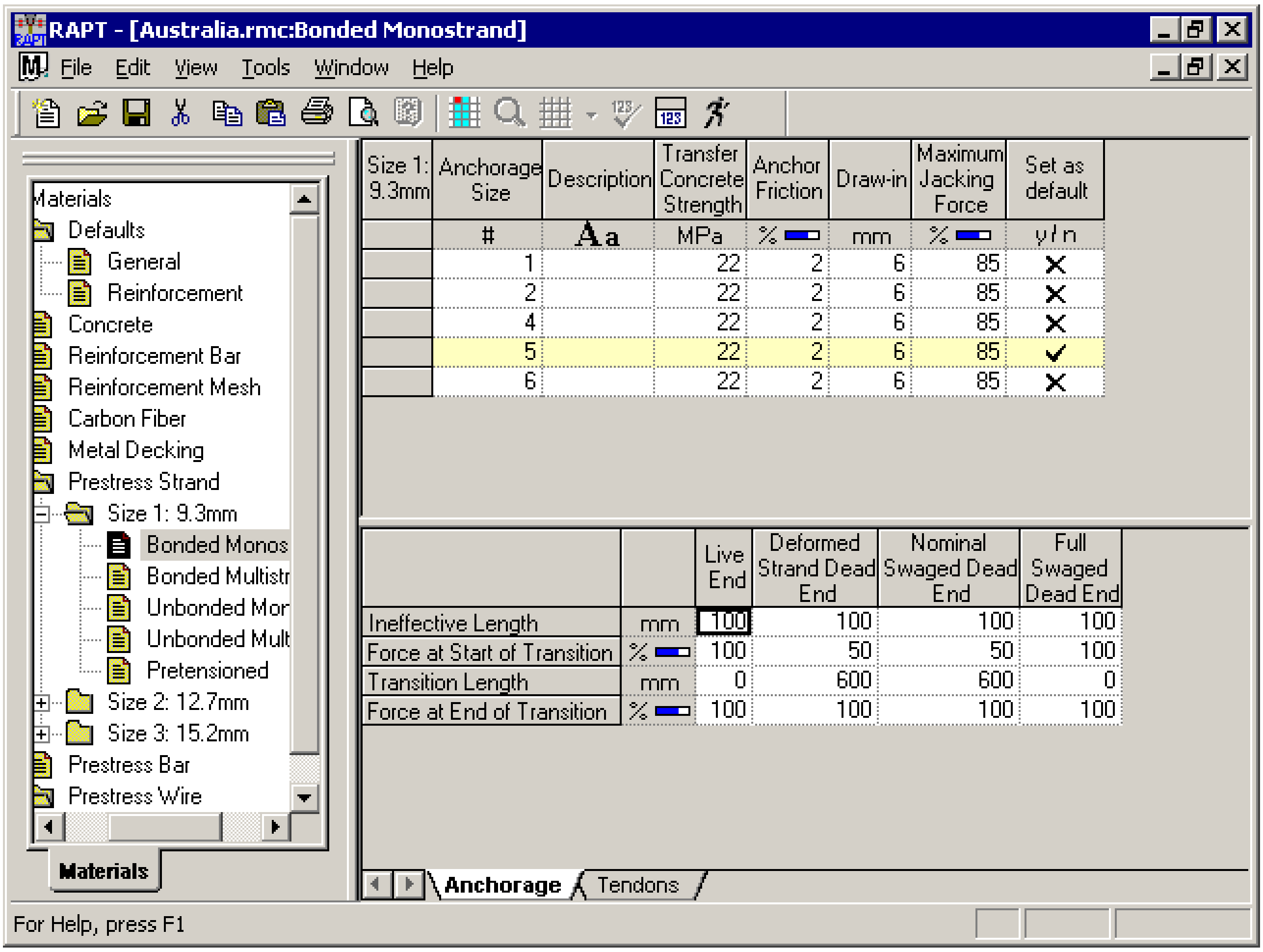
Anchor Friction
The friction loss experienced by the strand in the anchorage. Most anchorages (single strand anchorages normally do not) force the strand to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the strand past the anchorage.

Draw-in
The distance the strand slips through the anchorage when the jack is released.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the strand in the anchorage. This is normally defined by Design Standards.

Set as default
Set this anchorage size as the default size when this tendon type is selected.

6.8.1.1 Anchorage Properties

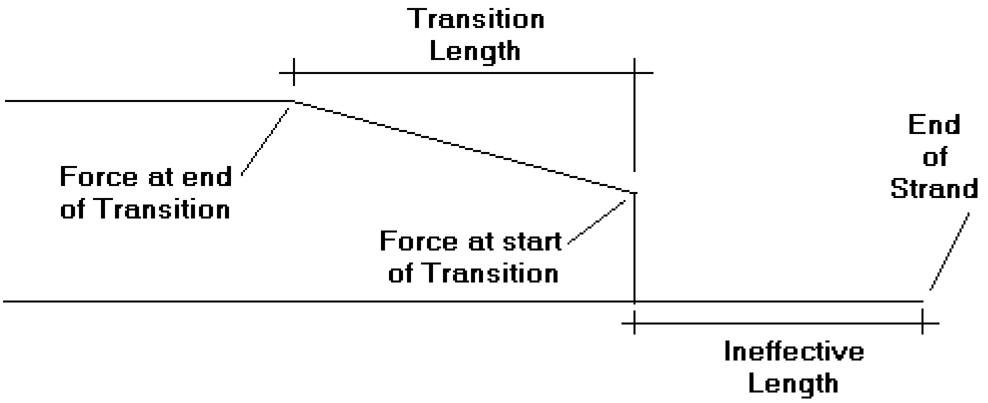


For each anchorage size in the control grid, RAPT provides one screen of Anchorage Properties data as defined below.

RAPT defines four types of anchorage. These are

1. Live End:- Stressing End
2. Deformed Strand Dead End:- Non stressing end in which the force is transferred to the concrete via a splayed section of strand at the end of the strand and a bonded length of strand. Sometimes called an Onion anchorage. The force is not transmitted through a plate, it is transmitted through bond along the length of bonded strand (nominally 60%) and by mechanical anchorage in the deformed "onion" strand at the end (nominally 40%) , though a thin plate is sometimes provided to space the strands.
3. Nominal Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a nominal thickness plate (in the order of 5 to 10mm thick) (nominally 50%) and also a bonded length of strand (nominally 50%).
4. Full Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a full bearing plate designed to distribute the load to the concrete (100%) and with the strand leading from the end of the duct debonded to allow the full force to reach the bearing plate.

For each of these anchorage types, RAPT allows the definition of four parameters defined below and shown graphically.



Ineffective Length

The length from the end of the strand to the first point at which the strand develops stress.

Force at Start of Transition

The maximum force as a percentage of the breaking force that can be developed in the tendon at the first point at which the strand develops stress.

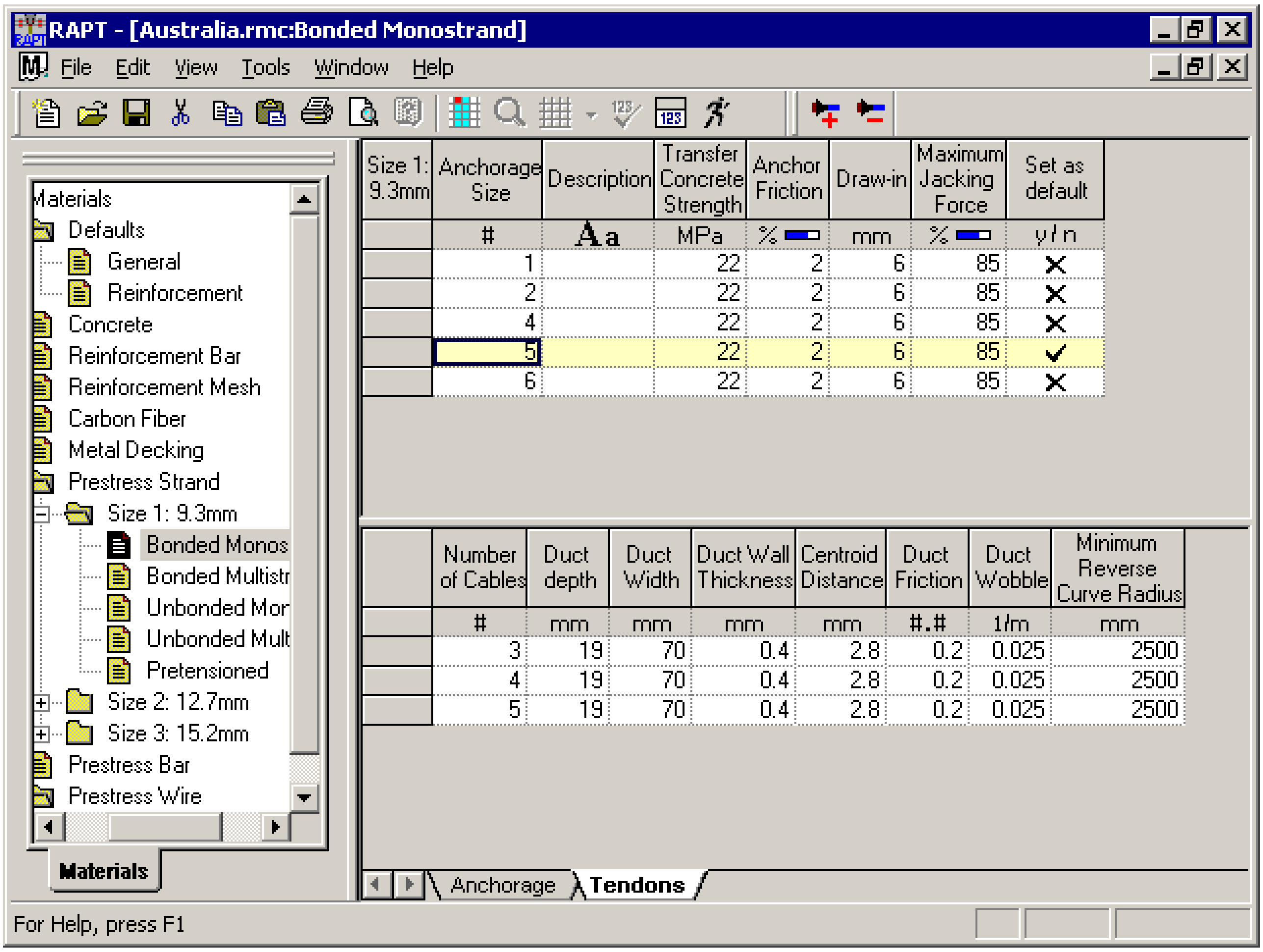
Transition Length

The length over which the development of force in the strand increases.

Force at End of Transition

The maximum force as a percentage of the breaking force that can be developed in the tendon at the end of the Transition Length. From this point onwards the tendon can develop the full breaking force in the tendon.

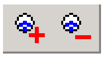
6.8.1.2 Tendon Data



For each anchorage size in the control grid, RAPT provides one screen of Tendon data which allows designers to nominate tendon sizes for the current anchorage in terms of a number of strands as defined below.

The default tendon size selected will be the last one in the list.

Toolbar



Add a new tendon size for the current anchorage. If a tendon size is selected, the new tendon size will be inserted in the list at the location of the selected tendon size and the tendon sizes at and below that location will be moved downwards. Otherwise, the new tendon size will be added at the end of the list of current sizes.

Delete the selected tendon sizes.

Data Definition

Number of Cables
The number of strands in this tendon.

Duct depth
The outside depth of the duct used for this tendon size.

Duct Width
The outside width of the duct used for this tendon size.

Duct Wall Thickness
The thickness of the wall of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the strand diameter so there is room for the strands to move vertically in the duct. When the strands sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the strands in this situation.

Duct Friction
The duct friction factor.

Duct Wobble

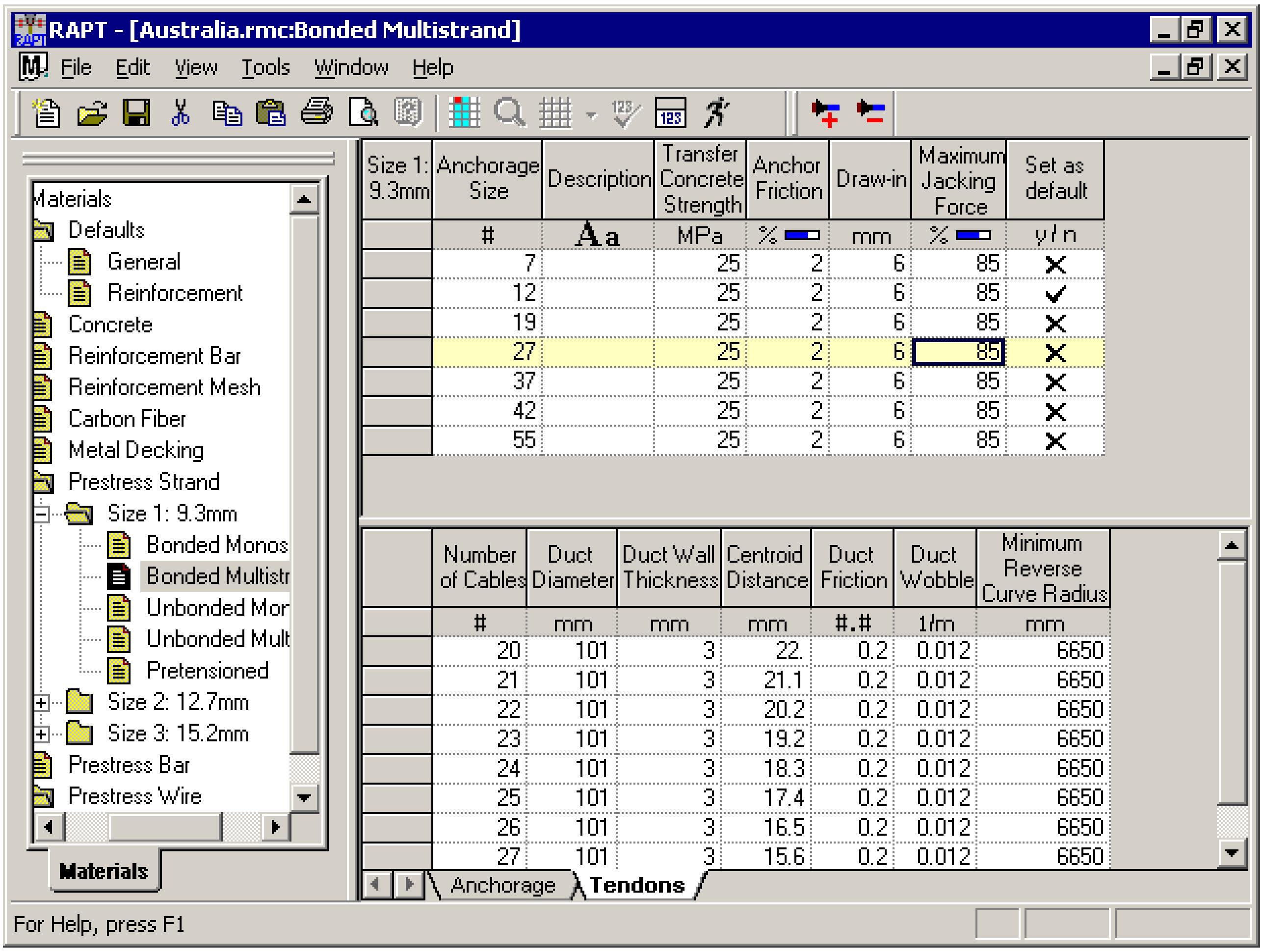
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view. Different design codes use one of two different methods to define wobble. Some define it as Unintentional Friction - *K*, which is an angle change multiplied by the duct friction factor. Others define it purely as an angle change, Unintentional Angle Change - *ß*, which must be multiplied by the duct friction factor in the friction equations. So $K = u * ß$.

Minimum Reverse Curve Radius

The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

6.8.2 Bonded Multistrand Anchorage Sizes

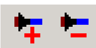


Multistrand anchorages are anchorages where the strands are stressed as a group, are generally square in shape with round duct and the strands bunch together. Sometimes the strands in round duct tendons are stressed individually also but they are still referred to as multistrand tendons in RAPT.

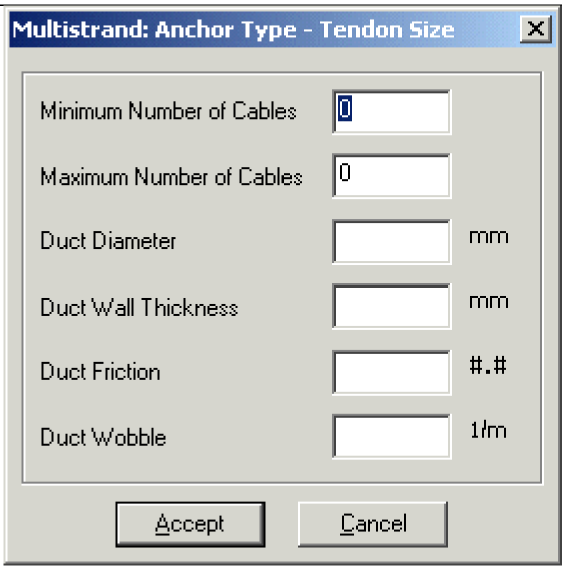
For each strand size the designer can define a range of anchorages to accommodate different numbers of strands. For each anchorage, the basic information to define the anchorage can be edited in this Control data screen. As well, for each anchorage size in the Control data (each row of the grid), there are two further screens of data, accessed via the View Tabs at the bottom of the Child Views, to define

1. 6.8.2.1 Anchorage properties
2. 6.8.2.2 Tendon properties

Toolbar



 Add a multistrand prestress anchor size for this strand size. If an anchor size is selected before adding, the new size will be inserted at the location of the selected size, otherwise it will be added at the end of the list. The following dialog will be presented to allow RAPT to automatically create the tendon data for this anchor size.



The data required in the dialog is

Minimum number of cables
The minimum number of strands that would be placed in this duct.

Maximum number of cables
The maximum number of strands that would be placed in this duct. In doing the calculations for strand centroid etc, if the number of strands nominated will not fit into the duct, RAPT will not create tendon data for the tendon sizes with more strands than can be physically fitted into the duct.

Duct Diameter
The diameter of the duct (external dimension)

Duct Wall thickness
The thickness of the duct wall material.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view.

Using this information RAPT will create the anchorage and tendon data for this anchorage size. This data can then be edited by the designer.

 Delete selected anchorage sizes. All data associated with a deleted anchorage size will be lost.

Data Definition

Anchorage Size
The size of this anchorage in terms of the maximum number of strands that can be accommodated.

Description
A text description of this anchorage.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

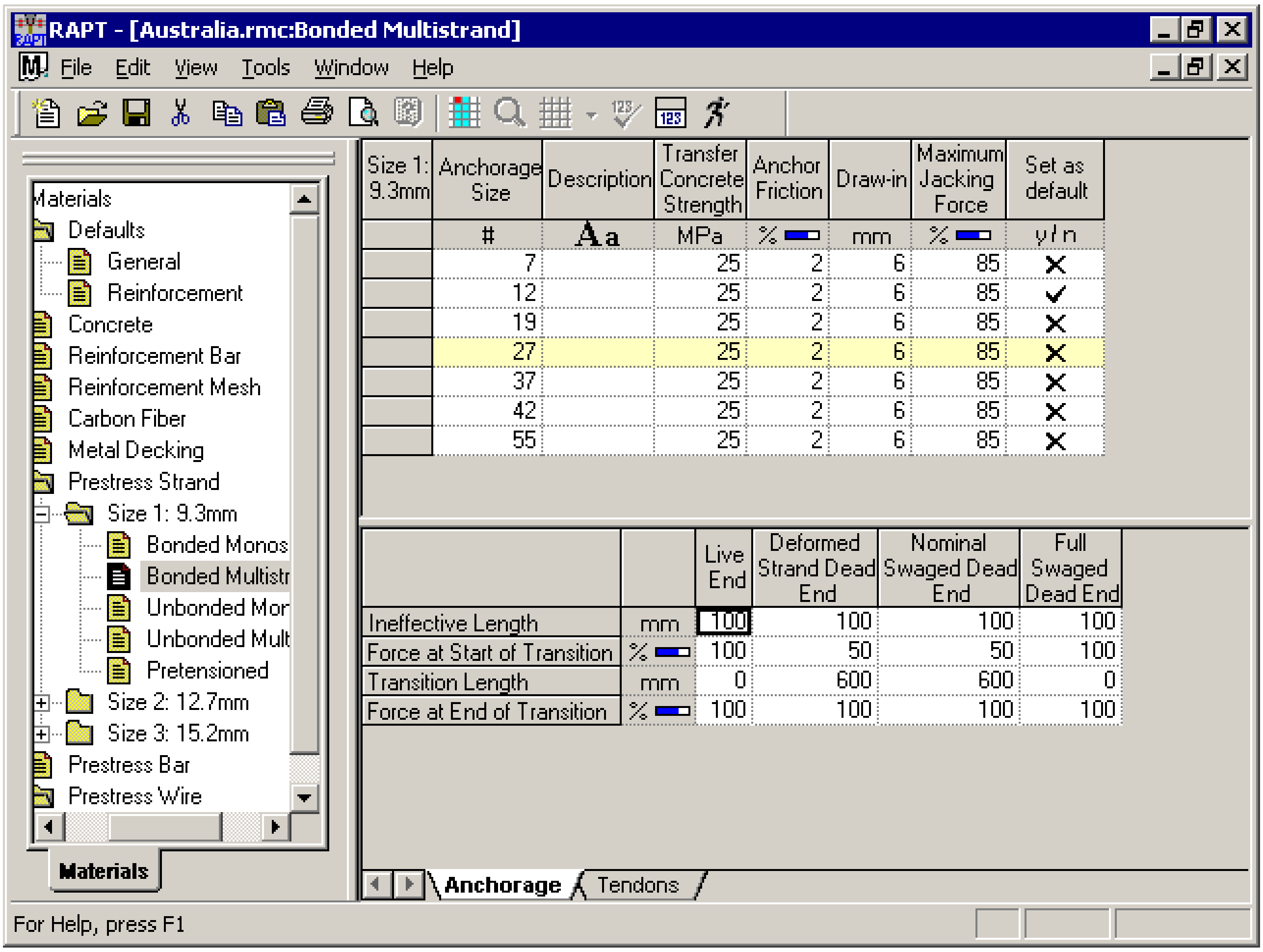
Anchor Friction
The friction loss experienced by the strand in the anchorage. Most anchorages (single strand anchorages normally do not) force the strand to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the strand past the anchorage.

Draw-in
The distance the strand slips through the anchorage when the jack is released.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the strand in the anchorage. This is normally defined by Design Standards.

Set as default
Set this anchorage size as the default size when this tendon type is selected.

6.8.2.1 Anchorage Properties

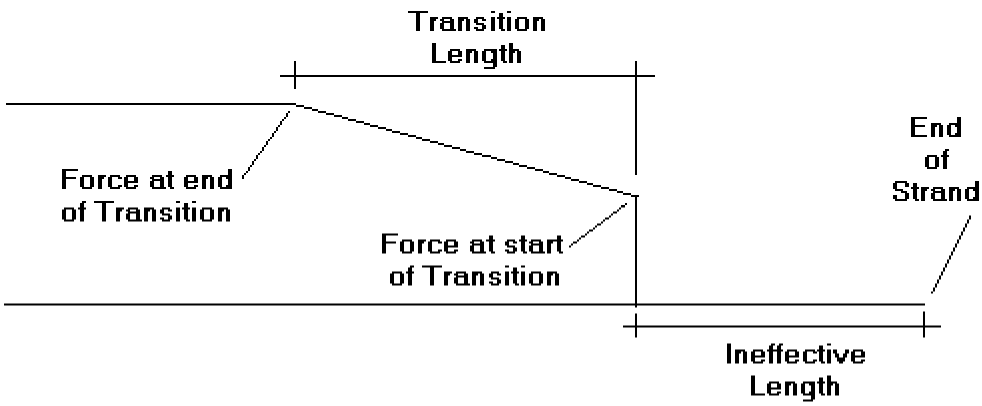


For each anchorage size in the control grid, RAPT provides one screen of Anchorage Properties data as defined below.

RAPT defines four types of anchorage. These are

1. Live End:- Stressing End
2. Deformed Strand Dead End:- Non stressing end in which the force is transferred to the concrete via a splayed section of strand at the end of the strand and a bonded length of strand. Sometimes called an Onion anchorage. The force is not transmitted through a plate, it is transmitted through bond along the length of bonded strand (nominally 60%) and by mechanical anchorage in the deformed "onion" strand at the end (nominally 40%) , though a thin plate is sometimes provided to space the strands.
3. Nominal Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a nominal thickness plate (in the order of 5 to 10mm thick) (nominally 50%) and also a bonded length of strand (nominally 50%).
4. Full Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a full bearing plate designed to distribute the load to the concrete (100%) and with the strand leading from the end of the duct debonded to allow the full force to reach the bearing plate.

For each of these anchorage types, RAPT allows the definition of four parameters defined below and shown graphically.



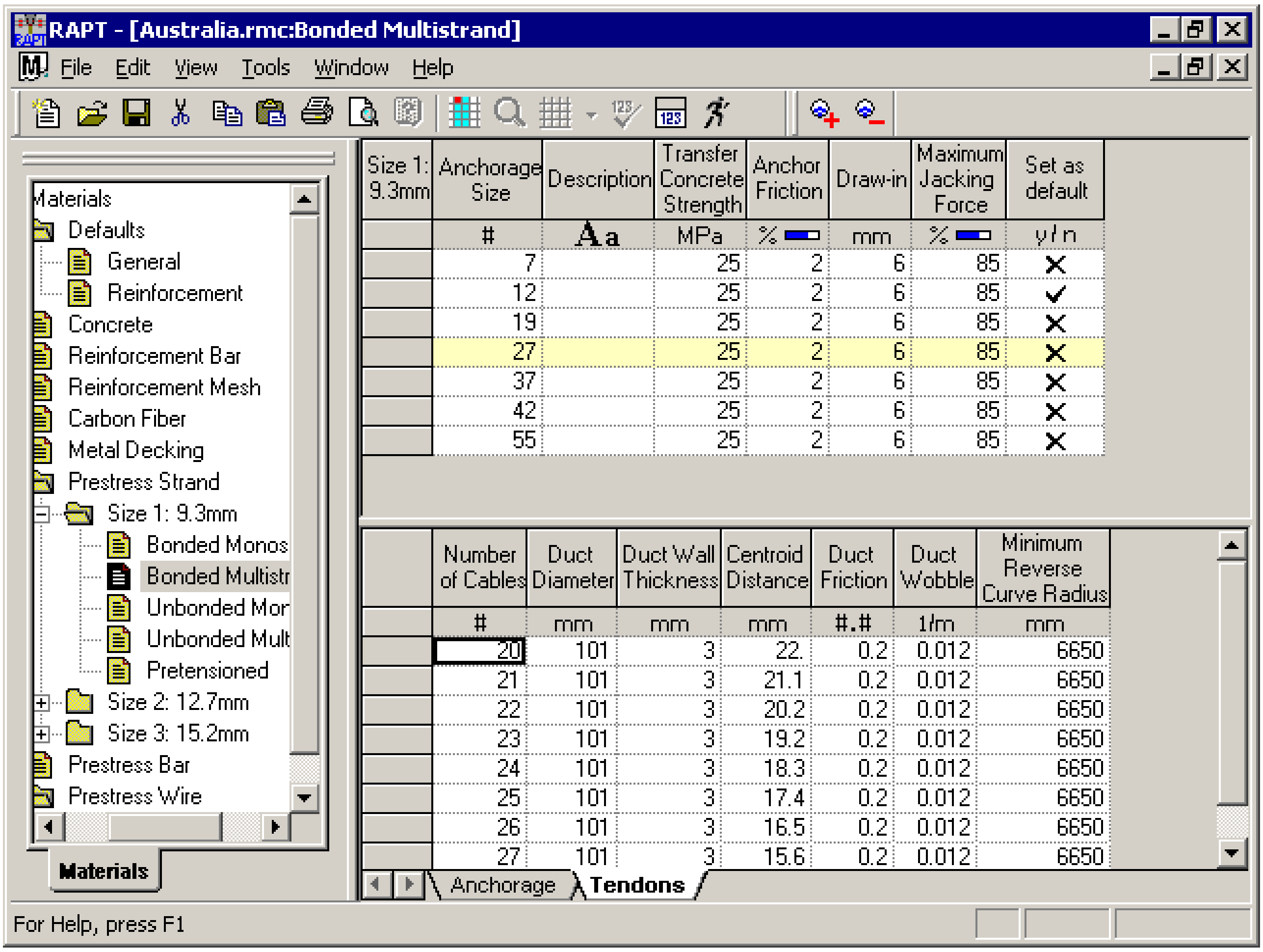
Ineffective Length
The length from the end of the strand to the first point at which the strand develops stress.

Force at Start of Transition
The maximum force as a percentage of the breaking force that can be developed in the tendon at the first point at which the strand develops stress.

Transition Length
The length over which the development of force in the strand increases. The lengths shown here assume that a square arrangement of strands is being used at the anchorage. For other arrangements a longer length may be required. Refer to specialist literature for further information.

Force at End of Transition
The maximum force as a percentage of the breaking force that can be developed in the tendon at the end of the Transition Length. From this point onwards the tendon can develop the full breaking force in the tendon.

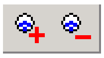
6.8.2.2 Tendon Data



For each anchorage size in the control grid, RAPT provides one screen of Tendon data which allows designers to nominate tendon sizes for the current anchorage in terms of a number of strands as defined below.

The default tendon size selected will be the last one in the list.

Toolbar



 Add a new tendon size for the current anchorage. If a tendon size is selected, the new tendon size will be inserted in the list at the location of the selected tendon size and the tendon sizes at and below that location will be moved downwards. Otherwise, the new tendon size will be added at the end of the list of current sizes.

 Delete the selected tendon sizes.

Data Definition

Number of Cables
The number of strands in this tendon.

Duct Diameter
The outside diameter of the duct used for this tendon size.

Duct Wall Thickness
The thickness of the wall of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the strand diameter so there is room for the strands to move vertically in the duct. When the strands sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the strands in this situation.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view. Different design codes use one of two different methods to define wobble. Some define it as Unintentional Friction -

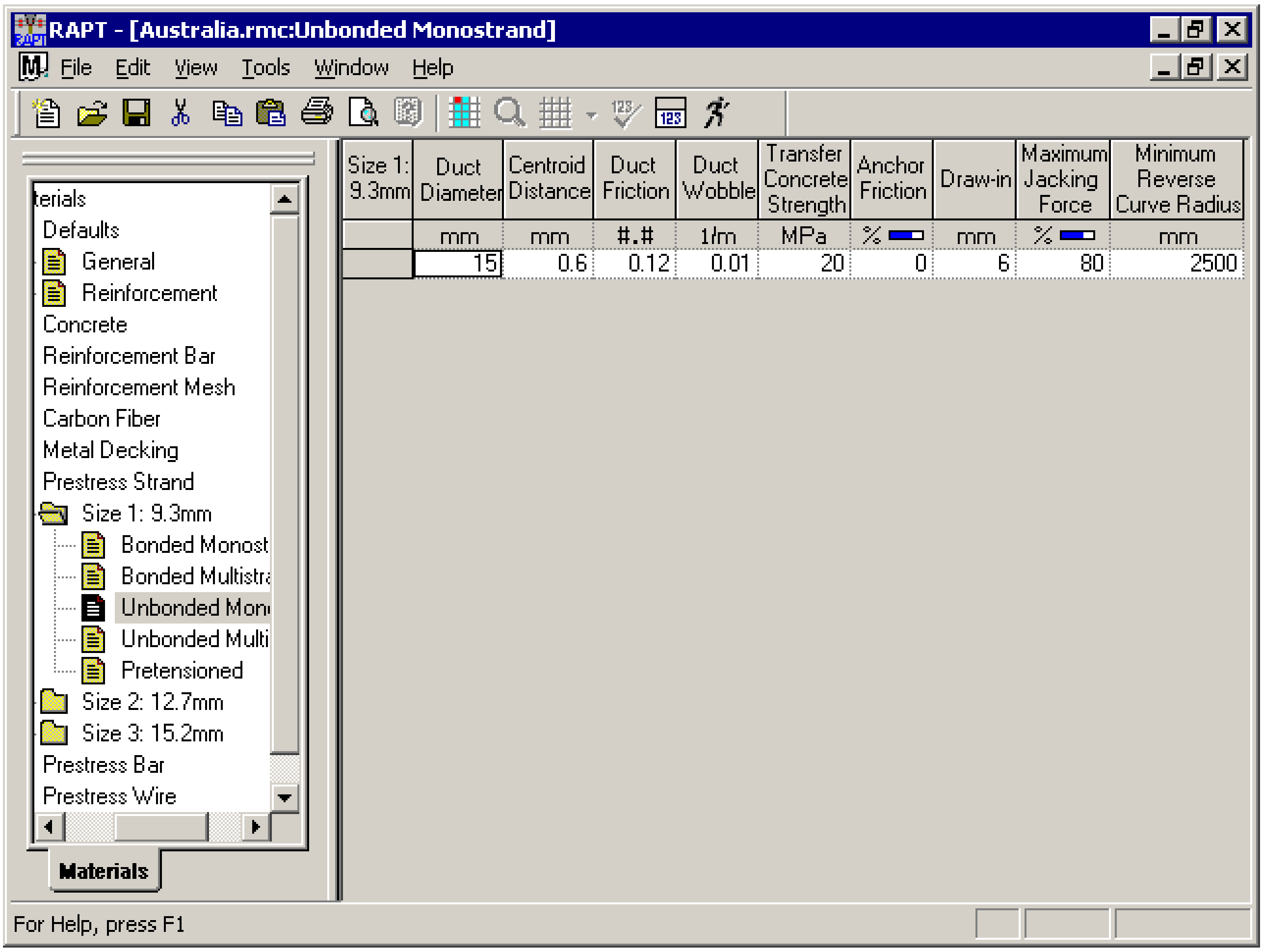
K, which is an angle change multiplied by the duct friction factor. Others define it purely as an angle change, Unintentional Angle Change - *ß*, which must be multiplied by the duct friction factor in the friction equations. So $K = u * ß$.

Minimum Reverse Curve Radius

The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

6.8.3 Unbonded Monostrand



Duct Diameter
The outside diameter of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the strand diameter so there is room for the strands to move vertically in the duct. When the strands sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the strands in this situation.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view. Different design codes use one of two different methods to define wobble. Some define it as Unintentional Friction - *K*, which is an angle change multiplied by the duct friction factor. Others define it purely as an angle change, Unintentional Angle Change - *ß*, which must be multiplied by the duct friction factor in the friction equations. So *K = u * ß*.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

Anchor Friction
The friction loss experienced by the strand in the anchorage. Most anchorages (single strand anchorages normally do not) force the strand to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the strand past the anchorage.

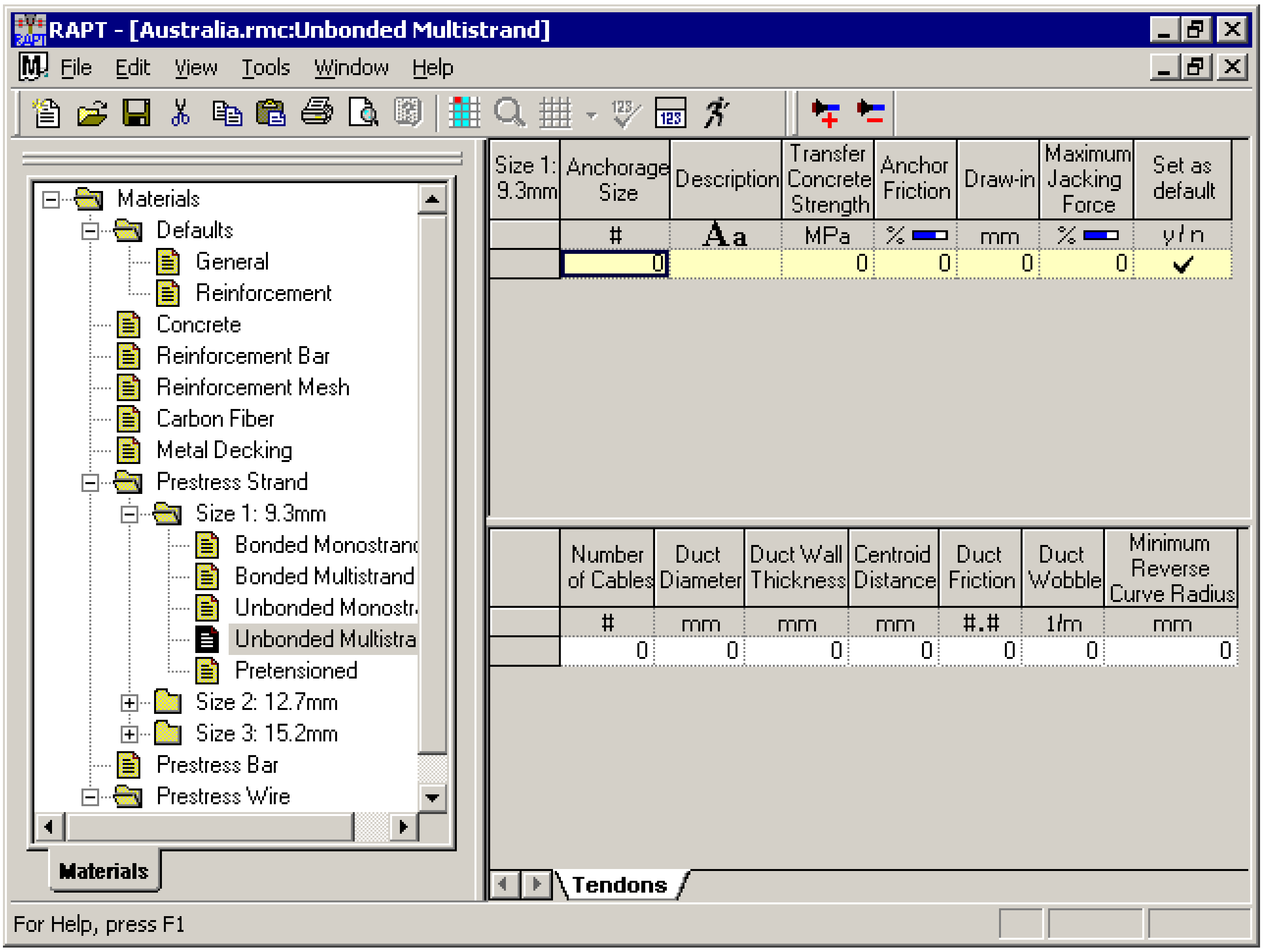
Draw-in
The distance the strand slips through the anchorage when the jack is released.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the strand in the anchorage. This is normally defined by Design Standards.

Minimum Reverse Curve Radius
The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

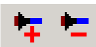
6.8.4 Unbonded Multistrand Anchorage Sizes



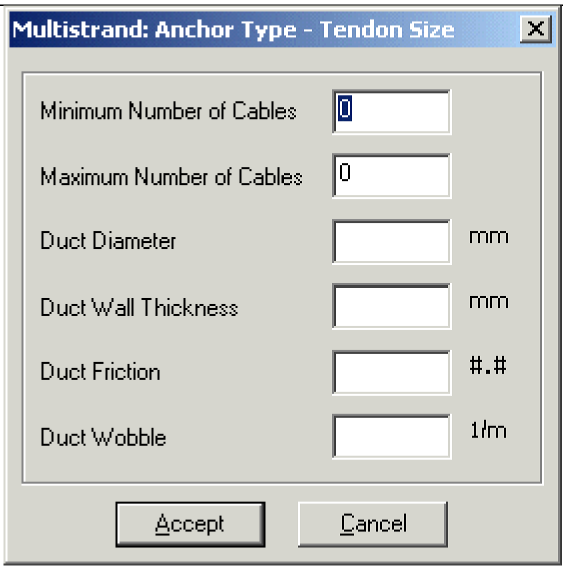
Multistrand anchorages are anchorages where the strands are stressed as a group, are generally square in shape with round duct and the strands bunch together. Sometimes the strands in round duct tendons are stressed individually also but they are still referred to as multistrand tendons in RAPT.

For each strand size the designer can define a range of anchorages to accommodate different numbers of strands. For each anchorage, the basic information to define the anchorage can be edited in this Control data screen. As well, for each anchorage size in the Control data (each row of the grid), there are one further screen of data in a Child view to define 6.8.4.1 Tendon properties.

Toolbar



Add a multistrand prestress anchor size for this strand size. If an anchor size is selected before adding, the new size will be inserted at the location of the selected size, otherwise it will be added at the end of the list. The following dialog will be presented to allow RAPT to automatically create the tendon data for this anchor size.



The data required in the dialog is

Minimum number of cables
The minimum number of strands that would be placed in this duct.

Maximum number of cables
The maximum number of strands that would be placed in this duct. In doing the calculations for strand centroid etc, if the number of strands nominated will not fit into the duct, RAPT will not create tendon data for the tendon sizes with more strands than can be physically fitted into the duct.

Duct Diameter
The diameter of the duct (external dimension)

Duct Wall thickness
The thickness of the duct wall material.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view.

Using this information RAPT will create the anchorage and tendon data for this anchorage size. This data can then be edited by the designer.

 Delete selected anchorage sizes. All data associated with a deleted anchorage size will be lost.

Data Definition

Anchorage Size
The size of this anchorage in terms of the maximum number of strands that can be accommodated.

Description
A text description of this anchorage.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

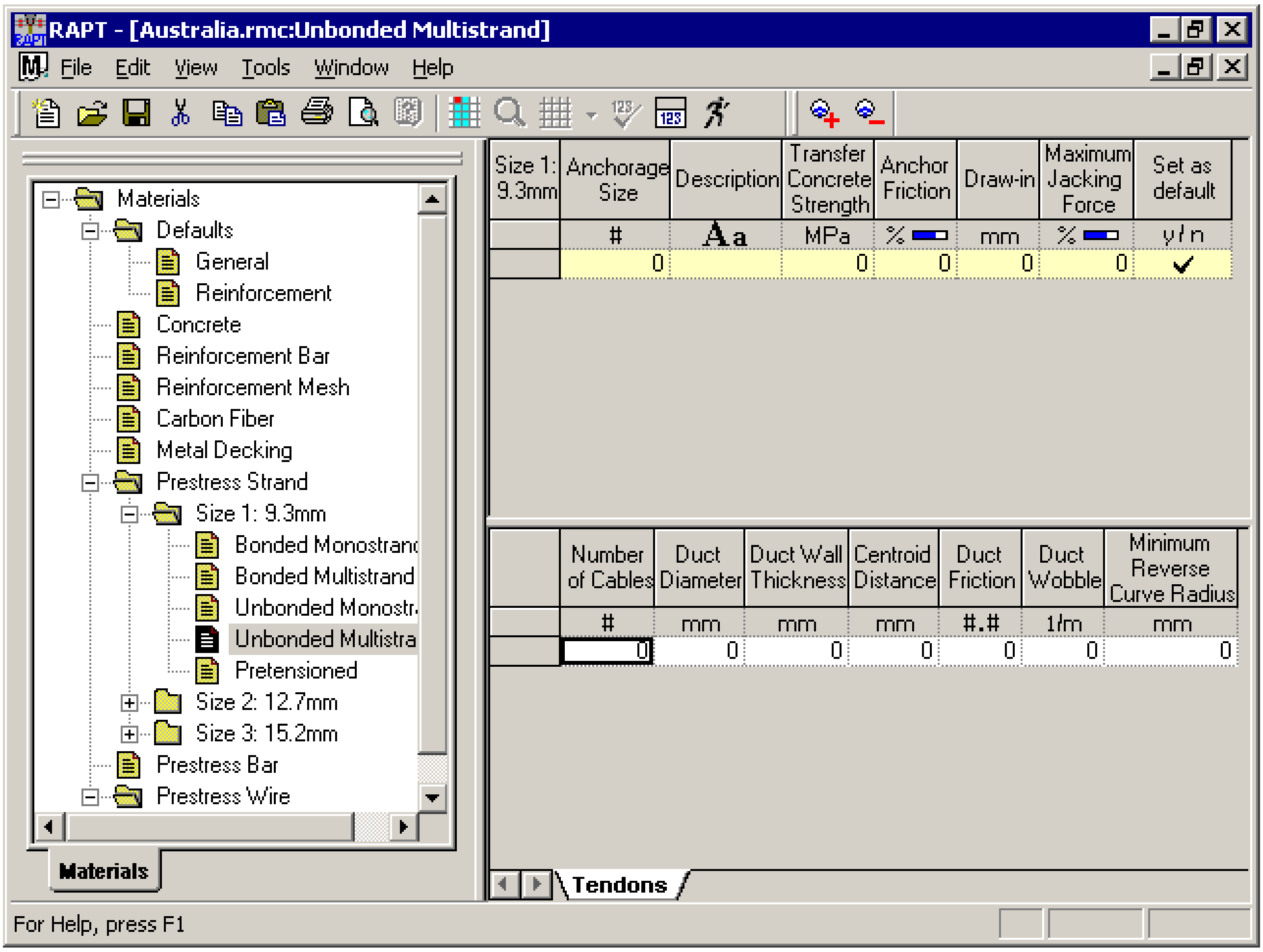
Anchor Friction
The friction loss experienced by the strand in the anchorage. Most anchorages (single strand anchorages normally do not) force the strand to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the strand past the anchorage.

Draw-in
The distance the strand slips through the anchorage when the jack is released.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the strand in the anchorage. This is normally defined by Design Standards.

Set as default
Set this anchorage size as the default size when this tendon type is selected.

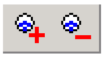
6.8.4.1 Tendon Data



For each anchorage size in the control grid, RAPT provides one screen of Tendon data which allows designers to nominate tendon sizes for the current anchorage in terms of a number of strands as defined below.

The default tendon size selected will be the last one in the list.

Toolbar



 Add a new tendon size for the current anchorage. If a tendon size is selected, the new tendon size will be inserted in the list at the location of the selected tendon size and the tendon sizes at and below that location will be moved downwards. Otherwise, the new tendon size will be added at the end of the list of current sizes.

 Delete the selected tendon sizes.

Data Definition

Number of Cables
The number of strands in this tendon.

Duct Diameter
The outside diameter of the duct used for this tendon size.

Duct Wall Thickness
The thickness of the wall of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the strand diameter so there is room for the strands to move vertically in the duct. When the strands sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the strands in this situation.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view. Different design codes use one of two different methods to define wobble. Some define it as Unintentional Friction -

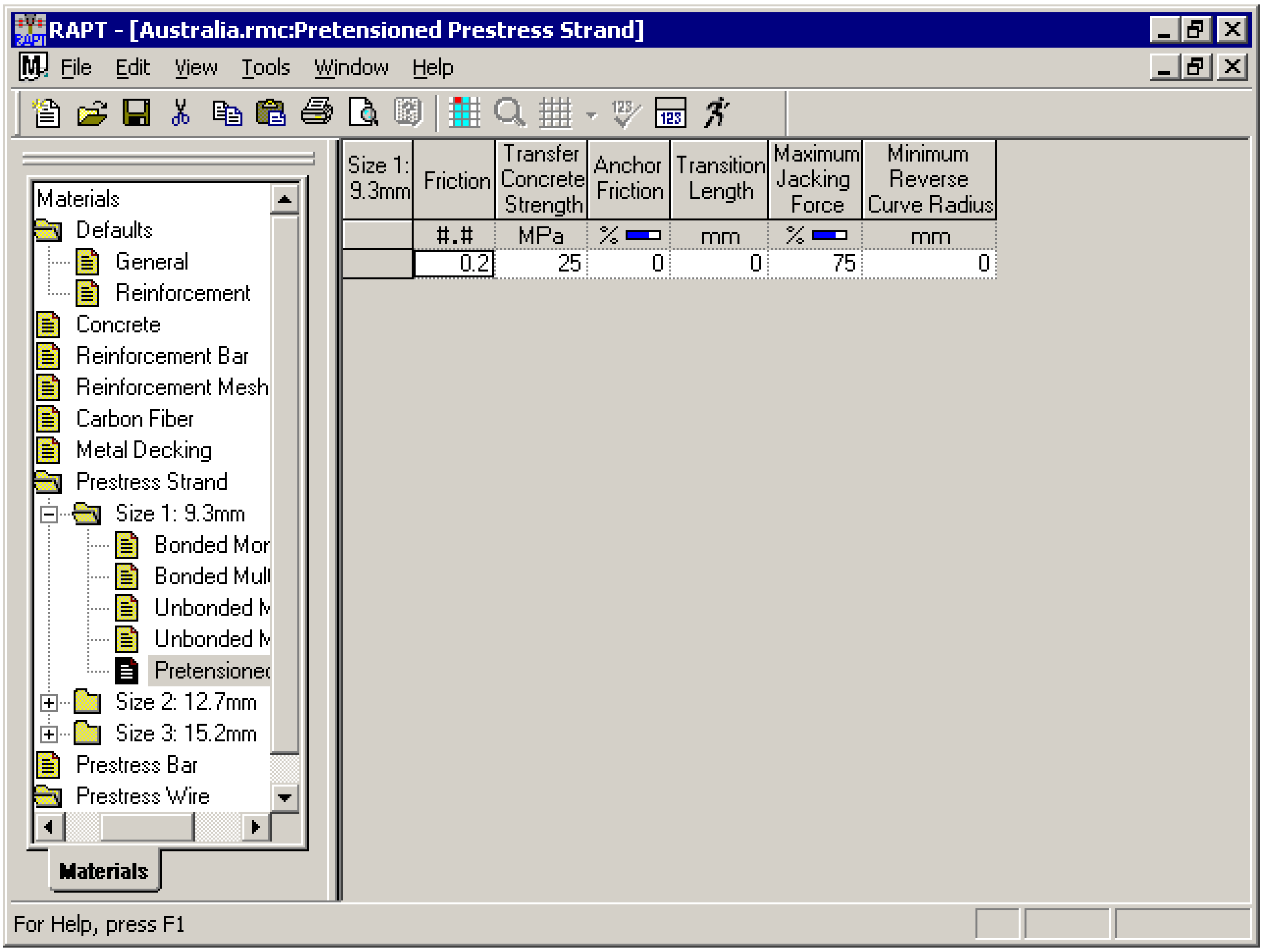
K, which is an angle change multiplied by the duct friction factor. Others define it purely as an angle change, Unintentional Angle Change - *ß*, which must be multiplied by the duct friction factor in the friction equations. So *K* = *u* * *ß*.

Minimum Reverse Curve Radius

The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

6.8.5 Pretensioned



Friction
The friction factor for the strand passing through angle changes.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

Anchor Friction
The friction loss experienced by the strand in the anchorage. Most anchorages (single strand anchorages normally do not) force the strand to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the strand past the anchorage.

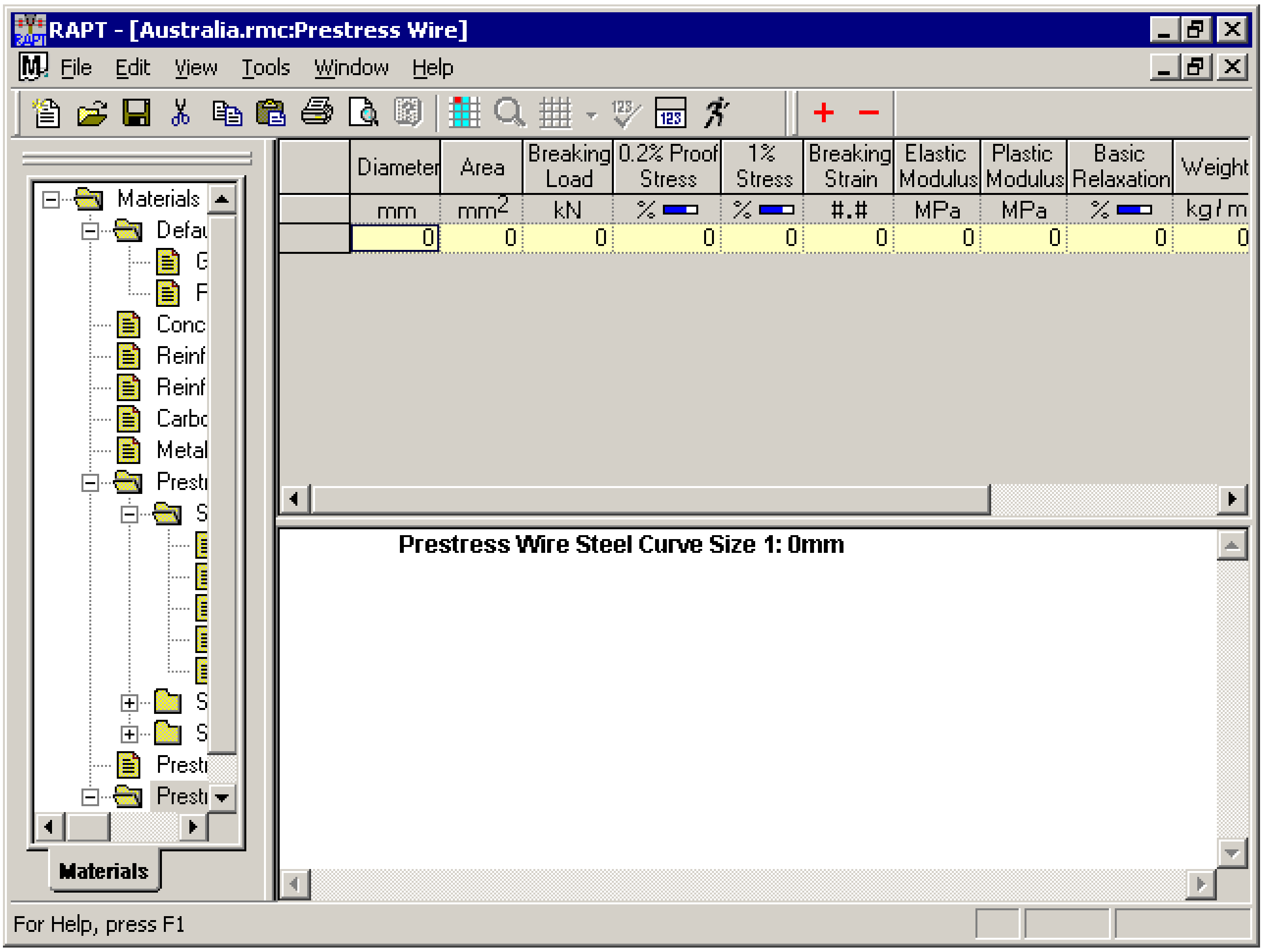
Transition Length
The length of strand over which the force is assumed to increase from zero at the end to the full tension force.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the strand in the anchorage. This is normally defined by Design Standards.

Minimum Reverse Curve Radius
The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

6.9 Prestressing Bar



For each bar size nominated in the Control Grid at the top of the screen the designer is able to define tendon properties in child grids accessed via the window tabs for two types of bar tendons

1. Bonded
2. Unbonded

Toolbar

 Add an new bar size. If an existing size is selected, the new size will be inserted at that location and the size at this location will me shifted down, otherwise the new size will be added after of the existing sizes.

 Delete the selected bar sizes.

Bar Data Definition

Diameter
The diameter of this bar size.

Area
The area of this bar size.

Breaking Load
The breaking load of this bar size.

Proof Stress
The percentage of the breaking stress at the proof strain. Depending on the design code being used, this could be either .1% or .2% proof stress.

1% Stress
The percentage of the breaking stress at 1% strain.

Breaking Strain

The breaking strain for this bar size.

Elastic Modulus
The elastic modulus of the bar. This defines the slope of the straight portion of the stress/strain curve shown below from the origin.

Plastic Modulus
The plastic modulus of the bar. This defines the slope of the near horizontal straight portion of the stress/strain curve to the breaking strain shown below.

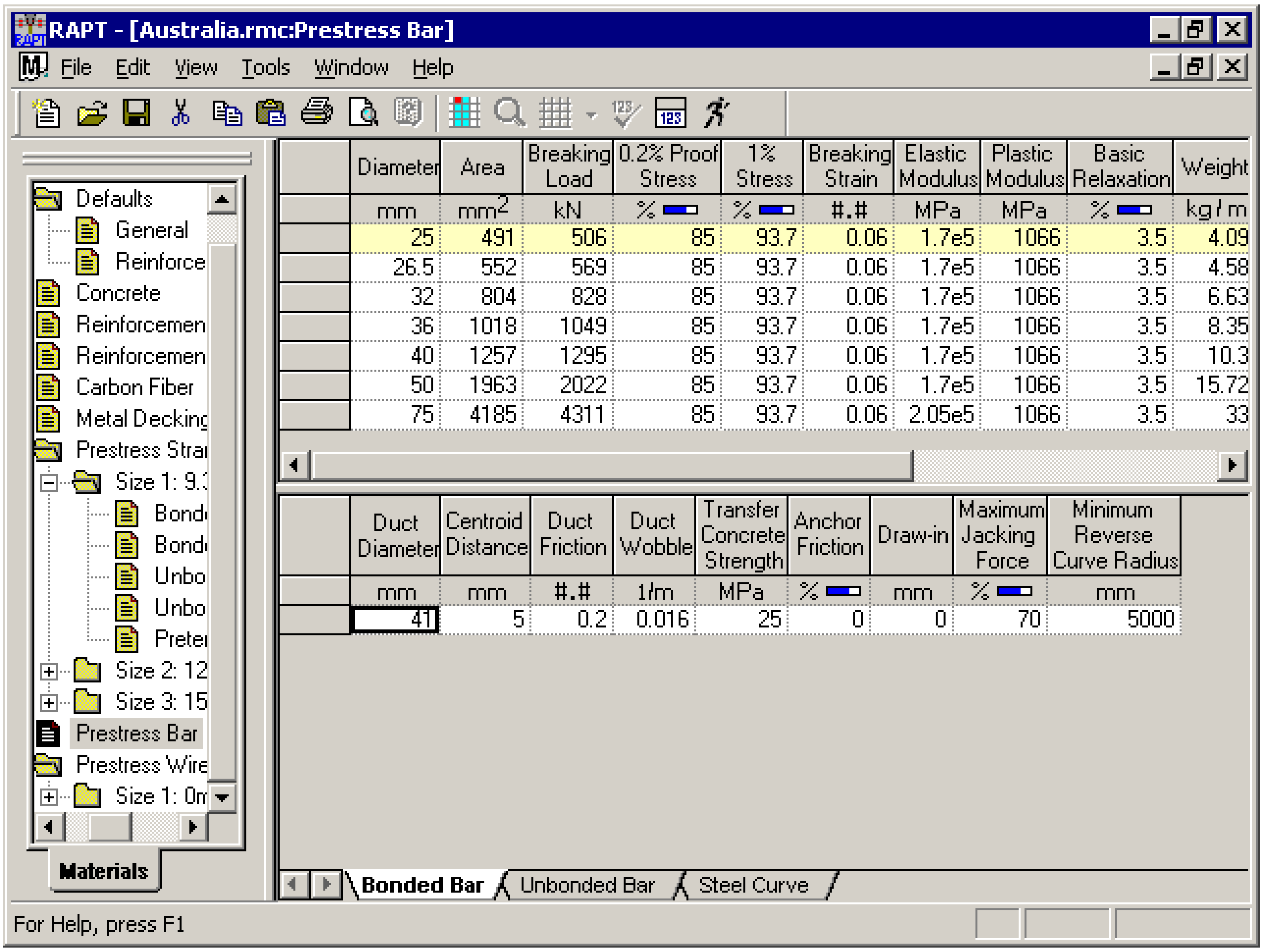
Basic Relaxation
The basic relaxation is normally defined as the bar relaxation after 1000 hours and at a load of 70% or breaking load at a standard temperature of 20 degrees Celsius

Weight
The weight of the bar per unit length.

Set as default
The bar size to be used as the default size in setting up new runs.

Description
A text description of the bar size.

Bonded Bar Tendons
Unbonded Bar Tendons



Duct Diameter
The outside diameter of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the bar diameter so there is room for the bars to move vertically in the duct. When the bars sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the bars in this situation.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view.

Transfer Concrete Strength

The minimum concrete strength required for full force application to this anchorage.

Anchor Friction

The friction loss experienced by the bar in the anchorage. Most anchorages (single bar anchorages normally do not) force the bar to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the bar past the anchorage.

Draw-in

The distance the bar slips through the anchorage when the jack is released.

Maximum Jacking Force

The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the bar in the anchorage. This is normally defined by Design Standards.

Minimum Reverse Curve Radius

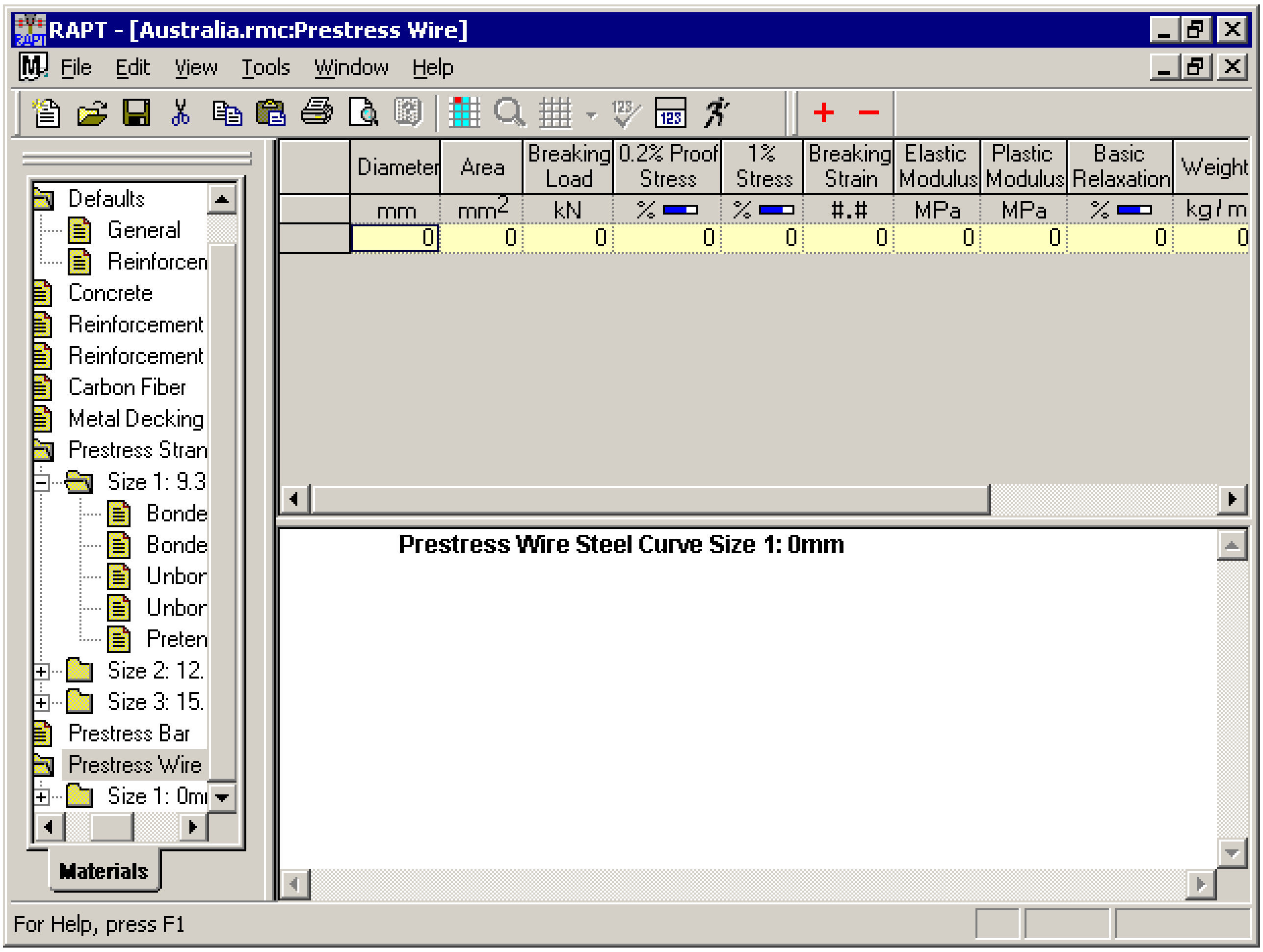
The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

Graphic View

The graphic view shows the curvilinear force/strain diagram developed from the data defined above for the current wire size. This curve will be used for the determination of the prestress force in the strain compatibility calculations for design codes that allow its use. For BS8110, SABS0100, CP65 and Eurocode 2, the stress/strain curves defined in those codes are used.

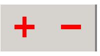
6.10 Prestressing Wire



Different wire sizes can be defined in this view along with the basic data need to define wire properties. When the tree folder associated with wire is opened, a branch is available for each of the wire sizes defined in this table. Each branch then allows the designer sub-branches to define different types of tendons. These are

1. 6.10.1 Bonded Multiwire
2. 6.10.2 Pretensioned

Toolbar




Add an new prestress wire size. If an existing size is selected, the new size will be inserted at that location and the size at this location will me shifted down, otherwise the new size will be added after of the existing types.

Delete the selected wire sizes.

Data Definition

Diameter
The diameter of this wire size.

Area
The area of this wire size.

Breaking Load
The breaking load of this wire size.

0.2% Proof Stress
The percentage of the breaking stress at the .2% proof strain.

1% Stress
The percentage of the breaking stress at 1% strain.

Breaking Strain

The breaking strain for this wire size.

Elastic Modulus

The elastic modulus of the wire. This defines the slope of the straight portion of the stress/strain curve shown below from the origin.

Plastic Modulus

The plastic modulus of the wire. This defines the slope of the near horizontal straight portion of the stress/strain curve to the breaking strain shown below.

Basic Relaxation

The basic relaxation is normally defined as the wire relaxation after 1000 hours and at a load of 70% or breaking load at a standard temperature of 20 degrees Celsius

Weight

The weight of the wire per unit length.

Set as default

The wire size to be used as the default size in setting up new runs.

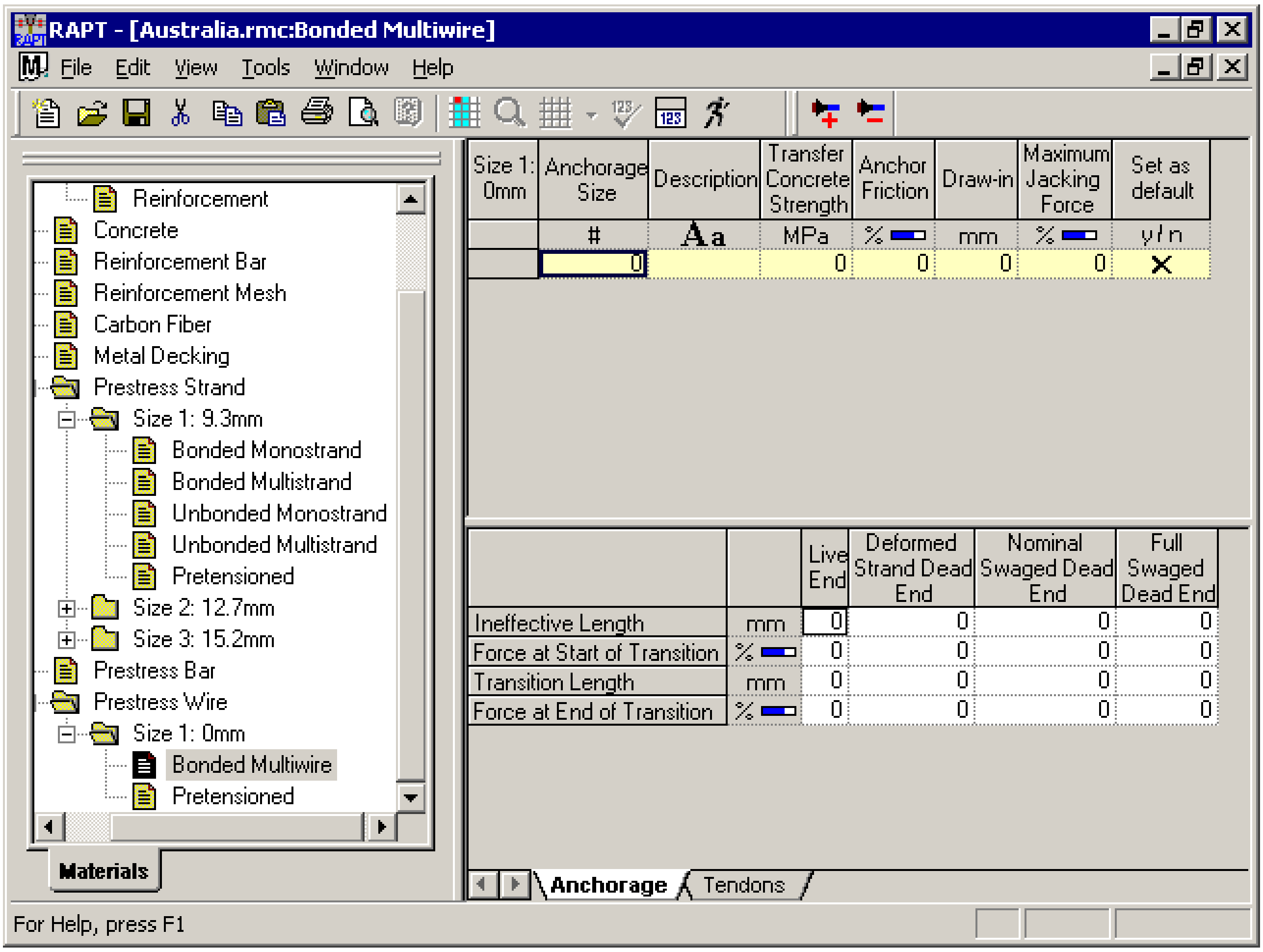
Description

A text description of the wire size.

Graphic View

The graphic view shows the curvilinear force/strain diagram developed from the data defined above for the current wire size. This curve will be used for the determination of the prestress force in the strain compatibility calculations for design codes that allow its use. For BS8110, SABS0100, CP65 and Eurocode 2, the stress/strain curves defined in those codes are used.

6.10.1 Bonded Multiwire Anchorage Sizes

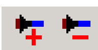


Multiwire anchorages are anchorages where the wires are stressed as a group, are generally square in shape with round duct and the wires bunch together. Sometimes the wires in round duct tendons are stressed individually also but they are still referred to as multiwire tendons in RAPT.

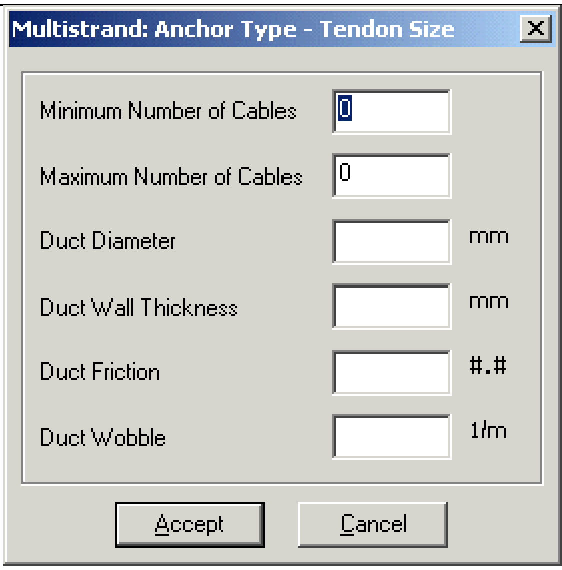
For each wire size the designer can define a range of anchorages to accommodate different numbers of wires. For each anchorage, the basic information to define the anchorage can be edited in this Control data screen. As well, for each anchorage size in the Control data (each row of the grid), there are two further screens of data, accessed via the View Tabs at the bottom of the Child Views, to define

1. 6.10.1.1 Anchorage properties
2. 6.10.1.2 Tendon properties

Toolbar



 Add a multiwire prestress anchor size for this wire size. If an anchor size is selected before adding, the new size will be inserted at the location of the selected size, otherwise it will be added at the end of the list. The following dialog will be presented to allow RAPT to automatically create the tendon data for this anchor size.



The data required in the dialog is

Minimum number of cables
The minimum number of wires that would be placed in this duct.

Maximum number of cables
The maximum number of wires that would be placed in this duct. In doing the calculations for wire centroid etc, if the number of wires nominated will not fit into the duct, RAPT will not create tendon data for the tendon sizes with more wires than can be physically fitted into the duct.

Duct Diameter
The diameter of the duct (external dimension)

Duct Wall thickness
The thickness of the duct wall material.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view.

Using this information RAPT will create the anchorage and tendon data for this anchorage size. This data can then be edited by the designer.

 Delete selected anchorage sizes. All data associated with a deleted anchorage size will be lost.

Data Definition

Anchorage Size
The size of this anchorage in terms of the maximum number of wires that can be accommodated.

Description
A text description of this anchorage.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

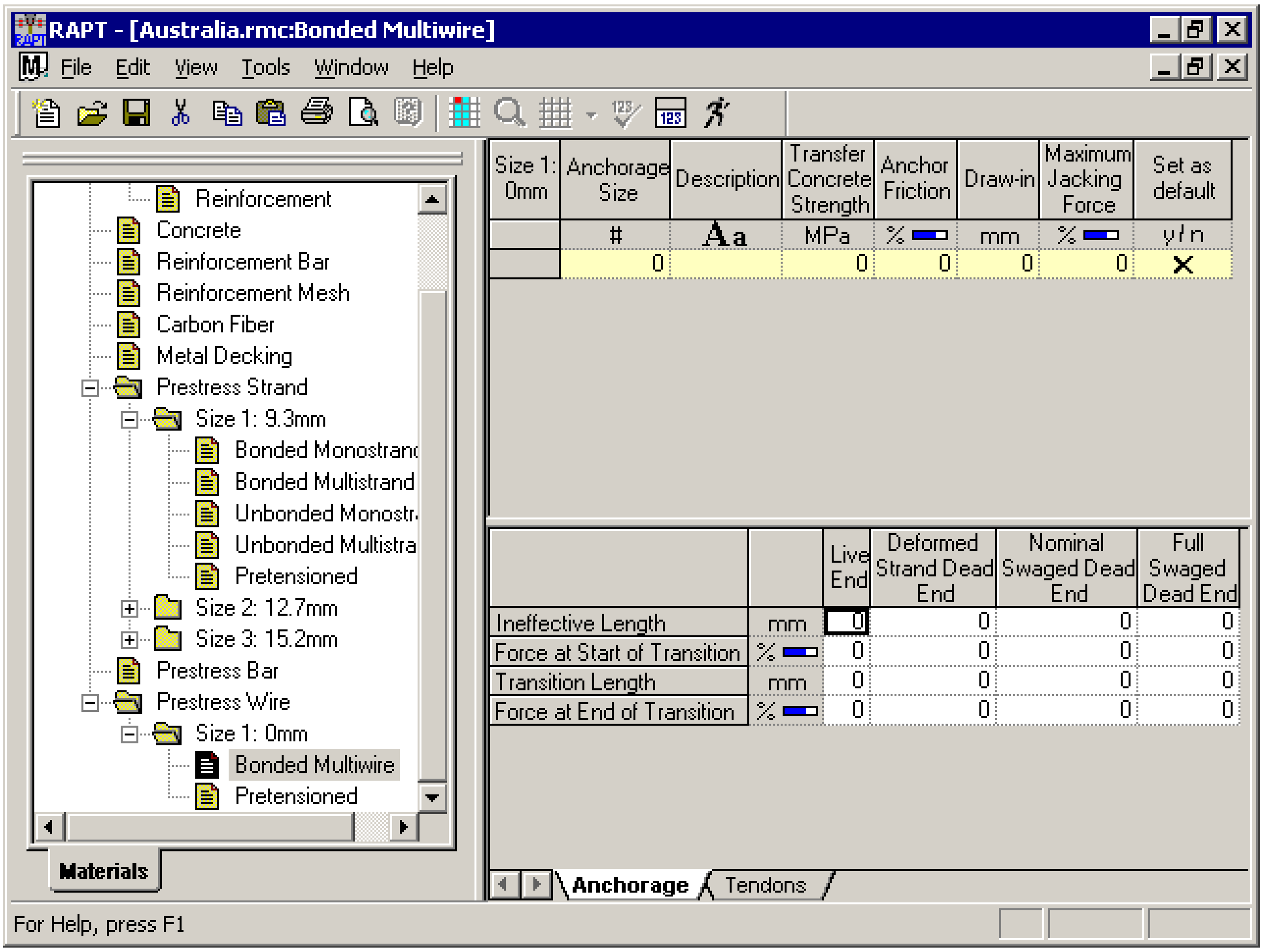
Anchor Friction
The friction loss experienced by the wire in the anchorage. Most anchorages (single wire anchorages normally do not) force the wire to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the wire past the anchorage.

Draw-in
The distance the wire slips through the anchorage when the jack is released.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the wire in the anchorage. This is normally defined by Design Standards.

Set as default
Set this anchorage size as the default size when this tendon type is selected.

6.10.1.1 Anchorage Properties

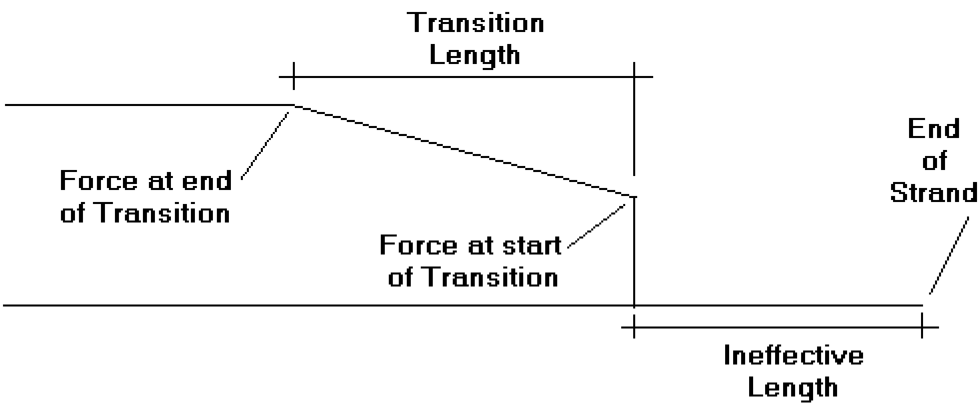


For each anchorage size in the control grid, RAPT provides one screen of Anchorage Properties data as defined below.

RAPT defines four types of anchorage. These are

1. Live End:- Stressing End
2. Deformed Wire Dead End:- Non stressing end in which the force is transferred to the concrete via a splayed section of wire at the end of the wire and a bonded length of wire . Sometimes called an Onion anchorage. The force is not transmitted through a plate through a thin plate is sometimes provided to space the wires.
3. Nominal Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a nominal thickness plate (in the order of 5 to 5mm thick) and also a bonded length of wire.
4. Full Swaged Dead End:- Non stressing end in which the force is transferred to the concrete via an end attached swage or barrel and wedge assembly which rests against a full bearing plate designed to distribute the load to the concrete and with the wire leading from the end of the duct debonded to allow the full force to reach the bearing plate.

For each of these anchorage types, RAPT allows the definition of four parameters defined below and shown graphically.



Ineffective Length

The length from the end of the wire to the first point at which the wire develops stress.

Force at Start of Transition

The maximum force as a percentage of the breaking force that can be developed in the tendon at the first point at which the wire develops stress.

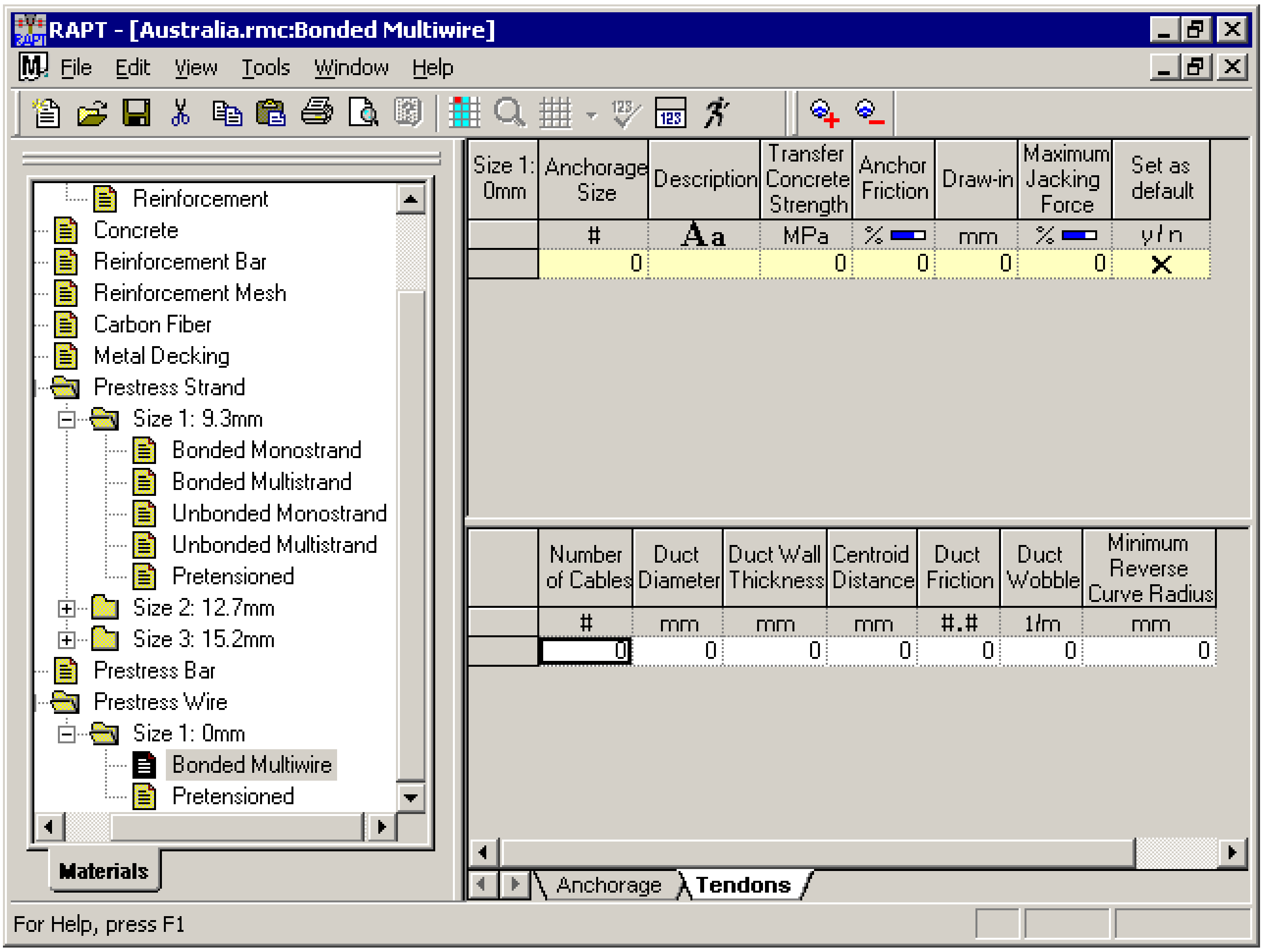
Transition Length

The length over which the development of force in the wire increases.

Force at End of Transition

The maximum force as a percentage of the breaking force that can be developed in the tendon at the end of the Transition Length. From this point onwards the tendon can develop the full breaking force in the tendon.

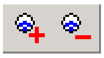
6.10.1.2 Tendon Data



For each anchorage size in the control grid, RAPT provides one screen of Tendon data which allows designers to nominate tendon sizes for the current anchorage in terms of a number of wires as defined below.

The default tendon size selected will be the last one in the list.

Toolbar



 Add a new tendon size for the current anchorage. If a tendon size is selected, the new tendon size will be inserted in the list at the location of the selected tendon size and the tendon sizes at and below that location will be moved downwards. Otherwise, the new tendon size will be added at the end of the list of current sizes.

 Delete the selected tendon sizes.

Data Definition

Number of Cables
The number of wires in this tendon.

Duct Diameter
The outside diameter of the duct used for this tendon size.

Duct Wall Thickness
The thickness of the wall of the duct used for this tendon size.

Centroid Distance
The duct is deeper than the wire diameter so there is room for the wires to move vertically in the duct. When the wires sit on the surface of the duct, they will be eccentric from the centre of the duct. This value is the distance from the centroid of the duct to the centroid of the wires in this situation.

Duct Friction
The duct friction factor.

Duct Wobble
The duct wobble factor consistent with the 6.2.1 wobble type defined in the Materials->Defaults->General data view. Different design codes use one of two different methods to define wobble. Some define it as Unintentional Friction -

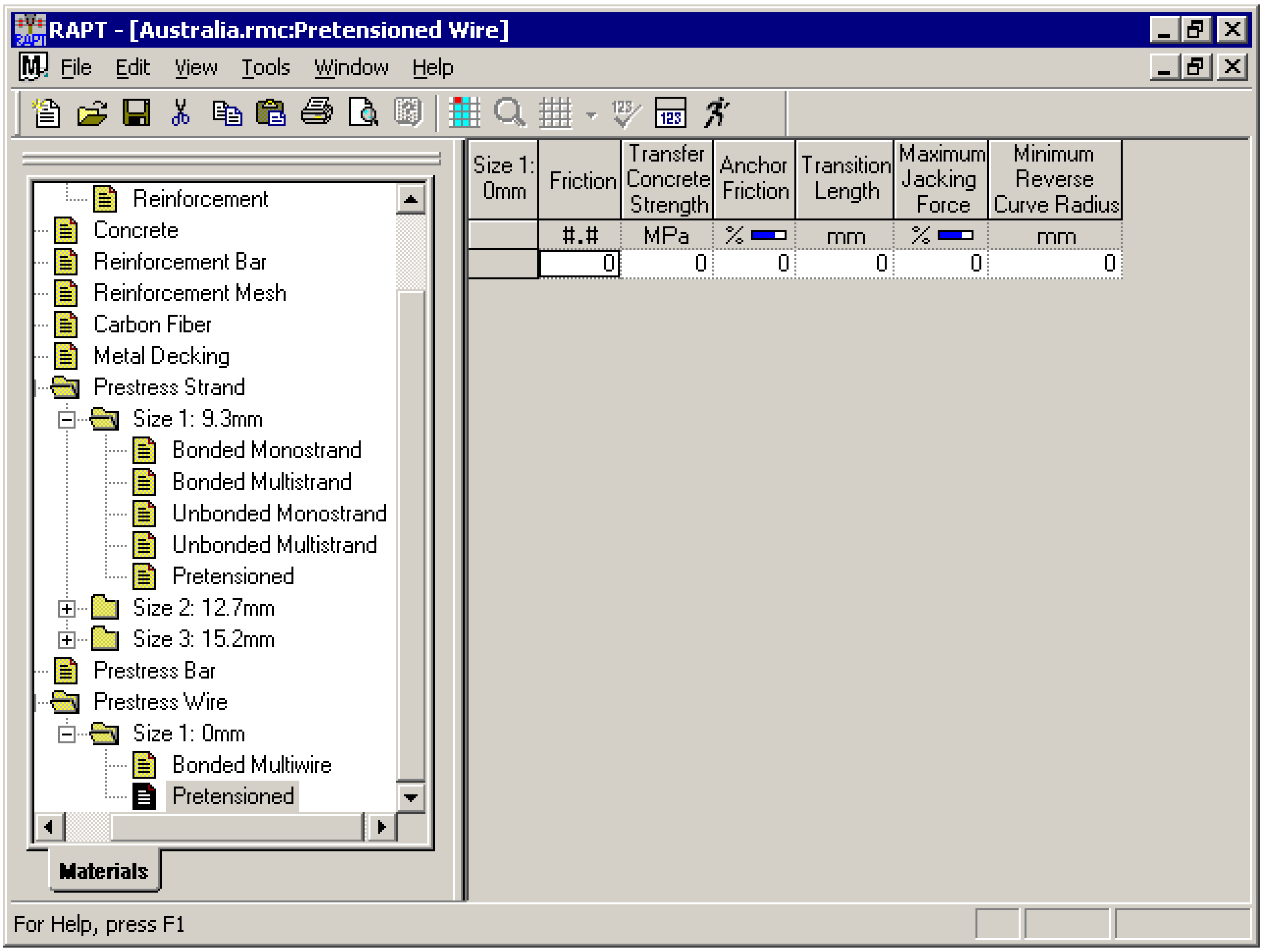
K, which is an angle change multiplied by the duct friction factor. Others define it purely as an angle change, Unintentional Angle Change - *ß*, which must be multiplied by the duct friction factor in the friction equations. So *K* = *u* * *ß*.

Minimum Reverse Curve Radius

The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

6.10.2 Pretensioned



Friction
The friction factor for the wire passing through angle changes.

Transfer Concrete Strength
The minimum concrete strength required for full force application to this anchorage.

Anchor Friction
The friction loss experienced by the wire in the anchorage. Most anchorages (single wire anchorages normally do not) force the wire to change angle over the length of the anchorage and so induce friction forces which reduce the force that can be achieved in the wire past the anchorage.

Transition Length
The length of wire over which the force is assumed to increase from zero at the end to the full tension force.

Maximum Jacking Force
The maximum jacking force that can be used with this anchorage as a percentage of the breaking load of the wire in the anchorage. This is normally defined by Design Standards.

Minimum Reverse Curve Radius
The minimum reverse curve radius that can be used with this tendon size. This size is available from prestressing companies and is set to limit the stresses induced in the concrete due to the bearing forces and resulting bursting stresses and also the wearing of the duct within the curve.

RAPT will allow a smaller radius to be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

7 Frame Definition and Design

7.1 Modelling Approach

RAPT can be used to analyse concrete frames of widely varying configurations. Consequently, it is difficult to discuss all possibilities in detail or even to list all of the possible combinations.

Instead, in the following sections we will attempt to explain the basic approach taken for different member/section types. It is then left to the designer to use their engineering expertise to develop structural systems to suit their purposes. The design staff of PCDC (all engineers experienced in the design of post-tensioned and reinforced concrete structures) are available to assist/advise both on the ability of RAPT to handle various structural systems and also on the best possible solution for a particular problem.

7.1.1 Tendon Profiles

In the profiling of tendons RAPT recognises that the cover limits applied to tendons are measured from the top and bottom surfaces of the slab or the beam. Drop panels, transverse beams and steps are treated as local additions to the concrete section as defined in Input Screen 2 for tendon profiling purposes.

INPUT: All profile measurements defined in input are measured from the member soffit at midspan by default and are given to the strand centroid. This reference level may be modified by the designer in 7.2.5.3 Prestress->Adopted Profiles->Profile Height Datum.

OUTPUT: All output tendon profile heights are given from the bottom concrete surface to the underside of the duct except at anchorages where the profile is measured to the centre line of the tendon. Note: In all cases, RAPT ignores transverse beams in defining profile heights and for two way slabs RAPT ignores the presence of drop panels when giving profile heights.

Default Profiles
Tendons will be defined to start and end at the extreme ends of the concrete frame with the anchorage types specified when selecting the tendon input type. The tendon profile will be automatically generated to utilize minimum cover at all high and low points with the anchorages placed at the centroid of the concrete section (ignoring drop panels and transverse beams). Where drop panels or transverse bands/beams are present. RAPT will automatically drape parabolic profiles to the face of these elements at internal columns if they are still enough (based on the length/depth ratio of the element). The designer can also select in the default setup for these profiles to be draped to the face of columns if desired.

Modified Profiles
The designer is free to modify

1. The locations of the ends of the tendon.
2. The tendon profile type in each span.
3. The locations of the profile control points in each span.
4. The height of the tendon at any profile control point.

It is possible for the user to lower an internal anchorage below the cover limit from the prestress menu in order to stress the cable from the underside of the slab but the tendon must be within the cover limits at the next defined profile point.

Each tendon profile shape can have any number of tendons in it. These will be assumed to be placed in the same plane over the length of the tendon. Each tendon profile shape can have any user defined start and end location within the frame length and the profile shape is controlled by the designer. RAPT will allow tendons to be outside the concrete section simulating the effect of external tendons as are sometimes used in concrete repair and bridge design.

Using Toolbar Icons RAPT allows the automatic re-profiling of tendons based on

1. Balanced Loads specified by the designer:- the user is asked to define loads to balance for each span that the tendon is active in. These loads can be any fraction of the self-weight extra dead load and live load cases defined by the designer plus an extra line load defined in this table. RAPT will generate a bending moment diagram for the loads selected and determine the minimum number of tendons and tendon profiles in each span required to balance this moment diagram using the profile types previously selected by the designer. This dialog also gives information regarding the efficiency of each span and the minimum number of tendons which would be required for each span. The designer can use this information to gain an idea of the need for extra short tendons in some areas.
2. Cover and Control point recalculation:- users can modify the allowable offsets and drape locations to allow for varying covers or special profiling options.

Reverse Curves
Reverse curves will be placed at all sudden changes in angle of a tendon. These will be placed at supports, at connections of parabolas to straight lengths and at connections between connecting straight lengths. These reverse curves will be based on the radius defined for each span. This method results in the calculation of transition points based on the logical length of transition curve needed for each situation. The locations of the transition points and the tendon heights at these locations are available in the output.

The radius can be modified by the designer but should never be set to a value less than the defined minimum unless the designer takes steps in the design to allow for increased friction, excessive bearing stresses on the duct and concrete through the curve and bursting of the concrete over the length of the reverse curve. RAPT will also ensure that the curvature of the main parabolic curve is also within the bounds set by the defined radius of curvature and will give an error message if this is violated.

RAPT does not use the logic of placing transition points in the curves at nominal locations (eg .1 * span length). This logic can very easily result in illogical and inconsistent reverse curve lengths and curvatures and can result in effective reverse curve radii which are far too small which could result in failure of the member.

7.1.2 Tendon Actions

RAPT calculates forces, reactions or moments from prestress in the following cases:

1. linear uplift forces from parabolic drapes
2. linear reaction forces at each transition curve at each intersection of straight sections of tendons
3. linear reaction forces at each transition curve
4. moments and/or force couples due to the change in concrete sections
5. moments due to eccentricity of tendon anchorages from the centroid of the concrete section

The tendon actions calculated by RAPT are based on the average prestress force in each span. These are used to calculate the secondary prestress moments.

The primary prestress moments are determined from the eccentricity of the prestress force at each point along the frame and the actual prestress force at that point, not the average prestress force as is used for the secondary moments. They are not determined from the tendon actions.

The total prestress effect at each point is then the sum of the primary and secondary prestress moments at that location.

7.1.3 Transverse Beams / Drop Panels / Steps

All 3 of these concrete shape types can be defined in RAPT as

1. Flexural:- Will act as part of the slab. Section change is small enough to allow full shear flow through the changes in section.
2. Rigid/Non-Flexural:- Will not flex with the slab. Section changes are too sudden to allow shear flow into the deeper section.

Transverse Beams/Bands

Transverse Beams can be defined in RAPT in two different ways

1. Transverse Beam:- RAPT will not calculate moments and forces due to the change in cross-section for prestress effects. Transverse beams are only taken into account in the calculation of moment of inertia of the equivalent column (not included in the full column stiffness or net column stiffness calculations), punching shear calculations and flexural critical section location if the dimensions allow and if requested. The stiffness of the slab or beam being analysed is not adjusted to take into account any possible effect from transverse beams. Their only effect on moments and deflections is through the column stiffness. To achieve a stiffening effect through haunch action a transverse beam should be modelled as a transverse band as described below as long as the designer is happy that the concrete section will act in flexure with the slab.
2. Transverse Band:- RAPT will calculate prestress moments due to section changes, and will use the transverse band concrete section in the calculations of the stiffness of the spanning member where appropriate, the cross section strength calculations and punching shear calculations. In the calculation of the moment of inertia of columns RAPT takes into account all concrete sections defined in the input. Transverse bands are treated like drop panels in accordance with ACI318-83 and AS1480 as detailed in section T.2.2.

Drop Panels/Drop Caps/Capitals

Drop Panels can be defined in RAPT in two different ways

1. Capitals:- Column capitals are simply an increase in the column plan dimensions at the underside of the concrete member. They have an effect on column stiffness, location of critical design sections and punching shear perimeter.
2. Drop Cap:- RAPT will not calculate moments and forces due to the change in cross-section for prestress effects. Drop Caps are only taken into account in the calculation of the member stiffness for analysis and punching shear calculations.
3. Drop panel:- RAPT will calculate prestress moments due to section changes, and will take into account the drop panel dimensions when calculating the stiffness of the member where appropriate, the cross section strength and beam and punching shear. In the calculation of the moment of inertia of columns RAPT takes into account all concrete sections defined in the input. Transverse bands are treated like drop panels in accordance with ACI318-83 and AS1480 as detailed in section 7.2.2. In two way slabs under AS3600, the drop panel depth is used in the calculation of the critical section for flexure.

Steps

Steps are used to model changes in the member shape both vertically and/or horizontally. A step is defined for each surface of a member i.e. top of slab, bottom of slab. The designer must check to ensure that the tendon profiles are within the concrete profile.

RAPT allows the designer top nominate whether a step is Flexible or Rigid for capacity calculations. If it is defined as flexible, it will be treated as any other area of the member for all design calculations. If defined as Rigid, no capacity calculations will be done over its length and the curvature over its length will be assumed to be zero for deflection calculations.

SUMMARY		
Transverse Beams	Drop Panels	Flexible Steps
Included in Self Weight calculations	Included in Self Weight calculations	Included in Self Weight calculations
No moment and force calculations due to change in cross sections	Change in cross section invokes moments and forces into design	Change in cross section invokes moments and forces into design
Does not affect stiffness of slab / beam	Affects slab / beam stiffness	Affects slab / beam stiffness
Affects moment of inertia of Equivalent Column	Affects Equivalent Column calculations by slab inertia	Affects Equivalent Column calculations by slab inertia

Affects Punching Shear Calculations	Affects Punching Shear Calculations	Affects Punching Shear Calculations
Used to define end drape tendon profiling positions if requested.	Used to define the end of tendon profile positions.	Not used to define end drape tendon profiling positions

7.1.4 Concrete Layers / Elements

RAPT uses two cross section input methods to model cross sections. These are

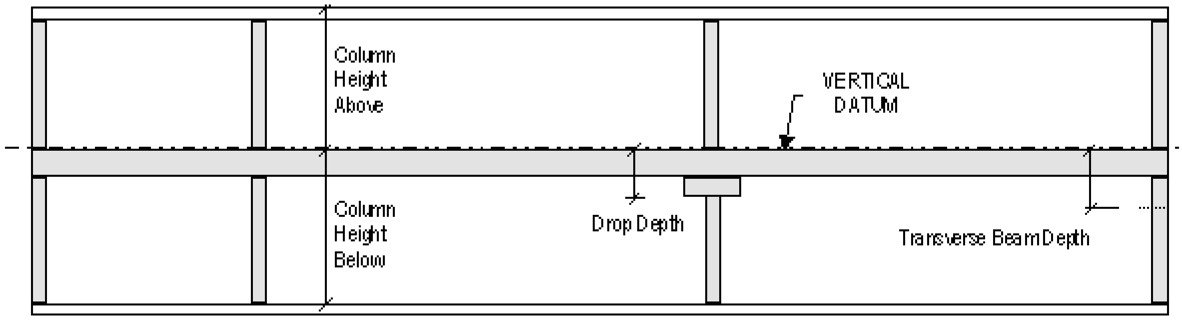
1. Layers:- Layers define the main cross section members. These always run full length of the structure and include the typical cross sections i.e. slab, beam, drop panels etc. When users input data into slabs, beams. drop panels and transverse beams data views, they are actually defining default layers.
2. Elements:- Elements are defined over a portion of the structure and are used to define voids, set-downs or plinths etc. They are complementary to layers and added to the shape defined by the layers.

Datum

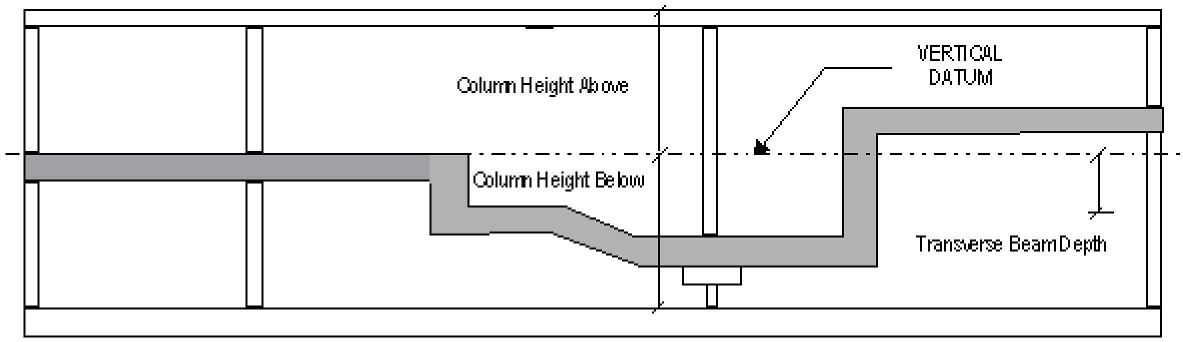
With the inclusion of steps, it has become necessary to define datum lines, from which all measurements are taken. They are defined as

1. Vertical Datum - For Vertical Steps - the top of the slab as defined in spans input screen . This was always the datum in earlier versions of RAPT although it was never defined as such.
2. Horizontal Datum - For Horizontal Steps - the line through the centre of the supports.

The following two diagrams show the vertical datum line before and after steps have been added.



Before Steps Added

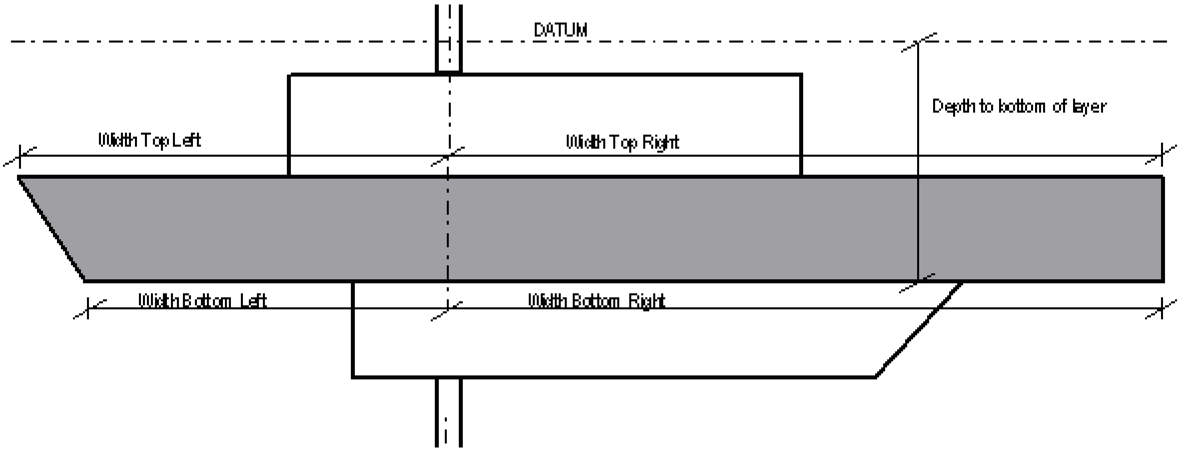


After Steps Added

Concrete Layers - General Description

In order to efficiently model structures, users need to have a clear understanding of how RAPT models a cross section. A cross section is made up of layers to which the user can add elements.

A Layer runs for the full length of the structure. Layers can not cross each other, but can have a zero depth and width. Data input in the spans/beams/drop panels and transverse beams data screens is used to define the default layers. Examples of layers include slab layer, beam layer, upturned beam layer, extra downturned layer, etc Users can add extra layers in the layers input screen and apply steps to layers, both vertical and horizontal.



A layer is defined in terms of (see above)

1. A depth to the bottom of a layer from the datum. The thickness of a layer is thus defined as the distance between the bottom of a layer and the bottom of the layer immediately above it.
2. Widths Left and Right / Top and Bottom. These dimensions are measured from a central control point which is the horizontal datum.

The slab is the only exception, as it is defined by two layers. These layers are

1. Slab Panel Width:- Has zero depth thus defining the top of the slab layer. The width defines the panel width of the slab layer.
2. Slab layer:- Depth to bottom of slab layer thus defining the bottom surface of the slab layer. The width defines the Beam flange width or Column strip width if a two way system is defined.

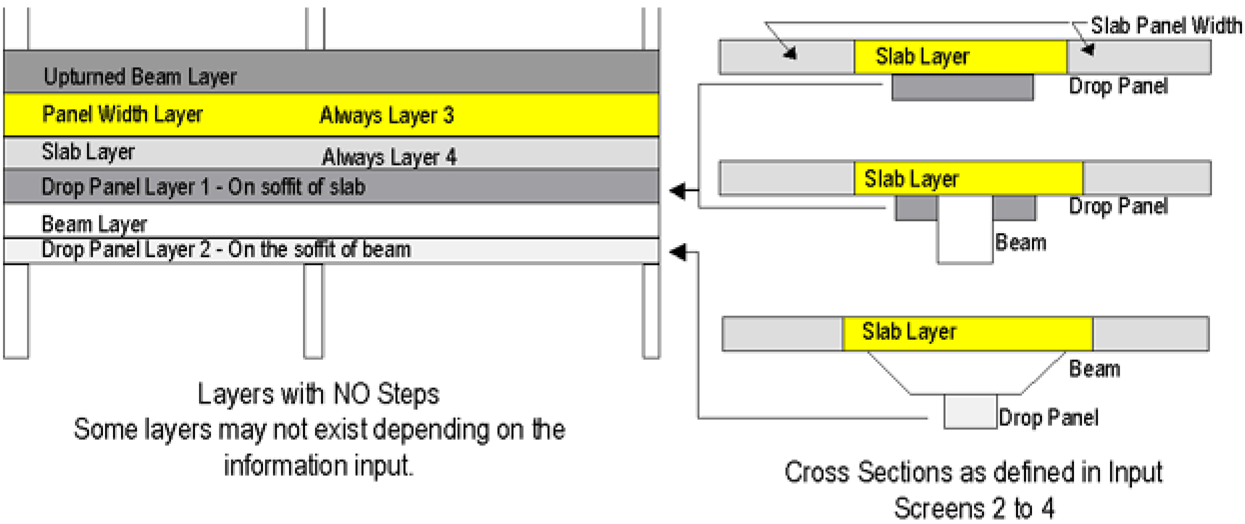
Upturned layers are above the Slab Panel Width layer and Downturned layers are below the Slab layer. A user can define up to 3 upturn layers and an unlimited number of downturn layers in modelling a cross section.

Layers from Input Spans to Transverse Beams

A cross section is firstly defined by the data input by the user in data screens Spans to Transverse Beams. This cross section can then be added to and or modified by using the Layers Input screen . Using Input screens Spans to Transverse beams users can define layers for

1. Slab (always defined as layer 3 and 4) (transverse beams/bands are included in the slab layer as they must extend full width of the panel).
2. Beam
3. Drop panel on under side of slab
4. Drop Panel on underside of beam
5. Upturned beam

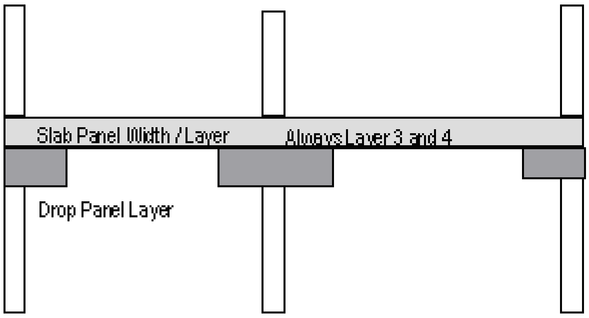
RAPT then takes the input and transforms it into layers defining them in the terms shown above. The top of the slab will always be set to the Vertical Datum by default. The figure below shows the basic layers that can be created from the input. RAPT will only use layers that are required.



When converting user input from the general slab/beam input to layer designation, RAPT will assign the layers as follows

Upturned Beam	Layer 2 or Not Present
Slab Panel Width	Always Layer 3
Slab Layer	Always Layer 4
Drop Panel on Slab	Layer 5 or Not Present
Down turned Beam	Layer 5 or 6 or Not Present (Layer 5 if No Drop panel on Slab is defined)
Drop Panel on Beam	Layer 6 or 7 or Not Present (Layer 6 if No Drop panel on Slab is defined)

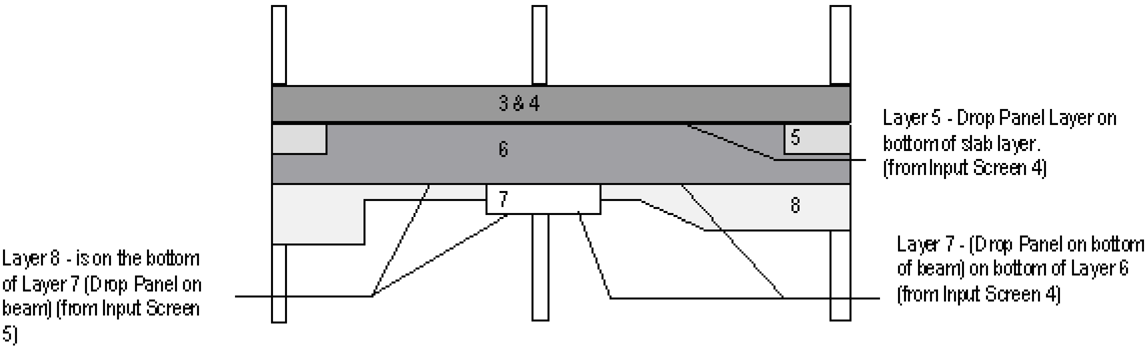
The figure above shows the layers for each member type. These have been shown at consistent depths to show that a layer once defined exists over the entire length of the structure. In reality though, drop panels are only defined for a portion of the structure as shown more realistically below. Where no drop panels are shown, the layer still exists but has a zero depth below the bottom of the layer above.



Slab and Drop Panel Layers
Steps set by Input Screen 4 [Drops]
Drop panel layer has 0 thickness
between faces of drop panels.

Layers defined in Layers Input Screen

The Layers Input Screen allows users to add extra layers. The figure below shows a combination of layers defined by In the General Section Input screens and the Layers Input Screen .



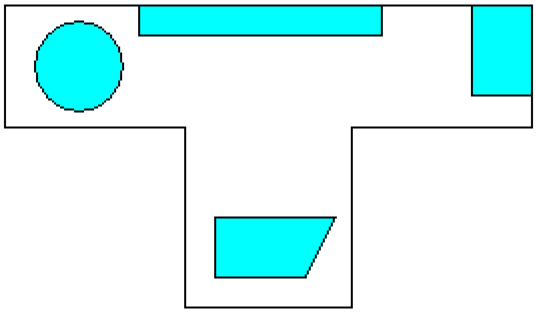
Layers with Steps. Some layers have 0
depth below the layer above.

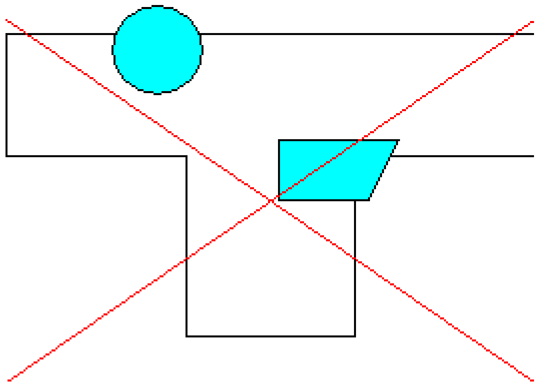
Elements

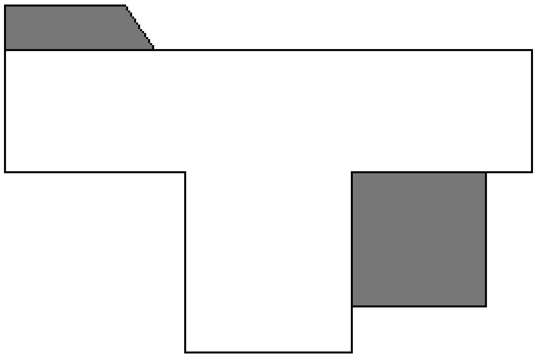
An element is a void in a concrete layer or an extra piece of concrete which is defined over a portion of the main structure. Elements are designed to allow the user to define voids, set-downs or plinths. Elements have a defined shape and a defined location within the structure. The rules that apply to elements are

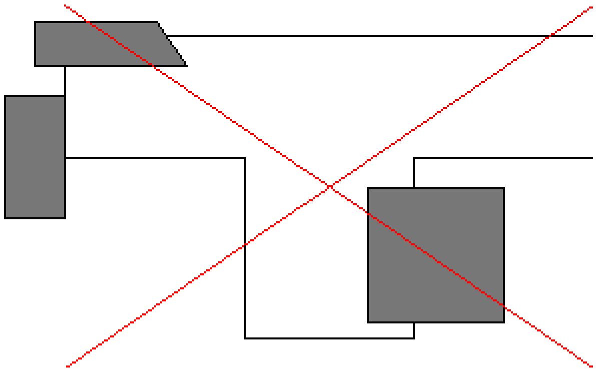
1. A void element must be contained within the existing layers.
2. A solid element must be defined so that it is outside of all layers, but touching at least one surface of the layers. For solid elements, the side of the element that is in contact with the layers must remain in contact with the layer for its full length / depth.
3. Steps applied to layers will not affect the elements unless specifically defined by the designer as a step in the element. Elements will hold their vertical and horizontal positions. Users must ensure that steps applied to layers do not cause the defined elements to conflict with rules 1. and 2. see figure below.
4. Elements can not overlap

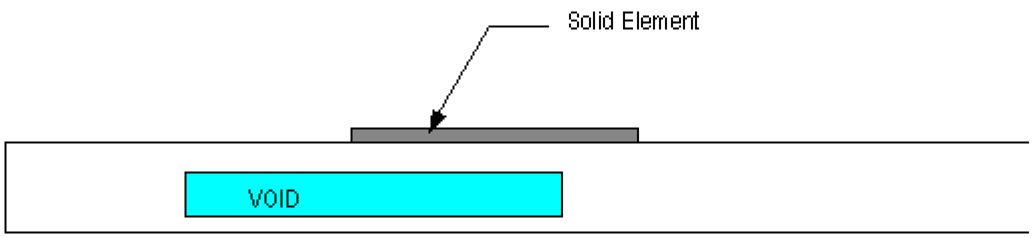
5. Steps can be added to elements to modify their shape over the length of the element and they can be reduced to zero depth (this disappearing over part of the length of the element) using vertical steps.


Allowable Voids

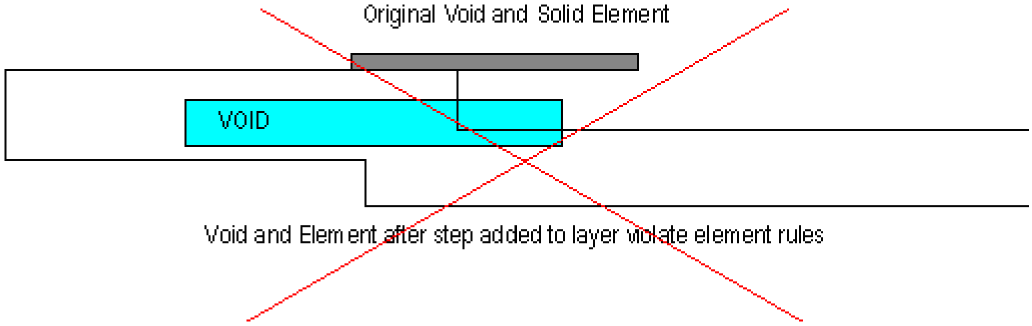

Voids NOT Allowed


Allowable Solid Elements


Solid Elements NOT Allowed



Original Void and Solid Element



Void and Element after step added to layer violate element rules

7.1.5 Frame Systems

Now let's consider the more general question of modelling an overall system.

Definition of Member Types eg Beam - Bonded Post-Tensioned - One Way

RAPT can analyse the following structural systems

1. Reinforced slab one way or two way
2. Reinforced beam one way or two way
3. Post-Tensioned slab one way or two way
4. Post-Tensioned beam one way only
5. Bonded and Unbonded Post-Tensioned
6. Pre-Tensioned

Reinforced Slab

One Way

Loads input in kPa are based on the one-way width (i.e. design width) in each span. If the design strip is to carry extra load from outside the design width, then these loads must be converted to a distributed load based on the design width or be defined as line loads. Point loads and Point Moments are applied to the one way width, not the panel width. They are assumed to extend over the full panel width at the same intensity for column reaction calculations and are multiplied by the ratio of panel width/design width to achieve this.

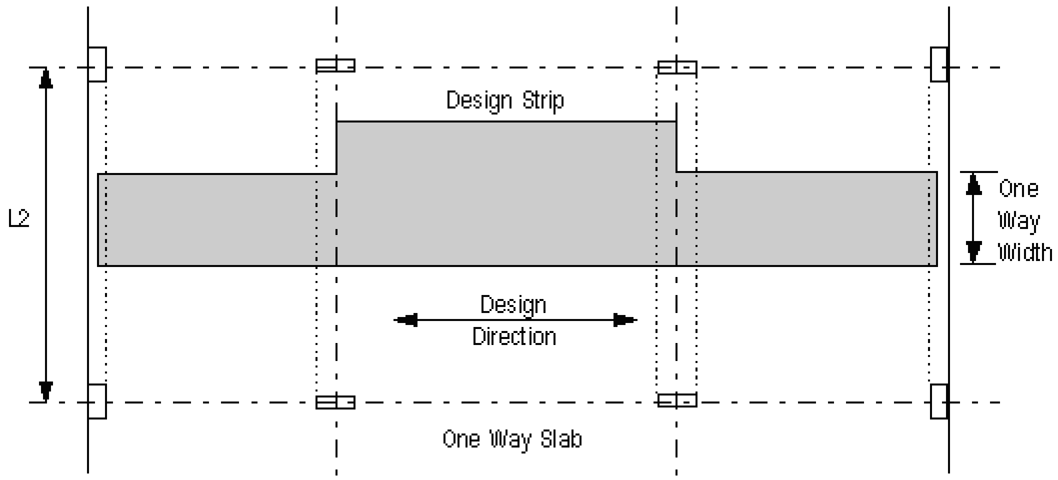
The One Way Width is used by RAPT in calculating the section properties and strength capacity.

Design widths can not vary over the length of a span, even using steps. i.e. the slab can not contain any horizontal steps.

Column loads, used to calculate Punching Shear, are calculated based on the panel width. The panel width is input in the second input screen. To ensure RAPT calculates the correct column loads, the panel width should equal the column spacing. Note: the panel width has no relation to the design load.

It would be unrealistic to allow RAPT to use the full column stiffness over the design width. RAPT proportions the stiffness to the design strip in the following way

Appropriate column stiffness for the design strip = design width / L2 x the Effective Column stiffness



One Way Full Width

Loads input in kPa are based on the panel width in each span.

Column loads are calculated based on the loads applied to the panel.

Design widths can vary over the length of a span using steps. i.e. the slab can contain horizontal steps.

Two Way average

This is a design method used in USA and espoused by the American PTI for the design of prestressed two way slabs. It is, in effect, the same as One Way Full Width design but uses the two way minimum reinforcement rules for unbonded prestress. This method is not allowed by some design codes and should only be used once the designer understands the approximations inherent in the method and the inherent limitations on the use of the method. This method is not allowed by AS3600.

Column loads are calculated based on the loads applied to the panel.

Loads input in kPa are based on the panel width in each span.

Design widths can vary over the length of a span using steps. i.e. the slab can contain horizontal steps.

Two Way

Loads input in kPa are based on the Panel Width at all locations along the length of the load. Thus RAPT will apply a total load of (kPa x panel width). If the panel width varies over the span, then RAPT will account for this when determining the applied loads.

Column loads are calculated based on the loads applied to the panel.

The column stiffness is calculated with the full panel width of the slab being considered. There is no reduction in column stiffness as in the case of the one way slab.

RAPT performs a single frame analysis based on the full panel width, then divides the moments into column and middle strips based on the distribution factors. (See Section 7.4 for various code distribution factors) The defaults factors may be manually overridden by the designer if desired.

Reinforced Beam

One Way

Loads input in kPa are based on the Panel Width at all locations along the length of the load. Thus RAPT will apply a total load of (kPa x panel width). If the panel width varies over the span, then RAPT will account for this when determining the applied loads.

In calculating the effective column stiffness, RAPT uses the full panel width. When calculating the slab inertias, RAPT uses the One Way Width (effective flange width). The One Way Width is used during the design process in calculating the section properties and strength capacities.

Two Way

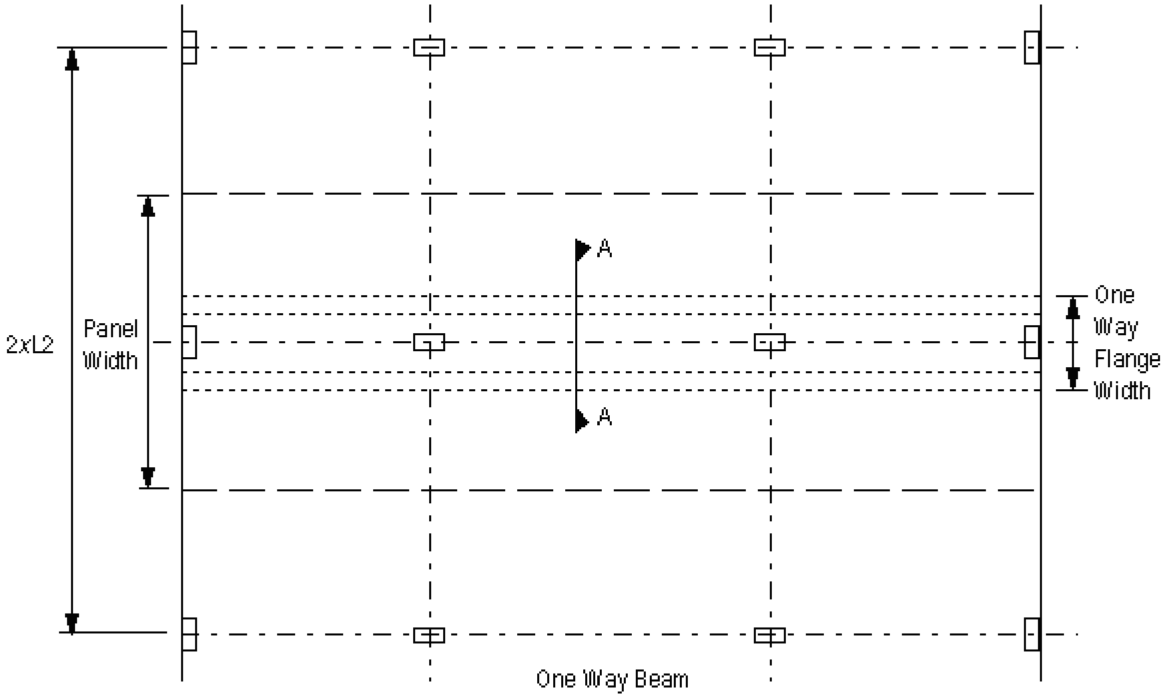
Loads input in kPa are based on the Panel Width at all locations along the length of the load. Thus RAPT will apply a total load of (kPa x panel width). If the panel width varies over the span, then RAPT will account for this when determining the applied loads.

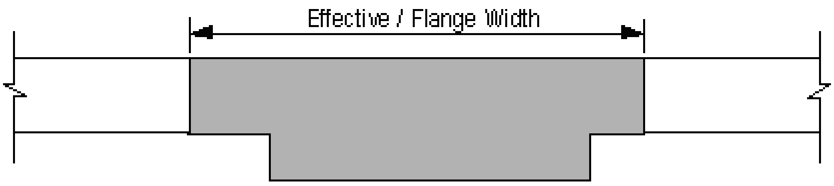
The full Panel width is used in calculating the effective column stiffness.

A single frame analysis is performed by RAPT, with the resulting moments being distributed between the beam, column strip and middle strip as per AS1480 and BS8110.

The lateral distribution factors allow the designer to specify the amount of column strip moment carried by the beam. RAPT does not design the column strip in this case. The designer can therefore take one of two approaches

1. Allow the beam strip to carry the extra column strip load. This is accomplished by choosing in lateral distribution factors to allow the beam to carry 100% of the column strip.
2. Continue the middle strip reinforcing into the column strip area.





SECTION A-A

Post-tensioned Slab

One Way

Same as Reinforced One Way Slab

Two Way

Loads input in kPa are based on the Panel Width at all locations along the length of the load. Thus RAPT will apply a total load of (kPa x panel width). If the panel width varies over the span, then RAPT will account for this when determining the applied loads.

RAPT performs the following tasks in order to calculate the Column and Middle Strip Moments

1. A separate analysis is carried out for the prestressing case for Column and Middle Strips. This allows different loads to be balanced in the Column and Middle Strips.
2. A single analysis for all other load cases, with the final moments being distributed to Column and Middle Strips with respect to the lateral distribution factors.

When Prestressed Two-Way is chosen, RAPT prompts the user for a % of the load to go to the column strip. This is used as the default value of the load to be carried by the column strip. Usually an average factor of 70 % of the total panel load is nominated for the column strip. This is amended by simply over-typing and deleting in the usual way. This option only appears for two-way prestressed slabs. For reinforced two-way systems the percentage taken to the column strip varies at every point as described in section T.4. Note: When designing two way slabs the designer must ensure that they design in a consistent manner (i.e. % of load carried in Column and Middle Strips in one direction complements the loads carried in the Column and Middle Strips in the other direction). RAPT has no way of checking if the two separate design directions are consistent.

Post-tensioned Beam

One Way

Same as Reinforced One Way Beam

Bonded And Unbonded Post-tensioned

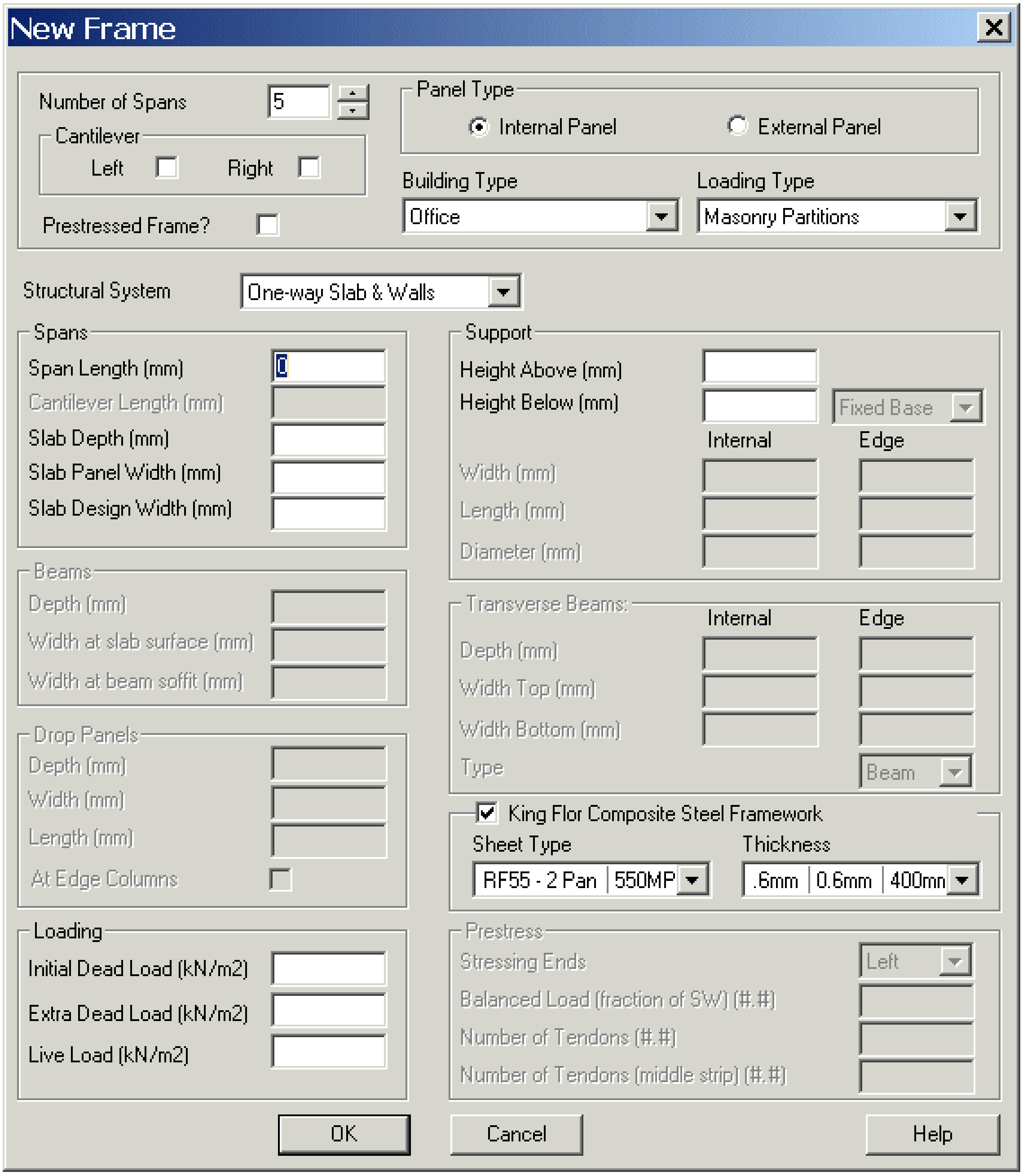
This option allows for the difference in grouting or greased tendons. RAPT designs unbonded and bonded according to each individual code. The stress in the tendon under ultimate conditions is determined from the appropriate code formula. Under service conditions it is conservatively assumed that there is no increase in stress in the tendons as load is applied to the member. (See Section T.7.5) RAPT will automatically update the strand properties in screen to suit the required tendon properties.

Pre-Tensioned

Pretensioned tendons differ to Post-Tensioned tendons in that they are stressed prior to the concrete pour. Usually Pre-Tensioning is associated with precast members. RAPT will automatically update the properties in screen to suit the required tendon properties.

7.2 Frame Definition

7.2.1 Input Dialog



When inputting a new frame, the dialog above is presented to the designer. The data cells initially available in the top area of the dialog provide the minimum data input required by RAPT to set up a frame. Depending on the options selected in this area and in Structural System, different data will be requested in the lower area Structural System. All of the data entered here can be modified once the data is accepted and the main data structure is created for the frame.

Data Definition

Number of Spans
The number of spans (not including cantilevers) in the frame.

Cantilever Left
Check the box to nominate a left cantilever in the frame.

Cantilever Right
Check the box to nominate a right cantilever in the frame.

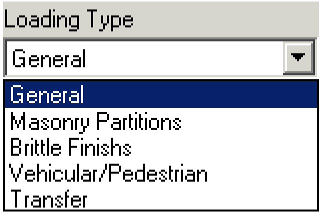
7.2.2 Panel Type
Select whether the Panel Type is to be an internal (there are other support lines on both sides of this support line) or an external panel (this is the edge support line of the floor.

Prestressed Frame
Check the box if prestressing strands or tendons of some sort are to be defined in the frame.

7.2.4.2 Building Type

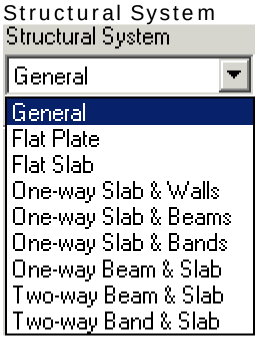
Nominate the building type for the creation of the load combinations.

Loading Type
The loading type controls the settings used for the deflection limits. The options are shown below.



1. General:- Only total deflection is of interest. Nothing is being attached to the slab which will be damaged by deflections of this magnitude so only visual and sensory limits are required.
2. Masonry Partitions: The member is supporting Masonry Partitions which many be damaged by deflection.
3. Brittle Finishes:- The member is supporting or has attached finishes which may be damaged by deflections, e.g. glass curtain walls.
4. Vehicular/Pedestrian:- The member is carrying vehicle or pedestrian traffic.
5. Transfer:- The member is a transfer member supporting other structural elements which require special deflection limitations. This would apply to any transfer member supporting and transferring a concrete structure above to a different support layout below.

Structural System



The list of structural systems above is available to select the system required for this design. When a structural system is selected different area of the Options for Structural System are made available for further input of data.

Structural Systems

The following structural systems can be selected for RAPT to create the main data file.

General System
This is the default setting and does not define an actual system. RAPT will create a default system in the data and the designer can modify it and all of the associated data in the main input screens. No extra data is required to be input other than that already defined.

Flat Plate
Two way slab supported by columns. No drop panels are provided. Data fields available are

1. Slab
2. Supports
3. Transverse beams (edge only)
4. Prestress if selected
5. Loadings

Flat Slab
Two way slab with drop panels supported by columns. Data fields available are

1. Slab
2. Supports
3. Transverse beams (edge only)
4. Drop panels
5. Prestress if selected
6. Loadings

One Way Slab and Walls

One Way slab with nominal design width supported by transverse walls. Data fields available are

1. Slab
2. Supports
3. Prestress if selected
4. Loadings

One Way Slab and Beams

One Way slab with nominal design width supported by transverse beams. Data fields available are

1. Slab
2. Supports
3. Transverse beams
4. Prestress if selected
5. Loadings

One Way Slab and Bands

One Way slab with nominal design width supported by transverse band beams. Data fields available are

1. Slab
2. Supports
3. Transverse beams
4. Prestress if selected
5. Loadings

One Way Beam and Slab

One way beam supporting attached slab supported by columns. Data fields available are

1. Slab
2. Supports
3. Beams
4. Transverse beams (edge only)
5. Prestress if selected
6. Loadings

Two Way Beam and Slab

Two way beam system on the columns supporting slab panels. Data fields available are

1. Slab
2. Supports
3. Beams
4. Transverse beams
5. Prestress if selected
6. Loadings

Two Way Band and Slab

Two way band beam system on the columns supporting slab panels. Data fields available are

1. Slab
2. Supports
3. Beams
4. Transverse beams
5. Prestress if selected
6. Loadings

Structural System Data

The following data options are available for the definition of a default structural system. Not all of the options are available in all cases, depending on the requirements of the system and other general options above that have been selected. The options available in each case are discussed below. The frame is defined using this data in all spans. Once the frame data file is created from this data, the designer can then make any modifications to the data to suit the frame to be designed before running the frame and viewing the results.

In cases where two numbers can logically be the same, eg, column height above and below, column width and length, beam width at top and bottom, RAPT will default the value of the second item to equal the first one if the user follows the input order programmed into RAPT. The standard way to move around this dialog is to use the TAB key. This will move through the data in a logical order.

7.2.3.1 Slab

1. Span Length:- Default span length. All spans will be set to this length. Span lengths can be modified later when the data file has been created.

2. Cantilever Length:- Default length of any cantilevers specified as Yes above. If no cantilever exists at an end, RAPT will automatically extend the edge of the slab to the outside face of the column.

3. Depth:- The depth of the slab.

4. Panel Width:- The overall width of the slab panel being designed. If an external panel is selected above, RAPT will automatically extend the external edge of the slab to the outside faces of the columns and the panel width will be dimensioned from this face.

5. Design Width:- For Nominal Width One Way Slabs Designs, the actual design width to be used.

7.2.3.2 Support

1. Height Above:- Column height above if there is one, otherwise 0.

2. Height Below:- Column height below if there is one, otherwise 0. Repeats height above by default.

3. Internal:- All columns except those at the extreme ends of the frame.

    1. Width:- Column dimension across the frame if rectangular.
    2. Length:- Column dimension in the direction of the frame if rectangular. Repeats width by default.
    3. Diameter:- Column diameter if circular.

4. External:- External End columns. Only where there is no cantilever at an end.

    1. Width:- Column dimension across the frame if rectangular. Repeats internal width by default.
    2. Length:- Column dimension in the direction of the frame if rectangular. Repeats internal length by default.
    3. Diameter:- Column diameter if circular. Repeats internal diameter by default.

7.2.3.3 Beams

1. Depth:- Depth of beam from top of slab. A negative depth signifies an upturned beam still measured from the top of the slab.

2. Width at slab surface:- Width of the beam at the connection to the slab.

3. Width at beam soffit:- Width of the beam at face extreme from the slab. Repeats width at face of slab by default.

7.2.3.5 Transverse Beams

1. Internal:- All columns except those at the extreme ends of the frame.

    1. Depth:- Overall depth of the transverse beams from the top of the slab.

    2. Width Top:- Width of the transverse beam at the soffit of the slab.

    3. Width Bottom:- Width of the transverse beam at the soffit of the transverse beam. Repeats width top by default.

2. Edge:- External End columns. Only where there is no cantilever at an end.

    1. Depth:- Overall depth of the transverse beams from the top of the slab.

    2. Width Top:- Width of the transverse beam at the soffit of the slab.

    3. Width Bottom:- Width of the transverse beam at the soffit of the transverse beam. Repeats width top by default.

    4. Edge Transverse beam:- The structural system selected will have set the transverse beam type depending on the system chosen. However, the edge transverse beam is often different to the internal ones so the following options are provided for the designer to over-ride the default setting for the edge transverse beams. This change can also be made later when the data file is created. See 7.2.3.5 Input->Transverse Beams for a definition of the difference in the handling of these by RAPT. The default setting will be the setting for the structural type selected.

        1. Band

        2. Beam

7.2.3.4 Drop Panels

1. Depth:- Depth of the drop panels from the top of the slab.

2. Width:- Total width of the drop panel. For external panels, this dimension is from the external edge of the concrete and so includes the external edge distance which defaults to half of the column width.

3. Length:- Total length of internal drop panels. End drop panels will use half of this length from the centre of the column.

4. At Edge Columns:- Drop panels will not be included at external end columns unless this option is checked.

Fielders KingFlor Composite Steel Formwork (Australian Materials) / Metal Decking (Other Materials)

1. If selected, metal decking will be placed automatically on the soffit of the slab for all spans. It will span to the faces of any transverse beams/bands, or to the centreline of wall supports. This option is only available for the three One Way Slab Structural Systems defined above.
   Note:- Metal Decking Sheets are supplied in specific widths. Normally the sheets will extend over the full width of the slab. RAPT cannot assume an area of sheet per unit of slab width, or a fraction of a sheet width. A whole number of sheets must be defined. To achieve this in RAPT, the Design Width for the slab must be defined as a multiple of the sheet width.
   When the sheet type and sheet thickness are defined below, RAPT will automatically update the Design Width to match the next whole number of sheets of the type selected. So, if the Design Width is 1000mm and the sheet width is 300mm, RAPT will modify the design width to 1200mm. The cursor will be moved to the Design Width data cell whenever RAPT automatically changes this value to indicate to the designer that the change has been made. All output results will then be the total for the new Design Width. The only situation in which RAPT will not adjust the Design Width to suit a whole number of sheets is if the new Design Width would be greater than the Panel Width. A warning will then be given when designing that the total sheeting width is less than the design width.

2. Sheet Type:- The List below shows all of the available Metal Decking Types defined in the Materials Properties set selected for this run. Select the type of Metal Deck to use in all spans. The Metal Decking Type can be modified in each span in the main input. The

Default Type selected will be the first type in the list.

Sheet Designation	Yield Stress	Description
RF55 - 2 Pan	550MPa	
RF55 - 3 Pan	550MPa	
KF57	550MPa	
KF40 - 2 Pan	550MPa	
KF40 - 3 Pan	550MPa	

3. Thickness:- The List below shows all of the available Sheet Thicknesses defined in the Materials Properties set selected for this run for the Metal Decking Type selected. Select the Sheet Thickness of the Metal Decking Type selected to be used in all spans. The Sheet Thickness can be modified in each span in the main input. The Default Sheet Thickness selected will be the first type in the list.

Nominal Size	Sheet Thickness	Sheet Width	Sheet Height	Sheet Area
.6mm	0.6mm	400mm	54mm	405.2mm2
.75mm	0.75mm	400mm	54mm	506.4mm2
.9mm	0.9mm	400mm	54mm	607.6mm2
1.0mm	1mm	400mm	54mm	675.2mm2

Prestress

1. Stressing Ends:-
    1. Left End
    2. Right End
    3. Both Ends
2. Balanced Load:- Either a fraction of the self weight to be balanced can be nominated to provide a starting number of tendons and tendon profiles.
3. Number of tendons:- Or a number of tendons can be specified. The tendons will be profiled to use maximum drape in each span based on the default covers in 5.3 Design Standards->Prestress for the different member types.
4. Number of Tendons (Middle Strip):- If a two way slab option has been selected, a number of tendons can also be defined for the middle strip.

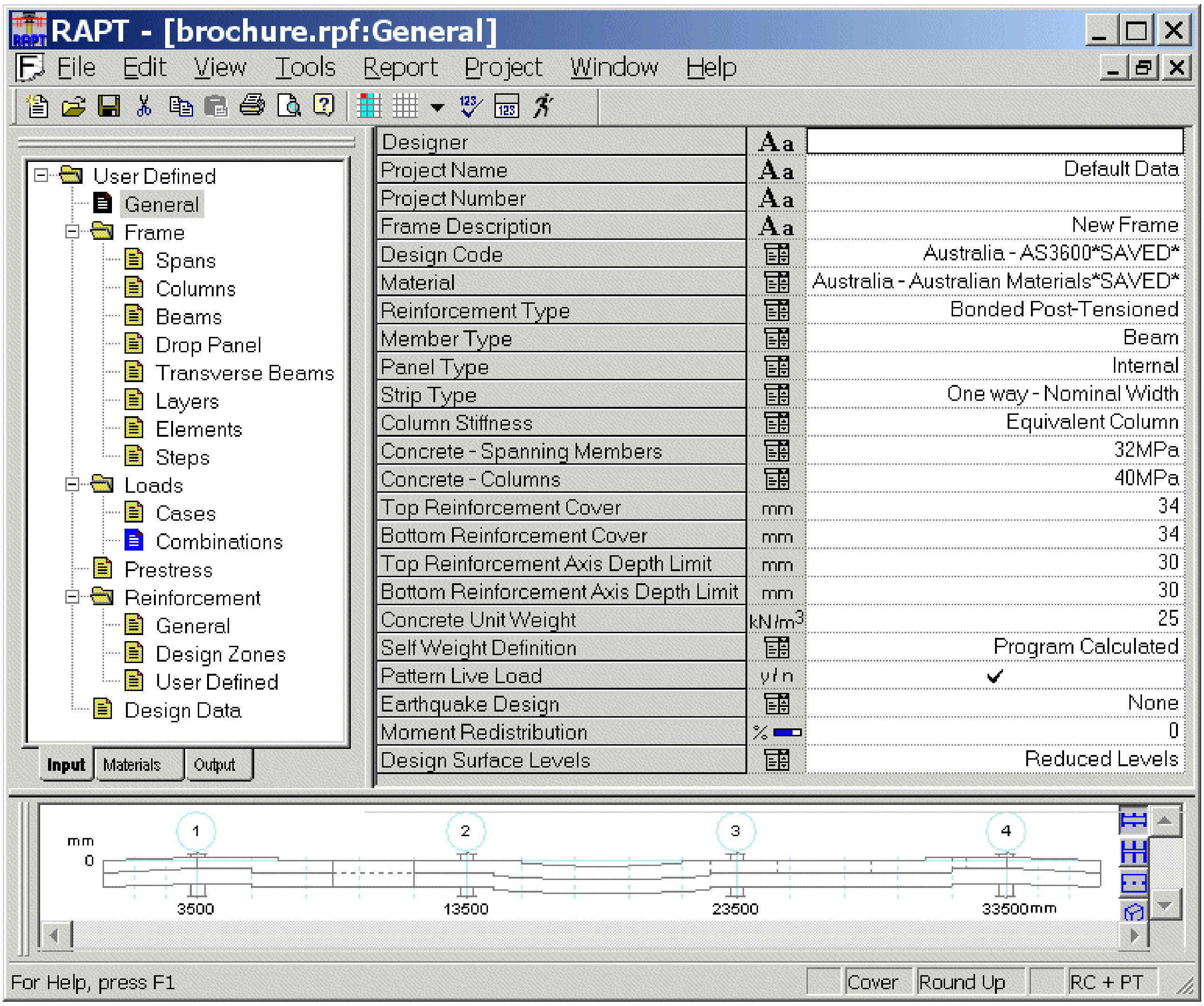
Loadings

1. Dead Load:- This will be input as the Initial Dead Load.
2. Live Load:- This will be input as the live load.

Data Checking

Some of this data is mandatory and some optional. If mandatory data is left out or data input is illogical eg a beam depth of 200mm with a slab depth of 300mm, RAPT will check the data on accepting this data with the OK button and will move the cursor focus to the data field which is in error. The user will not be able to proceed past this dialog until all of the data is acceptable.

7.2.2 General Screen



Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

Designer

The name of the designer in a 4.4.1 text field. This will default to the name in the 4.2.2 View->User Preferences->User Options->Default Designer. This is used in the output report.

Project Name

The project name in a 4.4.1 text field. This is used in the output report.

Project Number

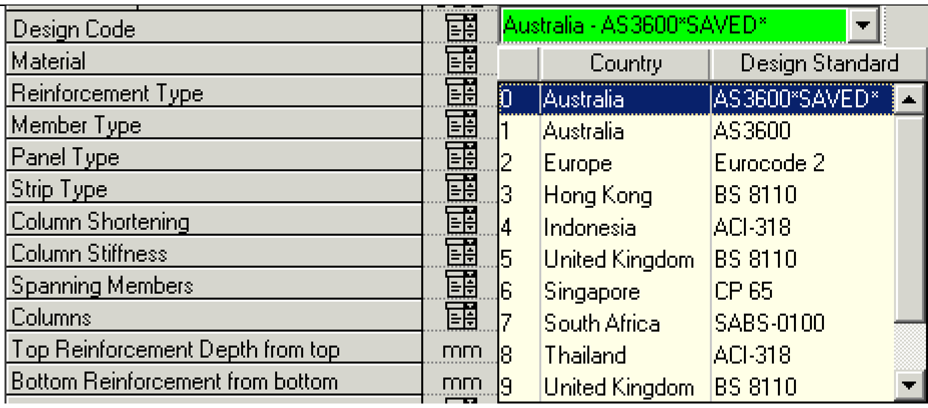
The project number in a 4.4.1 text field. This is used in the output report.

Frame Description

The project description in a 4.4.1 text field. This is used in the output report. This is also used as the default file name when saving the data for the first time.

Design Code

The designer can select the design code that the design is to be based on from the following drop down list of the available options.



RAPT can design to the rules in Six Design Standards. Users can adapt any of these Standards, creating their own 5.2 Design Codes based on the rules in the basic six. RAPT will list all current codes defined in the Design Codes Directory. The base Design Standards are

1. Australia AS3600-2009
2. Europe EUROCODE EN1992-1-1 - 2004
3. U.S.A. ACI318-2014
4. United Kingdom BS8110-97
5. South African SABS0100-1:1992
6. Singapore CP65 - 1999
7. Hong Kong Code of Practice for Structural Use of Concrete - CP2004
8. Indian Codes of Practice for Reinforced and Prestressed Concrete - IS456/IS1343

As an example of adapted Design Standards from these Base Standards, RAPT includes Singapore, UK, and Malaysian versions of Eurocode 2 in its supplied Standards list and Materials files based on these Standards as well. These Design Code options include the local Annex rules for Eurocode.

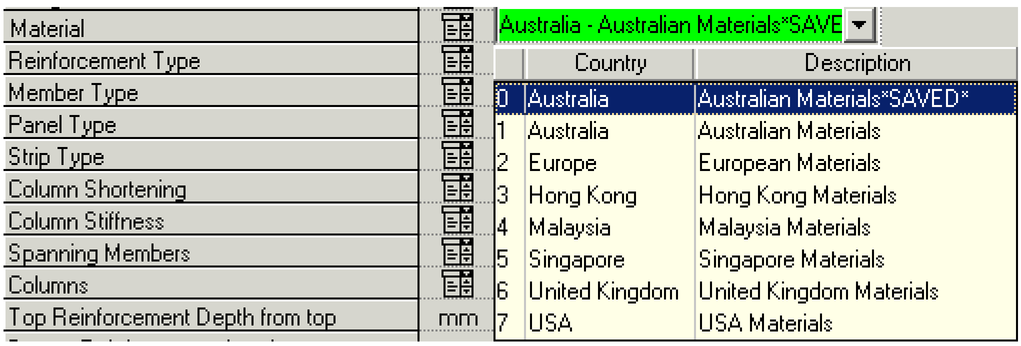
A RAPT Design Code nominates which design Standard the design is to be based on as well as a set of default settings and design parameters for that Design Code. When a Design Code is selected from the list, the settings in the data which are code dependent will automatically be changed to those for the selected design Code. The changed values will be highlight in blue and the tree folder colours will change to blue to indicate which data views have modified data.

The data from the current Design Code is saved with the run data. It is listed in the list of options as Design Code Name "SAVED" as shown above. Thus it is possible to reselect the same design code again and overwrite the Design Code settings in the data.

Some design Standards are based on Cube Strength for concrete (BS8110, CP65, SABS0100, CP2004, IS456/IS1343) and others on Cylinder strength (AS3600, ACI318, Eurocode 2). The Materials properties for those countries using the different Standards are set up based on the concrete strength type used. If a designer selects a Cube Strength based Design Standard with a Cylinder Strength based set of materials or vice versa RAPT will internally modify the concrete strengths by a factor of .8 (/.8 if the Design Standard is Cube Strength and *.8 if the Design Standard is Cylinder Strength) to adjust them to the Design Standard settings. This will show up in the output as a comment at the bottom of the output for the general screen if a conversion factor has been used. The Materials data will be shown as defined in the materials file and will not include this conversion factor. The concrete properties in the Materials Data screen will reflect the changes caused by this modification to the Concrete Strength Type.

Material

RAPT defines the materials for each country in separate 6.2.1 Materials files. The designer can select the materials file to use from a drop down list of the files available as shown below.



When a Materials file is selected as the file to be used, RAPT will check all of the materials currently used in the data and select the closest option available from the new materials file in each case. These checks will be done for concrete, prestressing materials and un-tensioned reinforcement.

The data from the current Materials Data is saved with the run data. It is listed in the list of options as Materials Name "SAVED" as shown above. Thus it is possible to reselect the same design code again and overwrite the Design Code

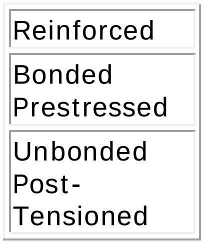
settings in the data. This materials data is accessible during the run and can be modified locally for use by this run data only. If a change is to be made to the Default Materials file, then that file must be loaded as the current file and saved. The modified materials File can then be reloaded into the Frame Data file by selecting it from the list.

The concrete properties, which are included in a countries Materials File, are by default defined for that countries Design Standard. If a different Design Standard is selected, RAPT will automatically recalculate the concrete properties to suit that new Design Standard. The designer can force RAPT to recalculate the default concrete properties for the current code in the Materials screen.

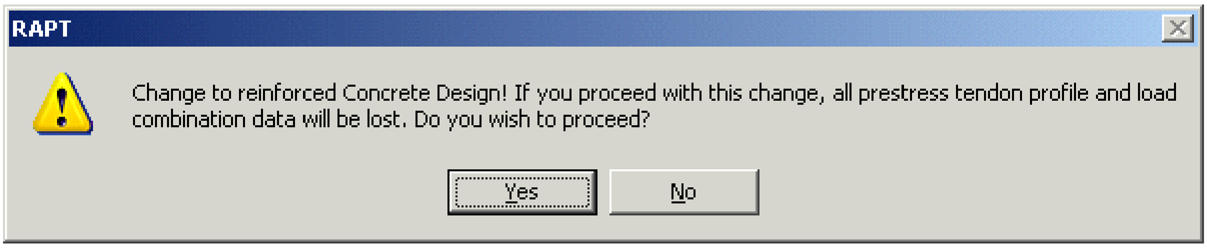
Reinforcement Type

The designer can select from a drop down list (shown below) to have RAPT design the member as

1. Reinforced Concrete
2. Bonded Prestressed:- Partial prestress design using bonded tendons. Pre-tensioned strands can also be defined in this type.
3. Un-bonded Post-tensioned:- Partial prestress design



When Reinforced Concrete is selected in a prestressed file, the following warning dialog will be presented to warn that all prestressed data will be lost if you continue (Yes). Selecting No will reverse the edit and leave the reinforcement Type as it was previously.



Member Type

The member type may be defined as

1. Slab
2. Beam

Selection of Beam does three things.

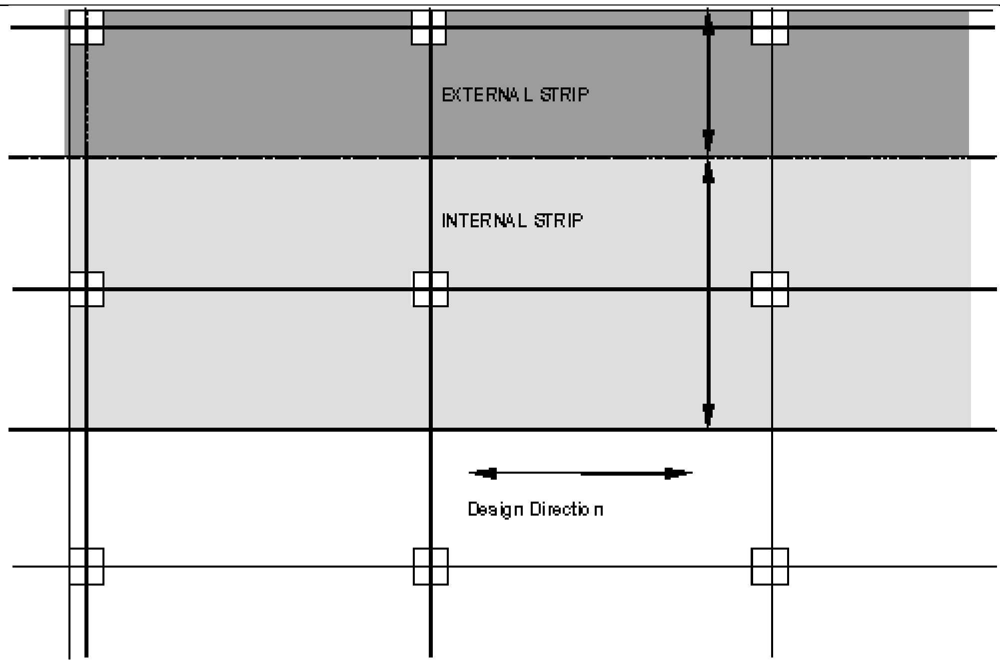
1. It makes a Data View available in the Input Tree for Beams
2. Self weight calculations and all Panel loadings are calculated based on the overall panel width, not on the effective width as is used for one way slabs. Thus the full panel load is applied to the beam.
3. The beam shear design will use the beam limits in most Design Standards for minimum shear reinforcement to decide when minimum shear reinforcement requirements can be used.

Beams do not need to be placed in every span but every span must have an effective width. In this case, leave all other dimensions as zero.

Panel Type

The panel type may be defined as

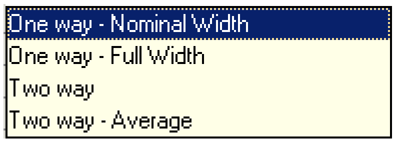
1. Internal:- Slab will extend on both sides of the column to other support lines on both sides.
2. External:- Slab will extend only to another support line on one side of the frame with the other side being a free edge.



This defines the location of the design panel in relation to the total floor plan. Using this choice, RAPT is able to default sensible information into other input screens. Values such as L2, column stiffness and punching shear are all affected by this choice. eg When an EXTERNAL PANEL is specified RAPT

1. Assumes that the columns have a torsional member on one side only, therefore affecting the Column Stiffness calculation.
2. Seeks information on external edge distances. These values are used in calculating the L2 default and in Punching shear calculations.
3. Calculates sensible default values of L2 for the defined system.
4. The outer edge of the slab, drop panels and beams are assumed to be at the free edge of the frame.

Strip Type
Defines how the bending moments and shears are to be distributed over the width of the frame being defined. The options are:-



1. One way - Nominal Width:-
    1. Slab systems:- Designs only a nominal width of the defined slab e.g. a 1000mm strip of a 8000mm panel of slab as a one way spaning slab. There is no transverse distribution of moments. Used typically for slabs spanning between beams and slabs spanning across walls. Self weight and panel loads are calculated for the nominal strip width in each span. Point loads are applied to the effective (design strip) width. RAPT adjusts the frame properties to be consistent with the design strip width. Column moments and reactions are calculated from the design strip results by factoring them by the ratio of the transverse column spacing to the effective width. No horizontal steps or tapers are allowed in the side of the slab.
    2. Beam Systems:- Will design the effective beam to support the full panel load. There is no transverse distribution of moments. Self weight and panel loads are calculated for the panel width. Point loads are applied to the effective beam.
2. One Way - Full Width:-
    1. Slab systems:- Designs the full panel width of the defined slab. The design width is the panel width at all locations. There is no transverse distribution of moments. Used typically for slabs spanning between beams and slabs spanning across walls. Self weight and panel loads are calculated for the full slab width. Point loads are applied to the full slab width. There is no limitation on the use of steps and tapers to any surface.
    2. Beam Systems:- Not applicable.
3. Two way:- The slab column strips and middle strips. The moments and shears are distributed to these strips using standard distribution factors defined in most Design Standards which are further controllable by the designer in 7.2.4.3 Loads->Lateral Distribution Factors. The designer can control the distribution of the effects at the supports and the maximum span moment points in each span. RAPT uses a parabolic distribution of the factors between these points to more closely match the distribution in a finite element analysis. Flexural Design, Reinforcement Layouts and Deflection calculations are provided for each design strip. Beam Shear calculations are only provided for the column strip.
    1. Slab systems:- Can be used for any reinforcement type.
    2. Beam Systems:- Only allowed for Reinforced Concrete systems. The effective beam is apportioned a fraction of the column strip moments and shears in each span. The effective beam is designed to carry these moments and the middle strip is designed to carry its share of the moments. Normally it is sufficient to extend the middle strip reinforcement to the face of the beam to allow the remainder of the slab in the column strip to carry the left over column strip moment. The designer should check to see if this will be adequate especially in situations where the portion of moment assigned to the effective beam is relatively small.
4. Two way - Average:- This is the ACI318 method of designing two way prestressed slabs. Even though moments and shears are distributed unevenly over the width of a slab panel, the "average" effect of this is used in the design. The method is not allowed by AS3600 and a modified version is allowed under British Code design rules where stricter limits are placed on the allowable tension stresses in the concrete in recognition of the effects of averaging of the moments compared to the real moment concentrations especially at supports. This method should not be used for partially prestressed slabs.

Percentage of load carried by Column Strip

For Two Way Prestressed Systems, RAPT asks for a percentage of load to be carried by the column strip. The default is 70%. This figure is used when doing load balancing calculations to determine numbers of tendons for each strip and is used as the default for the distribution factors in 7.2.4.3 Loads->Lateral Distribution Factors for the distribution of moments and shears between design strips. The designer may over-ride these values if desired.

Column Stiffness

RAPT users can choose to model their frame run based on

1. The Equivalent Column Approach. [See T.2.1 Section Theory T2.1] Also takes into account the infinite stiffness of the slab / beam at the column interface. This is the default and should be used for most slabs and beams in building floor systems.
2. Net Column Stiffness. The basic column stiffness. No attempt is made to allow for torsional rotations of slabs at the column or the effects at the slab/beam at the column interface.
3. Enhanced Column Stiffness. The stiffness of the columns taking into account the infinite stiffness of the slab / beam at the column interface. This option should only be used for concrete portal frames, where there is no transverse torsional member.

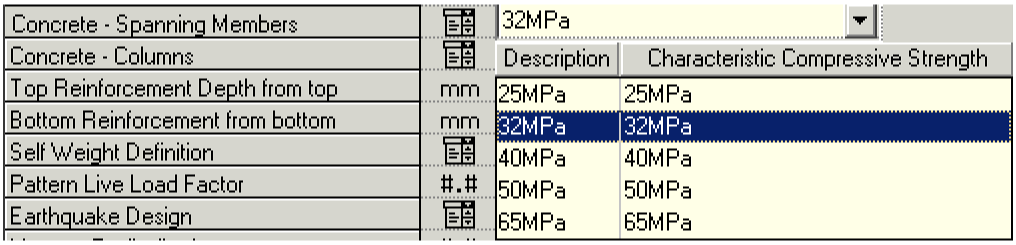
Note: The user can specify a Knife Edged support by defining column dimensions and a Column Stiffness or 0% or by having defining no column dimensions in the 7.2.3.2 Column Input Screen. RAPT calculates different critical sections for moment and shear depending on the method of input chosen. [See Section 7.5 for details]. In both cases, the vertical shortening of the column will restrained.

Concrete Type

The designer can define multiple Concrete Types in the 6.3 Materials Data->Concrete Properties to allow for variations in concrete properties in different areas of a country e.g. Brisbane/Sydney, Melbourne and General for Australian concretes and also to allow for special concrete mixes such as the Boral Envisia concretes now available in some areas in Australia. The selection from this list controls the list of concrete strengths available in the next two data items. There will always be at least one Standard Concrete Type available in each country materials set. The designer can add extra Concrete Types in the Materials data.

Concrete - Spanning Members

Concrete strength of the slabs and attached spanning members. This is selected from the list of available strengths in the Materials data for the Concrete Type selected.



Some design Standards are based on Cube Strength for concrete (BS8110, CP65, SABS0100) and others on Cylinder strength (AS3600, ACI318, Eurocode 2). The Materials properties for those countries using the different Standards are set up based on the concrete strength type used. If a designer selects a Cube Strength based Design Standard with a Cylinder Strength based set of materials or vice versa RAPT will internally modify the concrete strengths by a factor of .8 (/.8 if the Design Standard is Cube Strength and *.8 if the Design Standard is Cylinder Strength) to adjust them to the Design Standard settings. This will show up in the output as a comment at the bottom of the output for the general screen if a conversion factor has been used. The Materials data will be shown as defined in the materials file and will not include this conversion factor.

Concrete - Columns

Concrete strength of the columns. This is selected from the list of available strengths in the Materials data (see sample above).

Top Reinforcement Cover

This is the dimension to the top face of the top design reinforcement layer from the outside top face of the concrete. Should the user wish to use different covers at different points along the span it is possible to adjust this figure at each point in the 7.2.6.3 Reinforcement->Design Zones data screen. This figure is used as the default in creating the top reinforcement design zones. If this figure is modified after multiple design zones have been created, RAPT will modify the cover in the design zones whose cover was the same as the old default value with the new value. RAPT will also modify the cover to user defined bars whose original cover was the same as the original default value.

When designing cross-sections along the member, RAPT will calculate the depth to the top reinforcement to be designed based on the top cover value at that cross-section measured to the face of the bar size nominated as the Preferred Bar Size in the relevant 7.2.6.3 Reinforcement->Design Zones data screen.

Bottom Reinforcement Cover

This is the dimension to the bottom face of the bottom design reinforcement layer from the outside bottom face of the concrete. Should the user wish to use different covers at different points along the span it is possible to adjust this figure at each point in the 7.2.6.3 Reinforcement->Design Zones data screen. This figure is used as the default in creating the bottom reinforcement design zones. If this figure is modified after multiple design zones have been created, RAPT will modify the cover in the design zones whose cover was the same as the old default value with the new value. RAPT will also modify the cover to user defined bars whose original cover was the same as the original default value.

When designing cross-sections along the member, RAPT will calculate the depth to the bottom reinforcement to be designed based on the bottom cover value at that cross-section measured to the face of the bar size nominated as the Preferred Bar Size in the relevant 7.2.6.3 Reinforcement->Design Zones data screen.

Top Reinforcement Axis Depth Limit

This is the minimum depth from the top surface of the concrete to the centreline of any reinforcing or prestressing steel. In some situations, reinforcement depth is controlled by the depth to the centre of the steel, rather than the cover to the surface (eg fire on the underside of a concrete floor to some codes). RAPT will use this limit in the following ways

1. Top Reinforcement Zones - The covers shown in the input data for each zone will be calculated only from the Top Reinforcement Cover requirement above. When designing reinforcement for the top zone, RAPT will use a combination of the cover defined in reinforcement zones top at that location and this Top Reinforcement Axis Depth Limit in conjunction with the relevant bar size to determine the actual depth to the centre of the bar layer at that location. This will only be done at the time of calculation of section capacities and reinforcement detailing.
2. User Defined Top Reinforcement - RAPT will check that all bar layers conform with the axis distance limit and will give an Input warning if a bar violates this limit at any point along it's length. This will not stop program execution.
3. Prestressing Tendons - RAPT will check that all prestressing tendons conform with the axis distance limit (for prestressing tendons the limit is increased by 15mm in carrying out this check) and will give an Input warning if a tendon violates this limit at any point along it's length. This will not stop program execution.

In fire situations, in general the top reinforcement is not affected by fire and this limit can be ignored. If the default value is causing warnings when it should not be considered, then the designer should reduce this default value to a value that no longer causes warnings to occur.

In some situations, top reinforcement can be affected by fire e.g. an edge beam with no flange overhang. In these situations the designer should nominate a value consistent with the fire rating period required for the member being designed.

Bottom Reinforcement Axis Depth Limit

This is the minimum depth from the bottom surface of the concrete to the centreline of any reinforcing or prestressing steel. In some situations, reinforcement depth is controlled by the depth to the centre of the steel, rather than the cover to the surface (eg fire on the underside of a concrete floor to some codes). RAPT will use this limit in the following ways

1. Bottom Reinforcement Zones - The covers shown in the input data for each zone will be calculated only from the Bottom Reinforcement Cover requirement above. When designing reinforcement for the bottom zone, RAPT will use a combination of the cover defined in reinforcement zones bottom at that location and this Bottom Reinforcement Axis Depth Limit in conjunction with the relevant bar size to determine the actual depth to the centre of the bar layer at that location. This will only be done at the time of calculation of section capacities and reinforcement detailing.
2. User Defined Bottom and General Reinforcement - RAPT will check that all bar layers conform with the axis distance limit and will give an Input warning if a bar violates this limit at any point along it's length. This will not stop program execution.
3. Prestressing Tendons - RAPT will check that all prestressing tendons conform with the axis distance limit (for prestressing tendons the limit is increased by 15mm in carrying out this check) and will give an Input warning if a tendon violates this limit at any point along it's length. This will not stop program execution.

The designer should nominate a value consistent with the fire rating period required for the member being designed.

Concrete Unit Weight

This defines the unit weight of the floor slab or concrete member to be used in the calculation of the self weight load case. This weight should allow for the concrete plus any reinforcement in it. The Unit Weight of concrete by itself can vary significantly but normal weight concrete is normally taken to be in the order of 24KN/m3. The reinforcement weight will vary according to member type and the type of design. The reinforcement weight is approximately .6KN/m3 for every 1% by volume of steel reinforcement. For a slab with 120kg/m3 of reinforcement, this would equate to about .5KN/m3. For a beam with 350kg/m3 of reinforcement is would equate to approximately 2KN/m3. For floor slabs including beams an overall average would need to be calculated. Prestressed beams and slabs would have a much lower component of their unit weight from steel as the amount of steel used would be in the order of 1/3 of that used in reinforced concrete members. In earlier versions of RAPT where it was not possible to define this weight, concrete density was used in the calculation of self weight.

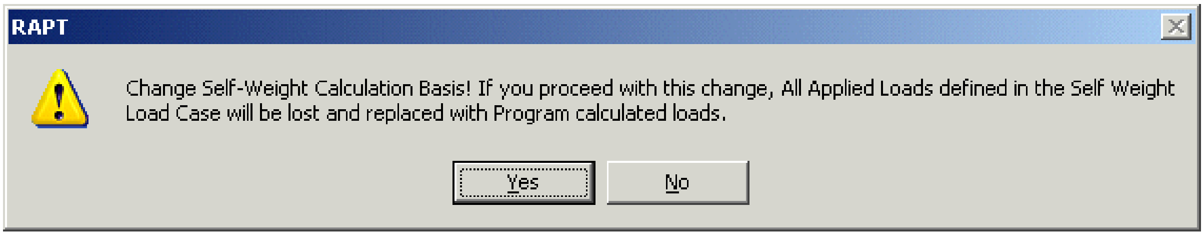
RAPT will calculate a default value equivalent to the concrete density plus 1KN/m3. If the concrete density is modified in the materials data, this data item will be adjusted to suit the new concrete density.

Self Weight Definition

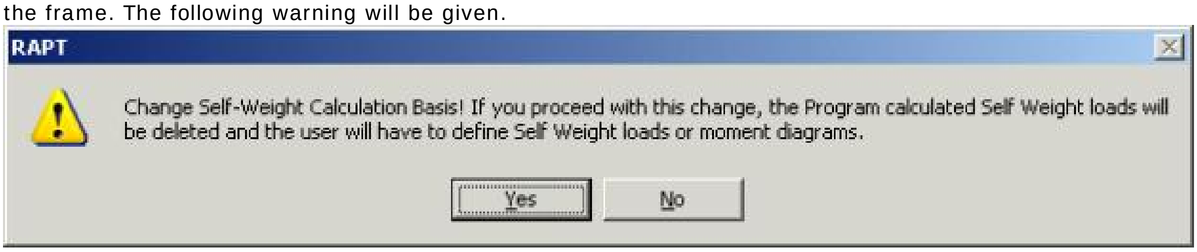
By default RAPT will calculate the self weight of the concrete frame that has been defined and place it in the self weight load case. These values are not editable but extra loads or either sign can be added by the designer. Here, RAPT gives the designer the option to ask RAPT not to calculate the self weight of the frame. It is then the designers responsibility to enter in the loads or a bending moment/shear diagram for the self-weight case and to modify this whenever changes are made to the input that would require loads to be recalculated. RAPT will adjust load positions and end locations when spans are added or span lengths change but cannot be expected to make other modifications to loads that may result from shape changes or if bending moment diagrams have been defined.

The options are

1. Program Calculated:- All loads in the self weight case will be deleted. RAPT will recalculate the frame self



2. User defined:- When this is selected, RAPT will delete all program calculated self weight loads. Other user defined self weight loads will be retained. The designer can then add extra loads to define the self weight of the frame. The following warning will be given.



Pattern Live Load
RAPT will automatically create the load cases required to satisfy the requirements of different design codes for the default patterning of Live Load on alternate and adjacent spans. The designer can select, in 7.2.7.5 Design data->Pattern Load, to use pattern live load cases in different areas of design. The default is to use patterning of live load in all areas if it is selected in the General Screen.

Design Standards define the percentage of load that should be applied for patterning of the live load. Also, the method used in patterning live loads varies from Standard to Standard. For more information on patterning of loads for various Design Standards see the relevant standard. RAPT follows the method defined in each standard in terms of load pattern and factors exactly as defined in that design code.

Only the load case named Live Load will be patterned. The combination factors used for this case are used for the envelope of actions created for the patterned live load. RAPT will create two live load pattern envelopes, one based on the envelope of moments at each design location with co-existing shears and one based on the shear envelope at each point with co-existing moments. For BS8110 based design codes, RAPT will also use different Dead Load Combination Factors for loaded and unloaded spans.

RAPT will automatically reduce continuous live loads to span based loads to achieve load patterning. It is not necessary for the designer to do this manually.

Designers should be aware that this patterning of live load only covers the default live load patterns defined in design standards, consisting of different combinations of loaded and unloaded spans. All design standards also require that designers account for any load pattern that can be expected to occur for the specific building that they are designing. The designer can create load cases and load combinations to account for any load pattern they wish to define, either as Alternate Live Loads which will be included automatically in any load combination that includes Live Loads or as Other Load Type for which the designer will have to create load combinations for any area of design that is to include that load pattern.

Earthquake Design
The designer can request that earthquake design rules be applied in designing and detailing for flexure and shear. The options are

1. None
2. Moderate Risk
3. High Risk

When options 2 or 3 are selected, RAPT will copy the 5.5 Design Standard->Earthquake Design default data into the 7.2.7.4 Design Data->Earthquake Design, replacing any settings that had been made previously.

Moment Redistribution
The designer can define a % of moment to be redistributed. RAPT will then redistribute the ultimate moment envelope and the associated shear envelope for Ultimate strength calculations. Serviceability load combinations remain unaffected.

Where the Ultimate moment at a design point is of a different sign to the service moment, RAPT will apply a factored Ultimate Moment of the same sign as the service moment equal to 1.2 x $M_{service}$ to ensure that there is sufficient strength on any face that will be tensile through any of the loading stages of the member.

If Moment Redistribution is requested, RAPT will automatically apply all limitations to the design for ductility as defined in the different design codes. It will over-ride any user defined setting for depth of neutral axis limit in the 7.2.7.1 Design Data in doing this unless the defined value is less than that calculated from the Design Standard rules.

Designers should be aware that the use of large amounts of redistribution of the moments in a member introduces severe limitations to the ductility requirements of a member due to the reliance in the strength calculations on increased rotations at plastic hinges. It also could introduce a requirement for extra reinforcement under service load

conditions to control crack widths. The resulting design could be less economical than a design without or with lesser redistribution of moments.

RAPT does not redistribute moments at end columns. To redistribute moment from an end column, a user should use the option to modify the I_{zz} of a column in the F3 Columns screen.

RAPT will only allow redistribution of moments for 7.2.3.2 Braced Frames. RAPT will limit the % redistribution defined in the input to the maximum allowed by the relevant design standard. It will allow both positive (reducing support moments) and negative (increasing support moments) values of the redistribution %.

The maximum limits are

1. AS3600:- 30% for Class N ductility reinforcement and 0% for Class L ductility reinforcement
2. ACI-318:- 15%
3. Eurocode 2:- 30% for class B or C reinforcement and 20% for class A reinforcement (Low Ductility).
4. BS8110, CP 65, IS456/IS1343 and SABS0100: 30% for reinforced concrete members and 20% for prestressed members.
5. Hong Kong CP2004: If concrete strength less than 70MPa, 30% for reinforced concrete members and 20% for prestressed members. If concrete strength is greater than 70MPa, no redistribution is allowed.

RAPT now uses two different redistribution methodologies depending on the load types defined

- Basic redistribution (old method pre version 6.3) The overall design envelope is calculated and then the whole envelope is redistributed by the requested percentage. This is now only done for cases where Moment Envelopes are defined. This includes Moving Load cases.

- Complex Redistribution is now done for all other load situations. In Complex Redistribution, each individual load combination (including each individual LL pattern load combination and Alternate LL combinations) is redistributed in the direction indicated by the designer and by a maximum of the amount requested but not by an amount that will result in an increase in the maximum moment causing tension on the other face of the member. To explain the procedure, For a redistribution from Support Hinges to Span,

- The un-redistributed Envelope is calculated. The maximum amount of redistribution possible is then calculated from the peak moments at the supports in each span, except at end columns where no redistribution is allowed.
- The individual load combinations are then recalculated and each combination is compared to the overall envelope. The redistribution is then applied to this combination but only to the extent that it reduces the critical Span Moment to the maximum span moment of the initial envelope of all of the combinations. So the Maximum Span Moment is never increased.

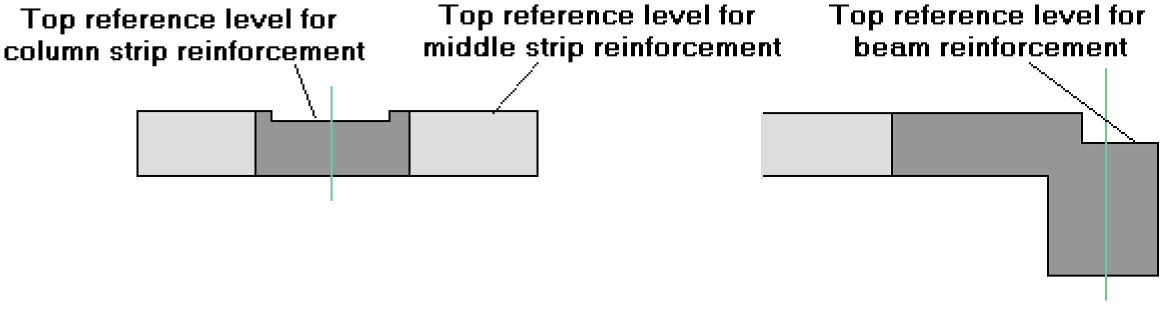
The overall effect is to compress the overall envelope towards the maximum moment causing tension on the other face, rather than shift the overall envelope. For +ve redistribution, the -ve moments are reduced towards the peak +ve moments and vice versa for -ve redistribution.

For negative redistribution, the same approach is taken but the redistribution is based on the maximum span moment. In this case, as increasing the support moments will increase the moment transfer to the columns, the redistribution is limited so that the moment transfer to the columns is not increased. This will normally severely limit the amount of -ve redistribution that is possible, especially in end spans.

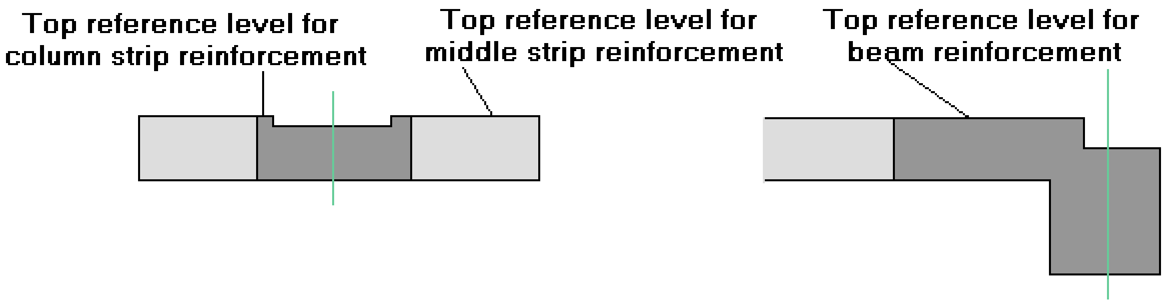
Design Surface Levels

The designer can request that RAPT use one of the following methods to define the concrete surface level from which RAPT will measure cover in the positioning of reinforcement and prestressing tendons in a member.

1. Reduced Surfaces - Will use the minimum depth of the the relevant surface (top or bottom) within 50mm either side of the centreline datum of the frame. This will be the extreme surface level in most cases. In situations where void elements have been defined to remove some of the top or bottom surface, this may result in RAPT selecting a reference level different to that required by the designer. Some examples of the effects of this selection are shown below.



2. Extreme Surfaces - Always selects the extreme concrete surface over the full width of the design strip on the relevant face.



RAPT will set the Design Surface Level as Extreme Surfaces until Concrete Elements are added to the input. When these are added RAPT will automatically change the setting to Reduced Surfaces to allow for the possibility that a void element will cause a logical reduction in the design surface level. No matter which option is selected for complicated frames, it may be necessary for the designer to modify the defined tendon covers or reinforcement design zone depths to ensure that they are positioned at the required depth in the section for a particular design example.

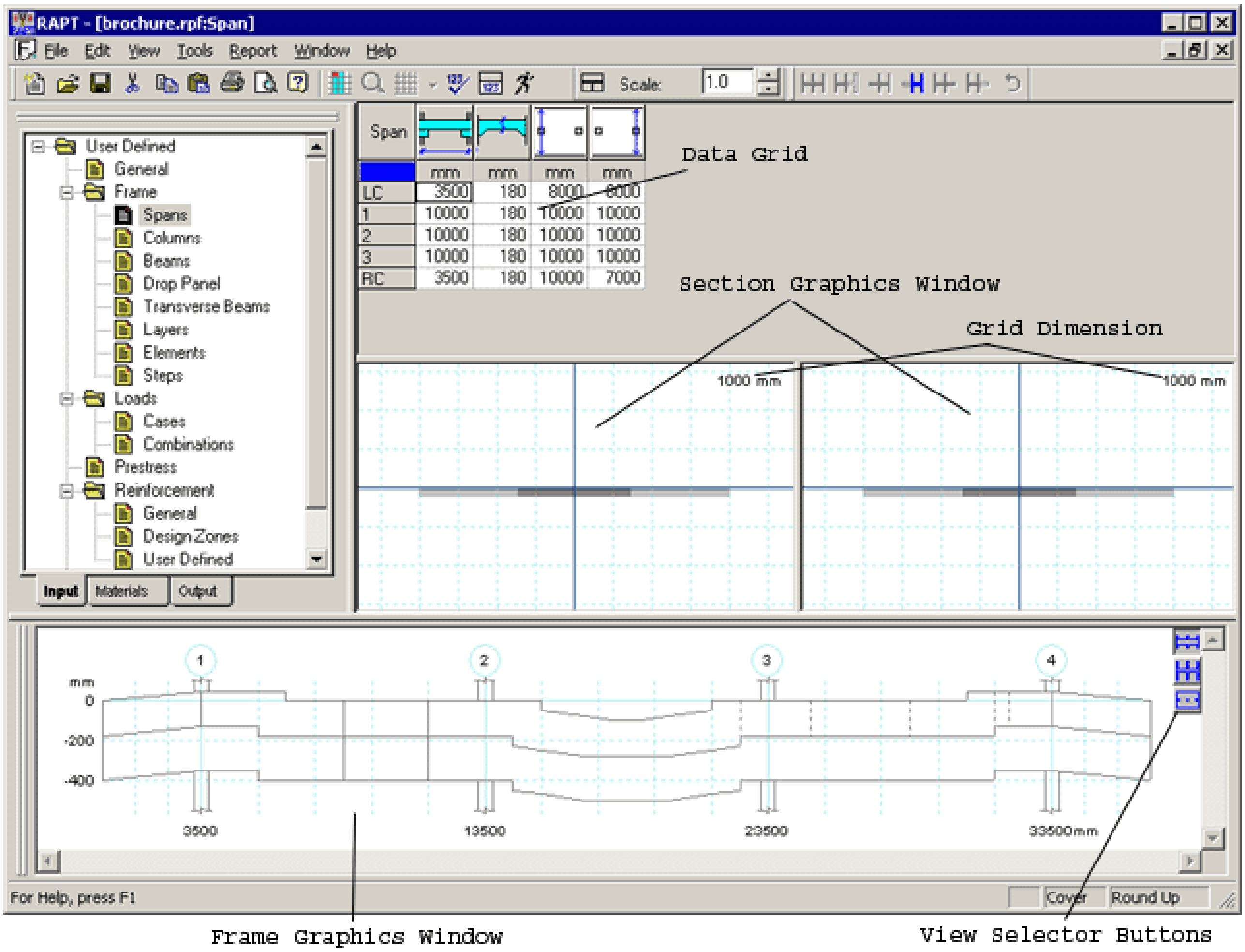
Eurocode Ultimate Load Combination Basis
Eurocode EN1990:2002-2005 in Table A2.4(B) defines two options for the Design Values of Actions for Strength. These are defined as

- Eqn 6.10 - Full factor is used on Dead Loads and Full factor used on the First Live Load and Combination Factor on all other Live Loads
- Eqns 6.10a - Full factor is used on Dead Loads with Combination factor used on all Live Loads
and 6.10b - Reduced factor used on Dead Loads with Full factor used on the First Live Load and Combination Factor on all other Live Loads

The standard makes no recommendations as to which set should be used and suggests that there will be guidance in the National Annex for each country.

This option is only available for Design Standards based on Eurocode 1992-1-1.

7.2.3 Frame Shape Screen Layout

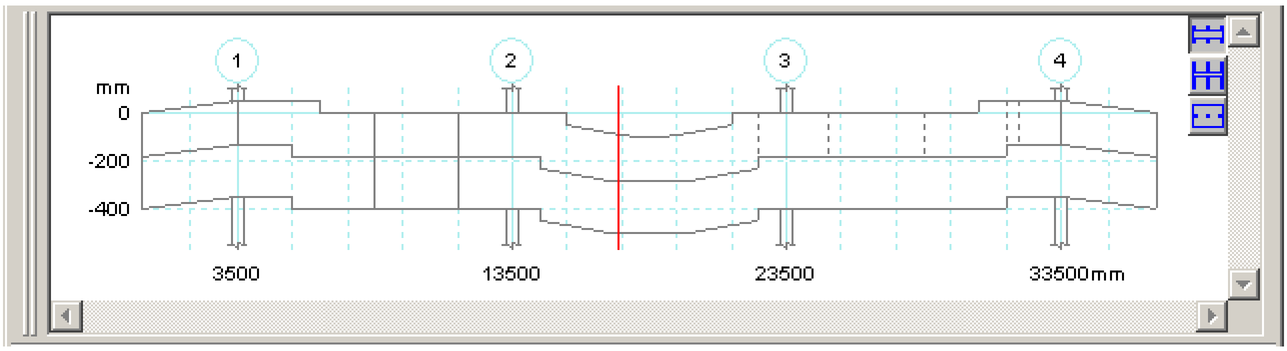


The general layout of the windows in the concrete frame input is as shown above. The data grid may be a single grid as shown above or Control/Child grid combination as is used for the 7.2.3.7 Steps input. Refer to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT. Two separate sets of graphics windows are provided to give the user various views of the concrete shape. These are described below.

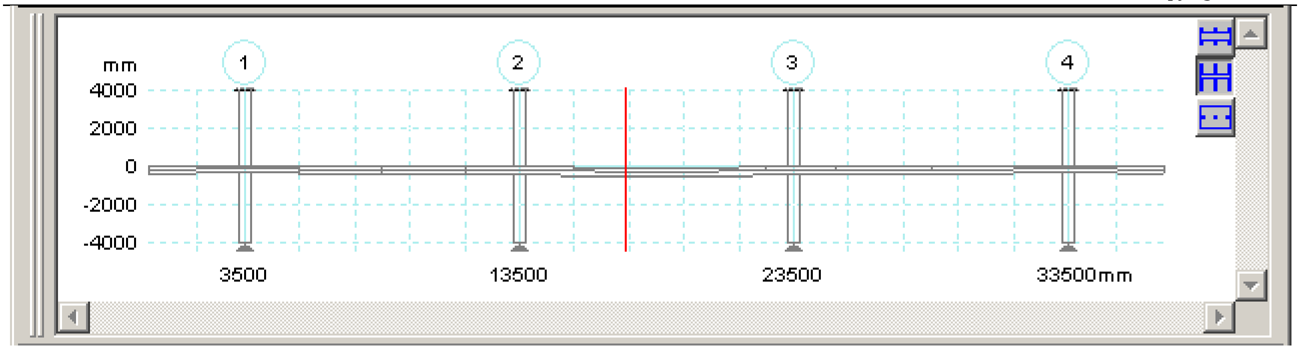
Frame Graphics Window

This window shows the graphic view of the full concrete frame. Three separate views of the frame are available controlled by the View Selector Buttons. These are

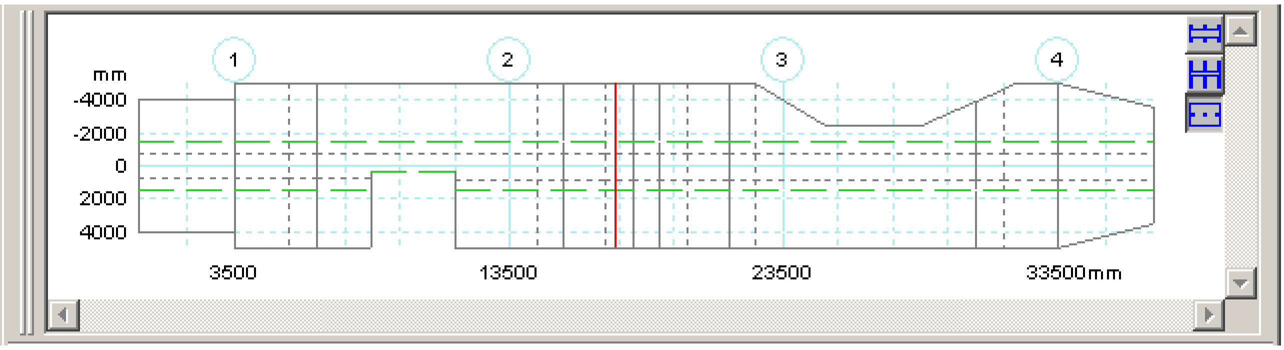
Elevation View:- Shows an expanded elevation view of the concrete member with only the connecting part of the columns. The frame is scaled to fit the view so the vertical to horizontal scale is distorted. The scales are shown in both directions. The elevation is shown viewed from the bottom of the plan view.



Full Elevation View:- Shows the full elevation of the concrete frame including the full height of the columns and the column end conditions. The frame is scaled to fit the view so the vertical to horizontal scale is distorted. The elevation is shown viewed from the bottom of the plan view.



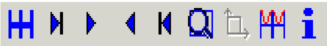
Plan View:- Shows the top plan view of the concrete panel. Columns and column capitals are not shown. The frame is scaled to fit the view so the vertical to horizontal scale is distorted.



FRAME GRAPHICS WINDOW TOOLBARS

When a location is selected in the graphics by clicking at a point or moving the cursor using the toolbar buttons, the location of the cursor from the left end of the frame will be shown in this toolbar data cell.

Alternatively, the user can move the cursor to a new position by entering the location here and pressing enter. The location can be defined as a value from the left end of the frame e.g. 10000 or as a location from a column e.g. 2;10000.



Zoom (Ctrl + Z). This button will toggle between full screen mode and span zoom mode for the graphics in a window. In span zoom mode, the current span will be shown with the half span either side (if a cantilever is the previous or next span, the full cantilever will show) scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window.

 Move to next item (Ctrl + Right Arrow). If in Full Screen Mode, changes to Span Zoom Mode and moves the next span to the centre if the window. In Span Zoom mode, the next span will move to the centre of the Window.

 Move to next point (Shift + Right Arrow). Move to the next change in section location in the frame. In Span Zoom mode, the span containing the next change in section point will move to the centre of the Window and the cursor will move to that point.

 Move to previous point (Shift + Left Arrow). Move to the previous change in section location in the frame. In Span Zoom mode, the span containing the previous change in section point will move to the centre of the Window and the cursor will move to that point.

 Move to previous item (span) (Ctrl + Left Arrow). If in Full Screen Mode, changes to Span Zoom Mode and moves the span to the left of the current span to the centre if the window. In Span Zoom mode, the span to the left of the current span will move to the centre of the Window.

 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

Then the user can then move around the graph using the Scroll Bars or the movement keys. and buttons will still move the cursor to the next and previous points. If the point is outside the viewable area it, the area of graph shown will adjust automatically to position the requested point near the left of the View Window.

Clicking this button again or on the or buttons will return the Window to Span Zoom Mode.

Clicking will change the mode to Span Zoom Mode in the span in which the cursor is positioned in the Select Zoom mode.

Show/Hide information dialog:- Not available in frame graphics windows.

Section Graphics Windows

The section graphics windows serve two functions

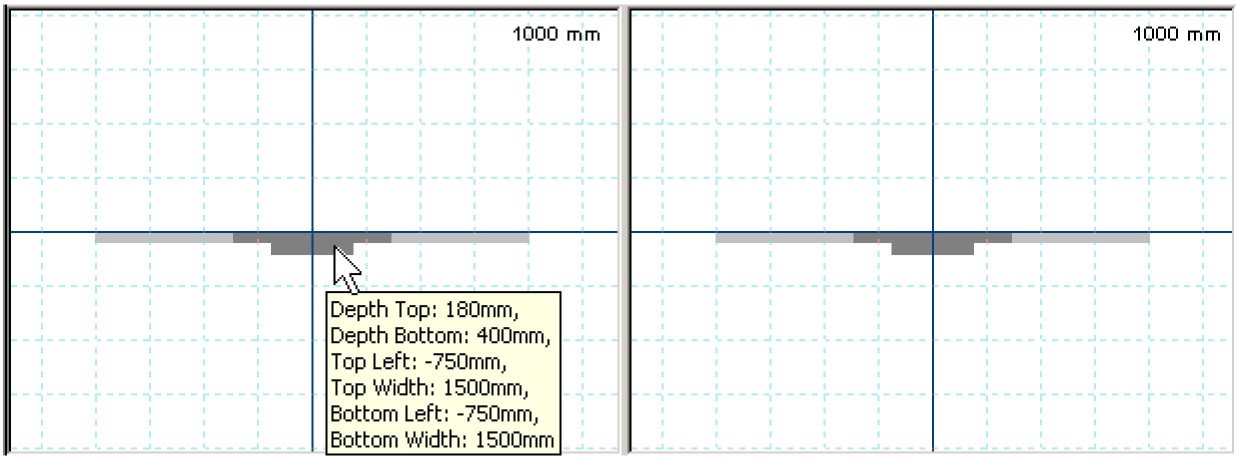
1. Focus in Data Windows:- The section graphics windows show representations of the data in the current data row in the Data Grid. See the discussion on what is shown in each case in the help section for each section of concrete frame input.
2. Focus in Frame Graphics Window:- Shows the complete frame cross section shape immediately each side of the cursor location. The cross-sections are shown as viewing from the right end of the frame. In most cases there is graphical interaction between graphics in this window and the data. See the discussion on what interaction is available in each case in the help section for each section of concrete frame input.

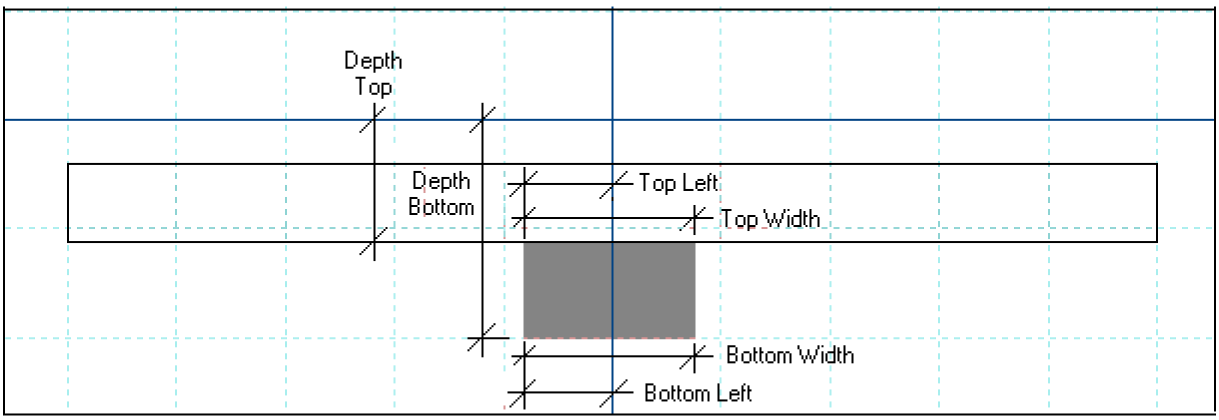
The section graphics windows show concrete sections in two equal sized windows to a consistent vertical/horizontal scale. The grid scale is printed in the top right corner of each section window and by default is equal in both directions and the scale factor is 1. This grid dimension will always apply for the direction which controls the scale in the window. For a wide flattish shape in the window the horizontal direction will control the scale and the horizontal grid dimension will always be equal to the scale dimension. For tall thin shapes, the vertical direction will normally control and the vertical grid dimension will always be equal to the scale dimension.

The centre splitter can be used to adjust the relative sizes of the two section windows.

The vertical axis line is the column/support centreline while the horizontal axis line is the zero depth datum.

If the mouse cursor is held steady over a and individual element shape in the section windows (do not click), an information window which defines the dimensions of that shape will appear as shown below. The cursor can be moved to each element comprising the overall shape to view the information for each element. Depths are positive downwards and widths are positive to the right. The information given is shown graphically below.

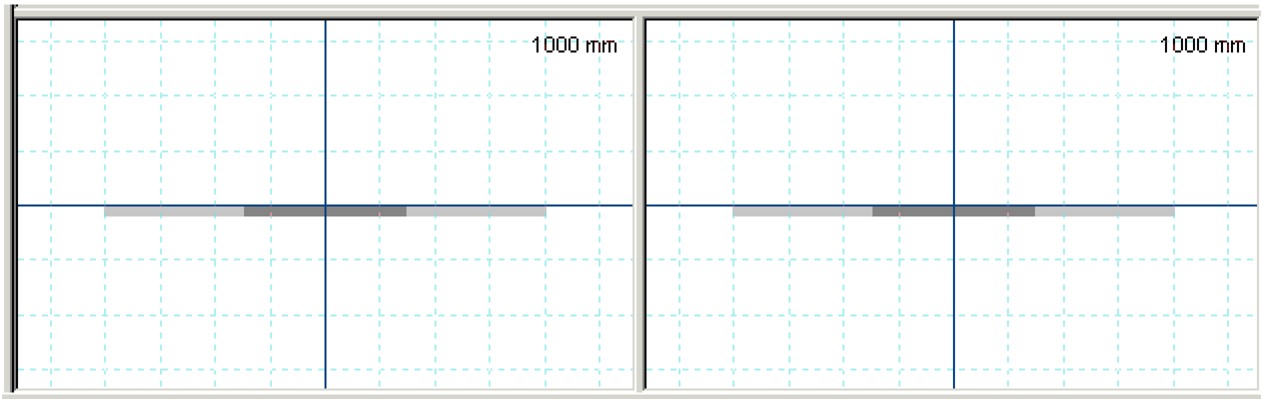


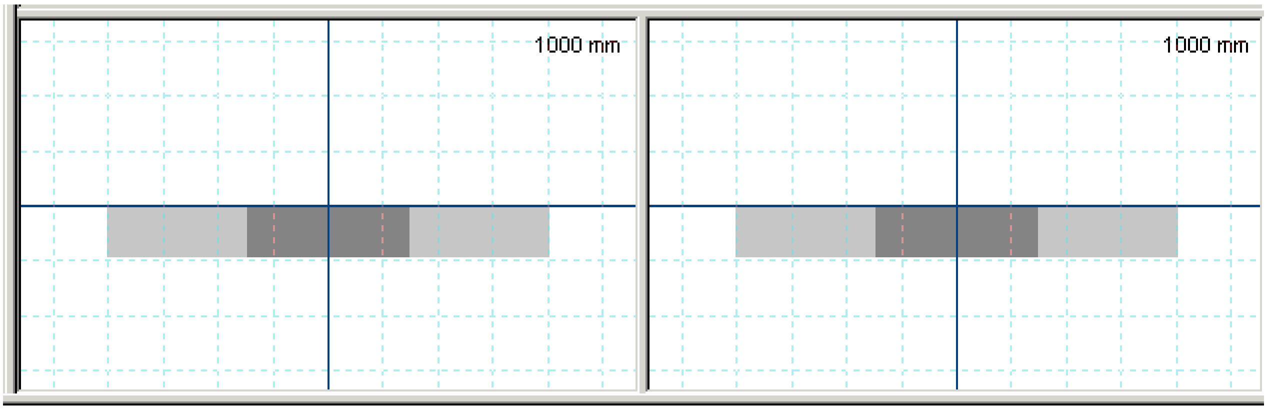


Section Graphics Window Toolbar

Reset Splitter and Scale:- Resets the window splitter location to divide the section windows equally within their space and resets the scale factor to 1

Set Scale Factor:- Allows the designer to set a scale factor that controls the ratio of the vertical/horizontal scale. A value of 2 will result in a distorted scale of 4:1 with the grid dimension in the non-controlling direction being divided by 4. The first diagram below shows the slab with a scale factor of 1 with the grids 1000mm in both directions. The second diagram below shows the same slab with a scale factor of 4. The horizontal grids are still 1000mm but the vertical grids are 1000 / 4 = 250mm each.





7.2.3.1 Span Data

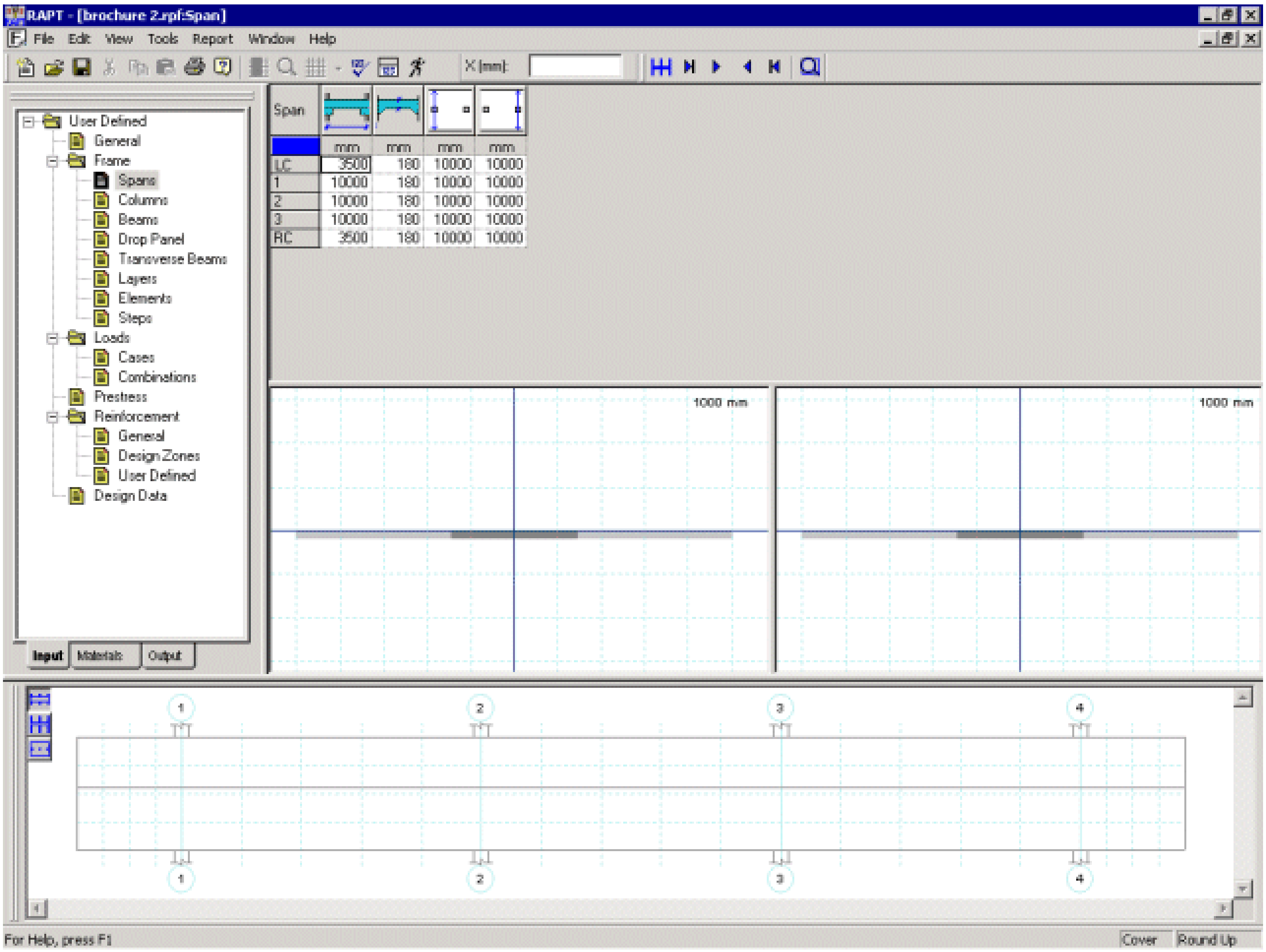


Figure 7.2.3.1 Span Data Screen

Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

This Input Screen is used to define general structural span and slab shapes. For more complicated cross sections, users should first enter their basic data in this screen and then refine the structure using Layers (7.2.3.6), Steps (7.2.3.7) and Elements (7.2.3.8).

The screen is set up with the span numbers down the left hand side and zero's for all data required. A left cantilever is identified by the term LC and a right cantilever by RC. When no cantilever exists, RAPT replaces the cantilever names with LE (left end) and RE (right end). The user can input a length in these fields which defines the location of the left and right slab ends from the respective column centre lines. This value can be negative and the slab can edge can be placed as far into the adjoining span as the front face of the column. The concrete shape in these end sections is a continuation of the shape in the adjoining spans. If ends are defined, tendons can extend into them and they are included in the analysis as cantilevers. No design is carried out in these end sections. The location of the ends affects the shear perimeter in the punching shear calculations.

Graphic Interaction

Clicking the left mouse button in a span in the graphics will automatically move the current row in the data to the row for that span. The section views will show the cross-section shape either side of the selected point.

Select a row in the data and the section graphics will show the data for that row in section. See below.

Section Views

When the program focus is in the spans data view, the Section Graphics Windows will show the slab shape as it is defined in this view, at the left (left panel) and right (right panel) ends of the span in which the cursor is positioned. Nothing is shown for Left Ends and Right Ends.

Ctrl + D, Ctrl + R

Refer to 4.4.4 Cell Repeating for a general discussion on repeating data automatically in other data cells. A few specific applications of the key apply to this screen when a range of repeat data cells is not selected. These are listed below

(i) Span Lengths:- This column is made up of two fields. These are

a) spans and

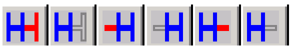
b) cantilevers/slab overhangs. (LC, RC, 0 and Number of spans + 1)

When the cursor is on a span number (excluding cantilevers and slab overhangs) the highlighted figure will be defaulted to all other blank spans, making all the spans the same length. No information will be placed in the cantilever/span overhang fields. Likewise when the cursor is in the cantilever/slab overhang field, key combination will default the highlighted figure to the other blank cantilever/slab overhang field.

(ii) Panel Width Left:- When a value is placed in the current cell and the repeat key combination is pressed, the value will be repeated in all zero cells in both the left and right panel width columns.

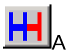
(iii) Panel Width Right, External Edge Distance Right, Column Strip Width Right:- When a repeat key combination is used on a current cell with a value of zero in it, the data will be copied from the similar data column for the left end of the span.

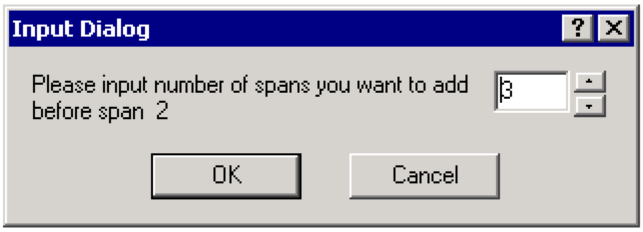
Spans Toolbar



The special Spans Toolbar allows the user control over adding and deleting spans and cantilevers. Only the toolbar buttons available at any time will be active for use. The remainder will be shown in background colour.

The buttons available are

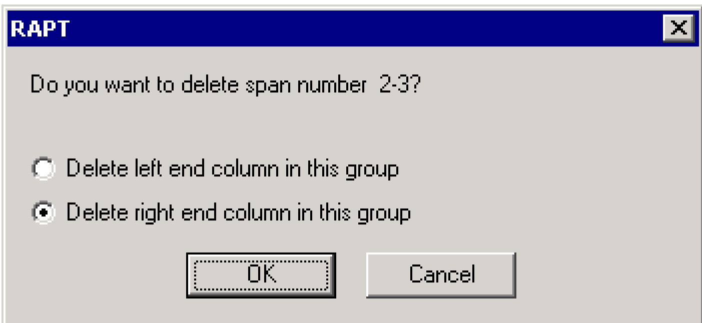
Add Spans: Adds spans at the current cursor location. All spans at and to the right of this span will be moved to the right. This button will only be available when the cursor is in a span or right cantilever/end. It will never be available at the left cantilever/end. The user is presented with the following dialog box to define the number of spans to be added.



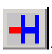
When spans are added, a default set of data will be inserted by RAPT based on the joining spans. All data associated with spans will be updated to suit this new data. This will include prestress, loads, reinforcement, steps and elements that are affected. Wherever data has been added/modified by RAPT the cell background colours will be modified to indicate that changes have been made (refer Section 4.4.3). The Tree Control will indicate views that have changes made with similar colour changes. The user should then go through all the screens where changes have been made to check and modify the data as appropriate.

Delete Spans: Deletes selected spans. To delete a group of spans, the user must select a group of whole data rows. The selection does not have to be continuous. This button will only be available when the cursor is in a span or when a group of spans has been selected. It cannot be used to convert cantilevers to ends.

The user is presented with the following Dialog nominating which spans will be deleted and asking which column to delete.

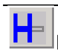


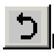
When spans are deleted, RAPT will adjust other affected data to suit the changed span arrangement. The main effects will be in prestress, loads, reinforcement, steps and elements. Any step, etc that is within a deleted span will be deleted. Loads prestress tendons and reinforcing bars that extend outside the deleted spans will be modified to suit and the cell background colours will be modified to indicate that changes have been made (refer Section 4.4.3).

Add Left Cantilever: Adds a cantilever at the left end. Effects on other data are as for adding spans.

Remove Left Cantilever: Removes a cantilever at the left end and replaces it with a Left End. Effects on other data are as for deleting spans.

Add Right Cantilever: Adds a cantilever at the right end. Effects on other data are as for adding spans

Remove Right Cantilever: Removes a cantilever at the right end and replaces it with a Right End. Effects on other data are as for deleting spans.

Recalculate Strip Width: Recalculates the column and middle strip widths, for two way slabs based on strip design methods, to the code calculated values. If no span selection is made, the values for all spans will be recalculated. If a span or group of spans is selected, the widths for those spans only will be recalculated.

Data Definition

Figure 7.2.3.1 shows all of the data columns available in this input screen. The data required in this screen varies depending on the structure type selected in the General Screen. Only those columns requiring data will be available for input for each case. Fig 7.2.3.2a -7.2.3.2c show the different data sets for different structure types. 7.2.3.2d shows the extra data items that are added for External Panels.

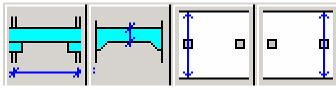


Figure 7.2.3.2a Span Data Screen Headers - 1-way slab full width, 1-way beam system

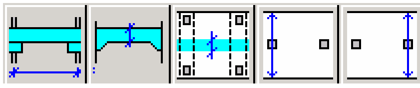


Figure 7.2.3.2b Span Data Screen Header s- 1-way slab Nominal Width

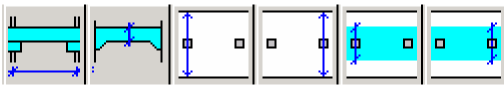


Figure 7.2.3.2c Span Data Screen Headers - 2-way, 2-way beam system

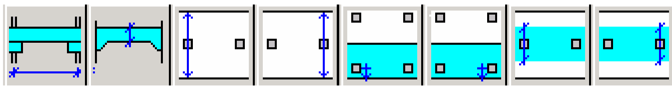
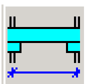
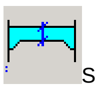


Figure 7.2.3.2d Span Data Screen Headers - 2-way, 2-way beam system - External Panel

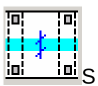
Span Length

The span length is measured from centre to centre of columns. Cantilever lengths are measured from the cantilever edge to the column centreline and must extend past the end of the columns, otherwise a Left/Right End should be selected. In end overhangs (i) The concrete shape in these end sections is a continuation of the shape in the adjoining spans. If the user wants to vary the shape a cantilever must be nominated or steps, etc can be added in the end overhangs. (ii) The length can be negative allowing the slab end to be placed at any point up to the inside face of the column.

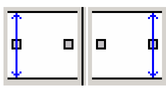
Slab Depth

The underside of the slab may step at the centre line of each column or at the faces of a transverse beam or band. If it varies either side of a longitudinal beam then the user should calculate an average value or users can use the Elements Input (7.2.3.8) to model the step correctly. Slab depth cannot equal 0 in any span. If the user wishes to run a beam without any slab, the user should input a nominal slab depth and restrict the one way width to the beam width by overriding the default figure.

The top of the slab as defined in this Input Screen is the DATUM for vertical measurements.

Slab Design Width

For a One Way Slab- Nominal Width a different slab design width may be input in each span. The Slab Design Width is used for all self weight and design calculations. In this way the self-weight of the slab need not be input manually by the user unless the load contributing area is not able to be defined in the RAPT input. All output results except Punching Shear will be based on this slab width.

Panel Width Left/Right End of Span

Define the transverse dimension of the slab at each column centre-line (left column and right column) in each span (see Figure 7.2.3.3 and Figure 7.2.3.4). This allows the user to input a tapered slab (in plan) or steps in the slab width at the column centre-lines. These dimensions are not used to calculate the torsional beam stiffness for the equivalent column calculation (see Transverse Column Spacing in Columns data).

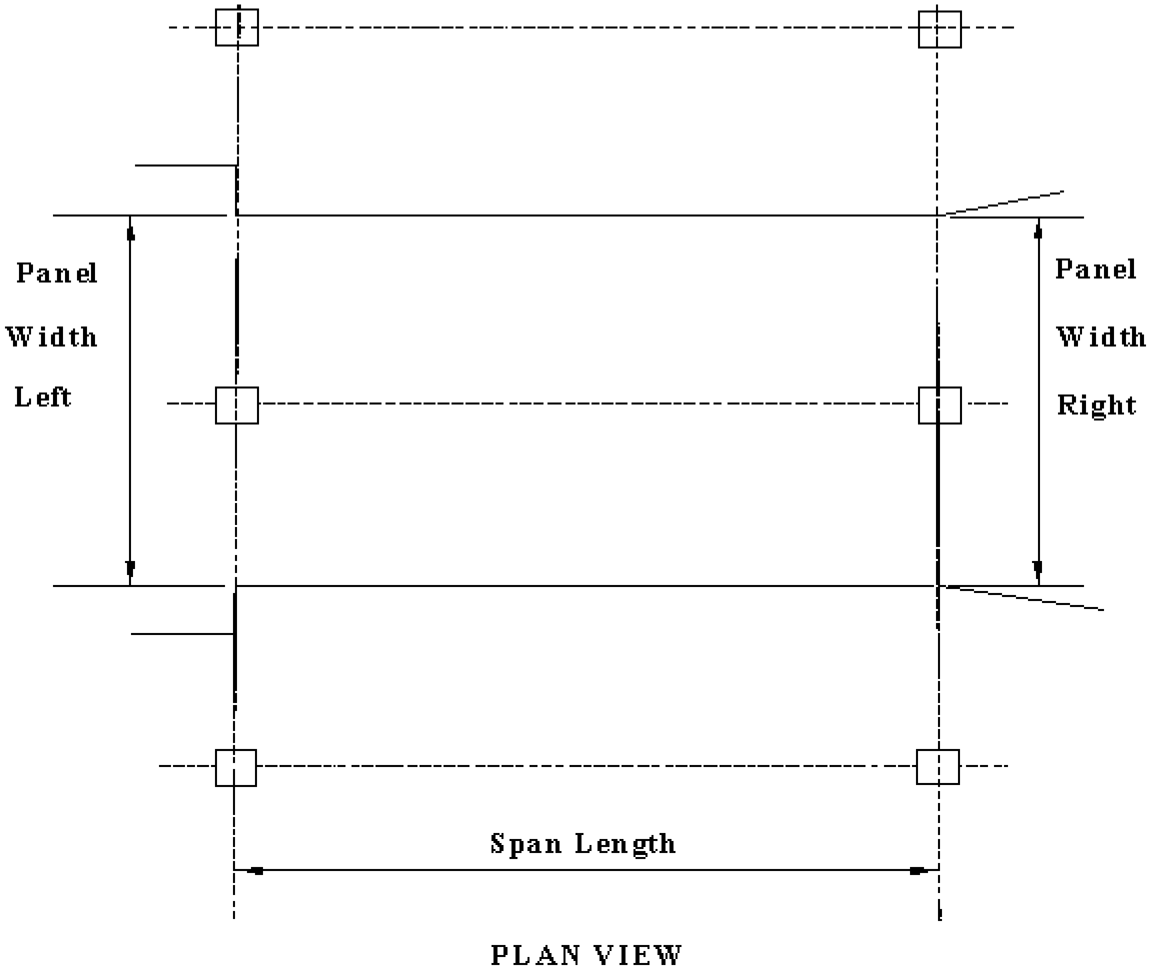


Figure 7.2.3.3 Panel Width Dimensions

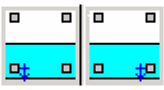
Note: Panel width should be input as:

(i) one-way slabs: At least the one-way width in each span. The self-weight and other load calculations for Panel Loads will be based on the one way width. For RAPT to default the correct value of Transverse Column Spacing in the Column Input Screen, the overall transverse width should be entered here.

(ii) beams: The overall transverse panel dimension at each column which the user considers to be the area of slab contributing self-weight loading to the beam. Other load calculations for Panel Loads will be based on the Panel Width (including the effects of any steps defined by the user.

(iii) two way slabs: The overall transverse panel dimension at each column which the user wishes to design. In this case self-weight and other load calculations for Panel Loads will be based on the panel widths.

(iv) external panels the total panel width will normally be equal to half the distance to the first internal column line plus the distance from the column centre-line to the outside edge of the slab or beam. The latter is referred to as the external edge distance.

 External Edge Distance

Left/Right End of Span The external edge distance is the perpendicular overhang (on the outside of the building) of the slab or beam past the centre-line of the columns at each end of the span. [See Figure 7.2.3.4] The concrete section at the column centre line is assumed to extend to this point. A negative value can also be specified.

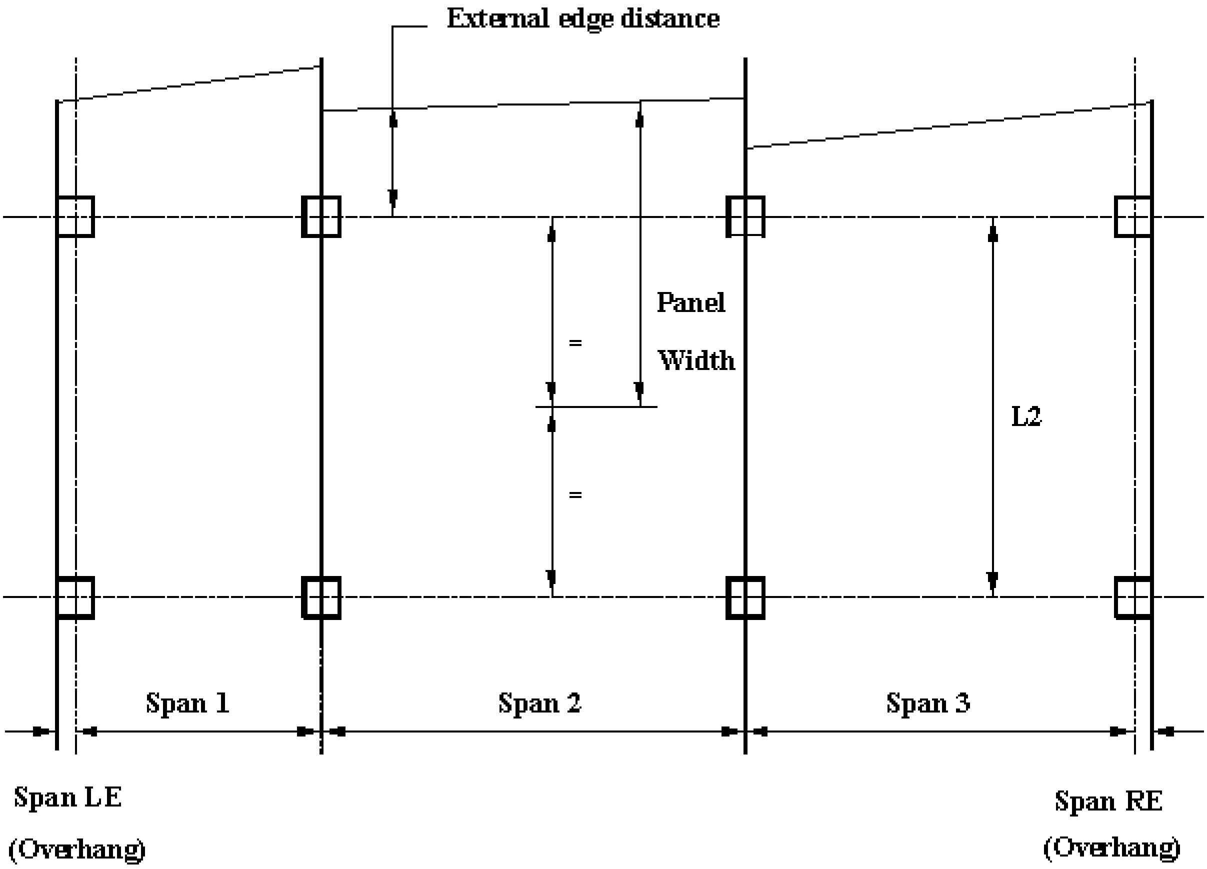
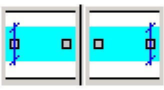


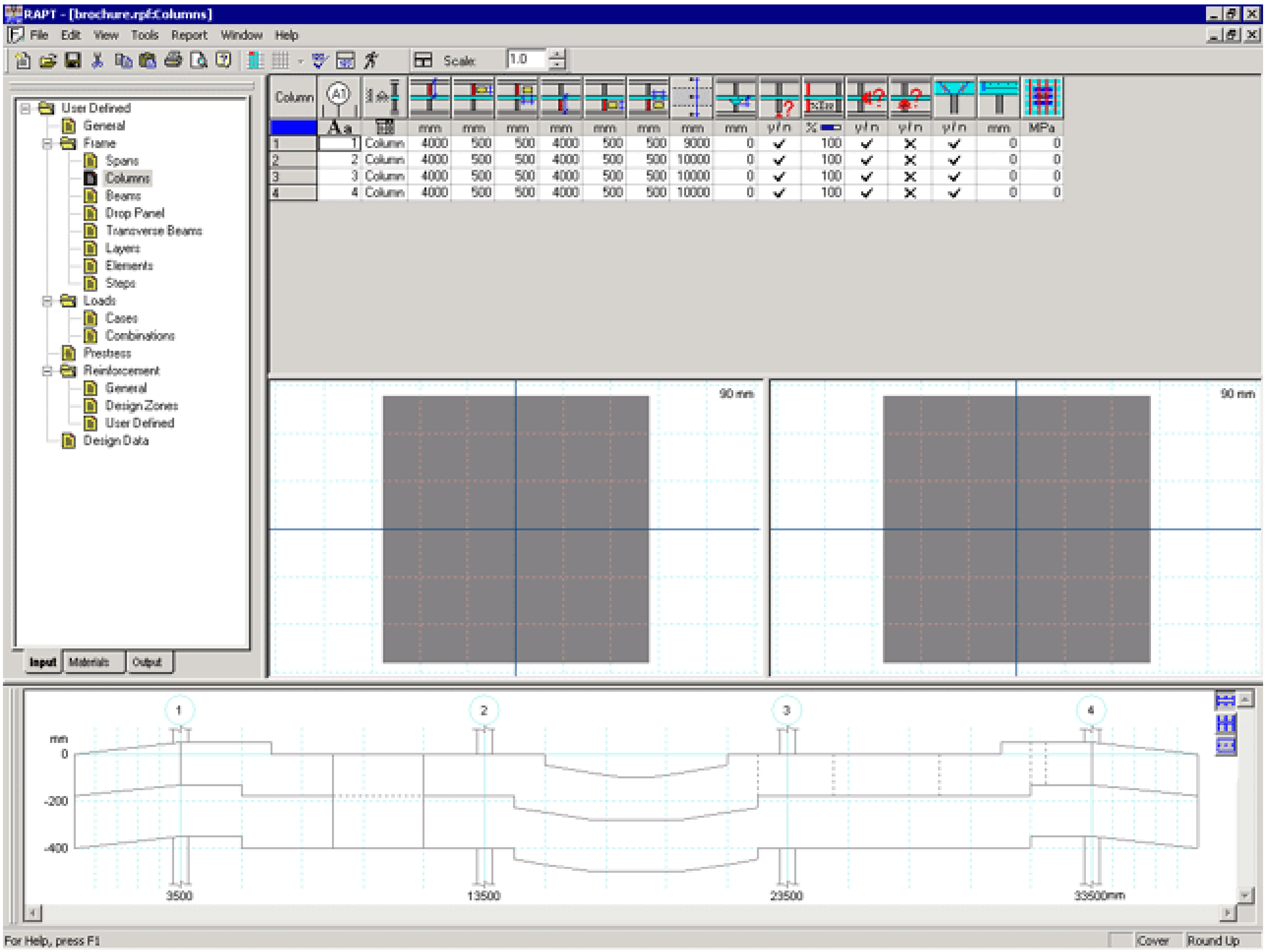
Figure 7.2.3.4 External Slab Dimensions

Values can only be entered in this field if external panel is specified in the General screen.

 Column Strip Width Left/Right End of Span

This option is only accessible when a two way slab has been specified. The value entered defines the Column Strip width. The Middle Strip width is then calculated as (Panel Width - Column Strip Width). The Column Strip Width is specified at the start and end of the span. This allows for a linearly varying column strip width along the span. More complicated Column Strip Widths can be entered using Steps Input. The values in these fields will default to the value defined in the relevant design code. It will be dependent on both span lengths and panel widths. Once these figures are calculated, designers may wish to rationalize the values where widths vary at column centrelines.

7.2.3.2 Column Data



Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

This screen is used to define the support types and details in RAPT.

The screen is set up with the support/column numbers down the left side and the data defining each support across the page. Supports can be

1. Knife Edge
2. Fixed support
3. Rectangular or Circular Columns above and/or below

Graphic Interaction

Clicking the left mouse button within the half span either side of a support/column in the graphics will automatically move the current row in the data to the row for that support/column. The section views will show the cross-section shape either side of the selected point.

Select a row in the data and the section graphics will show the cross-sections through the column above (left window) and column below (right window).

Section Views

When the program focus is in the columns data view, the Section Graphics Windows will show the cross-sections through the column above (left window) and column below (right window) if either has been defined.

Ctrl + D, Ctrl + R

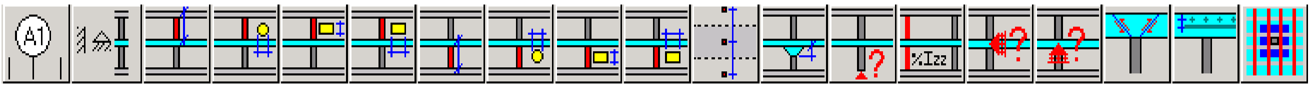
Refer to 4.4.4 Cell Repeating for a general discussion on repeating data automatically in other data cells. A few specific applications of the key apply to this screen when a range of repeat data cells is not selected. These are listed below

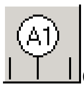
If the repeat key combination is pressed when the focus is in a blank cell (zero value) in

1. Column Length Above:- The values from Column Width Above will be copied into this data column
2. Column Height Below:- The values from Column Height Above will be copied into this data column
3. Column Width Below:- The values from Column Width Above will be copied into this data column
4. Column Length Below:- The values from Column length Below will be copied into this data column

These values will only be copied to columns that have a height.

Data Definition



Column Grid Reference

RAPT allows the designer to define the grid reference number of any support point to allow the designer to match the grid numbers in RAPT to those on the drawings for the member/panel being designed. This is a text field and the length of the text string is limited to 256 characters. RAPT will only use the first 3 characters in the graphics and text output. The Column Grid Reference will default to the column/support number e.g. 1, 2, 3, 4, 5 for a 4 span frame.

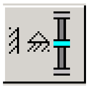
When spans are inserted, RAPT will match the new Column Grid Reference values created with the existing values if there is a simple logical consecutive pattern for all of the support locations in a frame. Otherwise, the column numbers of the inserted columns will be inserted as the default.

When spans are deleted, RAPT will match the new Column Grid Reference values with the existing values if there is a simple logical consecutive pattern for all of the support locations in a frame.

Logical consecutive patterns are

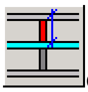
1. consecutive numbers - e.g 5, 6, 7, 8, 9
2. consecutive letter - e.g. D, E, F, G, H. When Z reached, the sequence will continue starting at AA, AB etc.
3. consecutive letters - e.g. BD, BE, BF, BG, BH. When BZ reached, the sequence will continue starting at CA, CB etc, or ABD, ABE, ABF, ABG, ABH. When ABZ reached, the sequence will continue starting at ACA, ACB etc.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

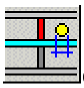
 Support Type

The type of support required at this support location. Types available are

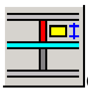
1. Knife Edge - No moment restraint. Vertical restraint is always provided and horizontal restraint is provided if Braced Column selected at this location.
2. Fixed Support - Full moment restraint provided at this column as well as vertical and horizontal force restraint.
3. Column - The defined columns above and/or below provide the moment restraint. If all of the column dimensions are defined as zero it will automatically be defined as a Knife Edge support. Vertical restraint is always provided and horizontal restraint is provided if Braced Column selected at this location.

Column Height Above

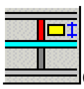
The height of the column from the top of slab datum level at the floor being analysed to the centroid of the concrete section in the floor above (see diagram below). The floor above is assumed to have the same concrete section as the floor being analysed.

Column Diameter Above

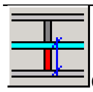
The diameter of the column if a circular column is to be defined above. To change to a rectangular column, delete the circular column dimension.

Column Width Above

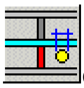
The width of a rectangular column above (c2 on diagram below). To change to a circular column, delete the rectangular column dimensions.

Column Length Above

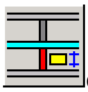
The length of a rectangular column above in the design direction (c1 on diagram below). To change to a circular column, delete the rectangular column dimensions.


Column Height below

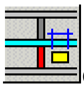
The height of the column from the top of slab datum level at the floor being analysed to the centroid of the concrete section in the floor below (see diagram below) if the column end is defined as fixed or to the top of a footing if the column end is defined as pinned. The floor below is assumed to have the same concrete section as the floor being analysed if the pinned end option is selected.


Column Diameter Below

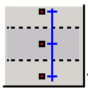
The diameter of the column if a circular column is to be defined below. To change to a rectangular column, delete the circular column dimension.


Column Width below

The width of a rectangular column below (c2 on diagram below). To change to a circular column, delete the rectangular column dimensions.

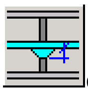

Column Length below

The length of a rectangular column below in the design direction (c1 on diagram below). To change to a circular column, delete the rectangular column dimensions.

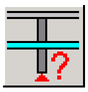

Transverse Column spacing

This is the average length of the transverse torsional member at each column which spans from this column to the columns on either side and is used for the calculation of the column properties (as distinct from the panel width which may vary from the transverse column spacing). RAPT defaults this figure as described below.

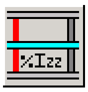
1. The default figure is only input by RAPT to make input easier in cases where the panel width is constant. If the panel width varies the user must change the default values to suit their needs.
2. For an internal panel RAPT will automatically insert the average panel width dimension at each column as the default. This will automatically cover the normal cases where the panel width is constant. For an internal panel the transverse column spacing should equal the average of the transverse column spacing on either side of a column.
3. For an external panel the transverse column spacing defaults to 2 x (Panel Width - External Edge Distance). For an external panel the transverse column spacing should equal the distance to the first internal column line.
4. If a one-way frame is being input then the transverse column spacing must be at least equal to the one way width even if no columns are input.
5. For beams and two way slabs the whole column inertia is applied in the frame analysis. For one-way slab structures the stiffness contribution of the column is proportioned to the ratio of the one-way width being run to the transverse column spacing.


Capital Depth below deepest section

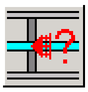
RAPT allows the designer to define a column capital or head at the top of the column below. This is defined as a depth below the deepest concrete soffit at either face of the column. The projection horizontally (outstand) is assumed the same on all four sides of the column as the vertical projection. This implies that the capital has a 45 degree taper and is of the same shape as the column.


Pin Base below

The base of any column below the slab may be either fixed or pinned. RAPT input to the setting in 5.2 Design Standard->General. Columns above the slab are assumed to be fixed ended under all circumstances.

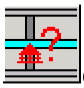

Percentage Column Stiffness

RAPT calculates a stiffness for each column. Users can then further modify this by specifying a % value of the calculated column stiffness. This portion of the column stiffness is applied to the frame analysis. The range of values allowed is 0% to 999%. Care should be taken in modifying this value especially for punching shear calculations. Redistribution of moments away form a column that is failing in punching shear is not recommended.


Braced frame at this column

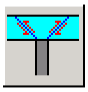
If the frame is braced at a column, RAPT will place horizontal pin supports (vertical rollers) at that column line of the

frame to prevent side sway. RAPT can not know from the general input if the frame being designed is braced by other members in the building such as shear walls so the designer must define this.

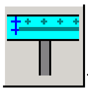
Column Shortening

RAPT gives the user the option to allow for the effects of Column shortening at each column. If Column Shortening is to be restrained, RAPT place a vertical restraint at each selected column beam node in the Frame Analysis. Thus no axial shortening can occur. The column shortening can, at times, have a large effect on moment distribution especially if high loads and columns of varying lengths or axial stiffness are input into a run. Where a concrete frame or the loading is not symmetric, allowing column shortening is normally the more accurate solution.

Always be careful when mixing knife edge supports and columns at different support locations as knife edge supports do not allow vertical movement while columns do. Knife-edge supports are often defined to represent walls and transverse beam supports. It is often better to model these as a column/wall but with zero stiffness to stop them from attracting moments but to still allow for the support settlement.

Punching Shear Check required

RAPT will carry out punching shear calculations for all columns unless told not to here. There are situations where punching shear calculations should be done for beam systems, especially band beams, and there may also be situations where local beams may negate the need for punching shear design in slab designs. These decisions are left to the designer by RAPT. The default setting is always for punching shear calculations to be done at all columns. RAPT will check the column shape in doing the calculations and, if the column width is greater than .8 times the panel width, RAPT will assume a wall is present and ignore punching shear calculations in this case.

Transverse effective depth difference

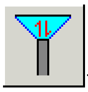
If a punching shear check has been requested, the designer may define punching shear properties for the transverse direction. Normally the effective depth in the transverse direction will be slightly different to allow reinforcement to pass over or under the reinforcement in this direction. At the end columns in prestressed frames for internal panels, the transverse tendons will be much higher in the slab than the longitudinal tendons thus requiring the input of an increased effective depth in the transverse direction. Conversely, for prestressed external frames, the transverse tendons will be much lower than the longitudinal tendons and a lower depth should be defined for them.

If a transverse effective depth difference is defined, RAPT will apply the longitudinal value calculated from the reinforcement and tendons defined at a column to the front and back faces of the punching perimeter and the transverse offset from this value to the side faces. An average effective depth will be reported in the output as

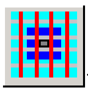
d = perimeter area / perimeter length.

The effective depth difference may be

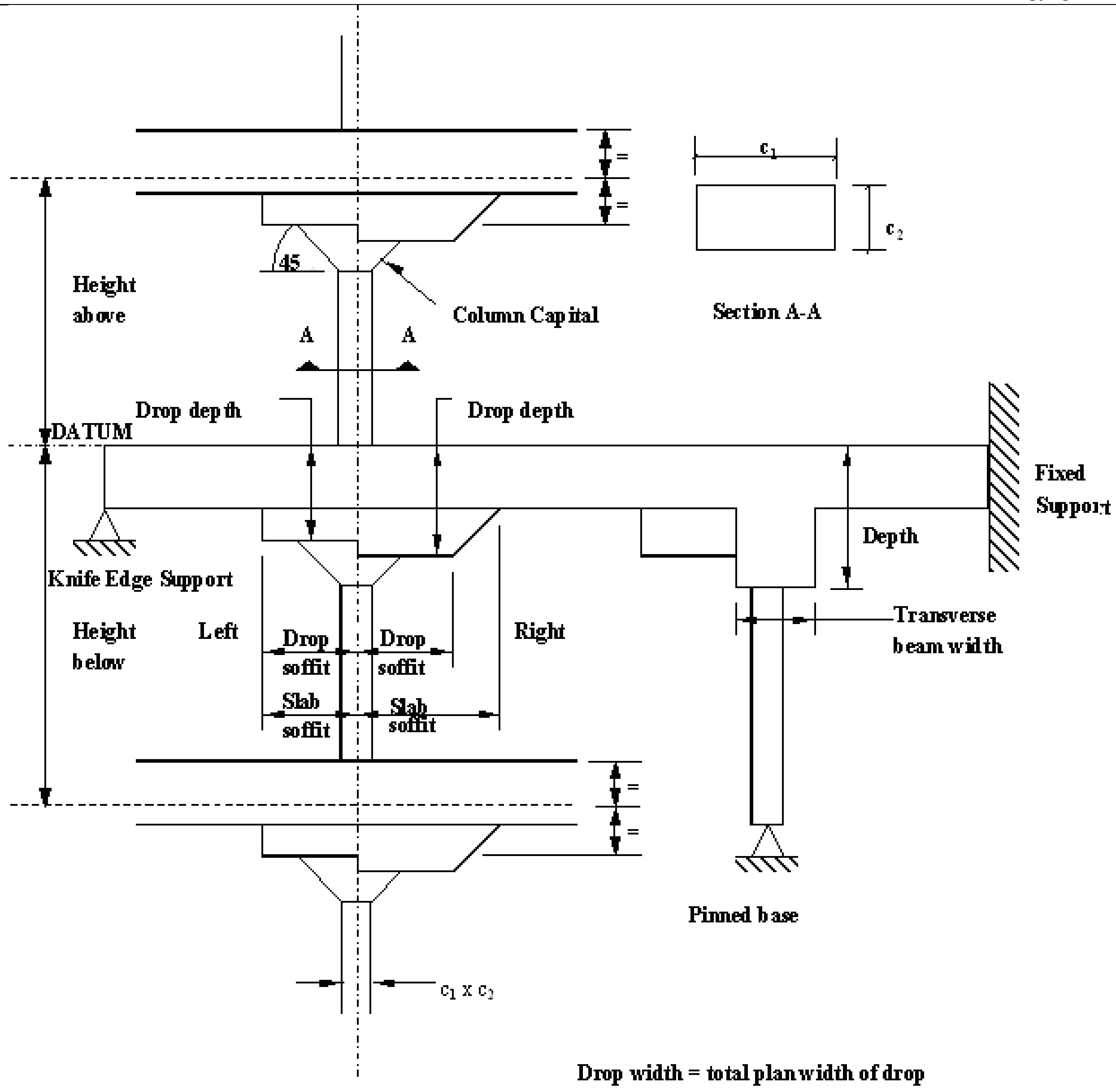
1. A positive value if the transverse effective depth is higher than the longitudinal effective depth (transverse reinforcement is above)
2. A negative value if the transverse effective depth is lower than the longitudinal effective depth (transverse reinforcement is below)

Transverse v'c

For BS8110, CP65, CP2004, IS456/IS1343 and SABS0100, the designer can define the concrete punching shear capacity in the transverse direction for each column. If a value is defined here, RAPT will apply the longitudinal value calculated from the reinforcement and tendons defined at a column to the front and back faces of the punching perimeter and the transverse punching shear value defined here to the side faces. Otherwise it will apply the calculated value for the frame being designed to all faces of the punching shear perimeter.

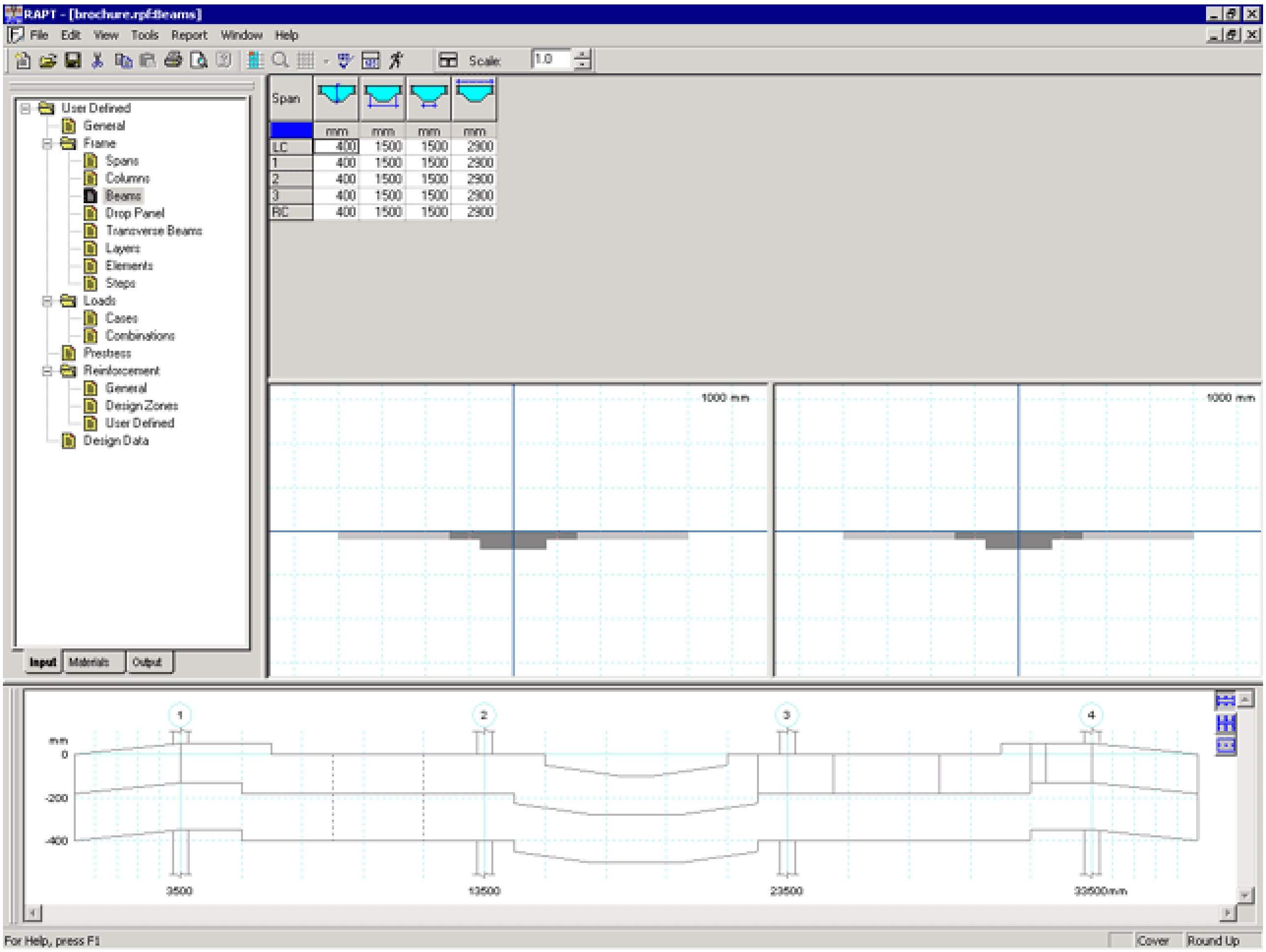
Transverse prestress (P/A)

For AS3600, ACI318 and Eurocode 2, the designer can define the axial prestress in the transverse direction for each column. If a value is defined here, RAPT will apply the longitudinal value calculated from the reinforcement and tendons defined at a column to the front and back faces of the punching perimeter and the transverse punching shear value defined here to the side faces. Otherwise it will apply the calculated value for the frame being designed to all faces of the punching shear perimeter.



Column and Drop Panel Data Input.

7.2.3.3 Beam Data



Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

Graphic Interaction

Clicking the left mouse button in a span in the graphics will automatically move the current row in the data to the row for that span. The section views will show the cross-section shape either side of the selected point.

Select a row in the data and the section graphics will show the data for that row in section. See below.

Section Views

When the program focus is in the beams data view, the Section Graphics Windows will show the slab shape as it is defined in spans data and the beam shape as it is defined in this view for this span, at the left (left panel) and right (right panel) ends of the span in which the cursor is positioned. Nothing is shown for Left Ends and Right Ends.

Ctrl + D, Ctrl + R

Refer to 4.4.4 Cell Repeating for a general discussion on repeating data automatically in other data cells.

If the repeat keys are used in the Beam Width (extreme surface) column of data in a data cell with a zero value, the Beam Width at Slab will be copied into each Beam Width cell following the rules in 4.4.4 Cell Repeating.

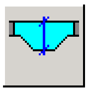
Beams Toolbar

Recalculate Effective Width: Recalculates the effective flange widths for beams based on the code requirements. If no span selection is made, the values for all spans will be recalculated. If a span or group of spans is selected, the widths for those spans only will be recalculated.

Simple/Detailed: In this case, simple/detailed refers to the definition of the effective flange widths.

1. Simple: A single effective flange width is defined in each span
2. Detailed: Three separate effective flange widths are defined in each span, one for each moment zone. This allows the definition of the effective flange widths required for Eurocode2 and HK CP2004. This option can be selected for any design code but will be calculated by the method defined in these codes.

Data Definition



Beam Depth

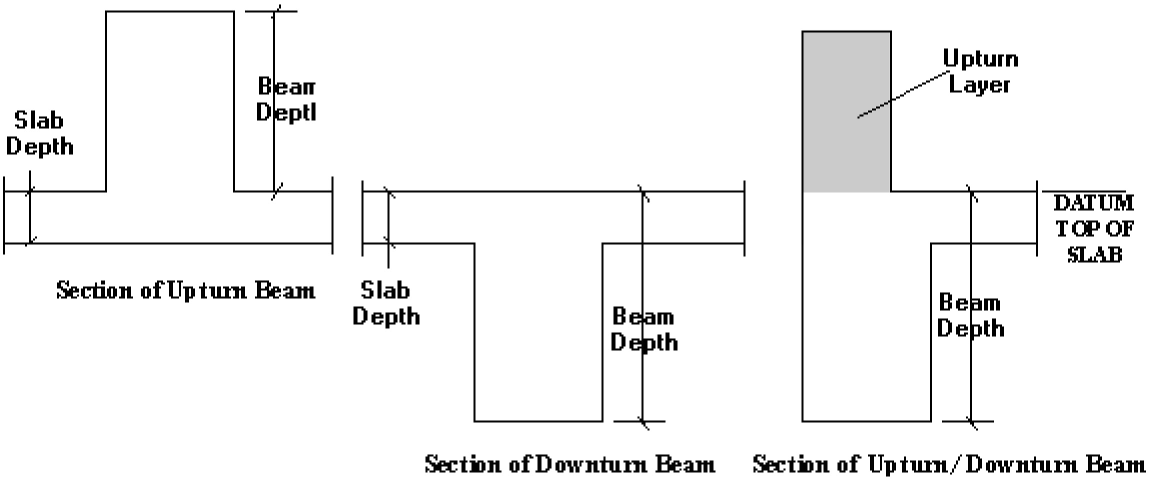
The Depth of the beam measured from the Top of Slab Datum. If the structural system has been defined as at least an effective flange width must be defined in each span. If the beam depth is left as zero, the slab with a width equal to the effective flange width will be used as the design section and should be made equal to the width the user considers to best model the way in which the system will act..

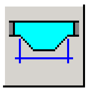
A negative value of beam depth represents an upturn beam (see figure below). In this case the depth of the beam is measured from the top of slab datum to the top of the beam.

Note:

1. Drop Panels are still placed on the underside of the slab if upturned beams are defined.
2. Users can not define a mixture of upturn and downturn beams. i.e. All beam depths must be positive or all must be negative, not a mixture. To define a combination of upturned and downturned beams users should use the extra 7.2.3.6 layer and 7.2.3.7 step input methods.

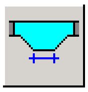
An upturn/downturn beam can be modelled by inputting an upturn beam and an extra 7.2.3.6 downturn layer or vice versa (see below). The method of defining upturn beams with continuous drop panels to define a downturn beam is not recommended by the authors.





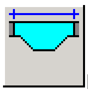
Beam Width at Slab

The width of the beam at the bottom of the slab. For an upturn beam the width is taken at the top of the slab.



Beam Width

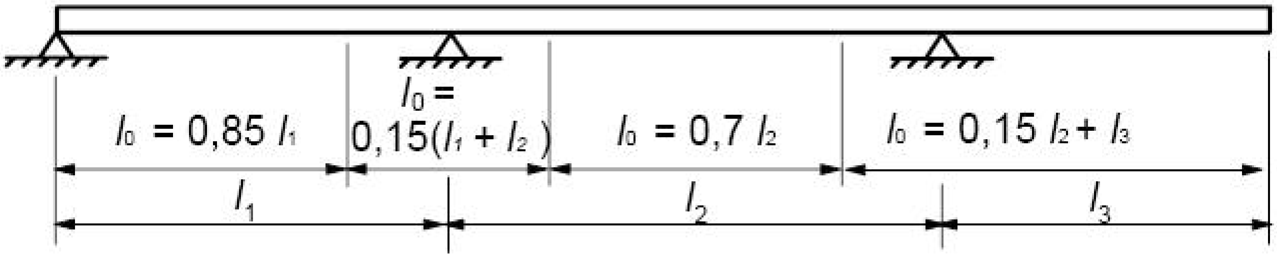
The width of the beam at the bottom of the beam. For an upturn beam the width is taken at the top of the beam.



Effective Flange Width

A different flange width may be defined in each span. As the web width at the slab is input by the user, RAPT defaults the effective flange width to that defined by the various design standards. For an internal panel, T-beam action is assumed. For an external panel the beam is defaulted as an L-beam. The flange widths may be modified if desired.

Some codes require a more complex calculation of effective flange width, resulting in three different widths in each span, one for each moment zone in the span. In Eurocode2 and HK CP2004, this more complex effective flange width is based on the l_o dimensions shown in the diagram below. To define this pattern, RAPT requires the definition of two lengths and three widths in each span as defined below. If the Detailed option is selected in the toolbar, the effective flange widths will be calculated using this method. This is the default for designs for Eurocode2 and HK CP2004.



Length of Left Effective Flange Width Zone
The length of the left end effective flange width zone in this span. Defaults to .15 of the span length if there is continuity or a column at the left end of the span. Zero for a left cantilever and at ends with no moment restraint and the length of the cantilever for a right cantilever.

Width of Left Effective Flange Width Zone
The effective flange width over the length of the length of the left moment zone.

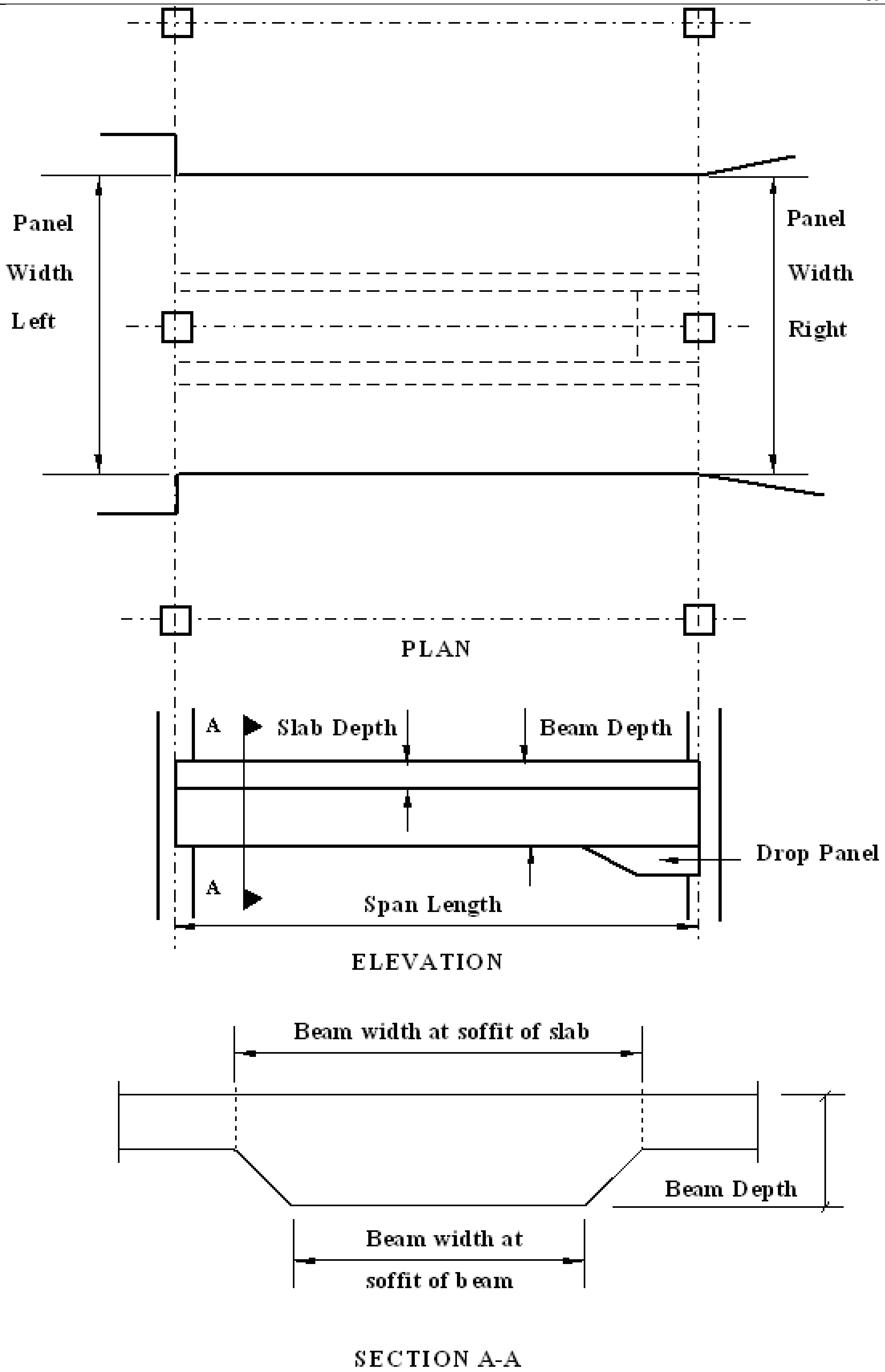
Width of Centre Effective Flange Width Zone
The effective flange width over the length of the length of the centre moment zone.

Length of Right Effective Flange Width Zone
The length of the right end effective flange width zone in this span. Defaults to .15 of the span length if there is continuity or a column at the right end of the span. The length of the cantilever for a left cantilever and zero for a right cantilever and at ends with no moment restraint.

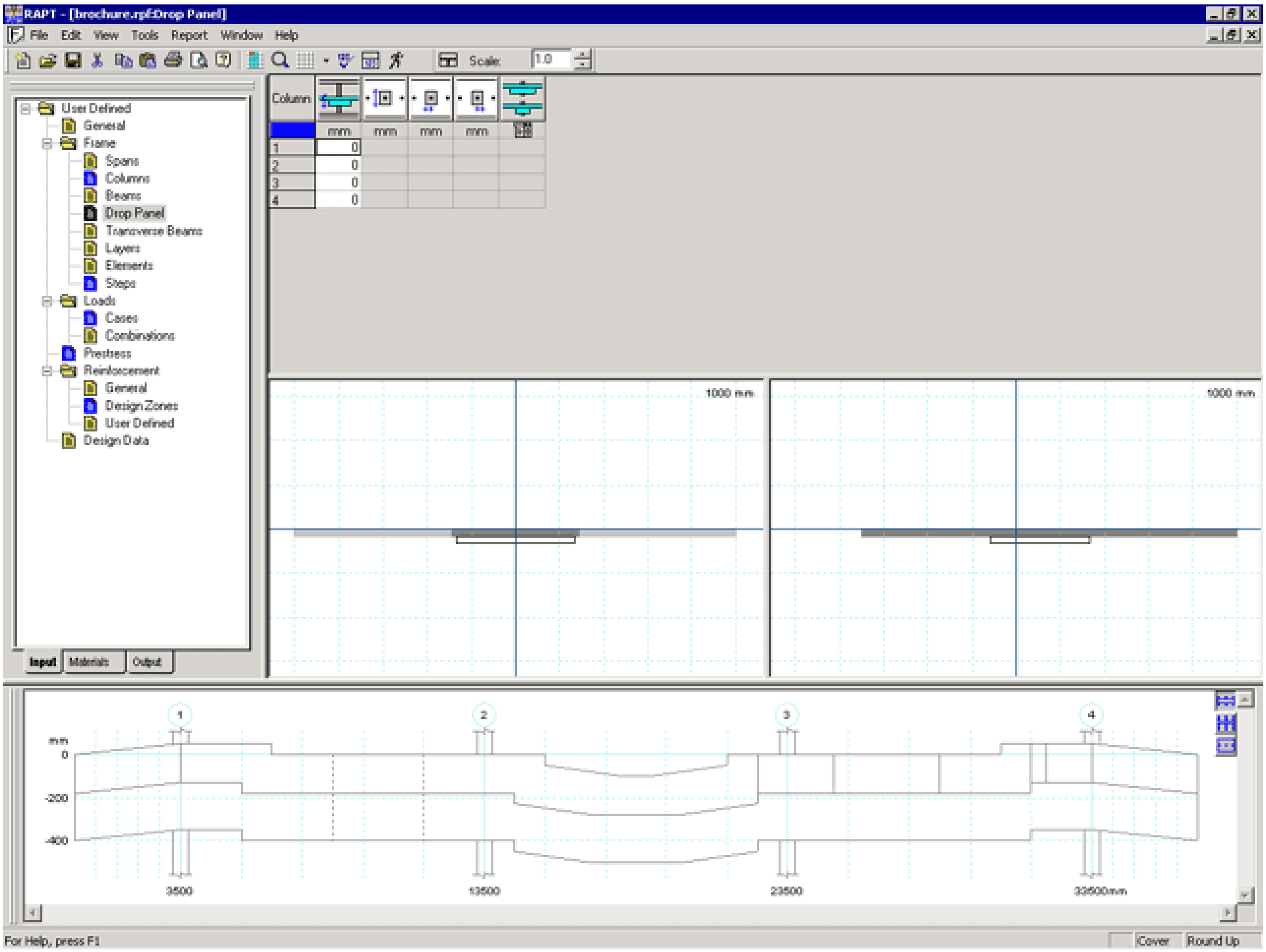
Width of Right Effective Flange Width Zone
The effective flange width over the length of the length of the right moment zone.

Note:

1. RAPT will check that the effective width that it calculates is less than the smaller of the transverse panel widths.
2. The flange widths are used for all strength and inertia calculations. The self-weight in a span is based upon the panel width defined in the Spans input screen except for one way slabs in which the Slab Design Width is used. In this way the self-weight of the slab need not be input manually by the user unless the load contributing area is not able to be defined in the RAPT input.
3. If the beam depth equals the slab depth in a span then the flange width defaults to the beam width.


Panel
Width
Left
Panel
Width
Right
PLAN
A
Slab Depth
Beam Depth
Drop Panel
A
Span Length
ELEVATION
Beam width at soffit of slab
Beam Depth
Beam width at
soffit of beam
SECTION A-A

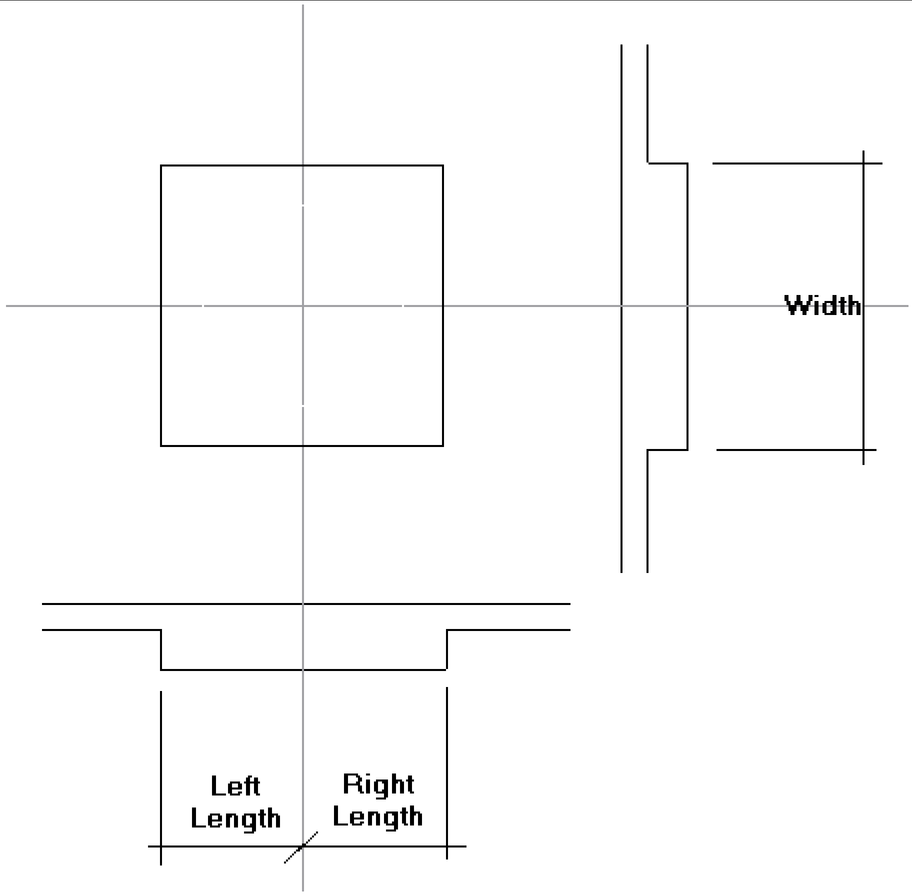
7.2.3.4 Drop Panel Data



Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

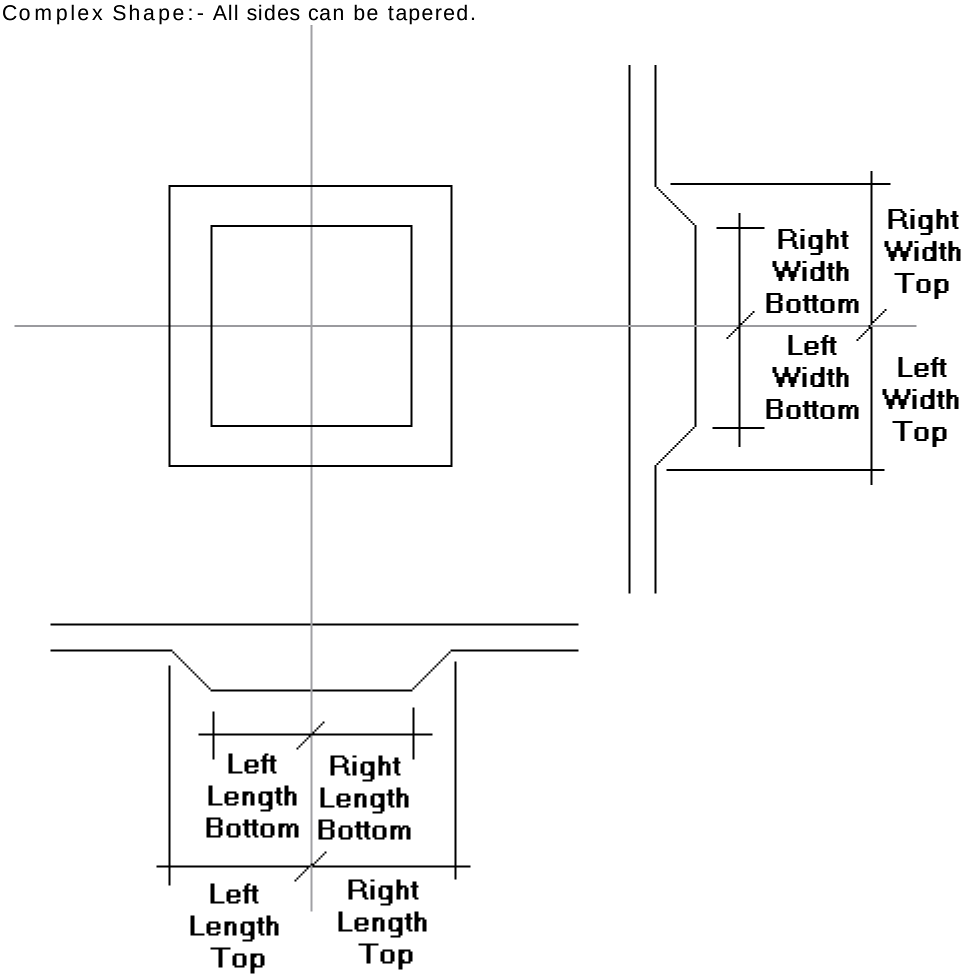
RAPT allows the definition of a drop panel of the shapes shown below at each column. All four sides may be tapered or vertical as desired. Two forms of data input are offered depending on the complexity of the drop panel shape.

1.



Simple Shape

2. Complex Shape:- All sides can be tapered.



Complex Shape

The desired input method can be selected by pressing the  button on the toolbar. When this button is expressed, the simple shape method will be available. When it is depressed, the complex shape will be available. When changing from Complex Shape to Simple Shape, RAPT will convert the drop panel sides to vertical using the top widths on all sides. The previous tapered side data will be lost.

The term Drop Panel has a specific meaning in concrete slab design and certain aspects of the design logic in RAPT will use these attributes. The use of drop panels to create complex shapes not related to the normal definition of a drop panel is not recommended. If they are used in this way the designer should check the results to make sure that RAPT is doing the calculations as you expect. RAPT will accept the use of drop panels joining at midspan (must join in the middle third of the span length) to model a local beam in a span though there are other ways to achieve this using 7.2.3.6 layers and 7.2.3.7 steps which do not require the use of drop panels.

Graphic Interaction

Clicking the left mouse button within the half span either side of a support/column in the graphics will automatically move the current row in the data to the row for that support/column. The section views will show the cross-section shape either side of the selected point.

Select a row in the data and the section graphics will show two cross-sections through the drop panel as discussed below.

Section Views

When the program focus is in the drop panels data view, the Section Graphics Windows will show the two cross-sections through the drop panel for the support/column line in which the cursor is positioned. The sections are based on the data as it is defined in the spans data for the slab and drop panels data for the drop panels. No account is taken of other concrete shapes that may occur at this location. The two sections are

1. Left Window:- Section across the frame showing the slab and drop panel, shows the width of the drop panel in section
2. Right Window:- Section along the frame showing the slab and drop panel, shows the length of the drop panel in section

Ctrl + D, Ctrl + R

Refer to 4.4.4 Cell Repeating for a general discussion on repeating data automatically in other data cells.

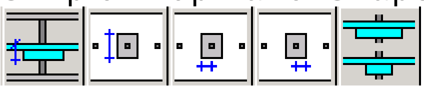
If the repeat key combination is pressed when the focus is in a blank cell (zero value) in

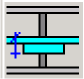
1. Drop Panel Length Right:- The values from Drop Panel Length Left will be copied into this data column making the drop panel symmetrical about the column.
2. Drop Panel Width Right Bottom :- The values from Drop Panel Width Right Top will be copied into this data column making the drop panel vertical sided.
3. Drop Panel Width Left Top:- The values from Drop Panel Width Right Top will be copied into this data column making the drop panel symmetrical about the column.
4. Drop Panel Width Left Bottom:- The values from Drop Panel Width Right Bottom will be copied into this data column making the drop panel symmetrical about the column.
5. Drop Panel Length Left Top :- The values from Drop Panel Width Right Top will be copied into this data column making the drop panel square.
6. Drop Panel Length Left Bottom :- The values from Drop Panel Width Right bottom will be copied into this data column making the drop panel square.
7. Drop Panel Length Right Top:- The values from Drop Panel Length Left Top will be copied into this data column making the drop panel symmetrical about the column.
8. Drop Panel Length Right Bottom:- The values from Drop Panel Length Left Bottom will be copied into this data column making the drop panel symmetrical about the column.

These values will only be copied to drop panels that have a depth and into data cells that are editable.

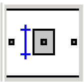
Data Definition

Simple Drop Panel Shape

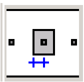



Drop Panel Depth

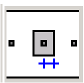
The depth of the drop panel from the top of the slab.


Drop Panel Width

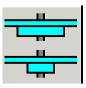
The overall width of the drop panel (refer diagram above). For internal panels, RAPT will place the drop panel symmetrically about the support line. For external panels, the drop panel width will be measured from the extreme outside edge at the centreline of the support.


Drop Panel Length Left

The length of the drop panel from the column line to the left end of the drop panel (refer diagram above). At a left end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The left end location can be moved in the 7.2.3.1 spans data.


Drop Panel Length Right

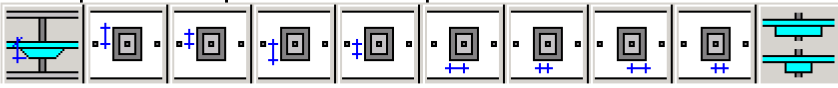
The length of the drop panel from the column line to the right end of the drop panel (refer diagram above). At a right end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The right end location can be moved in the 7.2.3.1 spans data.

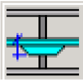

Drop Panel Type

RAPT allows the designer to define how the drop at each column is to be treated. Two options are available

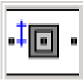
1. Drop Panel:- Normal rules for drop panels in flat slabs are that they extend at least span length / 6 from the column centreline in all directions. If this is the case, the drop will act both in flexural and in shear as a drop panel and will be included in all calculations for analysis, ultimate and service flexural design, deflection calculations and beam and punching shear calculations and the designer should nominate it as a Drop Panel. RAPT does not enforce the L/6 rule but will give a warning if at least one length is less than L/8 and Drop Panel is selected.
2. Drop Cap:- If a drop in a flat slab does not extend span length / 6 in all directions, it should not be considered to contribute to the flexural strength of the flat slab. It is then only included in the calculations for punching shear and the designer should nominate it as a Drop Cap. RAPT does not enforce the L/6 rule but will give a warning if all lengths are greater than L/6.5 and Drop Cap is selected suggesting that it could be changed to a Drop Panel to take advantage of the flexural strength benefits of a drop panel.

Complex Drop Panel Shape

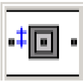



Drop Panel Depth

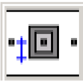
The depth of the drop panel from the top of the slab.


Drop Panel Width Right Top

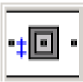
The width of the drop panel from the column line to the right side of the drop panel at the underside of the slab (refer diagram above). For an external panel, the value is set as the maximum 7.2.3.1 external edge distance to the edge of the concrete and cannot be modified by the designer. This edge distance can be modified in spans data.


Drop Panel Width Right Bottom

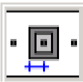
The width of the drop panel from the column line to the right side of the drop panel at the bottom of the drop panel (refer diagram above). For an external panel, the value is set as the maximum 7.2.3.1 external edge distance to the edge of the concrete and cannot be modified by the designer. This edge distance can be modified in spans data. Set equal to the Drop Panel Width Right Top for a vertical side on this side of the drop panel.


Drop Panel Width Left Top

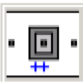
The width of the drop panel from the column line to the left side of the drop panel at the underside of the slab (refer diagram above).


Drop Panel Width Left Bottom

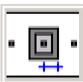
The width of the drop panel from the column line to the right side of the drop panel at the bottom of the drop panel (refer diagram above). Set equal to the Drop Panel Width Right Top for a vertical side on this side of the drop panel.


Drop Panel Length Left Top

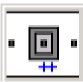
The length of the drop panel from the column line to the left end of the drop panel at the underside of the slab (refer diagram above). At a left end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The left end location can be moved in the 7.2.3.1 spans data.


Drop Panel Length Left Bottom

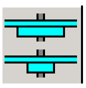
The length of the drop panel from the column line to the left end of the drop panel at the bottom of the drop panel (refer diagram above). At a left end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The left end location can be moved in the 7.2.3.1 spans data.


Drop Panel Length Right Top

The length of the drop panel from the column line to the right end of the drop panel at the underside of the slab (refer diagram above). At a right end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The right end location can be moved in the 7.2.3.1 spans data.


Drop Panel Length Right Bottom

The length of the drop panel from the column line to the right end of the drop panel at the bottom of the drop panel (refer diagram above). At a right end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The right end location can be moved in the 7.2.3.1 spans data.

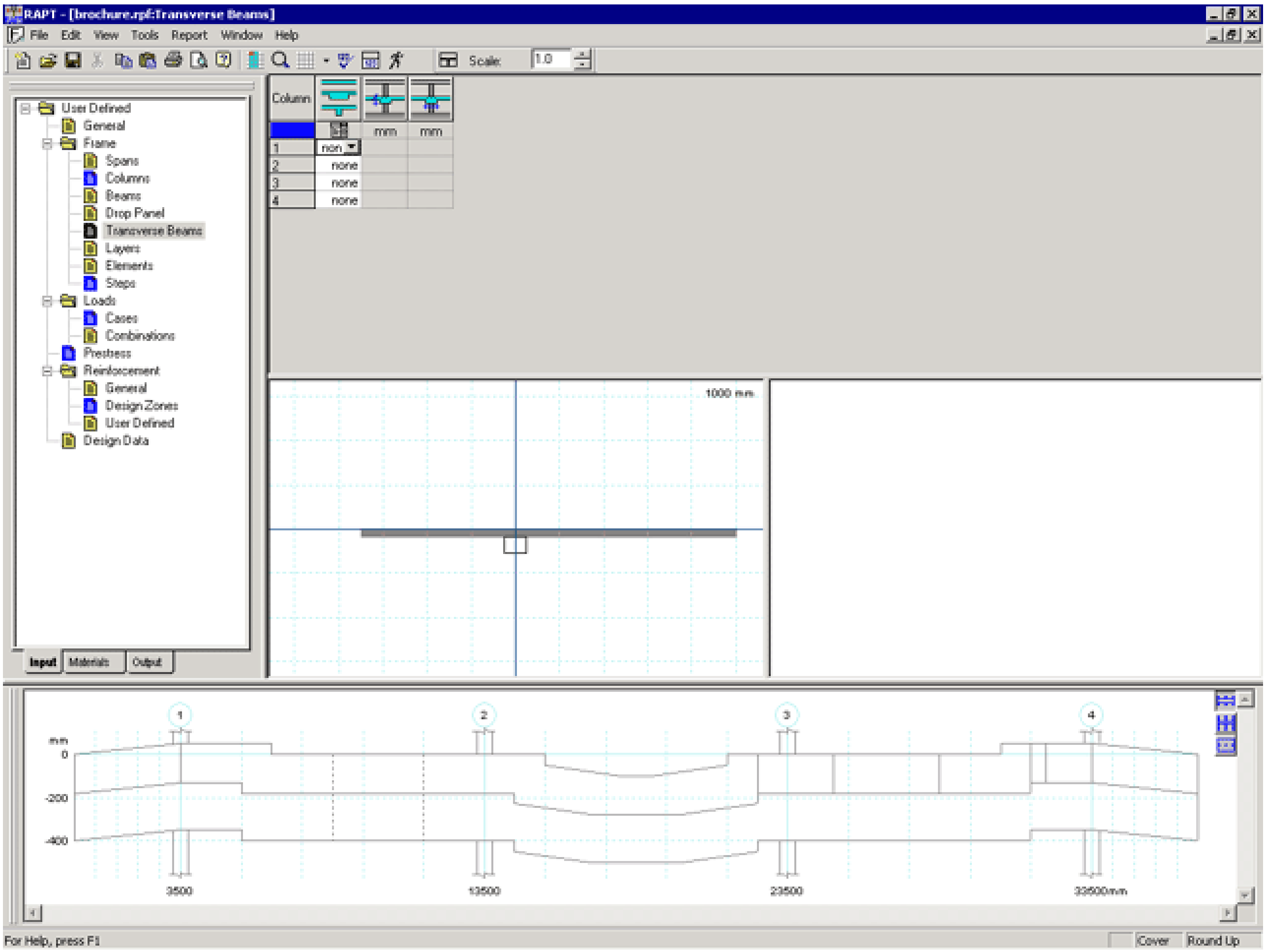

Drop Panel Type

RAPT allows the designer to define how the drop at each column is to be treated. Two options are available

1. Drop Panel:- Normal rules for drop panels in flat slabs are that they extend at least span length / 6 from the column centreline in all directions. If this is the case, the drop will act both flexurally and in shear as a drop panel and will be included in all calculations for analysis, ultimate and service flexural design, deflection calculations and beam and punching shear calculations and the designer should nominate it as a Drop Panel. RAPT does not enforce the L/6 rule but will give a warning at least one length is less than L/8 and Drop Panel is selected.

2. Drop Cap:- If a drop in a flat slab does not extend span length / 6 in all directions, it should not be considered to contribute to the flexural strength of the flat slab. It is then only included in the calculations for punching shear and the designer should nominate it as a Drop Cap. RAPT does not enforce the L/6 rule but will give a warning if all lengths are greater than L/6.5 and Drop Cap is selected suggesting that it could be changed to a Drop Panel to take advantage of the flexural strength benefits of a drop panel.

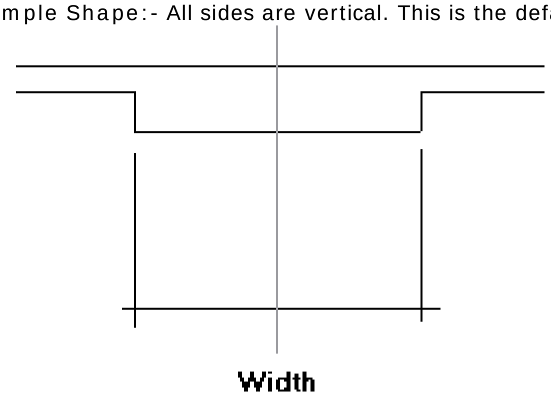
7.2.3.5 Transverse Beams/Bands



Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

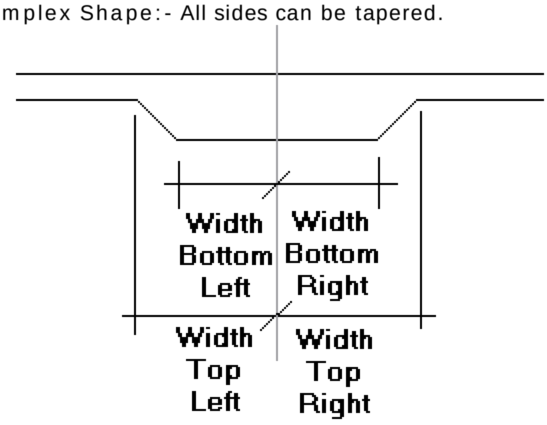
RAPT allows the definition of a transverse beam of the shapes shown below at each column. Each side may be tapered or vertical as desired. Two forms of data input are offered depending on the complexity of the transverse beam shape.

1. Simple Shape:- All sides are vertical. This is the default shape.



Simple Shape

2. Complex Shape:- All sides can be tapered.



Complex Shape

The desired input method can be selected by pressing the  button on the toolbar. When this button is expressed, the simple shape method will be available. When it is depressed, the complex shape will be available. When changing from Complex Shape to Simple Shape, RAPT will convert the transverse beam sides to vertical using the top widths on both sides. The previous tapered side data will be lost.

The terms Transverse Beam / Band have specific meanings in concrete slab design and certain aspects of the design logic in RAPT will use these attributes. The use of transverse beams to create complex shapes not related to the normal definition of a transverse beam is not recommended. If they are used in this way the designer should check the results to make sure that RAPT is doing the calculations as you expect. RAPT will not accept the use of transverse beams joining at midspan.

Graphic Interaction

Clicking the left mouse button within the half span either side of a support/column in the graphics will automatically move the current row in the data to the row for that support/column. The section views will show the cross-section shape either side of the selected point.

Select a row in the data and the left section graphics will show a cross-section through the transverse beam as discussed below.

Section Views

When the program focus is in the transverse beams data view, the Section Graphics Windows will show a cross-section across the transverse beam for the support/column line in which the cursor is positioned. The sections are based on the data as it is defined in the spans data for the slab and transverse beams data for the transverse beams. No account is taken of other concrete shapes that may occur at this location.

Ctrl + D, Ctrl + R

Refer to 4.4.4 Cell Repeating for a general discussion on repeating data automatically in other data cells.

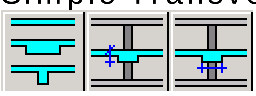
If the repeat key combination is pressed when the focus is in a blank cell (zero value) in

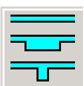
1. Transverse Beam Width Left Bottom:- The values from Transverse Beam Width Left Top will be copied into this data column making the transverse beam vertical sided.
2. Transverse Beam Width Top Right:- The values from Transverse Beam Width Left Top will be copied into this data column making the transverse beam symmetrical about the column.
3. Transverse Beam Width Bottom Right:- The values from Transverse Beam Width Left Bottom will be copied into this data column making the transverse beam symmetrical about the column.

These values will only be copied to transverse beams that have a depth and into data cells that are editable.

Data Definition

Simple Transverse Beam Shape

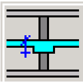


 Transverse Beam Type

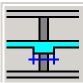
The user can select one of three options for a transverse beam at each column. These are

1. None:- No beam will be added and data cells are not accessible.

2. Transverse Band:- A transverse band will be added. A band is a wide shallow beam and will be modelled as a slab thickening. It will act flexurally with the slab. Design points will be placed at the face of the band and at the critical section at the face of the support within the band width. Prestress forces will be assumed to disperse into this depth. Its depth will contribute towards flexural, beam and punching shear calculations within its length.
3. Transverse Beam:-A transverse beam will be added. A beam will be modelled as a support element. It will act not flexurally with the slab. It will be considered in the calculation of the location of the critical section at the face of the support and will often control the location of the critical section at the face of the support. The transverse beam will be ignored in the prestress force and action calculations with the assumption that prestress will not disperse significantly into the depth of the beam. Its depth will not contribute towards flexural, beam and punching shear calculations within its length.

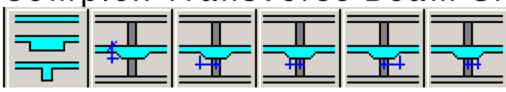
Transverse Beam Depth

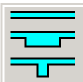
Depth of the transverse beam from the top of the slab. The depth must be greater than the slab depth either side of the column.

Transverse Beam Width

Width of the transverse beam (refer diagram above). At internal columns the transverse beam will be placed symmetrically about the column/support line. For edge transverse beams it will be measured from the external edge.

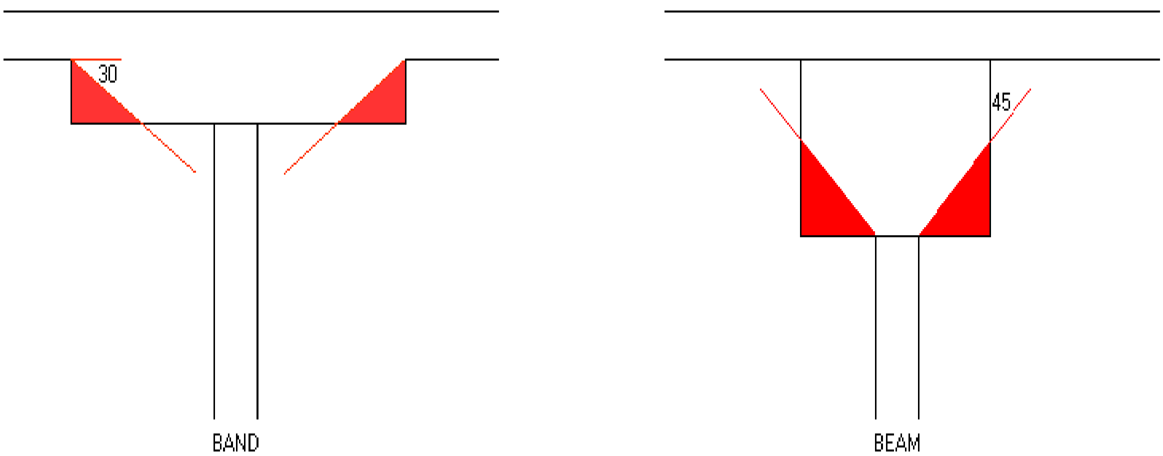
Complex Transverse Beam Shape

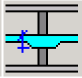


Transverse Beam Type

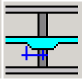
The user can select one of three options for a transverse beam at each column. These are

1. None:- No beam will be added and data cells are not accessible.
2. Transverse Band:- A transverse band will be added. A band is a wide shallow beam and will be modelled as a slab thickening. It will act flexurally with the slab. Design points will be placed at the face of the band and at the critical section at the face of the support within the band width. Prestress forces will be assumed to disperse into this depth. Its depth will contribute towards flexural, beam and punching shear calculations within its length. As shown in the left diagram below, if a 30 degree line from the intersection of the band and the slab does not reach the column, then it should be treated as a transverse band beam and therefore as part of the slab, similar in effect to a drop panel except that it is continuous in the transverse direction.
3. Transverse Beam:-A transverse beam will be added. A beam will be modelled as a support element. It will act not flexurally with the slab. It will be considered in the calculation of the location of the critical section at the face of the support and will often control the location of the critical section at the face of the support. The transverse beam will be ignored in the prestress force and action calculations with the assumption that prestress will not disperse significantly into the depth of the beam. Its depth will not contribute towards flexural, beam and punching shear calculations within its length. As shown in the left diagram below, if a 30 degree line from the beam/column corner extends through the side of the beam, then the transverse member should be defined as a transverse beam. It will act as part of the support.

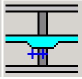



Transverse Beam Depth

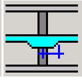
Depth of the transverse beam from the top of the slab. The depth must be greater than the slab depth either side of the column.


Transverse Beam Width Left Top

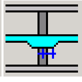
The width of the transverse beam from the column line to the left side of the transverse beam at the underside of the slab (refer diagram above). At a left end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The left end location can be moved in the 7.2.3.1 spans data.


Transverse Beam Width Left Bottom

The width of the transverse beam from the column line to the left edge of the transverse beam at the bottom of the transverse beam (refer diagram above). At a left end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The left end location can be moved in the 7.2.3.1 spans data.

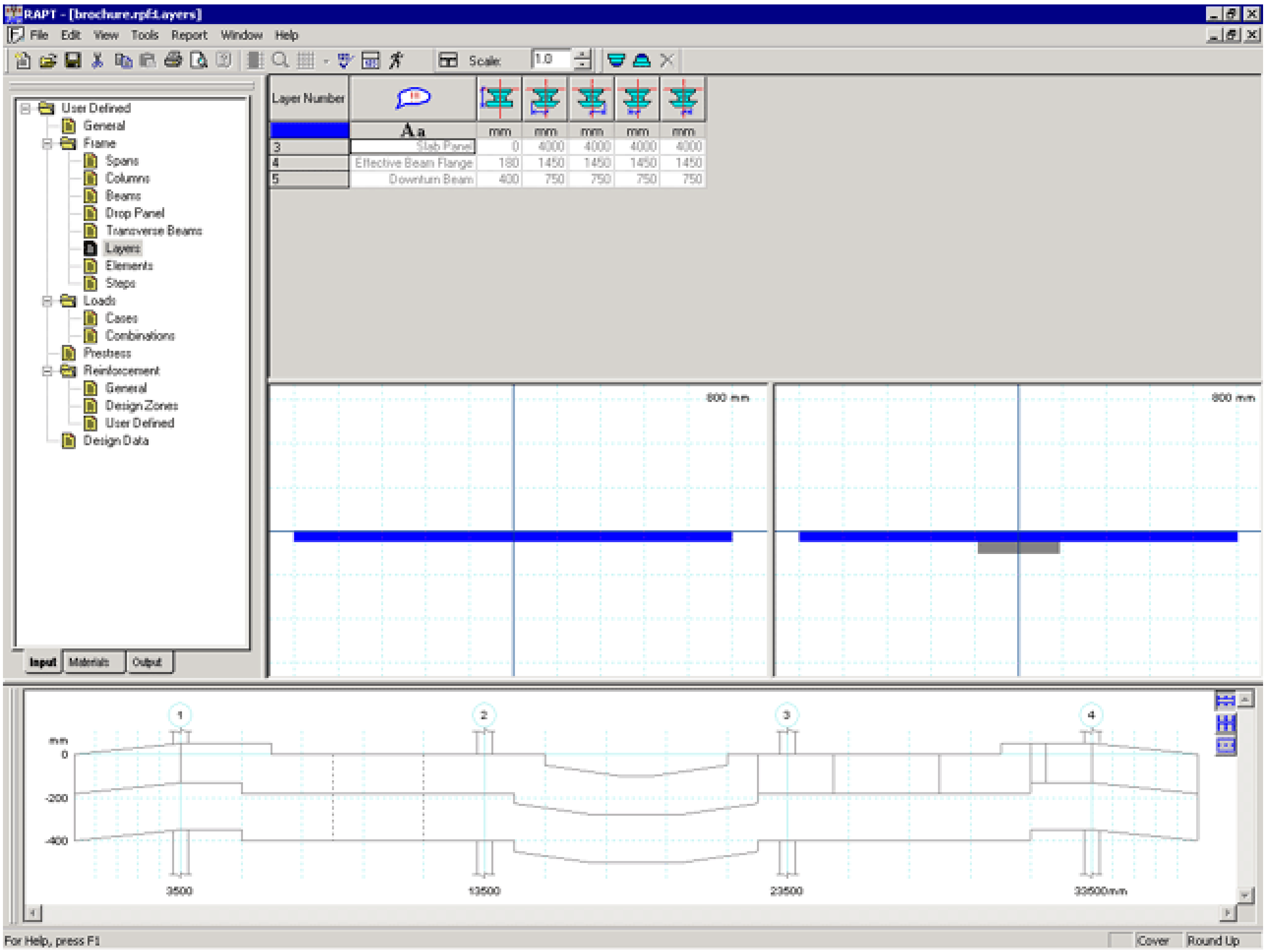

Transverse Beam Width Right Top

The width of the transverse beam from the column line to the right side of the transverse beam at the underside of the slab (refer diagram above). At a right end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The right end location can be moved in the 7.2.3.1 spans data.


Transverse Beam Width Right Bottom

The width of the transverse beam from the column line to the right edge of the transverse beam at the bottom of the transverse beam (refer diagram above). At a right end column, the value is set as the distance to the end of the concrete and cannot be modified by the designer. The right end location can be moved in the 7.2.3.1 spans data.

7.2.3.6 Layers



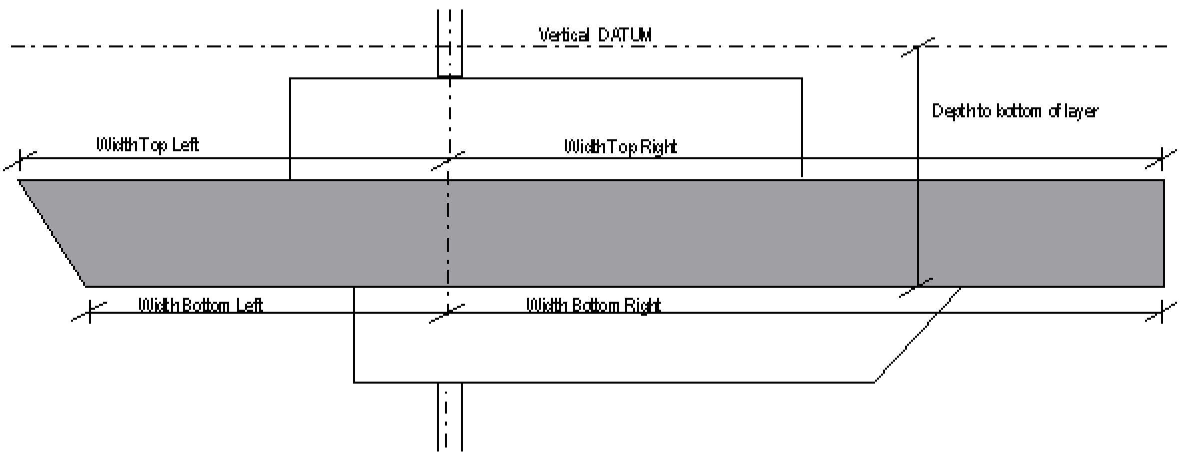
Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.

What is a Layer

A cross-section is made up of a series of Layers i.e. slab layer, beam layer, drop panel layer etc. Each layer presents a horizontal slice of the cross-section, defining the width properties for that portion of the cross-section. Each individual layer exists for the full length of the frame and can have vertical and horizontal steps applied to it to change its shape along the frame. Thus, a layer's dimensions from the datum can vary along the frame. The top and bottom surfaces of a layer must be horizontal whereas the side faces can be sloping. Thus each layer is a trapezium with flat horizontal surfaces.

A layer cross section is defined at the left hand end of the frame. This window defines the shapes of all of the layers at the left hand end of the frame. All steps, tapers etc defined in screens to and those defined in by the user are applied to these shapes.

The layers generated from the concrete frame input for slabs, beams etc is converted into layers and those layers are shown on this view but are not editable here. Editing them in their own views will modify their shapes here.



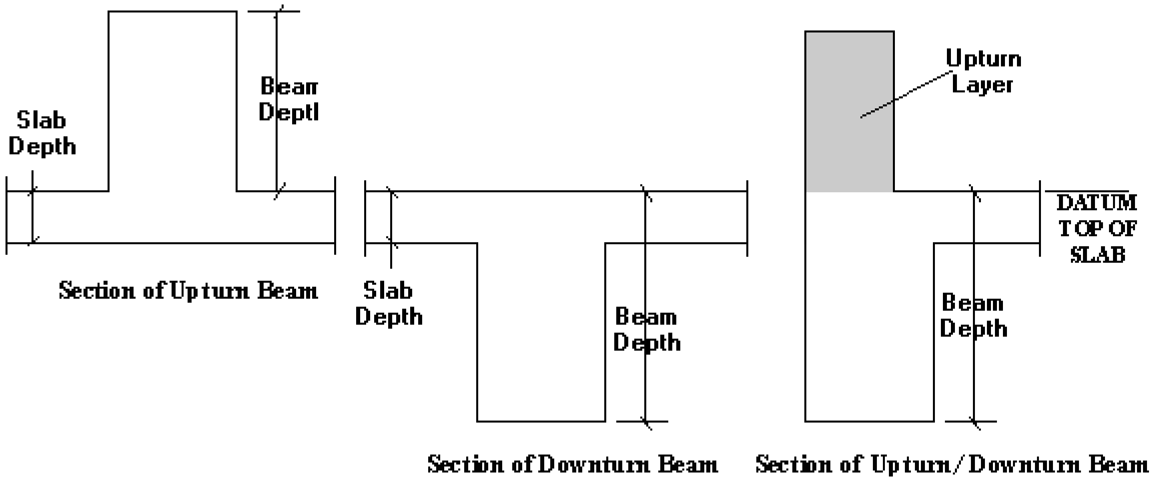
Thus, a layer is defined in terms of (see above)

1. A depth to the bottom (top for an upturn layer) of a layer from the datum. The thickness of a layer is thus defined as the distance between the bottom of a layer and the bottom of the layer above.
2. Widths Left and Right / Top and Bottom. These dimensions are measured from a central control point which is defined as the horizontal datum. (Column Centre Line)

Note: The Vertical Datum is defined as the top of slab as defined in the Spans window i.e. before any step effects are added. The Horizontal Datum is defined as the centreline of the supports.

Once a layer is defined it exists for the full length of the frame. It can be stepped vertically and horizontally using the other ALT menu options, so that it contains a zero depth, but it always exists once defined.

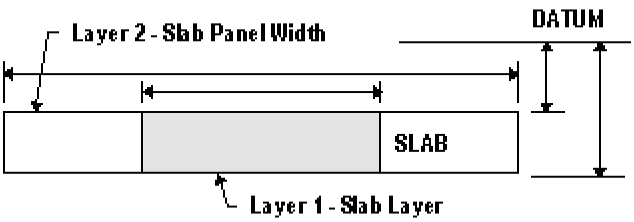
A useful application for layers is an upturn/downturn beam as shown below.



Special layers

The slab layer is special case. For every run a slab layer must exist. It is the basic layer upon which a cross section is built. It is also the only member defined by two layers. These layers are

1. Slab Panel Width:- Has zero depth thus defining the top of the slab layer from the datum The width defines the panel width of the frame being defined.
2. Slab Layer:- Depth to bottom of slab layer thus defining the bottom surface of the slab layer. The width defines the nominal design width for a one-way nominal width slab design, Beam flange width for a beam design or Column strip width if a two way system is defined.



Layers Toolbar

Extra layers added to the slab layer are defined as

1. Downturn Layers if they are added below the Slab Layer. (i.e. below bottom of slab)
2. Upturn Layers if they are added above the Slab Panel Width Layer (i.e. top of slab)

A new layer can be inserted between existing user defined layers. It cannot be inserted between program calculated layers. To insert a layer between existing layers, 4.4.3 select (4.4.3) the layer row at which you want the new layer added. The row selected and all rows below (downturn layer) or above (upturn layer) that row will move and a new data row will be inserted. If a program calculated row is selected, the toolbar buttons will not be available as program calculated cannot layers cannot move.

Add a layer below

Adds a downturn layer data row. Downturn layers must be below all of the program calculated layers. The depth to the downturn layer must be greater than or equal to the depth to the layer above it.

Add a layer above

Adds an upturn layer data row. Upturn layers must be above all of the program calculated layers. The depth to the downturn layer must be less than or equal to the depth to the layer above it so it must always be less than or equal to 0 and less than the depth to the top of an upturned beam if present.

Delete concrete layers

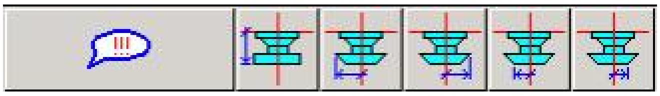
Any user defined layer or group of layers 4.4.3 selected (4.4.3) with the mouse can be deleted. Other layers will simply fill the void if outside the deleted layer/s. This button will not be available until a legal selection is made. If a program calculated layer is included in the selection to delete, no layers will be deleted.

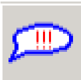
Section Views

When the program focus is in the layers data view, the Section Graphics Windows

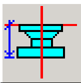
1. Left panel:- The shape of the layer defined by the current data row.
2. Right Panel:- The full cross-section shape at the left end of the frame with the shape of the layer defined by the current data row highlighted in the same colour as in 1 above.

Data Definition

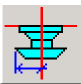


Description

Name of layer.

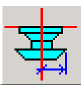
Extreme Depth

Depth to extreme face of layer from the Top of Slab Datum (see below). Depths to upturned layers should be negative.

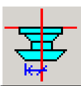
Width Top Left

Width from the centreline of the supports to the top left corner of the layer (see below). If the designer wants to define more than one of the widths as the same value, selecting multiple cells and using Ctrl + D or Ctrl + R or  will allow a group of widths to be set to the same value at the same time. The diagram below shows all four widths being set to 300mm.

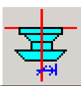
Layer Number						
	Aa	mm	mm	mm	mm	mm
3	Slab Panel	0	5000	5000	5000	5000
4	Effective Beam Flange	200	1450	1450	1450	1450
5	Downturn Beam	500	750	750	750	750
6	Downturn Layer	700	300	0	0	0

Width Top Right

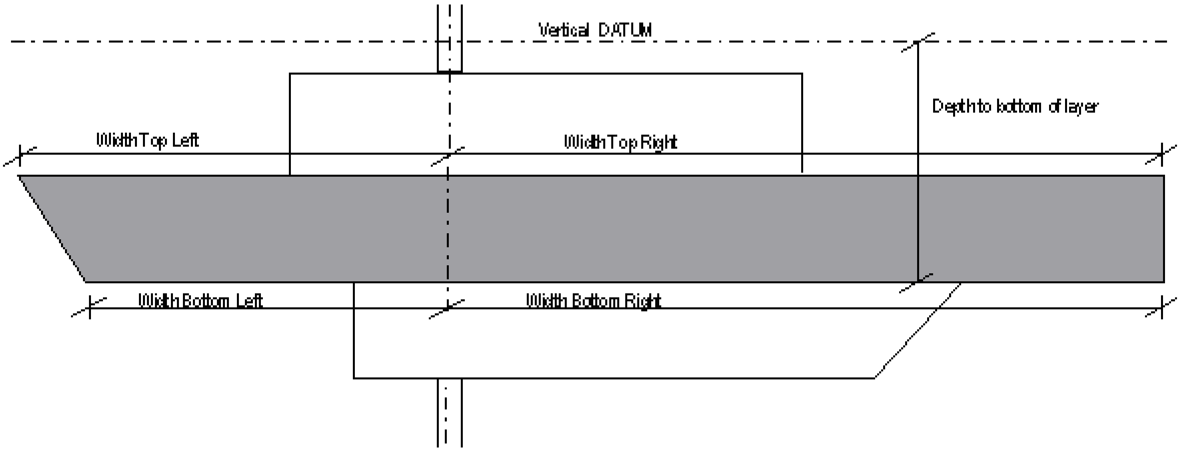
Width from the centreline of the supports to the top right corner of the layer (see below).

Width Bottom Left

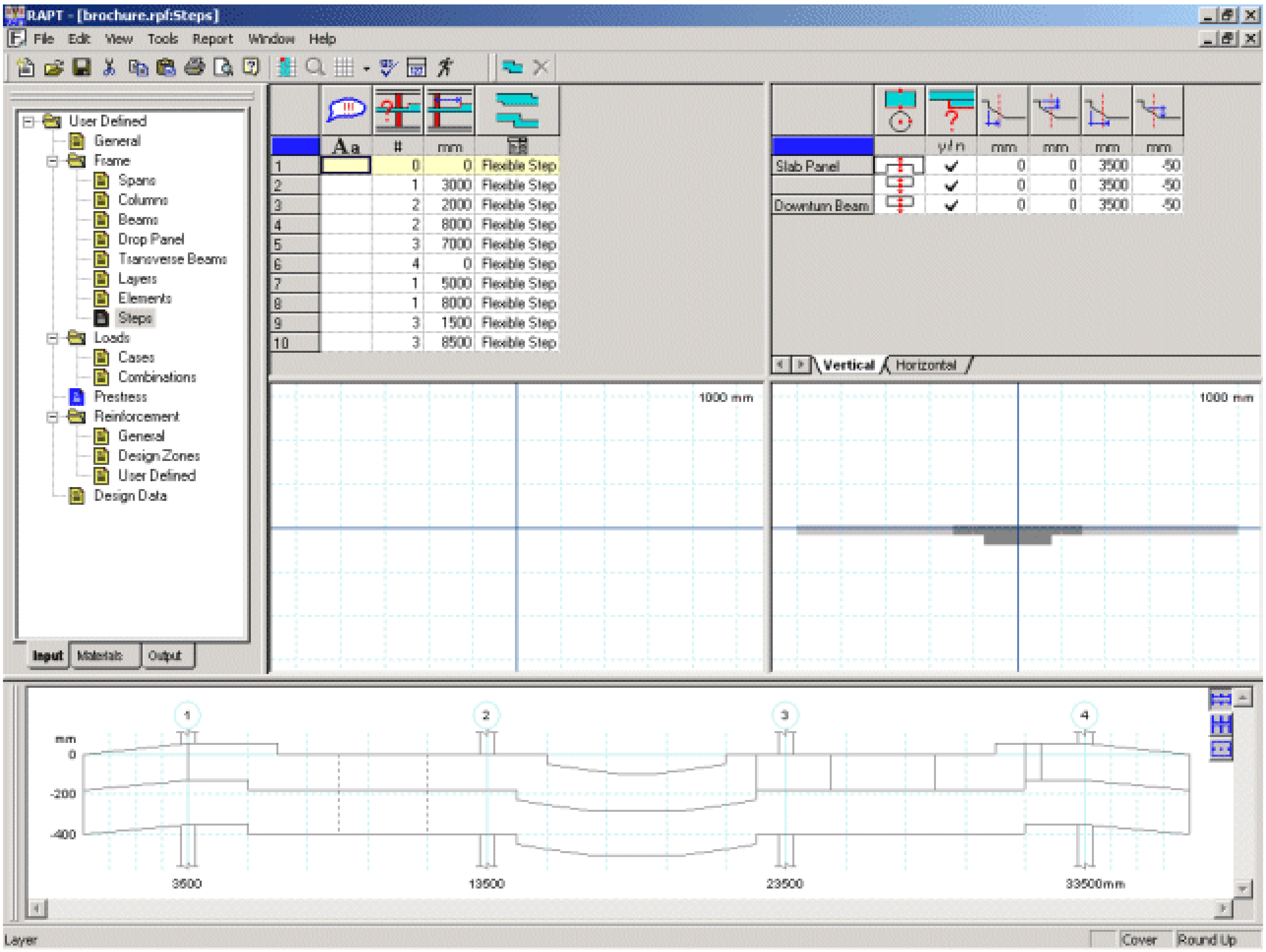
Width from the centreline of the supports to the top bottom corner of the layer (see below).

Width Bottom Right

Width from the centreline of the supports to the bottom right corner of the layer (see below).


Vertical DATUM
Depth to bottom of layer
Width Top Left
Width Top Right
Width Bottom Left
Width Bottom Right

7.2.3.7 Steps

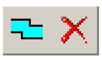


This input screen is used to define the step locations. The screen is divided into 3 separate windows

1. Steps Control Grid:- This grid lists the general location data for each step and controls which step is viewed in the Profile Data (Refer to 7.2.3 Frame Shape Screen Layout for discussion of the general principles of the RAPT Frame Screen layout and to 4.5.1 General Screen Layout Principles for discussion on the general layout features of windows in RAPT.). The functionality of this window is discussed in this section.
2. Vertical/Horizontal Steps Data:- Defines the specific step types and shapes for the step selected in the Control Grid. The functionality of the views in this window are discussed in the later sections.
3. Frame Graphics:- Shows the frame graphics views and section views as discussed in 7.2.3 Frame Shape Screen Layout and below.

At any defined step location, the designer is able to introduce a sudden step and/or tapered step, at an offset from this location, into any surface of any layer or element that exists at that step location. More details on the options available are provided in 7.2.3.7.1 Vertical Steps/Tapers and 7.2.3.7.2 Horizontal Steps/Tapers.

Step Toolbar




Add step

Adds a new line of data to this grid to allow the designer to nominate a new step location.


Delete step

To select a step to delete, the relevant row on the grid must be selected. It can be selected by a left mouse select in the step number column. Multiple steps can be deleted at once using the selection logic in 4.4.3 Cell Selection.

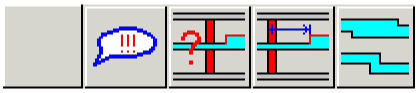
Section Views

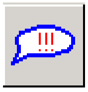
When the program focus is in the steps data view, the Section Graphics Windows

1. Left panel:- The section shape on the right side of the leftmost location defined for this step in any surface of any layer or element.

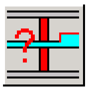
2. Right Panel:- The section shape on the left side of the rightmost location defined for this step in any surface of any layer or element.

Data Definition




Description

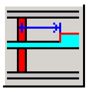
Descriptive name for the step.


Reference Column

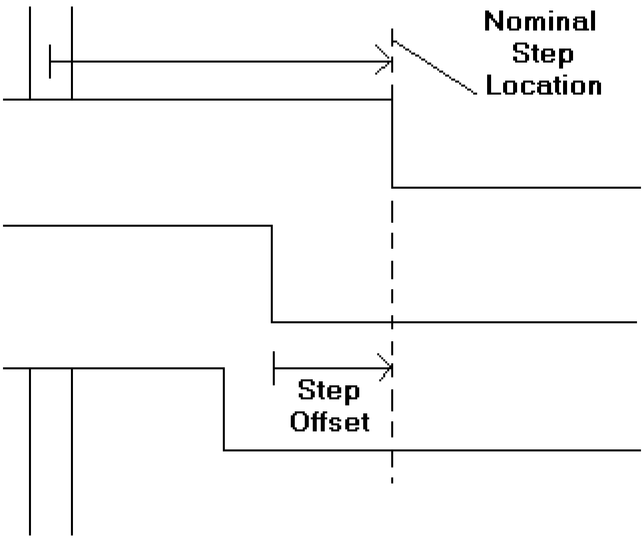
The 7.2.3.2 Column Grid Reference from which the step is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the step. This is necessary for the internal calculations for adding and deleting spans.

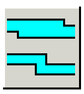
In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the location of a step in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the location of a step.


Distance from Reference Column

The distance from the reference column to the nominal step location. This location is logically the location of the vertical step in the topmost surface to have a step at this location but may be any reference location. Steps in each layer surface are defined at an offset dimension from this location as shown below.

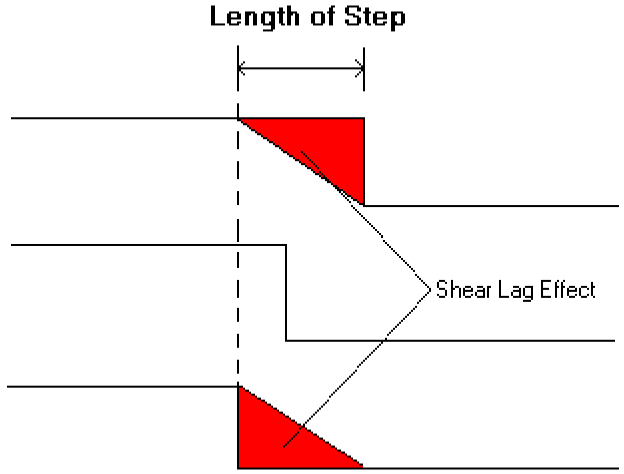


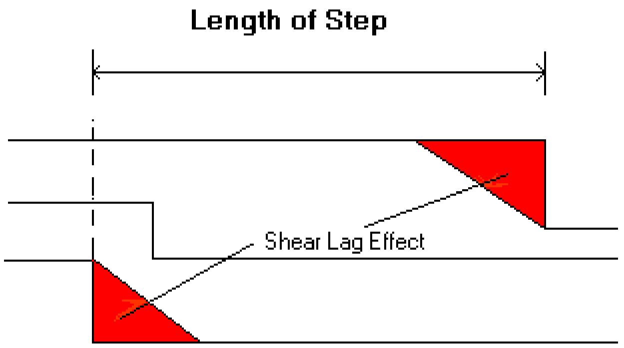

Flexible Step/Rigid Step

Option to tell RAPT how to treat the step in design. Options are

1. Flexible Step:- The default setting. RAPT will treat the area within the length of the step as a normal design area. At the minimum, design locations will be placed immediately inside each end of the step. If the step is sufficiently long extra design locations will be placed between these points. Calculations for flexure and shear will be carried out at all of these design points including checks on minimum reinforcement.
2. Rigid Step:- RAPT will treat the area within the length of the step as an inflexible area and will not carry out design at any of the design points within this length. Curvature over this length will be assumed to be zero for deflection calculations. This option should only be selected in cases where, after shear lag is considered, the member will be basically a constant section shape over the length of the step, as shown in the left diagram below. In longer steps, as shown in the right diagram below, the shear lag effect only affects the section shape at the ends of the step zone and, over the remainder of the length of the step zone, the full depth of the concrete section is active and must be treated as such.

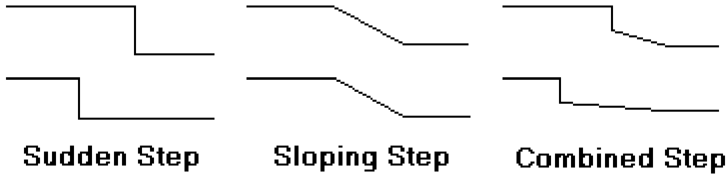
For a length to be calculated there must be a vertical step in both the top surface of the top layer and the bottom surface of the bottom layer. Otherwise the step will be assumed to have no length and all sections around the step will be treated as flexible. The length of a step is the length between the vertical step location in the top surface of the top layer and the vertical step location in the bottom surface of the bottom layer (see below).





7.2.3.7.1 Vertical Steps / Tapers

Vertical steps can be made in the top or bottom surface of any trapezoidal concrete layer or element and to the diameter and depth to centroid of circular elements. These can be sudden steps, sloping steps or combined sudden and sloping steps as shown below. At a step location, the designer can place steps in any of the concrete layer surfaces and any of the concrete element surfaces that exist at that location thus producing the complex steps shown below. This can be done by manually entering all of the step data in the Data Grid for each surface as described in the Data Definition below, or by using the Toolbar functions to add specific step types or by a combination of the two methods where the step dimensions are different in different layers.



Vertical Steps Toolbar

This toolbar offers the designer the means of quickly defining four different types of vertical steps using a single set of step data to place steps in several surfaces at once. Once the steps have been created using the toolbar functions the designer is free to modify the step dimensions to allow for any variations from the standard steps defined here as described in the Data Definition section below. The advantages to using these toolbar functions to add steps are

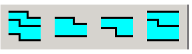
1. One set of data automatically adds the step data for steps in multiple surfaces
2. The recalculations required in loads, prestress and reinforcement to cater for the concrete shape changes are only performed once at the end of the creation of the steps rather than as every number is inserted as the

steps are defined manually as described below in Data Definition. Thus tendon re-profiling is only done for the final concrete shape rather than the series of shapes produced as the individual pieces of data are entered manually.

The diagrams at the bottom of this document show the data requirements for the dialogs shown below to define the different types of vertical steps. The four basic step groups are

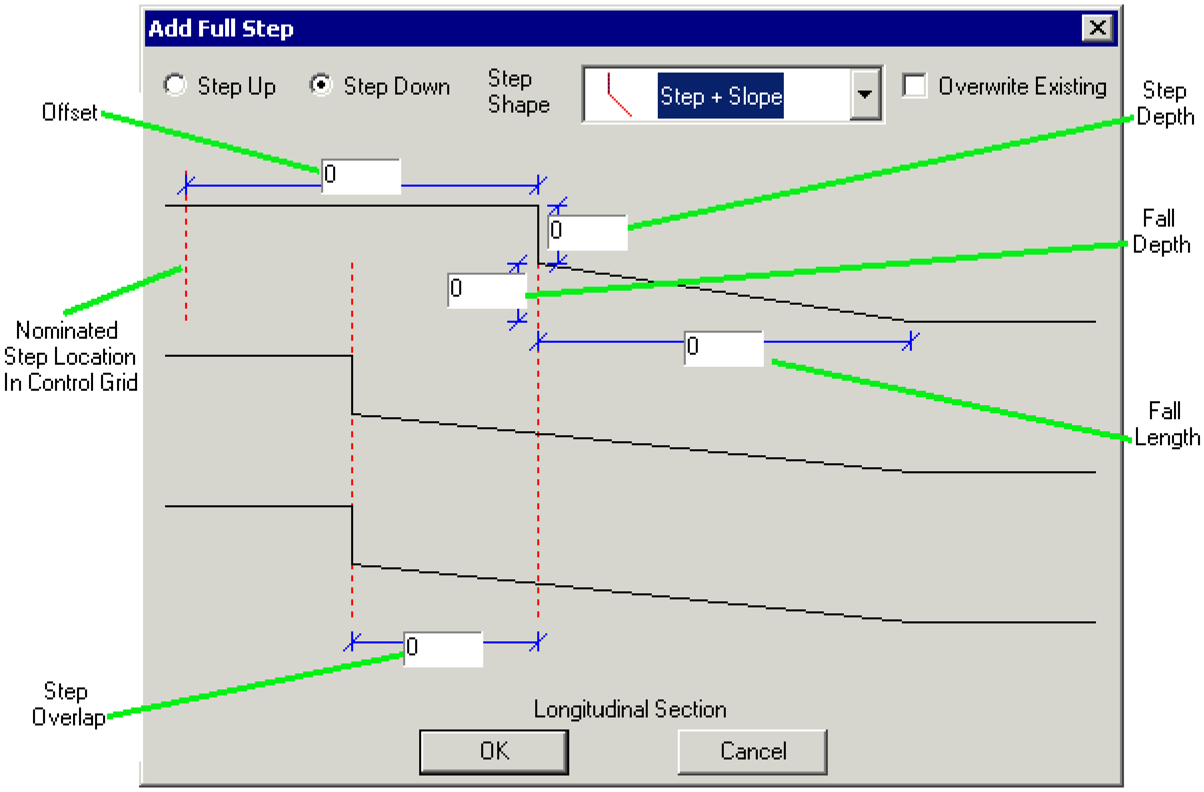
1. Step/Fall Down, Fall to the right of the step
2. Step/Fall Up, Fall to the right of the step
3. Step/Fall Up, Fall to the left of the step
4. Step/Fall Down, Fall to the left of the step

These diagrams show the effects of different methods of defining the data and the resulting step shapes.



Add Full Step

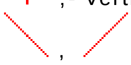
To step the full member section automatically as shown below. Allows the designer to define a step in the top and bottom surfaces of all of the layers and elements at the currently selected step location in the 7.2.3.7 Steps Control Grid. The following dialog will be presented for the designer to define the parameters of the step as discussed below. The diagram in the dialog is a longitudinal section and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested.

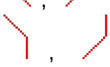


The data required is defined below.

- Step Direction:- The options available are Step Up or Step Down.
- Step Shape:- Four different basic step are possible.

 ;- Vertical step, down or up.

 :- Sloping Step, down or up.

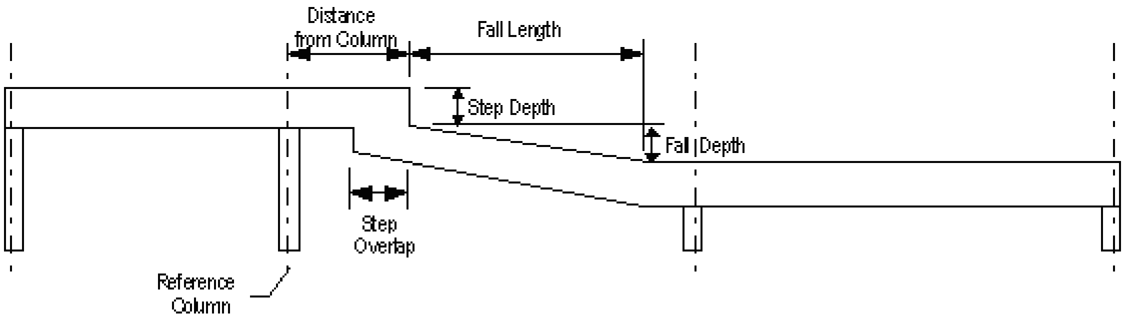
 :- Combined step with sloping section before vertical step, down or up.

 :- Combined step with vertical step before sloping section, down or up.

- Offset:- The distance to the sudden step location in the top layer from the nominated Step Location in the 7.2.3.7 Steps Control Grid.
- Step Depth:- The Vertical step dimension. The Value for this is always positive. Depending on the selection for Step Direction, RAPT will then create the step data from this value where Positive Values indicate a Downward Step and Negative Values indicate an Upward Step. The Sudden step in the top surface of the top

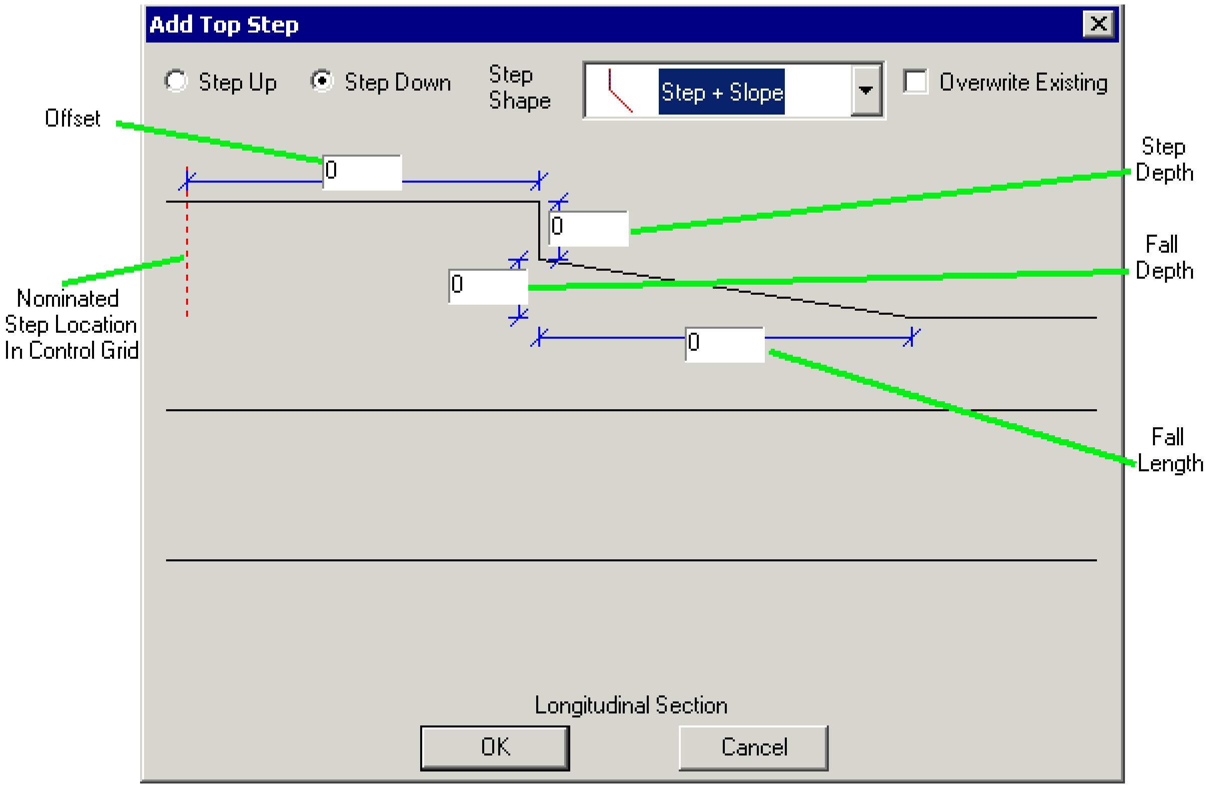
layer will be placed at the Offset from nominated Step Location in the 7.2.3.7 Steps Control Grid. The sudden step location in all other layers will be placed at the Step Overlap dimension from the sudden step location.

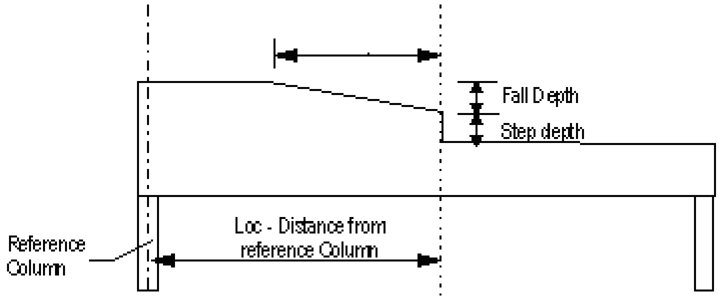
- Fall Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Direction and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.
- Fall Depth:- The Vertical dimension of the taper. The value is always positive. Depending on the selection for Step Direction and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Downward slope to the right and Negative Values indicate an Upward slope to the right.
- Step Overlap:- Allows users to specify an overlap length (a distance between the front and back of a step) with the top surface of the top layer having its step position at the defined step location and all other layers having their step location offset by the overlap dimension. See below. If no step overlap is specified, then all layers will have their step location at the defined step position. RAPT will only allow positive values to be input in this field. RAPT will determine which direction to apply the overlap. See diagrams at the bottom of this document.



Add Top Step

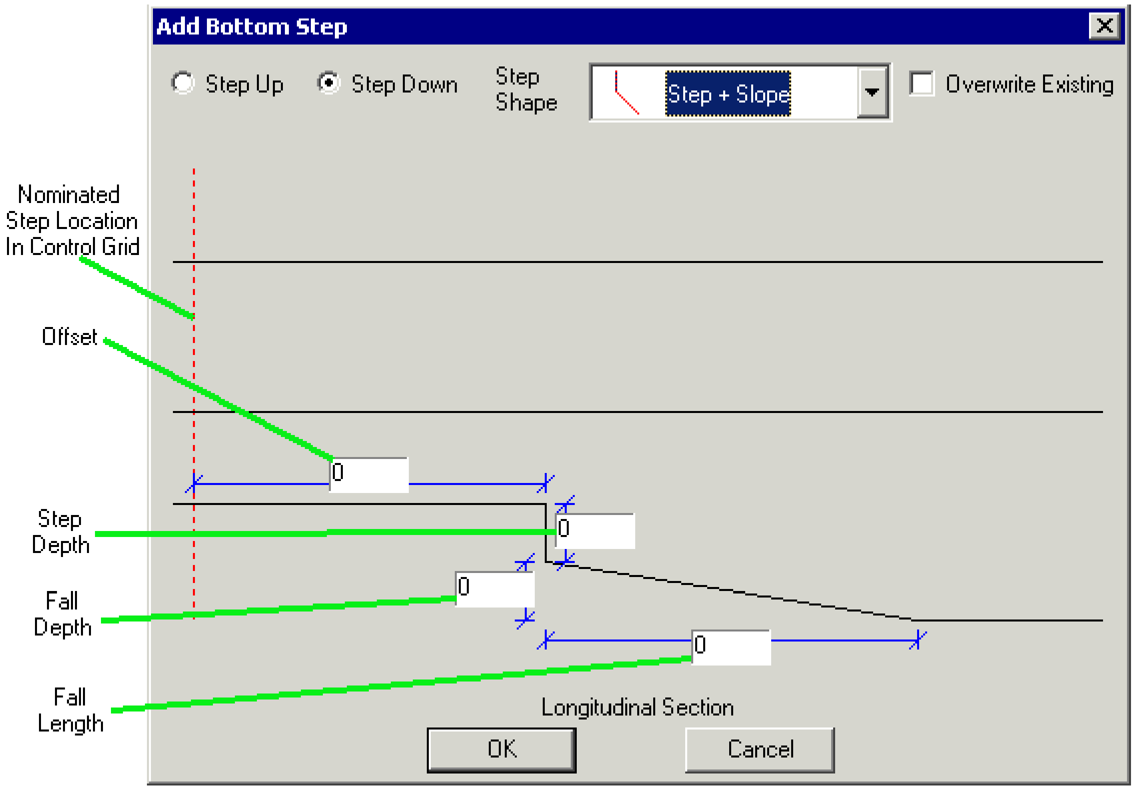
To place a step in the top surface of the top layer automatically as shown below. The following dialog will be presented for the designer to define the parameters of the step as discussed below. The diagram in the dialog is a longitudinal section and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested. For a description of the data required for this dialog See the above discussion for Add Full Step.

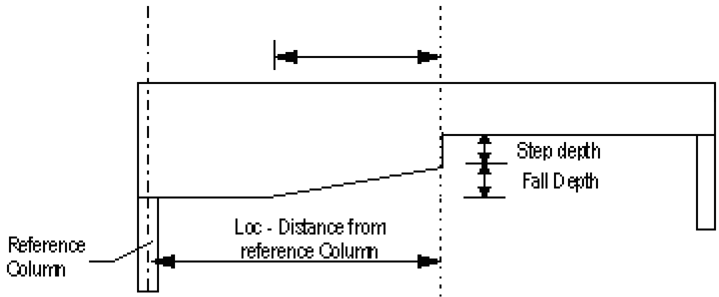





Add Bottom Step

To place a step in the bottom surface of the bottom layer automatically as shown below. The following dialog will be presented for the designer to define the parameters of the step as discussed below. The diagram in the dialog is a longitudinal section and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested. For a description of the data required for this dialog See the above discussion for Add Full Step.



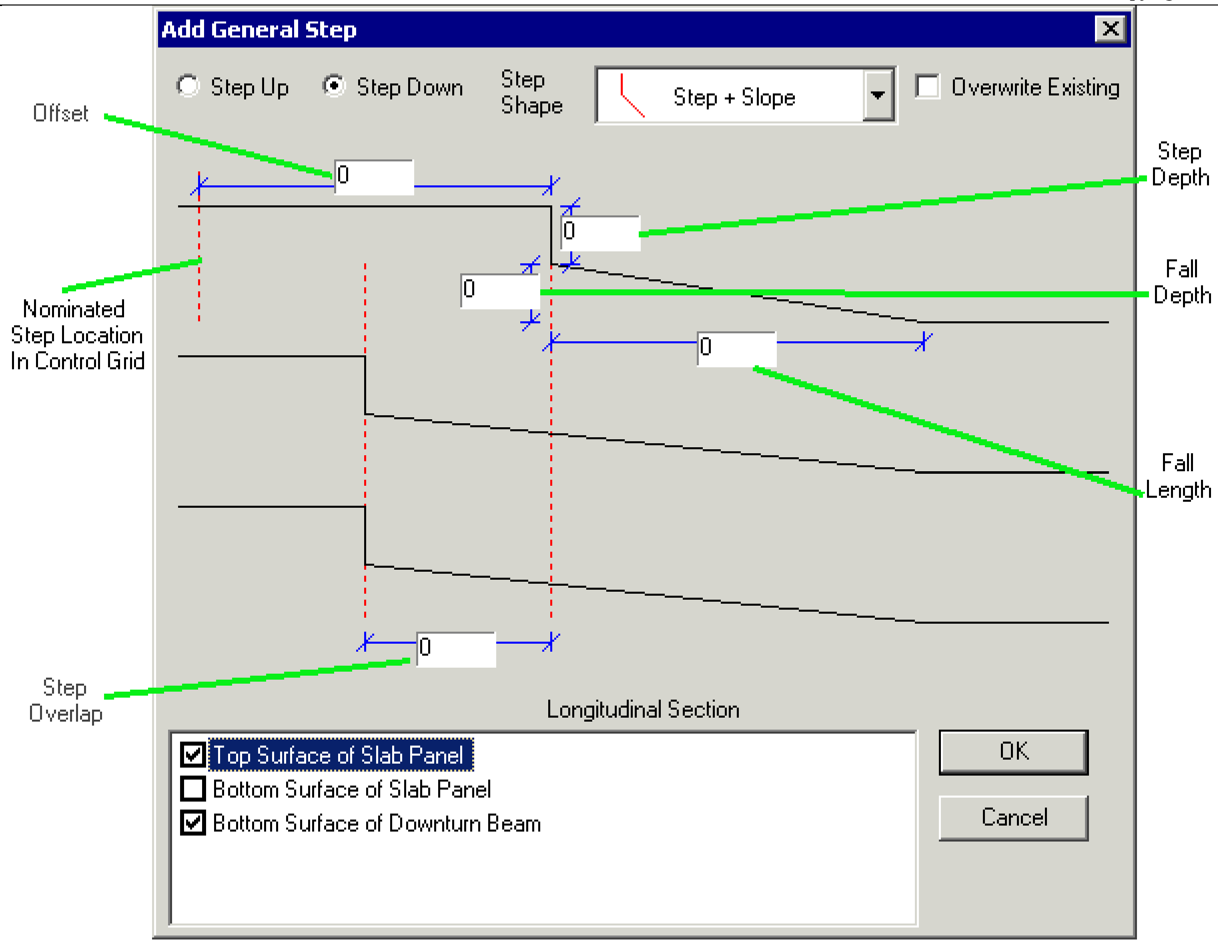



Add General Step

To place a step in any selected group of surfaces available at this step location. The following dialog will be presented for the designer to define the parameters of the step as discussed below. The diagram in the dialog is a longitudinal section and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested. For a description of the data required for this dialog See the above discussion for Add Full Step.

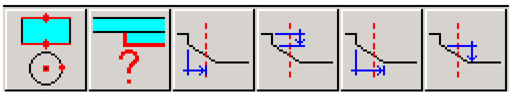
A list of the surfaces that can be stepped is offered at the bottom of the dialog. Select surfaces to be stepped by clicking the mouse in the check box to the left of the surface name. RAPT will add this step to any selected surfaces.

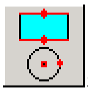
The Sudden step in the top surface of the top layer will be placed at the nominated Step Location in the 7.2.3.7 Steps Control Grid if this surface is selected to be stepped. The sudden step location in all other layers will be placed at the Step Overlap dimension from the nominated Step Location in the 7.2.3.7 Steps Control Grid.



Data Definition

All step attributes can be modified in this grid. The step profiles in each surface can be different. Whenever data is modified the recalculations required in loads, prestress and reinforcement to cater for the concrete shape changes are performed as every number is inserted and accepted as the steps are defined. Thus tendon re-profiling is done for the series of shapes produced as the individual pieces of data are entered manually. It is better to define general step shapes using the Toolbar functions described above and only to make the necessary modifications here for variations from the simple step information offered in the toolbar functions.

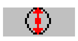


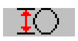
Shape Attribute to step

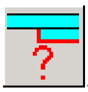
The attribute of the layer of the layer or element that is to be stepped. For trapezoidal shapes, the attribute could be the top or bottom surface. For circular shapes, the diameter or the vertical location of the centre of the circle can be stepped. This data cell cannot be edited by the designer. RAPT determines the options available from the layer and element data already defined.

 Step the top surface of this trapeziodal concrete layer or element. A positive step is downwards.

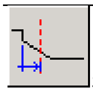
 Step the bottom surface of this trapeziodal concrete layer or element. A positive step is downwards.

 Step the location of the centroid of this circular element. A positive step is downwards.

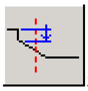
 Step the diameter of this circular element. A positive step will increase the diameter.

Step this attribute of this shape

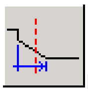
Indicates whether there is to be a step in this surface. If yes is selected, the remainder of the data cells for this surface will be available to define the step profile data.

Step Offset

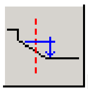
The distance to the sudden step location in this layer from the Nominated Step Location in the 7.2.3.7 Steps Control Grid. If there is only a tapered step, the distance is to the point where the tapered step is dimensioned from.

Step Depth

The depth of the sudden step in this layer. A positive value results in a step down at this point.

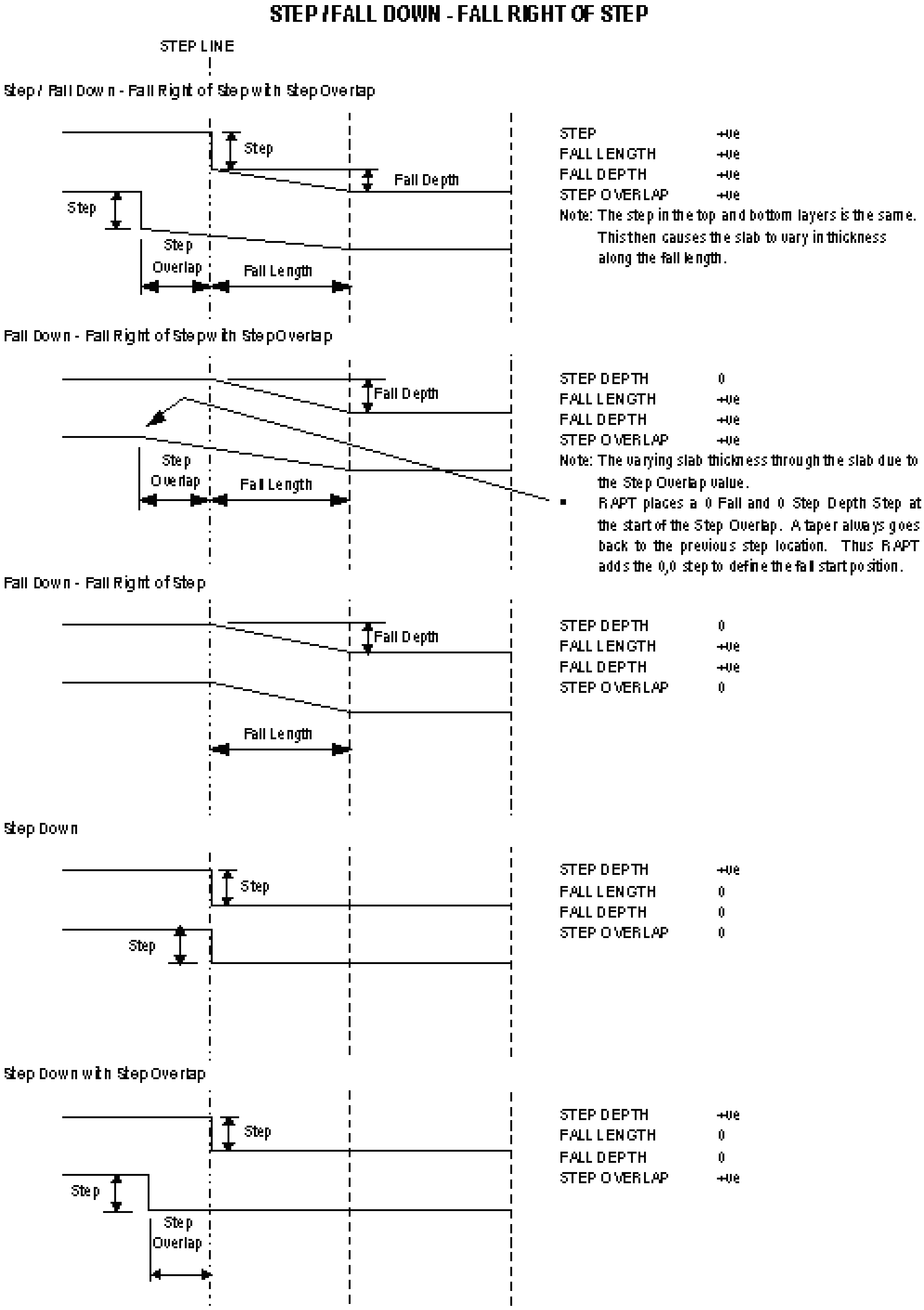
Fall/Taper Length

The length over which the sloping step falls. A negative value will place the sloping step to the left of the Step Offset location and a positive value will place it to the right of the Step Offset location.

Fall Depth

The depth of the taper/fall in this layer. A positive value results in a slope down to the right.

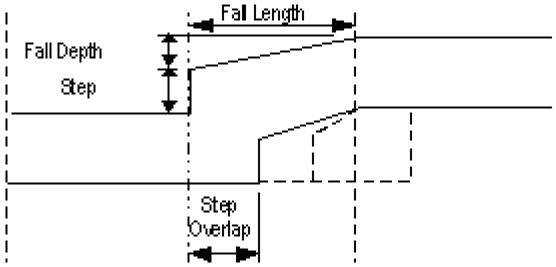
Step Shape Diagrams



STEP / FALL UP - FALL RIGHT OF STEP

STEP LINE

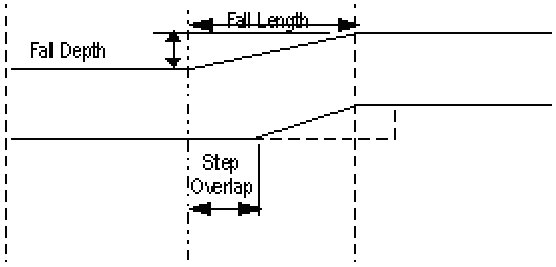
Step / Fall Up - Fall Right of Step with Step Overlap



STEP DEPTH -ve
FALL LENGTH +ve
FALL DEPTH -ve
STEP OVERLAP +ve

Note: If STEP OVERLAP ≥ FALL LENGTH then the Step and taper are replaced buy a single step New STEP DEPTH = FALL DEPTH + Input STEP DEPTH

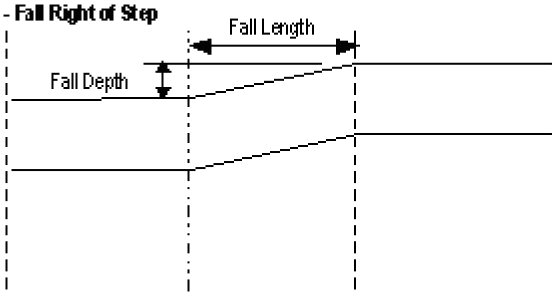
Fall Up - Fall Right of Step with Step Overlap



STEP DEPTH 0
FALL LENGTH +ve
FALL DEPTH -ve
STEP OVERLAP +ve

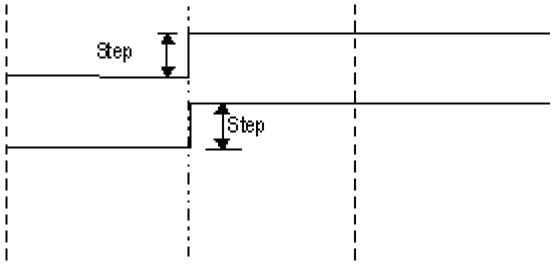
Note: If STEP OVERLAP > FALL LENGTH then RAPT replaces the STEP DEPTH with the FALL DEPTH. The FALL DEPTH is then set to 0.

Fall Up - Fall Right of Step



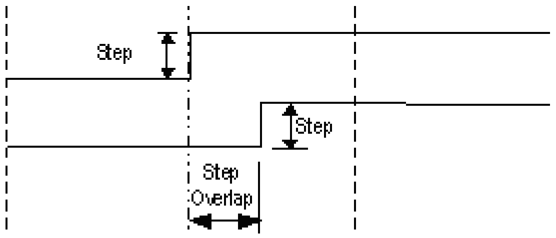
STEP DEPTH 0
FALL LENGTH +ve
FALL DEPTH -ve
STEP OVERLAP 0

Step Up



STEP DEPTH -ve
FALL LENGTH 0
FALL DEPTH 0
STEP OVERLAP 0

Step Up with Step Overlap

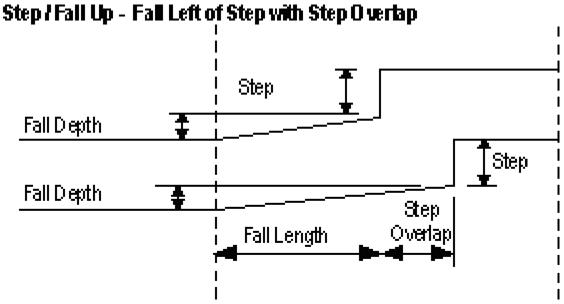


STEP DEPTH -ve
FALL LENGTH 0
FALL DEPTH 0
STEP OVERLAP +ve

STEP / FALL UP - FALL LEFT OF STEP

STEP LINE

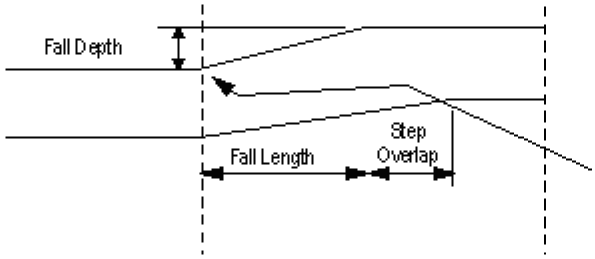
Step / Fall Up - Fall Left of Step with Step Overlap



STEP	-ve
FALL LENGTH	-ve
FALL DEPTH	-ve
STEP OVERLAP	+ve

Note: The step in the top and bottom layers is the same. This then causes the slab to vary in thickness along the fall length.

Fall Up - Fall Left of Step with Slab Overlap

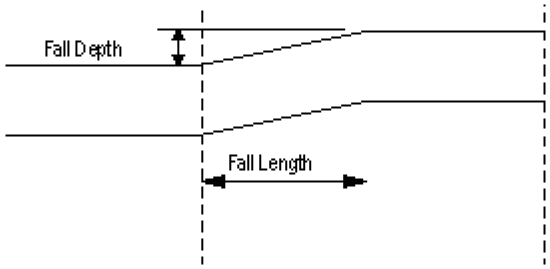


STEP DEPTH	0
FALL LENGTH	-ve
FALL DEPTH	-ve
STEP OVERLAP	+ve

Note: The varying slab thickness through the slab due to the Step Overlap value.

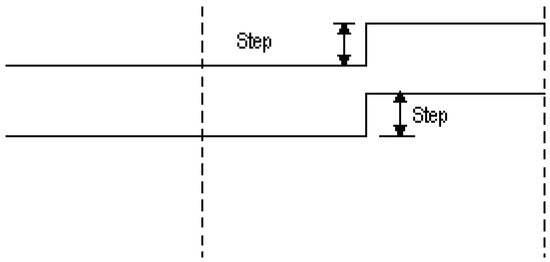
- RAPT places a 0 Fall and 0 Step Depth Step at the start of the Step Overlap. A taper always goes back to the previous step location. Thus RAPT adds the 0,0 step to define the taper start position.

Fall Up - Fall Left of Step



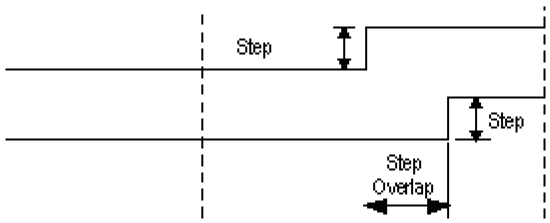
STEP DEPTH	0
FALL LENGTH	-ve
FALL DEPTH	-ve
STEP OVERLAP	0

Step Up

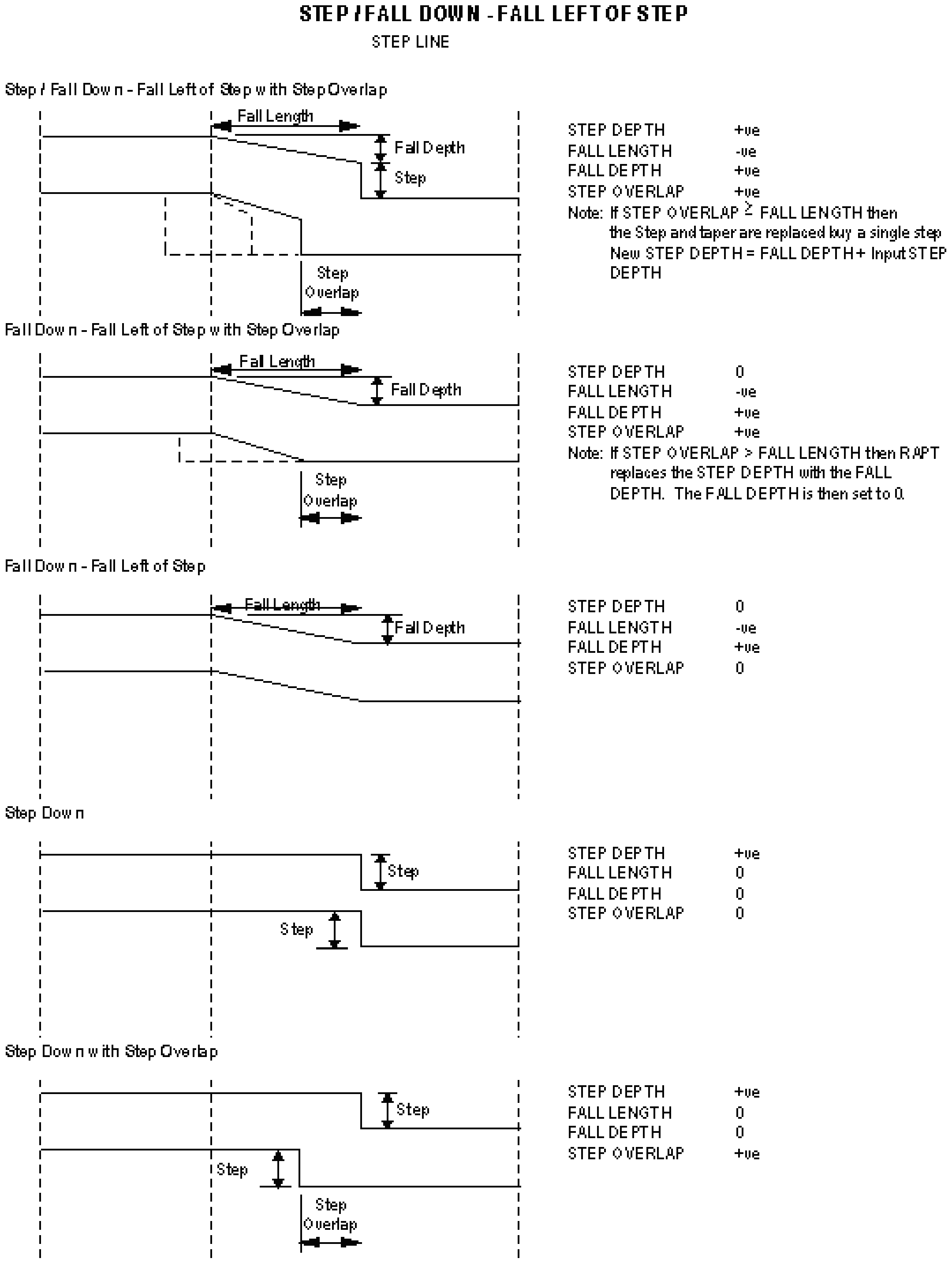


STEP DEPTH	-ve
FALL LENGTH	0
FALL DEPTH	0
STEP OVERLAP	0

Step Up with Slab Overlap



STEP DEPTH	-ve
FALL LENGTH	0
FALL DEPTH	0
STEP OVERLAP	+ve



7.2.3.7.2 Horizontal Steps / Tapers

Horizontal steps can be made in each corner of any trapezoidal concrete layer or element and to the horizontal location of the centroid of circular elements. These can be sudden steps, sloping steps or combined sudden and sloping steps. At a step location, the designer can place steps in any of the concrete layer corners and any of the concrete element corners that exist at that location thus producing complex steps. This can be done by manually entering all of the step data in the Data Grid for each shape attribute that is to be stepped as described in the Data Definition below, or by using the Toolbar functions to add specific step types or by a combination of the two methods where the step dimensions are different in different layers.

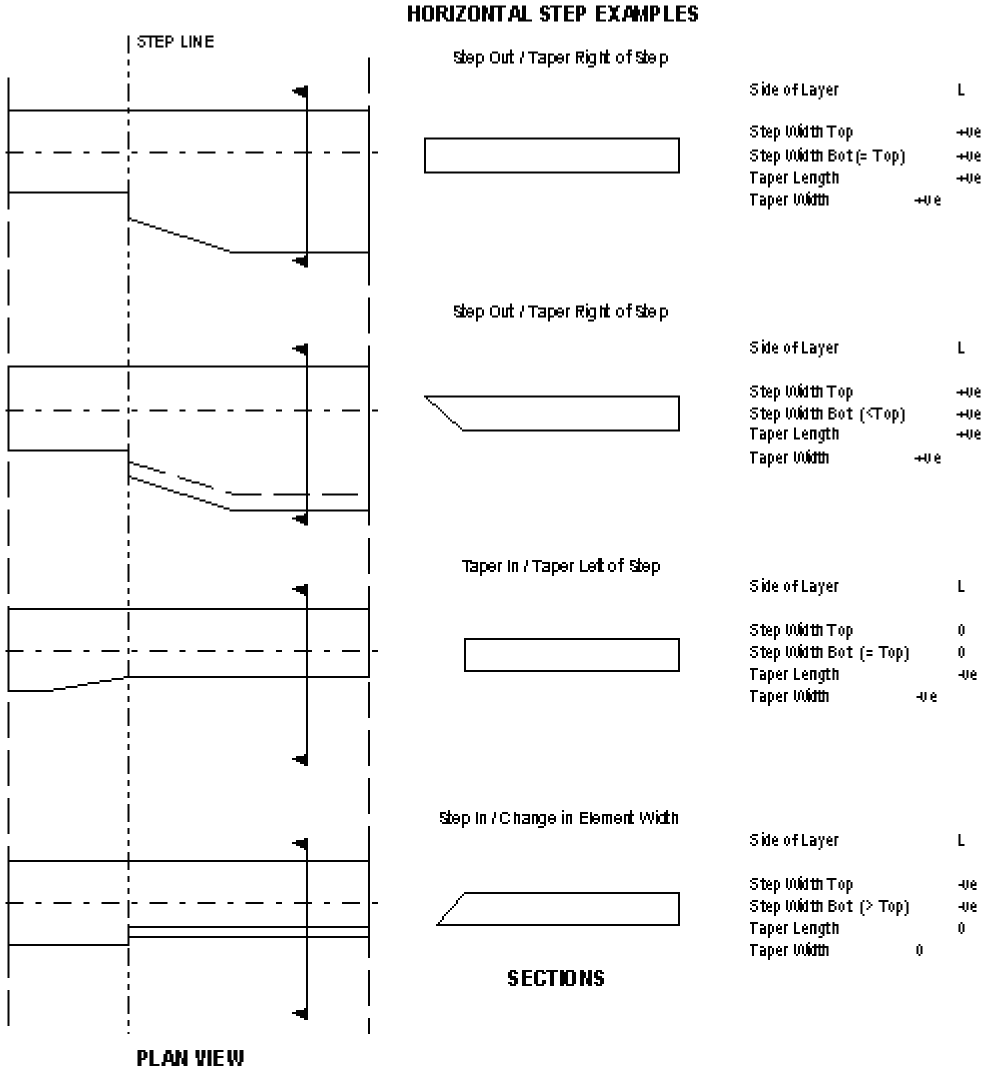
Horizontal Steps Toolbar

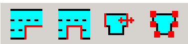
This toolbar offers the designer the means of quickly defining four different types of horizontal steps using a single set of step data to place steps in several surfaces at once. Once the steps have been created using the toolbar functions the designer is free to modify the step dimensions to allow for any variations from the standard steps defined here as described in the Data Definition section below. The advantages to using these toolbar functions to add steps are

1. One set of data automatically adds the step data for steps in multiple surfaces

2. The Step To functions allow the sides of different layers to be step by different amounts to a specific width as would be required where penetrations eat into the sides of beams etc.
3. The recalculations required in loads, prestress and reinforcement to cater for the concrete shape changes are only performed once at the end of the creation of the steps rather than as every number is inserted as the steps are defined manually as described below in Data Definition. Thus tendon re-profiling is only done for the final concrete shape rather than the series of shapes produced as the individual pieces of data are entered manually.

The diagram below shows the data requirements for the dialogs shown below to define the different types of vertical steps.

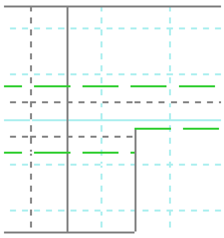


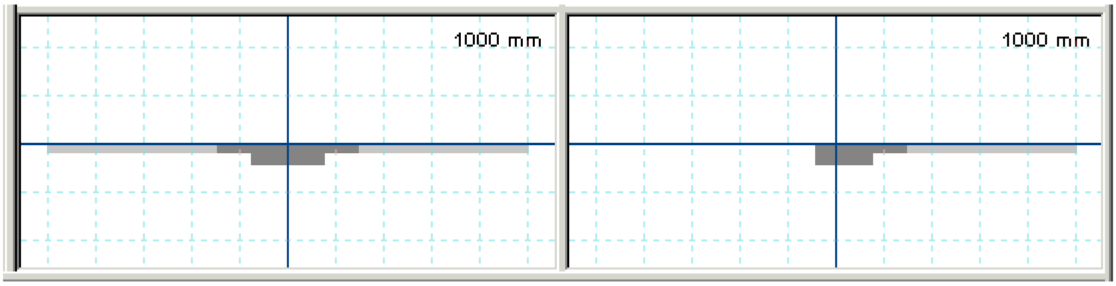


Full Horizontal Step to Point

To step the edge of the concrete to a new horizontal location automatically as shown below. Allows the designer to define a step in the side of the slab panel at the currently selected step location in the 7.2.3.7 Steps Control Grid to a new width location (not by an amount). The diagram below shows a step in the left side in which the edge of the slab has been stepped to a location within the beam width. Because the slab step has crossed both the edge of the beam and the effective flange width of the beam, they have both been stepped to the same point automatically. The cross-sections show the shape either side of the step.

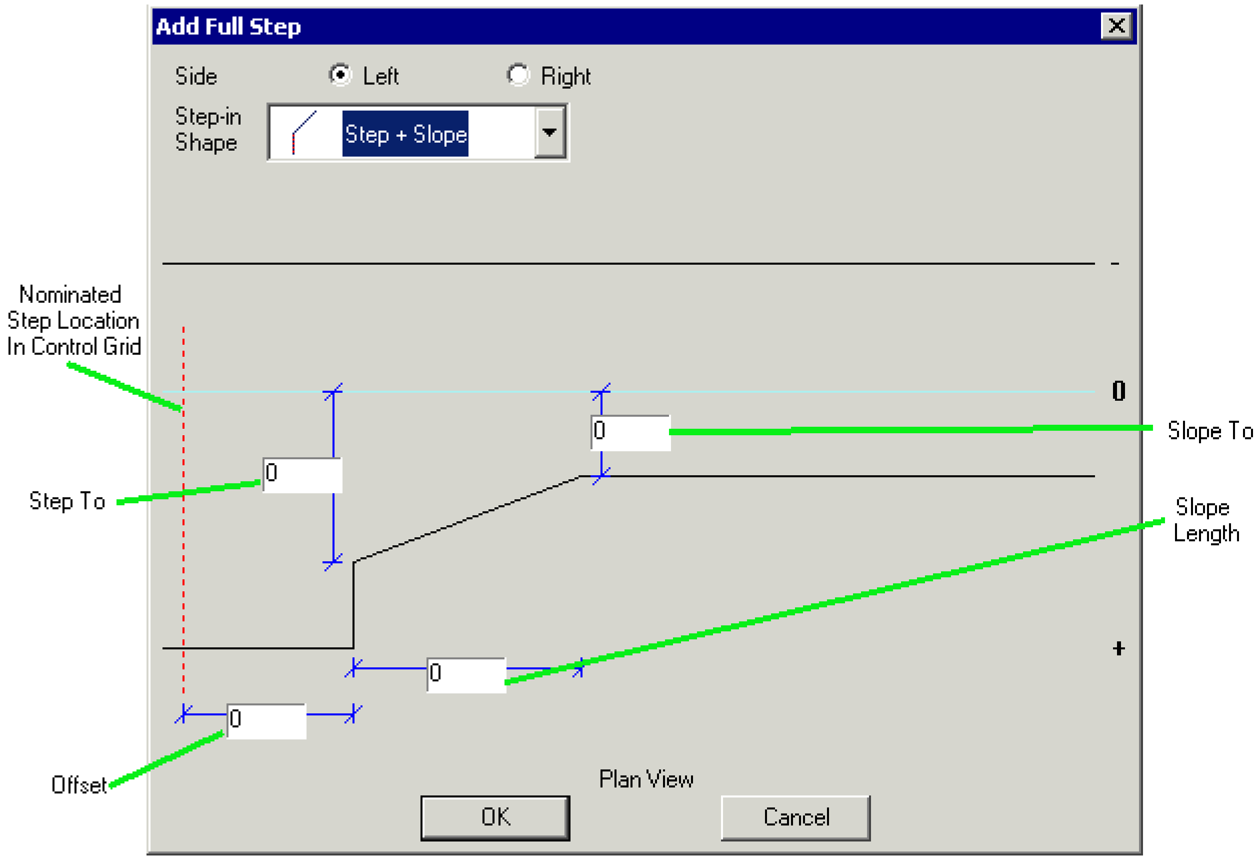
This icon will only be available if the current step in the Steps Control Grid has no horizontal steps in any layers on at least one side of the concrete shape at that step. A full step will only be able to be added on a side that currently has no horizontal steps in any layers at this step.





The step in the side of the slab must reduce the width of the slab. The Step To point cannot be outside the width of the slab. The step may be a sudden step, a sloping step or a combination of the two. The two options available for combined step shapes are shown below for a step in the left edge

The following dialog will be presented for the designer to define the parameters of the step as discussed below. The diagram in the dialog is a plan view and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested. The dialog shows a step in the left side.



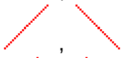
The data required is defined below.

1. Side:- The side which is to be stepped. The side is determined when looking from the right end of the frame (in plan left is at the bottom of the plan). See plan view below. Only the sides that currently have no

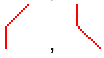
horizontal steps in any layers at this step will be available to select. Unavailable sides will be dulled and will not be able to be selected.

2. Step-in Shape:- The options available are shown below. The two pictures in each case are for a step in the left side and the right side respectively.

 ;- Sudden step

 :- Sloping Step on left/right side

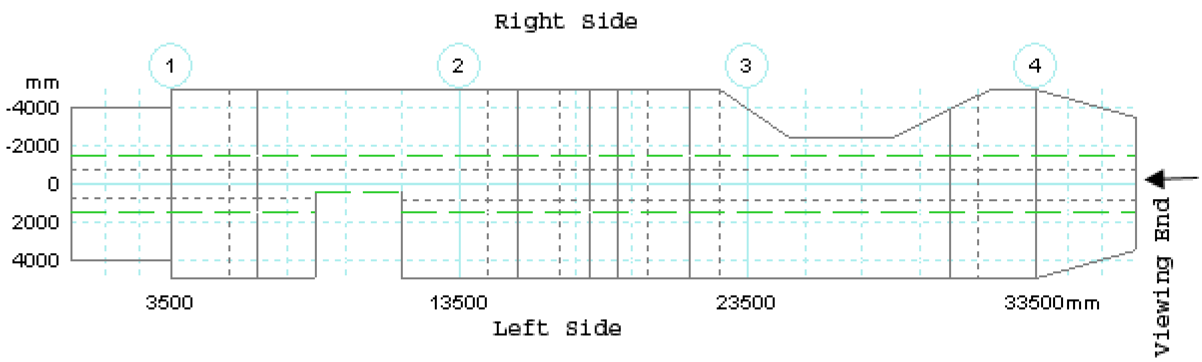
 :- Combined step with sloping section before sudden step on left/right side

 :- Combined step with sudden step before sloping section on left/right side

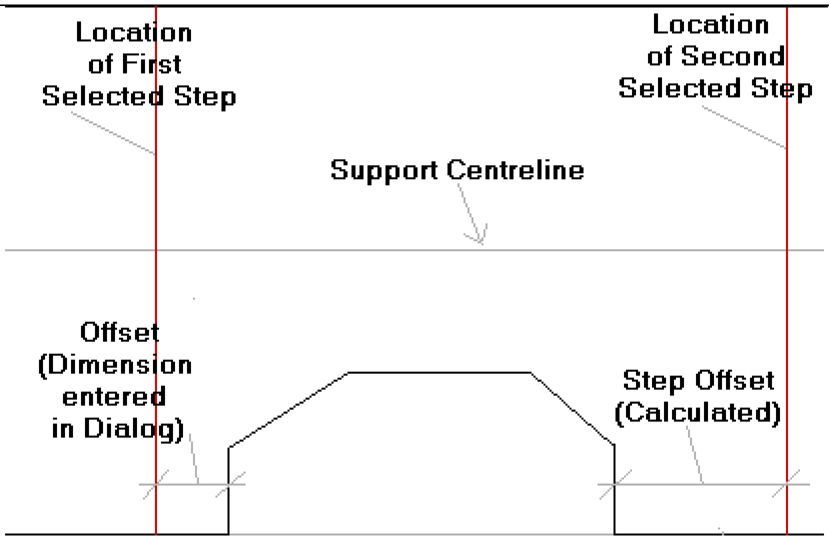
3. Offset:- Distance from the nominated Step Location in the 7.2.3.7 Steps Control Grid to the point at which the sudden step will be placed.
4. Step To:- Sudden step in the side of the concrete panel to this horizontal location. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below. The Sudden step in the side surface is placed at the Offset defined above from the nominated Step Location in the 7.2.3.7 Steps Control Grid.
5. Slope Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Side and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.
6. Slope to:- The horizontal location to slope the side of the concrete panel to. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below.

Full Horizontal Rebate to Point

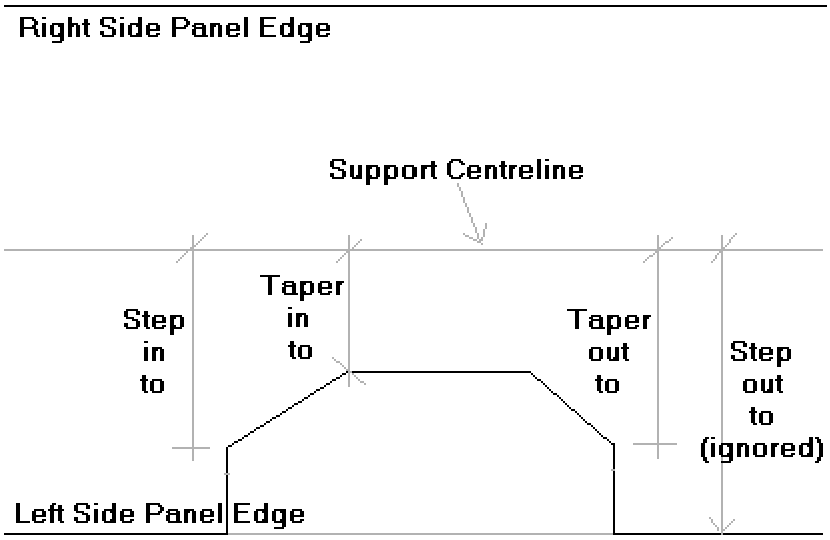
To place a rebate in the edge of the concrete automatically as shown below. Allows the designer to define a rebate in the side of the slab panel at the currently selected step location in the 7.2.3.7 Steps Control Grid. The diagram below shows a square sided rebate in the left side between grids 1 and 2 in which the edge of the slab has been stepped to a location within the beam width. Because the slab step has crossed both the edge of the beam and the effective flange width of the beam, they have both been stepped to the same point automatically. It also shows a sloping sided rebate in the right edge between grids 3 and 4.



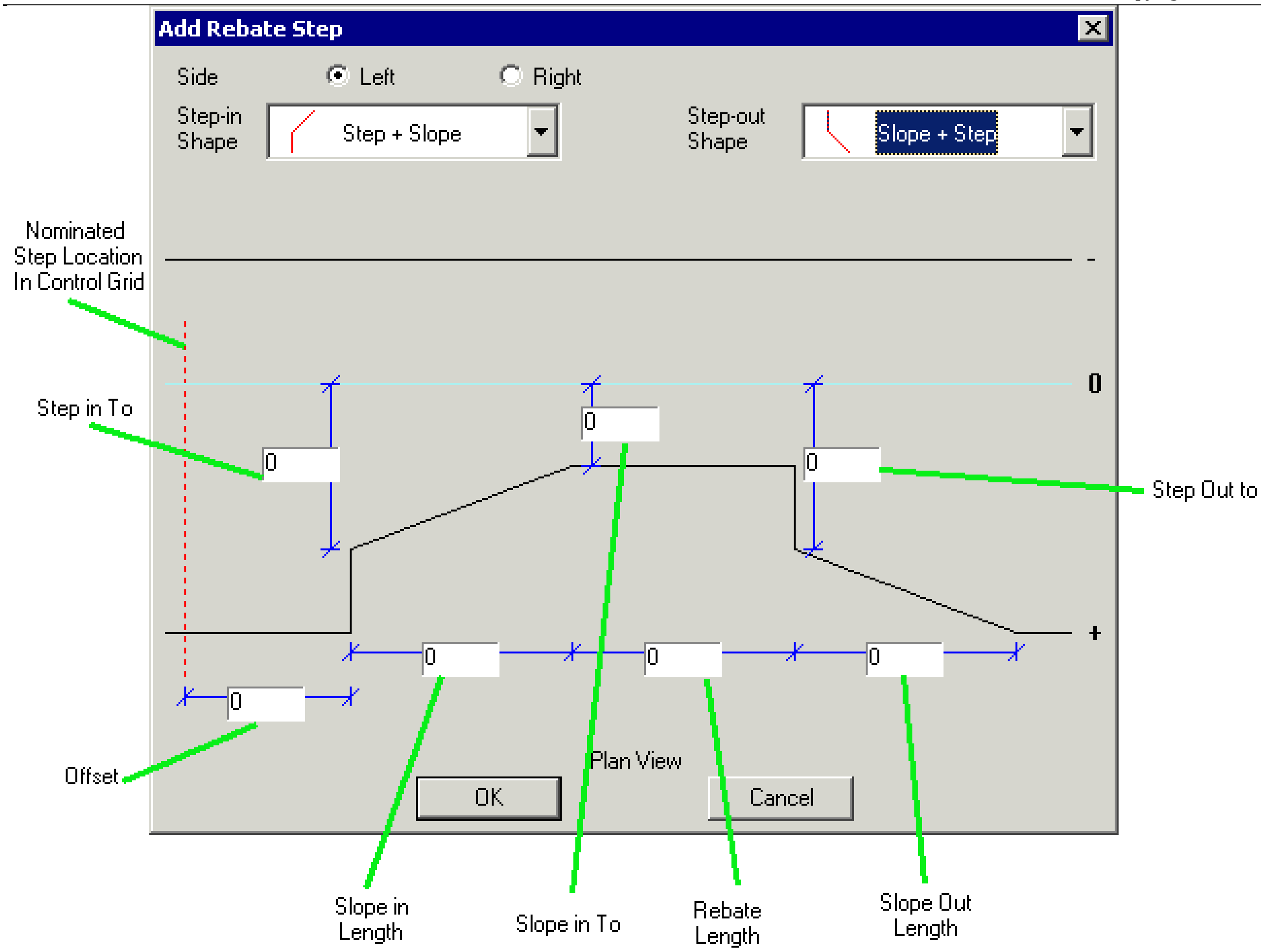
To define a rebate in the edge of a slab, two step locations must be nominated, the first (leftmost) to cater for the step in and the second (rightmost) for the step back out. In the 7.2.3.7 Steps Control Grid, these are selected by left clicking the mouse in the step number cell for the first step location and then using Ctrl Left click in the step number cell for the second step. Once two step locations are selected in this way, move program focus to the Horizontal Step data view by clicking the Horizontal tab and the Rebate icon will be available if the selected steps in the Steps Control Grid have no horizontal steps in any layers on at least one side of the concrete shape at that step (the same side must be available for both steps). A Rebate will only be able to be added on a side that currently has no horizontal steps in any layers at these steps.



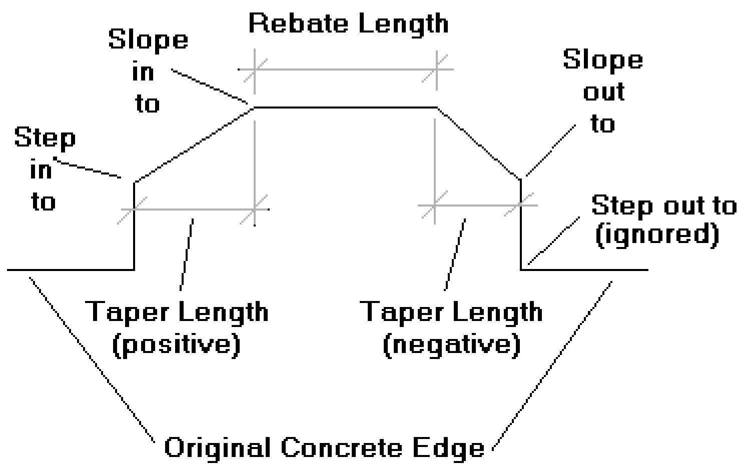
The rebate location will be determined by the location of the first step selected. The Sudden Step into the concrete edge will be placed at the Offset dimension from the Nominal Step Location defined by the left hand step of the two selected. RAPT will then determine the Step Offset dimensions for the steps in each surface to match the Nominal Step Location defined in the second step selected for the return of the rebate to the original concrete surface. (See above).

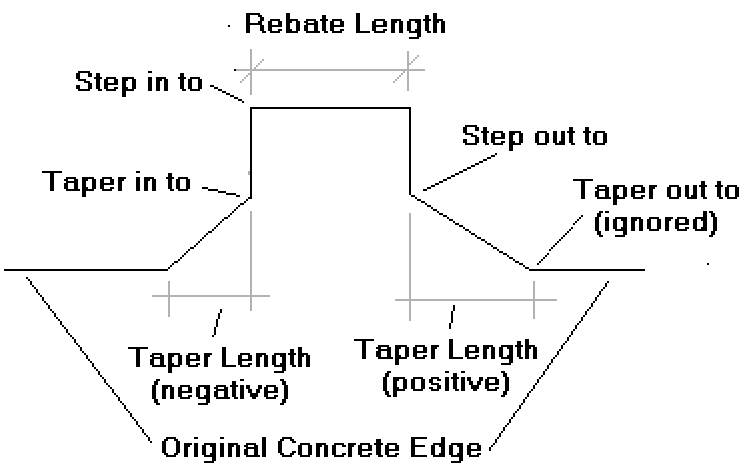


The dialog below is presented to the designer for the input of the data describing the step shape. The diagram in the dialog is a plan view and is diagrammatic only. It is not meant to be a true representation of your structure shape. It will adjust for the Step Direction and Step Shape selected to indicate the shape of step requested. The dialog shows a rebate in the left side.



The data required is defined below. The first half of the data (5 to item 7) defines the rebate shape into the side of the concrete panel. The last half of the data (from item 9 to 11) defines the rebate shape back out to the original side of the concrete panel. As the rebate starts and finishes at the existing concrete edge, the final step dimension is not required to reach the outside surface, RAPT will determine this automatically from the existing shape.





1. Side:- The side which is to be stepped. The side is determined when looking from the right end of the frame (in plan left is at the bottom of the plan). See plan view above. Only the sides that currently have no horizontal steps in any layers at the two selected steps will be available to select. Unavailable sides will be dulled and will not be able to be selected.
2. Step-in Shape:- The options available are shown below. The two pictures in each case are for a step in the left side and the right side respectively.

   ;- Sudden step

   , :- Sloping Step on left /right side

   , :- Combined step with sloping section before sudden step on left /right side

   , :- Combined step with sudden step before sloping section on left /right side
3. Step-out Shape:- The options available are shown below. The two pictures in each case are for a step out the left side and the right side respectively.

   ;- Sudden step

   , :- Sloping Step on left /right side

   , :- Combined step with sloping section before sudden step on left /right side

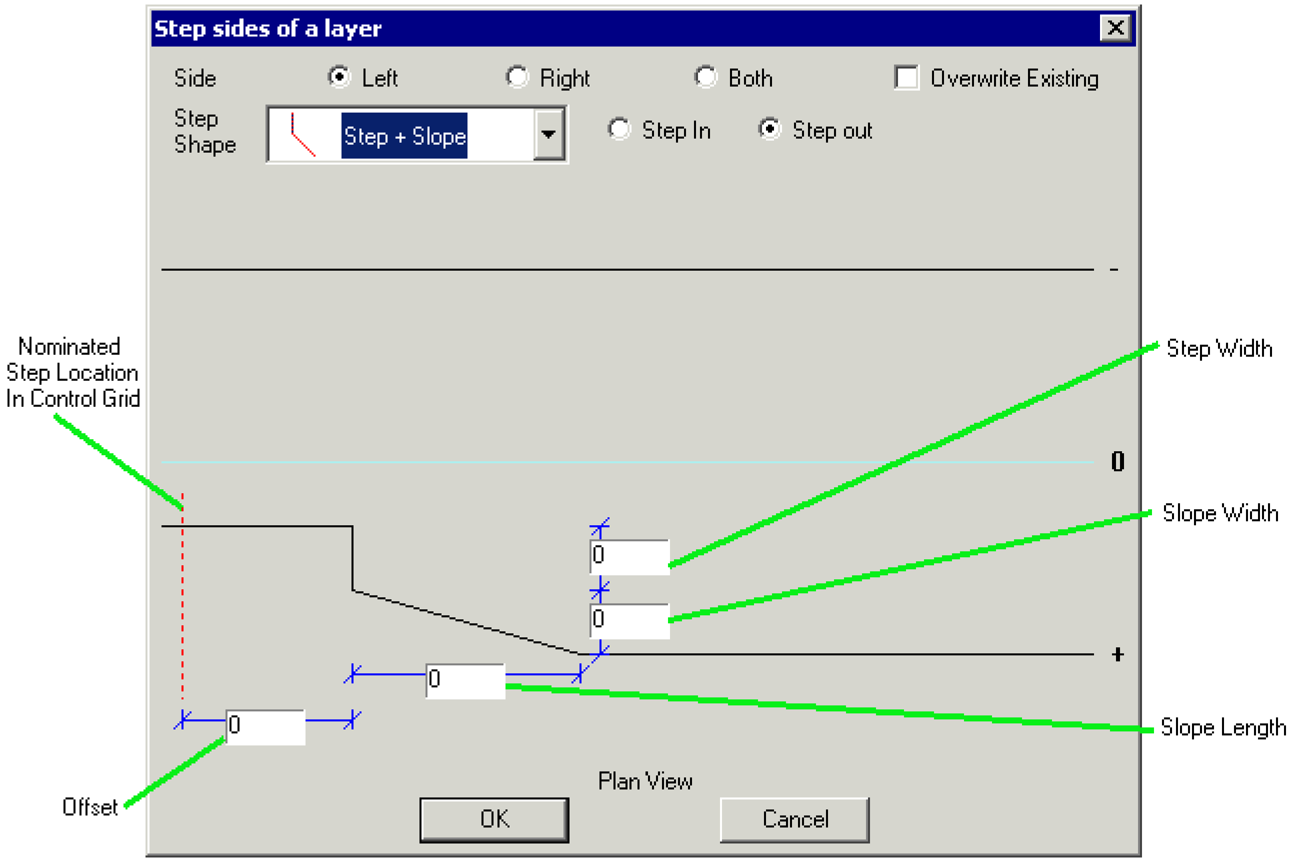
   , :- Combined step with sudden step before sloping section on left /right side
4. Offset:- Distance from the nominated Step Location in the 7.2.3.7 Steps Control Grid to the point at which the sudden step will be placed.
5. Step In To:- Sudden step in the side of the concrete panel to this horizontal location. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below. The Sudden step in the side surface is placed at the Offset defined above from the nominated Step Location in the 7.2.3.7 Steps Control Grid.
6. Taper In Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Side and Step Shape, RAPT will then create the step data from this value where Positive

Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.

7. Slope In To:- The horizontal location to slope the side of the concrete panel to. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below.
8. Rebate Length:- The length of the rebate parallel to the support line between the step/s in from the concrete edge and the step/s back out to the origin al concrete edge. Must be greater than or equal zero.
9. Taper Out Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Side and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.
10. Slope Out To:- The horizontal location to slope the side of the concrete panel to. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below. A value need only be placed here if there is a further sudden step from this point to the original edge of the concrete panel. Otherwise, the tapered step will automatically be dimensioned to the original concrete edge.
11. Step Out To:- The horizontal location to step the side of the concrete panel to. The location is relative to the support centreline. The locations are positive on the left side of the support line and negative on the right side when looking from the right end of the frame. See plan view below. The Sudden step in the side surface is placed at the Offset defined above from the nominated Step Location in the 7.2.3.7 Steps Control Grid. A value need only be placed here if there is a further tapered step from this point to the original edge of the concrete panel. Otherwise, the sudden step will automatically be dimensioned to the original concrete edge.

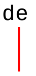

Step sides of a layer

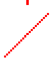
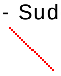
To place a step in one or both sides of a concrete layer. Allows the designer to define a step in the side/s of a layer at the currently selected step location in the 7.2.3.7 Steps Control Grid. Simply select a row belonging to the layer and RAPT will identify the layer to step from this.

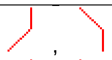


The data required is defined below.

1. Side:- The side/s which are to be stepped. The side is determined when looking from the right end of the frame (in plan left is at the bottom of the plan). See plan view below. The step can be applied to Left side, Right side or Both sides.
2. Step Shape:- The options available are shown below. The two pictures in each case are for a step in the left side and the right side respectively.

 ;- Sudden step

 ,  :- Sloping Step on left /right side

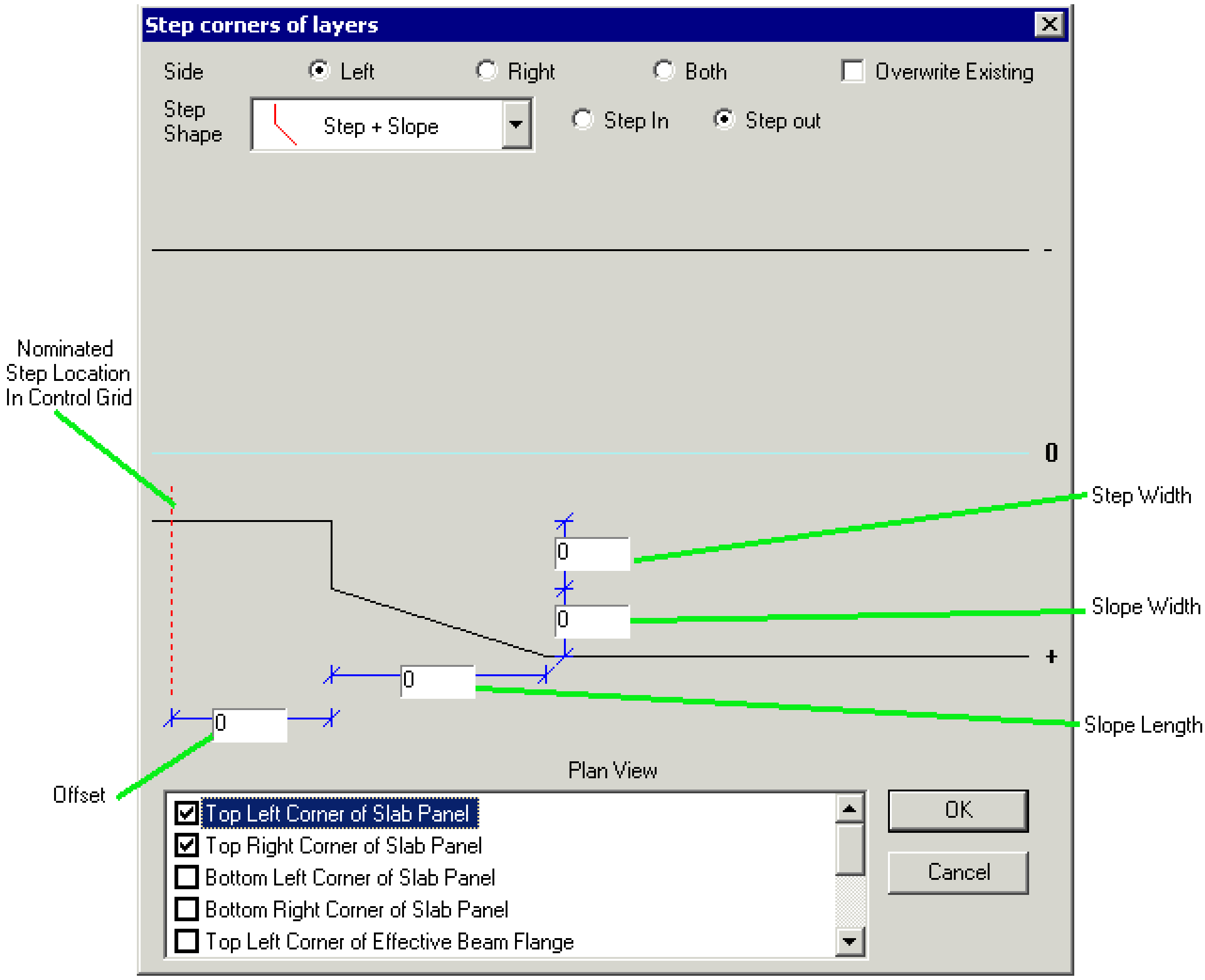
 :- Combined step with sloping section before sudden step on left /right side

 :- Combined step with sudden step before sloping section on left /right side

3. Offset:- Distance from the nominated Step Location in the 7.2.3.7 Steps Control Grid to the point at which the sudden step will be placed.
4. Step Width:- The width of the sudden step in each side. A positive step will increase the width while a negative step will reduce the width. The Sudden step in the side surface is placed at the Offset defined above from the nominated Step Location in the 7.2.3.7 Steps Control Grid.
5. Slope Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Side and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.
6. Slope Width:- The width change to occur over the Taper length. A positive value will increase the width while a negative value will reduce the width. The direction of the sloping step must be the same as the direction of the sudden step

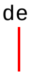
Step corners of layers

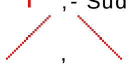
To place a step in a user selection of the corners of any concrete layers and elements at this step. Allows the designer to define a step in the corner/s of any layer at the currently selected step location in the 7.2.3.7 Steps Control Grid.

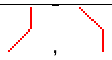


The data required is defined below.

1. Side:- The side/s which are to be stepped. The side is determined when looking from the right end of the frame (in plan left is at the bottom of the plan). See plan view below. The step can be applied to Left side, Right side or Both sides.
2. Step Shape:- The options available are shown below. The two pictures in each case are for a step in the left side and the right side respectively.

 ;- Sudden step

 :- Sloping Step on left /right side

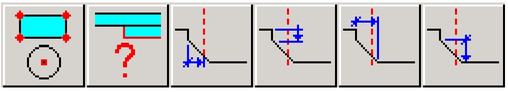
 :- Combined step with sloping section before sudden step on left /right side

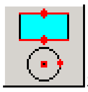
 :- Combined step with sudden step before sloping section on left /right side

3. Offset:- Distance from the nominated Step Location in the 7.2.3.7 Steps Control Grid to the point at which the sudden step will be placed.
4. Step Width:- The width of the sudden step in each side. A positive step will increase the width while a negative step will reduce the width. The Sudden step in the side surface is placed at the Offset defined above from the nominated Step Location in the 7.2.3.7 Steps Control Grid.
5. Slope Length:- The length over which the taper or slope acts. The value is always positive. Depending on the selection for Step Side and Step Shape, RAPT will then create the step data from this value where Positive Values indicate a Taper to the right of the step location and Negative Values indicate a Taper to the left of the step location.
6. Slope Width:- The width change to occur over the Taper length. A positive value will increase the width while a negative value will reduce the width. The direction of the sloping step must be the same as the direction of the sudden step
7. List of Shape Attributes to step:- A list of the shape attributes that can be stepped is offered at the bottom of the dialog. Select shape attributes to be stepped by clicking the mouse in the check box to the left of the relevant name. RAPT will add this step to any selected shape attributes.

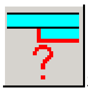
Data Definition

All step attributes can be modified in this grid. The step profiles for each corner/shape attribute can be different. Whenever data is modified the recalculations required in loads, prestress and reinforcement to cater for the concrete shape changes are performed as every number is inserted and accepted as the steps are defined. Thus tendon re-profiling is done for the series of shapes produced as the individual pieces of data are entered manually. It is better to define general step shapes using the Toolbar functions described above and only to make the necessary modifications here for variations from the simple step information offered in the toolbar functions.

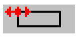


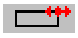
Shape Attribute

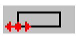
The attribute of the layer of the layer or element that is to be stepped. For trapezoidal shapes, the attribute would one of the corners of the shape or the effective width of the panel width. For circular shapes, the horizontal location of the centre of the circle can be stepped. This data cell cannot be edited by the designer. RAPT determines the options available from the layer and element data already defined.

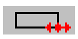
Step this attribute of this shape

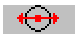
Indicates whether there is to be a step in this shape attribute. If yes is selected, the remainder of the data cells for this shape attribute will be available to define the step profile data.

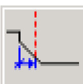
 Step the top left corner of this trapeziodal concrete layer or element. A positive step increases the width of the section.

 Step the top right corner of this trapeziodal concrete layer or element. A positive step increases the width of the section.

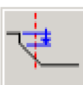
 Step the bottom left corner of this trapeziodal concrete layer or element. A positive step increases the width of the section.

 Step the bottom right corner of this trapeziodal concrete layer or element. A positive step increases the width of the section.

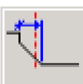
 Step the centroid of this circular element. A positive step moves the element to the right.

Step Offset

The distance to the sudden step location in this shape attribute from the nominated Step Location in the 7.2.3.7 Steps Control Grid.

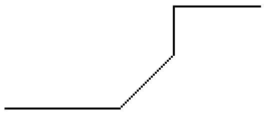
Step Width

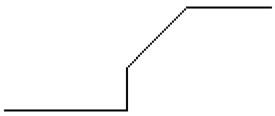
The width of the sudden step in this shape attribute. A positive value results in an increase in width.

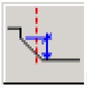
Taper Length

The length over which the sloping step falls. A negative value will place the sloping step to the left of the Step Offset

location as shown in the left diagram below and a positive value will place it to the right of the Step Offset location as shown in the right diagram below .

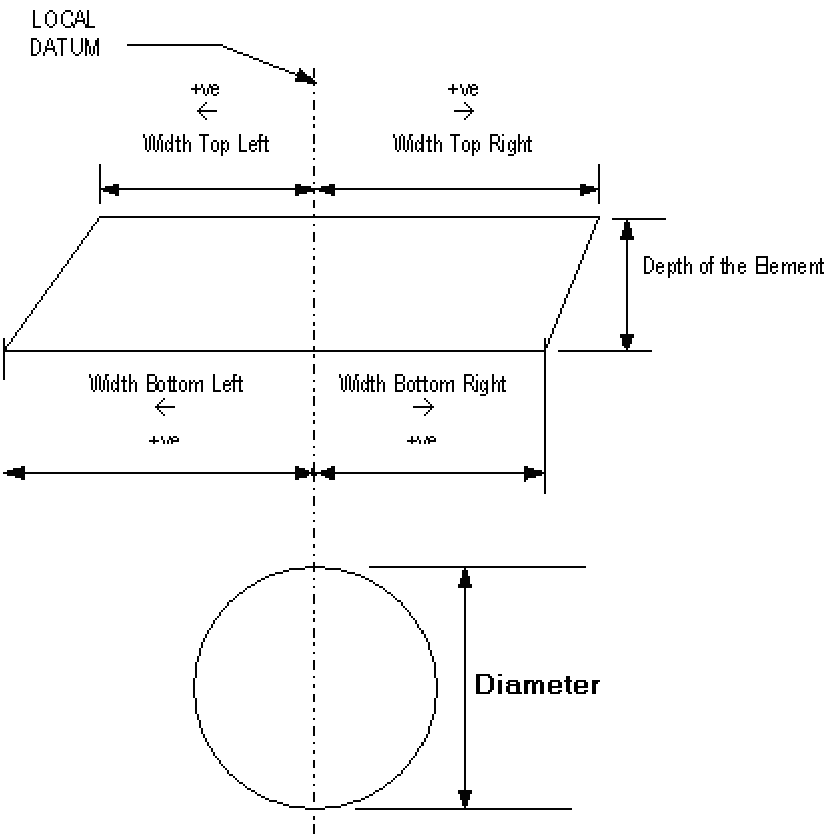




Fall Width

The width of the taper/fall in this layer. A positive value results in a slope outwards (increasing width) to the right.

7.2.3.8 Elements



In RAPT, concrete elements are local shapes added to the concrete layers. These can be

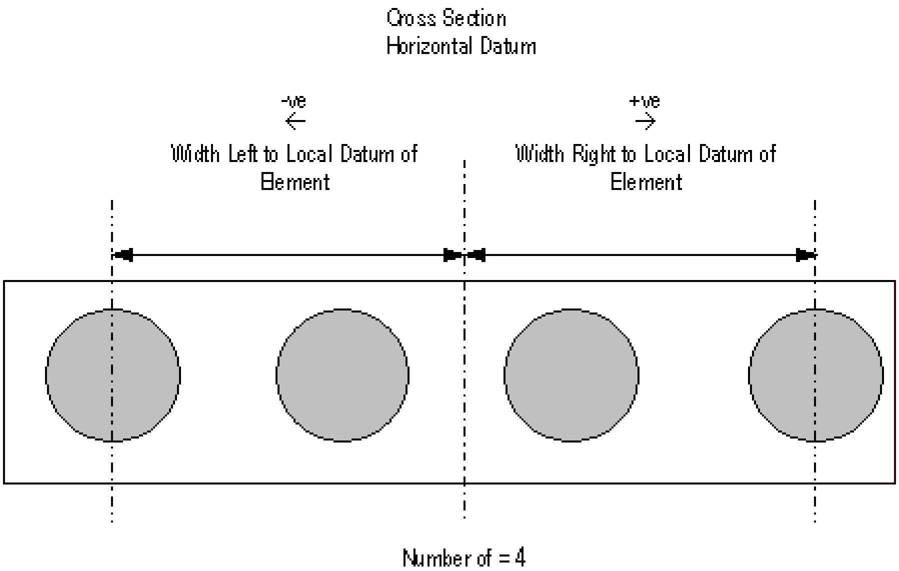
1. 7.2.3.8.1 Trapezium:- trapezoidal shape with horizontal, parallel top and bottom surfaces. The width of a top or bottom face can be zero to produce a triangular shape.
2. 7.2.3.8.2 Circular

shapes and can be either

1. solids added to the outside edge of the concrete layers
2. voids added internally in the layers.

Elements do not need to extend the full length of the frame. The designer specifies a start and end location. They can be used to model effects such as

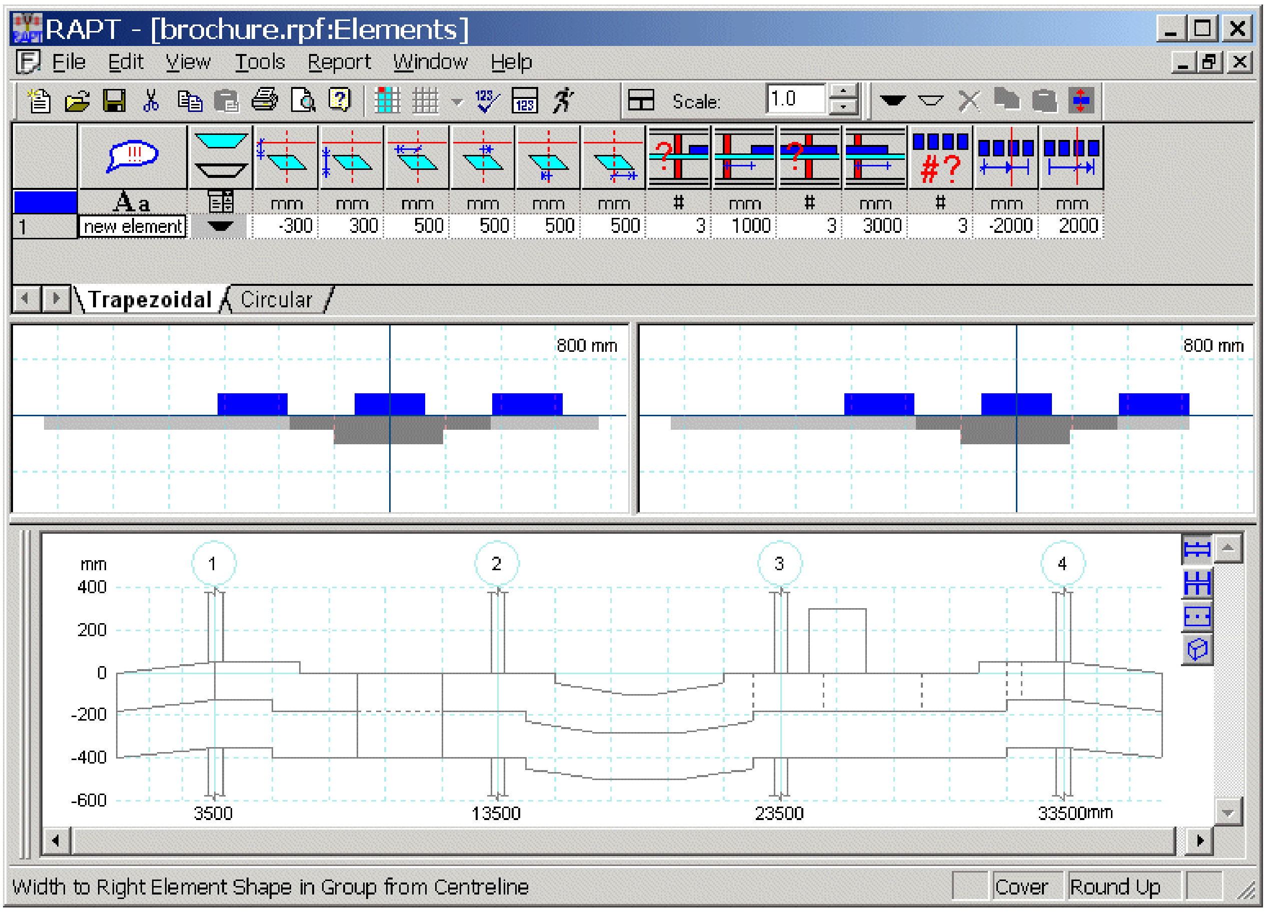
1. voided slabs (see below)
2. transverse penetrations in beams,
3. notches cut out of the sides of beams,
4. local increases in slab thickness,
5. set-downs in the tops of slabs,
6. etc.



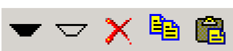
Steps in Elements

An element can vary in shape along it's length by using 7.2.3.7.1 Vertical and 7.2.3.7.2 Horizontal Steps in the same way as used for stepping the surfaces of layers and can be sudden or tapered. Using steps, an element can be reduced to zero depth thus removing the element over the length over which it's depth is zero. The width of an element should never be reduced to zero to achieve this.

7.2.3.8.1 Trapezoidal



Trapezoidal Element Toolbar



Add solid trapezoidal element

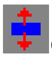
Add a new data row for a solid trapezoidal element. The elements will be inserted at a selected data row in the grid or added after the last data row in the grid if there is no selection.

Add void trapezoidal element
Add a new data row for a void trapezoidal element. The elements will be inserted at a selected data row in the grid or added after the last data row in the grid if there is no selection.

Delete trapezoidal elements
Delete one or more selected elements.

Copy:- Copy a selected group of elements for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any elements can be selected by clicking the element number with the left mouse button. Groups of elements can be selected by click and dragging the mouse. Extra elements can be added to the selected group using Ctrl + Left Mouse key. Changes to the element arrangement after the selection is made, by adding or removing elements or modifying concrete shapes, will void any selection.

Paste:- Paste the selected elements. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the element number cell, the copied elements will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted elements. If no data row is selected the copied elements will be added to the end of the existing elements. Multiple data rows cannot be selected for the paste operation. The paste can be into elements of the same type, e.g. trapezoidal elements can only be copied to trapezoidal elements. RAPT automatically creates the space in the data grid into which the elements are to be copied, so it is not necessary to create the rows for the data to be inserted into.

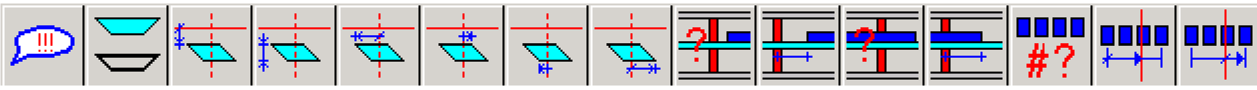
Graphic Input:- Allows the select element to be modified graphically. See Graphical Input below.

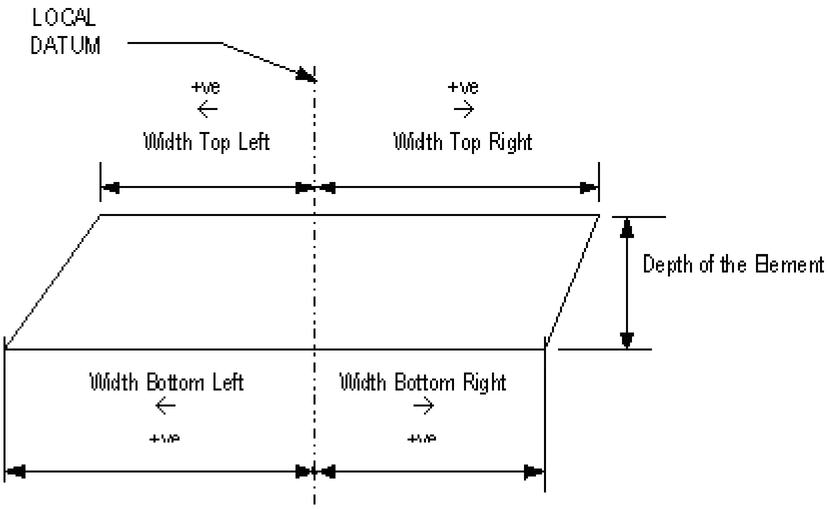
Section Views

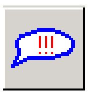
When the program focus is in the trapezoidal elements data view, the Section Graphics Windows

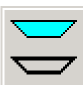
1. Left panel:- The shape of the elements defined by the current data row highlighted on the cross-section at the left end of the element.
2. Right Panel:- The shape of the elements defined by the current data row highlighted on the cross-section at the right end of the element.

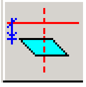
Data Definition



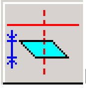


Description
Name of the element. The default name is "New Element". The designer should rename the element with a recognisable name.

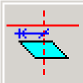
Solid / Void
Whether the element is a solid or a void shape. Solid elements should be fully outside the concrete shape and connected to the shape. Void elements should be fully within the solid concrete shape but may be connected to a surface thus forming a recess in the surface of the shape.

Depth Top

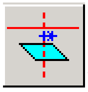
The depth to the top of the element from the top of slab datum. Positive is downwards.

Element Depth

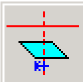
The overall depth of the element. It must be positive.

Width Top Left

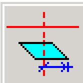
Width to the top left corner of the element from the Local Vertical Datum (see above).

Width Top Right

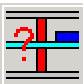
Width to the top right corner of the element from the Local Vertical Datum (see above).

Width Bottom Left

Width to the bottom left corner of the element from the Local Vertical Datum (see above).

Width Bottom Right

Width to the bottom right corner of the element from the Local Vertical Datum (see above).

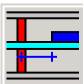
Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the element is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the element. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

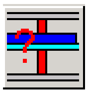
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of an element in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of an element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

Distance to Left End from Left Reference Column

The distance from the left reference column to the left end of the element. If the distance entered moves the element into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the start of the element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

Right End Reference Column

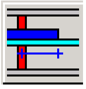
The 7.2.3.2 Column Grid Reference from which the right end of the element is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the element. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of an element in one operation rather than two

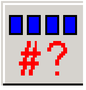
separate operations which would result in two complete calculations of the effect of moving the end location of an element.

Obviously the right end of an element must be to the right of the left end so make sure when modifying element ends that the order of modification will ensure this.

Distance to Right End from Right Reference Column

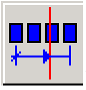
The distance from the right reference column to the right end of the element. If the distance entered moves the element into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the end of the element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

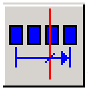


Number of Elements in Group

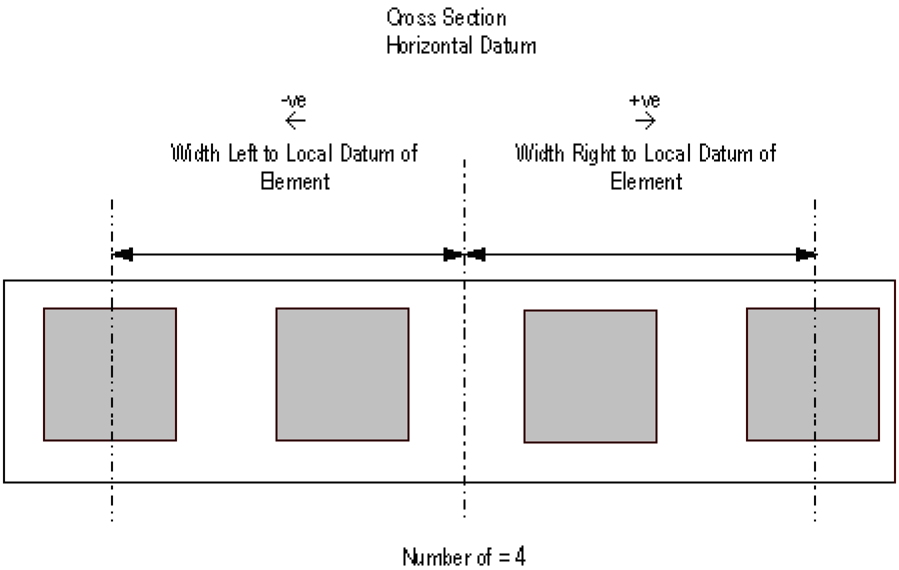
The number of this type of element to be placed at this location at an equal spacing across the width defined below (see below).

Width to Left Element Shape in Group from Centreline

The width from the centreline of the frame (line of supports) to the local datum of the leftmost element in the group. The dimension is negative to the left of the centreline and positive to the right (as shown below).

Width to Right Element Shape in Group from Centreline

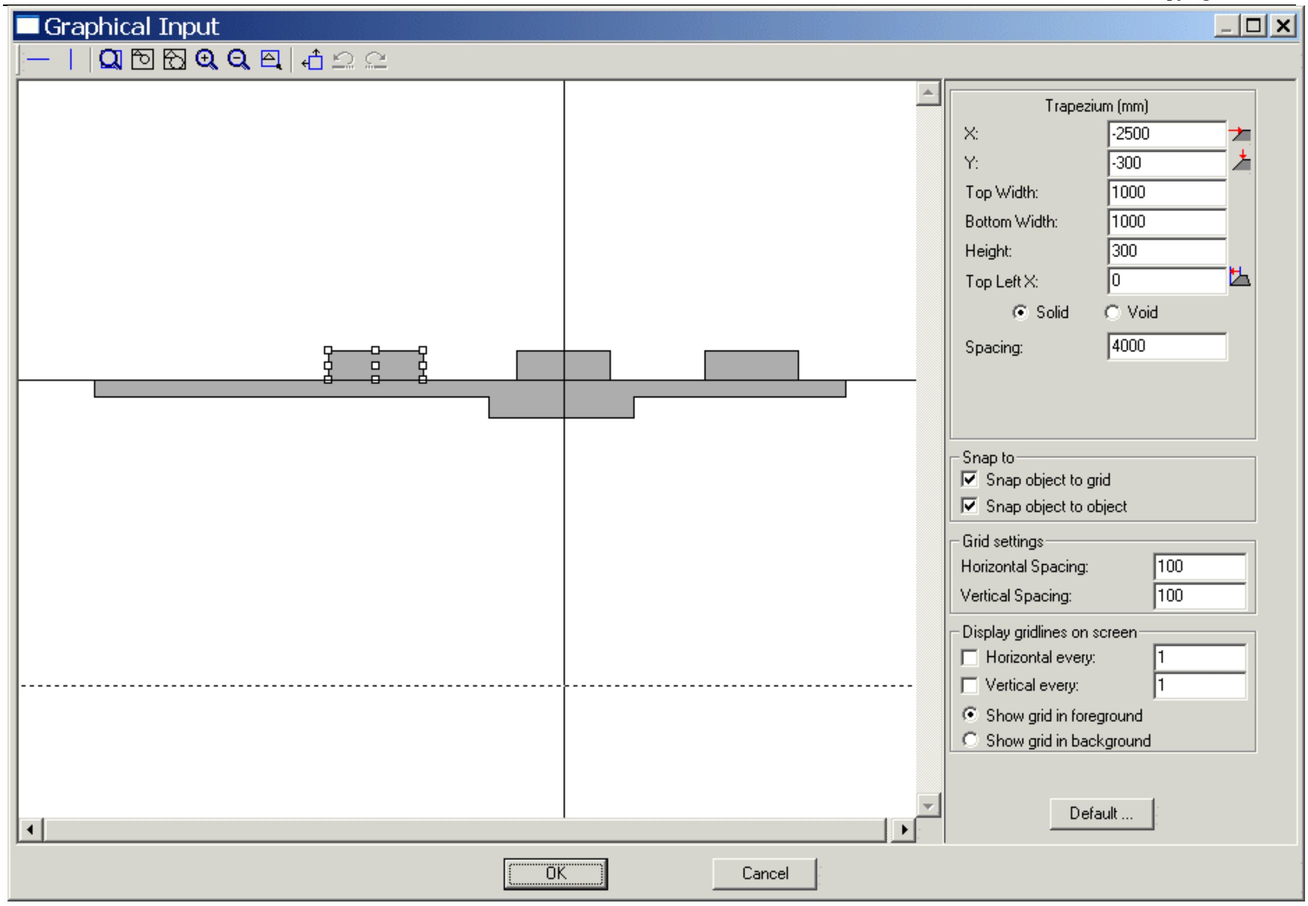
The width from the centreline of the frame (line of supports) to the local datum of the rightmost element in the group. The dimension is negative to the left of the centreline and positive to the right (as shown below).



Graphical Input

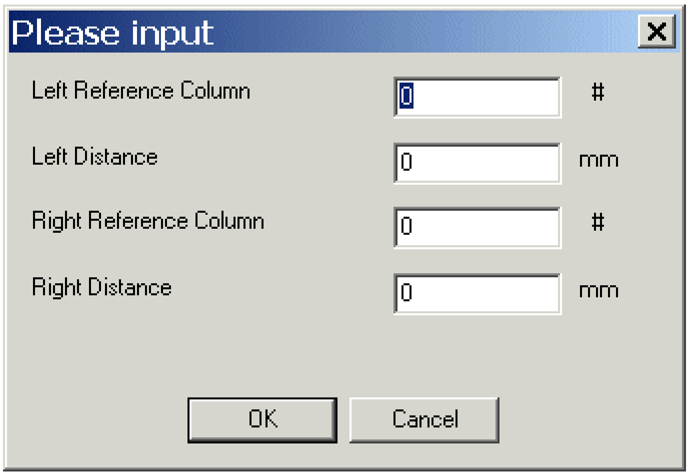
The shape and location of a selected element at the left end of the element can be modified graphically as described below. Selecting the element to be modified will present the dialog shown below. No other data can be changed in RAPT until this dialog is closed.

This diagram shows the concrete section at the left end of the element. This is not editable. It also shows the element shape and it is editable.

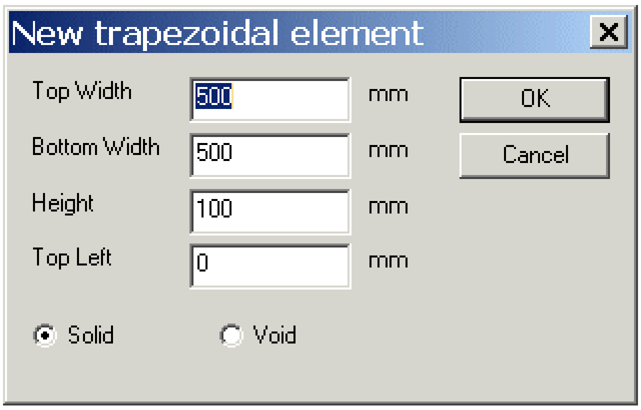


The dimensions and location of the element do not have to be pre-defined in the elements input screen

If no locations are defined, RAPT will present the following dialog on selecting the element. RAPT cannot proceed without element locations as it must define the overall concrete shape at the left end of the element, so the data required in this dialog must be provided.



If no element dimensions are defined, RAPT will then ask for default dimensions in the following dialog.



Once the main shape dialog is presented, the element shape and location can then me modified graphically or manually by adjusting the dimensions.

The shape can be modified by click and drag on any of the 9 gripper points shown. Where there are multiple shapes defined for 1 element only the left hand shape can be selected as shown above but all shapes will be modified in the same way. The shapes can also be modified using the Input Shortcut Palette defined below.

The shape or group of shapes can be moved by selecting any of the shapes (away from the grippers) and dragging. Void shapes cannot be moved outside solid shapes and solid shapes cannot overlap solid shapes.

Where there are multiple shapes for an element, the spacing between the shapes can be modified by using the Input Shortcut Palette or using the mouse by using Shift + Left Click and dragging the element sideways.

Graphical Shape Input Toolbar

The toolbar icons allow the user to manipulate the view as described below

Zoom to Selection: Click this button, then click the section input view and drag the mouse to select a rectangular area. Release the mouse and the view will be zoomed to show the selected area.

Zoom View to Fit: Zoom the entire View Size to fit the view window in at least one direction.

Zoom Shape to Fit: Zoom-out the entire Shape to fit the view window in at least one direction so that no scrolling is needed.

Zoom In: Zoom in the view by half with each click to see a more detailed view.

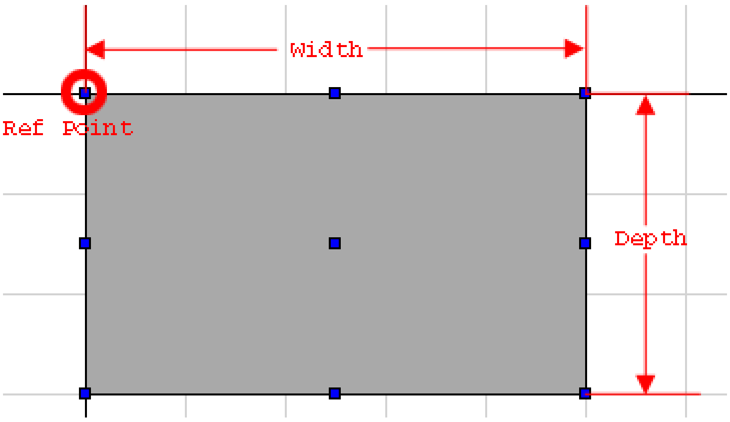
Zoom Out: Zoom out the view by two with each click to see more in the view window.

Section Input Definitions

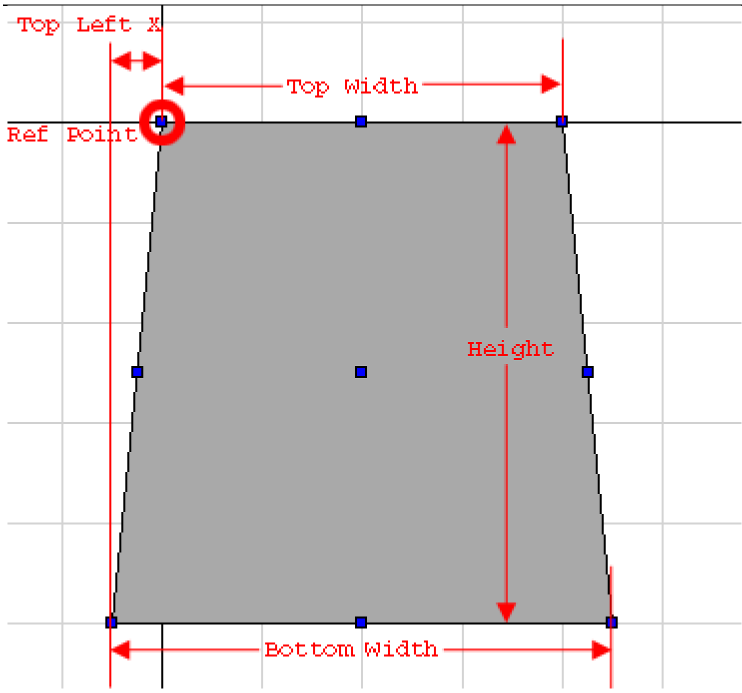
There are four basic shape types provided by RAPT to form more complex shapes. When selected, each shape type has 9 grippers indicated by small squares. Dragging those grippers with the mouse can change the shapes in the direction of the mouse cursor arrows (except for the centre gripper, which can not be dragged). The grippers can also be used as snap points. The centre gripper allows one shape's gripper to be snapped to the other shape's gripper, or one gripper to be snapped to a grid line or grid line intersection.

Each shape type has a reference point indicated in the following diagrams.

A rectangular shape's attributes are defined in the following diagram. Its reference point is the top-left point indicated by the red circle.



A trapezoidal shape's attributes are defined in the following diagram. The "top left X" is defined as the X coordinate difference between the bottom-left point and the top-left point. A negative value indicate the top-left point is at the left of the bottom-left point. The designer can also define a triangle by defining the Top Width or Bottom Width as zero. A trapezoidal shape's reference point is the top-left point indicated by the red circle.

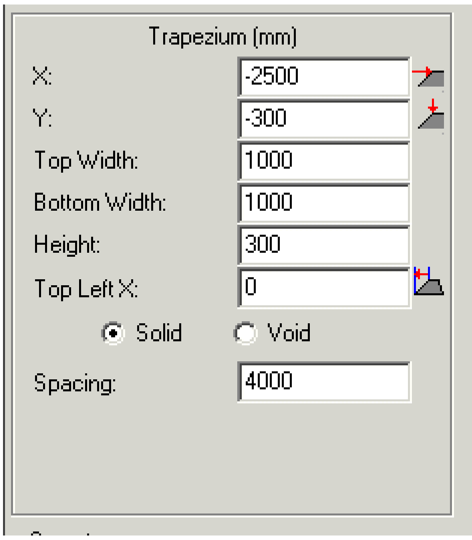


Section Input Palette

The section input palette on the right of the section input view consists of two main windows:

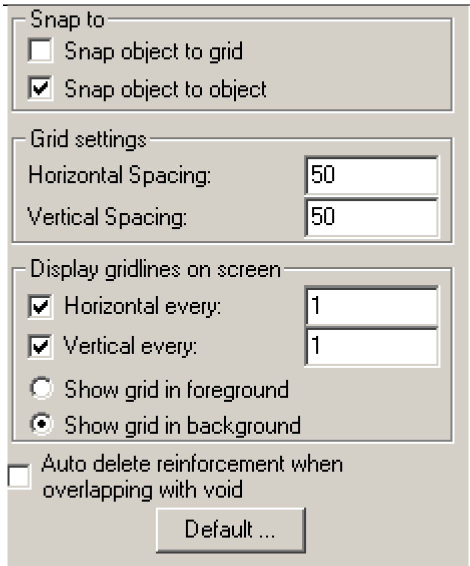
1. the input shortcut palette.
2. the section input configuration palette.

Input Shortcut Palette: The content of this window may change depending on the current selection on the section input view. It may reflect the data of a currently selected shape, the data of a group of selected shapes or a general reinforcing bar (not reinforcement patterns). It provides an accurate way of displaying and location changing data when mouse dragging is not accurate enough.



The above screen shot shows the Input Shortcut Window status when a rectangular shape is selected (shown with blue grippers). The shapes attributes are shown on the input shortcut window on the right. The designer can change the shape by either using a mouse to change the shape graphically in the left window by selecting a gripper and moving the mouse or make modifications to the shapes attributes in the input shortcut window on the right.

Section Input Configuration Palette: This window allows the designer to define section input settings such as grid lines and snap to options etc.



Snap To Options
When the "Snap object to grid" checkbox is checked, a moving shape will try to snap one of it's 9 gripper points to the closest grid line or grid line intersection.

When the "Snap object to object" checkbox is checked, a moving shape will try to snap one of it's 9 gripper points to one of the 9 gripper points of the closest shape if gripper points from the two shapes are close enough (within 4 pixels of each other).

When both "Snap object to grid" and "Snap object to object" are checked, the "Snap object to object" takes precedence than "Snap object to grid".

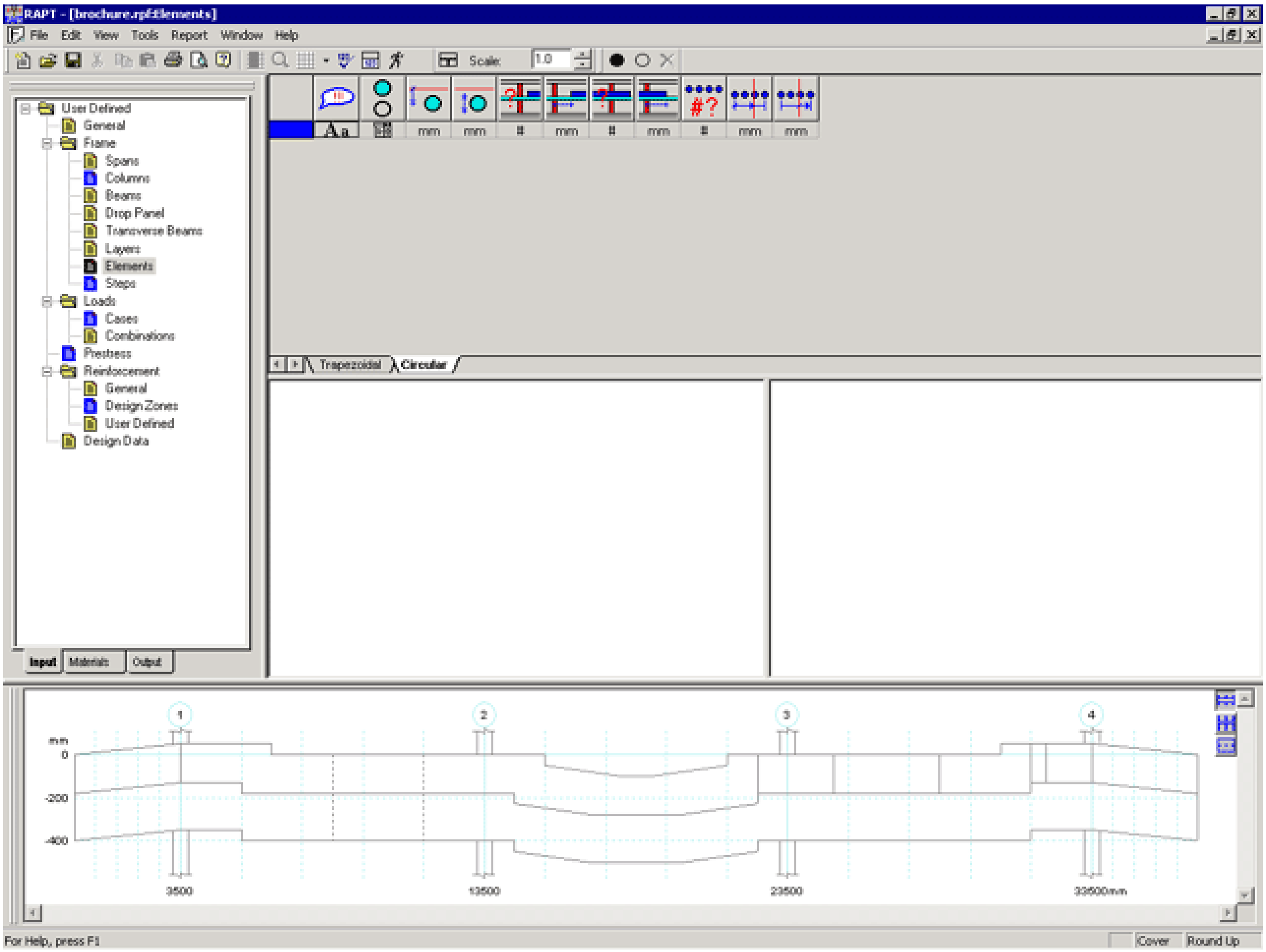
The shapes won't snap if the options are not turned on.

Grid Options
"Horizontal Spacing" and "Vertical Spacing" specify the spacing interval between grid lines in the current length unit.

"Horizontal every" and "Vertical every" specify how many spacing intervals are to be allowed between drawn grid lines in each direction. They can be a decimal number. The distance between grid lines is calculated as the spacing multiplied by number of spacing intervals. For example, 50mm horizontal spacing, 50mm vertical spacing with 1 horizontal interval and 0.5 vertical interval will result in a drawn grid of 50 mm x 25 mm.

"Show grid in foreground" will set the grid to be drawn in the foreground and be visible in the whole window. "Show grid in background" will set the shapes to cover the grid and grid lines will not be visible within the boundaries of any shapes in the window.

7.2.3.8.2 Circular



Circular Elements Toolbar

Add solid circular element

Add a new data row for a solid circular element. The elements will be inserted at a selected data row in the grid or added after the last data row in the grid if there is no selection.

Add void circular element

Add a new data row for a void circular element. The elements will be inserted at a selected data row in the grid or added after the last data row in the grid if there is no selection.

Delete circular elements

Delete one or more selected elements.

Copy:- Copy a selected group of elements for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any elements can be selected by clicking the element number with the left mouse button. Groups of elements can be selected by click and dragging the mouse. Extra elements can be added to the selected group using Ctrl + Left Mouse key. Changes to the element arrangement after the selection is made, by adding or removing elements or modifying concrete shapes, will void any selection.

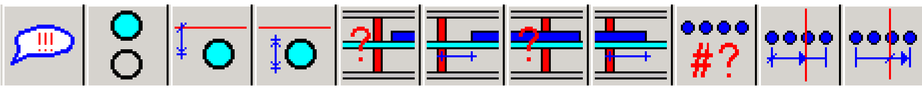
Paste:- Paste the selected elements. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the element number cell, the copied elements will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted elements. If no data row is selected the copied elements will be added to the end of the existing elements. Multiple data rows cannot be selected for the paste operation. The paste can be into elements of the same type, e.g. circular elements can only be copied to circular elements. RAPT automatically creates the space in the data grid into which the elements are to be copied, so it is not necessary to create the rows for the data to be inserted into.

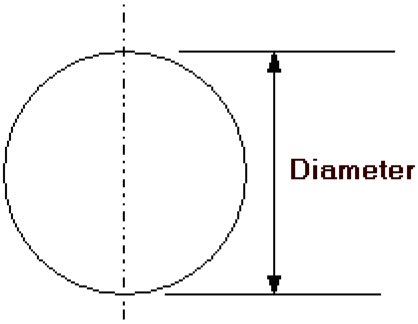
Section Views

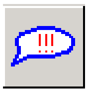
When the program focus is in the trapezoidal elements data view, the Section Graphics Windows

1. Left panel:- The shape of the elements defined by the current data row highlighted on the cross-section at the left end of the element.
2. Right Panel:- The shape of the elements defined by the current data row highlighted on the cross-section at the right end of the element.

Data Definition

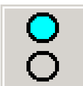






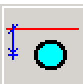
Description

Name of the element. The default name is "New Element". The designer should rename the element with a recognisable name.



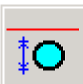
Solid / Void

Whether the element is a solid or a void shape. Solid elements should be fully outside the concrete shape and connected to the shape. Void elements should be fully within the solid concrete shape but may be connected to a surface.



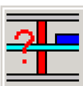
Depth to Centroid

The depth to the centroid of the element from the top of slab datum. Positive is downwards.



Element Diameter

Diameter of the element.



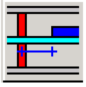
Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the element is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the element. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

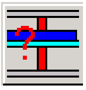
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of an element in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of an element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

Distance to Left End from Left Reference Column

The distance from the column grid reference to the left end of the element. If the distance entered moves the element into another span, RAPT will automatically adjust the column grid reference and the distance from it to refer to the nearest column grid reference at or before the start of the element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

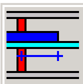
Right End Reference Column

The 7.2.3.2 Column Grid Reference from which the right end of the element is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the element. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

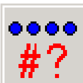
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of an element in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of an element.

Obviously the right end of an element must be to the right of the left end so make sure when modifying element ends that the order of modification will ensure this.

Distance to Right End from Right Reference Column

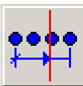
The distance from the right reference column to the right end of the element. If the distance entered moves the element into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the end of the element.

Obviously the left end of an element must be to the left of the right end so make sure when modifying element ends that the order of modification will ensure this.

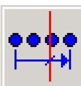


Number of Elements in Group

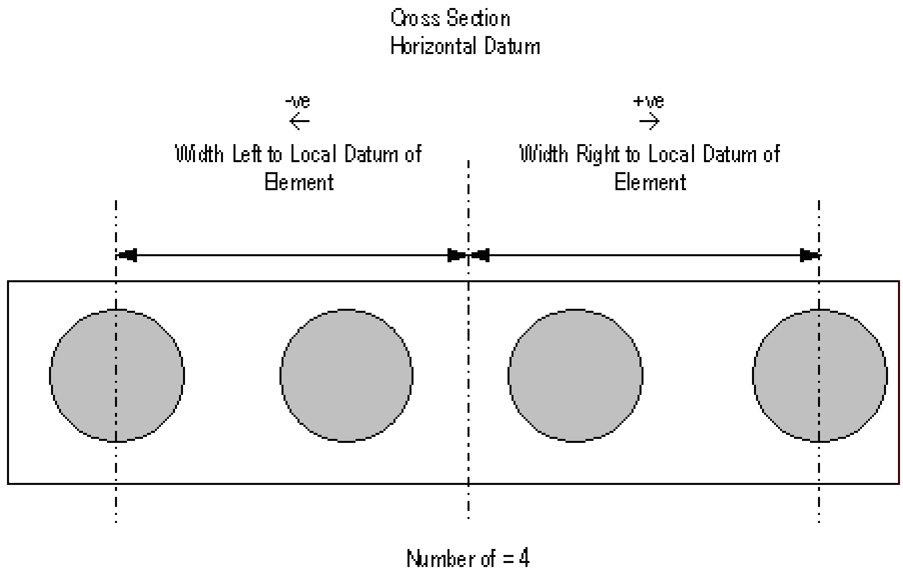
The number of this type of element to be placed at this location at an equal spacing across the width defined below (see below).

Width to Left Element Shape in Group from Centreline

The width from the centreline of the frame (line of supports) to the local datum of the leftmost element in the group. The dimension is negative to the left of the centreline and positive to the right (as shown below).

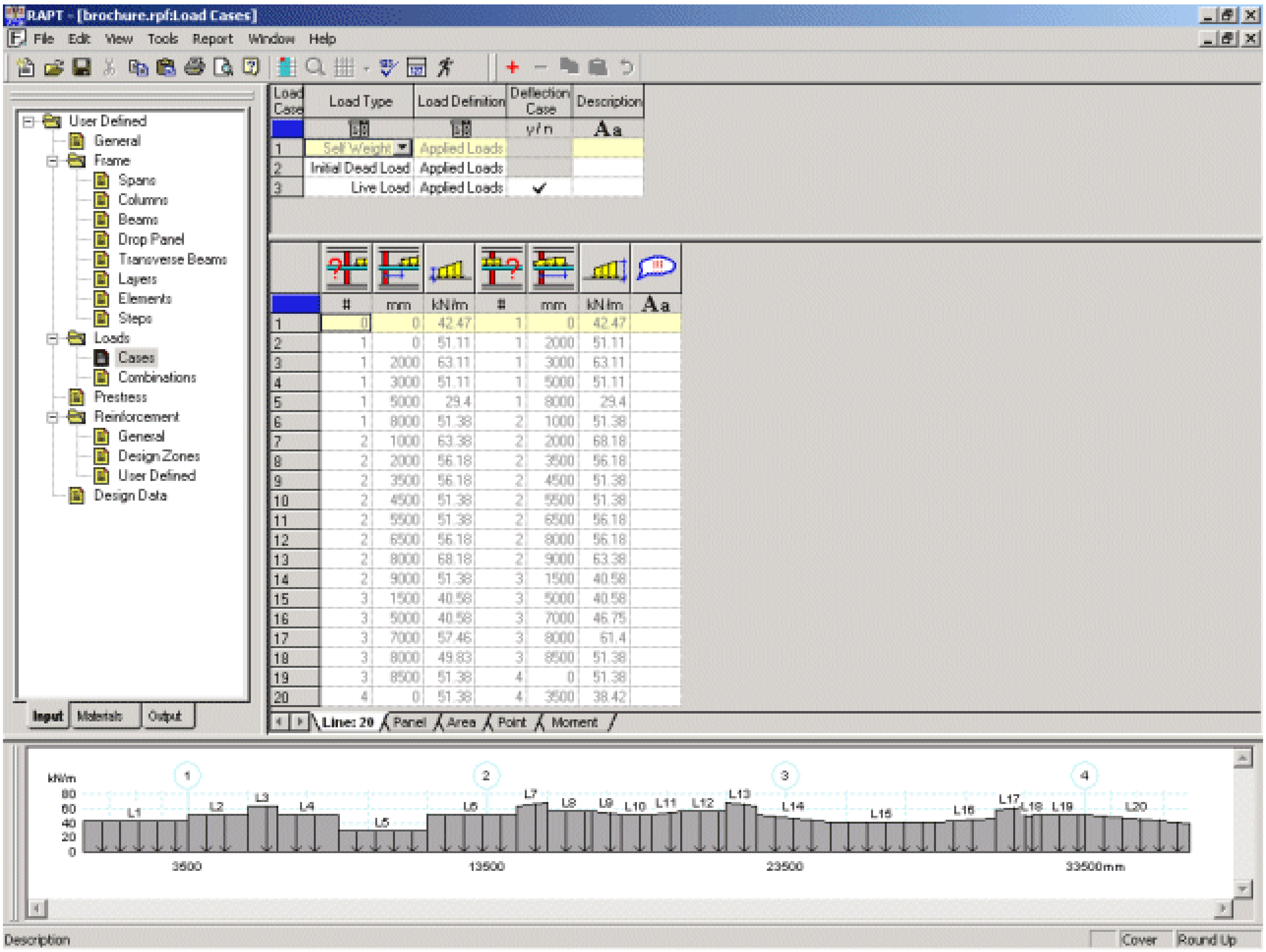
Width to Right Element Shape in Group from Centreline

The width from the centreline of the frame (line of supports) to the local datum of the rightmost element in the group. The dimension is negative to the left of the centreline and positive to the right (as shown below).


Cross Section
Horizontal Datum
-ve
+ve
Width Left to Local Datum of Element
Width Right to Local Datum of Element
Number of = 4

7.2.4 Loads

7.2.4.1 Load Cases



This input screen is used to define the various loadings applied to the frame. The screen is divided into 3 separate windows

1. Load Case Control Grid:- This grid lists the general data about each load case and controls which load case is viewed in the Load Case Data and Load Case Graphics windows (see 7.2.3 Screen Layout for general discussion). The functionality of this window is discussed in this section. Program focus must be in this window to access the toolbar to add load cases.
2. Load Case Data:- Defines the loads applied in the load case nominated in the Load Case Control Grid. The functionality of the views in this window is discussed in the later sections. Program focus must be in this window to access the toolbars to add loads to a load case.
3. Load Case Graphics:- Shows an elevation of the selected load case.

The load case control grid lists the load cases defined for the member. Each load case uses one row of the control grid.

Graphic Interaction

Clicking on a load in the graphics will open the page of data for that type of load and move the data focus to the data cells controlling that data.

Load Cases Control Toolbar

Add Load Case:- Adds a load case to the load case window. If a row in the data grid is select by left clicking on the load case number, the new load case will be inserted at that selected load case position and all of the load cases at and below that selected load case will be moved down by 1 case. Otherwise, the added load case will be the last load case in the grid.
If there is no Initial Dead Load case entered, RAPT will default the new case to Initial Dead Load if the number of load cases already existing is less than 4. If there is no Live Load case entered, RAPT will default the new case to Live Load if the number of load cases already existing is less than 4. Otherwise RAPT will default the Load Type to Other Load.

Delete Load Case:- Deletes the selected load cases. Multiple load cases can be deleted at once using the selection logic in 4.4.3 Cell Selection.

Copy Load Case:- Allows the complete contents of a load case to be copied for later pasting to another load case. The load case to be copied must be selected by clicking on the load case number in the left data column before the Copy Load Case icon is available. Click the Icon to copy this load case.

Paste Load Case:- Once a load case has been copied, it can be pasted to another load case. The "paste to" load case must be selected by clicking on the load case number in the left data column before the Paste Load Case icon is available. The Load Definition of the selected "paste to" load case and the "copy from" load case must be the same or the Paste Icon will not be available. Click the icon to paste all of the load data from the copied load case into this load case. For Applied Load cases, the copied loads will be appended to the loads already defined in the "paste to" load case. If you want to replace the loads, you will need to delete the loads from the "paste to" load case before performing the paste.

Clear Load Case:- Deletes all applied loads from all load types in a selected load case or a group of selected load cases. Resets moment diagrams and envelopes to their default state.

Data Definition

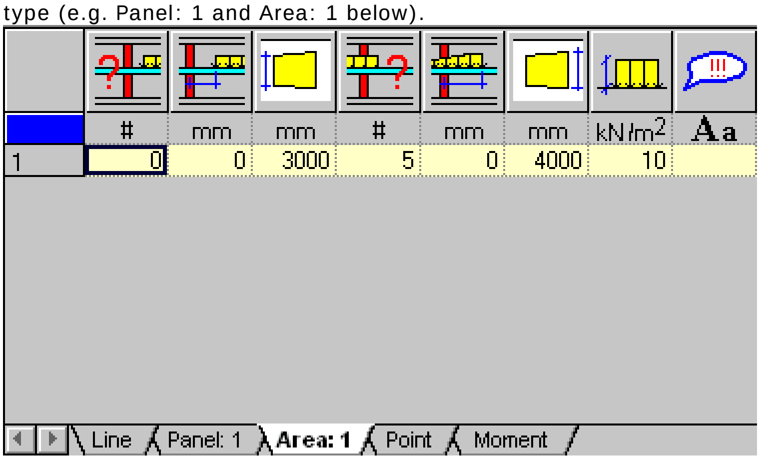
Load Type:- The load types available in RAPT are

1. Self Weight:- Self weight of the structure. RAPT will calculate the self weight of the frame. The designer can add extra loads to this if necessary. Alternatively, the designer can select in 7.2.2 General Input Screen to nominate all self weight loads. In this case RAPT will not calculate any self weight loads. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Only Applied Load or Moment Diagrams can be used for this load type.
2. Initial Dead Load:- The superimposed dead load that will be applied to the slab as the initial dead load in the deflection calculations. If finishes are to be applied that will be damaged by excessive deflection, this is the superimposed dead load that is to be applied to calculate the deflected state from which the incremental deflection is to be calculated. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Only Applied Load or Moment Diagrams can be used for this load type.
3. Extra Dead Load:- Any other superimposed dead load not included in Initial Dead Load above. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Only Applied Load or Moment Diagrams can be used for this load type.
4. Live Load:- The live loads on the member. If pattern loading has been selected in 7.2.2 General Input Screen, it will be applied to this load case. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Applied Load, Moment Diagrams, Moment Envelopes or Moving Loads can be used for this load type.
5. Alternate LL:- If there is more than one live load condition to consider, RAPT will create an envelope of the various live load and alternate live load cases. These load cases will NOT be added to the Live Load case, they will be compared to it at each design point to create an envelope. Default factors will be applied automatically for the various load combinations. These can not be modified directly by the user. When the Live Load Case factors are modified the factors for the alternate cases will be modified automatically. Applied Load, Moment Diagrams, Moment Envelopes or Moving Loads can be used for this load type.
6. Wind Load:- The wind loads or wind load moment diagram/envelope on the member. RAPT only uses 1 wind load case for service and ultimate strength combinations. The wind load case is a service case and a load factor is included to convert the service case to an ultimate case. For design codes that have variable factors between service and ultimate for wind load the designer should modify the factor used in combinations accordingly. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.
7. Earthquake Load:- The earthquake loads or earthquake moment diagram/envelope on the member. Default factors will be applied automatically for the various load combinations. These can be modified by the user as required to suit different design situations. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.
8. Snow Load:- Snow loading. At this stage RAPT does not apply load combination factors automatically for snow loading. Designers will need to create their own load combinations to handle this. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.
9. Alternate Wind L:- If there is more than one wind load condition to consider, RAPT will create an envelope of the various wind load and alternate wind load cases. These load cases will NOT be added to the Wind Load case, they will be compared to it at each design point to create an envelope. Default factors will be applied automatically for the various load combinations. These can not be modified directly by the user. When the Wind Load Case factors are modified the factors for the alternate cases will be modified automatically. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.
10. Alternate Eq L:- If there is more than one wind load condition to consider, RAPT will create an envelope of the various wind load and alternate wind load cases. These load cases will NOT be added to the Wind Load case, they will be compared to it at each design point to create an envelope. Default factors will be applied automatically for the various load combinations. These can not be modified directly by the user. When the Earthquake Load Case factors are modified the factors for the alternate cases will be modified automatically. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.

11. Other Load - Normal:- This is a load case type that is unknown to RAPT so RAPT does not apply load combination factors automatically for it. Designers will need to define their own combination factors or create their own load combinations in the various combination screens to handle this load type. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.
12. Other Load - Support Strip:- This load type is the same as "Other Load" above except that the moments and shears from this case are applied to the support (column) strip only in a two way design. No moment or shear will be distributed to the middle strip for this load type. Applied Load, Moment Diagrams or Moment Envelopes can be used for this load type.

Load Definition:- Loads can be defined in three different ways in a load case. These are

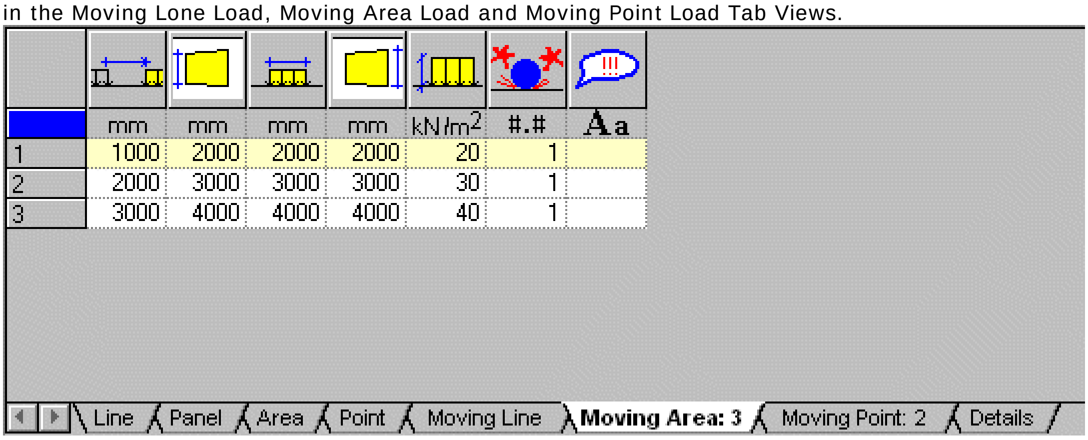
1. Applied Loads:- Five different types of applied load can be defined. For each load type, up to 999 individual loads can be defined in a load case. The number of loads in each Load Type is shown on the Tab for that load type (e.g. Panel: 1 and Area: 1 below).



The different applied load types are

   1. Line Loads:- A load defined as load/length. The load can vary in value linearly over it's length. Positive loads are downwards.
   2. Panel Loads:- A load defined as load/area. The load can vary in value linearly over it's length. The load is applied to the overall design panel defined for the frame. The exception to this is 7.2.2 One-Way Nominal Width Member Type where the width used is the design width in each span. RAPT will allow for varying panel widths along the length of the loading. Positive loads are downwards.
   3. Area Loads:- A load defined as load/area. The load is constant over the length of the loading. The load is applied to the widths nominated in the loading which can vary linearly over the length of the loading. Positive loads are downwards.
   4. Point Loads:- A load defined as a concentrated load. RAPT allows the user to define a length for point loads which will round the bending moment effect rather than resulting in a pure point load. A load length of zero will result in a pure point load, otherwise the load will be spread evenly over the load length resulting in a reduction in the peak moment. In a two-way slab, Its effect will be divided between column and middle strip as per the 7.2.4.3 lateral distribution factors defined for the frame. Positive loads are downwards.
   5. Point Moments:- A concentrated moment. Clockwise moments are positive.

2. Moment/Shear Diagram:- A bending moment and shear diagram with one value of moment and one value of shear at each nominated location along the frame.
3. Moment/Shear Envelope:- A bending moment and shear envelope with two values of moment and two values of shear at each nominated location along the frame.
4. Moving Loads:- A fixed pattern of loads (normally a distributed load) as defined for Applied Loads above in the Line Load, Panel Load, Area Load, and Point Load Tab Views and a pattern of moving loads can be defined in the Moving Lone Load, Moving Area Load and Moving Point Load Tab Views.



The pattern of moving loads can incorporate any combination of the following load types.

   1. Moving Line Loads:- Nominated as ML# in the graphics. A load defined as load/length. The load can vary in value linearly over it's length. Positive loads are downwards.
   2. Moving Area Loads:- Nominated as MA# in the graphics. A load defined as load/area. The load is constant over the length of the loading. The load is applied to the widths nominated in the loading which can vary linearly over the length of the loading. Positive loads are downwards.

3. Moving Point Loads:- Nominated as MP# in the graphics. A load defined as a concentrated load. RAPT allows the user to define a length for point loads which will round the bending moment effect rather than resulting in a pure point load peak moment. A load length of zero will result in a pure point load, otherwise the load will be spread evenly over the load length resulting in a reduction in the peak moment. In a two-way slab, Its effect will be divided between column and middle strip as per the 7.2.4.3 lateral distribution factors defined for the frame. Positive loads are downwards.

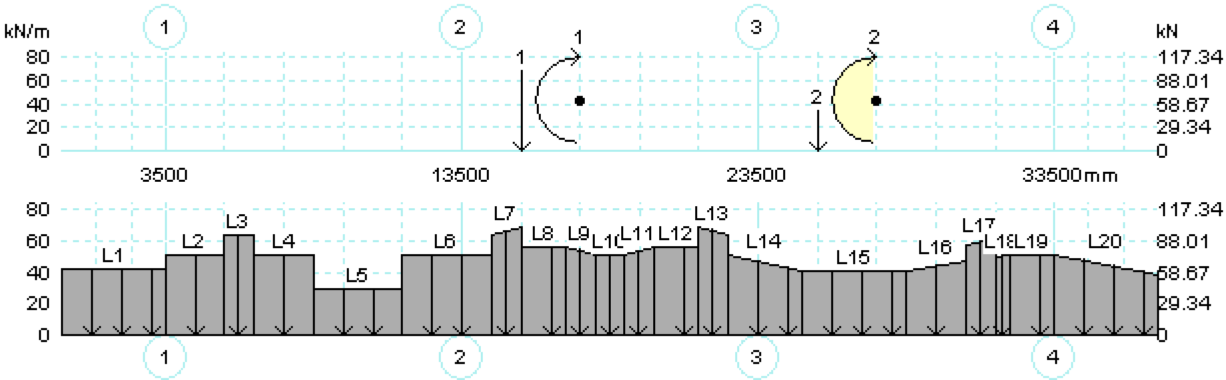
Deflection Case:- Nominates which of the load cases is to be used as the live load case in the deflection load combinations. The load case must be a Live Load case, Alternate Live Load case or an Other Load case and cannot be a Moment/Shear Envelope or a Moving Load case. Only one load case can be selected.

Description:- Designer's description of the load case. This is not a compulsory data field.

Load Case Graphics Window

The load case graphics window shows the load current load case as selected in the Load Case Control Window. Depending on the Load Definition Type for the current load case in the Loads Control Window, the view shows

1. Applied Loads and Moving Loads:- a series of line diagrams of the loads defined in the selected load case. For easier viewing, if two or more loads share the same location, they are shown on different layers in the graphics window. Use the vertical slider to view layers not in the view. When a load is selected in the Load Case Data window as the current load, the graphics will automatically adjust to show the layer containing that load.



Loads are shown with a black outline and a mid-grey background. If a load has been selected in the Loads Data Grids or has been selected in the graphics using the toolbar or by clicking in the span with the left mouse button, the current load will be shown with a pale yellow background. No cursor is available in the applied loads graphics view.

The load case graphics has two vertical scales. The left scale is in load/length and is for all Distributed Load Types (Line, Panel and Area: see note below) and the right scale is in load and is for Point Loads.

Note: Area and Panel Loads are converted to load/length in the graphics so all Distributed Loads are shown as load/length graphically. Panel loads are subdivided into separate loads for each segment of length of the member where the panel width varies.

In the graphics, each distributed load is provided with a label to indicate the source of the load (L = Line Load, P = Panel Load, A = Area Load) and the number of the load in the data list that generated this load (see above). Point Loads and Point Moments only have a number to indicate the load number. Moving loads will have the following labels: moving live load ML#, moving area loads MA# and moving point loads MP#.

Point loads will be shown as a single arrow. If a length is defined for a point load, it will be shown with a black outline but only a single central arrow.

Where a Panel Load is divided into several segments due to changes in panel width over the length of the load, each segment will have the same label referring to the same load in the data list. Each of these load segments is selectable and the Information button will show the data for each individual segment of the load and selection toolbar buttons will move to each individual segment of the load. In this case, the data screen will continue to show the same load data as the current load.

Selecting Data Cells
RAPT provides a logical connection between the data and the graphics for the designer. Clicking on a load in the graphics will open the page of data for that type of load and move the data focus to the data cells controlling that data.

Toolbar

Zoom (Ctrl + Z). This button will toggle between full screen mode and load zoom mode for the graphics in a window. In load zoom mode, the current load will be shown with the half span either side of the end spans in which the load is applied (if a cantilever is the previous or next span, the full cantilever will show) scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window.

Move to next load (Ctrl + Right Arrow).

Not available.

 Not available.

 Move to previous load (Ctrl + Left Arrow).

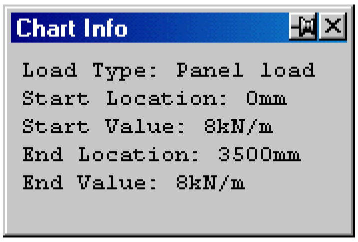
 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width of depth of the graph will be included than requested.
Then the user can then move around the graph using the Scroll Bars or the movement keys.

Clicking on the  or  buttons will move to the next or previous load and return the Window to Full Screen Mode.

Clicking this  button again will return the Window to Full Screen Mode.

Clicking  will change the mode to Span Zoom Mode on the load that was current in positioned in the Select Zoom mode.

 To view the information describing a load, open the Information Dialog from the graphics toolbar by clicking this button or press Ctrl + I, and then left click on the desired load in the graphics window. While the dialog is in view, click at any other load to view its information and the dialog data will be updated automatically.

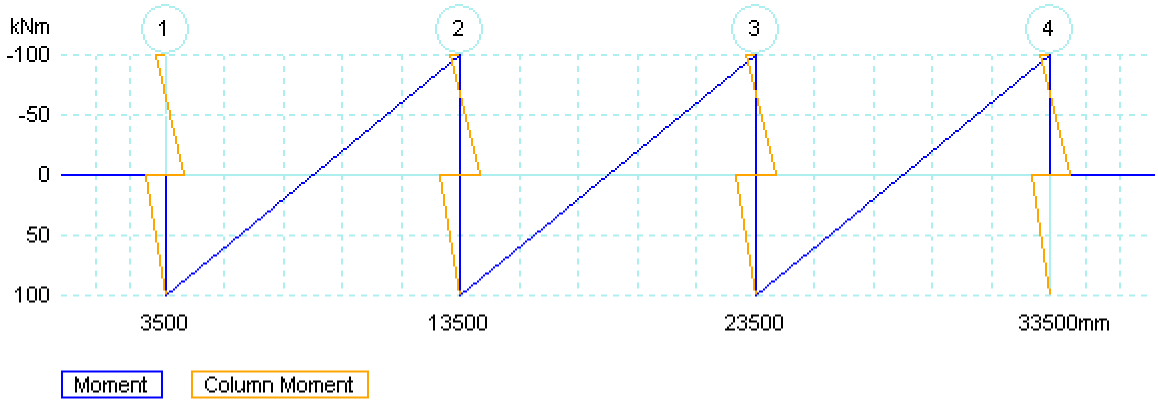


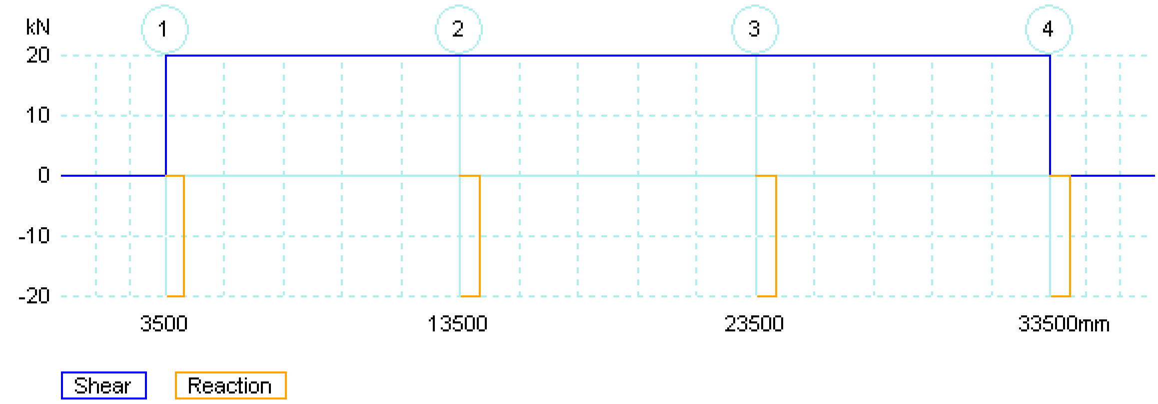
The dialog will list:

1. The load type e.g. Panel Load above.
2. The start location of the load from the left end of the frame.
3. The start load value in load/length.
4. The end location of the load from the left end of the frame.
5. The end load value in load/length.

Note that area and panel loads are converted to load/length in the graphics and in the info data and panel loads are subdivided into separate loads for each increment of length of the member with different panel widths.

2. Moment/Shear Diagrams:- A bending moment diagram and a shear/column reaction diagram as shown below.





Selecting Data Cells

RAPT provides a logical connection between the data and the graphics for the designer. Clicking in the graph will select the nearest moment definition point. The cursor will move to this point and the data focus will move to the data cells controlling that data showing the designer the data controlling that point.

Toolbar

 Zoom (Ctrl + Z). This button will toggle between full screen mode and span zoom mode for the graphics in a window. In span zoom mode, the current span will be shown with the half span either side (if a cantilever is the previous or next span, the full cantilever will show) scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window.

 Move to next span (Ctrl + Right Arrow). The cursor will move automatically to the first Moment Definition Point in the new span. In zoomed mode the new span will become the current span.

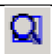
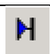
 Move to next point (Shift + Right Arrow). The cursor is moved to the next Moment Definition Point to the right. If it reaches the right end of the span in zoomed mode, the next span is moved to the centre of the Window.

 Move to previous point (Shift + Left Arrow). The cursor is moved to the Moment Definition Point to the left. If it reaches the left end of the span in zoomed mode, the previous span is moved to the centre of the Window.

 Move to previous span (Ctrl + Left Arrow). The cursor will move automatically to the last Moment Definition Point in the new span. In zoomed mode the new span will become the current span.

 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width of depth of the graph will be included than requested.

Then the user can then move around the graph using the Scroll Bars or the movement keys.  and  buttons will still move the cursor to the next and previous Moment Definition Points. If the next point is outside the viewable area the area of graph shown will adjust automatically to position the requested load near the left of the View Window.

Clicking this  button again or on the  or  buttons will return the Window to Full Screen Mode.

Clicking  will change the mode to Span Zoom Mode on the load that was current in positioned in the Select Zoom mode.

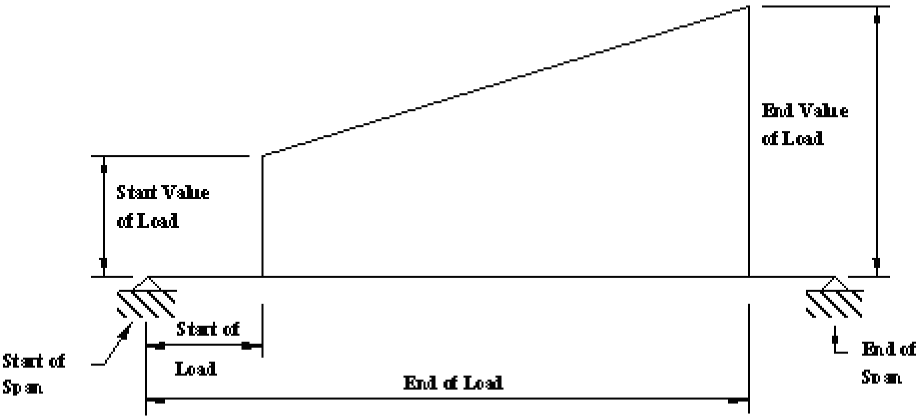
 Not Available. Selecting a defined moment point with the mouse or toolbar buttons will move the data focus to the line of data that controls the selected point in the data window so no Information Dialog is necessary.

3. Moment/Shear Envelopes:- Two pairs of bending moment diagram and a shear/column reaction diagram, one for each extreme of the moment envelope. Each of the pairs of diagrams is as shown above in Moment Diagram. The toolbar rules are also the same as above.

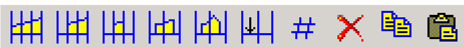
7.2.4.1.1 Applied Loads

7.2.4.1.1.1 Line Loads

A load defined as load/length. The load can vary in value linearly over it's length so a left load value and a right load value are required as shown below. Line loads can start and finish anywhere in the frame, independent of support locations. Internally RAPT will break each load down into span loads for pattern load calculations and member loads for frame analysis. Positive loads are downwards. In two-way slab analysis, line loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm except when the load case is defined as "Other Load - Support Strip" in which case they will only be applied to the support (column) strip.



Line Loads Toolbar



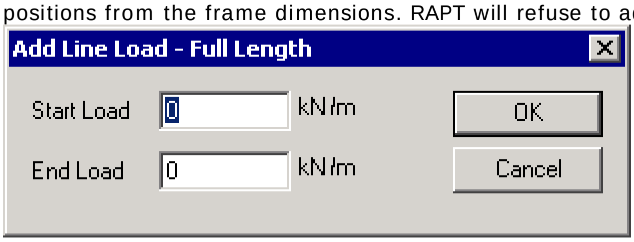
The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

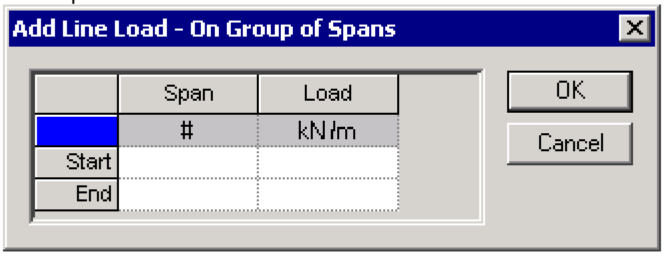
A blank cell is regarded as having a value of zero.

Where a Reference Column is asked for in any of the following dialogs, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

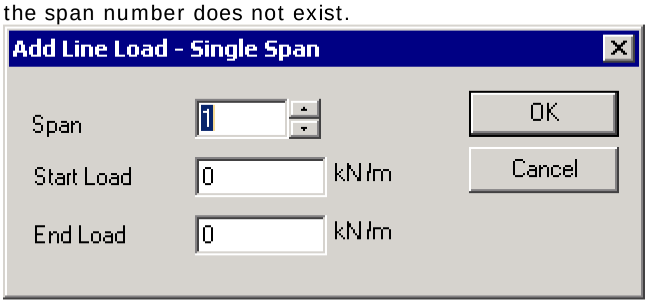
Add load full length of frame:- Will add a linearly varying load over the full length of the frame. The dialog below will be presented for input of the load values. Data focus is initially in the start load cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load values are zero.



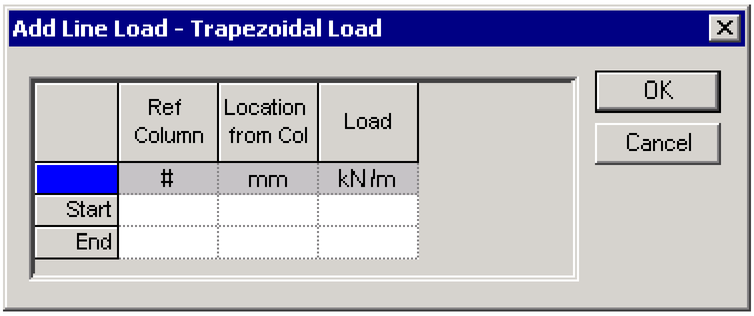
Add load on spans:- Will add a linearly varying load over the full length of the nominated group of spans. The dialog below will be presented for input of the span numbers and load values. Data focus is initially in the start span cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the end span is less than the start span or if one of the ends is outside the frame or if the two load values are zero.



Add load on span:- Will add a linearly varying load over the full length of a nominated span. The dialog below will be presented for input of the span number and load values. Data focus is initially in the span number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load values are zero or if the span number does not exist.

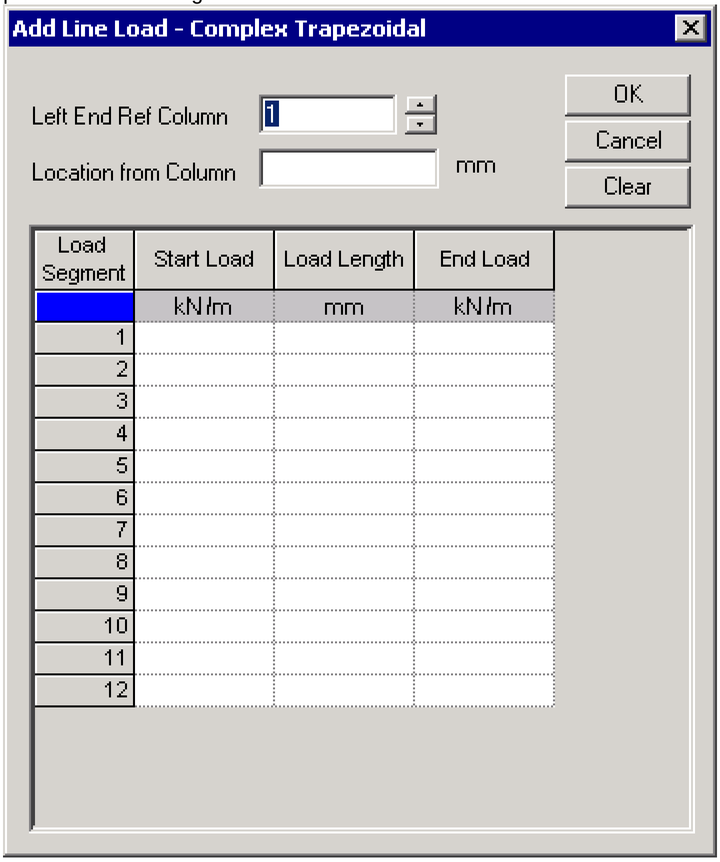


Add general trapezoidal load:- Will add a linearly varying load over the length nominated. The dialog below will be presented for input of the load start and end locations and load values. Data focus is initially in the start reference column number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame or if the two load values are zero.



Add multiple trapezoidal loads:- Will add multiple linearly varying loads over the lengths nominated. The dialog below will be presented for input of the load start, load lengths and load values. Data focus is initially in the left end reference column number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame. A load segment can have zero loads at both ends. RAPT will insert this as a gap in the load

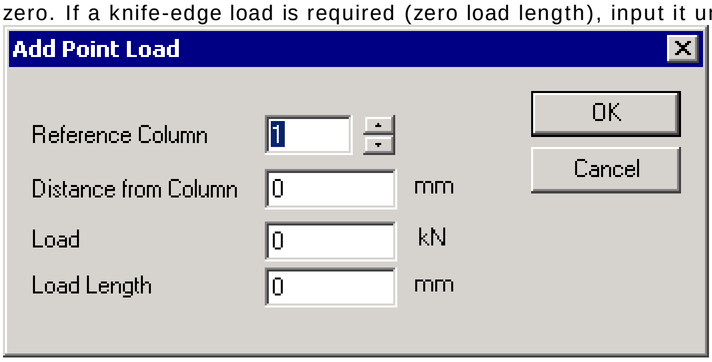
pattern of the length defined.



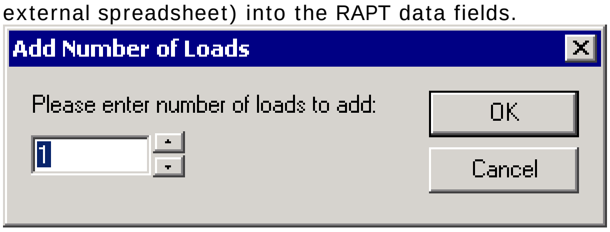
Add Concentrated load:- Most point loads actually apply load over at least the length of the item applying the load e.g. a transverse wall has a thickness. Many design codes also allow these loads to be distributed over an even wider area. Even though a point load is being defined, RAPT will apply it as a distributed load over the load length.

RAPT allows the designer to define a point load with a length over which the load is applied. The load will be applied centred at the defined load location. The designer should ensure that the full extent of the load will be within the frame length.

As well as defining a load location and a load value the designer is asked for a Load Length. This length cannot be zero. If a knife-edge load is required (zero load length), input it under the 7.2.4.1.1.4 Point Load input.



Add Number of Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields.



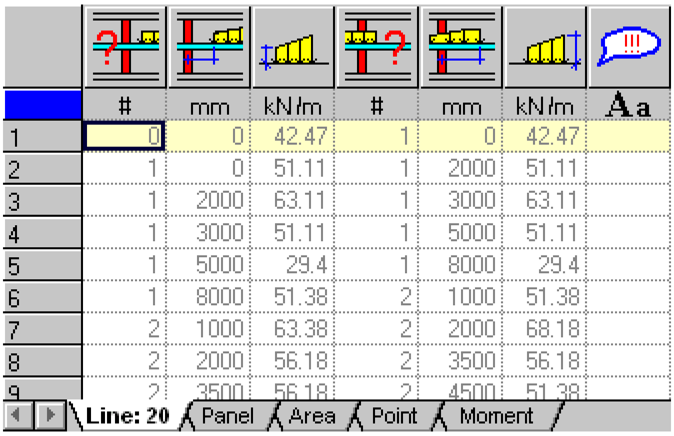
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

Copy:- Copy a selected group of loads to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

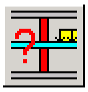
Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. line loads can only be copied to line loads. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

Data Definition

When editing or defining loads, RAPT will automatically convert the location entered to a location from the nearest column support to the left of or at the load.



	#	mm	kN/m	#	mm	kN/m	Aa
1	0	0	42.47	1	0	42.47	
2	1	0	51.11	1	2000	51.11	
3	1	2000	63.11	1	3000	63.11	
4	1	3000	51.11	1	5000	51.11	
5	1	5000	29.4	1	8000	29.4	
6	1	8000	51.38	2	1000	51.38	
7	2	1000	63.38	2	2000	68.18	
8	2	2000	56.18	2	3500	56.18	
9	2	3500	56.18	2	4500	51.38	

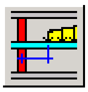
Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the load is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

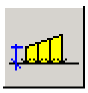
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

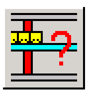
Left end of load from reference column

The distance from the left reference column to the left end of the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the start of the load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

Load at left end

Value of the load at the left end of the load in force/length.

Right End reference column

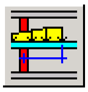
The 7.2.3.2 Column Grid Reference from which the right end of the load is dimensioned. RAPT will always adjust the

column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

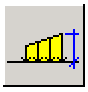
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

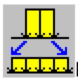
Right end of load from reference column

The distance from the right reference column to the right end of the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the end of the load.

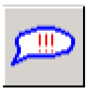
Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

Load at right end

Value of the load at the right end of the load in force/length.

Live Load reduction

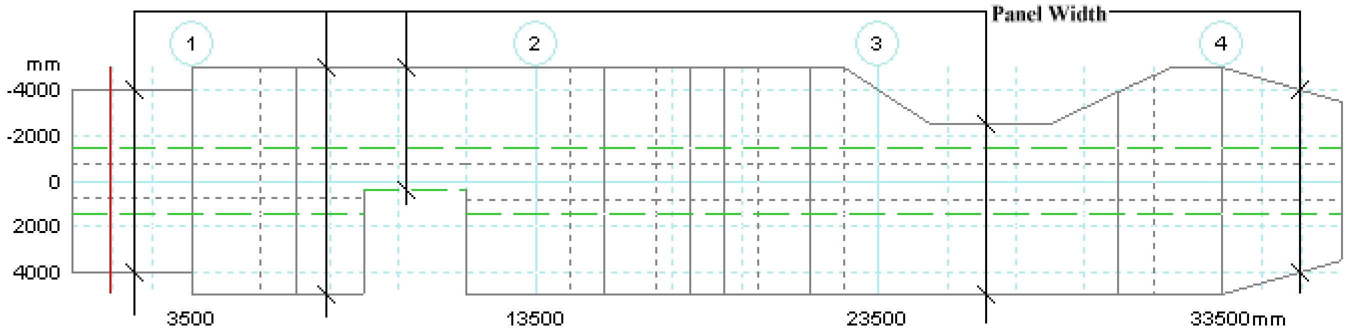
Only available for live load and alternate live load cases. The live load is defined as the full live load on the member before reduction. If a reduction factor is defined here, RAPT will use that reduction factor in all calculations using this load except the unbonded prestress minimum reinforcement calculations which require the full live load to be used. If the live load is defined as 3kPa and a live load reduction factor of .7 is specified then the actual live load used in strength and serviceability calculations will be .7 * 3 = 2.1kPa. This should only be used in situations where it is allowed by the design standard being used. The default value of the live load reduction is 1 (no reduction) and is the value used internally for all other load case types.

Description

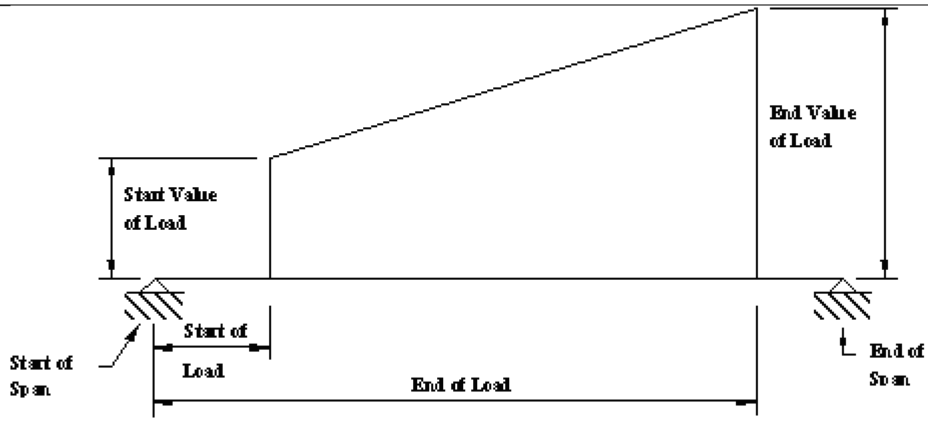
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.1.2 Panel Loads

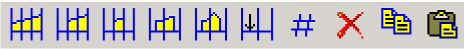
A load defined as load/length^2. The load can vary in value linearly over it's length so a left load value and a right load value are required as shown below. Panel loads can start and finish anywhere in the frame, independent of support locations. The load is applied to the panel width of the member (see below).



Internally RAPT will break each load down into loads/length for each different segment of panel width allowing for steps and tapers in the sides of the panel (except for one-way nominal width designs where the width used is the design width), then into span loads for pattern load calculations and member loads for frame analysis. Positive loads are downwards. In two-way slab analysis, panel loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm except when the load case is defined as "Other Load - Support Strip" in which case they will only be applied to the support (column) strip. In both cases the overall panel width is used to calculate the load/length.



Panel Loads Toolbar



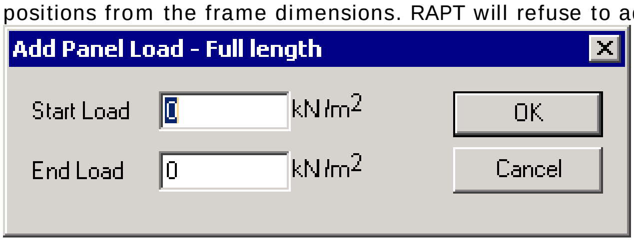
The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

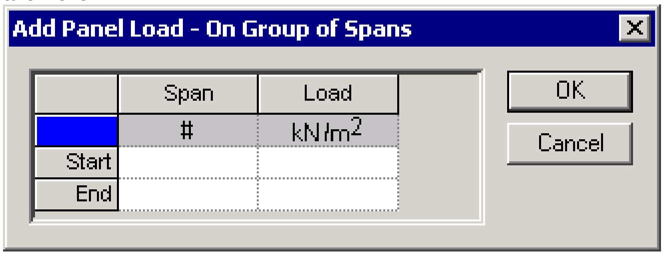
A blank cell is regarded as having a value of zero.

Where a Reference Column is asked for in any of the following dialogs, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

Add load full length of frame:- Will add a linearly varying load over the full length of the frame. The dialog below will be presented for input of the load values. Data focus is initially in the start load cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load values are zero.

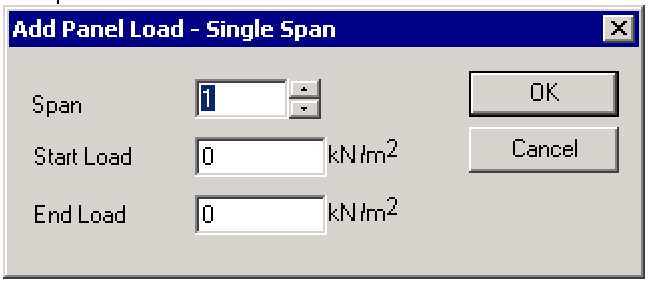


Add load on spans:- Will add a linearly varying load over the full length of the nominated group of spans. The dialog below will be presented for input of the span numbers and load values. Data focus is initially in the start span cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions and the start and end span numbers. RAPT will refuse to add the load if the end span is less than the start span or if one of the ends is outside the frame or if the two load values are zero.

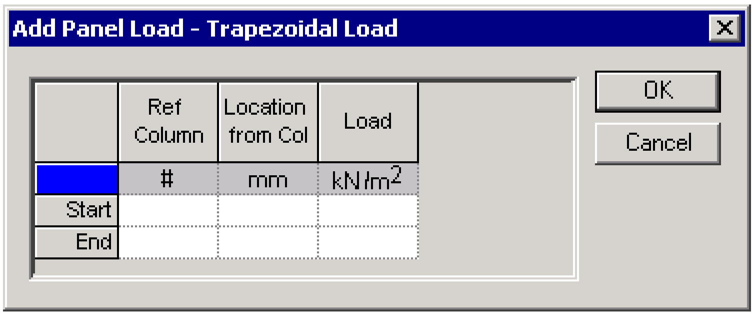


Add load on span:- Will add a linearly varying load over the full length of a nominated span. The dialog below will be presented for input of the span number and load values. Data focus is initially in the span number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end value to create a linearly varying load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load values are zero or if

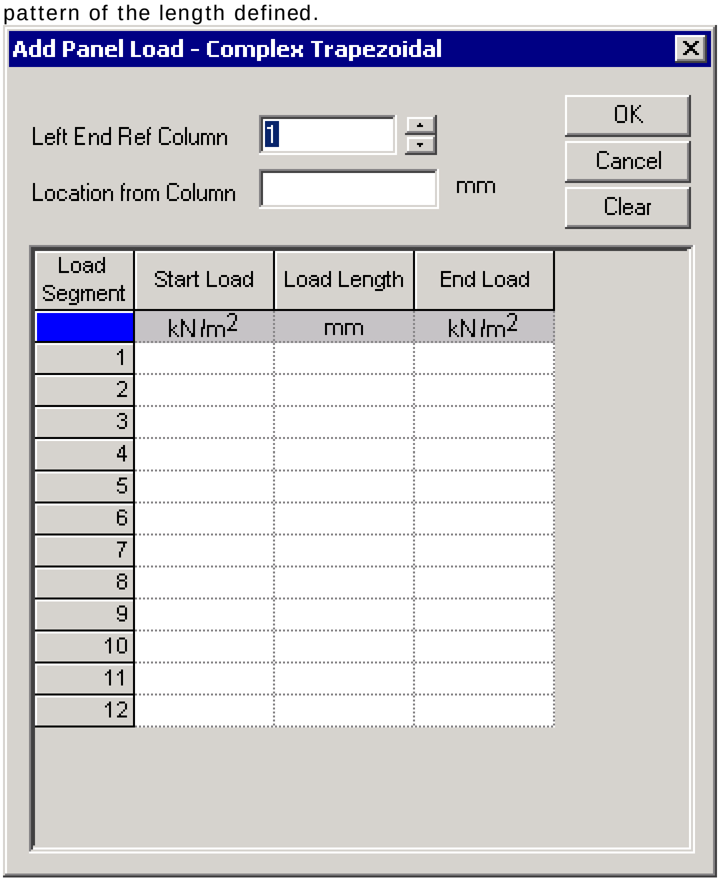
the span number does not exist.



Add general trapezoidal load:- Will add a linearly varying load over the length nominated. The dialog below will be presented for input of the load start and end locations and load values. Data focus is initially in the start reference column number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame or if the two load values are zero.



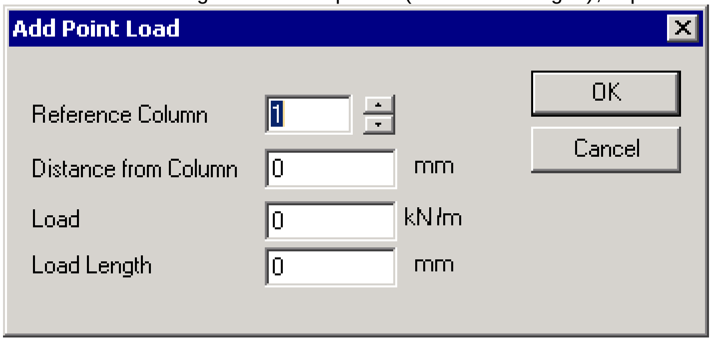
Add multiple trapezoidal loads:- Will add multiple linearly varying loads over the lengths nominated. The dialog below will be presented for input of the load start, load lengths and load values. Data focus is initially in the left end reference column number cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame. A load segment can have zero loads at both ends. RAPT will insert this as a gap in the load pattern of the length defined.



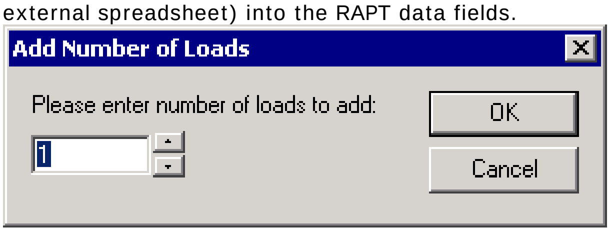
Add Concentrated load:- Most point loads actually apply load over at least the length of the item applying the load e.g. a transverse wall has a thickness. Many design codes also allow these loads to be distributed over an even wider area. Even though a point load is being defined, RAPT will apply it as a distributed load over the load length.

RAPT allows the designer to define a point load as a load/width with a length over which the load is applied. The width used is the panel width over the length of the load (except for one way nominal width designs where it is the effective width). The load will be applied centred at the defined load location. The designer should ensure that the full extent of the load will be within the frame length.

As well as defining a load location and a load value the designer is asked for a Load Length. This length cannot be zero. If a knife-edge load is required (zero load length), input it under the 7.2.4.1.1.4 Point Load input.



Add Number of Loads:-RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields.



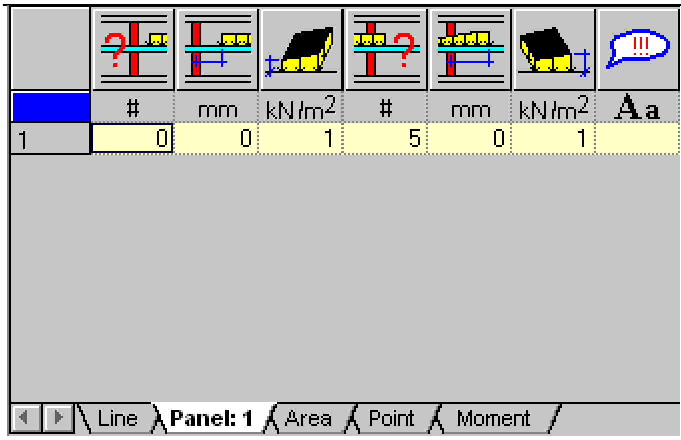
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

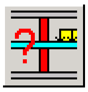
Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. panel loads can only be copied to panel loads. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

Data Definition

When editing or defining loads, RAPT will automatically convert the location entered to a location from the nearest column support to the left of or at the load.



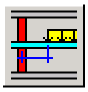

Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the load is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

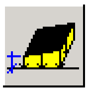
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

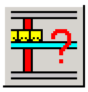

Left end of load from reference column

The distance from the left reference column to the left end of the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the start of the load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.


Load at left end

Value of the load at the left end of the load in force/length^2.

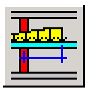

Right End reference column

The 7.2.3.2 Column Grid Reference from which the right end of the load is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a load.

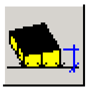
Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

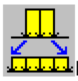

Right end of load from reference column

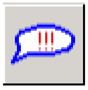
The distance from the right reference column to the right end of the load. If the distance entered moves the load into

another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the end of the load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

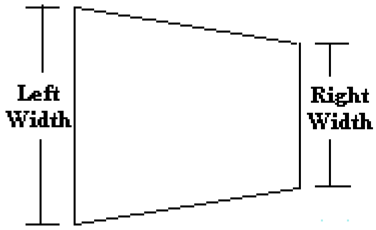
Load at right end
Value of the load at the right end of the load in force/length.

Live Load reduction
Only available for live load and alternate live load cases. The live load is defined as the full live load on the member before reduction. If a reduction factor is defined here, RAPT will use that reduction factor in all calculations using this load except the unbonded prestress minimum reinforcement calculations which require the full live load to be used. If the live load is defined as 3kPa and a live load reduction factor of .7 is specified then the actual live load used in strength and serviceability calculations will be .7 * 3 = 2.1kPa. This should only be used in situations where it is allowed by the design standard being used. The default value of the live load reduction is 1 (no reduction) and is the value used internally for all other load case types.

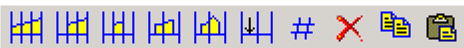
Description
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.1.3 Area Loads

A load defined as load/length^2. The load value is constant over the length of the load. The load is applied over a user defined width which can vary over the length of the load thus requiring the input of a width at the left end and a width at the right end of the load (as shown below). Area loads can start and finish anywhere in the frame, independent of support locations. Internally RAPT will break each load down into span loads for pattern load calculations and member loads for frame analysis. Positive loads are downwards. In two-way slab analysis, line loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm except when the load case is defined as "Other Load - Support Strip" in which case they will only be applied to the support (column) strip.



Area Loads Toolbar



The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

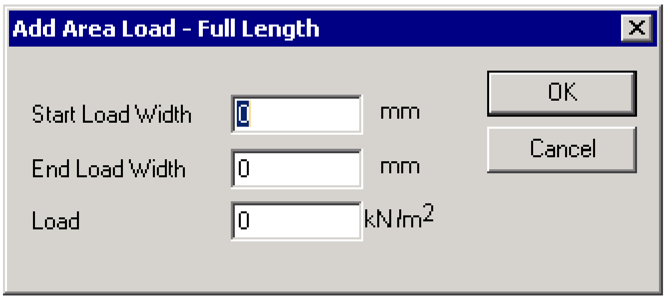
If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

A blank cell is regarded as having a value of zero.

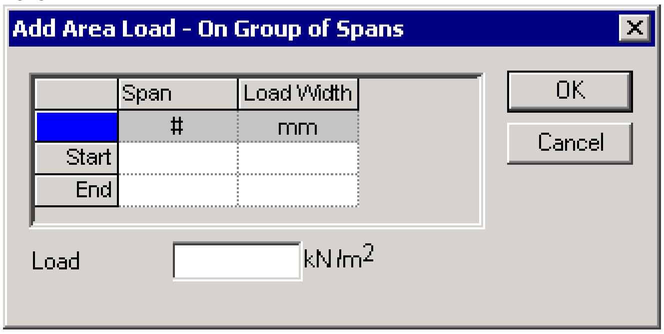
Where a Reference Column is asked for in any of the following dialogs, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

Add load full length of frame:- Will add a constant load defined as load/length^2 with a linearly varying width over the full length of the frame. The dialog below will be presented for input of the load value and the load widths. Data focus is initially in the start load width cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end width value to create a linearly varying width of load. RAPT will sort out the load positions from the frame

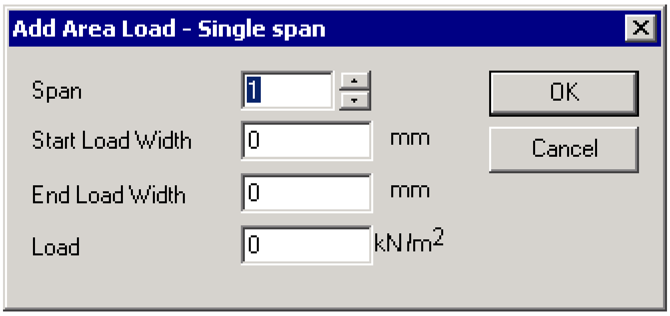
dimensions. RAPT will refuse to add the load if the two load width values are zero or if the load value is zero.



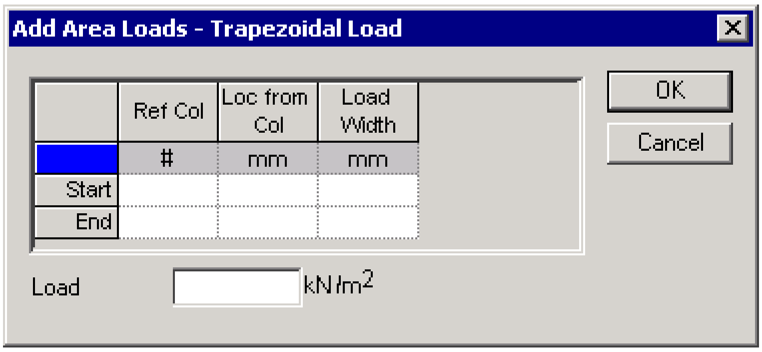
Add load on spans:- Will add a constant load defined as load/length^2 with a linearly varying width over the full length of the nominated group of spans. The dialog below will be presented for input of the span numbers, load value and the load widths. Data focus is initially in the start span cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end width value to create a linearly varying width of load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load width values are zero or if the load value is zero.



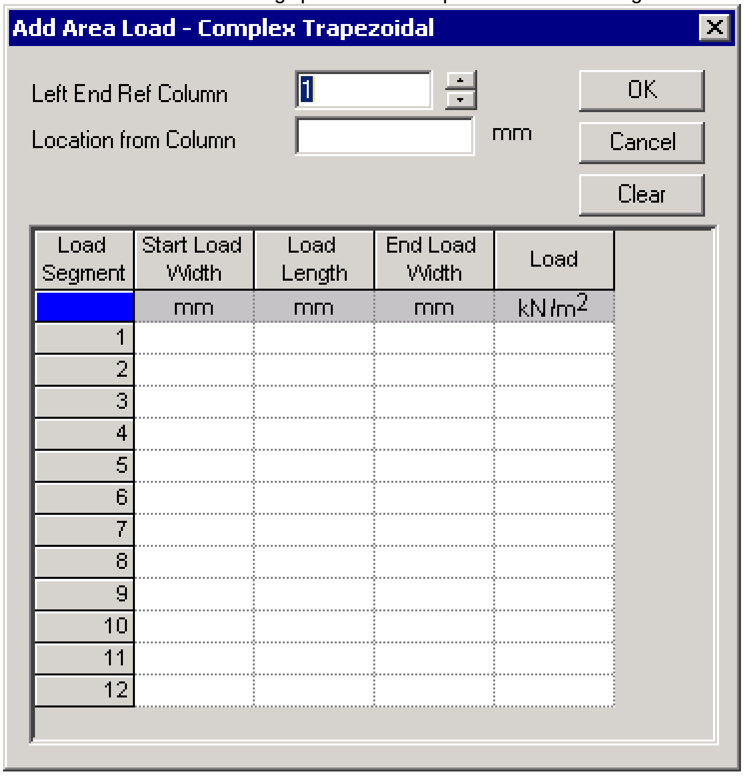
Add load on span:- Will add a constant load defined as load/length^2 with a linearly varying width over the full length of the nominated span. The dialog below will be presented for input of the span, load value and the load widths. Data focus is initially in the span cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can then over-ride this default end width value to create a linearly varying width of load. RAPT will sort out the load positions from the frame dimensions. RAPT will refuse to add the load if the two load width values are zero or if the load value is zero.



Add general trapezoidal load:- Will add a constant load defined as load/length^2 with a linearly varying width over the length nominated. The dialog below will be presented for input of the load start and end locations, load width values and the load. Data focus is initially in the start reference column number cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying width of load. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame or if the two load width values are zero or if the load is zero.



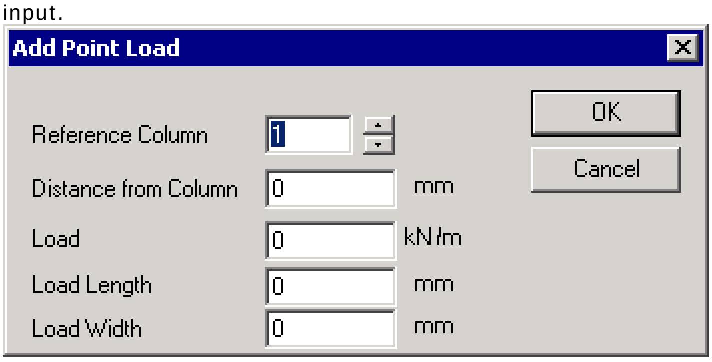
 Add multiple trapezoidal loads:- Will add multiple linearly varying load width over the lengths nominated. The dialog below will be presented for input of the load start, load lengths, load width and load values. Data focus is initially in the left end reference column number cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load width. RAPT will refuse to add the load if the end of the load is less than the start of the load or if one of the ends is outside the frame. A load segment can have zero load widths at both ends. RAPT will insert this as a gap in the load pattern of the length defined.



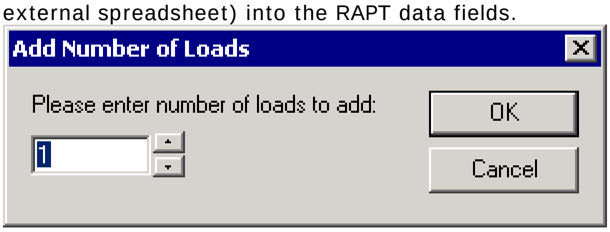
 Add Concentrated load:- Most point loads actually apply load over at least the length of the item applying the load e.g. a transverse wall has a thickness. Many design codes also allow these loads to be distributed over an even wider area. Even though a point load is being defined, RAPT will apply it as a distributed load over the load length.

RAPT allows the designer to define a point load as a load/width with a width of load a length over which the load is applied. The load will be applied centred at the defined load location. The designer should ensure that the full extent of the load will be within the frame length.

As well as defining a load location and a load value and load width, the designer is asked for a Load Length. The width and length cannot be zero. If a knife-edge load is required (zero load length), input it under the 7.2.4.1.1.4 Point Load input.



 Add Number of Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields.

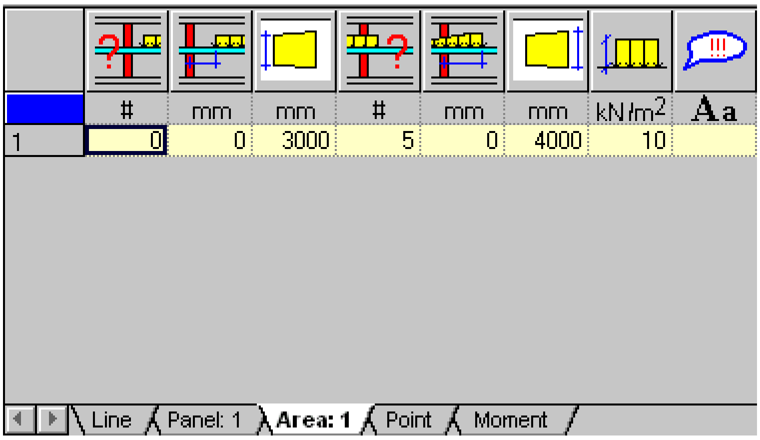


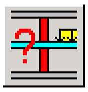
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. area loads can only be copied to area loads. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

Data Definition



Left End reference column

The 7.2.3.2 Column Grid Reference from which the left end of the load is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

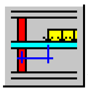
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

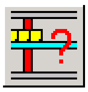
Left end of load from reference column

The distance from the left reference column to the left end of the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the start of the load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

Width at left end of load

Width of the loaded area at the left end of the load. RAPT does not attempt to relate this width to any dimension of the structure.

Right End reference column

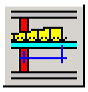
The 7.2.3.2 Column Grid Reference from which the right end of the load is dimensioned. RAPT will always adjust the

column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the load. This is necessary for the internal calculations for adding and deleting spans.

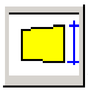
In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

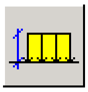
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a load.

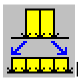
Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

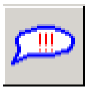
Right end of load from reference column
The distance from the right reference column to the right end of the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the end of the load.

Obviously the left end of a load must be to the left of the right end so make sure when modifying load ends that the order of modification will ensure this.

Width at Right end of load
Width of the loaded area at the right end of the load. RAPT does not attempt to relate this width to any dimension of the structure.

Load
The load in load/length^2 which is applied on the loaded area defined above.

Live Load reduction
Only available for live load and alternate live load cases. The live load is defined as the full live load on the member before reduction. If a reduction factor is defined here, RAPT will use that reduction factor in all calculations using this load except the unbonded prestress minimum reinforcement calculations which require the full live load to be used. If the live load is defined as 3kPa and a live load reduction factor of .7 is specified then the actual live load used in strength and serviceability calculations will be .7 * 3 = 2.1kPa. This should only be used in situations where it is allowed by the design standard being used. The default value of the live load reduction is 1 (no reduction) and is the value used internally for all other load case types.

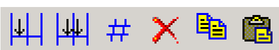
Description
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.1.4 Point Loads

A load defined as concentrated point load. RAPT allows a concentrated load to be defined with a length over which the load can be spread. If the Load Length is defined as zero, the load is treated as a knife-edge load so it's effect will be concentrated at the point of application. In design, RAPT will place design points immediately to each side of the load to trap the critical shear values. This type of load is very severe and rarely occurs like this in practice. Normally a concentrated load will have a length of application which will tend to smooth the effect of the load and reduce the design moments and shears caused by it. To achieve this, the user can define the load length in the Point Load screen, or use the various versions of 7.2.4.1.1.1 Concentrated Linear Loads for Line, Area and Panel loads.

Positive loads are downwards. In two-way slab analysis, point loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm except when the load case is defined as "Other Load - Support Strip" in which case they will only be applied to the support (column) strip.

Point Loads Toolbar



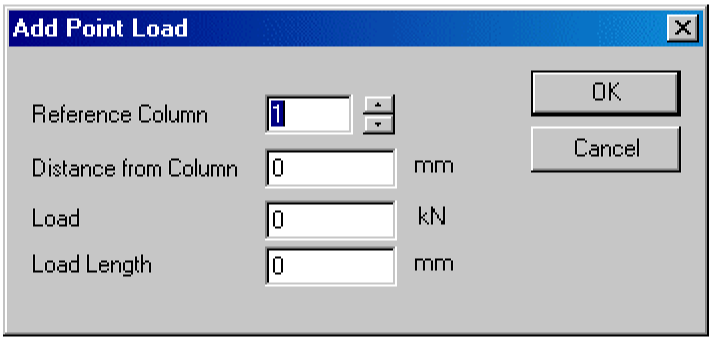
The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

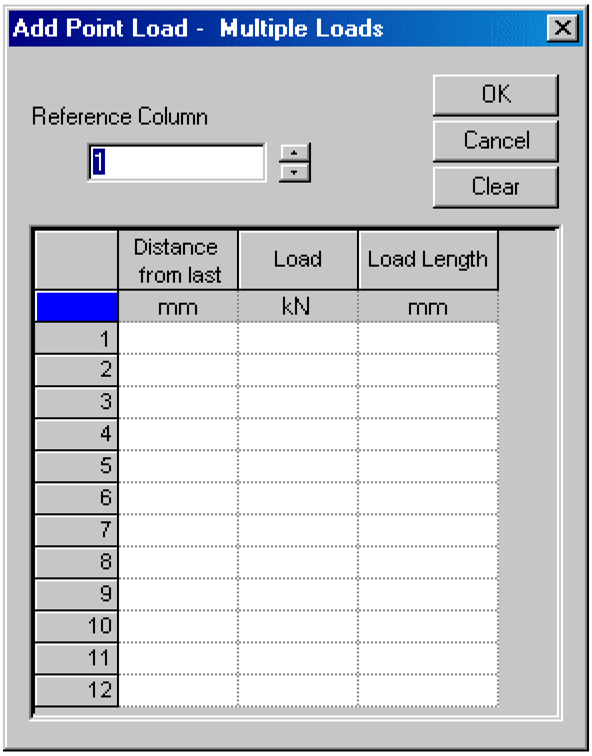
A blank cell is regarded as having a value of zero.

Where a Reference Column is asked for in any of the following dialogs, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.


Add single point load:- Will add a single point load at the location nominated. The dialog below will be presented for input of the load location, load value and load length. Data focus is initially in the reference column number cell. RAPT will refuse to add the load if the load or any part of the load length is outside the frame or if the load value is zero.

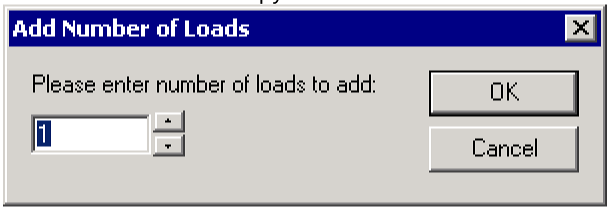



Add multiple point loads:- Will add a group of point loads at the locations nominated. The dialog below will be presented for input of the reference column number, the distance from the last load for each load (for the first load it is the distance from the reference column) and each load value and load length. Data focus is initially in the reference column number cell. RAPT will refuse to add a load if the load or load length is outside the frame or if the load value is zero.




Add Number of Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields. This is needed for the Windows Copy/Paste functions but is not

needed for the RAPT Copy/Paste functions described below which will automatically create the data rows as required.

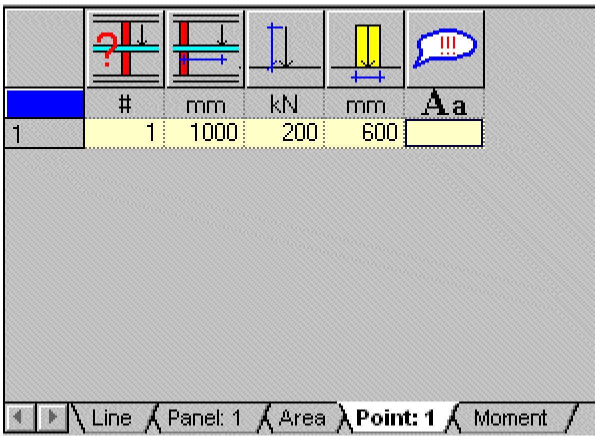


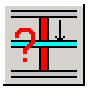
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. point loads can only be copied to point loads. RAPT automatically creates the space in the data grid into which the loads are to be pasted, so it is not necessary to create the rows for the data to be inserted into before the paste operation is performed.

Data Definition

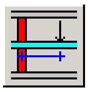


Reference column

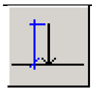
The 7.2.3.2 Column Grid Reference from which the load location is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the load. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

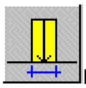
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the location of a load in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the location of a load.

Distance to Load from reference column

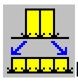
The distance from the reference column to the load. If the distance entered moves the load into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the load.

Load

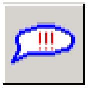
Value of the load.

Load Length

Length the point load will be spread over, centred on the load location. If set to 0, the load will be treated as a knife-edge point load. If a length is defined, the load will be spread as a distributed load over the length defined. This length must be fully within the frame.

Live Load reduction

Only available for live load and alternate live load cases. The live load is defined as the full live load on the member before reduction. If a reduction factor is defined here, RAPT will use that reduction factor in all calculations using this load except the unbonded prestress minimum reinforcement calculations which require the full live load to be used. If the live load is defined as 3kPa and a live load reduction factor of .7 is specified then the actual live load used in strength and serviceability calculations will be .7 * 3 = 2.1kPa. This should only be used in situations where it is allowed by the design standard being used. The default value of the live load reduction is 1 (no reduction) and is the value used internally for all other load case types.

Description

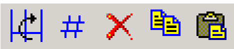
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.1.5 Point Moments

A moment defined as load * length. The moment is defined as concentrated at the point of application. In design, RAPT will place design points immediately to each side of the moment to trap the critical moment and shear locations.

Positive moments are clockwise. In two-way slab analysis, point moments are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm except when the load case is defined as "Other Load - Support Strip" in which case they will only be applied to the support (column) strip.

Point Moments Toolbar



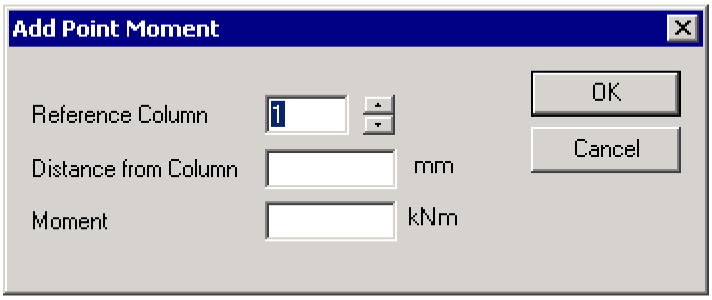
The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

A blank cell is regarded as having a value of zero.

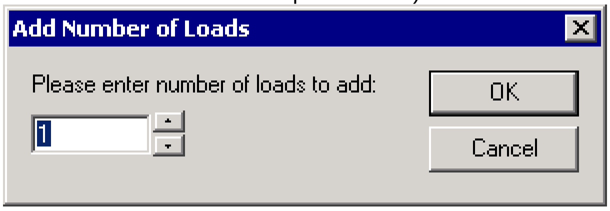
Where a Reference Column is asked for in any of the following dialogs, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

Add point moment:- Will add a single point moment at the location nominated. The dialog below will be presented for input of the moment location and moment value. Data focus is initially in the reference column number cell. RAPT will refuse to add the moment if the moment is outside the frame or if the moment value is zero.



Add Number of Loads:- RAPT will allow the user to define a number of moments and create the space for these moments in the data grid. No moment values will be input by RAPT. The designer must add all moment data for these loads. This has been done to allow designers to copy moments from another data source (either another loads grid in

RAPT or from an external spreadsheet) into the RAPT data fields.

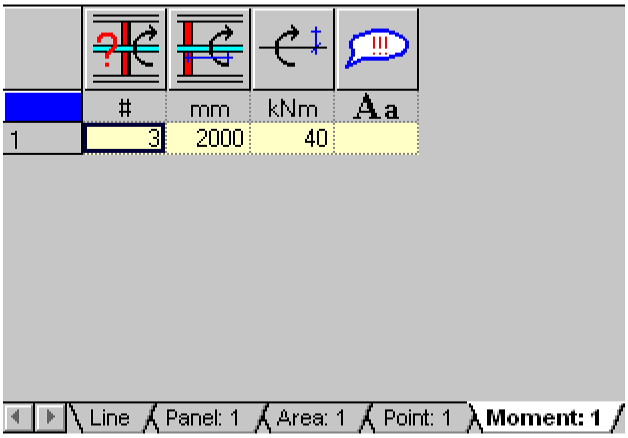


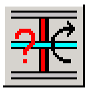
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. moments can only be copied to moments. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

Data Definition

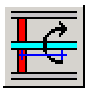


Reference column

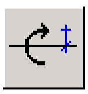
The 7.2.3.2 Column Grid Reference from which the moment location is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the moment. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

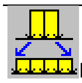
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the location of a moment in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the location of a moment.

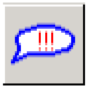
Distance to Moment from reference column

The distance from the reference column to the moment. If the distance entered moves the moment into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the moment.

Moment

Value of the load as load * length. Clockwise moments are positive sign.

Live Load reduction
Only available for live load and alternate live load cases. The live load is defined as the full live load on the member before reduction. If a reduction factor is defined here, RAPT will use that reduction factor in all calculations using this moment except the unbonded prestress minimum reinforcement calculations which require the full live load to be used. If the live load is defined as 3kPa and a live load reduction factor of .7 is specified then the actual live load used in strength and serviceability calculations will be .7 * 3 = 2.1kPa. This should only be used in situations where it is allowed by the design standard being used. The default value of the live load reduction is 1 (no reduction) and is the value used internally for all other load case types.

Description
Designer's description of the moment. This is not a compulsory data field.

7.2.4.1.2 Moment Diagrams

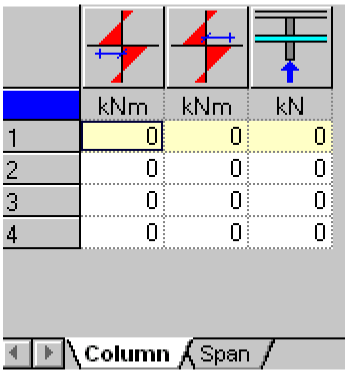
7.2.4.1.2.1 Bending Moment Diagram - Columns

When defining a moment/shear diagram, the designer can define column bending moments and reactions. These are included in the calculation of the various design combinations. If punching shear is not an issue, they will have no effect on the design.

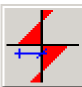
The sign convention is

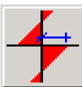
1. Reactions - Compression is positive
2. Moments - Clockwise positive applied to the member at the joint.

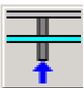
Data Definition



	kNm	kNm	kN
1	0	0	0
2	0	0	0
3	0	0	0
4	0	0	0

Moment in Column above
The moment in the column above, if it exists, at the centreline of the spanning member (at node connection point of the members).

Moment in Column below
The moment in the column below, if it exists, at the centreline of the spanning member (at node connection point of the members).

Reaction
The reaction at this support location.

Load Case Graphics Window

Refer to 7.2.4.1 Moment Diagram Graphics for a typical graphics view.

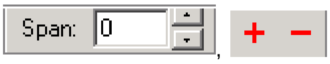
7.2.4.1.2.2 Bending Moment Diagram - Spans

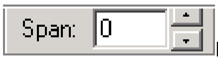
RAPT allows the designer to define moment/shear diagrams on a span basis and the data is presented to the designer individually for each span. The minimum number of points in each span is two, one at each end of the span. These two points would produce a straight line bending moment diagram over the length of the span (e.g. sway diagram). Any number of points can be defined by the designer to create more complicated moment/shear diagrams. The values of

moment and shear can be entered individually into the data cells or copied from another data source that is formatted in the same way.

RAPT uses this bending moment/shear diagram as it is defined. No analysis is carried out on it except to determine values at design locations. These are interpolated from a straight line between the defined values on either side of the design point. In two way slab designs the normal lateral distribution factors are used to divide the moments between column and middle strips.

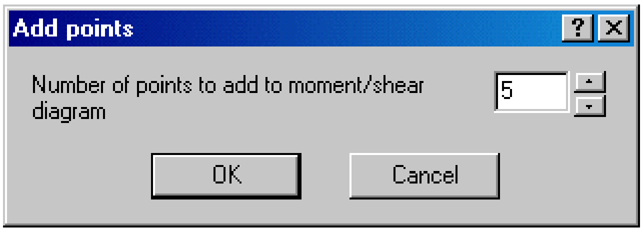
Bending Moment Diagram Spans Toolbar





Nominate Span to View:-
Allows the designer to select the span in which the bending moments and shears are to be viewed, added or modified. A span number can be typed in (press enter or click in the span window to finish edit and show the relevant data in the span window) or selected using the spin buttons (automatically shows the relevant data in the span window).


Add Moment Diagram Points:-
Allows the designer to add extra points to the bending moment diagram for this span using the following dialog.

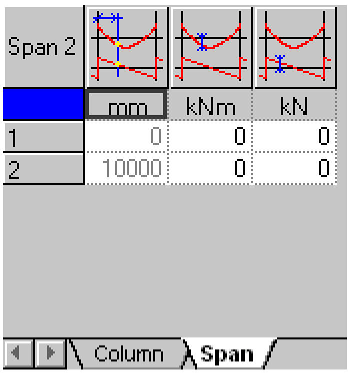


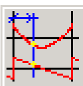
The span data grid starts with 2 points by default and this is the minimum number of points allowed in a span to define a bending moment/shear diagram. These are always the first and last points in the span and their locations are not editable and must be at location = 0 and location = span length respectively. These 2 points define a straight line bending moment and shear diagram in the span.

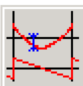
Extra points can be added between these points to define more complicated bending moment diagrams. The extra points will be added at the location selected (select a row by clicking the left mouse button in the point number data column) and the selected point will be moved down and the new points will be added between the point before the selected point and the selected point. RAPT will automatically place the new points equally spaced between the locations of the selected point and the point before it in the grid.

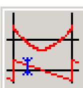
Delete Moment Diagram Points:-
Deletes the selected moment/shear points. Multiple moment/shear points can be deleted at once using the selection logic in 4.4.3 Cell Selection. The first and last points in the span cannot be deleted.

Data Definition




Location:- Location of this point from the left end of the span. The locations of the first and last points are not editable and are fixed at the left and right ends of the span.


Moment:- Bending Moment value at this point.


Shear:- Coexisting shear value at this point. If there are only 2 points nominated in a span (straight line moment and shear diagram) RAPT will automatically calculate the shears to be consistent with the defined bending moments as the moments are entered. The designer can over-ride the RAPT default shear values if desired.

Load Case Graphics Window

Refer to 7.2.4.1 Moment Diagram Graphics for a typical graphics view.

7.2.4.1.3 Moment Envelopes

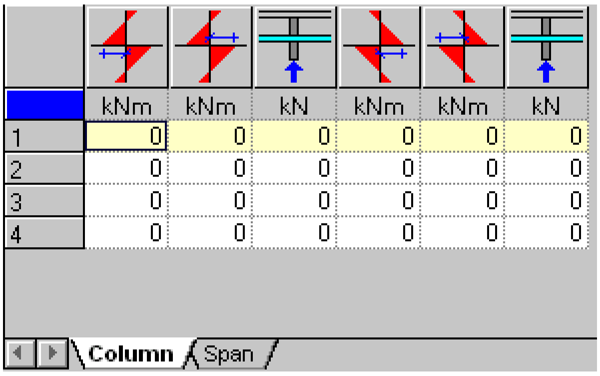
7.2.4.1.3.1 Bending Moment Envelope - Columns

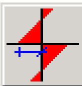
When defining a moment/shear envelope, the designer can define column bending moments and reactions for a primary and a reversal condition or an envelope of effects. These are included in the calculation of the various design combinations. If punching shear is not an issue, they will have no effect on the design.

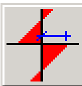
The sign convention is

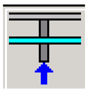
1. Reactions - Compression is positive
2. Moments - Clockwise positive applied to the member at the joint.

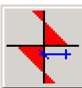
Data Definition

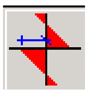


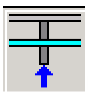
Primary Moment in Column above
The moment in the column above, if it exists, at the centreline of the spanning member (at node connection point of the members).

Primary Moment in Column below
The moment in the column below, if it exists, at the centreline of the spanning member (at node connection point of the members).

Primary Reaction
The reaction at this support location.

Reversal Moment in column above
The reverse moment in the column above, if it exists, at the centreline of the spanning member (at node connection point of the members).

Reversal Moment in column below
The reverse moment in the column below, if it exists, at the centreline of the spanning member (at node connection point of the members).

Reversal Reaction
The reverse reaction at this support location.

Load Case Graphics Window

Refer to 7.2.4.1 Moment Diagram Graphics for a typical graphics view.

7.2.4.1.3.2 Bending Moment Envelope - Spans

RAPT allows the designer to define moment/shear envelopes on a span basis and the data is presented to the designer individually for each span. The minimum number of points in each span is two, one at each end of the span. These two points would produce a straight line bending moment envelope over the length of the span (e.g. sway diagram). Any number of points can be defined by the designer to create more complicated moment/shear envelopes. The values of moment and shear can be entered individually into the data cells or copied from another data source that is formatted in the same way.

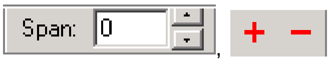
RAPT uses this bending moment/shear envelope as it is defined. No analysis is carried out on it except to determine values at design locations. These are interpolated from a straight line between the defined values on either side of the design point. In two way slab designs the normal lateral distribution factors are used to divide the moments between column and middle strips.

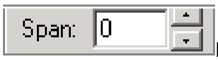
Note that to define an envelope of effects properly, it may be necessary to provide two separate envelopes of moment and shear. These would give

1. An envelope of the different bending moment cases with the coexisting shear forces at each point.
2. An envelope of the different shear force cases with the coexisting bending moments at each point.

For shear design, especially for prestressed members, the moments and shears must be coexisting as the concrete shear capacity calculation in most design standards relates the shear capacity to the relationship between the applied bending moment and shear force at a point.

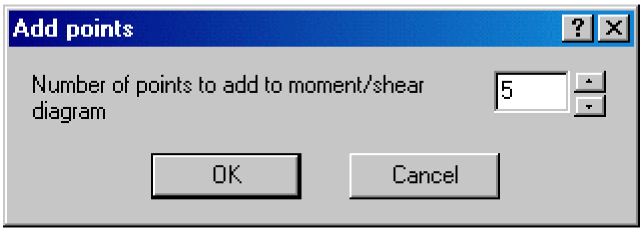
Bending Moment Envelope Spans Toolbar





Nominate Span to View:-
Allows the designer to select the span in which the bending moments and shears are to be viewed, added or modified. A span number can be typed in (press enter or click in the span window to finish edit and show the relevant data in the span window) or selected using the spin buttons (automatically shows the relevant data in the span window).

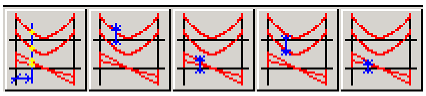

Add Moment Envelope Points:-
Allows the designer to add extra points to the bending moment envelope for this span using the following dialog.

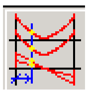


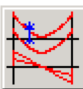
The span data grid starts with 2 points by default and this is the minimum number of points allowed in a span to define a bending moment/shear envelope. These are always the first and last points in the span and their locations are not editable and must be at location = 0 and location = span length respectively. These 2 points define a straight line bending moment and shear diagram in the span.

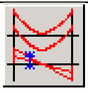
Extra points can be added between these points to define more complicated bending moment envelopes. The extra points will be added at the location selected (select a row by clicking the left mouse button in the point number data column) and the selected point will be moved down and the new points will be added between the point before the selected point and the selected point. RAPT will automatically place the new points equally spaced between the locations of the selected point and the point before it in the grid.

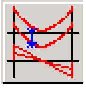
Delete Moment Envelope Points:-
Deletes the selected moment/shear points. Multiple moment/shear points can be deleted at once using the selection logic in 4.4.3 Cell Selection. The first and last points in the span cannot be deleted.

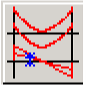



Location:- Location of this point from the left end of the span. The locations of the first and last points are not editable and are fixed at the left and right ends of the span.


Primary Moment:- Bending Moment value at this point for one side of the moment envelope.

Primary Shear:- Coexisting Shear value at this point for the primary side of the moment envelope. If there are only 2 points nominated in a span (straight line moment and shear diagram) RAPT will automatically calculate the shears to be consistent with the defined bending moments as the moments are entered. The designer can over-ride the RAPT default shear values if desired.

Reversal Moment:- Bending Moment value at this point for the other side of the moment envelope.

Reversal Shear:- Coexisting Shear value at this point for the Reversal side of the moment envelope. If there are only 2 points nominated in a span (straight line moment and shear diagram) RAPT will automatically calculate the shears to be consistent with the defined bending moments as the moments are entered. The designer can over-ride the RAPT default shear values if desired.

Load Case Graphics Window

Refer to 7.2.4.1 Moment Diagram Graphics for a typical graphics view.

7.2.4.1.4 Moving Loads

Moving load analysis often requires the combination of a fix pattern of loads, normally a uniform load, and a moving pattern of loads. To allow a designer to investigate all possible combinations of loads, RAPT allows the designer to specify both types of loads in a Moving Load case.

RAPT will analyse the fixed load pattern first as a partial live load case, resulting in the most critical moment controlled envelope and shear controlled envelope for this loading. The moving load pattern is then analysed at each location along the member and combined with the fixed load pattern as an extra partial load case to create the most critical envelopes for this combination of fixed load pattern plus this moving load position. The results of this combination for each load position are then compared to produce a final moment controlled envelope and a shear controlled envelope for the moving load case.

Fixed Load Pattern
Any number of loads can be included in the fixed load pattern. They may consist of

1. Line Loads - 7.2.4.1.1.1 7.2.4.1.1.1
2. Panel Loads - 7.2.4.1.1.2 7.2.4.1.1.2
3. Area Loads - 7.2.4.1.1.3 7.2.4.1.1.3
4. Point Loads - 7.2.4.1.1.4 7.2.4.1.1.4

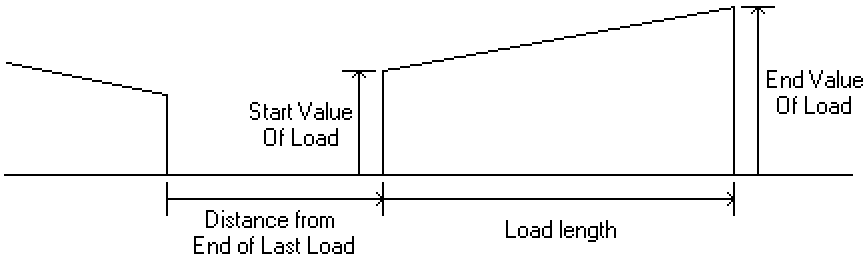
The definition of these loads is the same as for 7.2.4.1.1 Applied Loads.

Moving Load Pattern
RAPT's moving load analysis allows the user to define any pattern of loads desired. This pattern of loads is then placed at a series of positions along the frame to produce moment controlled and shear controlled envelopes of the effects of the moving load. The load positions are determined by RAPT in response to

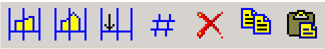
1. Information input by the user regarding the spacing of locations, the minimum number of positions on each span and the direction in which the load is to travel. Using this information, RAPT will initially place the load pattern with the first load 1 full increment in from the start of the frame. It will then move the load pattern along the frame in increments until the last load in the pattern clears the end of the frame. If the designer places limits on the extent over which the load is to be able to travel, the loading pattern will only be placed on the frame once the complete pattern can fit in between the limits defined.
2. Logical requirements near the ends of spans and cantilevers and at mid-span to determine the worst effect of the load pattern on moment and shear. To achieve this RAPT will place the load pattern so that the end of each span or cantilever or the mid-span point corresponds with
   - the centre of each point load
   - the mid distance between the ends of each pair of loads
   - the ends and the quarter points of each distributed load

7.2.4.1.4.1 Moving Line Loads

A load defined as load/length. The load can vary in value linearly over it's length so a left load value and a right load value are required as shown below. Line loads can start and finish anywhere in the frame, independent of support locations. Positive loads are downwards. In two-way slab analysis, line loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm.



Line Loads Toolbar

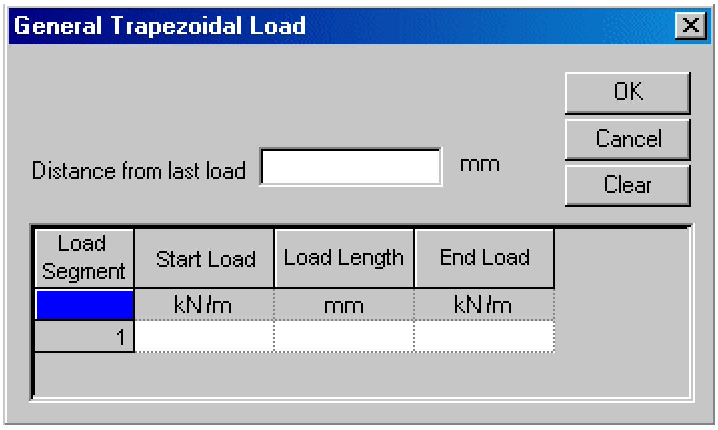


The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

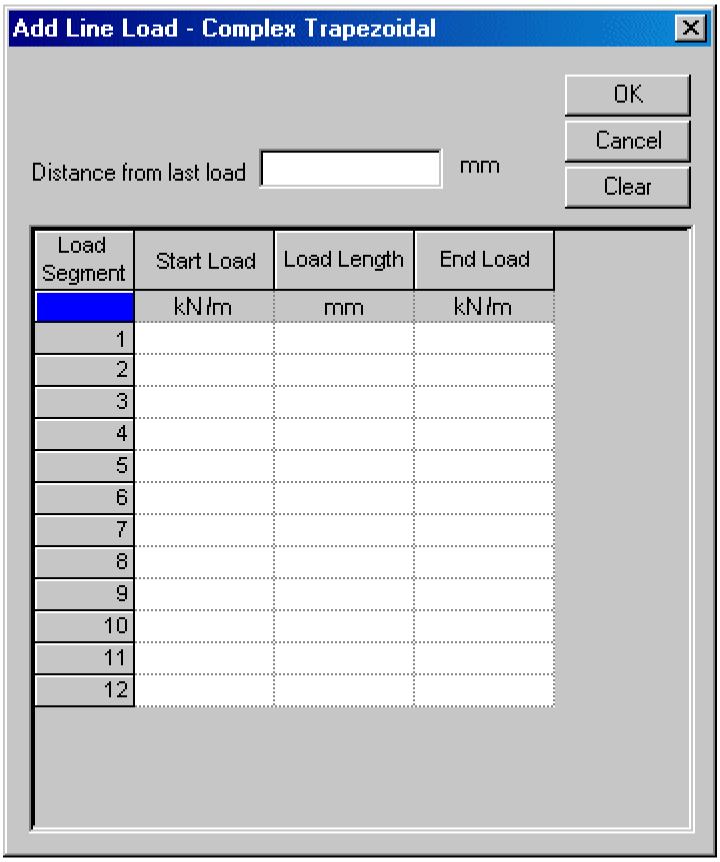
If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

A blank cell is regarded as having a value of zero.

Add general trapezoidal load:- Will add a linearly varying load over the length nominated. The dialog below will be presented for input of the load start and end locations and load values. Data focus is initially in the Distance from Last Load cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the load length is negative or if one of the ends is outside the frame or if the two load values are zero.



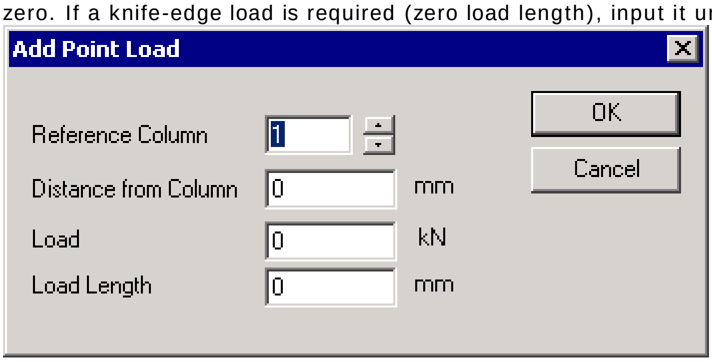
Add multiple trapezoidal loads:- Will add multiple linearly varying loads over the lengths nominated. The dialog below will be presented for input of the load start, load lengths and load values. Data focus is initially in the Distance From Last Load cell. When the start load value is initially entered, the end load value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load. RAPT will refuse to add the load if the load length is negative or if one of the ends is outside the frame. A load segment can have zero loads at both ends. RAPT will insert this as a gap in the load pattern of the length defined.



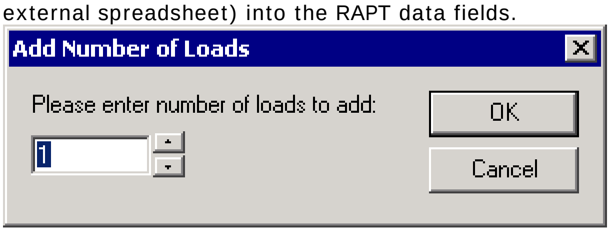
Add Concentrated load:- Most point loads actually apply load over at least the length of the item applying the load e.g. a transverse wall has a thickness. Many design codes also allow these loads to be distributed over an even wider area. Even though a point load is being defined, RAPT will apply it as a distributed load over the load length.

RAPT allows the designer to define a point load with a length over which the load is applied. The load will be applied centred at the defined load location. The designer should ensure that the full extent of the load will be within the frame length.

As well as defining a load location and a load value the designer is asked for a Load Length. This length cannot be zero. If a knife-edge load is required (zero load length), input it under the 7.2.4.1.1.4 Point Load input.



Add Number of Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields.

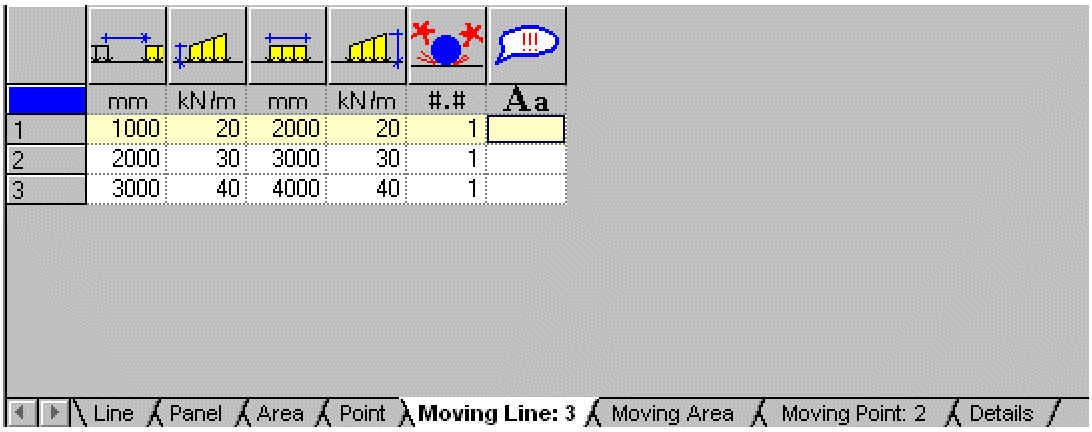


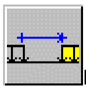
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

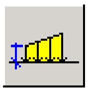
Copy:- Copy a selected group of loads to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

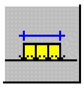
Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. moving line loads can only be copied to moving line loads. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

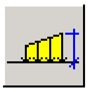
Data Definition

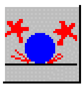


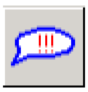
Left End of Load from End of Previous Load
The distance from the end of the moving line load to the left of this load to the left end of the load.

Load at Left End
Value of the load at the left end of the load in force/length.

Length of This Load
The length of this load.

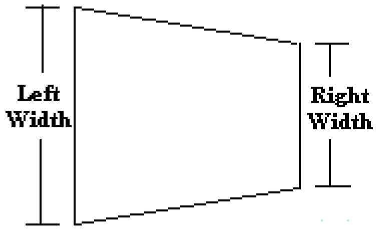
Load at right end
Value of the load at the right end of the load in force/length.

Impact Factor
The Impact factor associated with this load as defined in the relevant loading code. RAPT will multiple the load by this factor when including it in any load combination.

Description
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.4.2 Moving Area Loads

A load defined as load/length^2. The load value is constant over the length of the load. The load is applied over a user defined width which can vary over the length of the load thus requiring the input of a width at the left end and a width at the right end of the load (as shown below). Positive loads are downwards. In two-way slab analysis, line loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm.



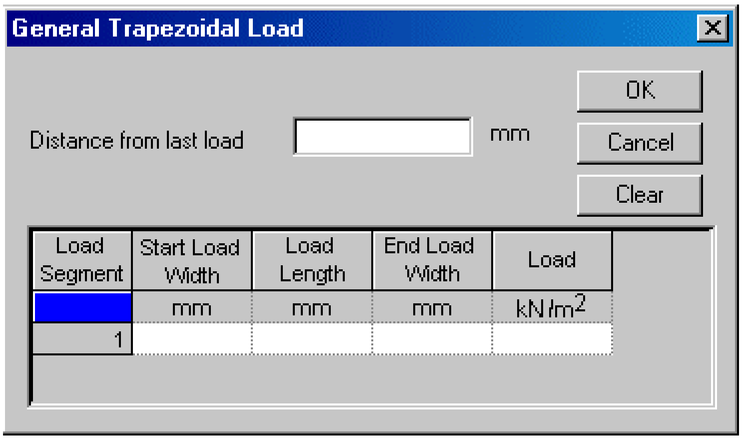
Area Loads Toolbar

The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

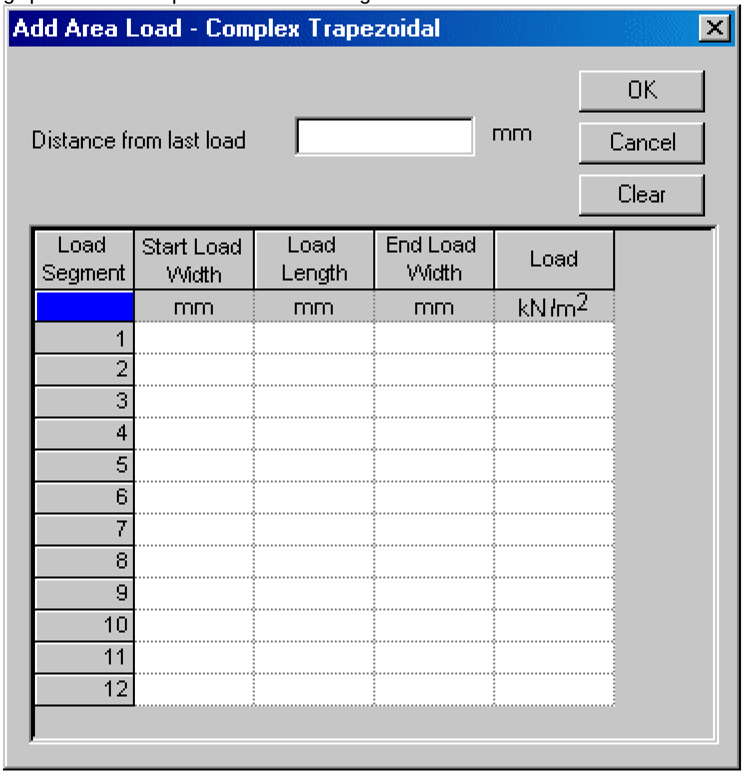
A blank cell is regarded as having a value of zero.

Add general moving trapezoidal load:- Will add a constant load defined as load/length^2 with a linearly varying width over the length nominated. The dialog below will be presented for input of the load start and end locations, load width values and the load. Data focus is initially in the Distance From Last Load cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying width of load. RAPT will refuse to add the load if the load length is negative or if one of the ends is outside the frame or if the two load width values are zero or if the load is zero.



Add multiple moving trapezoidal loads:- Will add multiple linearly varying load width over the lengths nominated. The dialog below will be presented for input of the load start, load lengths, load width and load values. Data focus is initially in Distance From Last Load cell. When the start load width value is initially entered, the end load width value will default to the same value creating a uniformly distributed load. The designer can over-ride this default end value to create a linearly varying load width. RAPT will refuse to add the load if the load length is negative or if one of the ends is outside the frame. A load segment can have zero load widths at both ends. RAPT will insert this as a

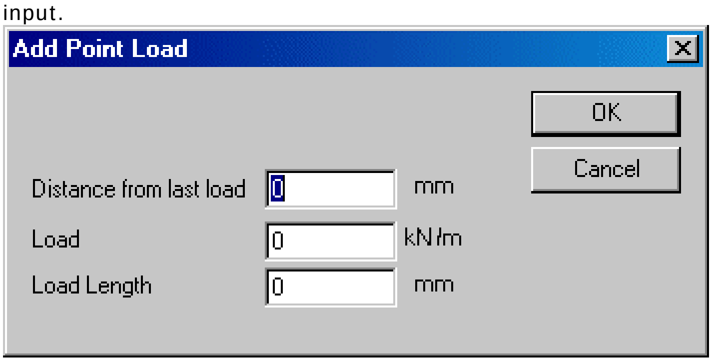
gap in the load pattern of the length defined.



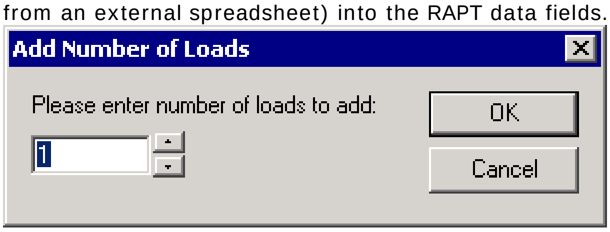
Add Concentrated moving load:- Most point loads actually apply load over at least the length of the item applying the load e.g. a transverse wall has a thickness. Many design codes also allow these loads to be distributed over an even wider area. Even though a point load is being defined, RAPT will apply it as a distributed load over the load length.

RAPT allows the designer to define a point load as a load/width with a width of load a length over which the load is applied. The load will be applied centred at the defined load location. The designer should ensure that the full extent of the load will be within the frame length.

As well as defining a load location and a load value and load width, the designer is asked for a Load Length. The width and length cannot be zero. If a knife-edge load is required (zero load length), input it under the 7.2.4.1.1.4 Point Load input.



Add Number of moving Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields.

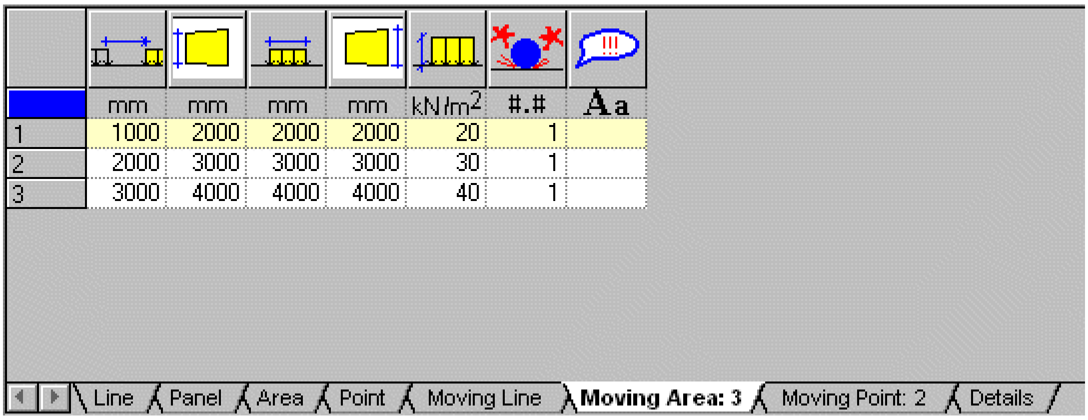


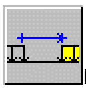
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

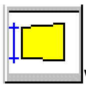
Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. moving area loads can only be copied to moving area loads. RAPT automatically creates the space in the data grid into which the loads are to be copied, so it is not necessary to create the rows for the data to be inserted into.

Data Definition

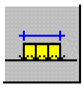


Left End of Load from End of Previous Load

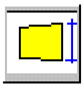
The distance from the end of the moving area load to the left of this load to the left end of the load.

Width at Left End of Load

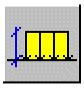
Width of the loaded area at the left end of the load. RAPT does not attempt to relate this width to any dimension of the structure.

Length of This Load

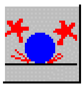
The length of this load.

Width at Right end of load

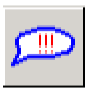
Width of the loaded area at the right end of the load. RAPT does not attempt to relate this width to any dimension of the structure.

Load

The load in load/length^2 which is applied on the loaded area defined above.

Impact Factor

The Impact factor associated with this load as defined in the relevant loading code. RAPT will multiple the load by this factor when including it in any load combination.

Description

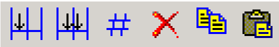
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.4.3 Moving Point Loads

A load defined as concentrated point load. RAPT allows a concentrated load to be defined with a length over which the load can be spread. If the Load Length is defined as zero, the load is treated as a knife-edge load so it's effect will be concentrated at the point of application. In design, RAPT will place design points immediately to each side of the load to trap the critical shear values. This type of load is very severe and rarely occurs like this in practice. Normally a concentrated load will have a length of application which will tend to smooth the effect of the load and reduce the design moments and shears caused by it. To achieve this, the user can define the load length in the Point Load screen, or use the various versions of 7.2.4.1.1.1 Concentrated Linear Loads for Line, Area and Panel loads.

Positive loads are downwards. In two-way slab analysis, point loads are divided between column and middle strips using the factors in 7.2.4.3 frame_input_lateral_factors.htm.

Point Loads Toolbar

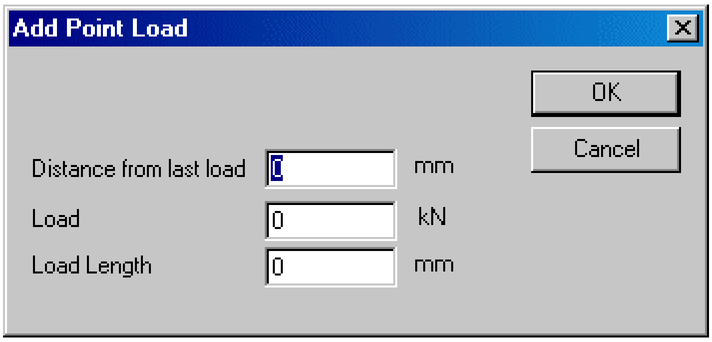


The default button in the loads dialogs is always OK (shows 3D or expressed). If the focus is in an edit cell, the OK button will always accept an Enter key press without having to move the focus to it. Pressing enter while in the edit cells will accept the data and insert the load as defined. Adding the load can be cancelled by clicking on or moving to and pressing enter on the Cancel Button.

If using the keyboard to navigate around the dialog, use the Tab key (forwards or Shift + Tab key (backwards) to move the focus. Using the Tab key at the end of a row will move focus to the start of the next row.

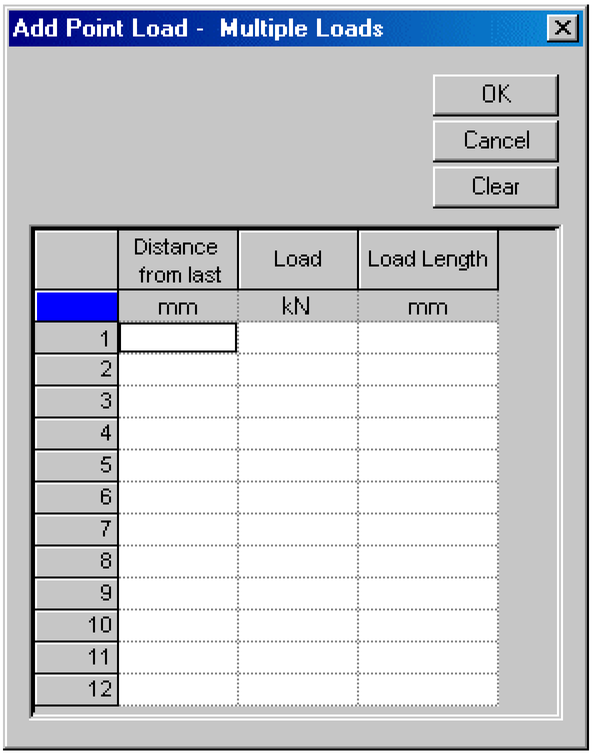
A blank cell is regarded as having a value of zero.

Add single point load:- Will add a single point load at the location nominated. The dialog below will be presented for input of the load location, load value and load length. Data focus is initially in the Distance from Last Load cell. RAPT will refuse to add the load if the load or any part of the load length is outside the frame or if the load value is zero.

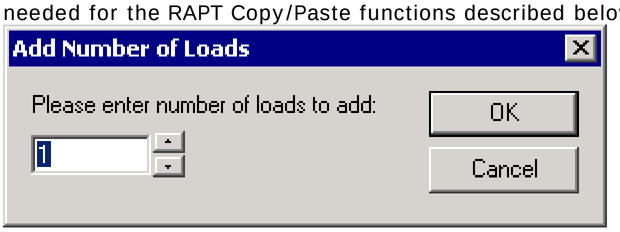


Add multiple point loads:- Will add a group of point loads at the locations nominated. The dialog below will be presented for input of the reference column number, the distance from the last load for each load (for the first load it is the distance from the reference column) and each load value and load length. Data focus is initially in the reference column number cell. RAPT will refuse to add a load if the load or load length is outside the frame or if the load value is

zero.



Add Number of Loads:- RAPT will allow the user to define a number of loads and create the space for these loads in the data grid. No load values will be input by RAPT. The designer must add all load data for these loads. This has been done to allow designers to copy loads from another data source (either another loads grid in RAPT or from an external spreadsheet) into the RAPT data fields. This is needed for the Windows Copy/Paste functions but is not needed for the RAPT Copy/Paste functions described below which will automatically create the data rows as required.

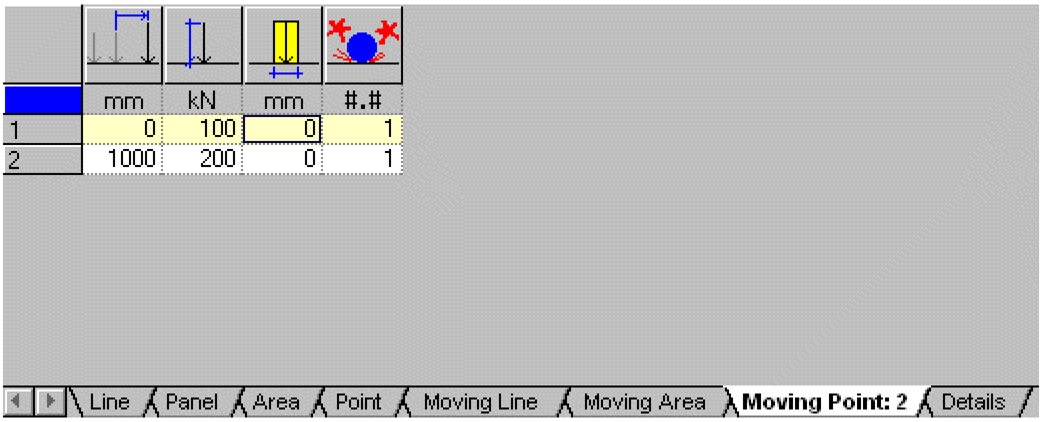


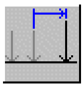
Delete load:- Deletes the selected loads. The loads must be selected by clicking on the load number cell. Multiple loads and non-continuous groups of loads can be selected using the 4.4.3 Cell Selection rules.

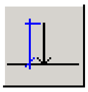
Copy:- Copy a selected group of loads for to memory for later Paste operation. The Copy option will only be available after a legal selection is made. Any loads in a load case can be selected by clicking the load number with the left mouse button. Groups of loads can be selected by click and dragging the mouse. Extra loads can be added to the selected group using Ctrl + Left Mouse key. Changes to the load arrangement after the selection is made, by adding or removing loads or modifying concrete shapes, will void any selection.

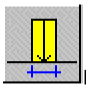
Paste:- Paste the selected loads. The Paste Option will only be available after a legal copy has been performed and the data grid into which the data is to be pasted is selected as follows; If a data row is selected by selecting the load number cell, the copied loads will be inserted at the row selected and all other data rows will be moved down to accommodate the pasted loads. If no data row is selected the copied loads will be added to the end of the existing loads. Multiple data rows cannot be selected for the paste operation. The paste can be into any load of the same type in any load case, e.g. point loads can only be copied to point loads. RAPT automatically creates the space in the data grid into which the loads are to be pasted, so it is not necessary to create the rows for the data to be inserted into before the paste operation is performed.

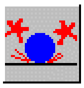
Data Definition

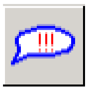


Distance to Load from previous load of this type
The distance from the centre of the last point load to the centre of this point load.

Load
Value of the load.

Load Length
Length the point load will be spread over, centred on the load location. If set to 0, the load will be treated as a knife-edge point load. If a length is defined, the load will be spread as a distributed load over the length defined. This length must be fully within the frame.

Impact Factor
The Impact factor associated with this load as defined in the relevant loading code. RAPT will multiple the load by this factor when including it in any load combination.

Description
Designer's description of the load. This is not a compulsory data field.

7.2.4.1.4.4 Moving Loads Details

This screen allows the designer to define the way in which the moving load pattern is to move along the frame.

RAPT determines the locations to position the moving load pattern in the following manner:

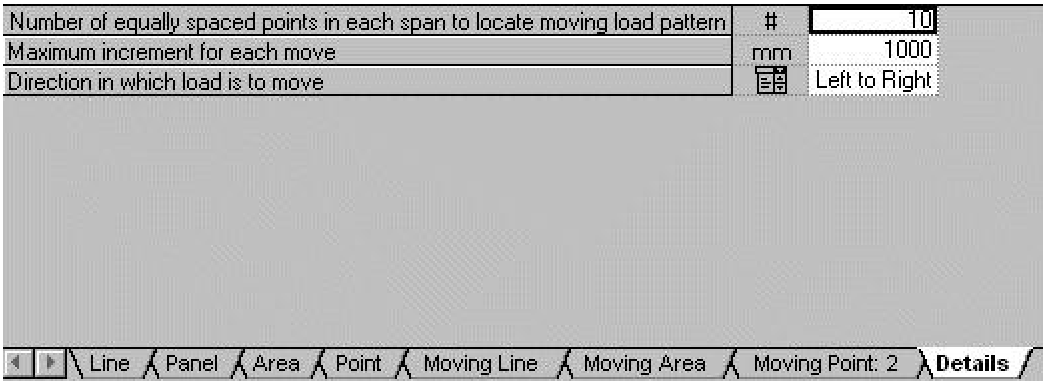
1. The start end of the loading pattern in the direction of movement is placed at one full increment from the start of the frame in that direction of movement. The load pattern is then moved in the direction of movement in increments until the entire moving load pattern clears the far end of the frame. If the designer places limits on the extent over which the load is to be able to travel, the loading pattern will only be placed on the frame once the complete pattern can fit in between the limits defined.
2. Special locations at each support as defined below.

The centre of the moving load pattern is determined as the mid-length between the start of the first load and the end of the last load in the pattern.

Special Locations:
RAPT determines the load locations required for Special Positions to achieve the following

1. All point loads are placed at the special location.
2. The 1/3rd points and locations of all distributed loads are placed at the special location.
3. The mid point between the end of one load (or a point load location) and the start of the next load is placed at the special location

Data Definition



Number of equally Spaced Points in each Span to Locate Moving Load Pattern

In each span, RAPT will initially determine the increment of movement of the moving load pattern by dividing the span length by the number of equally spaced points in each span.

Maximum Increment for Each Move

The Maximum Increment of Movement will be used to limit the increment of movement of the moving load pattern determined based on the number of points above.

Direction in which Load is to Move

RAPT offers 3 direction of movement options listed below.

1. Left to Right:- The defined moving load pattern will be placed at the locations defined above.
2. Right to left:- The moving load pattern will be turned around and placed at the locations defined above to simulate the same pattern travelling from right to left.
3. Return:- Both options 1 and 2 will be used.

Limit Left Extent of Moving Load

Allows the designer to define an area at the left end if the frame on which the moving load does not travel. The moving load pattern will not be placed on the frame until the complete length of the load pattern is clear of this length of the frame.

Left Extent Reference Column

Distance from Left Reference Column To Left Load Extent Limit

The location of the left end extent limit of the load pattern.

Limit Right Extent of Moving Load

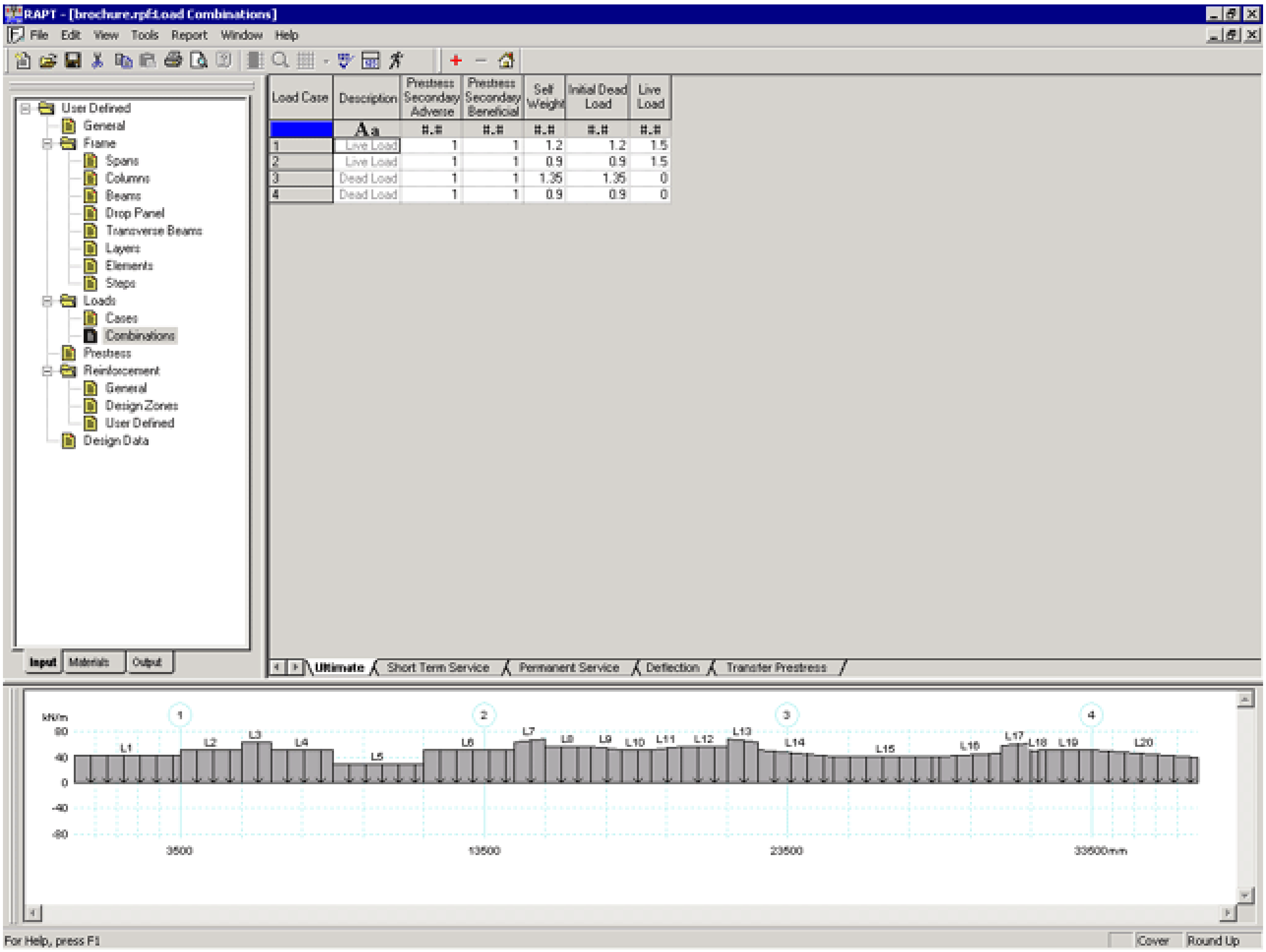
Allows the designer to define an area at the left end if the frame on which the moving load does not travel. The moving load pattern will not be placed on the frame until the complete length of the load pattern is clear of this length of the frame.

Right Extent Reference Column

Distance from Left Reference Column To Right Load Extent Limit

The location of the right end extent limit of the load pattern.

7.2.4.2 Load Combinations



The 7.2.4.1 load cases defined by the designer must be combined in various ways to be used in the design process. RAPT divides these combinations cases into 5 groups for design purposes.

1. 7.2.4.2.1 Ultimate:- Used for ultimate strength design for both flexure and shear. Multiple combination cases can be defined. RAPT will use the combination cases defined here to create two ultimate design envelopes. These are
    1. Moment controlled envelope in which the extreme moments at each design location are determined with their co-existing shears. This case is used for ultimate flexure design and ultimate shear design.
    2. Shear controlled envelope in which the extreme shears at each design location are determined with their co-existing moments. This case is used for ultimate shear design.
2. 7.2.4.2.2 Short term Service:- Used for serviceability design for flexure. Multiple combination cases can be defined. RAPT will use the combination cases defined here to create a short term load service design envelope.
3. 7.2.4.2.3 Permanent Service:- Used for serviceability design for flexure for some design codes. Multiple combination cases can be defined. RAPT will use the combination cases defined here to create a permanent load service design envelope.
4. 7.2.4.2.4 Deflection:- Used for three of the deflection cases. Individual combination cases each produce one bending moment/shear diagram. These combination cases are not enveloped.
5. 7.2.4.2.5 Transfer Prestress:- Used for prestress transfer calculations and Transfer deflection calculation for all members.
6. 7.2.4.2.6 Pre-Existing:- Used to define the load combination that determines the stress condition in the concrete when strengthening works are being carried out and reinforcement of some type is being attached to the concrete.

All of the available load cases are defined across the top of the data window. Factors can be applied to each load case for each combination case for each design option.

For all design cases, RAPT creates a set of default combination cases based on the requirements of the relevant design standard and the factors defined in 5.7 Design Standard->Load Combinations. These are created for the default building type (Office) or 7.2.1 requested building type in the input dialog. The designer is free to modify these combination factors as appropriate for the design situation. A different building type can be selected from the toolbar.

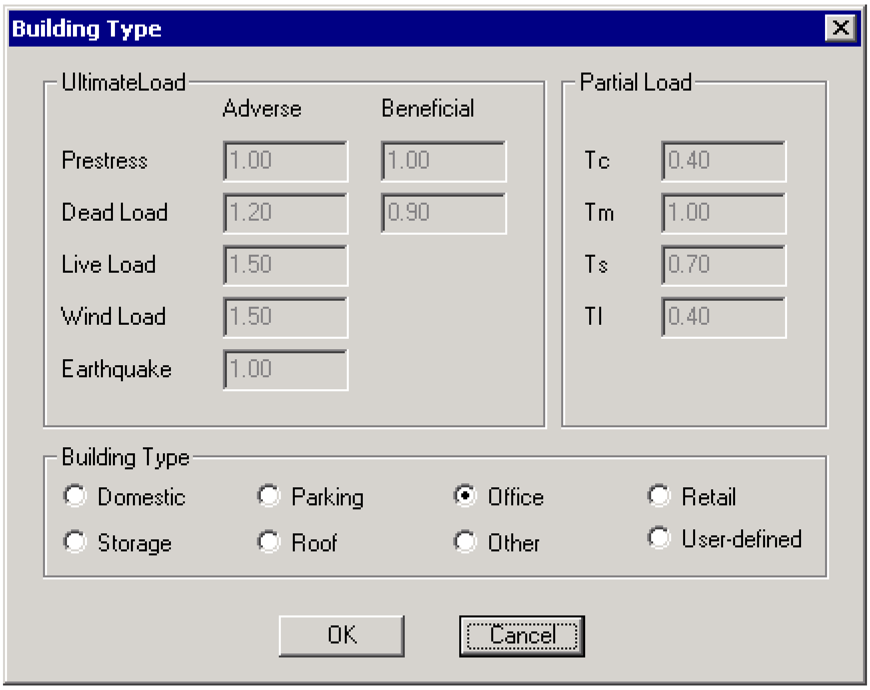
If an Alternate load case is defined in 7.2.4.1 Load Cases, it is shown in the data window with the combination values assigned to the basic load case it is associated with in each combination case. These values are not able to be edited. The same factor must apply to all loads of that type as an envelope of the load type is created. To change the factor do so in the data column for the basic load case for that type (e.g. Live Load and Alternate Live Load cases).

Load Combinations Toolbar

Add Load Combination:- Add combination case to the current design option. If a data row is selected by clicking on the left end cell in the row, the new combination case will be added at this location and the rows at and below this position will be shifted down. The combination case is added with all factors set to zero. The designer can then apply factors to any of the load cases available for this design option. This option is not available for deflection and transfer combinations.

Delete Load Combination:- Deletes the selected combination case/s from the current design option. Multiple combination cases can be deleted at once using the selection logic in 4.4.3 Cell Selection. Program calculated cases can be deleted. This option is not available for deflection and transfer combinations.

Change Building Type:- Resets the program defined load combinations to the defaults for the selected building type. Any user defined combination cases will be retained. This will reset the combinations for all five design cases, not only the one in which the toolbar button was selected. Selecting this toolbar button will indicate which Building Type was used as the basis for the current combination case settings. The default values for each building type are defined in 5.7 Design Standard->Load Combinations. For all except the User Defined Building Type, the factors are shown for information purposes only. They cannot be edited in this dialog. Once the building type has been accepted and the combination cases created with the new factors, the designer can then modify factors and add and remove combination cases for all building types.



The building types available are:-

1. Domestic
2. Parking
3. Office
4. Retail
5. Storage
6. Roof
7. Other
8. User-Defined - Allows the designer to edit the values for each factor and create their own building type.

The load factors that can be defined are

Ultimate Load Factors

Some load types require two factors. These factors are

1. Adverse:- The load effect in question is of the same sign as the sum of remainder of the load effects at this point and will lead to an increase in the overall effect.
2. Beneficial:- The load effect in question is of the opposite sign to the sum of remainder of the load effects at this point and will lead to an reduction in the overall effect.

The load types defined are

1. Prestress:- Two factors are required, one for Adverse effects and one for Beneficial. Some design codes do not differentiate for this and a value of 1.0 is used for both factors.
2. Dead Load:- Two factors are required, one for Adverse effects and one for Beneficial. Instead of applying the two factors for Dead Load as adverse/beneficial effects in a single calculation, RAPT will create separate combination cases for the two factors and will create an envelope of the effects. Dead load factors apply to all self weight and superimposed dead load cases.
3. Live Load:- Factor applied to live load cases in the Live Load Combination Cases.
4. Wind Load:- Factor applied to applied wind load to determine the ultimate wind loading. Some design standards actually specify separate loadings for service design and ultimate design. RAPT assumes a service wind loading is being applied and that the ultimate wind loading is determined as a factor times this. Sometimes the factor may vary depending on the building and the wind design parameters. It is up to the designer to make these changes for each particular design.
5. Earthquake Load:- The factor applied to the earthquake loading for ultimate strength. Some design standards define a service loading with a multiplying factor while others define an ultimate earthquake loading. The designer should ensure that the appropriate factor has been set for the loading type defined.

Partial Load factors

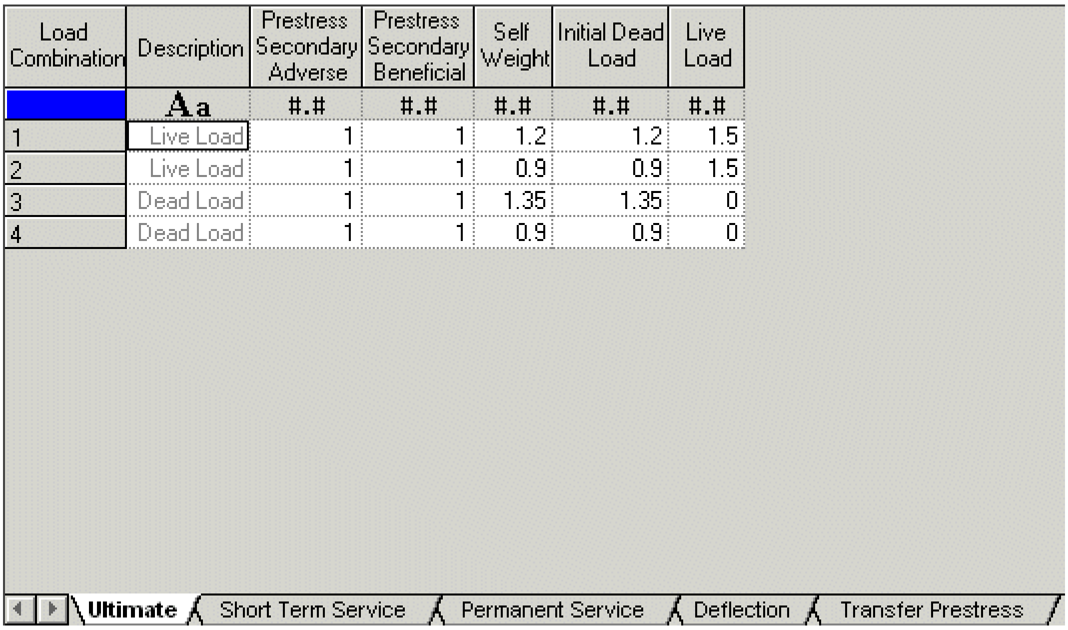
These factors are used on the live load for various conditions explained below.

1. *T*c:- Combination factor. Used for live load when it is combined with other variable actions such as wind load for ultimate strength combinations.
2. *T*m:- Combined Load Multiplying factor used in some codes.
3. *T*s:- Short Term Service Load factor.
4. *T*l:- Long Term (Permanent) Service Load factor.

The following shows how these factors are used in relation to the various codes.

1. AS3600
   *T*c - Combination live load factor used in assessing the design load for strength
   *T*m - Multiplying factor = 1.
   *T*s - Short term live load factor used in assessing the design load for serviceability
   *T*l - Long term live load factor used in assessing the design load for serviceability
2. Eurocode2
   *T*c - (To in Eurocode2) Combination coefficient for representative value of a variable action. [See Eurocode1 8.4 & Table 3, Notation)
   *T*m - Multiplying factor = 1.
   *T*s - (T1 in Eurocode2) Combination coefficient for frequent value of a variable action.
   *T*l - (T2 in Eurocode2) Combination coefficient for quasi-permanent value of a variable action.
3. ACI318
   *T*c - Combination factor = 1.
   *T*m - Multiplying factor used in assessing the design load for strength. [See ACI318 Clause 9.2.2 & 9.2.3 - Tm = 0.75].
   *T*s - Short term live load factor used in assessing the design load for serviceability.
   *T*l - Long term live load factor used in assessing the design load for serviceability.
4. BS8110, CP65, CP2004, IS456/IS1343 & SABS0100
   *T*c - Combination factor = 1.
   *T*m - Multiplying factor used in assessing the design load for strength. [See BS8110 Table 2.1 Case 3 - *m = 1.2].
   *T*s - Short term live load factor used in assessing the design load for serviceability.
   *T*l - Long term live load factor used in assessing the design load for serviceability.

7.2.4.2.1 Load Combinations - Ultimate



Load Combination	Description	Prestress Secondary Adverse	Prestress Secondary Beneficial	Self Weight	Initial Dead Load	Live Load
	Aa	#.#	#.#	#.#	#.#	#.#
1	Live Load	1	1	1.2	1.2	1.5
2	Live Load	1	1	0.9	0.9	1.5
3	Dead Load	1	1	1.35	1.35	0
4	Dead Load	1	1	0.9	0.9	0

7.2.4.2 TOOLBAR

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the 7.2.4.1 Load Cases data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the 7.2.4.1 Load Cases window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

Each combination case can have a different multiplying factor applied to each load case. The exception to this is Alternate Cases The values for Alternate Cases are read only and are not able to be edited. The same factor must apply to all loads of that type as an envelope of the load type is created. To change the factor, do so in the data column for the basic load case for that type (e.g. for Live Load and Alternate Live Load cases use the Live Load factor in each combination case). This will change the load factor for all related load cases.

Description
A text field allowing the designer to name each combination case. Program defined combination cases are named automatically and the name cannot be modified. The names used for the default combination cases are

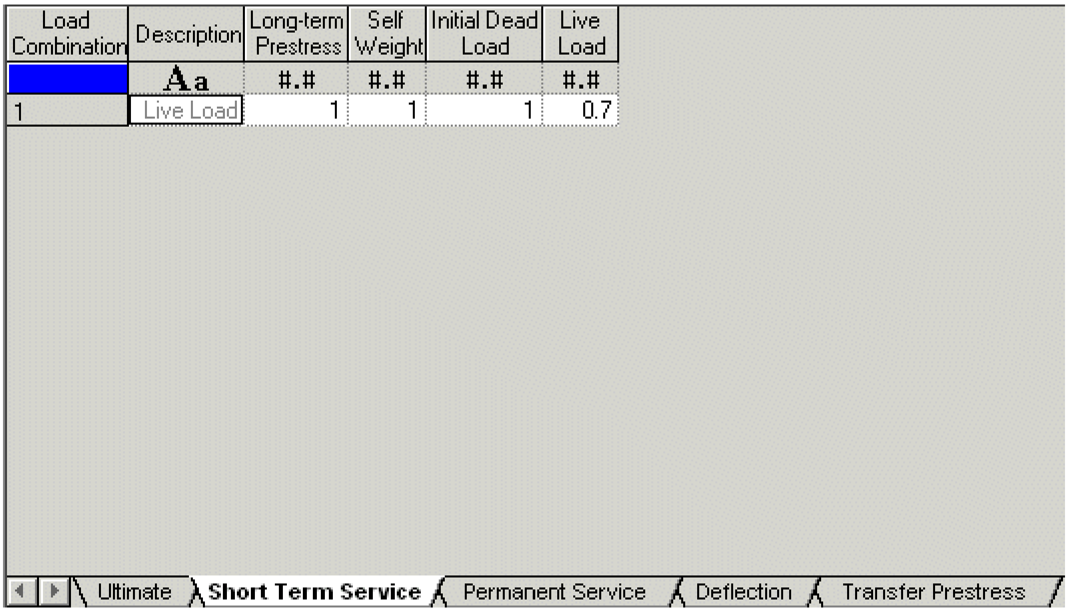
1. Live Load:- Cases from the relevant design standard which combine dead loads and live loads.
2. Dead Load:- Cases from the relevant design standard which combine dead loads only. For design standards that do not define such a combination, RAPT uses the average of the DL and LL factors and applies this factor to the dead loads only.
3. Wind Load:- Cases from the relevant design standard which combine dead loads, live loads and wind loads.
4. Earthquake:- Cases from the relevant design standard which combine dead loads, live loads and earthquake loads.

Prestress Secondary Adverse
Prestress Secondary Beneficial
For ultimate strength, only the long term Secondary Prestress load case can be selected. Some Design Standards place different factors on this depending on whether or not the secondary prestress effect at a point increases the applied moment (Adverse) or reduces the applied moment (Beneficial). RAPT will apply whichever of these factors is appropriate at each design location.

The other load cases are self-explanatory.

In design, two envelopes of the effects of all of the combination cases will be calculated, one based on an envelope of moments and the second based on an envelope of shears. These will be available for viewing in the output.

7.2.4.2.2 Load Combinations - Short-Term Service



7.2.4.2 TOOLBAR

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the 7.2.4.1 Load Cases data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the 7.2.4.1 Load Cases window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

Each combination case can have a different multiplying factor applied to each load case. The exception to this is Alternate Cases The values for Alternate Cases are read only and are not able to be edited. The same factor must apply to all loads of that type as an envelope of the load type is created. To change the factor, do so in the data column for the basic load case for that type (e.g. for Live Load and Alternate Live Load cases use the Live Load factor in each combination case). This will change the load factor for all related load cases.

Description

A text field allowing the designer to name each combination case. Program defined combination cases are named automatically and the name cannot be modified. The names used for the default combination cases are

1. Live Load:- Cases from the relevant design standard which combine dead loads and live loads.
2. Wind Load:- Cases from the relevant design standard which combine dead loads, live loads and wind loads.
3. Earthquake:- Cases from the relevant design standard which combine dead loads, live loads and earthquake loads.

Long Term Prestress

For serviceability calculations, only the long term full Prestress load case (P*e + Msec) can be selected.

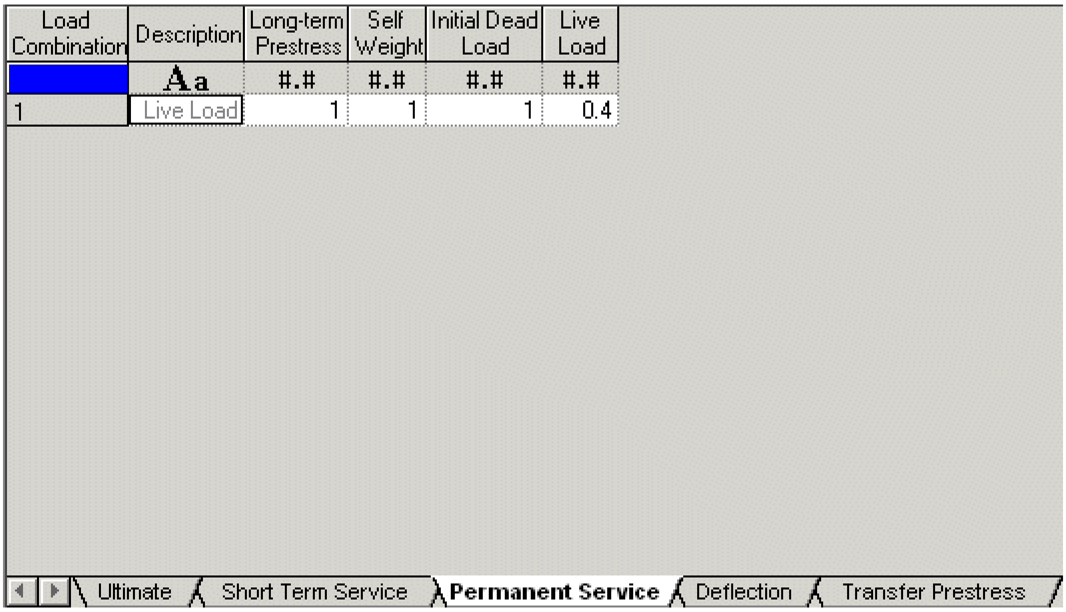
Live Load

The live load factor used is the short term factor defined for the relevant Design Standard and building type.

The other load cases are self-explanatory.

In design, an envelope of the effects of all of the Short Term Service combination cases will be calculated. This will be available for viewing in the output.

7.2.4.2.3 Load Combinations - Permanent Service



[7.2.4.2 TOOLBAR](#)

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the [7.2.4.1 Load Cases](#) data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the [7.2.4.1 Load Cases](#) window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

Each combination case can have a different multiplying factor applied to each load case. The exception to this is Alternate Cases The values for Alternate Cases are read only and are not able to be edited. The same factor must apply to all loads of that type as an envelope of the load type is created. To change the factor, do so in the data column for the basic load case for that type (e.g. for Live Load and Alternate Live Load cases use the Live Load factor in each combination case). This will change the load factor for all related load cases.

Description
A text field allowing the designer to name each combination case. Program defined combination cases are named automatically and the name cannot be modified. The names used for the default combination cases are

1. Live Load:- Cases from the relevant design standard which combine dead loads and live loads.
2. Wind Load:- Cases from the relevant design standard which combine dead loads, live loads and wind loads.
3. Earthquake:- Cases from the relevant design standard which combine dead loads, live loads and earthquake loads.

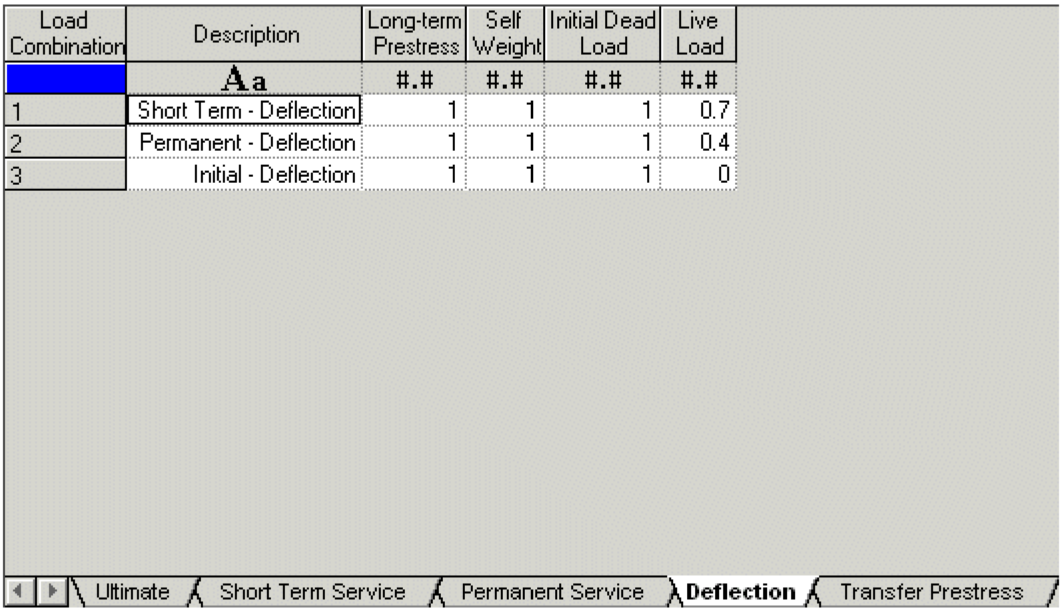
Long Term Prestress
For serviceability calculations, only the long term full Prestress load case (P*e + Msec) can be selected.

Live Load
The live load factor used is the permanent factor defined for the relevant Design Standard and building type.

The other load cases are self-explanatory.

In design, an envelope of the effects of all of the Permanent Service combination cases will be calculated. This will be available for viewing in the output.

7.2.4.2.4 Load Combinations - Deflection



Load Combination	Description	Long-term Prestress	Self Weight	Initial Dead Load	Live Load
	Aa	#.#	#.#	#.#	#.#
1	Short Term - Deflection	1	1	1	0.7
2	Permanent - Deflection	1	1	1	0.4
3	Initial - Deflection	1	1	1	0

7.2.4.2 TOOLBAR

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the 7.2.4.1 Load Cases data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the 7.2.4.1 Load Cases window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

Each combination case can have a different multiplying factor applied to each load case. Envelopes of effects cannot be included in the deflection combination cases.

NOTE:-
1 In design, the three deflection combination cases will be treated individually and will produce three separate bending moment diagrams. These will be available for viewing in the output. These will be used to calculate curvatures taking into account, where appropriate, the effects of cracking, tension stiffening, creep and shrinkage for the later calculation of deflections.

The three deflection load cases are used to calculate four different sets of curvatures. The curvature sets calculated from deflection combination cases are used in combination as described in the Theory T.7.7 Deflections. They are inter-related and the loadings defined in them should be consistent. They do NOT produce three mutually exclusive deflection results.

2 An extra deflection case will be calculated from the Transfer Load Combination. This will provide the deflection at transfer for a prestressed member and the initial self weight deflection at the time of loading for a reinforced concrete member.

Description
A text field allowing the designer to name each combination case. The three Deflection combination cases are named automatically and the name cannot be modified. No extra combination cases can be defined. The names used for the combination cases are

1. Short Term - Load Combination for Deflection:- Load Cases from the relevant design standard which combine dead loads and instantaneous live loads. This load combination is used to determine the instantaneous deflection in the short term and long term deflection calculations.
   Live Load
   By default, the load case selected for live load will be the one designated as the Deflection Live Load Case in 7.2.4.1 Load Cases. The live load factor used is the short term factor defined for the relevant Design Standard and building type.

   NOTE:- The same case should be selected for the Permanent - Deflection combination case.
2. Permanent - Load Combination for Deflection:- Load Cases from the relevant design standard which combine dead loads and permanent live loads. This combination is used to determine the effects of time dependent material properties such as creep and shrinkage on the long term deflections.
   Live Load

By default, the load case selected for live load will be the one designated as the Deflection Live Load Case in 7.2.4.1 Load Cases. The live load factor used is the permanent factor defined for the relevant Design Standard and building type.

NOTE:- The same case should be selected for the Short term - Deflection combination case.

3. Initial Deflection:- Load Cases which will be applied before the addition of finishes which will be adversely affected by deflection. This load combination is used to determine the initial deflected shape from which the incremental deflection is measured.
   Self Weight
   Should be included in the Initial Deflection Case.
   Initial Dead Load
   Will, by default, be included in the Initial Deflection Case. This represents dead load present before finishes which will be adversely affected by deflection are applied plus those finishes. If the designer has used this case for 7.2.4.1 a purpose other than is intended by RAPT (7.2.4.1), then it is up to the designer to define the use of this load case in this combination case by selecting which combinations it is to be included in and the factor to use.
   Extra Dead Load
   Will, by default, NOT be included in the Initial Deflection Case. This represents dead load not present before finishes which will be adversely affected by deflection are applied. If the designer has used this case for 7.2.4.1 a purpose other than is intended by RAPT (7.2.4.1), then it is up to the designer to define the use of this load case in this combination case by selecting which combinations it is to be included in and the factor to use.
   Live Load
   Should never be included in the Initial Deflection Case.

   NOTE:-
   1 If the incremental deflection is meant to be based on limiting live load deflection only, this combination should be modified to include all loading other than the live load. The resulting incremental deflection will then be the live load deflection. This is necessary for checks on vibration due to vehicle/pedestrian traffic.

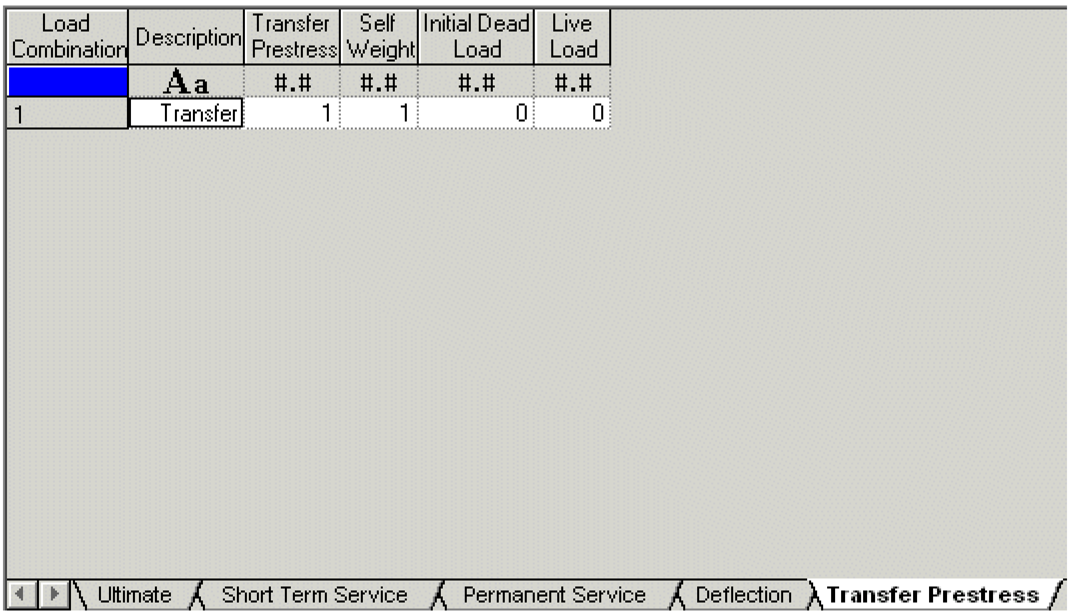
   2 If the member is propped when a brittle finish is added to the member then the initial deflection is zero and the incremental deflection is the total long term deflection.

Long Term Prestress - For service deflection calculations, only the long term full Prestress load case (P*e + Msec) can be selected. It should be selected in all three combination cases.

Alternate Load Cases and load cases defined as Moment Envelopes cannot be included in the deflection combination cases and are not available on the screen for inclusion.

The other load cases are self-explanatory.

7.2.4.2.5 Load Combinations - Transfer Prestress



7.2.4.2 TOOLBAR

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the 7.2.4.1 Load Cases data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the 7.2.4.1 Load Cases window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

Only one combination case can be defined for transfer. Extra combination cases cannot be added.

Description

A text field allowing the designer to name this combination case. The Transfer combination case is named automatically and the name cannot be modified.

This combination case should only include Load Cases which will be applied before the transfer of prestress. It should represent the minimum loading that could possibly be applied to the member at the time of stressing. For prestressed members, this combination is used for both the serviceability design checks at the final transfer of prestress and also the calculation of the deflections at this time. For reinforced concrete members, this combination case is used in the calculation of the deflection at the time of application of first load. This is normally the stripping of the falsework. The load cases will be treated in the following way by default:-

Self Weight

Should be included in the Transfer Case.

Initial Dead Load

Will, by default, NOT be included in the Transfer Case. If the designer is investigating the transfer effects in a member other than a standard slab or beam it is up to him to decide on the use of this load case in this combination case e.g. if the final stressing is going to be delayed until after some loads have been applied or if the if the designer is investigating various transfer stages for a transfer slab or beam part or all of the initial dead load may need to be applied for this combination case.

Extra Dead Load

Will, by default, NOT be included in the Transfer Case. If the designer is investigating the transfer effects in a member other than a standard slab or beam, it is up to the designer to decide on the use of this load case in this combination case and define which combinations it should be included in and the combination factor to use e.g. if the final stressing is going to be delayed until after some loads have been applied or if the if the designer is investigating various transfer stages for a transfer slab or beam part or all of the initial dead load may need to be applied for this combination case.

Live Load

Will, by default, NOT be included in the Transfer Case and should never be.

Long Term Prestress

For transfer calculations, only the transfer full Prestress load case (P*e + Msec) can be selected.

Alternate Load Cases and load cases defined as Moment Envelopes cannot be included in the transfer case and are not available on the screen for inclusion.

The other load cases are self-explanatory.

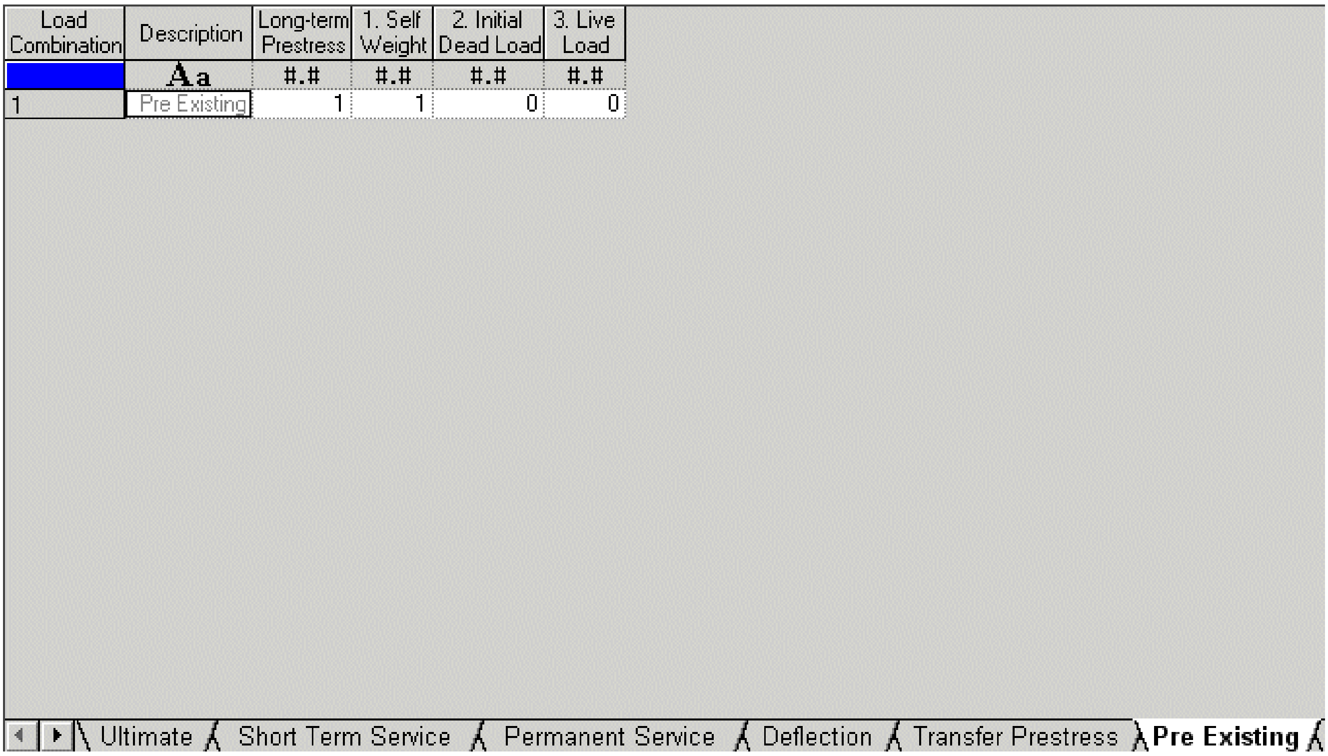
Transfer members

For transfer members there may be a need to investigate several stages of loading and stressing. To do this in RAPT, separate runs should be created (as a copy of the main run) but only including the prestress that has already been stressed or is to be stressed at this stage and the loads which will be applied at this stage.

RAPT provides three dead load cases which can help with defining the dead loading. The Self weight case should include the self-weight of the member and any attached slab areas that are supported by the member. The Initial Dead Load case includes any dead load that will be applied before attachments are made which could be damaged by deflection. Extra Dead Load is any dead loading added after this time. This would include any floors or supporting elements being added above and loading the transfer member. In the intermediate runs to check transfer stages, this case could be defined as the full load and reduced factors used in the combination cases or the load actually applied at each stage with the normal factors. As the building shape and the loading pattern normally over the height of the building, the second option is normally the best as the loading at each stage is not a true factor of the total load.

At each stage of loading, the transfer, service and ultimate capacities should be checked and the worst reinforcement result at each design location from all of the stages should be used in the design.

7.2.4.2.6 Load Combinations - Pre-Existing



Load Combination	Description	Long-term Prestress	1. Self Weight	2. Initial Dead Load	3. Live Load
	Aa	#.#	#.#	#.#	#.#
1	Pre Existing	1	1	0	0

Data Definition

Down the left side the combination cases are listed with a number and a description. Across the top of the data view, the available load cases are listed. A column of data will be made available for each load case defined in the 7.2.4.1 Load Cases data screens. If the member is prestressed, extra columns will be available for factors for prestressing effects.

As load cases are added in the 7.2.4.1 Load Cases window, they will be added to new columns in this window. If the load case is of a known type it will be given default values for the current building type from the Building Type Data. If the load case is not of a known type, it will be assigned a column of data but all factors will be zero. Factors must be added by the designer to the combination cases as required for this load case.

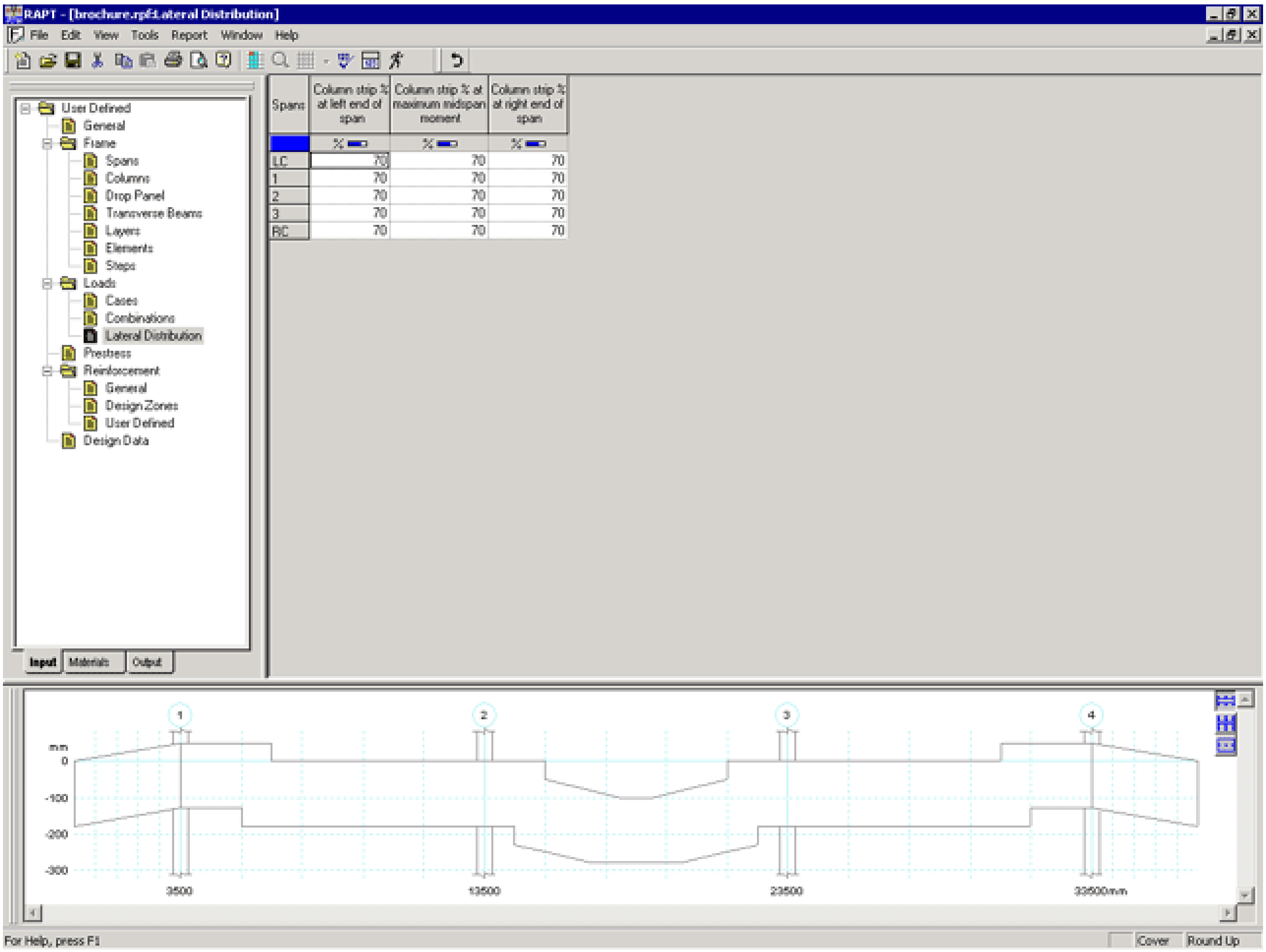
Only one combination case can defined for the pre-existing load condition. Extra combination cases cannot be added.

Description
A text field allowing the designer to name this combination case. The Pre-Existing combination case is named automatically and the name cannot be modified.

This combination case should only include Load Cases which will be applied before the attachment of strengthening materials. The load cases will be treated in the following way by default:-
Self Weight
Should be included in the Pre-Existing Case.
Initial Dead Load
Will, by default, NOT be included in the Pre-Existing Case.
Extra Dead Load
Will, by default, NOT be included in the Pre-Existing Case.
Live Load
Will, by default, NOT be included in the Pre-Existing Case.
Long Term Prestress
Will, by default, be included in the Pre-Existing Case.

The other load cases are self-explanatory.

7.2.4.3 Lateral Distribution Factors



For two way member designs, this screen allows the designer to define the transverse distribution of the bending moment and shears between the column and middle strips.

Lateral Distribution Factors Toolbar


Reset Factors

Recalculates the default factors based on the frame properties.

Data Definition

Column strip % at left end of span

The percentage of the moment at the left end of the span that will be apportioned to the column strip.

Column strip % at maximum mid-span moment

The percentage of the moment at the maximum span moment point that will be apportioned to the column strip.

Column strip % at right end of span

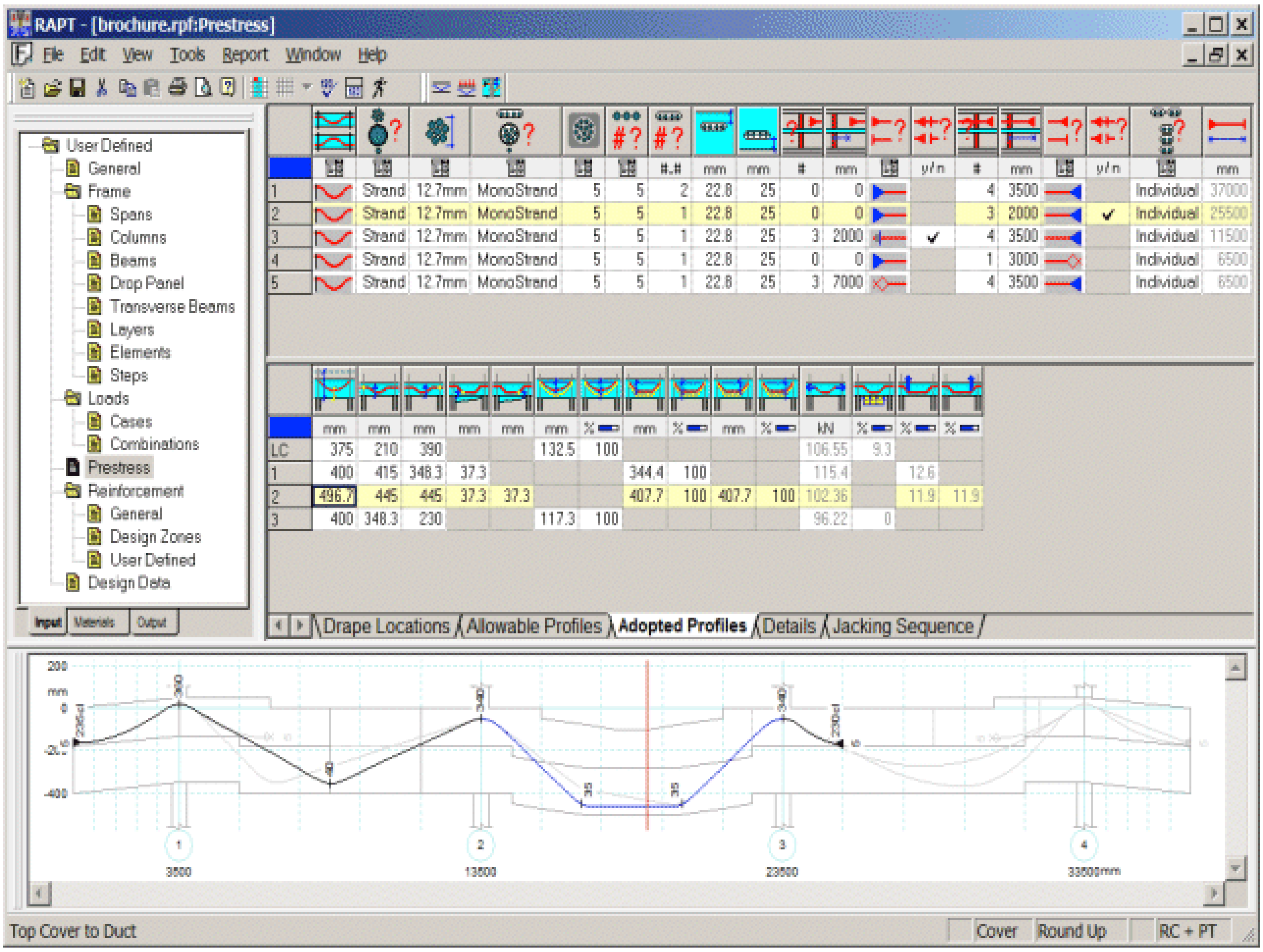
The percentage of the moment at the right end of the span that will be apportioned to the column strip.

% Column Strip moment to parallel beam

The percentage of the column strip moment that will be apportioned to the beam in a two way beam design case. For two-way reinforced concrete beams, all design output referring to the column strip is in fact referring to the beam of effective flange width entered in the input. Common design practice is to place the total column strip moments in the beam. The reinforcement detailed in the residual part of the column strip is then made identical to that in the middle strip. RAPT strictly follows the method from AS1480 which takes a percentage of the column strip moments, up to 85%, to the beam leaving the remainder in the column strip. The amount of reinforcement necessary in the residual part of the column strip is often less than that required in the middle strip hence it is recommended that the middle strip steel be carried into the column strip. No steel is detailed by RAPT in the residual part of the column strip for this type of structure. This is left to the designer.

These factors are applied at the peak moment points. Between these points RAPT applies a parabolic variation in the percentage between the peak points. If 100% is set as the percentage to the column strip at a support point, RAPT will ensure that all of the 100% is applied to all design points within that support zone in that span. This often occurs at end columns where it is common to apply the full moment to the column strip only as the torsion strip at the edge cannot generate moment a long way from the column.

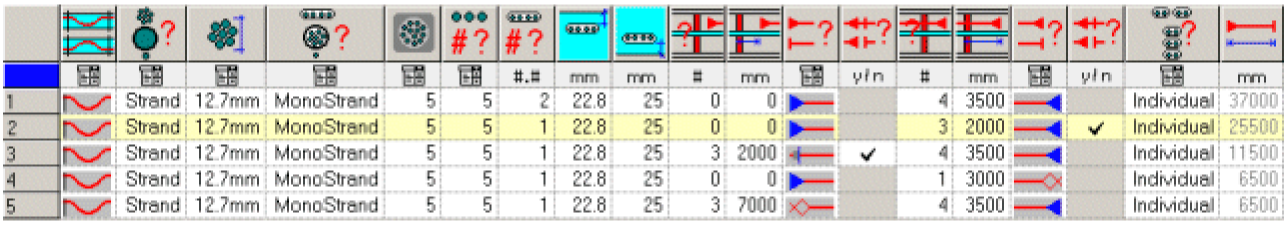
7.2.5 Prestress Data



This input screen is used to define the tendon locations and shapes. The screen is divided into 3 separate windows

1. Prestress Control Grid:- This grid lists the general data about each tendon and controls which tendon is viewed in the Profile Data and Graphics windows (see 7.2.3 Screen Layout for general discussion). The functionality of this window is discussed in this section.
2. Prestress Profile Data:- Defines the profile shape data for the tendon and strip selected in the Control Grid. The functionality of the views in this window are discussed in the later sections.
3. Prestress Graphics:- Shows an elevation of the selected tendon profile shape.

The prestress control grid lists the tendon profiles defined in the member. Each tendon profile uses one row of the control grid to define the tendon in each design strip. For a one-way member, one row will be added for each tendon profile as shown above, while two rows will be added for a two-way member as shown below. A tendon profile must use the same overall profile type, steel type, strand size and prestress type in both strips.



							#.#	mm	mm	#	mm		y/n	#	mm		y/n		mm
1		Strand	12.7mm	MonoStrand	5	5	2	22.8	25	0	0			4	3500			Individual	37000
2		Strand	12.7mm	MonoStrand	5	5	1	22.8	25	0	0			3	2000		✓	Individual	25500
3		Strand	12.7mm	MonoStrand	5	5	1	22.8	25	3	2000		✓	4	3500			Individual	11500
4		Strand	12.7mm	MonoStrand	5	5	1	22.8	25	0	0			1	3000			Individual	6500
5		Strand	12.7mm	MonoStrand	5	5	1	22.8	25	3	7000			4	3500			Individual	6500

Graphic Interaction

RAPT provides a logical connection between the data and the graphics for the designer.

Selecting Tendons

To select a prestress tendon profile as the current tendon profile from the graphics, click on a point along its length (within 4 pixels above or below the tendon) with the left mouse button. The selected tendon will be shown in black. The Control Grid data row controlling the selected tendon will become the current row and the data for the selected tendon will be shown in the data grids. If more than one tendon is found at the selected location, RAPT will show the first selected tendon after the currently selected tendon. Continuing to click at the same location (with a suitable pause to avoid double clicking) will continually select the next tendon in the selected list and assign that tendon as the current tendon. For two-way slabs where column strip and middle strip tendons exist, RAPT will select either column strip or middle strip tendons only at the clicked point depending on which strip type the current tendon belongs to.

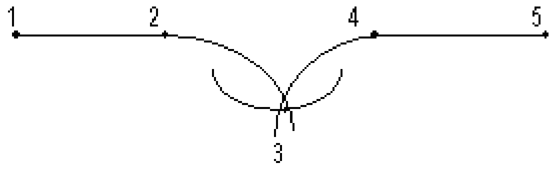
Selecting Data Cells

Clicking in certain areas of the graphics will move the data focus to the data cells controlling that data. Clicking the left mouse button in the graphics view will have the following effects

1. If the clicked location is near (within 4 pixels) of a Defined Profile Point,
    1. place a cursor at the Defined Profile Point.
    2. move the current span to the clicked span in the graphics and data windows.
    3. move the current cell in the Prestress Profile Data Grids to the cell which defines this Defined Profile Point.
    4. change the location in the Location Control in the toolbar to the location of the Defined Profile Point.
2. If the clicked location is not near (not within 4 pixels) of a Defined Profile Point,
    1. place a cursor at the clicked location.
    2. move the current span to the clicked span in the graphics and data windows.
    3. change the location in the Location Control in the toolbar to the current location.

Profile Generation Errors

RAPT will attempt to calculate tendon profiles to suit the profile shapes defined in the input screens. In some cases it will not be possible to calculate a profile to fit these requirements. The transition radius may be too large to allow the profile to be generated (see below).



Reverse Curves that do not fit profile.

Because the profile shape cannot be calculated properly RAPT will

1. attempt to calculate the profile using a zero reverse curve radius in the spans where the problem occurs. If this works you will notice that the drawing will show no transition curves in the offending spans. This is only a temporary calculation by RAPT to give you a picture to look at to help you to visualise where the problems lie.

2. If the attempt in 1 above fails, RAPT will not be able to generate the profile at all and the tendon will show as a straight line at the zero profile height. Temporarily modifying the reverse curve radii in the 7.2.5.1 Drape Locations screen in one or more spans to zero may allow RAPT to calculate a temporary profile to help you to visualise where the problems lie. These modified radii should be changed back to acceptable values before you proceed with the design.

In both of these cases RAPT will generate errors in the error window. Also the user can ask RAPT to check for errors using the  toolbar button. To toggle between the error window and the graphics window use the  toolbar button. When it is depressed the error window will show, when expressed, the graphics window will show. If the designer double clicks on an error line in the error window, RAPT will move the data focus in the control grid to that tendon and all of the Prestress Profile Data Grids will show the data for the offending tendon profile. The current data cell in the Adopted Offsets grid will move to the data item that most closely controls the attribute of the profile causing the problem. The current data cell in the Adopted Offsets grid will tell you which span is causing the problem. There are normally several solutions to this type of problem. The possible solutions may be

1. The reverse curve radius may be able to be reduced (do not reduce below the minimum allowed for this tendon type without taking the proper design and detailing precautions)

2. The low point profile height may be too low.

3. The length of the profile section of the tendon in this span may be too small to allow tendon to achieve the full available drape. Modifying the drape locations may help.

4. If the tendon starts in this span and the length of the tendon in this span is very small, it may be more logical to change the 7.2.5.1 Tendon Profile Type in this span to a passive tendon type without a controlling mid-profile point. Even with this type of profile it still may be necessary to raise the anchor height if the length of the tendon in the span is very small.

5. One or both high points may be too high. This would normally be a last resort except with a cantilever.

Remember that this must be sorted out before the member is built. If RAPT cannot calculate the profile shape then the profile shape cannot be built within the parameters of the tendon properties defined. It is not something that can be ignored.

Effects of Adding Spans

When a cantilever or end span is added and a tendon reaches that end of the frame in the old span configuration, the tendon will be

1. extended to the end of the cantilever or end span.

2. profiled to use full drape in the cantilever or end span.

3. re-profiled in the attached span to use full available drape in that span

When internal spans are added, RAPT will

1. maintain the end locations and terminating strand locations of the tendon relative to the moved columns

2. maintain the profile heights in other spans

3. profile the new spans to the allowable cover at the bottom and at new columns at the top and attach to the other previously existing spans at the previous profile heights at the connecting ends.

The designer should always check that the modifications that RAPT has made are suitable for the specific situation.

Effects of Deleting Spans

If all of the spans containing a single tendon are deleted then that tendon will be deleted. Otherwise, all possible attempts will be made to ensure a tendon profile is not deleted unless there is not logical solution to positioning the tendon in the new frame configuration.

If a tendon stops internally and the span in which it stops is deleted, the short tendon end will be moved to the next external span if logically possible, otherwise it will move to the next internal span.

If internal spans are deleted from a tendon, the new meeting ends are checked to ensure a smooth profile. No other re-profiling will be done.

When a cantilever or end span is deleted and the tendon extends into that span, RAPT will

1. move the tendon back into the new end span at that end.
2. re-profile the tendon in the new end span.

The designer should always check that the modifications that RAPT has made are suitable for the specific situation.

Effects of Modifying Concrete Shape

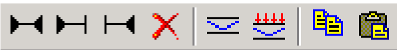
When concrete shapes are modified, RAPT will attempt to modify the tendon profile shape to match the new concrete shape. It will maintain the same cover to the top surface as was defined prior to the concrete shape change (except in cantilevers where the anchorage eccentricity will be maintained) and will profile the drape points based on the current setting selected in 4.2.2 User Preferences->User Options Dialog and shown in the Status Bar Pane (see diagram at the top of the document, set to %Drape).

The options available are

1. Percentage Drape:- use the same percentage drape in each span as was previously used to calculate the profile heights at the drape points
2. Cover:- use the same cover at each nominated profile point as was previously used to calculate the profile heights at the drape points

The designer should always check that the modifications that RAPT has made are suitable for the specific situation.

Prestress Control Toolbar



When program focus is in the Control Window, this toolbar will be available. Only the toolbar buttons available at any time will be active for use. The remainder will be shown in background colour.

The buttons available are

 Add double end stressed tendon profile.

 Add a tendon profile stressed from the left end.

 Add a tendon profile stressed from the right end.

When one of the add buttons is pressed, a new tendon will be added either at the location of the currently selected row if a row is selected or after the last tendon is no row is selected. The tendon will be created using the following

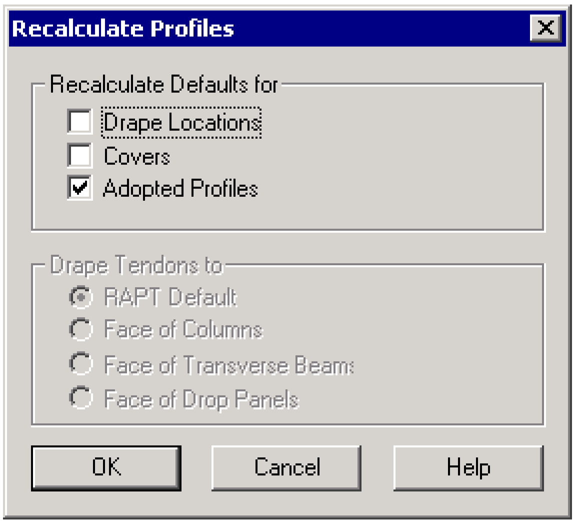
1. Strand will be set as the steel type.
2. The steel size nominated in the selected 6.8 Materials Data as the preferred size.
3. The Prestress type will be the default set for the selected 5.3 Design Standard->Prestress.
4. The anchor size will be the default set for the nominated steel type, size and prestress type as defined in the 6.8.1 Materials Data->Anchor Size.
5. The number of strands will be the maximum that can be used in the selected anchor size.
6. The number of tendons using this profile will be set to 1.
7. The covers will be the default set for the selected 5.3 Design Standard->Prestress and member type.
8. The tendon will be defined extending full length of the frame from the left end concrete edge to the right end concrete edge.
9. All tendons will be assumed to be included individually in the shear calculations.
10. Tendons will be profiled to use full drape in each span and to the concrete centroid at the anchorages rounded to the nearest 5mm or 1/8" depending on unit type. Parabola end locations will be determined based on the default setting in 5.3 Design Standard->Prestress or to the face of drop panels if they exist in the relevant design strip and if they are stiff enough.

 Delete selected tendon profiles.

To select a profile to delete, the relevant row on the grid must be selected. It can be selected by a left mouse select in the tendon number column. For tendons in two-way slabs where column and middle strips are defined, a tendon

profile uses 2 rows in this grid. Selecting either or both rows will result in the deletion of the tendon profile and both rows will be removed. Multiple tendons can be deleted at once using the selection logic in 4.4.3 Cell Selection.

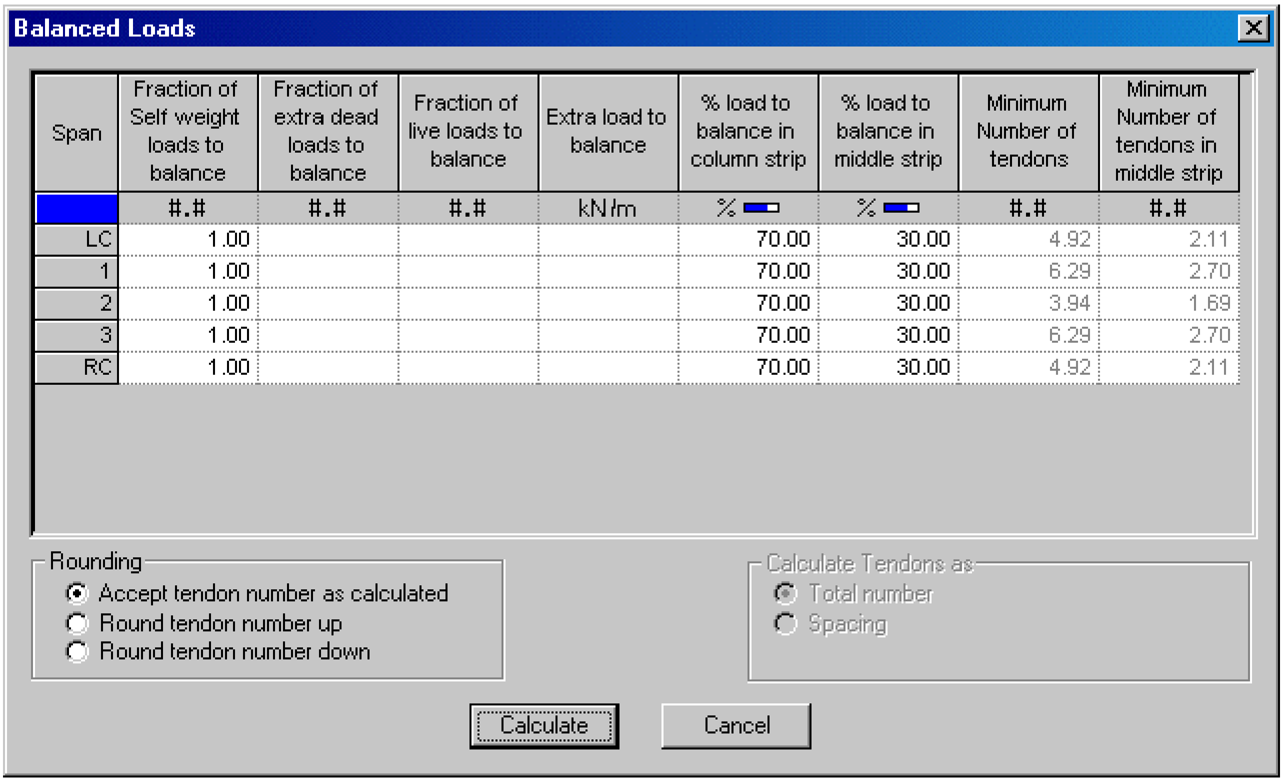
Recalculate Default Profile Data:-



Recalculates the default profile shape for the user based on the current tendon lengths and concrete shape for the selected rows in the Prestress Control Grid. The user can specify which aspect/s of profile shape is/are to be reset. Options are

1. Drape Locations:- Resets the end of parabola locations in each span. By default these are set to either the face of the drop panel if it exists and is stiff enough or to the centreline of the support. The user can modify the default centre of support location in 5.3 Design Standard->Prestress to be consider also face of column and/or face of transverse beams. In this dialog, the user can select to drape the tendons to
    1. RAPT Default:- as discussed above.
    2. Face of Columns
    3. Face of Transverse Beams
    4. Face of Drop Panels
2. Covers:- Resets the Allowable covers at each profile control point in each span to the default values based on the top and bottom covers defined in the control grid for that tendon profile.
3. Adopted Profiles:- Resets the adopted tendon profile heights for all control points for the profile types specified to the available cover at each point as defined in the Allowable Profiles.

Recalculate Tendons and Drapes from Balanced Loads



Recalculates the number of tendons and tendon profiles to suit a set of balanced loads defined by the user. In each span, the user can define the following

1. Fraction of Self-weight Load to Balance
2. Fraction of Extra Dead Load to Balance, includes both Initial Dead Load and Extra Dead Load cases
3. Fraction of Live Load to Balance
4. Extra load to balance in force/length of span.
5. Percentage Load to Balance in Column Strip (only for two-way designs)
6. Percentage Load to Balance in Middle Strip (only for two-way designs)

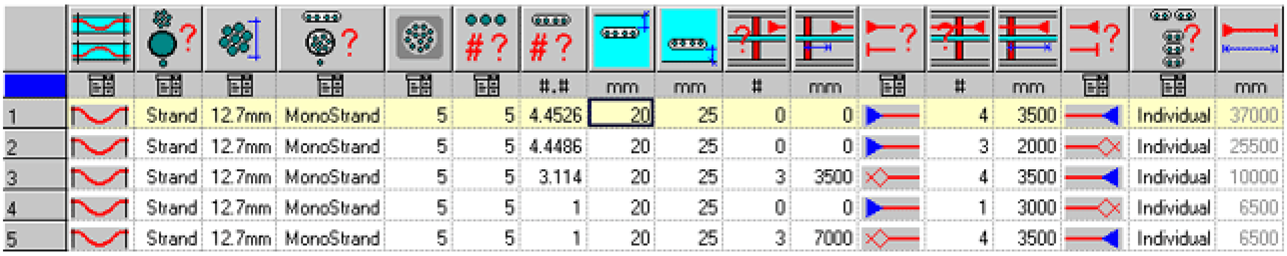
RAPT also defines the Minimum Number of Tendons in each span in each strip. This number of tendons is the minimum that would be required in each span to balance the nominated loads using full available drape in each span. It gives the designer guidance on the number of tendons that will be required in each span and therefore is an indicator of the areas where short tendons can be used to provide a more economical design. In the case above, 6.3 tendons would be required in the first internal spans and 4 tendons would be required in the middle span. It may therefore be more economical to use 4 tendons full length of the member and short tendons at each end of the member extending over the end spans and finishing approximately 20% of the span into the next span from each end.

In performing the load balance, RAPT will determine a pin-ended bending moment diagram in each span from the loads to be balanced. These loads include all loads defined in the load cases nominated. They will include the effects of partial uniform loads, trapezoidal loads, point loads and point moments and bending moment diagrams. In two way strip designs, the loads will be proportioned to the column and middle strips in each span in accordance with the factors in 5 and 6. From the bending moment diagram for each strip, RAPT will then determine the moment at critical profile points (eg the mid-length of parabolic profiles or the harp locations of harped profiles) and use these critical moments to determine the number of tendons required and the resulting tendon profile to balance these loads with the profile types selected. Thus it is possible to obtain a load balance for any combination of loads and the resulting tendon profiles and number of tendons required. RAPT will round the number of tendons to the next highest or lowest or, for one way slabs to the nearest 100mm spacing if requested in 4.2.2 User Preferences->User Options Dialog and shown in the Status Bar Pane (see diagram at the top of the document, set to Round Up).

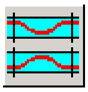
Copy Prestress Tendon- Allows the complete data for a tendon profile to be copied for later pasting to another tendon profile. The tendon profile to be copied must be selected by clicking on the tendon profile number in the left data column before the Copy Prestress Tendon icon is available. Click the Icon to copy this tendon profile. For two-way slabs with separate column and middle strip tendons, only the tendon in the selected strip will be copied.

Paste Prestress Tendon:- Once a tendon profile has been copied, it can be pasted to another tendon profile. The "paste to" tendon profile must be selected by clicking on the tendon profile number in the left data column before the Paste Prestress Tendon icon is available. Click the icon to paste all of the tendon profile from the copied tendon profile into this tendon profile. All data for the "paste to" tendon profile will be set exactly the same as the "copy from" tendon profile.

Data Definition

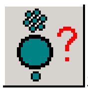


							#.#	mm	mm	#	mm		#	mm			mm
1		Strand	12.7mm	MonoStrand	5	5	4.4526	20	25	0	0		4	3500		Individual	37000
2		Strand	12.7mm	MonoStrand	5	5	4.4486	20	25	0	0		3	2000		Individual	25500
3		Strand	12.7mm	MonoStrand	5	5	3.114	20	25	3	3500		4	3500		Individual	10000
4		Strand	12.7mm	MonoStrand	5	5	1	20	25	0	0		1	3000		Individual	6500
5		Strand	12.7mm	MonoStrand	5	5	1	20	25	3	7000		4	3500		Individual	6500

Profile Orientation

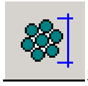
Allows the designer to define the orientation and reference surfaces for the tendon profile. For two-way slabs, a single cell controls both design strips. A Drop Down List with four options is available

1.  RAPT will profile the tendon as a parabolic profile with or without straight portions at the ends to high points at the supports based on the top cover and low points at mid drape based on the bottom cover.
2.  RAPT will profile the tendon as a parabolic profile with or without straight portions at the ends to low points at the supports based on the bottom cover and high points at mid drape based on the top cover.
3.  RAPT will profile the tendon as a straight profile between supports based on the top cover.
4.  RAPT will profile the tendon as a straight profile between supports based on the bottom cover.

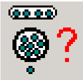
Steel Type

Three prestress steel type options are provided in a Drop Down List. These types and their properties are defined in the 6.1 Materials Data. For two-way slabs, a single cell controls both design strips. The default setting is always strand, The options are.

1. Strand
2. Bar
3. Wire

Steel Diameter

Lists the strand sizes for this Steel Type as defined in the 6.8 Material Properties. Three columns of data are provided which define the steel diameter, area and breaking strength. For two-way slabs, a single cell controls both design strips.

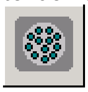
Prestress Type

Three prestressing types options are available for bonded prestressing and 2 for unbonded prestressing. These are

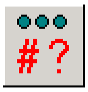
1. Monostrand:- Flat duct post-tensioning tendons used in both slabs and beams. Strands are normally stressed individually rather than as a tendon.
2. Multistrand:- Round duct post-tensioning tendons used mainly in larger beams. Strands are normally stressed as a group.
3. Pretensioned (not available for unbonded prestressing). Strands stressed between buttresses and released after the concrete has hardened.

These names are also used in the 6.1 Materials Data where the different tendon sizes available are defined. For two-way slabs, a single cell controls both design strips.

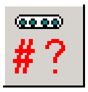
For two-way slabs, the remaining data to the right of this point is selected for each design strip. This means that the tendon size, length, stressing ends and profile shapes can be different in the 2 design strips.

Prestress Anchorage Size (not available for bar, pretensioned or unbonded monostrand tendons)

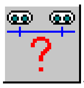
Lists the anchorage sizes available for the combination of steel type, steel diameter and prestressing type specified for this tendon.

Number of Strands (not available for bar, pretensioned or unbonded monostrand tendons)

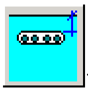
Lists the range numbers of strands that can be used in the anchorage type selected. These are defined in 6.8 Material Properties in the tendons tab.

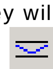
Number of Tendons

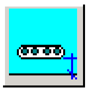
The number of tendons using this tendon profile in each strip. This number can be a decimal number.

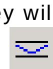
Spacing of Tendons

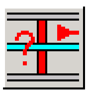
The spacing of tendons to be used in this design. This option is only available for one way slab designs.

Top Cover to Duct

The cover to the op surface of the tendon from the top surface of the concrete. When this value is modified, the Allowable Offsets will be automatically modified to suit. The Adopted Offsets at all critical profile points will then be checked to ensure they do not violate the allowable cover at each point. If they do violate the cover, they will be modified to suit the new cover. If the designer wants to re-profile the tendon to the new covers, use the  button on the toolbar as discussed above.

Bottom Cover to Duct

The cover to the bottom surface of the tendon from the bottom surface of the concrete. When this value is modified, the Allowable Offsets will be automatically modified to suit. The Adopted Offsets at all critical profile points will then be checked to ensure they do not violate the allowable cover at each point. If they do violate the cover, they will be modified to suit the new cover. If the designer wants to re-profile the tendon to the new covers, use the  button on the toolbar as discussed above.

Left Anchorage Reference Column

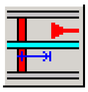
The 7.2.3.2 Column Grid Reference from which the left end of the tendon is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the tendon. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a tendon in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a tendon.

When the anchorage moves into another span, RAPT will create a default profile in the new spans and also in the last pre-existing span.

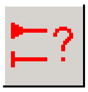
Obviously the left end of a tendon must be to the left of the right end so make sure when modifying tendon ends that the order of modification will ensure this.

Distance to Left Anchorage from Column

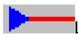
The distance from the left reference column to the left anchorage of the tendon. If the distance entered moves the anchorage into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the anchorage.

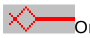
If the left anchorage location is modified but stays within the same span and is of the same type, RAPT will attempt to maintain the same profile as before the modification. If the anchorage moves into another span, RAPT will create a default profile in the new spans and also in the last pre-existing span.

Obviously the left end of a tendon must be to the left of the right end so make sure when modifying tendon ends that the order of modification will ensure this.

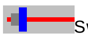
Left Anchorage Type

The anchorage type at the left end of the tendon can be defined as one of the following types

1. Live End. This indicates a stressing end for a tendon. At least one stressing end is required for all tendons, even pretensioned tendons. For pretensioned tendons with hold-down points, the stressing end is important, otherwise either end can be used as there is no friction or draw-in effect. Also for pretensioned tendons, the transmission length effects will be allowed for at live ends. This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

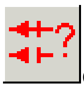
2. Onion Dead End. This indicates an onion type dead (non-stressing) end for a tendon. Only for bonded strand tendons. This type of dead end anchor does not transfer the full force to the end of the tendon and this will be reflected in all of the calculations. This type of anchorage cannot be used with couplers.

3. Swaged Dead end with nominal end plate. This indicates a dead (non-stressing) end for a tendon. Only for bonded strand tendons. This type of dead end anchor does not transfer the full force to the end of the tendon and this will be reflected in all of the calculations. The length of tendon from the end of the duct to the plate must be bonded to provide some anchorage as the end plate cannot develop the full force. This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

4. Swaged Dead end with full force transfer end plate. This indicates a dead (non-stressing) end for a tendon. In this case the end plate can carry the full force of the tendon and for bonded tendons the strand between the end of the duct and the end plate should be greased and unbonded to allow the full prestress force to reach the plate. This type of dead end must be used for unbonded tendons and pretensioned tendons. . This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

For pretensioned tendons, the transmission length effects will be allowed for at dead ends and live ends. No actual anchorage is assume to be provided other than bond of the strand to the concrete. This will be calculated in accordance with the design rules in each design standard allowing for an ineffective length, a transmission length and a development length.

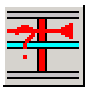
Note the different effects of the three dead end anchorage types have now been fully programmed into RAPT. It is important that the designer select the appropriate dead end type required for the purpose for which it is intended. Also, the live end and dead end ineffective lengths are now included in the design. It is important that the designer show the correct position of the slab edge relative to the supports so that RAPT can determine the effects of the reductions in force and also the requirements for development of the tension force required at end supports for shear.

Coupler at Left Anchorage

The tendon at this anchorage can be coupled to another tendon, providing continuity of these two tendons at this location. Another tendon must be defined with an anchorage at the right end that is also as a coupler to couple with this tendon.

Coupling is not possible with "Onion" type dead end anchorages or pretensioned strands. Anchorages which are normally coupled together are Dead End with Dead End or Live End with Dead End. While it is technically possible to couple two Live End anchorages together, it is very complicated and will not be a standard option with most post-tensioning companies. RAPT will allow this option, but will give a warning to check that you supplier can achieve this combination.

For two tendons to be coupled the tendons must be of the same type and size and be set to the same profile height at the anchorages. The total number of tendons connecting at a coupler can be divided between different tendon types in RAPT input. The total number of tendons connecting on either side of a coupler type must be the same. When Data Check is requested or on Running the data, RAPT will search through the tendon types defined and give an error message if it cannot find another tendon or group of tendons meeting the criteria above to couple to this one.

Right Anchorage Reference Column

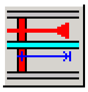
The 7.2.3.2 Column Grid Reference from which the right end of the tendon is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the tendon. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a tendon in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a tendon.

When the anchorage moves into another span, RAPT will create a default profile in the new spans and also in the last pre-existing span.

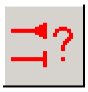
Obviously the right end of a tendon must be to the right of the left end so make sure when modifying tendon ends that the order of modification will ensure this.

Distance to Right Anchorage from Column

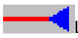
The distance from the right reference column to the right anchorage of the tendon. If the distance entered moves the anchorage into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the anchorage.

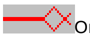
If the right anchorage location is modified but stays within the same span and is of the same type, RAPT will attempt to maintain the same profile as before the modification. If the anchorage moves into another span, RAPT will create a default profile in the new spans and also in the last pre-existing span.

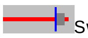
Obviously the right end of a tendon must be to the right of the left end so make sure when modifying tendon ends that the order of modification will ensure this.

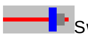
Right Anchorage Type

The anchorage type at the right end of the tendon can be defined as one of the following types (not all types are available for all tendon types)

1. Live End. This indicates a stressing end for a tendon. At least one stressing end is required for all tendons, even pretensioned tendons. For pretensioned tendons with hold-down points, the stressing end is important, otherwise either end can be used as there is no friction or draw-in effect. Also for pretensioned tendons, the transmission length effects will be allowed for at live ends. . This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

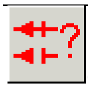
2. Onion Dead End. This indicates an onion type dead (non-stressing) end for a tendon. Only for bonded strand tendons. This type of dead end anchor does not transfer the full force to the end of the tendon and this will be reflected in all of the calculations. This type of anchorage cannot be used with couplers.

3. Swaged Dead end with nominal end plate. This indicates a dead (non-stressing) end for a tendon. Only for bonded strand tendons. This type of dead end anchor does not transfer the full force to the end of the tendon and this will be reflected in all of the calculations. The length of tendon from the end of the duct to the plate must be bonded to provide some anchorage as the end plate cannot develop the full force. This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

4. Swaged Dead end with full force transfer end plate. This indicates a dead (non-stressing) end for a tendon. In this case the end plate can carry the full force of the tendon and for bonded tendons the strand between the end of the duct and the end plate should be greased and unbonded to allow the full prestress force to reach the plate. This type of dead end must be used for unbonded tendons and pretensioned tendons. This anchorage type may be used with a coupler if it is internal (does not reach the slab edge) and another tendon is defined with a coupler to attach to it. The ineffective length will be considered to be zero in this case.

For pretensioned tendons, the transmission length effects will be allowed for at dead ends and live ends. No actual anchorage is assume to be provided other than bond of the strand to the concrete. This will be calculated in accordance with the design rules in each design standard allowing for an ineffective length, a transmission length and a development length.

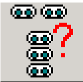
Note the different effects of the three dead end anchorage types have now been fully programmed into RAPT. It is important that the designer select the appropriate dead end type required for the purpose for which it is intended. Also, the live end and dead end ineffective lengths are now included in the design. It is important that the designer show the correct position of the slab edge relative to the supports so that RAPT can determine the effects of the reductions in force and also the requirements for development of the tension force required at end supports for shear.


Coupler at Right Anchorage

The tendon at this anchorage can be coupled to another tendon, providing continuity of these two tendons at this location. Another tendon must be defined with an anchorage at the left end that is also set as a coupler to couple with this tendon.

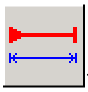
Coupling is not possible with "Onion" type dead end anchorages or pretensioned strands. Anchorages which are normally coupled together are Dead End with Dead End or Live End with Dead End. While it is technically possible to couple two Live End anchorages together, it is very complicated and will not be a standard option with most post-tensioning companies. RAPT will allow this option, but will give a warning to check that you supplier can achieve this combination.

For two tendons to be coupled the tendons must be of the same type and size and be set to the same profile height at the anchorages. The total number of tendons connecting at a coupler can be divided between different tendon types in RAPT input. The total number of tendons connecting on either side of a coupler type must be the same. When Data Check is requested or on Running the data, RAPT will search through the tendon types defined and give an error message if it cannot find another tendon or group of tendons meeting the criteria above to couple to this one.


Stacked Tendons

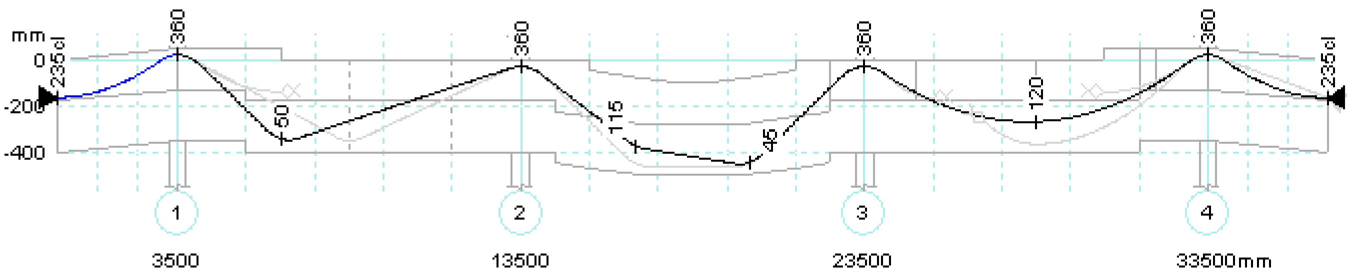
Shear design for prestressed members usually requires that the effective shear width of the member be reduced to allow for the width of the of the prestressing tendons at each section. In situations where tendons are actually placed in vertical layers (stacked), tendons in the same vertical line only need to be counted once. RAPT allows you to define the following options to indicate how the tendons in each tendon profile are allowed for in these calculations.

1. Individual:- All tendons will be included individually in the shear width calculations.
2. Group:- The tendons in this profile will be treated as 1 tendon in the shear width calculations. This option would normally be used where the designer has defined an "average" tendon and the tendons in the group will actually be stacked. In reality, "average" tendons should be avoided as the section design results will not be a true reflection of the actual stresses in these tendons.
3. None:- The tendons in this profile will be ignored in the shear width calculations. The designer must ensure that these tendons in this profile line vertically with other tendons whose width is included in the calculations.


Tendon Length

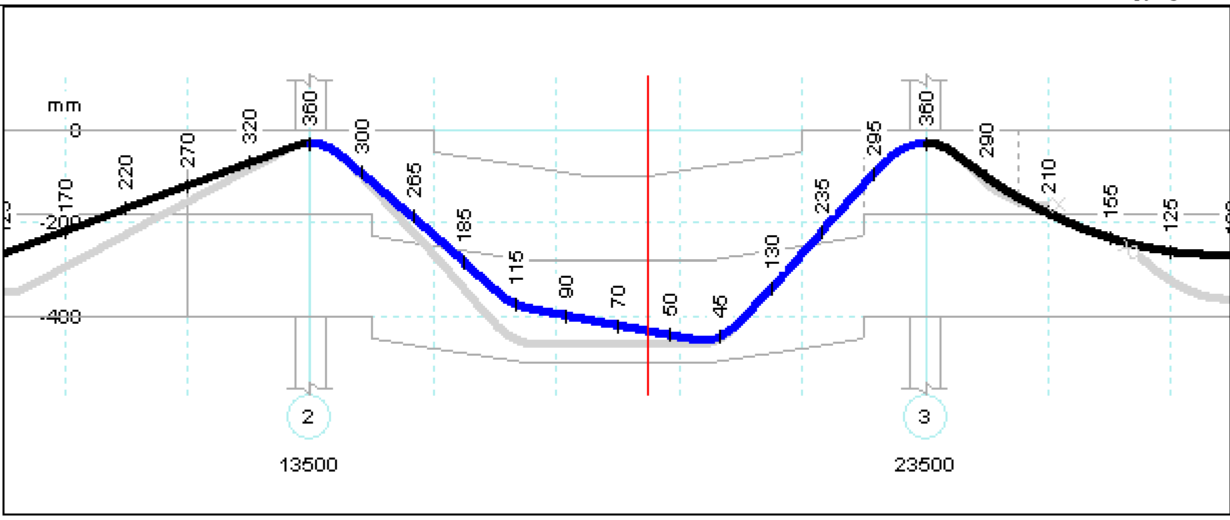
The length of the tendon between anchorages is reported for information only. It is not an editable data cell.

Graphics Window



The prestress graphics window shows an elevation view of the concrete member with prestress tendons drawn to scale in elevation with anchorage types shown as described in Left Anchorage Type and Right Anchorage Type. Profile heights are shown to the underside of the duct from the concrete soffit (ignoring drop panels for two-way slabs) except at anchorages where they are shown to the centreline of the anchorage.

In full screen mode, only the tendon profile heights at the Defined Profile Locations will be shown and the tendon is shown as a standard line width. In zoom mode, the profile heights will be shown at the nominated spacings for construction (nominally 1000mm but user selectable) and the tendon is drawn with a depth showing the depth of the tendon/duct to the current vertical scale (see below).



By default, all of the tendon profiles in a member are shown in the elevation. If redrawing is too slow, the designer can select to only show the current tendon in the graphics in the 4.2.2 User Preferences->User Options Dialog. The current tendon profile as controlled by the selected data row in the Control Grid is shown in black, while the other tendon profiles are shown in a background grey colour. If a span has been selected in the Prestress Profile Data Grids or has been selected in the graphics using the toolbar or by clicking in the span with the left mouse button, the current tendon in that span will be shown in blue. Selecting a different span in the graphics will move the current span in the graphics to the new span and the current data row in the Prestress Profile Data Grids to the data row for the newly selected span.

Prestress Graphics Toolbars

Location Control:- Defines the location of the cursor in the graphics window from the left end of the frame. If the designer enters a number here, the cursor will move to the nearest screen pixel location (a screen pixel has a finite length so the location requested will be within the width of the pixel shown).

 Zoom (Ctrl + Z). This button will toggle between full screen mode and span zoom mode for the graphics in a window. In span zoom mode, the current span will be shown with the half span either side (if a cantilever is the previous or next span, the full cantilever will show) scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window.

 Move to next span (Ctrl + Right Arrow). The cursor will move automatically to the first Defined Profile Point in the new span.

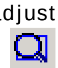
 Move to next point (Shift + Right Arrow). The cursor is moved to the next Defined Profile Point to the right. If it reaches the right end of the span, the next span is moved to the centre of the Window.

 Move to previous point (Shift + Left Arrow). The cursor is moved to the Defined Profile Point to the left. If it reaches the left end of the span, the previous span is moved to the centre of the Window.

 Move to previous span (Ctrl + Left Arrow). The cursor will move automatically to the last Defined Profile Point in the new span.

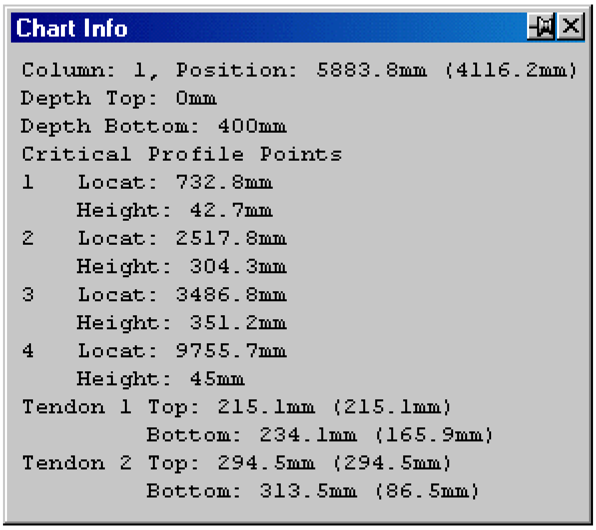
 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width of depth of the graph will be included than requested.

Then the user can then move around the graph using the Scroll Bars or the movement keys.  and  buttons will still move the cursor to the next and previous points. If the point is outside the viewable area it, the area of graph shown will adjust automatically to position the requested point near the left of the View Window.

Clicking this  button again or on the  or  buttons will return the Window to Full Screen Mode.

Clicking will change the mode to Span Zoom Mode in the span in which the cursor is positioned in the Select Zoom mode.

To view the information describing the tendon profile at any location over the length of the tendon, open the Information Dialog from the graphics toolbar by clicking this button or press Ctrl + I, and then left click at the desired location or enter the location in the Location Control described above. While the dialog is in view, click at any other location to view its information and the dialog data will be updated automatically.



The dialog will list:

1. The column number to the left of the location and distance to the cursor from that column and the distance to the next column in brackets.
2. Depth Top:- Depth to the lower top concrete surface at this location from the top of slab datum.
3. Depth Bottom:- Depth to the higher bottom concrete surface at this location from the top of slab datum.
4. Critical Profile Points:- The location of and profile height to strand centreline at each profile transition point in the span. For parabolic profiles, the lowest point on the profile is also nominated.
5. For each tendon that exists at this location, the depth at that location from the top of slab datum to
    1. the top of duct, and in brackets the depth to the top of the duct from the nearest top concrete surface
    2. the bottom of duct, and in brackets the depth to the bottom of the duct from the nearest bottom concrete surface

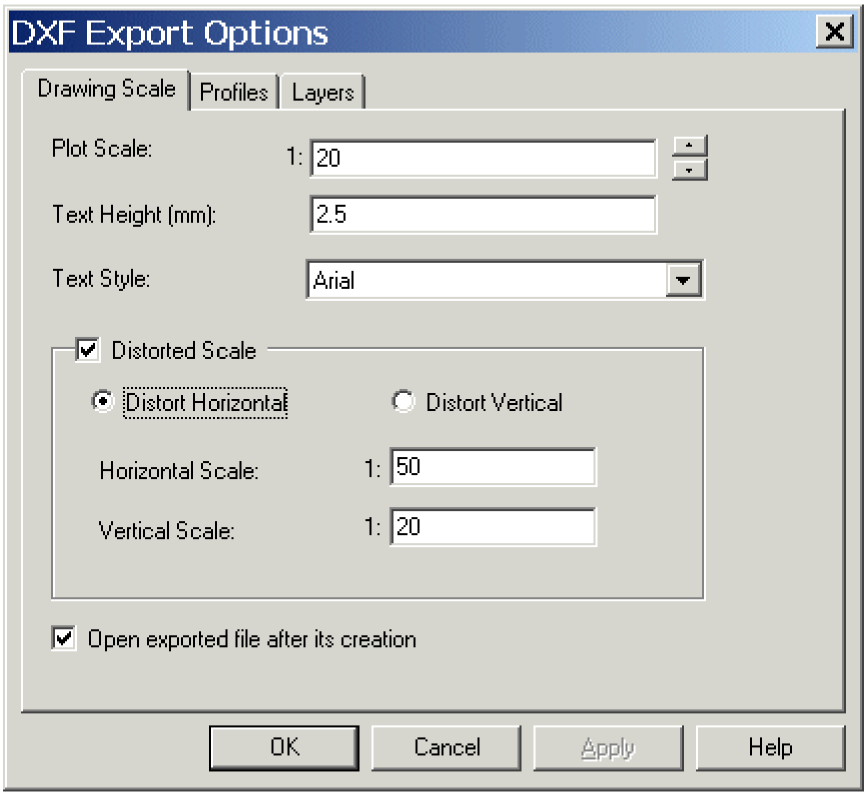
Profile DXF Output

Tendon profiles can be printed to a DXF file for transfer to CAD drawings. This can be done by Right Click with the mouse in the graphics window at the bottom of the prestress input screen and selecting Export to DXF or by selecting Export - Export to DXF from the File Menu. Either of these options will open the following Dialog for to define the requirements for the creation of the DXF file. Note that any changes made to the data in this dialog will be saved as the default options next time it is used.

The drawing will be created in 1:1 scale. RAPT allows the definition of horizontal and vertical scales for printing purposes as defined below.

The dialog has 3 tabs requiring the following information

Drawing Scale



Allows the user to define the following data related to the DXF drawing.

Plot Scale
While the drawing will be created in paper space at 1:1 scale, RAPT needs to know the planned plot scales to allow it to scale the text relative to the remainder of the drawing. In this way, when the drawing is finally printed, the text will be the correct size on the printed drawing.

Text Height
Final text size in mm. In creating the DXF file, the text height will be scaled so that the final text size on the scaled plot is equal to this text height.

Test Style
Style to be used for printing the text, selected from a list of available types.

Distorted Scale
Elevations are often drawn to a distorted scale. Normally in this case the length scale is more compressed than the height scale, but the reverse is also possible. To simulate this, RAPT allows the user to request a distorted scale. The distortion can be achieved in two ways, by reducing the length relative to the height, or by increasing the height relative to the length. The results are quite different. If the length is distorted, then length dimensions will not scale properly in your CAD program. In either case, one of the length/width scales must be equal to the Plot Scale.

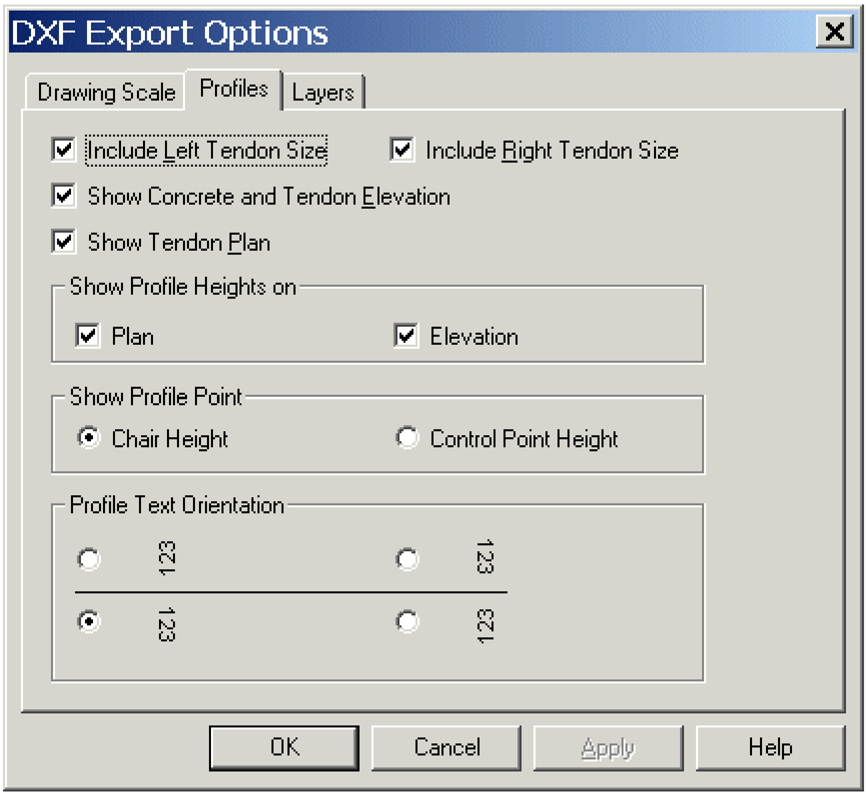
Distort Horizontal
If Distort Horizontal is selected, all horizontal dimensions will be divided by (1/V) / (1/H) = H/V.

Distort Vertical
If Distort Vertical is selected, all vertical dimensions will be multiplied by (1/V) / (1/H) = H/V.

Horizontal Scale
Horizontal scale to be used in the distorted scale calculations.

Vertical Scale
Vertical scale to be used in the distorted scale calculations.

Open Exported File
If this option is ticked, the program associated with the DXF file type on your Windows installation will be opened automatically and the DXF file loaded for viewing.



Allows the user to define the following profile/drawing attributes in creating the DXF drawing.

Include Left Tendon Size/Include Right Tendon Size
Include the tendon number and tendon size at the left, right or both ends of the tendon.

Show Concrete and tendon Elevation
Show the concrete profile and the tendon profile in elevation.

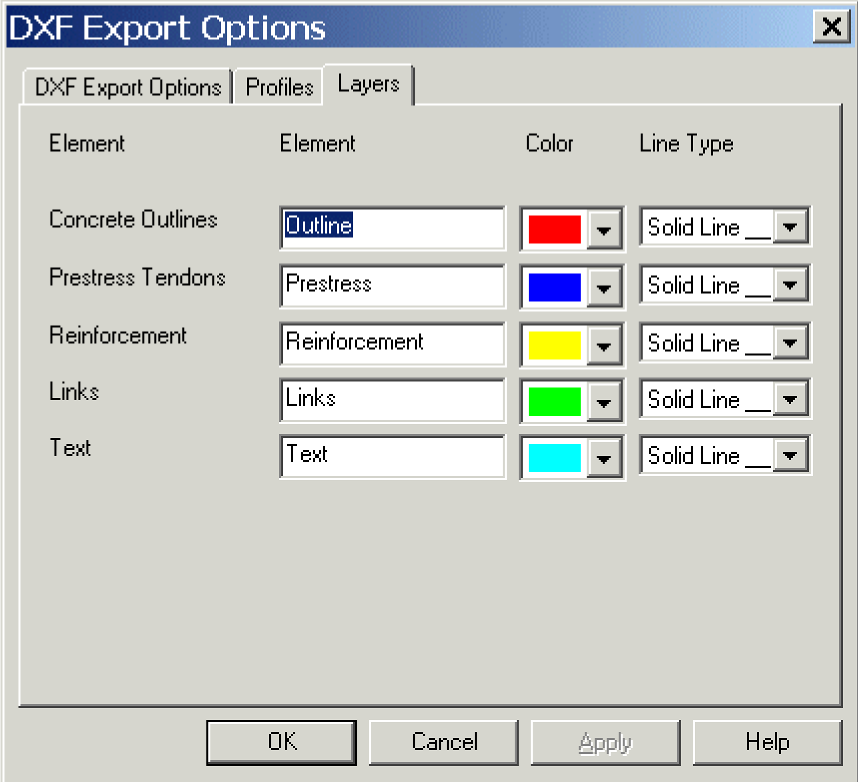
Show Tendon Plan
Show the tendon profile in plan view.

Show Profile heights on Plan/Elevation
Show the tendon profile heights on plan, elevation or both.

Show Profile Point - Chair height/Control Point Height
When showing the tendon profile heights show either the chair heights at nominal spacings or only show the critical heights at the high and low points.

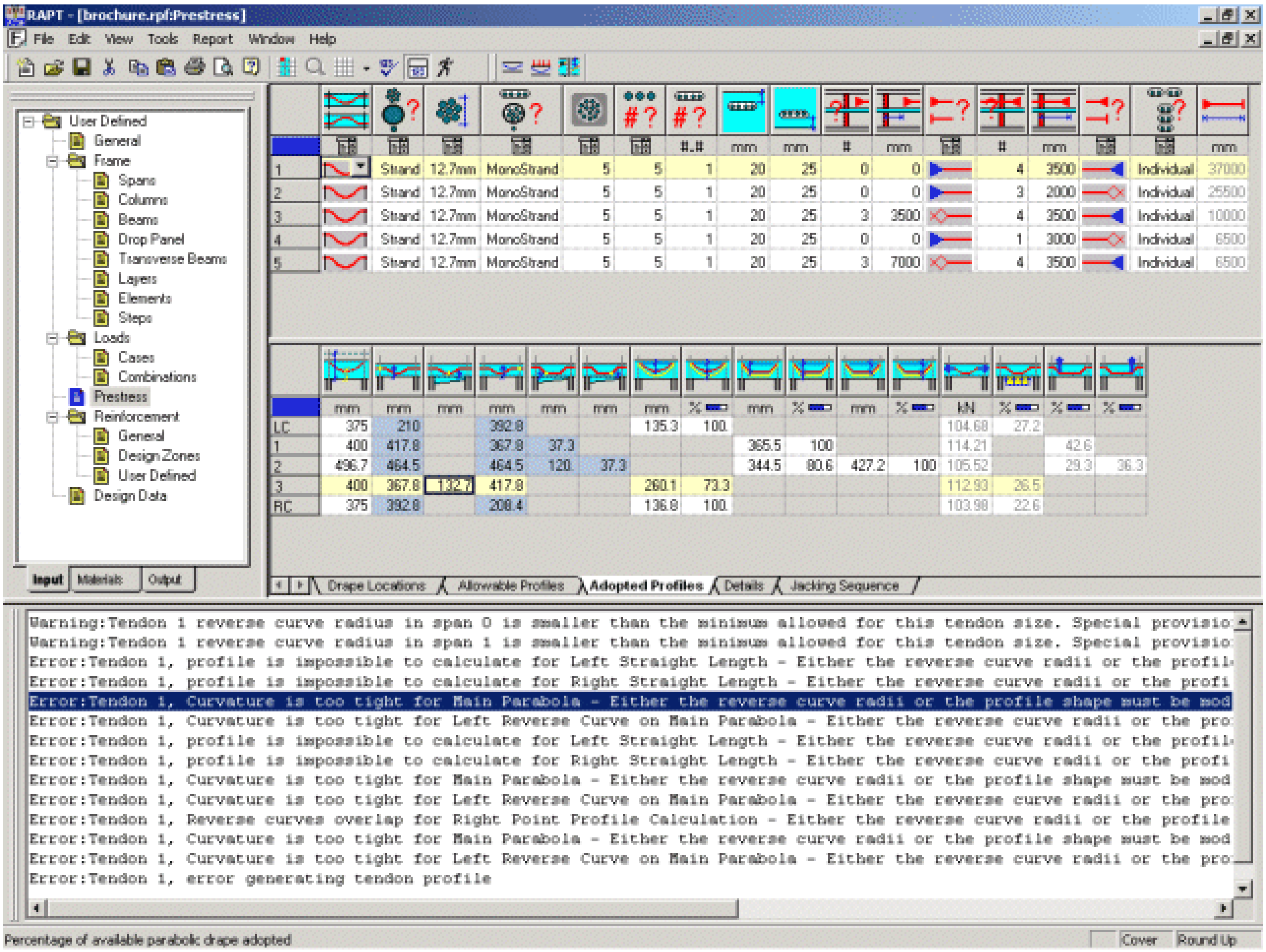
Profile Text Orientation
The orientation at which the profile height text is to be written.

Layers



Allows the user to define different colours and line types for the different DXF drawing layers.

Profile Errors



RAPT will not be able to generate the tendon profile for viewing or for design if the curvatures required to create the interacting curves are too small compared to the curvature specified by the Reverse Curve Radius. When this occurs, error messages will be presented in the 4.5.4 Message Window as shown above.

In the case above, the 5th message has been selected using a mouse double click. The message line is highlighted and the data cell focus has moved to

1. Prestress Data
2. Adopted Profiles data
3. for tendon 1,
4. span 3,
5. low point profile height for a parabolic profile.

This error occurred because the calculation of the main parabola could not be completed in span 3 of tendon 1 because the curvature of the main parabola was smaller than the minimum curvature set in Drape Locations for that span. RAPT has indicated the problem is with the low point profile and has nominated the span number and the tendon number but there are actually several data items that could possibly be modified to fix the problem. Selecting any of the Window Tabs to view other data will also show the span number causing the problem. The other data items that can be modified to affect tendon curvature problems are

1. Heights at each end of the parabola
2. Locations of each end of the parabola
3. Reverse Curve Radius. This should not be reduced to less than the minimum unless specific action is taken to allow for high bearing stresses, design of bursting reinforcement and investigation of the wear effects on the this duct. It should never be reduced to zero for a final design.
4. For end spans of a tendon, the reverse curve at the anchorages.
5. The profile requirements in 1 and 2 above in an adjoining span

When this type of error occurs RAPT will still attempt to produce a graphical view of the tendon profile so that the designer can also see the graphical representation of the data to help identify the problem. Clicking on the 

button will toggle between the Message View on Frame Graphics View to allow the designer to see the profile shape. To generate the profile RAPT will internally set the reverse curve radius for the offending spans to zero. If you look closely at the profile or zoom into the offending span this will be obvious. In some cases the error occurs at a stage where it is not possible for RAPT to set the radius to zero and recalculate. In these cases the profile will be shown as a straight line full length of the tendon at zero depth and RAPT will allow the designer to set the Reverse Curve Radius to zero in the Drape Locations View. This should only be a temporary measure to view the profile. A final design should never be produced with a reverse curve radius of zero as this is not able to be built and should only use a reverse curve radius less than the minimum if allowed for in the design and detailing as discussed in 3 above.

7.2.5.1 Drape Locations

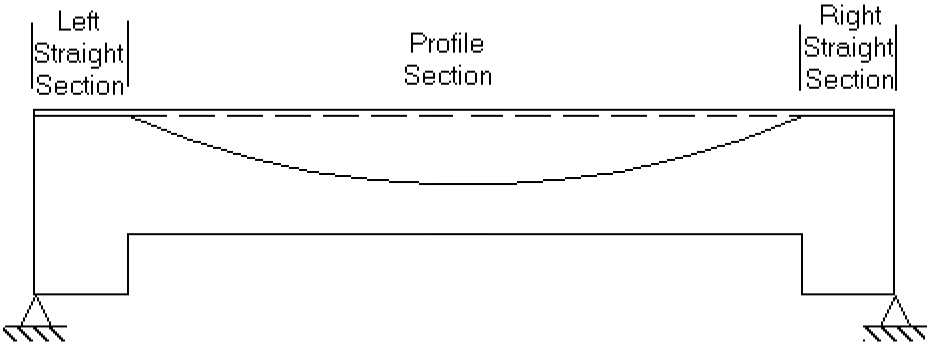
Drape locations define the end points of the different zones of a tendon profile in a span and points where individual strands in each tendon stop in spans. RAPT calculates a set of default drape locations as described below. The designer is free to modify these drape locations if desired to suit the particular design requirements of a run. The default values can be reset using the toolbar function defined below.

There is a separate table of this data for each design strip of each tendon profile nominated in the 7.2.5 Prestress Control Window.

Profile Sections

RAPT divides the tendon profile in each span into 3 sections as described below and in the diagram.

1. Left Straight Section - Extends from the left end of the tendon in the span to a user/program nominated location. Length may be zero. Profiled as a straight line from the defined point at the connection to the Profile Section to the left end point in the span with a transition curve to match the end of the line with the slope of the end of the Right Straight Section in the previous span. The tendon will always pass through the nominated end heights. For a tendon that does not reach the left end of the span and with no straight length, this location will be the start point of the tendon in the span.
2. Drape Section
3. Right Straight Section - Extends from the right end of the tendon in the span to a user/program nominated location. Length may be zero. Profiled as a straight line from the defined point at the connection to the Profile Section to the right end point in the span with a transition curve to match the end of the line with the slope of the end of the Left Straight Section in the previous span. The tendon will always pass through the nominated end heights. For a tendon that does not reach the right end of the span and with no straight length, this location will be the end point of the tendon in the span.



Default Drape Locations

By default RAPT will drape the tendons to the column centreline with the following exceptions:-

RAPT considers the default settings in 5.3 Design Standard->Prestress for tendon drape locations as well as the specific concrete shape for the member being input in calculates the most logical place to drape the tendons to. The options in the default settings are to drape to the face of the support by selecting either/or

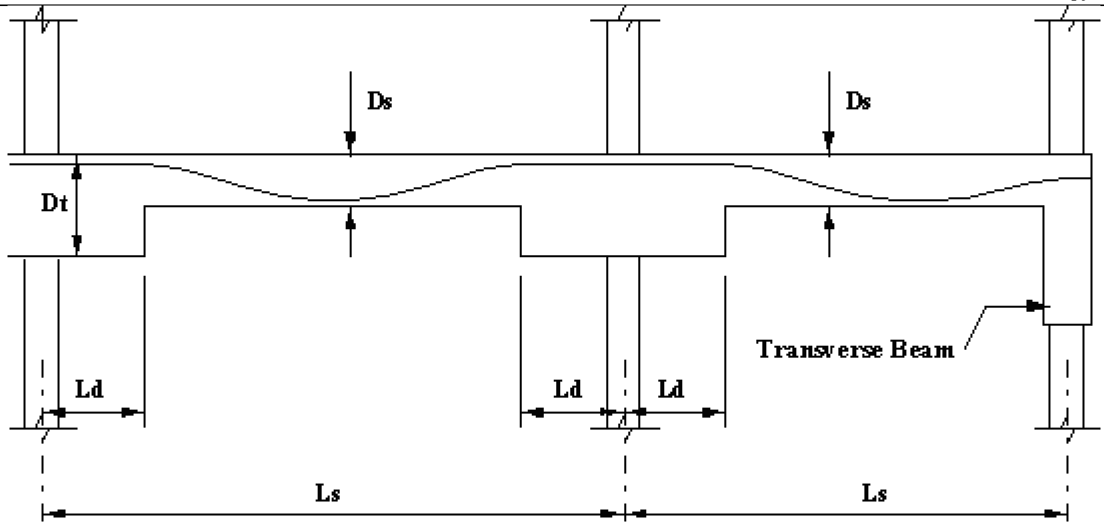
1. Face of columns
2. Face of Transverse beams

instead of the centreline of the column

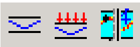
The following are considered as RAPT chooses the default drape locations:

1. for one-way systems:
    1. At internal columns, if (Dt / Ds) x (Ls / Ld)1/4 > 2.2 (see diagram below) then the tendons drapes to the face of the drop panel or transverse band beam, where Dt = Total Depth, Ds = Slab or Beam Depth, Ls = Span Length, Ld = Length of Drop.
    2. At end columns the tendon is draped to the column centre-lines irrespective of all other rules above.
    3. When there is a continuous drop panel along the span (local beam) the tendon is draped to the column centre-lines or support face as defined above.
2. for two-way systems:
   The rules are the same as for one-way systems with the following considerations also taken into account
    1. column strip: Drop panels are only considered in the column strip for calculating drape locations.
    2. middle strip: tendon draped to column centre-lines or support face, unless a drop panel extends full width of the panel or a transverse band beam is present.

The formula ((Dt / Ds) x (Ls / Ld)1/4 > 2.2) has been designed so that RAPT drapes the tendons to the most profitable place. Depending on the size of the drop panel, this may be at the edge of the drop panel or at the column centre-line.



Prestress Toolbar



When program focus is in the Drape Locations Window, this toolbar will be available. Only the toolbar buttons available at any time will be active for use. The remainder will be shown in background colour.

The buttons are

Recalculate Default Profile Data:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Default Profile Data

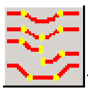
Recalculate Tendons and Drapes from Balanced Loads:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Tendons and Drapes from Balanced Loads

Not available in Drape Locations.

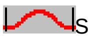
Data Definition

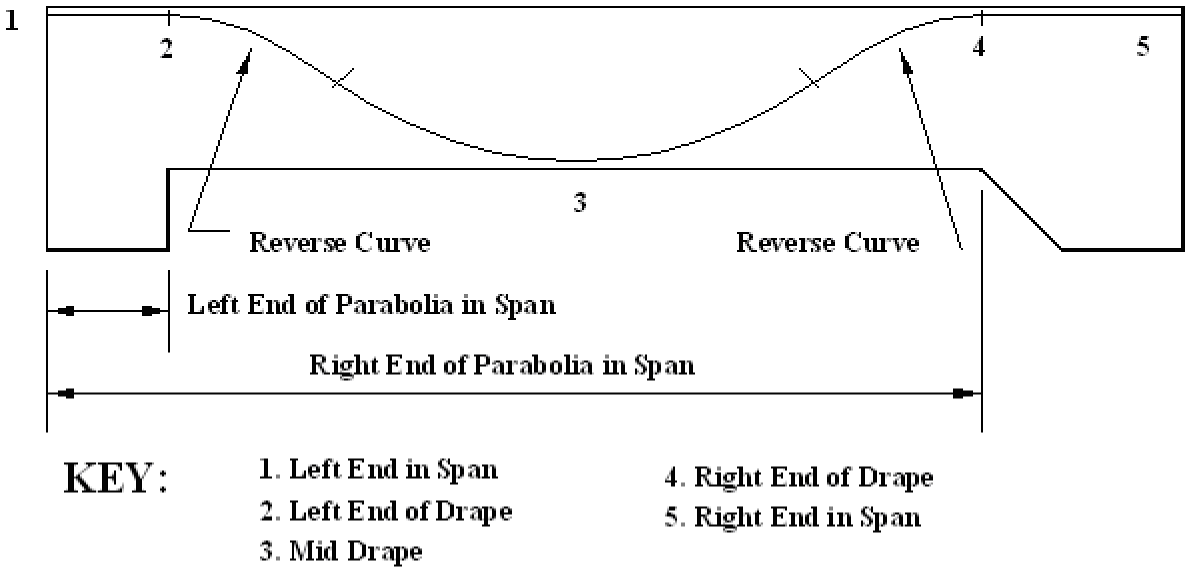
		mm	mm	mm	mm	mm	#	#	mm	#	mm	#.#	mm
LC		0	0			5000	0	0		0		0	1000
1		0	0	3000		5000	1	0	1500	0		0	1000
2		0	0	3333.3	6666.7	5000	0	0		0		0	1000
3		0	0			5000	0	0		1	3000	0	1000
RC		0	0			5000	0	0		0		0	1000

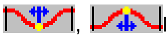
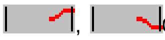
Drape Locations | Allowable Profiles | Adopted Profiles | Details | Jacking Sequence

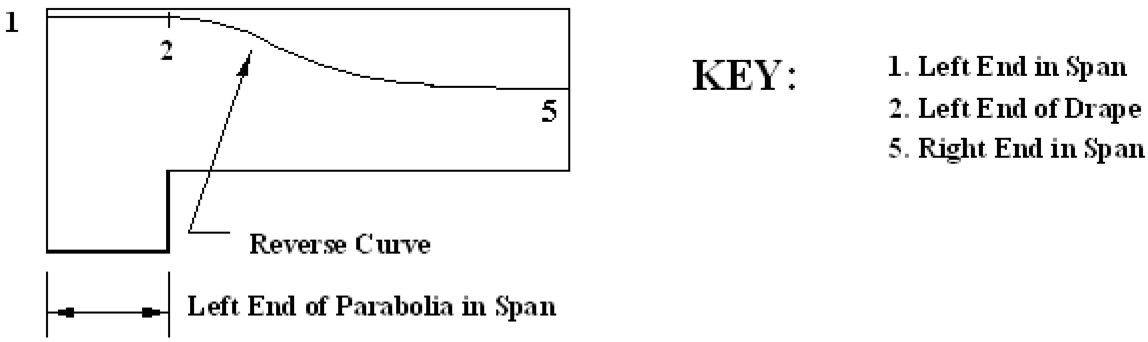


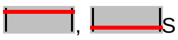
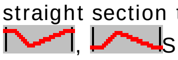
Tendon Profile Type

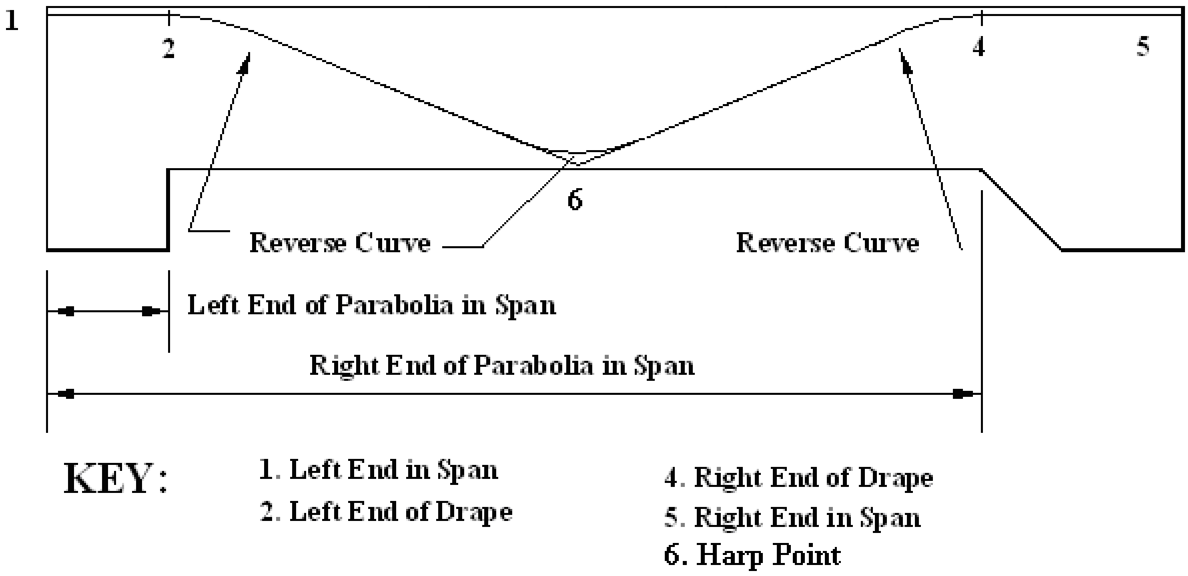
1. , Single Parabolic Profile defined by high points at each end of the profile section and a low point at the mid-distance between the end of the parabola points (in the diagram below, the horizontal location of point 3 is mid-length between points 2 and 4 and is fixed). If the end heights are different, the parabola will be skewed and the lowest point on the parabola will not be the mid length point used to define the parabola shape. Reverse curves will be supplied at the ends of the parabola to transition the parabola to the connecting profile sections. The curvature of the parabola will be checked to make sure it is not less than the curvature defined for the reverse curves. If it is less an error message will be generated and the profile will be drawn with zero reverse curve radius (see diagram in Reverse Curve Radius). This type of profile should not be used where the tendon only extends a short distance into a span. In this case use one of the Passive Parabolic Profiles below.

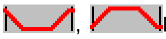


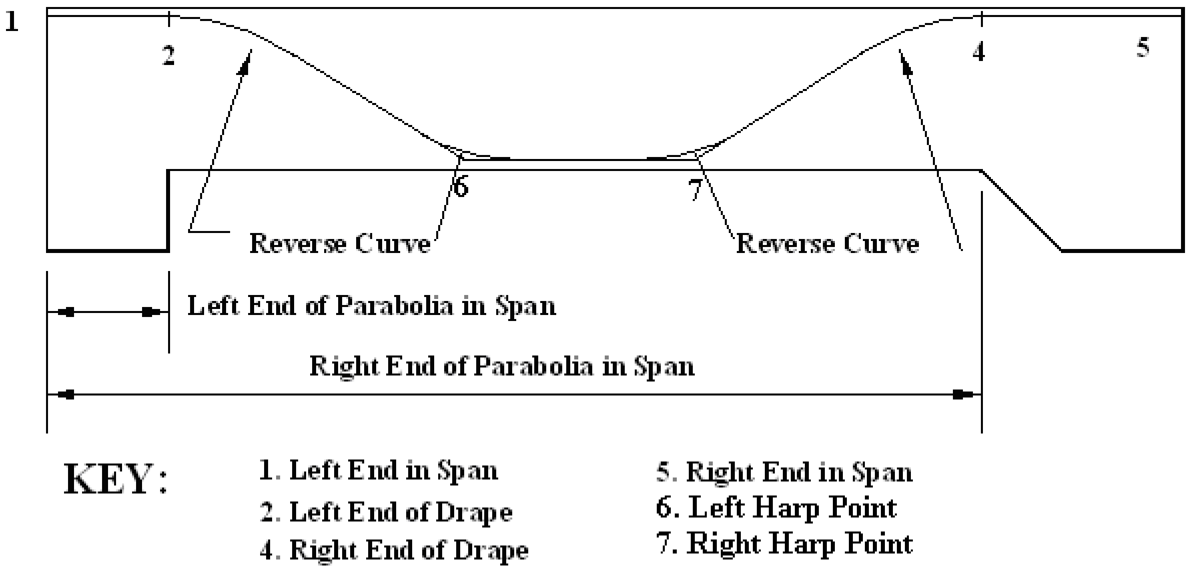
2.  Double Parabolic Profile defined by high points at each end of the profile section and a low point at a user defined location between the high points (in the diagram above, the horizontal location of point 3 is variable relative to points 2 and 4). The default low point location is at the mid-distance between the end of the parabola points. The 2 half parabolas are profiled with reverse curves at the extreme ends to transition the parabolas to the connecting profile sections. The parabolas join at the nominated location at zero slope. The curvature of each half parabola will be checked to make sure it is not less than the curvature defined for the reverse curves. If it is less an error message will be generated and the profile will be drawn with zero reverse curve radius (see diagram in Reverse Curve Radius). The curvatures of the two half parabolas will normally be different (unless the two half parabolas have exactly the same length and heights) and the resulting balanced loads for the two half parabolas will be different.
3.  Cantilever or Passive Parabolic Profile at the left end of a tendon defined by a high point at the right end of the profile section and a low point at the left end of the profile section. The slope at the low point is zero. This is commonly used in left cantilevers and at construction joints within approximately .35 of the span length to the left of a support. When used internally e.g. at a construction joint, the tendon in this span is not considered in the calculations to determine a number of tendons and tendon profiles from a set of balanced loads. Where the tendon only extends a very short distance into a span, it may not be possible to achieve the maximum profile available at the support due to curvature problems with the transition curve.
4.  Cantilever or Passive Parabolic Profile at the right end of a tendon defined by a high point at the left end of the profile section and a low point at the right end of the profile section. The slope at the low point is zero. This is commonly used in right cantilevers and at construction joints within approximately .35 of the span length to the right of a support. When used internally e.g. at a construction joint, the tendon in this span is not considered in the calculations to determine a number of tendons and tendon profiles from a set of balanced loads.

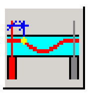


5.  Straight Line Profile over span length between defined profile heights at the left and right support centrelines. Reverse curves will be supplied at the ends of the straight section to transition the straight section to the connecting profile sections where necessary.
6.  Single Point Harped Profile defined by high points at each end of the profile section and a low point at a user defined location between the high points (point 6 in the diagram below). The default low point location is at the mid-distance between the end of the profile section points. A circular reverse curve which is tangential to the two straight line profile portions will be provided at the harp point. The actual low point of the tendon will not reach the defined low point to achieve this transition. The designer can define a low point which violates cover to bring the actual low point down to the desired cover. Reverse curves will be supplied at the ends of the straight sections to transition the straight sections to the connecting profile sections where necessary.

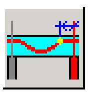


7. Double Point Harped Profile defined by high points at each end of the profile section and two low points at user defined locations between the high points (points 6 and 7 in the diagram below). The default low point locations are at the third points between the end of the profile section points. A circular reverse curve which is tangential to the two straight line profile portions will be provided at each harp point. The actual low point of the tendon will not reach the defined low point to achieve this transition unless the line between the two harp points is parallel to the bottom surface. The designer can define a low point which violates cover to bring the actual low points down to the desired cover. Reverse curves will be supplied at the ends of the straight sections to transition the straight sections to the connecting profile sections where necessary.



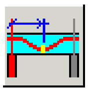
Length of Left End Straight Section/Start of Drape Section in Span

The dimension from the left column in the span to the left end of the profile section in the span. The tendon shape to the left of this point will be a straight line between the profile height at the left end of the tendon in the span and the profile height at the left end of the parabola section. If this straight portion has no length RAPT will force the profile heights at this point to be the same. A transition curve will be provided merge this straight section with the tendon in the previous span. The default location is as defined in the general discussion above.

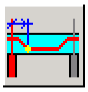
Length of Right End Straight Section/End of Drape Section in Span

The dimension from the right column in the span to the right end of the profile section in the span. The tendon shape to the right of this point will be a straight line between the profile height at the right end of the tendon in the span and

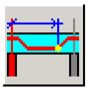
the profile height at the right end of the parabola section. If this straight portion has no length RAPT will force the profile heights at this point to be the same. A transition curve will be provided merge this straight section with the tendon in the next span. The default location is as defined in the general discussion above.

Location of Low Point of Parabola

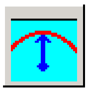
Defines the location, from the left column centreline of the span, of the low point of the parabola for a Double Parabolic Profile.

Location of Left Point Load from Left End of Span

Defines the location, from the left column centreline of the span, of the low point for a Single Point Harped Profile or the left low point for a Double Point Harped Profile. The location is the intersection of the two straight lines defining the harp profile shape.

Location of Right Point Load from Left End of Span

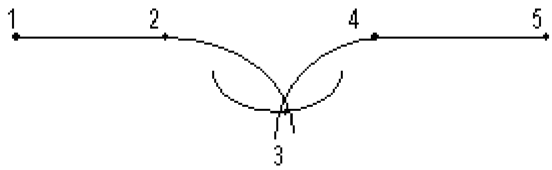
Defines the location, from the left column centreline of the span, of the right low point for a Double Point Harped Profile. The location is the intersection of the two straight lines defining the harp profile shape.

Reverse Curve Radius

The reverse curve radius used to define all of the transition curves in this span. RAPT automatically provides transition curves at all points where the different tendon sections join and at harped tendon points. For parabolic transition curves, a parabola of an equivalent curvature is used. For circular transition curves, a circle of this radius is used.

The default value or reverse curve radius used here for slabs and beams is defined in the 5.3 Design Standard->Prestress data. The tendon definitions in 6.8.1.2 Materials Data define a minimum reverse curve radius allowed for each tendon. RAPT will give a warning if the radius specified is less than that value. A smaller radius may be used if the designer can ensure that the tendons can be constructed properly and also if the designer allows for the increased bearing stress in the curve and the bursting forces induced in the concrete inside the curve. Special thicker duct may be needed and bursting reinforcement may be required in these areas where the duct curvature is less than this figure. This also could apply in the main parabola section as well as in transition curves.

In some cases, it will not be possible to achieve a transition curve with the radius specified as shown below. If the transition curve length is too great in a case with a parabolic profile, the curvature of the main profile will be too large and will violate the maximum curvature allowed based on the specified minimum radius. In the case of harped tendons the transition curves at each end of a straight section will overlap if the radius is too large. The profile as specified is therefore not buildable in these cases and will be rejected by RAPT and errors message will be generated.

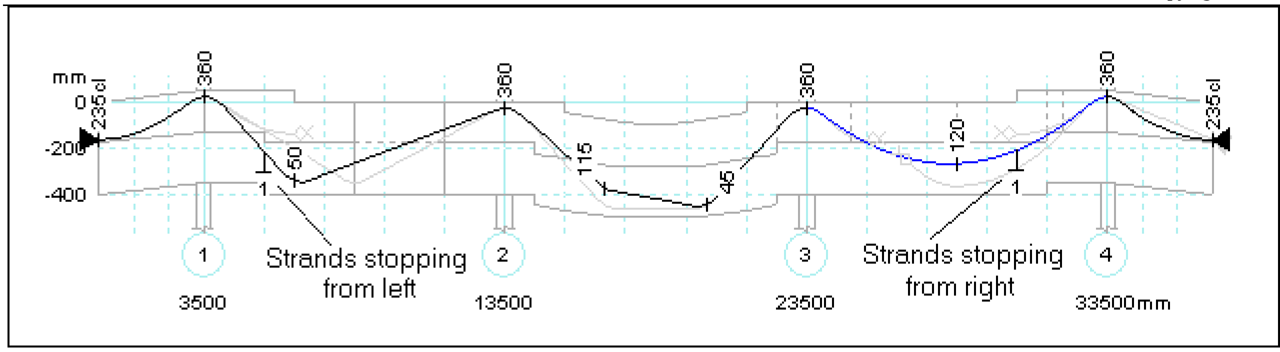


Reverse Curves that do not fit profile.

Terminating Strands

In the Prestress Control Grid, the user has defined a number of strands in each tendon for this profile. RAPT allows the designer to terminate some of the strands in a tendon at user defined locations along the member. The basic rules are

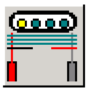
1. Strands can only terminate from a live end.
2. In double end stressed tendons, a strand can stop from either the left or the right end or from both ends.
3. There must be at least 1 strand at all points in the tendon length.
4. RAPT does not do separate friction loss calculations for the terminating strands. The friction loss calculation and especially the draw-in effect is calculated for the whole tendon. Strands stopping near a live anchor may actually be affected more by draw-in than the full length strands. It will be obvious from the 7.3.5.1 prestress forces diagram in the output if the termination point is within the draw-in zone of an anchorage.

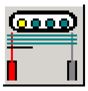


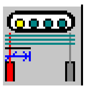
The stopping strands are shown on the graphics as shown above. To determine if a "strand stopping" symbol belongs to a strand stopping from the left or from the right, the symbol is drawn in direction of the slope of the tendon at the point on the tendon where the strands stop moving from the "anchor from which the strands stop". In the case above,

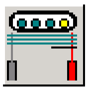
1. the symbol for "Strands Stopping from Left" in span 1 is downwards because the tendon is sloping downwards from left to right at that termination point.
2. the symbol for "Strands Stopping from Right" in span 3 is downwards because the tendon is sloping downwards from right to left at that termination point.

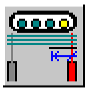
The next five data fields allow the designer to specify the information for terminating strands.

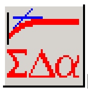
Number of Strands from Left End Terminating in Span and Restarting Before the End of the Tendon
In a double end stressed tendon, if a strand is to exist at both anchorages, then if must have two termination points. In this case, this terminating strand should be nominated in this data column as stopping in this span but starting again in this span or a span to the right of this one. It should also be included in the "Number of Strands from Right End Terminating in Span" data column for the span where it restarts.

Number of Strands from Left End Terminating in Span
In a left end stressed or a double end stressed tendon, this is the number of strands that exist at the left anchorage and terminate in this span and do not exist at the right anchorage.

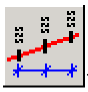
Location of terminating ends of strands stressed at the left end.
Location from the left end of the span of the strands stopping from the left in this span. The number of strands stopping at this point is the sum of the two numbers above.

Number of Strands from Right End Terminating in Span
In a right end stressed or a double end stressed tendon, this is the number of strands that exist at the right anchorage and terminate in this span. Each strand nominated here may or may not exist at the left anchorage depending on which of the strand stopping numbers above they relate to.

Location of terminating ends of strands stressed at the right end.
Location from the right end of the span of the strands stopping from the right in this span.



Extra angle change in this span
Angle change in radians that occurs in this span and is not part of the normal profile angle changes or the accidental wobble effects. This allows designers to make allowance for extra friction caused by horizontal curves etc. This angle change is averaged over the length of the span and is added to the normal friction effect on the basis of the square root of the sum of the squares of the angle changes.

Tendon Profile Point Spacing
The tendon profile spacing the designer wants to use for in this span. RAPT will base the number of profile spaces in each profile section on this value and will round the number of spaces in each profile section to a whole number based on the setting in the 4.2.2 User Preferences->User Options dialog. The default value used here is defined in the 5.3 Design Standard->Prestress data.

7.2.5.2 Allowable Profiles

Allowable Profiles defines the cover limits at each of the defined profile height locations in each span for a tendon profile. The face that the covers apply to depends on the Profile Orientation defined in the 7.2.5 Prestress Control Window. The default values are determined from the top and bottom covers defined in the 7.2.5 Prestress Control Window for the design strip of the tendon profile being investigated. At the anchorages, the values are the anchorage height.

RAPT uses these figures as the basis for the creation of the default adopted tendon profiles and also to provide warnings if the adopted profiles violate these cover limits at any point. The designer is free to modify these values to provide different covers at different points in each span. Modifying the default cover to top or bottom in the Prestress Control Window will reset the Allowable Profiles associated with the modified value. All of the default values can be reset using the toolbar function defined below. If the cover is modified at a point, the associated Adopted Profiles will be checked to see if they violate the cover. If they do violate cover they will be reset to the new cover automatically. Once the allowable offsets have been modified, the toolbar functions can be used to reset the Adopted Profiles to these figures for the whole tendon profile if desired.

There is a separate table of this data for each design strip of each tendon profile nominated in the 7.2.5 Prestress Control Window.

Allowable Profiles Toolbar

When program focus is in the Allowable Profiles Window, this toolbar will be available. Only the toolbar buttons available at any time will be active for use. The remainder will be shown in background colour.

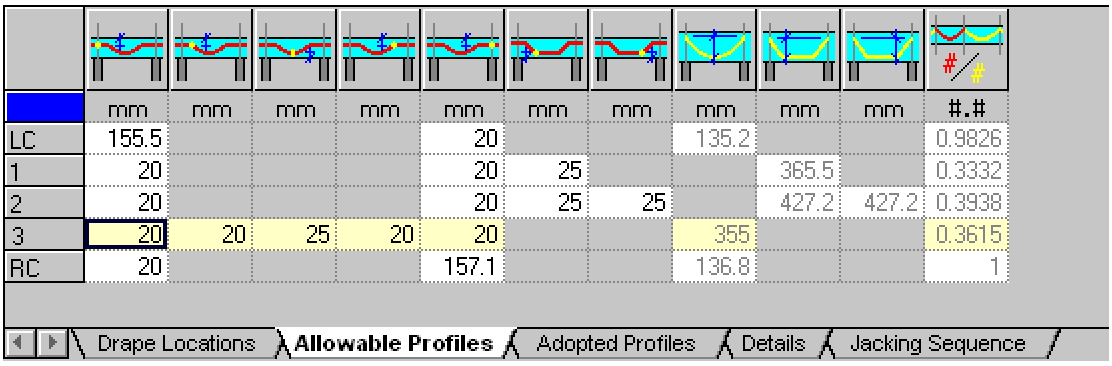
The buttons are

Recalculate Default Profile Data:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Default Profile Data

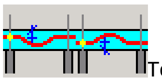
Recalculate Tendons and Drapes from Balanced Loads:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Tendons and Drapes from Balanced Loads

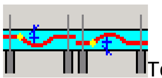
Not available in Allowable Profiles.

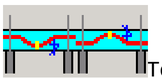
Data Definition

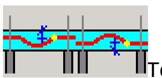


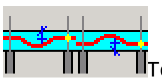
	mm	mm	mm	mm	mm	mm	mm	mm	mm	mm	#.#
LC	155.5				20			135.2			0.9826
1	20				20	25			365.5		0.3332
2	20				20	25	25		427.2	427.2	0.3938
3	20	20	25	20	20			355			0.3615
RC	20				157.1			136.8			1

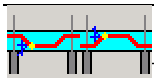
Tendon Cover to duct at left end of span:- Defines the allowable cover to the duct at the left end of the tendon in the span. If no left straight section is defined in drape locations, it also defines the cover at the left end of the 7.2.5.1 profile section.

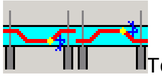
Tendon Cover to duct at left end of Parabola:- If a left straight section is defined in drape locations, defines the allowable cover to the duct at the left end of the 7.2.5.1 profile section.

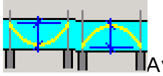
Tendon Cover to duct at low point of parabola:- Defines the allowable cover to the duct on the low point face of the parabola if a Single or Double Parabola Profile Type is defined in 7.2.5.1 profile section.

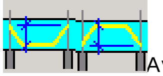
Tendon Cover to duct at right end of Parabola:- If a right straight section is defined in drape locations, defines the allowable cover to the duct at the right end of the 7.2.5.1 profile section.

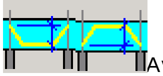
Tendon Cover to duct at right end of span:- Defines the allowable cover to the duct at the right end of the tendon in the span. If no right straight section is defined in drape locations, it also defines the cover at the right end of the 7.2.5.1 profile section.

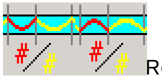
Tendon cover to duct at left point load point:- Defines the allowable cover at the harp point in a Single Point Harped Profile or at the left harp point in a Double Point Harped Profile as defined in 7.2.5.1 profile section.

Tendon cover to duct at right point load point:- Defines the allowable cover at the right harp point in a Double Point Harped Profile as defined in 7.2.5.1 profile section.

Available Parabola Drape:- Defines the maximum drape available in any of the parabola type profiles. The value is calculated at the mid-point of a Single or Double Parabola Type and at the external end of Cantilever or Passive Parabolic Profiles.

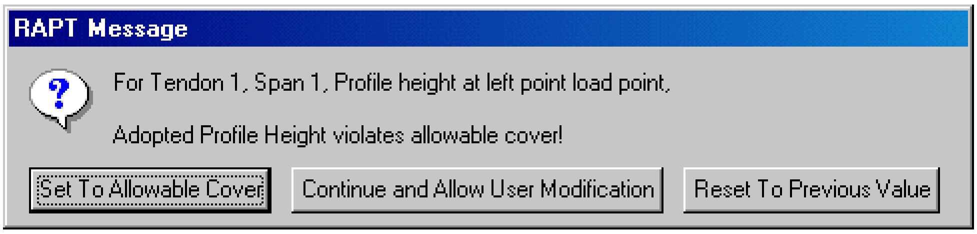
Available drape at left point load point:- Defines the maximum drape available at the harp point in a Single Point Harped Profile or at the left harp point in a Double Point Harped Profile as defined in 7.2.5.1 profile section.

Available drape at right point load point:- Defines the maximum drape available at the right harp point in a Double Point Harped Profile as defined in 7.2.5.1 profile section.

Relative Minimum Tendon Ratio:- Based on a nominal applied load on all spans, this is the ratio of the number of tendons required in this span to the maximum number required in all of the spans for this tendon profile based on the available drape in each span. It is not an editable data cell. It gives the designer guidance on the relative number of tendons that will be required in each span and therefore is an indicator of the areas where short tendons can be used to provide a more economical design.

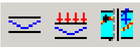
7.2.5.3 Adopted Profiles

The Adopted Profiles in RAPT define the profile shape by defining the heights of the tendon to the centreline of strand at the defined points in each profile type as defined in 7.2.5.1 Tendon Profile Type. For defined profile locations which are within 100mm of another location, RAPT will always ensure that the heights at the 2 locations are consistent accepting the last input value as the value for both. The designer is free to set the adopted offsets at any defined profile location to any reasonable value. If the height defined is outside the cover specified in the Allowable Profiles window, RAPT will give a warning to this effect and allow the designer to select one of three options to proceed (see diagram below). These are



1. Set to Allowable Cover:- Overrides the number input by the designer and sets the profile height at this location to the cover nominated in 7.2.5.2 Allowable Profiles.
2. Continue and Allow User Modification:- Accepts the new value. It is the designers choice to justify the cover violation. This can be used to define tendons which are external to the concrete as is often the case in designs involving the strengthening already constructed members.
3. Reset to Previous Value:- Cancels the previous edit and sets the value in the cell to its value before this edit operation.

Adopted Profiles Toolbar



When program focus is in the Adopted Profiles Window, this toolbar will be available.

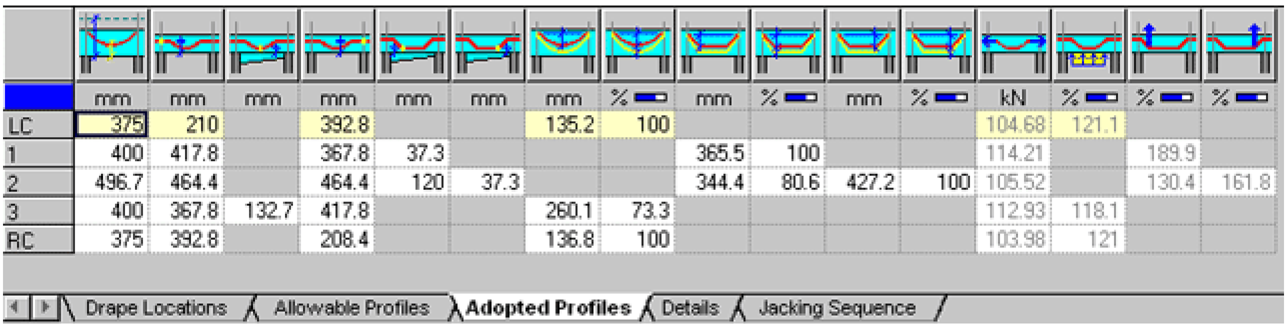
The buttons are

Recalculate Default Profile Data:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Default Profile Data

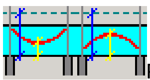
Recalculate Tendons and Drapes from Balanced Loads:- Refer to 7.2.5 Prestress Control Toolbar->Recalculate Tendons and Drapes from Balanced Loads

Toggle Covers Data:- When a profile height cell is in focus in this grid, selecting this icon will add an extra data column to the grid immediately to the right of the column in focus. This new column of data refers to the profile heights at the same defined profile point but is a depth from the logical reference surface for that point as defined by the 7.2.5 Profile Orientation to the centreline of the strand rather than as a depth from the Profile Datum Height for that span. Thus the designer can define the profile height either as a distance from the Profile Height Datum or a distance from the reference surface. This will sometimes make it easier for the designer to relate profiles to the actual concrete surface the tendon is near to and also make the definition of profile heights easier, especially with complicated member shapes.

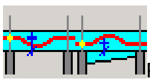
Data Definition



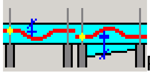
	mm	mm	mm	mm	mm	mm	mm	%	mm	%	mm	%	kN	%	%	%
LC	375	210		392.8			135.2	100					104.68	121.1		
1	400	417.8		367.8	37.3				365.5	100			114.21		189.9	
2	496.7	464.4		464.4	120	37.3			344.4	80.6	427.2	100	105.52		130.4	161.8
3	400	367.8	132.7	417.8			260.1	73.3					112.93	118.1		
RC	375	392.8		208.4			136.8	100					103.98	121		

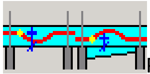


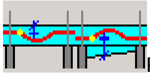
Profile Height Datum - The adopted profile heights are measured from a datum level in each span. This level defaults to the soffit of the concrete section at the low point location for draped profiles. The designer can set this level to any that he finds easier to use in a particular case to help to visualise the tendon profile and the profile heights.

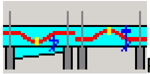


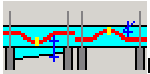
Profile height at left end of span:- Height from the Profile height Datum in this span to the centreline of the strand at the left end of the tendon in this span.

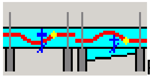
Profile cover to centreline at left end of span:- Height from the top concrete surface () and () or bottom concrete surface () or () at this location to the centreline of the strand at the left end of the tendon in this span.

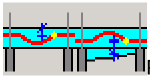
Profile height at left end of parabola:- Height from the Profile height Datum in this span to the centreline of the strand at the left end of the 7.2.5.1 Profile Section in this span.

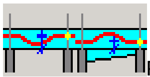
Profile cover to centreline at left end of parabola:- Height from the top concrete surface () or bottom concrete surface () at this location to the centreline of the strand at the left end of the 7.2.5.1 Profile Section in this span.

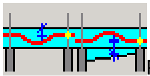
Profile Height at midpoint of parabola:- Height from the Profile height Datum in this span to the centreline of the strand at the location of the defined parabola low point in this span. This is only available for , , , Tendon Profile Types.

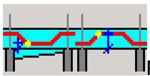
Profile cover to centreline of strand at midpoint of parabola:- Height from the bottom concrete surface () or top concrete surface () at this location to the centreline of the strand at the location of the defined parabola low point in this span.

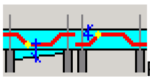
Profile height at right end of parabola:- Height from the Profile height Datum in this span to the centreline of the strand at the right end of the 7.2.5.1 Profile Section in this span.

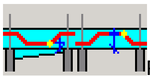
Profile cover to centreline of strand at right end of parabola:- Height from the top concrete surface () or bottom concrete surface () at this location to the centreline of the strand at the right end of the 7.2.5.1 Profile Section in this span.

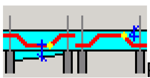
Profile height at right end of span:- Height from the Profile height Datum in this span to the centreline of the strand at the right end of the tendon in this span.

Profile cover to centreline of strand at right end of parabola:- Height from the top concrete surface () and () or bottom concrete surface () or () at this location to the centreline of the strand at the right end of the tendon in this span.

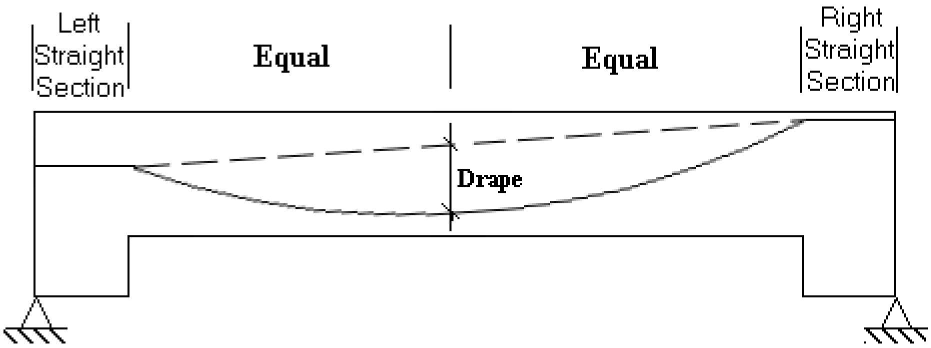
Profile height at left point load point:- Height from the Profile height Datum in this span to the intersection of the straight lines forming the left harped profile shape in this span. This is only available for , , , Tendon Profile Types.

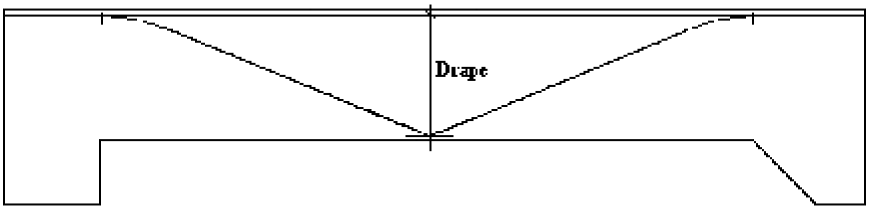
Profile cover to centreline at left point load point:- Height from the bottom concrete surface () or top concrete surface () at this location to the intersection of the straight lines forming the left harped profile shape in this span

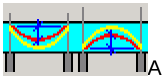
Profile height at right point load point:- Height from the Profile height Datum in this span to the intersection of the straight lines forming the right harped profile shape in this span. This is only available for , Tendon Profile Types.

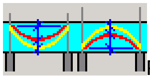
Profile cover to centreline at right point load point:- Height from the bottom concrete surface () or top concrete surface () at this location to the intersection of the straight lines forming the right harped profile shape in this span

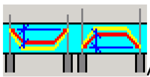
The Drape discussed below is the height from the defined low point on a tendon profile to the intersection with the straight line between a line joining the 2 high points on the profile at the location of the defined low point. See diagrams below

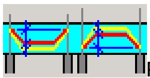


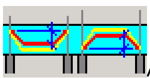


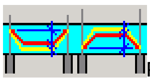
Adopted Parabolic Drape:- Drape provided by the parabolic profile in this span.

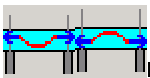
Percentage of available parabolic drape adopted:- The drape provided by the parabolic profile in this span as a percentage of the total available drape as defined in 7.2.5.2 Allowable Profiles.

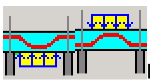
Adopted left point load Drape:- Drape provided at the left harped profile point in this span.

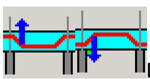
Percentage of available left point load drape adopted:- The drape provided at the left harped profile point in this span as a percentage of the total available drape at this point as defined in 7.2.5.2 Allowable Profiles.

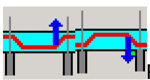
Adopted right point load Drape:- Drape provided at the right harped profile point in this span.

Percentage of available right point load drape adopted:- The drape provided at the right harped profile point in this span as a percentage of the total available drape at this point as defined in 7.2.5.2 Allowable Profiles.

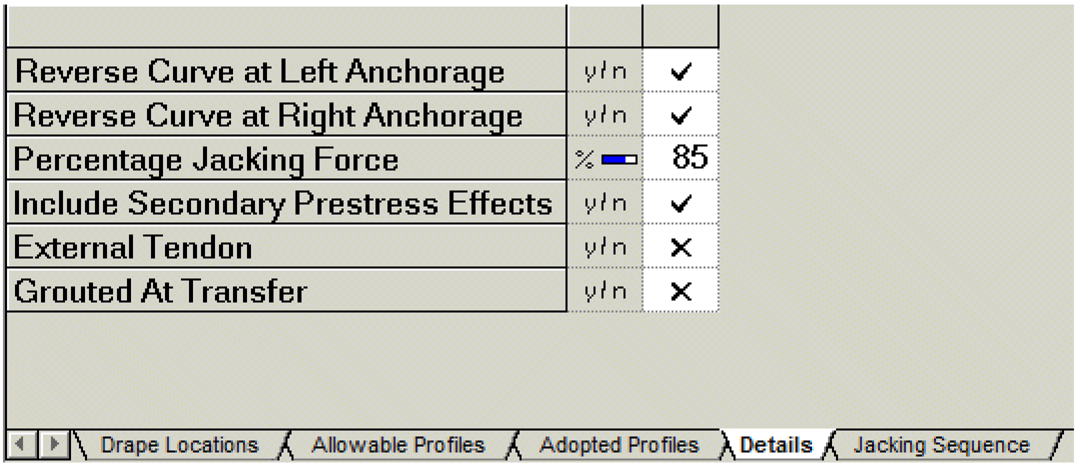
Prestress Force:- The approximate prestress force used in these calculations. Once the input is complete, more accurate prestress loss calculations will be done to determine the actual prestress forces and actions on the member for analysis and design.

Balanced Uniform Load:- The ratio of the moment balanced at the defined profile location for this parabola to the requested balanced moment at this point. If no balanced loads have been requested, the balanced load is assumed to be 1.0 * self weight.

Left Point Load Reaction:- The ratio of the moment balanced at the left harped profile point to the requested balanced moment at this point. If no balanced loads have been requested, the balanced load is assumed to be 1.0 * self weight.

Right point load reaction:- The ratio of the moment balanced at the right harped profile point to the requested balanced moment at this point. If no balanced loads have been requested, the balanced load is assumed to be 1.0 * self weight.

7.2.5.4 Details



Data Definition

Reverse Curve at Left Anchorage

Nominates whether the anchorage at left end of the tendon should be set horizontal with a transition curve to the main profile or set at the angle of the profile of the tendon at the anchorage. In bonded slab designs, it is normally not necessary to set the anchorage horizontal as the angles involved are very small and the anchorage block-out material can normally accommodate any small angle involved. Where larger tendons are involved, reverse curves may be needed or complicated anchorage recesses may need to be built and anchorage reinforcing detailing is often very complicated. If a reverse curve is going to be provided at an anchorage, it is often better to make sure that there is a horizontal straight length from the end of the anchorage for about 500mm to 1000mm so that the tendon actually enters the anchorage in a relatively straight line rather than risking a kink in the tendon and the strands at the entry to the anchorage.

The default condition is ✗except for members with long extensions past the end column.

Reverse Curve at Right Anchorage

Nominates whether the anchorage at right end of the tendon should be set horizontal with a transition curve to the main profile or set at the angle of the profile of the tendon at the anchorage. In bonded slab designs, it is normally not necessary to set the anchorage horizontal as the angles involved are very small and the anchorage block-out material can normally accommodate any small angle involved. Where larger tendons are involved, reverse curves may be needed or complicated anchorage recesses may need to be built and anchorage reinforcing detailing is often very complicated. If a reverse curve is going to be provided at an anchorage, it is often better to make sure that there is a horizontal straight length from the end of the anchorage for about 500mm to 1000mm so that the tendon actually enters the anchorage in a relatively straight line rather than risking a kink in the tendon and the strands at the entry to the anchorage.

The default condition is ✗except for members with long extensions past the end column.

Percentage Jacking Force

The jacking force as a percentage of the breaking force of the strand in the tendon. The maximum limit for this is set in most design standards and it varies for different types of prestressing. RAPT has absolute limits set in the Design Standard file for each design standard. If these are violated a warning will be given but the design will be allowed to proceed. The default values for this data are set in the 6.8.2 Materials->Prestress Steel->tendon type->anchorage size for each anchor type for each steel size.

Include Secondary Prestress Effect

RAPT automatically includes Secondary (Parasitic) Prestress effects in the calculations for all post-tensioned tendons.

Secondary Prestress effects are not included by default for pre-tensioned tendons as normally pretensioned members are stressed in a situation with no continuity, where there are no secondary prestress effects, but the precast members are often then included in an indeterminate frame as composite members. While the resulting composite member is indeterminate, as the prestressing was done while the member was determinate, there are no Secondary Prestress Effects induced by the pre-tensioned tendons.

There are situations where members are precast and stressed using post-tensioned also and then included in an indeterminate frame in a similar manner to pre-tensioned beams. In this case, there should be no Secondary Prestress effects but RAPT's default settings will always include Secondary prestress Effects for post-tensioned tendons.

This option now allows the designer to over-ride RAPT's default treatment of Secondary Prestress effects in all cases. When extra tendon types are added, RAPT sets the data for the new tendon to be the same as for the first tendon type added. The RAPT default will automatically adjust to the default settings when the different prestress type is modified.

External Tendon

For bonded post-tensioned tendons, the designer can now nominate that a tendon is External. If selected, the tendon will be treated as an Unbonded Tendon in design, but will still use the Bonded Tendon material properties such as friction etc. This allows a mix of bonded and unbonded tendons to be included in a design at the same time as often happens in box girder and superT bridges. The design will still be considered to be a bonded prestress design overall and the normal unbonded member minimum reinforcement rules will not be applied. It is up to the designer to ensure

that the resulting design is logically correct in terms of allowable stresses and minimum capacities as RAPT will allow all tendons to be external, effectively resulting in an unbonded member.

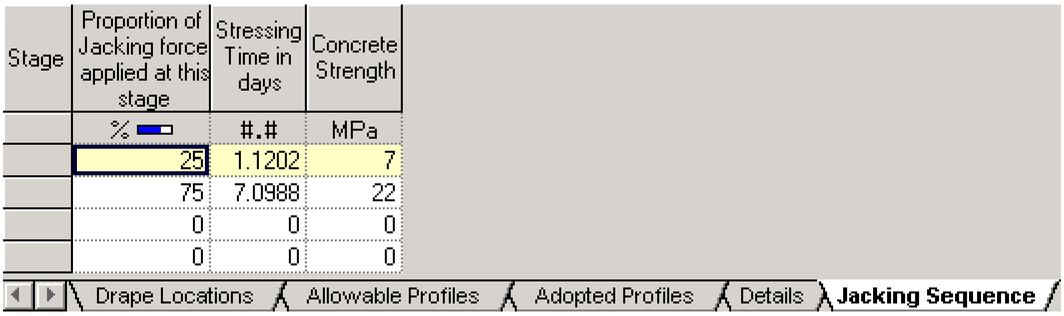
The tendon will be assumed to be external to the concrete section, so there will be no reduction for the area of the ducts from the area of concrete as there would be for normal unbonded tendons in internal ducts.

Grouted at Transfer

Normally bonded post-tensioned tendons will be ungrouted at transfer. However, in some transfer beams where stage stressing is used, at different stages some tendons will be grouted and others ungrouted. It is up to the designer to over-ride the RAPT default for the grouted tendons in this case for each stressing stage (we always recommend that tendons be grouted as soon after stressing as possible to ensure they are properly protected and are acting as bonded tendons for load carrying requirements during construction).

RAPT will, by default, assume that any bonded post-tensioned tendon is ungrouted at transfer. In special cases, the designer can nominate that any particular tendon is grouted. If ungrouted, the duct will be assumed to be a void in the concrete if it is in the compression zone, this reducing the compression area of the concrete and probably increasing the compression stress in the concrete as well as effecting the tension stress. Under deformation, the tendon will be assumed to not change strain as the concrete around it undergoes changes in strain as it is effectively an unbonded tendon at this stage.

7.2.5.5 Jacking Sequence



Stage	Proportion of Jacking force applied at this stage	Stressing Time in days	Concrete Strength
	%	#.#	MPa
	25	1.1202	7
	75	7.0988	22
	0	0	0
	0	0	0

Data Definition

RAPT allows the designer to specify up to four stages of stressing for each tendon profile. These stressing stages are used in the calculation of the prestress losses. A stage could mean that all of the strands are stressed to a percentage of the jacking load or a percentage of the strands are stressed to the full jacking load for each strand.

The highest transfer strength nominated for any of the tendon profiles in the member will be used by RAPT as the concrete strength at transfer for the calculation of transfer effects.

For onion dead ends and nominal plate dead ends it is recommended that full load not be applied to a strand until the concrete strength reaches at least the minimum recommended concrete strength for the anchorage (nominally 22 to 25 MPa). All of the strands in this type of tendon should be partially stressed if stage stressing is required at lesser concrete strengths because the anchorage of each individual strand is dependent on the bond strength of the concrete rather than the overall anchorage being dependent on the bearing strength of the concrete.

For transfer beams, it is recommended that tendons be fully stressed at a stage and grouted rather than each tendon being partially stressed at each stage.

Proportion of Jacking force applied at this stage

The percentage of the total jacking force that is to be applied to each tendon in this tendon profile at this concrete strength and time after pouring the concrete. The total of the values in this column of data must be 100%.

Stressing Time in days

The time in days after the concrete pour when this stage of the stressing will occur. The value in this cell will be defaulted to the time for the concrete to reach the nominated concrete strength in the next cell based on the concrete strength gain data defined for the concrete in the 6.3 Materials->Concrete data. If the designer overrides the calculated figure in this cell RAPT will accept it as the time at which the concrete will attain the concrete strength nominated in the next column.

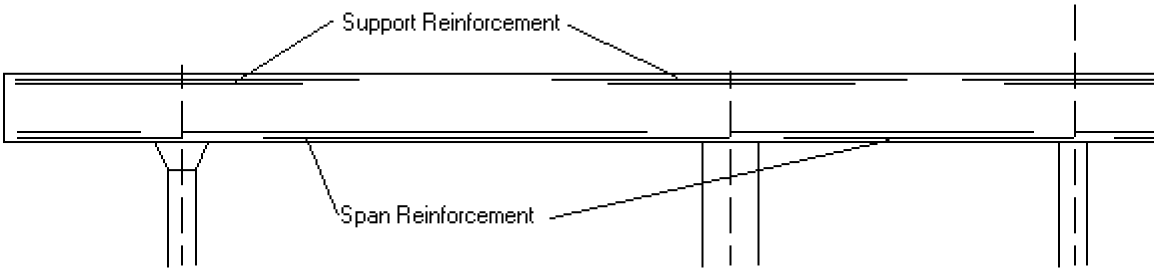
Concrete Strength

The concrete strength at which this stressing stage is completed. When a value is entered in this cell, the time of stressing will be recalculated based on the concrete strength gain data defined for the concrete in the 6.3 Materials->Concrete data.

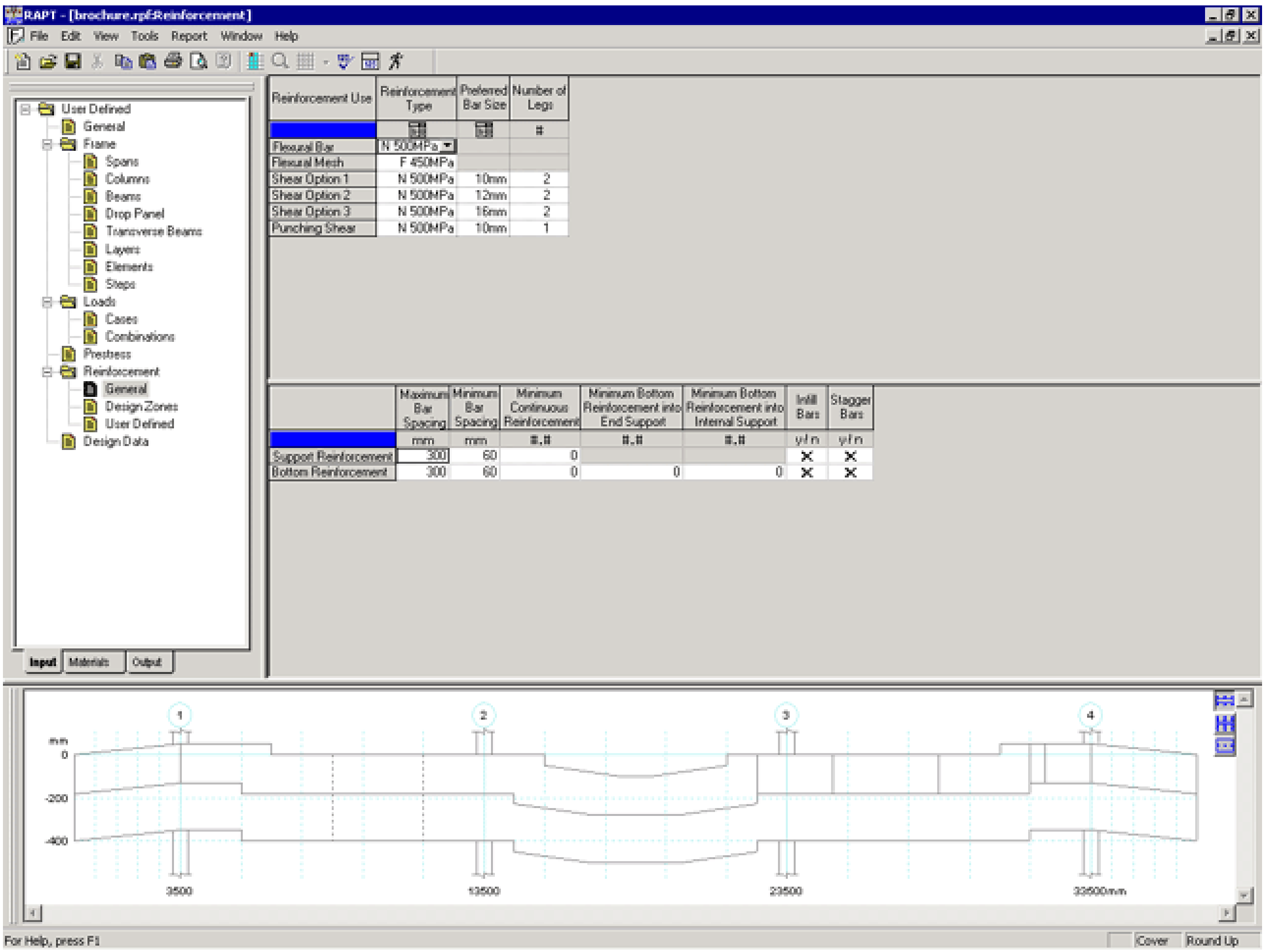
7.2.6 Reinforcement

7.2.6.1 Reinforcing Bar Types

RAPT allows the designer to specify data to define the reinforcement attributes for reinforcement it will add to a member during design (defined under [7.2.6.3 Reinforcement->Design Zones]) and also to define reinforcement that the designer wants to add to the member which is to be taken into account in the design (defined under [7.2.6.4 Reinforcement->User defined]). All user defined reinforcement will be taken into account in the design for flexure, shear and deflections. When defining reinforcement, the designer must define



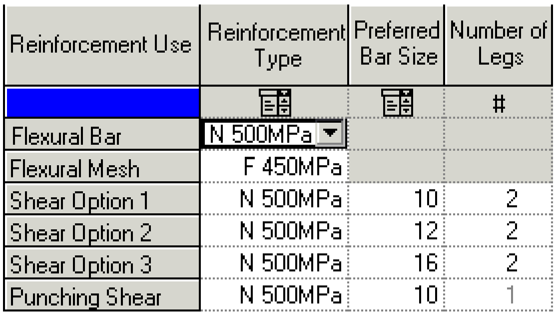
1. bar depth
2. bar type
3. bar size
4. number or spacing of bars
5. bar location:- When defining a bar, RAPT requires end locations. Also the designer can nominate that some of the bars will stop short of the end. The number stopping will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagram above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations. This called Staggering Bars.
6. end development:- A percentage development to allow for the presence of hooks, cogs or laps to other bars. e.g. a standard hook or cog would normally provide 50% tension development but 0% compression development while a full tension lap would provide 100% tension and compression development.



This input window contains 2 screens of data which define

1. Reinforcing Bar Type:- Defines the reinforcing bar types available to be used for flexural reinforcement designed by RAPT and for beam and punching shear reinforcement. The Flexural Bar type defined here is also used as the default bar type for user defined reinforcement.
2. 7.2.6.2 Reinforcing Bar Design Details:- Defines a set of limits used in the detailing of the flexural reinforcement.

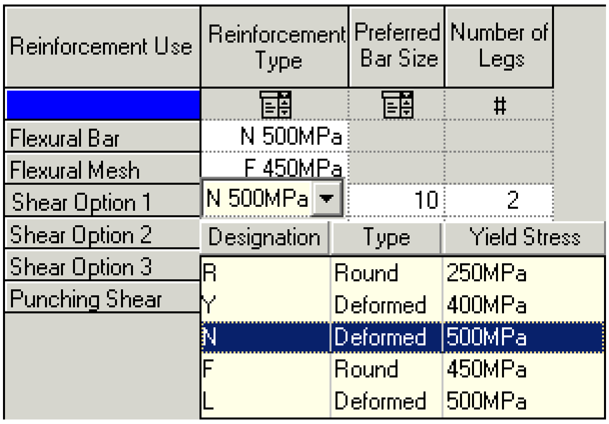
Data Definition



Reinforcement Use	Reinforcement Type	Preferred Bar Size	Number of Legs
Flexural Bar	N 500MPa		
Flexural Mesh	F 450MPa		
Shear Option 1	N 500MPa	10	2
Shear Option 2	N 500MPa	12	2
Shear Option 3	N 500MPa	16	2
Punching Shear	N 500MPa	10	1

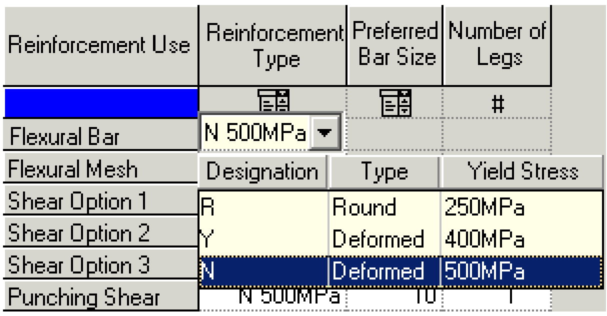
Flexural Bar:- The reinforcing bar type that will be used by RAPT for flexural design in the top and or the bottom of the member as defined in the 7.2.6.3 Design Zones input screens.

Flexural Mesh:- The reinforcing mesh type that will be used by RAPT for flexural design in the top and or the bottom of the member as defined in the 7.2.6.3 Design Zones input screens.



Designation	Type	Yield Stress
R	Round	250MPa
Y	Deformed	400MPa
N	Deformed	500MPa
F	Round	450MPa
L	Deformed	500MPa

Shear Option 1:- For flexural shear design RAPT allows the designer to nominate three sets of reinforcement type, bar size and number of legs for the tie sets. The designer may prefer to nominate a single bar type and obtain results for a series of different bar sizes or use a constant bar type and bar size and obtain results for various numbers of legs in the shear ties. RAPT will provide shear results for each shear reinforcement option at each design location.



Designation	Type	Yield Stress
R	Round	250MPa
Y	Deformed	400MPa
N	Deformed	500MPa

Shear Option 2:- Second flexural shear reinforcement type.

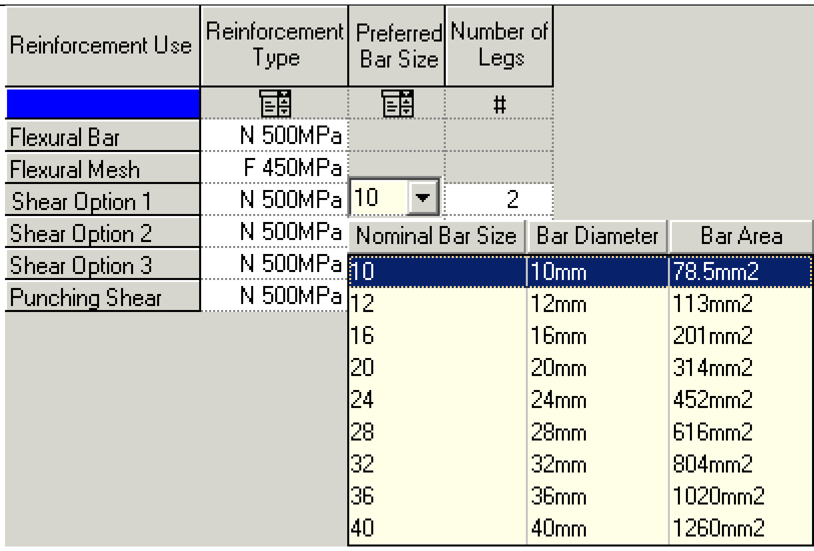
Shear Option 3:- Third flexural shear reinforcement type.

Punching Shear:- The reinforcing bar type and size that will be used for punching shear design.

Reinforcement Type:- The reinforcement type to be used for this purpose. This is selected from the list of defined reinforcement types for the relevant reinforcing steel type (bar and/or mesh) in the 6.4 Materials Data. The options available will be presented in a Drop Down Data List as shown above. The Shear list shows a combined list of reinforcing bar and reinforcing mesh options while the Flexural list shows reinforcing bar or mesh options only depending on the row of data being investigated. The list will show the Bar Designation, Bar Type and Yield Strength. If a different reinforcement type is selected, RAPT will check the bar size to find the bar size in the new type that has the closest area to the previous bar size and will substitute this new size.

The default settings are those set in the selected materials file under 6.2.2 Defaults->Reinforcement.

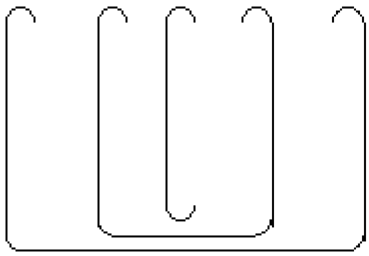
Preferred Bar Size:- For the shear reinforcement options, the designer must define a reinforcing bar size or mesh type. These are selected from a list of the sizes available for the reinforcement type selected as shown below. The information presented is Nominal Bar Size, Bar Diameter and bar area. The preferred bar sizes for flexural reinforcement are set in the 7.2.6.3 Design Zones input screens.



Reinforcement Use	Reinforcement Type	Preferred Bar Size	Number of Legs
Flexural Bar	N 500MPa		
Flexural Mesh	F 450MPa		
Shear Option 1	N 500MPa	10	2
Shear Option 2	N 500MPa		
Shear Option 3	N 500MPa		
Punching Shear	N 500MPa		

Nominal Bar Size	Bar Diameter	Bar Area
10	10mm	78.5mm2
12	12mm	113mm2
16	16mm	201mm2
20	20mm	314mm2
24	24mm	452mm2
28	28mm	616mm2
32	32mm	804mm2
36	36mm	1020mm2
40	40mm	1260mm2

Number of Legs:- For shear reinforcement options, the number of legs of shear reinforcement in each set of ties is nominated. RAPT will use this number of legs for the detailing of shear reinforcement at every point requiring reinforcement. A spin button is provided to scroll through the number of legs if desired.



= 5 LEGS

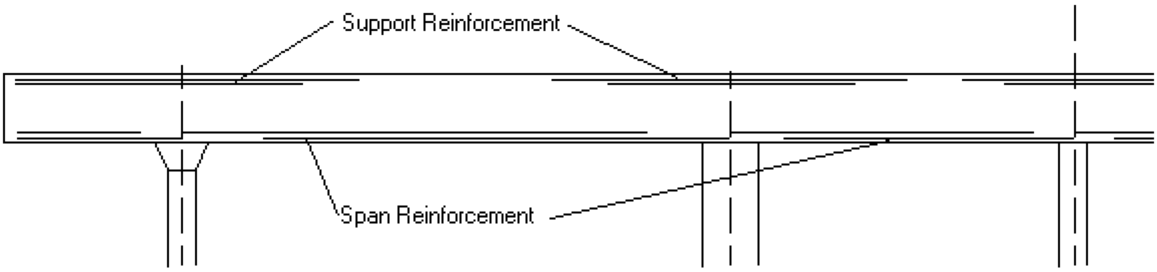
For punching shear to some codes (eg AS3600) the shear reinforcement consists of torsion reinforcement and therefore consists of only 1 leg for a tie. RAPT will not accept any other input in these cases.

7.2.6.2 Reinforcing Bar Design Details

	Maximum Bar Spacing	Minimum Bar Spacing	Minimum Continuous Reinforcement	Minimum Span Reinforcement into End Support	Minimum Span Reinforcement into Internal Support	Infill Bars	Stagger Bars
	mm	mm	#.#	#.#	#.#	y/n	y/n
Support Reinforcement	300	60	0			✕	✕
Span Reinforcement	300	60		0	0	✕	✕

Data Definition

Separate reinforcing detailing parameters can be set by the designer for both top and bottom reinforcement. RAPT asks for the information for Support Reinforcement and Span Reinforcement as described below.



Support Reinforcement:- The reinforcement on the face of the member which has a peak moment concentrated at a column and reducing away from the column. The main reinforcing bars normally are centred on the column and extend into the spans on either side of the column. For a member with downward loading, it would be the top face.

Span Reinforcement:- The reinforcement on the face of the member which has a maximum moment somewhere between the columns in a span and reducing towards the columns. The main reinforcing bars normally are centred on the middle of the span and confined to that span. For a member with downward loading, it would be the bottom face.

Maximum Bar Spacing:- The maximum bar spacing to use when determining bar sizes and detailing the reinforcement for the member. The default value is defined in 5.4.1 Design Standard Defaults->Reinforcement->Limits. If a crack control requirement requires a lesser bar spacing then it will control the maximum value. The design standard defined limits in 5.4.1 Design Standard Defaults->Reinforcement->Limits will also be applied when determining the maximum spacing to use.

Minimum Bar Spacing:- The minimum bar spacing to be used when determining bar sizes and detailing the reinforcement for the member. The default value is defined in 5.4.1 Design Standard Defaults->Reinforcement->Limits. The design standard defined limits in 5.4.1 Design Standard Defaults->Reinforcement->Limits will also be applied when determining the minimum spacing to use.

Minimum Continuous Reinforcement:- The minimum area of continuous reinforcement in the face of the member in a span as a fraction of the maximum area of reinforcement in that face in that span.

Minimum Span Reinforcement into End Support:- The minimum area of span face reinforcement extending into the support at an end support as a fraction of the maximum area of span face reinforcement in that span. This option is only used at end columns where no cantilever exists.

Minimum Span Reinforcement into Internal Support:- The minimum area of span face reinforcement extending into the support at internal supports as a fraction of the maximum area of span face reinforcement in that span.

Infill Bars:- Normally, the support face reinforcement is not continuous over the full length of the span. In beams it is often practical to add extra nominal bars to fill this area and provide supports for shear ties and cross reinforcement. RAPT allows the designer to decide whether these infill bars are to be added at either the support face or the span face in any type of member. The default settings are read from 5.4.1 Design Standard Defaults->Reinforcement->Limits. This will be used in the detailing of the reinforcement to determine bar patterns and cut-off locations.

Stagger Bars:- When curtailing reinforcing bars in areas away from the peak reinforcement location, some designers try to economize on the area of reinforcing steel by curtailing a bar type at multiple locations as the required area of reinforcement reduces. Others prefer to terminate all bars outside the relevant tension zone. If the designer wishes to stagger the end curtailment locations of reinforcing bars and have some terminate in a tension zone then setting this option to Yes for staggering of bars will achieve this. The number of bars stopping will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagram above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations. This called Staggering Bars.

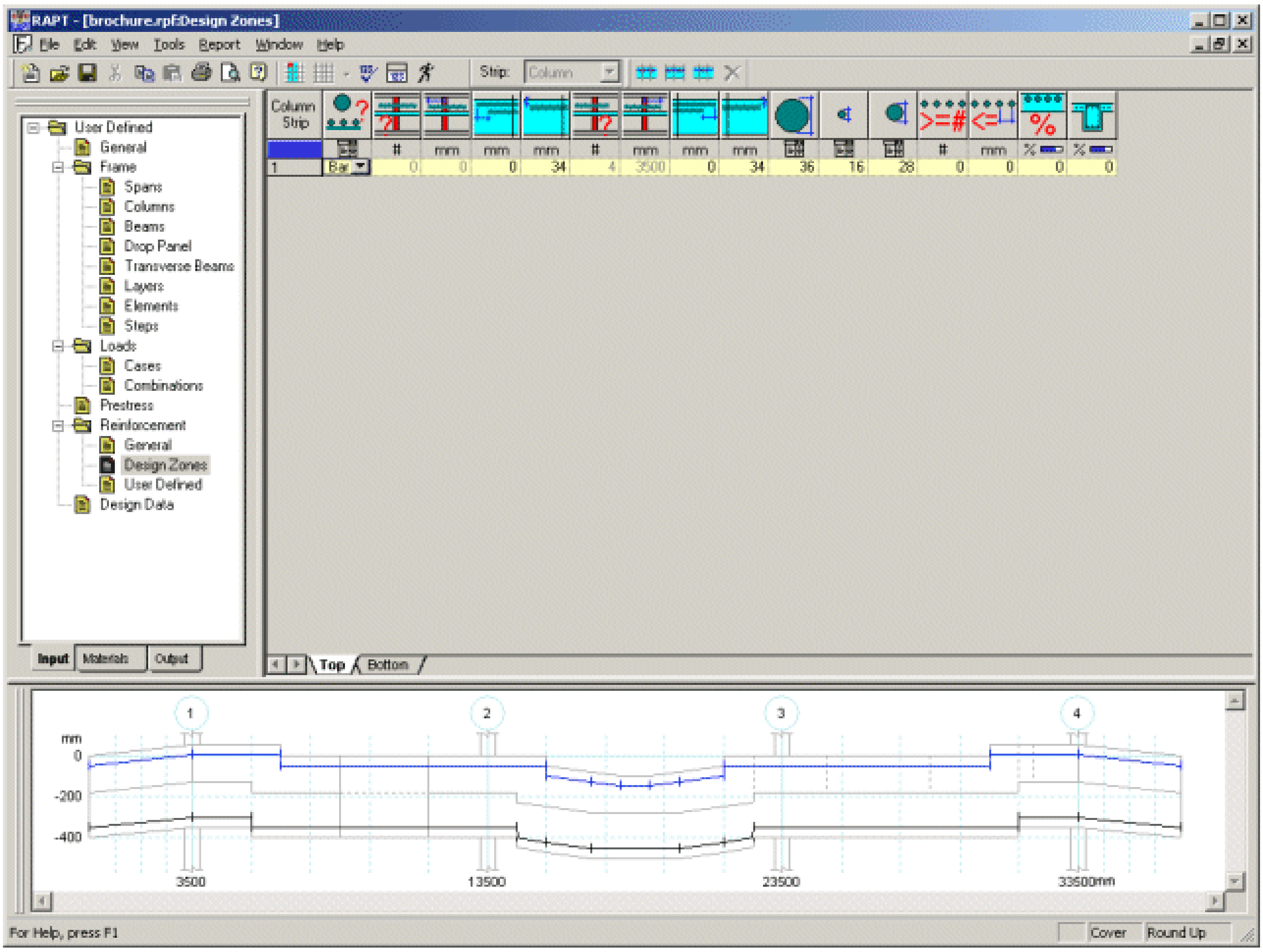
RAPT will then ensure that adequate reinforcement is provided at all locations and that bars are only terminated past the last point where they are required when it is detailing the reinforcement. RAPT will not use default curtailment locations from design standards for this, it will only use the calculation methods they provide.

In calculating termination locations for bars, RAPT will attempt to ensure that each bar will develop adequate capacity at each design location along it's length to provide the strength required at that point. As well, RAPT will apply the relevant clauses in each code

1. AS3600:- 8.1.8 (except 8.1.8.6) , 8.1.9 and 9.1.3.1, where relevant
2. ACI318:- 12.10
3. Eurocode:- 2 5.4
4. BS8110, SABS 0100, CP65:- 3.12.9 and 3.12.10
5. CP2004:- 9.2.1
6. IS456/IS1343:- 29.3

Some design codes require special shear reinforcement detailing at locations where tension bars are terminated in a tension zone. The designer should be aware of these rules and apply them where necessary. RAPT will not adjust shear reinforcement detailing for this automatically.

7.2.6.3 Design Zones



Design zones define the cover to the design flexural reinforcement layers and the preferred bar size and bar size limits in different areas of a member. They allow the designer to specify different reinforcement cover in different areas (zones) of the member. They also allow the designer to define minimum areas of reinforcement within each zone.

The design zones are divided into two areas controlled from the tabs at the bottom of the window,

1. Top Reinforcement Zones
2. Bottom Reinforcement Zones

Also, different design zones are required for the column strip and middle strip for two-way strip designs. The current strip is selected from the Strip Toolbar.

By default, there is one design zone in each face of the member extending full length of the face of the member. The cover used is the cover defined in the 7.2.2 General Screen for the member face. The maximum bar size, minimum bar size and preferred bar size default to the values in 6.2.2 Materials->Defaults->Reinforcement for the member type being designed.

The locations of the left end of the leftmost zone and the right end of the rightmost zone cannot be edited and are always the left and right ends of the member respectively. Any zones added by the designer will be fitted between these ends and will be listed in order from left to right.

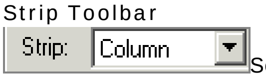
The concrete surface from which the cover is measured is within 50mm of the centreline of the member. The designer should check the graphic view to ensure that RAPT has selected the surface that the designer wants as the reference surface. Where rebates have been cut in the edges of beams or slabs or where the top surface of the concrete is set down locally from the general surface, the designer may have to adjust the cover in those areas to indicate the preferred design flexural reinforcement level.

Graphic Interaction

To select a zone, click on it with the left mouse button. The selected zone will be shown in blue. The window in which the selected design zone is defined will be selected and the data row controlling the selected zone will become the current row.

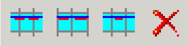
Design Zone Toolbars

Strip Toolbar

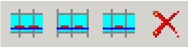


Select the strip to view/modify design zone information. This is only available for 7.2.2 two-way strip designs. The toolbar is in background only in other cases.

Top Design Zones Toolbar

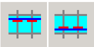


Bottom Design Zones Toolbar

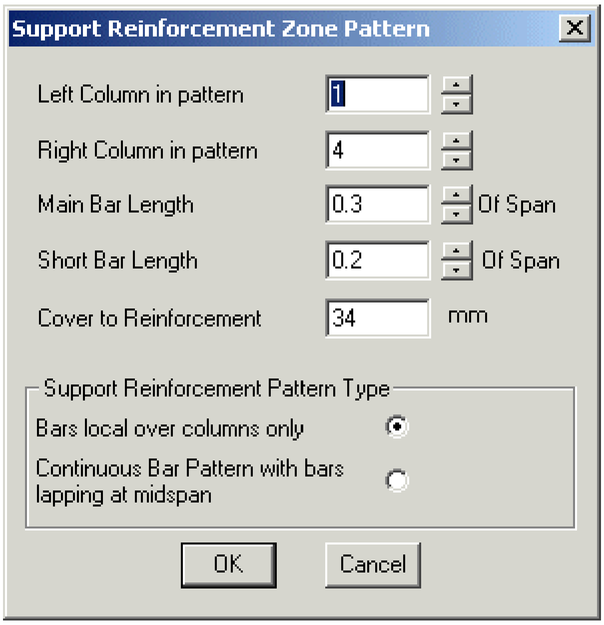


When adding zones, the current row in the data grid is irrelevant (this is different to the operation in all other data grids). RAPT will inset the added zones where they fit in location order from the left end.

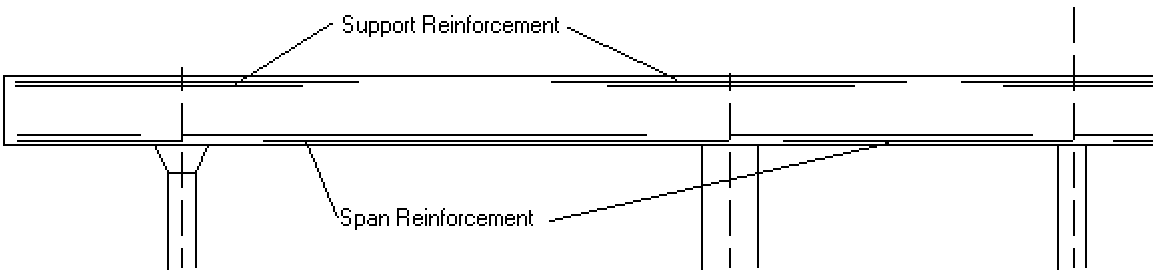
When design zones are added in one of the Patterns defined below (not single zone), the input type is remembered along with details regarding the ratio of span length to ends of the zone. When span lengths change or spans are added or deleted, RAPT will use this information to attempt to modify the zone details to suit the new span arrangements. The designer should always check that RAPT's modifications in this regard are as desired.


Add support zone pattern

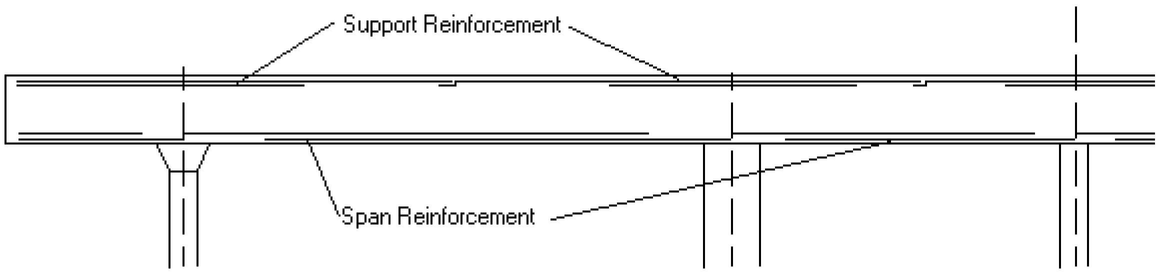
Allows the designer to nominate a group of design zones in the pattern of support reinforcement, i.e. reinforcement locally positioned over each column and extending into the spans on either side of the column. This zone pattern can be placed on either face of the member, top or bottom, and is available from both toolbars. The pattern of the design zones is shown in the diagrams below as Support Reinforcement. The designer is presented with the following dialog in which the following data is nominated

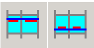


1. Left Column in Pattern:- The [7.2.3.2 Column Grid Reference] of the left column in the group of columns over which zones are to be placed
2. Right Column in Pattern:- The [7.2.3.2 Column Grid Reference] of the right column in the group of columns over which zones are to be placed
   In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.
3. Main Bar Length:- The fraction of the span over which the zone is to extend either side of the column. The span length used is the average of the span lengths either side of a column.
4. Short Bar Length:- The fraction of the span over which the staggered bars in the zone are to extend. The span length used is the longer of the span lengths either side of a column.
5. Cover to Reinforcement:- Cover to the surface of the reinforcement from the relevant face. Once the zones have been added the covers can be amended at each end of each zone.
6. Support Reinforcement Pattern Type:- There are two way the zones can be defined. These are
   1. Local over Columns:- New zones are created over the columns with the old zone properties still applying in the areas between the new zones. The new zone areas are shown in the diagram below for support reinforcement.

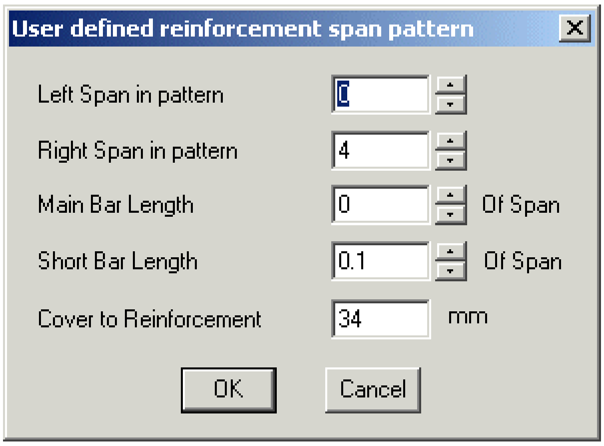


2. Continuous Bar Pattern with Bars Lapping at Mid-span:- New zones are created over the columns and joining at mid-span. The new zone areas are shown in the diagram below for support reinforcement.

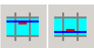


Add span zone pattern

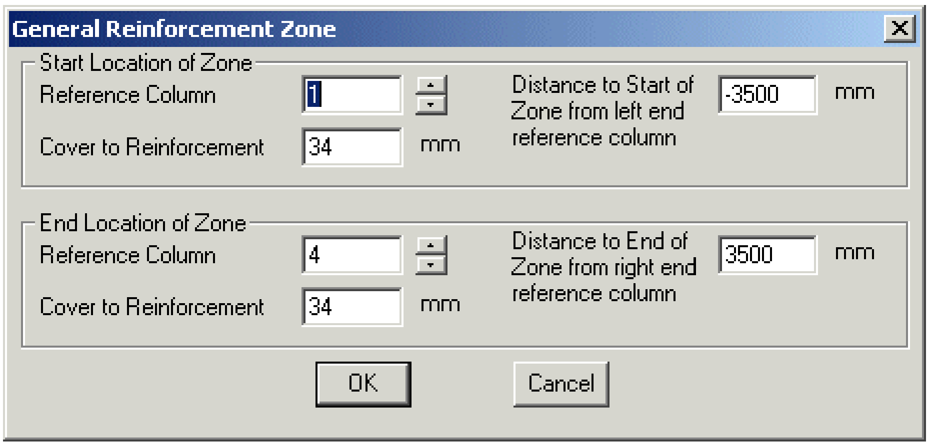
Allows the designer to nominate a group of design zones in the pattern of span reinforcement, i.e. reinforcement positioned between the centrelines of the columns at each end of the span. This zone pattern can be placed on either face of the member, top or bottom, and is available from both toolbars. The pattern of the zones is shown in the diagrams above as Span Reinforcement. The designer is presented with the following dialog in which the following data is nominated



1. Left Span in Pattern:- the left pan in the group of spans over which a zones are to be placed.
2. Right Span in Pattern:- the right span in the group of spans over which zones are to be placed.
3. Main Bar Length:- The distance from the column centreline to the start of the zone at each end of each span as a fraction of the span.
4. Short Bar Length:- The distance from the column centreline to the start of the staggered zones at each end of each span as a fraction of the span.
5. Cover to Reinforcement:- Cover to the surface of the reinforcement from the relevant face. Once the zones have been added the covers can be amended at each end of each zone.

Add single zone

Allows the designer to nominate a single random zone anywhere in the length of the member. The designer is presented with the following dialog in which the following data is nominated



1. Start Location of Zone

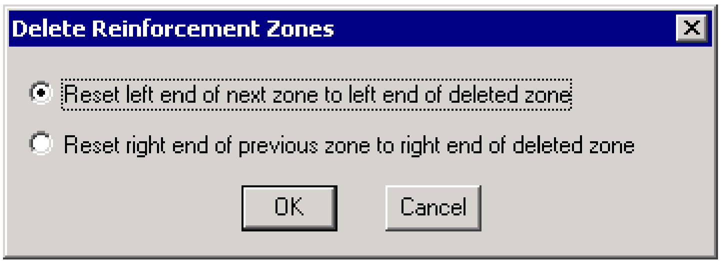
   1. The 7.2.3.2 Column Grid Reference of the left column from which the start location of the zone is measured.

    2. Distance to Start of Zone from Left End Reference Column
    3. Cover to reinforcement:- Cover to the surface of the reinforcement from the relevant face at the left end of the zone.
2. End Location of Zone
    1. The 7.2.3.2 Column Grid Reference of the left column from which the end location of the zone is measured.
    2. Distance to End of Zone from Right End Reference Column
    3. Cover to reinforcement:- Cover to the surface of the reinforcement from the relevant face at the right end of the zone.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

Delete top zone

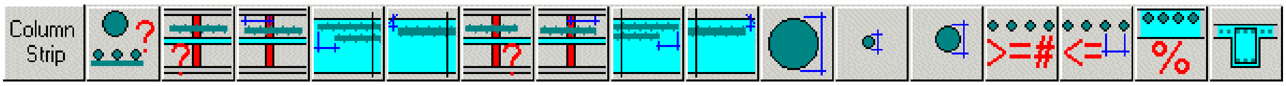
One design zone must remain at each design face under all circumstances so RAPT will not allow you to select and delete all design zones. If all zones are selected, RAPT will leave the leftmost zone and extend it full length of the member. Otherwise, any group of design zones can be selected and deleted using the selection logic in 4.4.3 Cell Selection. The designer is presented with the following dialog to indicate which end of the remaining zones is reset.



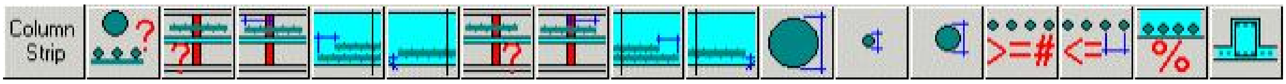
Data Definition

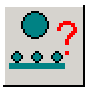
The data required for both the top and bottom reinforcement design zones is the same so it will only be described once.

Top Zone



Bottom Zone

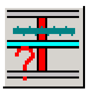


Reinforcement type

The reinforcement type to be used in this design zone. The options are

1. Bar
2. Mesh

The type of bar of mesh used is the one defined for Flexural reinforcement in the 7.2.6.1 Reinforcement->General window. All zones at a face must use the same reinforcement type, so, if this value is changed for a zone it will automatically be changed for all zones at that face. If a different reinforcement type is selected, RAPT will check the bar sizes to find the bar size in the new type that has the closest area to the previous bar size and will substitute this new size.

Left End Reference Column

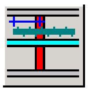
The 7.2.3.2 Column Grid Reference from which the left end of the reinforcement zone is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the reinforcement zone. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be

converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

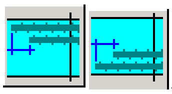
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a reinforcement zone in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a reinforcement zone.

Obviously the left end of a reinforcement zone must be to the left of the right end so make sure when modifying reinforcement zone ends that the order of modification will ensure this.

Distance to left end of bar from Reference Column

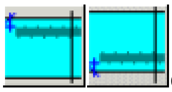
The distance from the left reference column to the left end of the reinforcement zone. If the distance entered moves the reinforcement zone into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the reinforcement zone.

Obviously the left end of a reinforcement zone must be to the left of the right end so make sure when modifying reinforcement zone ends that the order of modification will ensure this.

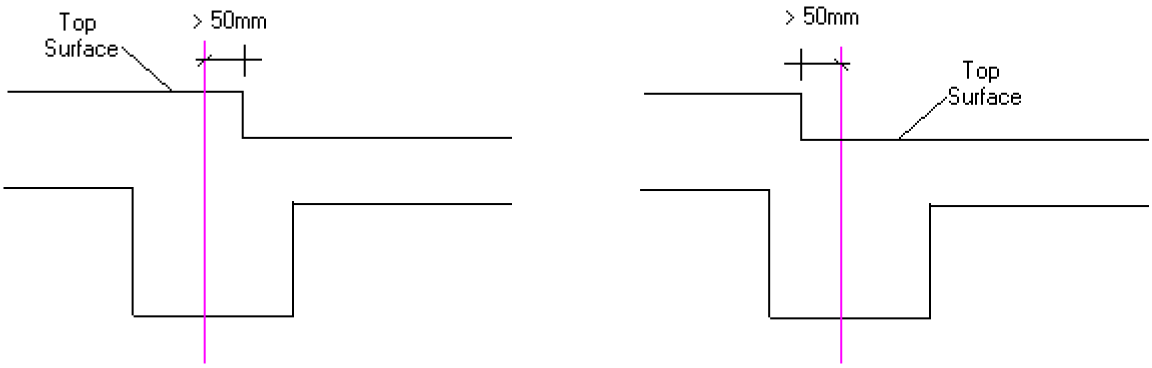
Alternate bar stagger length at left end:-

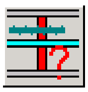
The distance from the left end of the design zone to the stagger location if bars are to be staggered. This value is 0 if there is no stagger. The net bar length of the short bar after allowing for staggers must be greater than 10% of the main bar length. The stagger will be used if the designer specifies a minimum quantity of reinforcement for a zone.

The number of bars stopping at the stagger location will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagrams above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations.

Cover From Concrete Surface to the Surface of the Bar at left end

The cover to the left end of the bar is measured to the top/bottom surface of the bar from the relevant concrete surface, Top or Bottom. The actual surface level used is based on the depth of the deepest surface that extends at least 50mm each side of the centreline of the supports (for external panels the width is only checked on the continuous side), see diagrams below. The bar size used to determine the actual bar depth is the Preferred Bar Size.



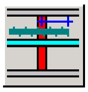
Right End Reference Column Number

The 7.2.3.2 Column Grid Reference from which the right end of the reinforcement zone is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the reinforcement zone. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

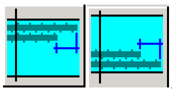
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a reinforcement zone in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a reinforcement zone.

Obviously the right end of a reinforcement zone must be to the left of the right end so make sure when modifying reinforcement zone ends that the order of modification will ensure this.

Distance to right end of bar from Reference Column

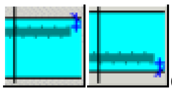
The distance from the right reference column to the right end of the reinforcement zone. If the distance entered moves the reinforcement zone into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the reinforcement zone.

Obviously the left end of a reinforcement zone must be to the left of the right end so make sure when modifying reinforcement zone ends that the order of modification will ensure this.

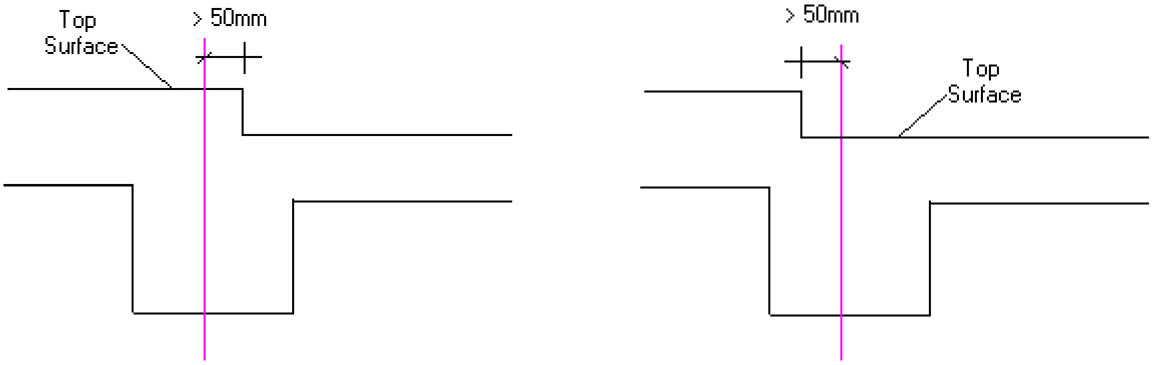
Alternate bar stagger length at right end

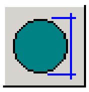
The distance from the right end of the design zone to the stagger location if bars are to be staggered. This value is 0 if there is no stagger. The net bar length of the short bar after allowing for staggers must be greater than 10% of the main bar length. The stagger will be used if the designer specifies a minimum quantity of reinforcement for a zone.

The number of bars stopping at the stagger location will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagrams above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations.

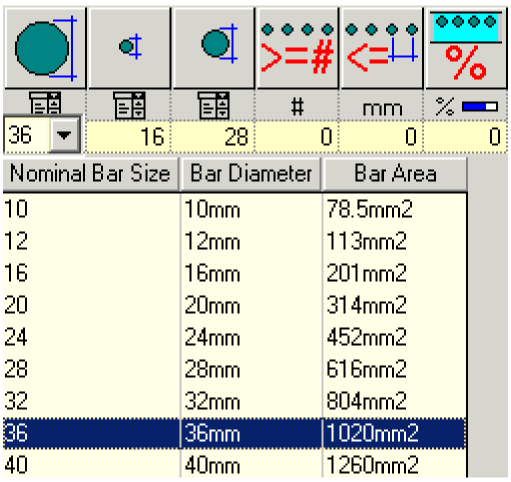
Cover From Concrete Surface to Surface of the Bar at right end

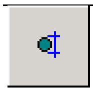
The cover to the right end of the bar is measured to the top/bottom surface of the bar from the relevant concrete surface, Top or Bottom. The actual surface level used is based on the depth of the deepest surface that extends at least 50mm each side of the centreline of the supports (for external panels the width is only checked on the continuous side), see diagrams below. The bar size used to determine the actual bar depth is the Preferred Bar Size.

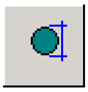


Maximum Bar Size

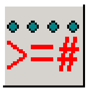
The maximum bar size to be used by RAPT in detailing the reinforcement in this zone. The bar size can be selected from a drop down list of available sizes for the reinforcement type and bar type selected as shown below. The list of bar sizes is defined in the 6.4 Materials Data->Reinforcement Bar/Mesh and can be modified there. Refer to 7.3.11 Reinforcement Detailing for discussion on how this data is used.

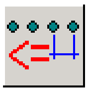


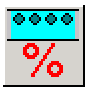
Minimum Bar Size
The maximum bar size to be used by RAPT in detailing the reinforcement in this zone. The bar size can be selected from a drop down list of available sizes for the reinforcement type and bar type selected as shown above. The list of bar sizes is defined in the 6.4 Materials Data->Reinforcement Bar/Mesh and can be modified there. Refer to 7.3.11 Reinforcement Detailing for discussion on how this data is used.

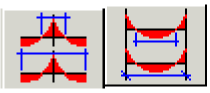
Preferred bar size
The preferred bar size to be used by RAPT in detailing the reinforcement in this zone. The bar size can be selected from a drop down list of available sizes for the reinforcement type and bar type selected as shown above. The list of bar sizes is defined in the 6.4 Materials Data->Reinforcement Bar/Mesh and can be modified there. RAPT will attempt to use this bar size in the detailing of the reinforcement if it fits within the spacing requirements of the design. Refer to 7.3.11 Reinforcement Detailing for discussion on how this data is used.

RAPT allows the designer to nominate a minimum amount of reinforcement that is to be added at design points in this zone (dependent on bar stagger above). The following 4 data items allow the designer to specify this in several ways.

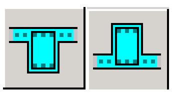
Minimum Number of Bars
RAPT will automatically add an area of reinforcement equivalent to this number of bars at all design points in this zone except as limited by the designers selection in "Minimum Reinforcement Placed at" below and as affected by bar staggers defined above for reinforcing bars only. This option is only available if Bar has been selected as the reinforcing type above.

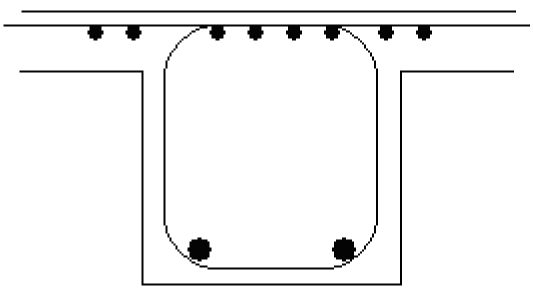
Maximum Spacing of bars
RAPT will automatically add an area of reinforcement equivalent to this number of bars calculated from this spacing based on the effective width at the bar level at each design section within this reinforcement zone except as limited by the designers selection in "Minimum Reinforcement Placed at" below and as affected by bar staggers defined above for reinforcing bars only. This option is only available if Bar has been selected as the reinforcing type above.

Minimum Reinforcement area as % of concrete area
RAPT will automatically add an area of reinforcement equivalent to this number of bars calculated from the area of steel based on this percentage of the total area of the effective cross-section at each design section within this reinforcement zone except as limited by the designers selection in "Minimum Reinforcement Placed at" below and as affected by bar staggers defined above for reinforcing bars only. This option is available for either Bar or mesh as the reinforcing type.

Minimum Reinforcement placed at
The minimum reinforcement requirement defined above can be added in 2 ways

1. Tension Points:- If there is a moment putting this surface into tension under ultimate strength conditions, RAPT will add the minimum reinforcement.
2. All Points:- RAPT will add the reinforcement to this face no matter what the applied moments or stress conditions.

Percentage Reinforcement in Flange
The percentage of the reinforcement calculated at this location that will be assumed to be placed in the flange of a beam (outside the shear cage) for beam shear calculations. Reinforcement in the flange of a beam is ignored in beam shear calculations thus reducing the shear capacity.



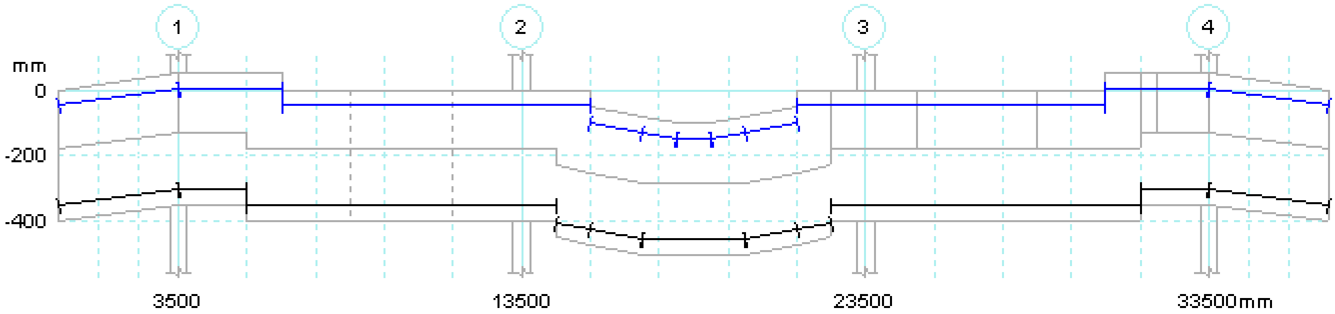
Graphical View

The graphical output consists of separate diagrams for each design strip.

Each reinforcing design zone is shown in elevation. If the surface level varies over a zone, the line will step also. Ends of each portion of a design zone line between steps and at the join with the next zone are indicated by a short vertical line. Stagger end locations are indicated with a diagonal line sloping towards the relevant bar end from the stagger end location on the bar.

To select a zone, click on it with the left mouse button. The selected zone will be shown in blue. The window in which the selected design zone is defined will be selected and the data row controlling the selected zone will become the current row.

Double-Click on a bar will move the focus into Zoom mode equivalent to pressing  (see below) and the reinforcing bars will then be shown with a diameter drawn to scale.



Graphics Toolbar 

A special graphics toolbar is provided to assist with viewing the data. This toolbar will only be available when program focus is in the Graphics Window. The functionality of this toolbar for reinforcement data is slightly different to the general functionality. The buttons available are

 Zoom (Ctrl + Z). This button will toggle between full screen mode and bar zoom mode for the graphics in a window. In design zone zoom mode, the spans in which the current design zone is placed will be shown scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window. In this mode, the bars are drawn to scale with their correct diameter of the preferred bar size. When not in zoom mode, the design zones are drawn as a 1 pixel wide line.

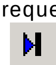
 Move to next Reinforcing Bar (Ctrl + Right Arrow). In Full Screen Mode the next reinforcing design zone in the text list will be shown in a blue colour (see graphical view above). In bar Zoom mode, the spans containing the next reinforcing design zone in the text list will move to the centre of the Window and the selected reinforcing design zone will be shown in a blue colour. The data will view will show the selected zone.

 Not Available.

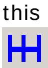
 Not Available.

 Move to previous Reinforcing Bar (Ctrl + Left Arrow). In Full Screen Mode the previous reinforcing design zone in the text list will be shown in a blue colour (see graphical view above). In Bar Zoom mode, the spans containing the previous reinforcing design zone in the text list will move to the centre of the Window and the selected reinforcing design zone will be shown in a blue colour. The data will view will show the selected zone.

 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

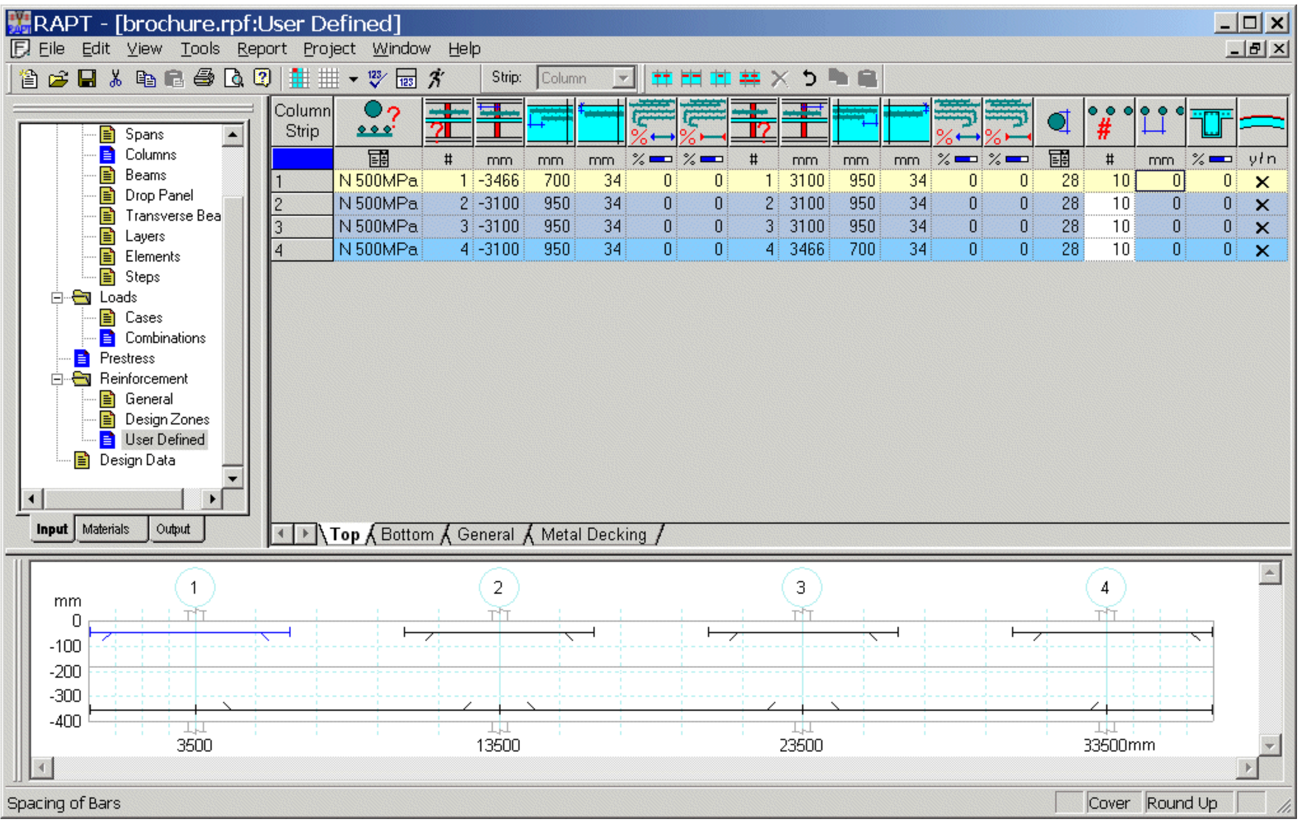
Then user can then move around the graph using the Scroll Bars or the movement keys.  or  buttons will still move the selection to the next or previous zone and will return the Window to Full Screen Mode.

Clicking this  button again or on the buttons will return the Window to Full Screen Mode.

Clicking  will change the mode to Bar Zoom Mode with the reinforcing bar selected in the Select Zoom mode.

Not available. Selecting a zone in the graphics automatically moves the data to the row controlling that design zone so there is no need for a dialog.

7.2.6.4 User Defined Reinforcement

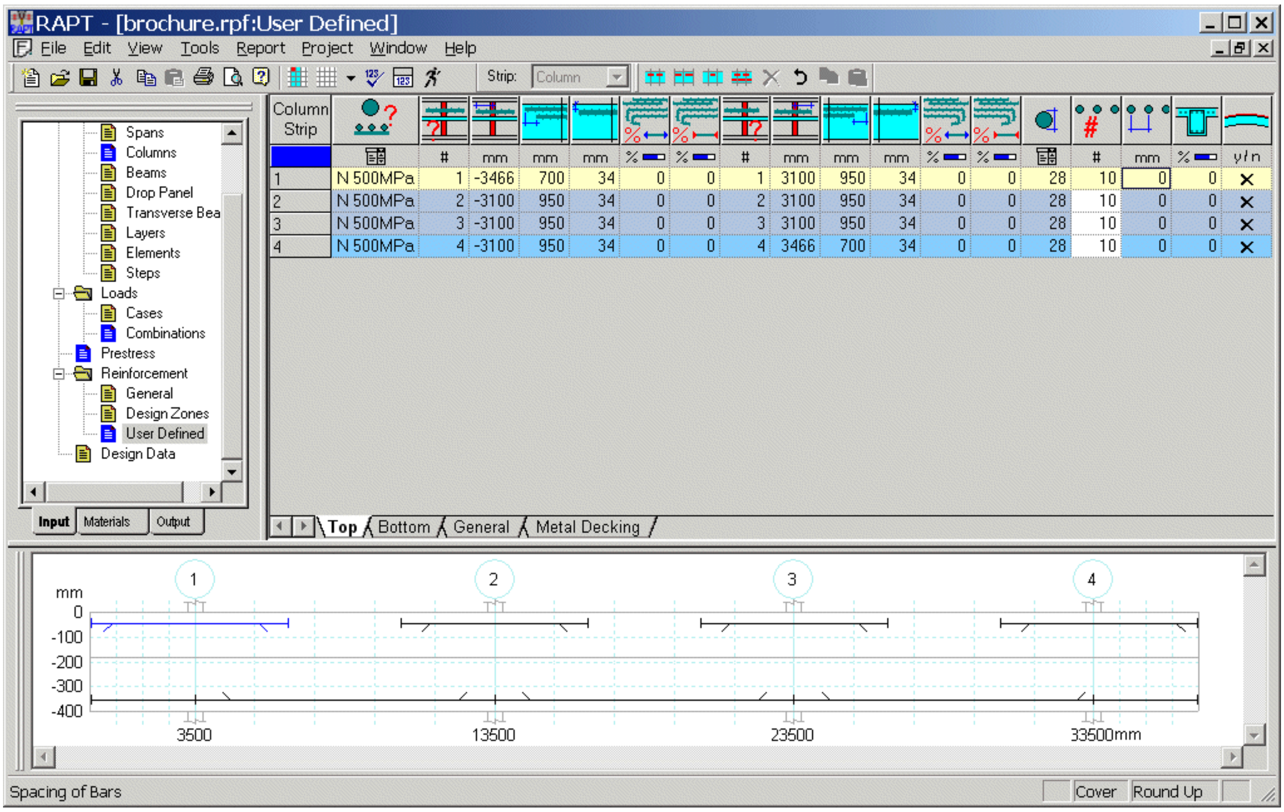


Input help for User Defined reinforcement is divided into 2 sections

7.2.6.4.1 7.2.6.4.1 Reinforcing Bars, Mesh and Fibre Reinforced Polymers

7.2.6.4.2 7.2.6.4.2 Metal Decking

7.2.6.4.1 Reinforcing Bars, Mesh and Fibre Reinforced Polymers

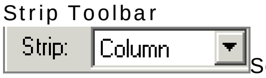


Graphic Interaction

To select a reinforcing bar, click on a point along its length (within 4 pixels above or below the reinforcing bar) with the left mouse button. The selected reinforcing bar will be shown in blue. The window in which the selected design reinforcing bar is defined will be selected and the data row controlling the selected reinforcing bar will become the current row. If more than one reinforcing bar is found at the selected location, RAPT will show the first selected reinforcing bar after the currently selected reinforcing bar. Continuing to click at the same location (with a suitable pause to avoid double clicking) will continually select the next reinforcing bar in the selected list and assign that reinforcing bar as the current reinforcing bar.

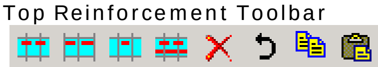
User Defined Reinforcement Toolbar

Strip Toolbar

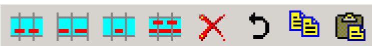


Select the strip to view/modify design reinforcing bar information. This is only available for 7.2.2 two-way strip designs. The toolbar is in background only in other cases.

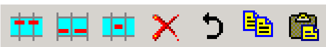
Top Reinforcement Toolbar



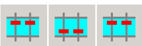
Bottom Reinforcement Toolbar



General Reinforcement Toolbar

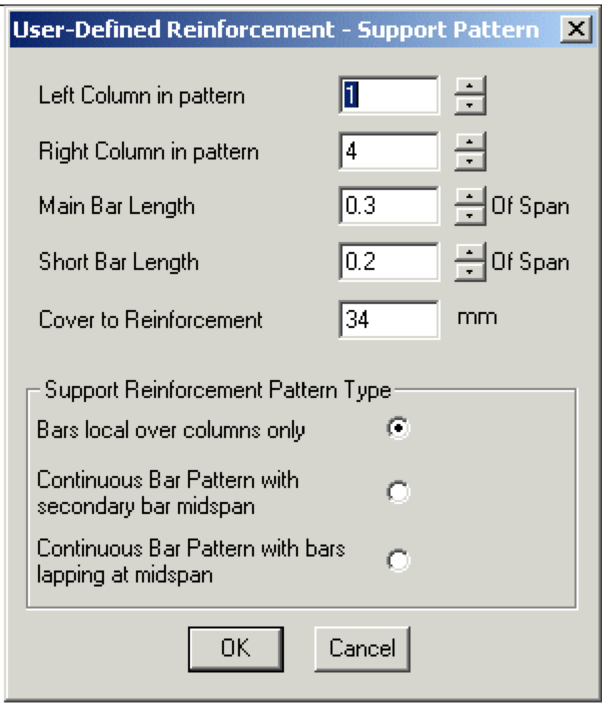


When reinforcing bars are added in one of the Patterns defined below (not single bar), the input type is remembered along with details regarding the ratio of span length to ends of the bars. When span lengths change or spans are added or deleted, RAPT will use this information to attempt to modify the bar details to suit the new span arrangements. The designer should always check that RAPT's modifications in this regard are as desired.

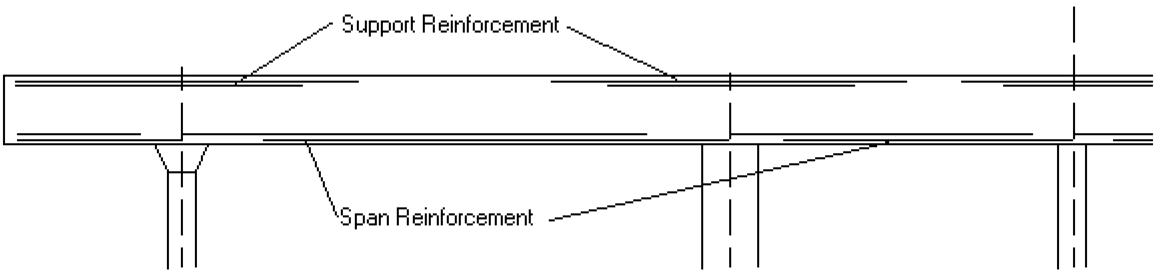


Add support bar pattern

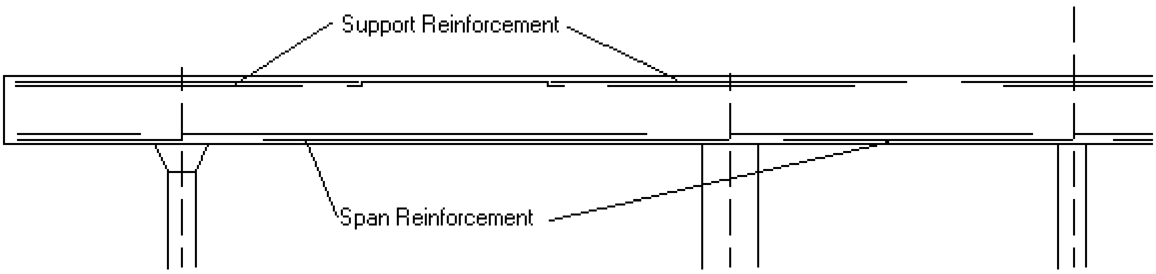
Allows the designer to nominate a group of reinforcing bars in the pattern of support reinforcement, i.e. reinforcement locally positioned over each column and extending into the spans on either side of the column. This reinforcing bar pattern can be placed on either face of the member, top or bottom, and is available from both toolbars. The pattern of the reinforcing bars is shown in the diagrams below as Support Reinforcement. The designer is presented with the following dialog in which the following data is nominated.



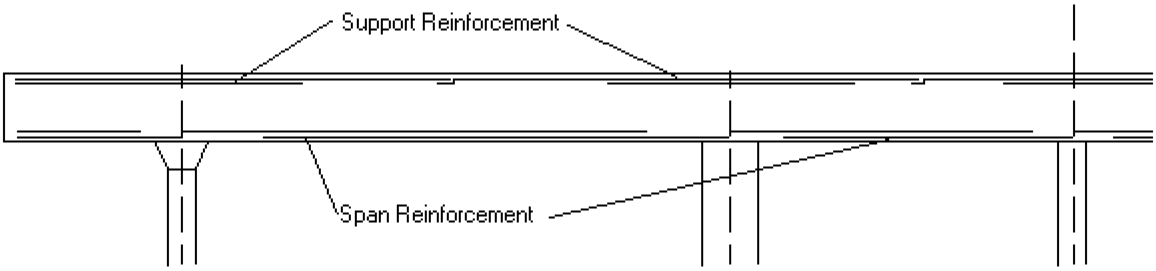
1. Left Column in Pattern:- The 7.2.3.2 Column Grid Reference of the left column in the group of columns over which reinforcing bars are to be placed
2. Right Column in Pattern:- The 7.2.3.2 Column Grid Reference of the right column in the group of columns over which reinforcing bars are to be placed
   In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.
3. Main Bar Length:- The fraction of the span over which the reinforcing bar is to extend either side of the column. The span length used is the average of the span lengths either side of a column.
4. Short Bar Length:- The fraction of the span over which the staggered bars in the reinforcing bar are to extend. The span length used is the longer of the span lengths either side of a column.
5. Cover to Reinforcement:- Cover to the relevant surface of the reinforcement from the relevant face. Once the reinforcing bars have been added the covers can be amended at each end of each bar.
6. Support Reinforcement Pattern Type:- There are three ways the reinforcing bars can be defined. If a span length is less than 75% of the length of the spans either side of it, RAPT will make the main bar continuous over the length of that span. They can be
   1. Local over Columns:- New bars are created over the columns and terminating at the Main Bar Length each side of the column. The new reinforcing bars are shown in the diagram below for support reinforcement.



   2. Continuous Bar Pattern with Bars Lapping at Mid-span:- New bars are created over the columns and terminating at the Main Bar Length each side of the column. Extra intermediate bars are provided between the ends of these bars to provide a continuous mat of reinforcement. The new reinforcing bars are shown in the diagram below for support reinforcement.

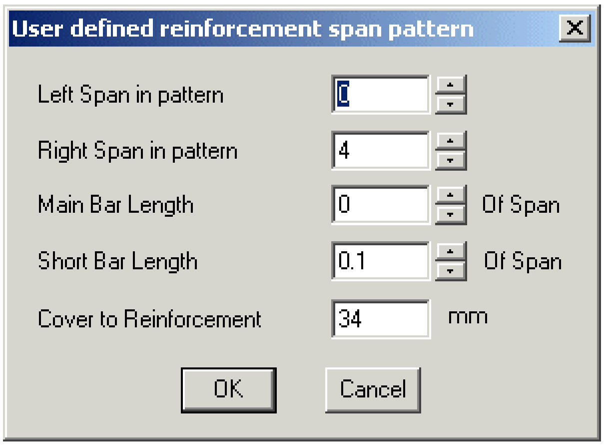


3. Continuous Bar Pattern with Bars Lapping at Mid-span:- New bars are created over the columns and joining at mid-span. The new reinforcing bars are shown in the diagram below for support reinforcement.



Add span bar pattern

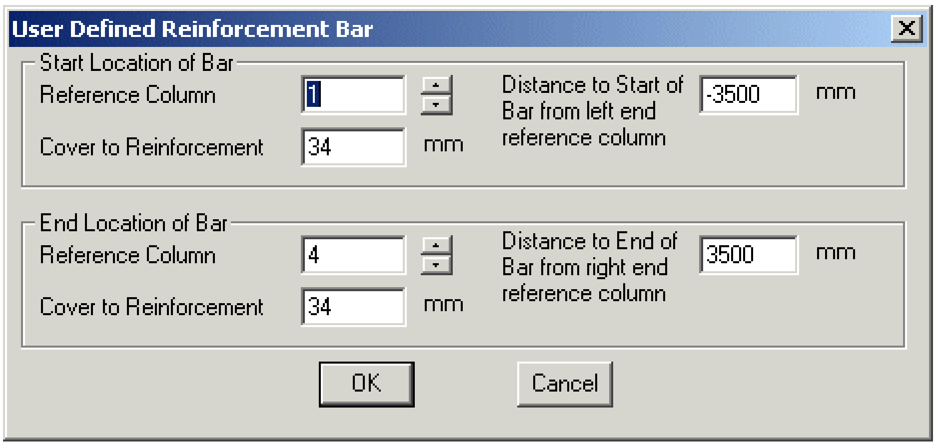
Allows the designer to nominate a group of reinforcing bars in the pattern of span reinforcement, i.e. reinforcement positioned between the centrelines of the columns at each end of the span. These reinforcing bars can be placed on either face of the member, top or bottom, and are available from both toolbars. The pattern of the reinforcing bars is shown in the diagrams above as Span Reinforcement. The designer is presented with the following dialog in which the following data is nominated



1. Left Span in Pattern:- the left span in the group of spans over which a bars are to be placed.
2. Right Span in Pattern:- the right span in the group of spans over which bars are to be placed.
3. Main Bar Length:- The distance from the column centreline to the start of the bar at each end of each span as a fraction of the span.
4. Short Bar Length:- The distance from the column centreline to the start of the staggered bars at each end of each span as a fraction of the span.
5. Cover to Reinforcement:- Cover to the relevant surface of the reinforcement from the relevant face. Once the bars have been added the covers can be amended at each end of each bar.

Add bar

Allows the designer to nominate a single random bar anywhere in the length of the member. The designer is presented with the following dialog in which the following data is nominated



1. Start Location of Bar

    1. The 7.2.3.2 Column Grid Reference from which the left end of the bar is to be measured
    2. Distance to Start of Bar from Left End Reference Column

3. Cover to reinforcement:- Cover to the relevant surface of the reinforcement from the relevant face at the left end of the Bar.

2. End Location of Bar

1. The 7.2.3.2 Column Grid Reference from which the right end of the bar is to be measured
2. Distance to End of Bar from Right End Reference Column
3. Cover to reinforcement:- Cover to the relevant surface of the reinforcement from the relevant face at the right end of the Bar.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

 Insert Design Reinforcement Pattern

Not available in General Reinforcement. If a RAPT run has been designed and extra reinforcement added in the design and recorded in the reinforcement layout, this button will insert the reinforcement detailed in Reinforcement layout into the input as User defined Reinforcement. This can then be rerun to see the effects of the actual reinforcement pattern on the design as distinct from the reinforcement required at each point. This should affect results for ductility, shear and deflections. Once the reinforcement has been inserted into the data grids, the user is free to modify it before rerunning.

Delete bar

Any group of reinforcing bars can be selected and deleted using the selection logic in 4.4.3 Cell Selection.

Delete All User defined Bars

All User Defined Reinforcing Bars defined in all three user defined reinforcement screens, Top, Bottom and General, in this input will be deleted.

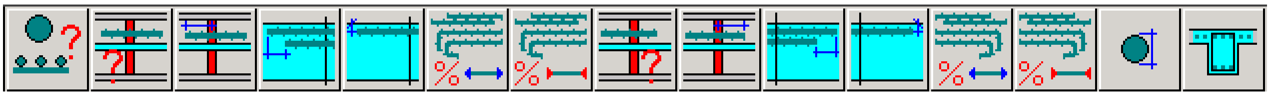
Copy

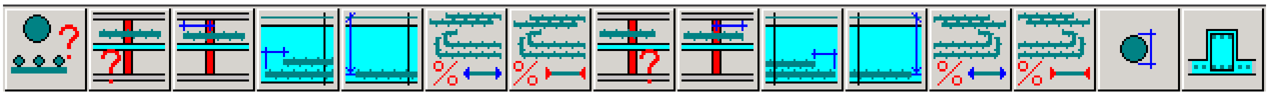
Copy the selected reinforcing bars to memory for later paste.

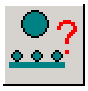
Paste

Paste the copied bars into the data. There is no need to create rows to paste into, RAPT will do this automatically. If a bar row is selected the new bars will be placed at that row, otherwise they will be placed at the end of the list of bars.

Data Definition

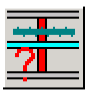




Reinforcement type

The user may select any type of reinforcing bar or mesh defined in the 6.4 Materials data. The default reinforcement type will be the bar type defined in 7.2.6.1 Reinforcement General as the Flexural Bar type. On clicking the data cell, a drop down list similar to the one below will appear to allow the designer to select the steel type desired. If a different reinforcement type is selected, RAPT will check the bar size list for that type to find the bar size in the new type that has the closest area to the previous bar size and will substitute this new size.

Designation	Type	Yield Stress
R	Round	250MPa
Y	Deformed	400MPa
N	Deformed	500MPa
F	Round	450MPa
L	Deformed	500MPa

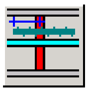
Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the reinforcing bar is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the reinforcing bar. This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

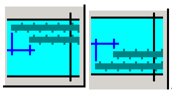
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a reinforcing bar in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a reinforcing bar.

Obviously the left end of a reinforcing bar must be to the left of the right end so make sure when modifying reinforcing bar ends that the order of modification will ensure this.

Distance to left end of bar from Reference Column

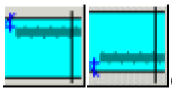
The distance from the left reference column to the left end of the reinforcing bar. If the distance entered moves the reinforcing bar into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the reinforcing bar.

Obviously the left end of a reinforcing bar must be to the left of the right end so make sure when modifying reinforcing bar ends that the order of modification will ensure this.

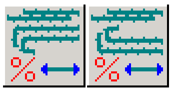
Alternate bar stagger length at left end

The distance from the left end of the reinforcing bar to the stagger location if bars are to be staggered. This value is 0 if there is no stagger. The net bar length of the short bar after allowing for staggers must be greater than 10% of the main bar length.

The number of bars stopping at the stagger location will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagrams above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations.

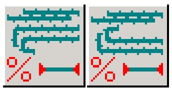
Cover From Concrete Surface to Bar Surface at left end

The cover to the left end of the bar is measured to the relevant bar surface from the relevant concrete surface at that point. When RAPT adds reinforcing bars it will determine a bar cover that attempts to provide the nominated cover on this face from the concrete surface. The designer may modify this to place the bar at any level (or slope) desired.



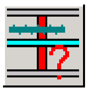
% Development of Left End of Bar in Tension

The percentage development at the left end of the full length bars (zero development is assumed for staggered bars) to allow for the presence of hooks, cogs or laps to other bars. e.g. a standard hook or cog would normally provide 50% tension development while a full tension lap would provide 100% tension.



% Development of Left End of Bar in Compression

The percentage development at the left end of the full length bars (zero development is assumed for staggered bars) to allow for the presence of hooks, cogs or laps to other bars. e.g. a standard hook or cog would normally provide 0% compression development while a full compression lap would provide 100% compression development. RAPT will calculate a reduced bar capacity if a design point is within the development length. It is up to the designer to ensure that the development nominated can be provided for the bars in practice.



Right End Reference Column Number

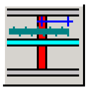
The 7.2.3.2 Column Grid Reference from which the right end of the reinforcing bar is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the reinforcing bar. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a reinforcing bar in one operation rather than

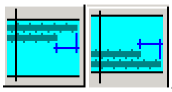
two separate operations which would result in two complete calculations of the effect of moving the end location of a reinforcing bar.

Obviously the right end of a reinforcing bar must be to the left of the right end so make sure when modifying reinforcing bar ends that the order of modification will ensure this.

Distance to right end of bar from Reference Column

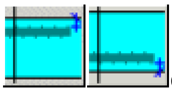
The distance from the right reference column to the right end of the reinforcing bar. If the distance entered moves the reinforcing bar into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the reinforcing bar.

Obviously the left end of a reinforcing bar must be to the left of the right end so make sure when modifying reinforcing bar ends that the order of modification will ensure this.

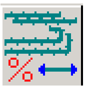
Alternate bar stagger length at right end

The distance from the right end of the reinforcing bar to the stagger location if bars are to be staggered. This value is 0 if there is no stagger. The net bar length of the short bar after allowing for staggers must be greater than 10% of the main bar length.

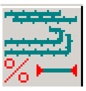
The number of bars stopping at the stagger location will be half of the total number for an even number of bars or half the number - 1 for an odd number of bars. These bars can then be placed in 2 ways as shown in the diagrams above. The Support reinforcement is using a long bar and a short bar while the Span reinforcement uses two equal length bars offset from each other. Both layouts result in the same bar capacity and the method used is not necessary for RAPT's calculations.

Cover From Concrete Surface to Bar Surface at right end

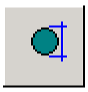
The cover to the right end of the bar is measured to the relevant bar surface from the relevant concrete surface at that point. When RAPT adds reinforcing bars it will determine a bar cover that attempts to provide the nominated cover on this face from the concrete surface. The designer may modify this to place the bar at any level (or slope) desired.

% Development of Right End of Bar in Tension

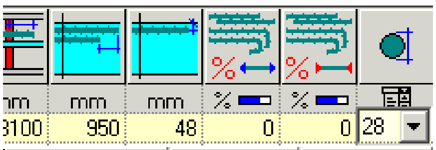
The percentage development at the right end of the full length bars (zero development is assumed for staggered bars) to allow for the presence of hooks, cogs or laps to other bars. e.g. a standard hook or cog would normally provide 50% tension development while a full tension lap would provide 100% tension.

% Development of Right End of Bar in Compression

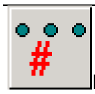
The percentage development at the right end of the full length bars (zero development is assumed for staggered bars) to allow for the presence of hooks, cogs or laps to other bars. e.g. a standard hook or cog would normally provide 0% compression development while a full compression lap would provide 100% compression development. RAPT will calculate a reduced bar capacity if a design point is within the development length. It is up to the designer to ensure that the development nominated can be provided for the bars in practice.

Bar Size

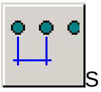
The bar size can be selected from a drop down list of available sizes for the reinforcement type and bar type selected as shown below. The list of bar sizes is defined in the 6.4 Materials Data->Reinforcement Bar/Mesh and can be modified there.



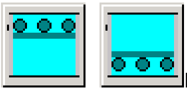
Nominal Bar Size	Bar Diameter	Bar Area
10	10mm	78.5mm2
12	12mm	113mm2
16	16mm	201mm2
20	20mm	314mm2
24	24mm	452mm2
28	28mm	616mm2
32	32mm	804mm2
36	36mm	1020mm2
40	40mm	1260mm2

Number of Bars

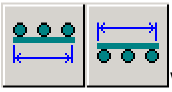
The number of bars to be used for this bar layer for design except in the end stagger lengths. Not available for mesh.

Spacing of bars

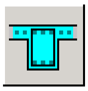
The spacing of bars to be used in this bar layer for design except in the end stagger lengths. RAPT will calculate a number of bars based on the width of the design section at the mid-depth of the bars at this location. Where the design width varies along the length of the bar, RAPT will calculate a different number of bars at each design location to suit the design width. Not available for mesh.

Mesh full width of Design Cross-section

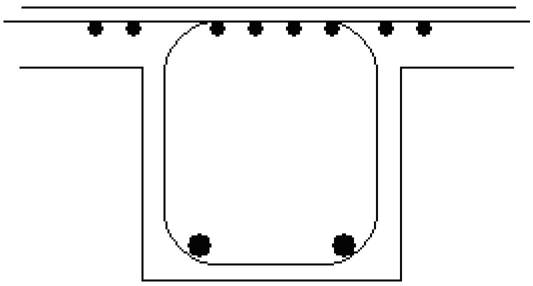
Allows for the mesh type that has been defined in this bar layer to be provided full width of the design section at this point based on the width of the design section at the mid-depth of the longitudinal bars at this location. Where the design width varies along the length of the bar, RAPT will calculate a different width of mesh at each design location to suit the design width. Not available for bar reinforcement.

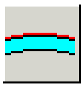
Width of mesh

The width of mesh to be used for this bar layer for design. Not available for bar reinforcement.

Percentage Reinforcement in Flange

The percentage of the reinforcement in this bar layer that will be assumed to be placed in the flange of a beam (outside the shear cage) for beam shear calculations. Reinforcement in the flange of a beam is ignored in beam shear calculations thus reducing the shear capacity.



Layer Attached after the PreExisting Load Case

This reinforcing layer is attached after the concrete member is placed under load as defined by the pre-existing Load Case. This would commonly be used in cases of strengthening of concrete members while in use. RAPT will calculate the strain in the concrete at this level of the member under the Pre-Existing load condition at all design points along the length of this reinforcement and use that as the initial strain condition in the concrete at each cross-section, rather than assuming zero strain, for the calculation of the strain and stress in this reinforcement under other load conditions.

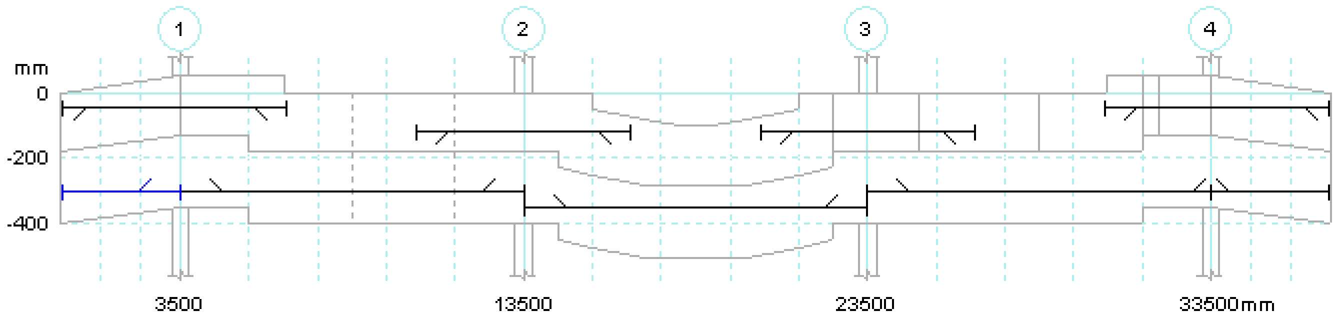
Graphical View

The graphical output consists of separate diagrams for each design strip.

Each reinforcing layer is shown in elevation. Ends of each reinforcing layer are indicated by a short vertical line. Stagger end locations are indicated with a diagonal line sloping towards the relevant bar end from the stagger end location on the bar and away from the surface the reinforcing layer is related to.

To select a reinforcing layer, click on it with the left mouse button. The selected reinforcing layer will be shown in blue. The window in which the selected reinforcing layer is defined will be selected and the data row controlling the selected reinforcing bar will become the current row.

Double-Click on a reinforcing layer will move the focus into Zoom mode equivalent to pressing  (see below) and the reinforcing layer will then be shown with a diameter of the reinforcing steel drawn to scale.



Graphics Toolbar 

A special graphics toolbar is provided to assist with viewing the data. This toolbar will only be available when program focus is in the Graphics Window. The functionality of this toolbar for reinforcement data is slightly different to the general functionality. The buttons available are

 Zoom (Ctrl + Z). This button will toggle between full screen mode and bar zoom mode for the graphics in a window. In bar zoom mode, the spans in which the current bar is placed will be shown scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window. In this mode, the bars are drawn to scale with their correct diameter. When not in zoom mode, the bars are drawn as a 1 pixel wide line.

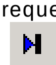
 Move to next Reinforcing Bar (Ctrl + Right Arrow). In Full Screen Mode the next reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In bar Zoom mode, the spans containing the next reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour. The data will view will show the selected reinforcing bar.

 Not Available.

 Not Available.

 Move to previous Reinforcing Bar (Ctrl + Left Arrow). In Full Screen Mode the previous reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In Bar Zoom mode, the spans containing the previous reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour. The data will view will show the selected reinforcing bar.

 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

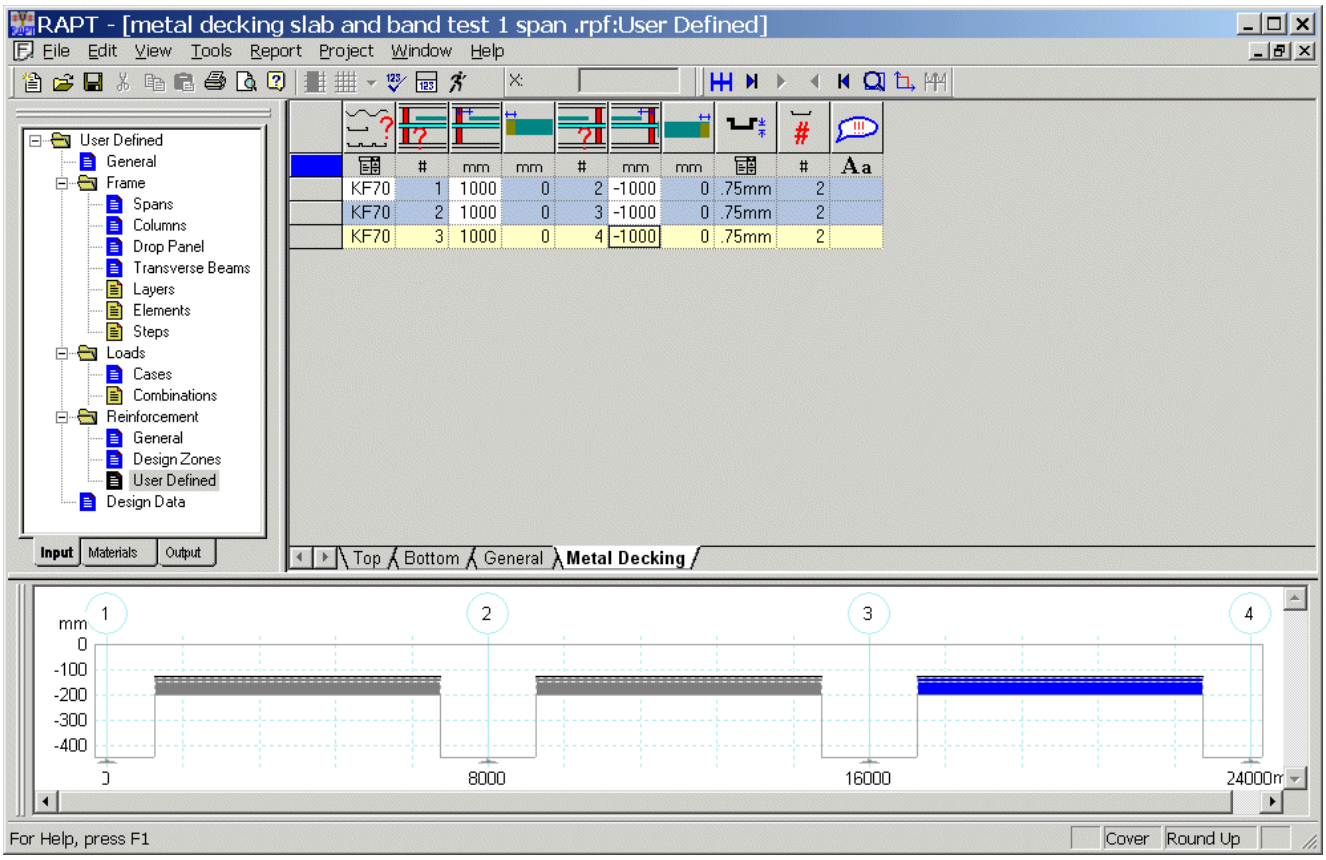
Then user can then move around the graph using the Scroll Bars or the movement keys.  or  buttons will still move the selection to the next or previous reinforcing bar and will return the Window to Full Screen Mode.

Clicking this  button again or on the buttons will return the Window to Full Screen Mode.

Clicking  will change the mode to Bar Zoom Mode with the reinforcing bar selected in the Select Zoom mode.

 Not available. Selecting a reinforcing bar in the graphics automatically moves the data to the row controlling that reinforcing layer so there is no need for a dialog.

7.2.6.4.2 Metal Decking



Graphic Interaction

To select a metal decking sheet or reinforcing bar, click on a point along its length (within 4 pixels above or below the reinforcing bar or metal decking sheet sheet) with the left mouse button. The selected metal decking sheet or reinforcing bar will be shown in blue. The window in which the selected metal decking sheet or reinforcing bar is defined will be selected and the data row controlling the selected metal decking sheet or reinforcing bar will become the current row. If more than one metal decking sheet or reinforcing bar is found at the selected location, RAPT will show the first selected metal decking sheet or reinforcing bar after the currently selected metal decking sheet or reinforcing bar. Continuing to click at the same location (with a suitable pause to avoid double clicking) will continually select the next metal decking sheet or reinforcing bar in the selected list and assign that metal decking sheet or reinforcing bar as the current metal decking sheet or reinforcing bar.

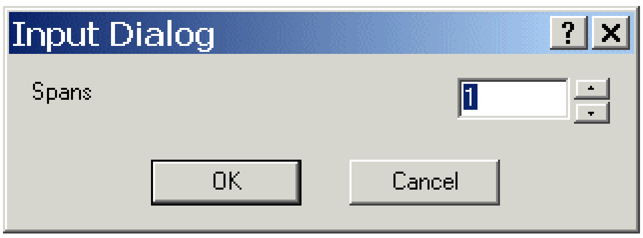
Metal Deck Toolbar

 Add Metal Decking to All Spans

Allows the designer to add metal decking to the soffit of all spans in the frame.

 Add Metal Decking Sheet to the Selected Span

Allows the designer to add metal decking to the soffit a selected span in the frame. The following Dialog will be presented requesting the span number.



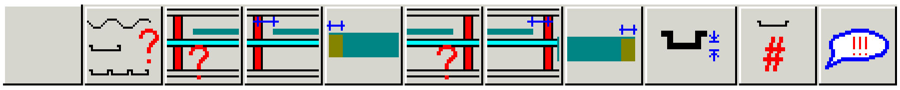
 Delete Metal Decking Sheet

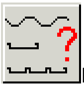
Any group of metal decking sheets can be selected and deleted using the selection logic in 4.4.3 Cell Selection.

Delete All User defined Sheets

All User Defined Reinforcing Bars defined in all three user defined reinforcement screens, Top, Bottom and General, and metal deck sheets in this input will be deleted.

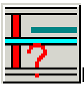
Data Definition



Metal Deck Type

The user may select any type of metal decking defined in the 6.7 Structural Deck Types. The default reinforcement type will be the first in the list. On clicking the data cell, a drop down list similar to the one below will appear to allow the designer to select the sheet type desired. If a different metal deck type is selected, RAPT will check the metal deck sheet size list for that type to find the sheet thickness in the new type that has the closest area to the previous sheet thickness and will substitute this new size.

Sheet Designation	Yield Stress	Description
RF55 - 2 Pan	550MPa	
RF55 - 3 Pan	550MPa	
KF57	550MPa	
KF40 - 2 Pan	550MPa	
KF40 - 3 Pan	550MPa	
KF70	550MPa	

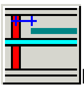
Left End Reference Column

The 7.2.3.2 Column Grid Reference from which the left end of the metal deck sheet is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the start of the metal deck sheet . This is necessary for the internal calculations for adding and deleting spans.

In input data screens, where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

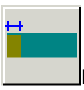
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable to allow the designer to modify the start location of a metal deck sheet in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the start location of a metal deck sheet.

Obviously the left end of a metal deck sheet must be to the left of the right end so make sure when modifying metal deck sheet ends that the order of modification will ensure this.

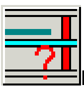
Distance to left end of Metal Deck Sheet from Reference Column

The distance from the left reference column to the left end of the metal deck sheet. If the distance entered moves the metal deck sheet into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the metal deck sheet.

Obviously the left end of a metal deck sheet must be to the left of the right end so make sure when modifying metal deck sheet ends that the order of modification will ensure this.

Distance from Start of Concrete Sheet to Start of Voids

Allows the designer to start the voids created in the concrete by the metal deck sheet to start at a distance along the sheet rather than at the end of the sheet. This allows the sheet to be embedded into a thicker concrete section at the end if required.

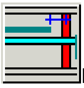
Right End Reference Column Number

The 7.2.3.2 Column Grid Reference from which the right end of the metal deck sheet is dimensioned. RAPT will always adjust the column grid reference and distance from it so that the column grid reference shown here is at or immediately before the end of the metal deck sheet. This is necessary for the internal calculations for adding and deleting spans.

In input data screens where a reference column number is required to define a location, the Column Grid Reference is used, not the column number. To input a new value, the Column Grid reference must be typed in full or the column number can be used preceded by a # symbol e.g. #5 for column number 5. When the value is accepted, it will be converted to the Column Grid Reference and this will be shown as the Reference Column. In the default case, 5 or #5 will both represent the column grid reference for column 5.

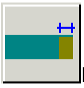
In view mode, the data in the cell shows only the column grid reference # (e.g. 1). In Edit mode, the cell shows both the column grid reference and the distance from it in the following format # ; ## (e.g. D;2000). Both of these data items are editable and allow the designer to modify the end location of a metal deck sheet in one operation rather than two separate operations which would result in two complete calculations of the effect of moving the end location of a metal deck sheet.

Obviously the right end of a metal deck sheet must be to the left of the right end so make sure when modifying metal deck sheet ends that the order of modification will ensure this.

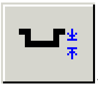
Distance to Right End of Metal Deck Sheet from Reference Column

The distance from the right reference column to the right end of the metal deck sheet. If the distance entered moves the metal deck sheet into another span, RAPT will automatically adjust the reference column number and the distance from the column to refer to the nearest column at or before the metal deck sheet.

Obviously the left end of a metal deck sheet must be to the left of the right end so make sure when modifying metal deck sheet ends that the order of modification will ensure this.

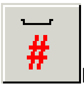
Distance from End of Concrete Sheet to End of Voids

Allows the designer to start the voids created in the concrete by the metal deck sheet to start at a distance along the sheet rather than at the end of the sheet. This allows the sheet to be embedded into a thicker concrete section at the end if required.

Sheet Thickness

The sheet thickness can be selected from a drop down list of available sizes for the metal deck type selected as shown below. The list of sheet thicknesses is defined in the 6.7 Structural Deck Types and can be modified there.

Nominal Size	Sheet Thickness	Sheet Width	Sheet Height	Sheet Area
.6mm	0.6mm	400mm	54mm	405.2mm2
.75mm	0.75mm	400mm	54mm	506.4mm2
.9mm	0.9mm	400mm	54mm	607.6mm2
1.0mm	1mm	400mm	54mm	675.2mm2

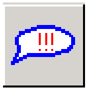
Number of Sheets

The number of sheets of metal decking to be used in this span. Only a whole number of sheets can be selected.

Note:- Metal Decking Sheets are supplied in specific widths. Normally the sheets will extend over the full width of the slab. RAPT cannot assume an area of sheet per unit of slab width, or a fraction of a sheet width. A whole number of sheets must be defined. To achieve this in RAPT, the Design Width for the slab must be defined as a multiple of the sheet width.
All output results will then be the total for this Design Width.

If the total width of metal decking sheets in a span is less than the nominated Design Width, a warning message will then be given when designing that the total sheeting width is less than the design width.

If the total width of metal decking sheets in a span is greater than the nominated Design Width, an Error message will then be given when designing that the total sheeting width is greater than the design width and the design will not proceed until this has been fixed

Description

Designers information about this metal deck sheet.

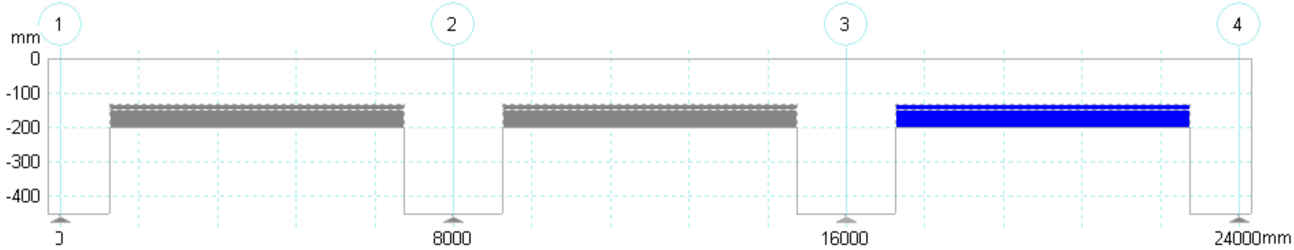
Graphical View

The graphical output consists of separate diagrams for each design strip.

Each metal deck sheet is shown in elevation to scale as a rectangle shaded mid grey. Voids will show in the background as shown below.

To select a metal deck sheet, or a reinforcing bar, click on it with the left mouse button. The selected reinforcing layer will be shown in blue. The window in which the selected reinforcing layer is defined will be selected and the data row controlling the selected reinforcing bar/metal decking sheet will become the current row.

Double-Click on a metal deck sheet will move the focus into Zoom mode equivalent to pressing (see below).



Graphics Toolbar

A special graphics toolbar is provided to assist with viewing the data. This toolbar will only be available when program focus is in the Graphics Window. The functionality of this toolbar for reinforcement data is slightly different to the general functionality. The buttons available are

Zoom (Ctrl + Z). This button will toggle between full screen mode and bar zoom mode for the graphics in a window. In bar zoom mode, the spans in which the current bar is placed will be shown scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window. In this mode, the bars are drawn to scale with their correct diameter. When not in zoom mode, the bars are drawn as a 1 pixel wide line.

Move to next Reinforcing Bar (Ctrl + Right Arrow). In Full Screen Mode the next reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In bar Zoom mode, the spans containing the next reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour. The data will view will show the selected reinforcing bar.

Not Available.

Not Available.

Move to previous Reinforcing Bar (Ctrl + Left Arrow). In Full Screen Mode the previous reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In Bar Zoom mode, the spans containing the previous reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour. The data will view will show the selected reinforcing bar.

Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

Then user can then move around the graph using the Scroll Bars or the movement keys. or buttons will still move the selection to the next or previous reinforcing bar and will return the Window to Full Screen Mode.

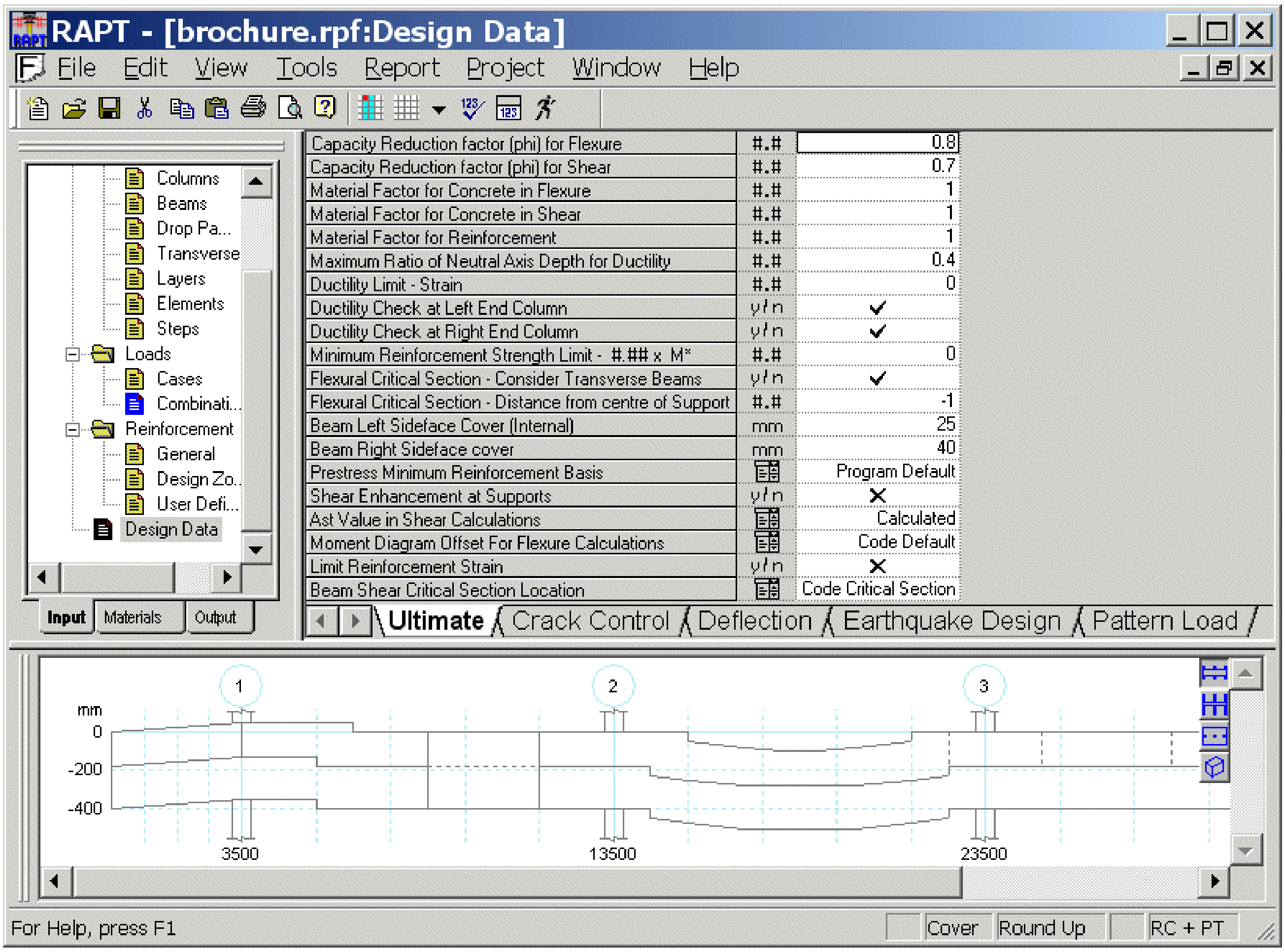
Clicking this button again or on the buttons will return the Window to Full Screen Mode.

Clicking will change the mode to Bar Zoom Mode with the reinforcing bar selected in the Select Zoom mode.

Not available. Selecting a metal deck sheet in the graphics automatically moves the data to the row controlling that metal deck sheet so there is no need for a dialog.

7.2.7 Design Data

7.2.7.1 Ultimate Design Data



Capacity reduction factors are used by many Design Standards, eg AS3600 and ACI318, to account for the variability of the material properties controlling strength and the likelihood of under-performance of a member. This factor also takes into account inaccuracies in design procedures and small deficiencies in workmanship on site (i.e. concrete dimensions etc). This factor should be set to 1 for other code types which use material factors.

Material factors are used by many Design Standards, eg BS8110, SABS0100, CP2004, IS456/IS1343 and Eurocode2, to account for the variability of the material properties controlling strength and the likelihood of under-performance of a member. This factor also takes into account inaccuracies in design procedures and small deficiencies in workmanship on site (i.e. concrete dimensions etc). This factor should be set to 1 for other code types which use capacity reduction factors.

Data Definition

Capacity Reduction factor (phi) for Flexure
The capacity reduction for flexural strength calculations.

Capacity Reduction factor (phi) for Shear
The capacity reduction factor for beam and punching shear strength calculations.

Material Factor for Concrete in Flexure
The material factor for concrete for flexural strength calculations.

Material Factor for Concrete in Shear
The material factor for concrete for beam and punching shear strength calculations.

Material Factor for Reinforcement
The material factor for all reinforcement types for flexural strength and beam and punching shear strength calculations.

Maximum Ratio of Neutral Axis Depth for Ductility
Ductility is an important aspect of the design of a section/member. A ductile member will undergo large deflections before failing. This acts as a safety measure in that members can be seen to be dangerous before actually failing. Ductility in most codes is controlled by limiting the depth of the neutral axis to a fraction of the effective depth of the section at plastic hinge locations. RAPT uses the term ku to define this fraction, so the depth limit is ku*d. Many design codes do not place a limit on ductility unless moment redistribution has been allowed for in design, either arbitrarily, as a percentage redistribution, or in a Plastic Design. This does not mean that designers should ignore

ductility in members. RAPT basically defaults the ductility limit for design for those codes to the upper limit of ductility where redistribution is used by using zero redistribution in the formula for each code. It is good design practice to make concrete members even more ductile than this. In all cases, at critical ductility locations (probable Plastic Hinge locations), RAPT will add compression reinforcement where necessary to limit the ductility as nominated by this user defined value, unless over-ridden by redistribution ductility rules as discussed below. Remember also, that as the depth of neutral axis rises above the point where the design is a Balanced design, the stress in the reinforcement at ultimate strength will drop below the yield strength of the steel and the steel is actually being wasted. Also, a member’s strength is reduced by most codes once the reinforcement ratio is greater than about 75% of the Balanced ratio.

Note the limits "without redistribution" are significantly higher for "Material Factor" codes. This is because the material factor for concrete and steel is applied to the properties at the start of the calculations, reducing the material strengths and increasing the neutral axis depth. In these codes, there is no need to think of a 66 or 75% balanced requirement as the neutral axis depth increase automatically accounts for this.

1. AS3600 - 2009 Clause 8.1.3, ku = 0.36
2. AS3600 - 2001 Clause 8.1.3, ku = 0.4
3. ACI318 - use Ductility Strain Limit below based on Clause 10.3.5
4. BS8110 and CP65 Clause 3.2.2.1, 3.4.4.4 and 4.2.3.1 ku = 0.5. Where there is no redistribution allowed for, a limit of .58 to ensure a balanced section may be more appropriate.
5. Eurocode2 Clauses 5.4 and 5.5, ku = 0.448 Grades <= C50/60 ku = 0.368 Grade > C50/60. Where there is no redistribution allowed for, a limit of .58 to ensure a balanced section may be more appropriate.
6. SABS0100 Clause 4.3.3.4, ku = 0.5. Where there is no redistribution allowed for, a limit of .58 to ensure a balanced section may be more appropriate.
7. CP2004 Clause 6.1.2.4b, ku = .5 for fcu <= 40MPa, ku = .4 for fcu <= 70MPa and ku = .33 for fcu <= 100MPa. Where there is no redistribution allowed for, a limit of .58 to ensure a balanced section may be more appropriate.
8. IS456 clause 37.1d and IS1343 clause 21.1.1d

RAPT allows the user to over-ride our interpretation of each code if they feel it is warranted.

If the designer nominates that 7.2.2 moment redistribution is to be allowed for in the design, RAPT use the formulae in each design standard to set a ductility limit based on the amount of redistribution used. RAPT will never use a limit higher than the one defined here no matter what the calculated value from the redistribution.

RAPT will add compression reinforcement in the design of potential plastic hinge locations to reduce the neutral axis depth to the value defined here. It will not limit the neutral axis depth at other locations in a member. Some design codes define limits for minimum compression reinforcement (e.g. AS3600 requires minimum compression reinforcement when ku = .36) but do not set a maximum limit on ku. It is not economical to design for a neutral axis depth above the values nominated above as full strength gain is not achieved for the extra materials used. Also, many design codes require even more stringent limits on ductility in Seismic/Earthquake design situations. Limits on ku <= .25 are more logical where a member is required to perform adequately under earthquake actions.

Ductility Limit - Strain

Some design codes, e.g. ACI-318 2008 and later, place a minimum limit on the tensile strain in the extreme reinforcement layer rather than a neutral axis depth limit to control ductility. This data item allows this alternate method of controlling ductility. If both a neutral axis depth limit and a strain limit are defined, RAPT will use the limit which results in a more ductile section to control the design. If this value is set to less than or equal to zero, it will be ignored.

If the designer nominates that 7.2.2 moment redistribution is to be allowed for in the design, RAPT use the formulae in each design standard to set a ductility limit based on the amount of redistribution used. RAPT will never use a limit higher than the one defined here no matter what the calculated value from the redistribution.

Ductility Check at Left End Column
Ductility Check at Right End Column

RAPT will check for ductility at the Support Moment Zone at end columns unless expressly told not to do so here. Users able to nominate if RAPT should check for ductility at the end columns. In prestressed design RAPT often has a ductility problem at the end columns as the tendon is at the centroid thus greatly affecting the ku value. RAPT will then add compression reinforcement to modify the ductility problem. Often in these cases there is only a small moment applied at the end columns (thus implying that a plastic hinge could not form here) and the extra reinforcement added due to ductility would not appear warranted. Thus RAPT now allows the designer to choose if the end columns are possible hinge locations or not. If users enter a NO value then RAPT will NOT check ductility at the end column

Minimum Reinforcement Strength Limit - #.## x M*

AS3600 BS8110, CP65, SABS0100, CP2004, IS456/IS1343 and ACI318 state that the minimum applied ultimate moment, M*, to be designed for is *1.2Mcr. Eurocode 2 uses 1.15*Mcr.

The codes require designers to include sufficient reinforcement to satisfy a moment which causes the member to crack. The ultimate strength in bending is calculated assuming a fully cracked section. For small percentages of steel, this moment could be less than the moment Mcr to cause cracking. Failure of such a member would be quite sudden. To prevent such a failure, the ultimate strength in bending must be greater than Mcr.

However there is no concession for the situation when the section's design strength in bending, ØMu is much greater than the applied ultimate moment, M* (ie ØMu >> M*).

In AS1480 designers were permitted to relax this rule if ØMu was greater than 1.33M* (ie ØMu > 1.33 M*). [Also see ACI318 Clause 10.5.2] This, in effect, gave an extra factor of safety on the failure condition but did not ensure that cracking occurred prior to failure. Some other codes allow similar concessions.

RAPT has given the user the option of accepting the code as read by leaving the value for this parameter as 0 and then minimum M* is taken as defined in each code.

If this rule is to be supplanted with something similar to AS1480 then a number may be entered in the table. eg. 1.33. This will instruct the program to accept the section as being of adequate strength, i.e. no need to provide extra reinforcement to satisfy minimum flexural reinforcement rules, as long as ØMu > 1.33M*. If this is not satisfied then reinforcement is added until ØMu > 1.33M* or Ø1.2Mcr whichever is the lesser. 1.33 is the minimum value allowed by RAPT.

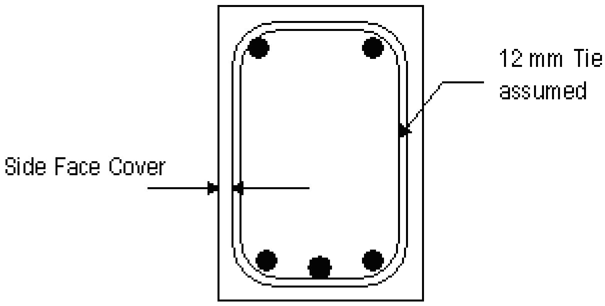
Flexural Critical Section - Consider Transverse Beams
Theory Section T5 defines how RAPT calculates the location of Flexural critical section near the face of the supports for different design standards. Users can have RAPT include or exclude any transverse beams entered in Input Screen in this determination. The YES option will include transverse beam dimensions in calculating the critical section for flexure.

Flexural Critical Section - Distance from centre of Support
RAPT allows users to pre-define the position of the flexural critical section. The critical section is defined as

Critical section = Factor x length of column to the face of the support.

where the factor is the value entered here. The input range allowed is 0 to 1.00. If users input -1 then RAPT defaults the critical sections to the code defaults. This factor is applied to all critical sections. The critical section is always measured from the column centre line. See Theory Section T5 for theory on calculation of Flexural Critical Sections for different design standards.

Beam Left Side face Cover (Internal)
Beam Right Side face Cover
Specified as the distance between the concrete surface and the outside of the shear tie. RAPT assumes that a 12 mm tie exists and then uses this data to decide what bar size and spacings are appropriate.



Prestress Minimum Reinforcement Basis
This field allows the user to define the calculation method to be used for minimum reinforcement calculations for prestressed members. In general the selected code rules will be used unless one of the options below is selected. The options available are

1. Program default (UBC for unbonded prestressed members)
2. ACI-318
3. UBC
4. TR43 (This is not considered by RAPT to be the default method for BS8110 design or Eurocode 2 design or for any derivatives of BS8110. The designer must select this option if it is to be used for a design).
5. None

Shear Enhancement at Supports
This field allows users to nominate whether shear enhancement near supports (β3 term in the beam shear equation) is to be used for AS3600, BS8110, SABS-0100, CP2004, IS456/IS1343 and CP 65 designs (2d/a term in shear design). The default is

- No for AS3600 - For AS3600 it should only be used under certain conditions defined in the standard and it's commentary.
- Yes for BS8110, SABS-0100, CP2004, IS456/IS1343 and CP 65.
- Other design standards are not affected by selection of this option.

Eurocode 1992 has a method of allowing for Shear Enhancement by reducing the effect of any load applied within 2d of the support face on the applied shear. This rule is not included in RAPT as it is very difficult to include it with the RAPT analysis logic. Using this logic we do not know how it is possible to create a moment/shear diagram that has shear and moment values from co-existing conditions. A designer could manually calculate an effect for this and apply the reduction in his own calculations to reduce the shear requirements.

When Shear Enhancement is selected, RAPT will calculate shear capacities at design locations including at the face of the support.

In situations where large concentrated loads are applied close to supports, the designer should also consider other design alternatives such as Deep Beam Theory using Strut-Tie Modelling to model the load transfer as this is often not a flexural design situation which RAPT considers.

Ast Value in Shear Calculations

This field allows users to nominate the basis on which RAPT will determine the area of flexural reinforcement that will be used in the shear strength calculations at each design point along the frame. The options available are

1. Calculated:- Uses the area of reinforcement at each point that has been calculated for strength at that point. This is the default setting.
2. Maximum:- Uses the maximum area of reinforcement at that face in that design zone. For support reinforcement areas, it will use the maximum area of reinforcement over a support between mid span on either side of the support in question and for span reinforcement areas will use the maximum area of reinforcement on that face in that span. It is then the users responsibility to ensure that the reinforcement is detailed so as to provide this reinforcement at all points as has been assumed by RAPT.

The area of reinforcement used in the calculations is defined in the Beam Shear text output at each point. This reinforcement represents only that provided by RAPT in its flexural strength calculations. It does not include user-defined reinforcement which is also taken into account in the flexural and shear calculations wherever appropriate depending on bar locations, development lengths etc.

A third option is also possible. The user can define the actual reinforcement pattern which is to be adopted for this member and RAPT will then check shear capacity, as well as strength and deflections based on this reinforcement pattern. This can be done automatically after a design has been run by getting RAPT to add the designed reinforcement layout into RAPT as 7.2.6.4 User Defined Reinforcement.

Moment Diagram Offset For Flexure Calculations

Most design codes require the reinforcement to be extended a distance, normally D, past the point where it is needed by design. This requirement is to satisfy the Truss Analogy for Shear Design. This is normally treated as a detailing requirement and not part of the design of the reinforcement requirements at each cross-section in reinforced concrete design. This approach is not consistent with prestressed concrete design where the end locations of tendons are fixed at the start of design and therefore cannot be "offset".

Instead, RAPT now ensures that the tension force required satisfy the applied moment at a design cross-section is provided for the offset distance required past that cross-section in the direction of reducing moment. This tension force could be provided by prestress tendons or un-tensioned reinforcement. If a tendon exists at a cross-section and is required there to resist the applied moment but does not extend by the offset distance past that cross-section, the portion of that prestress force that is required at the design cross-section will be compensated for at the offset location by extra reinforcement. The same will happen with user defined reinforcing bars that terminate in similar circumstances.

For shear design, the development requirements of any steel at a design cross-section will be based on the development at the offset location.

There is no longer an input option for this in RAPT.

Limit Reinforcement Strain

If this option is selected, the maximum strain in any reinforcement in a cross-section will be limited to the "Design Strain Limit" defined for that material type in the materials data. If a reinforcement strain is higher than this limit, the concrete strain will be reduced below the nominal maximum ultimate value for that concrete strength until the reinforcement strain limits are satisfied.

RAPT is now setting this as the default setting in all future runs. See also the discussion about Strain Hardening.

Include Strain Hardening of Reinforcement

Most design standards limit the stress in reinforcing steel to the Yield or Nominal Yield Stress. At the same time, most do not place a limit on the strain in the reinforcement, simply allowing the section to be designed on the assumption that the concrete strain will reach its maximum value and the reinforcing will yield if its stress reaches yield, otherwise a reduced reinforcing stress is used. A better understanding of the way a section acts can be obtained by limiting the strain in the reinforcing to a value below its peak strain (Eurocode recommends .9 of peak strain). While in many cases this will make little difference to the overall section capacity, it does give an idea of its ductility and, in some cases, can result in a reduced capacity.

When the reinforcing strain is increased above the yield strain, some codes allow the stress to increase above the yield stress also. This is normally referred to as Strain Hardening. Eurocode 2 specifically allows this for all reinforcing types. ACI318 and AS3600 allow it for bonded prestressing tendons without putting any direct limit on the strain, simply letting the minimum reinforcement rules control the maximum strain that can be achieved.

This option allows the designer to tell RAPT to include the effects of strain hardening in a design when the reinforcing strain is above the yield strain. The option can only be set if Limit Reinforcement Strain is also selected.

RAPT will use this setting to decide when strain hardening can be used for reinforcing bars, not for prestressing tendons.

For prestressing tendons, RAPT will allow strain hardening as defined in the code rules. For

- AS3600, ACI318, CSA and the Indian codes, the real stress/strain curve of the strand is used and the strain limit defined in the materials data will be applied to limit the stress in the prestressing independent of these two data settings.
- BS8110 and variants based on this code, the stress/strain relation defined in the code is used.
- Eurocode 2 and variants, prestressing steel is limited to yield if strain limitation is not applied. If strain limitation is applied, the strain hardening curve will be used.

Beam Shear Critical Section Location

The critical section for beam shear is normally located at a distance from the face of the support as long as diagonal compression exists between the loads and the support. This distance is defined in various design codes and may vary for different design conditions (eg reinforced and prestressed members have different critical shear locations in ACI318). RAPT will place the shear critical location at this code defined location by default, except for BS8110 where it will be placed at the face of the support and Shear Enhancement will automatically be included.

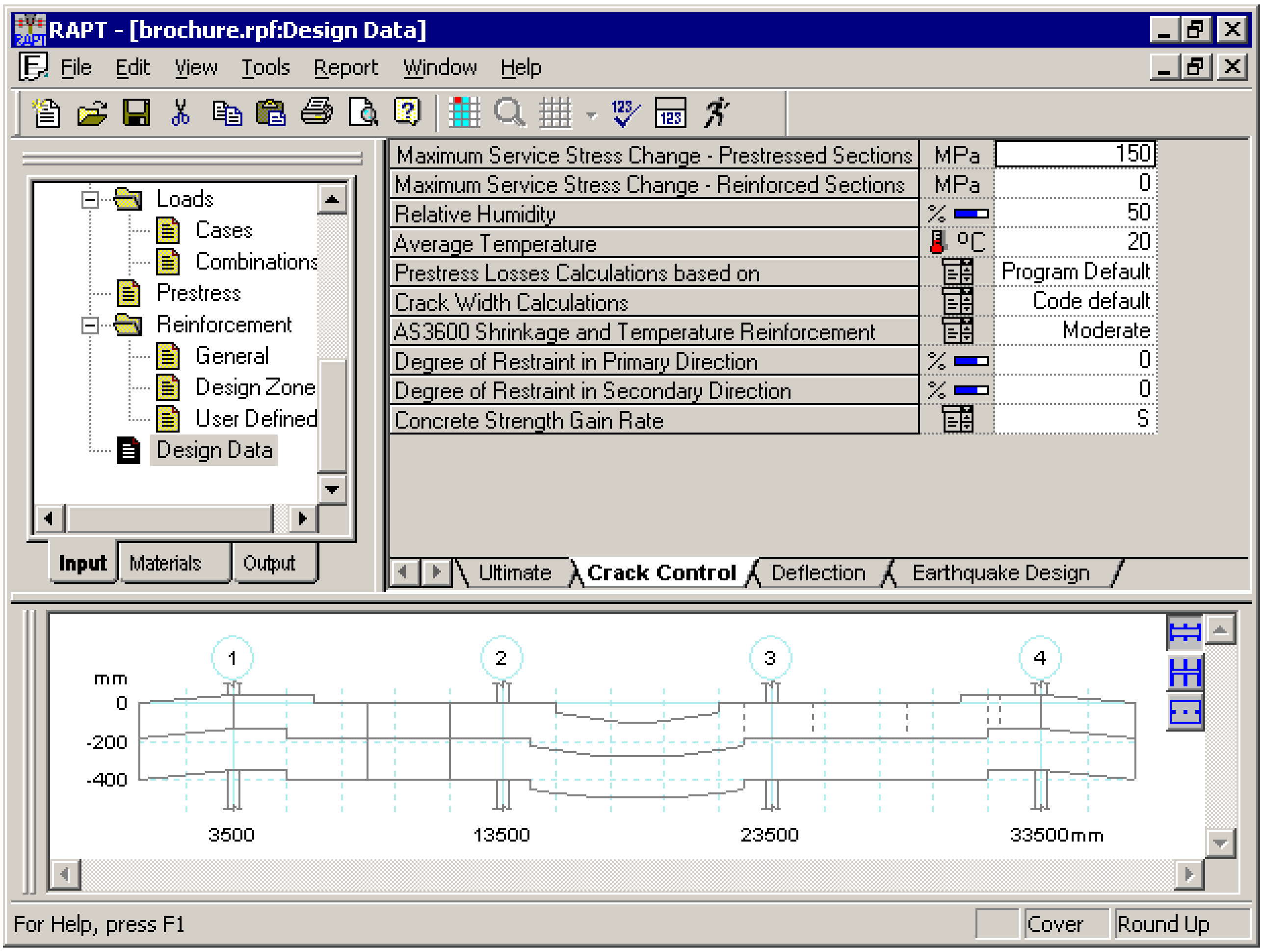
There are some conditions where this critical section is not appropriate and it is more logical to calculate shear capacities at the support face, eg where there is diagonal tension between the loading and the support, or in some earthquake design situations, especially where the total shear is to be carried by the shear reinforcement.

RAPT will over-ride this selection in cases where Shear Enhancement has been selected above and in those cases will use Face of Support.

Options available are

1. Code Critical Section :- RAPT's default setting will always be Code Critical Section.
2. Face of Support :- The Face of Support option should be turned on when a code require shear calculations to the face of the support, without the benefit of Shear Enhancement. If Shear Enhancement is selected above, shear calculations are automatically done to the face of the support. The Face of Support option may be necessary for some Earthquake shear design requirements. The designer will have to set this manually in all cases.

7.2.7.2 Crack Control Design Data



Data Definition

Maximum Service Stress Change - Prestressed Sections
Maximum Service Stress Change - Reinforced Sections
Designers can limit the change in stress in the tension reinforcement under service load conditions to the input value. Separate limits can be applied for prestressed and reinforced cross sections. In a prestressed design where a member is stressed along only part of its length, RAPT will use the reinforced sections figure where no tendons exist. If the stress limit is exceeded, RAPT will add extra reinforcement to bring the change in stress back to the specified value. The various codes give limits depending on the type of member. Limiting the change in stress in tendons is a serviceability crack control check and more information is provided in 7.3.7.3 Flexural Design Service.

If the values input for either stress limit is < 10MPa, RAPT will ignore the stress check for that type, accepting the stress level in the reinforcement required for ultimate conditions. RAPT will initially set these values to the values set in the default file. If a value greater than 80% of the tensile strength of the reinforcement is defined, RAPT will reset it to 80% of the tensile strength of the reinforcement during the design.

Some codes refer to crack widths instead of a stress in the reinforcement. Users will need to calculate a reinforcement stress limit from the allowable crack width if they want to use this option. For Code limits see

1. AS3600 Clause 8.6.2 (Beams 200 MPa), 9.4.2 (Slabs 150 MPa)
2. ACI318 Clause 10.6.4, 18.10.2
3. BS8110 and CP65 Clause 2.2.3.4, 4.3.4.3 (Users need to calculate from crack widths)
4. Eurocode2 Clause 7.3.3, Tables 7.2N and 7.3N
5. SABS0100 Clause 3.2.3.3 (Users need to calculate an appropriate value)
6. CP2004 Clause 7.2.1 and 7.2.3
7. IS456/ IS1343 clause 43.1 and Annex F

For design codes which limit reinforcing bar spacing and/or bar size based on the stress in the reinforcement to achieve crack control for reinforced concrete members, RAPT will assume by default that the designer does not want to add extra reinforcement to provide crack control so the value of the reinforced Sections will default to 0. Crack control must then be achieved by limiting the bar spacing or bar size as appropriate.

If the designer wants to add extra reinforcement at service to reduce the reinforcement stress and thus provide a wider range of bar spacings or bar sizes from which to choose for crack control, then the stress limit can be applied here.

Relative Humidity

This dictates the shrinkage and creep characteristics. The drier the air, the greater the tendency to lose water from the concrete and the greater will be the creep and shrinkage.

To simulate AS3600 Environment Variables, the following values can be used

1. tropical near coastal (65% Humidity)
2. temperate (55% Humidity)
3. interior (50% Humidity)
4. arid (40% Humidity)

Average Temperature

The average temperature for the design area. RAPT modifies creep and shrinkage for temperature affects.

Prestress Losses Calculations based on

Prestress Losses This field allows the user to define the calculation method to be used for prestress losses calculations. The options available are

1. Program Default
2. AS3600
3. Eurocode2
4. ACI-209
5. Zia et al
6. CEB-FIP 1970
7. CEB-FIP 1978

The default value in the field is Program Default which will cause RAPT to use the internal default method for each design code. This is

1. AS3600:- AS3600 method
2. ACI318:- ACI-209 method
3. Eurocode2:- Eurocode2 method
4. BS8110:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
5. SABS0100:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
6. CP 65:- AS3600 method with adjusted creep and shrinkage values to suit local conditions
7. HK CP2004:- CEB-FIP 1970 method. The Hong Kong code requires an extra multiplying factor cs = 4 to allow for shrinkage values in local concretes. This is included only for the Hong Kong Code and is used no matter which shrinkage/creep model is selected for designs to the Hong Kong Code.
8. IS456/IS1343:- CEB-FIP 1970 method

A method which is acceptable in a designer's local design community and which is consistent with the concrete manufactured in that area should be used.

In the materials data, Basic Shrinkage Strain and Basic Creep Factor values are nominated for different concrete strengths. These values are only used in models that accept such a value. The models based on CEB-FIP 1970, CEB-FIP 1978 and Eurocode 2 have formulae which calculate the shrinkage and creep factors. For these models, the nominated values, if defined in the materials data, will be ignored and the internal formulae used will always be used.

Crack Width Calculations

This field allows a designer to nominate whether or not crack width calculations are to be performed for a member and allows a user-defined crack width limitation to be defined for designs to BS8110, SABS0100 and CP 65. If the value is left at DEFAULT, RAPT will use the BS8110 default values of

1. Reinforced Concrete members - .3mm
2. Prestressed concrete members - .2mm

for designs to BS8110, SABS0100 and CP 65.

For design to Eurocode, the designer should specify the crack width to be used here. RAPT will default to the values in Clause 7.3.1 Table 7.1N, adopting .3mm for reinforced concrete and unbonded prestressed members and .2mm for bonded prestressed members. The designer can over-ride these values. RAPT will use this value to determine maximum bar size and bar spacing limits from Clause 7.3.3 Tables 7.2N and 7.3N. if worried about stresses in the concrete at decompression, RAPT provides the service stress results for Permanent Loading in the output but it is up to the designer to adjust the amount of prestress to achieve the required stress limitations.

For all other design standards, crack width calculations will be done based on limiting bar sizes and spacing as appropriate.

Selection of actual crack widths for Design Codes other than BS8110, SABS0100, CP2004, IS456/IS1343 and CP 65 will have the same effect as selecting DEFAULT, as the other codes base their crack width calculations on stresses in the reinforcement rather than on actual crack widths.

Some codes allow the designer to ignore crack width calculations in some circumstances, eg AS3600 clause 8.6.1 and 9.4.1 and Eurocode 2. RAPT will always default to carrying out crack width calculations. If the designer considers that they are not necessary for a member, then this option can be used to force RAPT to ignore crack width calculations.

AS3600 Shrinkage and Temperature Reinforcement

The options available are

1. No Check
2. Minor Control
3. Moderate Control
4. Strong Control

These are described in AS3600 Clause 9.4.3. The amount of reinforcement required for each of these options is dependent on the % restraint for the slab. See 7.3.12 Reinforcement Layout to see how this is applied in Detailing Reinforcement.

The reinforcement required to satisfy this rule is not included in the Flexural design results or the Detailed Reinforcement results. It is only included in the reinforcement layout results data. This is because the calculation of this reinforcement includes the top and bottom reinforcement at a cross-section and it is dependent on the extent of the bars and the cut-off locations of the top and bottom bars. This is not known until the detailing stage. Otherwise it is not possible to logically apportion the required reinforcement to the logical faces of the member to make full use of the reinforcement required in the member for design. To check for the effects of this reinforcement on the design, get RAPT to add the designed reinforcement layout into RAPT as 7.2.6.4 User Defined Reinforcement and then run the file again.

Degree of Restraint in Primary Direction

This option allows the designer to define a percentage restraint for the slab in the design direction for calculation of minimum shrinkage and temperature reinforcement to AS3600. See 7.3.12 Reinforcement Layout to see how this is applied in Detailing Reinforcement.

Degree of Restraint in Secondary Direction

This option allows the designer to define a percentage restraint for the slab perpendicular to the design direction for calculation of minimum shrinkage and temperature reinforcement to AS3600. This is not being used at present in RAPT

Concrete Strength Gain Rate

The options are

1. S:- Slow hardening cements as defined by Eurocode 2
2. N:- Normal hardening cements as defined by Eurocode 2
3. R:- Rapid hardening cements as defined by Eurocode 2
4. ACI209:- requires two constants defined below.

The first 3 options use the method in Eurocode 2 clause 3.1.2 (6).

Concrete Strength Gain Constant a
Concrete Strength Gain Constant ß

If ACI209 has been selected as the method for calculating the concrete strength at any time, two constants are required to define the rate of gain of strength. The following formula is used to estimate the compressive strength at a given time t according to ACI- 209 (1978)

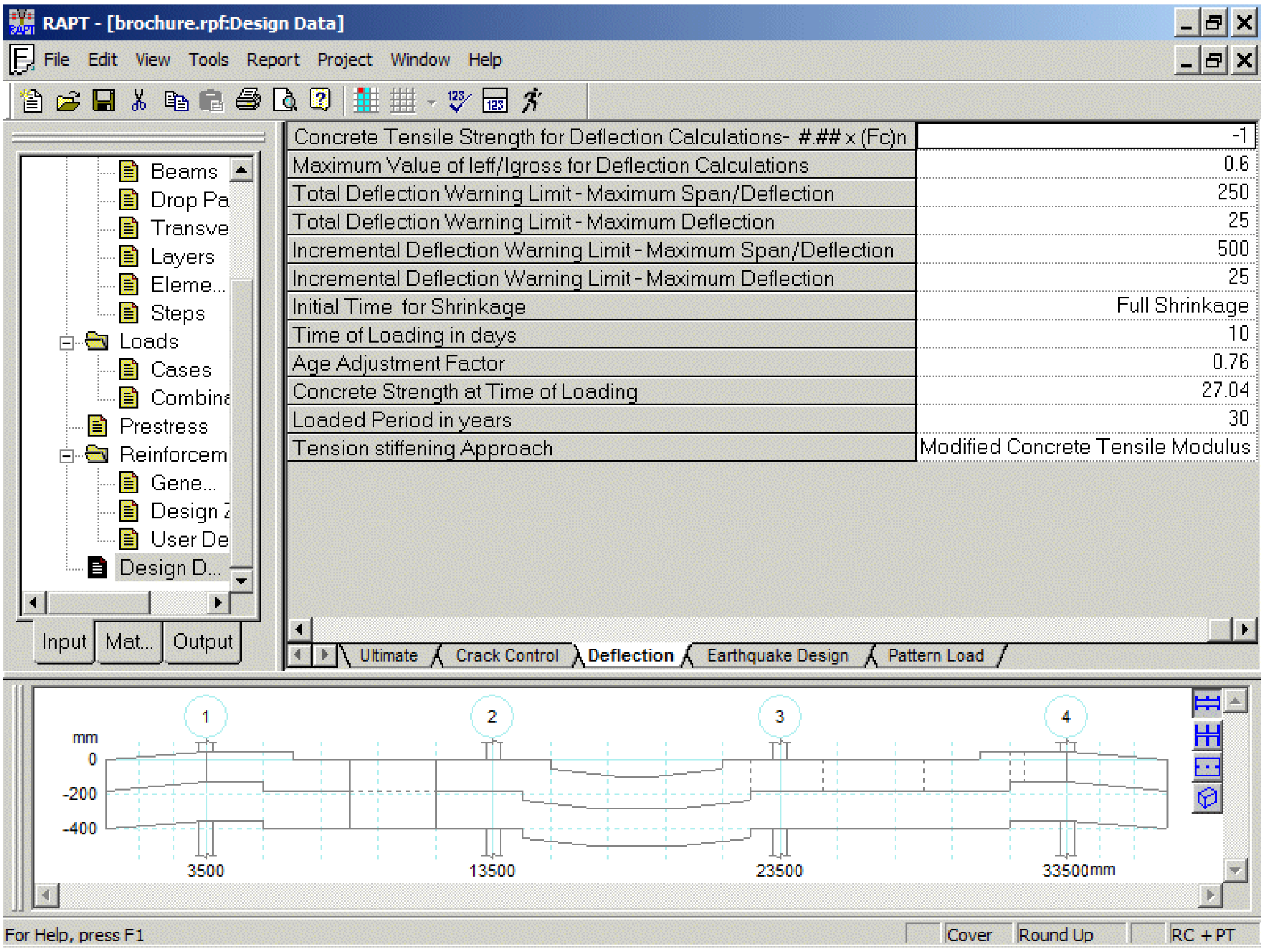
$f'_c = t / (a + ß * t) * f'_c(28)$

where

For normal Portland cement:
- For moist cured concrete: a = 4.0, ß = 0.85
- For steam cured concrete: a = 1.0, ß = 0.95

For high early strength cement:
- For moist cured concrete: a = 2.3, ß = 0.92
- For steam cured concrete: a = 0.7, ß = 0.98

7.2.7.3 Deflection Design Data



Data Definition

Concrete Tensile Strength for Deflection Calculations- #.## x (Fc)n

The designer may define a tensile strength of the concrete for deflection calculations. If the value of this factor is set to -1, RAPT will use the default lower bound tensile strength defined in 6.3 Material Properties for all designs and design codes except reinforced concrete members to BS8110 based codes (BS8110, CP65, SABS0100) for which a value of 1MPa at the level of the tension reinforcement will be used in accordance with Part 2 of BS8110. The value to be inserted here will be used as the factor in the tensile strength equation.

1. Eurocode 2:- factor * $fc^{2/3}$
2. Other codes:- factor * $fc^{1/2}$

In all cases, RAPT will reduce this value by the shrinkage restraint stress induced into the concrete by the bonded reinforcement.

Maximum Value of Ieff/Igross for Deflection Calculations

RAPT allows the user to specify a maximum $I_{effective}$ value. This value is specified as a fraction of I_{gross}. $I_{effective}$ is used by RAPT while calculating deflections. If a section is uncracked, RAPT will set $I_{effective} = I_{gross}$ unless specified in this input.

It is recommended by the authors that I_{gross} is not used by designers for deflection calculations unless a concrete member is completely unrestrained. The moments applied during the calculation of $I_{effective}$ normally only consider the vertically applied forces. Other forces such as restraint effects due to temperature, shrinkage and creep are not allowed for in the design moments. To make allowance for the effect of these additional forces on the cracked and effective inertia's in lightly loaded members, the authors recommend that designers restrict the $I_{effective}$ / I_{gross} ratio to 0.6 - 0.7.

Total Deflection Warning Limit - Maximum Span/Deflection

A limit at which RAPT will print warnings to the warning file telling the designer that the deflections have exceeded this value. RAPT will halve this figure when checking span / deflection ratios in cantilevers. If the value is set to zero RAPT will ignore the check.

Total Deflection Warning Limit - Maximum Deflection

RAPT will write a warning message in the warnings report if this value is exceeded. RAPT will use the same maximum deflection for supported spans and cantilevers. If the value is set to zero RAPT will ignore the check.

Incremental Deflection Warning Limit - Maximum Span/Deflection

A limit at which RAPT will print warnings to the warning file telling the designer that the deflections have exceeded this

value. RAPT will halve this figure when checking span / deflection ratios in cantilevers. If the value is set to zero RAPT will ignore the check.

Incremental Deflection Warning Limit - Maximum Deflection

RAPT will write a warning message in the warnings report if this value is exceeded. RAPT will use the same maximum deflection for supported spans and cantilevers. If the value is set to zero RAPT will ignore the check.

Initial Time for Shrinkage

RAPT calculates Total Long Term Deflection from the Time of Loading to the end of the Loaded Period allowing for one stage of loading. In this calculation the load is assumed to be applied at the Time of Loading and that is how the creep effects are calculated. But Shrinkage is not dependant on the Time of Loading, autogenous shrinkage begins almost immediately and is almost complete within 10-14 days while drying shrinkage begins as soon as the drying commences and continues at varying pace for about 30 years. So RAPT has always commenced the shrinkage calculation from time = 0, as that is the shrinkage that causes shrinkage deflection effects due to the restraint effects of bonded reinforcement over the life of the structure. The designer has had no control over this and thus has not been able to calculate the effects of multiple load stages.

However, sometimes designers will want to calculate deflection at various stages during the life of the structure to allow for different times of application of specific loads or supported elements. To do this the designer needs to be able to calculate the deflection that occurs over a specific period or multiple periods with different loads applied. To do this, the designer needs to be able to specify the start time for shrinkage for that period as well as for creep. To enable this, we have given 2 options for Initial Time for Shrinkage.

1. Full Shrinkage - Shrinkage is measured from day zero to the end of the loaded period. This is the normal situation for the calculation of the Total Long Term Deflection over the life of the structure. If manually doing stage loading calculations to check for the effects of the application of loads at different times, the first stage should use this setting with the Time of Loading set to the end of the first stage of the deflection calculations to fully account for the early shrinkage in the first stage.
2. Incremental Shrinkage - Shrinkage is measured from the Time of Loading to the end of the loaded period. If manually doing stage loading calculations to check for the effects of the application of loads at different times, set the Time of Loading to define the start of this stage and the Loaded Period as the time from day of pour to the end of this loading stage. The calculations for the first stage should use the Full Shrinkage option to fully account for the early shrinkage in the first stage.

Time of Loading in days

Time at which the structure is first loaded. This is used for deflection calculations to determine long term creep and shrinkage coefficients. For Reinforced Concrete members, it is also the time at which the Transfer Deflection is calculated.

Age Adjustment Factor

This factor is used to adjust the effective modulus to allow for the time of loading and rate of application of load/stress. RAPT calculates this factor based on

$$\chi(t,\tau_0) = \frac{\tau_0^{\,0.5}}{\left(1+\tau_0^{\,0.5}\right)} \qquad \text{Ageing coefficient}$$

$$\tau_0 \;=\; \text{Initial time of loading}$$

See Section T7.7.5 for more details.

Concrete Strength at Time of Loading

RAPT calculates this value based on the strength gain formula using the strength gain constants defined in the 6.3 material properties. Users can modify the calculated value if required.

Loaded Period in years

The time over which the structure is expected to be loaded in years, measured from the time of pouring the concrete. This figure is defaulted to 30 years as creep and shrinkage effects are minimal after this period. The long term losses are calculated at this time. The long term and incremental deflections are also calculated at this time. Estimates of effects at shorter times may be calculated by nominating the time at which calculations are required e.g. the expected deflection at 6 months can be calculated by substituting 6 months as the loaded period (.5 years).

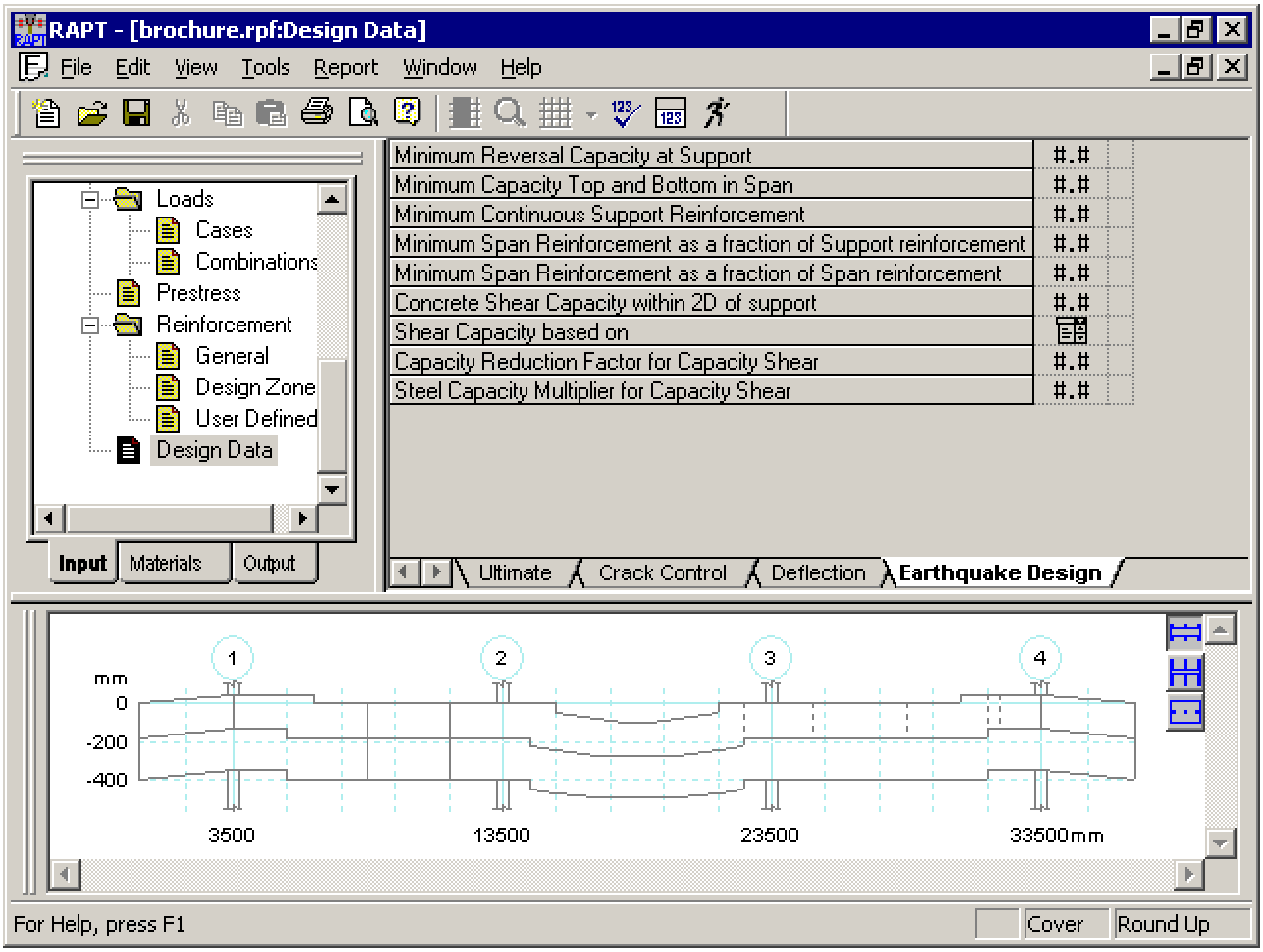
Tension stiffening Approach

RAPT can estimate the tension stiffening effect in concrete in three ways. Users can specify which method they would like RAPT to use. The three approaches are

1. Branson's Formula
2. Eurocode 2 Formula
3. Modified Concrete Tensile Modulus Method. RAPT's default and preferred method. We prefer this method as it actually tries to estimate a tension stiffening effect in the actual cracked section calculations rather than fudging a guess at the effect based on an averaging formula.

See Theory Section T7.7.7. for more detailed information on these methods.

7.2.7.4 Earthquake Design Data

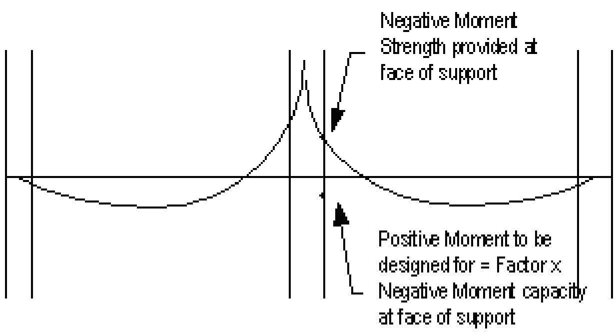


Allows the designer to specify the special requirements for earthquake design and detailing. This screen is only available if one of the earthquake design option is selected in the 7.2.2 General Data screen.

Data Definition

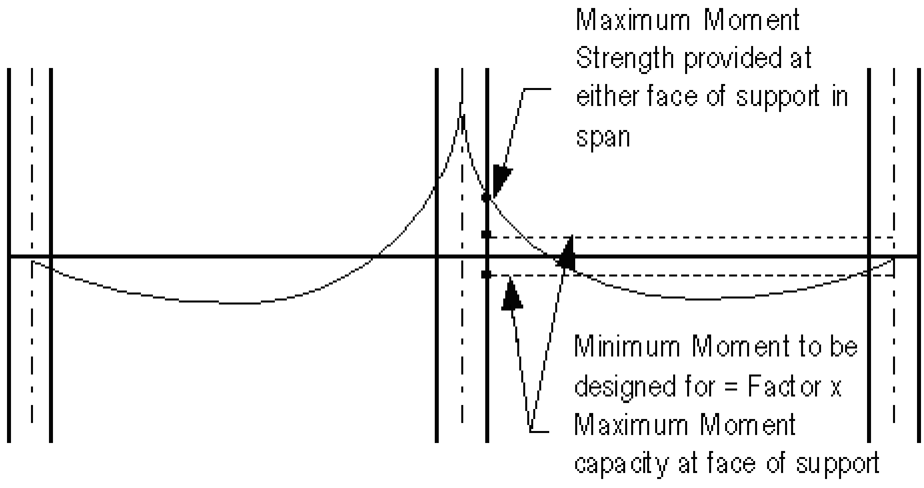
Min Reversal Capacity at Support

Some design codes require that the a fraction of the ultimate capacity at the critical section at the support be provided on the reverse face of the member for certain member types. At the critical sections at each support, this factor defines the minimum design moment for each face of the member. RAPT will look at the envelope of moments at the critical section and design for this fraction of the higher of the two moments as the minimum moment causing tension on each faces of the member. Even if the bending moment diagram indicates there is no tension stress on a face, this minimum moment will be designed for. The default value is the value from 5.5 Design Standard->Design->Earthquake Design.

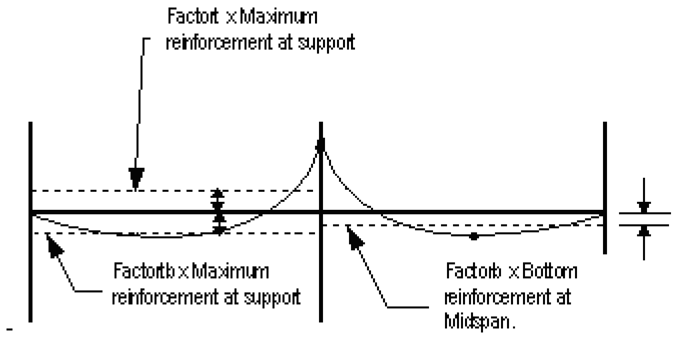


Min Capacity Top and Bottom in Span

Some design codes require that the a fraction of the ultimate capacity at the critical section at the support be provided at all points in both faces of the member for certain member types. At both faces of every design point in the member, the minimum design moment will be set to this factor multiplied by the largest support critical section moment. This will provide a minimum amount of continuous reinforcement in both faces of the member. The default value is the value from 5.5 Design Standard->Design->Earthquake Design.



Min Continuous Top Reinforcement
Some design codes require that the a fraction of the reinforcement required at the critical sections at the supports in a span be provided as continuous reinforcement on that face of the member for certain member types (Factort below). RAPT will ensure that the reinforcement at each design section in a span on the support face is at least equal to this factor times the larger area of reinforcement at each end of the span. The default value is the value from 5.5 Design Standard->Design->Earthquake Design.



Reinforcement Diagram

Min Bottom Reinforcement as a fraction of top reinforcement
Some design codes require that the a fraction of the reinforcement required at the critical sections at the supports in a span be provided as continuous reinforcement on the other face of the member for certain member types (Factortb above). RAPT will ensure that the reinforcement at each design section in a span on the span face is at least equal to this factor times the larger area of support reinforcement at each end of the span. The default value is the value from 5.5 Design Standard->Design->Earthquake Design.

Min Bottom Reinforcement as a fraction of bottom reinforcement
Some design codes require that the a fraction of the maximum reinforcement required in a span be provided as continuous reinforcement on that face of the member for certain member types (Factorb above). RAPT will ensure that the reinforcement at each design section in a span on the span face is at least equal to this factor times the largest area of span reinforcement required in the span. The default value is the value from 5.5 Design Standard->Design->Earthquake Design.

Concrete Shear Capacity within 2D of support
Some design codes specify that the concrete shear capacity should be reduced or ignored within 2D of the support. Input the factor to multiple the concrete shear capacity by here. It may be a requirement of your design code that the shear calculations be continued to the face of the support in this instance. If so, an option has been included to allow the designer to set this in 7.2.7.1 Design Data - Ultimate - Beam Shear Critical Section Location without including Shear Enhancement. To include Shear Enhancement, use 7.2.7.1 Design Data - Ultimate - Shear Enhancement at Supports. This will automatically include continuing shear calculations to the face of the support.

Shear Capacity based on
Some design codes specify that shear design be based on the capacity shear design in some earthquake members. This is normally required in areas of high seismic risk. RAPT gives the user the option of selecting

1. Applied Shear - Shear design is based on the elastic shear values calculated in the analysis.
2. Capacity Shear - Shear design is based on plastic shear values calculated from the Capacity Ultimate Moment Strength and the applied shear diagrams. This design option has not been added to RAPT at this stage.

If capacity design for shear is selected, the next two data fields are available.

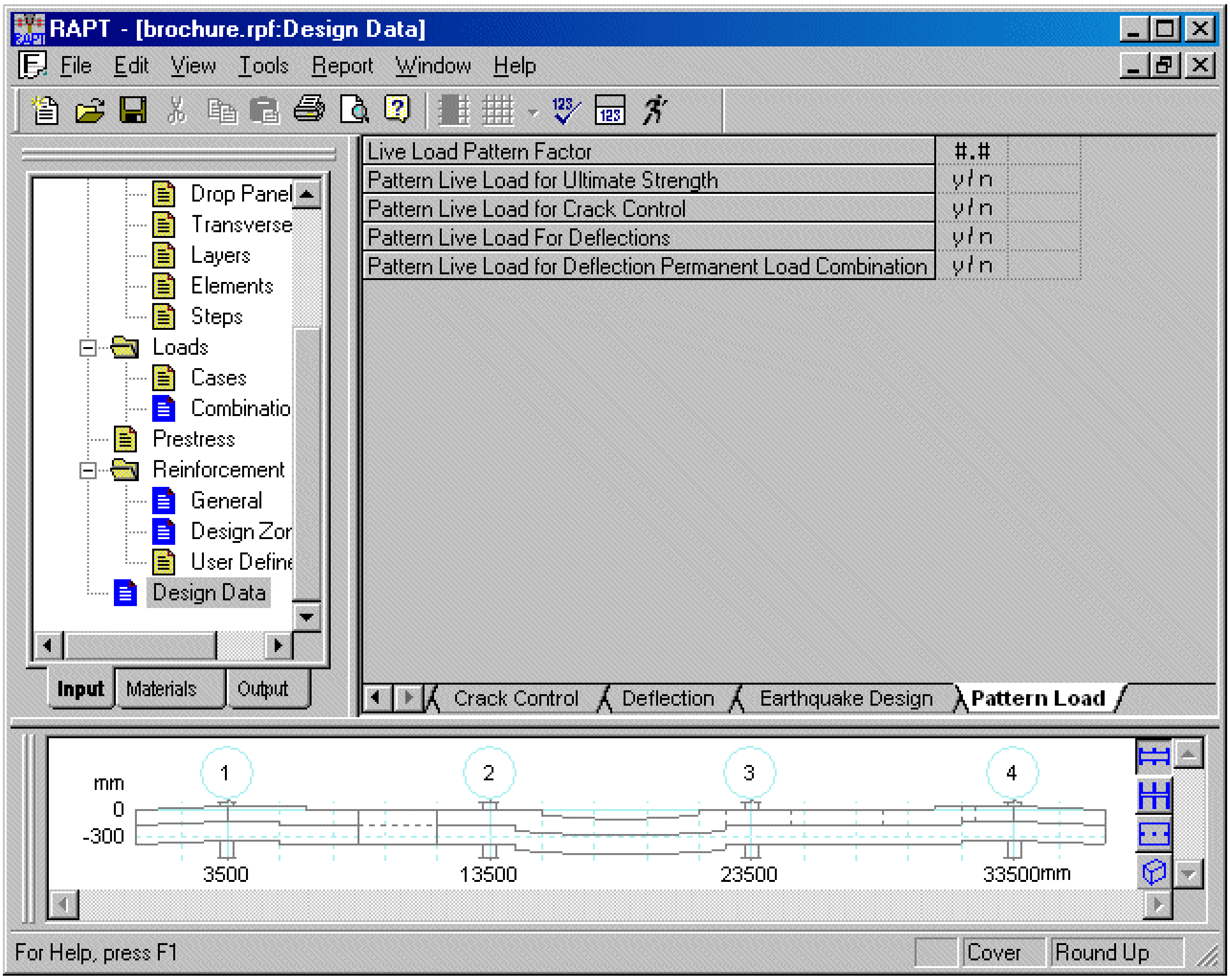
Capacity Reduction Factor for Capacity Shear

When doing a capacity shear design, RAPT will use this factor as the capacity reduction factor in place of the normal ultimate strength capacity reduction factor defined in 7.2.7.1 Design Data.

Steel Capacity Multiplier for Capacity Shear

When doing a capacity shear design, RAPT will use this factor to increase the reinforcing steel strength to a capacity strength rather than the normal yield strength defined in material properties. The Yield Strength will be multiplied by this factor.

7.2.7.5 Pattern Load Design Data



Allows the designer to specify the special requirements for pattern load analysis. This screen is only available if the pattern load design option is selected in the 7.2.2 General Data screen. Patterning of live load in RAPT is done in different ways depending on the design code selected. For all design codes available in RAPT the patterning of live loads and combination of the patterned live loads with other loads is done strictly in accordance with the provisions of that design code. It is not always necessary to pattern live load for all areas of design. The designer should determine this in accordance with the requirements of the design code being used. RAPT allows the designer to select which areas of design will use the Patterning of Live Load. The default will always be to pattern live load in all areas of design using the default pattern load factor defined in the relevant Design Code Default file. If this value is zero, RAPT will use a pattern live load factor of 1.0.

Partial Load Patterning Analysis for Shear

RAPT has also introduced automatic partial live load patterning analysis for shear. There is no user option to turn this calculation off. It is automatically applied to all load cases of the types Live Load and Alternate Live Load.

The normal assumption for live load design loading is for a uniformly distributed loading. This is hardly ever the case with partial loading being applied to some areas and no loading to others. Nearly all structures experience this type of random/partial live loading. A uniform loading will result in a value of live load shear at one point in any span of zero increasing linearly to a maximum at the supports. However, a partial loading, using the same value of load, could result in significant shear force at that zero shear point. Many design codes specify a minimum live load shear value (AS3600 specifies 25%) to allow for this discrepancy in the way we define live loads. RAPT's partial load patterning for shear overcomes this discrepancy. For the Ultimate Shear Envelope, RAPT analyses a large number of partial live load cases and calculates a design shear envelope for them. This envelope is then combined with the other load cases as required and the results compared to the shears from the normal loading patterns to determine the governing shears and associated moments at each design cross-section. All Live Load and Alternate Live Load cases will be analysed in this way in RAPT.

Data Definition

Live Load Pattern Factor

The Pattern Load factor is entered as a decimal (fraction of 1). The decimal refers to the amount of Live Load used in the pattern combinations i.e. 0.75 indicates that 75% of the Live Load is used in pattern combinations.

Pattern Live Load for Ultimate Strength Design

The designer can select to have the live load case patterned for ultimate flexural and shear strength design.

Pattern Live Load for Crack Control

The designer can select to have the live load case patterned for serviceability crack control design.

Pattern Live Load for Deflections

The designer can select to have the live load case patterned for deflection design.

Pattern Live Load for Deflection Permanent Load Combination

The designer can select to have the live load case patterned for permanent load deflection calculations. In most cases the permanent live load is assumed to be applied over the full length of the frame so patterning of permanent live load would be illogical and conservative. This would lead to higher creep deflections. In some cases, especially with storage buildings, factories and libraries, it may be logical to assume a pattern of the permanent live load for deflections. In cases where there is a known pattern of loading this should be used.

7.3 Frame Design / Results

7.3.1 Viewing Output Results

The input data and results data from a RAPT frame can be viewed and printed from the output tab on the tree. The Output Tree is used to control both what data is to be viewed on the screen and the data that is to be used to create a report. It is accessible by clicking the *Output* tab on the tree when a RAPT frame file is open.

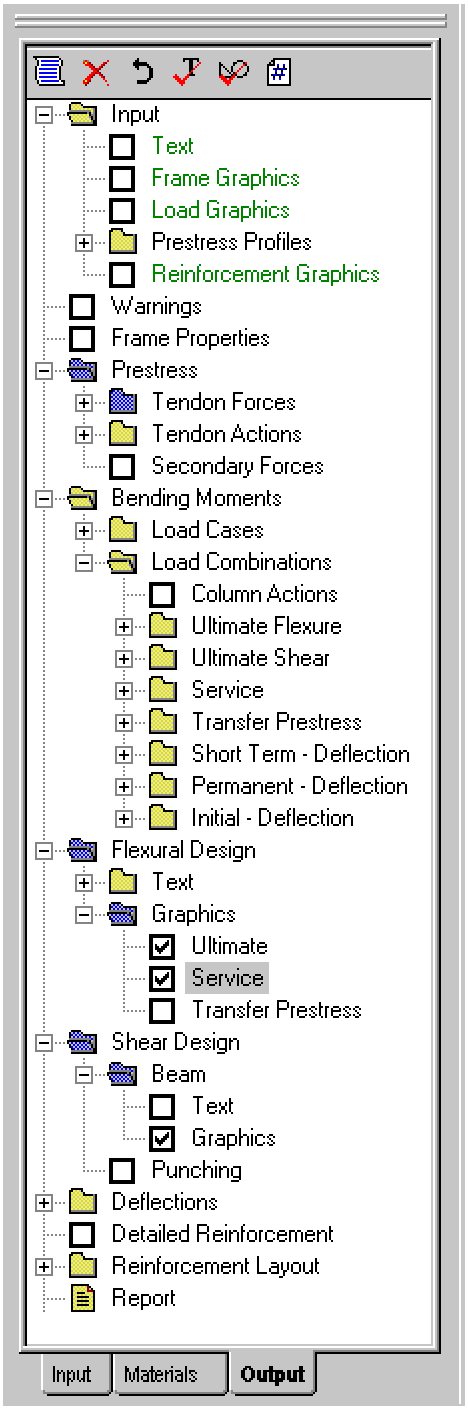
The Output Tree provides access to output in the form of text views and, in most cases, graphics views for all of the major areas of design. As well as the raw data in these views on which designers can make design decisions, RAPT will also make design comments in some areas to let the designer know what happened in some areas of the design.

Design Comments

In several areas of the output, Design Comments are attached to the output data. Design comments are not necessarily saying that there is a problem with the design. They are pointing out to the designer certain aspects of the results e.g. if compression reinforcement is required for ductility or if crack control reinforcement is required, RAPT will point this out in a comment to emphasize to the designer something that may not be immediately apparent by looking at the text or graphical output. The design has been done to the code rules and the results are acceptable but the designer may have a preference for another solution such as a deeper section, higher concrete strength or more prestress in the crack control example to remove this extra reinforcement and possibly produce a more economical design

When viewing individual areas of results on the screen, the design comments will be available with the text output. The associated graphical output will have a comment at the bottom that design comments are available for this area of results. When creating a report, If the text item is selected for a topic with design comments, the Design Comments will be placed with the text output and if the text output and comments are span based, will be placed at the end of each spans results. If graphics output only is selected for such a topic, the design comments will be placed after the graphics and, if span based, will be grouped in spans.

7.3.1.1 Output Tree



The major areas of design covered are

1. Input Data
2. Prestress Profiles
3. Warnings (text only)
4. Frame Properties (text only)
5. Prestress Forces and Actions
6. Bending Moments and Shear
7. Flexural Design
8. Shear design
9. Deflections
10. Detailed Reinforcement (text only)
11. Reinforcement Layout

In many cases there are multiple sub-reports for each of these topics which can be selected individually. Only the tree nodes with a check box to their left are selectable to produce text or graphics output. The other nodes can be expanded to give access to the selectable nodes that they control.

For a new data file, the tree structure will initially be opened according to the settings in 4.2.5 Output Report Settings. Nodes on the tree with a + or - sign can be open/closed by clicking on the sign or double clicking on the text of the node or by pressing the Right Arrow or Left Arrow keys. Up Arrow, Down Arrow and Page Up and Page Down keys can be used to move through the tree.

Node Colours
The tree node options will be shown in one of 3 colours:
1 Black - node is selectable for both on screen viewing and report creation and data is available
2 Green - node represents input data and is selectable for report creation only. Screen viewing is available in the Input tab on the tree.
3 Red - node is not selectable as there has been an error in the calculations at or before this point and no output is available. Output will only be available for the nodes before the first red node where calculations have been successful.

The data can be presented on the screen in 2 basic ways

The results for a single tree node.
The results for a single selectable node can be viewed and interrogated in the View Window by using a single left mouse click or moving to the node with the cursor and pressing return..

Selecting a text node will fill the text window with the results for that node. There is only one output text window. Selecting another node will replace the information currently in the text window with the results for the new node selected.

Selecting a graphics node will open a graphics window with the results for that node. Multiple graphics windows can be open at the same time and can be viewed simultaneously with each other and the text window by clicking the Restore Down button for the window. Either by arranging the required windows with the mouse and/or using the Window Menu options (Cascade, Tile, Arrange Icons) the user can then arrange the required open windows within the view area (in Win98 and previous a maximum of 4 windows can be open at once. RAPT will manage this for a single instance to the program running).

The graphics windows can be manipulated and interrogated to view the data in different ways by the user in this method. See 7.3.1.2 Output Graphics Windows for details.

The contents of a single window can be printed Print Preview and Print icons  or by selecting the Print or Print Preview options from the File menu.

If the current view is an individual text or graphics output view, Ctrl + Page Down / Ctrl + Page Up will open up the view for the next / previous checked output item.

A continuous report compiled from a list of selected nodes.
The output tree toolbar allows user to create a continuous report. This report is initially created in a graphic window and can be viewed there by the user. To move through the report the user can use

1. The standard movement keys (arrows, Page Up, Page Down).
2. Vertical and horizontal Slider bars

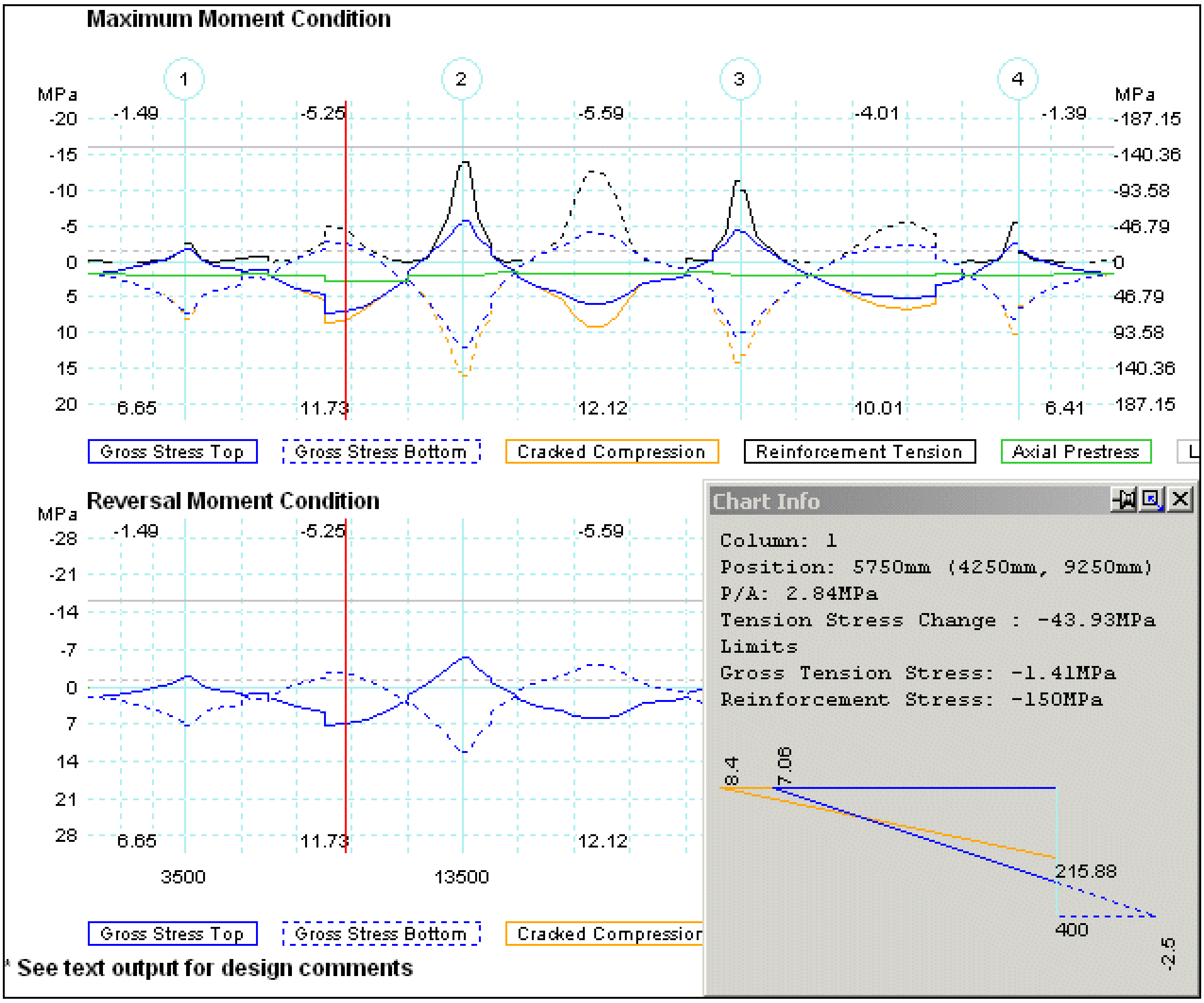
3. RIGHT click the text area of any selected item will move the current report position to the beginning of that right-clicked item. This function serves as a bookmark so that the user will be able to find the relevant sub-report quickly.
4. Ctrl + Page Down / Ctrl + Page Up will move the current report position to the beginning of the next / previous checked output item.

NOTE: the methods mentioned above in The results for a single tree node do not apply for viewing continuous reports.

The report can be printed using the Print Preview and Print icons or by selecting the Print or Print Preview options from the File menu.

For details on creating a report see 7.3.1.3 Creating A Report.

7.3.1.2 Viewing Output Results - Graphics Windows



When a graphics view is selected to view on the screen (not as part of a report) it is initially fitted to the View Window (as shown here but with the tree showing).

The horizontal axis is in length units and the locations of support locations, measured from the left end of the frame, are nominated at the bottom of the bottom graph in the window. Support numbers are nominated at the top of the top graph in the window. The main grid lines at supports are shown as a solid line and each span has 4 secondary dotted lines breaking each span into fifths.

The vertical axis varies according to the type of data being shown and the unit type and scale are shown on the left side axis. In some cases, e.g. Flexural Design->Graphic->Service shown here, two different scales are employed to show different data. In this case, the second scale and unit type are shown on the right side axis. In this case, the left axis is for concrete stresses and the right axis for reinforcement stresses which are normally an order of magnitude larger. The horizontal axis is shown as a solid line and the other scale lines are shown dotted. The left scale is calculated to give round numbers. If a right scale is required, it has to fit to the same grid lines so the scale will not be rounded.

Graph Legend
Underneath each graph, there is a legend which names each line in the graph and shows the line colour and line type (solid, dotted) Gross Stress Top. See 4.2.4 Line Options for information on setting line types.

Graph Values
When a cursor position is selected by the user, values can be placed on the graph for any plotted lines. For a single line graph, e.g. load case moment/shear diagram, the values will always be nominated at the cursor location. For graphs with more than one line, the values of each line at the cursor can be requested using the legend. Each legend name can be selected by left clicking the mouse within the legend boundary. When another name is selected with a left click the first will be turned off. The currently selected legend name is shown with a grey background Gross Stress Top while unselected legend names have a clear background Gross Stress Top. This legend name will remain selected until cleared by clicking it again or clicking another name in the legend list. The cursor can be moved to any location within the graph and the values selected will be shown.

Multiple legend names can be selected by using Ctrl + Left Mouse Click. All selected legend names will be cleared when a plain Left Mouse Click is made on any legend name in the list. If the newly clicked cell is currently unselected, it will be selected and all selected cells will be cleared. If the newly clicked cell is already selected then all cells will be cleared. In the case shown above 5 lines have been selected. The values are plotted at their correct location on the

curves. If numbers overlap, no attempt is made to move them from this location and some of the values may be unintelligible if too close together. The Chart Info button can then be selected to list all of the data on this graph at this location.

When at lease one legend name is selected, the horizontal location of the plotted point will be displayed at the top of the cursor line. This location is measured from the left end of the current span.

Critical Values

In most graphs, values will be nominated at the top and/or the bottom of the graph in each span. These are normally maximum values of a design parameter in that span. For each different graph type, the discussion on that type defines the values and their possible significance.

Graphics Toolbar

A special graphics toolbar is provided to assist with viewing the data. This toolbar will only be available when program focus is in the Graphics Window. The buttons available are

Zoom (Ctrl + Z). This button will toggle between full screen mode and span zoom mode for the graphics in a window. In span zoom mode, the current span will be shown with the half span either side (if a cantilever is the previous or next span, the full cantilever will show) scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window.

Move to next item (Ctrl + Right Arrow). The item referred to is normally a Span, except for Load and Reinforcement screens where it is a Load or a Reinforcing Bar. In Full Screen Mode the cursor will be moved to the next support line. In Span Zoom mode, the next span will move to the centre of the Window and the cursor will move to the start of that span.

Move to next point (Shift + Right Arrow). The cursor is moved to the next design point to the right. If it reaches the right end of the span, the next span is moved to the centre of the Window.

Move to previous point (Shift + Left Arrow). The cursor is moved to the next design point to the left. If it reaches the left end of the span, the previous span is moved to the centre of the Window.

Move to previous item (span) (Ctrl + Left Arrow). The item referred to is normally a Span, except for Load and Reinforcement screens where it is a Load or a Reinforcing Bar. In Full Screen Mode the cursor will be moved to the previous support line. In Span Zoom mode, the previous span will move to the centre of the Window and the cursor will move to the end of that span.

Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

Then the user can then move around the graph using the Scroll Bars or the movement keys. and buttons will still move the cursor to the next and previous points. If the point is outside the viewable area it, the area of graph shown will adjust automatically to position the requested point near the left of the View Window.

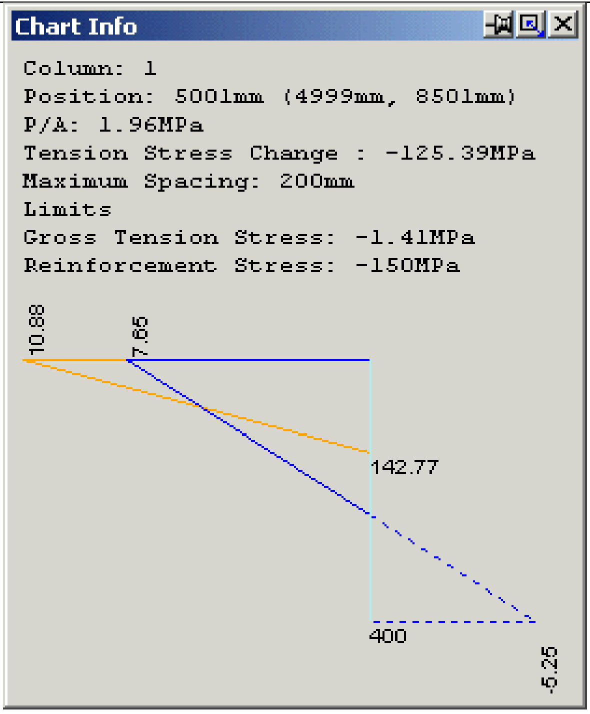
Clicking this button again or on the or buttons will return the Window to Full Screen Mode.

Clicking will change the mode to Span Zoom Mode in the span in which the cursor is positioned in the Select Zoom mode.

Toggle between Run Y Range and Local Y Range. In the Bending Moment/Shear output screens, the vertical scale is set by default to suit the largest moment and largest shear values recorded in all of the load cases and load combinations. If this icon is depressed, the vertical scale for the case/combination being viewed is set to suit the largest moment and largest shear values recorded in the diagrams only being shown for this case/combination. In this situation, the scale will be different for every load case and load combination.

The setting is remembered until the current RAPT session is closed. On starting a new RAPT session the default of "Run Y Range" will be set again.

Show/Hide information dialog (Ctrl + I) When this button is selected, a dialog will appear which defines the information which controls the current selected item (point at the cursor or selected load or reinforcing bar). In most cases it will give the numbers which have been used to plot the graphics at this point or item.



Position:- It will always define the Position from the nearest support line to the left of the point with the distance to the next support line and the distance to the left end of the frame (centre of left column if no cantilever) shown in brackets

Detailed discussion on what is shown in each graphic window is available in the relative discussions of the different areas of calculations and results.

The cursor will stay in the same location as different graphic Windows are opened for different sets of data and the information dialog will automatically update the contents with the information for this location on the new graph.

Clicking the mouse button inside the information dialog area will bring the dialog into focus. It will not move the cursor to the clicked position below the dialog. To do so, the dialog needs to be moved first by left click and hold the mouse in the title bar and dragging the mouse to the new position for the dialog.

Chart Info Dialog Buttons

Close Dialog

The dialog can be closed using the button on the title bar or by pressing again.

Auto Reposition Disabled
Auto Reposition Enabled

The dialog can be forced to stay in one position by pressing the button on the title bar. Generally the dialog will move horizontally to the opposite corner of the Window if the cursor position is under the dialog. When button this is pressed, it will change to and the dialog will not move and the cursor may move under the dialog without repositioning the dialog on the screen. If a window that does not require the dialog is in focus, it will hide itself until required again.

Auto Resize Disabled
Auto Resize Enabled

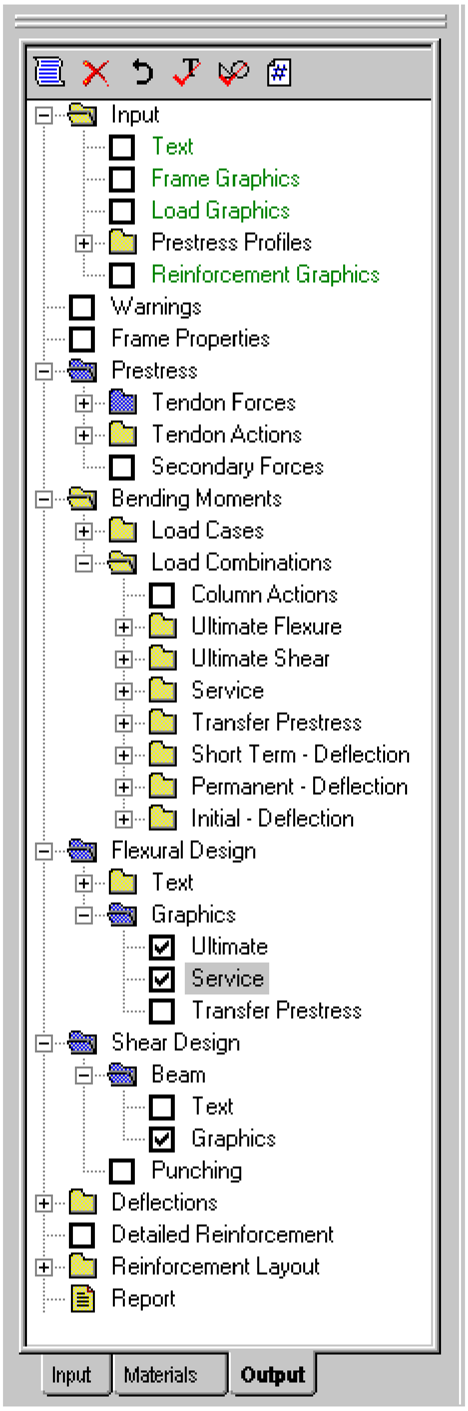
The dialog can be forced to stay the same size by pressing the button on the title bar. Generally the dialog will resize to suit the information to be printed into it when a new position or view is selected. When the button this is pressed, it will change to and the dialog will not resize and the data may not fit into the dialog or may only fill part of the dialog. If a window that does not require the dialog is in focus, it will hide itself until required again.

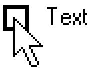
Design Comments

When viewing graphical output, if design comments exist for that area of the results, a comment will be added at the bottom that design comments are available for this area of results (see output graphics above). These can be viewed in the corresponding text output, normally for the span to which the comments refer.

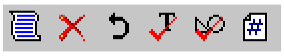
When creating a report, If the text item is selected for a topic with design comments, the Design Comments will be placed with the text output and if the text output and comments are span based, will be placed at the end of each spans results. If graphics output only is selected for such a topic, the design comments will be placed after the graphics and, if span based, will be grouped in spans.

7.3.1.3 Creating A Report



Reports are created from the items in the tree that are selected (see Flexural Design -> Graphics -> Ultimate Check Box in the diagram to the left). To select/unselect a Check Box, left click the check box on the left of its tree node, 

, or press the Space Bar when the cursor is on that items text.



The Toolbar at the top of the Output Tree provides several methods for selecting report settings.

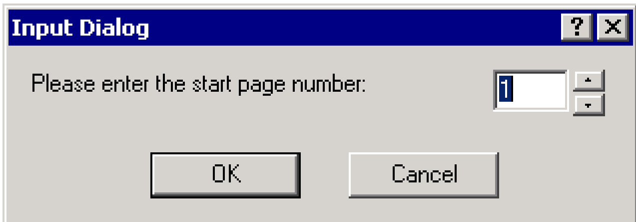
 Clears all selected report items.

 Select report items according to the saved format in 4.2.5 Output Report Settings

 Select all Text report items - any graphics items that are already selected will remain selected and all available text items will be selected

 Select all Graphics report items - any text items that are already selected will remain selected and all available graphics items will be selected

 Nominate the starting page number for this report. On clicking this Toolbar Icon, the dialog below will be presented asking for the starting page number. The default value is 1. This value will be reset to the default whenever a new file is created or an existing file is opened. It is not necessary to select this icon to start the page numbering at 1.



If the page numbering starts at 1, no header will be placed on the first sheet. The page numbering will be in the form XX/YY where XX is the current page and YY is the total number of pages in the report for this file. If the page numbering starts at any number other than 1, a header will be placed on the first sheet of this section of the report. The page numbering will be in the form XX-ZZ where XX is the current page and ZZ is the number of the last page in the report for this file.

The user can also perform these operations by right-clicking any blank area in the output tree and selecting from the popup menu, or clicking the *Report* Menu.

Note that the standard tree folder colour is yellow. When an item within a folder is selected to be part of the report, the folder colour changes to blue to indicate that there may be hidden selections within a folder if the folder is closed.

The standard tree node font colour is black, which indicates that this node is clickable and an associated sub-report view will show if it is clicked. A green tree node font colour indicates that there is no sub-report view associate with it. A red tree node font colour indicates that the associated sub-report view is not available due to calculation errors.

 Once the required items are selected, the user can create a report using this toolbar button. Alternately the user can also create a report by right-clicking any blank area in the output tree and selecting from the popup menu, or clicking the *Report* Menu and selecting *Report*, or by clicking *Report* node at the bottom of the tree.

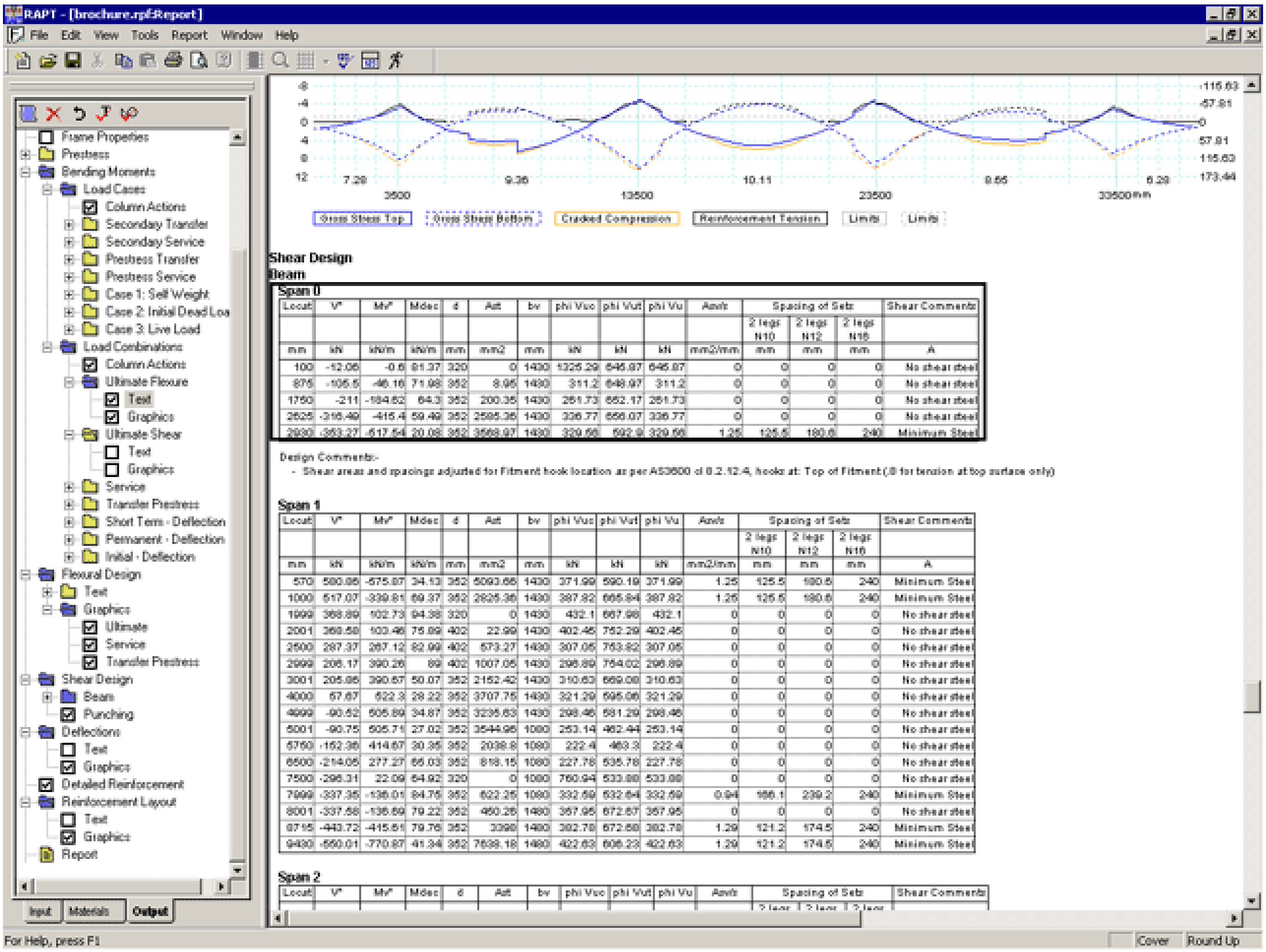
Design Comments

In several areas of the output, Design Comments are attached to the output data. When creating a report, If the text item is selected for a topic with design comments, the Design Comments will be placed with the text output and if the text output and comments are span based, will be placed at the end of each spans results. If graphics output only is selected for such a topic, the design comments will be placed after the graphics and, if span based, will be grouped in spans.

Printing a Report

Once the report has been created in the Viewing Window, it can be printed using the Print and Print Preview icons in the main toolbar or by selecting the *Print* or *Print Preview* options from the *File* menu. This can be printed to any printer in the Windows Printer list including file creating utilities such as Adobe Acrobat as long as the user's computer has the necessary drivers for that printer or utility.

7.3.1.4 Transferring Output Data to other Programs



The data in a table or grid or the text comments in the output window (graphics cannot be copied) can be copied to the Windows Clipboard and pasted to other programs which allow data input such as spreadsheet/database (e.g. Microsoft Excel, etc) and text editing programs (e.g. Microsoft Word, Notepad etc).

Complete tables of data must be copied. To copy a table, select the table from the Output Window using the left mouse button. A solid black border will surround the selected table of data as shown above. Groups of tables can be selected by using Ctrl + Left Mouse Click on subsequent tables of data. Clicking Ctrl + Left Mouse Click again on a selected table will unselect that table. Copy the selected tables using the standard Windows Copy methods (Ctrl + C or ). All of the data in each selected table will be copied to the Clipboard along with the column and row headers.

The contents of the Clipboard can then be inserted into the other program using the standard Windows Paste methods (Ctrl + V or ). The data will normally be inserted in the same tabular format as the original it is copied from, including header text fields.

7.3.2 Input

7.3.2.1 Input Text

The input text data can be viewed in the Input screens by selecting the Input Tab on the Control tree and then selecting the area of input to view from the tree. Separate viewing is not available in the output view.

This item can be selected for inclusion in a report by clicking the Check Box beside the name. The text output for all of the relevant areas of input will be included in the report along with any materials properties that are used in the frame.

7.3.2.2 Frame Graphics

The Frame Graphical views can be viewed in the Input screens by selecting the Input Tab on the Control tree and then selecting the area of Frame input to view from the tree. Separate viewing is not available in the output view.

This item can be selected for inclusion in a report by clicking the Check Box beside the name. The frame plan, and two elevation views will be included in the report.

7.3.2.3 Load Graphics

The Frame Load Graphical views can be viewed in the Input screens by selecting the Input Tab on the Control tree and then selecting the Loads-Cases option from the input tree and selecting the individual load case to be viewed from the control grid. Separate viewing is not available in the output view.

This item can be selected for inclusion in a report by clicking the Check Box beside the name. The graphics views of all of the load cases will be included in the report in load case order.

7.3.2.4 Prestress Profiles

7.3.2.4.1 Prestress Profiles

RAPT calculates profile heights to the underside of the duct (centreline at anchorages) at equal centres over the parabola length in each span. The calculation of these profile shapes allows for transition curves at all locations where the profile shape changes abruptly and ensures that the curvatures at any point on the profile are not greater than that calculated from the defined radius. In 4.2 User Preferences->User Options, the user can select rounding options for the spacing of the profile points. In 7.2.5.1 Prestress->Drape Locations, the user can nominate different spacings for each span and also can vary the reverse curve radius. The default values for these can be set for each design code in 5.3 Design Code->Prestress.

Text View

The text view gives a table of profile information in each design strip for each tendon in the member. For each tendon the following information is provided:-

In the header, the expected gross extension of the tendon and the size and type of the tendon.

Two tables of data are provided for each tendon in each strip.

1. Profile heights:- Defines the tendon profile heights to underside of duct at the nominated spacings
   1. Locat:- The location of the profile point from the left end of the span.
   2. Height:- The height of the tendon from the soffit of the member (from the closer soffit level if there is a sudden step at the point) and ignoring drop panels for two-way slabs rounded to the nearest 5mm or 1/8" depending on unit type selected. The height recorded at the anchorages is to the centreline.
2. Critical Profile Points:- Defines the location and profile height at transition points and peak profile points, including the ends of the tendon in each span, on the tendon profile.
   1. Locat:- The location of the critical profile point from the left end of the span.
   2. Height:- The height of the tendon from the top of slab datum level to the underside of the duct, without rounding.

Tendon 1 : - Gross Extension: 163.2mm - 3 - 5/12.7

Profile Heights

Span 0	Locat	mm	0	875	1750	2625	3500						
	Height	mm	235	235	265	310	360						
Span 1	Locat	mm	0	1000	2000	3000	4000	5000	6000	7000	8000	9000	10000
	Height	mm	360	285	205	150	115	100	115	150	205	285	360
Span 2	Locat	mm	0	1000	2000	3000	4000	5000	6000	7000	8000	9000	10000
	Height	mm	360	280	185	115	75	60	75	115	185	280	360
Span 3	Locat	mm	0	1000	2000								
	Height	mm	360	275	235								

Critical Profile Points

Span 0	Locat	mm	0	3112.7	3500		
	Height	mm	-174.5	-53.9	-39		
Span 1	Locat	mm	0	519	5000	9481	10000
	Height	mm	-39	-65.9	-298.5	-65.9	-39
Span 2	Locat	mm	0	597.4	5000	9402.6	10000
	Height	mm	-39	-74.6	-337.6	-74.6	-39
Span 3	Locat	mm	0	677.8	2000		
	Height	mm	-39	-84.9	-174.5		

Tendon 2 : - Gross Extension: 67.3mm - 3 - 5/12.7

Profile Heights

Span 3	Locat	mm	3500	4428.6	5357.1	6285.7	7214.3	8142.9	9071.4	10000
	Height	mm	235	130	120	135	170	225	295	360
Span 4	Locat	mm	0	875	1750	2625	3500			
	Height	mm	360	310	265	235	235			

Critical Profile Points

Span 3	Locat	mm	3500	5000	9510.4	10000
	Height	mm	-174.5	-283.8	-62.9	-39
Span 4	Locat	mm	0	387.3	3500	
	Height	mm	-39	-53.9	-174.5	

7.3.2.4.2 Graphics

The Prestress Profile Graphical views can be viewed in the Input screens by selecting the Input Tab on the Control tree and then selecting the Prestress option from the input tree and selecting the individual prestress profile to be viewed from the control grid. Separate viewing is not available in the output view.

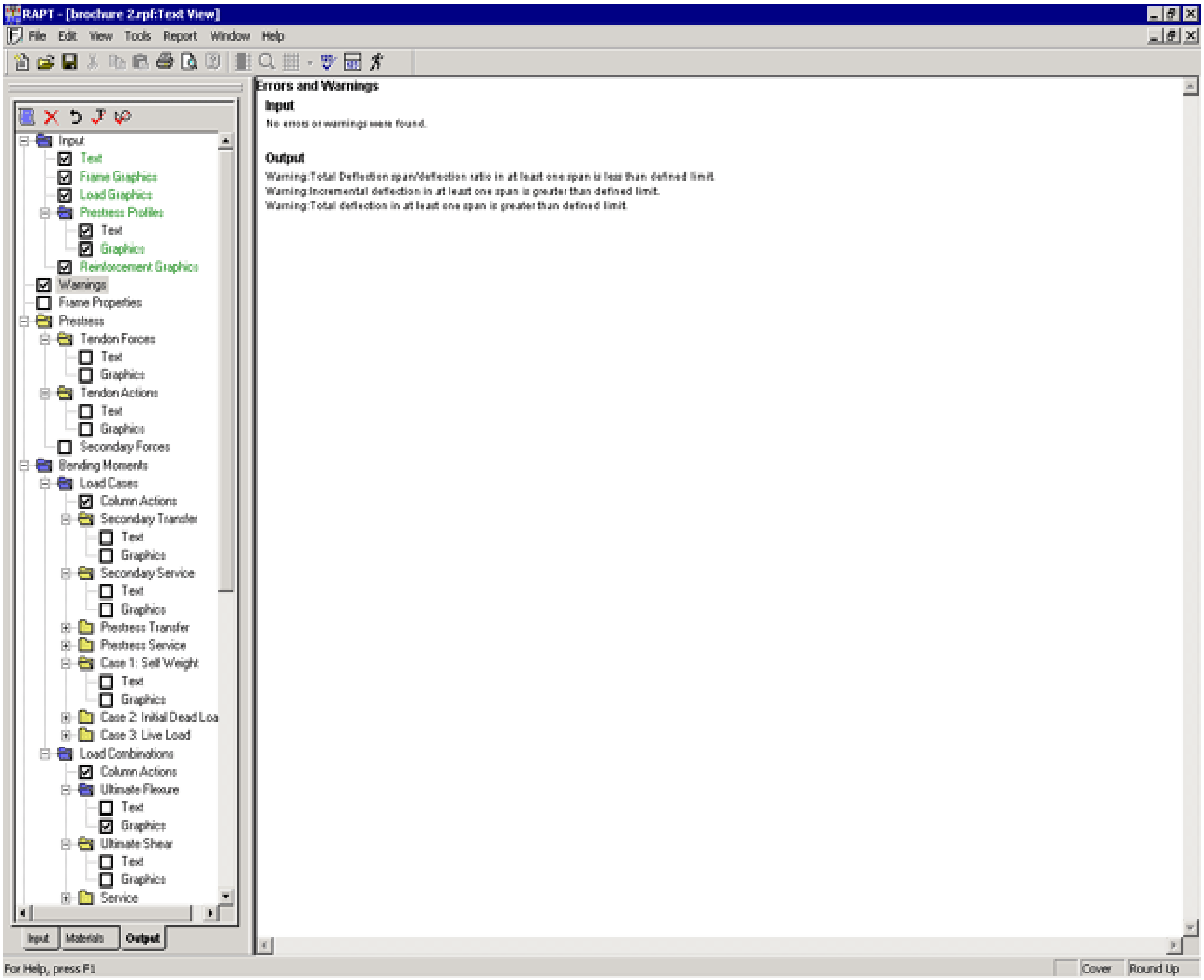
This item can be selected for inclusion in a report by clicking the Check Box beside the name. The graphics views of all of the prestress profiles will be included in the report in profile order.

7.3.2.5 Reinforcement Graphics

The Reinforcement Graphical views can be viewed in the Input screens by selecting the Input Tab on the Control tree and then selecting the Reinforcement->Zones or Reinforcement->User Defined options from the input tree. Separate viewing is not available in the output view.

This item can be selected for inclusion in a report by clicking the Check Box beside the name. The graphics views of the Reinforcement Zones and, if applicable, the User Defined Reinforcement will be included in the report.

7.3.3 Warnings



RAPT gives three types of messages regarding the results of the design. These messages could relate to input data inconsistencies or messages regarding the calculations that have been performed. These are

1. Design Comments:- These are listed with the related design results. They are comments on the results and do not imply in any way that there has been a problem with the design. An example would be the comments made when ductility reinforcement has been added in an over-reinforced member. RAPT will include the reinforcement in the design results but will make a comment noting the location at which the ductility check was performed and the amount of reinforcement that was added at that point for ductility. It is up to the designer to decide if the design is to be accepted or if modifications will be made to remove the need for the ductility reinforcement. The design comments that will be made are discussed in the relevant sections on output results.
2. Warning Messages:-
    1. Input:- a data incompatibility has been detected that should be considered by the designer but RAPT will still run the data. Double Clicking with the left mouse button on the warning text will open the data at the relevant data view and the data causing the problem will be in focus.
    2. Output:- A result has violated a defined limit. The designer is being warned to check this result and make sure that the design is acceptable to him e.g. deflections higher than a preset limit.
3. Error messages:-
    1. Input:- a data incompatibility is so severe that RAPT cannot not run the data. Double Clicking with the left mouse button on the error text will open the data at the relevant data view and the data causing the problem will be in focus.
    2. Output:- Either the calculations could not be completed or the member fails for some reason e.g. shear is greater than the maximum allowed at a section.

7.3.4 Frame Properties

RAPT calculates the frame stiffness properties based on the method specified by the user in the 7.2.2 General Data screen. The available options are

1 Equivalent Column
2 Net Column Stiffness
3 Enhanced Column Stiffness

The values shown in the results table represent the final inertias that will be applied to the frame. These inertias include any user defined change in stiffness due to the % Stiffness option in the 7.2.3.2 Column Data Screen.

For each method the frame is considered as a two-dimensional, plane-frame structure. The three-dimensional effect of more load being carried by the slab region centred along the column centre-lines in two way slabs is accounted for in the 7.2.4.3 lateral distribution of the bending moments. Steps included via the 7.2.3.7 Steps Data Screen are modelled by modifying the inertia over that region. Section T.2 provides details on the background theory to the Equivalent Column.

The Frame Properties Screen will display the frame properties for each Span Member followed by the Column Member properties in the following form

Frame Properties

Span 0

Length	mm	3250	250
Inertia	mm4	1.08e10	1.2e10
Area	mm2	8.52e5	8.52e5

Span 1

Length	mm	250	9500	250
Inertia	mm4	1.2e10	1.08e10	1.2e10
Area	mm2	8.52e5	8.52e5	8.52e5

Span 2

Length	mm	250	9500	250
Inertia	mm4	1.2e10	1.08e10	1.2e10
Area	mm2	8.52e5	8.52e5	8.52e5

Span 3

Length	mm	250	9500	250
Inertia	mm4	1.2e10	1.08e10	1.2e10
Area	mm2	8.52e5	8.52e5	8.52e5

Span 4

Length	mm	250	3250
Inertia	mm4	1.2e10	1.08e10
Area	mm2	8.52e5	8.52e5

Column Members

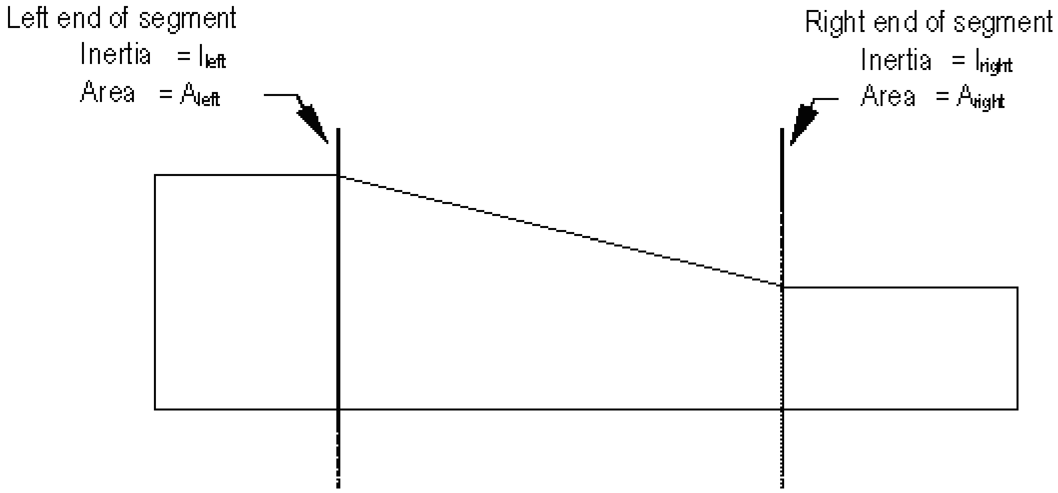
Col No.	Area Above	Inertia Above	Area Below	Inertia Below
#	mm2	mm4	mm2	mm4
1	2.5e5	3.66e9	2.5e5	3.53e9
2	2.5e5	3.66e9	2.5e5	3.53e9
3	2.5e5	3.66e9	2.5e5	3.53e9
4	2.5e5	3.66e9	2.5e5	3.53e9

Span Members

RAPT breaks each span up into a series of segments. Each segment represents a different section area and inertia. The number of segments depends on the number of changes in geometry in the span. The sum of the lengths of segments in a span will equal the span length. For each span the following information is listed

1 Length: Length of the span segment. The length over which the given Inertia and Area are applied
2 Inertia: Moment of Inertia of each segment.
3 Area: Cross sectional area of the design width.

For members tapered over their length, RAPT divides the taper length into smaller segments and uses an average inertia for each small length. The inertia and area given for the taper length are calculated as follows



$$I_{taper} = (I_{left} + I_{right}) / 2$$

$$A_{taper} = (A_{left} + A_{right}) / 2$$

Column Members

For each column RAPT lists

1 Area Above Cross Sectional Area of Column Above the slab

2 Inertia Above Inertia of the Column Above the slab. This inertia will be calculated based on the user inputs in Input Screen

3 Area Below Cross Sectional Area of Column Below the slab.

4 Inertia Below Inertia of the Column Below the slab.

When knife edge columns are specified, RAPT will display 0 values in each field. When a Fixed support is specified, RAPT will show 0 values for the area results and a large number for the inertias. This large number is only a representation. In the actual frame analysis, a fixed support is applied to the frame.

7.3.5 Prestress

7.3.5.1 Tendon Forces

RAPT calculates the forces in the tendons at each calculation point along the frame taking into account all short and long term losses based on the user selections in the input data. Friction is calculated allowing for the compound curve shapes generated to achieve the desired profiles and the effects of draw-in. Expected tendon extensions are also calculated. Both text and graphics output of the results are available.

Text Output
The text output for tendon forces is shown below and gives the following information:

General information for each tendon:

1. Tendon extension: the expected extension of the each strand in the tendon after allowing for anchorage and duct friction and anchorage draw-in.
2. Tendon size data: 3 - 5/12.7: 3 tendons make up of 5 strands of 12.7mm diameter strand. For each calculation point in each span of the tendon the following information is provided:

1 Locat:	Location of point from left end of span
2 Transfer Strand:	Force in a single strand at transfer, after all short term losses
3 Total:	Force in the tendon at transfer, after all short term losses
4 Service Strand:	Force in a single strand at service, after all short and long term losses
5 Total:	Force in the tendon at service, after all short and long term losses
6 Slope:	Slope of the tendon in radians at this point
7 Height:	Height of the tendon from the Top of Slab Datum to the centroid of the tendon. Negative downwards.

At points where a tendon does not exist in a span, all values will be set to zero.

Tendon 1 : - Gross Extension: 163.2mm - 3 - 5/12.7

Span 0

	Locat	mm	1	100	747	749	875	1750	2299	2301	2625	2930	3325	3499
Transfer	Strand	kN	122.16	122.28	123.06	123.06	123.22	124.28	124.96	124.96	125.36	125.74	127.14	128.14
	Total	kN	1832.37	1834.17	1845.91	1845.95	1848.24	1864.26	1874.38	1874.42	1880.41	1886.08	1907.08	1922.15
Service	Strand	kN	105.02	105.13	105.82	105.82	105.96	106.91	107.5	107.51	107.86	108.19	109.43	110.32
	Total	kN	1575.31	1576.91	1587.33	1587.36	1589.4	1603.6	1612.56	1612.59	1617.9	1622.91	1641.48	1654.78
Slope		#.#	0	0.0024	0.0182	0.0182	0.0213	0.0425	0.0559	0.0559	0.0638	0.0712	0.035	0
Height		mm	-165	-164.9	-158.2	-158.2	-155.7	-127.8	-100.8	-100.7	-81.3	-60.7	-35.3	-32.3

Span 1

	Locat	mm	1	175	570	1199	1201	2149	2151	2500	2999	3001	3099	3101
Transfer	Strand	kN	128.14	129.16	131.18	131.97	131.97	133.17	133.17	133.62	134.25	134.26	134.38	134.38
	Total	kN	1922.17	1937.35	1967.76	1979.58	1979.62	1997.57	1997.61	2004.26	2013.81	2013.84	2015.72	2015.76
Service	Strand	kN	110.32	111.2	112.93	113.59	113.6	114.61	114.61	114.99	115.53	115.53	115.63	115.64
	Total	kN	1654.79	1668.03	1693.89	1703.92	1703.95	1719.16	1719.19	1724.81	1732.88	1732.91	1734.5	1734.53
Slope		#.#	0	-0.035	-0.1002	-0.086	-0.0859	-0.0645	-0.0644	-0.0565	-0.0452	-0.0452	-0.043	-0.0429
Height		mm	-32.3	-35.3	-64.3	-122.8	-123	-194.3	-194.4	-215.5	-240.9	-241	-245.3	-245.4

Span 2

	Locat	mm	1	175	570	1199	1201	1500	2149	2151	2500	2999	3001	3099
Transfer	Strand	kN	127.22	126.22	124	123.13	123.12	122.75	121.93	121.92	121.49	120.86	120.86	120.74
	Total	kN	1908.28	1893.32	1859.97	1846.9	1846.86	1841.18	1828.89	1828.86	1822.29	1812.93	1812.89	1811.06
Service	Strand	kN	109.5	108.62	106.65	105.88	105.88	105.54	104.81	104.81	104.42	103.87	103.87	103.76
	Total	kN	1642.54	1629.32	1599.79	1588.21	1588.17	1583.13	1572.22	1572.19	1566.35	1558.03	1558	1556.37
Slope		#.#	0	-0.035	-0.114	-0.101	-0.1009	-0.093	-0.0757	-0.0757	-0.0664	-0.0532	-0.0531	-0.0505
Height		mm	-32.3	-35.3	-64.7	-133.5	-133.7	-162.7	-217.4	-217.6	-242.4	-272.2	-272.3	-277.4

Span 3

	Locat	mm	1	175	570	999	1199	1201	1999	2000	2001	2149	2151	2500	2999	3001
Transfer	Strand	kN	110.18	109.32	107.39	106.05	105.52	105.52	103.44	103.44	0	0	0	0	0	0
	Total	kN	1652.69	1639.73	1610.84	1590.74	1582.85	1582.77	1551.67	0	0	0	0	0	0	0
Service	Strand	kN	94.31	93.53	91.8	90.59	90.11	90.11	88.23	88.23	0	0	0	0	0	0
	Total	kN	1414.67	1403.01	1376.96	1358.81	1351.67	1351.6	1323.44	0	0	0	0	0	0	0
Slope		#.#	0	-0.035	-0.114	-0.0994	-0.0796	-0.0794	-9.9345e-5	0	0	0	0	0	0	0
Height		mm	-32.3	-35.3	-64.7	-115.2	-133.1	-133.3	-165	-165	0	0	0	0	0	0

Graphical Output

The graphic output for tendon forces is shown below and gives the following information:

General information for each tendon:

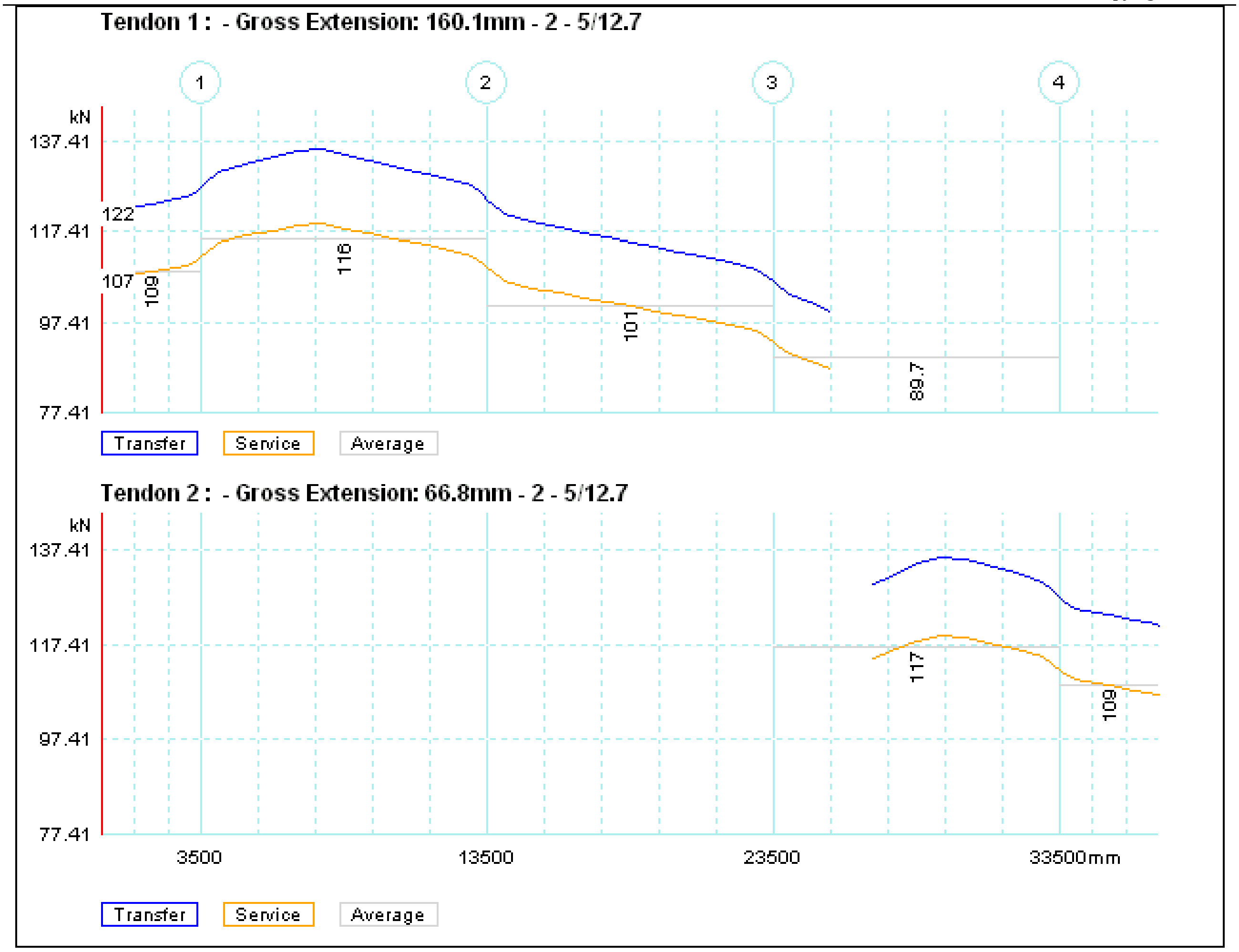
1. Tendon extension: the expected extension of the each strand in the tendon after allowing for anchorage and duct friction and anchorage draw-in.
2. Tendon size data: the
   1. Number of Tendons:- 3 - 5/12.7: 3 tendons made up of 5 strands of 12.7mm diameter strand.
   2. Spacing of Tendons:- 5/12.7@1000: tendons made up of 5 strands of 12.7mm diameter strand at 1000mm centres.

For each span The Average Service Force is plotted and a value is assigned to it at the centre of the span.

For each calculation point in each span of the tendon the following information is provided:

1. Transfer Strand: Force in a single strand at transfer, after all short term losses
2. Service Strand: Force in a single strand at service, after all short and long term losses
   At points where a tendon does not exist in the frame, no values are plotted.

The cursor will show the values at a selected point on each of the curves.

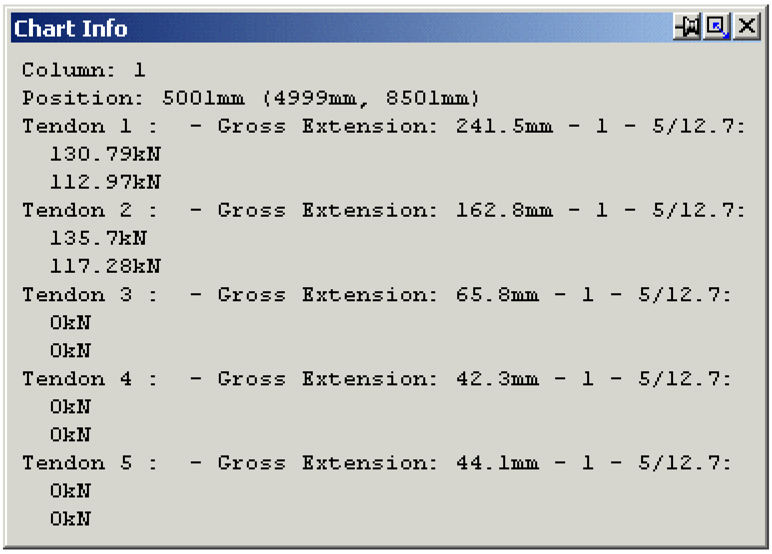


Information Dialog

A typical Info dialog is shown below As well as the tendon extension and size data, at the selected point, for each tendon it defines

1. Position of point in span
2. Transfer Strand: Force in a single strand at transfer, after all short term losses
3. Service Strand: Force in a single strand at service, after all short and long term losses

Tendons that do not exist at this point will be shown with forces equal to 0.



7.3.5.2 Tendon Actions

Knowing the accurate prestress forces, (i.e. transfer and long term (effective)), RAPT can calculate the true forces imposed on the structure by the prestress. When viewing text results, the following screens are presented to the user for each strip for each tendon:

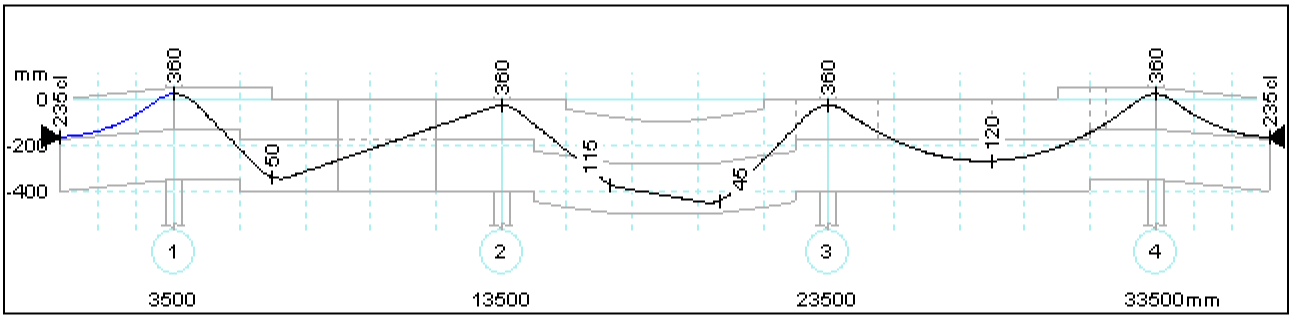
The method of calculating these forces is detailed in Theory Section T16.

The tendon actions calculated by RAPT are based on the average prestress force in each span. These are used to calculate the secondary prestress moments.

The primary prestress moments are determined from the eccentricity of the prestress force at each point along the frame and the actual prestress force at that point, not the average prestress force as is used for the secondary moments. They are not determined from the tendon actions.

The total prestress effect at each point is then the sum of the primary and secondary prestress moments at that location.

Text Output
The tendon actions are divided into four groups in the text output. The table of output for each group is only created if there is a force of that type induced in a specific run. The run we have used to generate the actions shown below is different to the one have used for all of the other output explanations. The tendon profile and concrete elevation used is shown below



Forces Induced by Tendon Shape
Forces imposed on the structure by changes of angle in the tendon. This is further divided into two separate tables for Distributed forces and Concentrated forces.

These forces are due to the change in angle of the tendon over the span length. The uniformly distributed loads are printed (positive down) together with any concentrated forces (positive down). The calculation of these forces is discussed in detail in Theory Section 16.1.

For parabolic profiles, RAPT uses a uniformly distributed load to model the uplift from the prestress in each span/strip calculated from the curvature of the upward parabola. It is calculated from the average prestress force within the drape length in that span/strip. The downward distributed loads from the reverse parabolas are also represented by distributed loads calculated from the curvature of the reverse parabolas. At the ends of tendons where the anchorage is placed at an angle, the resulting force will be treated as a concentrated force.

Forces induced by point load profiles (harped tendons) are represented by a uniform force over the length of the circular curve tangential to the 2 straight lines forming the harped shape.

If the centroid of the section slopes away from the point where a tendon ends, an axial force will be applied to account for the difference in slope between the prestress force and the centroid.

The following information is shown for each distributed load for each tendon profile and strip.

1. The average prestress force in the tendon (Weighted average)
2. The column line from which load locations are measured
3. The start (left end) location of the load
4. The load value at the start location
5. The end (right end) location of the load
6. The load value at the end location
7. Transfer factor - Factor to convert forces from effective to transfer conditions.

Distributed Forces Induced By Tendon Shape - Tendon 1 - 4 - 5/12.7

Load No.	Prestress Force	Ref Col No.	Start Location	Start Value	End Location	End Value	Transfer Factor
#	kN	#	mm	kN/m	mm	kN/m	#.#
1	2062.32	0	0	-72.32	2977.9	-72.32	1.1784
2	2062.32	0	2977.9	412.46	3500	412.46	1.1784
3	2197.13	1	0	439.43	720.7	439.43	1.181
4	2197.13	1	9759.8	439.43	10000	439.43	1.181
5	2197.13	1	2525.5	-439.43	3478.8	-439.43	1.181
6	2062.65	2	0	412.53	564.4	412.53	1.1683
7	2062.65	2	9281.9	412.53	10000	412.53	1.1683
8	2062.65	2	3115.6	-412.53	3552.4	-412.53	1.1683
9	2062.65	2	6247.2	-412.53	7082	-412.53	1.1683
10	2238.61	3	0	447.72	495.3	447.72	1.1728
11	2238.61	3	495.3	-52	9454.7	-52	1.1728
12	2238.61	3	9454.7	447.72	10000	447.72	1.1728
13	2077.6	4	0	415.52	526.6	415.52	1.1763
14	2077.6	4	526.6	-73.59	3500	-73.59	1.1763

The following information is shown for each concentrated load for each tendon profile and strip.

1. The average prestress force in the tendon (Weighted average)
2. The column line from which load locations are measured
3. The location of the load
4. The slope change causing the load
5. The load value
6. Transfer factor - Factor to convert forces from effective to transfer conditions.

Concentrated Forces Induced By Tendon Shape - Tendon 1 - 4 - 5/12.7

Load No.	Ref Col No.	Load Location	Prestress Force	Load Value	Transfer Factor
#	#	mm	kN	kN	#.#
2	0	0	2062.32	29.46	1.1784
12	4	3500	2077.6	29.68	1.1763

Moments from Anchorage Eccentricity

Forces imposed on the structure by the tendon at the tendon ends. If an anchor is placed eccentric to the centroid of the section at that point then a bending moment will be induced. The calculation of these forces is discussed in detail in Theory Section 16.2.

If there is no eccentricity at the anchors then the table is not printed.

The following information is shown for each tendon profile and strip. It shows values for the left and right anchors of the tendon: If there is eccentricity at only one end then the moment at the other end will be shown with a value of 0.

1. the value of the bending moment
2. the eccentricity of the anchor from the centroid of the section.
3. the span in which the anchor occurs
4. the distance from the left hand column centre-line in that span to the anchor.
5. The transfer factor. This is the conversion factor required to increase the moment from an effective moment to a transfer moment.

Moments From Anchorage Eccentricity - Tendon 1 - 4 - 5/12.7

Moment Start	e	Prestress Force	Span No.	Location	Transfer Factor	Moment End	e	Prestress Force	Span No.	Location	Transfer Factor
kNm	mm	kN	#	mm	#.#	kNm	mm	kN	#	mm	#.#
5.08	-2.5	2062.32	0	0	1.1784	-5.12	-2.5	2077.6	4	3500	1.1763

Forces from Changes in Concrete Centroid

Forces resulting from the change of the cross section shape.

At any variation in cross section where the centroid of the concrete section steps or changes slope (ie the ends of drop panels, steps, changing panel width from span to span, etc) there will be forces imposed by the prestress. If the

change in section is sharp (a step), a bending moment will be generated, if a tapered change, a force couple will be generated. Theory Section 16.1.3 describes the calculation of these forces.

The output screen lists all forces and moments that occur along the structure due to these changes in cross section. The properties listed are:

1. Reference Column
2. Distance from Col: Defines the position of the force or moment from the column specified in 1.
3. Prestress Force: The average prestress force at the location. If in the middle of a span, then the average value for the span is used. If near a column then the average force either side of the column is used.
4. Point Loads: Change of Slope: angle through which the centroid has deviated. (due to taper). A positive slope infers that the surface has rotated in an anticlockwise direction in comparison with the previous surface slope.
5. Point Loads: Vertical Load: Load applied to the structure due to the angle change in the concrete centroid. A positive force will result for a negative change in slope. A negative force is an upwards force.
6. Point Moments: Change of Centroid: the vertical step through which the centroid has deviated. A positive step indicates that the centroid has moved downwards from the last centroid position.
7. Point Moments: Applied Moment: The resulting moment applied to the structure due to the step in the centroid. A positive moment acts in a clockwise direction.
8. Transfer Factor: Factor applied to the effective moments and forces shown to give the transfer equivalent forces and moments.

Forces From Changes In Centroids - Tendon 1 - 4 - 5/12.7

			Point Loads		Point Moments		
Ref Col	Distance From Col	Prestress Force	Change Of Slope	Vertical Load	Charge Of Centroid	Applied Moment	Transfer Factor
#	mm	kN	#.#	kN	mm	kN/m	#.#
1	0	2129.72	-0.0143	-30.42	0	0	1.1797
1	2000	2197.13	0	0	22	48.36	1.181
1	3000	2197.13	0	0	28	61.5	1.181
1	5000	2197.13	0	0	8.9	19.52	1.181
1	8000	2197.13	0	0	-7.3	-16.09	1.181
2	1000	2062.65	-0.0091	-18.68	22.1	45.56	1.1683
2	2000	2062.65	-0.0109	-22.57	27.6	56.89	1.1683
2	3500	2062.65	0.0087	18	0	0	1.1683
2	4500	2062.65	0.0113	23.25	0	0	1.1683
2	5500	2062.65	0.0113	23.25	0	0	1.1683
2	6500	2062.65	0.0087	18	0	0	1.1683
2	8000	2062.65	-0.0109	-22.57	-27.6	-56.89	1.1683
2	9000	2062.65	-0.0091	-18.68	-22.1	-45.56	1.1683
3	7000	2238.61	0	0	-27.9	-62.48	1.1728
3	8000	2238.61	0	0	-22.1	-49.45	1.1728
4	0	2158.1	-0.0143	-30.83	0	0	1.1745

Graphical Output

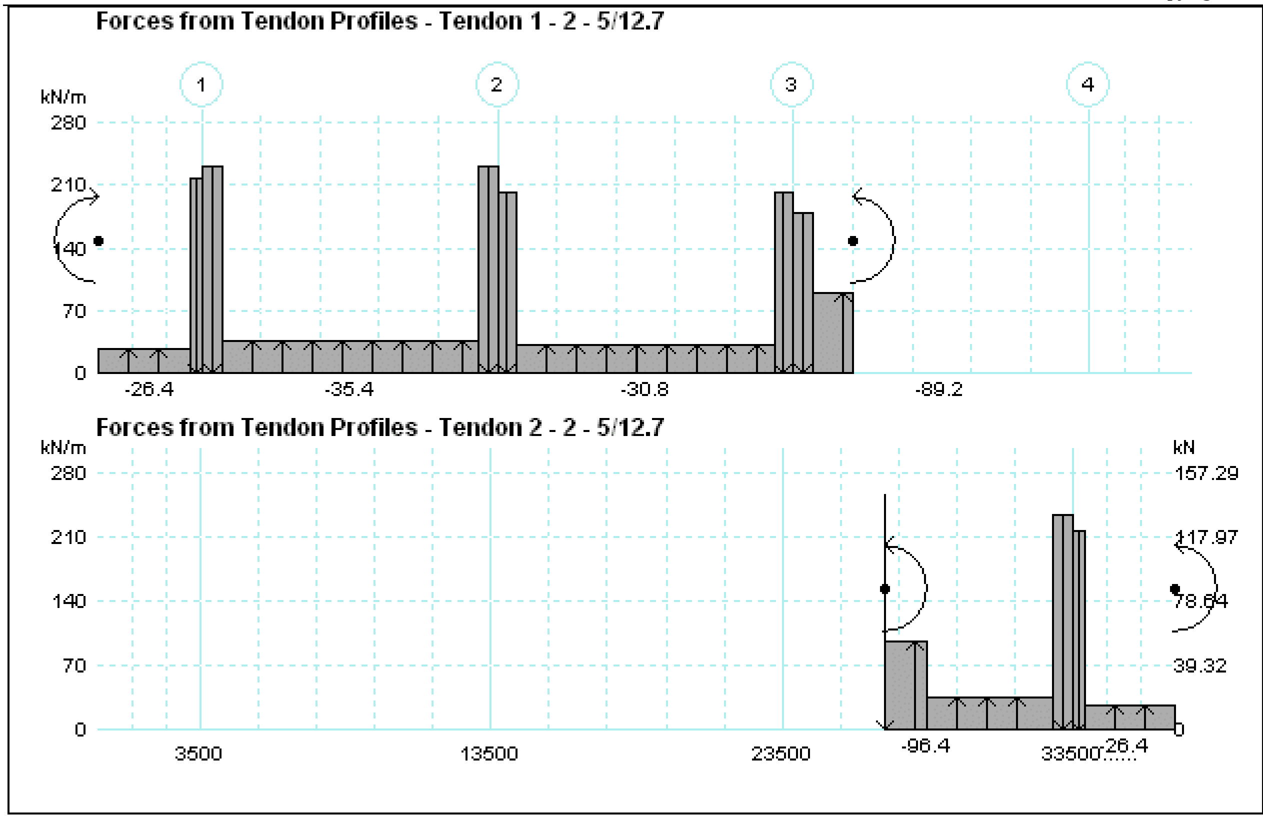
The three groups of forces described above are displayed on 2 separate graphs.

The Forces From Tendon Profiles graph includes the forces and moments discussed in

1. Forces Induced by Tendon Shape
2. Moments from Anchorage Eccentricity

The Forces from Changes in Section graph displays the forces and moments discussed in Forces from Changes in Concrete Centroid above.

Note that the left scale is for distributed loads (kN/m in this case) and the right scale is for Point loads (kN in this case). Point Moments are all drawn the same size as scaling these can often produce useless results for the user. Look at the Information Dialog or the text output for values for these.

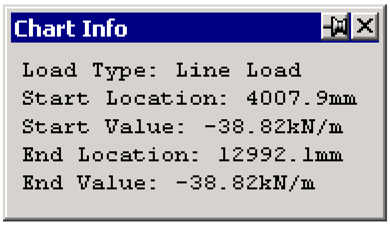


Information Dialog

To view the information describing any of the loads shown on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on a load to be viewed. While the dialog is in view, click on any other load to view its information and the dialog data will be updated automatically.

The dialog will list:

1. Load type: either line load, point load or point moment.
2. The locations of the load (measured from the left end of the frame)
3. The values of the load.



7.3.5.3 Secondary Forces

Most prestressed members experience secondary (parasitic) moments due to the action of the prestress on the member.

For determinate members and in-determinate members with concordant prestress profiles the secondary prestress moments be zero. See Theory Section 10 for a discussion on the calculation of secondary moments.

RAPT treats pretensioned tendons as a special case for the calculation of secondary prestress moments. Secondary prestress moments are caused by indeterminacy as discussed above. While a pretensioned member may be continuous and therefore in-determinate, the pretensioned element itself containing the pretensioned tendons will have been manufactured in a pretensioning bed and there will be no secondary prestressed effects induced during it's stressing. It may then become part of an indeterminate frame or have intermediate support points added during construction or in transport, but these will not induce secondary prestress moments. So, in the case of pretensioned tendons, RAPT sets the secondary moments for those tendons to zero. The total prestress moment at any point is equal to P * e for these tendons.

This section of output lists the information used to calculate the secondary bending moments and reactions for the prestress, long-term and transfer (via the transfer factor). The information provided for each end of each span of each tendon in each strip is:

1. tendon depth
2. centroid depth of the cross section
3. e = distance of the tendon from the centroid used to calculate P * e for the basic prestress moment
4. Peff = Effective prestress force
5. Mtotal = Moment caused at each section by the prestress forces (ie MTotal = Peff. e + Msec). This is calculated from the analysis of the 7.3.5.2 Prestress Actions
6. Msec = The secondary Moment calculated at each cross section = MTotal - P * e.
7. Reaction: the resulting reaction from the secondary moment, Msec, at either end of this span.
8. Transfer Factor. The factor used to convert results from results at long term to results at the Transfer condition.

No graphics output is available for this output file. Graphics output for the overall secondary moments Bending Moment/Shear diagram is available in Bending Moments->Load Cases.

Secondary Forces

Tendon 1 :

Span No.	End	Tendon Depth	Centroid Depth	e	Peff.	MTotal	MSec	Reaction	Transfer Factor
#	A	mm	mm	mm	kN	kNm	kNm	kN	#.#
0	Left	165	167.5	2.5	1605.2	-3.96	1.01e-6	-1.41e-7	0
0	Right	32.3	167.5	135.2	1605.2	-3.96	-6.89e-6	1.41e-7	0
1	Left	32.3	167.5	135.2	1716.6	-252.85	20.74	6.11	0
1	Right	32.3	167.5	135.2	1621.18	-252.85	73.42	-6.11	0
2	Left	32.3	167.5	135.2	1621.18	-299.95	80.74	-4.23	0
2	Right	32.3	167.5	135.2	1444.08	-299.95	44.63	4.23	0
3	Left	32.3	167.5	135.2	1444.08	-169.61	-25.65	3.34	0
3	Right	0	167.5	0	0	-169.61	2.92	-3.34	0
4	Left	0	167.5	0	0	7.11e-15	-7.11e-15	-6.09e-15	0
4	Right	0	167.5	0	0	7.11e-15	0	6.09e-15	0

Tendon 2 :

Span No.	End	Tendon Depth	Centroid Depth	e	Peff.	MTotal	MSec	Reaction	Transfer Factor
#	A	mm	mm	mm	kN	kNm	kNm	kN	#.#
0	Left	0	0	0	0	-4.44e-16	4.44e-16	-2.16e-15	0
0	Right	0	0	0	0	-4.44e-16	0	2.16e-15	0
1	Left	0	0	0	0	-1.11	1.11	-0.53	0
1	Right	0	0	0	0	-1.11	-3.44	0.53	0
2	Left	0	0	0	0	9.78	-9.78	4.08	0
2	Right	0	0	0	0	9.78	25.31	-4.08	0
3	Left	0	0	0	0	-68.67	68.67	-6.78	0
3	Right	32.3	167.5	135.2	1725.92	-68.67	10.29	6.78	0
4	Left	32.3	167.5	135.2	1605.17	-217.04	1.02e-4	-7.76e-5	0
4	Right	165	167.5	2.5	1605.17	-217.04	2.04e-8	7.76e-5	0

7.3.6 Bending Moments

In 7.2.4.1 Loads Cases and 7.2.4.2 Load Combinations users have input loads and defined the way in which the Primary Load Cases are to be combined to form the various Design Combinations.

RAPT uses this information to

1. Calculate the moment and shear diagrams for each Primary Load Case (using a frame stiffness analysis) (ie Self Weight, SDL, Prestress forces etc)
2. Calculates the moment and shear diagrams/envelopes for the Design Combinations (ie Ultimate, Deflection, Shear etc).

Both Load Cases and Load Combinations may be defined by an 7.3.6.3 envelope of moments and shears if the Design Combination is defined by more than one combination of Primary Load Cases, or if Live Load Patterning is requested or a bending moment envelope is defined in input, or by a single 7.3.6.2 moment and shear diagram in other cases

NOTE: Column moments are given at the mid-depth of the intersecting concrete member. They have not been reduced to the face of the intersecting concrete member

Pattern Loading

Only live loads which are defined in the load case whose Load Type is Live Load are considered for pattern loading. Loads in any other case including Alternate Live Loads are not patterned. All loads defined in the Live Load case are patterned including point loads and point moments. Distributed loads are internally converted to span loads so that any loads that extend over more than one span RAPT will be reduced to span loads and can be patterned properly. If the user does not wish some of the live loads to be patterned they should be placed in an extra load case (type : 7.2.4.1 Other Load). This case must then be added to the relevant load combinations by the user with the appropriate factors.

Specific live load moment and shear envelopes have not been produced for the pattern load cases because the requirements of several codes do not allow for the creation of an envelope independent of other loads such as dead load.

Skip (pattern) live load logic is as follows for different design codes,

For AS3600.

1. live load as defined
2. all combinations of live load on individual spans with reduced live load on alternate spans
3. all combinations of live load on pairs of adjacent spans with reduced live load on alternate spans

For ACI318, IS456/IS1343 and Eurocode2.

1. live load as defined
2. live load on odd numbered spans
3. live load on even numbered spans
4. live load on all pairs of adjacent spans

For SABS 0100

1. live load as defined by the users input
2. live load on odd numbered spans
3. live load on even numbered spans

For BS8110, CP2004 and CP65,
RAPT uses a different approach for pattern loads. These codes require the pattern load to include a pattern effect of the dead loads as well as the live loads for the ultimate load condition.

For BS8110, CP65,

1. factored dead and live loads as entered
2. live load on odd numbered spans
3. live load on even numbered spans.
4. dead load on odd spans
5. dead load on even spans

For CP2004,

1. factored dead and live loads as entered
2. live load on odd numbered spans
3. live load on even numbered spans.
4. dead load on odd numbered spans
5. dead load on even numbered spans
6. live load on each pair of adjacent spans

7. dead load on each pair of adjacent spans

The dead load includes self weight and superimposed dead loads. These load cases are then combined with the other load cases for ultimate strength with different dead load factors on loaded and unloaded spans as required by these codes.

7.3.6.1 Bending Moments - Column Actions

RAPT produces a tables of column reactions and moments as shown below. The load cases shown in the table will be defined in 7.3.6.4 Load Cases and 7.3.6.5 Load Combinations.

Load Case - Column Actions

Col No. 1		Secondary Transfer	Secondary Service	Prestress Transfer	Prestress Service	Self Weight	Initial Dead Load	Live Load
Moment Above	kNm	-24.27	-20.96	-24.27	-20.96	40.59	26.06	39.7
Moment Below	kNm	-18.49	-15.96	-18.49	-15.96	32.59	22.71	31.88
Reaction	kN	5.58	4.81	4.11	3.52	428.79	184.46	419.45

Col No. 2		Secondary Transfer	Secondary Service	Prestress Transfer	Prestress Service	Self Weight	Initial Dead Load	Live Load
Moment Above	kNm	-1.35	-1.01	-1.35	-1.01	-6.45	4.63	-6.31
Moment Below	kNm	-0.1	0.04	-0.1	0.04	-5.09	5.54	-4.98
Reaction	kN	-5.74	-4.91	-6.97	-6.02	516.81	264.93	505.55

Col No. 3		Secondary Transfer	Secondary Service	Prestress Transfer	Prestress Service	Self Weight	Initial Dead Load	Live Load
Moment Above	kNm	16.73	14.45	16.73	14.45	6.45	-3.6	6.31
Moment Below	kNm	14.43	12.47	14.43	12.47	5.09	-1.11	4.98
Reaction	kN	-3.29	-2.88	-1.81	-1.59	516.81	272.4	505.55

Col No. 4		Secondary Transfer	Secondary Service	Prestress Transfer	Prestress Service	Self Weight	Initial Dead Load	Live Load
Moment Above	kNm	18.52	15.88	18.52	15.88	-40.59	-9.88	-39.7
Moment Below	kNm	15.91	13.65	15.91	13.65	-32.59	-6.16	-31.88
Reaction	kN	3.44	2.98	4.67	4.09	428.79	233.21	419.45

For each column for each load case RAPT defines

1 Moment Above	- Bending moment in the column above the slab.
2 Moment Below	- Bending Moment in the column below the slab
3 Reaction	- Reaction at this support line

Moments at the far ends of the columns will be zero for pin ended columns and 50% of the nominated values for fixed ended columns.

7.3.6.2 Moment/Shear Diagram

Bending moment/shear diagrams give one moment and one shear at each design point in the frame as shown below.

Text View
Moments are given for each column above and below the floor and reactions are given for each column. Moment and shear values and the point location are listed at each calculation point in each span in each strip in the frame.

Total Panel Column Moments And Reactions

Col No.	Moment Above	Moment Below	Reaction
	kN/m	kN/m	kN
1	40.59	32.59	428.79
2	-6.45	-5.09	516.81
3	6.45	5.09	516.81
4	-40.59	-32.59	428.79

Moments Shear Diagram

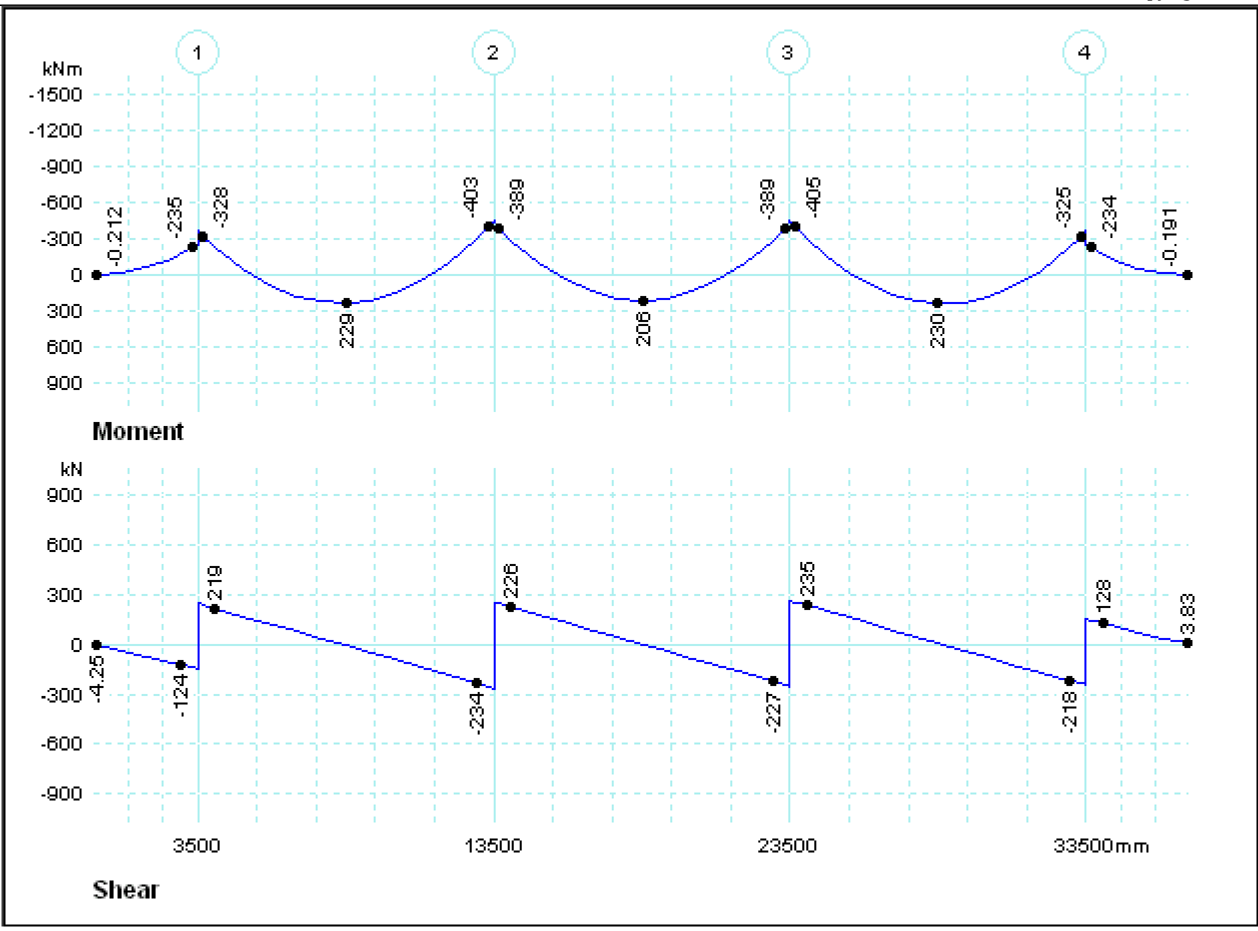
Span 0	Locat	mm	1	100	747	749	875	1750	2299	2301	2625	2930	3325	
	Moment	kNm	-2.56e-5	-0.26	-14.26	-14.34	-19.57	-78.27	-135.08	-135.31	-176.1	-219.4	-282.54	
	Shear	kN	-0.05	-5.11	-38.18	-38.28	-44.72	-89.45	-117.51	-117.61	-134.17	-149.76	-169.95	
Span 1	Locat	mm	1	175	570	1199	1201	2149	2151	2500	2999	3001	3099	
	Moment	kNm	-385.99	-343.3	-252.11	-123.37	-122.99	32.74	33.02	78.75	133.32	133.51	142.72	
	Shear	kN	249.84	240.94	220.75	188.6	188.5	140.05	139.94	122.11	96.6	96.5	91.49	
Span 2	Locat	mm	1	175	570	1199	1201	1500	2149	2151	2500	2999	3001	
	Moment	kNm	-431.22	-387.54	-294.11	-161.8	-161.41	-105.63	-0.29	-1.51e-3	47.71	105.11	105.3	
	Shear	kN	255.52	246.62	226.43	194.28	194.18	178.9	145.72	145.62	127.78	102.28	102.1	
Span 3	Locat	mm	1	175	570	999	1199	1201	1999	2000	2001	2149	2151	
	Moment	kNm	-442.76	-398.09	-302.42	-207.55	-166.53	-166.13	-22.92	-22.76	-22.6	0.36	0.67	
	Shear	kN	261.19	252.3	232.11	210.18	199.96	199.86	159.07	159.02	158.97	151.4	151.3	
Span 4	Locat	mm	1	175	549	551	570	875	1750	2625	2751	2753	3400	
	Moment	kNm	-312.89	-282.55	-222.56	-222.26	-219.4	-176.1	-78.27	-19.57	-14.34	-14.26	-0.26	-
	Shear	kN	178.85	169.95	150.84	150.73	149.76	134.17	89.45	44.72	38.28	38.18	5.11	

Graphics View
Separate diagrams are provided for bending moments and shears.

The support flexural critical section moments at or near the column face at each end of each span are identified on the moment plot along with a span moment value. The location of the span moment value is the same for the plots of all load cases and load combinations. It is determined by the location of the maximum span moment point (positive moment in this case) in each span for the ultimate flexure bending moment diagram.

The shear critical section values near the column face at each end of each span are identified on the shear plot

When the cursor is showing at a selected point, the values of moment and shear at that point are shown.

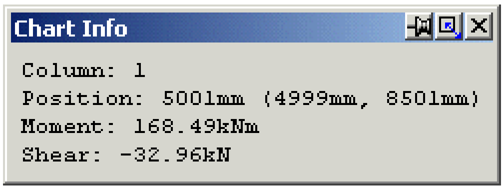


Information Dialog

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The dialog will list:

1. Location from left end of span (and to the right end of the span in brackets).
2. Moment at that point
3. Shear at that point.



7.3.6.3 Moment/Shear Envelope

Bending moment/shear envelopes generally give the highest and lowest value moments at each location from a range of values and the co-existing shear values i.e. they are a moment controlled envelope with co-existing shears. In these cases. the column results are based on the maximum and minimum moments with co-existing reactions. The exception to this is the Ultimate Shear Envelope which gives the highest and lowest value shears at each location from a range of values and the co-existing moment values i.e. it is a shear controlled envelope with co-existing moments. In this case the column results are based on the highest and lowest reactions with co-existing moments.

Text View
Two Moments are given for each column above and below the floor and co-existing reactions are given for the column. Two moment and two co-existing shear values shear values and the point location are listed at each calculation point in each span in each strip in the frame.

Total Panel Column Moments And Reactions

Col No.	Moment Above	Moment Below	Reaction	Reversal Moment Above	Reversal Moment Below	Reversal Reaction
	kN/m	kN/m	kN	kN/m	kN/m	kN
1	39.02	33.81	556.74	118.57	98.22	1369.88
2	-12.66	-6.9	1691.51	-2.65	0.44	698.66
3	17.01	16.06	707.41	27.33	24.73	1702.5
4	-104.23	-80.66	1426.54	-29.54	-21.22	598.77

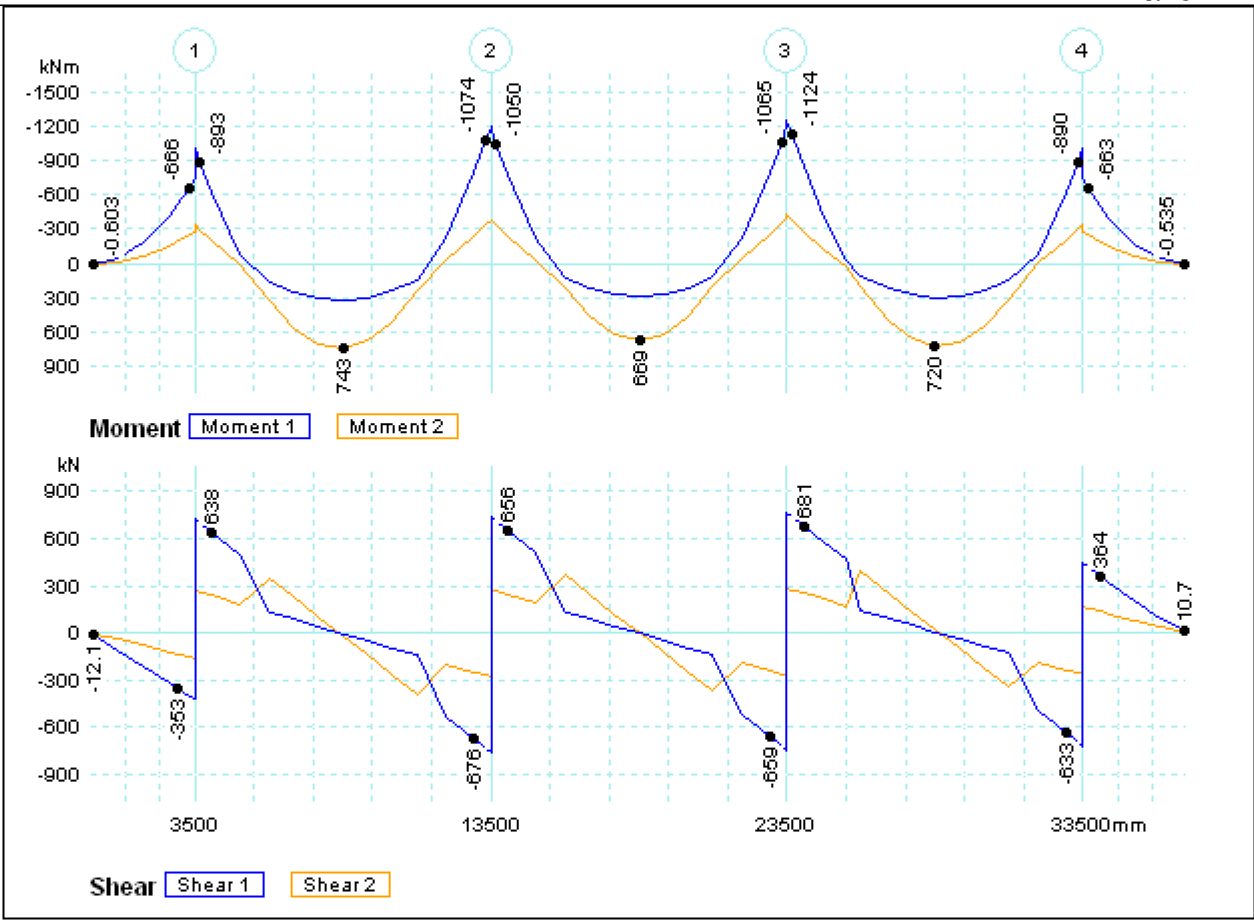
Moments Shear Envelope

Span 0	**Locat**	**mm**	**1**	**100**	**747**	**749**	**875**	**1750**	**2299**	**2301**	**2625**	**2930**
	Moment	**kNm**	-67.5	-67.91	-101.86	-102.09	-117.55	-293.26	-465.89	-466.61	-591.74	-725.
	Shear	**kN**	-0.08	-8.29	-112.72	-113.03	-132.42	-270.09	-359.15	-359.47	-413.01	-464.
	Reverse Moment	**kNm**	-45	-45.28	-60.7	-60.79	-66.63	-133.82	-200.77	-201.05	-250	-302.
	Reverse Shear	**kN**	-0.06	-5.53	-42.52	-42.64	-50.1	-104.13	-140.04	-140.17	-162.1	-183.
Span 1	**Locat**	**mm**	**1**	**175**	**570**	**1199**	**1201**	**2149**	**2151**	**2500**	**2999**	**300**
	Moment	**kNm**	-1248.6	-1110.16	-812.69	-387.77	-386.52	75.54	75.96	144.04	227.8	228
	Shear	**kN**	808.55	782.67	723.37	627.4	627.09	206.22	206.09	183.97	151.59	142
	Reverse Moment	**kNm**	-506.2	-449.15	-325.92	-148.16	-147.63	138.04	138.99	296.3	487.25	487
	Reverse Shear	**kN**	332.67	323.05	300.79	264.19	264.08	478.85	478.53	422.86	342.26	329
Span 2	**Locat**	**mm**	**1**	**175**	**570**	**1199**	**1201**	**1500**	**2149**	**2151**	**2500**	**2**
	Moment	**kNm**	-1398.76	-1253.34	-940.11	-490.76	-489.43	-297.71	68.82	69.73	143.78	[illegible]
	Shear	**kN**	848.7	822.77	763	665.25	664.93	617.35	511.61	334.81	199.11	1
	Reverse Moment	**kNm**	-556.77	-495.78	-363.69	-172.3	-171.73	-89.33	69.62	70.07	238.14	4
	Reverse Shear	**kN**	355.31	345.65	323.04	285.1	284.98	266.11	223.31	223.17	453.01	3
Span 3	**Locat**	**mm**	**1**	**175**	**570**	**999**	**1199**	**1201**	**1999**	**2000**	**2001**	**2**

Graphics View
Separate diagrams are provided for bending moments and shears.

The extreme support flexural critical section moments at or near the column face at each end of each span are identified on the moment plot along with the extreme span moment value. The location of the span moment value is the same for the plots of all load cases and combinations. It is determined by the location of the maximum span moment point (positive moment in this case) in each span for the ultimate flexure bending moment envelope.

The extreme shear critical section values near the column face at each end of each span are identified on the shear plot

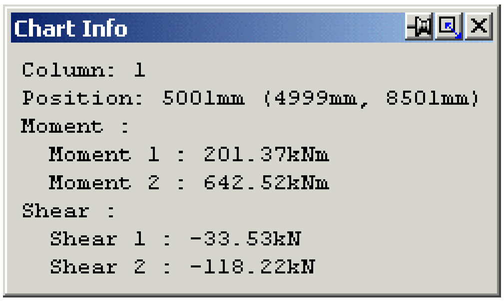


Information Dialog

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The dialog will list:

1. Location from left end of span (and to the right end of the span in brackets).
2. Minimum Moment at that point
3. Maximum Moment at that point
4. Shear coexisting with minimum moment at that point.
5. Shear coexisting with maximum moment at that point.



7.3.6.4 Bending Moments - Load Cases

RAPT provides bending moment and shear force diagrams for individual load cases for the following.

Prestress Effects
For prestressed runs, based on the 7.3.5.2 Prestress Actions calculated earlier

1. Prestress Secondary Moments at transfer
2. Prestress Secondary Moments at service
3. Prestress Moments at transfer
4. Prestress Moments at service

See 7.3.6.2 Moment/Shear Diagram and 7.3.6.3 Moment/Shear Envelope for details.

Input Load Cases
For every load case defined by the user in 7.2.4.1 Input Load Cases, RAPT will display

1. For prestress, applied loads or moment diagram cases, a 7.3.6.2 Moment/Shear Diagram
2. For moment envelope cases, a 7.3.6.3 Moment/Shear Envelope for each load case
3. For Moving Load cases,
    1. A moment controlled 7.3.6.3 Moment/Shear Envelope of critical moments and co-existing shears
    2. A shear controlled 7.3.6.3 Moment/Shear Envelope of critical shears and co-existing moments

as well as a single table of 7.3.6.1 Column Actions for all of the load cases.

Two Way Systems
For two-way systems the loads are combined separately for both the column and the middle strips and separate bending moment diagrams are provided for each strip.

Column Actions
A table of column actions is provided for each column giving column moments and reactions for each individual load case. See 7.3.6.1 Column Actions for details.

7.3.6.5 Bending Moments - Load Combinations

RAPT combines the bending moments and shears into design combinations in accordance with the information provided by the user in 7.2.4.2 Loads->Load Combinations.

The combinations may be a single bending moment/shear diagram, an envelope of moments with co-existing shears or an envelope of shears with co-existing moments, depending on the input requirements and the combination type. The combinations available for viewing are:-

1. Ultimate Flexure - Based on the Ultimate Combinations in the input, RAPT calculates a moment controlled envelope of moments and co-existing shears for ultimate flexural design. These results are also used as two of the four limiting cases for beam shear design.
2. Ultimate Shear:- Based on the Ultimate Combinations in the input, RAPT calculates a shear controlled envelope of shears and co-existing moments for beam shear design.
3. Service:- Based on the Short Term Service Combinations in the input, RAPT calculates a moment controlled envelope of moments and co-existing shears for serviceability crack control design.
4. Transfer Prestress:- Based on the Transfer Prestress Combination in the input, RAPT calculates a moment / shear diagram for transfer crack control design and for the transfer deflection calculation.
5. Short Term for deflection:- Based on the Short Term - Deflection Combination in the input, RAPT calculates a moment / shear diagram for use in the calculation of short term, long term and incremental deflections.
6. Permanent for Deflection:- Based on the Permanent - Deflection Combination in the input, RAPT calculates a moment / shear diagram for use in the calculation of long term and incremental deflections.
7. Initial for Deflection:- Based on the Initial - Deflection Combination in the input, RAPT calculates a moment / shear diagram for use in the calculation of incremental deflections.

See 7.3.6.2 Moment/Shear Diagram and 7.3.6.3 Moment/Shear Envelope for details.

Two Way Systems

For two-way systems the loads are combined separately for both the column and the middle strips and separate bending moment diagrams are provided for each strip.

Column Actions

A table of column actions is provided for each column giving column moments and reactions for the envelopes for combinations 1 to 3 above See 7.3.6.1 Column Actions for details.

7.3.7 Flexural Design

In Flexural Design, RAPT calculates any flexural reinforcement required by the structure to satisfy

1. ultimate strength
2. ductility
3. minimum reinforcement
4. serviceability criteria at both service loadings and transfer loadings

In this process, RAPT also calculates gross and cracked section concrete stresses, reinforcement tensile stresses, the cracked inertia of the member and the cracked curvatures under several load and conditions using both short term and long term material properties for deflection calculations at each design point .

The results of this section are reported in four separate sections:-

1. 7.3.7.1 Reinforcement - Total reinforcement requirements at each design point and all reinforcement and tendons defined in input that affect this point
2. 7.3.7.2 Ultimate - Reinforcement requirements and ultimate strength design data
3. 7.3.7.3 Service - Service condition concrete stresses and reinforcement stresses and crack control requirements
4. 7.3.7.4 Transfer - Transfer condition concrete stresses and reinforcement stresses and crack control requirements

Calculations
RAPT uses the Ultimate and Service Moments calculated in the analysis to design each cross section. The following design procedure is used by RAPT:-

1. RAPT identifies the possible plastic hinge locations and does a 7.3.7.2.1 ductility check to determine if the critical hinge locations in each zone are ductile and determines the area of compression reinforcement required at each critical hinge location. These areas of reinforcement are used as the maximum compression reinforcement for each plastic hinge zone.
2. Designs each cross section for ultimate strength requirements including ductility if the critical section for the zone that point falls in requires compression reinforcement. The design is first carried out for the largest moment on the section and the reinforcement requirements remembered. If the smaller moment is of reverse sign, the section is then designed for this moment also.
3. Calculates service stresses based on gross properties and, accounting for any reinforcement in stage 2, checks stresses based on cracked section and adds reinforcement to limit reinforcement stresses if necessary. Depending on the signs of the applied service moments this check may be done for one face or both faces at a point.
4. Calculates transfer stresses based on gross properties and, accounting for any reinforcement in stage 2, checks stresses based on cracked section and adds reinforcement to limit reinforcement stresses if necessary.
5. Calculates cracked section effects for initial, short term and long term loading using relevant material properties and long term effects to determine curvatures for deflection calculations.

RAPT performs cross section checks at a minimum of 13 points, including all possible critical locations (i.e. changes in cross-section, start and end of tendons etc), in each span. These nodal points are the same points as those specified in the Bending Moment diagrams.

7.3.7.1 Flexural Design - Reinforcement

The text output for this option gives lists the different reinforcement types at this design location which have been used in the design of the cross-section including the reinforcement added by RAPT to satisfy the design checks.

For each span, the data is defined at each design point located at the nominated distance from the left end of the span.

For each type of reinforcement, the table lists the total area of that type of reinforcement and the depth to the reinforcement from the top of the concrete at that point. The Types of reinforcement, listed at the top of each column are

1. Top Design (#) - The total area of top reinforcement required at this point to satisfy all flexural design criteria. This reinforcement must be fully developed at this point. The # is the number of the 7.2.6.3 Top Reinforcing Zone which controls the design reinforcement at this point.
2. Bott Design (#)- The total area of bottom reinforcement required at this point to satisfy all flexural design criteria. This reinforcement must be fully developed at this point. The # is the number of the 7.2.6.3 Bottom Reinforcing Zone which controls the design reinforcement at this point.
3. Top (#) - The total area of reinforcement from 7.2.6.3 Top User Defined Reinforcing from bar layer number # at this point. The percentage development of this reinforcement at this point has been calculated by RAPT and will be used in the design taking into account the end development defined by the designer. There could be more than one Top type at a point. If a reinforcing bar is defined with an end stagger and the percentage development of the 2 parts of the bar are different then this bar type will show twice with separate areas in each. If a bar type exists in a span then the column will show 0mm depth and 0mm2 area at design points where it does not exist in that span.
4. Bottom (#) - The total area of reinforcement from 7.2.6.3 Bottom User Defined Reinforcing from bar layer number # at this point. The percentage development of this reinforcement at this point has been calculated by RAPT and will be used in the design taking into account the end development defined by the designer. There could be more than one Top type at a point. If a reinforcing bar is defined with an end stagger and the percentage development of the 2 parts of the bar are different then this bar type will show twice with separate areas in each. If a bar type exists in a span then the column will show 0mm depth and 0mm2 area at design points where it does not exist in that span.
5. General (#) - (not shown here) The total area of reinforcement from 7.2.6.3 General User Defined Reinforcing from bar layer number # at this point. The percentage development of this reinforcement at this point has been calculated by RAPT and will be used in the design taking into account the end development defined by the designer. There could be more than one Top type at a point. If a reinforcing bar is defined with an end stagger and the percentage development of the 2 parts of the bar are different then this bar type will show twice with separate areas in each. If a bar type exists in a span then the column will show 0mm depth and 0mm2 area at design points where it does not exist in that span.
6. Tendon (#) - The total area of each tendon (row number # in 7.2.5 Prestress Control Grid) at each design point in the span and. If a tendon exists in a span then the column will show 0mm depth and 0mm2 area at design points where it does not exist in that span. For other prestress data at this location refer to the 7.3.5.1 Tendon Forces output.

For materials data for each of these types of reinforcing steel refer to text output of the input data or use the Materials Tab on the Control Tree.

Span 1

	Top Design (1)		Bott Design (1)		Top (1)		Top (2)		Bottom (1)		Tendon (1)	
Locat mm	Area mm2	Depth mm	Area mm2	Depth mm	Area mm2	Depth mm	Area mm2	Depth mm	Area mm2	Depth mm	Area mm2	Depth mm
175	465.5	48	0	402	452	48	0	0	452	402	1501.5	223.2
610	506.93	48	1322.28	402	452	48	0	0	452	402	1501.5	262.9
1379	0	48	0	402	452	48	0	0	452	402	1501.5	321.6
2149	0	48	0	402	226	48	0	0	452	402	1501.5	365.8
2151	0	48	0	402	226	48	0	0	452	402	1501.5	365.9
2500	0	48	390.59	402	226	48	0	0	452	402	1501.5	381.2
3099	0	48	390.59	402	0	0	0	0	452	402	1501.5	400.4
3101	0	48	390.59	402	0	0	0	0	452	402	1501.5	400.4
4050	0	48	599.28	402	0	0	0	0	452	402	1501.5	412.7
5000	0	48	719.16	402	0	0	0	0	452	402	1501.5	402.8
5949	0	48	628.32	402	0	0	0	0	452	402	1501.5	370.8
6899	0	48	0	402	0	0	0	0	452	402	1501.5	316.7
6901	0	48	0	402	0	0	0	0	452	402	1501.5	316.5
7500	0	48	0	402	0	0	226	48	452	402	1501.5	270.9
7849	0	48	0	402	0	0	226	48	452	402	1501.5	240.3
7851	0	48	0	402	0	0	226	48	452	402	1501.5	240.1
8799	1415.1	48	0	402	0	0	452	48	226	402	1501.5	141.8
8801	1423.3	48	0	402	0	0	452	48	226	402	1501.5	141.5
9390	3938.92	48	1792.62	402	0	0	452	48	226	402	1501.5	69.4
9825	3461.38	48	0	402	0	0	452	48	226	402	1501.5	35.3

No graphical representation of this data is available. The total design reinforcement area is plotted on the 7.3.7.2 Ultimate Flexural Design graphics results. A complete plot of the user defined reinforcement and the layout of the required reinforcement for design is available in 7.3.12 Frame_Reinforcement_Layout.

7.3.7.2 Flexural Design - Ultimate

RAPT checks the ultimate capacity for each cross section between the critical sections at the supports using the following procedure:-

1. Calculate the Applied Moment allowing for Moment Offset if necessary and the Minimum Moment.
2. Calculate the initial section capacity with all user defined reinforcement and tendons and report capacity. If acceptable then move on to other section calculations.
3. If the initial section capacity is not adequate or ductility problems exist, design the section either as singly reinforced or doubly reinforced section to achieve a moment capacity at least equal to the larger of the applied moment and the minimum moment for that point to satisfy the ductility requirements.
4. After the serviceability design is completed, repeat 2 now with any crack control reinforcement added to determine the final ductility condition and the ultimate capacity of the section with all reinforcement added. These values are recorded for the report.

Moment Reversal

It is often possible for a reversal of moments to occur at a cross-section. i.e. it is possible for both a positive and negative moment to occur at a cross-section due to the envelope effects. When designing for the ultimate capacity RAPT does the following

1. Carry out full design for ultimate conditions for the numerically larger ultimate moment, adding reinforcement if necessary to achieve the required capacity.
2. If the sign of the numerically smaller moment is opposite to that of the larger moment at the section, then RAPT again performs a complete ultimate design check, adding extra reinforcement if required to satisfy the ultimate moment requirements.

Text View

The following results are included in this table for each cross-section at which ultimate design has been carried out in each span:-

1. the location measured from the left end of the span. At some locations, RAPT will seem to repeat a point. This occurs whenever a moment reversal occurs, so that RAPT can display information for both moments.
2. Min Width - minimum section width at that point. Reinforcement spacings and shear calculations are calculated based on this value.
3. The controlling moments are listed in the report and include
    1. M*. Applied moment calculated from the loads and load combinations as specified in input and as analysed in Analysis and allowing for moment offset if necessary. If a service moment is less than 1.2 times the ultimate moment at a point, RAPT will increase the ultimate moment to 1.2 times the service moment.
    2. Mmin. The minimum moment capacity allowed by the relevant design code for each cross section.
4. Initial Condition. RAPT gives the user the existing capacity of the structure before any extra reinforcement has been added (Note:- the reversal case will include any reinforcement added for the main moment case). This initial condition is defined in terms of;
    1. Phi Mu - The existing moment capacity of the cross section prior to RAPT adding any extra reinforcement. Users would expect to see a non zero value here if they are using prestress or have specified user defined reinforcement in input.
    2. ku neutral axis depth (kd/d) for the cross-section at the initial conditions.
5. Final Design Condition RAPT prints the final design conditions in terms of
    1. Phi Mu - This is the final moment capacity of the cross section, taking into account all reinforcement (ie prestress, user defined and RAPT calculated for all flexural design conditions for this cross-section) .
    2. ku - The final neutral axis depth with all the reinforcement taken into account. The designer can use this information to ensure that ductility requirements have been satisfied. (see Section 7.6 for more details).
    3. dtens - the final effective depth, the distance from the extreme compressive fibre to the position of the resultant tensile force in all steel on the tensile side of the neutral axis;
    4. Max Strain Ratio - The strain ratio is the ratio of the strain in the reinforcement to the peak strain for that reinforcement type. RAPT reports the maximum value for this cross-section for all of the reinforcement types in the cross-section
6. Reinforcement
    1. Top - The reinforcement recorded in this table is the reinforcement in the top face, at each cross section, required to satisfy the ultimate condition only.
    2. Bottom - The reinforcement recorded in this table is the reinforcement in the bottom face, at each cross section, required to satisfy the ultimate condition only.

Span 1

		Design Moment		Initial Condition		Final Design Condition				Reinforcement	
Locat	Min Width	M*	Mmin	Phi Mu	ku	Phi Mu	ku	dtens	Max Strain Ratio	Top	Bottom
mm	mm	kNm	kNm	kNm	mm/mm	kNm	mm/mm	mm	#.#	mm2	mm2
175	1500	-1105.04	544.12	855.22	0.2858	1107.97	0.3961	358.8	0.1759	2741.49	0
570	1500	-1105.04	502.09	803.04	0.2805	1107.75	0.4	343.7	0.1705	3186.58	0
1199	1500	-653.02	407.02	673.63	0.3187	673.63	0.3187	294.4	0.1956	0	0
1201	1500	-651.67	406.73	673.25	0.3188	673.25	0.3188	294.3	0.1955	0	0
2149	1500	319.61	288.55	564.34	0.228	564.34	0.228	232.1	0.3391	0	0
2149	1500	-114.67	289.91	453.97	0.3493	453.97	0.3493	228.2	0.205	0	0
2151	1500	320.38	288.77	564.62	0.2279	564.62	0.2279	232.2	0.3391	0	0
2151	1500	-113.7	289.7	453.68	0.3494	453.68	0.3494	228	0.205	0	0
2500	1500	453.84	324.31	610.56	0.2139	610.56	0.2139	248.3	0.3377	0	0
2999	1500	594.63	367.39	665.25	0.2019	665.25	0.2019	267.7	0.3308	0	0
3001	1500	595.1	367.54	665.44	0.2018	665.44	0.2018	267.8	0.3308	0	0
3099	1500	618.15	374.92	674.69	0.1995	674.69	0.1995	271.1	0.3305	0	0
3101	1500	618.62	375.06	674.88	0.1995	674.88	0.1995	271.1	0.3305	0	0
4050	1500	778.17	427.73	739.65	0.1845	779.52	0.1914	297	0.3116	0	331.01
4999	1500	819.9	443.59	759.59	0.1803	922.17	0.2092	309.9	0.2728	0	516.37
5000	1500	819.9	443.58	759.59	0.1803	922.18	0.2093	309.9	0.2728	0	516.39
5001	1500	819.9	443.58	759.58	0.1803	922.11	0.2092	309.9	0.2728	0	516.41
5950	1500	759.24	421.6	734.57	0.1856	760.3	0.19	294.3	0.3177	0	213.4
6899	1500	567.02	365.52	664.76	0.202	664.76	0.202	267.6	0.3309	0	0
6901	1500	566.48	365.37	664.56	0.202	664.56	0.202	267.5	0.3309	0	0
6999	1500	540.06	357.66	654.79	0.2045	654.79	0.2045	264	0.3312	0	0
7001	1500	539.52	357.5	654.58	0.2046	654.58	0.2046	263.9	0.3312	0	0
7500	1500	381.56	312.79	597.18	0.2177	597.18	0.2177	243.6	0.3382	0	0
7849	1500	236.08	276.15	549.29	0.2327	549.29	0.2327	226.9	0.3399	0	0
7849	1500	-190.96	300.58	468.87	0.342	468.87	0.342	233.9	0.2041	0	0
7851	1500	235.24	275.93	549	0.2328	549	0.2328	226.8	0.3399	0	0
7851	1500	-192.13	300.79	469.17	0.3418	469.17	0.3418	234	0.2041	0	0
8799	1500	-796.21	420.36	693.59	0.3121	798.95	0.3477	308.6	0.1807	1043.56	0
8801	1500	-797.63	420.65	693.99	0.312	800.39	0.348	308.8	0.1806	1054.14	0
9430	1500	-1271.13	517.37	826.66	0.2742	1271.46	0.3995	349.6	0.172	4374.32	1392.86
9825	1500	-1271.13	541.14	854.88	0.2858	1377.37	0.4276	357.3	0.1743	4122.08	2152.06

Design Comments:-

- Compression reinforcement added at right support hinge location for ductility in accordance with AS3600 cl.8.1.3. Asc = 2152.06mm2 @ 9825m

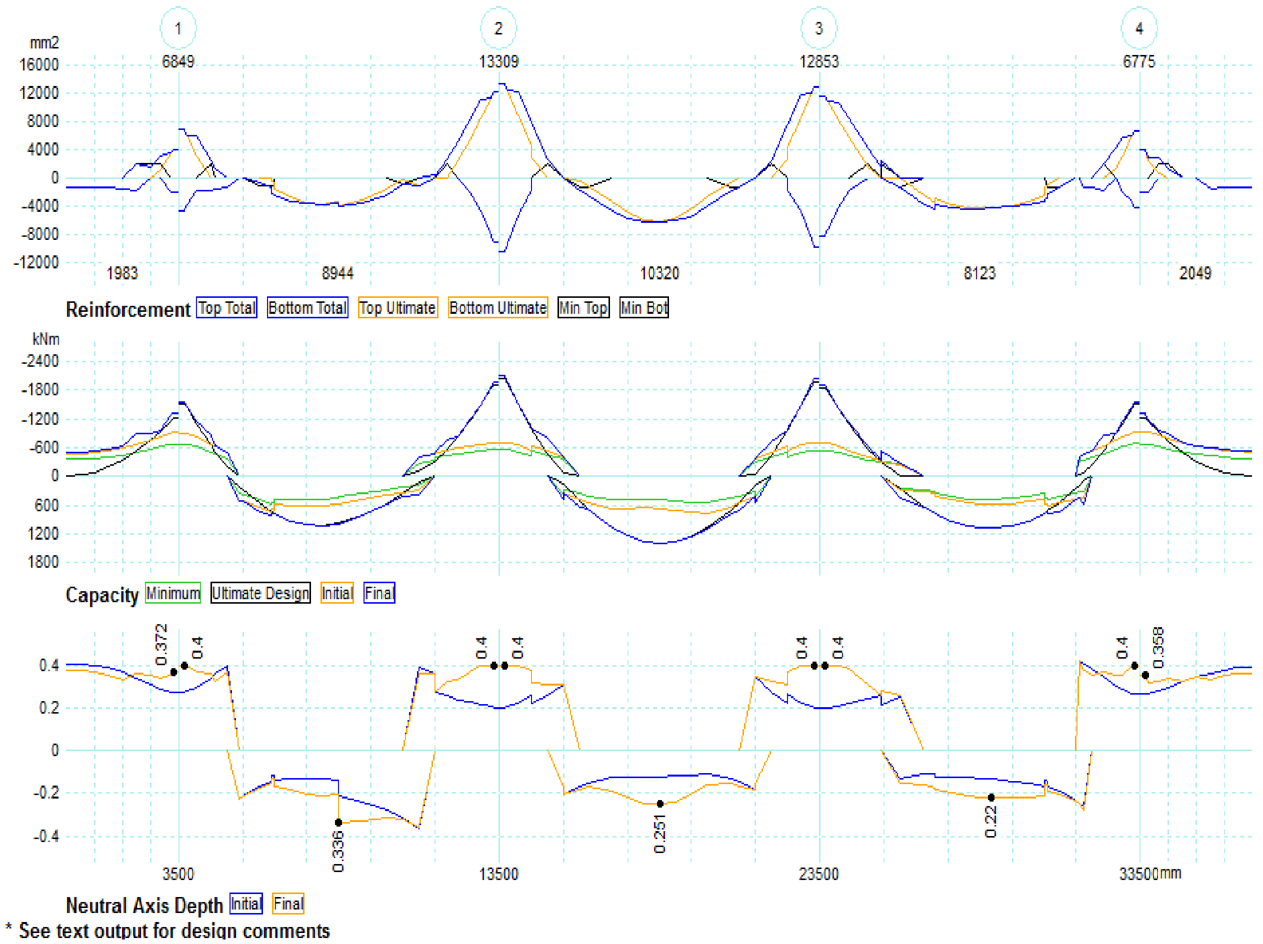
Graphical View

The graphical output consists of 3 separate diagrams for

1. Reinforcement - Top (above the zero axis) and Bottom (below the zero axis) reinforcement requirements are plotted at each design point. The nominated values are the maximum at the support surface between midspan of adjoining spans and maximum value on the span surface for each span between supports. Plotted values are
    1. Total Reinforcement - Total reinforcement required to be added to this face. It is required to be fully developed past this point. It does not include reinforcement defined as User Defined Reinforcement in the input. This reinforcement is the sum of the reinforcement required for ultimate strength and the extra flexural reinforcement required for shear based on the offset requirements of different design codes and service crack control and transfer crack control at each cross section where reinforcement is required by calculation. It does not include the extra reinforcement added as shrinkage and temperature reinforcement required for slabs to AS3600. These are catered for and included in the final detailing stage.
    2. Ultimate Reinforcement - reinforcement required to be added to this face for ultimate strength. Includes ductility reinforcement. The difference between the Total and Ultimate reinforcement is the extra reinforcement required to be added for crack control at service or transfer and for Shear.
    3. Minimum Reinforcement - Some codes require special minimum reinforcement quantities for some designs, eg Minimum reinforcement for members with un-bonded prestressing tendons or the Ast.min reinforcement required by some codes for crack control. The reinforcement shown here is

the actual calculated code requirement. It does not necessarily need to be added to the member as sufficient reinforcement may have been added by the user or in the design. This is simply indicating the total code requirement for these rules.

2. Capacity - This diagram shows a plot of the applied and minimum moments against the actual calculated section capacity both at the initial condition and the final design condition. It shows if minimum capacity is controlling or if excess capacity has been provided by the user when any prestressing tendons or user defined reinforcement have been added. If there are ultimate moments on both faces at a point, both sets of values will be plotted. Plotted values are
    1. Minimum Moment - Minimum moment capacity required by the design code. Normally related to Cracking moment.
    2. Ultimate Design Moment - M* - Applied design moment
    3. Initial Ultimate Capacity - capacity of the section with any user defined tendons and reinforcement only
    4. Final Ultimate Capacity - Phi M_u - final ultimate capacity including all reinforcement and tendons at the section.
3. Neutral Axis Depth - normally used as an indicator of ductility - If there are ultimate moments on both faces at a point, both sets of values will be plotted. Nominated values are the values at potential plastic hinge locations. Plotted values are
    1. Initial Neutral Axis Depth - neutral axis depth with any user defined tendons and reinforcement only, consistent with the Initial Capacity
    2. Final Neutral Axis Depth - neutral axis depth including all reinforcement and tendons at the section, consistent with the final capacity.



Design Comments

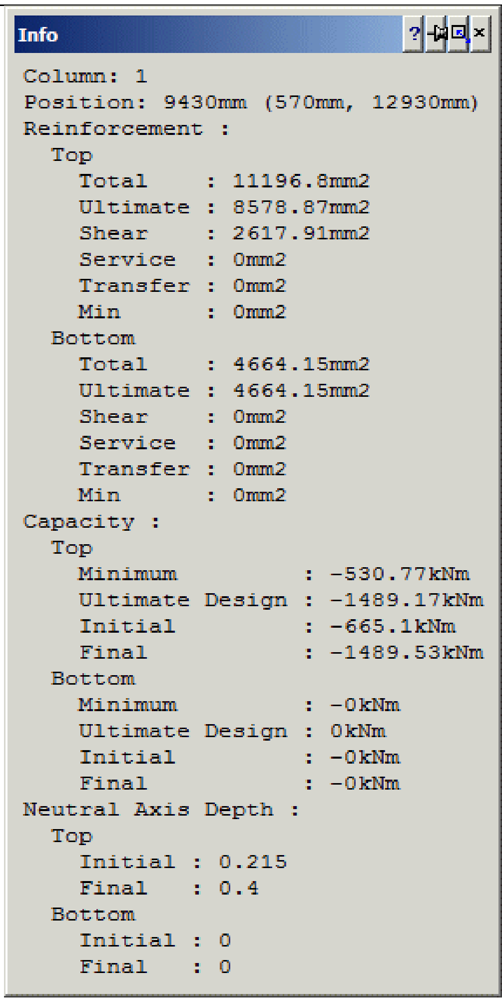
The design comments for the Ultimate Flexural report are generally to do with ductility (See text example above). In this case, it is simply reporting that ductility has been a problem at the right hinge location (at 9825mm from the left end of the span) in span 1 and that 2152mm2 of compression reinforcement has been added. The design is in accordance with the design code. The comment is simply letting the user know what happened when checking this section for ductility. The designer may not have realised that the member was having ductility problems and may wish to modify some of the input parameters, eg beam depth, in order to solve the ductility problem in another way.

Information Dialog

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The dialog will list:

1. Reinforcement - Top (above the zero axis) and Bottom (below the zero axis) reinforcement requirements are plotted at each design point. Plotted values are
    1. Total Reinforcement - Total reinforcement required to be added to this face. It is required to be fully developed past this point. It does not include reinforcement defined as User Defined Reinforcement in the input.
    2. Ultimate Reinforcement - reinforcement required to be added to this face for ultimate strength. Includes ductility reinforcement. The difference between the Total and Ultimate reinforcement is the extra reinforcement required to be added for crack control at service or transfer.
    3. Shear Reinforcement - extra flexural reinforcement above that listed for strength and serviceability, required to satisfy code requirements for the Truss Analogy for Shear design. In most design codes, this requirement is expressed as a simplified requirement for the extent of reinforcement past the point where it is required for strength, normally requiring the reinforcement to extend in the order of D past the point where it is required. This simplification does not work for prestressed members in many cases, so RAPT has been modified to use the more proper calculation for this, rather than the simplification of a reinforcement extent offset. This requires that the Tension force required at a point for ultimate strength be developed at the offset distance past that point in the direction of reducing moment. This tension force includes any prestressing tendons in the member plus the force from any reinforcement in tension. If extra tension force, above that required for ultimate strength at that point, is provided at a point in the form of tendons or extra reinforcement, only the proportion that is required for ultimate strength at that location needs to be developed at the offset point.
    4. Service Reinforcement - extra reinforcement, above that required for ultimate strength, required to limit crack control in accordance with code requirements and the user input in 7.2.7.1 Design Data
    5. Transfer Reinforcement - extra reinforcement, above that required for ultimate strength and service crack control, required to limit crack control in accordance with code requirements and the user input in 7.2.7.1 Design Data
    6. Minimum Reinforcement - Some codes require special minimum reinforcement quantities for some designs, eg Minimum reinforcement for members with un-bonded prestressing tendons or the Ast.min reinforcement required by some codes for crack control. The reinforcement shown here is the actual calculated code requirement. It does not necessarily need to be added to the member as sufficient reinforcement may have been added by the user or in the design. This is simply indicating the total code requirement for these rules.
2. Capacity - This diagram shows a plot of the applied and minimum moments against the actual calculated section capacity both at the initial condition and the final design condition. It shows if minimum capacity is controlling or if excess capacity has been provided by the user when any prestressing tendons or user defined reinforcement have been added. If there are ultimate moments on both faces at a point, both sets of values will be plotted. Plotted values are
    1. Minimum Moment - Minimum moment capacity required by the design code. Normally related to Cracking moment.
    2. Ultimate Design Moment - M* - Applied design moment
    3. Initial Ultimate Capacity - capacity of the section with any user defined tendons and reinforcement only
    4. Final Ultimate Capacity - Phi M_u - final ultimate capacity including all reinforcement and tendons at the section.
3. Neutral Axis Depth - normally used as an indicator of ductility - If there are ultimate moments on both faces at a point, both sets of values will be plotted. Plotted values are
    1. Initial Neutral Axis Depth - neutral axis depth with any user defined tendons and reinforcement only, consistent with the Initial Capacity
    2. Final Neutral Axis Depth - neutral axis depth including all reinforcement and tendons at the section, consistent with the final capacity.



7.3.7.2.1 Flexural Design - Ductility

RAPT checks the possible hinge locations in each span for ductility problems. To identify the hinge locations, three distinct moment zones are identified from the bending moment diagram as shown on the diagram below:-

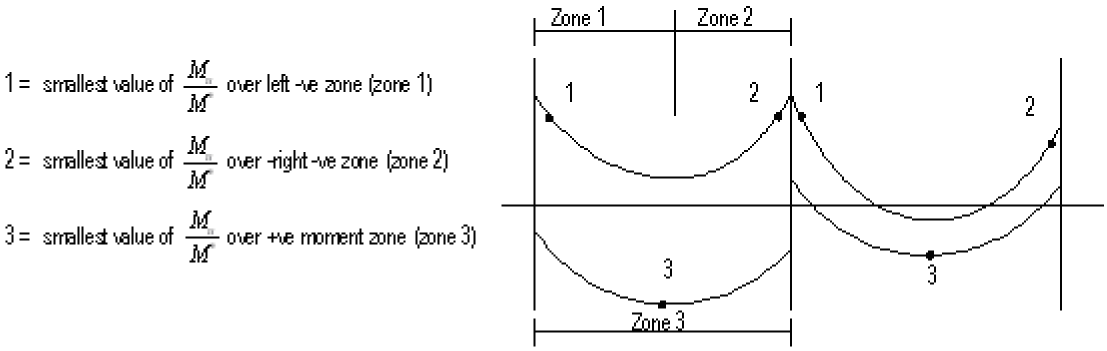
1. Left support zone
2. Right Support zone
3. Span zone

In ultimate strength calculations, k_u is the ratio of depth of the neutral axis to the d, the effective depth. RAPT calculates d based on the reinforcement in the tension zone of the member. The limit used for ductility is a limit on depth of neutral axis represented by k_u * d. This limit is set or implied by most codes. In cases where moment redistribution has been carried out, most codes require more ductile members and require more stringent limits on ductility, requiring lower limits on k_u * d than the general limit. The limit in 7.2.7.1 Design Data in the RAPT input is the general limit. If moment redistribution has been specified, RAPT will ignore any changes the designer has made to this limit and calculate a limit in accordance with the design rules in the code specified.

The following procedure is followed:-

1. An initial design check at each cross section in each zone placing no limit on ductility to determine the final section capacity, Mu,
2. Check each moment zone for the smallest value of M_u/M*, thus finding the three possible hinge locations. If two points have the same value, the point with the highest area of reinforcement required and then the point with the highest M* is selected.
3. Check the ductility properties of the three hinge locations by doing a complete ultimate strength and serviceability design at each of the hinge locations and determine the area of compression reinforcement required to make these sections ductile.

If the hinge location in a zone is found to be ductile, all points in the moment zone are considered to be ductile and no further check is done on ductility for points in that moment zone, even if a point away from the hinge location in that zone is not ductile. If a hinge location is found to be non-ductile and requires compression reinforcement, this reinforcement (from the ductility checks) is remembered and used as the maximum amount of compression reinforcement to be used at points within that moment zone.



The normal design procedure is then followed at all design points with ductility checks carried out at points within moment zones which were determined to be non-ductile. If the code ductility limit is exceeded at points within a non-ductile zone, RAPT will add compression reinforcement. It is possible to have a worse ductility problem (higher ku value) at an intermediate point along the span than at the hinge location which requires more compression reinforcement than the critical hinge location in that zone, especially in post-tensioned design with draped tendon profiles. At these locations RAPT will add compression reinforcement to improve ductility but the area of reinforcement added will be limited to the value calculated at the critical hinge location in that zone, so the final depth to neutral axis at those points may still be higher than the code limit.

Note: RAPT will sometimes add tension reinforcement to a section (during ductility check or ultimate check) in the following cases

1. When RAPT tries to add compression reinforcement, if ku * d < dc (ie thin sections with a small effective depth, where dc = distance to the compression reinforcement steel layer) and where the cover to the compression layer is too large, it is possible to get a situation where the compression reinforcement layer is in the tension zone. Thus RAPT can not add compression reinforcement as the compression reinforcement is layer in tension. To overcome this problem RAPT will add tension reinforcement in an attempt to increase the effective d, thus increasing ku * d. This may then move the compression reinforcement layer back into the compression zone.
2. In some cases, it is possible for the tendons to be in the compression zone with no reinforcement in the tension zone for the initial calculation of the ultimate capacity. As this case has no effective depth to the tension force, even though the section actually has an ultimate capacity, RAPT immediately adds an area of tension reinforcement sufficient, by itself, to provide the minimum moment capacity of 1.2*Mcr (or Mcr for BS8110, SABS 0100, EurocodeII). The section capacity is then recalculated. This check is independent of ductility requirements and will be carried out at any cross-section in the frame with the problem.

Design Errors

If 1 above is not successful, then the design cannot be completed using the current design parameters. At a design point where this occurs, RAPT will reduce the cover to the compression face reinforcement until a solution is possible. This is only done so that the designer will have some numbers on which to base his decision on how the problem can be fixed. The areas of reinforcement shown in the graphics will be zero and all output options past the Flexural Design options will not be available. Errors will be recorded in the Warnings file and, when printing is attempted, a dialog will be presented telling the designer that the results are not correct.

The designer must adjust the cover to the compression reinforcement or change the concrete cross section for the section to work.

7.3.7.3 Flexural Design - Service

RAPT checks service stress and crack control requirements for all members unless specifically instructed not to for reinforced concrete members. The following procedure is followed:-

1. Calculate steel stresses at decompression:- This calculation is only used by RAPT for prestressing calculations. When a reinforced cross section is being analysed, RAPT will set M_{decomp} to 0. The Decompression Moment is the moment required to cause a 0 stress state at the extreme tension face of the concrete. RAPT calculates the decompression moment, the stress at the extreme compressive fibre and the stress in each steel layer at this stress state. Most codes allow a maximum stress change in the reinforcement. This stress change is the change in stress induced in the steel as the member moves from the decompression moment to the applied service moments.
2. Calculate Stresses at service loading:- RAPT calculates the stresses at the top and bottom of the member, based on gross properties.
3. Cracked Section Analysis:- If the section is in tension (i.e. if the tension stress is > 0.1 * $f'c^{1/2}$) RAPT performs a cracked section analysis, calculating the concrete stress at the compressive fibre and the neutral axis (kd) depth that corresponds to this stress corresponding to the applied moments. The strain and curvature of the cross section are calculated. RAPT also calculates the stresses in each reinforcement layer based on a cracked section analysis. The change in stress for each reinforcement layer is calculated based on the stresses at decompression. Limiting the allowable change in stress controls the crack widths in the member. If this change in stress value is larger than the allowable stress change specified by the user in 7.2.7.2 Crack Control Design Data , then RAPT will added extra reinforcement until the stress in the reinforcement is less than or equal to the allowable stress change.
4. RAPT also makes a design comment if the tension stress exceeds nominal code limits or passes into another design category as set in the codes. [See T7.6]
   1. AS3600 clauses 8.6 and 9.4 specify the allowable stress changes for beams and slabs Note: other spacing requirements also apply in these clauses. See code.
   2. BS8110 clause 4.1.3 & 4.3.4.3. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   3. ACI318 clause 18.4.2 - RAPT warns the designer if these limits are exceeded.
   4. Eurocode2 clause 4.4.1 & Table 4.11. RAPT calculates results and warns users if the compressive strength in the concrete exceeds code allowable values.
   5. SABS 0100 clause 3.2.3. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   6. CP 65 clause 4.1.3 & 4.3.4.3. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   7. CP2004 clause 12.3.4. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   8. IS1343 clause 22.7. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
5. Under service loads it has been conservatively assumed that there is no increase in the stress in unbonded prestressed strand from the force in the strand after losses have taken place. RAPT uses moment curvature, strain compatibility and a Parabolic Rectangular Stress Strain curve to calculate the neutral axis depths. See T.7 for more information on these calculations.

Moment Reversal

It is often possible for a reversal of moments to occur at a cross-section. i.e. it is possible for both concrete faces to be in tension under different load conditions due to the envelope effects. RAPT will report results for the two extreme loading conditions if this occurs.

Crack Control- maximum reinforcement bar spacing/bar size

Calculation Logic

The crack control calculations in RAPT include the limitations in design codes on maximum spacing of reinforcement (all codes) and, in Eurocode 2 and AS3600, on maximum bar size. These limitations are often extensive and very different so we will cover them separately for the different code approaches. RAPT will not automatically add reinforcement to limit the stresses for crack control for reinforced concrete members. It will simply calculate the stresses and the corresponding bar size and cover limits. The designer can define a limit for the service stresses in 7.2.7.2 Crack Control Design Data "Maximum Service Stress Change - Reinforced Sections". If a value other than zero is defined here by a designer, RAPT will add reinforcement to limit service rebar stresses to this value. This can be useful in cases of concrete exposed to corrosive atmosphere or water retaining structures. The approach for different codes is. -

1. AS3600 - 2001 For reinforced concrete members the new AS3600-2001 design rules place limits on either bar size or bar spacing. These limits are dependent on the stress in the bar under service loads assuming that the concrete can carry no tensile stress. An upper limit on bar spacing of 2D or 300mm is applied.
   Amendment 2 to AS3600-2001 has removed the requirement for Ast.min. This calculation has been removed from RAPT in accordance with this amendment. If a designer wishes to comply with the old Ast.min requirements, the check will have to be done manually and any extra reinforcement added to the RAPT calculated requirements.
   AS3600 limits the types of structure for which crack control calculations are required provided the designer is willing to accept the presence of cracks wider than .3mm. The designer can nominate in 7.2.7.2 Crack Control Design Data to ignore crack width checks in this case. Allowing cracks of this size is not acceptable for surfaces of floors exposed to view or covered with brittle finishes, where the cracks will be visible and may be unsightly, and may not be acceptable to a designer's client in any case, but is left to the discretion of the designer by the AS3600 rules.

For prestressed concrete members, RAPT will automatically add reinforcement to keep the stress change in the reinforcement below the code limits for different tendon types. RAPT will report the maximum bar spacing which is 2D or 500mm for slabs and 200mm for beams. If the elastic stress (uncracked) under service loads is less than the limits in cl 8.6.2 (a) for beams or cl 9.4.2 (a) for slabs then no reinforcing spacing limit is applied (set to 0), otherwise the limit is set. No limit is placed on bar size for prestressed sections.

2. Eurocode 2 - For all concrete members with a depth greater than 200mm, Eurocode 2 places limits on either bar size or bar spacing. These limits are dependent on the stress in the bar under service loads assuming that the concrete can carry no tensile stress. An upper limit on bar spacing of 350mm or 1.5h is applied for all members. These rules also require a calculation for a minimum area of reinforcement to ensure that more than one crack forms. The calculated amount of this reinforcement is reported in the RAPT reinforcement graphics view as Minimum Reinforcement and the net amount required (above user defined and ultimate capacity reinforcement) is reported as serviceability reinforcement and is added to the cross-section before the reinforcement stresses are calculated. The net amount of reinforcement is also included in the Total Reinforcement required. . The amount reported here is the total needed in the Ast.min formula, not necessarily the amount that needs to be added to this member. The Total Reinforcement is still the amount of extra reinforcement that needs to be added to the member.
3. ACI 318-99 For reinforced concrete members, cl 10.6.4 provides a reinforcement spacing limit for crack control based on the stress in the reinforcement and the cover to the bar. The maximum limit for this spacing is also dependent on the reinforcement stress. ACI-318 suggest that this stress can be assumed to be 60% of the yield strength which is reasonable in normal situations but, where moment redistribution has been used or reinforcement exceeding that required for strength is supplied, the value will be incorrect. RAPT calculates the actual stress in the reinforcement under the service loading for this calculation. For prestressed concrete members, ACI318 also uses this rule for crack control with a couple of amendments. These are: No spacing limit is applied if the stress in the reinforcement is less than 20ksi and the limit is reduced if bonded tendons are included in the reinforcement being used to calculate the spacing. RAPT assumes that the spacing is being calculated based on the un-tensioned reinforcement only and therefore uses the full value of the spacing. If bonded tendons only are being used for crack control purposes then the maximum spacing produced should be reduced to 2/3 of the RAPT figure. If a combination of bonded tendons and un-tensioned reinforcement are being used to determine the actual spacing then the maximum spacing produced should be reduced to 5/6 of the RAPT figure.
4. BS8110, CP 65, CP2004, IS456/IS1343 and SABS-0100 For all concrete members, RAPT uses the crack width formula in Part 2 cl 3.8 equation 12. Strains in the reinforcement are calculated from RAPT's moment/curvature analysis of a cross-section rather than equation 13. RAPT uses a special case of the service load cross-section analysis, allowing for tensioning stiffening as per Part 2 of the code, to calculate these strains. The crack width used in the calculation is that defined in 7.2.7.2 Crack Control Design Data . The c_{min} used by RAPT is the depth to the "calculation reinforcement layer" defined by the user in 7.2.6.3 Reinforcement Design Zone. RAPT will attempt to calculate a value of acr and thus maximum bar spacing directly from a re-arranged version of this equation. Unfortunately, this is not always possible, depending on the strain in the reinforcement and the section dimensions, and can lead to misleading negative results. In those cases, RAPT will iterate in 25mm increments from a maximum spacing, as defined in the Code Defaults, downwards until the normal form of this equation results in a crack width less than the requested value. From the a_{cr} and c_{min} terms in this equation, RAPT calculates an allowable bar spacing. If the value of a_{cr} from this calculation is less than the cover, a negative value of the allowable spacing will result meaning that the crack control formula provides no solution and the designer will have to investigate other solutions to the design as discussed below. For prestressed cross-sections, if the elastic stress (uncracked) under service loads is less than the class 2 limit as defined in the code then no spacing limits will be applied by RAPT

Application of Crack Control Rules to Results

The bar spacings/maximum bar sizes determined by RAPT are based on any reinforcement defined by the designer plus any reinforcement added by RAPT for strength etc at each design cross-section all of which will affect the strains in the concrete and reinforcement and, thus, the crack widths in the member. The final results for a member may vary away from the critical strength cross-sections, as the design amount of reinforcement at these locations will be less than the final reinforcement supplied by the designer when the reinforcement is detailed. In some cases the spacing requirements may initially be worse at a point away from the critical strength cross-section but will not be critical once the final detailed reinforcing bar pattern is analysed. By iterating the design and defining the final reinforcing pattern in the 7.2.6.4 User Defined Reinforcement input the designer will get a better picture of the final pattern and also the effects of this reinforcement pattern can have a significant effect on the shear and deflection results.

Warning messages

If RAPT cannot find a bar size which fits within the bar size limits and/or bar spacing limits in a moment zone, a warning will be added to the warning report file and it will be up to the user to decide on the approach to be taken to solve the problem. In this case, using the concrete shape as defined and the bar size and bar spacing parameters defined by the user in the 7.2.6.1 Reinforcement->General input and using the maximum bar sizes and spacings determined by calculation, there is no solution.

Adding Reinforcement to Control Service stresses

1. reinforced concrete cross-sections RAPT will not automatically add reinforcement to limit the stress in the reinforcement and thus increase the allowable limits on bar size and bar spacing. If the only solution to a design is to limit these stresses, then the designer can set a maximum limit for reinforcement stress in 7.2.7.2 Crack Control Design Data and RAPT will add reinforcement where necessary to limit the reinforcement stresses as requested. This should be a last resort in a normal design as extra reinforcement is being added above that required for strength design and could lead to a less economical design. All other options should be investigated first. Some design situations (e.g. slabs exposed to corrosive atmosphere) may require stronger limits on crack control than those provided in the normal code rules (.3 - .4mm crack width) and this feature can be used to limit the stress in the reinforcement in these cases.

2. prestressed concrete cross-sections, RAPT will always limit the stress in the reinforcement by default. Designers can over-ride this default behaviour by modifying the stress limit for prestressed sections in 7.2.7.2 Crack Control Design Data. Setting this value to 0 will result in no stress limitation being applied; otherwise any desired limit can be set.

   If the concrete tension stress is greater than the nominal stress defined in the relevant design standard as the stress above which cracking must be considered, RAPT will automatically check to ensure that an area of bonded reinforcement at least equal to the smaller of .001 * Agross and 12@300 for slabs or 16@300 for beams is available near the tension face (within the tension quarter of the section depth but not greater than 200mm from the tension face or less than the tension reinforcement cover + 10mm). If this area of bonded reinforcement is not available, the area of bonded reinforcement will be increased to this level before the cracked section calculations are started.

Presentation of bar size and spacing limits

To view the values of maximum bar spacing and maximum bar size at each design cross-section, use Information Dialog while the Serviceability Graphics is showing on the screen. RAPT will show the maximum bar spacing and maximum bar size where applicable for that cross-section. These values are based on the reinforcement required at that cross-section and are independent of reinforcement requirements at other cross-sections in the span/member. To force RAPT to give results based on a specific reinforcement layout, the designer will need to define that reinforcement pattern in the input as 7.2.6.3 User Defined Reinforcement.

As discussed above, the spacing/bar size results are applied in determining the reinforcement bar sizes and spacings shown in the reinforcement summary. The maximum bar spacing and maximum bar size are listed at each cross-section along with the required area of reinforcement and the suggested bar size and spacings for both top and bottom reinforcement.

Reinforcement required to satisfy AS3600 Shrinkage and Temperature requirements for slabs according to clause 9.4.3 is not included in this reinforcement summary.

A value of maximum bar spacing of zero means one of 2 things

1. There was no tensile stress on this face at service so there was no need for a crack control check.
2. For prestressed members, in accordance to the following specific code rules crack control checks have not been carried out,
   1. AS3600, the gross concrete stress at that face is less than the limits in cl 8.6.2 (a) for beams or cl 9.4.2 (a) for slabs
   2. ACI318, the stress in the reinforcement is less than 20ksi
   3. BS8110, CP 65, CP2004, IS1343 and SABS-0100, the member is class 1 or class 2 at that cross-section

Text View

The following results are included in this table

1. The locations for the cross sections at which the results have been calculated. These are measured from the left end of each span. At some locations, RAPT will seem to repeat a point. This occurs whenever a moment reversal occurs, so that RAPT can design and report for both moments.
2. The moments listed in this report include
   1. M_{ser} - The Service Moment Combination using Mp (total prestress moments = p * e + M_{sec}) for the prestress effect. M_{ser} is used for calculations based on uncracked sections.
   2. M_{smod} - The Modified Service Moment Combination using M_{sec} (Secondary prestress moments) for the prestress effect is used in cracked section analysis. The difference between these moments is p * e
3. The stresses based on Gross Section Properties listed include
   1. P/A - The axial compression (P/A) is presented for prestressed systems and is the axial stress induced into the concrete by the tendons.
   2. Top and Bottom Stresses - RAPT calculates the top and bottom concrete fibre stresses for the concrete, based on gross section properties. This assumes no cracking occurs in the tension zone, thus these stresses are calculated at service conditions using M_{ser}.
   3. I_{gross} Transformed - The transformed gross section inertia allowing for all reinforcement in the section and short term concrete properties.
4. Cracked Section Properties. If the cross section has a face in tension, RAPT performs a cracked section analysis, calculating .
   1. Comp Stress - the compression stress at service at the compression face in the cracked section.
   2. k_{dser} - the depth to the neutral axis at service in the cracked section.
   3. Tension Stress Change - the maximum stress change in the tension reinforcement (final after any extra reinforcement is added).
   4. Extra Tension Reinforcement - the extra tension reinforcement (above that supplied in input and calculated by RAPT in the Ultimate calculation) required to limit stresses at service as required in input. A strain compatibility analysis is carried out to determine these values. The tension reinforcement shown in this table is the reinforcement added by RAPT to restrict the change in stress (from decompression to service) to the limit set by the designer. This reinforcement is used to control cracking in a member
   5. Maximum Spacing - The maximum bar spacing allowed by the code to ensure the crack width is within design limits - see theory above
   6. Maximum Bar Size - The maximum bar size allowed by some codes to ensure the crack width is within design limits - see theory above

7. The cracked inertia of the cross-section. If the cross-section is not cracked then the gross inertia (or factored inertia if input has a reduction factor applied) will be printed here. No allowance is made for tension stiffening in calculating this value

Span 1

	Design Moment		Gross Concrete Stress				Cracked Sect Results						
Locat	Mser	Msmod	P/A	Top	Bottom	Igross Transform	Comp Stress	Kdser	Tension Stress Change	Extra Tension Reinforcement	Maximum Spacing	Maximum Bar Size	Cracked Inertia
mm	kNm	kNm	MPa	MPa	MPa	mm4	MPa	mm/mm	MPa	mm2	mm	mm	mm4
175	-407.12	-560.36	1.33	-4.97	10.07	1.12e10	14.78	157.308	149.62	160.89	200	0	3.75e9
570	-270.13	-388.81	1.35	-2.83	7.15	1.12e10	9.45	195.578	59.9	0	0	0	5.14e9
1535	-48.66	-32.79	1.37	0.62	2.41	1.09e10	2.41	536.927	0.27	0	0	0	1.09e10
2500	115.13	233.73	1.38	3.16	-1.09	1.1e10	3.72	220.984	17.82	0	0	0	6.63e9
3333	211.01	391.82	1.39	4.66	-3.14	1.13e10	7.27	129.475	94.61	0	0	0	3.53e9
4166	264.73	483.22	1.4	5.49	-4.29	1.14e10	8.82	118.086	133.46	0	200	0	3.28e9
5000	276.28	507.91	1.39	5.66	-4.55	1.14e10	9.2	115.325	144.62	0	200	0	3.19e9
5833	245.63	465.84	1.37	5.17	-3.9	1.14e10	8.39	119.543	124.26	0	200	0	3.25e9
6666	172.82	357.08	1.36	4.04	-2.35	1.12e10	6.13	142.317	68.13	0	0	0	3.81e9
7500	57.69	181.38	1.35	2.24	0.11	1.09e10	2.24	420.854	0	0	0	0	1.09e10
8465	-128.26	-105.35	1.34	-0.65	4.09	1.1e10	3.96	335.612	4.06	0	0	0	1.05e10
9430	-371.9	-481.58	1.32	-4.43	9.3	1.18e10	11.54	177.844	90.1	0	200	0	5.16e9
9825	-517.97	-661.4	1.29	-6.72	12.42	1.14e10	16.15	170.466	143.11	1077.57	200	0	4.66e9

Design Comments:-

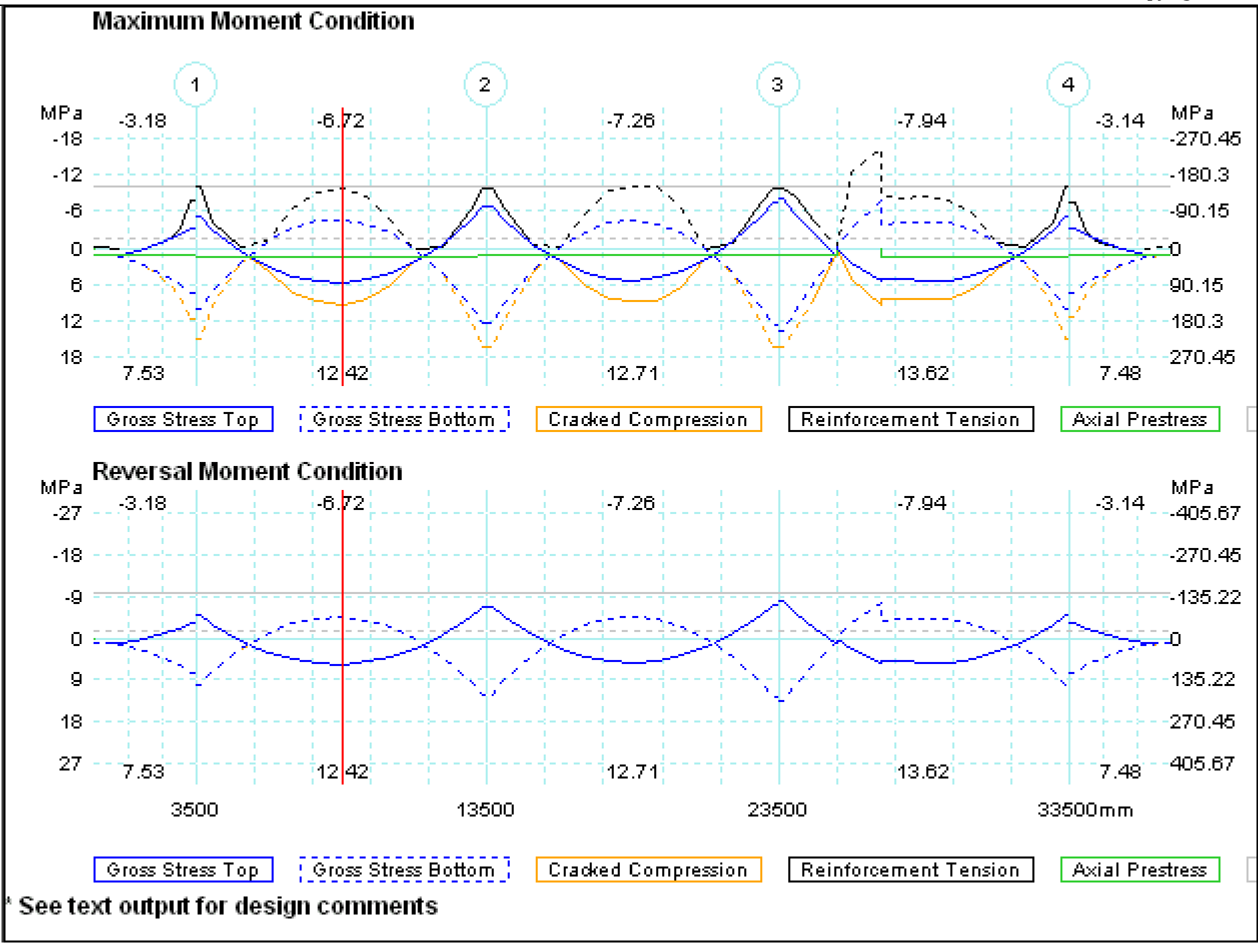
- SERVICE - Concrete tensile stresses greater than allowed in accordance with AS 3600 Clauses 8.6 _9.4
- Increase in stress from decompression to working exceeds 150MPa in at least one steel layer. Reinforcement has been added to reduce stress change below this maximum value. Refer RAPT manual Section 7.7.6

Graphical View

The graphical output consists of 2 separate diagrams for the main stress condition which represents the maximum service moment at each location and a reversal stress condition which represents the lesser moment at each location. The format of the data is the same in both. All values use the left scale unless noted otherwise. The data displayed is:-

1. Gross Concrete Stress Top/Bottom Stresses on each cross section. The top surface stresses are plotted as a solid line and the bottom stresses are plotted as a dotted line of the same colour. The maximum tensile and compressive stresses in each span are printed above and below the mid-span point in each span.
2. Cracked Section Concrete Compression Stress for the concrete compressive surface. RAPT plots the compressive surface stresses calculated based on a cracked section analysis. If a section has not cracked, then the gross section stresses will plot on the same line. The same line style convention applies as above with the top surface being plotted as a solid line and the bottom surface plotted as a dotted line of the same colour.
3. Cracked Section Tension Stress for reinforcement in tension. A plot of the tension stresses in reinforcement is plotted. RAPT will always plot the largest tension stress at each cross-section (i.e. If RAPT has more than one layer of steel, the largest tension stresses from all layers will be plotted). Top reinforcement stresses will be plotted as a solid line and bottom reinforcement tensile stresses as a dotted line of the same colour. The Vertical scale used for this line is the right hand scale.
4. P/A. The pre-compression stress from the tendons is plotted as a dotted line on the maximum moment case only. This shows the amount of pre-compression in the members and how it varies. Note that this line is the offset of the top and bottom fibre stresses from the zero stress line and actually passes through the intersection points of the top and bottom surface gross stress lines.
5. Reinforcement Stress limit. The allowable reinforcement stress limit, as specified in input , is plotted to show users how the reinforcement is working in the cross section. The Vertical scale used for this line is the right hand scale.
6. Tension Crack Control limit. The nominal allowable tensile stress in concrete before the concrete is classed as cracked is plotted according each code. This limit line may not show on the plot if the scale does not extend to the limit value.
7. Concrete Compression Limit. Some codes give a compressive limit for the concrete.

NOTE:- Concrete stresses use the left side vertical scale while steel stresses use the right side vertical scale.



Design Comments

Design comments will be displayed to indicate relative stress levels compared to code limits and also if reinforcement has had to be added for crack control. These are for information only and do not indicate that a design has failed. RAPT will have added sufficient reinforcement to ensure that crack control provisions have been complied with as long as relevant maximum bar size and spacing limits are complied with and that sections are ductile based on the design parameters defined by the designer in the input. In some cases RAPT will not have been able to define reinforcement layouts which conform to the crack spacing rules for spacing or bar size. In these cases warnings and design comments will be posted and it is up to the designer to modify the reinforcement layout to achieve a logical solution.

Information Dialog

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The dialog will list:

Text

1. Point Position from the left end of the span (and from the right end of the span and left end of the frame in brackets)
2. P/A - The axial compression (P/A) is presented for prestressed systems and is the axial stress induced into the concrete by the tendons.
3. Cracked Section Tension Stress Change for reinforcement in tension
4. Maximum Spacing - The maximum bar spacing allowed by the code to ensure the crack width is within design limits (not shown if 0) - see theory above
5. Maximum Bar Size - The maximum bar size allowed by some codes to ensure the crack width is within design limits (not shown if 0)- see theory above
6. Code Stress limits for concrete and reinforcement where applicable.

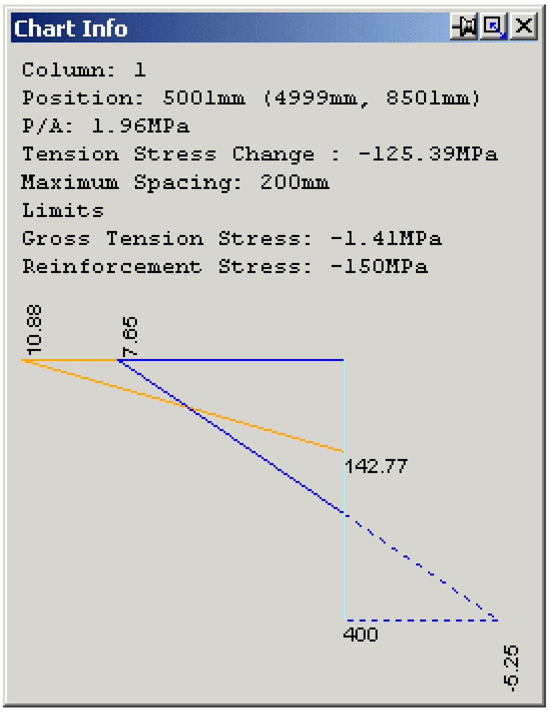
Graphics

This stress diagram is orientated the same as the member, the top level is the extreme top surface of the concrete section and the bottom level is the extreme bottom surface of the concrete section. The line colours and styles are the same as for the graphical representation above. The stresses are plotted as compression (positive) to the left of the axis and tension (negative) to the right of the axis. The stresses shown are

1. Gross Concrete Stress Top - solid line
2. Gross Concrete Stress Bottom - dotted line of the same colour as 1

3. Cracked Section Concrete Compression Stress for the concrete compressive surface. Solid (as shown) or dotted line depending on the line type for the face in compression, top face solid, bottom face dotted. If this line does not show then the section is uncracked and it is hidden by the uncracked section line.

As well, the total depth of the section and the depth of the neutral axis, both measured from the top, are shown on the vertical axis.



7.3.7.4 Flexural Design - Transfer

RAPT checks service stress and crack control requirements for all prestressed members. The following procedure is followed:-

1. Calculate steel stresses at decompression:- . The Decompression Moment is the moment required to cause a 0 stress state at the extreme tension face of the concrete. RAPT calculates the decompression moment, the stress at the extreme compressive fibre and the stress in each steel layer at this stress state. Most codes allow a maximum stress change in the reinforcement. This stress change is the change in stress induced in the steel as the member moves from the decompression moment to the applied service moments.
2. Calculate Stresses at transfer loading:- RAPT calculates the stresses at the top and bottom of the member, based on gross properties.
3. Cracked Section Analysis:- If the section is in tension (i.e. if the tension stress is > 0.1 * $f'ct^{1/2}$) then RAPT performs a cracked section analysis, calculating the concrete stress at the compressive fibre and the neutral axis (kd) depth that corresponds to this stress corresponding to the applied moments. The strain and curvature of the cross section are calculated. RAPT also calculates the stresses in each reinforcement layer based on a cracked section analysis. The change in stress for each reinforcement layer is calculated based on the stresses at decompression. Limiting the allowable change in stress controls the crack widths in the member. If this change in stress value is larger than the allowable stress change specified by the user in 7.2.7.1 Design Data , then RAPT will added extra reinforcement until the stress in the reinforcement is less than or equal to the allowable stress change.
   If the tensile stress based on gross properties is greater than .25 * $f'ct^{1/2}$, RAPT will ensure that a minimum amount of un-tensioned reinforcement is provided at the tension face before the cracked section analysis is done. For the ACI code this minimum amount is based on clause 18.9.3.2, otherwise the minimum is set to .1% of the area of the concrete in the section but not greater than .2 times the tension face width. If the stress in the reinforcement is greater than the limit, extra reinforcement will be added to reduce the stresses to the limit.
4. RAPT also makes a design comment if the tension stress exceeds nominal code limits or passes into another design category as set in the codes. [See T7.6]
   1. AS3600 clauses 8.6 and 9.4 specify the allowable stress changes for beams and slabs Note: other spacing requirements also apply in these clauses. See code.
   2. BS8110 clause 4.3.5.1 and 4.3.5.2 - RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   3. ACI318 clause 18.4.1 - RAPT warns the designer if these limits are exceeded.
   4. SABS 0100 clause 5.3.2.3.2. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   5. CP 65 clause 4.3.5.1 and 4.3.5.2 - RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   6. CP2004 clause 12.3.4. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
   7. IS1343 clause 22.7. RAPT calculates the tension stress and informs the designer of the class of member according to the code.
5. Under transfer loads it has been conservatively assumed that there is no increase in the stress in unbonded prestressed strand from the force in the strand after losses have taken place. RAPT uses moment curvature, strain compatibility and a Parabolic Rectangular Stress Strain curve to calculate the neutral axis depths. See T.7 for more information on these calculations.

Text View
The following results are included in this table

1. The locations for the cross sections at which the results have been calculated. These are measured from the left end of each span. At some locations, RAPT will seem to repeat a point. This occurs whenever a moment reversal occurs, so that RAPT can design and report for both moments.
2. The moments listed in this report include
   1. M_{tran} - The Transfer Moment Combination using Mp (total prestress moments = p_t * e + M_{sec}) for the prestress effect. M_{tran} is used for calculations based on uncracked sections.
   2. M_{tmod} - The Modified Transfer Moment Combination using M_{sec} (Secondary prestress moments) for the prestress effect is used in cracked section analysis. The difference between these moments is p_t * e
3. The stresses based on Gross Section Properties listed include
   1. P/A - The axial compression (P_t / A) is the axial stress induced into the concrete by the tendons.
   2. Top and Bottom Stresses - RAPT calculates the top and bottom concrete fibre stresses for the concrete, based on gross section properties. This assumes no cracking occurs in the tension zone, thus these stresses are calculated at transfer conditions using M_{tran} .
4. Cracked Section Properties. If the cross section has a face in tension, RAPT performs a cracked section analysis, calculating .
   1. Comp Stress - the compression stress at service at the compression face in the cracked section.
   2. k_{dser} - the depth to the neutral axis at service in the cracked section.
   3. Tension Stress Change - the maximum stress change in the tension reinforcement (final after any extra reinforcement is added).
   4. Extra Tension Reinforcement - the extra tension reinforcement (above that supplied in input and calculated by RAPT in the Ultimate and service stress calculations) required to limit stresses at transfer as required in input. A strain compatibility analysis is carried out to determine these values. The tension reinforcement shown in this table is the reinforcement added by RAPT to restrict the

change in stress (from decompression to service) to the limit set by the designer. This reinforcement is used to control cracking in a member

Span 1

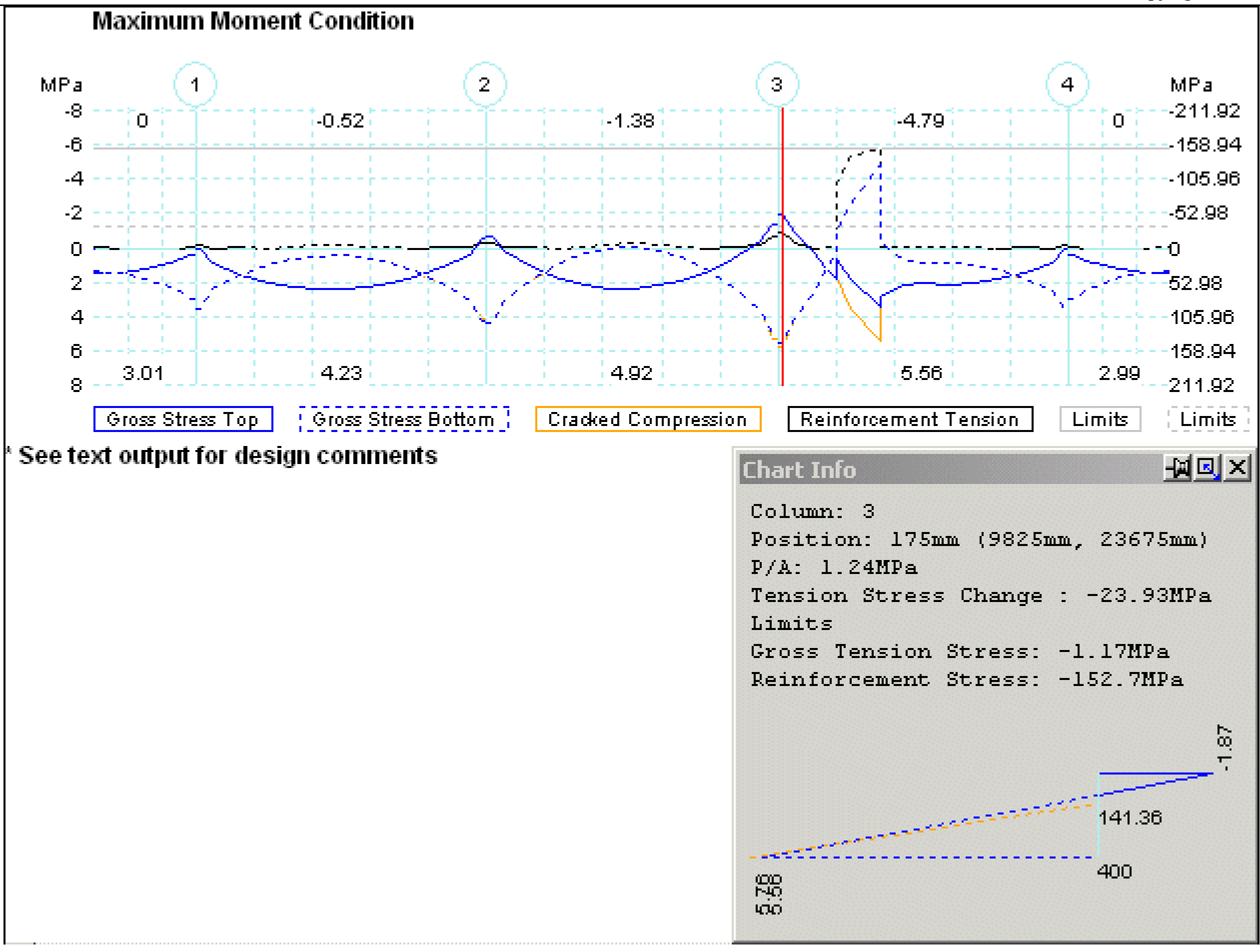
	Design Moment		Gross Concrete Stress			Cracked Sect Results			
Locat	Mtran	Mtmod	P/A	Top	Bottom	Comp Stress	Kdser	Tension Stress Change	Extra Tension Reinforcement
mm	kNm	kNm	MPa	MPa	MPa	MPa	mm/mm	MPa	mm2
175	-92.4	-266.74	1.51	0.08	3.49	3.49	409.23	3.16	0
570	-39.31	-174.35	1.53	0.93	2.38	2.38	655.345	0	0
1535	-0.19	17.84	1.56	1.55	1.56	1.56	87909.9	0.62	0
2500	27.52	162.43	1.57	2.	0.98	2.	786.264	0	0
3333	43.3	248.96	1.59	2.26	0.66	2.26	564.507	1.46	0
4166	51.51	300.02	1.59	2.39	0.49	2.39	502.425	3.27	0
5000	52.17	315.62	1.58	2.39	0.46	2.39	495.163	3.84	0
5833	45.26	295.71	1.56	2.26	0.59	2.26	541.673	2.76	0
6666	30.8	240.33	1.55	2.03	0.89	2.03	712.271	0.48	0
7500	8.76	149.35	1.54	1.67	1.35	1.67	2065.99	0	0
8465	-26.2	-0.28	1.52	1.11	2.08	2.08	860.376	0.43	0
9430	-72.56	-197.51	1.5	0.38	3.06	3.06	456.142	0.9	0
9825	-128.62	-291.97	1.47	-0.52	4.23	4.16	348.194	6.94	0

Graphical View

The graphical output consists of a single diagram .All values use the left scale unless noted otherwise. The data displayed is:-

1. Gross Concrete Stress Top/Bottom Stresses on each cross section. The top surface stresses are plotted as a solid line and the bottom stresses are plotted as a dotted line of the same colour. The maximum tensile and compressive stresses in each span are printed above and below the mid-span point in each span.
2. Cracked Section Concrete Compression Stress for the concrete compressive surface. RAPT plots the compressive surface stresses calculated based on a cracked section analysis. If a section has not cracked, then the gross section stresses will plot on the same line. The same line style convention applies as above with the top surface being plotted as a solid line and the bottom surface plotted as a dotted line of the same colour.
3. Cracked Section Tension Stress for reinforcement in tension. A plot of the tension stresses in reinforcement is plotted. RAPT will always plot the largest tension stress at each cross-section (i.e. If RAPT has more than one layer of steel, the largest tension stresses from all layers will be plotted). Top reinforcement stresses will be plotted as a solid line and bottom reinforcement tensile stresses as a dotted line of the same colour. The Vertical scale used for this line is the right hand scale.
4. Tension Crack Control limit. The nominal allowable tensile stress in concrete before the concrete is classed as cracked is plotted according each code. This limit line may not show on the plot if the scale does not extend to the limit value.
5. Reinforcement Stress limit. The allowable reinforcement stress limit, as specified in input , is plotted to show users how the reinforcement is working in the cross section. The Vertical scale used for this line is the right hand scale.
6. Concrete Compression Limit. Some codes give a compressive limit for the concrete.

NOTE:- Concrete stresses use the left side vertical scale while steel stresses use the right side vertical scale.



Design Comments
Design comments will be displayed to indicate relative stress levels compared to code limits and also if reinforcement has had to be added for crack control. These are for information only and do not indicate that a design has failed. RAPT will have added sufficient reinforcement to ensure that crack control provisions have been complied with as long as relevant maximum bar size and spacing limits are complied with and that sections are ductile based on the design parameters defined by the designer in the input. In some cases RAPT will not have been able to define reinforcement layouts which conform to the crack spacing rules for spacing or bar size. In these cases warnings and design comments will be posted and it is up to the designer to modify the reinforcement layout to achieve a logical solution.

Information Dialog
To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The dialog will list:

Text

1. Point Position from the left end of the span (and from the right end in brackets)
2. P/A - The axial compression (P/A) is the axial stress induced into the concrete by the tendons.
3. Cracked Section Tension Stress Change for reinforcement in tension
4. Code Stress limits for concrete and reinforcement where applicable.

Graphics
This stress diagram is orientated the same as the member, the top level is the extreme top surface of the concrete section and the bottom level is the extreme bottom surface of the concrete section. The line colours and styles are the same as for the graphical representation above. The stresses are plotted as compression (positive) to the left of the axis and tension (negative) to the right of the axis. The stresses shown are

1. Gross Concrete Stress Top - solid line
2. Gross Concrete Stress Bottom - dotted line of the same colour as 1
3. Cracked Section Concrete Compression Stress for the concrete compressive surface. Solid or dotted line depending on the line type for the face in compression, top face solid, bottom face dotted. If this line does not show then the section is uncracked and it is hidden by the uncracked section line (as shown).

As well, the total depth of the section and the depth of the neutral axis (if cracked), both measured from the top, are shown on the vertical axis.

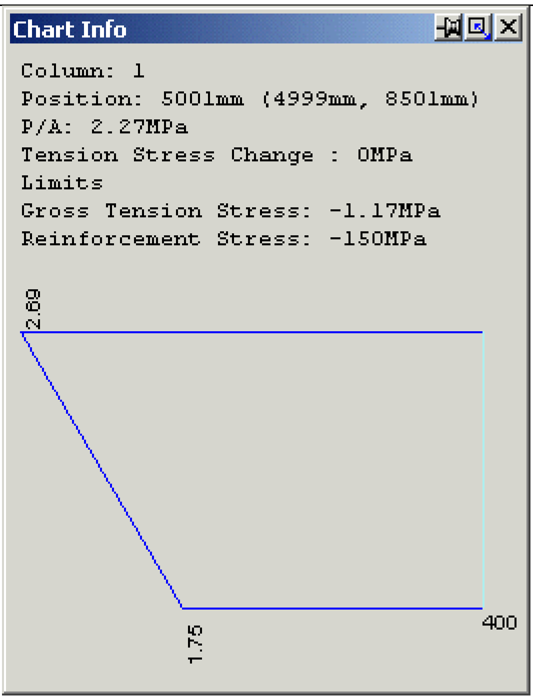


Chart Info
Column: 1
Position: 5001mm (4999mm, 8501mm)
P/A: 2.27MPa
Tension Stress Change : 0MPa
Limits
Gross Tension Stress: -1.17MPa
Reinforcement Stress: -150MPa
2.69
1.75
400

7.3.8 Shear Design - Beam

RAPT checks beam shear for all frame members.

Load Combinations
When checking shear, RAPT uses the shears and moments from the Load Combinations for Ultimate Flexure and Ultimate Shear. Thus RAPT will check four shear force / moment combinations during design. These are

1. Maximum shear on section with co-existing moment
2. Reversal shear on section with co-existing moment.
3. Maximum moment on section with co-existing shear
4. Reversal moment on section with co-existing shear

All four checks are always carried out, even if reversal cases are of the same sign as the maximum case.

The results reported will be those for the controlling case from the following in order of significance

1. the case requiring the largest area of shear reinforcement of the four cases.
2. If more than one case requires the same area of reinforcement but one requires a smaller spacing of reinforcement then the case with the smaller spacing is reported
3. If more than one case requires the same area of reinforcement or no cases require shear reinforcement, the case with the highest applied shear force will be reported..

In calculating the beam shear forces and moments, for

1. Two Way Slabs - RAPT uses the load applied to the column strip using the section shape of the column strip.
2. Two Way Beams - RAPT applies the percentage of load specified on the effective beam shape to check beam shear.
3. All other systems - RAPT uses the total applied load and applies it to the effective cross-section.

The shear width is taken as the thinnest width of concrete in the cross-section that extends the full depth between the top and bottom design reinforcement layers at that cross-section. The depths of the top and bottom design reinforcement layers are defined in 7.2.6.3 Reinforcement Design Zones in the input and can be viewed graphically there.

Beam shear is checked at all design cross-sections between points .8 * D from the face of the supports (column face or column capital face if it exists) unless shear enhancement has been requested in which case it is checked to the face of the support.

RAPT also reports a shear check at the face of the column, even if one is not requested. This is to check the maximum shear which should be checked at all points between the faces of the supports. Shear reinforcement will not be checked for or added at this location unless the designer requests it in the input.

The designer can specify the 3 combinations of stirrup size and number of legs in the 7.2.6.1 Reinforcement General input screen. As well, RAPT calculates the minimum number of legs required to satisfy code transverse leg spacing rules at each cross-section. RAPT gives stirrup set spacing results for all 4 combinations. The "minimum legs" combination uses the smallest bar size of the highest strength reinforcement of the three options nominated in the input to determine the stirrup set spacing.

Text View
The following results are included in this table for each cross-section at which beam shear design has been carried out in each span:-

1. the location measured from the left end of the span
2. V* - the applied shear force at each point
3. M_v* - the co-existing applied moment at each point
4. M_{dec} - the decompression moment at each point except for ACI318 for which the cracking moment is reported. The sign of the decompression moment indicates the sign of the stress from the prestress at the face that is tensile under the ultimate moment, M_v*. A negative decompression moment indicates that the prestress is inducing tension at this face and a positive decompression moment indicates that the prestress is inducing compression at this face.
5. d - the effective depth used in shear calculations. For BS8110, CP65 and SABS0100, this is the effective depth, not the shear tie depth d_t which is used in calculations where appropriate but not reported
6. A_{st} - the area of tension reinforcement calculated by RAPT, in excess of that defined in input, that has been used for the shear calculations at this cross-section. To see the user defined reinforcement and tendons also taken into account see 7.3.7.1 Flexural Design Reinforcement Text output.
7. b_v - the effective shear width used at each point. This value allows for deductions for prestressing ducts as appropriate. If the net width is less than 20mm, RAPT will ignore shear calculations at this point and leave it to the designer to justify the shear design at this point.
8. Phi V_{uc} - the flexural cracking shear capacity at this cross-section
9. Phi V_{ut} - the principal tensile shear capacity at this cross-section. If the value is 99999, then no check was carried out. The value printed is the lowest of the values at the centroid of the section and those at any sudden section change point in the depth of the member, e.g. at flange/web junction.
10. Phi V_u - the controlling concrete shear capacity at each point. The concrete shear strength, without any shear reinforcement, calculated according to the code type specified.

11. Phi Vumax - The maximum shear the section can carry. This is normally limited by the compression stress in the diagonal strut.
12. Phi Vumin - the section shear capacity with minimum shear reinforcement added.
13. Phi Vus - Shear strength provided by the shear reinforcement Asv/s. This could be different to the shear strength provided by the nominated stirrups as spacing limits may have controlled requiring extra shear reinforcement over that required for strength.
14. Theta - Shear Strut angle used in the calculation of the shear reinforcement required.
15. the shear reinforcement requirements in two forms
    1. Generally as the area of reinforcement / spacing (for a specified reinforcement yield strength). This value is the larger of the calculated Asv/s and Asv/s based on the limiting longitudinal spacing requirements as set in the relevant design code.
    2. The tie spacings for four different bar types and size and numbers of legs, the first three as defined in the 7.2.6.1 Reinforcement General input screen and the fourth using the minimum number of legs at that cross-section and the smallest bar size of the largest steel strength of the three options defined in the input.
    3. The minimum number of shear tie legs required to satisfy code transverse spacing limits for this cross-section.
16. Comments on the shear design at this cross-section.
    1. No shear steel.
    2. Web width too small. Design web width is less than 20mm and RAPT cannot design the section for shear. This could be because there is a horizontal penetration full width of the beam, in which case the designer must design the shear transfer across the penetration based on the shear data supplied here, or because the sum of the prestress duct diameters is too large, possibly because ducts that are arranged vertically have been counted individually, see 7.2.5 Prestress Input Control Grid.

Span 1

Locat	V*	Mv*	Mdec	d	Ast	bv	phi Vuc	phi Vut	phi Vu	Phi Vumax	phi VuMin	phi Vus	Theta	Asv/s	Spacing of Sets				Minimum Legs	Shear Comments
															2 legs N10	2 legs N12	2 legs N16	Min legs N10		
mm	kN	kNm	kNm	mm	mm2	mm	kN	kN	kN	kN	kN	kN	#.#	mm2/mm	mm	mm	mm	mm	#	A
250	1155.36	-1438.78	526.45	361.5	6110.43	1360	0	0	0	2284.33	0	0	0	0	0	0	0	0	0	
570	1068.68	-1082.94	471.48	346	3169.43	1360	816.36	99999	816.36	2278.69	1010	252.32	30.6937	1.24	126.9	182.7	200	200	5	
1000	952.19	-648.45	348.67	346	1809	1360	830.91	99999	830.91	2293.32	1024.56	121.28	30	0.95	164.9	237.4	300	300	5	Minimum Steel
1144.3	913.1	-513.87	308.88	320	0	1360	787.39	99999	787.39	2130.91	966.48	125.71	30	0.95	164.9	237.4	300	300	5	Minimum Steel
1521.3	814.71	-209.88	211.87	320	0	1360	968.14	956.31	956.31	2122.71	1135.4	0	30	0	0	0	0	0	0	No shear steel
1900	712.13	79.22	175.37	320	0	1360	99999	950.93	950.93	2116.8	1130.02	0	30	0	0	0	0	0	0	No shear steel
1999	685.31	148.39	198.98	320	0	1360	99999	949.1	949.1	2115.29	1128.19	0	30	0	0	0	0	0	0	No shear steel
2001	684.75	149.76	164.27	360	0	1360	99999	1008.62	1008.62	2361.89	1210.09	0	30	0	0	0	0	0	0	No shear steel
2500	555.91	390.02	276.1	396	603	1360	693.64	968.17	693.64	2572.69	915.26	0	30	0	0	0	0	0	0	No shear steel
2899	441.84	589.08	350.07	396	603	1360	533.17	852.39	533.17	2505.72	754.8	0	30	0	0	0	0	0	0	No shear steel
2901	441.27	589.96	339.45	396	603	1395	523.13	99999	523.13	2567.21	750.46	0	30	0	0	0	0	0	0	No shear steel
2999	413.25	631.83	351.38	396	603	1395	489.42	99999	489.42	2550.36	716.75	0	30	0	0	0	0	0	0	No shear steel
3001	412.69	632.66	289.77	346	1528.39	1395	479.19	99999	479.19	2236.96	677.81	0	30	0	0	0	0	0	0	No shear steel
3500	285.6	702.5	309.39	346	2609.96	1395	382.95	99999	382.95	2155.63	581.58	0	30	0	0	0	0	0	0	No shear steel
4000	168.9	818.47	306	346	3411.35	1395	357.88	99999	357.88	2155.87	556.51	0	30	0	0	0	0	0	0	No shear steel
4499	-98.1	908.96	302.78	346	3627.83	1395	354.01	99999	354.01	2168.67	552.64	0	30	0	0	0	0	0	0	No shear steel
4999	-233.55	826.05	292.44	349.1	3338.49	1395	415.68	99999	415.68	2219.76	616.08	0	30	0	0	0	0	0	0	No shear steel
5001	-233.92	825.58	282.21	349.1	3982.31	1045	369.08	99999	369.08	1672.48	519.2	0	30	0	0	0	0	0	0	No shear steel
5751	-325.43	679.36	214.12	346	3130.35	1045	399.72	99999	399.72	1688.82	548.51	0	30	0	0	0	0	0	0	No shear steel
6500	-426.55	395.22	139.67	346	1603.07	1045	399.2	99999	399.2	1687.1	547.99	27.35	30	0.73	214.6	300	300	300	4	Minimum Steel
7000	-498.77	211.88	91.03	346	804	1045	420.65	676.55	420.65	1685.62	569.45	78.12	30	0.73	214.6	300	300	300	4	Minimum Steel
7500	-421.61	119.93	45.41	346	988.77	1045	387.1	675.65	387.1	1684.95	535.9	34.51	30	0.73	214.6	300	300	300	4	Minimum Steel
7999	-638.87	-316.88	221.34	320	0	1045	549.7	681.64	549.7	1563.69	687.32	89.16	30	0.73	214.6	300	300	300	4	Minimum Steel
8001	-639.29	-318.16	215.29	320	0	1445	582.88	837.68	582.88	2138.43	773.16	56.41	30	1.01	155.2	223.4	300	300	5	Minimum Steel
8359	-736.39	-564.41	250.52	346	2010	1445	589.33	99999	589.33	2307.51	795.07	147.06	30	1.01	155.2	223.4	300	300	5	Minimum Steel
8716	-833.22	-844.58	287.41	346	2411.56	1445	570.95	99999	570.95	2308.61	776.69	262.27	30.5535	1.28	122.8	176.8	200	200	5	
9073	-927.27	-1142.16	320.56	346	5305.02	1445	616.45	99999	616.45	2308.57	822.19	310.82	31.0604	1.55	101.6	146.2	200	200	5	
9430	-1024.1	-1490.48	353.35	346.2	8578.87	1445	655.47	99999	655.47	2309.66	861.32	368.63	31.6858	1.88	83.6	120.3	200	200	5	
9750	-1110.89	-1832.07	381.67	361.6	11479.5	1445	0	0	0	2400.85	0	0	0	0	0	0	0	0	0	

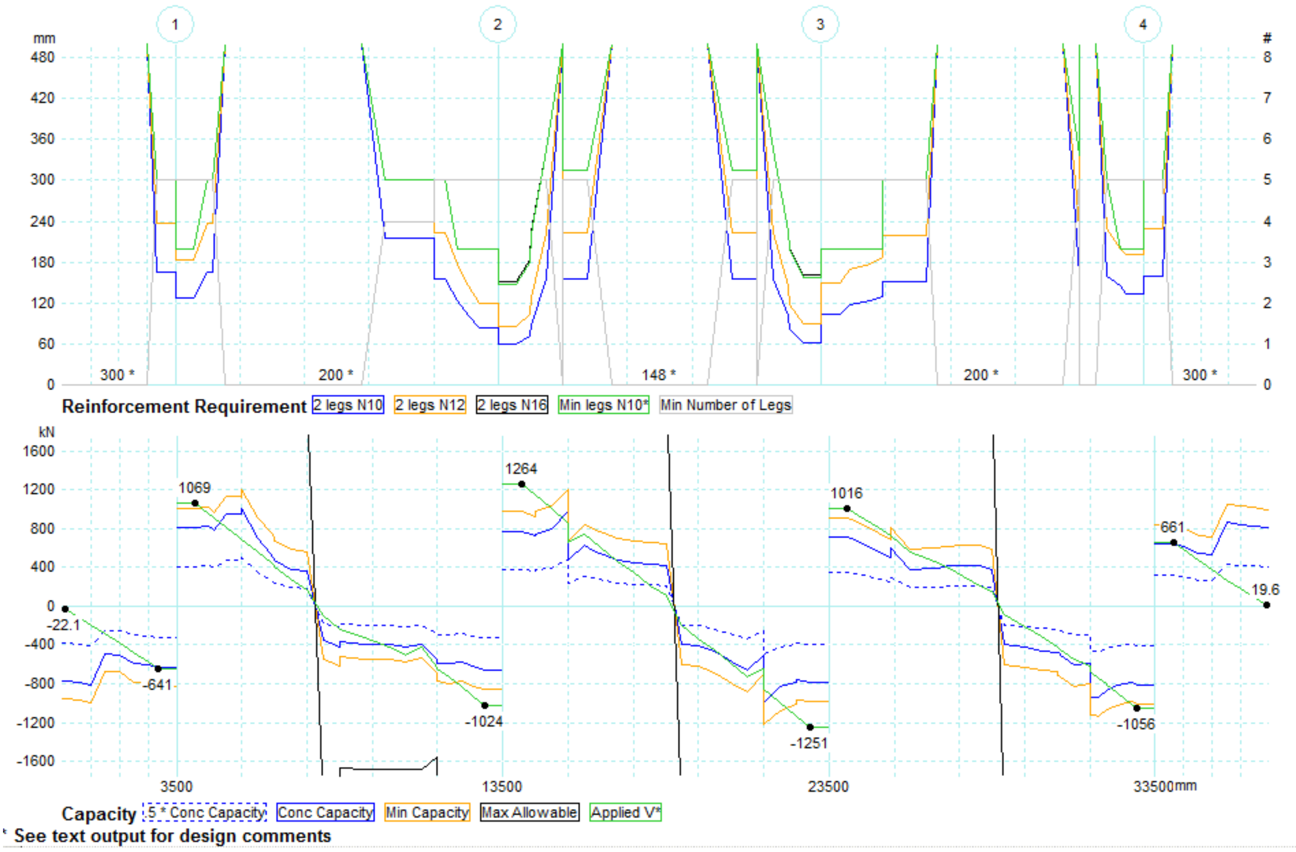
Graphical View

The graphical output consists of 3 separate diagrams for

1. Shear Reinforcement Requirements - Longitudinal Spacing of shear ties required at each design location for the 3 different shear reinforcement combinations, consisting of a bar size and number of legs, defined in the input and the combination using minimum number of legs for this cross-section. The values shown for each span are the minimum spacing in each span for the combination using minimum number of legs (* in the legend).
2. Shear Capacity
    1. .5 * Concrete Capacity - .5 * Phi Vu - For beams, the minimum shear above which shear reinforcement is required for shear strength (some codes, such as BS8110 and Eurocode, require shear reinforcement for all sections except for beams of secondary importance such as lintels).
    2. Concrete Capacity - Phi V_u - the controlling concrete shear capacity at each point
    3. Min Capacity - Shear Capacity with minimum area of shear reinforcement defined in the relevant design code.
    4. Max Allowable - maximum shear allowed to be applied at a cross-section. This is normally limited by web crushing and is defined in the relevant design code.
        - BS 8110- 85 cl.3.4.5.2 && cl.4.3.8.2
        - SABS 0100- 1:1992 cl.4.3.4.1.1 && cl.5.3.4.1
        - CP 65 - 2000 cl.4.3.4.1.1 && cl.5.3.4.1
        - Eurocode 2 cl.6.2.3(3)
        - ACI 318- 99 cl.11.5.6.8
        - AS 3600 cl.8.2.6

If the applied shear is greater than this value RAPT will report a shear failure at this point in the design comments.

5. Applied V* - the applied shear force at each point. The shear force from the case which has controlled the shear design at this design point.



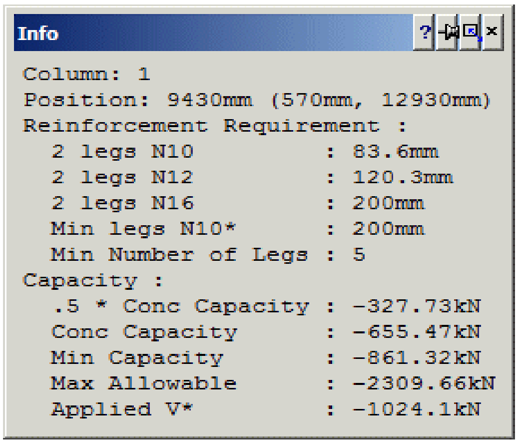
Design Comments

Design comments will be made in cases where RAPT cannot complete the design because insufficient concrete width is available or where the shear design fails where the applied shear is greater than the maximum allowed shear at a cross-section.

Information Dialog

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar, or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The information dialog reports exactly the same information as is shown on the graphical view as explained above.



7.3.9 Shear Design - Punching

RAPT checks punching shear at the punching shear critical section as defined in the relevant design code at all support locations where a column has been defined. If no support is defined or the support width is greater than 80% of the panel width RAPT will assume the support is a wall and will not check for punching shear.

RAPT will also check for punching shear at a critical section around a Drop Panel or Drop Cap at a column if one has been defined.

When checking punching shear, RAPT uses the shears and moments from the Load Combinations for Ultimate Flexure and Ultimate Shear. Thus RAPT will check four shear force / moment combinations during design. These are

1. Maximum shear on section with co-existing moment
2. Reversal shear on section with co-existing moment.
3. Maximum moment on section with co-existing shear
4. Reversal moment on section with co-existing shear

All four checks are always carried out, even if reversal cases are of the same sign as the maximum case.

The results reported will be those for the controlling case from the following in order of significance

1. the case requiring the largest area of shear reinforcement of the four cases.
2. If more than one case requires the same area of reinforcement or no cases require shear reinforcement, the case with the highest column axial load will be reported..

In all run types, the forces and moments used by RAPT to check punching shear, are based on the panel width. e.g. If a one way slab is run with an effective width of 1000mm, the user should input the panel width as the distance between transverse columns. This will ensure that the punching shear calculations are based on the total loaded area (defined by panel width) and not just on the load attributed to the effective width.

The ties specified by RAPT for Punching Shear are set in the 7.2.6.1 Reinforcement General Input Screen.

Text View

A detailed report applicable to each code (ie the results for each code type are presented in a form appropriate to the code chosen) is available in text format. RAPT specifies all the information used in the process of checking the punching shear. The reports contain notation, the results and a comment defining each term.

The following results are included in this table for each cross-section at which beam shear design has been carried out in each span (Each code uses slightly notation as shown below):-

1. Bh - longest / shortest dimension of effective loaded area
2. a - width of torsion strip
3. a_t - average depth of the torsion strip
4. u - shear perimeter length
    1. critical shear perimeter at d/2 from face of column (AS3600, ACI318, IS456/IS1343)
    2. critical shear perimeter at 1.5d from face of column (BS8110, CP65, SABS, Eurocode 2, CP2004)
5. d - average effective depth both sides of column
6. concrete shear strength
    1. f_{cv} - concrete shear strength (AS3600, BS8110, CP65, SABS, Eurocode 2, CP2004)
    2. phi f_c - concrete shear strength (ACI318, IS456/IS1343)
7. P/A - total effective prestress force / area of full panel
8. $A_{sw/s\ min}$ - minimum torsional reinforcement (based on steel yield stress as input in or code as specified.)
9. V* - applied ultimate punching shear force on the column
10. M_v* -
    1. ultimate bending moment transferred from the slab to column (AS3600, BS8110, CP65, SABS, Eurocode 2, CP2004)
    2. bending moment transferred by shear at CL of shear perimeter (ACI318, IS456/IS1343)
11. V_{uo} - ultimate shear capacity when there is no moment transfer (AS3600)
12. V*eff - design effective shear force allowing for moment transfer (BS8110, CP65, SABS, Eurocode 2, CP2004)
13. Ultimate Punching Shear Capacity
    1. V_u - design ultimate punching shear strength (AS3600)
    2. V_u - design ultimate punching shear strength without reinforcement (BS8110, CP65, SABS, CP2004)
    3. V_{Rd1} - design ultimate punching shear strength without reinforcement (Eurocode 2)
14. V_{umin} - ultimate shear capacity when provided with minimum ties (AS3600)
15. Maximum shear allowed on shear perimeter
    1. V_{umax} - ultimate shear strength limited by web crushing (AS3600)
    2. V_{umax} ultimate shear strength limited by web crushing (BS8110, CP65, SABS, CP2004)
    3. V_{Rd2} ultimate shear strength limited by web crushing (Eurocode 2)
    4. V_{umax} maximum allowed ultimate shear on the column (ACI318, IS456/IS1343)
16. side beam - defines as yes if a transverse beam has been specified in input screen and if this transverse beam is deeper than the cross section. (AS3600)
17. transfer - Yes = Moment transferred to column (AS3600)
18. $A_{sv/s}$ reqd - The area of punching shear reinforcement required (AS3600, BS8110, CP65, SABS, Eurocode 2, CP2004)
19. J_c - moment of inertia of shear perimeter (ACI318, IS456/IS1343)
20. y_c - centroid of shear perimeter from the centre of column (ACI318, IS456/IS1343)
21. v_u L - maximum applied shear stress at the left side of the column (ACI318, IS456/IS1343)
22. v_u R - maximum applied shear stress at the right side of the column(ACI318, IS456/IS1343)
23. Result - Column capacity is either OK or FAILED

Punching

Column Head Critical Section

Column No.	Bh	a	at	u	d	fcv	P/A	Asw/s min	V*	Mv*	phi Vuo	phi Vu	phi Vumin	phi Vumax	side beam	Moment Transfer	Asw/s reqd	result
#	#.#	mm	mm	mm	mm	MPa	MPa	mm2/mm	kN	kNm	kN	kN	kN	kN	A	A	mm2/mm	A
1	1	852	400	3408	352	1.92	1.32	0.31	1144.9	269.02	1947.61	1460.23	1506.29	3096.27	No	Yes	0	O K
2	1	852	400	3408	352	1.92	1.28	0.31	1500.59	27.87	1938.57	1888.75	2229.11	4582.07	No	Yes	0	O K
3	1	852	400	3408	352	1.92	1.09	0.31	1509.21	63.28	1890.48	1784.21	2065.3	4245.36	No	Yes	0	O K
4	1	852	400	3408	352	1.92	1.32	0.31	1163.66	263.28	1947.6	1473.91	1526.42	3137.66	No	Yes	0	O K

For all codes RAPT will calculate the amount of reinforcement required to satisfy the shear loads if possible. If the concrete can not withstand the shear forces, RAPT will display the result "Punching Shear FAILS". It is up to the designer to calculate any extra reinforcement in this case.

Helpful Hints If a design fails in punching shear, the following amendments may overcome the Punching Shear problem. (i) Provide or increase the drop panel depths or column capitals. (ii) Design a shear head.

7.3.10 Deflections

RAPT calculates 4 deflections cases for each design strip. These deflections are based on the 7.2.4.2 Load Combinations defined in the input. The deflections are calculated at different times and take into account the different material properties at those times, the effects of cracking of the concrete and tension stiffening, and the effects of reinforcement in the section on cracking, shrinkage and creep. All deflections are measured from a straight line between the supports at each end of a span. The effects of support settlement are not included in these figures.

For two-way slabs incorporating column strips and middle strips, RAPT will report expected deflections for both the column strip and the middle strip. The middle strip deflection is a net deflection measured from the deflected line of the supporting column strips.

The four deflections calculated are:-

1. Transfer - Effects of creep and shrinkage are not considered. Cracking and tension stiffening are considered as appropriate.
    1. For prestressed members, this deflection is based on the 7.2.4.2.5 Transfer Load Combination and is calculated at the time of final transfer as defined in 7.2.5.5 Stressing Sequence. The concrete properties are calculated at this time.
    2. For Reinforced Concrete members, this deflections is based on the 7.2.4.2.5 Transfer Load Combination and is calculated at the Time of Loading as defined in 7.2.7.1 Design Data. The concrete properties are calculated from the concrete strength at this time.
2. Short Term - This deflection is based on the 7.2.4.2.4 Short Term - Deflection load combination. The concrete properties are the 28 day properties. Effects of creep and shrinkage are not considered. Cracking and tension stiffening are considered as appropriate.
3. Incremental - A deflected shape is calculated for the 7.2.4.2.4 Initial Load - Deflection load combination. Effects of creep and shrinkage are not considered for this deflected shape. Cracking and tension stiffening are considered as appropriate. The Incremental Deflection is the increment of deflection from this Initial Load deflected shape to the Total Long Term Deflected shape.
   The Incremental Deflection is often used by designers to determine the deflection affecting attached partitions. It should only be used if the member being analysed is un-propped at the time of construction of the attached element. If the member is propped at this time, the Total Long Term deflection should be used.
   In some cases, the Initial Deflected Shape may be negative (upwards). This would happen in prestressed members where more than the dead load has been balanced by the prestress. In these cases, the Incremental Deflection will be greater than the Total Long Term Deflection as it is being measured from the upwardly Initial Deflected shape.
   Some designers pre-camber concrete members to help with deflections. This does not reduce incremental deflections, only the visible total deflection, and cannot be used to reduce deflections experienced by attached elements. The Incremental Deflection is measured from the deflected shape before attachment (pre-cambered upwards) to the Total Long Term deflected shape and is therefore unaffected by the pre-camber.
4. Total Long Term - The total long term deflection is based on the 7.2.4.2.4 Short Term - Deflection and 7.2.4.2.4 Permanent - Deflection load combinations as described in Theory Section T.7.7. Effects of creep and shrinkage are considered in calculating the permanent part of this deflection. Cracking and tension stiffening are considered as appropriate.

Deflections are reported as negative downwards.

Checking Deflections

In checking deflections, the designer should consider the deflections of the individual strips over their length as well as the diagonal panel deflection over the diagonal support spacing. The diagonal panel deflection can be calculated as the sum of the column strip deflection in one direction and the middle strip deflection in the other direction. If the 2 values of this that can be calculated for the 2 column strip/middle strip combinations vary significantly, the relative load distribution could be varied until they are similar or an average of the 2 figures could be used.

For one way slabs supported by beams, the diagonal panel deflection should also be checked as the sum of the beam deflection and the transverse slab deflection over the diagonal support spacing.

A deflection of L/250 in both directions will result in a deflection on the diagonal of approximately L/177. Thus, to achieve a diagonal deflection of L/250 the deflections in each direction will have to be reduced by about 20% or the deflection in one direction reduced by about 40%.

Most design standards provide deflection limits for different types of buildings. RAPT allows the designer to 7.2.7.1 define limits in the input, defaulted from the relevant 5.6 default design code figures. RAPT will give warnings if these limits are violated in any span.

Reducing Deflections

There are several strategies that can be used to reduce deflections. The logical strategies are

1. Define the true reinforcement pattern in the input rather than relying on the designed reinforcement at each design location. The 7.2.6.4 designed reinforcement layout can be added using the  toolbar button.
2. Add extra tension reinforcement. This will only be effective if the section is significantly cracked under the short term loading. In this case, a significant increase in effective inertia will result from the increased depth of neutral axis resulting in a decrease in deflections.

   However, if the section is uncracked or only just cracked, the increase in effective inertia due to the added

reinforcement will normally be very small and will be more than offset by the increase in shrinkage warping due to the shrinkage restraint of the reinforcement resulting in increased long term deflections.

3. Add compression face reinforcement. If the section is reasonably heavily reinforced in tension and the compression face reinforcement is in an area of high compression stress, it can be effective in reducing deflections. There are several limitations to the use of compression face reinforcement to limit deflections.
    1. If the section is not deep enough, the compression face reinforcement will actually be in the tension zone so will do nothing to reduce creep deflection.
    2. Even with a deeper section, at positive moment areas the depth of the neutral axis is normally very small due to the wide compression flange and the compression face reinforcement may still be in the tension zone or only be in an area of low compression stress and have little effect especially as it is often the second reinforcement layer.
    3. If the section is only lightly stressed, the effect of the compression face reinforcement on deflections will be low.
4. Increase the level of prestress or modify the prestress tendon profiles to provide more uplift in the critical spans. Modifying the profile in a span beside the critical span can have an effect on the deflection in the critical span. Raising the anchorage above the centroid at the end of a cantilever can reduce the deflections in a cantilever.
5. Add short tendons in the critical spans.
6. Increase the depth of the concrete or the compressive strength of the concrete.
7. Introduce shrinkage reducing additives to the concrete to reduce the shrinkage characteristics of the concrete.

Text View

The following results are included in this table for each deflection case for each span:-

1. Deflection - maximum deflection in that span for that deflection case. The absolute maximum is reported.
2. Span/Deflection - The effective span length divided by the Deflection for the maximum deflection in the span.
3. Location - the location of the maximum deflection in the span measured from the left end of the span.

	Span	#.#	0	1	2	3	4
Transfer	Deflection	mm	0.5	-1.4	0.2	-3.2	1.1
	Span/Defection	#.#	6555.11	7055.97	47842.9	3118.57	3088.63
	Locat	mm	1	4999	8500	3499	3499
Short Term	Deflection	mm	1	-13.6	-11.5	-16.4	-1.1
	Span/Defection	#.#	3371.01	737.581	869.131	611.047	3301.82
	Locat	mm	2299	4999	4999	4999	3499
Incremental	Deflection	mm	2.8	-32.2	-30.6	-36.6	2.5
	Span/Defection	#.#	1264.14	310.307	327.005	273.258	1396.68
	Locat	mm	1750	4999	4999	4999	875
Total Long Term	Deflection	mm	3	-38.1	-35.7	-44.6	2.8
	Span/Defection	#.#	1166.44	262.507	280.25	224.314	1245.06
	Locat	mm	2299	4999	4999	4999	875

Span 1

Design Comments:-

- Incremental Deflection greater than user defined limit = -32.2mm
- Total Deflection greater than user defined limit = -38.1mm

Span 2

Design Comments:-

- Incremental Deflection greater than user defined limit = -30.6mm
- Total Deflection greater than user defined limit = -35.7mm

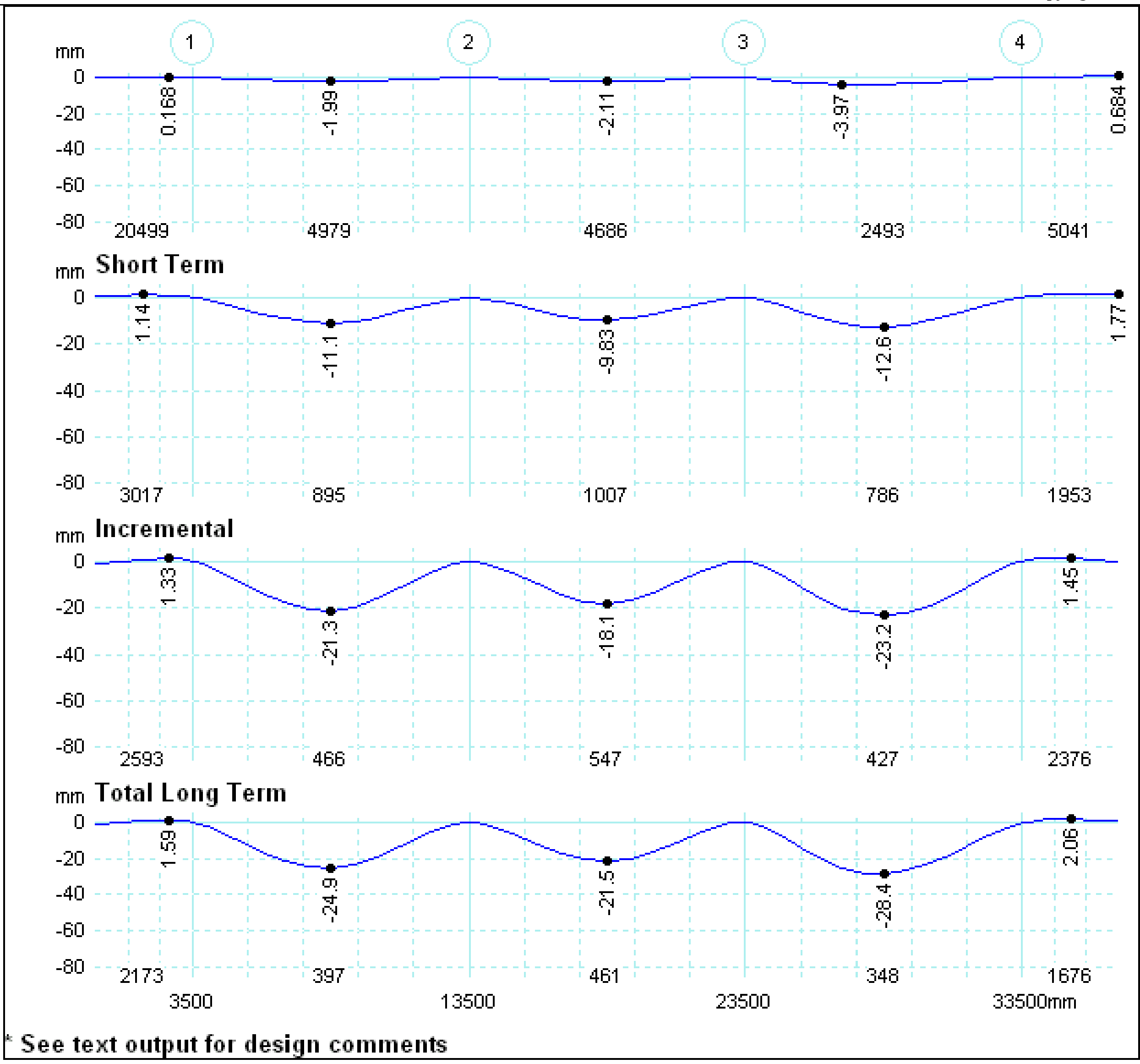
Span 3

Design Comments:-

- Total Span/Deflection ratio less than user defined limit = 224.314
- Incremental Deflection greater than user defined limit = -36.6mm
- Total Deflection greater than user defined limit = -44.6mm

Graphical View

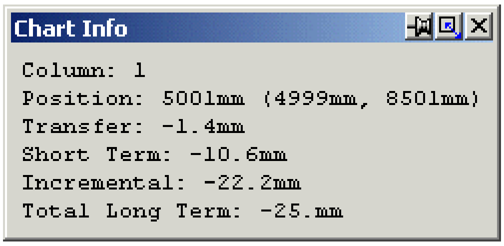
The graphical output consists of separate diagrams for each deflection case. For each deflection case, the deflection is available at each design location. For each deflection case, the maximum deflection in each span is shown on the plot at the location at which it occurs and the corresponding span/deflection ratio is recorded at the bottom centre of the span of each plot.



Design Comments
Design comments will be made in cases where the deflections are outside the deflection or span/deflection ratio limits set in the 7.2.7.1 Design Data in the input. See Text View above for examples.

Information Dialogs
To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The information dialog reports the deflection for each deflection case at the location selected.



7.3.11 Detailed Reinforcement

The Detailed Reinforcement Report summarises all the reinforcement designed by RAPT in a member. This does not include prestressing strands or reinforcement defined by the user in input. Nor does it include extra reinforcement required to provide crack control for restraint effects. This information is a summary of Flexural Design and Beam Shear Outputs.

No Graphical output is available are available for this output option.

Application of Crack Control Rules to Results

The bar spacings/maximum bar sizes determined by RAPT are based on any reinforcement defined by the designer plus any reinforcement added by RAPT for strength etc at each design cross-section all of which will affect the strains in the concrete and reinforcement and, thus, the crack widths in the member. The final results for a member may vary away from the critical strength cross-sections, as the design amount of reinforcement at these locations will be less than the final reinforcement supplied by the designer when the reinforcement is detailed. In some cases the spacing requirements may initially be worse at a point away from the critical strength cross-section but will not be critical once the final detailed reinforcing bar pattern is analysed. By iterating the design and defining the final reinforcing pattern in the 7.2.6.4 User Defined Reinforcement input the designer will get a better picture of the final pattern and also the effects of this reinforcement pattern can have a significant effect on the shear and deflection results.

Determining Bar Sizes

RAPT initially decides which is the support reinforcement surface and which is the span reinforcement surface based on the shears on the entire member. The support reinforcement surface is then analysed in terms of reinforcement areas to divide the surface into "reinforcement zones" over which RAPT will use a single bar size. These zones will be localised over each column and will normally extend to the point of zero area of reinforcement on each side of each column. If the reinforcement is continuous over a span then the mid-point of the span is selected as the end of the zone. For the span surface each span is treated as a separate zone.

In the detailing of reinforcement, RAPT applies the crack control results as described in Design rules which limit either Bar Size or Bar Spacing and Design rules which limit only Bar Spacing at the cross-section in a moment zone which has the critical amount of reinforcement. This provides a bar size for that zone which is used throughout the zone. The results for each design point in the zone are then presented in terms of a number of bars of this size.

Design rules which limit either bar size or bar spacing

For design rules which limit either bar size or bar spacing (eg AS3600-2009 and Eurocode 2) - The procedure used by RAPT to estimate the suggested bar size/spacing for a moment zone uses the following logic steps,

1. RAPT first checks spacings based on the 7.2.6.3 Preferred Bar Size at this location unless from the code rules unless the "maximum bar size" from the calculations at this point is less than this bar size. If this bar size produces a bar spacing within the limits defined in 7.2.6.1 Reinforcement->General input and also within the overall Design Code maximum and minimum spacings (not the maximum spacing based on the stress level in the bar), RAPT will accept this bar size and continue on to determine the bar numbers and spacings at any point within this moment zone using this bar size and ignore steps 2 and 3, otherwise
2. RAPT then checks spacings based on the Maximum Bar Spacing and RAPT will step through the range of bar sizes as defined in 7.2.6.3 Reinforcement Design Zone from the Maximum Bar Size to the Minimum bar Size. If a trial bar size is larger than the Crack Control Maximum Bar Size then the Maximum Bar Spacing is used as the upper limit to the reinforcement spacing, otherwise the user defined limit in 7.2.6.1 Reinforcement->General input is used as the upper limit. When RAPT is attempting to determine a bar size that will fit within the maximum and minimum bar spacing range in a design zone, it will determine a sub-range of bar sizes which fit within the spacing limits. If the 7.2.6.3 Preferred Bar Size defined in 7.2.6.3 Reinforcement Design Zone is within this sub-range of bar sizes, RAPT will adopt this bar size as the solution, otherwise RAPT will determine the bar size within this sub-range of bar sizes which closest in diameter to the 7.2.6.3 Preferred Bar Size as the proposed solution for that design zone, Otherwise
3. If no solution is found within the Range of Bar Sizes defined in 7.2.6.3 Reinforcement Design Zone and if the user has nominated to do so in User Preferences, RAPT then will then check spacings based on the Maximum Bar Spacing either from the Largest Bar Size available for the bar type to the Maximum Bar Size or from the Minimum bar Size to the Smallest Bar Size available in the bar type to try to find a solution. If a trial bar size is larger than the Crack Control Maximum Bar Size then the Maximum Bar Spacing is used as the upper limit to the reinforcement spacing, otherwise the user defined limit in 7.2.6.1 Reinforcement->General input is used as the upper limit. When RAPT is attempting to determine a bar size that will fit within the bar spacing range in a design zone, it will determine a sub-range of bar sizes which fit within the spacing limits. RAPT will determine the bar size within this sub-range of bar sizes which provides the smallest total area of reinforcement using a whole number of bars and present this as the proposed solution for that design zone

Remember, RAPT is only giving one solution out of a range of possible combinations of bar size and bar spacing for a moment zone. Other solutions may work and may be adopted for a design. For some designs, it may be necessary to modify the maximum and minimum reinforcement spacings defined in 7.2.6.1 Reinforcement->General input to achieve an acceptable result. The default values for these spacing and bar size ranges are defined for normal design situations and will not necessarily suit all designs. RAPT will give warnings if it cannot find a solution to the reinforcement detailing for a moment zone. These warnings will try to indicate the types of changes that may be necessary to the input to achieve a result. Normally, in a case where a solution could not be found and warnings are given, the minimum bar size nominated in the range to be considered will be adopted.

It is up to the designer to ensure that the final reinforcement arrangement fits within the design parameters and that any modifications made to the reinforcement requirements calculated by RAPT will result in the same design solution. Adding extra reinforcement without checking its effect on the design can be as bad as adding less reinforcement than RAPT has calculated in terms of ductility and deflection. For this reason, it is recommended that the final reinforcement

pattern be entered into RAPT as 7.2.6.3 User Defined Reinforcement in input and the effects of this pattern of reinforcement be checked for compliance with the relevant code rules.

Design rules which limit only Bar spacing
For design rules which limit only Bar spacing (eg BS8110, ACI-318, CP65, SABS0100, CP2004, IS456/IS1343)- if the "maximum bar spacing" is less than the user defined maximum bar spacing in 7.2.6.1 Reinforcement->General input then it is used as the upper limit to the reinforcement spacing, otherwise the user defined limit in input is used as the upper limit. Required bar spacing for all bar sizes in the range defined in 7.2.6.3 Reinforcement Design Zone are investigated. When RAPT is attempting to determine a bar size that will fit within the maximum and minimum bar spacing range in a design zone, it will determine a sub-range of bar sizes which fit within the spacing limits. If the 7.2.6.3 Preferred Bar Size defined in 7.2.6.3 Reinforcement Design Zone is within this sub-range of bar sizes, RAPT will adopt this bar size as the solution, otherwise RAPT will determine the bar size within this sub-range of bar sizes which provides the smallest total area of reinforcement using a whole number of bars and present this as the proposed solution for that design zone

If no solution is found within the bar size range defined in 7.2.6.3 Reinforcement Design Zone, RAPT will then investigate bar sizes either from the Largest Bar Size available for the bar type to the Maximum Bar Size or from the Minimum bar Size to the Smallest Bar Size available in the bar type to try to find a solution.

Beam Flanges
Remember that, in the reinforcement layer at the flange face of a beam, the reinforcement can be spread across the effective flange width to reduce congestion and improve crack control in the flange outstand. In all cases, it is necessary to provide crack control in the flange so at least minimum crack control reinforcement should be placed in the flange outstand and be detailed with full tension laps. RAPT allows the designer to specify that a portion of the reinforcement is to be placed in the flange outstand in the 7.2.6.3 Reinforcement Design Zone input. This will result in less congestion in the beam shear cage in these areas but will reduce the shear capacity of the member in these areas and could result in more shear reinforcement being required. RAPT will still nominate the total number of bars needed at all sections in these zones but the nominated spacing will require that reinforcement be placed outside the beam shear cage as requested.

Text View
The following results are included in this table for each span for each design strip:-

The results can be broken into three sections. For each design location, RAPT gives

1. Top Reinforcement. This part contains six columns of data
    1. Max Size:- for design standards which define a maximum bar size for crack control, this is the calculated maximum bar size at this cross-section based on the reinforcement defined by the user and that added by RAPT at this cross-section. Final detailing may provide more reinforcement at this cross-section which could increase this bar size.
    2. Max Space:- for design standards which define a maximum bar spacing for crack control, this is the calculated maximum bar spacing at this cross-section based on the reinforcement defined by the user and that added by RAPT at this cross-section. Final detailing may provide more reinforcement at this cross-section which could increase this bar spacing.
    3. Area:- the area of reinforcement calculated by RAPT to satisfy ultimate strength, serviceability and minimum reinforcement requirements at this cross-section.
    4. Depth:- depth to reinforcement from top of concrete member
    5. Section Width:- The width of the concrete section at the depth of the design reinforcement on which the bar size requirement is calculated at each cross section.
    6. the number, size, type and spacing of reinforcement required to satisfy the amount of reinforcement needed. Refer to the discussion above in Determining Bar Sizes for details on how RAPT determines the bar size nominated.
2. Bottom Reinforcement. This part also contains six columns, as for the top reinforcement.
3. Shear Reinforcement. This part has 5 columns. The shear reinforcement requirements are shown in 2 forms in the first 4 columns of data
    1. Area:- As the area of reinforcement / spacing (for a specified reinforcement yield strength). This value is the larger of the calculated Asv/s and Asv/s based on the limiting longitudinal spacing requirements as set in the relevant design code.
    2. Legs of Bar Size:- The tie spacings for three different bar types and size and numbers of legs, as defined in the 7.2.6.1 Reinforcement General input screen .
    3. Comments:- Comments on the shear design at this cross-section.
        1. No shear steel.
        2. Web width too small. Design web width is less than 20mm and RAPT cannot design the section for shear. This could be because there is a horizontal penetration full width of the beam, in which case the designer must design the shear transfer across the penetration based on the shear data supplied here, or because the sum of the prestress duct diameters is too large, possibly because ducts that are arranged vertically have been counted individually, see 7.2.5 Prestress Input Control Grid.

Span 1

	Top Reinforcement						Bottom Reinforcement						Shear Reinforcement				
														Spacing of Sets			
Locat	Max Size	Max Space	Area	Depth	Section Width	Rebar Reqd	Max Size	Max Space	Area	Depth	Section Width	Rebar Reqd	Area	2 legs N10	2 legs N12	2 legs N16	Shear Comments
mm	mm	mm	mm2	mm	mm	A	mm	mm	mm2	mm	mm	A	mm2/mm	mm	mm	mm	A
175	0	200	3917.41	48	2900	7 N28 @ 230.5	0	0	0	352	1500	No Steel Reqd	0	0	0	0	
570	0	0	4190.35	48	2900	7 N28 @ 230.5	0	0	0	352	1500	No Steel Reqd	1.25	125.5	180.6	240	Minimum Steel
1535	0	0	300.15	48	2900	1 N28 @ 999	0	0	0	352	1500	No Steel Reqd	0	0	0	0	No shear steel
2500	0	0	0	48	2900	No Steel Reqd	0	0	625.59	352	1500	2 N20 @ 999	0	0	0	0	No shear steel
3333	0	0	0	48	2900	No Steel Reqd	0	0	1534.99	352	1500	5 N20 @ 347.8	0	0	0	0	No shear steel
4166	0	0	0	48	2900	No Steel Reqd	0	200	1893.49	352	1500	7 N20 @ 231.8	0	0	0	0	No shear steel
5000	0	0	0	48	2900	No Steel Reqd	0	200	1920.51	352	1500	7 N20 @ 231.8	0	0	0	0	No shear steel
5833	0	0	0	48	2900	No Steel Reqd	0	200	1749.7	352	1500	6 N20 @ 278.2	0	0	0	0	No shear steel
6666	0	0	0	48	2900	No Steel Reqd	0	0	1162.26	352	1500	4 N20 @ 463.7	0	0	0	0	No shear steel
7500	0	0	551.22	48	2900	1 N28 @ 999	0	0	0	352	1500	No Steel Reqd	1	156.8	225.8	300	Minimum Steel
8465	0	0	1578.17	48	2900	3 N28 @ 691.5	0	0	0	352	1500	No Steel Reqd	0	0	0	0	No shear steel
9430	0	200	6044.46	48	2900	10 N28 @ 153.7	0	0	2076.13	352	1500	7 N20 @ 231.8	1.25	125.5	180.6	240	Minimum Steel
9825	0	200	6764.73	48	2900	11 N28 @ 138.3	0	0	2136.56	352	1500	7 N20 @ 231.8	0	0	0	0	

7.3.12 Reinforcement Layout

RAPT will determine a reinforcing bar pattern to match the reinforcement requirements for a run and the relative code development and detailing rules using the bar sizes and spacings determined above to suit the crack control requirements of the code. This reinforcing bar pattern can be viewed in text form in the output screens and can be automatically added to the run as 7.2.6.4 user defined reinforcement.

The reinforcing pattern includes all reinforcement calculated by RAPT as being need to satisfy code design requirements for;

1. strength and ductility
2. minimum strength
3. crack control
4. AS3600 Shrinkage and Temperature requirements for slabs

In calculating termination locations for bars, RAPT will attempt to ensure that each bar will develop adequate capacity at each design location along it's length to provide the strength required at that point. As well, RAPT will apply the relevant clauses in each code

1. AS3600 clauses 8.1.8 (except 8.1.8.6), 8.1.9 and 9.1.3.1, where relevant
2. ACI318 clause 12.10
3. Eurocode clause 2 5.4
4. BS8110, SABS 0100, CP65 clauses 3.12.9 and 3.12.10
5. CP2004:- 9.2.1
6. IS456/IS1343:- 29.3

Support reinforcing bars will normally be terminated at a point past the point of contraflexure sufficient to develop the bar where required. If staggering of bars is requested in the input and if appropriate, 50% of bars (lesser whole number if odd number of bars) will be terminated where possible in the tension zone. If reinforcement is required past this point for some reason, RAPT will either add intermediate bars (Ast at support > .3 Ast at middle) or extend the main support bars to lap at mid-span (100% development) with some bars stopping past the point of contraflexure. If more reinforcement is required at mid-span than at the supports, RAPT will lap the support bars at mid-span and add extra bars at mid-span.

Span reinforcing bars will normally be continued to the support and terminated at the centre of the support. A development percentage will be assigned at each end sufficient to satisfy development requirements along the bars. If staggering of span bars is requested in the input and if appropriate, 50% of bars (lesser whole number if odd number of bars) will be terminated where possible in the tension zone at each end of the span. If a greater area is required at an end than at mid-span, the full number of bars required at mid-span will be continued to the support and fully developed and extra bars will be added at the support.

Remember, this reinforcement pattern is determined from the design results and detailing requirements and is an attempt to provide a logical reinforcement layout for the design. It is not the only possible reinforcement layout and may result in more reinforcement than is shown in the design calculations. The design results, strength, serviceability, shear and deflections, will not reflect the actual layout of reinforcement shown here. To achieve final design results that reflect this reinforcement pattern, go back into input 7.2.6.4 user defined reinforcement and add this reinforcing pattern to the user defined reinforcement using .

Text View
The reinforcing shown in the text output includes all of the reinforcement used in and calculated in the design of the member. It includes all user defined reinforcement from the input (shown first in the table) and all reinforcement added by RAPT in the design and detailing.

The following results are included in this table for reinforcing bar:-

1. Bar Type:-
    1. Type:- Reinforcing type as selected in 7.2.6.1 Reinforcement General and 7.2.6.3 Reinforcement Zones, or 7.2.6.4 Reinforcement User Defined and defined in materials.
    2. Size:- Size of reinforcing bar or mesh used.
    3. Number of:- Number of reinforcing bars or width of reinforcing mesh.
2. Left End Of Bar:-
    1. Ref Col No:- Number of the Column from which the left end of the bar is dimensioned.
    2. Distance from Column:- Distance to left end of the bar from the left reference column.
    3. Stagger Length:- Distance from the left end of the bar to the left end of the staggered bars in the group. Zero if there is no stagger.
    4. Depth Down:- Depth to left end of bar from the top of slab datum.
    5. % Development
        1. Tension:- Percentage of full development required for the left end of this bar in tension. This could be provided with a lap, hook, cog or bar coupler or straight bar extension past this point.
        2. Compression:- Percentage of full development required for the left end of this bar in compression. This could be provided with a lap, or bar coupler or straight bar extension past this point. Hooks and cogs cannot provide compression development.
3. Right End of Bar

1. Ref Col No:- Number of the Column from which the right end of the bar is dimensioned.
2. Distance from Column:- Distance to right end of the bar from the right reference column.
3. Stagger Length:- Distance from the right end of the bar to the right end of the staggered bars in the group. Zero if there is no stagger.
4. Depth Down:- Depth to right end of bar from the top of slab datum.
5. % Development
    1. Tension:- Percentage of full development required for the right end of this bar in tension. This could be provided with a lap, hook, cog or bar coupler or straight bar extension past this point.
    2. Compression:- Percentage of full development required for the right end of this bar in compression. This could be provided with a lap, or bar coupler or straight bar extension past this point. Hooks and cogs cannot provide compression development.

Bar				Left end of bar						Right end of bar					
Layer No.	Type	Size	No. Of	Ref Col No.	Distance From Column	Stagger Length	Depth Down	% Development		Ref Col No.	Distance From Column	Stagger Length	Depth Down	% Development	
								Tension	Compression					Tension	Compression
#	A	A	#.#	#	mm	mm	mm	#	#	#	mm	mm	mm	#	#
1	N500	20	7	1	0	0	352	0	0	1	10000	0	352	100	100
2	N500	20	6	2	0	0	352	100	100	2	10000	0	352	100	100
3	N500	16	22	3	0	0	352	57	100	3	10000	0	352	0	0
4	N500	20	1	2	-795.4	0	352	0	0	2	4459.3	0	352	0	0
5	N500	20	1	3	-1519.5	0	352	0	0	3	795.4	0	352	0	0
6	N500	28	7	1	-3499	0	48	43	43	1	2679.4	0	48	0	0
7	N500	28	11	2	-2859	0	48	0	0	2	2906.7	0	48	0	0
8	N500	28	15	3	-2942.7	0	48	0	0	3	2826.4	0	48	0	0
9	N500	28	7	4	-2702.7	0	48	0	0	4	3499	0	48	43	43

Graphical View

The graphical output consists of separate diagrams for each design strip.

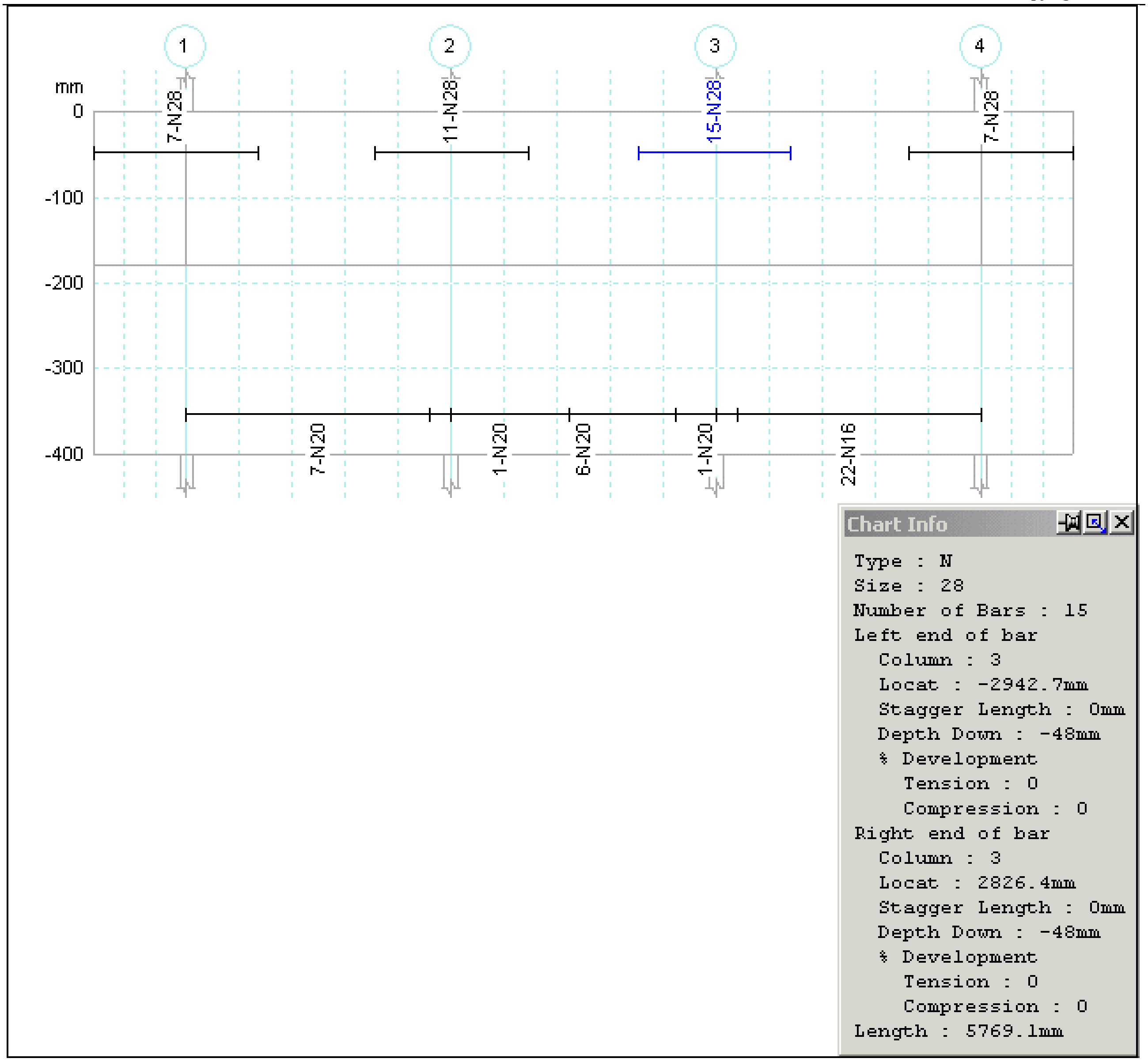
Each reinforcing bar is shown in elevation. The ends of a bar a indicated with a short vertical line. Stagger end locations are indicated with a diagonal line sloping towards the relevant bar end from the stagger end location on the bar. All reinforcing bars used in the design and designed by RAPT are shown here. The difference in representation is

1. Program Calculated Reinforcing Bar (see general discussion above):- the number of bars, type of bar and size of bar are indicated thus 4-N16, printed perpendicular to the bar at the midpoint of the length of the bar.
2. 7.2.6.4 User Defined Reinforcing Bar, the number of bars, type of bar and size of bar are indicated thus (5-N20).

Where there are 2 or more bars overlapping in elevation as shown below, the bar descriptor will be moved from the midpoint of the length of the bar if necessary to avoid overlapping of the descriptors.

To select a bar, click on it with the left mouse button. The selected bar and its attached attributes will be shown in blue. Where there are 2 or more bars overlapping in elevation as shown below, the first bar in the text list will be selected when clicked on with the mouse. If the mouse is clicked again (use sufficient delay to avoid a double-click), the next bar at this location in the text list will be shown and so on. This will loop through the overlapping bars at the clicked point continuously. Alternatively, the toolbar buttons can be used to select each reinforcing bar in turn as described in the Graphics Toolbar section below.

Double-Click on a bar will move the focus into Zoom mode equivalent to pressing  (see below) and the reinforcing bars will then be shown with a diameter drawn to scale.



Graphics Toolbar 

A special graphics toolbar is provided to assist with viewing the data. This toolbar will only be available when program focus is in the Graphics Window. The functionality of this toolbar for reinforcement data is slightly different to the general functionality. The buttons available are

 Zoom (Ctrl + Z). This button will toggle between full screen mode and bar zoom mode for the graphics in a window. In bar zoom mode, the spans in which the current bar is placed will be shown scaled to fill the whole window. The scales will still show at the left and right sides of the window. The horizontal scale will change to suit the new length of the graph being shown in the Window. In this mode, the bars are drawn to scale with their correct diameter. When not in zoom mode, the bars are drawn as a 1 pixel wide line.

 Move to next Reinforcing Bar (Ctrl + Right Arrow). In Full Screen Mode the next reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In bar Zoom mode, the spans containing the next reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour.

 Not Available.

 Not Available.

 Move to previous Reinforcing Bar (Ctrl + Left Arrow). In Full Screen Mode the previous reinforcing bar in the text bar list will be shown in a blue colour (see graphical view above). In Bar Zoom mode, the spans containing the previous reinforcing bar in the text bar list will move to the centre of the Window and the selected reinforcing bar will be shown in a blue colour.

 Zoom to user defined rectangle. This button is only available in Full Screen Mode. Click the button and then use the mouse to select a rectangle within the graphics into which you want to zoom. To do this, click and hold the left mouse button at the top left or bottom right corner of a rectangle and move the mouse to create a rectangle that encloses the area you wish to zoom into. The relative scale of the zoomed area will be the same as that for the Full Screen Mode so, if you make the selected rectangle shape exactly the same relative proportions as the Window, the rectangular shape you have selected will fill the entire window. Otherwise, the relative scale will still be maintained so a larger shape will be shown to ensure that the full selected rectangle is in the view: depending on the relative shapes of the rectangle and the Window, more width or depth of the graph will be included than requested.

Then user can then move around the graph using the Scroll Bars or the movement keys.  or  buttons will still move the selection to the next and previous bars but will not move the view location.

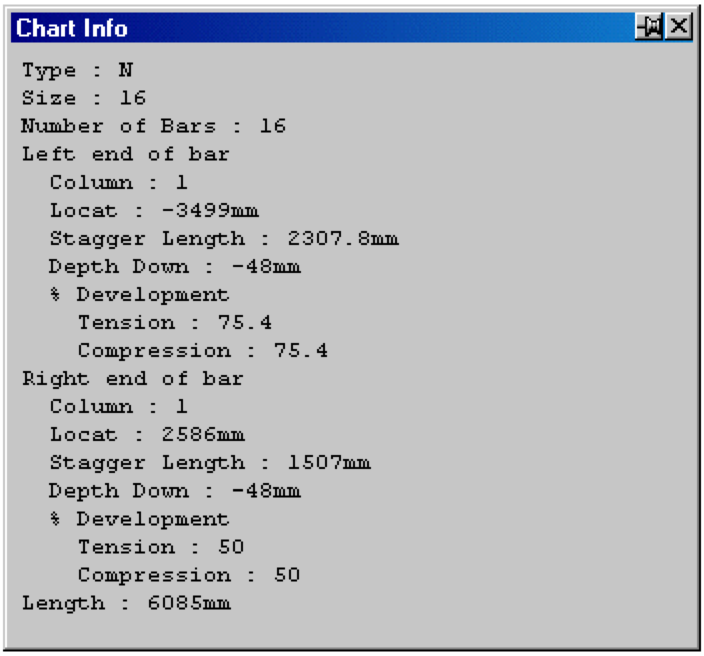
Clicking this  button again or on the buttons will return the Window to Full Screen Mode.

Clicking  will change the mode to Bar Zoom Mode with the reinforcing bar selected in the Select Zoom mode.

Information Dialogs

To view the information used to plot the curves on these graphics views, open the Information Dialog from the graphics toolbar , or press Ctrl + I, and then left click on design location desired. While the dialog is in view, click on any other point or move with the toolbar buttons or quick keys to view its information and the dialog data will be updated automatically.

The information dialog reports the information defining the selected reinforcing bar. The information provided is exactly the same as that described above in Text View.



8 Column Definition and Design

This section is not available yet.

9 Cross-section Definition and Design

This section is not available yet.

10 Tendon Profile Definition and Design

This section is not available yet.

T Theory

The theory section of this manual has been written to give the designer background on the design methods used in RAPT. Reasons for assumptions, theory used and helpful design hints can be found throughout this section. We have endeavoured to set this section out in a systematic way working through the design procedure as a designer would by hand.

RAPT allows designers the versatility of designing to 8 codes.

1. Australia AS3600
2. United Kingdom BS8110
3. USA ACI318
4. Europe EUROCODE 2
5. South Africa SABS 0100
6. Singapore CP 65
7. Hong Kong CP2004
8. India IS456/IS1343

In some instances the various codes approach the design of members in the same way and in other areas they use totally different design approaches. Thus RAPT at times uses common routines for each code and at other times uses separate design techniques. In writing this section of the manual, we have given code references for the user to examine where the codes differ. In sections where no code references appear the user can assume that the approach being discussed is used by all codes.

In many instances the codes give the designer the option to use a simplified approach to analysis or design and an option to design by "first principals". Wherever possible RAPT uses the second option, which is often code independent, along with the various factors required by each code, using “first principals” to gain a more accurate picture of the structure under review. This section aims to explain these methods used by RAPT.

T.1 Preliminary Sizing

Before starting a RAPT run the designer should have some idea of the member sizes and a feel for the structure. Below are listed some suggested guide-lines for designers, in order to establish a reasonable structural system first up.

T.1.1 Initial Selection of Section Depth

Prestressed Systems

Below are the suggested limits proposed by T.Y.Lin and N.H.Burns for span to total depth, L/D, ratios for prestressed systems. These ratios are for light office loading only.

Application	Simple Span		Continuous	
	roof	floor	roof	Floor
one-way solid slabs	48	44	52	48
two-way solid slabs	44	40	48	44
(supported on columns only)				
two-way waffle slabs				
(900 mm waffles)	36	32	40	36
(3600 mm waffles)	32	28	36	32
one-way slabs with small cores	46	42	50	46
one-way slabs with large cores	44	40	48	44
single tees (spaced 6000 mm c/c)	32	28	36	32
double tees and single tees				
(spaced side by side)				

For prestressed slabs, in order to fit anchorages within the depth of the slab, a minimum depth of 140 mm is recommended. This may vary between different prestressing companies and reference should be made to the companies in different countries. Span to depth ratios (L/D) ACI (ref 14) applicable for prestressed slabs continuous over 2 or more spans

1. roof slabs 48-52
2. floor slabs 42-48

Span to depth ratios (L/D) for prestressed slabs ACI (ref 13)

1. roof slabs 45-48 (max 52)
2. floor slabs 40-45 (max 48)

For banded slab systems the clear span is taken as being the distance from edge to edge of the band tapers except in end spans where it is measured at the discontinuous end to the column centre-line. It is not uncommon to deepen the slab in end spans. The greatest economy will be achieved if the end span length is reduced to about 80% of the length of the internal spans and all slab depths made equal.

The band beam is commonly proportioned to L/D = 25 - 30 (for office construction but may drop to 18 for storage conditions) where L is the largest of the span lengths in either direction. Usually the depth of the band is about 1.5 - 2.0 the depth of the slab.

The web width of the band, measured at the slab soffit, is taken as about 0.25 - 0.33 times the transverse span width. This dimension is rationalised to suit standard plywood sheets. i.e. 1200, 1500, 1800, 2100 or 2400 mm. For long span bands (in either direction) consideration should be given to introducing a taper on the sides of the band beam. The width of the taper should ideally suit standard plywood sheets (600 or 900 mm).

The flange width of bands may be calculated from the smaller of

1. the effective flange width formula for beams even though the band is designed more as a wide slab than a beam.
2. the column strip width as if it were a two-way slab.

Prestressed beams may initially be proportioned by

L/D = 20 for b approx. = D/3
L/D = 30 for b approx. = 3D (where b = beam width)

In contrast to these figures the authors would recommend the following for light office loadings.

Flat plate slabs	L/D = 40	
Flat slabs with drop panels	L/D = 45-50 drop = 1.5 - 2 x D	All values for continuous spans. Reduce by 10% for simple support
Band Beams	L/D = 25 - 30 L_T/D = 25 - 30 width = 0.20 - 0.3 x L_T	Reduce by 5% for end spans

where
D = Depth of the member
L = The span length
L_T = The transverse column spacing width

Note:

1. For retail loading, these figures should be reduced by 5 - 10%
2. For storage loading, these figures should be reduced by 10 - 35% depending on the level of load.

In all cases, these figures are only a starting point. We recommend that the best way to "size" a slab for design is to start with these figures and do a few runs to optimise them for a particular design.

Reinforced Systems

Earlier versions of the Australian standard (CA2-1963) gave the following table for the initial selection of depth for reinforced beams and slabs. There has been criticism that these are not always conservative however they give a quick initial estimate of the depth required.

Beams	L/D
Simply supported beams	20
Continuous beams	25
Cantilever beams	10
Slabs	**L/D**
Slabs spanning in one direction, simply supported	30
Slabs spanning in one direction, continuous	35
Slabs spanning in two directions, simply supported	35
Slabs spanning in two directions, continuous	40
Cantilever slabs	12

Note: for a slab spanning in two directions, L is that of the shorter direction.

For two-way reinforced slabs using AS1480 (ref 33) (assuming continuous spans where the span length £ 9000 mm, Ast/bd = 0.0075 and the steel stress = 210 MPa under dead plus live load) the following Span to Depth ratios (L/D) are obtained

1. two way reinforced flat slabs 25 - 30
2. if provided with drop panels 30 - 37

There has been criticism that these are not conservative in all cases. Rangan (ref 34) proposed the following method, which appears in AS3600 (clause 9.3.4) in the following form

where
$F_{d.ef}$ = $(1 + k_{cs})(G + SW) + (y_s + k_{cs}\, y_l)Q$ for total deflection
= $k_{cs}(G + SW) + (y_s + k_{cs}\, y_l)Q$ for incremental deflection

Beams

$$L_{ef} / d = \frac{[k_1 b_{ef} (d / L_{ef}) E_c]^{0.33}}{k_2 F_{d.ef}}$$

where

k_1 = I_{ef} / bd3 which may be taken as

= 0.045 for rectangular sections

= $0.045\ (0.7 + 0.3\ b_w / b_{ef})^3$ for T and L sections

k_2 = a deflection constant, taken as -

1. for simply supported beams, 5/384 or;
2. for continuous beams, where in adjacent spans the ratio of the longer span to the shorter span does not exceed 1.2 and where no end span is longer than an interior span.
   - 1/185 in an end span;
   - 1/384 in interior spans

Slabs

$$L_{ef} / d = \frac{k_3\, k_4\ [b\ (d\ /\ L_{ef})\ E_c\ ]^{0.33}}{F_{d.ef}}$$

where

k_3 = 1.0 for a one-way slab

= 0.95 for a two-way slab without drop panels

= 1.05 for a two-way flat slab with drop panels which extend at least L/6 in each direction on each side of the support centreline and have an overall depth not less than 1.3 D where D is the slab thickness beyond the drops

= 1.0 for a slab supported by beams on four sides

k4 = the deflection constant which may be taken as -

a) for simply supported slabs 1.6; or

b) for continuous slabs, where in adjoining spans the ratio of the longer span to the shorter span does not exceed 1.2 and where no end span is longer than an interior span -

2.1 in an end span; or

2.6 in interior spans

c) for a slab supported by beams on four sides from the table below

Edge condition	Deflection Constant k_4			
	Ratio of long to short side (L_y / L_x)			
	1.00	1.25	1.50	2.00
1. 4 edges continuous	4.00	3.40	3.10	2.75
2. 1 short edge discontinuous	3.75	3.25	3.00	2.70
3. 1 long edge discontinuous	3.75	2.95	2.65	2.30
4. 2 short edges discontinuous	3.55	3.15	2.90	2.65
5. 2 long edges discontinuous	3.55	2.75	2.25	1.80
6. 2 adjacent edges discontinuous	3.25	2.75	2.50	2.20
7. 3 edges discontinuous (1 long edge contin)	3.00	2.55	2.40	2.15
8. 3 edges discontinuous (1 short edge contin)	3.00	2.35	2.10	1.75
9. 4 edges discontinuous	2.50	2.10	1.90	1.70

T.1.2 Determination of Load to be Balanced

The selection of the load to balance is an important factor in the economics of prestressed systems. One of the major advantages of prestressing is to reduce the long-term deflection of the concrete. However selection of too high a load to balance may incur excessive prestressing costs reducing the economy of the prestressed solution. A combination of a lower level of "balanced load" and the addition of normal reinforcement at peak moment regions will prove to be a more economical solution in most applications (ie a partial prestress design).

If the basis of design is that the structure be flat under long-term loading then the load to balance should be the self-weight plus a proportion of the long-term applied loading. Over balancing, i.e. balancing self-weight plus a large superimposed loading, may result in problems at transfer. In this instance the full prestress is present (accentuated by the as yet not realised long-term losses) and the live load is yet to be applied. This problem may be overcome if stage stressing is adopted depending on the loading type. Generally the loads to balance in the following table will usually prove satisfactory.

Occupancy of Building	Partition and Other Super Imposed Dead Load (kPa)	Design Live Load (kPa)	Load to Balance (kPa)
Car Parks	------------	3.0	(0.7 - 0.9) sw
Shopping Centres	0.0 - 2.0	5.0	(0.9 - 1.1) sw
Residential	2.0 - 4.0	2.0	sw + 50% of partition load
Office Buildings	0.5 - 1.0	3.0	(0.8 - 1.0) sw
Storage (check transfer carefully)	------------	2.4 kPa/m height	sw + 20 - 30% LL

In all cases the level of "load to balance" required is influenced by the L/D chosen. The lower the L/D selected for a particular design, the lower the "load to balance required". The "low end" figures nominated in this table would normally apply for the L/D ratios nominated by the authors in the earlier section on selection of section depth. Higher L/D ratios would require the "high end" figures nominated in this table.

T.1.3 Selection of Level of Prestress

The level of average prestress (the total effective force divided by the net concrete area) serves as a useful check on the economics of the design and the serviceability of the structure. Below are some guidelines on different stress levels. From ref 13

Below 1.4 MPa	Amount of compression is generally inadequate to resist cracking, and is much too low for waterproofing. The tendons will probably not have enough ultimate capacity alone and will have to be supplemented with reinforcement at both positive and negative regions. Economical for lightly loaded slabs.
1.4 - 3.5 MPa	Generally accepted "proper" range for post-tensioned slabs. Check for very stiff vertical elements and very long distances between expansion joints or stiff elements. For typical conditions, slabs stressed in this range will perform well with respect to cracking and shortening. Use 2.0 MPa minimum for watertight slabs. Will usually mean that no extra reinforcement is required for ultimate flexure at positive regions (but perhaps in end spans) but may be required for negative flexure except in heavily loaded members.
Above 3.5 MPa	Slabs stressed in this range will undergo excessive shortening due to axial creep and elastic shortening due to axial creep and elastic shortening. Check all connections to vertical elements, limit pour size to as small as practicable, pour walls after slab is stressed (if possible). In short, take every available precaution to avoid shortening restraint problems.

T.2 Frame Properties

RAPT calculates the stiffness of both the slab / beam and columns.

T.2.1 Column Calculations

The columns are modelled as either

- Equivalent Column T.2.1.1
- Net Column Stiffness T.2.1.2
- Enhanced Column Stiffness T.2.1.3

T.2.1.1 Equivalent Column Calculations

Note (ref 3) that "all of the equations regarding the equivalent column stiffness concept are general in nature and are equally applicable to slab systems supported on precast concrete beams or structural steel members as long as there is frame action designed into the beam-column system (a rigid joint) and composite action between slab and beam is also provided for."

The basis for the calculation of an equivalent column is that studies have shown that the positive moment in a slab may increase under pattern loading, even if rigid columns are used because of the flexibility of the slab away from the column. However, if a two-dimensional frame analysis is applied to a structure with rigid columns, pattern loads will have little effect. To account for this difference in behaviour between slab structures and frames, the equivalent column torsional member is introduced. The transverse slab-beam can rotate even though the column is infinitely stiff, thus permitting moment distribution between adjacent panels. It can be seen that the stiffness of the columns is affected by both the flexural stiffness of the columns and the torsional stiffness of the slab and beams framing into the columns.

The equivalent column stiffness is therefore less then the sum of the stiffness of the columns above and below the slab thereby distributing moment back into the slab-beams.

For hand checks of RAPT, the tables for equivalent frame properties proposed by Simmons and Misic (1971 ref 17) will generally prove sufficient for simple geometric layouts.

The equivalent column calculations follow the procedure detailed in references 2 and 3 and are based on gross section properties throughout.

Torsional Member
At every column / slab joint determine the dimensions of the attached torsional member (see Figures T.2.1 and T.2.2). This member consists of

1. in its most basic form a beam, within the depth of the slab, with a width equal to the transverse column width, c1, and a depth equal to the depth of the slab.
2. when a column capital is present then the width is that of the capital.
3. when a drop panel is present then the depth is that of the drop panel and the width of the column or capital.
4. when a transverse beam frames into the joint then the transverse torsional member consists of the beam projection above and below the slab plus flanges extending on either side of the beam a distance equal to the projection of the beam above or below the slab but not greater than four times the thickness of the slab.

Figure T.2.1 shows details of typical attached torsional members.

Torsional Constant
Calculate the cross-sectional torsion constant, C, used to approximate the St.Venant torsion constant as given by equation T.2.1.

$$C = \sum\left(1 - 0.63\frac{x}{y}\right)\frac{x^3 y}{3}$$

Where x and y are the smaller and larger dimensions of the attached torsional member respectively. The cross-section is divided into separate rectangular parts and the C of each component part is then added to give the total torsional constant.

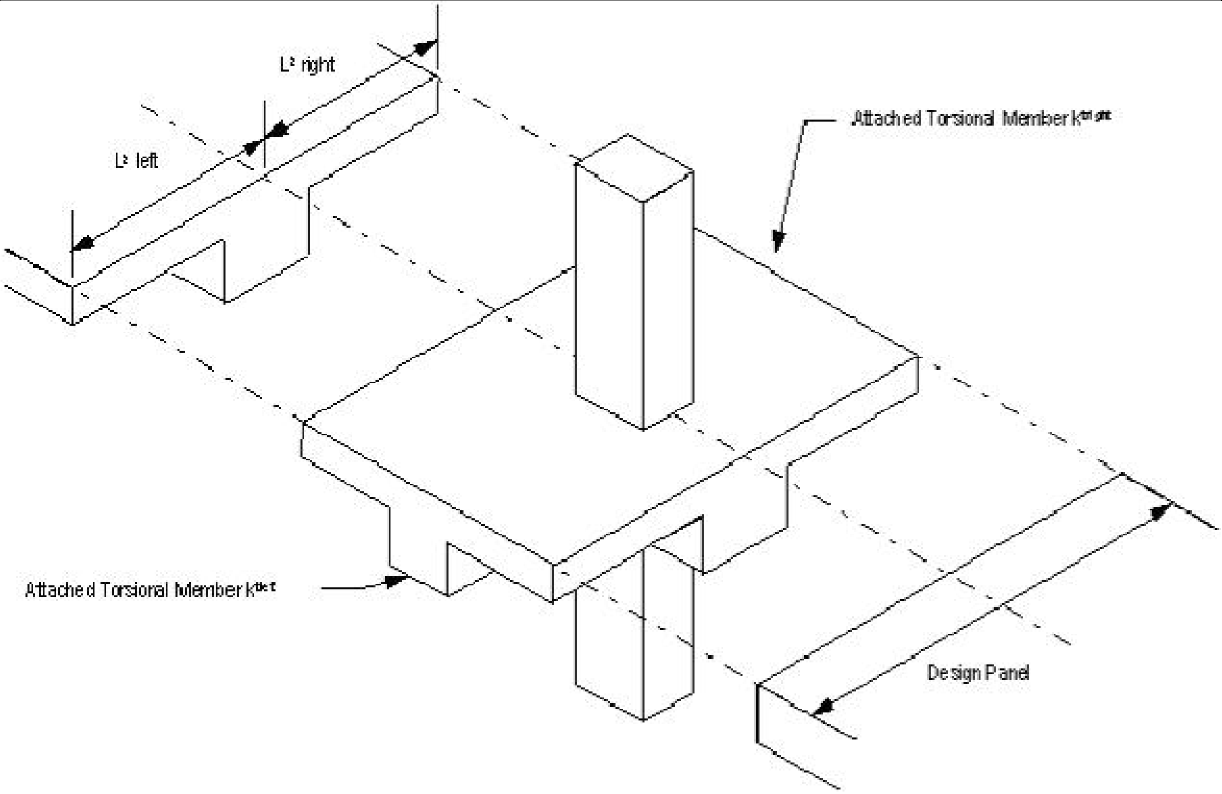


Figure T.2.1

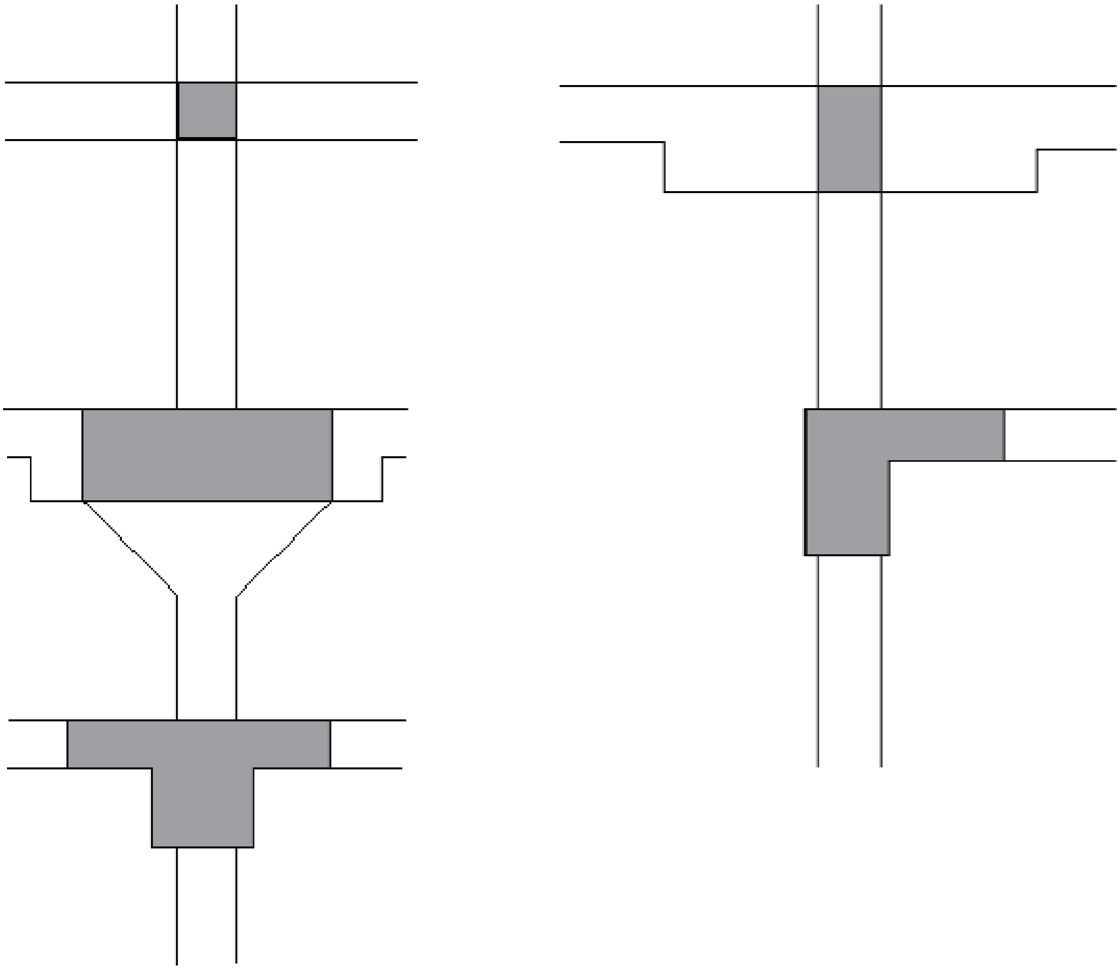


Figure T.2.2

RAPT handles non-typical cases in the following manner.

1. When the drop panel does not extend one-sixth of the transverse span on either side of the column then the depth of the drop (forming part of the transverse beam), for the calculation of C, is reduced linearly down from the full drop depth to the depth of the slab.
2. When the transverse width of the drop is less than or equal to two times the drop depth and the drop width is less than lt/3 then the drop is assumed to be acting solely as a beam haunch and does not contribute to the torsional stiffness.
3. The average parallel beam and drop panel depths each side of the column centre-line are used for the torsional stiffness calculation.

Transverse Torsional Stiffness

Calculate the transverse torsional member's stiffness on one side of the column, kt, given by equation.

$$k_t = \frac{9E_sC}{l_2\left(1-c_2/l_2\right)^3}$$

Es = Young's modulus of the slab-beam concrete

Note:

1. L2 is the average transverse centre-line to centre-line dimension on either side of the column. More strictly, for an internal panel .ie. having transverse members on either side of the column, there will be a k2left and a k2right and equation T.2.2 should be evaluated twice and summed. For an internal panel RAPT evaluates equation T.2.2 once based on the average dimension and multiplies by 2. Note that for an edge panel L2 is equal to the distance to the next adjacent transverse column centreline.

   Therefore for an internal panel:

   $$\sum k_t = 2 \times k_t$$

2. the transverse support width dimension, c2, is generally the column dimension however where a column capital exists it is taken as the width of the capital at the soffit of the slab.

Column Stiffness Above and Below

Calculate the stiffness of the columns above and below the joint (see Fig T.2.3). Note that the floors above and below the floor being designed are assumed as being identical as the design level.

The column is assumed to have infinite stiffness over the deepest of the (ref 3)

- slab depth
- average parallel beam depth
- average drop panel depth

Note: The depth of the transverse beams is ignored.

The inertia of the column capital is taken as that at mid-depth of the capital projection below the deepest member. The projection of the capital is assumed to be uniform around the perimeter of the column. The same capital is assumed to be present at the slab / column connection above and below the slab being analysed.

The stiffness is calculated using moment-area principles. Note that the horizontal centre-line of the equivalent frame is modelled as the centroid of the slab / beam at midspan, denoted as the datum.

The procedure is summarised as follows (ref 20):

P is the total area of the 1/I diagram for the column

$= \sum x / I_x$

X is the distance to the centroid of the 1/I diagram from the datum. This is obtained by taking first mo column about the datum and dividing by P.

R is the second moment of area of the 1/I diagram about X

$= \sum \frac{x^3}{3I_x}$ areas about X

calculate the column stiffness, k_c

$$k_c = \frac{E_c\left(X^2 P + R\right)}{PR}$$

If the column were to have the remote end pin-ended then the stiffness is taken as being three-quarters that given by the above equation.

The same procedure is followed for the column above and below the slab thereby giving the column stiffness kca and kcb respectively.

As a check on the above the approximation proposed by Cross and Morgan (ref 19) may be used as described below.

$$k_c = \frac{E_c I_{col}\left[1 + 3\left(L / L'\right)^3\right]}{L'}$$

where
I_{col} = the column inertia remote from the column ends
E_c = Youngs modulus of the column
L = floor to floor height
L' = clear column height

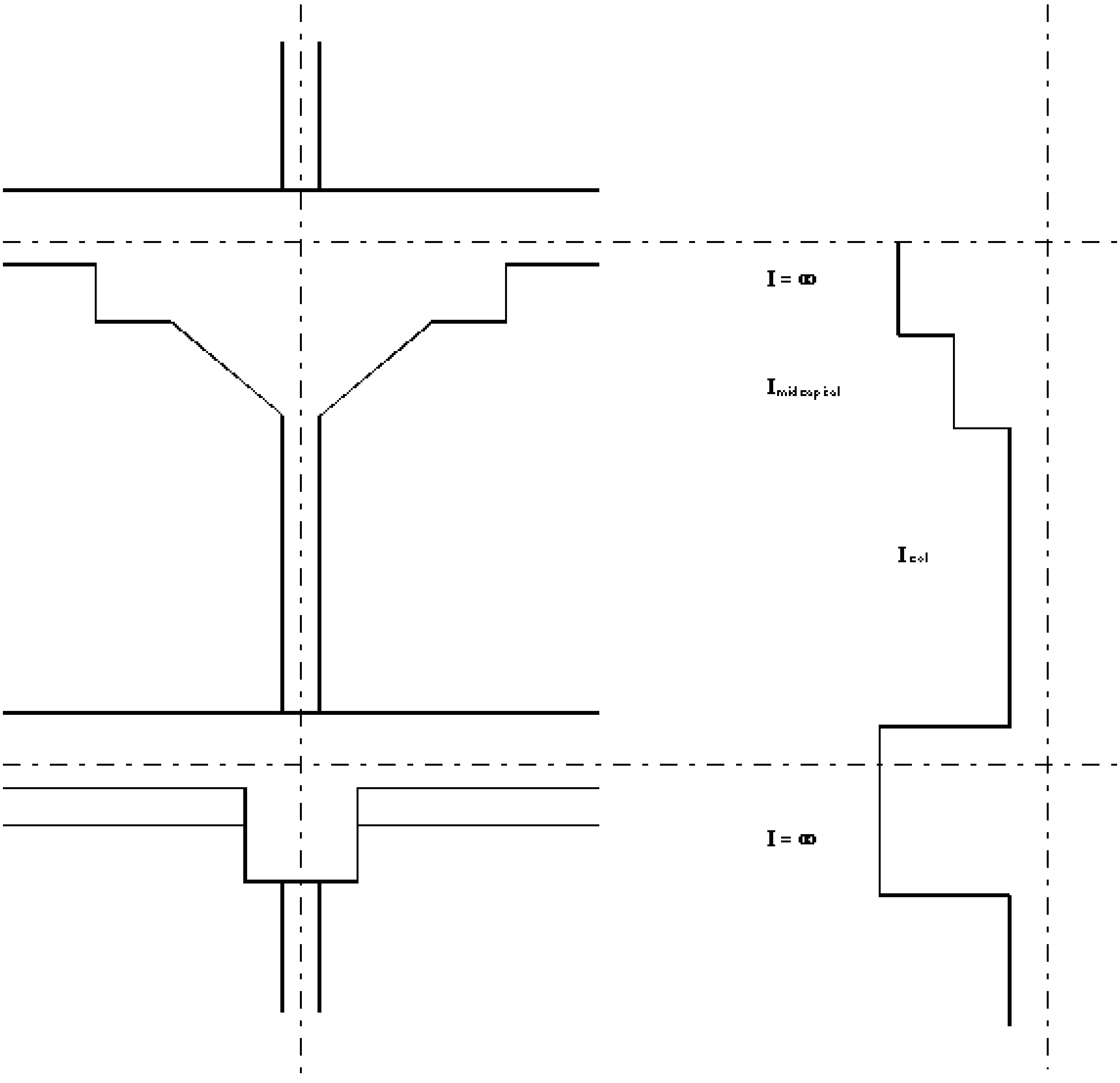


Figure T.2.3

Alternatively Rice and Hoffman (ref 20) proposed

$$k_c = \frac{4E_c I_{col}}{L-2h}$$

where
h = the depth of the increased stiffness region at the top of the column.

Equivalent Column Stiffness
Combine the column stiffness (k_{ca} + k_{cb} = Sk_c) and the stiffness of the transverse torsional member (k_t) to give the "equivalent column" stiffness using equation T.2.5.

$$\frac{1}{k_{ec}} = \frac{1}{\sum k_t} + \frac{1}{\sum k_c} \qquad \text{(T.2.5)}$$

Note that at the edge of the frame the transverse member must be designed to resist, in torsion, a portion of the moment attracted to the equivalent edge column. RAPT, in its output, distributes the total moment in the equivalent column to the columns above and below.

If the design is that of a typical strip of a one-way slab then the inertias of the frame are based on the width of the strip being considered. This then requires that the column stiffness should be reduced to reflect the ratio of the width of the strip being considered to the full panel width , Lt, at each column. RAPT uses the ratio of the one-way width to the L2 dimension at each column line. For a one way beam the full equivalent column stiffness's are used.

The column stiffness above and below can be represented in the following simplified form

k_c = x E_c I_c / L

where x is a factor defining the stiffness. For a column of uniform cross-section fixed at both ends x would have a value of 4. Define x_a and x_b as the stiffness factors for the columns above and below the joint respectively.

Express the stiffness of the columns as a ratio, a_a and a_b, of the total column stiffness

a_a = k_{ca} / (k_{ca} + k_{cb})
a_b = k_{cb} / (k_{ca} + k_{cb})

If the assumption is made that a_a and a_b will also be the ratio of the equivalent column stiffness above and below, with respect to the total equivalent column stiffness, then the equivalent column stiffness's above and below are

k_{eca} = a_a k_{ec}
k_{ecb} = a_b k_{ec}

This implies that the contribution of the transverse member is shared to the two columns in the ratio of their own stiffness's.

The I_{ec} printed to the screen is based on an equivalent prismatic member using a stiffness factor of 4 ie.

$$I_{ec} = \frac{k_{ec} L}{4E_c}$$

As the length of the equivalent column is undefined we can achieve this result from

$$I_{ec} = \frac{\alpha_a k_{ec} H_a}{4E_c} + \frac{\alpha_b k_{ec} H_b}{4E_c}$$

$$I_{ec} = \frac{k_{eca} H_a}{4E_c} + \frac{k_{ecb} H_b}{4E_c}$$

For checking of the program using a plane frame computer program, the moment of inertia to be entered for the equivalent column will be given by the above equation. If columns are to be modelled above and below then the actual lengths, Ha and Hb, should be used and the akec factor applicable.

The cross-sectional area for the frame analysis is taken as the gross sectional area of the main column section. Columns above are analysed as having vertical roller supports. If the frame is braced then a vertical roller support is placed at the two extreme ends of the frame.

T.2.1.2 Net Column Stiffness

This option uses the actual column stiffness calculated as

1. For Rectangular Columns $I = \frac{bd^3}{12}$
2. For Circular Columns $I = \frac{\pi \times d^4}{64}$

These calculations are repeated for each column above and below and then applied to the frame analysis.

T.2.1.3 Enhanced Column Stiffness

For Enhanced Column stiffness RAPT allows for the infinite stiffness of the column over the slab/beam depth. Thus RAPT uses the approach as already explained under "COLUMN STIFFNESS ABOVE AND BELOW" (See Equation T.2.4) to calculate kc. This stiffness is then converted to an inertia by

$$I_{col} = \frac{k_c \times L}{4 \times E_c}$$

This calculation is repeated for each column above and below and these inertias are applied to the frame analysis.

T.2.2 Slab / Beam Calculations

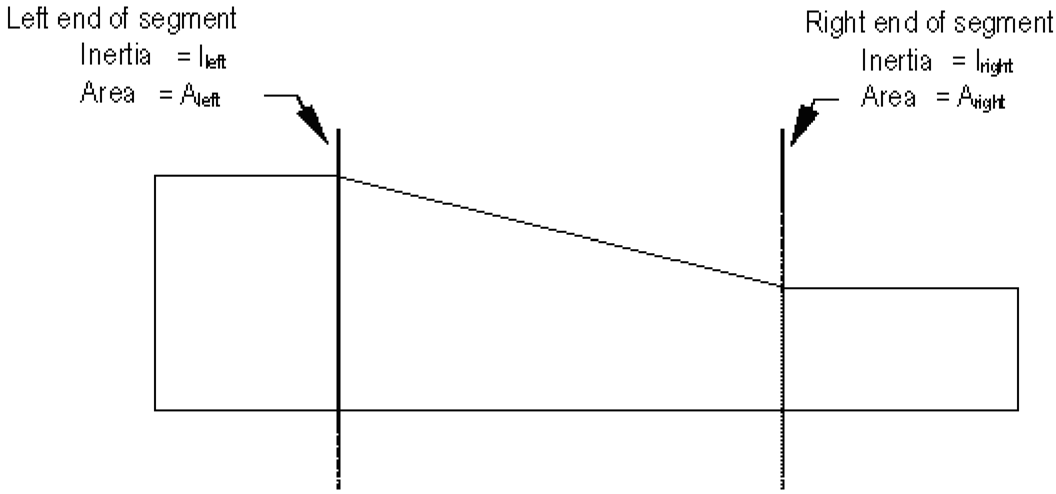
RAPT divides each span into segments. A new segment is created for each change in inertia along the span. In the case of a taper (constantly changing inertia), RAPT will define a series of segments to approximate the inertia along the taper. RAPT will also define an inertia over the length of the columns.

Inertias
Inertias are calculated based on first principals.

Inertia within Columns
The inertia within a column region is obtained from equation below (ref 4).

$$I = \frac{I(\text{just outside of column})}{\left(1 - c_2 / L_2\right)^2}$$

Tapers
For tapered elements, RAPT divides the taper length into smaller even segments. (equal segments of approx length = span/15). The inertia and area given for the taper length are calculated as follows



$$I_{taper} = \frac{I_{left} + I_{right}}{2}$$

$$A_{taper} = \frac{A_{left} + A_{right}}{2}$$

Frame Application
The calculated inertias for each segment within a span are applied to the frame analysis. The frame is modelled as a straight line (no vertical members beside the columns) with the increased inertias at the steps modelling the steps in the frame.

T.3 Blank

T.4 Lateral Distribution Factors

RAPT uses the procedure described in BS8110, AS1480 and ACI318 to determine the percentage of the total panel moments resisted by the column and middle strips. AS3600 implies the same procedure as AS1480 and ACI318, but falls short of providing concise percentages.

For specific information see

- AS3600 and AS1480/81 see AS1480 Clause 21.4.4
- ACI318 Clause 13.6
- BS8110 Table 3.20
- SABS 0100 Table 17

For post-tensioned structures the default distribution is performed solely on the factor nominated in Input F1 [Input].

As set out in AS1480 and ACI318, a percentage of the panel moment at each column centre-line and at the maximum span moment location are obtained from the following tables.

where

$$\alpha_1 = \frac{I_b L_t}{I_s L_1} \tag{T.4.1}$$

$$\beta_t = \frac{C}{2 I_s} \tag{T.4.2}$$

= 0 if no transverse beam (ref 24)

I_b is the inertia of the beam defined by the projection of the beam above and below the slab plus flanges extending on either side of the beam a distance equal to the projection of the beam above or below the slab but not greater than four times the thickness of the slab.
= 0 if no parallel beam

I_s is the inertia of a width of slab equal to the full width between panel centre-lines, Lt, excluding that portion of the beam stem extending above and below the slab.

C is the transverse torsional constant (as per the column calculations)

L_1 is the length of a span in the direction in which moments are being determined, measured centre to centre of supports.

For cantilevers the percentage along the full span is taken as the value at the exterior support. For internal columns the a1 and bt values are based on the average slab and beam inertias either side of the column.

Column Strip: Negative Moment Factors for an Exterior Support

Interpolations between the values shown shall be linear

		Negative moment factor for an exterior support for L_t /L_l equal to		
a_1 L_t /L_1	b_t	0.50	1.00	2.00
0	0	1.00	1.00	1.00
	³ 2.5	0.75	0.75	0.75
³ 1	0	1.00	1.00	1.00
	³ 2.5	0.90	0.75	0.45

Column Strip: Negative Moment Factors for an Interior Support

Interpolations between the values shown shall be linear

$a_1 L_t / L_1$	Negative moment factor for L_t / L_1 equal to		
	0.50	1.00	2.00
0	0.75	0.75	0.75
³1	0.90	0.75	0.45

Column Strip: Positive Moment Factors for an all spans

Interpolations between the values shown shall be linear

$a_1 L_t / L_1$	Positive Moment Factor for L_t / L_1 equal to		
	0.50	1.00	2.00
0	0.60	0.60	0.60
³1	0.90	0.75	0.45

BS 8110-85 & SABS 0100: Distribution Factors

Negative Moment	Positive Moment
Column / Middle	Column / Middle
75 / 25	55 / 45

EUROCODE2 : Distribution Factors

Negative Moment	Positive Moment
Column / Middle	Column / Middle
75 / 25	55 / 45

The remaining proportion of the total panel moment is taken by the two half middle strips comprising the design strip. RAPT selects the maximum moment point within each span and applies the above positive column strip factor. The percentage between this point and the two support centre-lines is taken as varying parabolically A plot of a two-way reinforced slab bending moment diagram for a middle strip demonstrates this effect.

When there is a beam spanning in the design direction and within the column strip it is designed to resist a percentage of the load taken by the column strip. The percentage reflects the relative stiffness of the beam to that of the column strip. The factor is the minimum of the following

0.85 or

$$0.85\ a_1\ L_t/L \qquad \text{(T.4.3)}$$

Normal design office practice and the method adopted by RAPT is as follows.

1. Take all the column strip load to the parallel beam.
2. Assume that the middle strip moment per metre applies to the middle strip and also to residual part of the column strip outside of the flange of the beam.

Where a beam spans in the design direction, references to the column strip within RAPT refer to the beam.

The different codes specify different reinforcement and tendon distribution in two way slabs. ie for AS3600 -

> Reinforcement and tendon distribution in two-way flat slabs (Clause 9.1.2 AS3600) requires that at least 25 percent of the total of the design negative moment in the column strip and adjacent half middle-strips shall be resisted by reinforcement or tendons or both, located in a cross-section of slab centred on the column and of a width equal to twice the overall depth of the slab or drop panel plus the width of the column.
>
> Note: These concentrations are not checked or detailed by RAPT.

T.4.1 Column Strip Widths

The width of the column strip is free to vary in each span. Regardless of the width selected the frame will still stand up provided the steel in each strip is detailed to support the load assumed to be taken to the strip. However non-judicious selection of column strip widths may result in unsatisfactory performance in service eg. large deflection. The method of calculating column strip widths used by RAPT is based on that of Section T.1.2 of AS3600. However the standard when read literally would result in sharp steps in the column strip width at the column lines if span lengths vary in the design direction. The authors believe this is not only impractical but also not the intention of the standard and have adopted the method described below.

The width of the panel is defined in Input F2 [Spans], at either end of each span. Therefore we may have a different panel width on either side of each column. In normal design situations these two values will be equal however in some cases the panel being analysed may step sharply (in plan) at the column line which will result in different panel widths on either side of the column and hence different column strip widths.

The calculations at each column line proceed as follows. (for all code types)

- for an internal column strip
  The average span (in the design direction) on either side of the column is calculated, then this value is divided by two.
  This value is then compared with the panel widths on either side of the column divided by two and the smaller values are adopted for the column strip widths.
- for an external column strip
  The average span (in the design direction) on either side of the column is calculated, then this value is divided by four.
  This value is then compared with the panel widths on either side of the column (towards the interior of the building) divided by two plus the overhanging region (from the column centre-line) and the smaller values are adopted for the column strip widths.

Note:

1. For cantilevers the column strip width is taken as uniform extending from the exterior column. It may be argued that the shorter the cantilever the smaller should be the column strip width. The codes do not make allowance for this.
2. the width of the column strip may vary linearly in plan in each span.

T.5 Critical Sections

T.5.1 Flexure

In a linear-elastic analysis, column supports are modelled as isolated support lines of zero width. The support is in fact of finite width and hence the column reaction is not a single point reaction but rather a trapezoidal distributed load over the width of the column. The peak bending moment obtained from the analysis at the column centre-line is therefore an over-estimate of the maximum bending moment as the distributed reaction forces will have caused a reduction in this value.

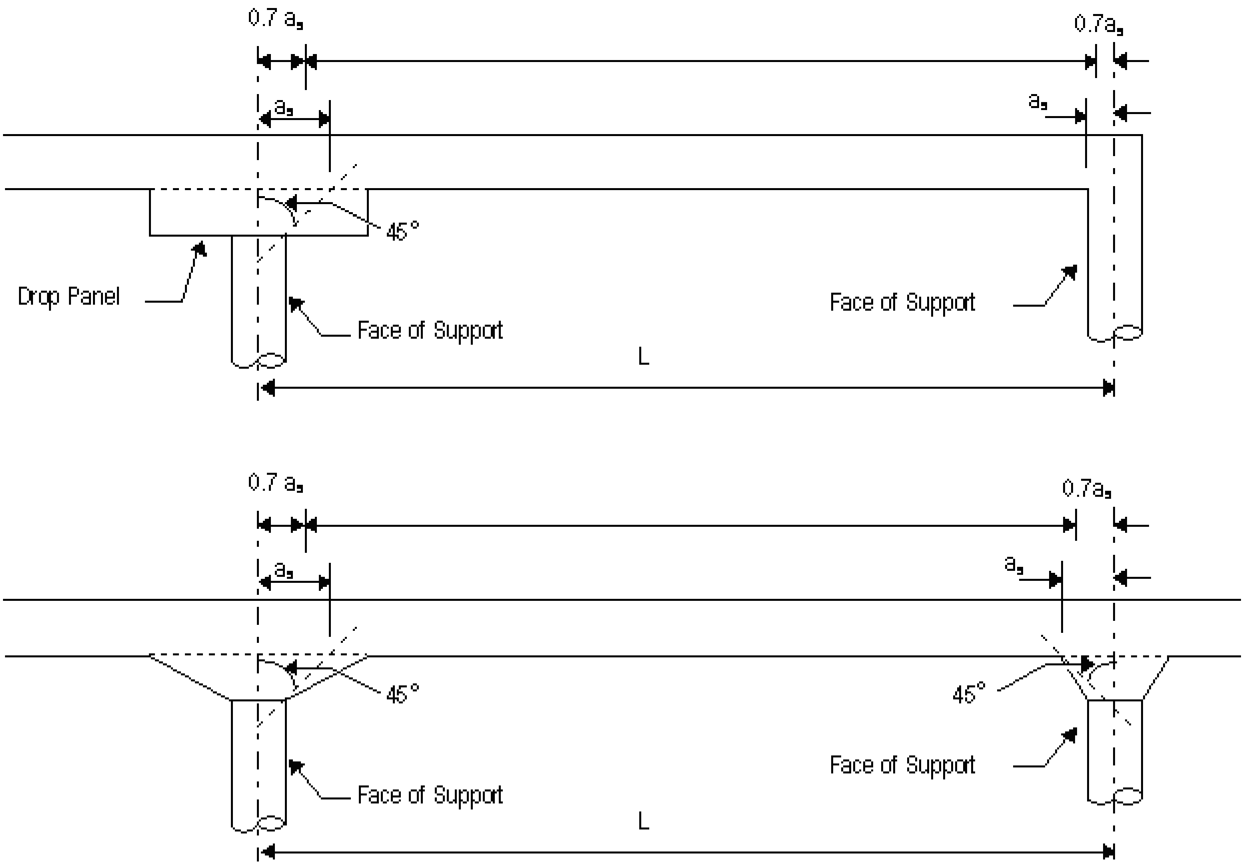
This effect is certainly present at a section centred over the column, the column strip, but must be reduced as we move further towards the centre of the panel.

It has been common to allow for this moment reduction effect by designing the slab for the moment at "the critical section". This section is at the same location in both the column and middle strips. In previous Australian and American codes the location of the critical section has varied from the face of the column to other intermediate points within the column width. RAPT uses the appropriate critical section for each code.

In calculating the critical section, RAPT will, if specified by the user in Input Screen F9, take account of transverse beams. The critical section is taken at the projection of a 45° line from the column face to the under side of the slab / beam. If this 45° line crosses the side of the transverse beam then the critical section is limited to the width of the transverse beam.

AS3600
Presently AS3600 (reference 7) states that

"The critical section for maximum negative bending moment shall be taken at 0.7 times the span support length, as, from the centre-line of the support."



FigureT.5.1 - Critical Sections for Flexure

AS3600 (reference 6) defines a span support as
"the length of a support in the direction of the span, as, taken as -

1. for beams or
   for flat slabs without either drop panels or column capitals,
   the distance from the centre line of the support to the face of the support; or
   Note: Drop panels on the underside of beams are therefore disregarded in the determination of as.
2. for flat slabs with drop panels or column capitals or both,
   the distance from the centre-line of the support to the intersection with the plane of the slab soffit of the longest line, inclined at 45 degrees to the centre-line of the support, which lies within the surfaces of the slab and the support."

RAPT will only consider drop panels in the calculation of as for slabs defined as two way slabs.

BS8110
Clause 3.7.2.6 states for flat slabs
*Negative moments greater than those at a distance hc/2 (where hc = effective diameter of a column or column head < 1.5 * min column dimension) from the centre line of the column may be ignored.*

For beams RAPT takes the critical location at the column / capital face.

ACI318
Clause 8.7.3 states
For beams built integrally with supports, design on the basis of moments at faces of support is permitted.

RAPT assumes the critical moment occurs at the face of the columns / capitals / transverse beams.

Eurocode2
Clause 2.5.3.3.(5) states
Where a beam or slab is cast monolithically into its supports, the critical design moment at the support may be taken as that at the face of the support.

RAPT assumes the critical moment occurs at the face of the columns / capitals / transverse beams.

SABS 0100
Clause 4.3.2.1.4 states
For continuous beams over supports, the design hogging moment need not to be taken as greater than the moment at a distance d/2 from the face of the support.

Clause 4.6.5.1.3 states for flat slabs
*Negative moments greater than those at a distance hc/2 (where hc = effective diameter of a column or column head not > 1.5 * the smallest dimension of the column head) from the centre line of the column may be ignored.*

T.5.2 Shear

For shear the critical location is given as

- AS3600 d past the face of the support AS3600 Clause 8.2.4
- BS8110 d past the face of the support BS8110 Clause 3.4.5.10
- ACI318 d past the face of the support ACI318 Clause 11.1.3.1
  For prestressed members ACI318 specifies (Clause 11.1.3.2) that the critical section is at h/2 past the face of the support. (where h = member thickness). This is assumed due to the varying nature of d in a prestressed member. RAPT however still assumes that the critical section is at d past the face of the support.
- Eurocode 2 d past the face of the support Eurocode 2 Clause 4.3.2.2(10)
- SABS 0100 d past the face of the support SABS 0100 Clause 4.3.4.2.3

Note: In each code the above conditions apply for generally uniform loads. RAPT does not calculate any enhancements for shear strength close to the supports when points loads are applied near the supports. Designers will need to check these relevant code conditions by hand if applicable.

In each case RAPT assumes d = 0.8D (ie d=0.8 x member thickness).

T.6 Ultimate Flexure

RAPT makes the usual strength of material assumptions for bending ie.

1. Plane sections normal to the axis remain plane after bending.
2. The concrete has no tensile strength.
3. The distribution of compressive stress is determined from a stress-strain relationship for the concrete as given by a Parabolic Rectangular Concrete Stress Strain Curve.

The program complies with each code

1. AS3600 Clause 8.1
2. BS8110 Clause 3.4.4
3. ACI318 Clauses 10.2 and 10.3
4. Eurocode2 Clause 4.3
5. SABS 0100 Clause 4.3.3

Design Procedure
The Ultimate Design is based on the following procedure

1. RAPT defines the concrete section shape and tabulates the existing reinforcement.
2. Gross section properties are calculated. The properties are
    1. centroid
    2. section modulus (top and bottom fibres)
    3. area of cross section
    4. inertia of the cross section
    5. Pe (if it exists)
    6. P/A precompression (if it exists)
3. The existing section capacity is calculated. In doing this RAPT uses the known compressive strain (at Ultimate conditions from each code), guesses a neutral axis depth and then uses a strain compatibility analysis, iterating kd until C = T.
   RAPT calculates the compression force in the concrete based on the Parabolic Rectangular Stress Strain Concrete curve. See 7.7.3 for more information on Parabolic Rectangular Concrete Curves. Note: RAPT will also include compression forces from reinforcement in the compression zone in the total compressive force.
   The tension force in the reinforcement is calculated, with RAPT determining the stress in the reinforcement layers based on the neutral axis depth (taking into account the portion of yield stress). For information on the reinforcement see section 7.7.4. For information on the prestressing strand see section 7.7.5.
   RAPT will continue its iteration process until a neutral axis depth is found which gives C=T. RAPT will then calculate a section capacity based on the neutral axis depth and the forces in the section.
4. RAPT does a ductility check at critical sections (ie sections at which a plastic hinge may form). Ductility checks are based on the ratio of the neutral axis to the effective depth (ie ku). See Ductility below for methods used by RAPT.
5. RAPT compares the initial moment capacity against the applied capacity and then, if required, adds extra reinforcement until the final moment capacity is greater than the applied moment. This will also involve strain compatibility analysis as RAPT searches for a new compatible relationship between C=T for the extra reinforcement.
   During this stage RAPT will also check to ensure that the capacity of the section is greater than the minimum moment allowed for the cross section. The codes require designers to include enough reinforcement to satisfy a moment which causes the member to crack. The ultimate strength in bending is calculated assuming a fully cracked section. For small percentages of steel, this moment could be less then the moment Mcr to cause cracking. Failure of such a member would be quite sudden. To prevent such a failure, the ultimate strength in bending must be greater than Mcr.

Ductility
Codes specify ductility limits so that structures do not become over reinforced, thus having the possibility of failing in a brittle and sudden manner. RAPT checks each span for the 3 most probable hinge (first failure points) locations (see Determination of possible Hinge Locations) and then checks these for ductility problems. ku gives this measure of ductility, with all codes specifying an allowable ku limit or an allowable steel ratio. ku is the ratio of depth of the neutral axis to the effective depth. RAPT calculates d based on the reinforcement in the tension zone of the member. If the code ductility limit is exceeded, RAPT will add compression reinforcement.

> Note: RAPT will also add tension reinforcement to a section (during ductility check or ultimate check) in the following cases
>
> 1. When RAPT tries to add compression reinforcement, if $k_{ud} < d_c$ (ie thin sections with a small effective depth, where d_c = distance to the compression reinforcement steel layer) and where the cover to the compression layer is too large, it is possible to get a situation where the compression reinforcement layer is in the tension zone. Thus RAPT can not add compression reinforcement as all layers are in tension. To overcome this problem RAPT will add tension reinforcement in an attempt to increase the effective d, thus increasing k_{ud}. This may then move the compression reinforcement layer back into the compression zone.
>   If this is not successful, then the designer must adjust the cover to the compression reinforcement or change the concrete cross section for the section to work. RAPT will only make this adjustment when doing a ductility check.

2. Where a prestressing tendon is well away from the tension face and ductility is a problem, it is sometimes possible to add reinforcement near the tension face to increase the effective depth and reduce ku, thus removing the ductility problem. RAPT will attempt to do this before adding compression reinforcement.
   Tension reinforcement added in (i) and (ii) above is only added to the cross section at which the ductility was checked. The designer, when detailing the reinforcement, should add this reinforcement over the length of the moment zone in question.
3. In some cases, it is possible for the tendons to be in the compression zone with no reinforcement in the tension zone for the initial calculation of the ultimate capacity. As this case has no effective depth to the tension force, even though the section actually has an ultimate capacity, RAPT immediately adds an area of tension reinforcement sufficient, by itself, to provide the minimum moment capacity of 1.2fMcr (or Mcr for BS8110). The section capacity is then recalculated. This check is independent of ductility requirements and will be carried out at any cross-section in the frame with the problem.

AS3600
Clause 8.1.3 of AS3600 which allows a neutral axis greater than 0.40 but artificially reduces the design strength in bending down to what one would obtain if the neutral axis was 0.40, fM_{ud}. RAPT follows the procedure as shown below

1. Calculate the design bending strength of the initial section.
2. If k_u >= 0.40 then add compression reinforcement of at least 0.01 times the area of concrete in compression.
3. Recalculate fM_u and k_u
4. If ku >= 0.40 then proceed with the following steps.
5. Reduce the design bending strength in bending down to be what would be given if k_u = 0.40. Set k_u= 0.40 and calculate the compressive forces in the concrete and compression steel. Assume that the total tensile force equilibrates the total compressive force and that its line of action remains at where it was last calculated. Calculate fM_u and call it fM_{ud}.

BS8110
BS8110 Clauses 3.2.2.1, 3.4.4.4 and 4.2.3.1 suggest that ku be limited to 0.5 with zero redistribution of moments. RAPT thus limits k_u to 0.5.

ACI318
ACI318 Clauses 10.3.3 and 18.8.1 give limits for the steel ratios in a cross section. These limits can be converted to an allowable k_u value. RAPT limits k_u to 0.42, which is equivalent to a steel ratio of 0.36 b1 with zero redistribution of moments.

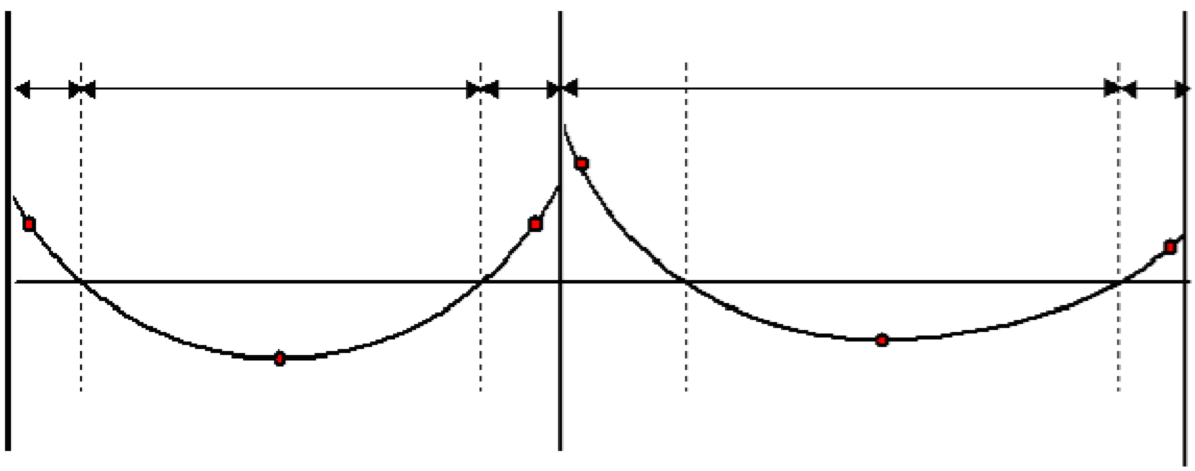
Eurocode2
Clause 2.5.3.4.2(5) limits k_u to the following values

for concrete strengths < C35/45 k_u = 0.45
for concrete strengths >= C35/45 k_u = 0.35

SABS 0100
Clause 4.3.3.1 and 5.2.2.1 suggest that k_u be limited to 0.5 with zero redistribution of moments.

Determination of possible Hinge Locations
For a member to fail, it must form 3 plastic hinges. These 3 positions can occur anywhere along the span, especially in a span with varying member element thickness due to steps. There will be one possible hinge in each of the 3 moment areas along the span.

RAPT determines the most likely hinge location in each of the moment areas shown below



Bending Moment Diagram

Figure T.6.1 - Determination of Hinge Locations

To define the most likely hinge locations RAPT does an Ultimate Moment check at all locations along the span and calculates the ratio $\frac{M_{capacity}}{M_{applied}}$ at all locations. For each moment region, the calculated ratio closest to 1 is then defined as the hinge location for that moment region. If one or more points within the moment region share the same ratio and they have the closest ratio to 1, then RAPT will define the point with the largest applied moment of this group as the hinge location.

AXIAL Compression / Tension

Axial Compression / Tension can be applied to the structure through the Load Input Menu in Input Screen F8. The axial force is applied to each cross section effecting the strain compatibility calculations ie (C-T = Axial Force). This then also effects the stresses in the concrete and reinforcement as well.

Minimum Moment

Each code gives a value for the minimum moment capacity of each cross section. These values are all based on the cracking moment. For more information on the cracking moment see section 7.7.2.

- AS3600 Clause 8.1.4.1
  M_{min} = 1.2 x f x M_{cr}
- BS8110 RAPT uses
  M_{min} = M_{cr}
- ACI318 Clause 18.8.3
  M_{min} = 1.2 x f x M_{cr}
- EUROCODE2 Clause 4.3.1.3 warns users of brittle failure. Thus RAPT assumes
  M_{min} = M_{cr}
- SABS 0100 RAPT uses
  M_{min} = M_{cr}

Unbonded Tendons

ACI318 gives a minimum area of steel when Unbonded tendons are used. No other codes specify a limit. RAPT uses the ACI318 rules and applies the following to all codes.

ACI318 Clause 18.9 (summarised)

A_s = 0.004A
where A = area of that part of cross section between flexural tension face and centre of gravity of gross section.
except for two way flat plates then

- Bonded reinforcement shall not be required in positive moment areas where computed tensile stress in the concrete at service loads (after losses) does not exceed 0.17Öf'c.
  If positive moment exceeds 0.17Öf'c minimum reinforcement is computed by

  $$A_s = \frac{N_c}{0.5 f_y}$$

  where N_c = tensile force in concrete due to unfactored dead load plus live load
- In negative moment areas at column supports, minimum area is computed by
  A_s = 0.00075hl
  where
  l = length of span in direction parallel to that of the reinforcement being determined.
  h = overall thickness of member

Effective Beam Widths

For T or L beam action the effective flange width used is the width input by the user in the input. RAPT defaults this value to the code defaults

- AS3600 clause 8.8
- BS8110 clause 3.4.1.5
- ACI318 clause 8.10
- Eurocode2 clause 2.5.2.2.1
- SABS 0100 clause 4.3.1.5

From AS3600 clause 8.8

"a) T-beams bef = bw + 0.2a; and
b) L-beams bef = bw + 0.1a
where a is the distance between points of zero bending moment, which for continuous beams, may be taken as 0.7L.

In both items a) and b) above, the overhanging part of the flange considered effective shall not exceed half the clear distance to the next member. The effective width so determined may taken as constant over the whole span."

Detailing Considerations

The different codes specify different reinforcement and tendon distribution in two way slabs. ie for AS3600 -

Reinforcement and tendon distribution in two-way flat slabs (Clause 9.1.2 AS3600) requires that at least 25 percent of the total of the design negative moment in the column strip and adjacent half middle-strips shall be resisted by reinforcement or tendons or both, located in a cross-section of slab centred on the column and of a width equal to twice the overall depth of the slab or drop panel plus the width of the column.

Note: These concentrations of reinforcement are NOT checked or detailed by RAPT.

T.7 Serviceability

Serviceability takes into account all the working stress and deflection checks. In this section we will consider each element of the section in turn giving a detailed discussion explaining how RAPT performs these calculations.

T.7.1 General

RAPT uses code combinations to define the working load cases. (Crack control cases) A detailed list of the default combinations for each code type as given in Section 4.10 of this manual.

The total prestress moment is comprised of the prestress force multiplied by the eccentricity of the tendon from the centroid of the section plus the secondary prestress moment (see Section T.10) due to frame continuity.

$$PW = P_{sec} + P.e \qquad \text{(T.7.1)}$$

When there is more than one layer of prestressing the prestress force is the total force of all tendons and the eccentricity is the location of the resultant prestress tensile force.

For an uncracked section, stresses may be based on the gross section properties. Therefore

$$\sigma_t = \frac{P}{A} \pm \frac{M_s}{Z_t}$$

$$\sigma_b = \frac{P}{A} \pm \frac{M_s}{Z_b} \qquad \text{(T.7.2)}$$

where $M_s = M_{DL} + M_{LL} + PW$

If the simplification is made that the section is cracked as soon as the extreme tensile fibre goes into tension then, if either the top or bottom fibres are in tension, a cracked section analysis should be carried out.

For this calculation we find the unique combination of the compressive top fibre strain, e_t, and the neutral axis depth, kd, that will produce an internal moment in the section equal to $M_{s'}$. The compressive fibre is defined as the most compressive face from the stress calculations based on gross section properties from above (equation T.7.2).

where $M_{s'} = M_{DL} + M_{LL} + P.e$

The reason that we converge towards $M_{s'}$ and not M_s is that the strain compatibility calculations include the P.e term in the strength of the section.

Applied Moment	=	Internal Moment	
$DL + y_sLL + P_{sec}$	=	$C.y - P.e - T.e_t$	(Ms' loading case)
$DL + y_sLL + (P_{sec} + P.e)$	=	$C.y - T.e_t$	(Ms loading case)
$DL + y_sLL + PW$	=	$C.y - T.e_t$	(T.7.3)

where
C = The Compressive Force
y = Eccentricity of the Compressive Force
T = Change in tensile force of all Reinforcement (including prestress)
e_t = Eccentricity of force T
P = The Total Net Force in the Tendons.
e = The Eccentricity of the Total Prestressing Force, P
DL = Dead Load
LL = Live Load
y_s = Short Term Live Load Factor
P_{sec} = Secondary Moment
PW = Total Prestress Moment

Note:

1. That the P/A term does not appear in the above equations. The force is still present but is represented in the compressive force term. The logic applied above for cracked sections is identical to that used at the ultimate limit state where again the section is cracked.
2. As the section is in internal equilibrium the eccentricity may be measured from any location as long as consistency is maintained and due account is taken of the signs of the forces.

Strain COMPATIBILITY Calculations
By trial and error we find the unique top fibre strain and neutral axis depth that will give an internal moment equal to the applied bending moment, Ms', whilst simultaneously satisfying internal force equilibrium within the cross-section.

T.7.2 Cracking Moment

Once the correct solution to the working stress calculation is found, the Cracking Moment, Mcr, is calculated based on the orientation of the section to suit M*. This will be the orientation when the first crack appears even though the tensile fibre may switch over as the load increases (for post-tensioned sections).

Mcr is calculated using the limiting tensile strength, ft, at the extreme tensile fibre of the concrete, given by

1. AS3600 clause 6.1.1.2 as 0.6 Öf'c
2. BS8110 defaults to 0.5Öf'c (as recommended by "Reinforced Concrete Designer's Handbook" Charles E Reynolds and James C Steadman)
3. ACI318 clause 18.4 as 0.6Öf'c
4. Eurocode2 clause 3.1.2.3 as 0.21 $f_{ck}^{2/3}$
5. SABS 0100 defaults to 0.5Öf'c (as recommended by "Reinforced Concrete Designer's Handbook" Charles E Reynolds and James C Steadman)

RAPT calculates M_{cr} as

$$\therefore M_{cr} = Z_{tens}\left(f_t + \frac{\sum (P \times e)}{Z_{tens}} + \frac{\sum P}{A_g} \right) \qquad \text{(T.7.4)}$$

If $M_{s'} > M_{cr}$ (ie section cracks at service) then the cracked moment of inertia must be calculated. The centroid (centre of area), y_c, of the transformed (equivalent concrete area calculated by multiplying the steel area concerned by E_s/E_{conc}) section may be found by taking first moments of the transformed areas about the top fibre and dividing by the total transformed area.

For reinforced sections y_c will equal kd for the cracked section analysis due to the absence of the prestress force. The cracked inertia is then calculated using the parallel axis theorem about y_c.

Cracked Moment of Inertia
The cracked moment of inertia may be readily obtained for a singly reinforced rectangular section (non-prestressed) from considering equilibrium of the section and is summarised below. This calculation assumes that the top fibre strain remains within the linear region of the concrete stress-strain function ie <= 0.45 f'c

y_c = Ö(p^2n^2 + 2pn) - pn (T.7.5)

$I_{cr} = b(y_c)^3 / 3 + nA_{st} (d - y_c)^2$

where

$p = A_{st} / bd$

$n = E_s / E_c$

T.7.3 Concrete

T.7.3.1 Stress / Strain Curve

The compressive fibre strain will only be within the linear region of the stress / strain curve for the concrete for low levels of strain in the concrete. This assumption is inaccurate for concrete stresses above 0.3 -0.4 f'c. Also, under ultimate strength conditions, all codes allow the use of an equivalent rectangular (simplified) concrete compression force. These simplified stress blocks become less accurate as the section shape becomes less regular. Also, they cannot be used for service limit state calculations. For these reasons, RAPT uses the more general parabolic stress block stress / strain function as set out in the British and European Codes.

$s_1 = A \times e^2 + B \times e$ From $e = 0$ to e_o

$s_2 = f_{con} \times f_{act}$ From $e = e_o$ to e_t

where

A = Curvature of the parabola

B = Initial Youngs Modulus for the Concrete Curve

e = Strain at any point

e_o = Strain at which f'c is first reached. Different Codes specify different strains for this point.

e_t = End Strain Point as defined by Codes (Ultimate strain limit).

f_{act} = Factor applied to limit stress in concrete as defined in each code.

The concrete stress strain curve is defined by each code where

(i)	AS3600	e_t = 0.003 AS3600 clause 8.1.3	
		B	AS3600 clause 6.1.2
(ii)	BS8110	e_t = 0.0035 BS8110 Part 2 figure 2.1.	
		B & e_o	BS8110 Part 1 figure 2.1
	RAPT applies a cube to cylinder factor to the concrete strength as discussed below.		
(iii)	ACI318	e_t = 0.003 ACI318 10.2.3	
(iv)	Eurocode2	e_t = varies with concrete strength. Eurocode2 clause 4.2.1.3.3	
		A & B	Eurocode2 Figure 4.2
(v)	SABS 0100	e_t = 0.0035 SABS 0100 Figure 1.	
		B & e_o	SABS 0100 Figure 1.

This allows us to more accurately predict the stress condition under any loading condition from zero stress to the full ultimate capacity for any cross section shape. This level of accuracy is absolutely necessary for the design of partially prestress sections and for calculation of the curvatures for deflection calculations and is also helpful for all other concrete designs.

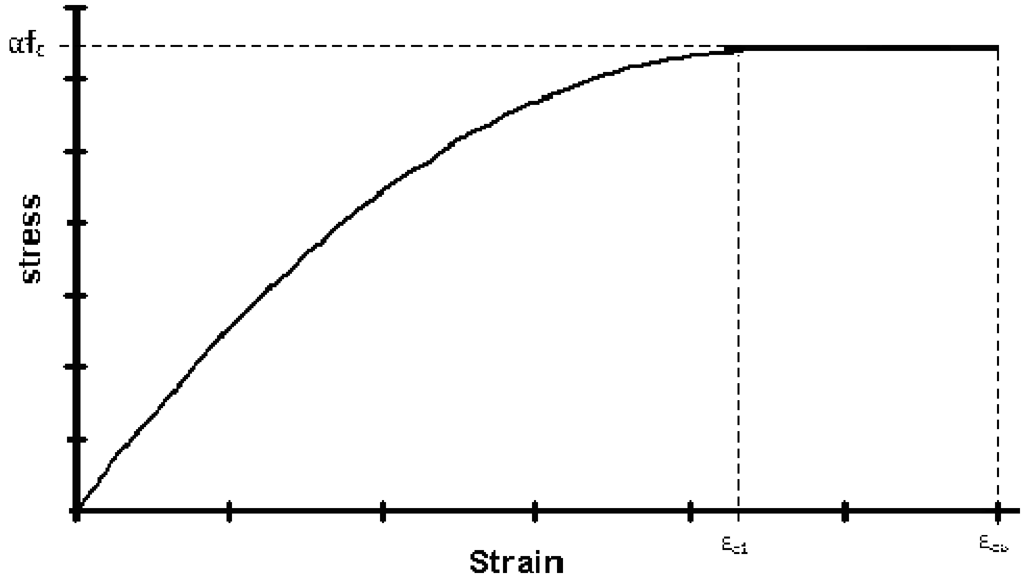
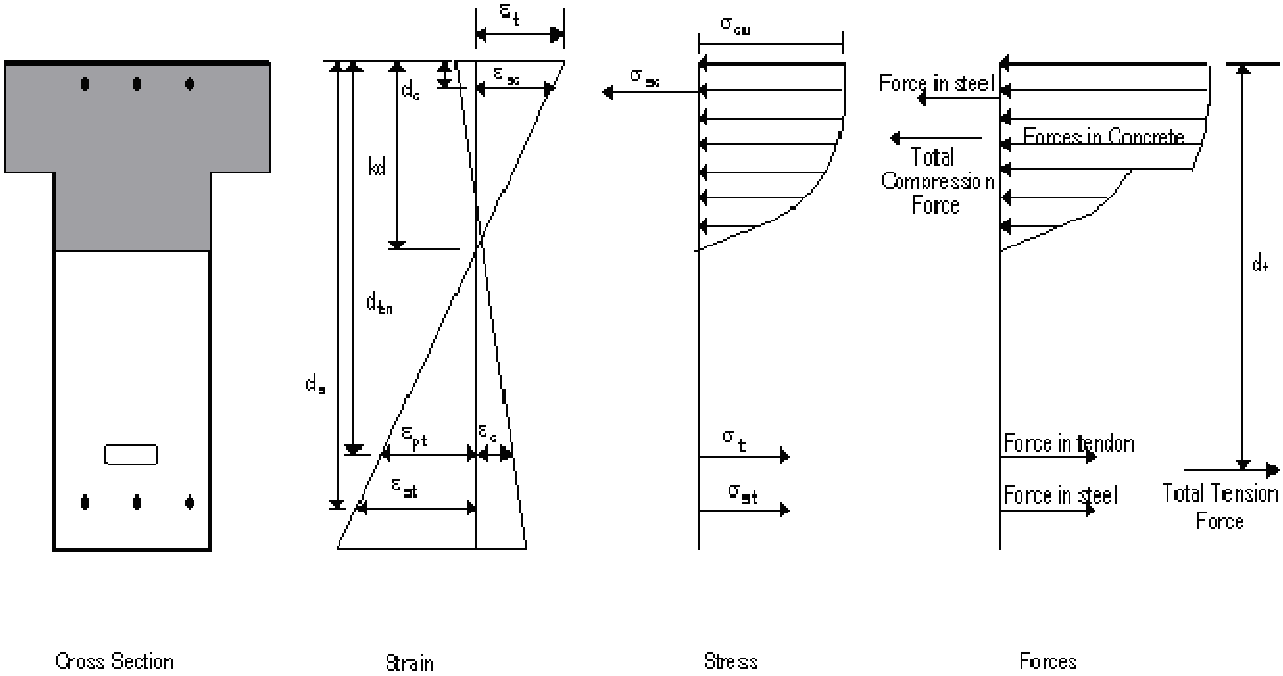


Figure T.7.3.1 - Parabolic Rectangular Stress / Strain Concrete Curve

T.7.3.2 Forces in a Cross Section

Figure 7.7.2 shows the strain, stress and force diagrams used by RAPT in calculating the moment properties of a section. Refer to section 7.7.4 and T.7.5 for information on reinforcement and tendons.

Notice the compression stresses and forces generated by the concrete during bending.



FigureT.7.3.2 - Strain, Stress and Forces Diagrams

T.7.3.3 Youngs Modulus of Concrete

The modulus of concrete is calculated differently for each code. RAPT will use the code default unless the user has specified a value in Input screen .

(i)	AS3600	Clause 6.1.2	E_c = 0.043 r$^{1.5}$ Öf''c
(ii)	BS8110 Part2	Clause 7.2	E_c = 5500Ö(f_{cu} / g_m)
(iii)	ACI318	Clause 8.5	E_c = 0.043 r$^{1.5}$ Öf''c
(iv)	Eurocode2	Figure 4.2	E_{cm} = 1000 x f_{ck} x 0.85 / g_m
(v)	SABS 0100	Appendix C.1.3	E_{con} =5500Ö(f_{cu} / g_m)

The Youngs Modulus used in calculating frame properties and deflections is a mean value calculated from Cement and Concrete Association records. Figure 7.7.3.3 shows the graph used by RAPT. This can be expressed as

f_{cm} = 1.085 f'c + 2.5

This mean concrete stress is applied to the above Youngs moduli formula to get the Mean Youngs Moduli.

The compressive strength of concrete varies over time and the following formula is used to estimate the compressive strength at a given time t according to ACI- 209 (1978)

$$f'_c(t) = \frac{t}{\alpha + \beta \times t} \times f'_c(28)$$

where

For normal Portland cement:

For moist cured concrete:	a = 4.0	b = 0.85
For steam cured concrete:	a = 1.0	b = 0.95

For high early strength cement:

For moist cured concrete:	a = 2.3	b = 0.92
For steam cured concrete:	a = 0.7	b = 0.98

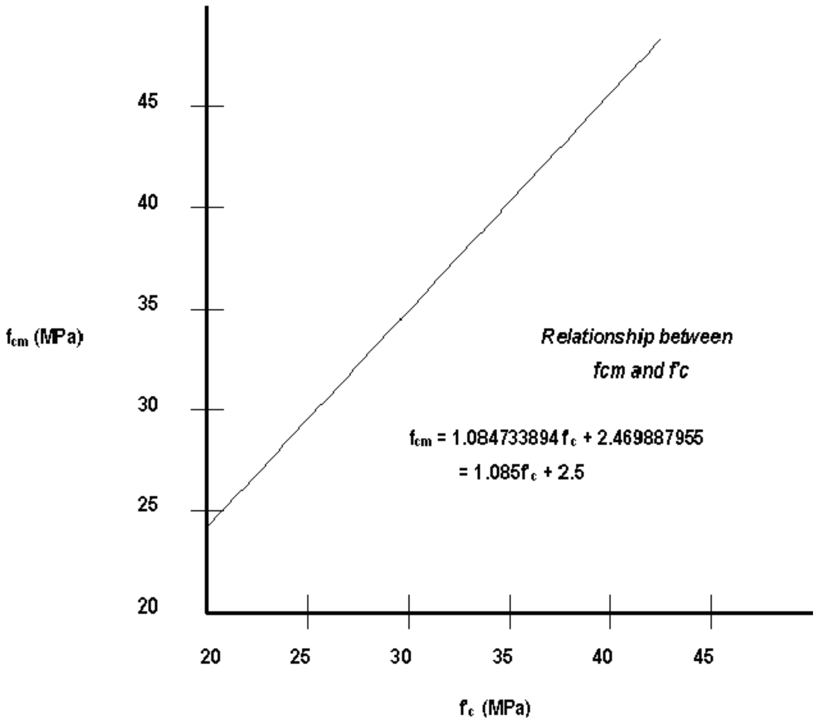


Figure T.7.3.3

T.7.3.4 Creep

Creep in concrete is made up of

1. creep when concrete is kept at hygral equilibrium
2. drying creep when water is lost from the matrix.

Under constant applied loading the concrete will creep ie experience extra strain without the application of extra load. Each code defines a f factor (creep factor) which is used to modify to the Youngs Modulus, thus taking into account the creep affects.

AS3600 / General Approach
To define creep, AS3600 uses a factor, defined as the basic creep factor, $f_{cc.b}$, based on the ratio of the ultimate creep strain to the elastic strain for a specimen loaded at 28 days under a constant stress of 0.4f'c.

AS3600 suggests that the basic creep factor, $f_{cc.b}$ may be taken as follows

f'_c (MPa)	20	25	32	40	50
$f_{cc.b}$	5.2	4.2	3.4	2.5	2.0

To account for the effects of ambient humidity, member proportions and load duration a term k2 is introduced. To account for the maturity of the concrete at the time of the initial loading the term k_3 is introduced. The design creep factor f_{cc} is then given by

$f_{cc} = f_{cc.b}\ k_2\ k_3$

where

k_2 from Figure 6.1.8.2A of AS3600
k_3 from Figure 6.1.8.2.B of AS3600.
= 1.85 - 0.75R for 0 <= R <= 1
= 1.60 - 0.50R for 1 <= R <= 1.4
= 0.90 for R >= 1.4

The creep of the concrete is dependent upon the age of the concrete when the loading is applied hence use of the compressive strength at transfer in the k3 term for prestress loss calculations.

Note: RAPT internally calculates the k values from formula representing the curves given in AS3600.

To calculate k_3 you must know a value for f_{cm}. The value of f_{cm} is the taken as the mean compressive strength in the concrete at loading.

The creep curves as defined in AS3600 are estimated in RAPT by the following formulas

$$k_2 = \frac{k_7 \times k_8 \times t^{0.7}}{t^{0.7} + k_9}$$

where

$$k_7 = 0.76 + 0.9e^{-0.008t_h}$$

$$k_8 = 1.37 - 0.011h$$

$$k_9 = 0.15 \times t_h$$

$$t_h = \frac{2A_g}{u_e}$$

Hypothetical thickness of a member
A_g = The gross cross section area of a member
u_e = The exposed perimeter of a member cross section
h = Relative Humidity (%)

Using the above formulae, users can model the code curves by setting

- Relative Humidity = 40% = for Arid conditions
- Relative Humidity = 50% = for Interior Environment conditions

- Relative Humidity = 55% = for Temperate conditions
- Relative Humidity = 65% = for Tropical conditions

BS8110, ACI318 and SABS 0100

As stated in the operations section of this manual, BS8110 and ACI318 creep results will match the code requirements using the formula as set out in AS3600. Users will however need to make the following adjustments if using BS8110 or SABS 0100.

To simulate these codes set the creep factor to 3.4 in input screen F1.

Eurocode2 creep

RAPT calculates the creep design factor in accordance with Appendix A.1.1.2 of Eurocode2. RAPT uses the same formula to calculate P_4

where
$k_2 = f_{RH} \times b_0(t-t_o)$
$k_3 = b(t_o)$
$f_{cc.b} = b(f_{cm})$

From Appendix A1.1.2
$f(t,t_o) = f_o \times b_c(t - t_o)$
where
$f_o = f_{RH} \times b(f_{cm}) \times b(t_o)$ notional creep coefficient

$f_{RH} = 1 + \left(\frac{1 - \frac{RH}{100}}{0.10 \times \sqrt[3]{h_o}} \right)$ factor to allow for effect of relative humidity

$b(f_{cm}) = \frac{16.8}{\sqrt{f_{cm}}}$ factor to allow for effect of concrete strength

$b(t_o) = \frac{1}{\left(0.1 + t_o^{\;0.20}\right)}$ factor to allow for effect of concrete loading age

$h_o = \frac{2A_c}{u}$ notional size of member in mm.

$b_c(t-t_o) = \left(\frac{(t - t_o)}{\beta_H + t - t_o} \right)^{0.3}$ factor describing the development of creep with time

$b_H = 1.5 \times \left(1 + (0.012RH)^{18}\right) \times h_o + 250 \leq 1500$

T.7.3.5 Shrinkage

Each code gives a method to calculated the shrinkage strain of concrete. For each code type, we have simplified their formula to a common expression of

$e_{cs} = e_{cs.b} \times k_1$

AS3600
AS3600 suggests that a suitable mean value of basic shrinkage strain, $e_{cs.b}$, be taken as 700 E-6. This is converted to a design shrinkage strain, e_{cs}, by multiplying by k_1 (see below).

The factor k_1 may be obtained from Figure 6.1.7.2 of AS3600. The theoretical thickness is a parameter modelling the sensitivity of the cross-section to moisture loss and hence shrinkage. The formula therefore includes allowance for the length of the perimeter of the cross-section which is exposed to the atmosphere. Figure 6.1.7.2 provides for different severities of drying conditions.

$t_h = 2 A_g / u_e$ - theoretical thickness
A_g = gross area of the cross-section
u_e = exposed perimeter of the cross-section + half the perimeter of any closed voids contained therein.

Using formulas suggested by Gilbert (reference 43), k_1 can be estimated from

$$k_1 = \frac{k_4 \times k_5 \times t^{0.7}}{t^{0.7} + k_6}$$

where

$$k_4 = 0.62 + 1.5e^{-0.005t_h}$$

$$k_5 = \frac{4.0 - 0.04h}{3}$$

$$k_6 = 0.15 \times t_h$$

$$t_h = \frac{2A_g}{u_e}$$

Hypothetical thickness of a member
A_g = The gross cross section area of a member
u_e = The exposed perimeter of a member cross section
h = Relative Humidity (%)

Using the above formulae, users can model the code curves by setting

- Relative Humidity = 40% = for Arid conditions
- Relative Humidity = 50% = for Interior Environment conditions
- Relative Humidity = 55% = for Temperate conditions
- Relative Humidity = 65% = for Tropical conditions

BS8110, ACI318, AS1480 and SABS 0100
BS8110, ACI318 and AS1480 all use the same approach as explained above. The above formula will simulate the code shrinkage values when users input $e_{cs.b}$ as suggested in input screen F5 or shown below (see section 4.7). Users will need to set this figure based on materials in their region. Below are the code references and their equivalent recommended basic shrinkage strain values.

Code References

- AS3600 Clause 6.1.7.2 code suggested value 700E-6
- BS8110 Clause 4.8.4 & Part 2 clause 7.4 use 385E-6 (for British Conditions)
- ACI318 Clause 18.6 use 700E-6
- SABS 0100 Appendix C.3 use 385E-6

Eurocode2
RAPT calculates the shrinkage design factor in accordance with Appendix A.1.1.3 of Eurocode2. If this value is entered as 0, RAPT will use the formula described in Appendix A1.1.3 to calculate the shrinkage value. This value is dependent

on f'_c and the Basic Shrinkage Strain $e_s(f_{cm})$ will be recalculated whenever the concrete strength is modified in screen F1. The following formulae are used by RAPT.

$e_{cs}(t-t_s) = e_{cso} \times b_s(t-t_s)$

where
$e_{cso} = e_s(f_{cm}) \times b_{RH}$ (notional shrinkage coefficient)
$e_s(f_{cm}) = (160 + 5 \times (90 - f_{cm}))$ 10-6 a factor to allow for concrete strength on shrinkage

$b_{RH} = -1.55 \times \left(1 - \left(\frac{RH}{100}\right)^3\right)$ a factor to allow for the relative humidity

$b_s(t-t_s) = \left(\frac{(t-t_s)}{(0.035 \times h_o{}^2 + t - t_s)}\right)^{0.5}$ coefficient describing development of shrinkage with time

$h_o = 2A_c / u$ notional size of member in mm

where
A_c = the area of the cross section
u = the perimeter of the member in contact with the atmosphere.

RAPT then redefines this in our standard format as

$e_{cs} = e_{cs.b} \times k_1$
where
$k_1 = b_{RH} \times b_s(t-t_s)$
$e_{cs.b} = e_s(f_{cm})$

T.7.4 Reinforcement

The strain in normal reinforcement is obtained assuming strain compatibility of the concrete and steel and that plane sections remain plane. If the assumption is made that the steel will exhibit elastic-plastic behaviour then by similar triangles

$$\sigma_{st} = E_s \varepsilon_t \frac{(d_t - kd)}{kd}$$

= stress in a tensile steel layer with a modulus of elasticity Es, at depth ds, measured from the compressive fibre

For a compressive layer of steel

$$\sigma_{sc} = E_s \varepsilon_t \frac{(kd - d_c)}{kd}$$

The calculation of the compressive force in the concrete assumed that no steel was present in the compressive zone. The stress in compressive steel must be modified to erase the double count of the force due to the compression in the concrete at that level. Therefore

$s_{s'} = s_s - s$

The force in the steel is found simply by multiplying the stress by the area of steel at that location. Figure 7.7.3.2 shows the various variables used in the formula above

Development Lengths
RAPT allows users to define existing reinforcement for a design in Input F9 [Reinf]. When RAPT sets up the Default Bars, the end of each bar is assumed to be fully developed. Thus when detailing the bars, designers must allow for the extra portion of bar needed to develop the yield strength.

When entering reinforcement in the User Defined Reinforcement menu (Input F9 [Reinf]), the user is required to specify the yield condition at the end of the bar. This value can range from 0% (no yield stress) to 100% (full yield stress). If the end condition is specified as 0% yield stress, then RAPT will calculate the development length required to obtain full yield stress according to the chosen code type. RAPT will then use the appropriate stress at different points in the span, when calculating the extra reinforcement required.

The code development length formulae can be found as follows

- AS3600 Clause 13.1.2
- BS8110 Clause 3.12.8
- ACI318 Clause 12.3
- Eurocode2 Clause 5.2.2
- SABS 0100 Clause 4.11.6

T.7.5 Tendons

For tensioned steel the total strain in the steel will be the summation of

e_{pt} The Tensile or Compressive Strain from the applied moment, Ms' (which induces a top fibre compressive strain, e_t, in the concrete) (See figure T.7.2)

e_{pe} The Tensile Strain from the initial tensioning of the tendon

e_{ce} The Strain from the initial axial compression and hog of the concrete section prior to grouting of the duct. Even though this is initially a compressive strain after grouting of the duct it is thereafter considered a tensile strain as e_{ce} must be overcome by applying tensile strain prior to achieving applied tensile strains. (See figure T.7.2)

NOTE: This is not considered for unbonded tendons

where

$$\varepsilon_{pt} = \frac{\varepsilon_t\left(d_{ten} - kd\right)}{kd}$$

as long as it lies in the tension zone, ie below kd

$$\varepsilon_{pe} = \frac{P}{A \times E_p}$$

where

P = Total Force in the tendon

E_p = Youngs modulus of the tendon

A_p = Area of the tendon

$$\varepsilon_{ce} = \frac{1}{E_{con}} \times \left(\frac{\sum\left(P.e_i\right) \times e_i}{I_g} + \frac{\sum P}{A_g}\right)$$

where

E_{con} = According to code.

r = Concrete Density (kg/m3) commonly taken as 2400.

e_i = The Eccentricity of that tendon from the centroid of the uncracked concrete section.

I_g = Gross Inertia of the cross section.

It may be argued that the self-weight bending moment should be taken into account in calculation of e_{ce}. However e_{ce} is usually less than one tenth the value of the other strains in the steels and does not play a major role.

The calculation of the compressive force in the concrete assumed that no steel was present in the compressive zone. The stress in compressive steel must be modified to erase the double count of the force due to the compression in the concrete at that level.

Therefore

$s_{s'} = s_s - s$
where
s is the stress in the concrete at the steel level given by equation T.7.7.

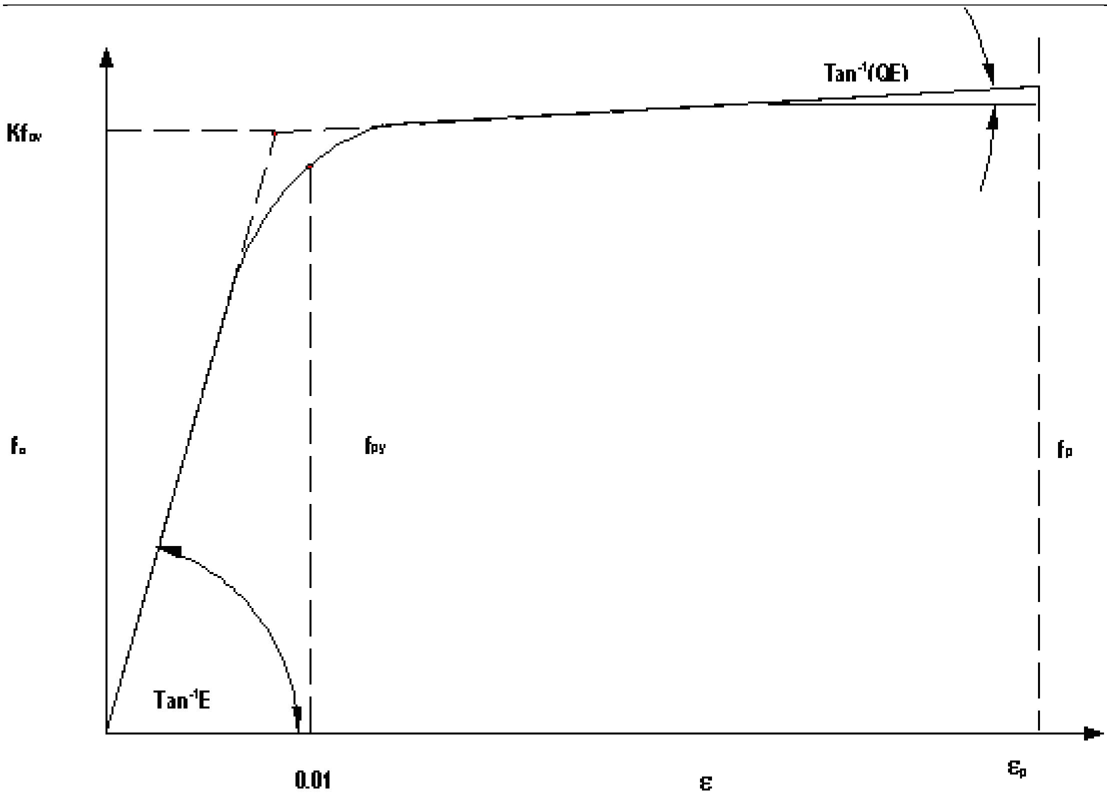


Figure T.7.5.1 - Default Stress/Strain curve for Prestressing Steel AS3600

The above stress / strain curve is the general curve used to estimate the forces in tendons for AS3600. The curve proposed by Naaman (ref 36) is described below.

The curve is taken as being a straight line rising at a slope equal to the Youngs modulus of the steel, E, which strikes the continuation of the strain hardening modulus (of slope QE) at a stress of K f_{py} (which does not lie on the curve). The two straight lines are joined by equation T.7.16 given below. Equation T.7.16 is in fact valid for any strain whether it lies in the linear regions or not.

To define the curve one must know

f_p = the Stress at Ultimate Tensile Failure
e_p = the Strain at Ultimate Tensile Failure
E = initial modulus in Elastic Range (Youngs modulus)
E' = final modulus in Strain Hardening Range = Q E
f_{py} = the Yield Stress taken as the stress at 1% strain

Suppliers of prestressing wire will supply test certificates upon request. The terms f_{py}, f_p, e_p, E and a graph of stress versus strain will be given. The strain hardening modulus, QE, will have to be calculated off the graph.

Solve equation T.7.17 for the co-efficient K (about 1.04)

$$Q = \frac{\left(f_p - Kf_{py}\right)}{\left(\varepsilon_p E - Kf_{py}\right)}$$

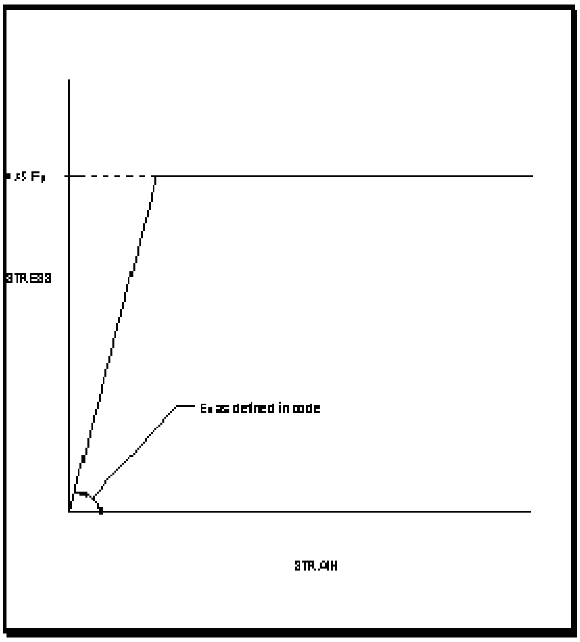
Solve for R in equation T.7.16 using the result at 1% strain ie fs = fpy and e = 0.01 and then use the equation below for any known strain value, e.

$$f_s = \varepsilon E \left[Q + \frac{1 \quad Q}{\left(1 + \left(\frac{\varepsilon E}{Kf_{py}}\right)^R\right)^{1/R}} \right]$$

The force in the steel is found simply by multiplying the stress by the area of steel at that location.

Designers can also use one of the code simplifications. (see section 4.7 for procedure).

Below are a list of the available curves / simplifications depending on the code type chosen.



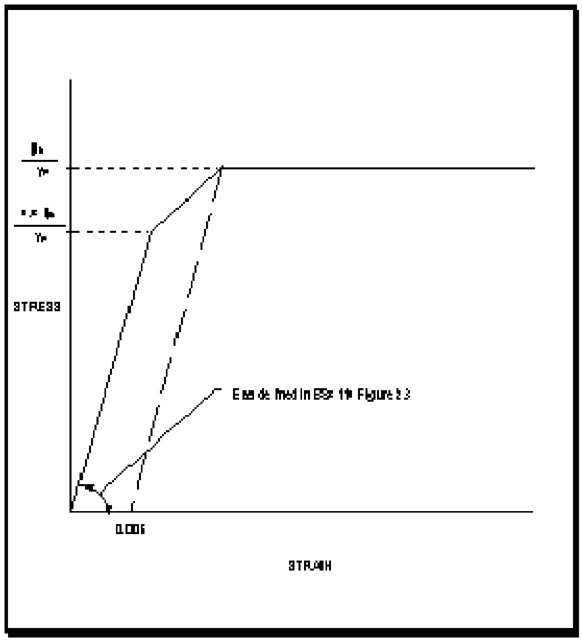


Figure T.7.5.2 - AS3600 Elastic Plastic Curve Figure T.7.5.3 - BS8110 and SABS 0100 Approximation

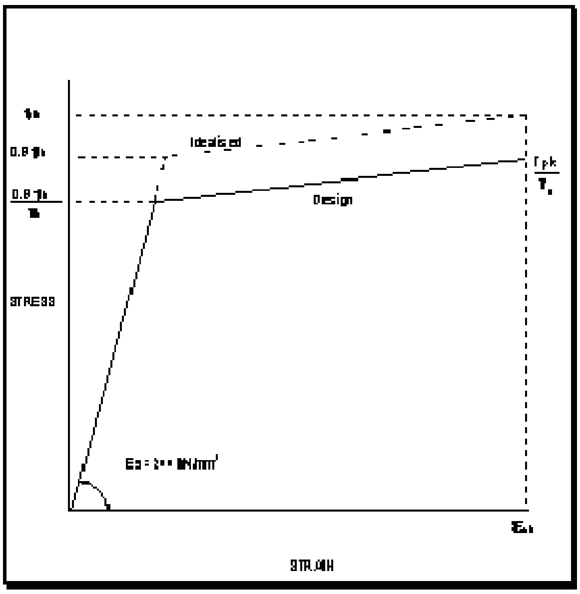


Figure T.7.5.4 - Eurocode2 Approximation

Tendon properties
Below is a list of various tendon properties, sizes and other relevant information. Users should check with their local supplier for more detailed information on tendon properties. The list below will give users an idea of what is available.

DUCT DIAMETER
Duct sizes have been standardised by the various prestressing companies to the following.

- 70 mm (wide) x 19 mm (up to 5 strands)
- 90 mm (wide) x 19 mm (up to 6 strands)

These are both termed flat or oval ducts.

For beams, round ducts are used if flat ducts are not feasible. The round duct sizes (multi-strand systems) have not been standardised between prestressing companies. Wherever possible flat ducts are used as the strands are stressed using single strand jacks, operable by one man, whereas multi-strand tendons require a jack which stresses all the strands simultaneously. These jacks have significant weight and require lifting by a crane. This may cause site complications as the site crane will be involved with tensioning work for a significant period and also access for the jack may be limited.

Typical sizes of round (multi-strand) ducts are shown below. Note that these vary depending upon which prestressing company is used.

duct outside diameter (mm)	suitable number of strands
56	5 - 7
75	8 - 12
90	13 - 19
101	20 - 27
121	28 - 37
124	38 - 42
141	43 - 55

Note:

1. For Pretensioned strands RAPT defaults the duct diameter to the strand diameter.
2. For Unbonded Post-Tensioned strands RAPT defaults the duct diameter to the strand diameter plus 1mm

Strand CG

For flat bonded round (ie. only one strand in elevation) the maximum offset of the cg of the strands to the cg of the duct will be

- for 12.7 mm strand offset is 3.15 mm = (19 - 12.7)/2
- for 15.2 mm strand offset is 1.9 mm = (19 - 15.2)/2

For bonded beam ducts the centroid of the strand group depends on how many strands are actually placed in the duct. A common approximation is that the offset is as follows

- +/- 0.15 times the outside diameter of the duct when the maximum permissible number of strands is placed in the duct or
- +/- 0.23 times the outside diameter of the duct when the minimum number of strands is placed in the duct.

Drawin

The amount of drawin is dependant upon the type of anchorage system being used however the following are typical figures and defaulted by RAPT.

- Bonded 6mm
- Unbonded 6mm
- Pretensioned 0mm

Friction

For multi-strand tendons, if a strand in the middle of the group is stressed separately then it would experience a different friction to a strand on the outside of the group if it were stressed separately. The co-efficients are tabulated for all strands being stressed simultaneously. This does not happen for flat ducts where each strand is stressed individually but is true for beam ducts. For flat ducts the factor none-the-less provides good correlation with measured results in the field.

In the absence of specific test data the following co-efficients are used.

(i) for greased-and-wrapped coating	0.10 - 0.15
(ii) for bright and zinc-coated metal sheathing	0.15 - 0.20
(iii) for bright and zinc-coated flat metal ducts	0.20
(iv) machined, cast steel	0.15

saddles	
(v) lead-coated metal sheathing	0.15

Anchorage Friction
The loss of prestress is dependant upon the anchorage system being used. The following values are suggested values

- Bonded 2% (of force measured at Jack)
- Unbonded 2%
- Pretensioned 0%

Wobble Factor

(i) for sheathing containing tendons other than bars and having an internal diameter of

< 50 mm	0.016 to 0.024
> 50 mm and < 90 mm	0.012 to 0.016
> 90 mm and < 140 mm	0.008 to 0.012

(ii) for flat metal ducts containing tendons other than bars (McAlloy)
0.016 to 0.024 (commonly taken as 0.025)

(iii) for sheathing containing bars and having an internal diameter of 50 mm or less
0.008 to 0.0016

(iv) for bars of any diameter in a greased and wrapped coating
0.008

(v) for unbonded tendons ACI 318 gives the following recommendations for duct wobble
Mastic Coated 0.001 to 0.002
Pre-Greased 0.0003 to 0.002

Creep
AS3600 uses a factor, defined as the basic creep factor, $f_{cc.b}$, based on the ratio of the ultimate creep strain to the elastic strain for a specimen loaded at 28 days under a constant stress of 0.4f'c. This figure is modified to account for the environment (humidity) and time.

AS3600 gives the following table (Table 6.1.8.1)

Characteristic strength f'_c, MPa	20	25	32	40	50
Creep factor $f_{cc.b}$	5.2	4.2	3.4	2.5	2.0

Eurocode formula are as specified in the code in Appendix A1.1.2

Relaxation
The basic relaxation value is measured by the strand suppliers and for Australian materials may be taken as

- for low relaxation wire 1 %
- for low relaxation strand 2 %
- for alloy steel bars 3 %

Cover to tendons
AS3600 allows figures as low as 15mm for exposure cover. The authors consider that any figure below 25mm will not allow adequate concrete penetration and compaction below the duct. Cover of less than 25 mm could result in bony concrete leading to corrosion. Another possible problem is that reasonably high pressures are achieved during grouting and a concrete cover of 15mm to 20mm is not sufficient to resist these pressures when it is realised that the duct itself is not pressure sealed.

The specified covers should be the minimum of those required for corrosion and for fire resistance. It is usual for the architect to nominate the required fire rating for a building from reference to the local government ordnance.

T.7.6 Crack Control for Flexure

RAPT allows the designer to control the cracking in Prestressed systems or reinforced by specifying an allowable stress change in the reinforcement / tendons. Each code has different limits / classes for crack control. For specifics on each code see section 5.1.4 in this manual or

- AS3600 clause 8.6 and 9.4 (see below)
- BS8110 clause 4.1.3, 4.3.4.3 and Part II Section 3. Also see Section 6 of Concrete Society Technical Report No. 23 - Partial Prestressing.
- ACI318 clause 18.4.2 (RAPT informs the designer if the limits are exceeded)
- Eurocode2 clause 4.4.2.3 (RAPT will inform users if the stress limit set is exceeded)

Below (section T.7.6.1 to 4), the crack control limits for AS3600 have been shown with an explanation of how RAPT interacts with the code rules to give the designer control over cracking. RAPT uses the same procedure for all codes ie RAPT will check the reinforcement stresses, and if required add extra reinforcement to reduce the stresses to an acceptable level. How this extra reinforcement is applied depends on the code being used. The explanation of AS3600 rules follows.

T.7.6.1 Prestressed Slabs

Designers may select to ensure compliance of their designs for crack control to clause 9.4.2 AS3600 in a number of ways. RAPT cannot pre-empt the designer's decision.

If the maximum tensile stress in the concrete (on gross or cracked section as applicable), under the action of the short-term service loads, is less than 0.25 Öf'c then no further action is required by the code. If this is not satisfied then the designer must place some reinforcement or bonded tendons near the tensile face to control any cracking that may occur. The code gives no guide-lines on how much to provide or at what spacing. Further to this one must satisfy either one of the following

- limit the maximum tensile stress under the short-term service loads to 0.5 Öf'c or
- complying with both the following
  - limiting the stress change (in the reinforcement or tendon closest to the tensile face) from decompression to short-term service loading to 150 MPa and
  - ensuring that the tendons or steel reinforcement are no further apart than 500 mm centre to centre.

If the tensile stress exceeds 0.25 Öf'c then RAPT will print a warning to the report file indicating the span, strip and location where it occurred. It will also print a warning if the stress change in i) above exceeds 150 MPa.

The user must ensure that the design is in accordance with the code rules.

In the secondary direction it is normal to provide prestress such that the P/A (for internal environment slabs) is approximately 1.0 MPa. This, according to AS3600 clause 9.4.3.4, will provide minor to moderate crack control for shrinkage and temperature effects. For the above calculation take the width of slab as the distance between edges of effective beam flanges.

T.7.6.2 Prestressed Beams

The code provisions for prestressed beams are very similar to the provisions for prestressed slabs. If the tensile stress under short-term service loads exceeds 0.25 Öf'c then one must provide reinforcement or bonded tendons near the tensile face (clause 8.6.2 AS3600). Further to this requirement the designer must comply with either a) or b) below.
"

- limit the maximum tensile stress under the short-term service loads to 0.6 Öf'c or
- complying with both the following
    - limiting the stress change (in the reinforcement or tendon closest to the tensile face) from decompression to short-term service loading to 200 MPa and
    - ensuring that the tendons or steel reinforcement are no further apart than 200 mm centre to centre.

"

If the tensile stress exceeds 0.25 Öf'c then RAPT will print a warning to the report file indicating the span, strip and location where it occurred. It will also print a warning if the stress change in i) above exceeds 200 MPa.

The user must ensure that the design is in accordance with the code rules.

T.7.6.3 Reinforced Slabs

RAPT treats reinforced slabs identically to prestressed slabs. The users attention is drawn to clause 9.4.1 of AS3600 which indicates that cracking due to flexure is deemed to be controlled if the centre-to-centre spacing of bars in each direction does not exceed the lesser of 2.5 D or 500 mm.

If a higher level of crack control is required, RAPT allows the user to specify a limit to the tensile stress in the reinforcement. (Set in Input Screen F9)

T.7.6.4 Reinforced Beams

Cracking will be deemed to comply to AS3600 if both of the following requirements are satisfied;

a) The centre-to-centre spacing of bars near the tension face of the beam shall not exceed 200 mm.
b) The distance from the side or soffit of a beam to the centre of the nearest longitudinal bar shall be not greater than 100 mm.

For the purpose of a) and b) above, a bar having a diameter less than half the diameter of the largest bar in the cross-section shall be ignored.

If a higher level of crack control is required, RAPT allows the user to specify a limit to the tensile stress in the reinforcement. (Set in Input Screen F9)

T.7.7 Deflections

RAPT uses the same procedure to calculate deflections for all code types. Users can specify the load cases used by RAPT for this calculation. Below is a detailed explanation of how RAPT calculates deflections.

T.7.7.1 Summary of Method

The deflected shape of a structure is related to the curvature of the structure. In calculating the deflections, RAPT calculates the curvature at each node location and then sums the first moment of area of these curvatures about the desired point to obtain the deflections.

In summary the method used is as follows

1. Perform the frame analysis for the full width panel based on gross inertias.
2. Proportion bending moments laterally to column and middle strips. The proportion varies at every point along each span. The bending moments are based on the combinations defined by the user in Input screen F8. See below for more information.
3. Calculate the curvatures at each nodal point. (Detailed Reinforcement) RAPT calculates 4 curvatures which, combined, define the 3 deflection cases. Note that the deflections will include any effects due to tension or compression reinforcement entered by the user in Input Screen F9.
4. For internal and end spans calculate the deflection of the right hand support, B, with respect to the tangent at the left hand support, A, termed $d_{B|A}$. This is equal to the area of the curvature diagram between these two points. The right hand support cannot deflect, hence, using small angle theory we can say that the rotation at joint A is given by

   $$\theta_A = \frac{\delta_{B/A}}{L}$$ (ie arc = radius x enclosed angle)

   For cantilevers the rotation at the support is found from the first internal span.
5. Calculate the deflection, $d_{x/A}$ off this tangent at all intermediate points, x, in the span. This is equal to the 1st moment of area of the curvature diagram between A and x, about point x.
6. The actual deflection, d_x, at point x is therefore equal to
   $d_x = q_A \times L_x - d_{x/A}$
   where
   L_x = the distance from support A to point x

Note that this method will produce different joint rotation values on either side of the same support as the rotation in each span is calculated using the curvatures in that span with no regard to adjacent spans. However the shape of the deflected structure indicates that this discrepancy is small.

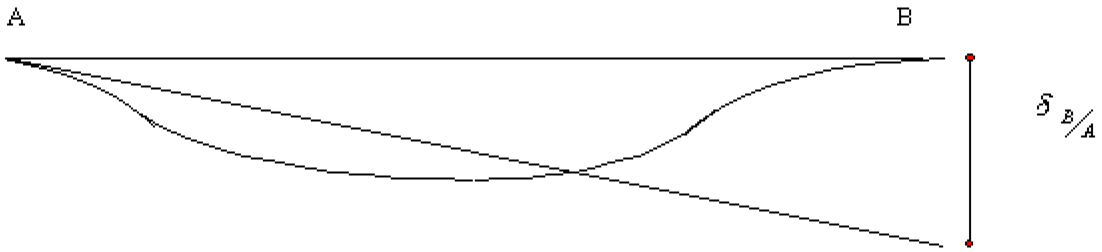


Figure T.7.7.1

T.7.7.2 Loading Cases

Users define the deflection loading cases in Input Screen F8 under the Combinations Menu. The 3 cases are

(a) Short Term Load	SW + SDL + y_s LL + PW
(b) Permanent LOAD (with Long Term LL)	SW + SDL + y_L LL + PW
(c) Initial Load (Short term load before incremental deflection)	SW + SDL + PW

As can be seen in section T.7.3, these cases are not independent of each other. The three cases must be consistant. In the default cases, we are investigating the effects using a variation in the live load between initial loading, short-term loading and long-term or permanent loading.

T.7.7.3 Curvature Conditions

The curvatures used by RAPT for calculating deflections are calculated in Detailed Flexural Reinforcement using the four following load and modulus combinations:

1. Short Term Load (A) with Short Term Concrete Modulus
2. Permanent Load (B) with Short Term Concrete Modulus
3. Permanent Load (B) with Long Term Concrete Modulus plus shrinkage curvature.
4. Initial Load (C) with Short Term Concrete Modulus

In each of these cases curvature is calculated as $\frac{strain}{kd}$ where the strain and kd are calculated from a strain compatibility calculation which determines the strain condition in the section under the applied moment for each case. In case (iii) the shrinkage curvature is added to the curvature calculated from the strain compatibility results.

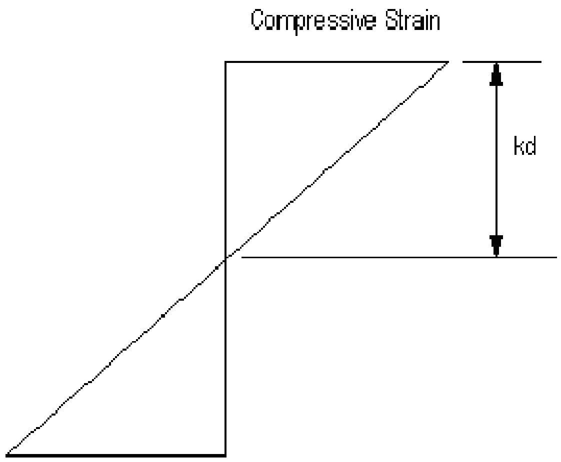


Figure T.7.7.2 - Curvature Calculation

T.7.7.4 Deflection Cases

RAPT calculates the deflections for each of the four cases shown above. The final deflections reported by RAPT are calculated based on combinations of these deflections. RAPT gives 3 deflection case results. These are

- Maximum short-term deflection
  The deflection that occurs when the short term service loading is applied prior to any long-term creep and shrinkage effects. Deflection for the curvatures is calculated from combination (from T.7.7.3)
  1
- Incremental deflection
  The deflection that will occur after the addition of the superimposed dead loads. This deflection is that component of the deflection that will result in cracking of brittle partitions. It is the difference between the deflection of the structure upon completion of the construction of the brittle partitions (DL + PW) and the deflection after all long-term effects are accounted for assuming that the member is un-propped at the time of the addition of the partitions. It is assumed that, as the partitions are constructed, they incorporate the immediate deflection that they induce. Deflections are calculated from the sum of combinations (from T.7.7.3)
  3 + (1 - 2) - 4
- Long-term deflection
  The deflection that will be measured (from zero datum) after all long-term effects. Note that it is not the summation of the initial and incremental deflections. Deflections are calculated from the sum of combinations (from T.7.7.3). Note that this deflection would be the critical case for partitions built on a member that is still propped at the time of construction of the partitions.
  3 + (1 - 2)

T.7.7.5 Creep Curvature

In order to account for the creep affects in the concrete, in the calculation of deflections, a reduced or effective modulus is used. Faber's Effective Modulus Method (ref 44) gives

$$E_e(t,\tau) = \frac{E_{cm}(\tau)}{1+\phi(t,\tau)}$$

where
E_{cm} = Mean Concrete Modulus. See Section T.7.3.3
f (t,t) = Creep factor. See Section T.7.3.4

The Effective Modulus Method gives good answers when a constant loading history has been applied to the structure as it depends only on the current stress at the creep strain time t. No consideration is given for the variations of load after time t_0. Thus in the case of an increasing stress history, the Effective Modulus method will overestimate the creep, and, for a decreasing stress history, the Effective Modulus Method will underestimate the creep.

In order to account for the aging effect due to creep of concrete, an Age - Adjusted Effective Modulus can be used for the Long term modulus. For Short Term conditions we use E_{cm} as defined in section T.7.3.3. The Age - Adjusted Effective Modulus can be defined as

$$\overline{E}_e(t,\tau_0) = \frac{E_{cm}(\tau_0)}{1+\chi(t,\tau_0)\phi(t,\tau_0)\times\dfrac{E_{cm}(\tau_0)}{E_{cm}(t_{28})}}$$

where
$E_{cm}(t_0)$ = Short Term Modulus at initial loading time. See T.7.3.3.

$$\chi(t,\tau_0) = \frac{\tau_0^{\;0.5}}{\left(1+\tau_0^{\;0.5}\right)}$$ Aging co-efficient.

f (t,t_0) = Creep factor. See 7.7.3.4 for information on creep factor.
t_0 = Initial time of loading
t = Time at which deflections are calculated.

Proposed by Trost (ref 41) and developed by Bazant (ref 42) as defined in CEB-FIP Model Code 1990 (ref 44)

The Aging co-efficient is defaulted to the value determined from the above formula, which was taken from the CEB FIP Model code. In RAPT we assume that a constant load is applied to the structure for it's entire life. Users can modify this value in Input Screen F1. The usual range for this value is 0.6 - 1. The value input reflects the loading history of the structure. The "Rate of Creep Method" as detailed in (ref 43) allows users to calculate a value for this constant appropriate to the loading of a structure.

T.7.7.6 Shrinkage Curvature

The shrinkage curvature is calculated from

$$\frac{1}{r_{cs}} = \frac{\varepsilon_{cs} \times n \times S_s}{I_t}$$

where
e_{cs} = Shrinkage strain modified for shape, environment and time effects. See T.7.3.5

$$= \frac{E_s}{E_e}$$

n = Modular ratio
E_s = Steel Modulus
E_e = Effective Concrete Modulus
S_s = the first moment of area of the reinforcement about the centroid of the cracked or gross section, whichever is appropriate
I_t = The transformed Inertia based on Èe.

T.7.7.7 Tension Stiffening

RAPT allows users the choice of 3 methods to account for tension stiffening effect.

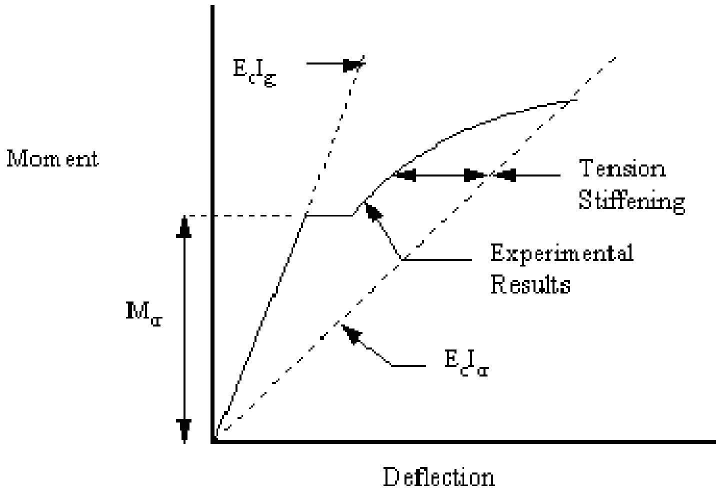


Figure T.7.7.3 - Moment vs Deflection Curve (Reinforced Member)

Bransons Formula

To define an Ieff which assimilates the slope of the deflection curve after cracking. The formula for the determination of the effective inertia was originally developed by Branson and was based on a series of tests on reinforced concrete beams.

$$I_{ef} = \left(\frac{M_{cr}}{M_s}\right)^4 \times \left(I_g - I_{cr}\right) + I_{cr}$$

To extrapolate the method to apply to partially prestressed sections one must modify the effective inertia equation to bring the curvature of the section back to the zero datum. We should therefore subtract the total prestress bending moment. Readers are referred to the work by Branson and Trost (ref 32). RAPT uses the decompression bending moment as a conservative lower bound for the total prestress bending moment as given by

$$I_{ef} = \left(\frac{M_{cr} - Pe}{M_s - Pe}\right)^4 \times \left(I_g - I_{cr}\right) + I_{cr}$$

where $M_s = M_{DL} + M_{LL} + M_{SEC}$

Eurocode II Approach

The Eurocode II approach is given in EurocodeII Section A4.3. The following formula accounts for tension stiffening by giving an effective curvature.

$$\kappa_{eff} = \xi \times \kappa_{cr} + (1 - \xi)\kappa_{uncr}$$

$$\xi = 1 - \beta_1 \beta_2 \left(\frac{M_{cr}}{M_{app}}\right)^2$$

where
b_1 = 1 for deformed reinforcement bars
= 0.5 for plain reinforcement bars
b_2 = 0.5 for sustained loads.
Note: If the section is uncracked then x = 0, thus the Effective Curvature is set to the uncracked curvature.

Modified Concrete Tension Modulus

The Modified Concrete Tension Modulus method is based on the method specified in BS8110, Part2 Clause 3.6 and SABS 0100. This method modifies the Youngs modulus of the concrete in tension to allow for cracking, then uses a strain compatibility analysis to calculate the equilibrium condition, thus giving a kd and strain to calculate the

curvature for the applied moment. This is the preferred method by the writers as it more accurately describes what is happening in the concrete cross section.

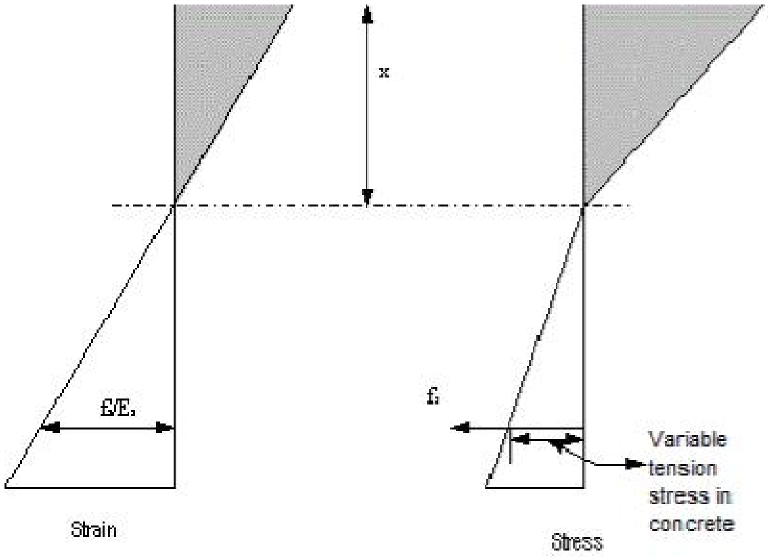


Figure T7.7.4 - Modified Concrete Tension Modulus.

For BS8110 and SABS 0100, RAPT limits the concrete stress at the bottom layer of tension reinforcement to 1N/mm2 as defined in these codes. A section is defined as uncracked before this limit is reached and cracked once the limit is reached. For all other codes, RAPT uses the curvatures based on gross section properties (ie uncracked) until the applied moment causes the concrete tensile stress at the outer face to exceed the allowable concrete tensile stress as specified by the user in Input Screen F9 (cracked condition). When the applied stress in the concrete is greater than the tensile strength of the concrete, the tension stress in the concrete at the bottom tension steel layer is limited to 1N/mm2thus assuming that the tension stiffening effect after cracking occurs is equivalent to a tensile strength of the concrete of 1N/mm2.

T.7.7.8 Two Way Systems

For beam runs RAPT assumes a stiff element over the column length. This also applies to two way beams, but only to the beam strip. For two-way systems the bending moment used in the calculation of deflections, for each of the nodal points, is factored to account for the percentage taken to the column strip at that point (see section T.4).

Two Way Slab Deflection
For two-way slab systems the maximum deflection is at the geometrical centre of a slab panel bounded by four columns. This deflection will be the summation of the average column strip deflection of two adjacent panels in one direction plus the deflection in the middle strip of the design strip running in the transverse direction. RAPT considers only one design strip at a time hence the user must manually combine these two deflection components.

RAPT calculates 3 deflection cases for 2 way slabs

1. column strip based on curvatures of the column strip
2. middle strip based on curvatures of the middle strip
3. average based on curvatures of the panel width.

T.7.7.9 Deflection Limits

RAPT gives deflections and a span/deflection ratio. Each code gives guidelines on deflections as follows

- AS3600 clause 9.33 c) and Table 2.4.2
- BS8110 clause 3.2 Part II
- ACI318 clause 9.5
- Eurocode2 clause 4.4.3.1

AS3600 provides the following table for deflection limits.

LIMITS FOR CALCULATED DEFLECTION OF BEAMS AND SLABS

(a) Where provision is made to minimise cracking (commonly taken as when brickwork joints are spaced less than or equal to 5000 mm)

Type of	Deflection to	limitation for	
member	be considered	Spans	Cantilevers
All members	total deflection	1/250	1/125
members supporting masonry partitions	the deflection which occurs after the addition or attachment of the partitions.	1/500	1/250

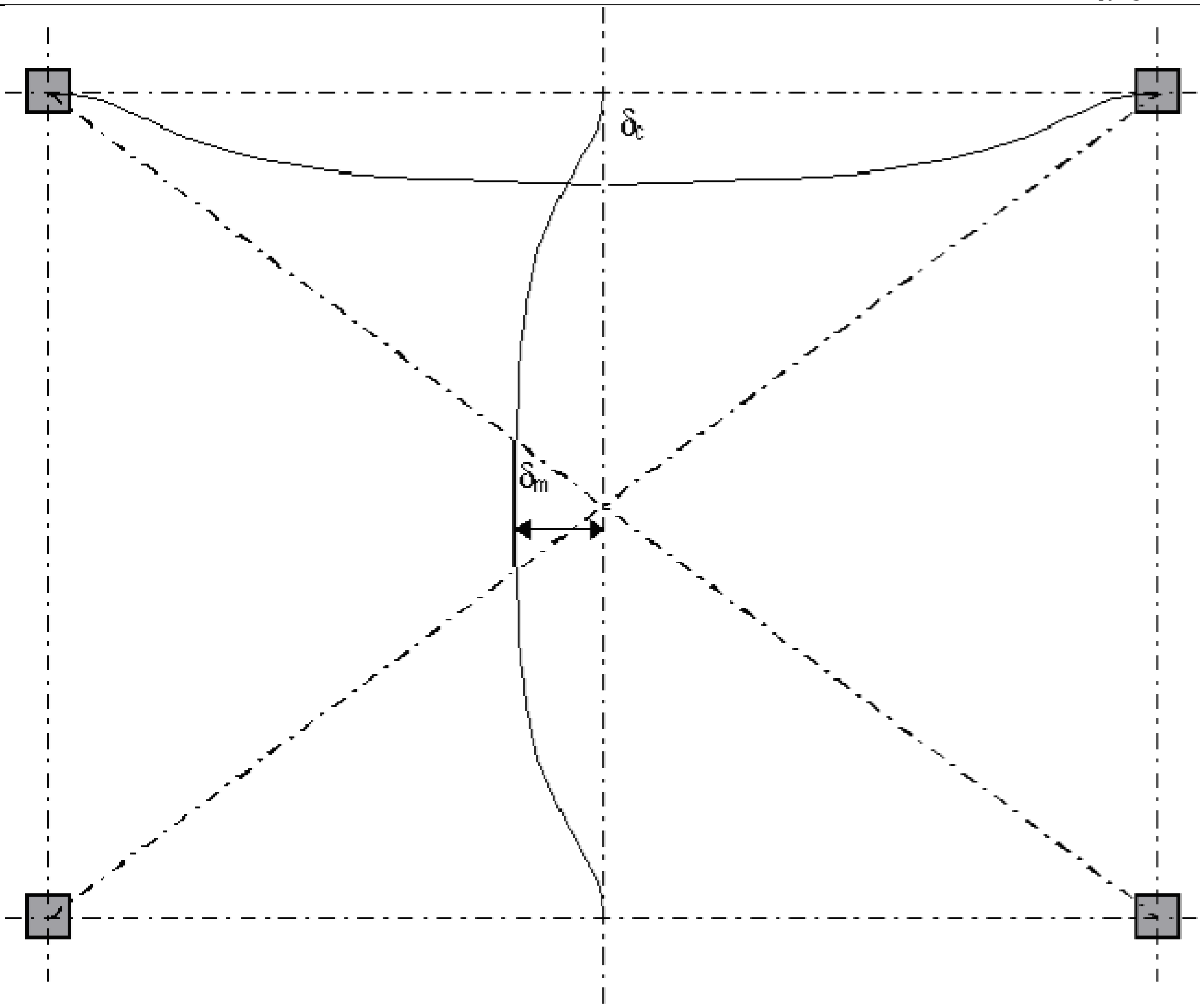
(b) Where provision is NOT made to minimise cracking

Type of	Deflection to	limitation for	
member	be considered	Spans	Cantilevers
members supporting masonry partitions	the deflection which occurs after the addition or attachment of the partitions.	1/1000	1/500

According to AS3600 TABLE 2.4.2 NOTE 1, deflection limits given are to apply to the theoretical deflection of the line diagram representing the idealised frame for flat slabs. This implies that it is the deflection of the full panel width prior to the proportioning of the load to the column and the middle strips. The deflection of the full panel line diagram would therefore be a weighted average deflection of the column and middle strip deflections. Applying the deflection limits to the separate strips should provide conservative results as the column strip deflection would be higher than this "average" deflection.

It is suggested that one should limit the total deflection to an absolute limit of 20 - 30 mm. This is proposed as the limit where one may notice the deflection by visual inspection.

The designer must decide if the average deflection, or the column strip deflection or the total panel deflection is critical in each case. It should be remembered that in many cases with brickwork on slabs, the walls are placed on column lines. In this case the column strip deflection would be more critical.



$d_{diagonal} = d_c + d_m$

$$\delta_{ave} = \frac{\delta_c + \delta_m}{2}$$

Figure T.7.7.5

T.8 Transfer

RAPT carries out a serviceability check for the transfer condition. The loading used to check the transfer case is

SW + PT

RAPT calculates the top and bottom fibre stresses based on gross section properties. If the stress at transfer is greater than this limit, RAPT will carry out a cracked section analysis and add reinforcement to satisfy maximum allowable stresses in the reinforcement. because of this, for transfer members where stage stressing is being used, it is important that the designer investigate all stages in the construction with the relevant loads applied. The final stage will have all tendons stressed and the structure weight at the final stressing stage included in the transfer load case and the full loading of the building included in the ultimate and service cases.

RAPT will inform the user of the state of the member. The users attention is drawn to

- AS3600 Clause 8.1.4.2 (RAPT complies with the second paragraph of this clause only)
- BS8110 and CP65 Clause 4.3.5
- ACI318 Clause 18.4
- Eurocode2 Clause 4.2.3.5.4
- SABS0100 Clause 5.3.2.3
- CP2004 clause 12.3.5
- IS1343 clause 22.8.2

In essence there is no difference between checking at transfer and checking at working, except that at working an ultimate design has been carried out first and there will usually be sufficient steel detailed to ensure compliance at working.

Comments on AS3600
When the only loads acting are self-weight and transfer prestress then the usual worst case would be 0.8 times self-weight and 1.15 times the prestress. The authors are not at this stage clear as to what the code is referring to in this clause for the prestress bending moment. If one compares the ultimate check at transfer to the ultimate check at full working then presumably the secondary prestress at transfer is the bending moment to be used however the factor of 1.15 then appears inappropriate as generally the secondary moment are acting in the opposite sign as applied moments.

The method suggested by Faulkes (ref 25) is that M^*_t be calculated using an ultimate factor of 0.8. The internal equilibrium of the section is then found in terms of an unknown ultimate prestress force, P_u. This force is then multiplied by the capacity reduction factor of 0.6 to give the design prestress force to cause failure at transfer, fP_u. This is then compared to an ultimate applied prestress force, P*, using an ultimate factor of 1.15.

This method is limited in that it is only easily applicable to cross-sections which have only one layer of prestressing.

A Helpful hint

If users wish they could perform a cracked section analysis on the transfer moments by using the Cross Section Design Utility to perform a cracked section analysis on the transfer moments by inputting the transfer moments as service moments.

T.9 Ultimate Shear

RAPT checks cross-sections for both Punching and Beam Shear. Each is discussed in detail below.

T.9.1 Punching Shear

RAPT designs punching shear in accordance with the chosen code. For specific information on each code see

- AS3600 Clause 9.2
- BS8110 and CP65 Clause 3.7.6, 3.7.7
- ACI318 Clause 11.12
- Eurocode2 Section 4.3.4
- SABS 0100 Clause 4.6
- CP2004 clause 6.1.5.6
- IS456 and IS1343 clause 31.6

The program will automatically check punching shear at every column and save the results to the output file. For edge columns where no cantilever is present the user must define the extent of the edge of the slab with respect to the column centre-line.[see Input F2 [Spans]] The frame is input with all span lengths and widths centred along the centre-line of the frame. The critical shear perimeter around an edge column would, in the case where no cantilever was input, be under-estimated if modelled to the column centre-line. For each column RAPT calculates the critical perimeter. RAPT will calculate this perimeter for both rectangular and circular columns. In all cases RAPT will calculate the shortest failure perimeter.

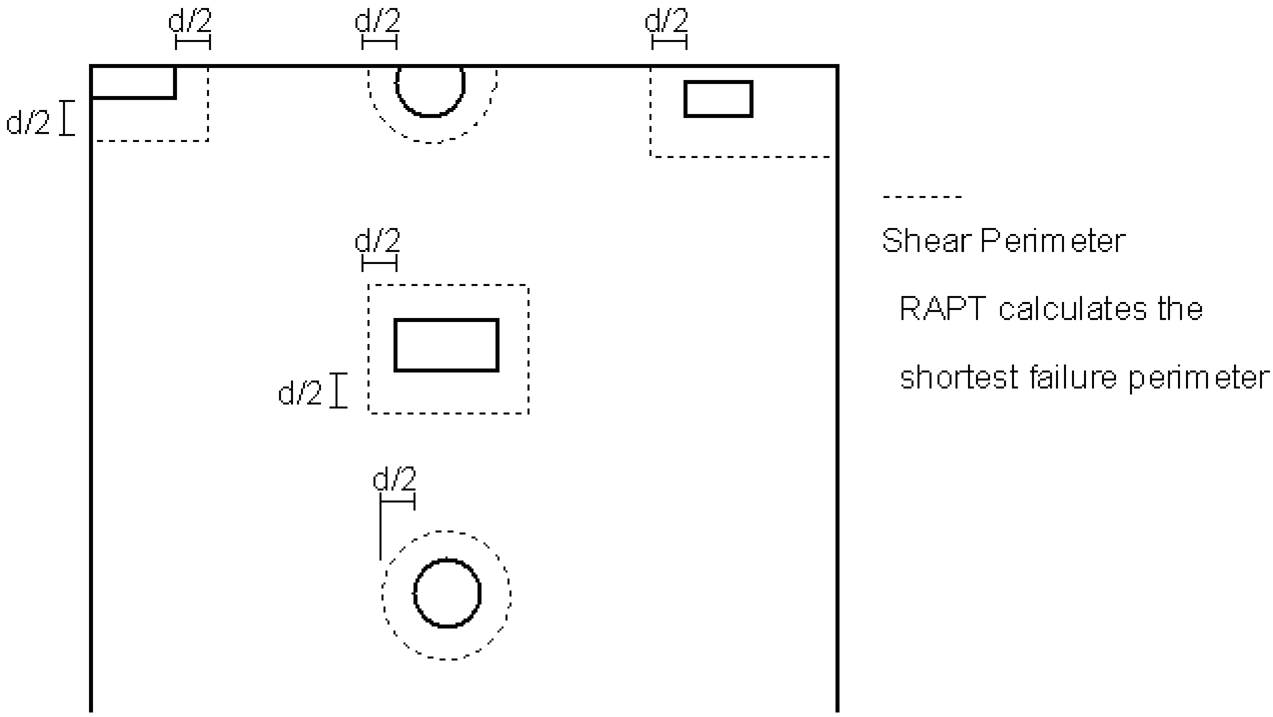


Figure T.9.1 - Shear Perimeters

The effective depth is the average value either side of the column in the column strip or one-way width and the average prestress force divided by the average concrete area is taken for the design direction only. Both of the above should more correctly be for the average of the two directions at the column.

For AS3600, stirrups will automatically be nominated in the transverse direction in order to satisfy punching shear requirements. Note that this may occur even if no transverse beam is present. The centre-line of the ties is taken as 55 mm from all faces of beams and 35 mm from all faces of slabs. The term y1 is the longest dimension of the closed tie.

For the other codes (BS8110, SABS0100 and Eurocode II, CP3004, IS456, IS1343, ACI318) RAPT calculates shear head reinforcement, not torsion strip reinforcement.

The output provides the design parameters that are necessary to check the RAPT calculations. The design punching shear force at the column is the sum of the total panel span reactions on either side of the column, not simply the load on the tributary area.
To design punching shear RAPT considers 3 load cases

1. Shear Case f1 DL + f3 LL + f6/7 Psec [see section 4.10]
2. The maximum envelope from the ultimate case.
3. The minimum envelope from the ultimate case.

Thus RAPT will take patterning effects into account from the ultimate combinations cases. The critical case for the maximum punching shear force will generally occur for full live loading on all spans.

Note:

1. RAPT will present the results for the case which gives the worst punching shear conditions.
2. These results will include a moment transferred to the column. RAPT does not check that the column can adequately sustain the moment transferred to it or check that the slab has adequate reinforcement to transfer the moment to the column.

Secondary moments are included in the column moments as follows.

The calculation of secondary moments in each span is based on the average prestress force in that span (see Section T.10). Strictly speaking the secondary moment at the column centre-line should be calculated using the prestress force at the column centre-line, not the average force within the span length. Similarly secondary moments at any location within the span length should be calculated using the force at that location within the span. However, as the forces imposed by the prestress (distributed loads, moments and point loads) have been calculated using the average force within the span length (as opposed to having a continually varying distributed load from the prestress), the secondary moments should then be calculated using the same average force.

The slab-beam secondary moments on either side of column are calculated on the average force in those spans and the difference between the two moments is then distributed to the columns above and below the floor according to the relative stiffness of the columns. There will be a discrepancy in the secondary moment calculated at the column if the calculation is based on the prestress force at the column centre-line as opposed to the average force as discussed above.

For one-way systems the total punching shear force and bending moment transferred to each column is factored up in the ratio of the one-way width being run to the transverse average column spacing, l_2.

T.9.1.1 Punching Shear to AS3600

The formulation of the rules for punching shear owe a great deal to Rangan and Hall et al and the reader is referred to their text (ref 35) for a more complete description. Below is a condensed form of the background to the code rules. Note that the capacity reduction factor for shear is taken as 0.70.

Punching shear strength when no beams present
As in previous codes it is assumed that the shear and moment transferred to the column through a section of slab surrounding the column of width $d_o/2$. The column has dimensions c_1 and c_2 in directions parallel and transverse to the direction that moments are being considered respectively. This thereby defines the failure surface when the column "punches" through the slab as a truncated cone. For an internal column the cone has, in plan, a dimension parallel to the design direction equal to a (where $a = c_1 + d_o$) and a transverse dimension b (where $b = c_2 + d_o$). The total plan length of the cone is referred to as the critical perimeter, u. Note that do is the average depth of the slab averaged around the critical perimeter. RAPT can only run one direction at a time hence it will take do as being the average either side of the column in the design direction only.

> d_o = the distance from the extreme compressive fibre of the concrete to the centroid of the outermost layer of tensile reinforcement or tendons but not less than 0.8D.

The previous Australian concrete code (and the current ACI code) method was to determine the percentage of the load taken to the four faces of the column in accordance with the proportion of the polar moment of inertia of that face to the total polar moment of inertia of the critical section.

Rangan et al have proposed that

1. The direct shear force, V*, on the column is distributed to each of the four faces of the critical perimeter in the direct ratio of the length of a side to the total length of the perimeter.
   Therefore
   $V_1^* = (a/u)\, V^*$
   = the direct shear force on a side face
   a = dimension of the critical shear perimeter measured parallel to the direction of M^*_v
   u = length of the critical shear perimeter
   The ultimate direct shear strength, V_{uo}, of the critical perimeter is given by

   $$f_{cv} = 0.17\left(1 + \frac{2}{\beta_h}\right)\sqrt{f'c}$$

   $$\leq 0.34\sqrt{f'c}$$

   $$V_{uo} = ud_o\left(f_{cv} + 0.3\sigma_{cp}\right)$$

   where
   b_h = the ratio of the longest dimension of the column, y, to the shorter dimension, x, measured at right angles
   s_{cp} = the P/A due to prestress at the column based on the full panel width
   The direct shear capacity of one side face of the critical perimeter is therefore

   $$V_{uc} = \left(\frac{a}{u}\right)V_{uo}$$

2. The greater proportion of the bending moment transferred to the column, M_v^*, is taken by the faces of the critical perimeter which are parallel to the design direction as 0.4 M_v^* on each side face. This is transferred to the column in a torsional mode on the side faces.
   The ultimate strength of the side face torsional member, T_{uc}, which is the depth of the slab and of width equal to the parallel column width, a, is given by

   $$T_{uc} = \frac{3.2ad_oV_{uo}}{u}$$

Combining the shear and torsion in a linear relationship the column is considered satisfactory to resist the loading if the following is satisfied

$$\frac{V_1}{V_{uc}} + \frac{T_1}{T_{uc}} \leq 1.0$$

Substituting in the formulae above we find that the maximum bending moment that may be transferred simultaneously with the direct shear force is given by

$$\mathbf{M_v}^* = 8d_o\left(\frac{a}{u}\right)\left(\phi V_{uo} - V^*\right)$$

or alternatively we may say that the ultimate design shear capacity is

$$\phi V_u = \frac{\phi V_{uo}}{1+\left[\dfrac{M_v^*/a}{8V^* d_o/u}\right]}$$

which for the case of no bending moment being transferred to the column reduces to

$$\phi V_{uo} = \phi \mathbf{u}\mathbf{d}_o\left(f_{cv} + 0.3\sigma_{cp}\right)$$

If V* > fV$_u$ then one must provide closed ties in the transverse direction. The minimum requirement for the ties is that they have a minimum cross-sectional area per spacing $(A_{sw}/s)_{min}$ as given below and that they are spaced no further apart than the minimum of the depth of the slab; or 300 mm.

$$\frac{\left[A_{sw}\right]_{min}}{s} = \frac{0.2y_1}{f_{sy}}$$

where
y_1 = the longest overall dimension of the closed tie
$f_{sy.f}$ = the yield stress of the tie
A_{sw} = the cross-sectional area of the bar forming a closed tie (1 leg)
s = the spacing of the ties

The shear strength of the connection including the capacity of the torsional member which has minimum closed ties is then given by

$$V_{u.min} = \frac{1.2V_{uo}}{\left[1+\dfrac{M_{v*}}{2V^*a^2/u}\right]}$$

If the applied ultimate loading, V*, exceeds the design strength with minimum ties, f$V_{u.min}$, then one must provide closed ties which are greater than the minimum given above.

Setting the design ultimate capacity fV_u to the applied ultimate shear force, V*

$$\frac{\left(A_{sw}\right)_{req}}{s} = \frac{\left(A_{sw}\right)_{min}}{s} \times \left(\frac{V^*}{\phi V_{u.min}}\right)^2$$

The closed ties have a width equal to a.

Punching shear strength when beams are present
Whenever a beam is subjected to torsion, ie. when M_v* < > 0 then one must provide at least the minimum closed ties as described above. However the strength of the connection is then given by

$$V_{u.min} = \frac{D_b/D_s 1.2V_{uo}}{\left[1+\dfrac{M_v^*}{2V^*b_w a/u}\right]}$$

where
D_b = the depth of the transverse beam
D_s = the depth of the slab. Taken as the average drop panel depth if drop panels are present
b_w = the width of the beam web

Similarly if V* > f$V_{u.min}$ then ties greater than the minimum are required as given above.

T.9.1.2 Punching Shear to BS8110 & SABS 0100

RAPT applies the punching shear force to the critical perimeter. This is defined as being 1.5d from the loaded area. (see BS8110 figure 3.17). When calculating the punching shear requirements of prestressed slabs, RAPT converts the Prestressing tendons into an equivalent area of reinforcement. The following equations are then used to check punching shear. [see BS8110 section 3.7.6 & 7]

$$V_{eff} = V_t\left(1 + \frac{1.5M_t}{V_t x}\right)$$ shear stress for slab/column internal connections

$$V_{eff} = V_t\left(1.25 + \frac{1.5M_t}{V_t x}\right)$$ shear stress for other slab/column connections

where
x = the length of the side of the perimeter considered parallel to the axis of bending
M_t = the design moment transmitted from the slab to the column at the connection
V_t = the design shear transferred to the column

V_{eff} is then checked against code limits to evaluate the punching shear condition. The following limits are applied by the code

$$V_{max} < 0.8\sqrt{f_{cu}} \text{ or } 5 \text{ N/mm}^2$$ whichever is less

where

$$V_{max} = \frac{V}{u_o d}$$ Note: Vmax is based on the loaded area

u_o = effective length of the perimeter which touches a loaded area
d = effective depth of slab

The design shear strength for the shear stress failure zone is given by

$$v = \frac{V}{ud}$$

where
u = effective length of the perimeter of the critical zone (based on 1.5 d from column)

The shear capacity vc of the failure zone is determined by the following equations (see BS8110 table 3.9)

$$v_c = \frac{0.79\left(\frac{100A_s}{b_v d}\right)^{1/3} \times \left(\frac{400}{d}\right)^{1/4}}{\gamma_m}$$

where

$$\frac{100A_s}{b_v d}$$ should not be taken as greater than 3

$$\frac{400}{d}$$ should not be taken as less than 1

For characteristic concrete strengths greater than 25 N/mm2, v_c should be multiplied by

$$\left(\frac{f_{cu}}{25}\right)^{1/3}$$ but the value of f_{cu} should not be taken as greater than 40.

Following the code rules, using the results calculated above, RAPT determines what shear reinforcement is required. ie

If $v < v_c$ then no shear reinforcement is required.
If $v > v_c$ and the slab is over 200mm deep then the following A_{sv} can be added

$$A_{sv} = \frac{(v - v_c)ud}{0.87 f_{yv}}$$

where
$(v - v_c)$ should not be taken as less than 0.4 N/mm2
f_{yv} is the characteristic strength of the shear reinforcement
A_{sv} is the area of shear reinforcement. This reinforcement should be distributed around the column in accordance with the code.

T.9.1.3 Punching Shear to ACI318

Section 11.12 of ACI318 defines the requirements for punching shear. The critical perimeter is defined to occur at d/2 from the column face.

d = depth to the centroid of the tension reinforcement but d shall not be taken as < 0.8D

The code sets out the following requirements for punching shear

V_u <= fV_n
where
$V_n = V_c + V_s$
V_c = the nominal shear strength provided by the concrete
V_s = the nominal shear strength provided by shear reinforcement.

For non prestressed slabs vc is the smallest of

$$V_c = \left(\frac{1}{6} + \frac{1}{3\beta_c}\right)\sqrt{f'_c}\,b_o d$$

(i)
where
b_c is the ratio of the long side to short side of the column

$$V_c = \left(\frac{\alpha_s d}{12 b_o} + \frac{1}{6}\right)\sqrt{f'_c}\,b_o d$$

(ii)
where
a_s is 40 for interior columns
is 30 for edge columns
is 20 for corner columns

$$V_c = \frac{\sqrt{f'_c}\,b_o d}{3}$$

(iii)

For two-way prestressed slabs

$$V_c = \left(\beta_p \sqrt{f'_c} + 0.3 f_{pc}\right) b_o d + V_p$$

where

b_p is the smaller of 0.291667 or $\left(\frac{\alpha_s d}{12 b_o} + 0.125\right)$
a_s is 40 for interior columns
is 30 for edge columns
is 20 for corner columns
b_o is the perimeter of critical section
f_{pc} is P/A
V_p is the vertical component of the effective prestress forces crossing the critical section. RAPT assumes 0 for this figure.

This formula can only be used if
(i) no portion of the column cross section shall be closer to a discontinuous edge than 4 times the slab thickness
(ii) f'_c shall not be taken greater than 35MPa
(iii) f_{pc} shall not be less than 0.86 MPa, nor greater than 3.5 MPa
If these conditions are not met, than RAPT uses Vc based on the non prestressed section formulae.

When moment is transferred to the column, some of it is transferred by flexure and the rest is transferred by eccentricity of shear about the centroid of the critical section. Thus gvMu is transferred by shear. gv is defined as

$$\gamma_v = 1 - \frac{1}{1 + \left(\frac{2}{3}\right)\sqrt{\frac{b_1}{b_2}}}$$

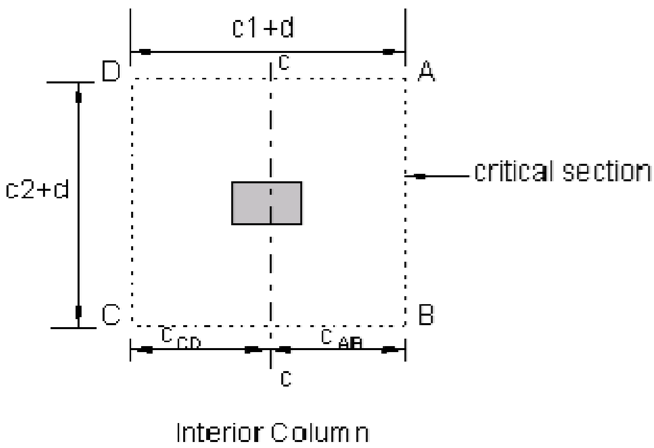
where
b_1 = width of the critical section measured in the direction of the span for which moments are determined
b_2 = width of the critical section measured perpendicular to b_1

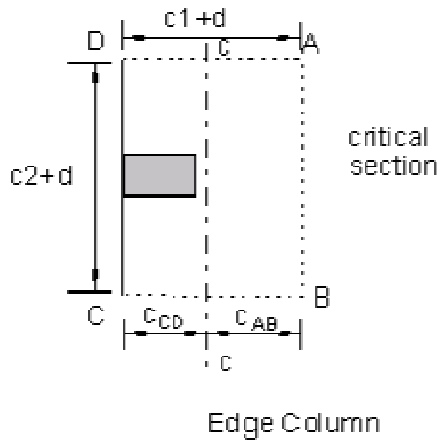
The shear stress resulting from moment transfer by eccentricity of shear shall be assumed to vary linearly about the centroid of the critical sections. The maximum factored shear stress can be calculated from

$$v_{u(AB)} = \frac{V_u}{A_c} + \frac{\gamma_v M_u c_{AB}}{J_c}$$

$$v_{u(CD)} = \frac{V_u}{A_c} - \frac{\gamma_v M_u c_{CD}}{J_c}$$

where
A_c = area of concrete of assumed critical area
J_c = property of assumed critical section analogous to polar moment of inertia.





RAPT has the ability to calculate J_c for rectangular or circular columns. J_c will be calculated for any portion of a circular column at an edge.

RAPT does not calculate a value for V_s. Users can use the V_c value calculated by RAPT to decide if extra shear reinforcement is required but the calculation of the reinforcement required is left to the designer.

To help the designer in this calculation RAPT also calculates V_{nmax}. The following formula is used

$$V_{n\,max} = \frac{\sqrt{f'_c}\, b_o d}{2}$$

T.9.1.4 Punching Shear to EUROCODE2

Section 4.3.4 of the code defines the punching shear requirements used by RAPT.

The critical perimeter is defined at a point 1.5 d from the loaded area. The code has limits defining acceptable loaded areas. To comply with the code the

1. Shape must be within the following limits (where d is the effective depth of the slab)
   circular, with diameter < 3.5 d
   rectangular, with perimeter < 11 d and the ratio of length to breadth < 2
   any shape, the limiting dimensions being fixed by analogy with the shape mentioned above.
2. Loaded area can not have its critical perimeter intersect with another critical perimeter from another load.

If the loaded area does not comply with the conditions above (ie for wall or rectangular column supports) then the following diagram will define the critical perimeter.

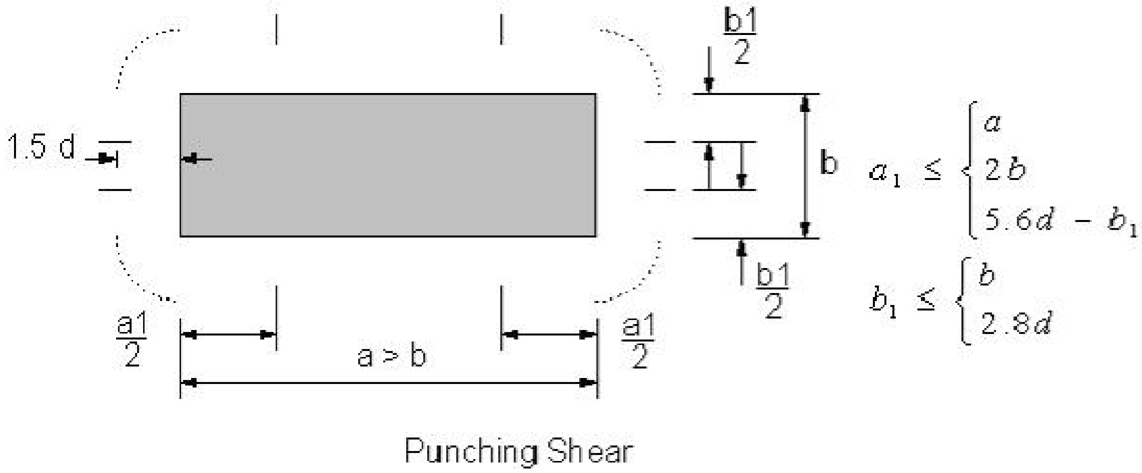


Figure T.9.3

Moment Transfer

The total design shear force is calculated from

$$V_{sd} = \frac{V_{sd} \times \beta}{u}$$

where
Vsd = total design shear force
u = the perimeter of the critical section
b = a coefficient which takes account of the effects of eccentricity of loading. (Moment transfer to column)
 = 1.50 for corner columns
 = 1.40 for edge columns
 = 1.15 for internal columns

The shear resistance of slabs are calculated as follows

(i) Non Prestressed slab

$$v_{Rd1} = \tau_{Rd} k(1.2 + 40 p_1) d$$

where
t_{Rd} varies according to concrete strength f_{ck} ie

f_{ck}*ck*	12	16	20	25	30	35	40	45	50
t_{Rd} *Rd*	0.18	0.22	0.26	0.30	0.34	0.37	0.41	0.44	0.48

Values for t_{Rd} (N/mm2) Table 4.8 Eurocode2

k = (1.6 - d) >= 1.0 (d in meters)
r1 = reinforcement ratio of the tension steel < 0.015
d = effective depth of the slab ie (depth to the centroid of the tension reinforcement but d shall not be taken as < 0.8D

(ii) For Prestressed Slabs the same equation applies but

$$\rho_1 = \rho_1 + \frac{\sigma_{cpo}}{f_{yd}} \rangle 0.015$$

where

$$\sigma_{cpo} = \frac{N_{pd}}{A_c}$$ (ie P/A)

f_{yd} = design yield stress of the reinforcement

The maximum design shear resistance for a slab containing shear reinforcement is (per unit length of critical perimeter)

v_{Rd2} = 1.6 x v_{Rd1}

The design shear resistance for a slab containing shear reinforcement is (per unit length of critical perimeter)

$$v_{Rd3} = v_{Rd1} + \frac{\sum A_{sw} \times f_{yd}}{u}$$

RAPT calculates the minimum reinforcement based on

$A_{sv\ min}$ = r w x (A_{crit} - A_{load})

where
A_{crit} = the area within the critical perimeter
A_{load} = the area within the loaded area.
r w = reinforcement ratio taken from clause 5.4.2.2 and 5.4.3.3
RAPT assumes that (A_{crit} - A_{load}) = 1.5 x u

T.9.2 Beam Shear

RAPT designs Beam Shear in accordance with each code. For specific information see

- AS3600 Clause 8.2 (and section T.9.2.2 to 4 of this manual)
- BS8110 and CP65 Clause 3.4.5 and 4.3.8
- ACI318 Chapter 11
- Eurocode2 Section 4.3.2
- SABS 0100 Clause 4.3.4 & 5.3.4
- CP2004 clause 6.1.2.5 and 12.3.8
- IS456 and IS1343 clause 40 and 22.4

It should be noted that RAPT assumes that all loads are applied at the top and centre of the members. If this is not the case then the designer should ensure that their design accounts for this condition. ie Add hanging shear reinforcement to transfer the load to the top and torsion reinforcement to transfer the load to the centre of the member.

> Note: ACI318 and AS3600 limit the effective depth to 0.8D for beam shear calculations. Thus in theory it is possible to design the shear requirements for a beam assuming that tensile reinforcement exists at 0.8D when there is no tension reinforcement at all. Users should acknowledge this and consider placing some tension reinforcement in the tension face when these parameters occur.

Critical Sections
For all code types RAPT places the critical shear section at 0.8D from the face of the columns.

T.9.2.1 RAPT Summary

The summary output details

1. the required spacing of 3 bar sizes (relevant bar specifications used for each code)
2. when minimum STIRRUP spacing is controlling
3. when no stirrups are required
4. the shear steel area divided by the shear steel spacing, Asv/s, required for strength ie before spacings are checked.
5. the web shear width used, bv

The width of the ducts for the shear width, b_v, are assumed to be

- For duct diameters of 18 to 26 mm, RAPT assumes a duct width b_w of
  70 mm if strand dia <= 13 mm and
  < 5 strands in duct
  70 mm if strand dia >14.5 mm and
  < 4 strands in duct
  for all other cases RAPT assumes a duct width bw of 90 mm.
- For duct diameters > 27 mm RAPT defaults bw to the duct diameter.

RAPT conservatively assumes all ducts detailed are within the web width.

RAPT calculates the required shear reinforcement based on the maximum reinforcement in the tension face for each moment region as calculated from "Detail Flexural Reinforcement".

A warning will also be printed if a section fails in beam shear due to crushing of the compressive strut of the truss model, ie if $V^* > fV_{u.max}$.
Other decisions made by the program are

- If V* < 1 kNm then P_v is added to the sections design capacity in shear.
- If P/A = 0 at a section then principal tensile checks are not carried out. In this case RAPT will default the principal tensile stress to a large number ie 9999.

The decompression bending moment is calculated taking due account of secondary bending moments.

If the principal tensile failure check does not need to be made at the flange / web junction ie. the neutral axis of the gross-section does not lie within the flange, then the principal tensile force is still calculated at the flange / web junction but is not compared with the force calculated at the centroid.

T.9.2.2 Principal Tensile Strength

There are essentially two forms of shear failures: one in which the cracking of the concrete starts in the web as a result of high principal tension and another in which vertical flexural cracks occur first and gradually develop into inclined shear cracks. AS3600, BS8110 and ACI318 check each of these failure modes. The Eurocode uses other methods, described later in this chapter.

The failure is due to high tensile stresses exceeding the concrete's inherent tensile strength, which is approximately 10% of its compressive strength, acting on a plane inclined (at approximately 45 degrees) to the axis of the member. The plane on which the failure occurs is known as the principal tensile plane and its orientation may be found from elastic strength of materials theory for a homogeneous, uncracked section. Mohrs circle of stresses is a convenient tool for this calculation. Note that in the creation of Mohrs circle the axial compressive force from prestress and the axial and shear stresses from the applied loading should be considered.

It is worth noting that as the foregoing relies on the section being uncracked then, at high overload, the strength limit state, a higher factor of safety may be appropriate. However it is usually found that principal tensile failure occurs at regions of low bending moment and hence the section may be essentially uncracked at these regions.

Both the current and previous Australian codes, American, British and South African codes do not require principal tensile stress checks to be made for sections which are not prestressed.

The maximum principal tensile stress does not necessarily occur at the centroidal axis, where the maximum vertical shearing stress occurs. For I-sections the junction of the flange and the web is frequently the critical location. ACI 318 (ref 23) and AS1481 both state that the check at the web / flange intersection need only be made when the neutral axis lies within the flange itself.

Principal tensile failure is sudden and is very rarely preceded by a noticeable crack forming in the web of the member.

The calculation of the applied ultimate shear force, V_t required to induce the allowable principal tensile stress proceeds as follows.

We define V_t as the shear force we need to apply to the cross-section in order to produce, in combination with the applied service loads, a principal tensile stress equal to the tensile strength of the concrete. The assumption is made that the applied bending moment does not increase with an increase in the applied shear force. This simplification is not strictly correct. However the shear force we calculate, V_t, when compared to the applied ultimate shear force, V*, will yield a measure of the factor of safety for the loads as applied to the section.

If we set the principal tensile stress, s1 equal to the code maximum allowable stress (tensile positive)

(i)	AS3600 clause 8.2.7.2	s1 = 0.33 Öf'$_c$	(T.9.9)
(ii)	BS8110 and CP65 clause 4.3.8.4	s1 = 0.24 Öf'$_{cu}$	
(iii)	ACI318 R11.4.2	s1 = 0.33 Öf'$_c$	
(iv)	SABS 0100 clause 5.3.4.2.1	s1 = 0.23 Öf'$_{cu}$	
(v)	CP2004 clause 12.3.8.4	s1 = 0.24 Öf'$_{cu}$	
(vi)	IS1343 clause 22.4.1	s1 = 0.24 Öf'$_{ck}$	

The applied longitudinal axial stress is given by

$$\sigma = \frac{-P}{A} \pm \frac{M_s y}{I} \pm \frac{A_f}{A}$$

where
P = the total axial force from the prestress
A = the gross cross-sectional area of the concrete
y = the distance from the centroidal axis to the cut
I = the gross second moment of area
Ms = is the total applied service bending moment and therefore must include secondary bending moments, Msec, from the prestress (due to frame continuity).
= MSW + MDL + ysMLL + Mp
Mp = (S P.e) + Msec
= total prestress bending moment
= the bending moment obtained from the frame run of the continuous structure with the equivalent prestress loads applied.
Af = Axial force applied to the structure. (Compression / Tension force on members)

In all equations where the sign is optional as positive or negative one must assess whether the design action is placing compressive stress at the level of the cut (-ve) or tensile stress (+ve).

The shearing stress is given by

$$\tau = \frac{V_t Q}{It}$$

where
V_t = the shear force to cause principal tensile cracking.
Q = first moment of area above or below the cut with respect to the centroidal axis.
I = second moment of area of the total cross-section
t = shear width of the section at the cut
= b_v ie. allowance must be made for the presence of prestressing ducts crossing the shear plane.
b_v = b_w - 0.5 Sd_d AS3600 clause 8.2.6
= b_w - Sd_d BS8110 clause 4.3.8 for ungrouted tendons
= b_w - 0.667 Sd_d BS8110 clause 4.3.8 for grouted tendons
= b_w ACI318 gives no adjustment for tendons in the web width.

With reference to Mohrs circle shown in Figure T.9.4 we find that

$$\sigma_1 = \sqrt{\left(\left(\frac{\sigma}{2}\right)^2 + \tau^2\right)} + \frac{\sigma}{2}$$

(tensile +ve)

rearranging

$$\sigma_1 - \frac{\sigma}{2} = \sqrt{\left(\left(\frac{\sigma}{2}\right)^2 + \tau^2\right)}$$

$$\left(\sigma_1 - \frac{\sigma}{2}\right)^2 = \left(\frac{\sigma}{2}\right)^2 + \tau^2$$

$$\sigma_1{}^2 - \sigma_1\sigma + \frac{\sigma^2}{4} = \frac{\sigma^2}{4} + \tau^2$$

$$\tau^2 = \sigma_1{}^2 - \sigma_1\sigma$$

$$\tau = \sqrt{\left(\sigma_1{}^2 - \sigma_1\sigma\right)}$$

multiply through by s1 / s1

$$\tau = \sigma_1\sqrt{\left(1 - \frac{\sigma}{\sigma_1}\right)}$$

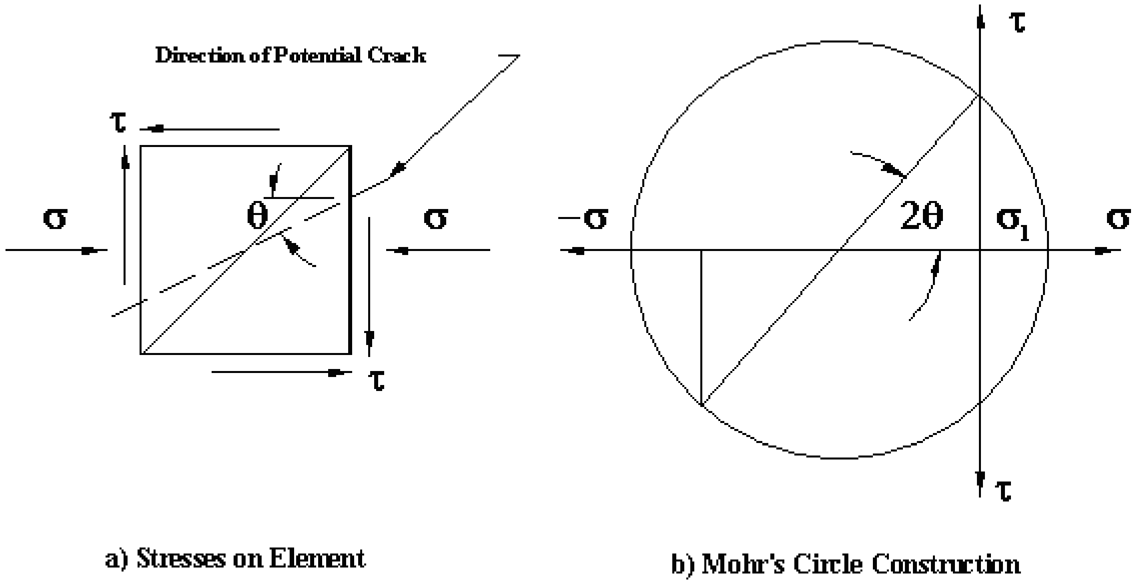


Figure T.9.4

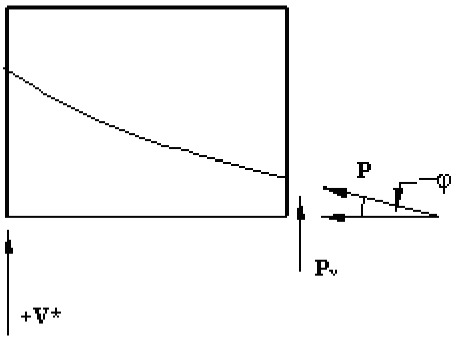


Figure T.9.5

then (remembering compression is -ve)

$$V_t = \frac{It\sigma_1}{Q}\sqrt{\left(1 \quad \frac{\sigma}{\sigma_1}\right)} \pm P_v$$

The web-shear cracking strength at the centroidal axis may be checked by

$V_t = b_v\, d_o$ [0.292 Ö(f'$_c$) + 0.33 s]

The above equation (ref 23) is based on a straight line simplification of the previous equation. This was given in a more conservative form in the draft of AS3600 at the centroidal axis as

$V_t = b_v\, d_o$ (0.25 Ö(f'$_c$) + 0.25 P/A) ± P_v

The vertical component of the prestressing force, P_v, must be included as an additional force component. P_v is calculated by multiplying the prestress force by the inclination of the tendon (in radians). In the usual case this will increase the shear capacity of the section as shown in figure T.9.5. As a general rule where the sign of V* and P_v are opposite then P_v increases the sections shear capacity.

In summary for principal tensile failure to occur

fV_t < V*

where
f is the capacity reduction factor for shear of 0.70. AS3600 table 2.3
Partial safety factors are used BS8110. Table 2.2 gives a g_m factor of 1.25
f is the capacity reduction factor for shear of 0.85. ACI318 clause 9.3.22

Equation T.9.13 is the equation used by RAPT to calculate the Principal Tensile Failure shear force. If necessary RAPT will calculate this value at the centroid and web / flange intersection. (As discussed prior). RAPT will present the results in the Detailed Shear Report. Each code uses different notation to describe the same shear force. Below is a summary of the different code notation.

- AS3600 V_{ut}
- BS8110 V_{co}
- ACI318 V_{cw}

T.9.2.3 Flexure-Shear Strength AS3600

For lightly reinforced sections, the common economical design solution, and in regions where the bending moment is large with respect to the shear force (towards centre-span) flexure-shear failure will usually precede principal tensile failure. The cracks initially form as flexural cracks on the tensile face of the section and as the load increases proceed up the section. If the section were under the action of bending alone the cracks would remain vertical. In the presence of shear however eg near supports, the cracks diverge towards the plane of principal tensile failure. Failure will eventually occur when the concrete in compression above the crack crushes.

For beams which contain stirrups this failure may be preceded by buckling of the compressive reinforcement between the stirrups. This induces spalling of the concrete above the buckle leading to total failure from the sudden loss in the compressive concrete area. When the flexural cracks begin to develop into inclined tensile cracks, sudden and violent failure may occur.

The ultimate capacity of a beam in shear is comprised of the shear forces to

(i) firstly initiate a flexural crack, Vo. This will occur under the load which will cause decompression of the extreme tensile fibre. The decompression bending moment, Mo, must therefore be calculated. For reinforced sections decompression occurs as soon as the section is loaded therefore Mo = 0. For prestressed sections Mo is influenced by the effect of the prestressing overcoming the applied loads. Also for continuous prestressed beams the effects of secondary actions must also be accounted for.

The longitudinal stress at the extreme tensile fibre (at decompression) is found by

$$\sigma = \frac{-P}{A} + \frac{\sum (P \times e)y}{I} - \frac{M_o y}{I}$$

= 0

hence

$$M_o = Z_{tens}\left(\frac{P}{A} + \frac{\sum (P \times e)y}{I}\right)$$

Due account must be taken here of any secondary bending moments if they exist.

$$M_o = Z_{tens}\left(\frac{P}{A} + \frac{\sum (P \times e)y}{I}\right) \pm M_{sec}$$

As

$$M_{sec} = M_p - \sum (P \times e)$$

$$M_o = Z_{tens}\left(\frac{P}{A}\right) \pm M_p$$

If the assumption is made that the ratio of the shear force and the bending moment remains constant for all loadings states and for simple and continuous spans we may state that the shear force at the decompression moment will be

$$V_o = \frac{M_o}{(M^* / V^*)}$$

(ii) then to cause the crack to develop into an inclined tensile crack extending a distance do. The formulation of the empirical equation for this capacity is based on extensive experimental work considering the many variables which effect the shear strength. For normal design this formula reduces to

$$V_{uc} = \beta_1 \times \beta_2 \times b_v \times d_o \left[\left(\frac{A_{st} + A_p}{b_v d_o} \right) f'_c \right]^{1/3}$$

V_{uc} may be considered as the shear capacity of the compressive member at the top of an equivalent concrete truss where the compressive members are comprised of concrete and the tensile members are of steel as shown in figure T.9.6 and T.9.7.

where

d_o = the greater of the depth to the centroid of the outermost layer of steel or 0.8 times the overall depth of the section.

b_1 = the larger of 1.1 and 1.6 - d_o / 2000

b_2 = 1 or

$$1 - \left(\frac{N^*}{3.5 \times A_g} \right) \geq 0$$

for members subjected to significant axial tension

$$1 + \left(\frac{N^*}{14 \times A_g} \right)$$

for members subjected to significant axial compression

N* can be applied by users in Input Screen using the Axial forces option.

d_d = the sum of the diameters of prestressing ducts crossing the web

b_v = the web shear width

= bw - 0.5 Sd_d

(iii) The vertical component of the prestressing force, P_v.

P_v = S P x q

S = the inclination of the tendon (radians)

As a general rule when the inclination of the tendon is of opposite sign to the applied ultimate shear force, V*, then P_v will add to the shear capacity of the beam.

In summary for flexural-shear failure to occur

f (V_{uc} +V_d + P_v) < V*

where f is the capacity reduction factor for shear of 0.70.

T.9.2.4 Reinforcing for Shear to AS3600

For beams where the stirrups are vertical we may consider the internal actions within the beam to be behaving in a similar manner to a truss as shown in figure T.9.6. When an inclined shear cracks form at the strength limit state as shown in figure T.9.7 the vertical tensile force in the truss model must be resisted by the vertical legs of the stirrups. For a crack inclined at an angle q the horizontally projected length of the crack will be given by

$L = d_o$ / TAN q

The number of stirrup legs crossing this projected length will be

n = L / s
= d_o / (s tanq)

If the assumption is made that all the stirrups are at yield then the tensile force resisted by the stirrups will be

$$V_{us} = \frac{A_{sv} f_{sy.f} d_o}{s \tan \theta}$$

where
A_{sv} = the area of the stirrups per spacing s
$f_{sy.f}$ = the yield stress of the stirrups

The angle of the inclined crack, q, is dependent upon the level of shear present with respect to the shear capacity of the cross-section. For lightly stressed sections this approaches 30 degrees and for heavily stressed sections 45 degrees. For intermediate percentages we may linearly interpolate between the two limits.

$$q = 30 + (45 - 30)\left[\frac{V^* - \phi V_{u.min}}{\phi V_{u.max} - \phi V_{u.min}}\right]$$

where
fV_{umin} = the design strength in shear when minimum stirrups are provided ie where the area of stirrups per spacing s equals
A_{sv} / s = 0.35 b_v / $f_{sy.f}$
f$V_{u.max}$ = the design strength in shear when the diagonal compressive strut of the truss model crushes. This is given by the following empirical formula
f$V_{u.max}$ = f (0.2 f'_c b_v d_o + P_v)

The total design strength in shear is then given by the summation of the shear strength of the concrete, given by the minimum of the principal tensile strength and the flexural shear strength, plus the capacity of the stirrups. Therefore for failure to occur

f(V_{uc} + V_{us}) < V*

AS3600 provides rules for the detailing of the stirrups where they are required. Note that in some situations regardless of whether the applied ultimate shear force is less than the design strength in shear, stirrups must be provided.

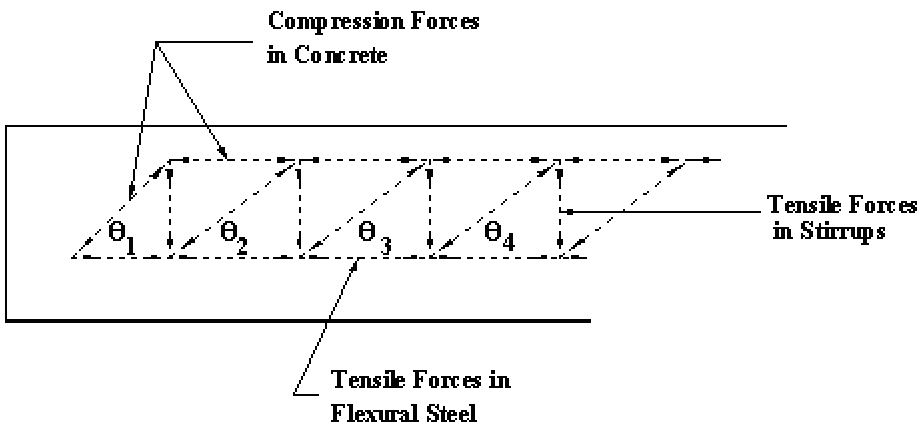


Figure T.9.6

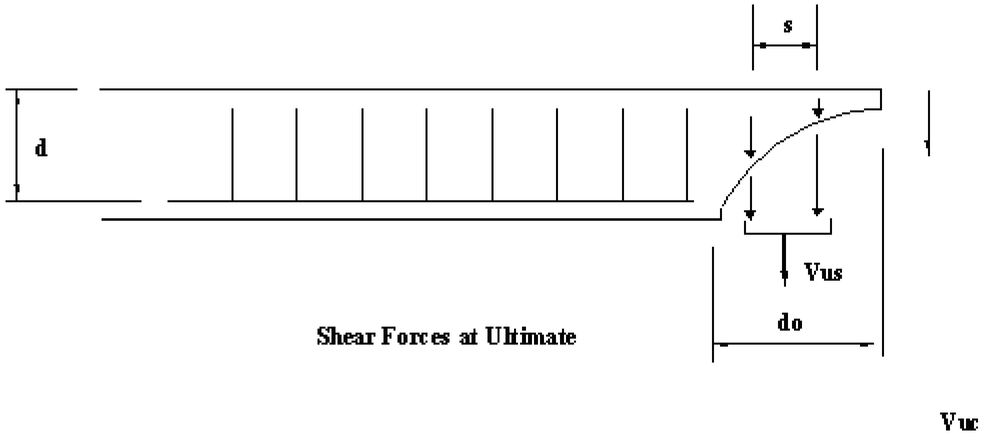


Figure T.9.7

T.9.2.5 Reinforcing for Shear to BS8110 & SABS 0100

BS8110 clause 4.3.8 details beam shear requirements. RAPT calculates the principal tensile and flexure forces as discussed in sections T.9.2.2 & 3. The following requirements are also specifically met by RAPT for BS8110 and SABS 0100. The shear strength V_c is calculated as the lesser of V_{co} and V_{cr}.

Principal Tensile Failure VCO
See T.9.2.2

Flexure - Shear Failure V_{CR}
BS8110 gives the following formulae to calculate this value

$$V_{cr} = \left(1 - \frac{0.55 f_{pe}}{f_{pu}}\right) v_c b_v d + M_o \frac{V}{M}$$

where
d = effective depth (distance from the extreme compression fibre to the centroid of the steel area (Aps + As) in the tension zone).

$$V_{cr} \geq \quad 0.1 b_v d \sqrt{f_{cu}}$$

fpe = effective prestress force in tendons
fpu = characteristic strength of a prestressing tendon
vc = design concrete strength obtained from table 3.9 of code, in which As is replaced by (As + Aps) where As and Aps are the respective areas of reinforcement and tendons in the tension zone.

$$= \frac{0.79}{\gamma_m}\left(\frac{100 A_s}{b_v d}\right)^{1/3}\left(\frac{400}{d}\right)^{1/4}$$

where

$\frac{100 A_s}{b_v d}$ should not be taken as greater than 3

$\frac{400}{d}$ should not be taken as less than 1

For characteristic concrete strengths greater than 25 N/mm2, the values in table 3.9 may be multiplied by (f_{cu}/25)1/3. The value of f_{cu} should not be taken as greater than 40.

M_o = moment necessary to produce zero stress in the concrete at the extreme tension fibre. In this calculation only 0.8 of the stress due to prestress should be taken into account.
V = Applied shear force
M = Applied moment force at that section.

Axial Compression Forces

If axial compression forces are included in the design, RAPT will modify the v_c value as follows:

$$v'_c = v_c + 0.75 \times \frac{N}{A_c} \times \frac{V \times d}{M} \leq 0.8\sqrt{f_{cu}}$$

or 5 N/mm2 whichever is the lesser.

where $\frac{V \times d}{M} < 1$
See BS8110 Clause 3.4.5.12
SABS 0100 Clause 4.3.4.4

BS8110 and SABS 0100 do not give any indication of what to do for Axial Tension forces for reinforced members or axial Tension / Compression forces for prestressed members. RAPT uses the AS3600 formula for these cases (AS3600 Clause 8.2.7.1 or see section T.9.2.3 in this manual).

Minimum Design Shear Stress V_{UMIN}

RAPT specifies a V_{umin} value which is calculated as

$V_{umin} = V_c + 0.4\ b_v\ d$.

This is the minimum shear value possible with a minimum area of ties.

Maximum design shear stress V_{UMAX}

The following formula is given

For BS8110 V_{umax} = 0.8 Öf_{cu} or 5 N/mm2 whichever is the lesser. (This includes an allowance of 1.25 for gm.)
For SABS 0100 V_{umax} = 0.75 Öf_{cu} or 4.75 N/mm2 whichever is the lesser. (This includes an allowance of 1.25 for g_m.)

Shear reinforcement
RAPT assumes that the lesser of V_{co} and V_{cr} is equal to V_c. RAPT will calculate the required shear ties depending on the size of the applied shear in comparison to the shear capacity of the member. RAPT gives the user two forms of results

1. $\frac{A_{sv}}{s}$ (Area of reinforcement divided by spacing of ties)
2. A tie size and spacing using the above ratio for different sized tie bars.

$\frac{A_{sv}}{s}$ is calculated according to the following rules

- If $V < 0.5\ V_c$ then no reinforcement is calculated
- If $V < V_c + 0.4\ b_v\ d$
  then $$\frac{A_{sv}}{s} = \frac{0.4 b_v}{0.87 f_{yv}}$$
- If $V > V_c + 0.4\ b_v\ d$
  then $$\frac{A_{sv}}{s} = \frac{V - V_c}{0.87 f_{yv} d_t}$$
  where
  d_t = depth from the extreme compression fibre either to the longitudinal bars or to the centroid of the tendons, whichever is greater.

For reinforcement detailing rules the user should refer to the code.

T.9.2.6 Reinforcing for Shear to ACI318

RAPT designs shear reinforcement fully to ACI318. Below is a summary of the formulae used by RAPT. The shear strength V_c is calculated as the lesser of V_{cw} and Vci.

Principal tensile failure V_{CW}
See T.9.2.2 for theory and calculation method.

Flexure - shear failure VCI
For non prestressed concrete members

$$V_c = \left(0.158\sqrt{f'_c} + 17.241\frac{A_s}{b_w d} \times \frac{V_u d}{M_u}\right) b_w d$$

where
V_c is not greater than 0.291Öf'$_c$ b_w d

$\frac{V_u d}{M_u}$ shall not be taken greater than 1.0
M_u is the factored moment occurring simultaneously with V_u at the section considered.

For prestressed members

$$V_{ci} = \frac{0.6\sqrt{f'_c}\, b_w d}{12} + V_d + \frac{V_i M_{cr}}{M_{\max}}$$

where
Vci need not be taken less than 1.7 Öf'$_c$ b_w d

$$M_{cr} = \left(\frac{I}{y_t}\right)\left(0.5\sqrt{f'_c} + f_{pe} - f_d\right)$$

y_t = distance from centroidal axis of gross section, neglecting reinforcement, to extreme fibre in tension
I = moment of inertia of section resisting externally applied factored loads.
f_{pe} = compressive stress in concrete due to effective prestress forces (after losses) at extreme fibre of section where tensile stress is caused by externally applied loads.
f_d = stress due to unfactored dead load, at extreme fibre of section where tensile stress is caused by externally applied loads.
V_i = factored shear force at section due to externally applied loads occurring simultaneously with Mmax.
M_{max}= maximum factored moment at section due to externally applied loads
V_d = shear force at section due to unfactored dead load.

Axial Compression / Tension
For Axial Compression, the above equations are modified so that M_m is substituted for M_u and V_{ud}/M_u is not limited to 1.0.

$$M_m = M_u - N_u \times \left(\frac{4 \times h - d}{8}\right)$$

h = member thickness
but Vc shall not be greater than

$$V_c = \frac{3.5}{12} \times \sqrt{f'_c} \times b_w \times d \times \sqrt{1 + \frac{N_u \times 145}{500 \times A_g}}$$

See ACI318 Clause 11.3.2.2

For Axial Tension, the Flexure Shear formula should be modified as follows

$$V_c = \frac{2}{12} \times \left(1 - \frac{N_u \times 145}{500 \times A_g}\right) \times \sqrt{f'_c} \times b_w \times d$$

See ACI318 Clause 11.3.2.3

ACI318 does not give any indication of what to do for Axial Tension or Compression forces for prestressed members. RAPT uses the AS3600 formulas in these cases. AS3600 Clause 8.2.7.1 or see section T.9.2.3 in this manual.

Minimum Design Shear Stress V_{UMIN}

V_{umin} is minimum design capacity taking into account minimum ties.

$fV_{umin} = fV_c + fV_s$

where

$$V_s = \frac{50 \times b_w d}{145}$$

Maximum Design Shear Stress V_{UMAX}

$fV_{umax} = fV_c + fV_s$

where
V_s = 0.667Öf'$_c$ b_w d

Shear Reinforcement
If Vu exceeds fVc then ties must be provided as follows

$$\frac{A_v}{s} = \frac{(V_u - \phi V_c)}{\phi f_y d}$$

For spacings and detailing users should refer to the code for specifics.

T.9.2.7 Reinforcing for Shear - EUROCODE2

According to the Eurocode2 code designers need to comply with three design shear resistances. V_{Sd} is the design value of the applied shear force at the ultimate limit state.

1. VRd1 = the design shear resistance of the member without shear reinforcement.
2. VRd2 = the maximum design shear force that can be carried without crushing of the notional compressive struts.
3. VRd3 = the design shear force that can be carried by a member with shear reinforcement.

RAPT does not calculate

1. Shear between web and flanges [see Eurocode2 clause 4.3.2.5]
2. The tensile force in the longitudinal reinforcement. [see Eurocode2 clause 4.3.2.4.4(5)]

It is up to the designer to check that their design is within the code limits. Designers should also ensure that they detail the shear reinforcement in accordance with the code rules.

V_{RD1}

If V_{Sd} <= V_{Rd1} then no shear reinforcement is required [see Eurocode2 clause 4.3.2.3]

$$V_{Rd1} = \left(\tau_{Rd} k\left(1.2 + 40\rho_1\right) + 0.15\sigma_{cp}\right) b_w d$$

where

t_{Rd} = basic design shear strength

= 0.25 $f_{ctk0.05}$ / g_c

$f_{ctk0.05}$ = 0.7 f_{ctm}

f_{ctm} = mean value of the tensile strength

g_c = should be taken as 1.5

k = 1 for members where more than 50% of the bottom reinforcement is curtailed otherwise

k = 1.6 - d but not less than 1

r_1 = A_{s1} / (b_wd) but not greater than 0.02

where

A_{s1} = the area of tension reinforcement extending not less than d + $l_{b.net}$ beyond the section considered.

$l_{b.net}$ = required anchorage length

b_w = minimum width of the section over the effective depth

s_{cp} = N_{Sd}/A_c (Accounts for prestress and axial forces in a member)

N_{sd} = Longitudinal force in section due to loading or prestressing.

V_{RD2} (V_{UMAX})

In calculating V_{Rd2} RAPT assumes that all ties are vertically aligned. The maximum web crushing value can be calculated from [see Eurocode2 clause 4.3.2.3 (3)]

V_{Rd2} = 0.5 n f_{cd} b_w 0.9 d

where

n = 0.7 - f_{ck} / 200 but not less than 0.5

f_{ck} = Characteristic compressive cylinder strength of concrete at 28 days

f_{cd} = Design value of concrete cylinder compressive strength

V_{RD3}

The Eurocode2 gives designers two methods of calculation

1. Standard Method
2. Variable strut inclination method

RAPT uses the Standard method in which

V_{RD2} = V_{cd} + V_{wd}

where

V_{cd} = the contribution of the concrete and is equal to VRD1.

V_{wd} = A_{sw} / S x 0.9 x d x f_{ywd}

V_{wd} = the contribution of vertical shear reinforcement

f_{ywd} = the design yield strength of the shear reinforcement

Axial Compression

If a member is subjected to an axial compressive force, then Vrd2 should be reduced as follows

$$V_{Rd2.red} = 1.67 \times V_{Rd2} \times \left(1 - \frac{\sigma_{cp.eff}}{f_{cd}}\right) \leq V_{Rd2}$$

$$\sigma_{cp.eff} = \frac{\left(N_{Sd} - \frac{f_{yk} \times A_{s2}}{\gamma_s}\right)}{A_c}$$

where
N_{sd} = the design axial force
A_{s2} = the area of reinforcement in the compression zone at the ultimate limit state
f_{yk} = the yield strength of the compression steel <= 400 N/mm2
A_c = the total area of the concrete cross section
See Eurocode2 Clause 4.3.2.1(4)

Prestressed members

When checking shear of a prestressed member RAPT also takes into account the prestressing effects.

$V_{Sd} = V_{od} + V_{pd}$
where
V_{od} = the design shear force in the section
V_{pd} = donates the force component of the inclined prestressed tendons, parallel to Vod. Vpd is taken as positive in the same direction as Vod.

Shear Reinforcement
To calculate the required area of shear reinforcement, RAPT rearranges V_{Rd3} as follows

$$\frac{A_{sw}}{s} = \frac{V_{Sd} - V_{Rd1}}{0.9 \times d \times f_{ywd}}$$

Note: At least 50% of V_{Sd} shall be resisted by vertical stirrups.

Minimum Reinforcement
For minimum reinforcement the Eurocode2 specifies that [see Eurocode2 clause 5.4.2.2 (5)]

$$\frac{A_{sw}}{s} = \frac{\rho_w}{b_w}$$

where
r_w = the shear reinforcement ratio with minimum values as shown below

Concrete Classes	Steel Classes		
	S220	S400	S500
C12/15 and C20/25	0.0016	0.0009	0.0007
C25/30 to C35/45	0.0024	0.0013	0.0011
C40/50 to C50/60	0.0030	0.0016	0.0013

T.10 Secondary Moments

Usually the action of prestress on continuous slab or beam systems will produce what are known as secondary reactions.

The reactions, sometimes referred to as hyperstatic reactions, are the forces induced into the system to maintain equilibrium. Hence we can see that the forces are real and must be considered in the design.

Consider the simply supported beam as shown in Figure 7.10.1. The tendon drape of 317.9 - 83 = 234.9 mm will balance 8.4 kN/m if we assume an average effective force of 447.1 kN.

Taking a cut at mid span and considering the free body to the left, we find the internal moment from the eccentricity of the tendon, the primary moment Mp, has a value of

$$M_p = P.e$$
$$= 447.1 \times (317.9 - 83)E\text{-}3$$
$$= 105 \text{ kNm}$$

The equivalent applied loading (induced from the change in angle of the tendon) will result in a bending moment, the total moment Mt, at mid span of

$$Mt = 8.4 \times 10^2 / 8$$
$$= 105 \text{ kNm}$$

Therefore we can see that equilibrium is satisfied for a simply supported member and there are no secondary forces. The same will apply for cantilevers.

If the same beam were now made continuous over two spans and taking the average effective forces of 447.1 kN and 429.5 kN for span one and two respectively and the drapes as shown in Figure 7.10.2 a).

At the centre support if we take first the free body to the left and use the average force in span one we find

$$M_p = [(447.1 + 429.5) / 2] \times (317.9 - 512)E\text{-}3$$
$$= 85.07 \text{ kNm}$$

By some method of analysis (bending moment distribution or a frame analysis) we find that the total bending moments induced from the equivalent load due to the tendon curvature will be as shown in Figure 7.10.2 b) giving a total bending moment at the centre support of M_t = 105 kNm.

Notice the discrepancy between the total moment, M_t, and the primary moment, M_p, at the cut, this is the termed secondary moment, M_s.

$$M_s = M_t - M_p$$
$$= 105 - 85.07$$
$$= 19.93 \text{ kNm}$$

which is the moment at the section required to maintain equilibrium. The secondary moment has been induced due to the continuity of the system which has resulted in the development of hyperstatic reactions. Therefore we may calculate the value of the reaction from the value of the bending moment that it has induced by considering the free body of the span, Therefore

$$R = M_s / 10$$
$$= 1.99 \text{ kN}$$

The reactions at either ends of the span will be equal and opposite. For an internal span the reaction may be calculated as the difference between the two end secondary moments divided by the length.

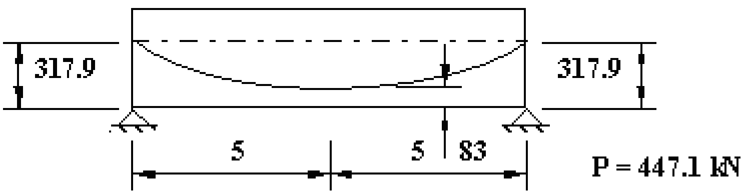


Figure T.10.1

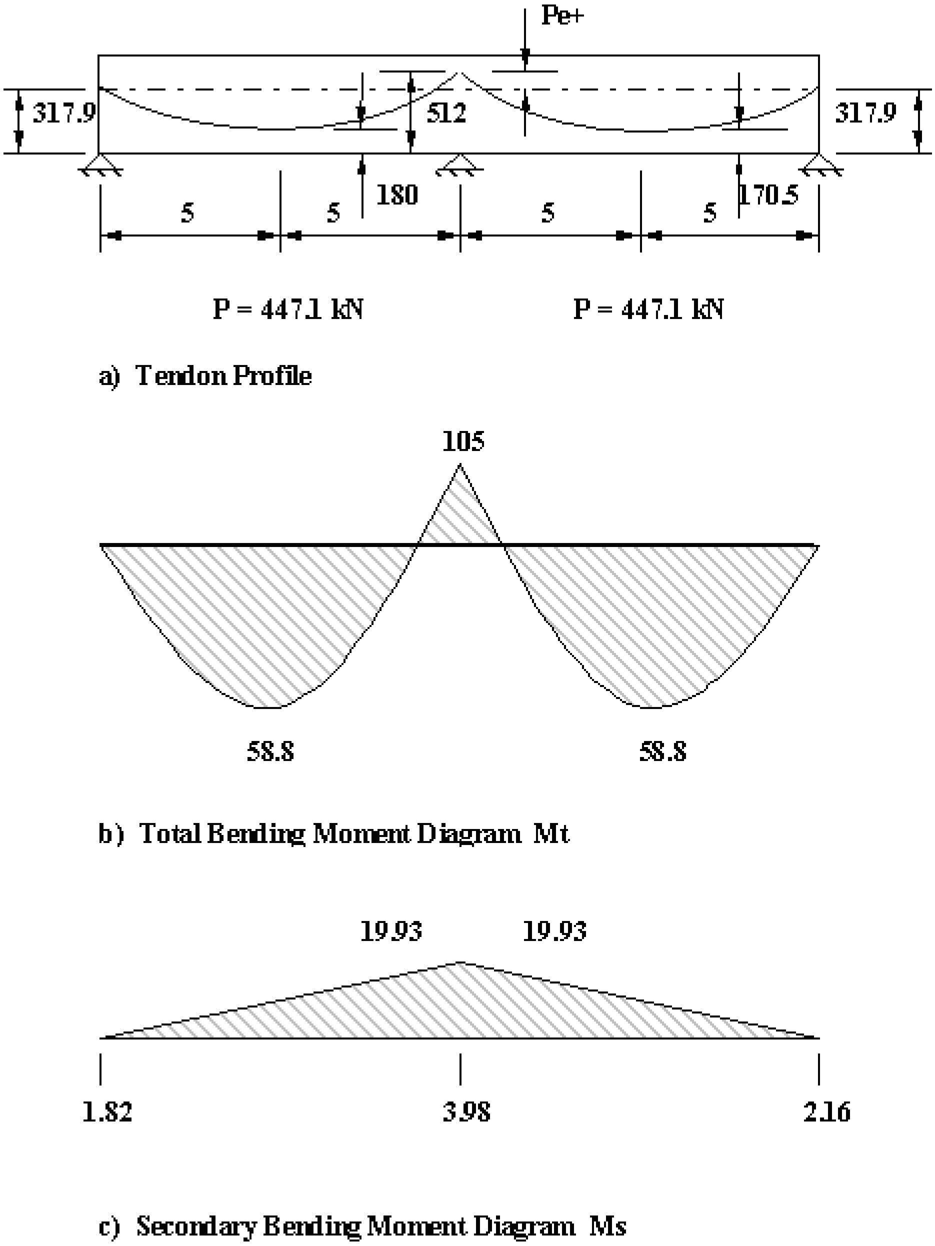


Figure T.10.2

Note that since the secondary moments are only due to the hyperstatic reactions at the supports that the secondary bending moment diagram will always be linear between support points.

It should be observed that the force at the support is considered to be the average of the forces in the spans on each side of the support. Therefore if we were to consider the free body to the right at the centre support then

M_p = 438.3 x (317.9 - 512)E-3
= 85.07 kNm
M_s = 105 - 85.07
= 19.93 kNm

When do we use the total bending moment diagram as opposed to the secondary moment diagram ?
For a thorough design we must consider the load history of the structure from the initial, uncracked state to the ultimate strength limit state. As long as a section remains uncracked we may base the calculation of stresses on gross section properties. This simple analysis obviates the need to determine the separate forces in the tendon and / or steel, though we note that the summation of the internal forces will be in equilibrium.

$$s = \frac{P}{A} \pm \frac{M}{Z}$$

The bending moment in the above equation will be due to all applied loadings (dead and live loads denoted Mg and Mq respectively), including equivalent loads from the prestress, M_t.

$$M = M_g + M_q + M_t$$

Following the onset of cracking the above analysis may not be applied as the compressive cross-sectional area and elastic modulus no longer remain constant. A more detailed analysis is required which assumes strain compatibility of the concrete and the steel bonded to the concrete. This entails the calculation of all forces within the cross-section.

We may use the same applied loadings as those applied to the uncracked cross-section however one must ensure that a "double count" of the primary bending moment does not occur. From the earlier discussion we noted that the total bending moment from the effects of prestressing (distributed loads, point loads and moments) gave M_t. The total bending moment was found to be comprised of the primary bending moment, M_p = P.e, and the secondary bending moment, M_s. Hence it would be incorrect to include the primary bending moment as a component of the internal force set within the cross-section if it were already included as an applied loading. The primary moment is more correctly considered as a force within the cross-section contributing to the internal capacity.

$$M = M_g + M_q + M_t$$

At first glance one may query the deletion of the bending moments induced by the total prestress, for example the upward effect of the equivalent loading, in the above equation. The total prestress bending moments are implicitly included as, noted previously, the calculation of the secondary bending moments were based on the total prestress bending moment diagram. Similarly the primary bending moments have not been deleted but are included in the cross-section capacity terms.

T.11 Losses of Prestress

Losses of prestress are subdivided into those which occur up until completion of the jacking of the strands (transfer of the load to the concrete) and those which will occur over the remaining life of the structure, termed the immediate and long-term losses respectively. RAPT calculates all loses in accordance with the specified code.

For code specific information see

- AS3600 clause 6.3
- BS8110 clauses 4.7 - 4.9
- ACI318 clause 18.5 & 6
- Eurocode2 clause 4.2.3.4 and 5
- CP65 clause 4.8
- CP2004 clause 12.8
- IS1343 clause 18.5

Note: All of section T.11 has AS3600 values and formulae where appropriate throughout.

Normally the strands are stressed from one end only where the total length of the strand is less than about twenty metres. Twenty-five per cent of the jacking force is usually applied at one day following the concrete pour (concrete strength about 7 MPa) with the remaining seventy-five per cent being applied at fourteen days (concrete strength about 22 MPa).

For design purposes a diagram is usually constructed with the force in the strand as the vertical axis and the distance from the jack along the horizontal axis. As we move further from the jacking end there will be a reduction of the force in the strand due to the losses outlined below.

T.11.1 Immediate Losses

Immediate losses calculations in RAPT account for

- friction within live anchorage
- friction along the length of the duct
- drawin of the strand at the live anchorage
- elastic shortening of the concrete

Once all the losses have been determined an assessment of whether the difference in the as calculated effective force and the assumed force warrants another iteration.

The most commonly used strand is the 7-wire super strand (stress-relieved) having a nominal area of 100 mm^2, diameter 12.7 mm, minimum breaking load of 184 kN, Youngs modulus of 1.95E5 MPa and a minimum tensile strength of 1840 MPa. The strands are usually jacked to the maximum permissible by the code

- 85% of the minimum breaking load for AS3600 clause 19.3.4.6
- 80% of the minimum breaking load for BS8110 clause 4.7.1
- 80% of the minimum breaking load for ACI318 clause 18.5
- 80% of the minimum breaking load for Eurocode 2 clause 4.2.3.5.4
- 80% of the minimum breaking load for CP65 clause 4.7.1
- 80% of the minimum breaking load for CP2004 clause 12.7.1
- 80% of the minimum breaking load for IS1343 clause 18.5.1

The following notes will refer to standard or average values where appropriate. Users can default these figures to whatever they like using the Default Menus.

T.11.1.1 Friction within the Live Anchorage

As the strand is drawn through the live anchorage, (the jack) it loses some of the force at the dial gauge side of the jack due to sharp angle changes causing friction within the jack itself. The loss is dependant upon the type of jack being used but is normally taken as being a 2% loss of prestress (refer to prestress companies for actual figures).

P_{jack} = 0.85 x 184
= 156.4 kN
P_1 = 0.02 x P_{jack}
= 3.13 kN (typically per strand)
where
P_1 = the loss due to friction within the jack

This point lies on the vertical axis of the diagram at zero distance from the jack.

T.11.1.2 Friction along the Duct

As the strand is pulled through the duct the friction of the strand rubbing against the duct will reduce the effective force in the strand. The amount of the friction is dependent upon the coefficient of friction between the strand and the duct and the inherent "wobble" of the duct (a function of the type of duct) and the angular changes that we are forcing the strand to go through. All codes use a similar equation to estimate friction losses. For specific code information see

- AS3600 clause 6.4.2.3 (see below)
- BS8110 clause 4.9.3.2 and 4.9.4.2
- ACI318 clause 18.6
- Eurocode 2 clause 4.2.3.5.5 (8)
- CP65 clause 4.9.3.2 and 18.9.4.2
- CP2004 clause 12.9.3.2 and 12.9.4.2
- IS1343 clause 18.5.2.6

At any distance, Lpa from the live anchorage we can use the expression

Pa = $P_j \times e^{-\mu\left(\alpha_{tot}+\beta_p \times L_{pa}\right)}$

where

m = the co-efficient of friction between the strand and the duct. This is dependent on how rusted the strand is and the type of duct. Usually taken as 0.20. If pretensioned, then 0.15.

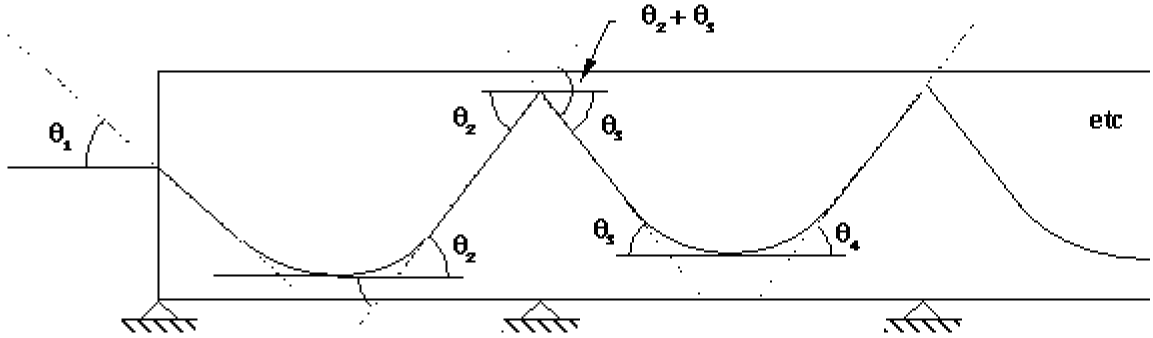
a_{tot} = the total angular change (radians) that we forced the strand to go through up till that point Lpa (see Figure T.11.1).

b_p = the wobble term of the duct normally 0.025 rad/m. This term reflects the "stiffness" or rigidity of the duct. Therefore ducts of bigger diameter and thicker walls will have a reduced bp. It is also affected by lack of care in placement of ducts or displacement or distribution of the ducts during construction. For prestressed systems b_p = 0.

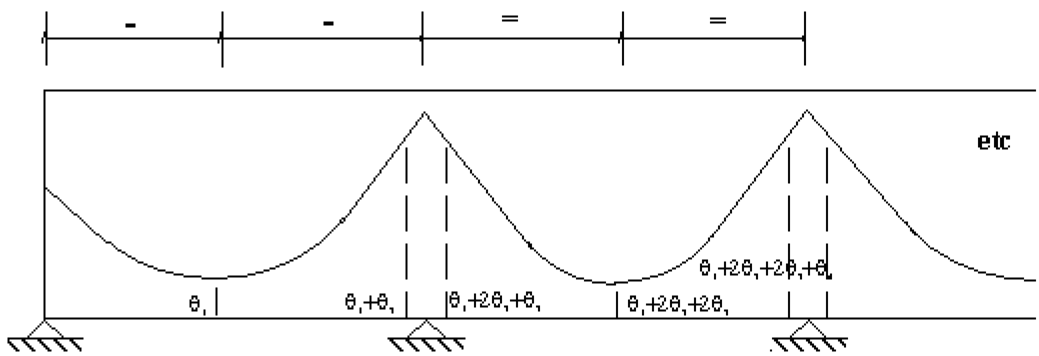
> For BS8110, CP65, CP2004, IS1343 b_p = K/m (see BS8110 clause 4.9.3.2 for recommended K value)
> For ACI318 b_p = K/m (see ACI318 clause 18.6.2.1 for recommended K value)
> For Eurocode2 b_p = K/m (see Eurocode2 clause 4.2.3.5.5 (8) for recommended K value)

P_j = the force at the jack (slab side) ie less P1 and P2 only.

L_{pa} = the distance from the jack where the loss due to friction is being found (metres).



a) Angle Changes in the Profile



b) Cumulative Angle Changes in the Profile

Figure T.11.1

We can find the losses due to friction at mid span and at the supports of every span using the approximate model described below.

For simple parabolas the equation of the simple parabolic profile from the straight line connecting the two end high points is given by

$$y = 4h\left(\frac{x}{L} - \frac{x^2}{L^2}\right)$$

$$\frac{dy}{dx} = \frac{4h}{L} - \frac{8hx}{L^2} = \theta$$

The slope of the strand at the supports, where both high points are at the same height, is therefore 4h/L (radians) and zero at mid-drape (assumed low point of the parabola) where h is the parabolic drape at mid drape length and L is the total length of the drape.

For skewed parabolas (high points at different heights) then we must add or subtract the inclination angle to the angle calculated above for the non-skewed case. Note that the total angular change within the span will remain the same as for the non-skewed case.

At internal supports, assuming we can sharply kink the strand to form the next simple parabola, we must force the strand through another angle change on either side of the support. An average total angular change is taken at the supports due to the two different total values of angular change that we have bent the strand through at this point. In fact the strand adopts a reverse parabolic profile over the supports however the net angular change is closely modelled by the above approximation.

Note that at end supports the strand is not usually bent to come out horizontal but remains at a slope.

Summarising the total angular change in

simple spans 8h/L (no reversal)
ends spans is 12h/L (h = average of high points - low point, 1 reversal)
internal spans 16h/L (h = average of high points - low point, 2 reversals)
cantilevers 4h/L (h = high point - low point)
(where L = actual span length.)

T.11.1.3 Drawin at the Anchorage

(Not for pretensioned)

When the strands are "locked off" ie when the jack is disconnected from the strands, there is a small slip (or draw-in) of the strands through the jack before the anchorage wedges "bite" into the strand and hold it firm. This will only need be considered for post-tensioned systems. The slip is normally about 6 mm (dependant upon the jacking system used). The strand possesses energy due to the strain induced by the jacking with a peak, prior to locking off, behind the anchorage. Hence the strand will try to dissipate energy to relieve the highest strained regions when the strands are locked off. In other words the strands will draw-in, slip along the length of the duct, attempting to bring the energy along the full length of the strand to a constant value. This cannot be achieved. For the strand to slide through the duct, it most overcome the friction mobilised against the walls of the duct. In effect the friction effect is the reverse of when the strands were initially jacked. It gets as far as it can before the friction effect overcomes the energy relieved by the draw-in.

The draw-in thus reduces the force in the strand over this distance. The point where the friction overcomes the draw-in may be found by firstly considering the deflection of an axially loaded member where the axial deflection, d, is given by

$$d = \frac{PL}{E_p A_p}$$

The length, L, is the length where the friction has overcome the draw-in deflection, d. The energy to be dissipated is given by the area under the force vs distance diagram of the strand.

$$\text{Energy} = \text{force x distance}$$

$$= P \times L$$

$$= \frac{\delta \times E_p \times A_p}{L} \times L$$

$$= d \times E_p \times A_p$$

where
Ep = the Youngs modulus of the strand
Ap = the area of the strand
d = the draw-in deflection

If we consider an average slope of the effective force line (the slope is due to the cumulative effect of the friction along the strand reducing the force measured at the jack) we may draw the force vs distance diagram. The line has a constant slope downwards to the right as we move away from the jacking end and at some distance, L, is met by an upward rising line representing the effective force line due to the draw-in which is progressively being overcome by friction. Both lines will have the same slope as they are both only affected by the friction. The distance L may be found by finding the area enclosed by the two lines which will give the energy required to dissipate the draw-in.

Expressing the slope of the force lines as Q in kN/m and leaving all other units in mm and MPa we derive

$$L = (d \times E \times A \times 1E\text{-}6 / Q)^{0.5} \text{ (metres)}$$

Normally L will be in the order of 8 - 11 m. For most applications this will lie either fully within the first span or just into the second. As the majority of energy loss occurs in the first span then it is accurate enough for hand calculations to assume that one may model Q as being simply the average slope of the force diagram in the first span.

This completes the immediate losses of prestress and we may plot the transfer force .vs. distance diagram from which the transfer force at any distance from the jack may be read. We may now calculate the average force at transfer Ppi along the full length of the tendon. If this differs greatly from the assumptions made during the foregoing then one should start again using the latest calculated average force.

T.11.1.4 Elastic Shortening

As the load is transferred from the jack to the concrete, the concrete elastically shortens in direct proportion to the applied force, P (at transfer).

As the concrete section elastically shortens the steel contained within the section will elastically shorten as well ie the steel will experience the same change in compressive strain as the concrete - hence relieving some of the prestress force.

When there is more than one stressing sequence things get a little more complicated eg when there is more than one tendon / strand and they are not all stressed at the same time. The deformation due to elastic shortening in each tendon / strand is dependent upon when it was stressed in relation to the application of stressing to the other tendon / strands. The tendon first stressed will undergo the largest losses and the last tendon will experience no losses.

We usually take an average elastic shortening value and apply it to all strands. (ie. 0.5 x total losses) Looking at the effect of a single strand on another strand

e = d / E(averaged)

let

A = the concrete area of the design strip at each point

P_{tr} = the prestress force in one strand after transfer along the full length of the strand with losses only from anchorage friction and friction losses along the full strand length.

E_{ct} = average Youngs modulus of the concrete at transfer. RAPT calculates an Ect relevant to the stressing sequence specified in Input F6 [P/S -2]. The average compressive stress is calculated then applied to the relevant formulae for Youngs Modulus for concrete.

P_2 = force loss in 1 strand due to elastic shortening.

A_p = Area of 1 strand

considering the section as all concrete

$$e = \frac{P_{tr}}{AE_{ct}}$$

Transform the stress in the concrete to the stress in the steel by multiplying by Ep, Youngs modulus of the strands, (note that the strains are the same for both materials)

$$Ds = \frac{P_{tr}E_p}{AE_{ct}}$$

Now for post-tensioned systems account for the different stressing sequences by taking a simple average of the stresses during the whole stressing operation by introducing the 0.5 (as discussed above) in the equation below.

For pre-tensioned systems when the force is transferred to the concrete it is done in one operation when the section is removed from the formwork. In this case the averaging factor of 0.5 in the formula should not be applied. This is similar to the case for a beam with one round duct when the strands are all stressed simultaneously.

$$Ds_{av} = \frac{\sum P_{tr} \times 0.5 \times E_p}{A_{effconcrete} \times E_{ct}}$$

$P_2 = A_p \times Ds_{av}$

T.11.2 Long-term Losses of Prestress

Calculations of long term losses need to account for

- shrinkage of the concrete
- creep of the concrete
- relaxation of the strands

These cumulatively will reduce the average force at transfer by about 10 - 15%.

T.11.2.1 Shrinkage of the Concrete

The strain experienced from the shrinkage is dependant upon the environment in which the member is drying and the surface area of the member from which moisture is lost. e_{cs} (design shrinkage strain) for each code is given in section 7.7.3.5.

The loss of prestress due to concrete shrinkage is then given by

$P_3 = E_p \times e_{cs} \times A_p$ (Typically about 9 kN)

T.11.2.2 Creep of the Concrete

For specific information on creep calculations see section T.7.3.4.

The creep loss in a prestress tendon is dependent on the compressive stress in the concrete at the level of the tendon under the sustained or permanent loading.

To simplify this calculation, some designers consider that in prestress designs we usually balance nearly all the self-weight of the slab so we may approximate the sustained concrete stress at the level of the centroid of the strands, s_{ci}, (along the full length of the strand) prior to any time dependent losses and the sustained portions of the live load as being the total average force at transfer along the full length of the tendon in the design strip divided by the average concrete area, P_{tr} / A_{av}. This is very approximate and can lead to conservative or un-conservative results depending on the situation

RAPT calculates the moments at each design location due to the permanent service load (the permanent deflection case is used) and the transfer prestress forces. At each design location, the stress s_{ci} is determined from this moment and the axial prestress force at transfer at the level of each tendon. If s_{ci} is compressive for a tendon the creep loss is calculated as shown below,

$s_{in\ conc}$ = f_{cc} x s_{ci}
$e_{in\ conc}$ = f_{cc} x s_{ci} / E_c
= e in the strand

therefore the force loss in the strand is

P4 = $$\frac{\phi_{cc} \times \sigma_{ci} \times E_p \times A_p}{E_c}$$

where
Ec = Youngs modulus of the concrete at long-term

T.11.2.3 Relaxation of the Strands

Each code treats relaxation of strands in a slightly different way. The methods used by RAPT are given below.

AS3600 / ACI318 / BS8110 / SABS 0100 and general information
ACI318 only specifies that relaxation of tendons should be accounted for but gives no further information. RAPT thus uses the method as set in AS3600 for the ACI318 code. The code references referring to relaxation of strands are

- AS3600 Clauses 6.3.4 and 6.4.3.4
- BS8110 and CP65 Clause 4.8.2
- ACI318 Clause 18.6.1
- SABS0100 Clause
- CP2004 clause 12.8.2
- IS1343 clause 18.5.2.3

Under the continued long term high level of strain imposed on the strands a limited amount of strain relaxation will occur. The amount of relaxation, R, taken to occur for design purposes is based on a datum (basic) relaxation, R_b, occurring in a standard test.

The test procedure is to load the strand to 70 % of the strand's characteristic strength, f_p (usually 1840 MPa), at a constant temperature of 20 degrees Celsius for a period of 1000 hours. The basic relaxation value for strands is supplied by the strand supplier. For commonly used super grade, stress relieved strand 2% is typical.

The design relaxation, R, is obtained from the following expression (for AS3600)

$R = k_4\ k_5\ k_6\ R_b$ (a percentage of the force after transfer commonly about 3.4 %)

where
k_4 = a co-efficient dependent upon the duration of the prestressing force.
= $\log(5.38\ (j)^{1/6})$
j = duration of loading in days. For 30 year loading $k_4 = 1.404$
k_5 = a co-efficient dependent upon the stress in the strand after all losses (assume here 15% long-term losses) as a proportion of the strand characteristic strength. k5 is obtained from figure 6.3.4.3 of AS3600.
If $>= 0.7\ f_p$ and $<= 0.85\ f_p$
= 1 + (f / fp - 0.7) / 0.2
if $0.4 >= (f/f_p) <= 0.7$ then k_5 is taken as
= (f / fp - 0.4)/0.3 or
= 0 if < 0.4
k_6 = a co-efficient dependent upon the average annual temperature in degrees Celsius, taken as T / 20 but not less than 1.0.

One can appreciate that the force in the strand will gradually abate due to the loss in prestress from the other long-term effects from creep and shrinkage. The strands are therefore not kept at constant length and strain as assumed earlier. The total long-term relaxation in the strand hence will not reach the figure calculated above. We obtain a modified relaxation from

R' = R x (1 - force lost from the creep, P_3 and shrinkage, P_4) / force immediately after the transfer

commonly this would be in the order of R' = 1.5 - 3%

Therefore the force lost from strand relaxation, P_5 , may be taken in most design applications as

$P_5 = (0.015 \text{ to } 0.03) \times P_{pi}$
where
P_{pi} = the average force in the strand immediately after transfer ie after all immediate losses.

EUROCODE2

RAPT calculates the relaxation losses based on

(i) EUROCODE2 clause 4.2.3.4.1 and 4.2.3.5.5

The long term values for relaxation losses are assumed to be 3 times the relaxation losses after 1000 hours. The relaxation losses are taken from Figure 4.8 of the EUROCODE2. Figure 4.8 in EUROCODE2, which graphs the different relaxation values for different class wires, bars and strands, is modelled internally in RAPT. Users need to specify the 70% class value and RAPT will calculate the relaxation for the strand at each node point according to the stress in the tendon at each point.

The final stress loss in the tendon is calculated taking into account the losses due to creep and shrinkage. The code gives the following formula to calculate the initial stress

$sp = s_{pgo} - 0.3 \times Ds_{p,c+s+r}$

where
sp = initial stress
s_{pgo} = the initial stress in the tendons due to prestress and permanent actions
$Ds_{p,c+s+r}$ = the variation of stress in the tendons due to creep, shrinkage and relaxation. Initially RAPT does not have a value for relaxation. Thus RAPT does the relaxation calculation twice, assuming 0 relaxation the first time to obtain an estimation of relaxation. RAPT then applies the estimation to

the formula above to get a new (sp / f_{pk}) ratio and thus a better estimation of the relaxation value. Note: The final relaxation value obtained form figure 4.8 is multiplied by 3 in both calculations in accordance with code requirements.

The ratio of initial stress to characteristic tensile stress (sp / f_{pk}) is then applied to the graph in figure 4.8 of the EUROCODE2 to find the variation in stress in the tendon at each node point.

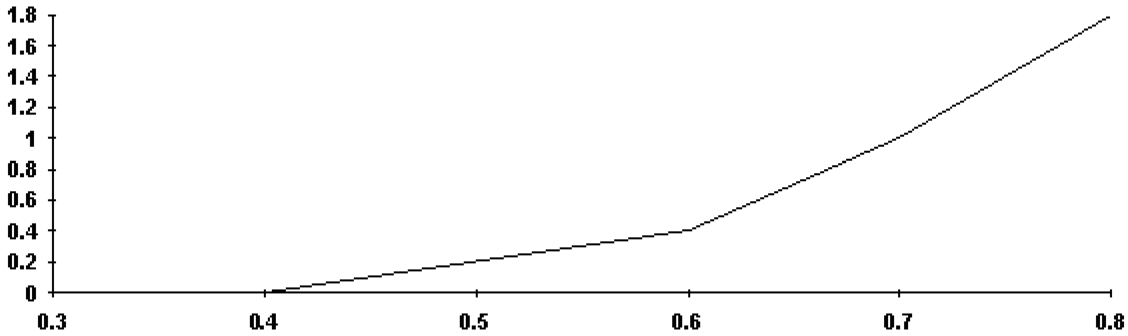


Figure T.11.1 - Relaxation

T.12 Strand Extension

As an on-site check that the stressing operation is progressing correctly the strand extension is measured as it is jacked. The calculated extensions should correlate to the site measurements. This check is used because

- The duct may have been punctured during the slab pour and concrete may have flowed into the duct effectively bonding the strand to the concrete. When the strand is stressed the force measured at the jack would then only be the force in the strand on the jack side of the break in the duct.
- The calibration of the jack may have been disrupted and may not be reading the correct force for a given extension.
- As a check on the assumed friction losses during the design eg. the duct may be corroded
- One or more of the wires in the strand may be broken
- The staff on site may be misreading the force gauge.

Hand calculations for the strand extension are normally based on considering each span separately and taking an average prestress force in that span, applying Hooke's law then summing the total extension, d

$$d = \frac{P_{av} L}{E_p A_p}$$

where
P_{av} = the average force at transfer in that span and strand
L = the span length
A_p = area of the strand
E_p = Youngs modulus of the strand

RAPT calculates extensions by doing a segmental calculation, using the force at each nodal point. Thus RAPT calculates the force at each nodal point and calculates the average extension over half the length to each node point either side of the point in consideration. The figures provided by RAPT also include any draw-in effects.

As a guide for normal slab work one would expect about 7 mm extension for every 1000 mm of span length.

It is good practice not to provide the prestressing contractor with the expected extensions prior to the stressing operation.

It has been common practice to accept variations between the on-site and the calculated measurement of ± 8% between the on-site and calculated extensions. The various codes give the following limits

- AS3600 The previous prestressed concrete code gave a limit of ± 5% variation and that elongation's were to be read to ± 3 mm. AS3600 states (clause 19.3.4.5) that the prestressing force is to be measured to ± 3% and that if the disparity between the on-site and calculated extensions exceeds 10% then appropriate action be taken.
- BS8110 and CP 65 clauses 8.7.3 and 8.7.5.4 state that the elongation should be measured to within 2% or 2mm whichever is lesser and if the measured extensions differ by more than 6% to the calculated extensions, corrective action should be taken.
- ACI318 clause 18.18.1 state that corrective action should take place if there is greater than 5% difference in measured to calculated elongation's for pretensioned elements and 7% for post-tensioned elements.
- Eurocode2 clause 6.3.4.5 only states that elongation measurements need to be recorded.

If the extensions printed from the RAPT prestressing report file are not being used then one may follow the simplified technique described below to obtain strand extension estimates.

1. Calculate the total angular change of the tendon within each span
   simple spans 8h/L
   end spans 12h/L
   internal spans 16h/L
2. Assuming
   1. 12.7 mm strand
   2. the strands are being jacked to 85% of breaking stress
   3. 2% losses within jack

   Giving a force behind the jack (slab side) of Pj = 153.3 kN per strand

3. Using the formula below calculate the loss of prestress at one extreme end of the tendon (assuming being jacked from one end)

   $$P_a = P_j e^{-\mu\left(\alpha_{tot} + \beta_p \times L_{pa}\right)}$$

   where
   a_{tot} = total angular change over full length

m = 0.20 co-efficient of friction
b_p = 0.025 duct wobble factor
L_{pa} = total length of tendon (metres)

4. Calculate the average force along the full length of the tendon.
   If the tendon is stressed from one end only then the average force is taken as simply
   $P_{av1} = (153.3 + P_a) / 2$
   If stressed from both ends then this is averaged again
   $P_{av2} = (P_a + P_{av}') / 2$
5. Taking the average force as applying along the full length of the tendon calculate the expected extension from Hookes Law.
   $$d_{total} = \frac{P_{av} L}{EA}$$
   where
   E = 1.95 x 10^5 N/mm^2
   A = 100 mm^2
   L = the full length of the tendon
6. Subtract the draw-in from the locking off operation
   6 mm when stressed from one end
   12 mm when stressed from both ends

T.13 Columns

RAPT allows designers to design column to any chosen code.

T.13.1 Columns - Stocky

A short column is one where the length effect or deflection response is very small and is considered negligible. The great majority of braced columns and about half of unbraced columns may be considered as being stocky columns. The maximum length of a short column depends upon its deflected shape and is defined in more detail in Section T.13.2.

The axial load capacity of a column decreases when a moment is present. A plot of the column axial load capacity, N_u, against the moment that it can simultaneously carry, M_u, is called a column interaction diagram (see Fig T.13.1). Any loading that plots within the area enclosed by the curve and the axis is a permissible loading; any combination that falls outside this area is a failure combination.

Since all columns are subject to some moment eg. from out of vertical column construction, the code requires that all columns be designed for a minimum eccentricity about either axis.

- AS3600 0.05 x the column dimension in that direction. clause 10.1.2
- ACI318 (15.2 + 0.03h) mm clause 10.11.5.4
- BS8110 and CP 65:- 0.05 x the column dimension in that direction but < 20 mm clause 3.8.2.4
- EurocodeII (ea) an equivalent geometrical imperfection eccentricity clause 4.3.5.4
- SABS 0100 0.05 x the column dimension in that direction but < 20 mm clause 4.7.2.3
- CP2004:- 0.05 x the column dimension in that direction but < 20 mm clause 6.2.1.2d
- IS 456:- This code bases the minimum value on the unsupported length of the column which is unknown by RAPT in clause 25.4 with a minimum of 20mm. Instead RAPT uses 0.05 x the column dimension in that direction but < 20 mm

Each code has different detailing rules for columns. Designers should check their codes for these rules. RAPT provides helpful information such as the reinforcement ratio so designers can meet their code requirements easily.

AS3600 requires columns to contain longitudinal reinforcement sufficient to make the reinforcement percentage, p = A_s / bD, at least equal to 0.01 (due to creep and shrinkage effects on smaller areas) but normally not greater than 0.04 (ie. 0.08 at splice locations which may cause congestion problems). At least 6 bars must be used in a circular arrangement and 4 bars in a rectangular arrangement. Ties must be at least 6 mm diameter for Y12 longitudinal bars and 10 mm for Y24 - Y36 and 10 mm for bundled bars. Every corner bar and every alternate bar must be laterally braced by a tie and no bar shall be more than 150 mm from such a laterally supported bar.

Plotting the Interaction Diagram.
RAPT applies the general rules as stated below for each code, allowing for each codes different factors of safety and rules on bending and axial compression.

Point 1 Maximum Axial Capacity
Although in design, axial load without moment is not a practical case, N_{uo} is a convenient theoretical limit and one well documented experimentally.

N_{uo} = 0.85 f'_c (A_{conc} - SA_s) + Sf_y A_s
where
A_{conc} = total concrete area
A_s = total steel area
f_y = steel yield stress

For this loading condition using the rectangular stress block representation, the neutral axis position, kd, is at an infinite distance from the extreme compressive fibre. If a curvilinear stress-strain function is used for the concrete then the extreme compressive fibre strain and kd should be manipulated until the maximum axial capacity is found.

We calculate the plastic centroid position, dp, for this neutral axis location. It is defined as the distance from the extreme compression fibre to the line of action of the resultant of all internal forces in the cross-section.

Point 2 Pure Bending
We consider the column as if it were acting in purely flexural action ie the sum of the internal forces will be in equilibrium (C=T ,N_u=0). By trial and error or using a closed form solution for simple steel arrangements we find the kd such that the above statement is true. We may then take internal moments about any fibre and hence calculate Mu. For consistency we should do so about the plastic centroid as for all other points on the interaction diagram internal equilibrium is not satisfied and we will have to take moments about the plastic centroid. For all other points equilibrium is achieved by the applied axial force acting at the plastic centroid balancing the misclose between C-T.

At this point we calculate the "effective depth" of the section, d. The effective depth is defined in all codes as

"d = the line of action of the resultant tensile force of all steel layers which are tensile for the pure bending condition."

Point 3 "Balanced" Condition
This point is found by setting the strain in the extreme reinforcement layer to 0.002. We calculate the imbalance in C-T = N_{ub} and the internal bending moment about the plastic centroid, M_{ub}.

Note that for beams we avoid balanced failure defined as the point where the concrete reaches its crushing strength simultaneously with the steel yielding, as it is in fact on the limit of a brittle failure condition ie sudden collapse. However for columns, which are primarily axial load carrying members, brittle or compression failure, at ultimate loading, frequently cannot be avoided.

Point 4

Not calculated as a special point any more.

Point 5 Point of Decompression

The extreme fibre which will be in tension for the pure bending case is given a strain of zero as it is on the verge of passing from compression to tension (decompression) thereby defining kd = D. Calculate N_u and M_u as above. The interaction diagram is essentially linear from this point to N_{uo}.

Point 6 Minimum Eccentricity

For any given axial load, N_u, the minimum moment to be taken as acting simultaneously is given by each code. See 7.13.1 for code minimum eccentricities.

Point 7 Pure Axial Tension

The force in the reinforcement at full tension yield.

Intermediate Points

These may be obtained by either selecting kd values intermediate of previously calculated points or by finding the kd value that will yield an imbalance in C-T equal to a force that one would like to plot.

Design Strength Curve

To account for bad workmanship and in the case of columns, a further factor of safety due to the brittle collapse mode, a capacity reduction factor, f , / factor of safety is introduced as detailed below. The strength in bending curve N_u vs M_u is reduced by multiplying by the appropriate capacity reduction factor / factor of safety for that loading condition. For code values see

- AS3600 clause Table 2.3
- BS8110 and CP65 clause 2.4.4 and 3.8
- ACI318 clause 8.8 and 9.3.2.2
- Eurocode clause 2.3.3
- SABS 0100 clause 3.3.3.2
- CP2004 clauses 2.44 and 6.2
- IS456 clauses 36.4 and 39

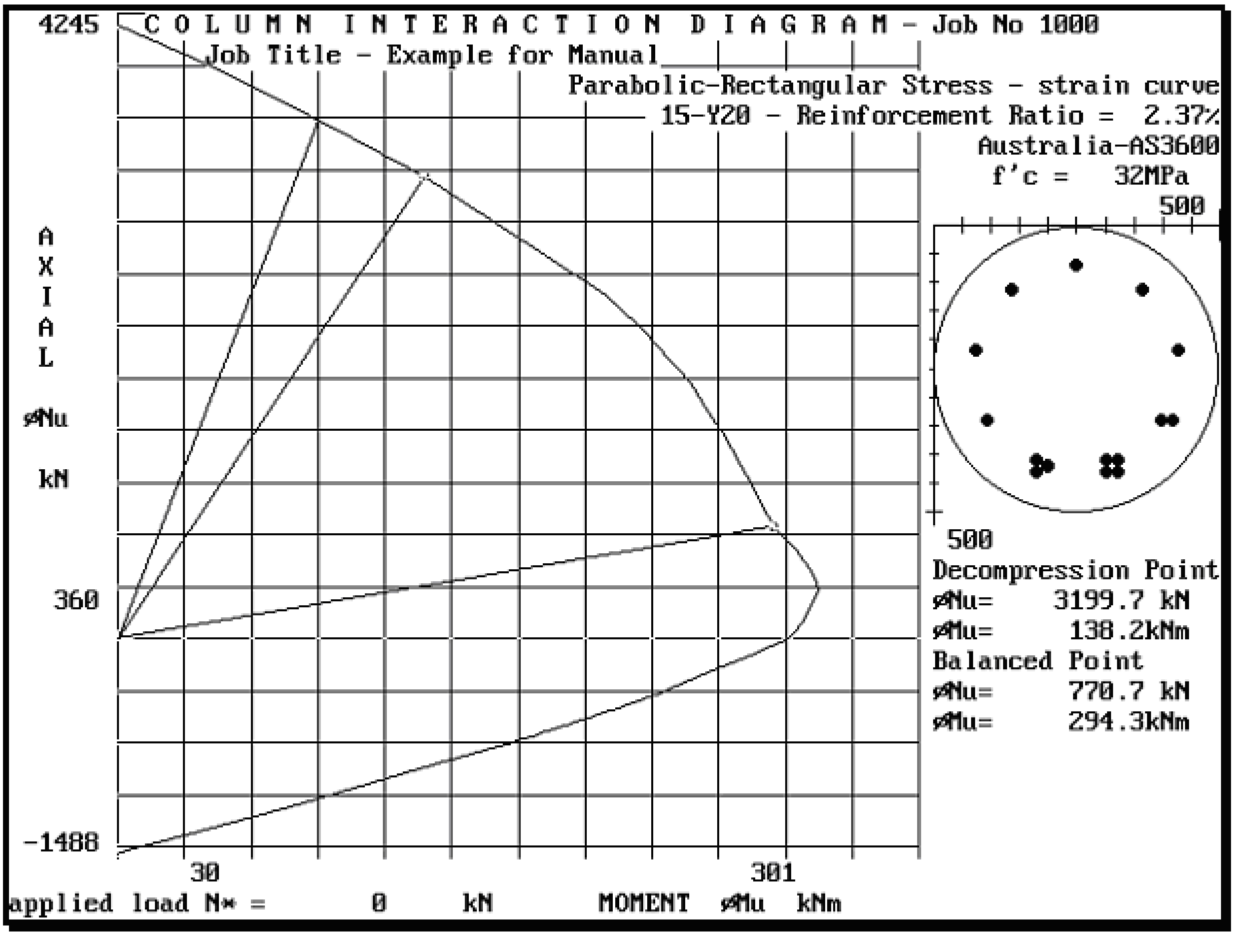


Figure T.13.1

T.13.2 Columns - Slender

RAPT checks slender columns according to each code type as specified. From the codes, RAPT calculates a magnified moment which is plotted onto the interaction diagram. To calculate this magnifier, users are required to specify extra column information as set out in the slenderness option

Slenderness limits
A column is deemed to be slender if

	CODE	BRACED	UNBRACED	CLAUSE
(i)	AS3600	Le/r > 25 or Le/r > $60(1+M^*_1/M^*_2)(1.0-N^*/0.6N_{uo})$	Le/r > 22	10.3.1
(ii)	Eurocode 2	25 x (2 - e_{o1} / e_{o2}) or l_o/i > 25 or 15/Ön_u	l_o/i > 25 or 15/Ön_u	4.3.5.3.5
(iii)	BS8110 and CP65	l_e / h or l_e / b > 15	l_e / h or l_e / b > 10	3.8.1.3
(iv)	ACI318	kl_u/r > 34-12M_{1b}/M_{2b}	kl_u/r > 22	10.11.4.1 & 2
(v)	SABS0100	l_{ex} / h or l_{ey} / b > 17-7(M_1 / M_2)	l_{ex} / h or l_{ey} / b > 10	4.7.1.4
(vi)	CP2004	l_e / h or l_e / b > 15	l_e / h or l_e / b > 10	6.2.1.1
(vii)	IS456	l_e / h or l_e / b > 12	l_e / h or l_e / b > 12	25.1.2

Moment Magnifier
There are two basic approaches to calculating the moment magnifier. AS3600 and ACI318 calculate it based on the critical buckling load where as BS8110, SABS 0100 and Eurocode 2 calculate the magnified moment based on the applied axial force multiplied by the eccentricity induced into the slender column. Each of these methods and code approaches are shown below.

Buckling Load Method
AS3600 and ACI318 calculate the moment magnifier based on the critical buckling load. The Buckling load is calculated as

(i) $$N_c = \left(\frac{\pi^2}{L_e^2}\right)[EI]$$ AS3600 clause 10.4.4

(ii) $$P_c = \left(\frac{\pi^2}{(kl_u)^2}\right)[EI]$$ ACI318 clause 10.11.5.1

where EI = M / curvature

The resulting moment magnifier is calculated from

(i) AS3600

$$\delta_b = \frac{k_m}{\left(1 - \frac{N^*}{N_c}\right)} \geq 1.0$$

AS3600 clause 10.4.2 for a braced column.

$$\delta_s = \frac{1}{\left(1 - \frac{\sum N^*}{\sum N_c}\right)} \geq 1.0$$

AS3600 clause 10.4.3 for an unbraced column.

where $k_m = \left(0.6 - 0.4 \frac{M^*_1}{M^*_2}\right)$ but shall not be taken as not less than 0.4.

M^*_1 / M^*_2 is the ratio of the smaller to the larger of the design bending moments at the ends of the column. The ratio is taken as negative when the column is bent in single curvature and positive when the column is bent in double curvature.

If users input a value into the User Defined Nu/Nc option in the slenderness menu, RAPT will calculate d_s based on the input value rather than calculate P_c based on the sway effective length factor.

(ii) ACI318

$M_c = d_b M_{2b} + d_s M_{2s}$ Magnified factored Moment Clause 10.11.5.1

M_{2b} = value of larger factored end moment on compression member due to loads that result in no appreciable sidesway.

M_{2s} = value of larger factored end moment on compression member due to loads that result in appreciable sidesway.

$$\delta_b = \frac{C_m}{\left(1 - \frac{P_u}{\phi P_c}\right)} \geq 1.0$$

ACI318 clause 10.11.5.1 for a braced column.

where $C_m = 0.6 + 0.4 \frac{M_{1b}}{M_{2b}}$ but not less than 0.4

M_{1b} = value of smaller factored end moment on a compression member due to the loads that result in no appreciable sidesway, calculated by conventional elastic frame analysis, positive if member is bent in single curvature, negative if bent in double curvature.

M_{2b} = value of larger factored end moment on a compression member due to the loads that result in no appreciable sidesway, calculated by conventional elastic frame analysis.

where P_c calculated using the braced effective length factor

$$\delta_s = \frac{1}{\left(1 - \frac{\sum P_u}{\phi \sum P_c}\right)} \geq 1.0$$

ACI318 clause 10.11.5.1 for an unbraced column.

where P_c is calculated based on the sway effective length factor.

If users input a value into the User Defined Nu/Nc option in the slenderness menu, RAPT will calculate d_s based on the input value rather than calculate P_c based on the sway effective length factor.

After the calculation of the moment magnifier, this is applied to the larger moment and plotted on the interaction diagram at the applied axial load.

Eccentricity Method

BS8110, SABS 0100, CP65, CP2004 and Eurocode 2 use this approach. In general the magnified moment is Applied Axial load multiplied by the eccentricity specified by the code.

(i) BS8110 Braced Column

Design Moment = the largest of

M2
$M_i + M_{add}$
$M_1 + M_{add}$
e_{min} x N
where $M_i = 0.4M_1 + 0.6M_2 >= 0.4M_2$

$M_{add} = N \times a_u$

where
$$a_u = \left(\frac{l_e}{\pi}\right)^2 \times \frac{1}{r}$$
clause 3.8.3.1

1/r = curvature of the column = M/EI

BS8110 Unbraced Column

For sway effects users can input an average au for the group of columns in the storey. If no value is input then RAPT assumes the isolated column is an average column for the building.
For sway cases the design moment is calculated from

$M_2 + M_{add}$

(ii) SABS 0100 Braced Column

Design Moment = largest of (Slenderness Moment)

M_2
$M_i + M_{add}$
e_{min} x N
where $M_i = 0.4\ M_1 + 0.6\ M_2$

$M_{add} = N \times a_u$

where
$$a_u = \left(\frac{l_e}{\pi}\right)^2 \times \frac{1}{r}$$
clause 3.8.3.1

and 1/r = curvature of the column = M/EI

SABS 0100 Unbraced Column

For sway effects users can input an average au for the group of columns in the storey or RAPT will calculate the sway case based on:
Design Moment = larger of

$$M_V + M_H\left(1 + \frac{M_{add\,,unbr}}{(M_V + M_H)}\right)$$

$$0.6 \times M_2 + 0.4 \times M_1 + M_{add\,,braced}$$

where $M_2 = M_V + M_H$

M_1 and M_2 are the smaller and larger column end moments respectively
M_V and M_H are the initial column end moment due to vertical load and horizontal load respectively.
$M_{add,unbr} = M_{add}$ based on the unbraced (sway) effective length.
$M_{add,braced} = M_{add}$ based on the braced effective length.

(iii) Eurocode 2 Braced Column

$M_{mag} = N_{SD} \times e_{tot}$
$e_{tot} = e_e + e_a + e_2$
where e_e is taken as the larger of
$e_e = 0.6 \times e_{o2} + 0.4 \times e_{o1}$ or
$e_e = 0.4 \times e_{o2}$ Clause 4.3.5.6.2

where e_{o1} and e_{o2} denote the first order eccentricities at the two ends and $e_{o2} >= e_{o1}$

ie $e_{o1} = M_1 / N_{SD}$ and $e_{o2} = M_2 / N_{SD}$

$e_a = (n \times l_o) / 2$ eccentricity due to imperfections. Clause 4.3.5.4

where l_o denotes the effective length of the isolated element

n denotes the inclination from the vertical = $\frac{1}{\left(100\sqrt{l}\right)}$ in radians. Users are required to calculate this value and enter it into the slenderness menu as the Accidental Inclination.

$$e_2 = \left(\frac{l_o}{\pi}\right)^2 \times \left(\frac{1}{r}\right)$$ eccentricity due to second order effects. Clause 4.3.5.6.3

1/r = curvature of the column = M/EI

Eurocode 2 Unbraced Column

For sway effects users can input an average au for the group of columns in the storey. If no value is input then RAPT assumes the isolated column is an average column for the building.

Creep

All slenderness column calculations are required to include effects for creep. See section 7.7.3.4 for theory on creep calculations. The modified Youngs modulus due to creep is then used in the E values during the curvature calculations.

T.14 Composite Steel Beams

RAPT designs simply supported, single span composite steel beams to AS2327, Part 1-1980. All clause references below refer to this code. It is recommended that the designer adopt the plastic design method to achieve the optimum beam design. However the elastic section properties must still be calculated in order to calculate deflections, vibration response and stresses under service loads.

The most common profiled sheeting used is BONDEK which has the following profile

hr = height of rib
= 53 mm

w = width between ribs at top of rib
= 200 - 32 = 168 mm

The following theory has been limited to symmetrical universal beams however RAPT can be used to design composite beams of user-defined cross-section plus a concrete haunch above the top steel flange and below the slab may also be incorporated. If a haunch is present the program assumes that it extends the full width of the top steel flange and that it has a slope of 45 degrees up to the slab soffit. Use of haunches is not common practice as, especially when profiled sheeting is used, it reduces the floor to floor construction cycle. They are used for bridge girders where cross-falls are required and are obtained by varying the depth of the haunch on each girder across the width of the deck. Previous design office practice has been to neglect the capacity of the haunch in strength and section property calculations as its contribution is minor in comparison to the contribution of the other elements.

If user-defined sections are nominated RAPT assumes that the section is fully effective ie the steel flange outstand and web unsupported depth rules are complied with.

T.14.1 Effective Flange Width

For flexural action the flange widths for L-beams (ie beams where the slab extends on one side of the beam only) and for T-beams (where there is slab on either side of the beam) are given in

- AS3600 clause 8.8.2.
- Eurocode 2 clause 5.3.2.1
- ACI318 Clause 8.10
- BS8110, CP65 clause 3.4.1.5
- SABS0100 clause 4.4.1.5
- CP2004 clause 5.2.1.2
- IS456 and IS1343 clause 23.1

T.14.2 Elastic Section Properties

Assuming that under the action of service loads the steel and concrete remain within their respective linear stress / strain regions we may calculate the elastic section properties using the transformed section (or linear elastic) method. We transform the composite section ie after the concrete has bonded to the concrete to an equivalent steel area. Consider a concrete element A. The force acting on this element must remain equal regardless of what material we transform it into. Therefore it has a force acting on it of P then the stress will be given by P/A. Converting it to an equivalent steel area we multiply by the ratio of the Youngs modulus of the steel, E_s, divided by the Youngs modulus of the concrete, E_c. This ratio is termed the modular ratio n.

$n = E_s / E_c$

where
E_s = 2E5
E_c = 5056 Öf'$_c$

Therefore the area of the element A will be increased to A' = A n.

The strain diagram may be used directly to get the strain in the element, considering it to be composed of steel, acting as part of a fully homogeneous section composed solely of steel. In a similar manner section properties may be calculated using the transformed section.

However when we wish to find the true stress in the concrete we must convert back to concrete by dividing by the modular ratio.

Considering the transformed section we find the centre of area of the concrete and steel elements from the top fibre, kd, by taking first moments of area about the top fibre of each element and dividing by the total area.

Note that concrete in tension is ignored.

$$kd = \frac{\sum(Ay)}{\sum A}$$

The code requires that where BONDEK is present and

(i) orientated parallel to the beam axis
consider only the thickness of the slab above the top of the ribs.

(ii) orientated perpendicular to the beam axis
calculate section properties assume a solid slab equal to the total slab depth but modify the properties as follows

$$I_{ce} = I_c\left(1 - \frac{h_r}{5D_{cs}}\right)$$

$$Z_{ce} = Z_c\left(1 - \frac{h_r}{2D_{cs}}\right)$$ at the top concrete fibre

$Z_{bce} = Z_{bc}$ at the bottom steel fibre
$Z_{tce} = Z_{ts}$ at top steel fibre use unmodified composite properties

As no criteria is given in the code for the composite elastic modulus for the compressive fibre of the steel beam one may assume that the stress at this location it is not a design parameter. If it is checked then the elastic section modulus used should be based on the as calculated modulus of the composite section for this location, Z_{tce}.

To obtain kd by initially assuming that it lies within the flange depth, case (i) below (normal case). If this proves to be false then use case (ii) below.

(i) neutral axis within effective slab thickness [see (i) and (ii) above]
defining

$$r = \frac{2A_s}{bd_s}$$

$$kd = 0.5d_s\left[\left(nr(4+nr)\right)^{1/2} - nr\right]$$

check that kd is within effective slab depth. If not use equation theory below.

$$I_c = \frac{b(kd)^3}{3n} + I_s + A_s\left(d_s - kd\right)^2$$

(ii) neutral axis below effective slab depth

$$k_d = \frac{\left[\left(bt^2\right)/\left(2n\right)\right] + \left(A_s d_s\right)}{\left[\left(bt\right)/n\right] + A_s}$$

$$I_c = \frac{bt^3}{12n} + \frac{bt}{n}\left(kd - 0.5t\right)^2 + I_s + A_s\left(d_s - kd\right)^2$$

T.14.3 Design based on Elastic Methods

It is not common to design beams elastically. Beams designed using a limit state (plastic) approach will generally be more economical. A check list of the elastic design criteria follow.

1. Under construction loads, ie prior to the composite action taking effect, (beam self-weight + construction load) the flexural stresses (compressive and tensile) in the steel beam, f1, must be less than or equal to 0.66 times the flange yield stress. This assumes that the critical buckling flange, the top steel flange, is restrained by the sheeting or formwork during the construction phase. The bending moment is determined taking due account of the propping making the beam continuous.
   $s_1 <= 0.66\ f_y$ based on Z_s
2. If the beam is UNpropped during construction then a limit is placed on the tensile and compressive steel stresses under maximum working loads of 0.9 times the flange yield stress.
   $s_1 + s_2 <= 0.9\ f_y$
   where
   s_2 = stress due to superimposed loads based on Z_{ce}
   = superimposed DL + LL
3. Under maximum working loads the tensile and compressive steel stresses are less than 0.66 times the flange yield stress.
4. That the shear stress in the web is less than or equal to 0.37 times the yield stress of the web. For this calculation we assume that all of the vertical shear force is taken solely by the web of the steel beam.

T.14.4 Design based on Plastic (Strength Limit State) Methods

A check list of the design criteria follow.

If BONDEK is used then, irrespective of its orientation with respect to the beam axis, the concrete below the top of the rib shall be ignored. RAPT includes the concrete haunch if present.

1. Under construction loads, ie. prior to the composite action taking effect, (beam self-weight + construction load) the stress in the steel beam, f1, must be less than or equal to 0.66 times the flange yield stress. This assumes that the top steel flange is restrained by the sheeting or formwork during the construction phase.
   $f_1 <= 0.66\ f_y$ based on Zs
2. If the beam is UNpropped during construction then a limit is placed on the tensile and compressive steel stresses under maximum working loads of 0.9 times the flange yield stress.
   $f_1 + f_2 <= 0.9\ f_y$
   where
   f_2 = stress due to superimposed loads based on Z_{ce}
   = DL + LL
3. Under maximum ultimate loading the design strength in bending, M_r' (also known as moment of resistance), is greater than or equal to the applied ultimate bending moment, M*. The ultimate load factor is taken as 1.67 on live and dead loads.
   For this calculation the stress in the concrete may be assumed to be modelled using a simplified compressive stress distribution with the concrete stress set at 0.85 times the compressive strength of the concrete, f'_c, and a depth equal to the neutral axis depth, kd. Note that this divergs from the simplified rectangular stress block for normal reinforced concrete design which limits the depth of the stress block to gamma times kd, where gamma is dependant upon the compressive strength of the concrete and varies between 0.85 and 0.65.
   The concrete strength in tension is ignored.
   The stress in the steel beam is taken as being at yield.
   The capacity reduction factor, phi, is taken as
   0.95 when kd lies within the concrete slab (normal case) and
   0.90 when it lies below the concrete slab. Again this varies from AS 3600 which takes phi as 0.80 for bending.

The moment of resistance is dependant upon the kd location. The location of kd is found by the following procedure.

1. kd lies within the concrete flange.
   This is the initial assumption which must be checked. Assume that internal equilibrium is satisfied (C=T) with kd within the flange.
   kd = $\dfrac{A_s \times f_y}{0.85 \times f'c \times b}$ Note: consider separate fy values for flanges and web
   <= t_e if not then try 2 below
   where
   t_e = the effective depth of concrete .ie. modified for the presence of BONDEK if required.
   f = 0.95
   $M_r' = 0.95\ A_s\ f_y[\ D/2 + t - kd/2\ ]$
2. kd within top steel flange
   Check whether the compressive force that can be mobilised with kd set at the bottom face of the top steel flange is sufficient to equilibrate the corresponding tensile force.
   $C = 0.85\ f'_c\ t_e b + B\ t_f\ f_y$
   $T = f_y\ B\ t_f + [\ D - 2\ t_f\ ]\ t_w\ f_y$
   If C >= T then kd will lie within the top steel flange otherwise go to 3 below
   f = 0.90
   Find the depth of the compressive region of the top steel flange, x.
   x = $\dfrac{t_w f_y\left(D - 2t_f\right) + Bt_f f_y - 0.85bf'ct_e + Bt_f f_y}{2Bf_y}$
   $C_c = 0.85\ f'_c\ t_e\ b$
   $C_s = x\ f_y B$
   $T_w = t_w\ f_y\ (\ D - 2\ t_f)$
   $T_{fb} = B\ t_f\ f_y$
   $T_{ft} = B\ (\ t_f - x\ )\ f_y$
   $M_r' = 0.90\ [\ -C_s\ (DC + x/2\ ) - C_c\ DC/2 + T_{ft}(\ DC + t_f - (t_f - x)/2) + T_{fb}(DC + D - t_f/2) + T_w(DC + D - t_f/2)\ ]$
3. kd within beam web
   x = $\dfrac{t_f f_y B - 0.85\ bf'ct_e - Bf_y t_f + t_w f_y (D - 2t_f)}{2t_w f_y}$
   f = 0.90
   $C_c = 0.85\ b\ f'_c\ t_e$
   $C_{ft} = B\ f_y\ t_f$

$C_w = t_w f_y x$
$T_w = t_w f_y (D - 2t - x)$
$T_{fb} = B t_f f_y$
$M_r' = 0.90 [-C_c t_e/2 - C_{ft}(DC - t_f/2) - C_w(DC + t_f + x/2) + T_w(DC + D - \{D - 2 t_f - x\} / 2 - t_f) + T_{fb}(DC + D - t_f/2)]$

4. That the shear stress in the web is less than or equal to 0.37 times the yield stress of the web. For this calculation we assume that all of the vertical shear force is taken solely by the web of the steel beam.

T.14.5 Shear Stud Design

The code requires that regardless of the design approach adopted for the design of the steel beam the shear studs are to be designed using the ultimate design method with a capacity reduction factor, phi, of 0.85.

RAPT selects the spacing required for shear studs of the following diameters and detailed in triples, pairs and single studs per stud spacing
13, 16, 19, 22, 25 mm (Note: 22 and 25 diameter studs are not to be used with profiled sheeting.)

The method adopted for design of the spacing required for each stud diameter is as follows.

1. Determine the maximum ultimate compressive force that can be mobilised within the concrete flange.
   H_{cc} = 0.85 f'_c b kd
   This has an upper limit of kd being the slab depth for solid slabs and kd being equal to the slab depth minus the rib height for slabs incorporating profiled sheeting.
2. Read the characteristic strength of the stud, P_k, from Table 7.1 of AS 2327 which is abbreviated below.

Characteristic Stud strength P_k (kN)

Nominal Stud Diameter					
f'_c MPa	13 mm	16 mm	19 mm	22 mm	25 mm
20	35	50	70	90	110
25	40	55	75	95	120
30	45	60	80	105	130
40	50	70	95	120	145

Note that P_k must be modified where

1. the concrete density is other than 2400 kg per cubic metre by multiplying by the reduction factor R below.
   R = $(\text{density})^{0.67}$ / 2400
   <= 1.0
2. Profiled sheeting is present and is orientated parallel to the beam axis.
   $P_k = P_k\ 0.6\ w_r / h_r\ (h_e/h_r - 1)$
   where
   w_r = 200 mm for BONDEK
   h_r = 53 mm for BONDEK
   h_e = length of stud after welding (usually lose 10 mm in welding process)
   <= h_r + 75 mm
3. profiled sheeting is present and is orientated transverse to the beam axis.
   Pk = $$\frac{P_k 0.85 w_r}{(N_e)^{1/2} h_r (h_e / h_r - 1)}$$
   where
   N_e = the number of studs. Therefore this equation must be used iteratively.
4. Assume that the load sharing factor (see v) below) is unity, hence Pd = Pk, and calculate the first estimate of the number of studs, Ni.
   $N_i = H_{cc} / P_k$
   NOTE: that the number of studs calculated is the number required between the point of zero bending moment and the location of the maximum ultimate bending moment along the beam length, L. RAPT determines the minimum value for the above length from the flexural analysis and assumes that the final stud spacing calculated will be carried through for the full length of the beam.
   Users may wish to adjust the spacing of the studs manually to satisfy clause 7.1.6 of the code which states that where there are sharp discontinuity's in the shear force diagram (ie from point loads) the studs should be distributed in accordance with the respective areas of the shear force diagram.
5. Calculate the design strength of each stud, P_k. The design strength of a stud is dependant upon the amount of load sharing between all studs. The more studs that are present the greater the design strength, P_d, of each stud (where profiled sheeting is present and orientated transverse to beam axis recalculate P_k from 2) above).
   Pd = $$0.85 P_k \left(1.45 - \frac{0.45}{(N_e)^{1/2}}\right)$$

6. Recalculate the required number of studs
   $N_i = H_{cc} / P_d$
7. Iterate from 5 to 6 above until $N_j = N_i$. When this is satisfied $N_e = N_j$
8. Finally calculate the spacing required for the shear studs over the distance L (see above) from
   $s = L / (N_e / j)$
   where
   j = the number of studs at stud spacings

T.14.6 Stud Detailing

The code requires that the following rules be complied with regarding the detailing of the shear studs. RAPT checks these rules and flags where they need be considered.

1. The shank diameter of the stud must not exceed twice the thickness of the flange of the beam to which it is connected.
2. The distance between the edge of the stud and the edge of the beam flange must not be less than 25 mm.
3. The longitudinal spacing of shear connectors must not be greater than 4 times the slab thickness.
4. The longitudinal spacing of shear connectors must not be greater than 600 mm.
5. The transverse spacing of the studs must be such that the clear spacing between the studs is not less than 1.5 times the shank diameter of the stud.
6. The cover to the stud shall not be less than 25 mm from the top of the slab.
7. The cover to the stud shall not be less than 75 mm from any side face of the slab including haunches.
8. The minimum stud heights are as follows

Nominal Stud diameter (mm)	Minimum Height of Stud after Welding (mm)
13	52
16	64
19	76
22	88
25	100

T.14.7 Ductility

RAPT does not design beams to comply with the ductility rules of the code. The ductility check is reported for the present rules in the code and according to the proposals by Bridge and Patrick (ref 21). Both guide-lines are described below and are more fully discussed in reference 21.

1. Present code rule
   The neutral axis location, kd, (at ultimate strength limit state) is limited to the following
   kd < 0.16 x the overall depth of the composite section
   This applies regardless of the yield stress of the steel and whether the beam is a T or an L-beam. However if kd lies within the concrete slab only then the following rule governs
   kd < 0.136 x the overall depth of the composite section
2. Proposal by Bridge and Patrick
   In summary the proposals are as follows.
   a) Ensure that the section is at least compact ie. that it can develop the plastic moment capacity but rotation may be limited by local buckling, distortion or concrete crushing. This is defined by the following
   d/t <= (30 / bd) (235 / F_{yw})0.5
   where
   d = the clear depth of the beam between root fillets
   t = the thickness of the web
   F_{yw} = the yield stress of the web
   bd = the depth to the plastic neutral axis of the composite cross-section measured below the top root fillet.
   This is generally complied with.
3. Ensure that the deflection at ultimate flexure is less than or equal to the span length divided by 100. This is purely a parameter which was found to provide satisfactory ductility for the beams tested by ref 21, it in itself does not reflect the beam ductility. This must be obtained using a moment-area integration along the beam length using the moment-rotation response curve for the beam under the action of the ultimate loading.
4. Ensure that deflection under serviceability loads is within satisfactory limits.

T.15 Anchorage Zones

The following information is based on AS3600.

Anchorage zones are usually at the ends of the beam or slab where the force in the tendon is transferred onto the concrete by steel bearing plates or proprietary anchorages. However in some cases tendons are stopped short within the length of a span and these are treated in the same manner.

The forces are high and are applied over a small area and hence complex stress distributions are set up locally immediately behind the anchorages. As we move away from the anchorage the stress isobars readjust and eventually become linear giving a trapezoidal stress distribution on the concrete section and stresses may be then calculated based on the normal elastic theory assuming an uncracked cross-section.

If the anchorage zone has more than one anchorage then the design should investigate the possibility of any of the tendons being stressed separately (normal case) or all tendons being stressed simultaneously. If two anchors are closer together than 0.3 times the total depth, or breadth of the member then the effect of the two tendons will be similar to that of a single tendon with the total force of both the single tendons.

Within the anchorage zone two forms of stresses must be checked. They are known as "spalling" and "bursting" stresses.

Bursting refers to the tensile force which is set up within the depth of the member away from the outside face. Maximum bursting usually occurs directly behind the anchorage and is calculated using an empirical formula which has been shown to give reliable results.

$$T = 0.25 \times Pj \times (1 - kr) \qquad (T.15.1)$$

where

T = the tensile bursting force transverse to the longitudinal axis of the member and perpendicular to the plane being considered.

P_j = the maximum jacking force in the tendon

k_r = the ratio of the depth or breadth of the anchorage bearing plate to the corresponding depth or breadth of the symmetrical prism.

The symmetrical prism is a notional prism with an anchorage at the centre of the end face and a depth or breadth, taken as twice the distance from the centre of the anchorage to the nearer concrete face.

The above formula may only be justified by statical considerations for the simple case of a single concentric anchor.

Note that the bursting tensile force tends to split the concrete apart in both the horizontal and the vertical planes. Hence the above calculation must be done for both planes separately taking due account of the shape of the prism and the size of the bearing plate in the plane being investigated.

The code (AS3600 section 12.2) states that Eq.(T.15.1) is to apply, to each anchor separately, for bursting calculations regardless of the number and configuration of the anchors in the anchorage zone. This recommendation is based on experimental evidence. The authors conservatively recommend, that at this stage, that the worst tensile force derived from Eq.(T.15.1) and that obtained from statical considerations be used for design purposes.

The magnitude of the force is calculated using a concrete beam analogy which considers a free-body of the anchorage zone. The force in the tendons is represented by large distributed loads over the depth of the anchorage plates which are resisted by a single distributed load remote from the anchorage zone (where stresses are again linearly distributed) which is equal to the total force in all tendons divided by the depth of the member. Using the free-body one may calculate the bending moment diagram through the depth of the member. A spalling bending moment will reveal itself where the sign of the bending moment is negative. Bursting moments are positive.

Note that for a single anchorage placed symmetrically in the end zone the maximum bursting moment will occur directly behind the anchor. Dividing by a lever-arm of 0.5 D one arrives with Eq.(T.15.1) above. The code gives no guide-lines for the value of the lever-arm for bursting moment calculations when there are multiple anchors in the anchorage zone. Ref.25 recommends that when there are two or more anchors in a rectangular end block that a lever-arm of D/n be adopted where D = the overall depth of the section and n is number of anchors.

Spalling refers to a tensile force which may occur at the outside face (loaded face) of the concrete member

- between tendons for anchorage zones with multiple anchorages or
- for anchorage zones where the tendon is eccentric from the member centreline

The bending moment is then divided by an assumed lever-arm (transverse to the free-body) of the resisting internal concrete forces. The lever-arm distance (based on experimental evidence) is calculated as follows

- For a single anchorage; by dividing the peak transverse moment by a lever-arm assumed to be one half the overall depth of the member.
- Between pairs of anchorages; by dividing the peak transverse moment by a lever-arm assumed to be 0.6 times the spacing of the anchorages.
- For more than two anchorages; by dividing the peak transverse moment by "n" where n is the number of anchorages.

Distribution of reinforcement

The total cross-sectional area of stirrups required is calculated by dividing the tensile force, T, by 150 MPa regardless of the yield stress of the steel. This is an attempt to limit the crack widths in the anchorage zone.

The bursting moment reinforcement is to be placed from $0.2D_e$ to $1.0D_e$. Determine a suitable combination of bar size and spacing over this length and then continue this all the way to the loaded face and an extra $0.2D_e$ past the $1.0D_e$ point. It is recommended that bar sizes of 20 mm or less be used. This is to forestall the designer using large bar sizes to give the required cross-sectional area but in so doing spread the spacing out too far give wider crack widths.

Spalling reinforcement is spread over $0.2D_e$ from the loaded face.

Example : Anchorage Zone

The end block for a rectangular post-tensioned concrete beam is 2400 mm deep and 400 mm wide, and has two anchorages which are spaced 1200 mm apart vertically (the top anchorage is 600 mm from the top of the section), and they are both 200 mm from the side face of the member. Anchorage plates are 300 mm square.

If the two anchorages are stressed simultaneously with jacking forces in each tendon of 1500 kN, design the end block reinforcement crossing the horizontal and vertical planes to satisfy the requirements of AS3600.

In each case symmetrical prisms D_e = 1200 mm

(i) Vertically

$T = 0.25 P_i (1 - k_r)$

$= 0.25 \times 1500 \times (1 - 300 / 1200)$

$= 381.3$ kN

$A_{st} = T / 150$

$= 281.3E3 / 150$

$= 1875$ mm^2

spread this steel over a length of
$0.2D_e = 240$ to $1.0D_e = 1200$ ie over 960 mm
therefore use 2 legs of Y12 (220 mm^2) which means we need a total of 9 stirrups.
ADOPT Y12 - 100 mm (from loaded face to 1200)

(ii) Horizontally

for each anchorage

$T = 0.25 \times 1500 \times (1 - 300 / 400)$

$= 93.8$ kN

$A_{st} = T / 150$

$= 93.8E3 / 150$

$= 625$ mm^2

using Y12 stirrups we require 3 off spread over $0.2D_e = 80$ to $D_e = 400$ ie provide at 100 c/c
ADOPT Y12 - 100 mm

T.16 Prestress Forces Imposed

T.16.1 From Angle Change

Wherever the slope of the tendon varies or at the ends of the tendon (anchorages) the tendon will impose loads upon the concrete. For ease of computation the value of the force used in all calculations is taken as the average force within the span length and design strip (column or middle for two-way systems).

Consider firstly a simply supported beam with a tendon having a "harped" or sharply kinked profile as shown in Figure T.16.1a). The tendon is brought out at the centroid at the ends and has an eccentricity from the centroid of "e" at mid span. A simple analogy is if we had a skipping rope and were to pull both ends whilst holding down the rope at mid length we will get an upward force generated at the mid length location. The value of the upward force is given by the change in slope of the tendon multiplied by the force in the tendon at that point. For small angles sin q = q and cos q = 1.0. The set of loads imposed by the curvature of the tendon is shown in Figure T.16.1d).

T.16.1.1 Forces in Spans

Consider now a simple parabola as shown in Figure T.16.3a). The offset from the straight line connecting the two high points is given by

$$y = 4h\left(\frac{x}{L} - \frac{x^2}{L^2}\right)$$

The slope of the line at any point is obtained by differentiating once with respect to x

$$\frac{dy}{dx} = \frac{4h}{L}\left(1 - \frac{2x}{L}\right)$$

= q

The curvature, the rate of change of the slope, is given by differentiating again with respect to x

f = -8h / L^2 = constant over the length of the parabola

If we consider a small length dx the angle change over that length is

d_q = f dx

and the upward force (normal to the slope of the tendon at that point) is given by

dF = P d_q
= P f dx

therefore
dF / dx = Pf

=-8Ph / L^2

which is the uniformly distributed load imposed normal to the tendon. For small angles of q (slabs and generally all beams) we assume that the force acts vertically.

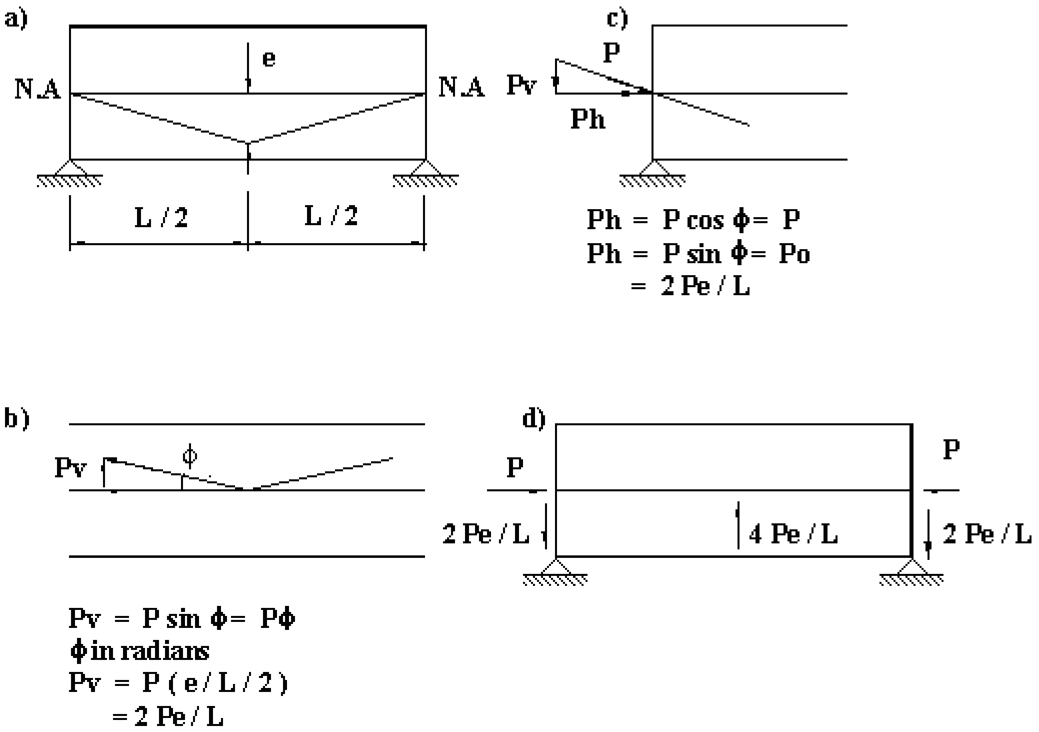


Figure T.16.1

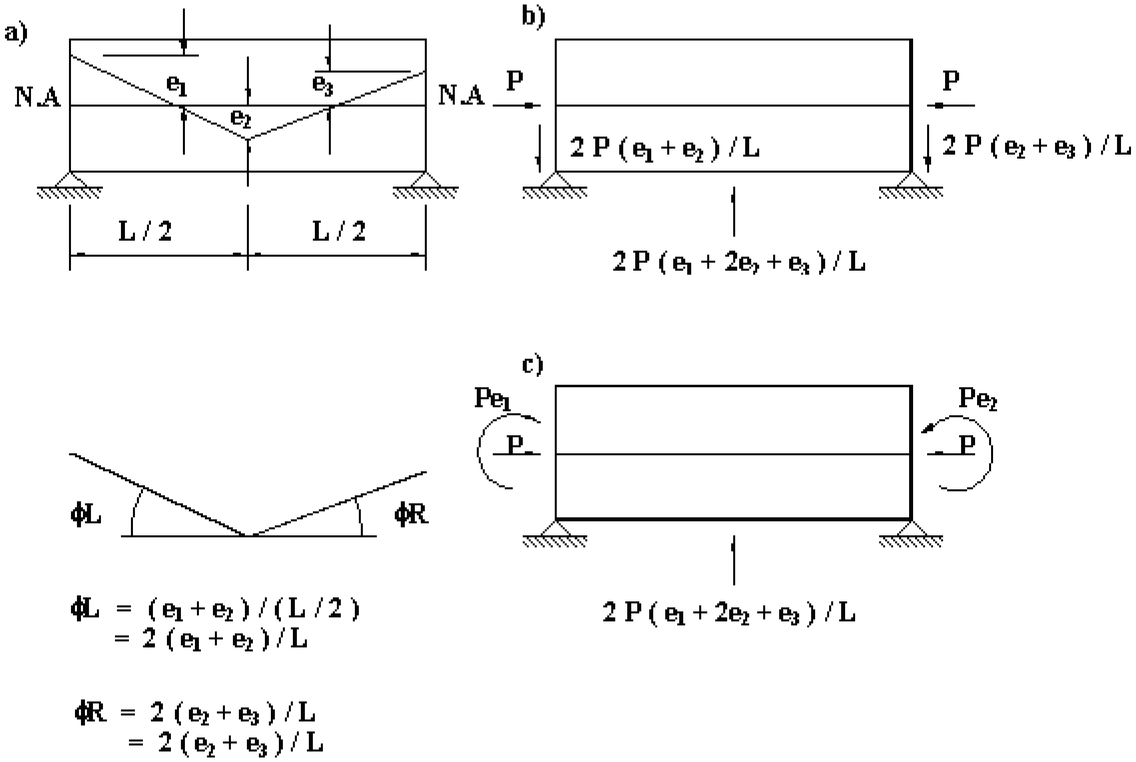


Figure T.16.2

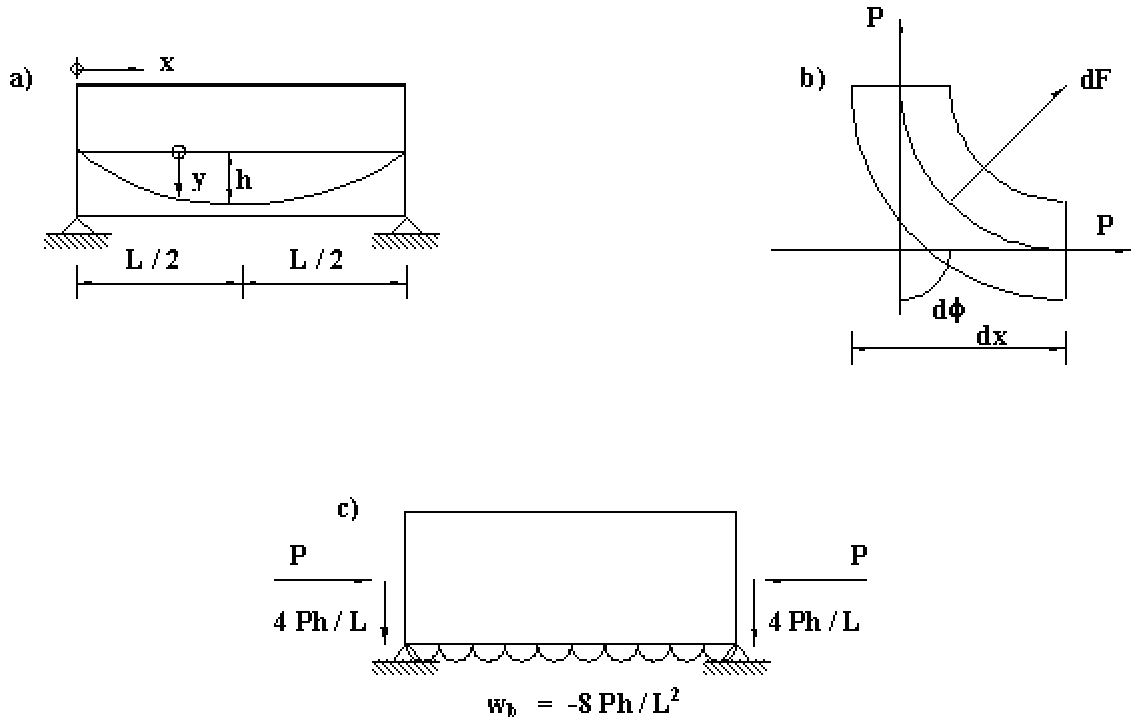


Figure T.16.3

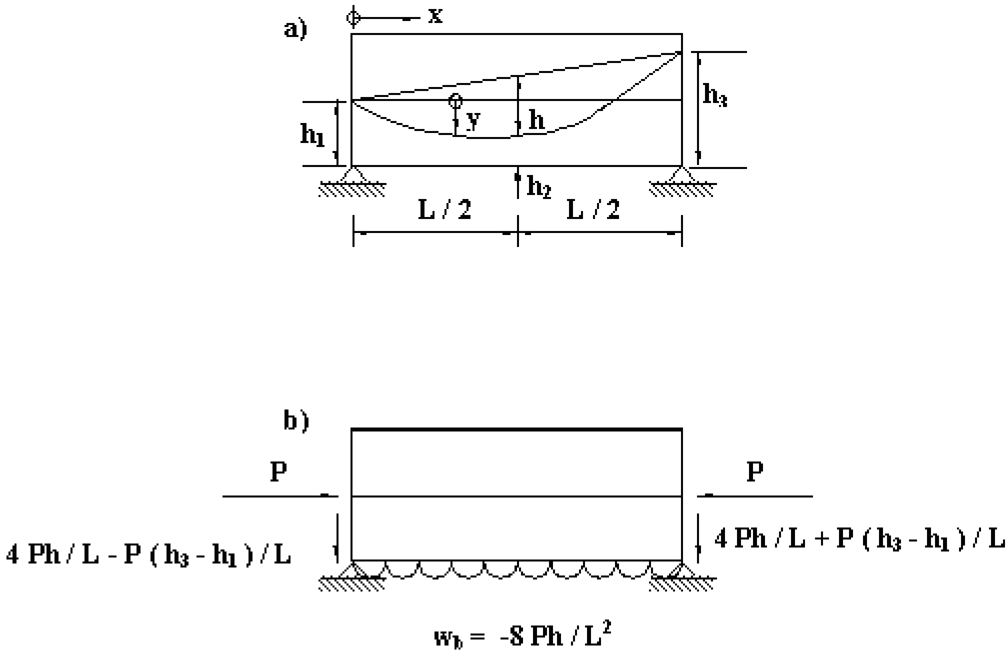


Figure T.16.4

At the anchorages the forces are obtained as previously ie the force components are obtained from the angle of the tendon multiplied by the force. In this case, at the anchorages, the slope is

$q = 4h / L$
therefore the vertical force is
$P_v = 4Ph / L$

Note this result may also be obtained by considering equilibrium of the span freebody diagram.

A more general case is that of a "skewed" parabola ie a parabola where the high points at either end of the drape are at different heights as in Figure T.16.4a). The rate of change in the angle of the tendon remains constant at -8h/L2 therefore the equivalent balanced load is given by

$w_b = -8Ph / L^2$

However the slope of the tendon has altered from the simple non-skewed case. The slope of the straight line connecting the high points is given by

$q = (h3 - h1) / L$ radians
which thereby induces an extra force of magnitude
$P_v = P(h3 - h1) / L$

This force must be added or subtracted from the force imposed by the slope of the non-skewed case depending upon the signs of the slopes as shown in figure T.16.4b).

It is worth noting at this point that if we were to lift the tendon up or down whilst retaining the same slopes then the distributed load would remain unaltered.

If the tendon is anchored eccentrically from the members centroid then a bending moment will be induced. This of course applies whether such an anchorage is located at the end of a member or within the member's length. There will also be an accompanying vertical force induced unless the tendon is terminated horizontally .ie. by making the low point (zero slope for this profile) of the parabola coincide with the anchorage eg this would occur typically for an anchorage located at the end of a cantilever.

T.16.1.2 Forces in Cantilevers

The most important thing to remember is that regardless of what one does with the profile within the span length of the cantilever, at the support, the moment balanced is simply given by the prestressing force multiplied by the eccentricity at that point, P.e.

Consider a distributed load along the full length of the cantilever.

M = P.e
= wL^2 / 2
where
L = the drape length within the cantilever span
w = the load balanced
e = the eccentricity from the centroid at the support.

If the drape did not extend the full length of the cantilever span then the bending moment induced from the sharp angle change at the end of the drape length will, in tandem with the load balanced within the drape length, give the same result as the above ie P.e at the support line. To calculate the value of the distributed load for this case consider the drape length as being half of a symmetrical parabola. Take the drape length as being twice the actual drape length ie the low point of the actual parabola is on the point of symmetry. The balanced load is then given by

M = wL^2 / 8
= P.h
where
L = twice the actual drape length.
h = the difference between the high and low point offsets.

The are no secondary prestress bending moments in cantilevers as they are caused by hyperstatic reactions of which there are none.

T.16.1.3 Forces due to Change in Centroids

Wherever there is a change in the location of the centroidal axis the tendon will impose loads upon the concrete. For ease of computation the value of the force used in all calculations is taken as the average force within each span length and the strip (column or middle for two-way systems). The centroid is also taken as that applicable to the strip being investigated.

Consider firstly the case of a sharp change in centroid as may occur at the edge of a drop panel as shown in Figure T.16.5 a). If we idealise the prestress force as acting along the line of the centroids it may be seen that a bending moment will be induced of magnitude

M = P (h2 - h1)

Similarly for the case of a sloping taper, as may occur at the edge of a band beam as shown in figure T.16.6a), a set of forces as shown in Figure T.16.6b) will be induced. We may consider this problem in two ways. Firstly it may be said that the two forces will be statically equivalent to the bending moment that would be induced if it were a sharp step similar to Figure T.16.5a). The value of the forces may then be calculated from

$$P_v = P\left(\frac{h_2 - h_1}{x}\right)$$

Alternatively we may consider the prestress force to be following a hypothetical centroidal axis location drawn between the two centroid locations at either end of the taper and thereby inclined at an angle q. The value of the vertical component of the prestress force will again be given by the equation above.

The second method may more readily be demonstrated by Figure T.16.7 which depicts a section of the same dimensions but inclined to each other. Note that in figure T.16.7 the force induced from the change in the centroid exactly counter-balances the force induced by the change in angle of the tendon.

T.16.2 From Anchorage Eccentricity

Consider now a sharply kinked tendon similar to Figure T.16.1 but having eccentricities at the anchorages as shown in Figure T.16.2a). The procedure for calculating the forces imposed by the tendon is similar to the previous example however with the further complication of different angles of inclination of the tendon on either side of the sharp central kink. The forces at the anchorages are more commonly replaced with the statically equivalent moments as shown in Figure T.16.2c).

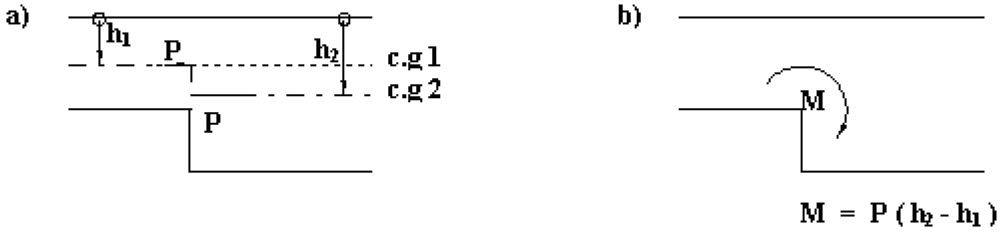


Figure T.16.5

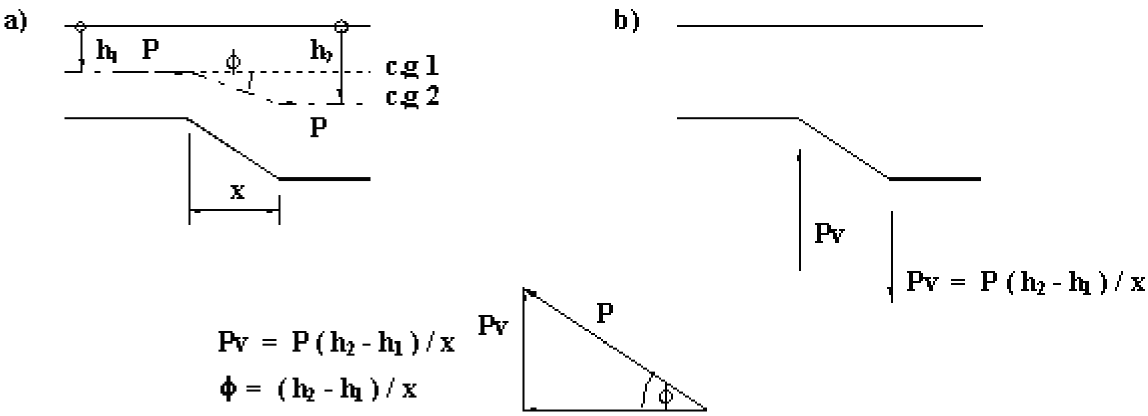


Figure T.16.6

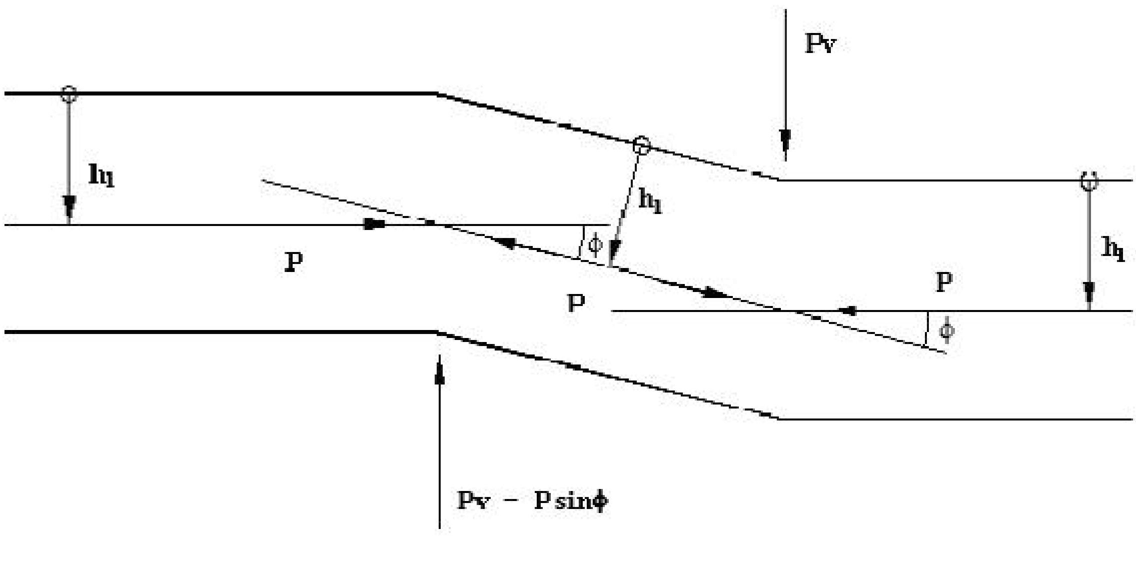


Figure T.16.7

T.17 Fibre Reinforced Polymers

As there is no real design code at this stage for the design of members using Fibre reinforced Polymers, we have provided the designer with the ability to control the design in several different ways to be consistent with the various suggested design methods. The designer is responsible for ensuring that the design requirements of his design method are being covered. As more formalised design guidelines are produced in each country we will modify RAPT to include these. To date, we have based the logic available in RAPT on methods discussed in

- SSUK - TR55 - Design guidance for strengthening concrete structures using fibre composite materials by the British Concrete Society
- FIB Bulletin 14 - Externally Bonded FRP reinforcement for RC structures
- ACI 440-F

The main design points are listed below

1. RAPT does not currently allow for a development length for FRP reinforcement. The reinforcement is assumed to be fully bonded over the length defined. The designer needs to add an appropriate development length when detailing the design. The logic for this is covered in the design guides. RAPT does not do any specific checks on horizontal shear, end peeling or the effects around cracks or uneven surfaces, all of which is discussed in the design guides. However, RAPT will allow the designer to specify a maximum strain in the FRP reinforcement to limit bond failures (see below).
2. The capacity of FRP sheet is controlled by the overall capacity of the material and the bond of the material to the concrete. RAPT allows the designer to control both of these.
    - Sheet Capacity:- FRP sheets are not ductile. They have no plastic region in their stress/strain curve. Also, the properties of the sheet are variable and the capacity depends a lot more on installation procedures than normal reinforcing. Consequently material factors are significantly higher than those for normal reinforcing and capacity reduction factors much lower.

      Material factors:- Because of this, the logic of material factors for them is more complex. This is explained in detail in TR55. They suggest multiple material factors for both strength and elastic modulus. These are then combined into a single material factor for strength and a separate one for elastic modulus. RAPT allows the user to define these separate material factors for strength and elastic modulus. Note that in calculating the material factors TR55 includes the elastic modulus material factors in the calculation of the strength material factor.

      Capacity Reduction Factors:- ACI 440 suggest either the use of separate capacity reduction factors for different materials or the use of a hybrid factor that varies depending on the proportion of the strength being provided by the FRP compared to other tensile reinforcement. If they designer wishes to use a hybrid factor, they can define this in the normal RAPT input screens, but this will be a fixed figure for something that will vary within a span and in different spans. We have also modified RAPT to allow the definition of different Capacity Reduction Factors for different tensile materials. These actually operate internally as a material factor. If the Material Capacity Reduction Factor is less than the overall Capacity reduction factor, RAPT will internally introduce a Material factor for that material equal to

      Material Factor = Capacity Reduction Factor / Material capacity reduction Factor.

      This factor will be applied to the force from that material before the overall Capacity Reduction Factor is applied to the overall capacity. If there is already a material factor defined for this material, it will be multiplied by the above factor if a capacity reduction factor is also defined, so make sure you do not double up on factors.

      A value of -1 for any of these factors means that the factor will be ignored.
    - Bond and anchorage:- Bond will normally limit the design strength available from the FRP sheet. RAPT has given the designer the option to limit the maximum strain in the FRP sheet to a percentage of the breaking strain. This allows the designer to limit the strain ion the FRP sheet and thus the bond stresses as is suggested in the above guides to ensure adequate bond. This option is in the materials data. When defining the material properties, the designer should ensure that this data item is set correctly in accordance with the set of design rules he is using. There is also now an option in Design Data - Ultimate to limit the reinforcement strain in the design. This must be turned on manually by the designer, but RAPT will insist on it being turned on for design using FRP sheet.
3. FRP reinforcement is normally applied when a member is already in service. Because of this there is already some strain in the concrete to which the FRP is being attached and which will not induce strain in the FRP. To allow for this, RAPT allows the designer to nominate a Pre-Existing load combination. Any user defined reinforcing layer can then be nominated as being applied after the Pre-Existing load condition. Any reinforcement applied after this load condition will be assumed to have zero strain in it when the strain in the concrete is at this pre-existing strain and the strain in that reinforcement will increase from that point, rather than from zero strain in the concrete.

T.18 Metal Decking

Australia has no design code for the design of concrete members with metal decking used to provide strength. Design of slabs with Metal Decking for Australian buildings is covered in Design Booklet 31. Design of Composite Slabs for Strength, published by Onesteel Manufacturing Pty Ltd in 2001.

The design methodology included in RAPT also conforms with the design logic in Eurocode 4.

The major difference in RAPT's treatment of design for composite metal decking is that RAPT treats the decking as a series of webs and flanges. The strain profile through the depth of the decking is determined and resulting stress and force profiles calculated. Flanges are assumed to have no variation in strain over their depth, which is the nominal sheet thickness. Flanges are assumed to have linearly varying strain over their depth. In all cases the peak strain is limited to the yield strain of the material and the maximum strain that can be developed assuming bond between the decking and concrete has failed and that horizontal shear can be transmitted between the decking and the concrete due to friction and mechanical interlock.

To be considered in RAPT as structural composite decking, the decking must have been tested in accordance with one of these documents or similar and have defined friction/interlock performance to quantify the interaction between decking and concrete. The assumption of bond is not acceptable under ultimate strength conditions.

12 References

1. "AS 3600" clauses 7.5.4 and 8.8.2
2. "Notes on ACI 318-77" PCA, edited by Gerald B Neville. 1980
3. "Commentary to ACI 318-77" clause 13.7.5
4. "AS 1480" clause 21.5.1.4c)
5. "AS 3600" clause 9.1.2
6. "AS 3600" clause 7.1.2
7. "AS 3600" clause 7.6.10
8. "AS 3600" clause 8.5.3.1
9. "AS 1480" clause 10.1.3.1
10. "Design of Prestressed Concrete" R.I. Gilbert, N.C. Mickleborough (1990)
11. "AS 3600" clause 6.1.1.2
12. "Minimum thickness requirements for control of two-way slab deflections." Thompson D P, Scanlon A, ACI Structural Journal Title No. 85-S2, Jan-Feb 1988
13. "Tentative recommendation for Prestressed concrete flat plates" Journal of the ACI, Proceedings, Vol 71, No. 2, Feb 1974
14. ACI clause 13.7
15. "Design Procedures for computing deflections" Branson D E, ACI Journal Sept 1968, No. 65-53, pp 730-742
16. "A study of combined bending and axial load in reinforced concrete members." Hognestad E, Univ of Illinois, Eng expt station Bull, No. 899
17. "Design factors for the equivalent frame" Simmonds S, Misic J, ACI Journal Vol 68 Nov 71
18. "Continuous frames of reinforced concrete" Cross, Hardy and Morgan, Newlin-Dolbey, John Wiley and Sons Inc, New York 1954
19. "Structural design to the ACI building code", Rice P F, Hoffman E S, Van Nostrand Reinhold New York
20. "Design of slab systems" Drewe I S, Master of Eng Sc thesis, Univ Sydney 1986
21. "Ductility of Simply-Supported Composite Beams", Patrick, M and Bridge, R. AISC Steel Construction. Vol. 22 No. 2 May 1988
22. "PCI Design Handbook" 2nd edition p3-63
23. "ACI Manual of Concrete Practice Part 3" 1982 clause 11.4.2.2
24. "Reinforced Concrete" 2nd edition Warner Rangan & Hall p353
25. "Prestressed Concrete" Warner & Faulkes p208. Longman Cheshire 1989
26. "Deflections of Continuous Beams", ACI Committee 435, sub-committee 7, ACI Journal, Dec 1973
27. "Approximate Calculation of Deflections of Two-Way Floor Systems", Nilson A.H & Walters B.D, ACI Journal May 1975
28. "State-of-the-art Report, Deflection of Two-Way Floor Systems", special publication SP43, ACI Detroit Michigan 1974
29. "Short-Term Deflections of Flat Plates, Flat Slabs and Two-Way Slabs", Kripanarayanan K.M & Branson D.E. ACI Journal 1976
30. "Long-Term Multipliers For Estimating Two-Way Slab Deflections", ACI Journal Dec 1986, Graham C & Scanlon S
31. "Initial and Long-Term Deflections of a Reinforced Concrete Flat Plate Structure", paper No.2794 Conference on the Deformation of Concrete and Concrete Structures", the Institution of Engineers Australia, Brisbane September 1969
32. "Unified procedures for predicting the Deflection and Centroidal Axis Location of Partially Cracked, Non-Prestressed and Prestressed Concrete members", Branson D.E and Trost H, ACI Journal March/April 1982
33. AS1480-1974, clause 10.1.4
34. "Control of Beam Deflections by Allowable Span-to Depth Ratios", Rangan V, ACI Journal Sept-Oct 1989
35. "Reinforced Concrete" Warner Rangan & Hall, Chapter 6
36. "Ultimate Analysis of Prestressed and Partially Prestressed Sections by Strain Compatibility", Naaman A.E, PCI Journal, Jan-Feb 1977
37. "An Interactive Analysis and Design Program for Rectangular Reinforced Concrete beams",Van Leeuwen M.J & Wilson J.K, under-graduate thesis, University of Technology, Sydney, 1988
38. Ibid., Chapter 1, p2. and also Dan E Branson, op. cit. Chapter 1, p3
39. BS8110
40. "Reinforced Concrete Designer's Handbook", Charles E Reynolds and James C Steadman
41. Trost, H., "Auswirkungen des Superpositionsprinzips auf Kriech- und Relaxations- Probleme bei Beton und Spannbeton", Beton- und Stahlbetonbau, 62, No 10, 1967, pp 230-238, No 11, 1967, pp 261-269.42
42. Bazant, Z.P., "Predicition of Concrete Creep Effects using Age-Adjusted Effective Modulus Method", ACI Journal, 69, April 1972, pp 212-217.43
43. R.I.Gilberts, "Time effects in concrete structures", Elsevier 1988, pp 59-7644
44. "CEB-FIP Model Code 1990", Thomas Telford ISBN: 0 7277 1696 4, pp 144

13 Appendix

13.1 Local Adaptation of Codes

13.2 Example RAPT Runs