Portal Frame Design Release 4.0

Fastrak Portal - Design of Portal Frame Buildings

Disclaimer

Computer Services Consultants (UK) Limited does not accept any liability whatsoever for loss or damage arising from any errors which might be contained in the documentation, text or operation of the programs supplied.

It shall be the responsibility of the customer (and not CSC):

- to check the documentation, text and operation of the programs supplied,
- to ensure that the person operating the programs or supervising their operation is suitably qualified and experienced,
- and to ensure that program operation is carried out in accordance with the user manuals,

at all times paying due regard to the specification and scope of the programs and to the CSC Software Licence Agreement.

Proprietary Rights

Computer Services Consultants (UK) Limited, hereinafter referred to as the **OWNER**, retains all proprietary rights with respect to this program package, consisting of all handbooks, drills, programs recorded on ***CD*** and all related materials. This program package has been provided pursuant to an agreement containing restrictions on its use.

This publication is also protected by copyright law. No part of this publication may be copied or distributed, transmitted, transcribed, stored in a retrieval system, or translated into any human or computer language, in any form or by any means, electronic, mechanical, magnetic, manual or otherwise, or disclosed to third parties without the express written permission of the **OWNER**.

This confidentiality of the proprietary information and trade secrets of the **OWNER** shall be construed in accordance with and enforced under the laws of the United Kingdom.

Trademarks

Fastrak Portals® is a registered trademark of Computer Services Consultants (UK) Limited,

Microsoft is a registered trademark,

Windows is a trademark of Microsoft Corporation.

Credits

Computer Services Consultants (UK) Limited would like to acknowledge:

- the assistance of the DTI, through the Eureka CIMsteel project, in the research and development of Fastrak Portals.

-

Table of Contents

Portal Frame Design
Release 4.0

Portal Frame Design Release 4.0

Table of Contents

User's Guide

Portal Modeller

Engineer's Handbook

User's Guide

Portal Frame Design
Release 4.0

1 Introducing Portal Frame Design

Introducing Portal Frame Design features

Welcome to ***Portal Frame Design***, an application which allows you to define, design and/or check a comprehensive range of portal frames.

Portal Frame Design integrates seamlessly with other ***Fastrak*** applications such as ***Moment Connection Design***[1], ***Column Base Design***[1], ***Wind Load Generator***[1], ***Snow Load Generator***[1] and ***Crane Beam Design***[1].

Portal Frame allows you to:

- set preferences for the portals you define to minimise the time you spend defining the frames details,
- define single span or multi span portals of the following types:
  - Symmetric pitched, Asymmetric pitched, Monopitch, Propped, Flat Top, Mansard.
- add supplementary features to the spans that you define as follows:
  - Haunches, Cranes, Ties, Floors, Parapets,

Note Some features are not appropriate for some span types.

- define complete portal buildings with orthogonal grids containing a number of different frames of any of the above types,
- create a series of loadcases containing loads that are applied to:
  - the entire building including:

1. *These are additional applications which you must purchase separately to the main* ***Portal Frame Design*** *program.*

- dead, imposed and service loads,
- wind loads calculated from the geometry of the building, (this requires the optional ***Wind Load Generator***[1] add-in),
- snow loads calculated from the geometry of the building, (this requires the optional ***Snow Load Generator***[1] add in).

- individual frames including:
  - area loads, point and line loads and point couples (these can be used to model the effects of wind and snow loading on the structure in the absence of the optional ***Wind Load Generator***[1] and ***Snow Load Generator***[1] add-ins).

- incorporate individual applied load cases (Dead, Imposed, Crane and Wind types are available) to produce combinations that will be used in the design (***design combinations***). ***Portal Frame*** will automatically calculate the factors for each combination based on the type of the applied load cases that it contains (the calculated factors can be changed as necessary),
- define and position cladding rails.
- design or check the building in accordance with current design codes, including:
  - overall member strength checks at ***Ultimate Limit State*** (ULS),
  - local member strength checks at ULS,
  - design to limit deflections at ***Serviceability Limit State*** (SLS),
  - local and overall frame stability checks,
- design or check the individual members for local stability utilising the position and type of any restraints that you have defined, and adding new ones as required to make the member pass these checks,

1. *These are additional applications which you must purchase separately to the main* ***Portal Frame Design*** *program.*

- design any or all of the connections using ***Fastrak Portal Connection Design***, the resulting designs being held as part of the current frame design,
- design any or all of the column bases using ***Fastrak Column Base Design***, the resulting designs being held as part of the current frame design,
- produce a report of your design, containing only those calculations that you deem necessary for each separate part of the design, including:
  - a summary,
  - a pictorial representation of the plastic hinges, the frame and member bending moments, the frame and member shear forces, the frame and member axial loads,
  - the hinge history leading to collapse,
  - the ULS strength checks,
  - the SLS deflections,
  - the frame stability checks,
  - the member stability checks,
  - the results of the connection designs that you have performed,
  - the results of the fire designs that you have performed,

Introducing Portal Frame documentation

Portal Frame comes with the following online information:

Online Help - Procedural steps for all *Portal Frame* based on the text of the ***User's Guide*** and an ***Engineer's Handbook*** that tells you what theory and assumptions the application is using.

Electronic Manual - The ***User's Guide*** and ***Engineer's Handbook*** are provided in electronic format as an *Adobe Acrobat* (`.pdf`) file. The current version of the *Adobe Acrobat Reader* is provided on the CD.

2 Getting Started

Checking your package

This section lists your *Portal Frame* package's contents. Please check it now and if it is not complete contact your software dealer or supplier.

Discs

Your *Portal Frame* package contains a CD.

System requirements

This section describes the hardware, system software, memory and disc space requirements that your computer needs in order to run *Portal Frame*.

Hardware

Portal Frame requires a computer capable of running *Microsoft Windows*[1].

System software

You must have *Windows* installed and running on your computer to install and use *Portal Frame*.

Memory

Portal Frame needs 32 MB of RAM although 64 MB is preferable.

Disc space

Portal Frame requires a minimum of 125 MB of available hard disc space.

What next ?

If you like reading, the following chapters cover all aspects of *Portal Frame*. Alternatively you can start using *Portal Frame* and only refer to the manual if you have a specific query.

1. *Either **Microsoft® Windows 98 ™** (or higher) or **Microsoft® Windows NT ™ Version 4.0** (or higher). All versions are referred to by the term **Windows**. **Portal Frame** may not run correctly on **Windows** emulators.*

Note Remember that extensive online help is available when you are running *Portal Frame*. Simply select *Help / Help Topics* or press *F1*.

3 Starting to use Portal Frame

Launching Portal Frame

In order to launch *Portal Frame* you must already have *Windows* running.

To launch Portal Frame

1. Click **Start** on the *Windows Taskbar* that appears along one edge of your *Windows* screen. A menu appears with various options.
2. Click the *Programs* option. This will show another list or table of all the applications that are installed on your system.
3. Click the *Fastrak* option to see all the *Fastrak* applications.
4. Click the *Portal Frame* option to launch the application.

Becoming familiar with Portal Frame

In *Portal Frame* you always work on a *Project* which can contain all the frames for a particular job[1].

For each *Project* there are three distinct windows:

- the *Structure* window which shows a graphical representation of your entire structure, based on the individual frames you have defined and placed in your project,
- the *Project Workspace* window which shows all the frames that the project contains,

1. *Limited only by the memory and resources available on your computer.*

- the ***Report*** window which shows what your report will contain and how it will look when printed. You can ensure that your report is complete before you print it, transfer it to ***TEDDS*** or to ***Microsoft Word***.

For each ***Frame*** in the project there are four distinct windows:

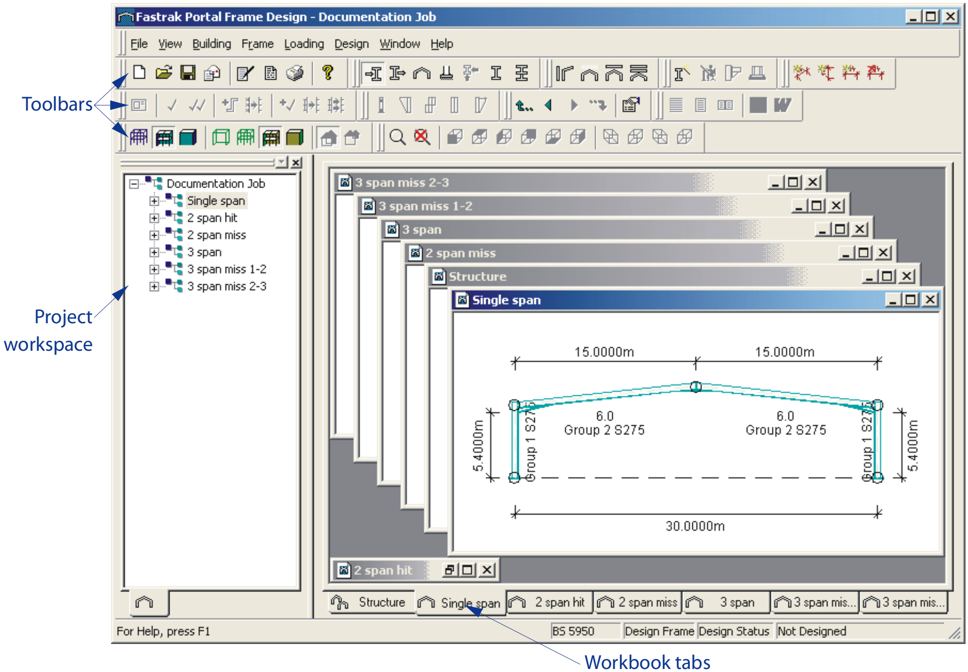
- The ***Geometry*** window shows a graphical representation of your frame. You can click on specific items to modify particular details quickly.
- The ***Loading*** window shows a graphical representation of the loads on your frame. You can choose the load case whose details you want to see using the ***Project Workspace***.
- The ***Results*** window shows the hinge history, shear force, bending moment and deflection diagrams for any loadcase or design combination. You choose the diagram which you want to see using the ***Analysis*** toolbar, and the load case using the ***Project Workspace***.
- The ***Stability*** window shows a single member of your frame with its bending moment. You can also see the restraints you have specified, the stability checks you have performed for any design combination which you choose using the ***Project Workspace***.

In addition to these frame based windows, there is a ***Design Summary*** property sheet for each frame. This shows that frame's design results including the status of each check, you can use the ***Design Summary*** to see particular results in more detail.

If you are a first time user it is a good idea to take a few seconds to acquaint yourself with each of these views. Familiarise yourself with the terms and concepts introduced in the following sections before you start using the application. Such familiarisation will reward you in the long run, because you will be able to use ***Portal Frame*** quickly and efficiently.

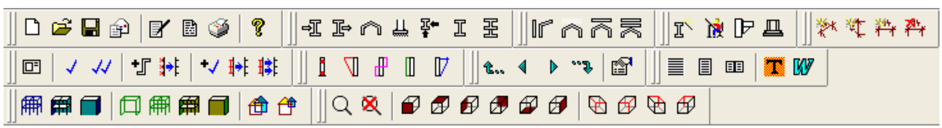
Portal Frame Main window

You will see an annotated representation of the *Portal Frame* main window below.

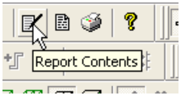


Toolbars

These show sets of icons which are short-cuts to many *Portal Frame* commands.



Tip If you let the pointer rest over an icon for a short while you will see a ***Tool Tip*** explaining that icon's function.

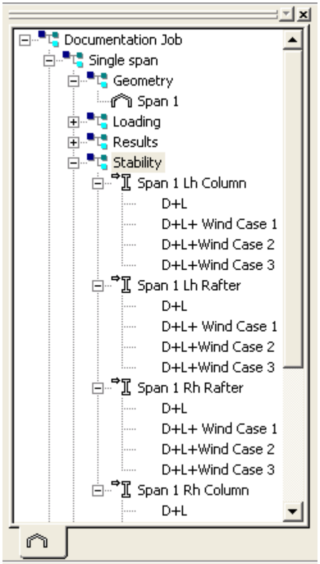


Help For further details on **Toolbar** icons ***see*** *"Portal Frame Toolbars"* on page 38.

Help You can show or hide the **Toolbars** at will, ***see*** *"Controlling the Toolbars"* on page 100.

The Project workspace window

This window shows all the frames that you have defined in the project and the available windows for each frame.



You can also use the ***Project Workspace*** to perform many actions in ***Portal Frame***, for example you can:

- change your project's details,
- change a frame's reference,
- open a particular window for a frame.

? **Help** For further information ***see*** *"Using the project workspace"* on page 509.

Workbook tabs

These show the open windows. You can select a particular view by clicking on its tab.

Help You can show or hide **Frame Tabs** at will, *see* *"Controlling the Workbook tabs"* on page 101.

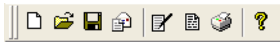
Tip You can open a frame by double clicking its *Project Workspace* reference.

Portal Frame Toolbars

The ***Toolbars*** provide instant access to many features.

There are fourteen ***Toolbars***:

- the ***Standard*** toolbar,



- the ***Building*** toolbar,

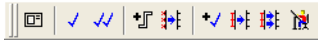
- the ***Frame*** toolbar,

- the ***Loading*** toolbar,

- the ***Design*** toolbar,

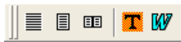
- the ***Analysis*** toolbar,

- the ***Stability*** toolbar.

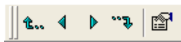


- the ***Dimensions*** toolbar.

- the ***Report*** toolbar,



- the ***View*** toolbar,



- the ***Graphics*** toolbar,

- the ***Scheme*** toolbar.

- the ***Select*** toolbar.

- the ***Animate*** toolbar.

All the toolbars operate in a similar manner:

- they all contain icons that you choose to access particular features,
- they can all be switched on or off at will,
- they can all be placed wherever you like on the screen,
- You can dock the toolbars against any edge of the ***Portal Frame*** window,
- If a toolbar is not docked you can change its shape.

Help For further information on the toolbars ***see***:

- *"The Standard Toolbar"* on page 41,
- *"The Building toolbar"* on page 42,
- *"The Frame Toolbar"* on page 43,
- *"The Loading Toolbar"* on page 45,
- *"The Design Toolbar"* on page 45,
- *"The Analysis Toolbar"* on page 46,
- *"The Stability Toolbar"* on page 47,
- *"The Dimensions toolbar"* on page 48,
- *"The Report Toolbar"* on page 49,
- *"The View Toolbar"* on page 49,
- *"The Graphics toolbar"* on page 53,
- *"The Scheme toolbar"* on page 55,
- *"The Select toolbar"* on page 57,
- *"The Animate toolbar"* on page 71.

Note The corresponding toolbar icon is shown to the left of the descriptive text later In this ***User's Guide*** where the functions are covered in more detail.

To move a toolbar to a particular position

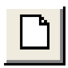
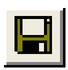
1. Point between icons on a toolbar (or the toolbar title for a floating toolbar).
2. Hold down the left mouse button and drag the toolbar to a new location. If you drag it to the edge of the window, the toolbar will dock to the edge of the window automatically. If you drag it to any other point on the screen the toolbar will become a ***floating toolbar*** at that position.

To change the shape of a floating toolbar

1. Point over the edge of the toolbar, and then, holding the left mouse button down drag the edge of the toolbar. The shape of the toolbar will change in steps which fit the icons that have to be displayed most efficiently.

The Standard Toolbar

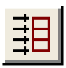
Gives you access to the following features:

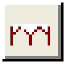
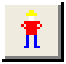
Icon	*Name*	*Function*
	**New project**	Create a new project, give the project details, define the building details and set the details for the first portal frame in the project – ***see*** ***"To define a new project"*** on page 145.
	**Open project**	Open a previously saved project recovering its details from disk – ***see*** ***"To open a project"*** on page 155.
	**Save project**	Save the current project to disk. You can recall it later for amendments or to perform additional calculations – ***see*** ***"To save a project for the first time"*** on page 154 and ***"To save a project which has been changed"*** on page 155.

Icon	Name	Function
	**Send mail**	Send the content of the current project as an email attachment – *see "To send mail"* on page 157.
	**Report contents**	Select the information that you want to include in your report– *see "To set the content of the frame design report"* on page 488.
	**Report view**	See the report as it will appear when printed – *see "To view a frame design report"* on page 500.
	**Print project**	Print the entire report, or individual pages – *see "To print the report"* on page 504.
	**About**	Show *Portal Frame's* release details.

The Building toolbar

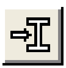
Gives you access to the following features:

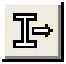
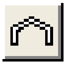
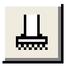
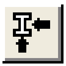
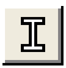
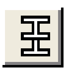
Icon	Name	Function
	**Frame Grid**	Allows you to manipulate your building grid – *see "To define grid details"* on page 175.
	**Building Load**	Allows you to manipulate your building loads – *see "To modify the building loads"* on page 160.
	**Building Wind Load**	Allows you to manipulate your building wind loads – *see "To modify the wind loads"* on page 161.

Icon	Name	Function
	Building Snow Load	Allows you to manipulate your building snow loads – *see "To modify the snow loads"* on page 162.
	Design all Frames	Allows you to design all the frames in your project – *see "To design or check all frames in a project"* on page 429.
	Floor Wizard	Allows you to create a floor over a part, or over all your structure – *see "To use the floor wizard"* on page 541.
	Hip Wizard	Allows you to create a hip at a gable end of your structure, or even within the length of it – *see "To use the hip wizard"* on page 562.
	Building Cold-Rolled Wizard	Allows you to convert the restraints on the members of your structure into a series of appropriately positioned cold-rolled sections which you can then export to a DXF file or to *3D+* – *see " To use the cold rolled sections wizard"* on page 604.

The Frame Toolbar

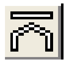
Contains the following icons:

Icon	Name	Function
	Design frame mode	Sets *Portal Frame* into *Design Frame Mode* (*Portal Frame* will choose those sections which result in a satisfactory design).

Icon	*Name*	*Function*
	**Check frame mode**	Sets ***Portal Frame*** into ***Check Frame Mode*** (***Portal Frame*** will check sections that you specify to see if they are adequate).
	**Frame span geometry**	Allows you to define the geometry of the current frame – ***see*** ***“To define span geometry”*** on page 181.
	**Frame base fixity**	Allows you to edit the details for all bases in the current frame – ***see*** ***“To define base properties”*** on page 209.
	**Frame valley beams**	Allows you to edit the details for all valley supports in the current frame – ***see*** ***“To define valley beam properties”*** on page 218.
	**Frame member properties**	Allows you to set the section details for each member in the current frame – ***see***: • ***“To define rafter properties”*** on page 240, • ***“To define column properties”*** on page 245, • ***“To define valley beam properties”*** on page 264, • ***“To define tie properties”*** on page 252, • ***“To define parapet properties”*** on page 260.
	**Frame haunch properties**	Allows you to set the fabrication type and details for each haunch and end plate of the current frame – ***“To define haunch properties”*** on page 280.

The Loading Toolbar

Accesses the following features:

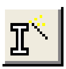
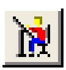
Icon	*Name*	*Function*
	**Wind loading**	Links to the *Wind Load Generator*[a] – *see "Wind Load Generator"* on page 369.
	**Snow loading**	Links to the *Snow Load Generator*[b] – *see "Snow Load Generator"* on page 401.
	**Frame loading**	Define the applied loading for the frame – *see "To define a frame loadcase"* on page 292.
	**Design combinations**	Create design combinations from your loadcases – *see "To define design combinations"* on page 406.

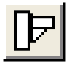
a. *The **Wind Load Generator** is a separate application which you need to purchase separately.*

b. *The **Snow Load Generator** is a separate application which you need to purchase separately.*

The Design Toolbar

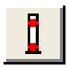
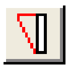
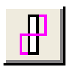
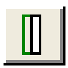
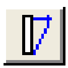
Allows you to access the following features:

Icon	*Name*	*Function*
	**Design wizard**	Change various factors which influence the design – *see "To set design checks"* on page 415.
	**Design frame**	Designs the portal frame automatically or checks the adequacy of a frame that you have defined – *see "To perform the design"* on page 428.

Icon	*Name*	*Function*
	**Connection design**	Allows you to design a connection using ***Moment Connection Design*** – *see* ***“Sharing details with Moment Connection Design”*** on page 512.
	**Base plate design**	Allows you to design a base using ***Column Base Design*** – *see* ***“Sharing details with Column Base Design”*** on page 516.

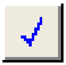
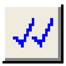
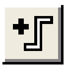
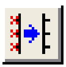
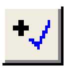
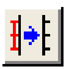
The Analysis Toolbar

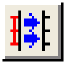
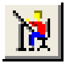
Allows you to access the following functions:

Icon	*Name*	*Function*
	**Hinge history**	Show the position of the plastic hinges around the frame.
	**Bending moments**	Show the bending moments.
	**Shear forces**	Show the shear forces.
	**Axial loads**	Show the axial loads.
	**Deflections**	Show the deflections.

The Stability Toolbar

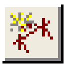
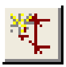
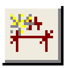
Gives you access to the following features:

Icon	Name	Function
	**Check member summary**	Allows you to view a summary of the results of the stability checks for all members under the current design combination – *see "To view the results summary for a design combination"* on page 480.
	**Check current combination**	Performs the stability checks that you have defined for the current combination.
	**Check all combinations**	Performs the stability checks that you have defined for all combinations.
	**Member restraints**	Shows the *Restraints* dialog for the current member – *see "Defining restraints"* on page 445.
	**Copy restraints to members**	Copies the restraints from the current member to other members – *see "To copy restraints between frame members"* on page 464.
	**Stability checks**	Shows the *Stability Checks* dialog for the current member – *see "Defining checks"* on page 466.
	**Copy stability checks to members**	Copies the stability checks from the current member to other members – *see "To copy stability checks to a different member"* on page 475.

Icon	Name	Function
	**Copy stability checks to design combinations**	Copies the stability checks from the current design combination to other design combinations – *see* ***"To copy stability checks to a different combination"*** on page 476.
	**Perform automatic stability checks**	Performs automatic stability checks for the current member – *see* ***"To automatically check a member's stability"*** on page 442.

The Dimensions toolbar

Gives you access to the following features:

Icon	Name	Function
	**Create dimension**	Allows you to create a dimension parallel to two points you choose – *see* ***"To define a dimension parallel to two points"*** on page 237.
	**Create vertical dimension**	Allows you to create a dimension giving the projected vertical dimension between two points you choose – *see* ***"To define a vertical projected dimension between two points"*** on page 238.
	**Create horizontal dimension**	Allows you to create a dimension giving the projected horizontal dimension between two points you choose – *see* ***"To define a horizontal projected dimension between two points"*** on page 238.

Icon	Name	Function
	Delete dimension	Allows you to delete a dimension from your structure – *see* ***"To delete a dimension"*** on page 239.

The Report Toolbar

Provides the following functions:

Icon	Name	Function
	Page width	Sets the display so that the width of the report takes up the full width of the report window – *see* ***"To view in page width format"*** on page 502.
	Single page	Sets the display so that a single page of the report appears in the report window – *see* ***"To view in full page format"*** on page 503.
	Double page	Sets the display so that two pages of the report appear in the report window – *see* ***"To view in double page format"*** on page 504.
	Export to TEDDS	Allows you to export the report for inclusion in a *TEDDS* calculation document – *see* ***"To transfer the report to TEDDS"*** on page 504.
	Export to Word	Allows you to export the current report directly to *Microsoft Word* – *see* ***"To transfer the report to Microsoft Word"*** on page 505.

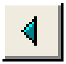
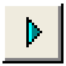
The View Toolbar

This toolbar controls all views on the screen, its functionality depends on the active window.

Note If an option is not possible, then the appropriate view icon is dimmed.

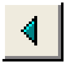
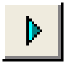
Example If you are already viewing the first page of a report, then both the **First** and **Previous** icons will be dimmed.

Frame Definition window

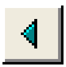
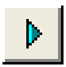
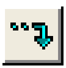
Icon	Name	Function
	First	Shows the first available view of the frame.
	Previous	shows the previous view of the frame.
	Next	shows the next view of the frame.
	Last	Shows the last view of the frame.
	Properties	Shows the ***Properties*** dialog which allows you to control the content of the ***Frame Definition*** window display.

Analysis Results window (excluding hinge history)

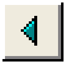
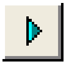
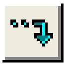
Icon	Name	Function
	First	Shows the first available view of the ***Analysis Results*** window.

Icon	Name	Function
	**Previous**	shows the previous view of the ***Analysis Results*** window.
	**Next**	shows the next view of the ***Analysis Results*** window.
	**Last**	Shows the last view of the ***Analysis Results*** window.
	**Properties**	Shows the ***Properties*** dialog which allows you to control the content of the ***Analysis Results*** window.

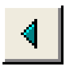
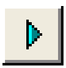
Analysis Results window (hinge history)

Icon	Name	Function
	**First**	shows the first hinge event that occurs for your frame.
	**Previous**	shows the previous hinge event that occurs for your frame.
	**Next**	shows the next hinge event that occurs for your frame.
	**Last**	Shows the last hinge event that occurs for your frame.
	**Properties**	Shows the ***Properties*** dialog which allows you to control the content of the ***Analysis Results*** window.

Member stability window

Icon	*Name*	*Function*
	**First**	Shows the first available view of the ***Member Stability*** window.
	**Previous**	shows the previous view of the ***Member Stability*** window.
	**Next**	shows the next view of the ***Member Stability*** window.
	**Last**	Shows the last view of the ***Member Stability*** window.
	**Properties**	Shows the ***Properties*** dialog which allows you to control the content of the ***Member Stability*** window.

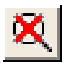
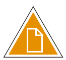
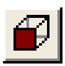
Report window

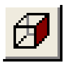
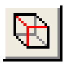
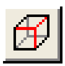
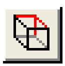
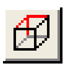
Icon	*Name*	*Function*
	**First**	Shows the first page of the report.
	**Previous**	- shows the previous page of the report.
	**Next**	shows the next page of the report.

Icon	Name	Function
	**Last**	Shows the last page of the report.
	**Properties**	Shows the ***Properties*** dialog (which informs you that there are no properties for the ***Report*** window!).

The Graphics toolbar

Provides access to the following features:

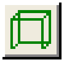
Icon	Name	Function
	**Zoom Area**	Allows you to see a particular part of the beam in more detail interactively from the graphical display – *see* ***"To zoom into an area of the display"*** on page 124.
	**Note**	*You can use the zoom tool progressively to see a part of the display in more and more detail.*
	**Zoom Extents**	Returns the view of the ***Analysis Results*** window to its initial state (not zoomed) – *see* ***"To return a 2D window to its initial state"*** on page 129.
	**Note**	*This icon duplicates the* **Previous** *icon on the View toolbar.*
	**Front View**	- View the beam from the front – *see* ***"To use standard views"*** on page 128.

Icon	*Name*	*Function*
	Note	If you click this or any of the **View** icons listed below the display resets to show the entire beam from that direction.
	**Top View**	View the beam from the top – *see "To use standard views"* on page 128.
	**Left View**	View the beam from the left – *see "To use standard views"* on page 128.
	**Back View**	View the beam from the back – *see "To use standard views"* on page 128.
	**Bottom View**	View the beam from the bottom – *see "To use standard views"* on page 128.
	**Right View**	View the beam from the right – *see "To use standard views"* on page 128.
	**View South-West**	View the beam from the southwest (bottom left) – *see "To use standard views"* on page 128.
	**View South-East**	View the beam from the southeast (bottom right) – *see "To use standard views"* on page 128.
	**View North-East**	View the beam from the northeast (top right) – *see "To use standard views"* on page 128.
	**View North-West**	View the beam from the northwest (top left) – *see "To use standard views"* on page 128.

The Scheme toolbar

The icons of this toolbar allow you to choose the way you want to view a particular window both when it is static and when you manipulate it (zoom, pan and rotate[1]):

View Type	*Icon*	*Name*	*Function*
Static		**Axis**	Show the columns and rafters in the current window by their centre-lines – ***see*** ***“Static - Axis”*** on page 130.
		**Wired**	Show a wireframe view of the columns and rafters in the current window – ***see*** ***“Static - Wired”*** on page 131.
		**Solid**	Use a fully rendered view to show the columns and rafters in this window. (This may give a slow display for large structures on less powerful computers.) – ***see*** ***“Static - Solid”*** on page 132.

View Type	*Icon*	*Name*	*Function*
Dynamic		**Box**	Show the extents of the content of the window as a shaded box as you manipulate it – ***see*** ***"Dynamic - Box"*** on page 133.
		**Axis**	Show the content of the current window by the centre-lines of the members as you manipulate the display – ***see*** ***"Dynamic - Axis"*** on page 134.
		**Wired**	Show the content of the current window by a wireframe view of the members as you manipulate the display – ***see*** ***"Dynamic - Wired"*** on page 135.
		**Solid**	Use a fully rendered view to show the columns and rafters in this window as you manipulate the display. (This may give a slow display for large structures) – ***see*** ***"Dynamic - Solid"*** on page 136.
Structure		**Isometric**	Shows the 3D display of the structure using an isometric view – ***see*** ***"To view as isometric"*** on page 137.
		**Perspective**	Shows the 3D display of the structure using a perspective view – ***see*** ***"To view as perspective"*** on page 138.

The Select toolbar

Gives you access to the following features:

Icon	Name	Function
	**Create**	Creates members of the current element type in locations which you choose using the current method. For further information see under the various element types below.
	**Delete**	Deletes elements of the current element type in locations which you choose using the current method. For further information see under the various element types below.
	**Set Attributes**	Resets the properties of the current element type in locations which you choose using the current method. For further information see under the various element types below.
	**Move**	Moves objects of the current element type in locations which you choose using the current method. For further information see under the various element types below.

Icon	Name	Function
	**Grid Points**	Determine which elements are to be actioned by picking grid points in your structure. For further information see under the various element types below.
	**Grid Line**	Determine which elements are to be actioned by picking a grid line in your structure. For further information see under the various element types below
	**Area**	Determine which elements are to be actioned by dragging across an area in your structure. For further information see under the various element types below.
	**Single**	Determine which elements are to be actioned by picking a single element/area. For further information see under the various element types below.

Icon	Name	Function
	**Gable Post**	Perform the action using the current method on the gable posts in your structure. ***See:*** • *"To create gable posts by grid line"* on page 609, • *"To create gable posts singly"* on page 613, • *"To delete gable posts by grid line"* on page 616, • *"To delete gable posts by area"* on page 616, • *"To delete gable posts singly"* on page 617, • *"To modify gable post attributes by grid line"* on page 618, • *"To modify gable post attributes by area"* on page 621, • *"To modify gable post attributes singly"* on page 624, • *"To move gable posts singly"* on page 627.

Icon	Name	Function
	Roof Bracing	Perform the action using the current method on the roof bracing in your structure. ***See:*** • *"To create roof bracings by grid line"* on page 629, • *"To delete roof bracings by grid line"* on page 637, • *"To modify roof bracing attributes by grid line"* on page 639, • *"To modify roof bracing attributes by area"* on page 642, • *"To modify roof bracing attributes singly"* on page 645, • *"To move roof bracings singly"* on page 648.

Icon	Name	Function
	Side Bracing	Perform the action using the current method on the side bracing in your structure. ***See:*** • *"To create side bracings by grid line"* on page 650, • *"To create side bracings singly"* on page 655, • *"To delete side bracings by area"* on page 658, • *"To delete side bracings singly"* on page 659, • *"To modify side bracing attributes by area"* on page 660, • *"To modify side bracing attributes singly"* on page 663, • *"To move side bracings singly"* on page 666,

Icon	*Name*	*Function*
	**Gable Bracing**	Perform the action using the current method on the gable bracing in your structure. ***See:*** • *"To create gable bracings singly"* on page 668, • *"To delete gable bracings by area"* on page 672, • *"To delete gable bracings singly"* on page 673, • *"To modify gable bracing attributes by area"* on page 674, • *"To modify gable bracing attributes singly"* on page 677, • *"To move gable bracings singly"* on page 680.
	**Eaves Tie**	Perform the action using the current method on the eaves ties in your structure. ***See:*** • *"To create eaves ties singly"* on page 686, • *"To delete eaves ties by area"* on page 688, • *"To delete eaves ties singly"* on page 689, • *"To modify eaves tie attributes by area"* on page 690, • *"To modify eaves tie attributes singly"* on page 692.

Icon	Name	Function
	**Standard Purlin**	Perform the action using the current method on the purlins in your structure. ***See:*** • *"To create cold-rolled purlins by grid line"* on page 696, • *"To create cold-rolled purlins singly"* on page 700, • *"To delete cold-rolled purlins by grid line"* on page 701, • *"To delete cold-rolled purlins by area"* on page 702, • *"To delete cold-rolled purlins singly"* on page 702, • *"To modify cold-rolled purlin attributes by grid line"* on page 703, • *"To modify cold-rolled purlin attributes by area"* on page 705, • *"To modify cold-rolled purlins attributes singly"* on page 707.

Icon	Name	Function
	**Standard Rail**	Perform the action using the current method on the sheeting rails in your structure. ***See:*** • *"To create cold rolled side rails by grid line"* on page 715, • *"To create cold rolled side rails singly"* on page 719, • *"To delete cold rolled side rails by grid line"* on page 720, • *"To delete cold rolled side rails by area"* on page 721, • *"To delete cold rolled side rails singly"* on page 721, • *"To modify cold rolled side rail attributes by grid line"* on page 722, • *"To modify cold rolled side rail attributes by area"* on page 724, • *"To modify cold rolled side rails attributes singly"* on page 726.

Icon	Name	Function
	**Hip Purlin**	Perform the action using the current method on the hip purlins in your structure. ***See:*** • *“To create hip purlins by grid line”* on page 729, • *“To create hip purlins singly”* on page 733, • *“To delete hip purlins by grid line”* on page 735, • *“To delete hip purlins by area”* on page 735, • *“To delete hip purlins singly”* on page 736, • *“To modify hip purlin attributes by grid line”* on page 737, • *“To modify hip purlin attributes by area”* on page 738, • *“To modify hip purlin attributes singly”* on page 740.

Icon	Name	Function
	**Gable Rail**	Perform the action using the current method on the gable rails in your structure. ***See:*** • *"To create gable rails by grid line"* on page 743, • *"To create gable rails singly"* on page 749, • *"To delete gable rails by grid line"* on page 752, • *"To delete gable rails by area"* on page 752, • *"To delete gable rails singly"* on page 753, • *"To modify gable rail attributes by grid line"* on page 754, • *"To modify gable rail attributes by area"* on page 755, • *"To modify gable rail attributes singly"* on page 757.

Icon	Name	Function
	Eaves Beam	Perform the action using the current method on the eaves beams in your structure. ***See:*** • *“To create eaves beams by grid points”* on page 760, • *“To create eaves beams by grid line”* on page 762, • *“To create eaves beams singly”* on page 764, • *“To delete eaves beams by grid line”* on page 766, • *“To delete eaves beams by area”* on page 767, • *“To delete eaves beams singly”* on page 767, • *“To modify eaves beam attributes by grid line”* on page 768, • *“To modify eaves beam attributes by area”* on page 770, • *“To modify eaves beam attributes singly”* on page 772.

Icon	Name	Function
	**Jack Rafter**	Perform the action using the current method on the jack rafters in your structure. ***See:*** • *"To create jack rafters singly"* on page 774, • *"To delete jack rafters by grid line"* on page 777, • *"To delete jack rafters by area"* on page 777, • *"To delete jack rafters singly"* on page 778, • *"To modify jack rafter attributes by grid line"* on page 779, • *"To modify jack rafter attributes by area"* on page 781, • *"To modify jack rafter attributes singly"* on page 783, • *"To move jack rafters singly"* on page 785.

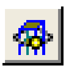
Icon	Name	Function
	**Hip Raker**	Perform the action using the current method on the hip rakers in your structure. ***See:*** • *"To delete hip rakers by grid line"* on page 787, • *"To delete hip rakers by area"* on page 788, • *"To delete hip rakers singly"* on page 788, • *"To modify hip raker attributes by grid line"* on page 790, • *"To modify hip raker attributes by area"* on page 791, • *"To modify hip raker attributes singly"* on page 794.

Icon	*Name*	*Function*
	**Floor joist**	Perform the action using the current method on the floor joists in your structure. ***See:*** • *"To create floor joists singly"* on page 796, • *"To delete floor joists by grid line"* on page 799, • *"To delete floor joists by area"* on page 799, • *"To delete floor joists singly"* on page 800, • *"To modify floor joist attributes by grid line"* on page 801, • *"To modify floor joist attributes by area"* on page 804, • *"To modify floor joist attributes singly"* on page 807.

Icon	*Name*	*Function*
	**Floor area**	Perform the action using the current method on the floor areas in your structure. ***See:*** • *"To create floor areas singly"* on page 811, • *"To delete floor areas by grid line"* on page 813, • *"To delete floor areas by area"* on page 814, • *"To delete floor areas singly"* on page 814, • *"To modify floor area attributes by grid line"* on page 816, • *"To modify floor area attributes by area"* on page 819, • *"To modify floor area attributes singly"* on page 822.

The Animate toolbar

Gives you access to the following features:

Icon	*Name*	*Function*
	**Animate**	Animates the current view of your structure – ***see*** *"To animate the view"* on page 138.

Using help

Portal Frame provides online help for every function. You can resize, move, tile, or cascade the *Help* window and the *Portal Frame* main window so that you can see both of them.

The extensive help contains all the information from this ***User's Guide***. Extensive cross-referencing means that you can quickly find the information that you require.

There are several ways to obtain online help:

- You can see context sensitive help for all dialogs and property sheet by clicking **Help** which will take you straight to the information on that dialog or property sheet.
- You can also access help by selecting *Help/Help Topics*.

Exiting Portal Frame

Before leaving ***Portal Frame*** ensure that you save any information you require later. If you forget, ***Portal Frame*** will prompt you.

To close Portal Frame

1. Select *File/Exit*.

4 Understanding Portal Frame

Portal Frame is an extremely advanced product, using state of the art analysis, design and ***Windows*** controls. This enables you to achieve extremely competitive solutions for a wide range of conditions easily. At CSC we are rightly proud of this application, and hope that it will increase your productivity, while at the same time making your work more enjoyable.

Take the time now to familiarise yourself with the terminology and concepts below and you will find that you are rewarded with increased productivity later.

Portal Frame – Basics

Portal Frame is a powerful ***Windows*** based portal frame design and checking tool based on the requirements of BS 5950-1: 2000.

You can define the frames necessary for your building and then assemble these to create the overall building geometry.

You can then apply building loads to your frames and can run the ***Wind Load Generator***[1] and ***Snow Load Generator***[1] applications on them to calculate the wind and snow loads. These loads are applied to your frames based on their location within the building, and on the frame centres perpendicular to the span direction.

You can add other loads to your frames to allow for local effects, or to include loads which do not apply on a building wide basis.

For any frame in a building you can define the positions of restraint and transfer these to the other frames in the building giving a consistent layout.

1. *These are additional applications which you must purchase separately to the main* ***Portal Frame Design*** *program.*

Once you have defined the geometry and loading for a frame you can perform a complete analysis and design for it. Alternatively you can check a known set of sections for adequacy. You can also check the overall stability of the frame and the local stability of each member.

Portal Frame is based around the concept of a ***Project*** which can contain anything from a single frame to all the frames required for a building[1]. You define the geometry and loading for each frame (from scratch or by copying and modifying existing frames). Once you have defined a frame you can design it.

Portal Frame – Symbols

Symbols show you the design status at a glance. You will see the following symbols in *Portal Frame*.

Symbol	*Meaning*	*Action*
✔	The member, span, frame etc. passes this check.	
✘	The member, span, frame etc. does not pass this check.	**Examine** the check to see the corrective action you need to take.
⇨	This check is critical to the design of the frame or member.	

1. *Constrained only by the memory and resources available on your computer.*

Symbol	*Meaning*	*Action*
	An error condition has occurred for this check.	**Examine** the check to see the corrective action you need to take.
	There is further information available for this check.	Use **Examine** to see what this is.
	A condition has occurred which invalidates the current check.	Use **Examine** to see what the problem is.
	The member has only been partially checked, however all the checks that have been performed pass.	
	A condition has occurred which has generated a warning for the current check.	Use **Examine** to see what this is before taking corrective action.

The other symbols listed below give you information about the frame or the current member.

Symbol	*Meaning*
	A plastic hinge has formed at this load factor.
	A plastic hinge has reversed (unformed) at this load factor.
I	The section is an I section at this point.
H	The rafter is haunched at this point (i.e. it is a three flanged section). This point on the column is above the bottom flange of the haunch (or the bottom flange of the rafter if there is no haunch).

5 Setting Preferences

Understanding preferences

We know that there are nearly as many preferred ways of working as there are engineers. You can configure *Portal Frame* to suit yourself using *Preferences* to choose the:

- units that you prefer to work in,
- basic design settings for each country,
- colours used for the various parts of the graphical display.

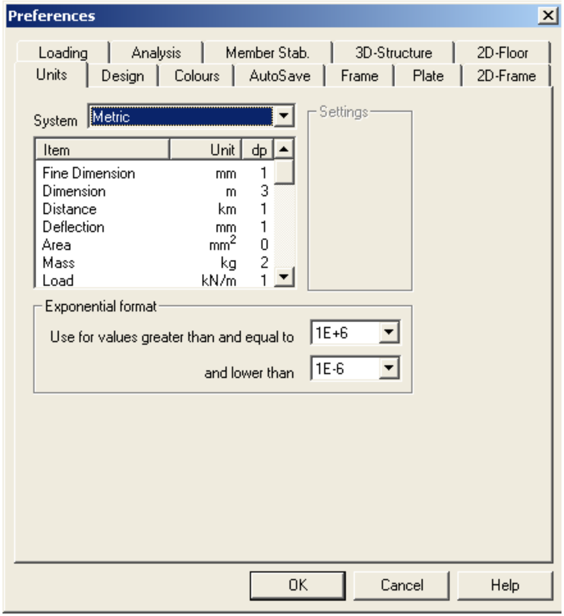
Setting preferences for Portal Frame

Initially, *Portal Frame's* preferences are set, based on our many years experience in the industry. You can change these at will knowing that the preferences you set will be remembered the next time you use *Portal Frame*.

Note You can change your preferences as often as you like.

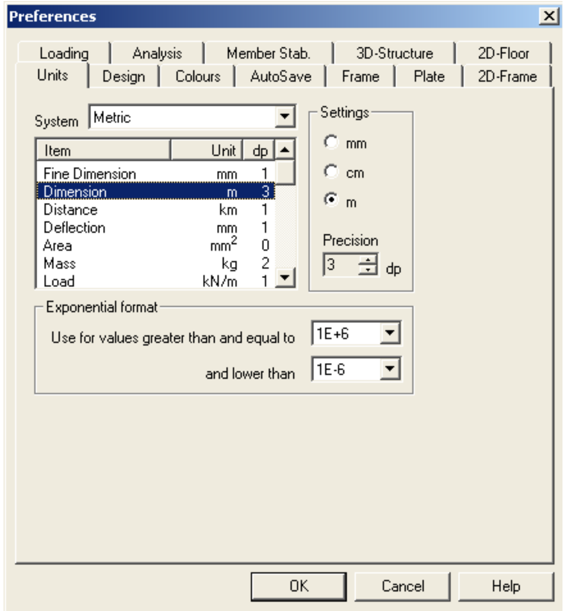
To set unit preferences

1. Select *File/Preferences…* to see the ***Units*** page of the ***Preferences*** property sheet.



2. The only **System** of units that you can use in ***Portal Frame*** is the *Metric* one. The **Items** list shows the units and preferences in this **System**.

3. Initially **Settings** is dimmed and empty. However click an item in the **Item** list and **Settings** will show the available options and the current **Precision**.



4. Pick your preferred **Unit** and set the **Precision** using the spin buttons.

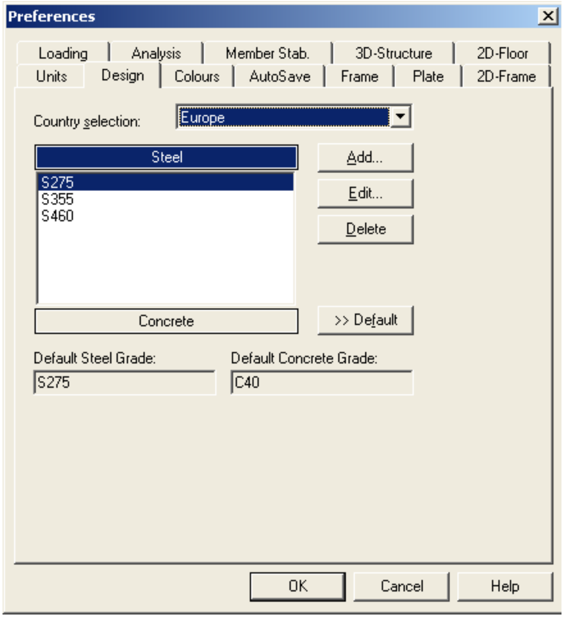
Note *You can change the precision at any time and the new precision will be used immediately.*

5. You can display small or large numbers using **Exponential Format**. This setting applies to all units i.e. you can not set a different threshold for each unit.

6. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set design preferences

1. Select *File/Preferences* and then pick ***Design***

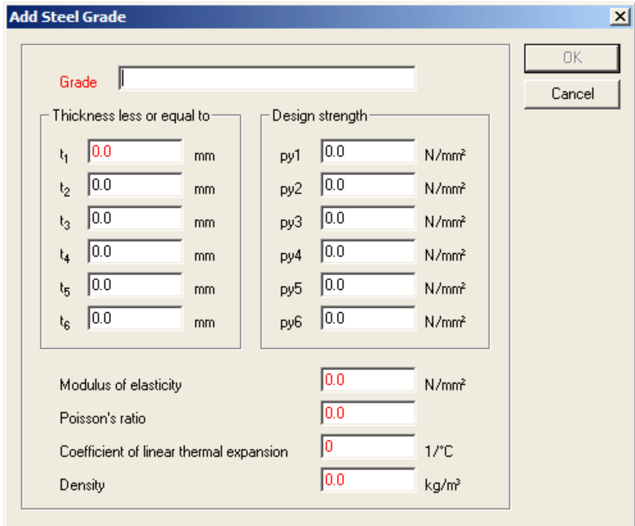


2. Select your **Source of Sections**. The list of **Steel Grades** will show those for this country.

3. If you want to use one of the listed steel grades as your default pick it and click **Set Default**.

Add steel grade dialog

4. If you want to add a new steel grade, then click **Add…** while you are viewing the list of steel grades. You will see the ***Add Steel Grade*** dialog.



Enter the description for the **Grade** that you want to add.

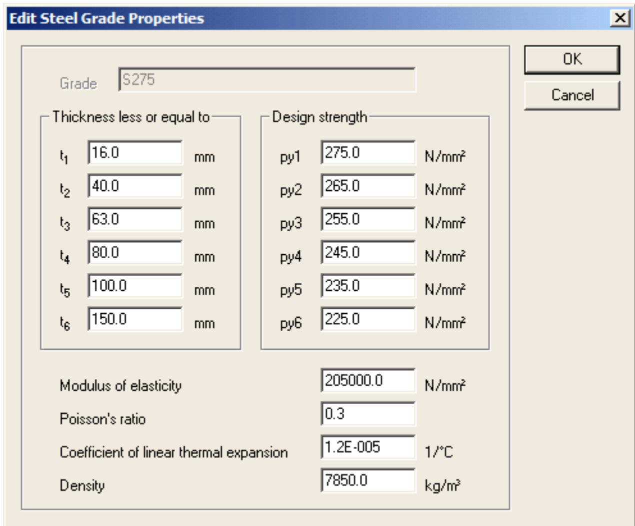
If the design strength depends on the material thickness, then complete the table of **Thicknesses** and **Design Strengths**.

If the design strength does not depend on the material thickness, then enter a large value for the first **Thickness** (***t1***), the constant design strength for the first **Design strength** value (***fy1***) and leave the other thicknesses and strengths at zero.

Enter the values for **Modulus of Elasticity** (Young's modulus), **Poisson's ratio**, **Coefficient of linear thermal expansion** and **Density**.

Once your settings are complete click **OK** to return to the ***Preferences*** property sheet. The grade that you have just added will be sorted into the list of available grades.

5. If you want to change the data for an existing steel grade, then click **Edit…** while you are viewing the list of steel grades. You will see the ***Edit Steel Grade*** dialog.



In this case the name of the grade cannot be changed and so it is dimmed.

If the design strength depends on the material thickness, then complete the table of **Thicknesses** and **Design Strengths**.

If the design strength does not depend on the material thickness, then enter a large value for the first **Thickness** (*t1*), the constant design strength for the first **Design strength** value (*py1*) and leave the other thicknesses and strengths as zero.

Caution You should only change settings to cater for modifications to design codes etc. The changed settings will be used for all designs using this steel grade. Your changes might affect the work of others.

If you want to use specific values resulting from material tests etc. – then we recommend that you add a new grade containing this information and name it explicitly.

Enter the values for **Modulus of Elasticity** (Young's modulus), **Poisson's ratio**, **Coefficient of linear thermal expansion** and **Density**.

Once your settings are complete click **OK** to return to the ***Preferences*** property sheet. The changes you have made to that grade will be used from now on.

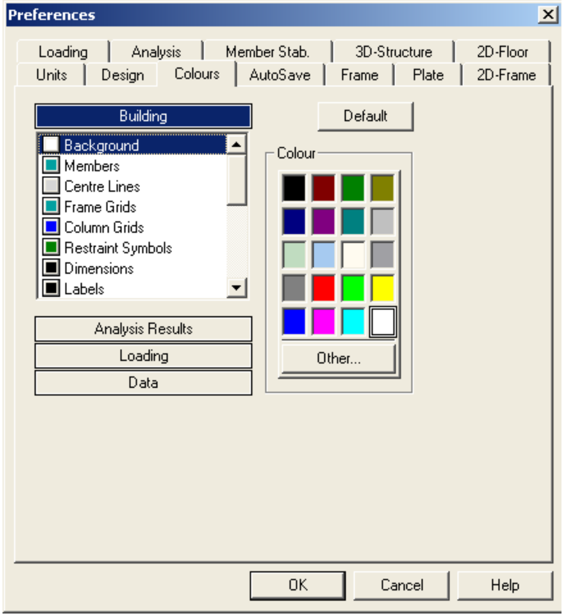
6. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set colour preferences

You use colour preferences to control the colour of the various parts of the graphical display.

For a solid display the colour you choose is the base colour for that part, however this will be affected by the lighting to give you a rendered view of the member. For a wire frame view the colour you choose is the colour of the lines used to represent the member. For an axis view the colour you choose is the colour of the line used to represent the centre-line of the member.

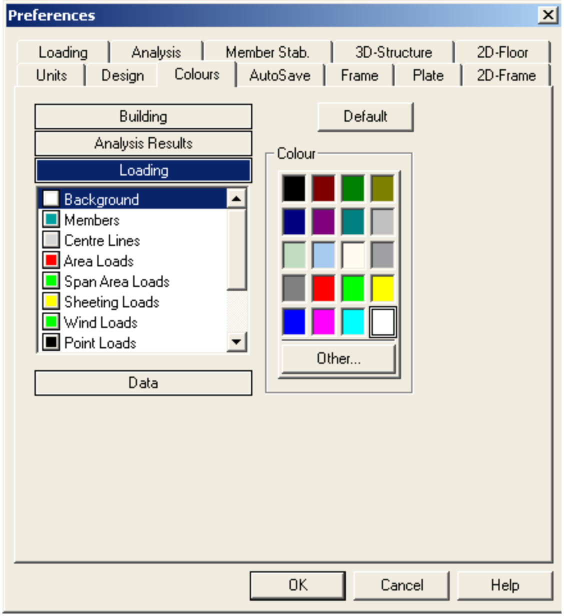
1. Select *File / Preferences…* and pick ***Colours***.



The colour swatch to the side of each component shows the base colour that will be used for that component.

2. To select the colour for an item choose the **text** relating to that item. You can then either choose a colour from the swatches in the **Colours** part of the dialog, or you can click **Other…** to select the colour using the standard ***Windows Colour*** dialog. Alternatively you can select the **swatch** for an item which will take you to the ***Windows Colour*** dialog directly.

3. If required, select the colour settings for the items on the **Analysis Results**, **Loading** or **Data** drop down lists.

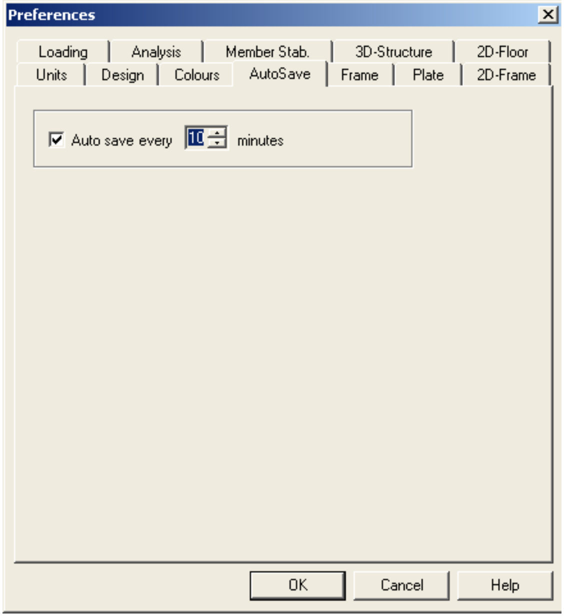


4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set autosave preferences

You use these preferences to control the autosave feature. If you switch this option on, then ***Portal Frame*** will save your structure at the intervals you specify. If you then experience a problem with your computer or model, ***Portal Frame*** asks if you want to open the most recent autosaved file.

1. Select *File / Preferences…* and pick ***AutoSave***



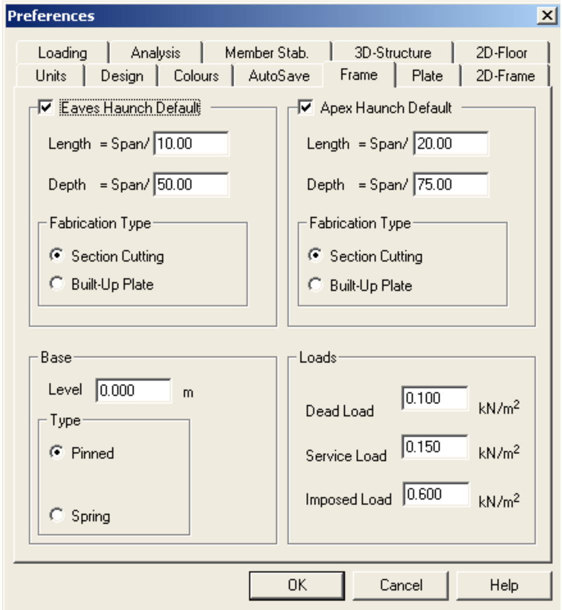
2. To enable the auto save feature ensure that the **Auto save** box is ticked.

3. Set the frequency of saving by setting the number of **minutes** between saves.

4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set frame preferences

1. Select *File/Preferences* and pick ***Frame***



This page allows you to control the parts of your frame which ***Portal Frame*** can generate automatically for you, together with the initial values which it will assume. You can also set default values for base details and the initial loads which ***Portal Frame*** will create on your frame.

Note *You can change any automatically generated value which is not appropriate to the current frame.*

2. If you want ***Portal Frame*** to automatically create eaves haunches, then check **Eaves Haunch Default** and enter the appropriate **Length** ratio, **Depth** ratio and **Fabrication type**.

3. Adopt a similar approach for your apex haunches.

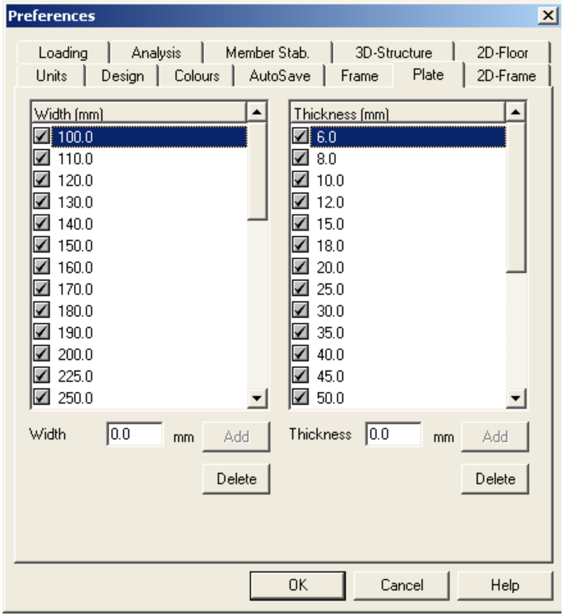
4. Enter the **Level** of your default bases and specify their **Type**.

5. Enter the values for the loads that you want ***Portal Frame*** to use as its default.

6. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set plate preferences

1. Select *File/Preferences…* and pick ***Plate***



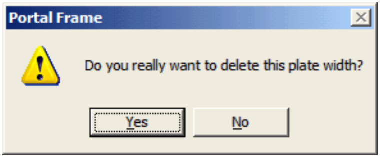
2. The **Width** and **Thickness** lists show all the available widths and thicknesses. If you want to stop use of a particular width or thickness, then remove the check to its side.

Adding plate details

3. If you want to add a new width or thickness enter the value in the **Width** or **Thickness** field and click **Add**.

Deleting plate details

4. If you want to permanently delete a width or thickness click on that width or thickness in the **Width** or **Thickness** lists and then click **Delete**. You will see a confirmation dialog.



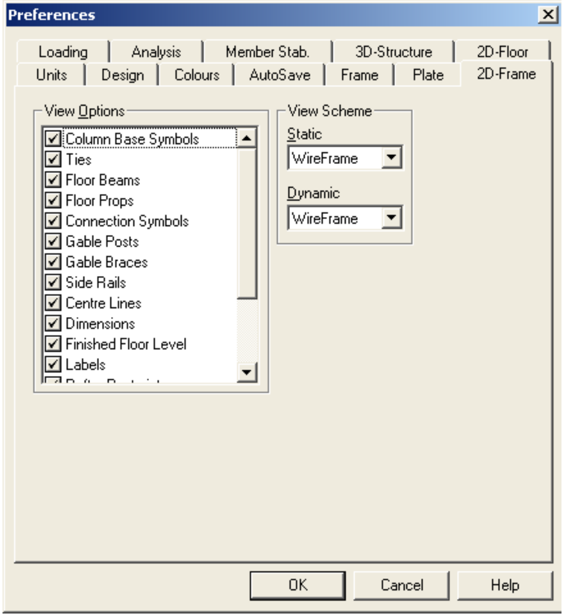
If you do want to delete the width or thickness click **Yes**.

5. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set 2D-frame window preferences

You use these preferences to control the default display that ***Portal Frame*** will show for a 2D ***Geometry*** window.

1. Select *File / Preferences…* and click ***2D-Frame***



2. Pick the items that you want the initial view to contain from the left hand list.

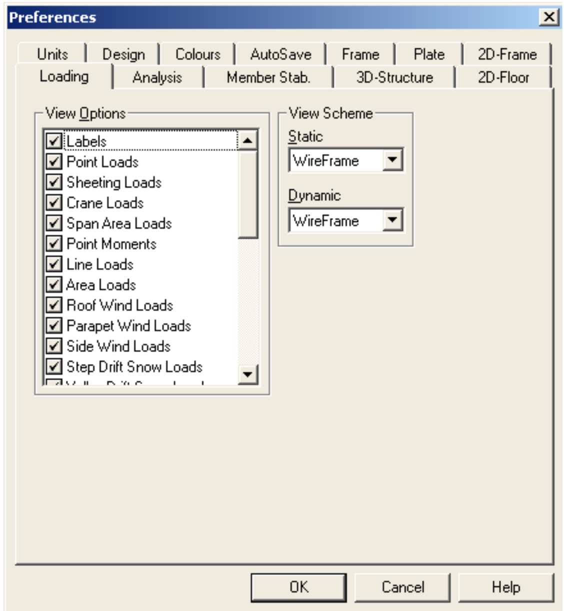
3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view, which again (for the dynamic view only) takes more time than the box view.

4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set frame loading window preferences

You use these preferences to control the default display that *Portal Frame* will show for a frame results window.

1. Select *File / Preferences…* and pick ***Frame Loading***



2. Pick the items that you want the initial view to contain from the left hand list.

Note If a particular floor view does not contain items of a particular type, then these will not be shown irrespective of your settings here.

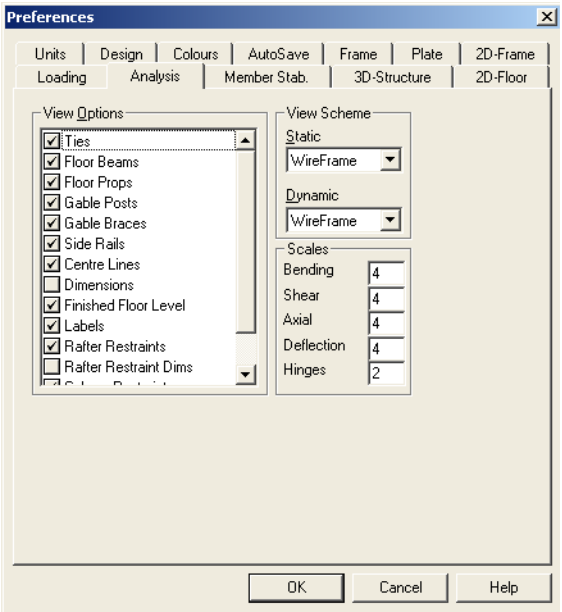
3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view.

4. Pick another page of the *Preferences* property sheet or click **OK** to save your current preferences.

To set Analysis window preferences

You use these preferences to control the default display that *Portal Frame* will show for an analysis results window.

1. Select *File / Preferences…* and pick ***Analysis***



2. Pick the items that you want the initial view to contain from the left hand list.

Note If a particular analysis view does not contain items of a particular type, then these will not be shown irrespective of your settings here.

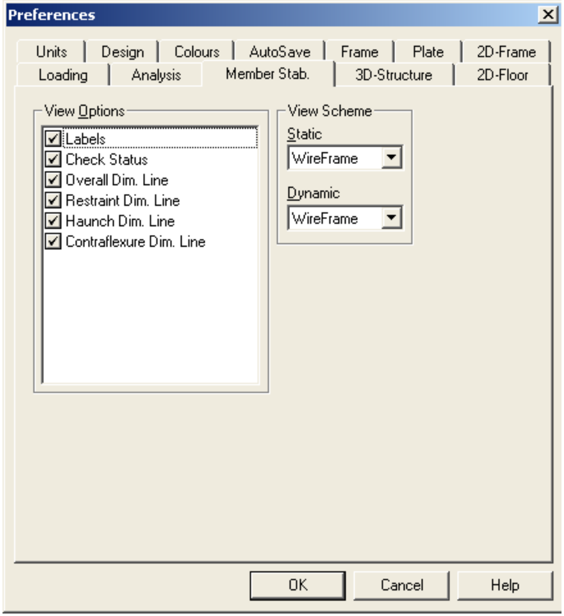
3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view.

4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set member stability window preferences

You use these preferences to control the default display that ***Portal Frame*** will show for a member stability window.

1. Select *File / Preferences…* and pick ***Member Stability***



2. Pick the items that you want the initial view to contain from the left hand list.

Note *If a particular floor view does not contain items of a particular type, then these will not be shown irrespective of your settings here.*

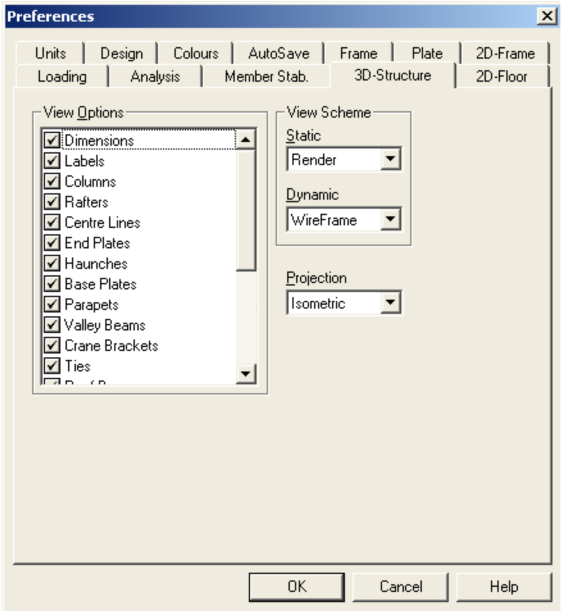
3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view.

4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

To set 3D-building window preferences

You use these preferences to control the default display that ***Portal Frame*** will show for the 3D ***Structure*** window.

1. Select *File / Preferences…* and pick ***3D-Structure***



2. Pick the items that you want the initial view to contain from the left hand list.

3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view, which again (for the dynamic view only) takes more time than the box view.

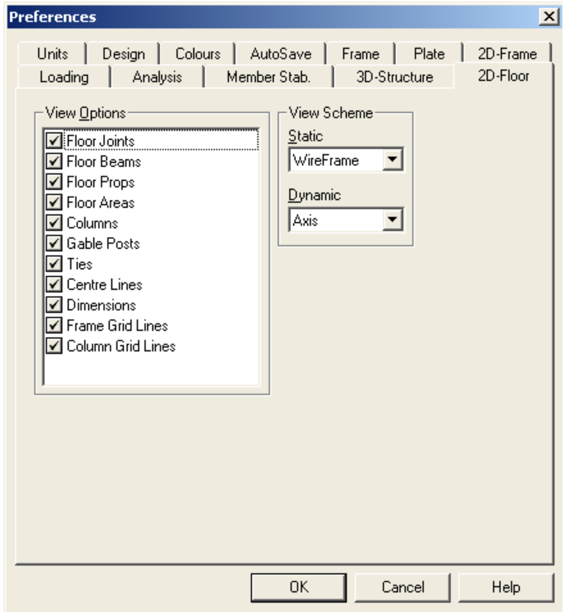
4. Select the **Projection** type that you want to use.

5. Pick another page of the *Preferences* property sheet or click **OK** to save your current preferences.

To set 2D-Floor window preferences

You use these preferences to control the default display that *Portal Frame* will show for a 2D floor window.

1. Select *File / Preferences…* and pick ***2D-Floor***



2. Pick the items that you want the initial view to contain from the left hand list.

Note *If a particular floor view does not contain items of a particular type, then these will not be shown irrespective of your settings here.*

3. Select the view scheme that you want the initial view to use, both when static, and when you are manipulating it using the mouse. Remember that the rendered view will take more time than the wire frame one, which will in turn take more time than the axis view.

4. Pick another page of the ***Preferences*** property sheet or click **OK** to save your current preferences.

6 Controlling display content

Normally *Portal Frame* displays every possible item on the screen which might lead to a cluttered display. If this is so, then you can remove specific items to improve clarity.

Controlling the Toolbars

Portal Frame displays its ***Toolbars*** immediately under the menu bar. You can switch the toolbars off to give a larger area for the graphical display.

To remove a Toolbar

1. Select *View/Toolbar/Toolbar name*

   If a toolbar is displayed its name is ticked. Click the toolbar name to remove it.

To reinstate the Toolbar

1. Select *View/Toolbar/Toolbar name*

   If a toolbar is not displayed its name is not ticked. Click the name to reinstate it.

Controlling the Status Bar

Portal Frame displays its ***Status bar*** at the bottom of its window. The status bar is used to give essential information about the current design.

Tip *We recommend that you do not switch the status bar off except as a temporary means of increasing the area available for the graphical display.*

To remove the Status Bar

1. Select *View/Status Bar*

If the **Status Bar** is displayed its menu option is ticked. Click the *Status Bar* menu option to remove the status bar,

To reinstate the Status Bar

1. Select *View/Status Bar*

   If the **Status Bar** is not displayed its menu option is not ticked. Click the *Status Bar* menu option to reinstate the status bar,

Controlling the Workbook tabs

Normally *Portal Frame* displays **Workbook tabs** at the bottom of its graphics area. The workbook tabs allow you to switch between the various open graphical displays.

To remove the Workbook tabs

1. Select *View/Workbook*

   If the **Workbook tabs** are displayed their menu option is ticked. Click the *Workbook* menu option to remove them,

To reinstate the Workbook tabs

1. Select *View/Workbook*

   If the **Workbook tabs** are not displayed their menu option is not ticked. Click the *Workbook* menu option to reinstate them,

Controlling the Project Workspace

Normally *Portal Frame* displays the **Project Workspace**. You can use the workspace to see exactly which frames your project contains, to select the frame that you want to work on and to access the *Beam Details* or *Analysis Results* windows for that frame quickly. Furthermore you can choose whether the **Project Workspace** can be *Docked* against the side of the main *Portal Frame* window, or whether it is forced to be a *Floating* window which can be positioned anywhere on your *Windows* desktop.

To remove the Project Workspace

1. Select *View/Project Workspace*

Mouse Click anywhere over the **Project Workspace** with the right mouse button, and pick *Hide* on the context menu.

If the **Project Workspace** is displayed its menu option is ticked. Select *Project Workspace* to remove the **Project Workspace**,

To reinstate the Project Workspace

1. Select *View/Project Workspace*

If the **Project Workspace** is not displayed its menu option is now ticked. Select *Project Workspace* to reinstate the **Project Workspace**,

To choose a docked or a floating Project Workspace

You can only choose this option using the mouse. You must be able to see the **Project Workspace** to make this choice.

Mouse Click anywhere over the **Project Workspace** with the right mouse button. If the *Allow docking* option is ticked, then the **Project Workspace** can be docked, otherwise it cannot. Selecting the option switches *Allow docking* on and off.

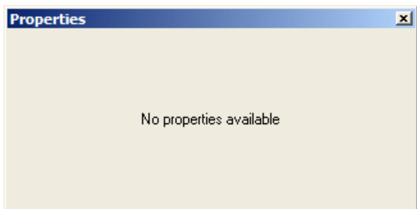
Note If **Project Workspace** docking is allowed, and you switch the *Allow docking* option off, then the **Project Workspace** will become a normal window at the point on your *Windows* desktop where it was positioned before it was docked.

Note You can still position a docked **Project Workspace** anywhere on your *Windows* desktop, by positioning the cursor over its border, and then dragging using the left mouse button. If *Allow docking* is switched on and you drag and drop the **Project Workspace** over the edge of the *Portal Frame* window the **Project Workspace** will dock to that side of the window.

Controlling the content of the display

You can choose the information that you want to see in any graphical window through the ***Properties*** sheet. These are described texturally below, however we recommend that you take time to investigate the effects of the different combinations of settings for yourself.

If your current window has no properties, but you have the ***Properties*** sheet open it simply indicates that there are no properties available.

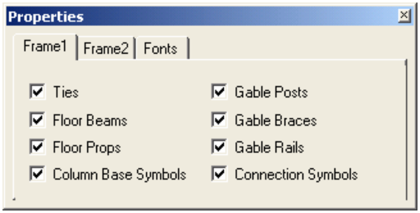


To set Frame window display properties

1. Ensure that the ***Frame*** window whose contents you want to set is active.

Frame1 page

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***Frame 1*** tab.



This page allows you to choose the elements which you want to see in your ***Frame*** window.

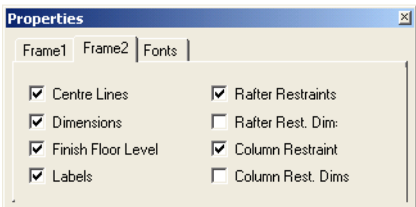
The **Column Base Symbols** and **Connection Symbols** control the display of hollow circles at the bases, eaves or apex of the frame. These indicate the area in which you must click to invoke the link to ***Column Base Design*** or ***Moment Connection Design*** respectively.

Note *If the Properties sheet does not show these options, then the active window is not a Frame window.*

3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Frame2 page

4. Click the ***Frame2*** tab to see this page of the ***Properties*** sheet.

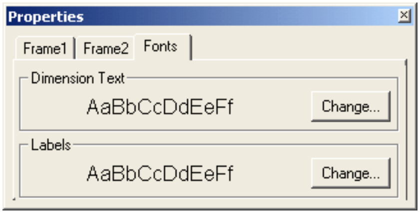


This page allows you to control the display of those items in the window which are not frame members.

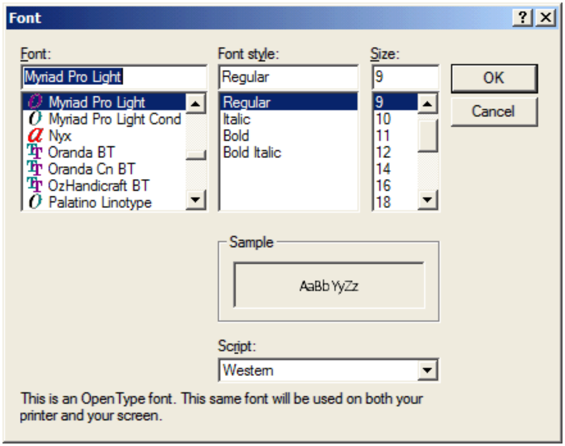
5. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

6. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Frame*** window. You choose whether or not the text is displayed using the options on the ***Frame2*** page. If you want to change the font, click the appropriate **Change...** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

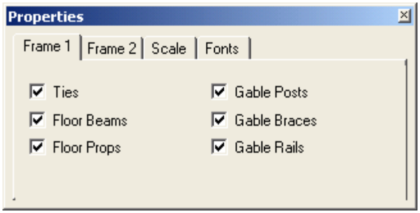
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

To set Analysis Results window display properties

1. Ensure that the ***Analysis Results*** window whose contents you want to set is active.

Frame1 page

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***Frame 1*** tab.

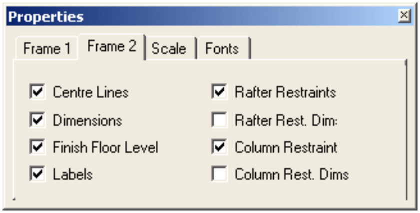


This page allows you to choose the elements which you want to see in your frame window.

3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Frame2 page

4. Click the ***Frame2*** tab to see this page of the ***Properties*** sheet.

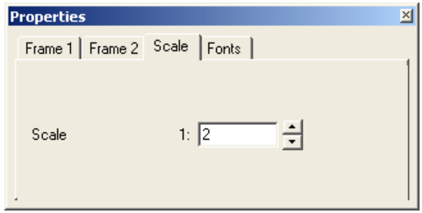


This page allows you to control the display of those items in the window which are not frame members.

5. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Scale page

6. Click the ***Scale*** tab to see this page of the ***Properties*** sheet.



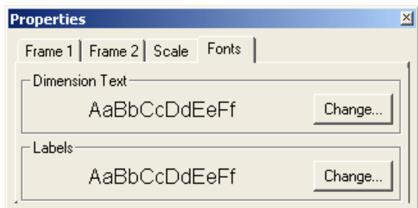
This page allows you to set the scale by which the analysis result diagrams are to be scaled, in order to accentuate them.

Note *You can specify a separate scale for each analysis result diagram.*

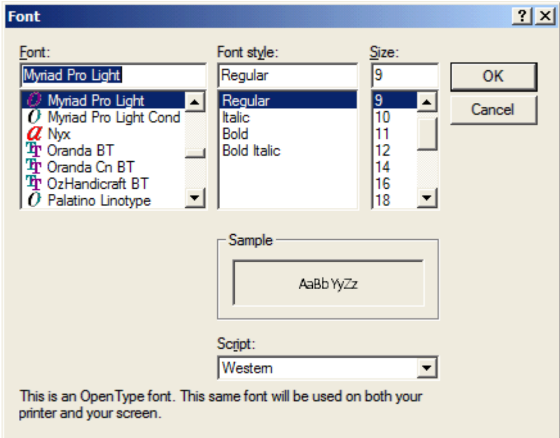
7. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

8. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Analysis Results*** window. You choose whether or not the text is displayed using the options on the ***Frame2*** page. If you want to change the font, click the appropriate **Change…** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

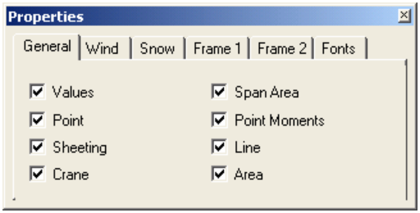
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

To set Loading Diagram window display properties

1. Ensure that the ***Loading Diagram*** window whose contents you want to set is active.

General

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***General*** tab.

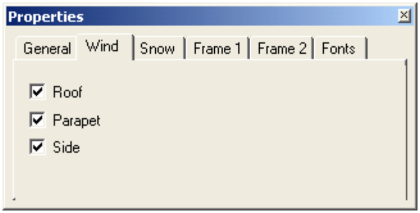


This page allows you to control which types of loads you want to see in the ***Loading Diagram*** window. You can also choose whether or not you want to see the values of the loads, or just their graphical representation.

3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Wind

4. Click the ***Wind*** tab to see this page of the ***Properties*** sheet.

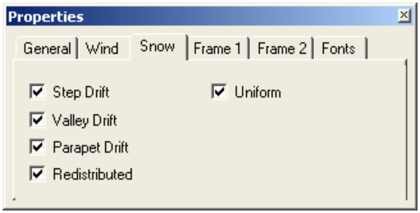


This page allows you to control the display of the wind loads which you want to see.

5. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Snow

6. Click the ***Snow*** tab to see this page of the ***Properties*** sheet.

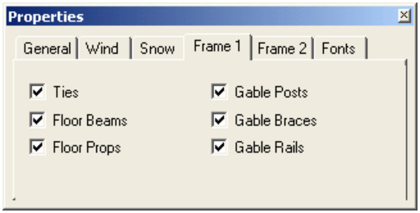


This page allows you to control the display of the snow loads which you want to see.

7. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Frame1 page

8. Click the ***Frame1*** tab to see this page of the ***Properties*** sheet.

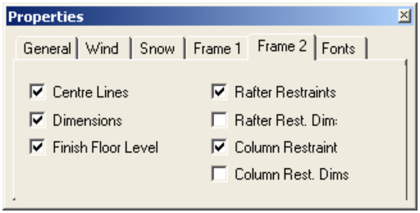


This page allows you to choose the elements which you want to see in your applied loading window.

9. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Frame2 page

10. Click the ***Frame2*** tab to see this page of the ***Properties*** sheet.

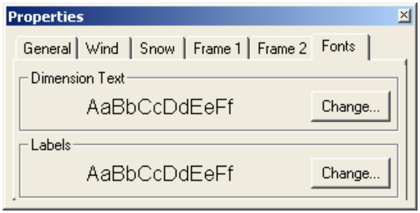


This page allows you to control the display of those items in the window which are not frame members.

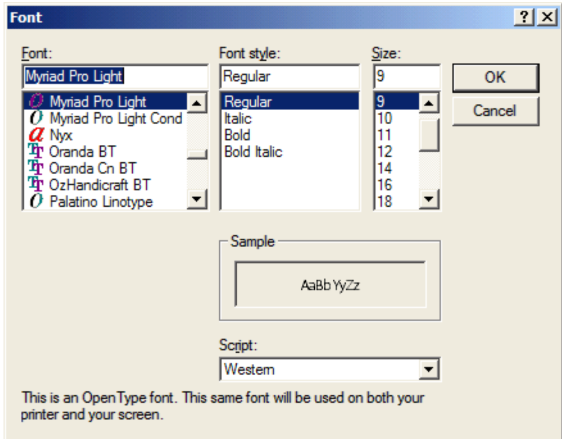
11. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

12. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Frame*** window. You choose whether or not the text is displayed using the options on the ***General*** and ***Frame2*** pages. If you want to change the font, click the appropriate **Change…** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

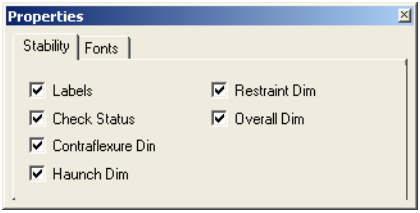
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

To set Stability window display properties

1. Ensure that the ***Stability*** window whose contents you want to set is active.

Stability page

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***Stability*** tab.

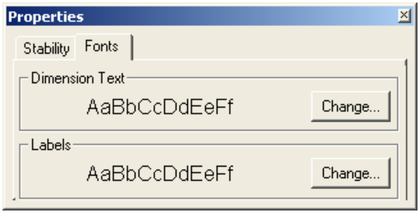


This page allows you to choose the information which you want to see in your ***Stability*** window.

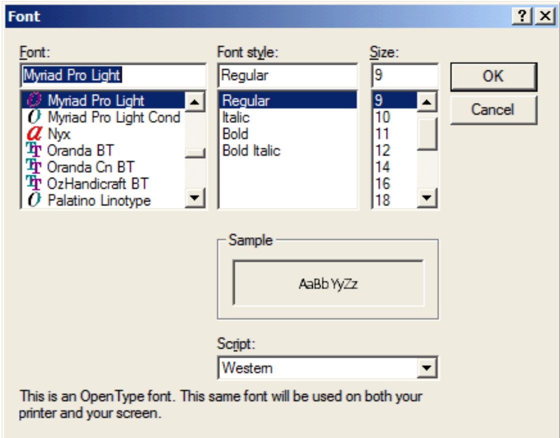
3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

4. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Stability*** window. You choose whether or not the text is displayed using the options on the ***Stability*** page. If you want to change the font, click the appropriate **Change…** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

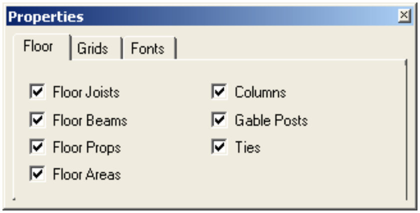
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

To set Floor window display properties

1. Ensure that the ***Floor*** window whose contents you want to set is active.

Floor page

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***Floor*** tab.

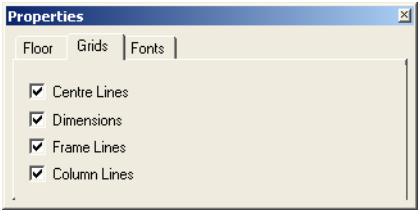


This page allows you to choose the members and floor areas which you want to see in your ***Floor*** window.

3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Grids page

4. Click the ***Grids*** tab to see this page of the ***Properties*** sheet.

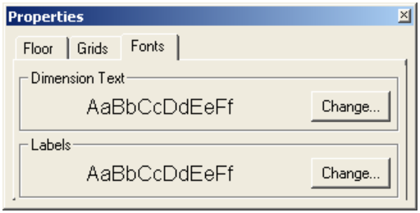


This page allows you to control the grid lines and dimensions which you want to see in your ***Floor*** window.

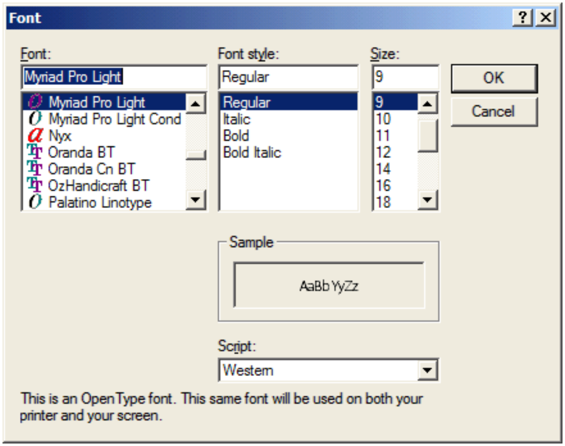
5. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

6. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Floor*** window. You choose whether or not the text is displayed using the options on the ***Grids*** page. If you want to change the font, click the appropriate **Change…** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

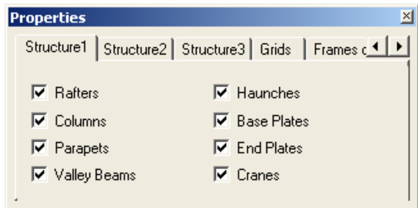
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

To set Structure window display properties

1. Ensure that the ***Structure*** window is active.

Structure 1 page

2. Select *View/Properties…* and you will see the ***Properties*** sheet – ***Structure1*** tab.

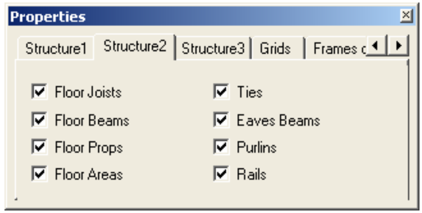


This page allows you to choose the main frame members which you want to see in your ***Structure*** window.

3. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Structure2 page

4. Click the ***Structure2*** tab to see this page of the ***Properties*** sheet.

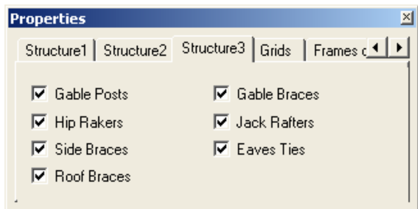


This page allows you to control the first set of ancillary steelwork which you want to see in your ***Structure*** window.

5. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Structure3 page

6. Click the ***Structure3*** tab to see this page of the ***Properties*** sheet.

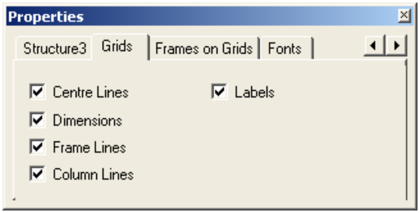


This page allows you to control the second set of ancillary steelwork which you want to see in your ***Structure*** window.

7. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Grids page

8. Click the ***Grids*** tab to see this page of the ***Properties*** sheet.

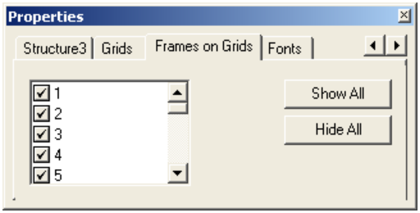


This page allows you to control the grid lines and dimensions which you want to see in your ***Structure*** window.

9. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Frames on Grids page

10. Click the ***Frames on Grids*** tab to see this page of the ***Properties*** sheet.

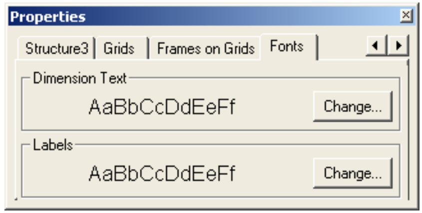


This page allows you to control the frames in your building grid which you want to see in the ***Structure*** window. If you remove the tick against a frame, then not only will that frame be removed from the ***Structure*** window, but any ancillary steelwork related to that frame will also be removed.

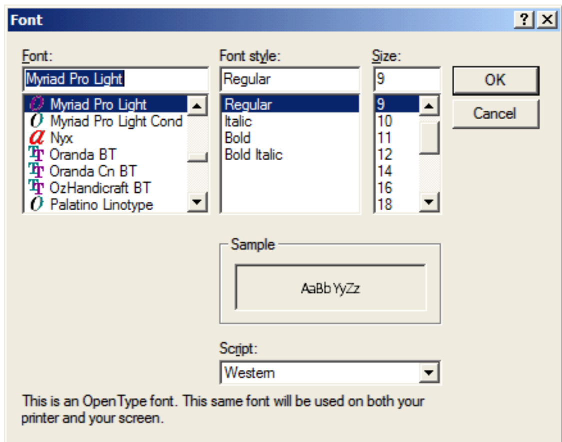
11. Ensure that the items that you want to display are ticked. As you tick items or remove ticks the display is updated instantly.

Fonts page

12. Click the ***Fonts*** tab to see this page of the ***Properties*** sheet.



This page allows you to control the fonts which ***Portal Frame*** will use for the textural information in the ***Floor*** window. You choose whether or not the text is displayed using the options on the ***Grids*** page. If you want to change the font, click the appropriate **Change…** button to see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

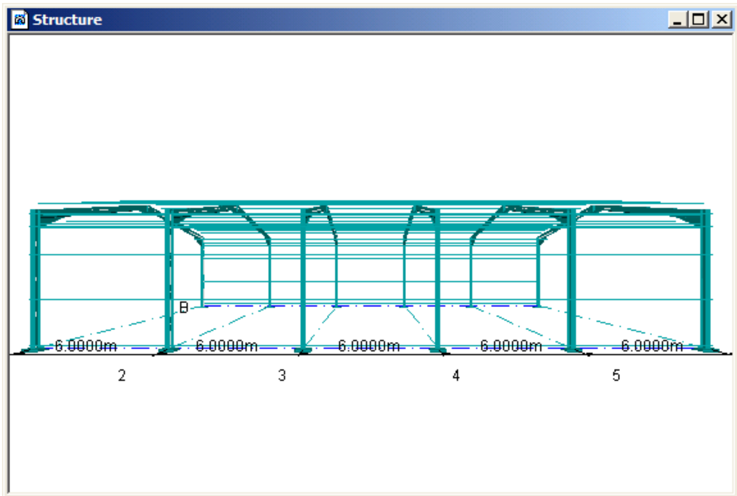
When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

7 Manipulating the graphical displays

When you create a frame and define its details, or if you change to a different orientation of the *Structure* window *Portal Frame* will show the details at a scale which causes them to fill the window. For many frames this display will be adequate for you to pick major elements. However for very large frames the display may be too small for you to do this efficiently.

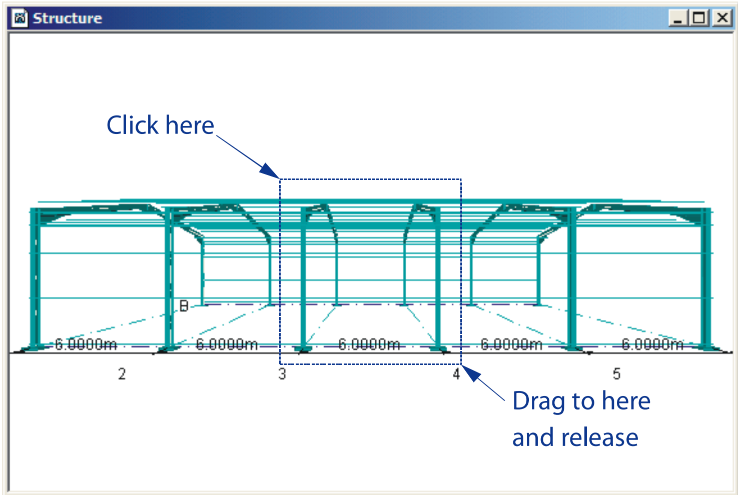
Zooming the display

Zooming allows you to enlarge a particular area of the display. There are several ways to zoom in and out. Simply choose the option that you find easiest and then stick with it. You can use zooming in all graphical windows.



To zoom into an area of the display

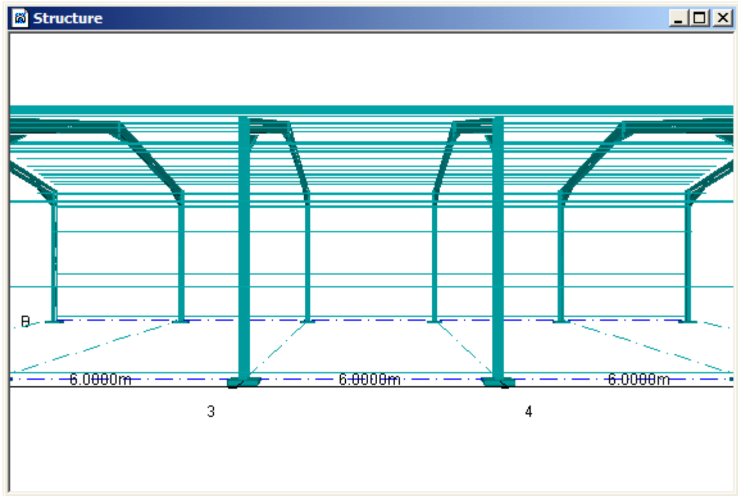
Select *View/Beam Details/Zoom Area*. You can then select the area you want to see zoomed.



Mouse Right click the display, and pick *Zoom Area* from the context menu.

1. Move the cross hairs cursor to the top left corner of the area you want to see.
2. Hold down the left mouse button and drag across to the diagonally opposite corner of the area.

3. Release the button and the area will be expanded to fill the frame window.



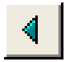
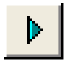
Note When the display is zoomed you can pan around by left clicking, holding the mouse button down and dragging.

Tip You can use the icons on the *View* toolbar to move between different views of the frame or of the analysis results.

To move between different views

Portal Frame stores the views of its windows. You can use the icons of the ***View*** toolbar to move backward and forward between the views you have accessed recently.

1. Click the icon on the *View* toolbar to get to the view you require. The options are given in the table below.

Icon	*Action – move*
	move to the first available view of the window
	move to the previous view of the window
	move to the next view of the window
	move to the last available view of the window

To zoom in-to or out-from the centre of the display

You can zoom in-to, or out-from, the centre of the graphical display using the mouse. The method to use depends on whether you want to zoom the 3D *Structure* window or one of the other (2D) windows.

The Structure window

1. Right click and hold the mouse button down, now
2. Left click and again hold the mouse button down (you should now be holding both left and right mouse buttons down, then
3. Drag downwards to zoom in-to the centre of the display or drag upwards to zoom out-from the centre of the display.

Any other window

1. Right click and hold the mouse button down, then

2. Drag downwards to zoom in-to the centre of the display or drag upwards to zoom out-from the centre of the display.

To rotate the view of the Structure window

You can rotate the graphical display of the ***Structure*** window using the mouse.

Note Rotation is not applicable to the 2D windows.

Mouse Right click and hold over the *Beam Definition* display, and then drag to rotate the view in wireframe mode until you have achieved the orientation you require, then release the button.

To pan the view of any window

When you have zoomed into any window, then you can move the display around, so that you can see other parts of the display. You do this using the mouse in any window.

1. Left click and hold the mouse button over the window, towards the side opposite to the direction in which you want to move.

2. While holding the button down drag the display until you can see the part of the display in which you are interested.

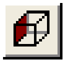
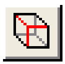
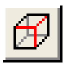
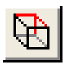
Using standard views

Portal Frame allows you instant access to a series of standard views of the ***Structure*** window, If you choose one of these options, then ***Portal Frame*** will show the structure from the appropriate direction and will also make it fit into the window.

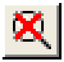
To use standard views

1. Ensure that the ***Structure*** window is active.

2. Click on the appropriate icon as summarised in the table below.

Icon	*Action*
	View the structure from the front.
	View the structure from the top.
	View the structure from the left.
	View the structure from the back.
	View the structure from the bottom.
	View the structure from the right.
	View the structure from the southwest.
	View the structure from the southeast (bottom right).
	View the structure from the northeast (top right).

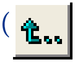
Icon	*Action*
	View the structure from the northwest (top left).



To return a 2D window to its initial state

You have a choice of two options to return any 2D window to its initial state (not zoomed):

1. Click the **Zoom Extents** icon.

2. Click the **First** icon () from the ***View*** toolbar.

Display Modes

You can choose the display mode that you want to use for the normal static view of any graphical window, and also the view that you want to use while you are adjusting it dynamically (zooming any window, panning any window or rotating the ***Structure*** window).

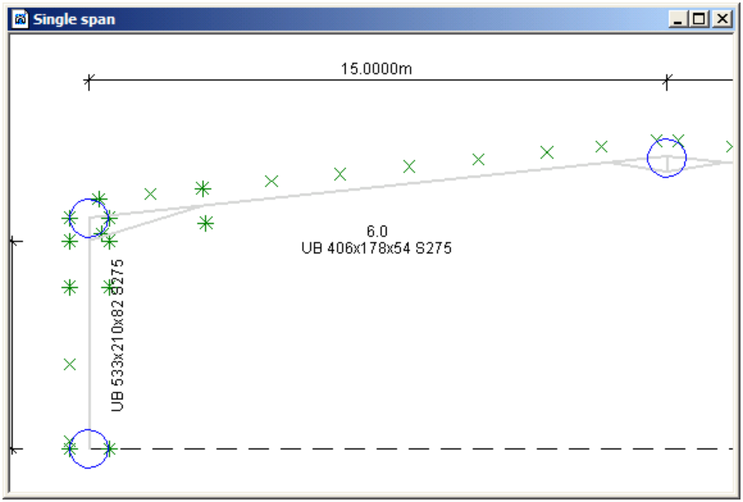
Your choice will depend on the speed of your computer, and on the size and complexity of the building you are dealing with. The options below are listed in order of quickest to slowest and include screen captures of the ***Geometry*** window in each mode.

To set the display mode

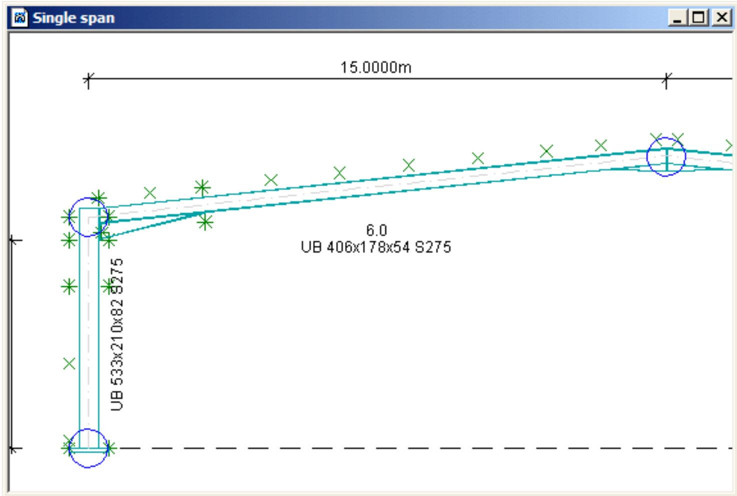
1. Pick the display mode that you want to use for your static and dynamic views from the toolbar as shown below.

Static - Axis

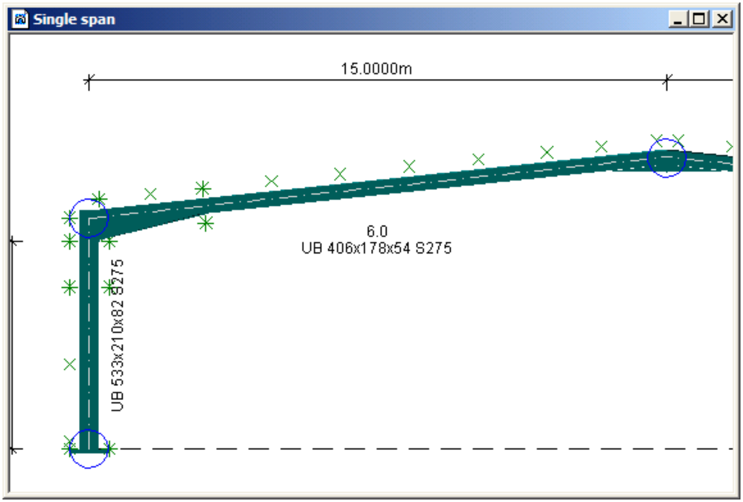
Show the elements in the window by their centre-lines.

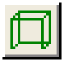


Static - Wired Show the elements in the window by their outlines.



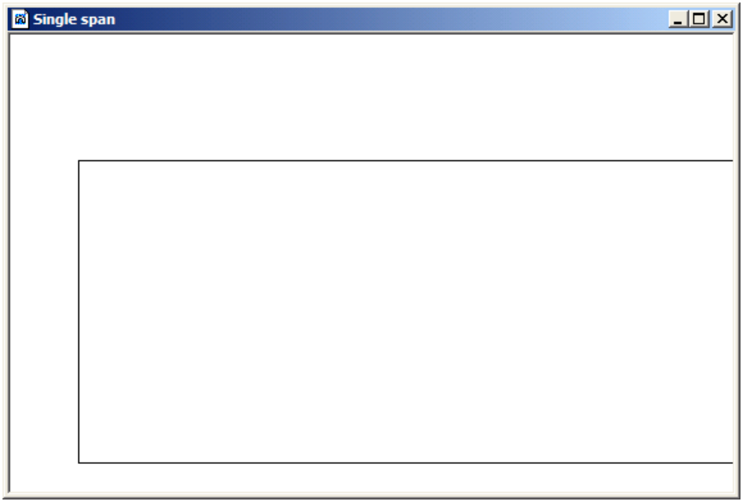
Static - Solid Show the elements in the window fully rendered.





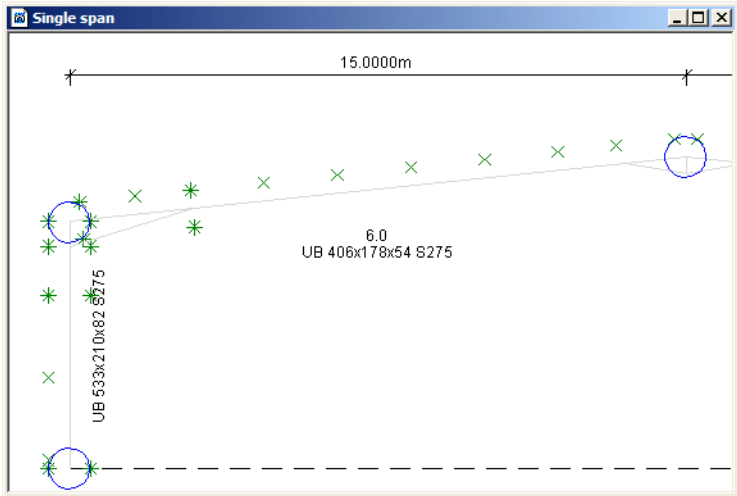
Dynamic - Box

Show the elements in the window as a simple box as you manipulate them.



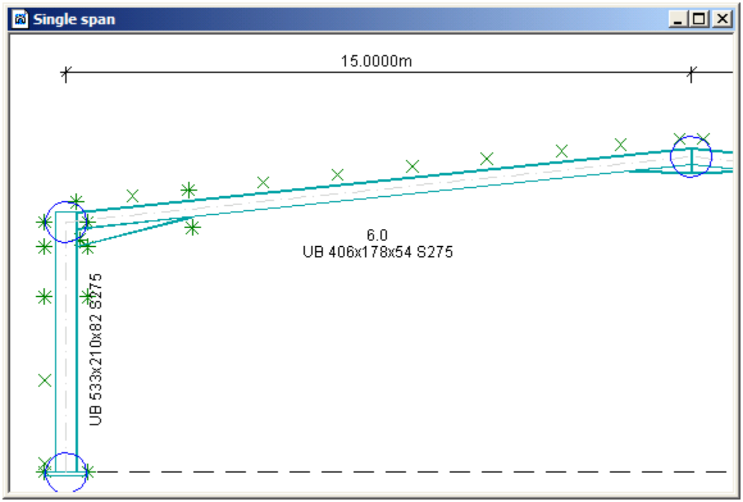
Dynamic - Axis

Show the elements in the window by their centre-lines as you manipulate them.

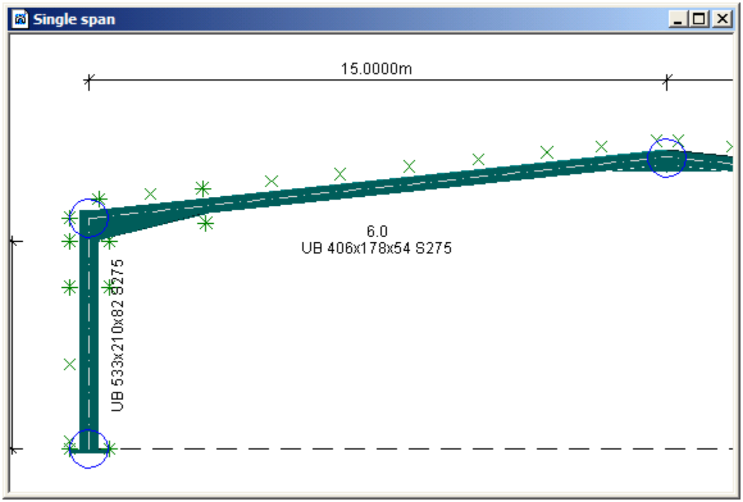


Dynamic - Wired

Show the elements in the window by their outlines as you manipulate them.



Dynamic - Solid Show the elements in the window fully rendered as you manipulate them.

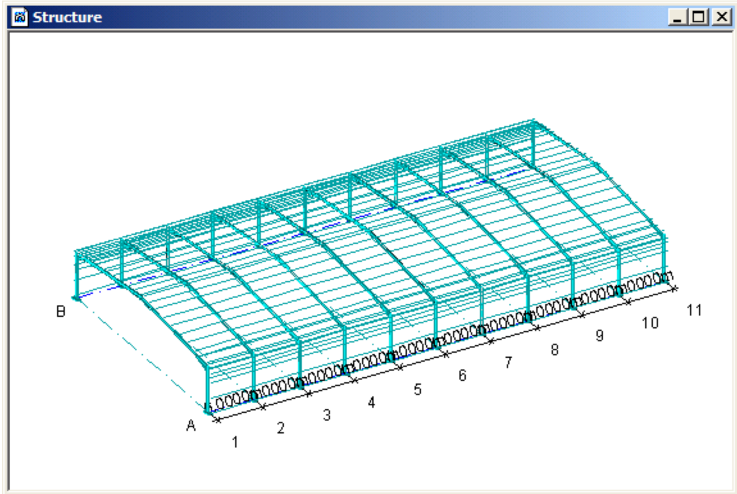


Isometric and Perspective views

You can set the ***Structure*** window to show an **Isometric** or a **Perspective** view.

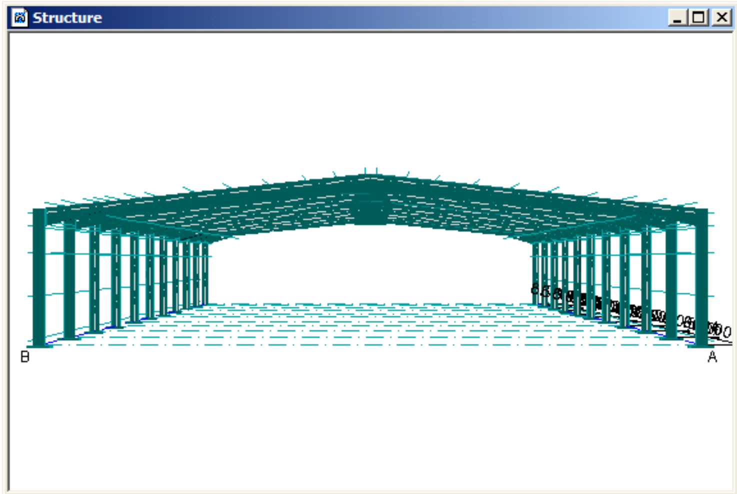
To view as isometric

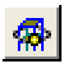
1. Select the ***Structure*** window and click the **Isometric** icon from the toolbar.



To view as perspective

1. Select the ***Structure*** window and click the **Perspective** icon from the toolbar.





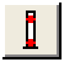
To animate the view

When you are viewing the ***Structure*** window, you can animate the display. Simply click the **Animate** icon to start the animation and click it again to stop the animation.

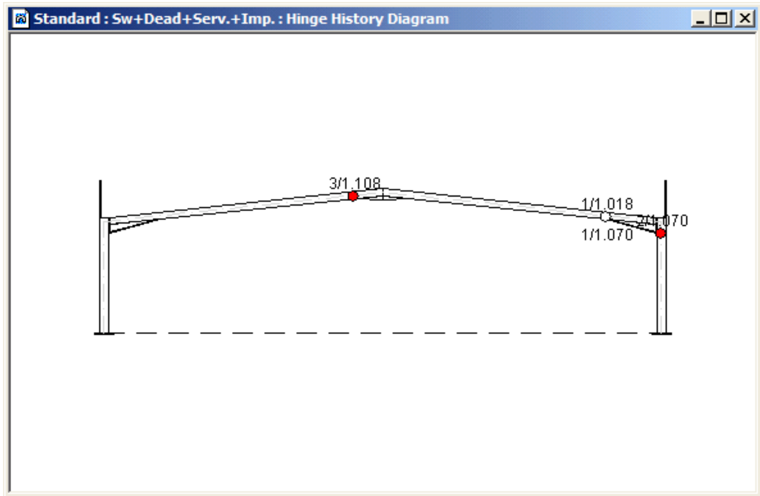
Controlling the analysis results window

Once you have completed the design of a frame you can review the results of the analysis for that frame. When the design completes you will see the ***Design Summary*** dialog, and the ***Analysis Results*** window (which may lie behind the ***Design Summary*** dialog). You can control the information you want to see in the window, and apply scaling factors to enable you to view the window's contents as clearly as possible.

To view the hinge history



1. With the *Analysis Results* window open, click **Hinge History**, and you will see the details for the current combination.



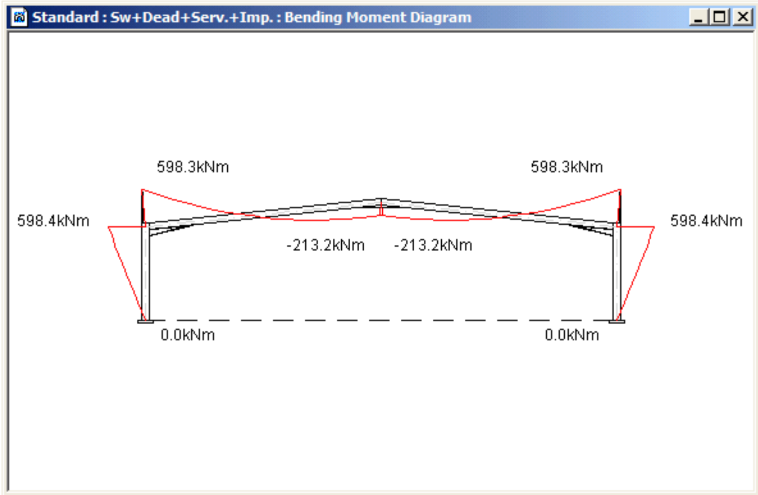
2. Use the *Properties* dialog if you want to change the size of the hinge history dots, to remove the text shown in the window, or change the font used for the text.

3. If you want to see another combination, then double click its name in the *Project Workspace*.

4. If you want to see another diagram, click the appropriate icon from the *Analysis* toolbar.

To view the bending moments

1. With the ***Analysis Results*** window open, click **Bending Moments**, and you will see the details for the current combination.

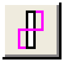


2. Use the ***Properties*** dialog if you want to change the scale of the bending moment diagram, to remove the text shown in the window, or change the font used for the text.

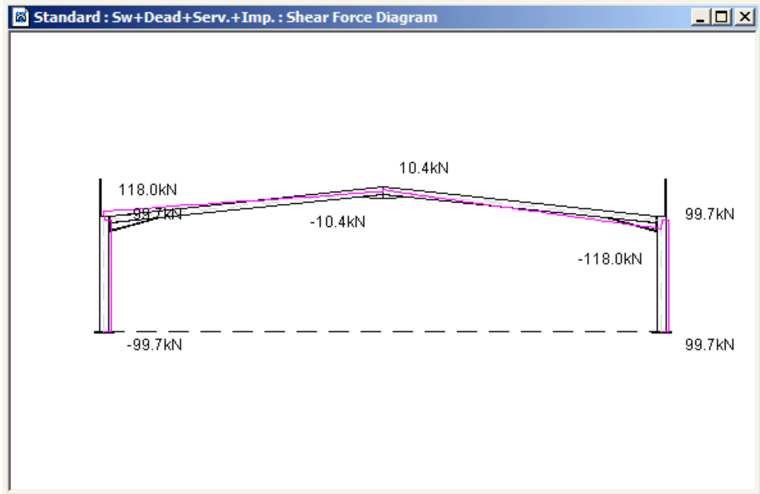
3. If you want to see another combination, then double click its name in the ***Project Workspace***.

4. If you want to see another diagram, click the appropriate icon from the ***Analysis*** toolbar.

To view the shear forces



1. With the ***Analysis Results*** window open, click **Shear Forces**, and you will see the details for the current combination.



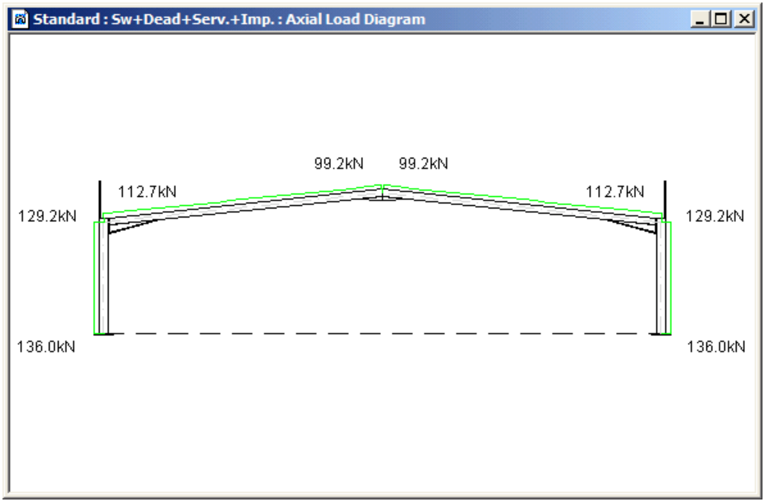
2. Use the ***Properties*** dialog if you want to change the scale of the shear force diagram, to remove the text shown in the window, or change the font used for the text.

3. If you want to see another combination, then double click its name in the ***Project Workspace***.

4. If you want to see another diagram, click the appropriate icon from the ***Analysis*** toolbar.

To view the axial loads

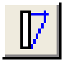
1. With the ***Analysis Results*** window open, click **Axial Loads**, and you will see the details for the current combination.



2. Use the ***Properties*** dialog if you want to change the scale of the axial load diagram, to remove the text shown in the window, or change the font used for the text.

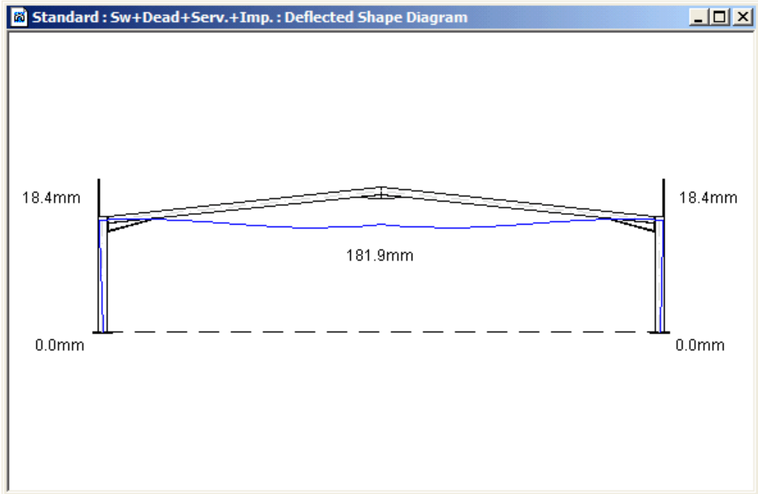
3. If you want to see another combination, then double click its name in the ***Project Workspace***.

4. If you want to see another diagram, click the appropriate icon from the ***Analysis*** toolbar.



To view the deflected shape

1. With the ***Analysis Results*** window open, click **Deflected Shape**, and you will see the details for the current combination.



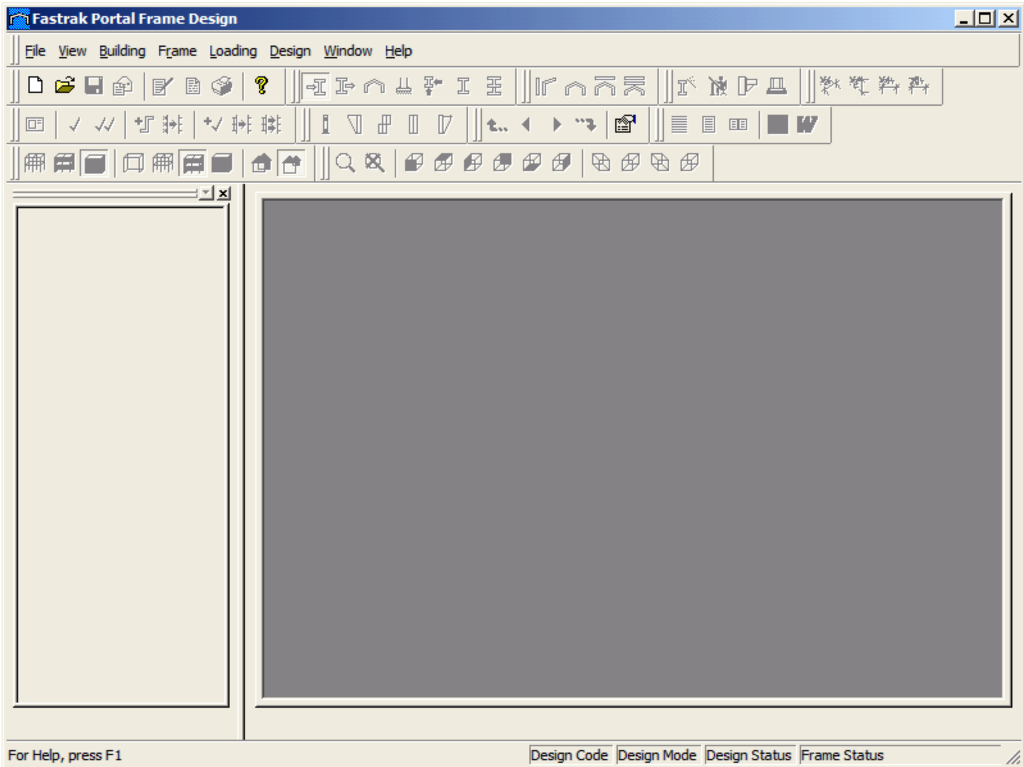
2. Use the ***Properties*** dialog if you want to change the scale of the deflected shape diagram, to remove the text shown in the window, or change the font used for the text.

3. If you want to see another combination, then double click its name in the ***Project Workspace***.

4. If you want to see another diagram, click the appropriate icon from the ***Analysis*** toolbar.

8 Understanding Projects

When you launch *Portal Frame* without opening an existing project you will see the main window.



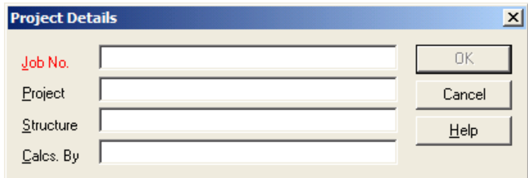
Defining a new project

As covered previously ***Portal Frame*** works around the concept of a project. Before you can define the frames you must first create the project and give its reference information.

Until you do this, and define the details for the building and the first frame, many toolbar icons are dimmed and many menu options are not available.

To define a new project

1. Select *File/New Project…*



2. Complete this dialog by entering the **Job**, **Project**, **Structure** and **Calcs. by** details for the project and then click **OK**.

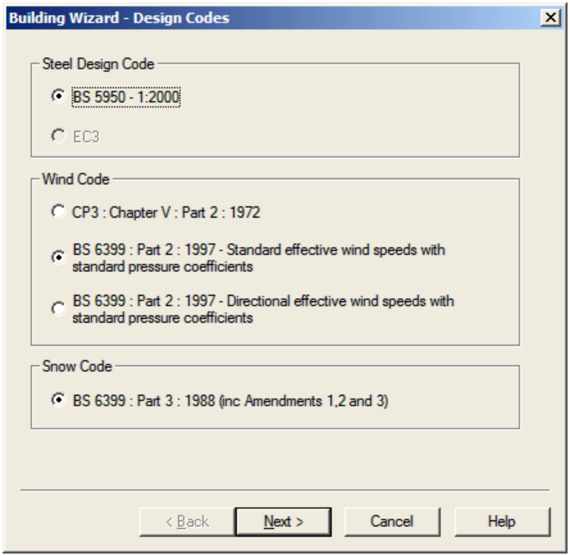
Note You must enter a Job reference, but the other details are optional.

Tip If you want to change these details later you can do so by selecting *File / Project Details…* or by secondary clicking the existing job details in the *Project Workspace* and then selecting *Project Details…* from the context menu that appears.

Building Wizard You will now see the various dialogs of the ***Building Wizard***.

Building Wizard – Design Codes

The ***Building Wizard - Design Codes*** dialog lets you choose the codes that you want to use for this building.

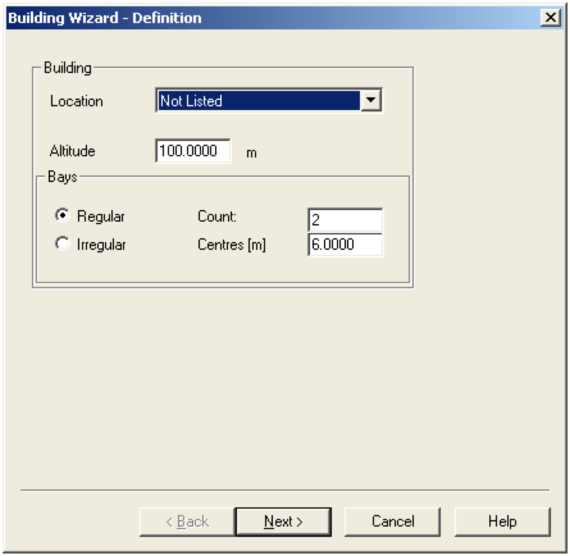


3. Pick the appropriate **Steel**, **Wind** and **Snow** options and then click **Next>**.

Caution *The first BS 6399 option is automatically selected for new projects since CP3 : Chapter V : Part 2 : 1972 has been withdrawn by the BSI. You should only use this code of practice for legacy designs.*

Building Wizard – Building Definition

The ***Building Wizard – Building Definition*** dialog allows you to set the details for your building which are used to generate the wind and snow load information.

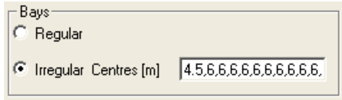


4. Select the **Location** of your building, enter its **Altitude** above sea level.

Note *If your town is not listed, then you can set the basic wind and snow load details yourself later.*

5. If the portal frames in your structure are at equal centres, then pick the **Regular** option and then simply enter the number of frames that your building contains as the **Count** and enter the distance between consecutive frames as the **Centres**.

6. If the portal frames in your structure are ***not*** at equal centres, then pick the **Irregular** option, the dialog reconfigures to allow you to define the irregular frame centres you require.



Enter the distances between the frames, separating them with commas as shown above.

Example If you enter centres of *4.5,6,6,6,6,6,6,6,6,6,6,6,6,4.5* you will create frames at *0*, *4.5, 10.5*, *16.5*, ... *70.5*, *76.5* and *81* metres.

You can enter successive, identical values as a multiple separated by an asterisk, *. So, for example, *6,6,6,6* is the same as *4*6*, so you could enter *4.5,12*6, 4.5* to create frames at the same centres as shown above.

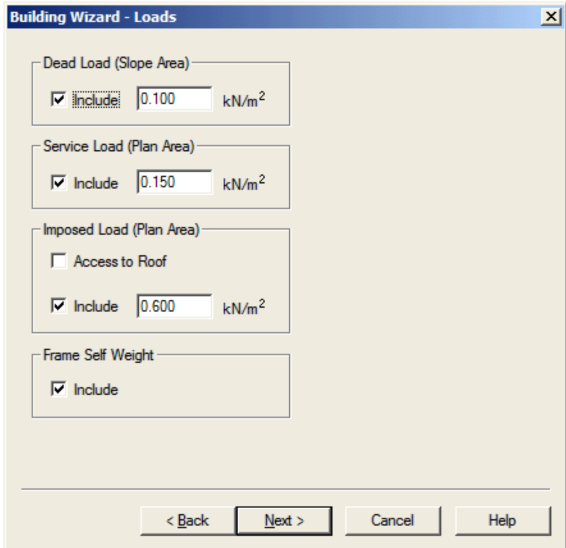
Note You can alter the bay centres later using the *Building Grid* dialog.

Help ***see*** *"To define grid details"* on page 175.

7. Once you have defined your centres click **Next>**.

Building Wizard – Building Loads

The ***Building Wizard - Building Loads*** dialog allows you to define the basic loads that will be applied to all frames in the building.

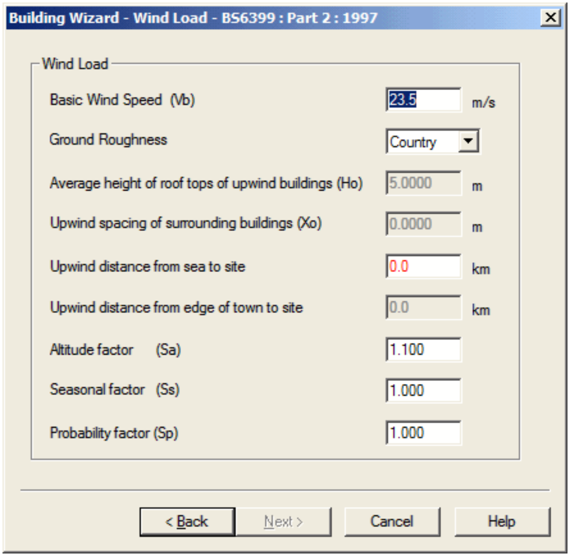


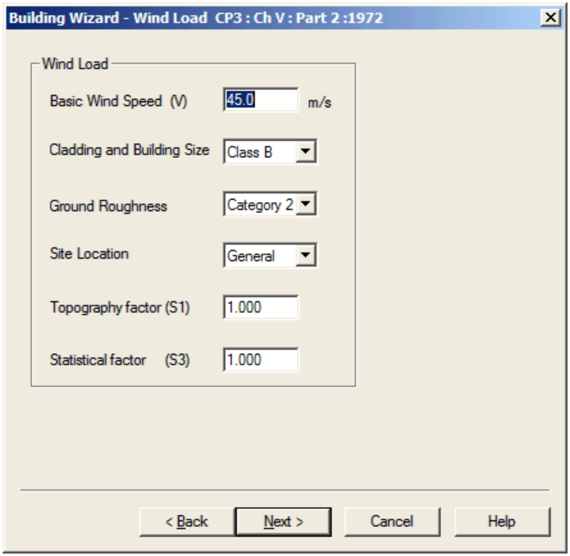
Help *The values which Portal Frame proposes come from your Preferences* ***see*** *"To set frame preferences" on page 87.*

8. Tick the **Include** box for the load type that you want ***Portal Frame*** to generate automatically for you and enter a value where appropriate. Once your loading details are correct click **Next>**.

Building Wizard – Building Wind Load

The ***Building Wizard – Building Wind Load*** dialog that you see depends on the **Wind Code** that you chose earlier. The two options are shown below. You use this dialog to define the basic wind load details for your site.



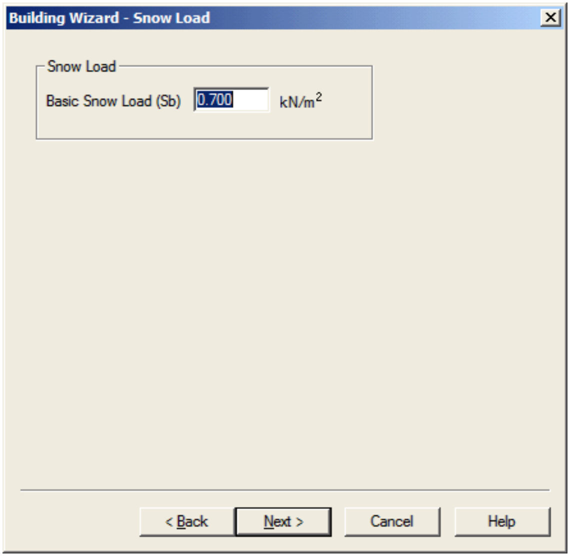


9. The **Basic Wind Speed** default is based on the **Location** that you chose earlier. Change this if necessary and define the other details appropriate to the wind code that you have selected.

10. Once your details are correct click **Next>**.

Building Wizard – Building Snow Load

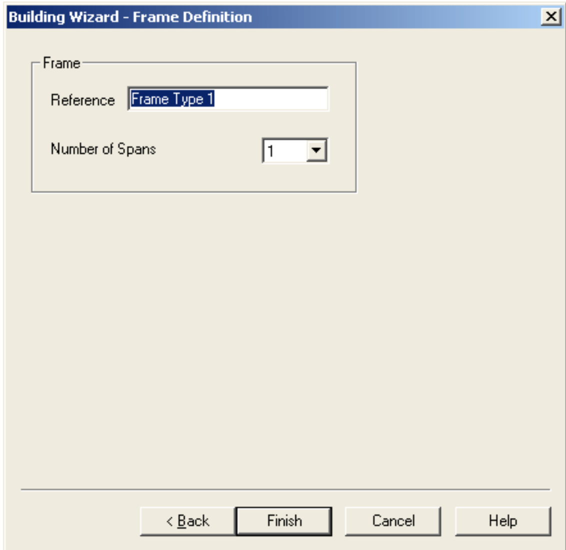
The ***Building Wizard – Building Snow Load*** dialog lets you define the basic snow load for your site.



11. The default value for the **Basic Snow Load** is based on the **Location** that you chose earlier. Change the default value if necessary and then click **Next>**.

Building Wizard – Frame Definition

The ***Building Wizard – Frame Definition*** dialog allows you to define the building details for the first frame in the building.



12. Give the **Reference** for the frame and set the **Number of Spans**.

Note *Portal Frame* automatically calculates the frame centres for each frame based on the building grid you define, you no longer need to define them here.

Help ***see*** *"To define grid details"* on page 175.

13. Once your details are correct to finish with the ***Building Wizard*** and proceed to define the details for the frame.

Help For further information on the definition of frames *see "Defining the project's first frame"* on page 180.

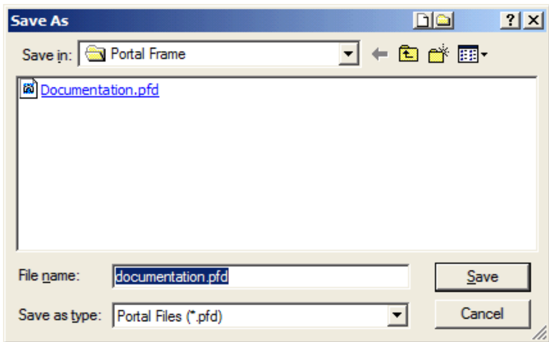
Caution *Your project definition is not complete until you have defined the first frame's details. If you do not complete these frame details you will lose the other details that you gave in the Building Wizard.*

Saving a project

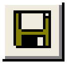
You will probably need to save your project t o disc so that you can pick it up later to make changes without having to enter all the data again.

To save a project for the first time

1. Select *File/Save Project As…*



2. Navigate to the folder where you want to save the project, enter its name and then click **Save**.



To save a project which has been changed

If you change a project you can easily save your changes.

1. Select *File/Save*

   The existing file is overwritten automatically.

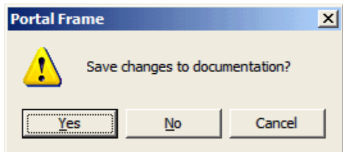
Closing a project

You may want to work on several projects during a single session in ***Portal Frame***, however you can only have one project open at once. When you want to move from one project to another you must close the current project before opening the new one.

To close a project

1. Select *File/Close Project*

   If you have changed, but have not saved the current project ***Portal Frame*** prompts you to save it.



   Respond to this dialog and you can then create a new project or open an existing one.

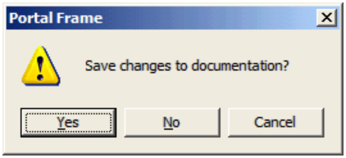
Opening a project

If you want to make changes to a saved and closed project you will first need to open it.

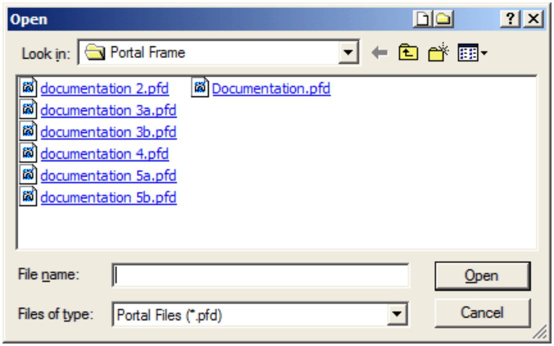
To open a project

1. Select *File/Open…*

If you already have a project open which has been changed but not saved you need to decide whether or not to save the changes.



Respond to this dialog and you can then pick the project to open.



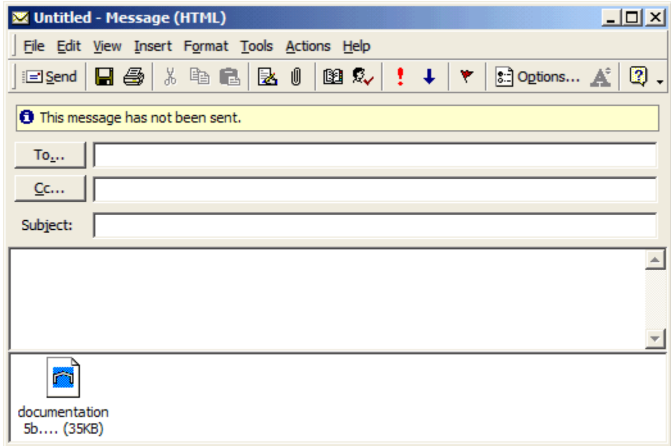
2. Navigate to the folder where the project is saved. Select it and then click **Open**.

Sending mail

If you have any queries about the designs you create in ***Portal Frame***, or if you encounter unexpected features you will need to contact our support department for assistance. Generally you will be asked to provide a copy of your project. If you have email, you can use ***Send Mail*** to create an e-mail with your project as an attachment.

To send mail

1. Select *File/Send Mail…*



Note *Portal Frame* has automatically attached a copy of your project file to the new message.

2. Enter the e-mail address of the recipient in the **To…** box, and optionally enter a **Subject** and a **Message** to accompany the project.

Note The e-mail address for the UK CSC Support Department is *support@cscworld.com*

3. Click **Send** to send the e-mail.

Note If you do not have a permanent e-mail connection you might need to trigger the sending of the e-mail from your e-mail application.

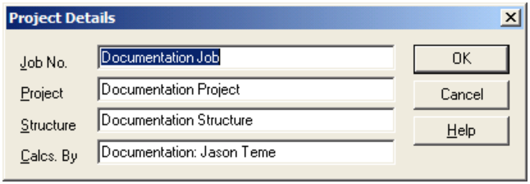
Modifying the information for a project

Once you have defined a project you may subsequently need to alter some of its information should circumstances dictate.

To modify the project details

You may need modify the project details should you need to submit calculations for the same building to more than one potential client.

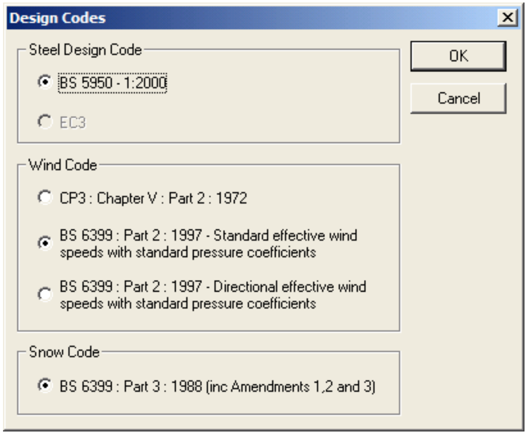
1. Select *File/Project Details…*



2. Make your changes and click **OK**.

To change the design codes

1. Select *Building/Design Code…*



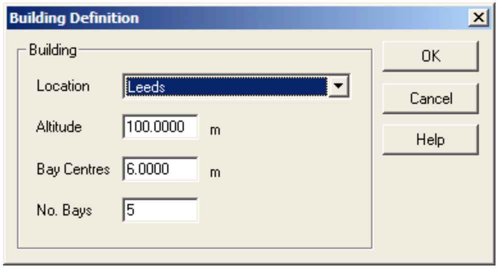
2. Make the changes you require and then click **OK**.

Caution *CP3 : Chapter V : Part 2 : 1972 has been withdrawn by the BSI. You should only use this code of practice for legacy designs.*

Caution *Don't forget that if you change the wind code you'll also need to modify the wind load details.*

To modify the building definition

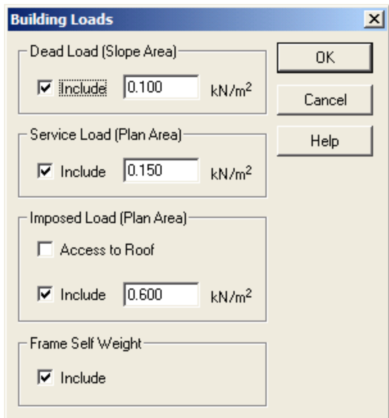
1. Select *Building/Definition…*



2. Change the details as necessary and then click **OK**.

To modify the building loads

1. Select *Building/Loads…*



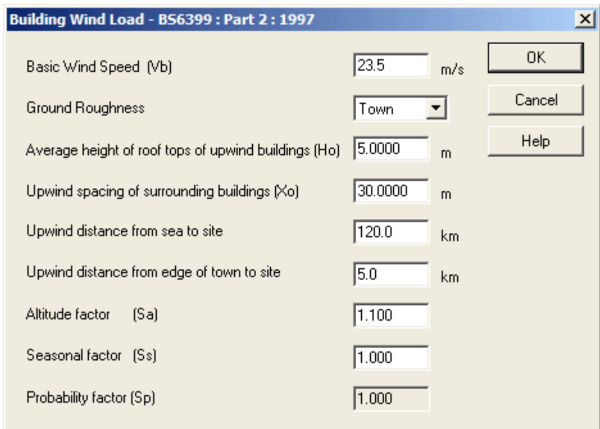
2. Make your changes and click **OK**.

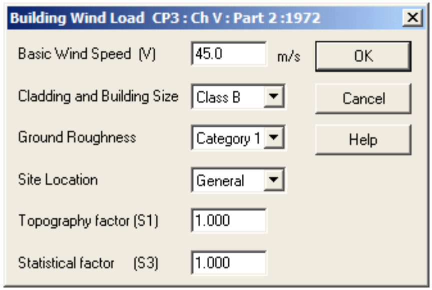
Caution If you change a value here, *Portal Frame* recalculates the associated value in the frame loadcase overwriting any manual changes that you have made.

Caution If you remove the check against **Include** for any item *Portal Frame* removes the automatically generated load from the loadcase. If the loadcase is then empty *Portal Frame* deletes it automatically.

To modify the wind loads

1. Select *Building/Wind Load…*





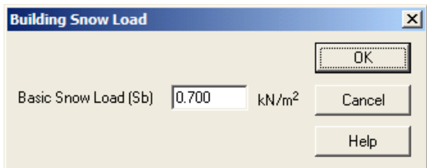
Note The dialog depends on the Wind code.

2. Make any changes you require and then click **OK**.

Help For further information on the details for CP3 **see** *"To modify building wind load data"* on page 390 or for BS 6399 **see** *"To modify building wind load data"* on page 371.

To modify the snow loads

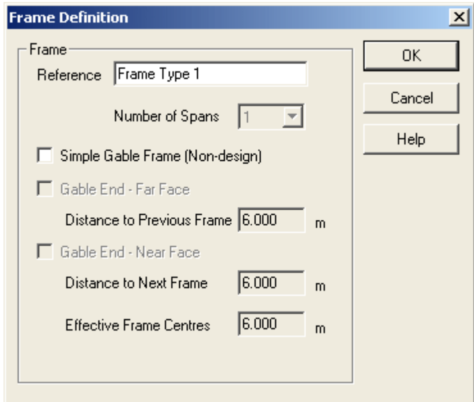
1. Select *Building/Snow Load...*



2. Change the details as necessary and then click **OK**.

To modify the frame definition details

1. Select *Frame/Frame Definition…*



Note This option allows you to change the basic definition details of the frame shown in the current *Frame* window. If the active window is not that for a frame, then the menu option is not available.

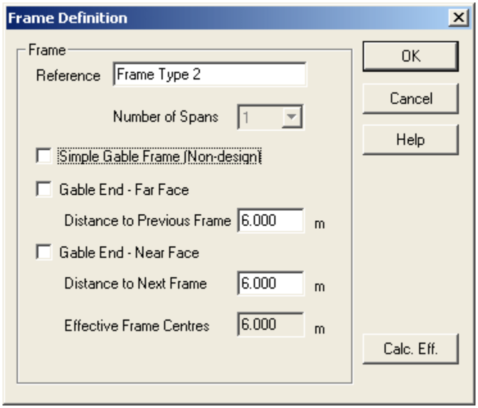
2. Change the **Reference** details as necessary.

Note You cannot change the number of spans here, since you cannot control where a span is to be added or which span is to be deleted. However when you are editing the details for the span geometry you can add or delete spans as necessary.

3. If you do not want ***Portal Frame*** to design the frame, but you want to define its details directly (for model completeness and without design tick **Simple Gable Frame (Non-design)**. and then click **OK**. The remaining details in the dialog are not appropriate if you tick this option, and so they are removed.

Note Although this option is primarily intended for gable frames (as its title indicates), you can use it for any frame in your structure which you don't want to design.

4. If the current frame is involved in the building grid system, then the remaining details are dimmed, since they are automatically determined from the grid. If the frame is not involved in the building grid, then you can enter the appropriate details.



Note By default the initial frame that you define (*Frame Type 1*) is involved in the grid system and so these details are dimmed. If you want to define your own frame centres rather than have *Portal Frame* use those calculated from the building grid, then we suggest that you delete *Frame Type 1* (you must first define another frame, since your project must always contain at least one frame) and then not involve your other frames in the building grid.

If this frame is a gable frame, then tick the appropriate **Gable End** field (**Near Face** if the gable will lie at the start of the building (grid line 1), **Far Face** if it will lie at the end of the building). If you tick one of these options, then the associated **Distance** field is not appropriate and is dimmed.

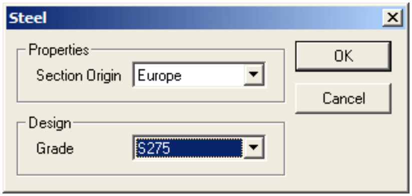
Enter the **Distances** that are required and click **Calc. Eff.** to calculate the effective width of loading which is carried by this frame.

Once your details are correct click **OK**.

To change the steel for the building

When you define a portal building the region whose steel sections you want to use and the default grade are taken from your preferences. You can change this information for the entire building at once if necessary

1. Select *Building/Design Steel…*



2. Select the region of origin and the grade that you want to use then click **OK**. The region and grade that you specify will be applied to all members in all frames in your project.

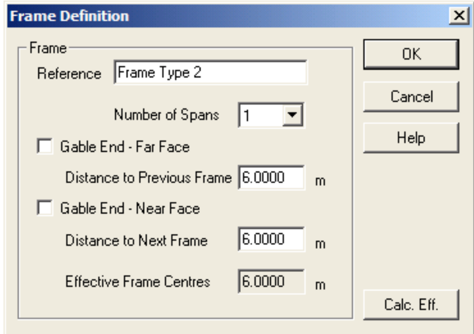
Adding a new frame

Your project may contain as many frames as you like[1]. You can create a new frame from scratch, or you can copy an existing frame and then modify it appropriately.

1. *Limited only by the memory and resources available to your computer.*

To add a new frame

1. Select *Frame/New…*



2. Give the **Reference** of the frame, define the appropriate details and click **OK** to define its details.

Help For further information on the definition of frames ***see*** *"Defining the project's first frame"* on page 180.

Note If you place your frame onto the building grid, then the frame centres that are used will be those calculated from the grid system, and these will override any values that you specify here. If this occurs, then when you close the *Building Grid* dialog you will see a warning message to this effect.

Help ***see*** *"To define grid details"* on page 175.

To copy an existing frame

1. Select the ***Frame Definition*** or ***Analysis Results*** window for the frame that you want to copy.

2. Select *Frame/Copy*

   A new frame will be created which is identical to the selected frame. This frame will have a new reference based on the number of frames that already exist in your project, you can then change this as necessary.

Help For further information on modifying a frame reference **see** *"To modify the frame definition details"* on page 163.

Selecting the frame on which to work

A project can contain as many frames as you like[1]. When you add a frame, or copy an existing one you will automatically start to work on that new frame. You can pick any frame in order to work on it.

To pick the frame on which to work

1. Click the title of a window for the frame on which you want to work.

Help You can also pick the frame using the *Project Workspace*. For further details **see** *"To choose the frame on which to work"* on page 510.

Removing a frame

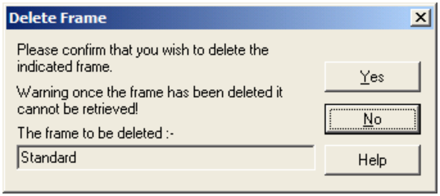
In some cases you may find that your project contains frames that you no longer need. You can delete these from the project.

To delete a frame

1. Make sure that you are working on the frame you want to remove.

1. *Limited only by the memory and resources available to your computer.*

2. Select *Frame/Delete…*



Check that the reference is for the frame that you want to remove and then click **Yes** to delete it.

Caution *Once a frame has been removed from the project in this way its information* ***can not be recovered. Please take care when you use this feature.***

Exporting information from Portal Frame

Portal Frame provides a wide range of options to export information from your model in formats which you can use with other programs. These include other CSC programs as well as those from external sources.

These export facilities are in addition to the options to export material lists to ***Excel***.

Help For further information **see** *"To export a materials list to Excel"* on page 174

They are also in addition to the options to export reports to ***TEDDS*** or ***Word***.

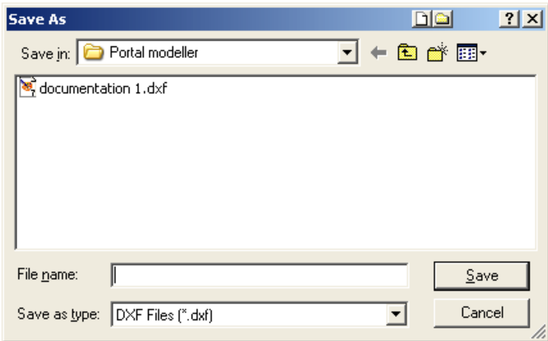
Help For further information **see** *"To transfer the report to TEDDS"* on page 504 and *"To transfer the report to Microsoft Word"* on page 505.

To export a frame to a dxf file

When you have created a model in *Portal Frame*, you can immediately transfer the details of any frame to a `dxf` file. You can only obtain this information for the frame *Definition* window.

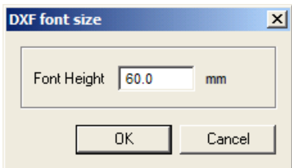
As well as giving details of the sections (reference marks and sizes) the `dxf` file includes the dimensions of the frame.

1. Click *File/Export/Export to DXF...*



2. Enter the **File name** that you want to use and then click **Save.**

3. You will now see a dialog which allows you to give the size of the text that is to be used in the `dxf` file.

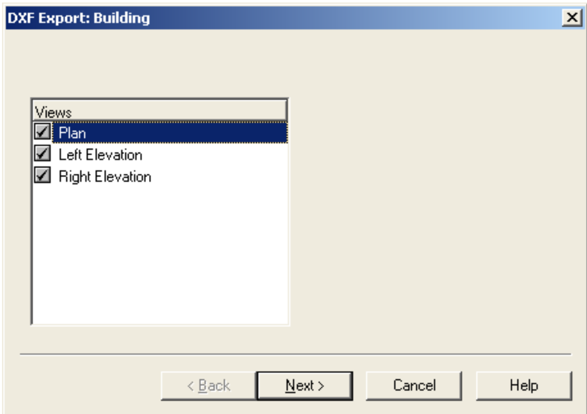


4. Enter an appropriate value for the text size (which depends on the scale that you feel appropriate for the `dxf` file) and then click **OK** to create the file.

To export the Structure view to a dxf file

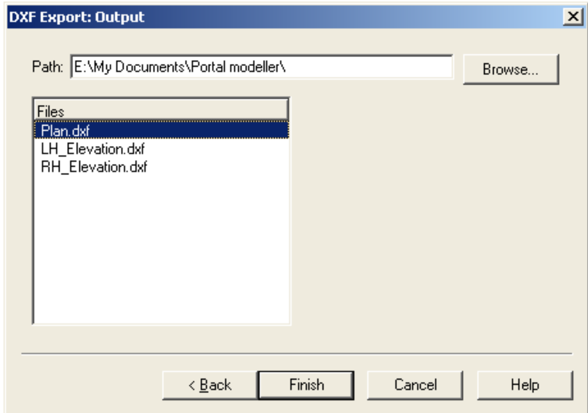
When you have created a model in ***Portal Frame***, you can immediately create `dxf` files containing a plan and elevations of your structure. You can only obtain this information when the ***Structure*** window is active.

1. Click *File/Export/Export to DXF...* and in this case you will see the ***DXF Export: Building*** dialog.



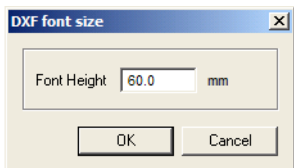
2. Tick the boxes for the views you want to create and then click **Next>**.

3. You will see the ***DXF Export: Output*** dialog which gives the names of the files which contain the views which you have requested and the folder in which they will be placed.



Note *The default path is that of the folder containing your model. You can change this as necessary.*

4. Click **Next>** to see a dialog which allows you to give the size of the text that is to be used in the `dxf` file.

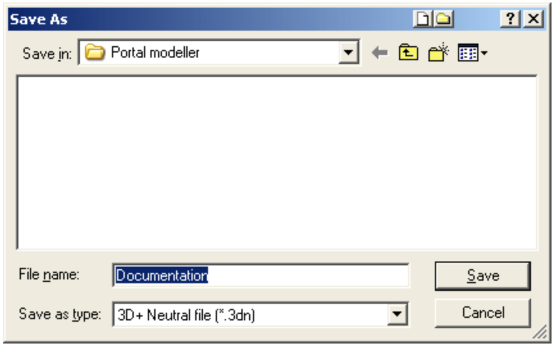


5. Enter an appropriate value for the text size (which depends on the scale that you feel appropriate for the `dxf` file) and then click **OK** to create the file.

To export model details to 3D+

When you have created a model in ***Portal Frame***, you can immediately transfer the physical model to ***3D+***[1] for drawing production.

1. Click *File/Export/Export to 3D+…*

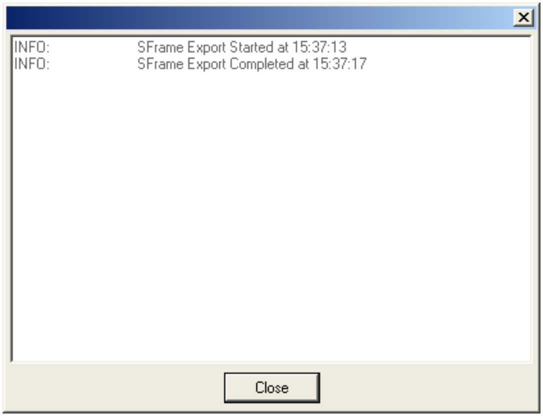


To export a model to S-Frame

When you have created a model in ***Portal Frame***, you can immediately transfer the model to ***S-Frame***[2]. If you have performed a design in ***Portal Frame***, then all the section sizes will be transferred to ***S-Frame*** for you. You can use ***S-Frame*** to analyse your entire structure and then use its link to ***S-Steel***[3] to perform the design of any members you require. There is no risk of changing the sizes that ***Portal Frame*** has already determined for the main frame members, since these are locked and excluded from an ***S-Steel*** design.

1. *Assuming that you have purchased, installed and licensed a copy of* ***3D+****.*
2. *Assuming that you have purchased, installed and licensed a copy of* ***S-Frame****.*
3. *Assuming that you have purchased, installed and licensed a copy of* ***S-Steel****.*

1. Click *File/Export/Export to S-Frame…* You will see a dialog which tells you of any issues that have arisen during the export process.

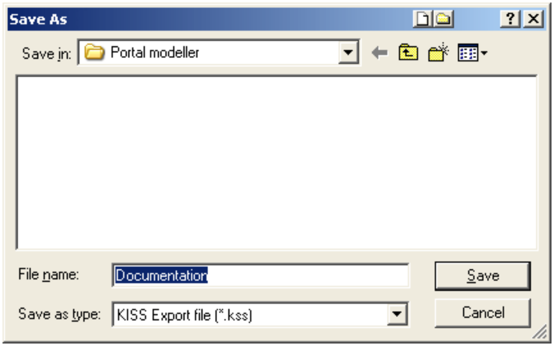


Once the export is complete ***S-Frame*** launches and automatically loads the exported ***Portal Frame*** model.

To export a model to a MIS system

When you have created a model in ***Portal Frame***, you can immediately transfer it to any MIS system which can read the industry standard KISS file format.

1. Click *File/Export/Export to MIS…*



2. Enter the **File name** that you want to use and then click **Save** to create the file.

To export a materials list to Excel

When you have created a model in ***Portal Frame***, you can immediately export the material list details of the members it contains to ***Excel***, where you can work on it further.

1. Click *File/Export/Export Material List to Excel*.

2. ***Excel*** will launch and the transferred material list details will be shown.

9 Building Grids in Portal Frame

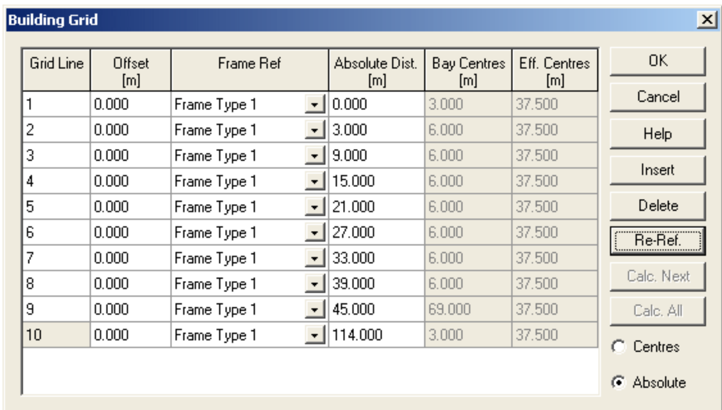
When you have defined the different frames in your building you can assemble these into a pseudo-3D building. You can then use the ***Wind Loading Generator*** and ***Snow Loading Generator*** to calculate the wind and snow loads on the building, or you could use this information to transfer the building data to a 3D structural detailing package.

Defining grid details

You define your building grid by specifying the frame type on each grid line the number of which depends on your ***Building Definition's*** **No. of Bays**.

To define grid details

1. Select *Building/Grid…*

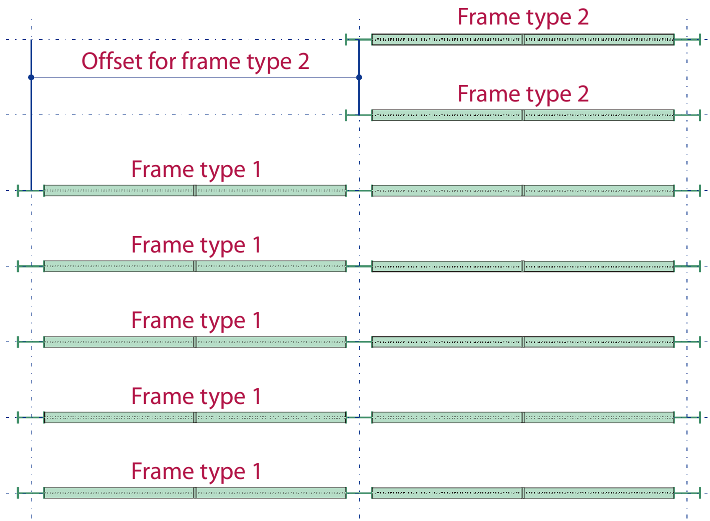


This shows all the grid lines in your structure along which the portal frames lie. The initial settings for this dialog are based on the information that you entered in the ***Building Wizard*** as you created the project and on the details for the first frame which you defined.

?

Help *see "Building Wizard – Building Definition"* on page 147 and *"Defining span geometry"* on page 180.

2. For each grid line you can set the **Offset** of the frame as illustrated below. This allows you to define a frame with fewer spans, and align these appropriately.



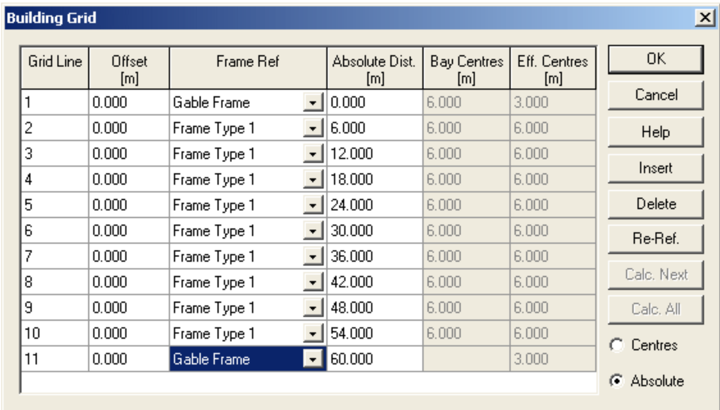
3. You can then pick the **Frame** type that you want to place on the grid line by choosing its reference from the list which will show the references of the different frames which you have created for this structure.

4. With respect to the centres you can choose whether you want to define the distances between the frames in terms of an **Absolute** dimension from some set-out point, or in terms of the **Centres** between adjacent frames.

Note If you change the frame centres from those you defined initially you will see a warning message to tell you that those loads which depend on the frame centres have been automatically recalculated for you.

Caution If you have a series of frames of the same type (*Frame Type 1* and so on), but these are at different centres, then *Portal Frame* will determine the most onerous centres and will use this value as the **Effective Centres** for all frames of this type.

If you do not want this to happen, then you will need to create differing frame types with identical geometry (you can copy one frame type to create as many variations as you like), and then place these appropriately in your building so that each frame type picks up the loading you intend.



Grid Line	Offset [m]	Frame Ref	Absolute Dist. [m]	Bay Centres [m]	Eff. Centres [m]
1	0.000	Gable Frame	0.000	6.000	3.000
2	0.000	Frame Type 1	6.000	6.000	6.000
3	0.000	Frame Type 1	12.000	6.000	6.000
4	0.000	Frame Type 1	18.000	6.000	6.000
5	0.000	Frame Type 1	24.000	6.000	6.000
6	0.000	Frame Type 1	30.000	6.000	6.000
7	0.000	Frame Type 1	36.000	6.000	6.000
8	0.000	Frame Type 1	42.000	6.000	6.000
9	0.000	Frame Type 1	48.000	6.000	6.000
10	0.000	Frame Type 1	54.000	6.000	6.000
11	0.000	Gable Frame	60.000		3.000

As soon as you change the frame reference the new effective centres are calculated automatically.

5. If you need to change the number of frames in a structure, then you can pick the line for one frame, and click **Insert** to add a new frame immediately before the selected frame. If you need to remove a frame, then you can pick its line in the table and then click **Delete**. If you are adding or deleting frames

and you need to change the frame centres, then you can ripple the changes through your structure quickly by judicious clicking of **Calc. Next** and/or **Calc All**.

6. Once you have the grid arrangement that you require click **OK**.

10 Span Definition in Portal Frame

When you create a new project in ***Portal Frame*** you give its reference details and use the ***Building Wizard*** to define the basic building details. Only when you have done this can you continue and define the span details for the first frame.

Defining the project's first frame

When you create a new project and have given the basic building details ***Portal Frame*** automatically allows you to define the first frame's geometry.

Help For further information ***see*** *"To define a new project"* on page 145 and *"Building Wizard"* on page 145.

Adding a new frame

If you are adding a new frame to a project, then you define the frame details and then continue to define the geometry of the frame.

Help For further information ***see*** *"Adding a new frame"* on page 165.

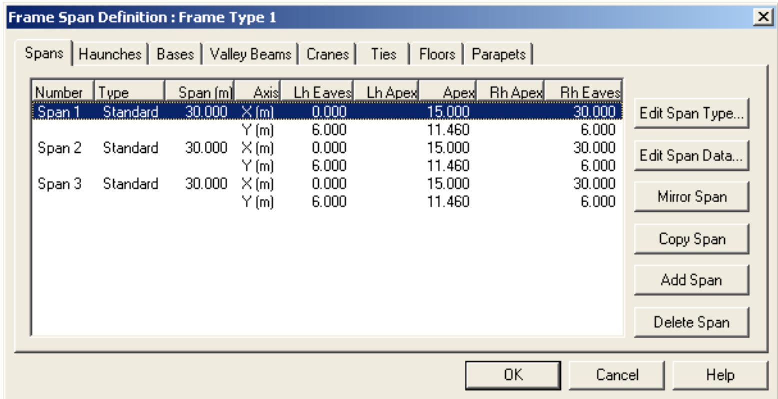
Defining span geometry

If you are defining a new project, or adding a new frame you will be asked to define the geometry of the span as part of the creation process.

Caution Your project definition is not complete until you have defined the first frame's details. If you do not complete these frame details you will lose the other details that you gave in the *Building Wizard*.

To define span geometry

1. Select *Frame/Span Definition* to see the ***Span Definition*** property sheet. (This step is only required if you are changing the details for a frame as otherwise you will automatically see the ***Span Definition*** property sheet.) The property sheet will be configured to show the number of spans that you have specified or previously defined.



To pick a span type

When you are defining a new frame the ***Spans*** page has all span types set to standard and their details are zeroed. The information for a span depends on its type and you must therefore set this first.

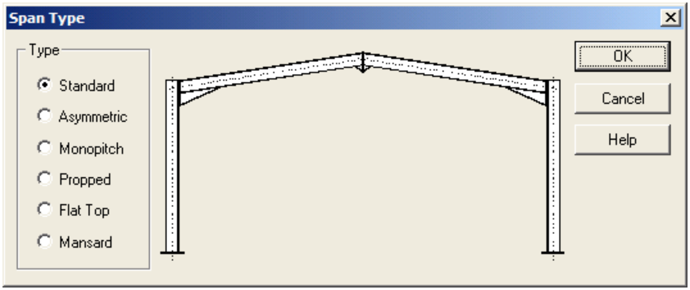
Note *The information also depends on any existing information for the spans to either side of it.*

Example *If you have defined the right base and eaves level of the span to the left of this one the left base and eaves level of this span will show these values.*

1. Select a line in the table that shows the details for the span whose type you want to change.

Note If you want to use a standard span, then you do not need to change it, you can click **Edit Span Data** to give its details immediately.

2. Click **Edit Span Type...** to see the ***Span Types*** dialog.



3. Select the **Type** of span and then click **OK** to proceed to the ***Span Input*** dialog for that type.

Help For further information on the dialog for the different span types ***see***:

- *"To define a standard span"* on page 184,
- *"To define an asymmetric span"* on page 186,
- *"To define a monopitch span"* on page 189,
- *"To define a propped span"* on page 191,
- *"To define a flat top span"* on page 194,
- *"To define a mansard span"* on page 197.

4. Complete this dialog and click **OK** to return to the ***Span Definition*** property sheet which will reflect the span details.

5. Continue this process until you have defined all spans.

?

Help

The **Copy Span** and **Mirror Span** options enable you to duplicate or reflect details of existing spans to build up your total frame layout. For further information ***see***:

- *"To mirror span details"* on page 200,
- *"To copy span details"* on page 201.

6. Now you can continue and define the details for the haunches, bases, valley beams and any other additional steelwork that you require.

?

Help

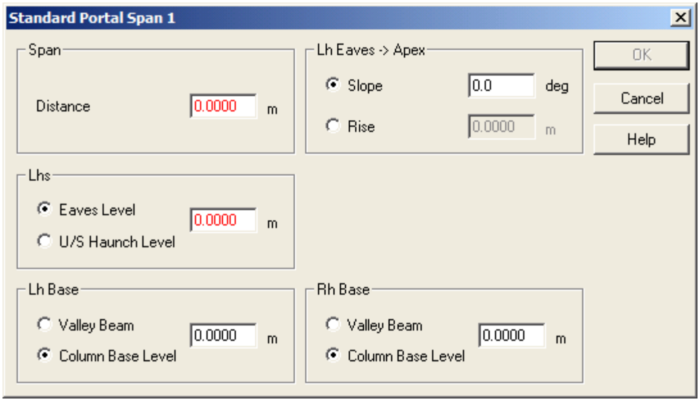
For further information ***see***:

- *"Haunch Geometry in Portal Frame"* on page 204,
- *"Column Base Fixity in Portal Frame"* on page 209,
- *"Valley Support Fixity in Portal Frame"* on page 218,
- *"Crane Geometry in Portal Frame"* on page 221,
- *"Tie Geometry in Portal Frame"* on page 225,
- *"Parapet Geometry in Portal Frame"* on page 234.

7. Once your span details are complete click **OK** to return to the main ***Portal Frame*** window which will show your frame diagrammatically.

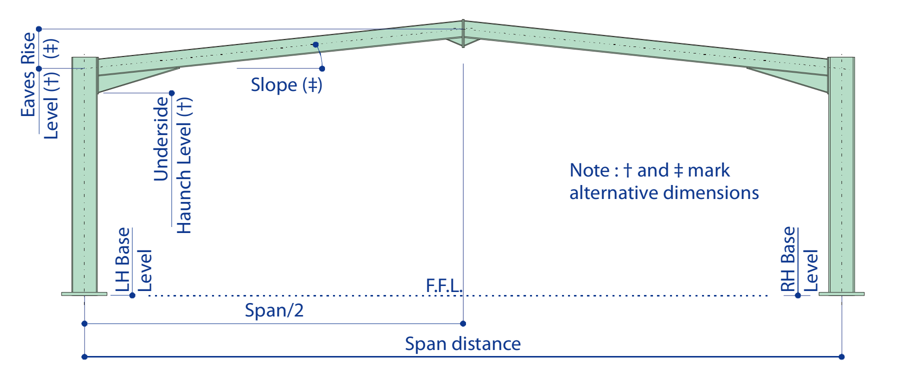
To define a standard span

You define or edit a standard span using the ***Standard Portal*** dialog.

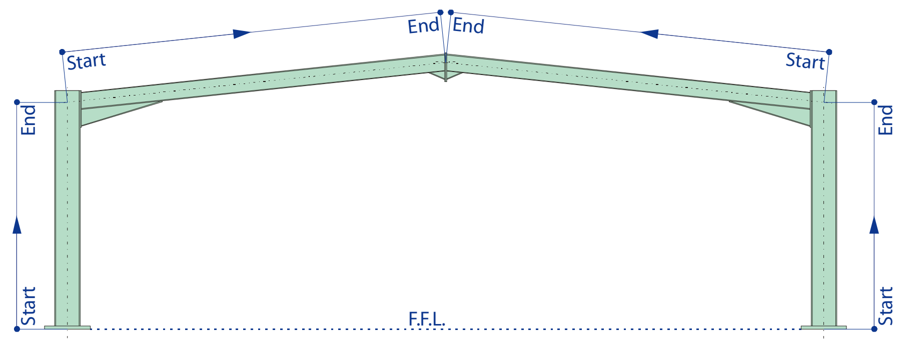


This allows you to define portal frames of the type shown below.

Standard span dimensions



Standard span start and end points



1. Enter the **Span Distance**.

2. Pick the way in which you want to define the apex position and then enter the **Slope** or the **Rise**.

3. Pick the method that you want to use to define the **Left Hand Eaves Level** and enter the appropriate value.

Example You would probably use the **Underside of Haunch** option in order to achieve a minimum clear height.

Note The right and left hand haunches are at the same level for this type of portal.

4. Pick the left hand base **Type** (and for a column base enter its **Level**).

Help You can set the default level for your bases, ***see*** *“To set frame preferences”* on page 87,

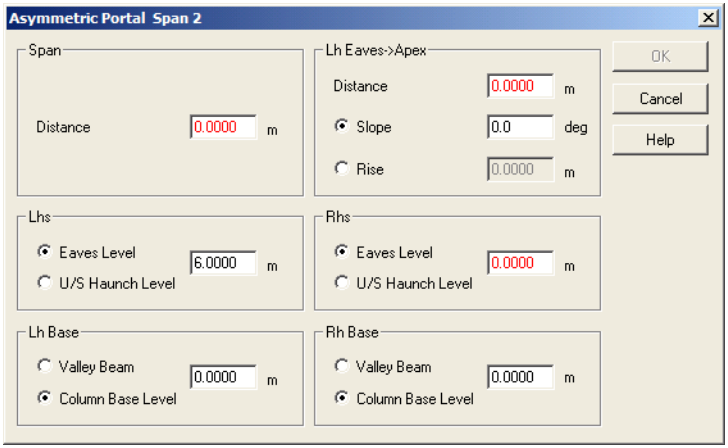
5. If necessary change the **Type** and **Level** of the right hand base.

6. Once your details are correct click **OK** to return to the ***Span Definition*** property sheet.

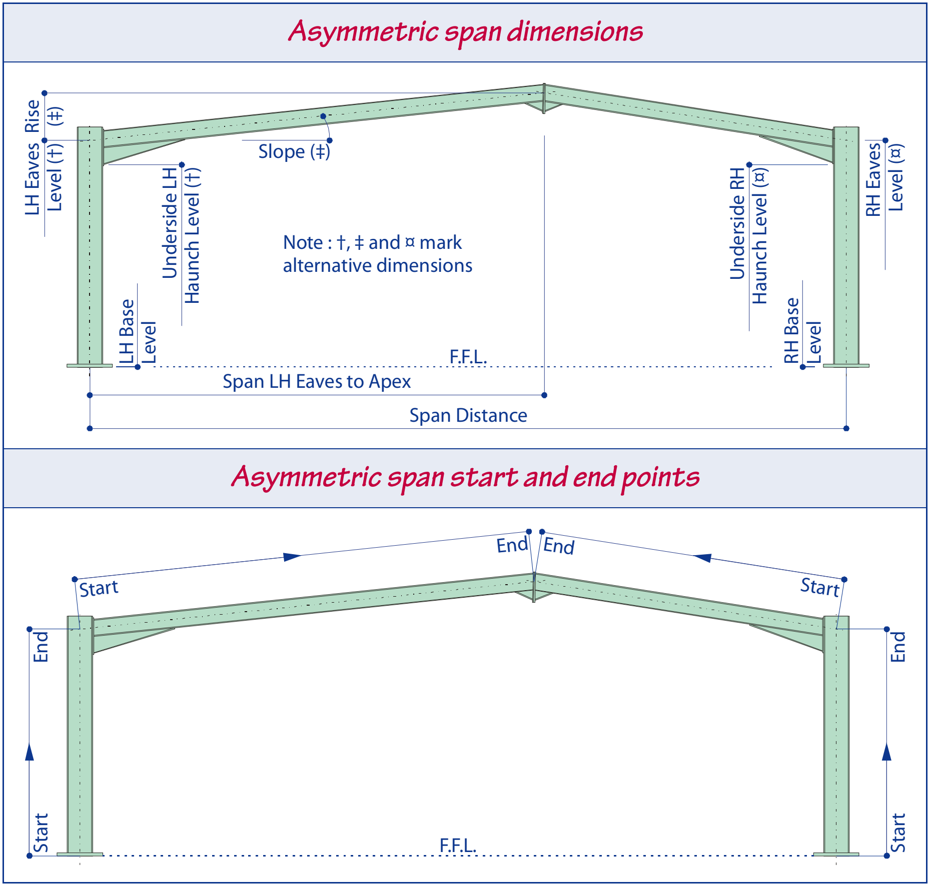
Note If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.

To define an asymmetric span

You define or edit an asymmetric span using the ***Asymmetric Span*** dialog.



This allows you to define portal frames of the type shown below.



1. Enter the **Span Distance**.

2. Enter the horizontal **Distance** to the apex from the eaves. Pick the way in which you want to define its vertical position and then enter the **Slope** or the **Rise**.

3. Pick the method that you want to use to define the **Left Hand Eaves Level** and enter the appropriate value.

Example *You would probably use the* **Underside of Haunch** *option in order to achieve a minimum clear height.*

4. Enter the **Right Hand Haunch Level** similarly.

5. Pick the left hand base **Type** (and for a column base enter its **Level**).

Help *You can set the default level for your bases,* ***see*** *"To set frame preferences" on page 87,*

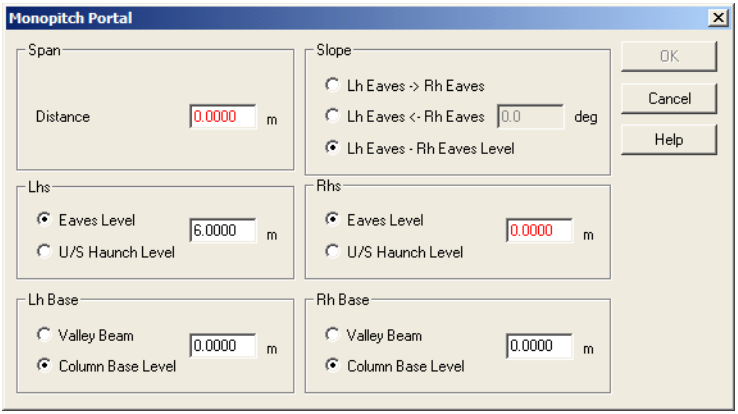
6. If necessary change the **Type** or **Level** of the right hand base.

7. Once your details are correct click **OK** to return to the ***Span Definition*** property sheet.

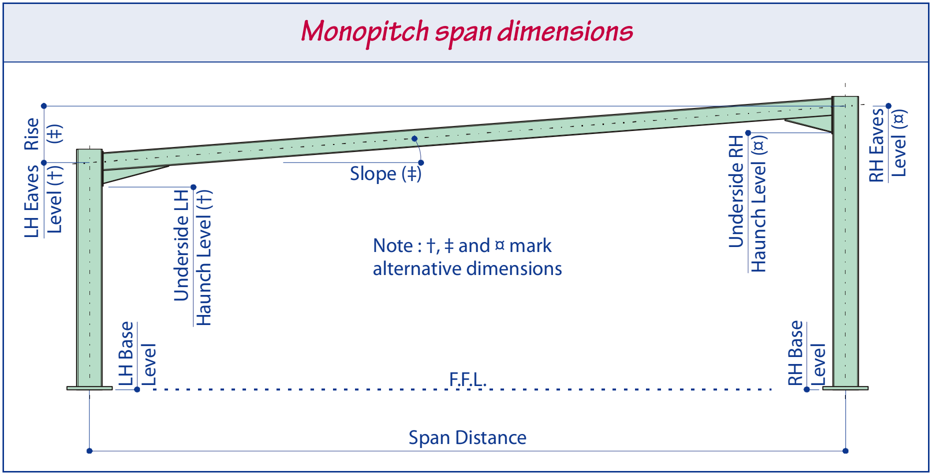
Note *If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.*

To define a monopitch span

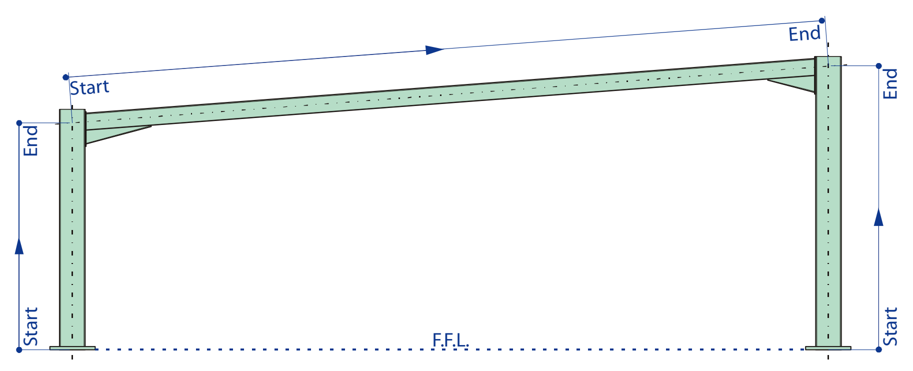
You define or edit a monopitch span using the *Monopitch Span* dialog.



This allows you to define portal frames of the type shown below.



Monopitch span start and end points



1. Enter the **Span Distance**.

2. Pick the way in which you want to define the vertical position of the right hand eaves and then enter the **Slope**, **Rise**. If you pick the options to use the **Levels**, then the details you define for the eaves define this data.

3. Pick the method that you want to use to define the **Left Hand Eaves Level** and enter the appropriate value.

E

Example *You would probably use the* **Underside of Haunch** *option in order to achieve a minimum clear height.*

4. Pick the left hand base **Type** (and for a column base enter its **Level**).

?

Help *You can set the default level for your bases,* ***see*** *"To set frame preferences" on page 87,*

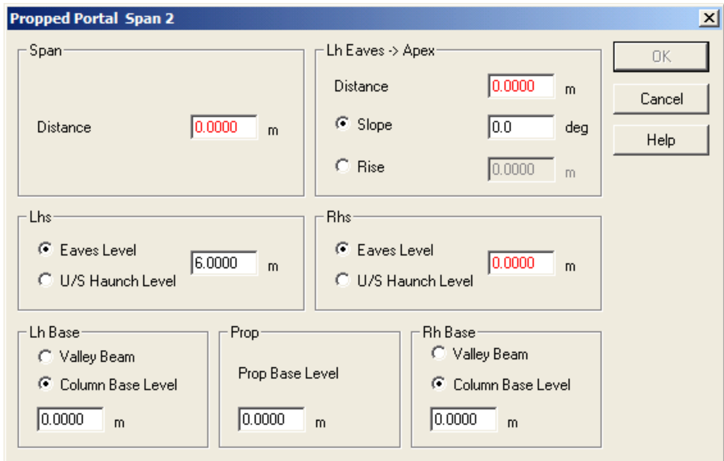
5. Similarly set the **Type** or **Level** of the right hand base.

6. Once your details are correct **OK** to return to the *Span Definition* property sheet.

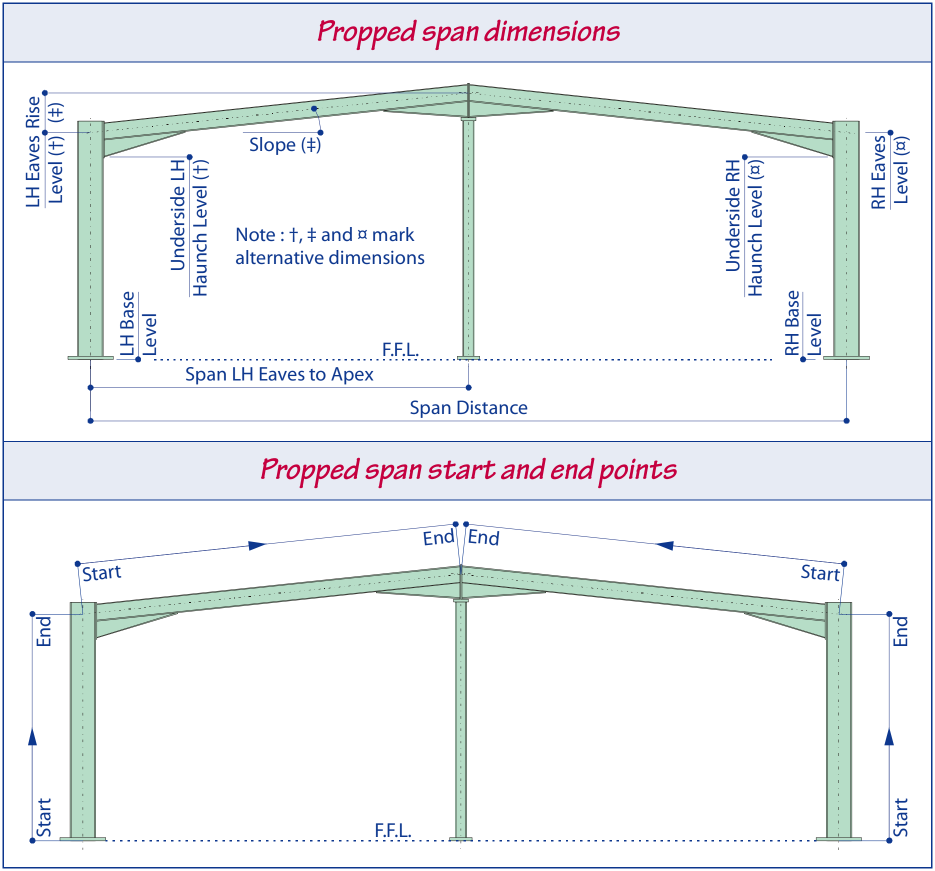
Note *If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.*

To define a propped span

You define or edit a propped span using the *Propped Span* dialog.



This allows you to define portal frames of the type shown below.



1. Enter the **Span Distance**

2. Enter the horizontal **Distance** to the apex from the eaves. Pick the way in which you want to define its vertical position and then enter the **Slope** or the **Rise**.

3. Pick the method that you want to use to define the **Left Hand Eaves Level** and enter the appropriate value.

Example *You would probably use the* **Underside of Haunch** *option in order to achieve a minimum clear height.*

4. Define the **Right Hand Haunch Level** similarly.

5. Pick the left hand base **Type** (and for a column base enter its **Level**).

Help *You can set the default level for your bases,* ***see*** *"To set frame preferences" on page 87.*

6. If necessary change the **Type** or **Level** of the right hand base.

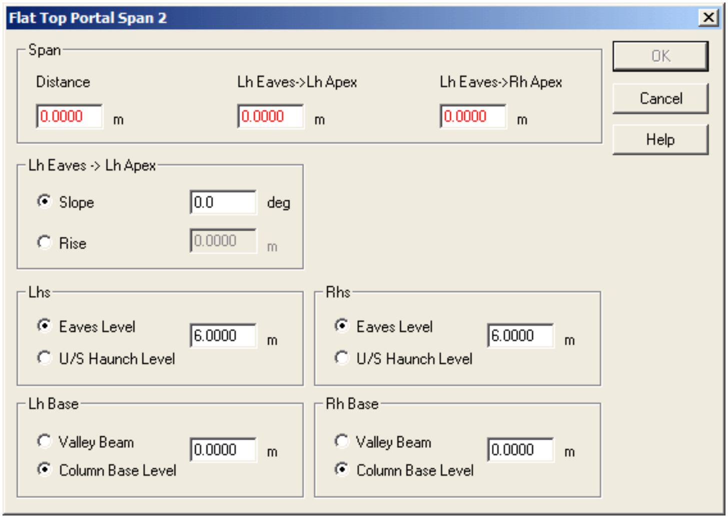
7. Enter the prop base level.

8. Once your details are correct click **OK** to return to the ***Span Definition*** property sheet.

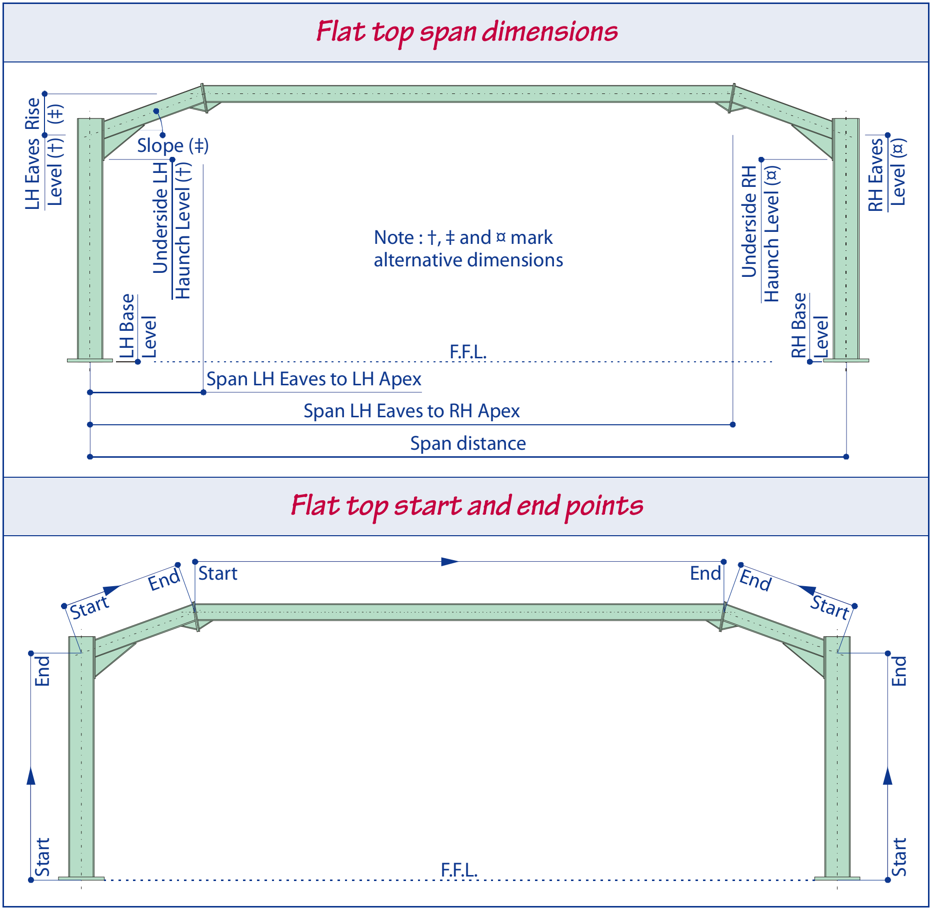
Note *If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.*

To define a flat top span

You define or edit a flat top span using the ***Flat Top Span*** dialog.



This allows you to define portal frames of the type shown below.



1. Enter the **Span Distance**.

2. Enter the horizontal **Distance** from the left hand eaves to the left hand apex and that from the left hand eaves to the right hand apex.

3. Pick the method that you want to use to define the vertical position of the left hand apex and then enter the **Slope** or the **Rise**.

4. Enter the **Left Hand Eaves Level** and enter the appropriate value.

Example *You would probably use the* **Underside of Haunch** *option in order to achieve a minimum clear height.*

5. Similarly enter the **Right Hand Eaves Level**.

6. Pick the left hand base type (and for a column base enter its **Level**).

Help *You can set the default level for your bases,* ***see*** *"To set frame preferences" on page 87.*

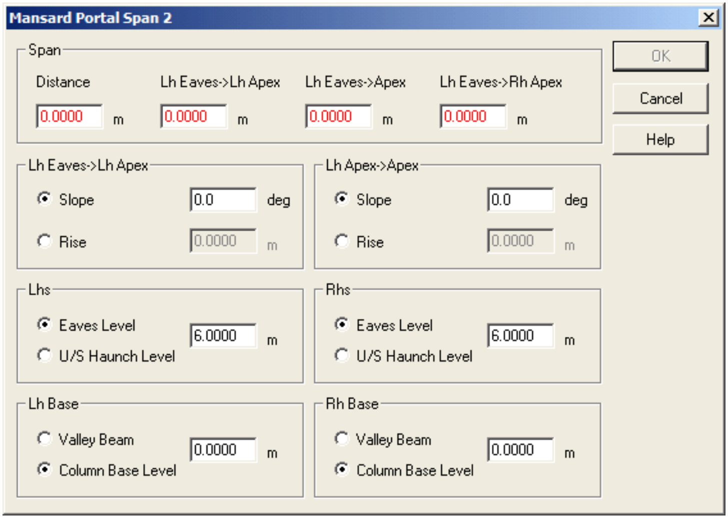
7. If necessary change the **Type** or **Level** of the right hand base.

8. Once your details are correct click **OK** to return to the ***Span Definition*** property sheet.

Note *If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.*

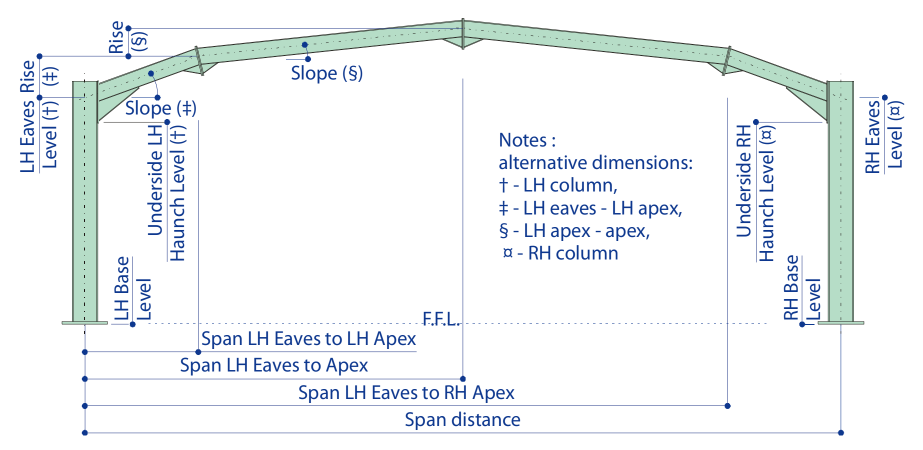
To define a mansard span

To define or edit a Mansard span you use the ***Mansard Span*** dialog.

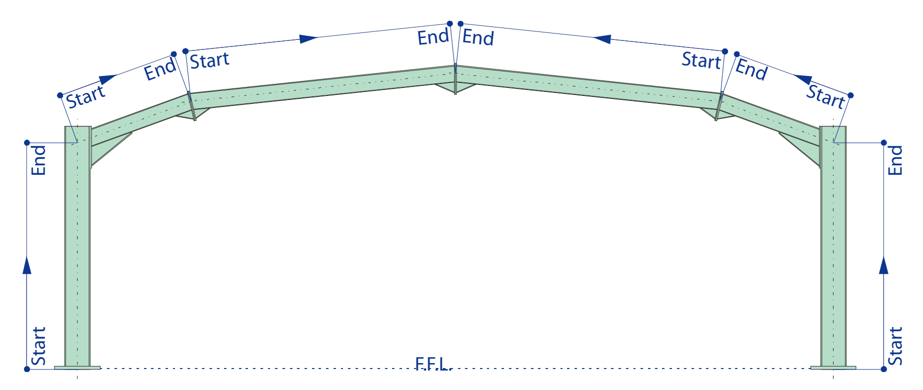


This allows you to define portal frames of the type shown below.

Mansard span dimensions



Mansard span start and end points



1. Enter the **Span Distance**.

2. Define the horizontal **Distances** from the left hand eaves to the left hand apex, the apex and the right hand apex.

3. Pick the way in which you want to define the vertical position of the left hand apex and then enter the **Slope** or the **Rise**.

   Note *These details are also used to determine the vertical position of the right hand apex from the right hand eaves.*

4. Similarly define the vertical position of the Apex from the left hand apex.

5. Pick the method that you want to use to define the **Left Hand Eaves Level** and enter the appropriate value.

6. Define the **Right Hand Eaves Level** similarly.

7. Pick the left hand base **Type** (and for a column base enter its **Level**).

   Help You can set the default level for your bases, **see** *"To set frame preferences"* on page 87.

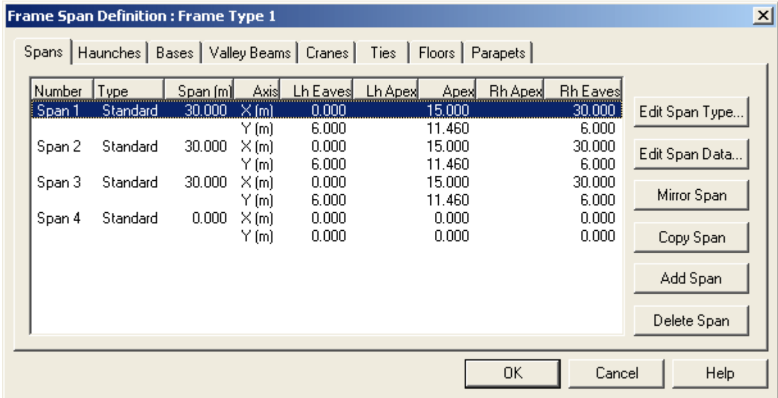
8. If necessary change the **Type** or **Level** of the right hand base.

9. Once your details are correct click **OK** to return to the ***Span Definition*** property sheet.

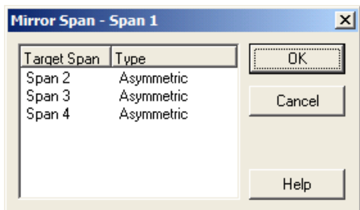
   Note *If any of your details are invalid they will show as red text. Simply rest the pointer over the text to see the valid range.*

To mirror span details

Portal Frame allows you to mirror the information from one span to one or more of the others.



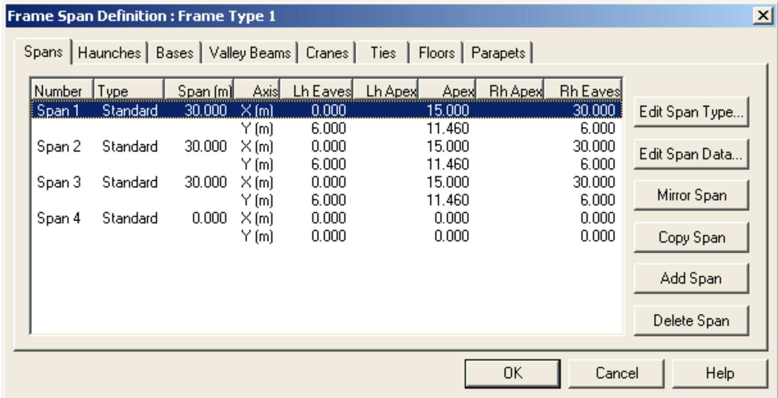
1. First define the span that you want to mirror. Now select a line relating to that span on the *Span Definition* property sheet and click **Mirror Span**. You will see the *Mirror Span* dialog.



2. Select the span(s) to which you want to mirror the selected spans details and click **OK**. The details for the mirrored spans will be reflected in the *Span Definition* property sheet.

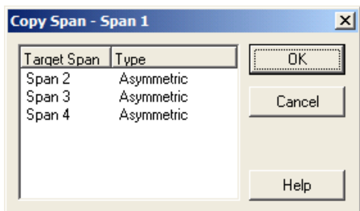
To copy span details

Portal Frame allows you to copy the information for one span into any of the others.

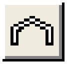


1. Completely define the span that you want to copy, and return to the ***Frame Span Definition*** property sheet.

2. First define the span that you want to copy. Now select a line relating to that span on the ***Span Definition*** property sheet and click **Copy Span**. You will see the ***Copy Span*** dialog.



3. Select the span(s) to which you want to copy the selected spans details and click **OK**. The details for the copied spans will be reflected in the ***Span Definition*** property sheet.



To modify the span geometry

1. Select *Frame/Span Definition*

Hot spot Click the **Span** hot spot (shown by blue text and surrounded by a blue box) for the span that you want to change.

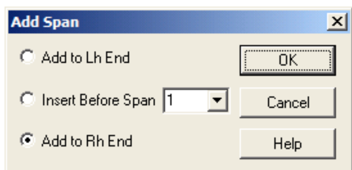
2. You will see the ***Span Definition*** property sheet. Select a line relating to the span you want to change and then click **Edit Span Data**. You will see the dialog appropriate to that span type.

Note If you want to change the **Type** of span, then select a line relating to that span and click **Edit Span Type**. Select the new **Type** of span then click **OK**. You will then see the dialog for the new span type.

3. When you have made all your changes click **OK** to close the ***Span Definition*** property sheet. The graphical display will show the amended frame details.

To add a span to a frame

1. Click **Add Span…** you will see the ***Add Span*** dialog.



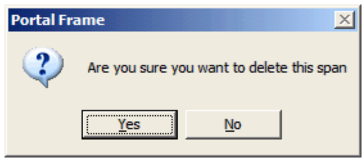
Note If your frame already has the maximum number of spans[1] **Add Span...** is dimmed.

2. Select the position where you want to add the new span and then click **OK**.

3. The ***Span Definition*** property sheet will show the new span. You can define its details directly in any way you like.

To delete a span from a frame

1. Select a line relating to the span which you want to delete and then click **Delete Span**. You will be asked to confirm the deletion.



Take note of the message and respond appropriately.

2. If you click **OK** the span will be deleted with all its details.

Note You can not delete the only span in a frame,

Caution You can not undo this action. **Please take due care when using this feature.**

1. *This depends on the version of* ***Portal Frame*** *you are using.*

11 Haunch Geometry in Portal Frame

Portal Frame lets you define the length and depth of each haunch.

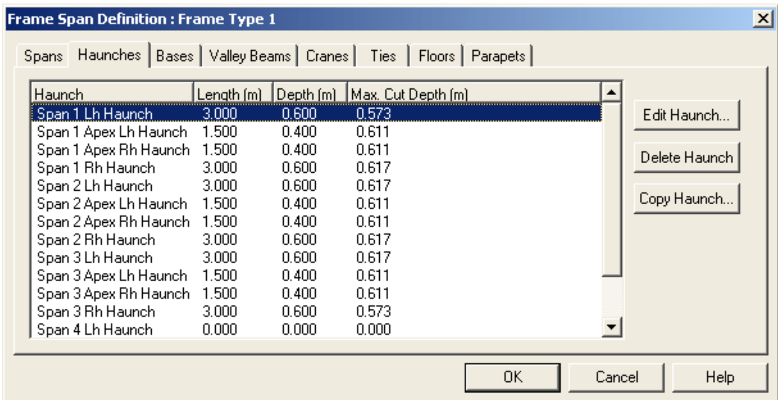
Defining haunch geometry

If you have set eaves or apex haunch preferences the ***Span Definition Haunches*** page will show the values calculated in accordance with these. Otherwise you will see *No haunch* against each haunch.

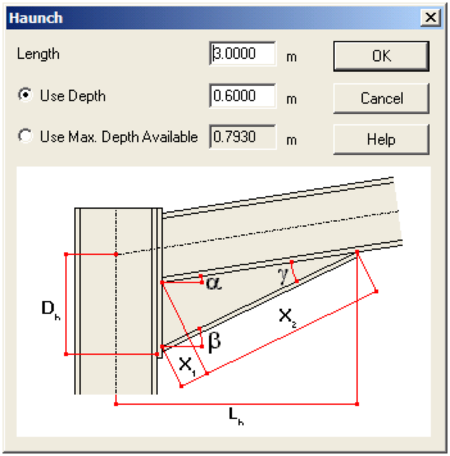
Help For further information **see** *"To set frame preferences"* on page 87.

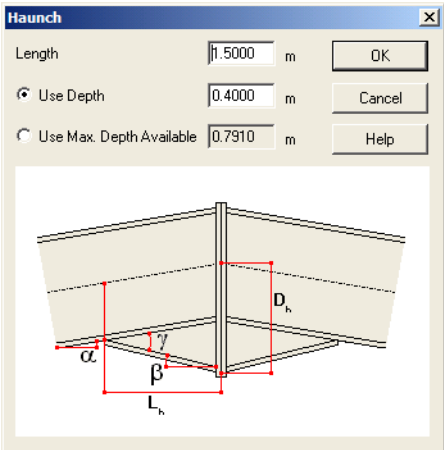
To modify the haunch geometry

1. Select *Frame/Span Definition…* and the ***Haunches*** tab.



2. Select the haunch you want to amend and then click **Edit Haunch** to see the *Haunch* dialog.





3. Enter the **Length** and **Depth** of your haunch.

4. If the column (eaves haunch only), rafter and haunch section sizes are known (as for a check design or after an automatic one) you can pick **Use Maximum Available Depth** to use the calculated maximum possible depth.

Caution The maximum depth is calculated for the current section sizes. If these change you will need to check that the depth of haunch can be achieved with the new sections.

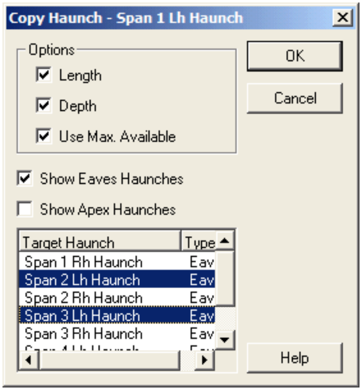
Note The haunch is only checked in the context of the entire frame when you **OK** the *Span Definition* property sheet.

5. Once your haunch details are correct click **OK** to return to the ***Haunches*** page of the ***Span Definition*** property sheet.

To copy haunches

Portal Frame allows you to copy the information for one haunch to all other haunches within the frame.

1. Select the line for the haunch you want to copy on the ***Haunches*** page and then click **Copy Haunch** to see the *Copy Haunch* dialog.



2. Tick the options to indicate the information you want to copy.

Note *If you tick **Use Max. Available** then the maximum depth available for each target haunch will be calculated and used and not the maximum depth of the source haunch.*

3. Tick the haunches that you want to see in the target haunch list.

4. Select the target haunches and then click **OK**. The details of the copied haunches will be updated on the ***Haunches*** page.

To delete a haunch

1. Select the haunch that you want to delete and click **Delete Haunch.** The ***Haunches*** page is updated to show *No Haunch*.

Tip

If you want to delete several haunches, then you will find it quicker to delete a single one, and then copy it to all other locations which need haunches deleting.

12 Column Base Fixity in Portal Frame

You can define each base in your frame as being pinned, fixed or spring and give the associated details.

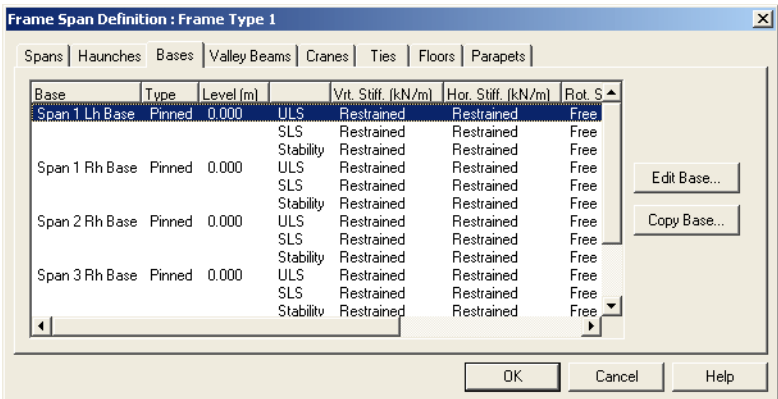
Help Valley beams are treated differently, *see* *"Valley Support Fixity in Portal Frame"* on page 218.

Defining base properties

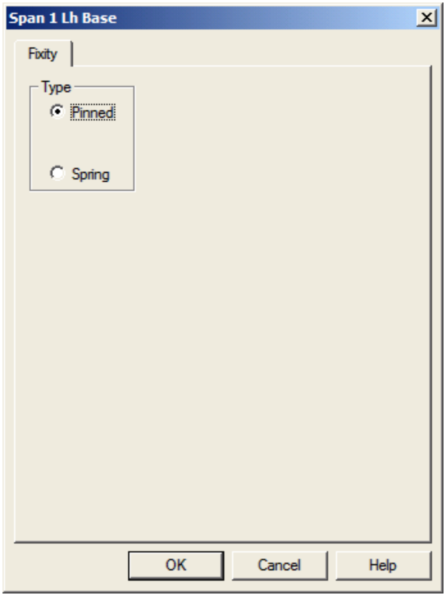
The ***Span Definition Bases*** page shows a summary for each column base and prop base in your frame. You define the details for each of these bases identically.

To define base properties

1. Select *Frame/Span Definition…* and pick ***Bases***.



2. Select a base and click **Edit Base…** to change its details. You will see the ***Base*** property sheet.



3. Pick the type of base that you want to use and give any necessary additional information (as covered in the following sections of the ***User's Guide***) then click **OK** to return to the ***Span Definition*** property sheet.

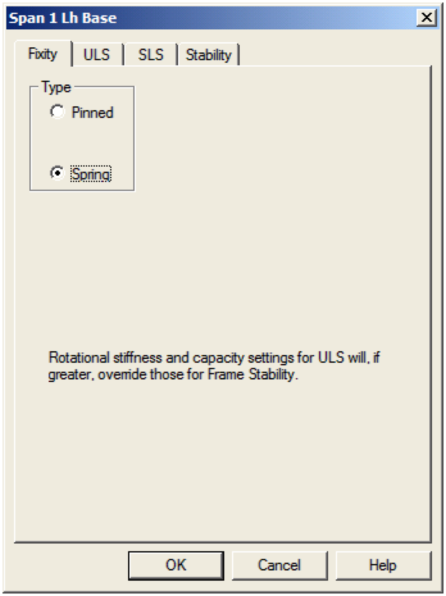
To define a pinned base

1. Pick **Pinned**.

2. No further information is required so click **OK** return to the ***Span Definition*** property sheet.

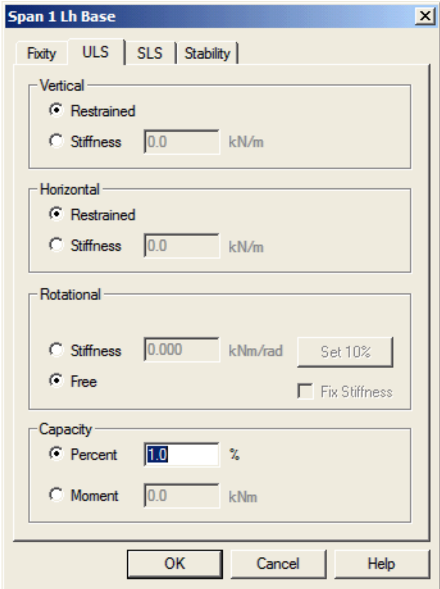
To define a spring base

1. Pick **Spring**. Three new pages are added to the property sheet so that you can define the details at the Ultimate and Serviceability Limit States and also the details that you want to use for the P292 frame stability check.



Ultimate Limit State

2. Pick the ***ULS*** tab to set the details for the ultimate limit state.



3. For the vertical and horizontal directions pick **Restrained** or pick **Stiffness** and enter a value which models the movement of the base in that direction.

4. For the rotational direction pick **Stiffness** and enter a value which models the rotational characteristics of the base or pick **Free** if the base provides no rotational stiffness.

Note *Once the column size is known you can use* ***Set 10%*** *to set the stiffness to 10% of the stiffness of the column section.*

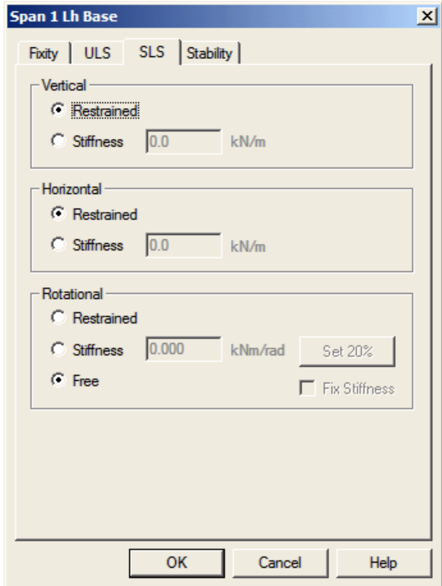
Note If you don't want *Portal Frame* to automatically change the stiffness of the base if you change the frame geometry or sections then tick **Fix Stiffness**.

5. If you want to limit the ULS moment capacity of the base to less than 100% of that of the column to which it is connected, or to a specific value if site conditions necessitate this, then pick the appropriate option and enter the requisite value.

Caution If you specify the Moment capacity your base must be able to provide this capacity at the Ultimate Limit State.

Serviceability Limit State

6. Pick e the ***SLS*** tab to set the stiffnesses of the base at the Serviceability Limit State.



7. For the vertical and horizontal directions pick **Restrained** or pick **Stiffness** and enter a value which models the movement of the base in that direction.

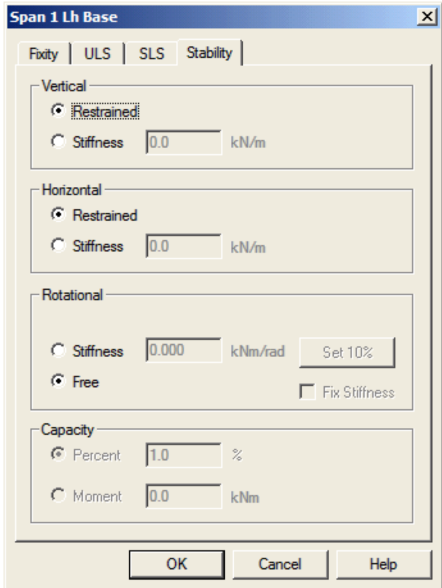
8. For the rotational direction pick **Restrained**, **Free**, or pick **Stiffness** and enter a value which models the rotational characteristics of the base.

Note *Once the column size is known you can use* **Set 20%** *to set the stiffness to 10% of the stiffness of the column section.*

Note If you don't want *Portal Frame* to automatically change the stiffness of the base if you change the frame geometry or sections then tick **Fix Stiffness**.

Frame Stability Condition

9. Pick the ***Stability*** tab to set the stiffnesses of the base at the P292 Frame Stability check.



10. The vertical and horizontal directions are ignored for this check. They are shown to keep the dialog layout similar between pages.

11. For the rotational direction pick **Stiffness** and enter a value which models the rotational characteristics of the base or pick **Free** if the base has no rotational stiffness.

Note Once the column size is known you can use **Set 10%** to set the stiffness to 10% of the stiffness of the column section.

Note If you don't want *Portal Frame* to automatically change the stiffness of the base if you change the frame geometry or sections then tick **Fix Stiffness**.

The rotational capacity of the base is set at the nominal value specified for this check. This is 1% and you can not change this. However the stiffnesses and rotational values are compared with those set on the ***ULS*** page, and where the value on that page is greater than the one set here, then the larger value will be used in the P292 check.

Example If you have *Restrained* set on the *ULS* page and you enter a value of 1000 kNm/rad here, then the larger value from the ULS page will be used.

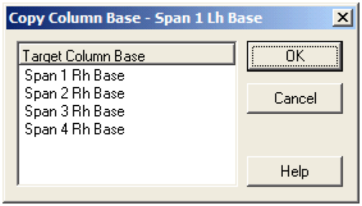
If you have a percentage capacity of 10% set on the *ULS* page, then this will be used rather than the default value of 1% set on this page.

Help For further information on the P292 check *"SCI publication P292"* on page 858.

12. Once your spring base details are complete click **OK** to return to the ***Span Definition*** property sheet which will summarise its details.

To copy the details for a base

1. Select the line for the base you want to copy and then click **Copy Base…** to see the ***Copy Column Base*** dialog.



2. Select the target bases and then click **OK** to perform the copy.

13 Valley Support Fixity in Portal Frame

You can set fixities for each valley beam in your frame.

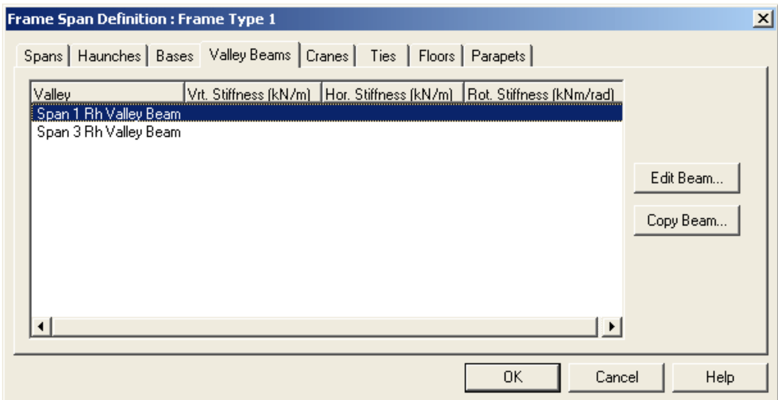
Help Column and prop bases are treated differently, *see* *"Column Base Fixity in Portal Frame"* on page 209.

Defining valley beam Properties

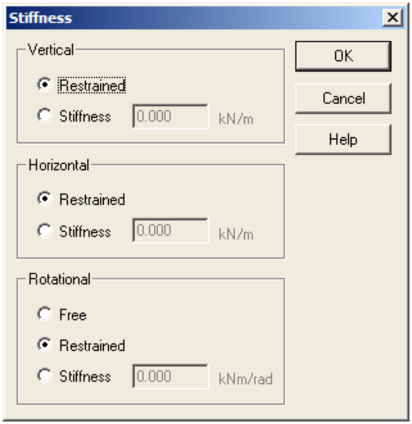
The *Span Definition* ***Valley Beams*** page shows a summary for each valley beam in your frame.

To define valley beam properties

1. Select *Frame/Span Definition…* and pick the ***Valley Beams*** tab.



2. Select a beam and click **Edit Beam…** to change its details. You will see the ***Stiffness*** property sheet.



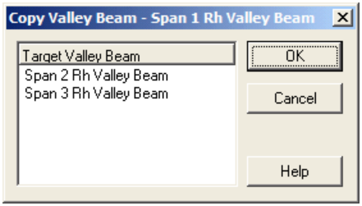
3. For the vertical and horizontal directions pick **Restrained** or pick **Stiffness** and enter a value which models the movement of the base in that direction.

4. For the rotational direction pick **Restrained**, **Free**, or pick **Stiffness** and enter a value which models the rotational characteristics of the base.

5. Once your valley beam details are complete click **OK** to return to the ***Span Definition*** property sheet which will summarise its details.

To copy the details for a valley beam

1. Select the line for the valley beam you want to copy and then click **Copy Beam…** to see the *Copy Valley Beam* dialog.



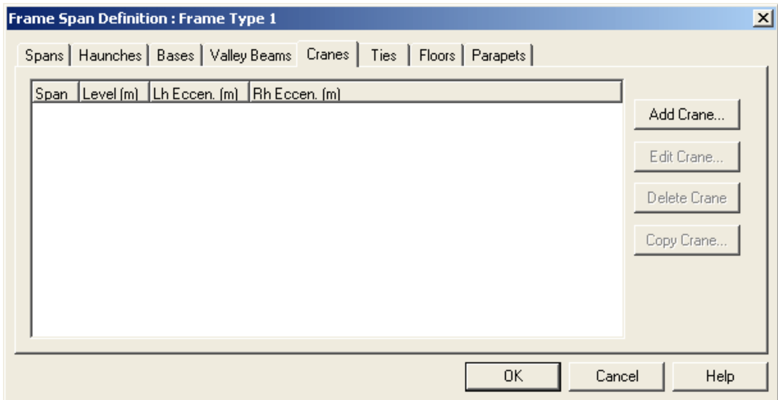
2. Select the target valley beams and then click **OK** to perform the copy.

14 Crane Geometry in Portal Frame

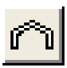
You can add a single crane to any span of your frame. When you have added cranes are defined you can then define the loads they apply and include these in your design.

Defining crane geometry

You add cranes to your frame from the ***Cranes*** page of the *Span Definition* property sheet.

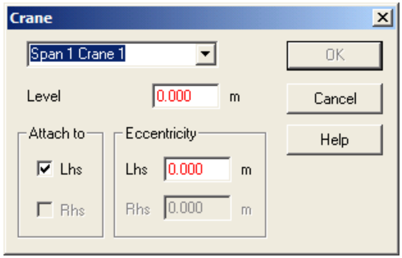


To add a crane



1. Select *Frame/Span Definition…* and pick the ***Cranes*** tab.

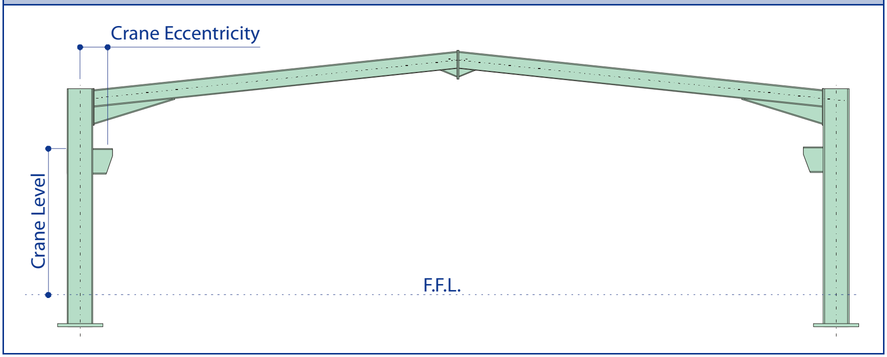
2. Click **Add Crane** to see the ***Crane*** dialog.



Note *If you have a crane in every span* ***Add Crane*** *will be dimmed.*

The dimensions that you need to give are clearly indicated on the diagram.

Crane dimensions



1. Pick the **Span and Crane Number**. (If a span already has a crane, then it will not appear in the list.)

2. Enter the **Level** of the crane and its left and right hand side **Eccentricities**.

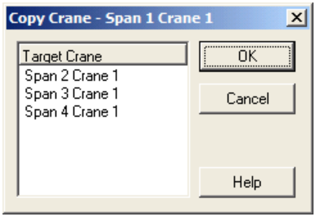
3. If there is no column for the crane to attach to at one side of the span (because you put a valley beam there), then the **Attach to** box for that side is dimmed.

Note *The crane is only checked in the context of the entire frame when you* **OK** *the Span Definition property sheet.*

4. Once the details are correct click **OK** to return to the ***Span Definition*** property sheet.

To copy cranes

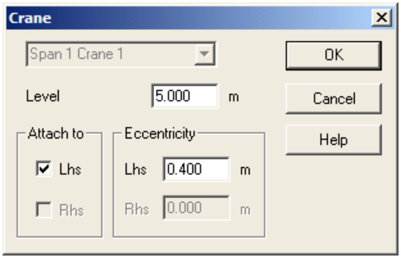
1. Select the line for the crane that you want to copy and then click **Copy Crane** to see the ***Copy Crane*** dialog.



2. Select the target crane(s) and click **OK** to return to the ***Span Definition*** property sheet which will show your changes.

To modify the details for a crane

1. Select the crane you want to modify on the ***Cranes*** page and then click **Edit Crane** to see the ***Crane*** dialog completed with the details for that crane.



2. You cannot change the span and crane number so this is dimmed.

3. Make the modifications you require and click **OK** to return to the ***Span Definition*** property sheet.

Tip *You can use* **Copy** *to update cranes as well as create them.*

To delete a crane

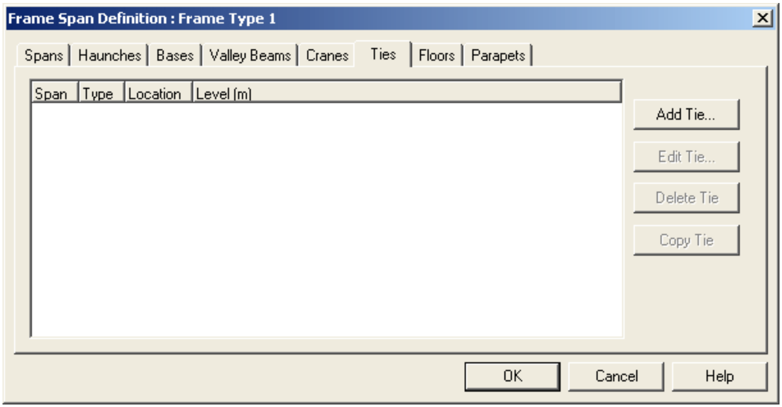
1. Select the crane that you want to delete on the ***Cranes*** page and then click **Delete Crane**.

15 Tie Geometry in Portal Frame

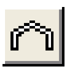
You can define ties in any span of the frame and include their effects in your design.

Defining tie geometry

You add ties to your frame from the ***Ties*** page of the ***Span Definition*** property sheet.

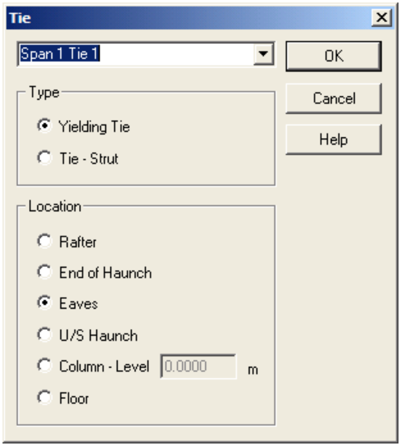


To add a tie



1. Select *Frame/Span Definition…* and pick the ***Ties*** tab.

2. Click **Add Tie** to see the *Tie* dialog.



Note *If you the maximum number of ties in every span* **Add Tie** *will be dimmed.*

3. Select the **Span and Tie Number**. If you have a tie in a location, then it will not be shown in the list.

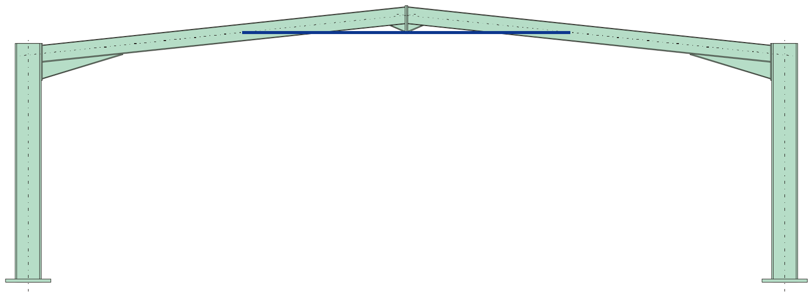
Note *The options are limited by the number of spans in your frame, the ties that you have already defined and the maximum number of ties that your edition allows.*

4. Pick the **Type** of tie that you require:
   - A yielding tie is one that can only take tension. If it would go into compression *Portal Frame* handles the frame as though the tie does not exist. Such ties would usually have yielded at ultimate limit state, although this depends on the properties that you give them.

- A tie-strut can take both tension and compression. If the tie goes into compression ***Portal Frame*** takes account of this in the design of the frame. If you use a tie-strut you must ensure that the section that you use can carry the applied compressive force without buckling or yielding.

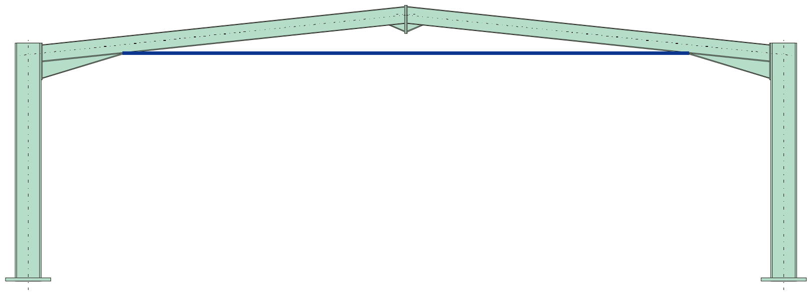
5. Enter the **Location** of the tie.

   The diagrams below show the various locations in a standard span.

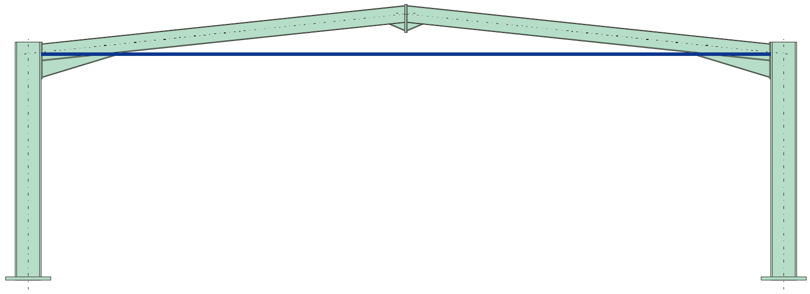
Standard Span Tie Locations
Rafter tie


Standard Span Tie Locations (Continued)

Haunch end tie

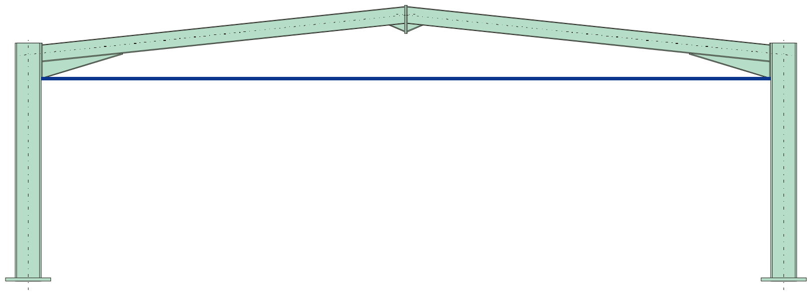


Eaves tie

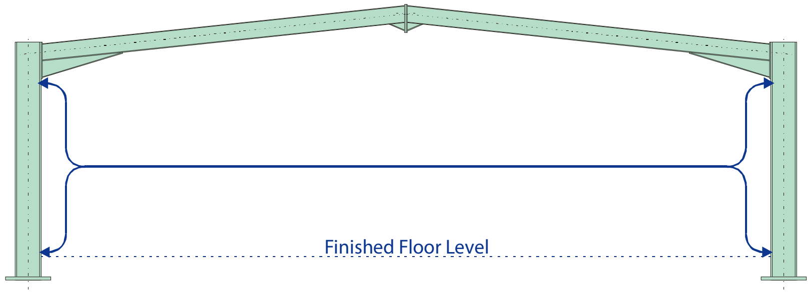


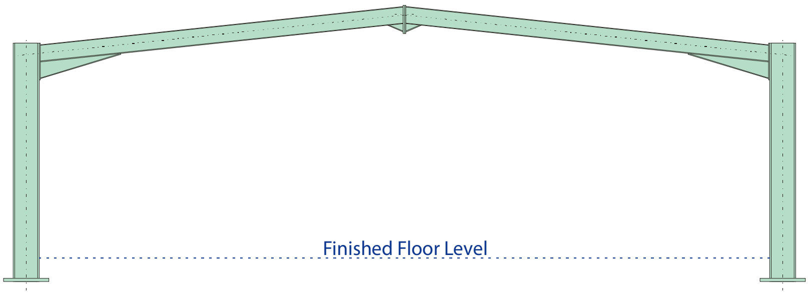
Standard Span Tie Locations (Continued)

Haunch underside tie



Column tie



Standard Span Tie Locations (Continued)
Floor Tie


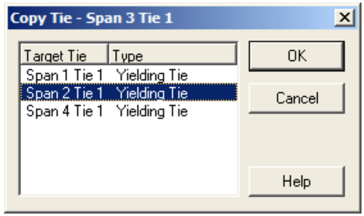
6. You need to enter the **Level** for a column tie. This must be at least 0.1 metres below the bottom of the lowest haunch and the same distance above the floor level (taken as 0.000 metres).

Note The tie is only checked in the context of the entire frame when you **OK** the *Span Definition* property sheet.

7. Once the details are correct click **OK** to return to the ***Span Definition*** property sheet.

To copy ties

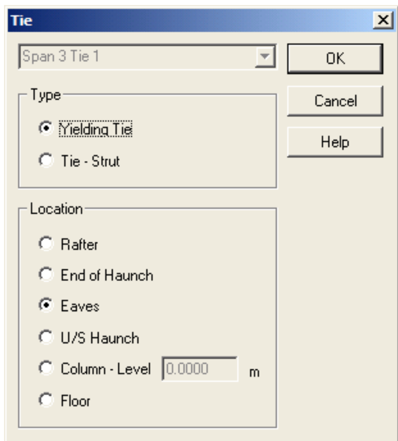
1. Select the line for the tie that you want to copy and then click **Copy Tie** to see the ***Copy Tie*** dialog.



2. Select the target ties and click **OK** to perform the copy and return to the ***Span Definition*** property sheet which will show your changes.

To modify the details for a tie

1. Select the line for the tie you want to amend and click **Edit Tie** to see the ***Tie*** dialog with the current tie details.



2. You cannot change the span and tie number so this is dimmed.

3. Make the modifications you require and click **OK** to return to the ***Span Definition*** property sheet.

Tip You can use **Copy** to update ties as well as create them.

To delete a tie

If you have generated a tie that is not required for any reason, then it is easy to delete it in ***Portal Frame***.

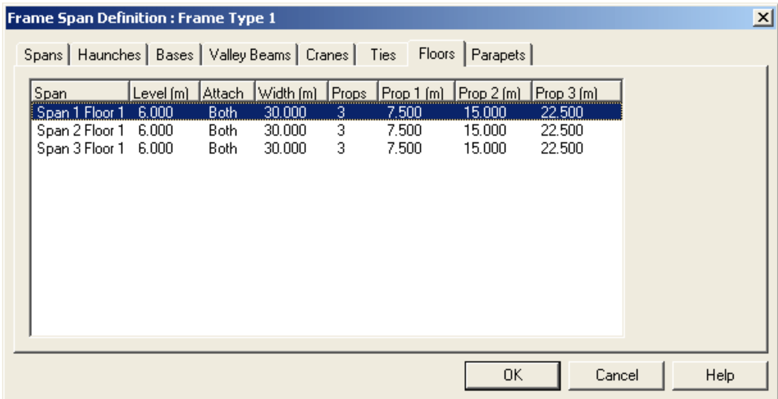
1. Select the tie that you want to delete on the ***Ties*** page and then click **Delete Tie**.

16 Floor Geometry in Portal Frame

You can define floors in any span of the frame and include their effects in your design.

Defining floor geometry

You view the floors in your frame through the ***Floors*** page of the ***Span Definition*** property sheet. You add floors into your frame using the ***Floor Wizard***[1], or by adding the appropriate beams and joists using the ***Portal Modeller***[1] toolbar.



Caution **In this release of *Portal Frame* the floors are not involved in the design of the portal frames.** Floors are merely a way of adding ancillary steelwork into your model.

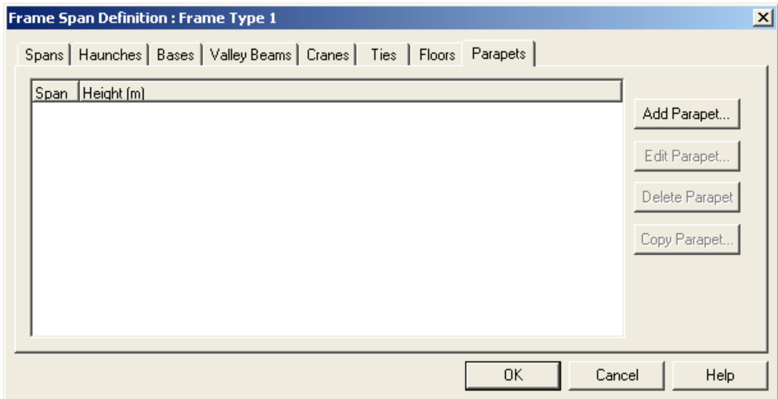
1. *This is only available if you have installed and licensed the* ***Portal Modeller*** *add-in.*

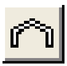
17 Parapet Geometry in Portal Frame

You can add a single parapet to every eaves point in a frame.

Defining parapet geometry

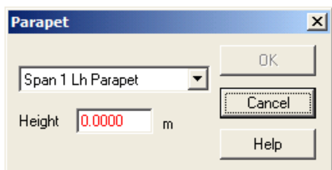
You add parapets to your frame from the ***Parapets*** page of the ***Span Definition*** property sheet.





To add a parapet

1. Select *Frame/Span Definition…* and the ***Parapets*** tab.
2. Click **Add Parapet** to see the ***Parapet*** dialog.



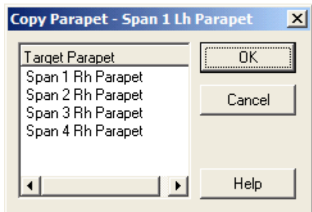
Note *If you have added a parapet to every eaves point of your frame* **Add** *will be dimmed.*

3. Select **Position of the Parapet** and enter its **Height** (which must be at least 0.1 metres).

4. Once your parapet details are correct click **OK** to return to the ***Span Definition*** property sheet.

To copy parapets

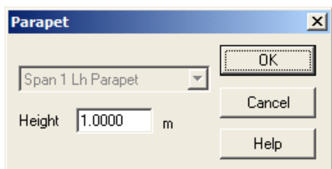
1. Select the parapet which you want to copy and then click **Copy Parapet** to see the ***Copy Parapet*** dialog.



2. Select the target parapets and click **OK** to return to the ***Span Definition*** property sheet which will show your changes.

To modify the details for a parapet

1. Select the parapet which you want to modify and click **Edit Parapet** to see the ***Parapet*** dialog with the current details.



You cannot change the parapet location and so the **Span** is dimmed.

2. Change the **Height** of the parapet and then click **OK** to return to the ***Span Definition*** property sheet.

Tip You can use **Copy** to update parapets as well as create them.

To delete a parapet

1. Select the parapet that you want to delete and click **Delete Parapet**.

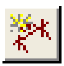
18 Dimensions in Portal Frame

While you are defining portal frames, you may want to review particular dimensions which are of interest. *Portal Frame* allows you to add dimensions between many points around each span of each frame. The allowable points include:

- Base,
- Crane (if present),
- Underside of haunch,
- intersection of rafter/column, rafter/rafter (depending on span type),
- end of haunch,
- apex,
- intersection of tie, and so on.

Using dimensions

Before you can add dimensions to your frame layout drawing, you must have defined the frame in sufficient detail for the intersection points you require to be included on the display.



To define a dimension parallel to two points

1. Click the **Create Aligned Dimension** icon.

2. Move the pointer over the part of the display where you want the dimension to start. When the pointer is over a point you can pick you will see that the icon changes. Left click to select the point you want.

3. Move the pointer over the part of the display where you want the dimension to end. Left click to select the point you want. *Portal Frame* will add a dimension parallel to the two points. This will initially lie on the line connecting the two points.

4. Move the pointer to move the dimension line and text to the location you require, and *Portal Frame* will move the dimension line to that location.

To define a vertical projected dimension between two points

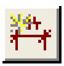
1. Click the **Create Vertical Dimension** icon.

2. Move the pointer over the part of the display where you want the dimension to start. When the pointer is over a point you can pick you will see that the icon changes. Left click to select the point you want.

3. Move the pointer over the part of the display where you want the dimension to end. Left click to select the point you want. *Portal Frame* will add a vertical dimension at the second of the two points.

4. Move the pointer to move the dimension line and text to the location you require, left click and *Portal Frame* will move the dimension line to that location.

5. By default *Portal Frame* draws the end lines for your dimension right back to the points which you clicked to define the start and end of the dimension. If you want to reduce the clutter which these lines can cause, right click the dimension and then click *Offset Dim* on the context menu which appears. This is a simple toggle which switches the option on or off.



To define a horizontal projected dimension between two points

1. Click the **Create Horizontal Dimension** icon.

2. Move the pointer over the part of the display where you want the dimension to start. When the pointer is over a point you can pick you will see that the icon changes. Left click to select the point you want.

3. Move the pointer over the part of the display where you want the dimension to end. Left click to select the point you want. *Portal Frame* will add a horizontal dimension at the second of the two points.

4. Move the pointer to move the dimension line and text to the location you require, left click and *Portal Frame* will move the dimension line to that location.

5. By default *Portal Frame* draws the end lines for your dimension right back to the points which you clicked to define the start and end of the dimension. If you want to reduce the clutter which these lines can cause, right click the dimension and then click *Offset Dim* on the context menu which appears. This is a simple toggle which switches the option on or off.

To delete a dimension

1. Click the **Delete Dimension** icon.

2. Move the pointer over the dimension line you want to delete, then left click and *Portal Frame* will delete the dimension.

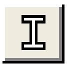
3. By default *Portal Frame* draws the end lines for your dimension right back to the points which you clicked to define the start and end of the dimension. If you want to reduce the clutter which these lines can cause, right click the dimension and then click *Offset Dim* on the context menu which appears. This is a simple toggle which switches the option on or off.

19 Member Properties in Portal Frame

When you have defined the geometry of your frame you can assign properties to the parts that need them. These depend on whether you are designing a frame or checking it. In either case after the design/check the *Member Properties* sheet gives you basic information about the frame's members.

Defining member properties

In all cases you use the *Member Properties* sheet to define the details that are required.

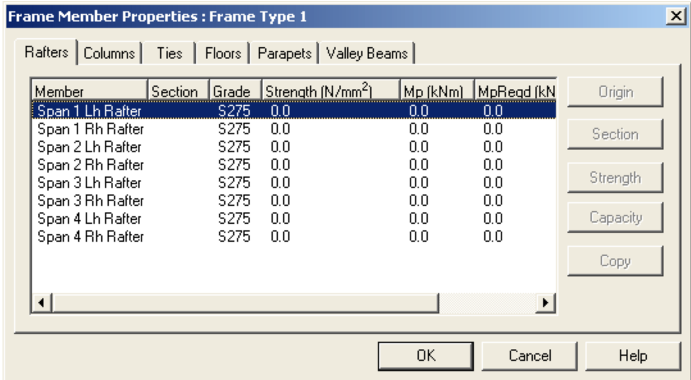


To define rafter properties

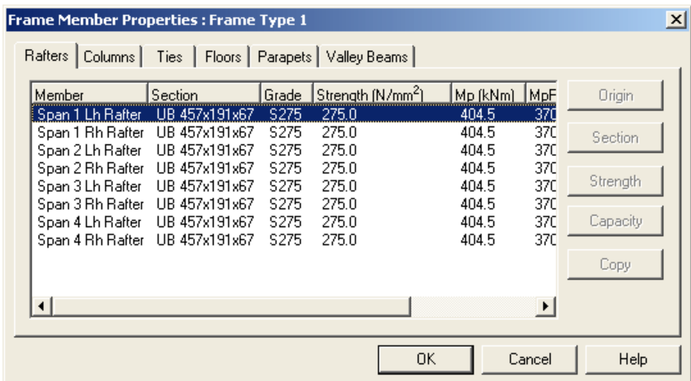
The properties you need to define depend on whether you are designing a frame (in which case *Portal Frame* will choose appropriate section sizes for you) or checking a frame (in which case you will need to give the details of the sizes which *Portal Frame* is to check). Both options are covered below.

When designing a frame

1. Select *Frame/Member Properties…* Until you have performed a design there will be no section details, and so you will see the page as shown below.

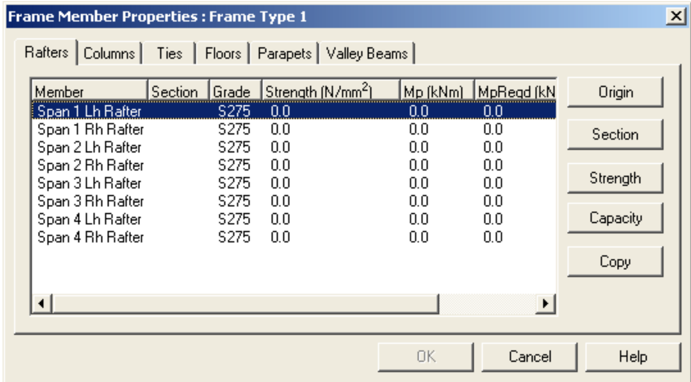


When you have performed the design, then the ***Rafters*** page allows you to see the sections resulting from the design and is provided for reference only. A typical post-design display is shown below.



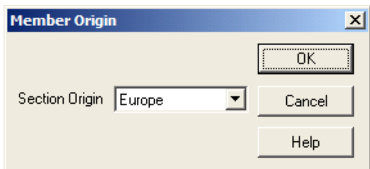
When checking a frame

1. Select *Frame/Member Properties…*



For any rafter you can choose the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one rafter to any other rafters in the frame. To do any of these you will first need to select the rafter to work on.

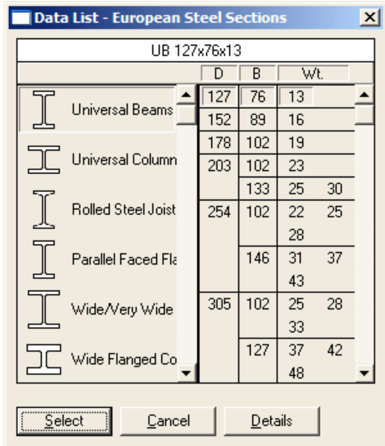
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then click **OK** to return to the ***Rafters*** page.

Help You can set the country of origin in preferences ***see*** *"To set design preferences"* on page 80.

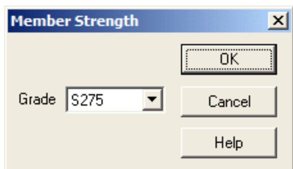
3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



Note *For rafters only I and H sections are allowed.*

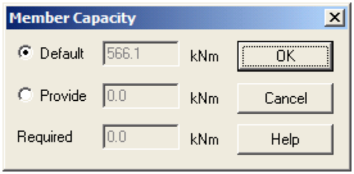
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Rafters*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use and then click **OK** to return to the ***Rafters*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

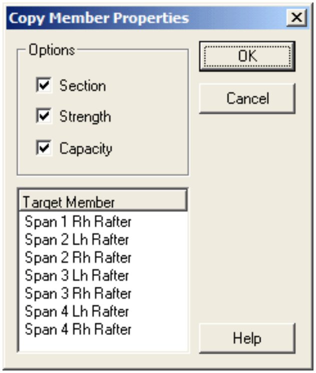


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the rafter can carry the this moment in the presence of the axial loads that it carries. Click **OK** to the ***Rafters*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

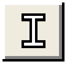
To copy the properties for a rafter

1. Select *Frame/Member Properties…* and pick the rafter you want to copy from the ***Rafters*** page. Now click **Copy** to see the *Copy Member Properties* dialog.



2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target rafters for the copied information and then click **OK** to perform the copy.

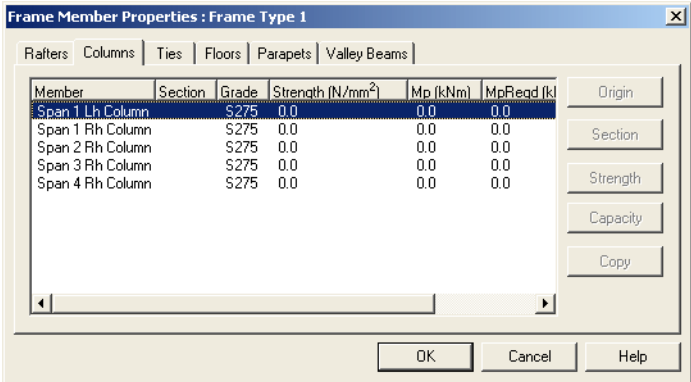


To define column properties

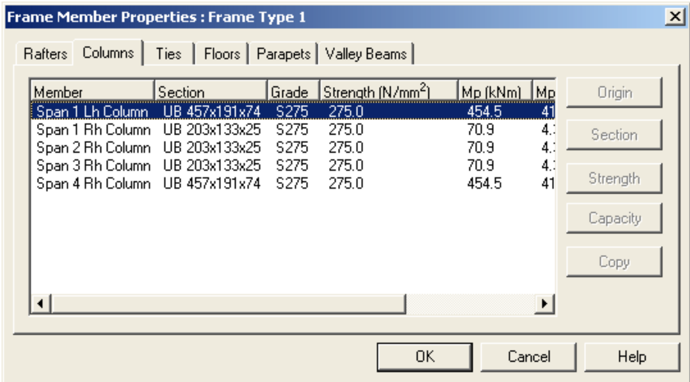
The properties you need to define depend on whether you are designing a frame (in which case *Portal Frame* will choose appropriate section sizes for you) or checking a frame (in which case you will need to give the details of the sizes which *Portal Frame* is to check). Both options are covered below.

When designing a frame

1. Select *Frame/Member Properties…* and the ***Columns*** tab.

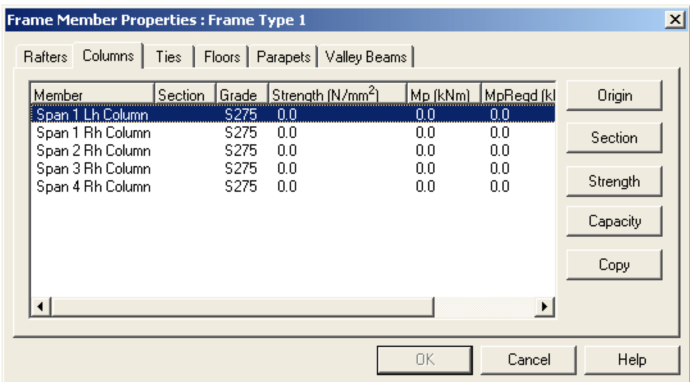


The ***Columns*** page shows all the columns in your frame. For the main columns it allows you to see the sections resulting from the design and is provided for reference only. For valley columns the ***Columns*** page allows you to select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for any valley column to any other valley columns in the frame. This is identical to the functionality for all columns when you are defining a frame to check. A typical post-design display is shown below.



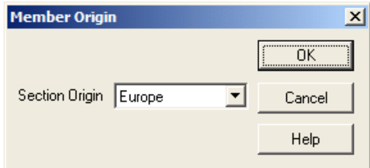
When checking a frame

1. Select *Frame/Member Properties…* and the ***Columns*** tab.



For any column you can select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one column to any other columns in the frame. To do any of these you will first need to select the column to work on.

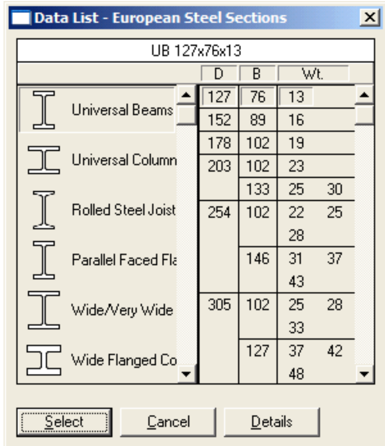
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then **OK** this to return to the ***Columns*** page.

Help You can set the country of origin in preferences ***see*** *"To set design preferences"* on page 80.

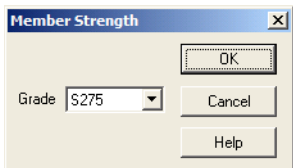
3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



Note *For columns only I and H sections are allowed.*

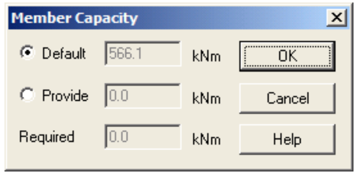
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Columns*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use followed by **OK** to return to the ***Columns*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

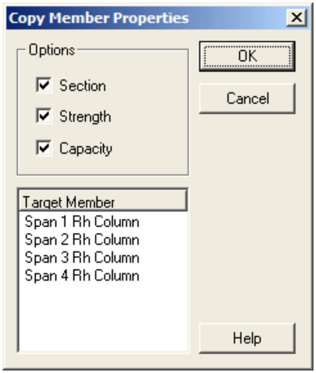


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the column can carry the this moment in the presence of the axial loads that it carries. Click **OK** to return to the ***Columns*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

To copy the details for a column

1. Select *Frame/Member Properties…* followed by the ***Columns*** tab. Now pick the column whose details you want to copy and click **Copy** to see the *Copy Member Properties* dialog.

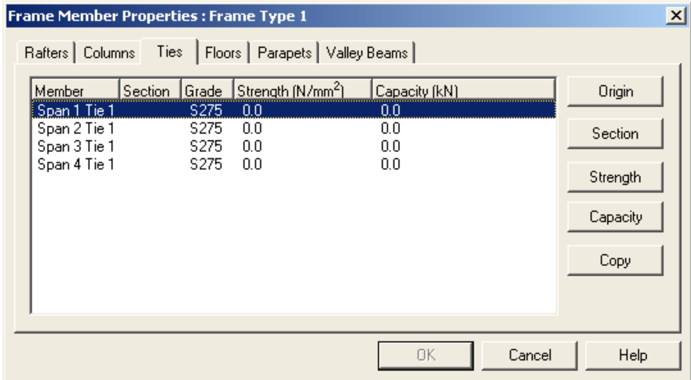


2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target columns for the copied information and then click **OK** to perform the copy.

To define tie properties

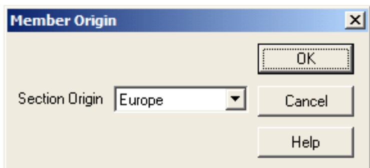
1. Select *Frame/Member Properties…* and the ***Ties*** tab.



Note *If your frame has no ties the Ties page will be empty.*

For any tie you can select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one tie to any other ties in the frame. To do any of these you will first need to select the tie to work on.

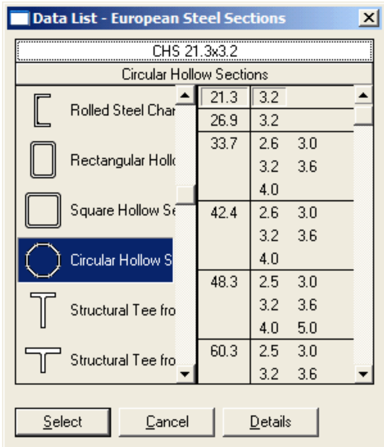
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then **OK** this to return to the ***Ties*** page.

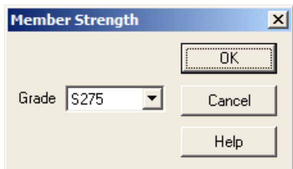
Help You can set the country of origin in preferences ***see*** *"To set design preferences"* on page 80.

3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



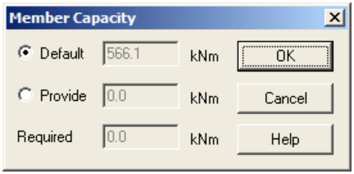
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Ties*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use followed by **OK** to return to the ***Ties*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

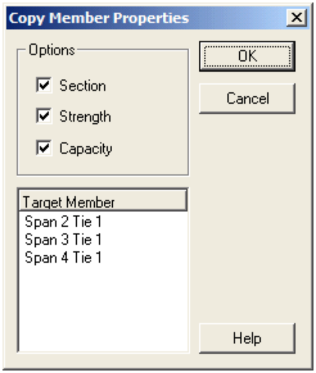


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the tie can carry the this moment in the presence of the axial loads that it carries. Click **OK** to return to the ***Ties*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

To copy the properties for a tie

1. Select *Frame/Member Properties…* followed by the ***Ties*** tab. Now pick the tie whose details you want to copy and click **Copy** to see the ***Copy Member Properties*** dialog.

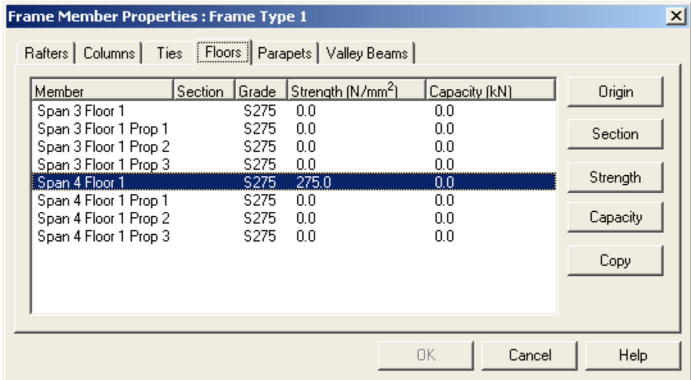


2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target ties for the copied information and then click **OK** to perform the copy.

To define floor properties

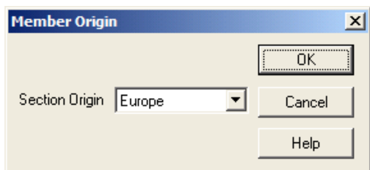
1. Select *Frame/Member Properties…* and the ***Floors*** tab.



Note If your frame has no floors the *Floors* page will be empty.

For any floor you can select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one floor to any other floors in the frame. To do any of these you will first need to select the floor to work on.

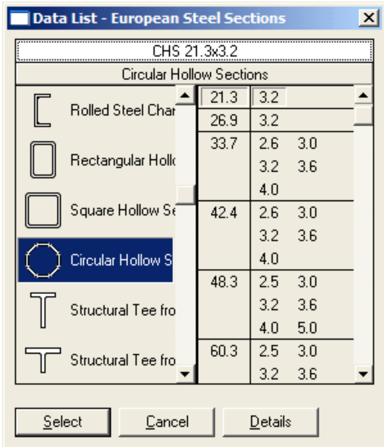
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then **OK** this to return to the ***Floors*** page.

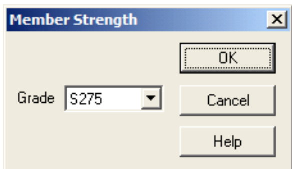
Help You can set the country of origin in preferences *see* *"To set design preferences"* on page 80.

3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



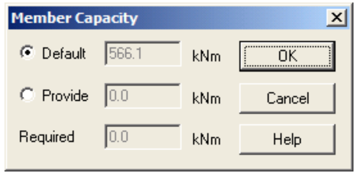
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floors*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use followed by **OK** to return to the ***Floors*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

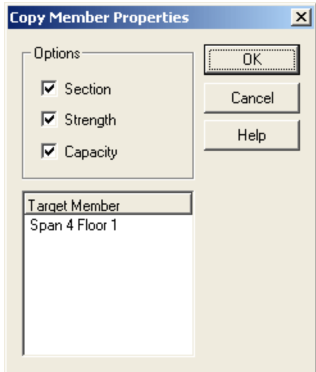


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the floor can carry the this moment in the presence of the axial loads that it carries. Click **OK** to return to the ***Floors*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

To copy the properties for a floor

1. Select *Frame/Member Properties…* followed by the ***Floors*** tab. Now pick the floor member whose details you want to copy and click **Copy** to see the ***Copy Member Properties*** dialog.



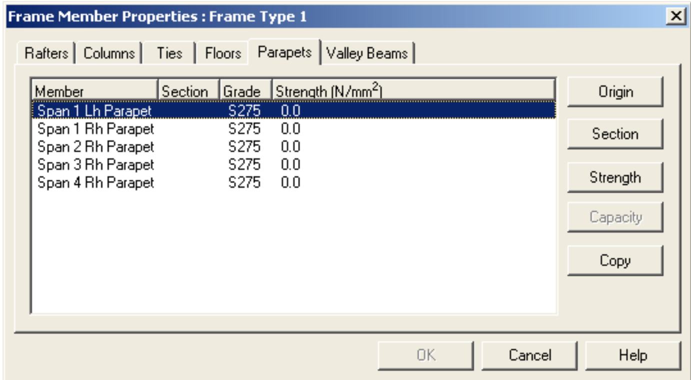
Note *If you pick the main floor beams from the list of Members, then you will only see floor beams in the list. Similarly if you pick a floor prop from the list of members, then you will only see floor props in the list.*

2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target floors for the copied information and then click **OK** to perform the copy.

To define parapet properties

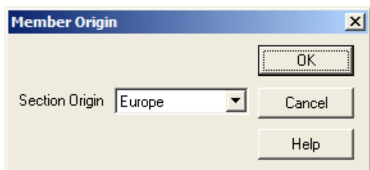
1. Select *Frame/Member Properties…* and the ***Parapets*** tab.



Note If your frame has no parapets the *Parapets* page will be empty.

For any parapet you can select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one parapet to any other parapets in the frame. To do any of these you will first need to select the parapet to work on.

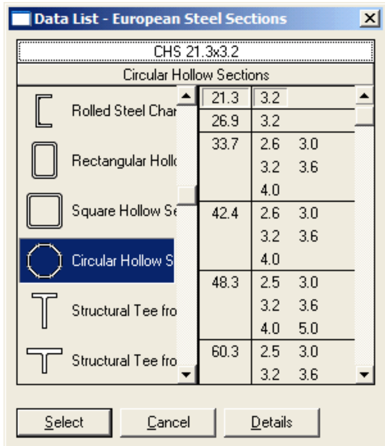
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then **OK** this to return to the ***Parapets*** page.

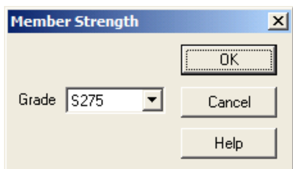
Help You can set the country of origin in preferences *see "To set design preferences" on page 80.*

3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



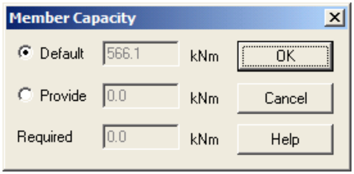
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Parapets*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use followed by **OK** to return to the ***Parapets*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

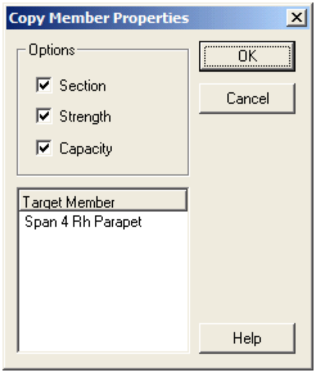


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the parapet can carry the this moment in the presence of the axial loads that it carries. Click **OK** to return to the ***Parapets*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

To copy the details for a parapet

1. Select *Frame/Member Properties…* followed by the ***Parapets*** tab. Now pick the parapet whose details you want to copy and click **Copy** to see the ***Copy Member Properties*** dialog.

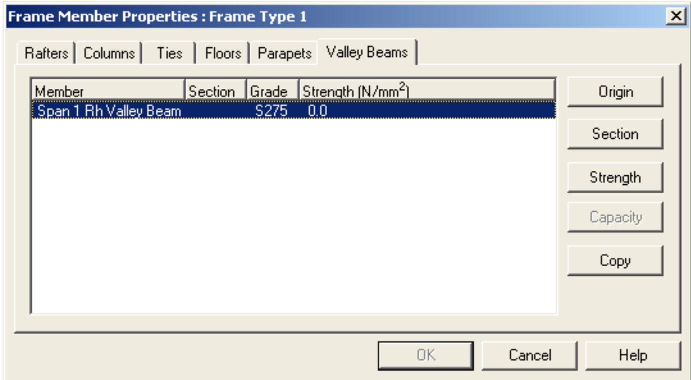


2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target parapets for the copied information and then click **OK** to perform the copy.

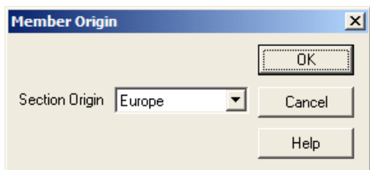
To define valley beam properties

1. Select *Frame/Member Properties…* and the ***Valley Beams*** tab.



For any valley beam you can select the region of **Origin** for its section, select its **Section** type and size and set its **Strength** and **Capacity**. You can also copy the details for one valley beam to any other valley beams in the frame. To do any of these you will first need to select the valley beam to work on.

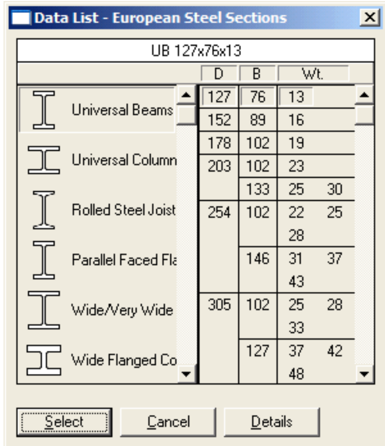
2. Click **Origin** to see the ***Member Origin*** dialog.



Select the **Origin** and then **OK** this to return to the ***Valley Beams*** page.

Help You can set the country of origin in preferences ***see*** *"To set design preferences"* on page 80.

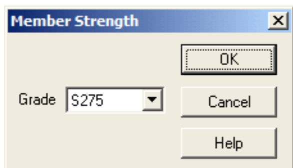
3. Click **Section…** to see the ***Steel Section Data List*** for the region you selected above.



Note *For valley beams only I and H sections are allowed.*

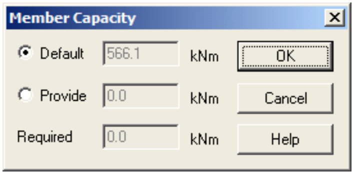
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection pick **Select** to return to the ***Valley Beams*** page which will show your selected size.

4. Click **Strength…** to see the ***Member Strength*** dialog.



Select the **Grade** of steel that you want to use followed by **OK** to return to the ***Valley Beams*** page which shows the grade and strength.

5. Click **Capacity…** to see the ***Member Capacity*** dialog which allows you to set the capacity directly, rather than allowing ***Portal Frame*** to calculate it for you based on the axial load in the member.

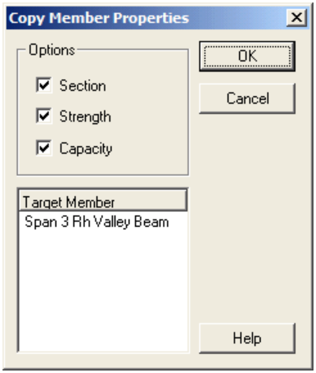


If you pick **Default** then ***Portal Frame*** will calculate the capacity for you based on the axial load in the member. If you pick **Provide** you can then enter the capacity yourself in which case you must ensure that the valley beam can carry the this moment in the presence of the axial loads that it carries. Click **OK** to return to the ***Valley Beams*** page which shows the capacity.

6. Either pick another tab of the property sheet or click **OK** to return to the graphical display of the frame.

To copy the details for a valley beam

1. Select *Frame/Member Properties…* followed by the ***Valley Beams*** tab. Now pick the valley beam whose details you want to copy and click **Copy** to see the ***Copy Member Properties*** dialog.



2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target valley beams for the copied information and then click **OK** to perform the copy.

Modifying member properties

You will usually use the previous methods to define member properties, however if you want to make changes to a single member you might find the following graphical approach easier.

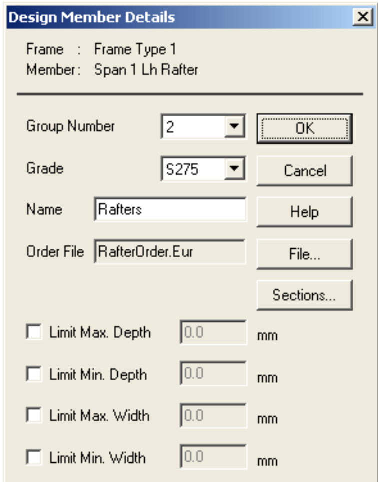
In all cases you must make your selections from the ***Geometry*** window for the frame whose details you want to modify.

To modify rafter properties using graphics

1. Choose the rafter from the graphical display. The subsequent dialog depends on whether you are designing or checking a frame. The two options are covered separately below.

When designing a frame

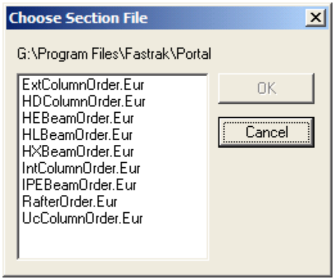
In this case you need to specify to which group the member belongs. All members in the same group take the same section size which is that for the most onerous member in the group.



Help For further details on grouping ***see***:

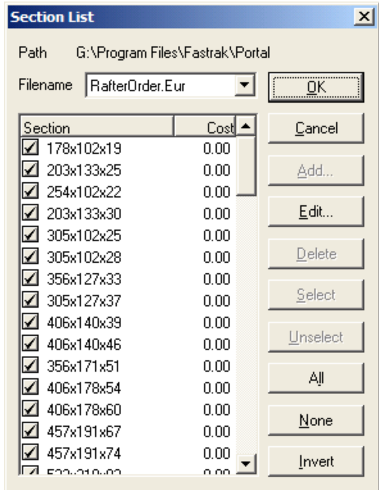
- *“To set design groups”* on page 418,
- *“To set grouping”* on page 422.

If you want to change the order file which lists the sections you want to consider during the design click **File…** to see the ***Choose Section File*** dialog.

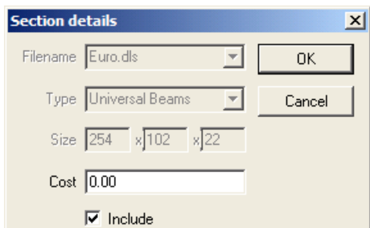


Pick the file that you want to use from the list of available files and then click **OK** to return to the ***Design Member Details*** dialog.

If you want to limit the sections within the file that you want to use, or to define a cost for a particular section, then you should first select the file and then click **Sections…** You will see the ***Section List*** dialog.



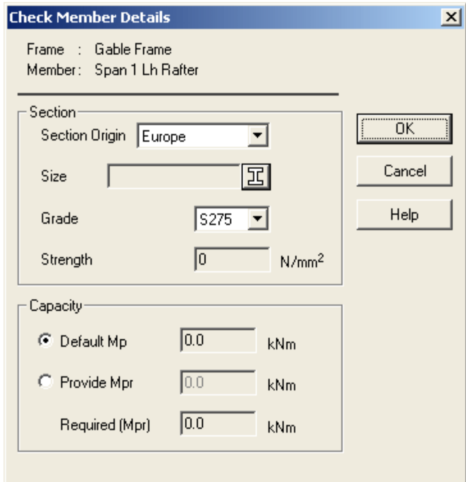
To include a section in the design process, make sure that its entry is ticked in this list. If you want to enter a cost for the section, then pick the line for that section and click **Edit…** to see the ***Section details*** dialog.



Enter the cost for the section. and then click **OK**.

When checking a frame

In this case you need to define the details of the member, namely its section size and grade.

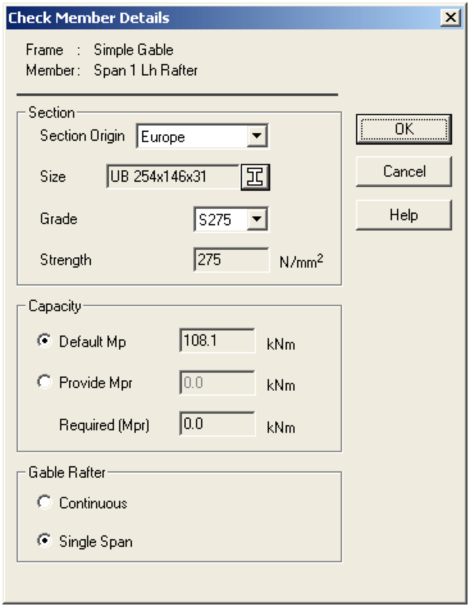


Help For further details **see** *"To define rafter properties"* on page 240.

1. When your changes are complete click **OK** to return to the graphical display which will show your changes.

When using a simple gable frame

In this case you need to define the details of the member, namely its section size and grade[1].



Help For further details *see* "To define rafter properties" on page 240.

You can also specify whether the gable rafter is **Continuous** or is made up of several **Single Span** members. In the latter case when you define any gable posts for the frame the rafter will be split to span between these.

1. *Since the members of a simple gable frame are not designed the grade is only used when you create a DXF file for the frame or the structure or when you transfer the model to* ***3D+****.*

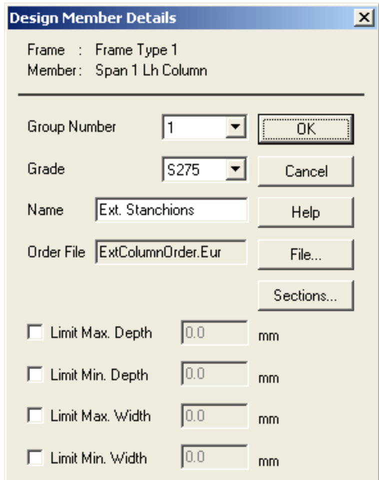
1. When your changes are complete click **OK** to return to the graphical display which will show your changes.

To modify column properties using graphics

1. Choose the column from the graphical display. The subsequent dialog depends on whether you are designing or checking a frame. The two options are covered separately below.

When designing a frame

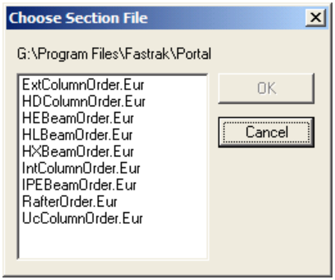
In this case you need to specify to which group the member belongs. All members in the same group take the same section size which is that for the most onerous member in the group.



Help For further details on grouping ***see***:

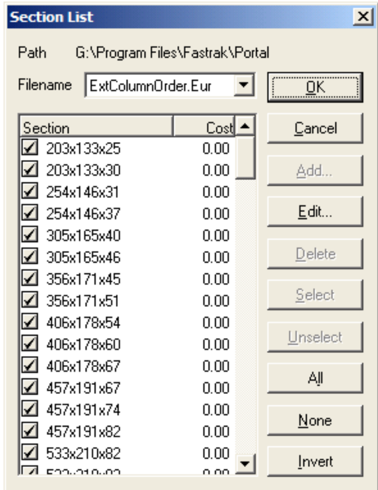
- *"To set design groups"* on page 418,
- *"To set grouping"* on page 422.

If you want to change the order file which lists the sections you want to consider during the design click **File...** to see the ***Choose Section File*** dialog.

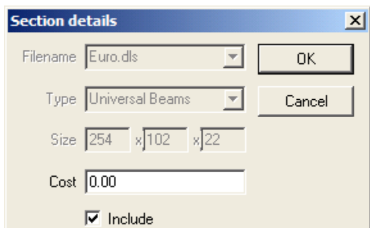


Pick the file that you want to use from the list of available files and then click **OK** to return to the ***Design Member Details*** dialog.

If you want to limit the sections within the file that you want to use, or to define a cost for a particular section, then you should first select the file and then click **Sections…** You will see the ***Section List*** dialog.



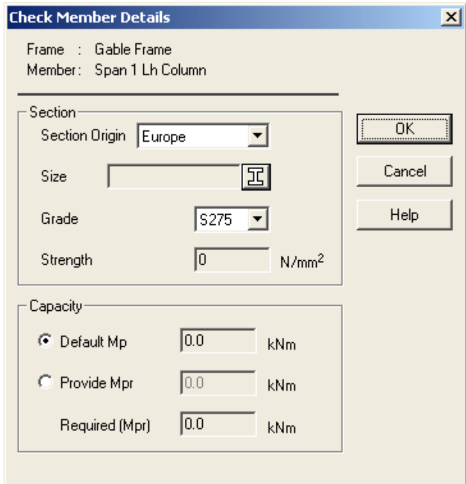
To include a section in the design process, make sure that its entry is ticked in this list. If you want to enter a cost for the section, then pick the line for that section and click **Edit…** to see the ***Section details*** dialog.



Enter the cost for the section. and then click **OK**.

When checking a frame

In this case you need to define the details of the member, namely its section size and grade.

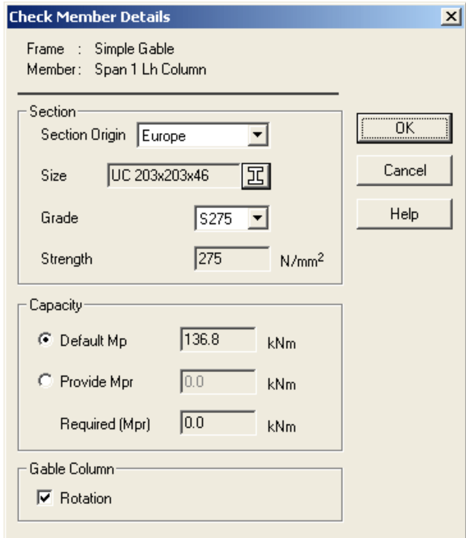


Help For further details ***see*** *"To define column properties"* on page 245.

1. When your changes are complete click **OK** to return to the graphical display which will show your changes.

When using a simple gable frame

In this case you need to define the details of the member, namely its section size and grade[1].



Help For further details **see** *“To define rafter properties”* on page 240.

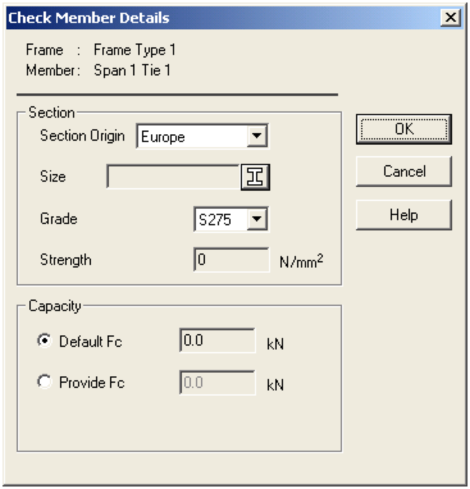
You can also specify whether the gable column is oriented with its web or flanges are oriented parallel to the span direction. If you click **Rotation**, then the column will be oriented so that its flanges are parallel to the span direction, otherwise the web will be parallel to the span direction.

1. When your changes are complete click **OK** to return to the graphical display which will show your changes.

1. *Since the members of a simple gable frame are not designed the grade is only used when you create a DXF file for the frame or the structure or when you transfer the model to* ***3D+****.*

To modify tie properties using graphics

1. Choose the tie from the graphical display.

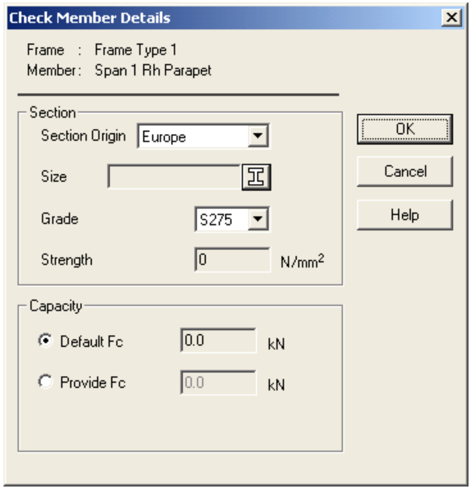


Help For further details *see* *"To define tie properties"* on page 252.

2. When your changes are complete click **OK** to return to the graphical display which will show your changes.

To modify parapet properties using graphics

1. Choose the parapet from the graphical display.



Help For further details *see* *"To define parapet properties"* on page 260.

2. When your changes are complete click **OK** to return to the graphical display which will show your changes.

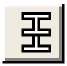
20 Haunch Properties in Portal Frame

You can assign properties to your frame's haunches. These properties depend on the fabrication method for your haunches – cutting them from sections, or building them up from plates.

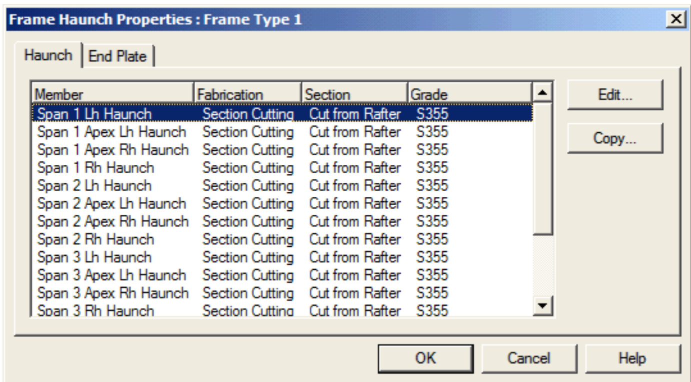
Defining haunch properties

In all cases you use the ***Haunch Properties*** sheet to define the details that are required.

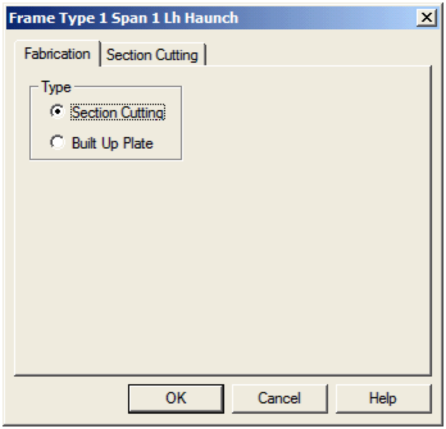
To define haunch properties



1. Select *Frame/Haunch Properties…*



2. Select the haunch you want to define or modify and then click **Edit...** to see the ***Haunch*** property sheet.

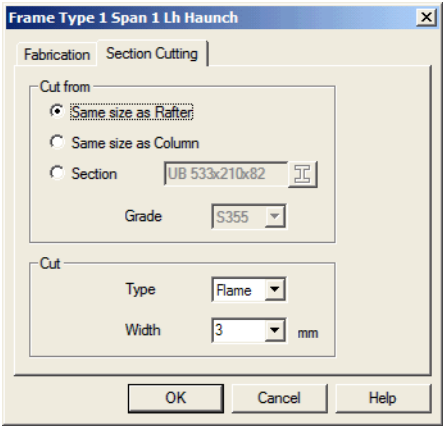


3. Pick the **Type** of fabrication for this haunch. You will see that the other tab changes to match your selection.

4. Use the other tab to define the details for your haunch and then click **OK** to return to the ***Haunch Properties*** sheet.

To define a section cutting haunch

1. Set the details for your section cutting haunch.



2. Use the options to specify how you want to set the section size of the haunch.

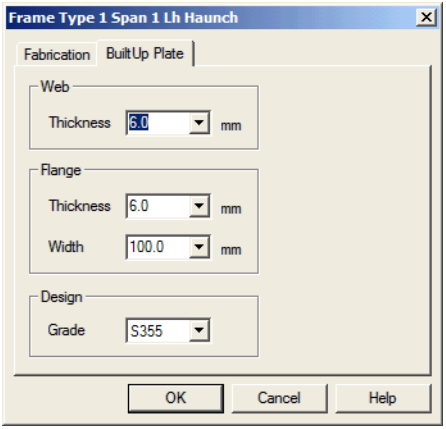
Note *The* **Same size as Column** *option is not available for an apex haunch.*

3. Specify the way that the section is to be cut to form the haunch, and detail the amount of material that is lost during the cutting process.

4. Once these details are correct click **OK** to return to the ***Haunches*** page.

To define a built up from plates haunch

1. Enter the details for your built up haunch.



2. Select the **Web plate thickness**, the **Flange plate thickness** and the **Flange plate width** either by selecting a value from the lists, or by entering a value directly.

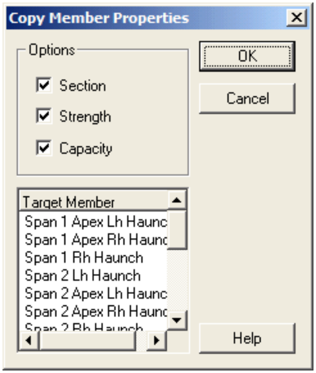
3. Select the **Design grade** for the steel that you want to use.

Note *Portal Frame forces the haunch grade to be the same as that of the rafter to which it is attached in order to avoid strain incompatibility problems.*

4. Once these details are correct click **OK** to return to the ***Haunches*** page.

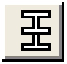
To copy the details for a haunch

1. Pick the haunch you want to copy and then click **Copy…** to see the ***Haunch Copy*** dialog.



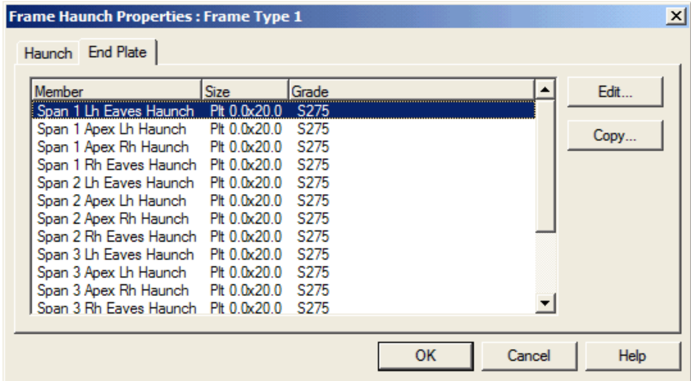
2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

3. Pick the target haunches for the copied information and then click **OK** to perform the copy.

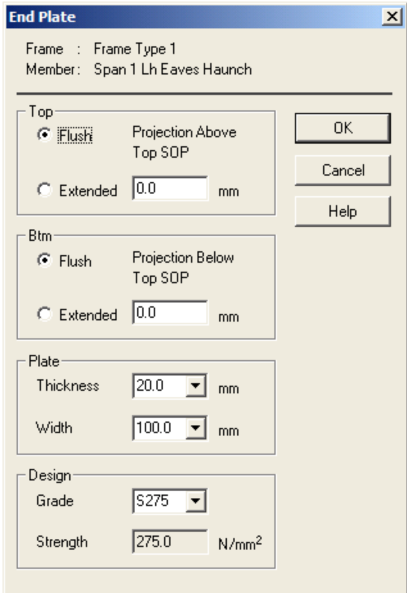


To define the end plate details

1. Select *Frame/Haunch Properties…* followed by the ***End Plate*** tab.



2. Select the end plate you want to define or modify and click **Edit** to see the ***End Plate Details*** dialog.



3. Pick whether you want the end plate to be **Flush** or **Extended** at the top and bottom. If you want an extended end plate, then enter the projection from the appropriate set out point.

4. Select the end plate **Thickness** and **Width**, or enter a value directly.

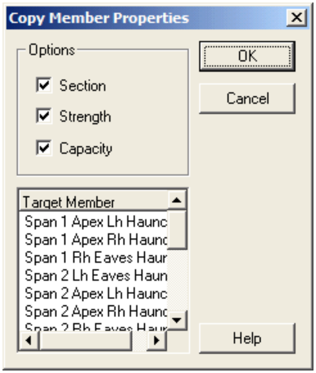
5. Select the **Grade** of your end plate.

Note *Portal Frame* forces the haunch grade to be the same as that of the rafter to which it is attached in order to avoid strain incompatibility problems.

6. Once these details are correct click **OK** to return to the ***End Plates*** page.

To copy the details for an end plate

1. Pick the end plate to copy and click **Copy…** to see the ***End Plate Copy*** dialog.



2. Check the options that you require (**Section**, **Strength** and the **Capacity**).

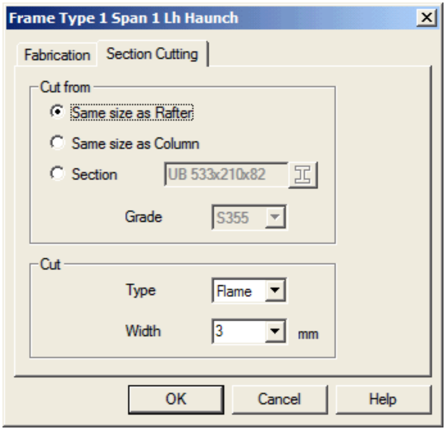
3. Pick the target end plates for the copied information and then click **OK** to perform the copy.

Modifying haunch and end plate properties

You will usually use the previous methods to define haunch properties, however if you want to make changes to a single haunch you might find the following graphical approach easier.

To modify haunch properties using graphics

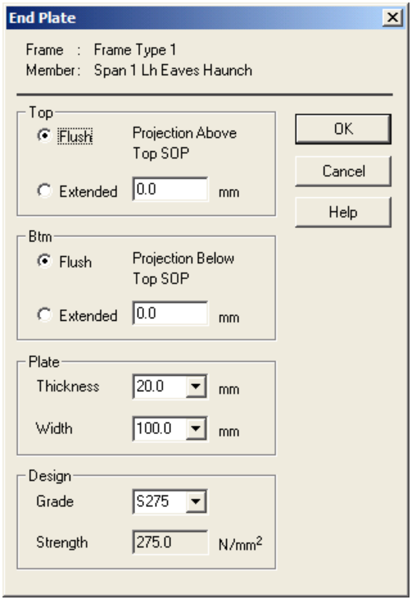
1. Choose the haunch from the graphical display to see its ***Haunch*** property sheet which will be configured for that haunch.



2. Make the changes you need and then click **OK** to return to the graphical display.

To modify end plate properties using graphics

1. Choose the end plate from the graphical display to see the ***End Plate*** dialog.



2. Make your changes and click **OK** to return to the graphical display.

21 Frame Loading in Portal Frame

You define the loading that is applied to your frame as a series of individual loadcases. You then assemble these loadcases into the combinations that you want to investigate during the design, applying the appropriate factors, depending on the types of load they contain.

Using the wind load generator and snow load generator

These are two ***Fastrak Portals Plus*** plug-ins.

- the ***Wind Load Generator*** calculates the wind loading that your frame must carry in accordance with ***BS 6399 : Part 2 : 1997*** or ***CP3 : Chapter V : Part 2 : September 1972***,

Caution CP3 : Chapter V : Part 2 : 1972 has been withdrawn by the BSI. You should only use this code of practice for legacy designs.

- the ***Snow Load Generator*** calculates the snow loading in accordance with ***BS 6399 : Part 3 : 1988***.

When you purchase and install these plug-ins ***Portal Frame*** recognises them automatically and activates the *Frame/Wind Loading…* and *Frame/Snow Loading…* menu options and the equivalent ***Toolbar*** icons to provide seamless integration.

Help For further details on these plug-ins ***see*** *"Wind Load Generator"* on page 369 and *"Snow Load Generator"* on page 401.

If you have not purchased these plug-ins, you can still apply wind and snow loading to your frame. However you will have to calculate and enter the loads directly yourself.

Understanding frame loadcases

The ***Frame Loadcases*** dialog allows you to define Dead, Imposed, Wind, etc. loadcases on your frame. When you choose the type of loads that your loadcase contains you will find that the loads that you can define are those that you would expect. You can also add the more general area, point and line loads to any loadcase type.

When you first see the ***Frame Loadcases*** dialog you may find that some loadcases are already defined, these are automatically created by ***Portal Frame*** from your building loads.

Help For further information **see** *“Building Wizard – Building Loads”* on page 149.

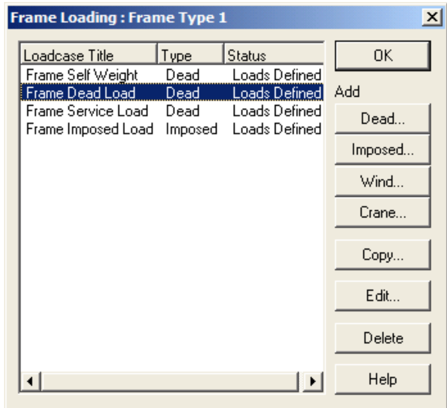
Note You cannot change the automatically generated *Self Weight Frame Loadcase* in any way. This loadcase includes the self weight of the main steel members only (i.e. not haunches, end plates, base plates etc.). To allow for the self weight of these and the other minor elements – sheeting rails, purlins etc. you can either edit the automatically generated *Frame Dead Load* loadcase or add a new *Frame Loadcase* to contain these loads.

Caution You can edit, delete and add loads in the other automatically generated frame loadcases, However if you edit or delete the **automatically generated** loads and then access and **OK** the building loads, the automatically generated loads will be recreated with their calculated values.

Help For further information *see* *"To modify an existing frame loadcase"* on page 297.

To define a frame loadcase

1. Select *Loading/Frame Loading...* to see the ***Frame Loading*** dialog.



This dialog shows the titles of all your frame loadcases and allows you to add new loadcases as well as editing, viewing or deleting existing ones.

Note **Crane...** is dimmed if your frame has no cranes.

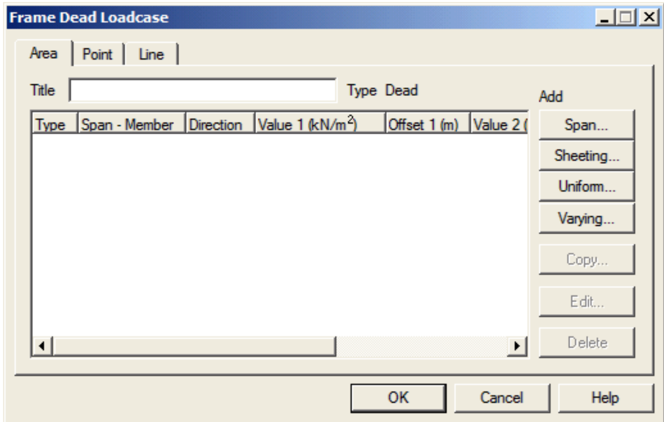
2. Once your frame loadcases are complete click **OK**.

Help For further information on adding loadcases *see*:

- *"To add a new dead frame loadcase"* on page 293,
- *"To add a new imposed frame loadcase"* on page 294,
- *"To add a new wind frame loadcase"* on page 295,
- *"To add a new crane frame loadcase"* on page 296.

To add a new dead frame loadcase

1. Click **Dead…** to see the ***Frame Dead Loadcase*** property sheet. The graphical display changes to the ***Loading*** diagram which shows a picture of the loads that you define.



2. Enter a descriptive **Title** for the loadcase which should enable you to differentiate it from the other loadcases that you may define.

3. Use the various pages of the property sheet to add your loads. As you define them you will see that they are added to the graphical display.

Tip You can close the *Frame Dead Loadcase* property sheet so that you can see the graphical display more clearly.

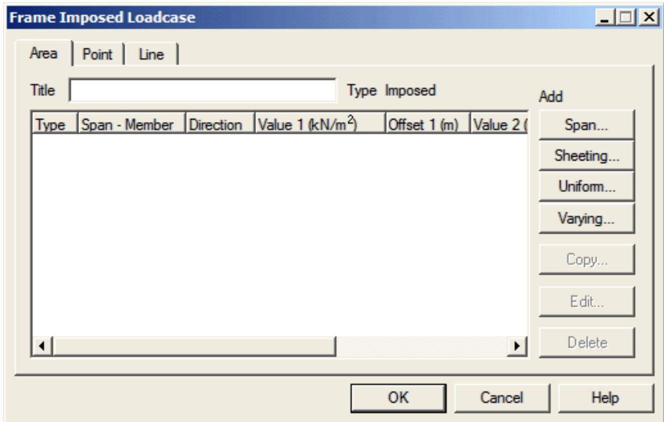
Help For further information on the various load types ***see***:

- *"Understanding area loads"* on page 299,
- *"Understanding point loads"* on page 314,
- *"Understanding line loads"* on page 321.

4. Once your loadcase is complete click **OK** to return to the ***Frame Loadcases*** dialog.

To add a new imposed frame loadcase

1. Click **Imposed…** to see the ***Frame Imposed Loadcase*** property sheet. The graphical display changes to the ***Loading*** diagram which shows a picture of the loads that you define.



2. Enter a descriptive **Title** for the loadcase which should enable you to differentiate it from the other loadcases that you may define.

3. Use the various pages of the property sheet to add your loads. As you define them you will see that they are added to the graphical display.

Tip — *You can close the Frame Imposed Loadcase property sheet so that you can see the graphical display more clearly.*

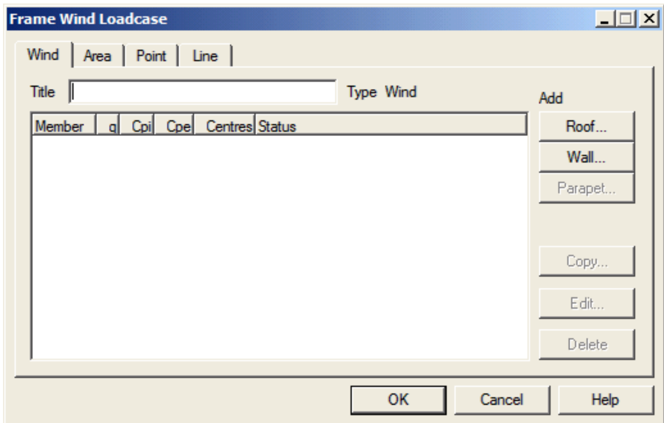
Help For further information on the various load types ***see***:

- *"Understanding area loads"* on page 299,
- *"Understanding point loads"* on page 314,
- *"Understanding line loads"* on page 321.

4. Once your loadcase is complete click **OK** to return to the ***Frame Loadcases*** dialog.

To add a new wind frame loadcase

1. Click **Wind…** to see the ***Frame Wind Loadcase*** property sheet. The graphical display changes to the ***Loading*** diagram which shows a picture of the loads that you define.



2. Enter a descriptive **Title** for the loadcase which should enable you to differentiate it from the other loadcases that you may define.

3. Use the various pages of the property sheet to add your loads. As you define them you will see that they are added to the graphical display.

Tip You can close the *Frame Wind Loadcase* property sheet so that you can see the graphical display more clearly.

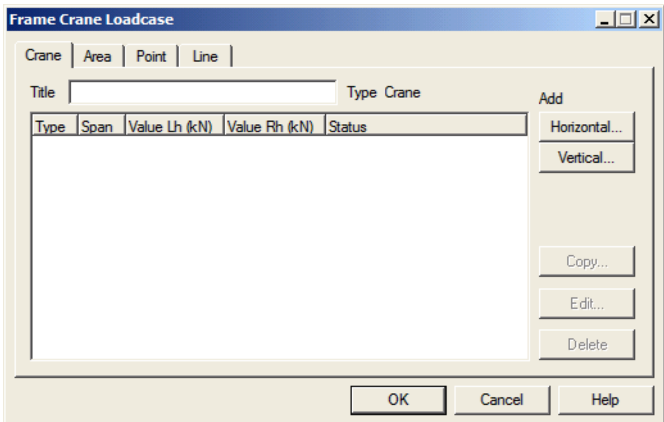
Help For further information on the various load types ***see***:

- *"Understanding wind loads"* on page 331,
- *"Understanding area loads"* on page 299,
- *"Understanding point loads"* on page 314,
- *"Understanding line loads"* on page 321.

4. Once your loadcase is complete click **OK** to return to the ***Frame Loadcases*** dialog.

To add a new crane frame loadcase

1. Click **Crane…**, to see the ***Frame Crane Loadcase*** property sheet. The graphical display changes to the ***Loading*** diagram which shows a picture of the loads that you define.



2. Enter a descriptive **Title** for the loadcase which should enable you to differentiate it from the other loadcases that you may define.

3. Use the various pages of the property sheet to add your loads. As you define them you will see that they are added to the graphical display.

Help For further information on the various load types ***see***:

- *"Understanding crane loads"* on page 359,
- *"Understanding area loads"* on page 299,
- *"Understanding point loads"* on page 314,
- *"Understanding line loads"* on page 321.

4. Once your loadcase is complete click **OK** to return to the ***Frame Loadcases*** dialog.

To modify an existing frame loadcase

1. Select the frame loadcase you want to edit and click **Edit…** to see its ***Frame Loadcase*** property sheet. for the frame loadcase that you have selected. The graphical display changes to the ***Loading*** diagram which shows a picture of the loads in the loadcase.

2. Make the changes that you need, again the graphical display is updated instantaneously.

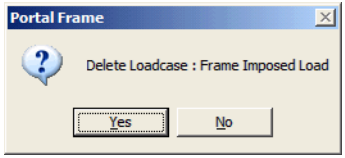
Help For further information on the various load types ***see***:

- *"Understanding area loads"* on page 299,
- *"Understanding point loads"* on page 314,
- *"Understanding line loads"* on page 321,
- *"Understanding wind loads"* on page 331,
- *"Understanding crane loads"* on page 359.

3. Once your changes are complete Click **OK** to return to the ***Frame Loadcases*** dialog.

To delete an existing frame loadcase

1. Select the frame loadcase that you want to delete and then click **Delete…** ***Portal Frame*** prompts you to confirm the deletion.



Caution Deletion removes the loadcase and all its associated loads **permanently. Please take care when using this feature.**

Caution Deletion also removes the loadcase from any design combinations that use it. Again **you can not recover these deleted details.**

2. To delete the frame loadcase click **Yes** to return to the ***Frame Loadcase*** dialog. If you do not want to delete the frame loadcase click **No**.

Understanding member ends

You apply many loads to the individual members of your frame. ***Portal Frame*** uses a consistent approach for the position and direction of all such loads:

- For a column you give the vertical distance from the base of the column to the point for the load.
- For a rafter you give the horizontal (plan) distance from the start of the rafter to the particular point for the load. Alternatively you may specify that a particular point of the load lies at the start or end of the member.

In the diagrams for each load type the dimensions you need to specify are clearly marked.

Understanding area loads

Area loads allow you to define the loads which normally cover large areas of your building. These loads are usually transferred from the sheeting through the sheeting rails, actually creating a series of point loads. It is traditional to treat such loads as distributed loads over the length of the member in portal frame design. In ***Portal Frame*** you apply these loads in kN/m^2 and they are multiplied by the frame centres to calculate the loading on the frame.

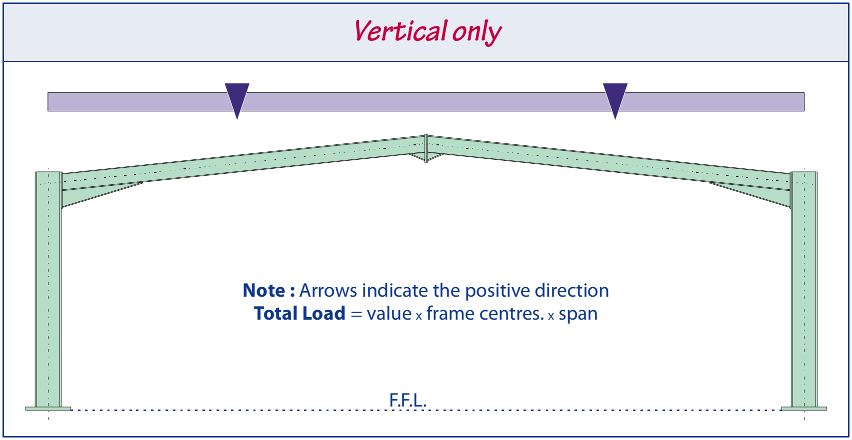
Note You can set the frame centres for each area load. This lets you cater for a particular load that might not be as extensive as the others.

You can define four types of area load – ***Span***, ***Sheeting***, ***Uniform*** and ***Varying*** as covered below.

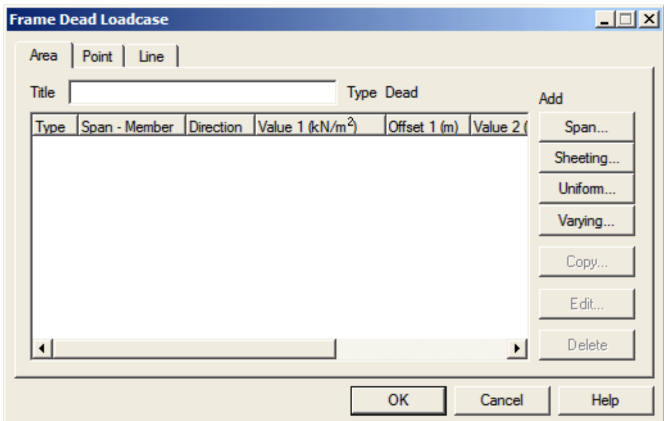
Note You apply *Span* and *Sheeting* loads to the rafters of a complete span and *Uniform* and *Varying* loads to individual members of that span. You can apply a uniform or varying load to the whole or part of a member.

To add span area loads

The sign convention for span area loads is shown in the table below. Span area loads can only be applied in the vertical direction.

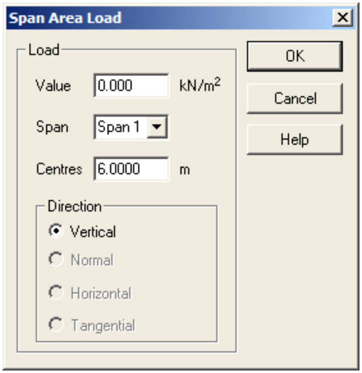


1. If the ***Area*** page is not visible pick the ***Area*** tab.



Tip You can close the *Frame … Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Span…** to see the ***Span Area Load*** dialog.



3. Enter the **Value** of the load, select the **Span** and enter the **Centres**.

Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you change the **Effective Frame Centres** this will reset the value here.

Help For further information ***see*** *"To modify the frame definition details"* on page 163.

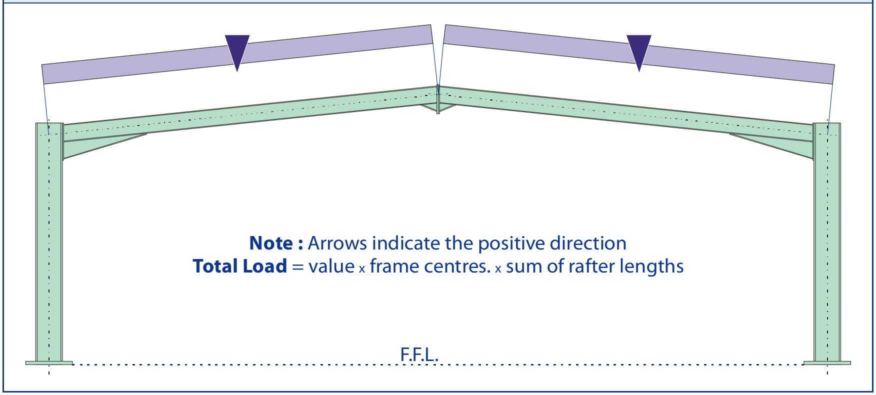
4. The only **Direction** for span area loads is vertical.

5. Once you have defined your load click **OK** to return to the ***Frame … Loadcase*** property sheet.

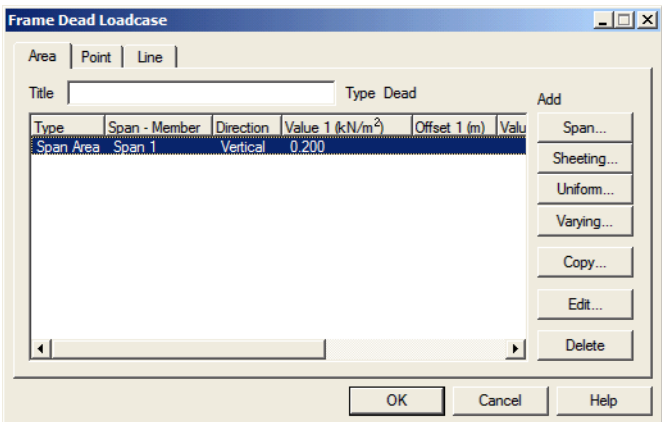
To add sheeting loads

The sign convention for sheeting loads is shown in the table below. Sheeting loads can only be applied in the vertical direction.

Vertical only

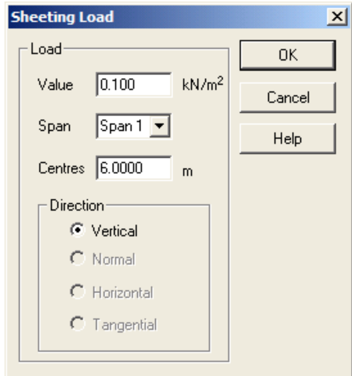


1. If the ***Area*** page is not visible pick the ***Area*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Sheeting...** to see the ***Sheeting Load*** dialog.



3. Enter the **Value** of the load, select the **Span** and enter the **Centres**.

Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you to change the **Effective Frame Centres** this will reset the value here.

Help For further information ***see*** *"To modify the frame definition details"* on page 163.

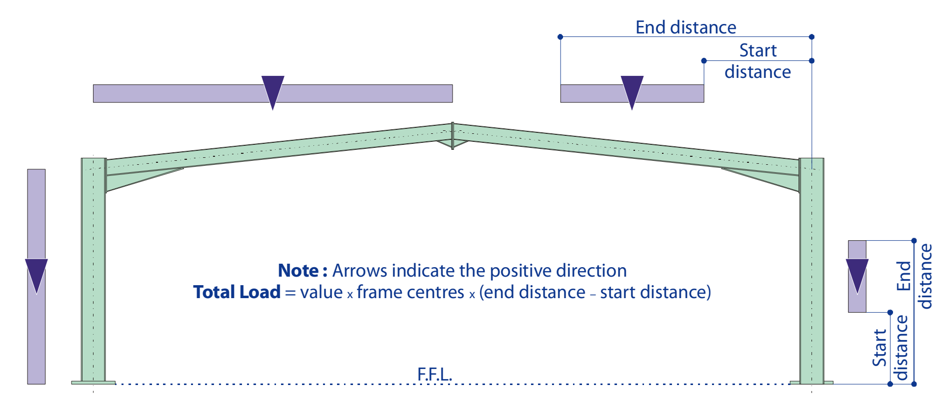
4. The only **Direction** for sheeting loads is vertical.

5. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

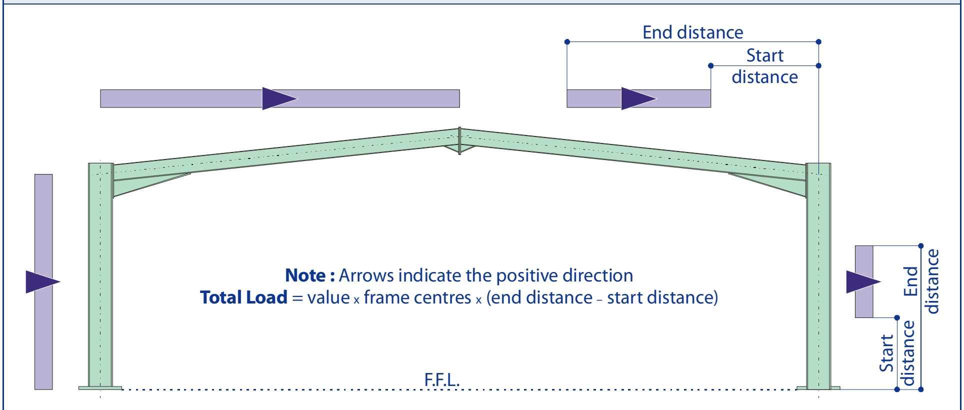
To add uniform area loads

The sign conventions for uniform area loads are shown in the table below. Uniform area loads can be applied in several directions.

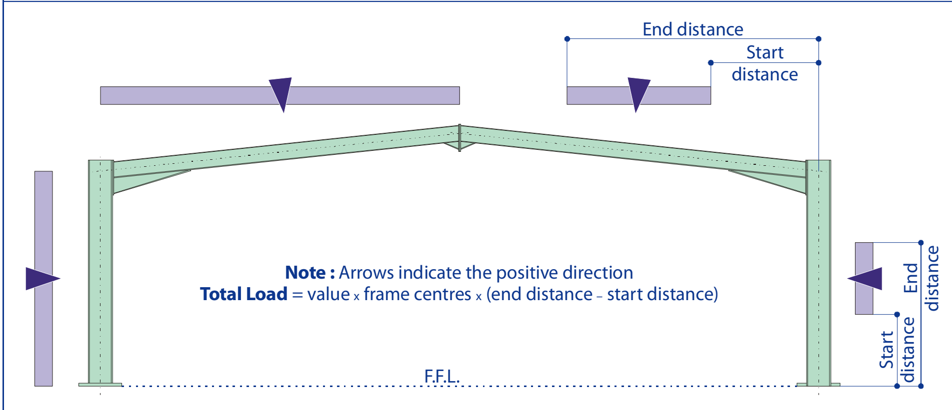
Vertical



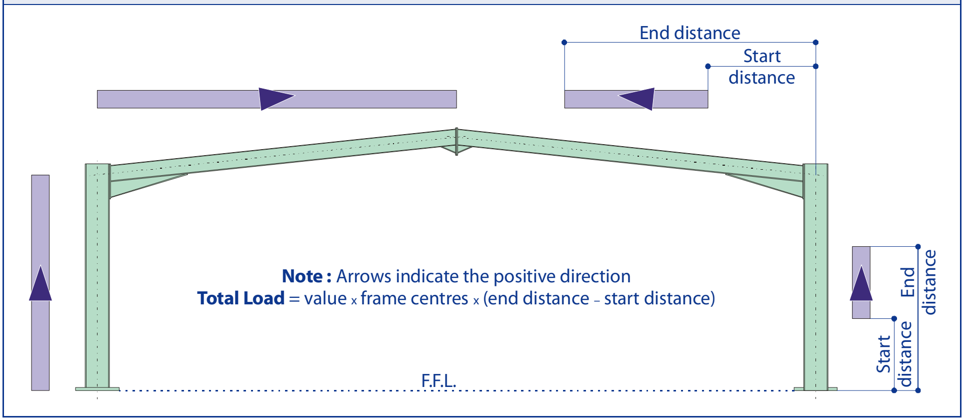
Horizontal



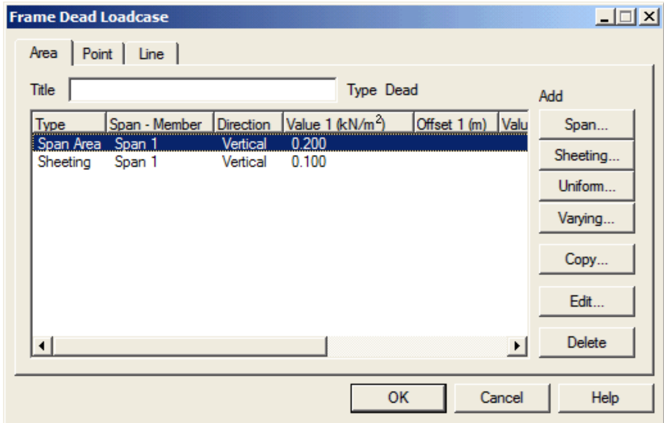
Normal



Tangential

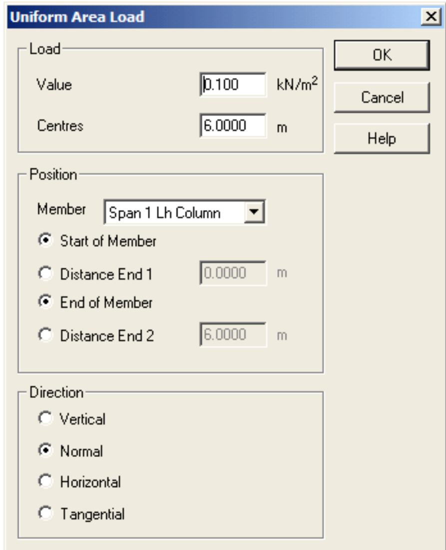


1. If the ***Area Loads*** page is not visible pick the ***Area*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Uniform…** You will then see the ***Uniform Area Load*** dialog.



3. Enter the **Value** of the load, the **Centres** and select the **Member.**

Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you to change the **Effective Frame Centres** in *Frame Definition* this will reset the value here.

Help For further information ***see*** *"To modify the frame definition details"* on page 163.

4. Set the position for the start of the uniform area load. If you pick the **Distance End 1** option then enter the distance from the start of the member to End 1 of the load.

Help For further information on start points and distances *see* *"Understanding member ends"* on page 298.

Note If you enter a **Distance to End 1** of the load, and change the member length you must ensure that your entered distance is still correct.

5. Select the position for the end of the uniform area load. If you pick the **Distance End 2** option then enter the distance from the start of the member to end 2 of the load.

Note If you enter a **Distance to End 2** of the load, and change the member length you must ensure that your entered distance is still correct.

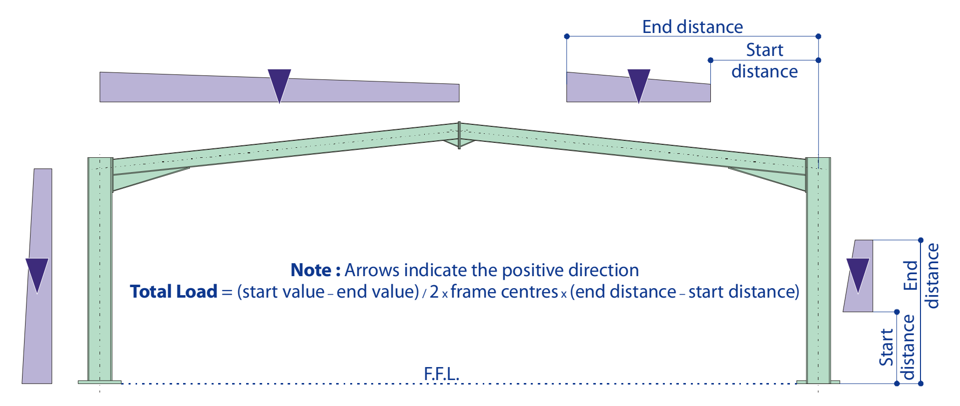
6. Select the **Direction** of the load as shown above.

7. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

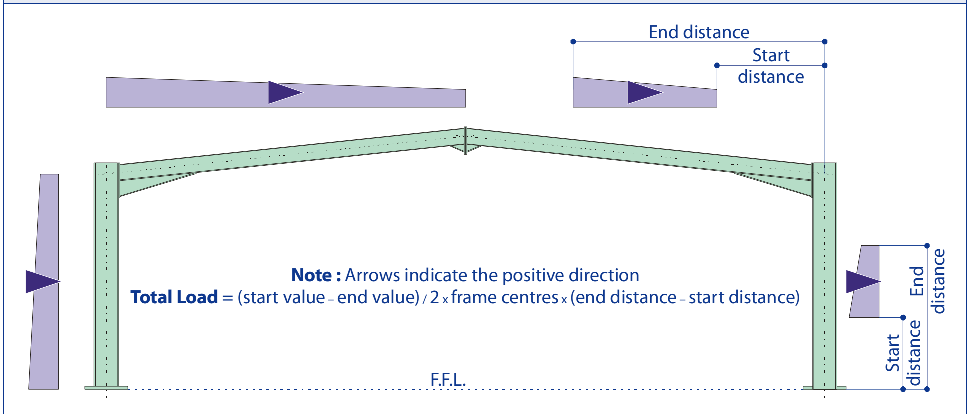
To add varying area loads

The sign conventions for varying area loads are shown in the table below. Varying area loads can be applied in several directions.

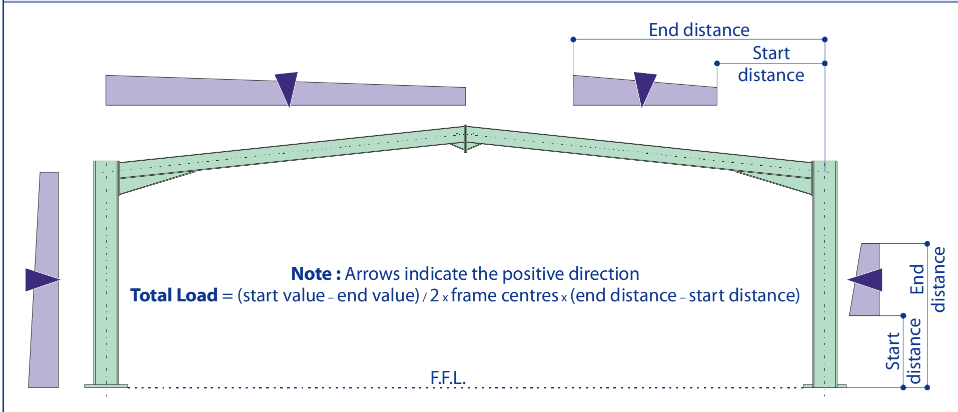
Vertical



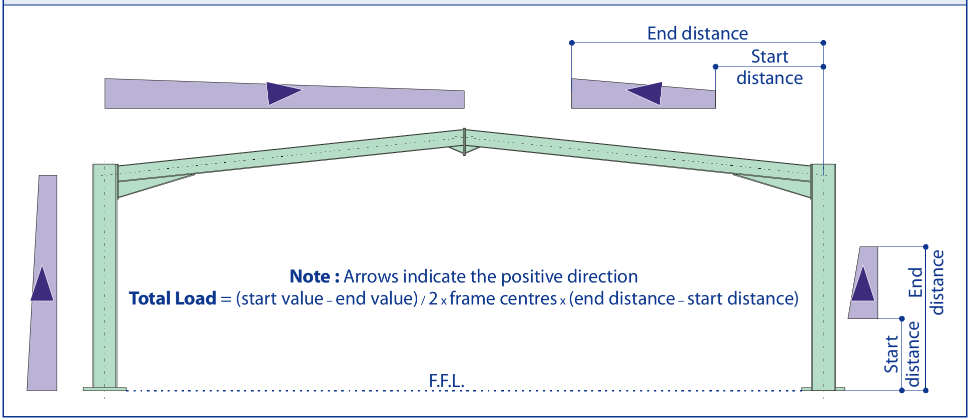
Horizontal



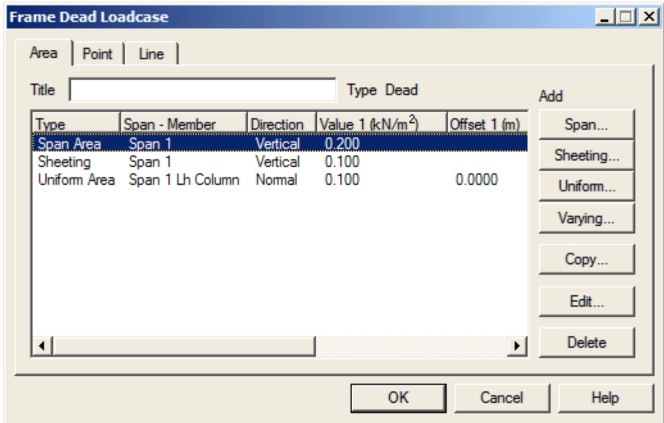
Normal



Tangential

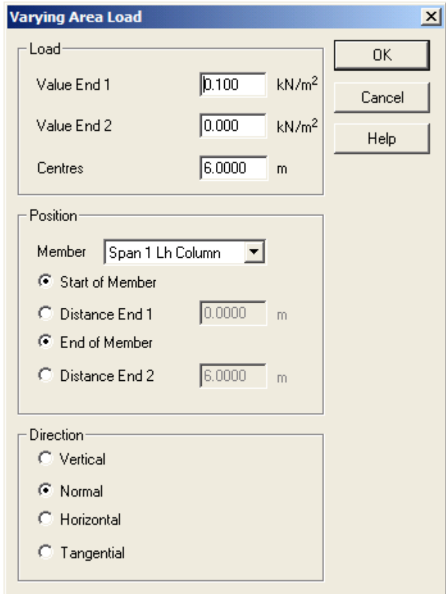


1. If the ***Area Loads*** page is not visible pick the ***Area*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Varying…** to see the ***Varying Area Load*** dialog.



3. Enter the **Values** of the load at End 1 and End 2, the **Centres** and select the **Member.**

Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you to change the **Effective Frame Centres** in *Frame Definition* this will reset the value here.

Help For further information ***see*** *"To modify the frame definition details"* on page 163.

4. Set the position for the start of the varying area load. If you pick the **Distance End 1** option then enter the distance from the start of the member to End 1 of the load.

Help For further information on start points and distances ***see*** *"Understanding member ends"* on page 298.

Note If you enter a **Distance to End 1** of the load, and change the member length you must ensure that your entered distance is still correct.

5. Select the position for the end of the varying area load. If you pick the **Distance End 2** option then enter the distance from the start of the member to end 2 of the load.

Note If you enter a **Distance to End 2** of the load, and change the member length you must ensure that your entered distance is still correct.

6. Select the **Direction** of the load as shown above.

7. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

To modify area loads

1. Select the area load you want to modify and click **Edit…** to see the area load dialog for that load.

Note You can not change the type of area load using this feature. You have to delete the load and then add a new one of the type you require.

?

Help

For further information ***see***:

- *"To add span area loads"* on page 300,
- *"To add sheeting loads"* on page 302,
- *"To add uniform area loads"* on page 304,
- *"To add varying area loads"* on page 309,
- *"To delete area loads"* on page 314.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete area loads

1. Select the area load that you want to remove and click **Delete**.

2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

Understanding point loads

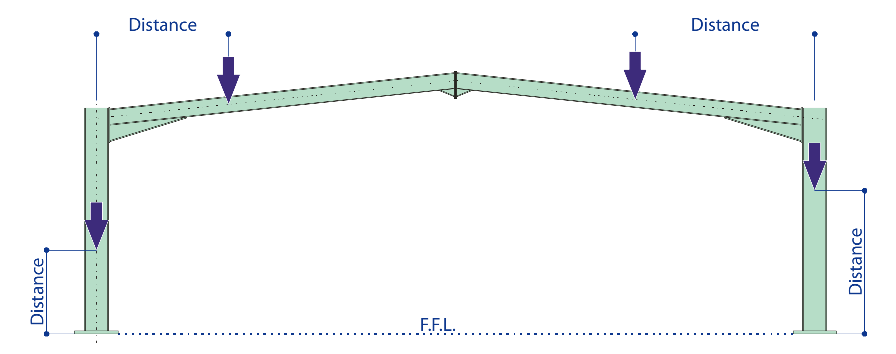
Point loads allows you to define individual forces and moments at a particular point on any member of the frame. Forces are applied in kN while moments are applied in kNm. The loads that you specify are applied directly to the member i.e. they are not multiplied by the frame centres.

You can define ***Loads*** and ***Moments*** as covered below.

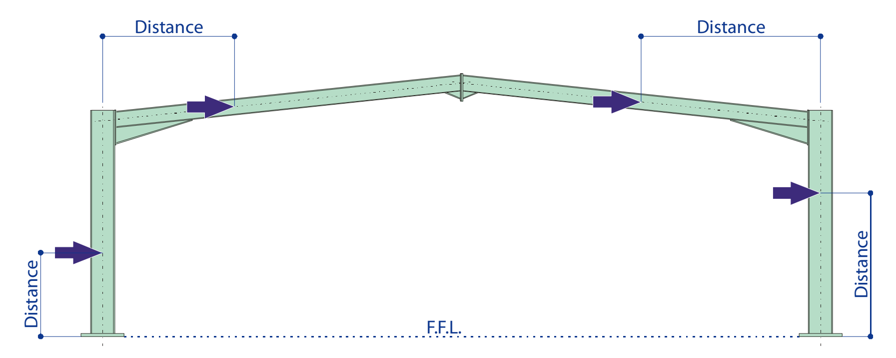
To add point loads

The sign conventions for point loads are shown in the table below. Point loads can be applied in several directions.

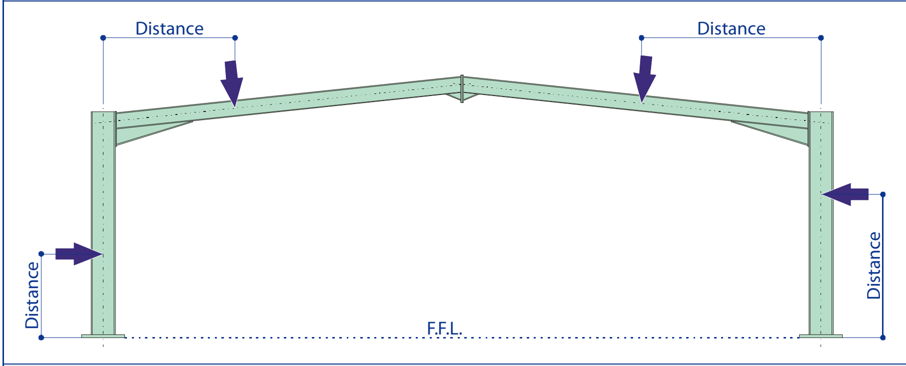
Vertical



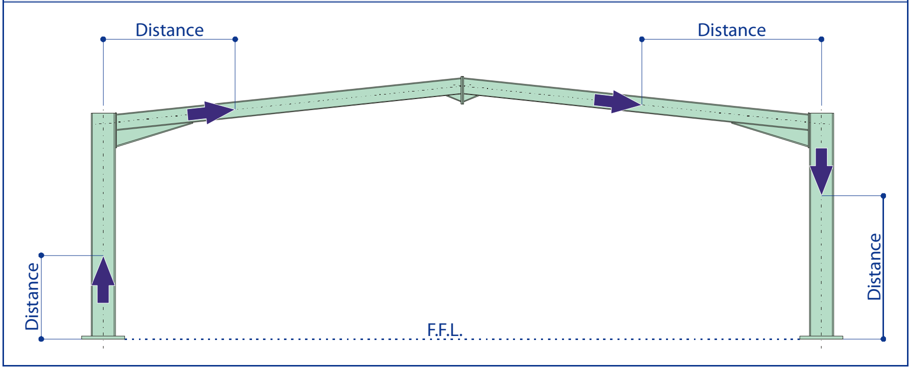
Horizontal



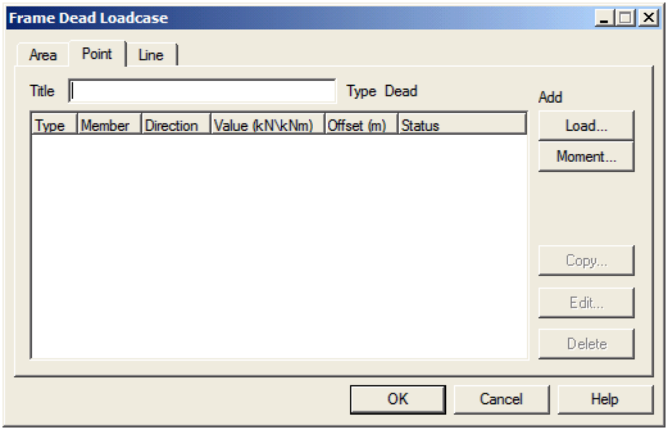
Normal



Tangential

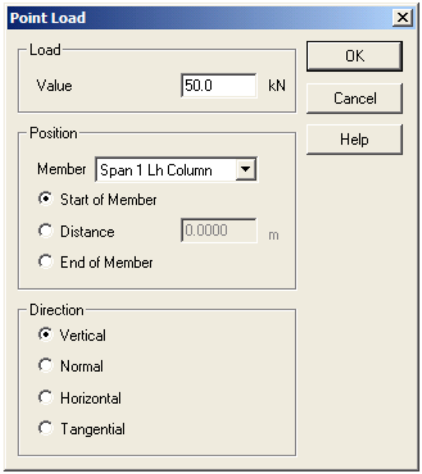


1. If the ***Point*** page is not visible pick the ***Point*** tab.



Tip You can close the *Frame … Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Load...** to see the ***Point Load*** dialog.



3. Enter the **Value** of the load and select the **Member** it is applied to.

4. Set the position for the point load. If you pick the **Distance** option then enter the distance from the start of the member to the load.

Help For further information on start points and distances ***see*** *"Understanding member ends"* on page 298.

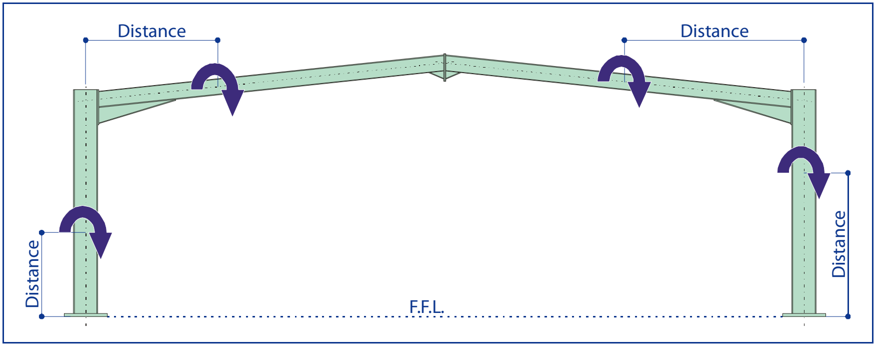
Note If you enter a **Distance** to the load, and change the member length you must ensure that your entered distance is still correct.

5. Select the **Direction** in which the load is to apply.

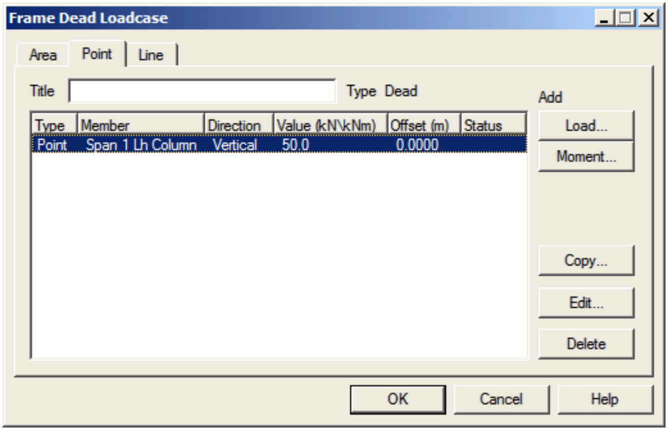
6. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

To add point moments

The sign conventions for point moments are shown in the table below.

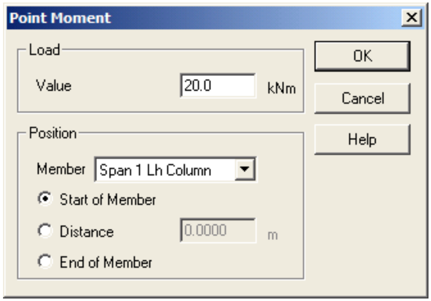


1. If the ***Point*** page is not visible pick the ***Point*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Moment…** to see the ***Point Moment*** dialog.



3. Enter the **Value** of the load and select the **Member** it is applied to.

4. Set the position for the point moment. If you pick the **Distance** option then enter the distance from the start of the member to the load.

Help For further information on start points and distances ***see*** *"Understanding member ends"* on page 298.

Note If you enter a **Distance** to the load, and change the member length you must ensure that your entered distance is still correct.

5. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

To modify point loads

1. Select the load or moment load that you want to modify and click **Edit…** to see the dialog for the selected load.

Note You can not change the type of load using this feature. You have to delete the load and then add a new one of the type you require.

Help For further information ***see***:

- *"To add point loads"* on page 315,
- *"To add point moments"* on page 319,
- *"To delete point loads"* on page 321.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete point loads

1. Select the load or moment that you want to remove and click **Delete**.
2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

Understanding line loads

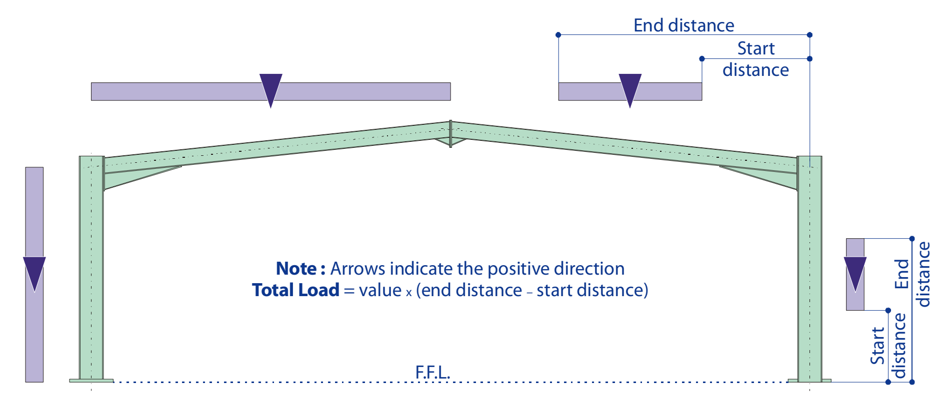
Line loads allows you to define uniform and varying distributed loads to the members of your frame. These loads are applied in kN/m and you specify the actual load on the member.

You can define ***Uniform*** and ***Varying*** line loads as covered below.

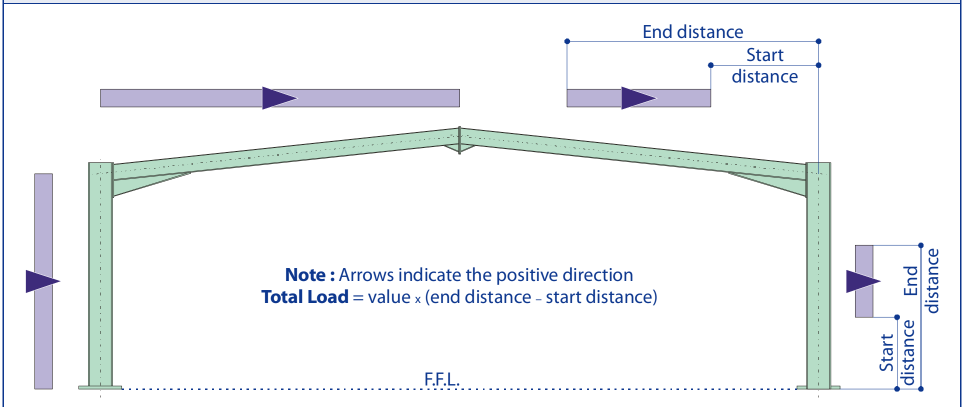
To add uniform line loads

The sign conventions for uniform line loads are shown in the table below. Uniform line loads can be applied in several directions.

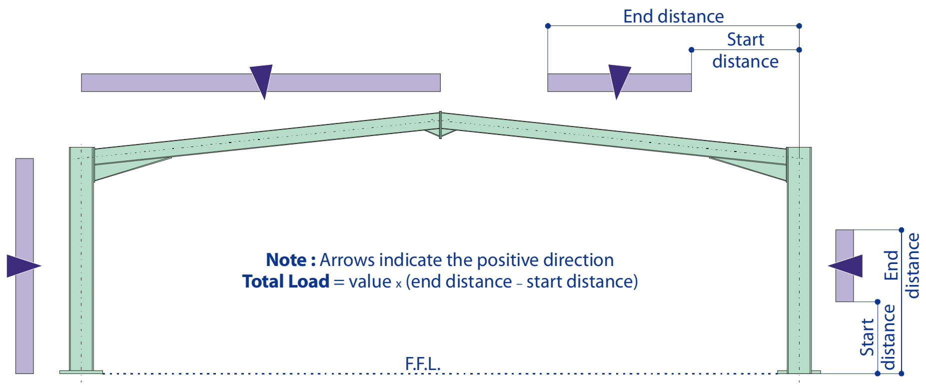
Vertical



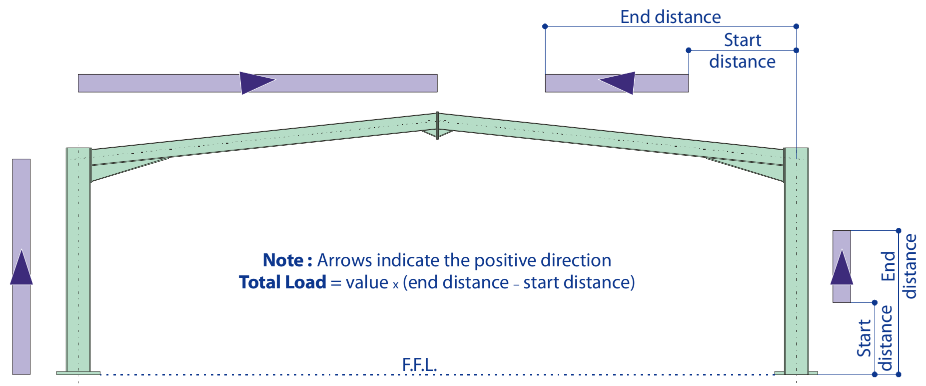
Horizontal



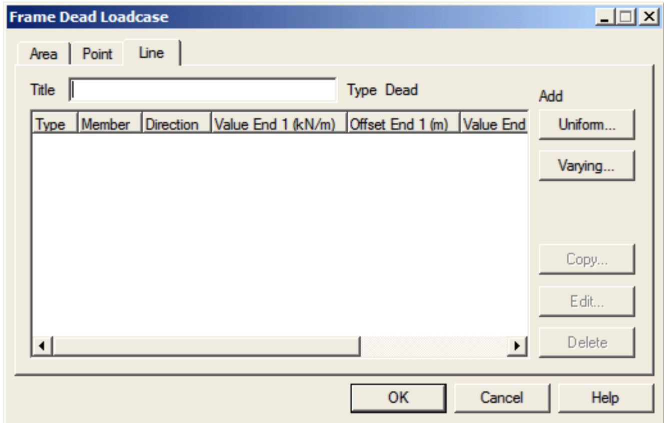
Normal



Tangential

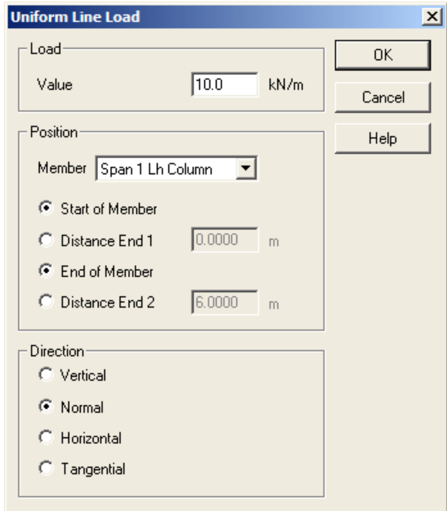


1. If the ***Line*** page is not visible pick the ***Line*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Uniform…** to see the ***Uniform Line Load*** dialog.



3. Enter the **Value** of the load and select the **Member** it is applied to.

4. Set the position for the start of the uniform line load. If you pick the **Distance End 1** option then enter the distance from the start of the member to End 1 of the load.

Help For further information on start points and distances ***see*** *"Understanding member ends"* on page 298.

Note If you enter a **Distance to End 1** of the load, and change the member length you must ensure that your entered distance is still correct.

5. Select the position for the end of the uniform line load. If you pick the **Distance End 2** option then enter the distance from the start of the member to end 2 of the load.

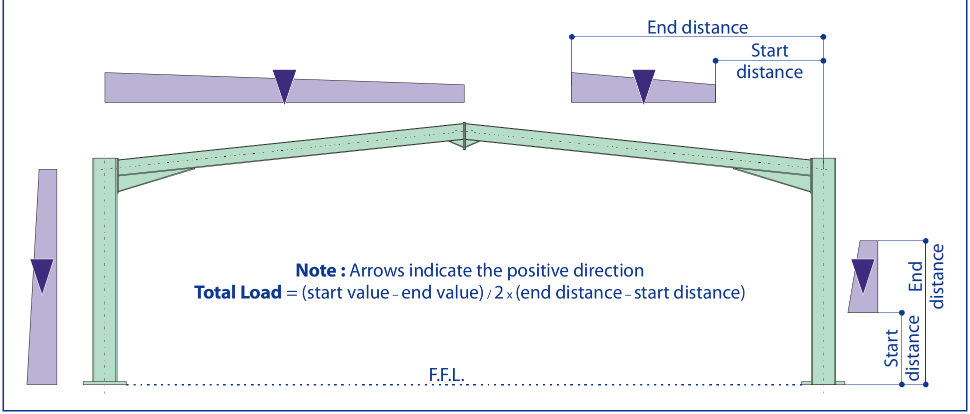
Note If you enter a **Distance to End 2** of the load, and change the member length you must ensure that your entered distance is still correct.

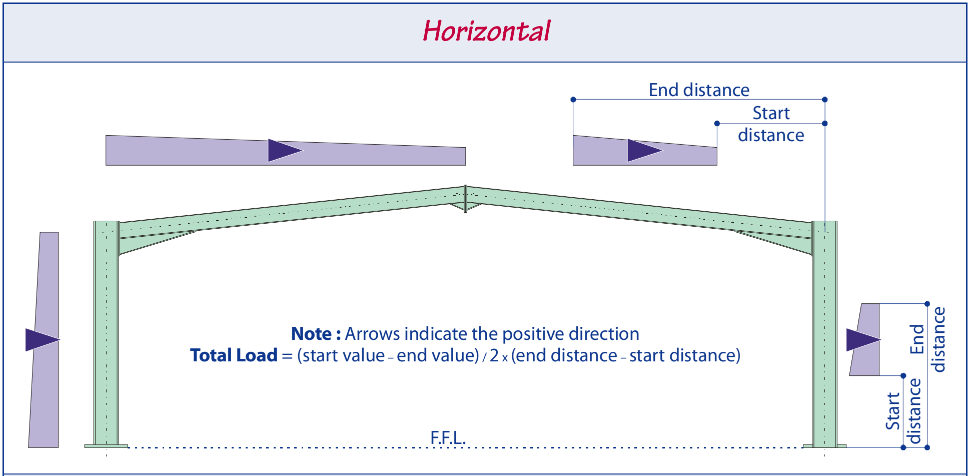
6. Select the **Direction** of the load as shown above.

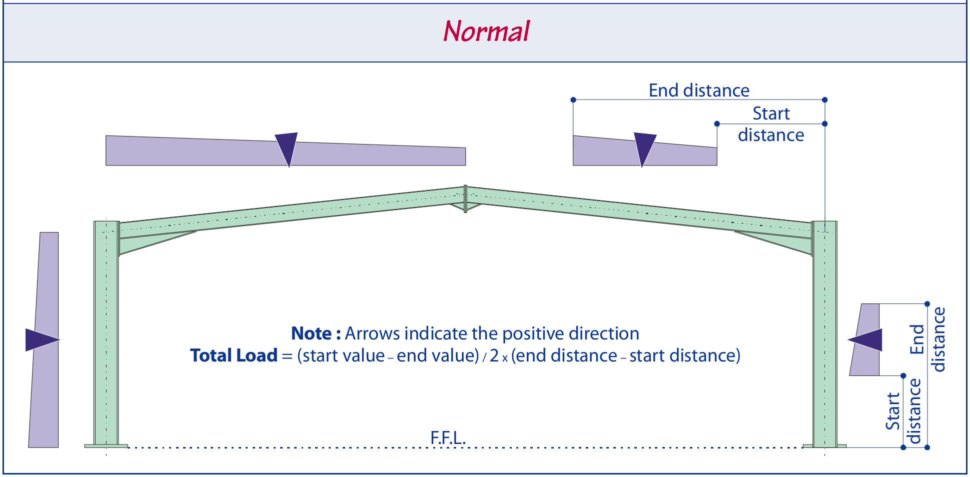
7. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

To add varying line loads

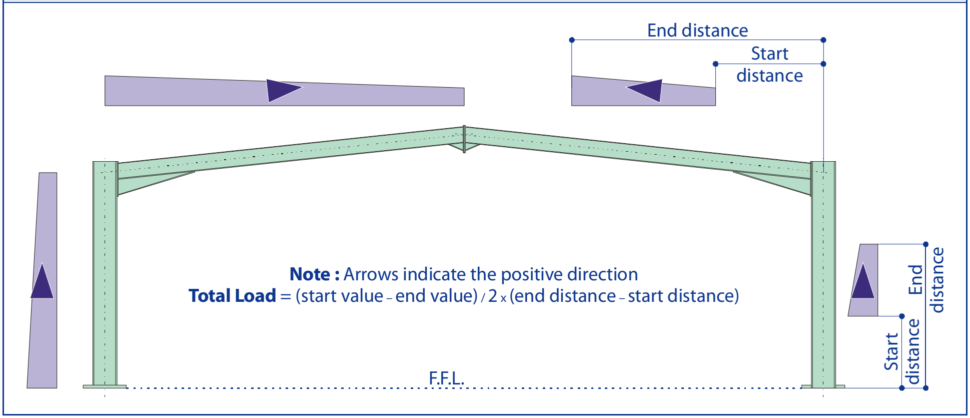
The sign conventions for varying line loads are shown in the table below. Varying area loads can be applied in several directions.

Vertical



Horizontal
End distance
Start
distance
End
distance
Start
distance
Note : Arrows indicate the positive direction
Total Load = (start value - end value) / 2 x (end distance - start distance)
F.F.L.

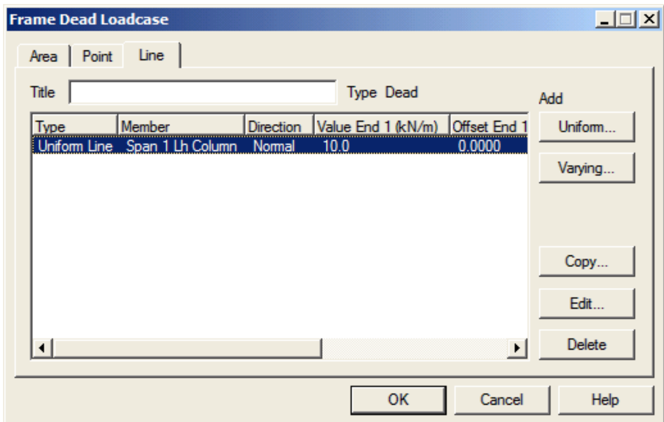

Normal
End distance
Start
distance
End
distance
Start
distance
Note : Arrows indicate the positive direction
Total Load = (start value - end value) / 2 x (end distance - start distance)
F.F.L.

Tangential



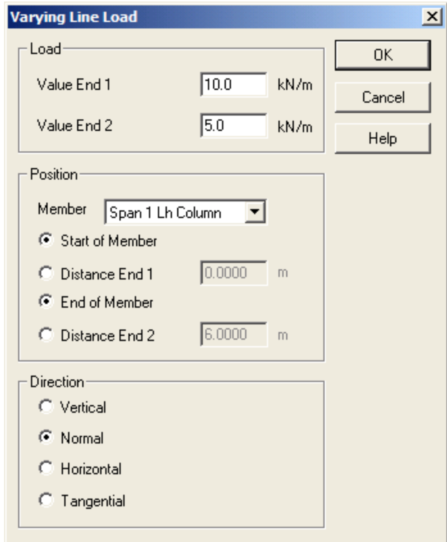
Note : Arrows indicate the positive direction
Total Load = (start value - end value) / 2 x (end distance - start distance)

1. If the ***Line*** page is not visible pick the ***Line*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Varying…** to see the ***Varying Line Load*** dialog.



3. Enter the **Values** of the load at End 1 and End 2 and select the **Member.**

4. Set the position for the start of the varying line load. If you pick the **Distance End 1** option then enter the distance from the start of the member to End 1 of the load.

Help For further information on start points and distances ***see*** *"Understanding member ends"* on page 298.

Note If you enter a **Distance to End 1** of the load, and change the member length you must ensure that your entered distance is still correct.

5. Select the position for the end of the varying line load. If you pick the **Distance End 2** option then enter the distance from the start of the member to end 2 of the load.

Note If you enter a **Distance to End 2** of the load, and change the member length you must ensure that your entered distance is still correct.

6. Select the **Direction** of the load as shown above.

7. Once you have defined your load click **OK** to return to the ***Frame ... Loadcase*** property sheet.

To modify line loads

1. Select the line load that you want to modify and click **Edit...** to see the line load dialog for the type of line load that you have selected.

Note You can not change the type of load using this feature. You have to delete the load and then add a new one of the type you require.

Help For further information ***see***:

- *"To add uniform line loads"* on page 322,
- *"To add varying line loads"* on page 326,
- *"To delete line loads"* on page 330.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete line loads

1. Select the line load that you want to remove and click **Delete**.

2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

Understanding wind loads

You can only add wind loads to wind frame loadcases, i.e. you must have chosen **Wind…** from the ***Frame Loadcase*** dialog.

?

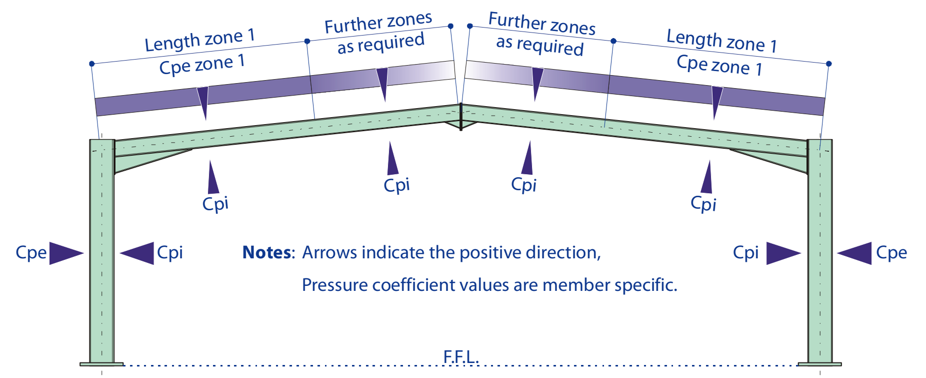
Help For further information ***see*** *"To add a new wind frame loadcase"* on page 295.

You can choose to define wind loads either to BS 6399 or to CP3, however you should note that ***CP3 has now been officially withdrawn, and you should therefore not use this code for designs in the United Kingdom***. The CP3 option has not been removed from the program since it may be appropriate in other geographical locations, or you may want to use it to check an existing building. The details for BS 6399 and CP3 are covered below.

BS 6399 wind load coefficients

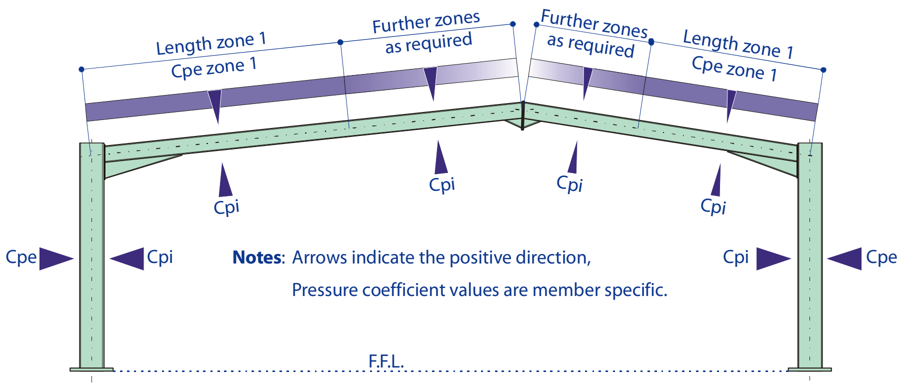
You define the loads that apply to the members of the frame by entering the ***Wind Pressure*** (q_s) and the ***Internal Pressure Coefficient*** (C_{pi}) for the entire building. For each wind zone[1] on a member you give its ***External Pressure Coefficient*** (C_{pe}) and its length. For the entire member you then give the information needed to calculate the size effect factor for that member.

BS 6399 wind load coefficients – Standard span

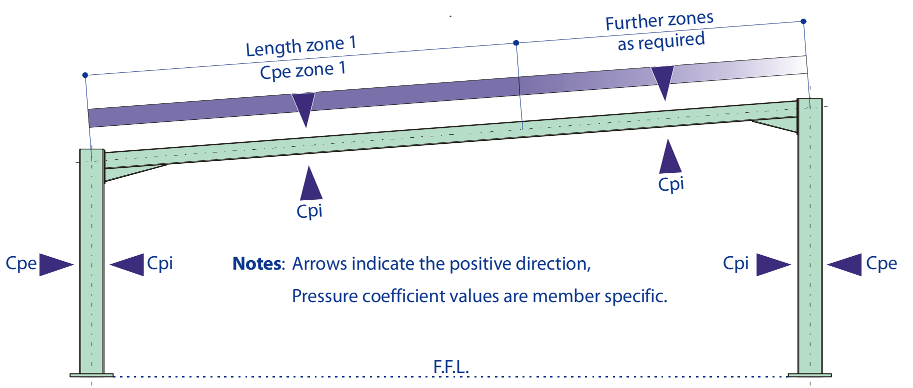


1. *The number of zones depends on the type of frame, the member whose loads you are defining, and the wind load condition you are defining.*

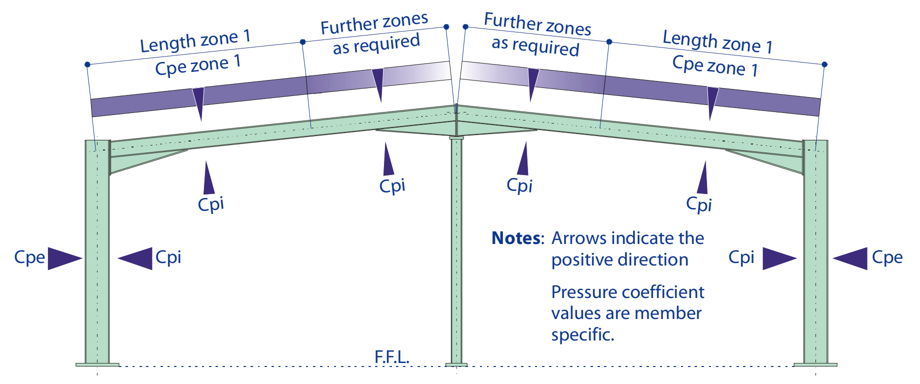
BS 6399 wind load coefficients – Asymmetric span



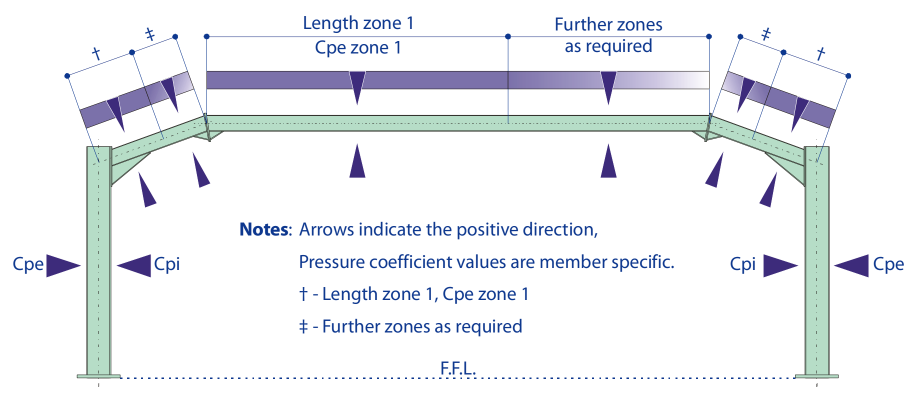
BS 6399 wind load coefficients – Monopitch span



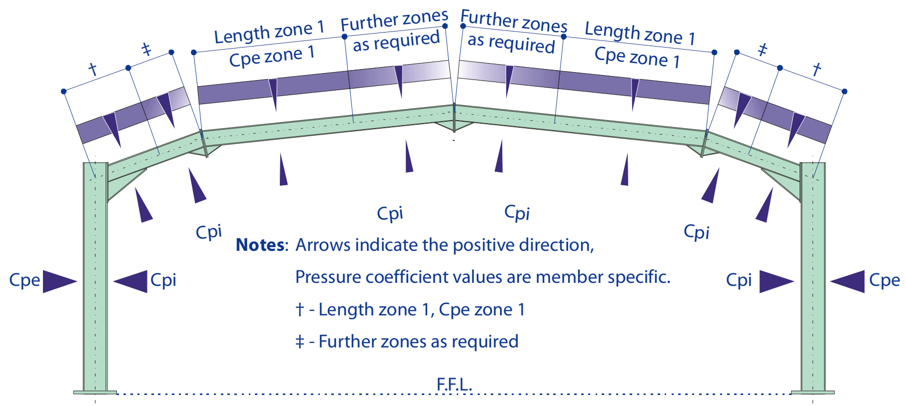
BS 6399 wind load coefficients – Propped span



BS 6399 wind load coefficients – Flat top span

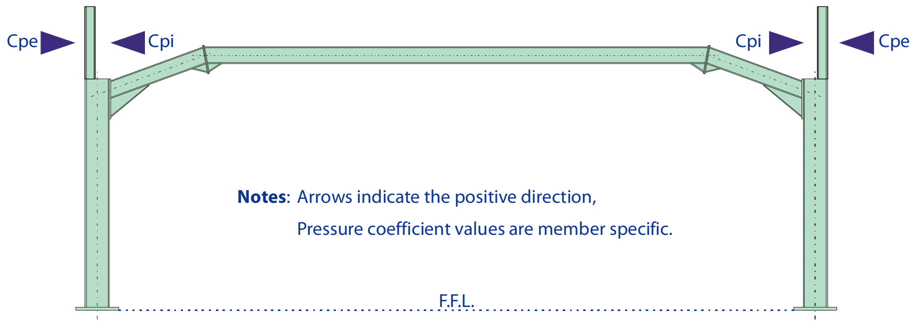


BS 6399 wind load coefficients – Mansard span



Notes: Arrows indicate the positive direction,
Pressure coefficient values are member specific.
† - Length zone 1, Cpe zone 1
‡ - Further zones as required

BS 6399 wind load coefficients – Parapets (any span type)

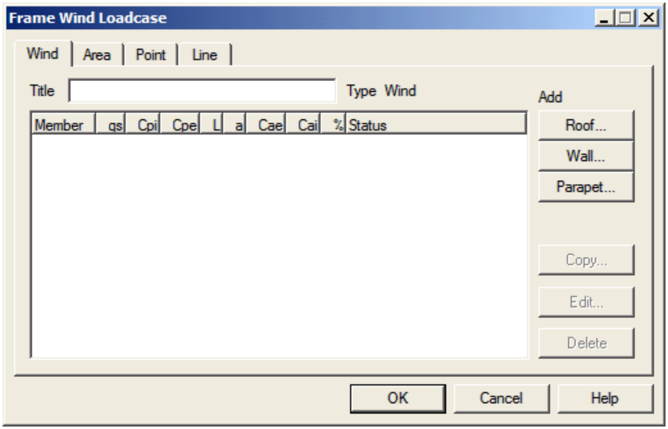


Notes: Arrows indicate the positive direction,
Pressure coefficient values are member specific.

You can apply wind loads to the ***Roof***, ***Walls*** and ***Parapets*** as detailed below.

To add roof wind loads to BS 6399

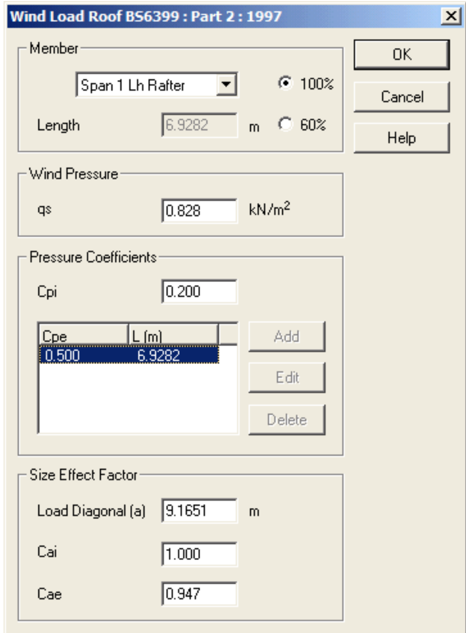
1. If the ***Wind*** page is not visible pick the ***Wind*** tab.



Tip

You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Roof...** to see the ***Roof Wind Load*** dialog.



3. Select the **Member** and choose whether it is to be fully- or partially-loaded (to account for asymmetric loading).

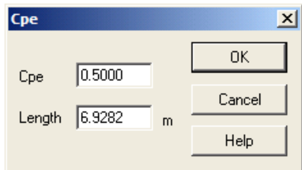
Note BS 6399 states that you should consider asymmetric loading by reducing the load on each member of your frame in turn to 60% of the full design load. *Portal Frame* allows you to set the 60% factor for as many members as you want since you may want to investigate conditions such as both windward rafters of a mansard portal having the 60% load.

4. Enter the wind pressure **qs** which applies to this member.

Note *The **Wind Pressure** is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.*

5. Enter the value of the **Internal Pressure Coefficient (Cpi)**,

6. To add the internal pressure coefficient(s) click **Add**

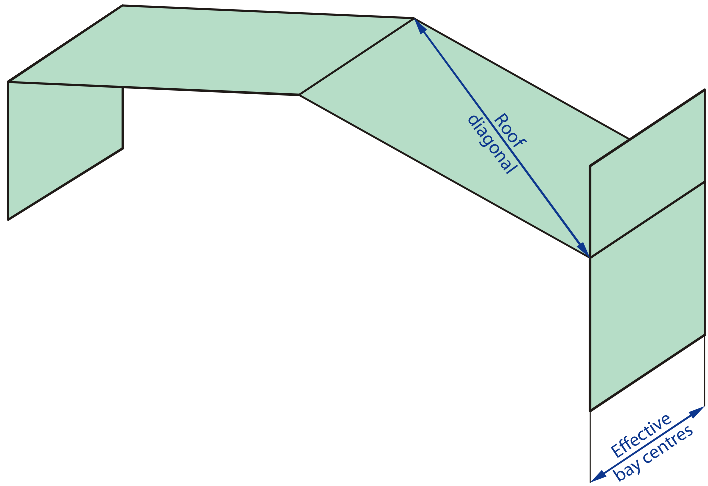


7. Enter the **External Pressure Coefficient (Cpe)** and the length over which it applies and then click **OK**.

Help *For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types **see** “Understanding wind loads” on page 331.*

8. If you need to define several zones on the member, then repeat step 7 for each zone.

9. Enter the **Load Diagonal (a)** dimension for the rafter as indicated in the diagram below.

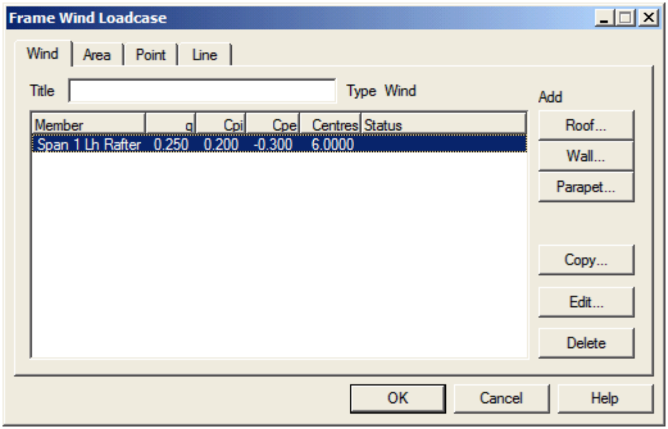


10. Enter the internal and external size effect factors (C_{ai} and C_{ae}) which enable the effective wind load on the rafter to be calculated.

11. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

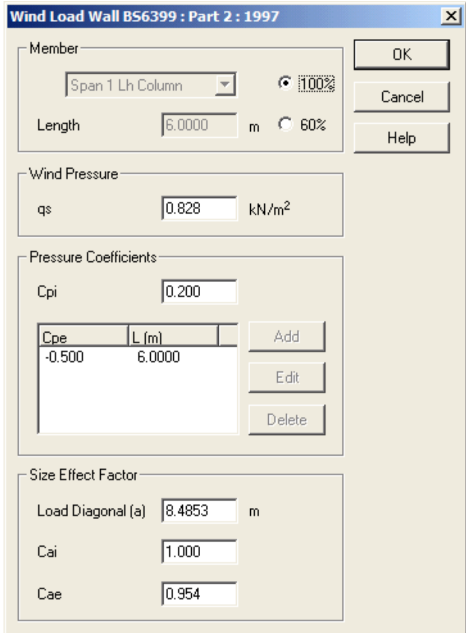
To add side wind loads to BS 6399

1. If the ***Wind*** page is not visible pick the ***Wind*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Wall...** to see the ***Wall Wind Load*** dialog.



3. Select the **Member** and choose whether it is to be fully- or partially-loaded (to account for asymmetric loading).

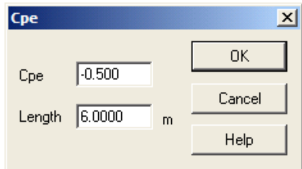
Note BS 6399 states that you should consider asymmetric loading by reducing the load on each member of your frame in turn to 60% of the full design load. *Portal Frame* allows you to set the 60% factor for as many members as you want since you may want to investigate conditions such as both windward rafters of a mansard portal having the 60% load.

4. Enter the wind pressure **qs** which applies to this member.

Note *The* **Wind Pressure** *is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.*

5. Enter the value of the **Internal Pressure Coefficient (Cpi)**,

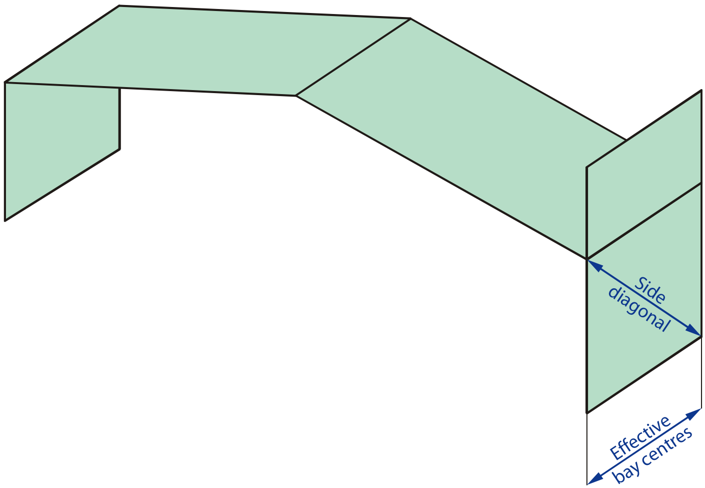
6. To add the internal pressure coefficient(s) click **Add**



7. Enter the **External Pressure Coefficient (Cpe)** and the length over which it applies and then click **OK**.

Help *For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types* **see** *“Understanding wind loads”* on page 331.

8. Enter the **Load Diagonal (a)** dimension for the column as indicated in the diagram below.

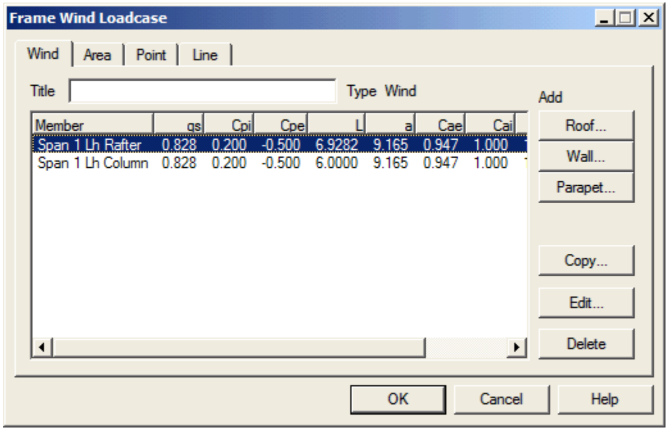


9. Enter the internal and external size effect factors (***C_{ai}*** and ***C_{ae}***) which enable the effective wind load on the column to be calculated.

10. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

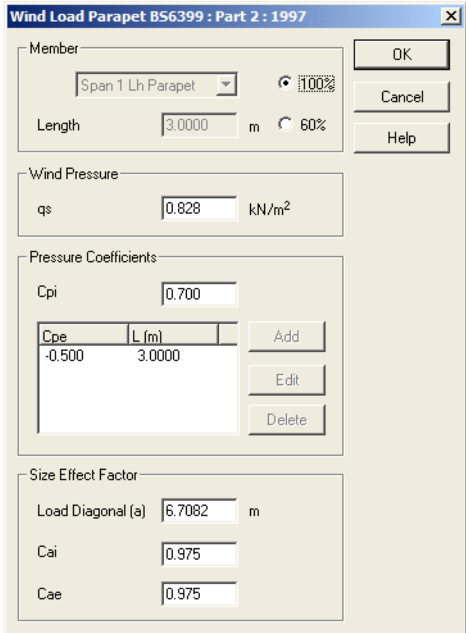
To add parapet wind loads to BS 6399

1. If the ***Wind Loads*** page is not visible, then pick the ***Wind*** tab to see it.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Parapet...** to see the ***Parapet Wind Load*** dialog.



Note *If you have not defined any parapets on your building, then* **Parapet...** *is dimmed.*

3. Select the **Member** and choose whether it is to be fully- or partially-loaded (to account for asymmetric loading).

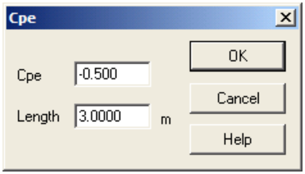
Note BS 6399 states that you should consider asymmetric loading by reducing the load on each member of your frame in turn to 60% of the full design load. *Portal Frame* allows you to set the 60% factor for as many members as you want since you may want to investigate conditions such as both windward rafters of a mansard portal having the 60% load.

4. Enter the wind pressure **qs** which applies to this member.

Note The **Wind Pressure** is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.

5. Enter the value of the **Internal Pressure Coefficient (Cpi)**,

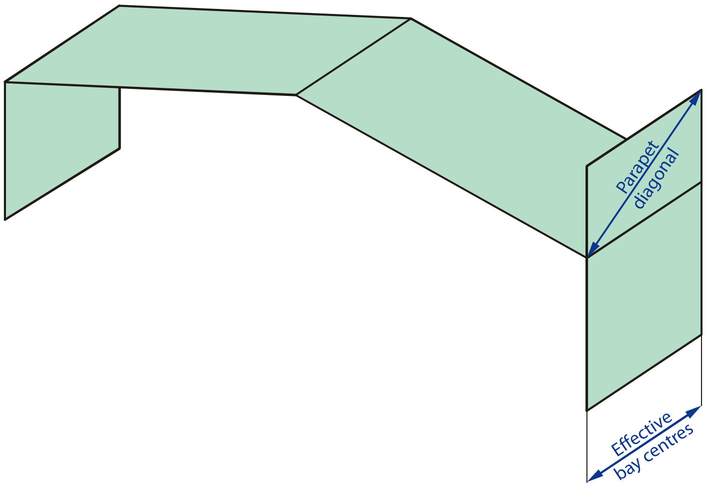
6. To add the internal pressure coefficient(s) click **Add**



7. Enter the **External Pressure Coefficient (Cpe)** and the length over which it applies and then click **OK**.

Help For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types **see** *"Understanding wind loads"* on page 331.

8. Enter the **Load Diagonal (a)** dimension for the column as indicated in the diagram below.



9. Enter the internal and external size effect factors (C_{ai} and C_{ae}) which enable the effective wind load on the column to be calculated.

10. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

To modify wind loads

1. Select the wind load that you want to modify and click **Edit…** to see the wind load dialog for the selected load.

Note You can not change the type of load using this feature. You have to delete the load and then add a new one of the type you require.

?

Help

For further information ***see***:

- *"To add roof wind loads to CP3"* on page 352,
- *"To add side wind loads to CP3"* on page 354,
- *"To add parapet wind loads to CP3"* on page 356,
- *"To delete wind loads"* on page 358.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete wind loads

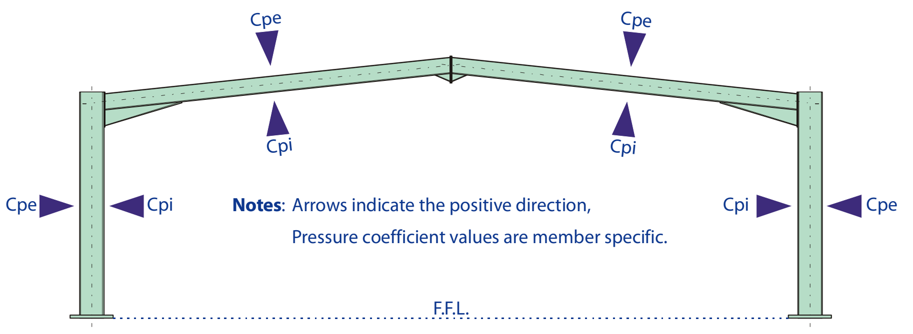
1. Select the wind load that you want to delete and click **Delete**.

2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

CP3 wind load coefficients

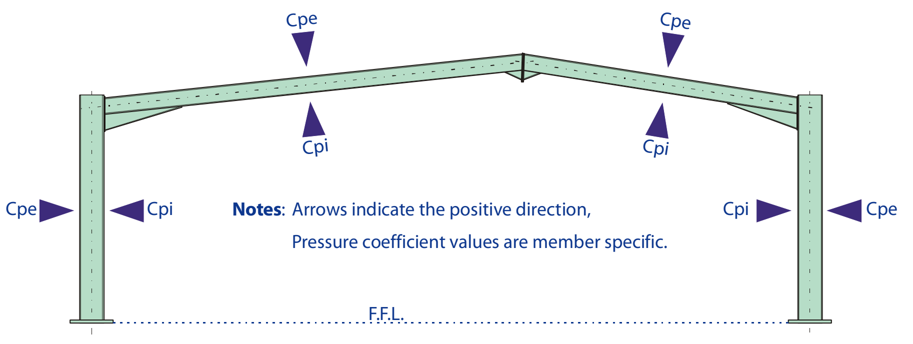
You define the loads that apply to the members of the frame by giving the ***Wind Pressure (q)*** and entering the ***Internal*** and ***External Pressure Coefficients*** (***Cpi*** and ***Cpe***) for the particular member.

CP3 wind load coefficients – Standard span

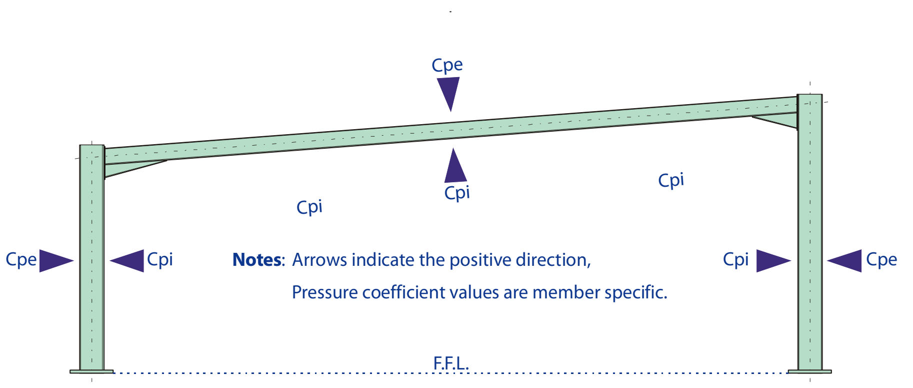


Notes: Arrows indicate the positive direction,
Pressure coefficient values are member specific.

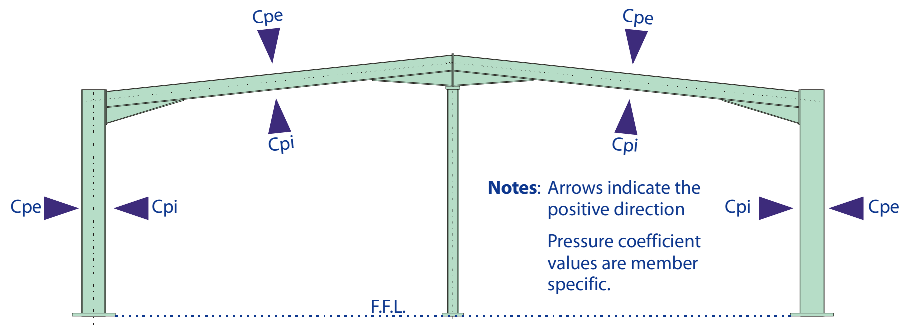
CP3 wind load coefficients – Asymmetric span



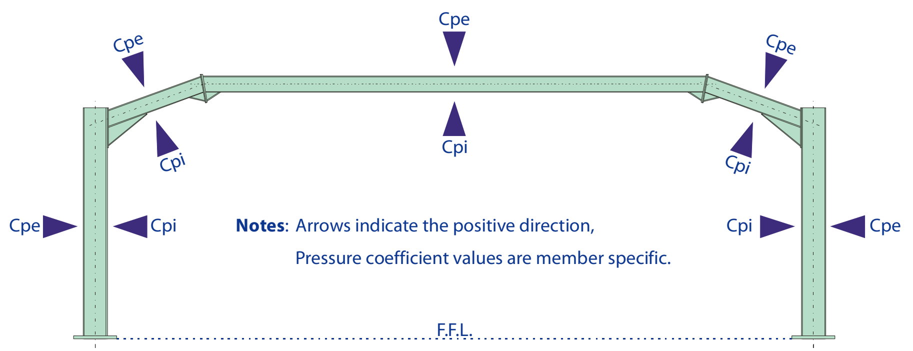
CP3 wind load coefficients – Monopitch span



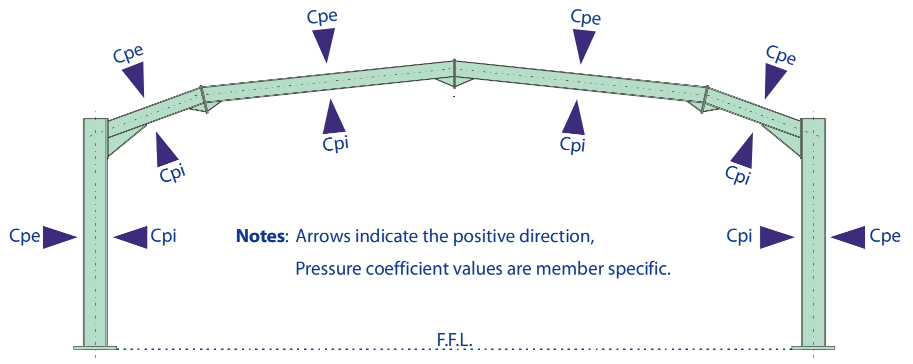
CP3 wind load coefficients – Propped span



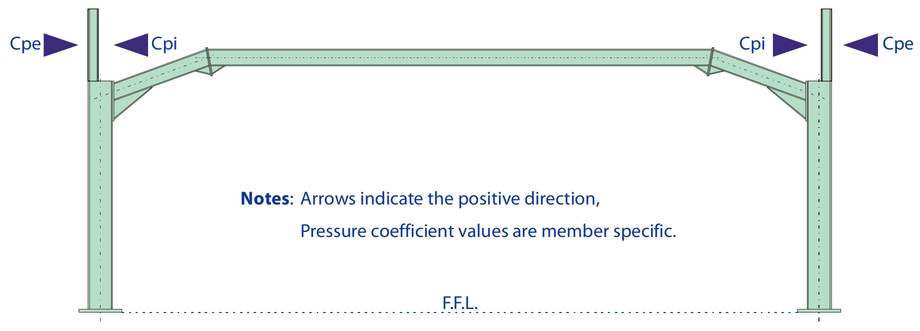
CP3 wind load coefficients – Flat top span



CP3 wind load coefficients – Mansard span



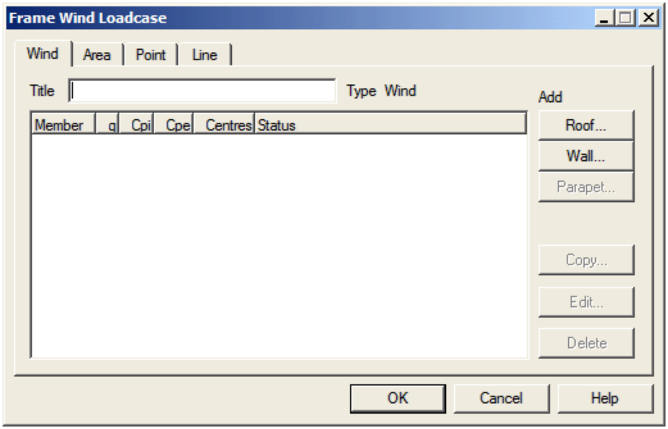
CP3 wind load coefficients – Parapets (any span type)



You can apply wind loads to the ***Roof***, ***Walls*** and ***Parapets*** as detailed below.

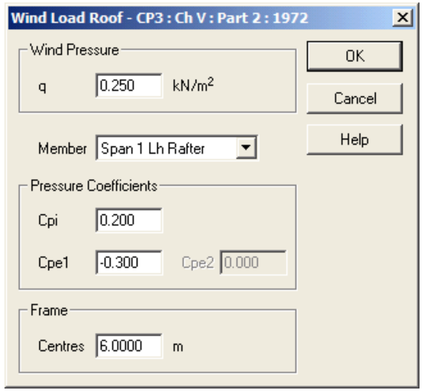
To add roof wind loads to CP3

1. If the ***Wind*** page is not visible pick the ***Wind*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Roof...** to see the ***Roof Wind Load*** dialog.



3. Give the **Wind Pressure** and select the **Member**.

Note *The **Wind Pressure** is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.*

4. Enter values for the **Internal Pressure Coefficient (Cpi)**, the **External Pressure Coefficient (Cpe1)** and if necessary the **Internal Pressure Coefficient (Cpe2)**.

Help *For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types **see** "Understanding wind loads" on page 331.*

5. Enter the **Centres**.

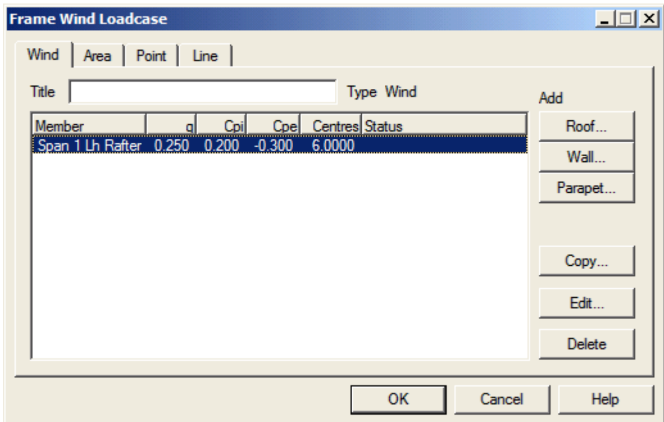
Caution `Centres` is set to the **`Effective Frame Centres`** that you specified in *`Frame Definition`*, you can change this if necessary. However if you to change the **`Effective Frame Centres`** in *`Frame Definition`* this will reset the value here.

Help For further information **see** *“To modify the frame definition details”* on page 163.

6. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

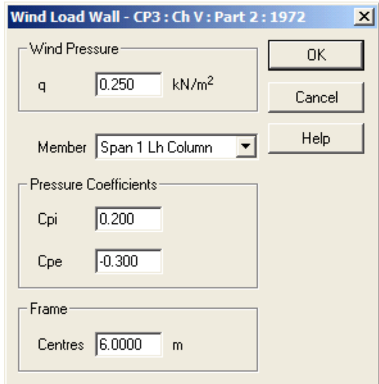
To add side wind loads to CP3

1. If the ***Wind*** page is not visible pick the ***Wind*** tab.



Tip You can close the *`Frame ... Loadcase`* property sheet so that you can see the graphical display more clearly.

2. Click **Wall…** to see the ***Wall Wind Load*** dialog.



3. Give the **Wind Pressure** and select the **Member**.

Note *The **Wind Pressure** is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.*

4. Enter values for the **Internal Pressure Coefficient (Cpi)** and the **External Pressure Coefficient (Cpe)**.

Help *For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types* ***see** “Understanding wind loads” on page 331.*

5. Enter the **Centres**.

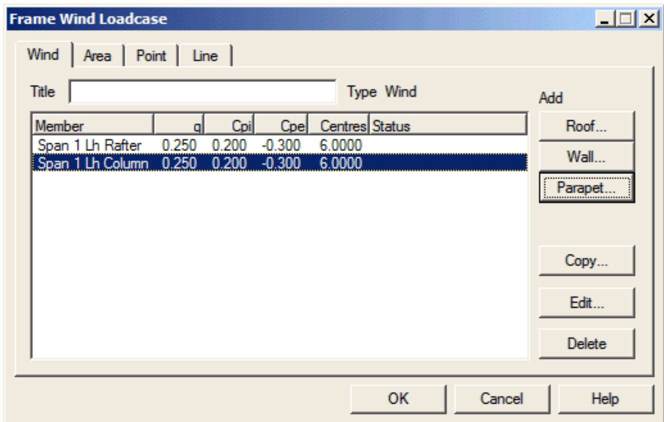
Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you to change the **Effective Frame Centres** in *Frame Definition* this will reset the value here.

Help For further information **see** *"To modify the frame definition details"* on page 163.

6. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

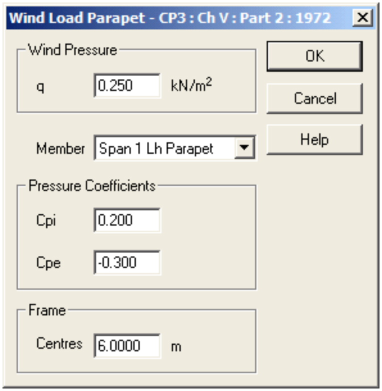
To add parapet wind loads to CP3

1. If the ***Wind Loads*** page is not visible, then pick the ***Wind*** tab to see it.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Parapet...** to see the ***Parapet Wind Load*** dialog.



Note If you have not defined any parapets on your building, then **Parapet...** is dimmed.

3. Give the **Wind Pressure** and select the **Member.**

Note The **Wind Pressure** is always positive, the direction of the load applying to a particular member is determined by the internal and external pressure coefficients.

4. Enter values for the **Internal Pressure Coefficient (Cpi)** and the **External Pressure Coefficient (Cpe)**.

Help For details on the sign conventions for pressure coefficients and those that are required for the members of various frame types **see** *"Understanding wind loads"* on page 331.

5. Enter the **Centres**.

Caution **Centres** is set to the **Effective Frame Centres** that you specified in *Frame Definition*, you can change this if necessary. However if you to change the **Effective Frame Centres** in *Frame Definition* this will reset the value here.

Help For further information ***see*** *"To modify the frame definition details"* on page 163.

6. Once your wind load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

To modify wind loads

1. Select the wind load that you want to modify and click **Edit…** to see the wind load dialog for the selected load.

Note You can not change the type of load using this feature. You have to delete the load and then add a new one of the type you require.

Help For further information ***see***:

- *"To add roof wind loads to CP3"* on page 352,
- *"To add side wind loads to CP3"* on page 354,
- *"To add parapet wind loads to CP3"* on page 356,
- *"To delete wind loads"* on page 358.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete wind loads

1. Select the wind load that you want to delete and click **Delete**.

2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

Understanding crane loads

You can only add crane loads to crane frame loadcases, i.e. you must have chosen **Crane…** from the ***Frame Loadcase*** dialog.

Note *You can only define Crane loadcase if your frame contains at least one crane.*

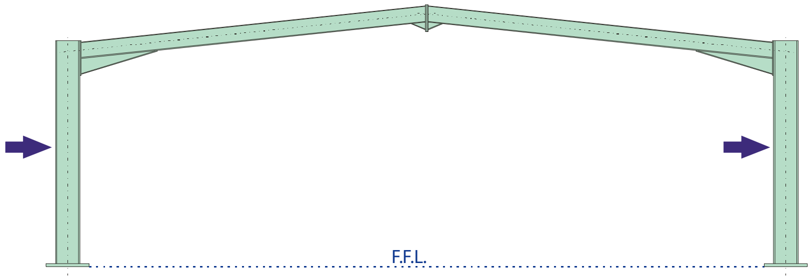
Help For further information on cranes ***see:***

- *"To add a crane"* on page 221,
- *"To add a new crane frame loadcase"* on page 296.

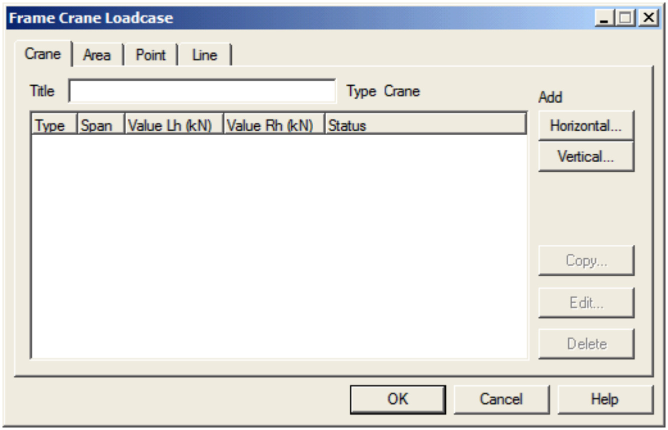
You can define ***Horizontal*** and ***Vertical*** loads to any cranes as described below.

To define horizontal crane loads

The sign convention for horizontal crane loads is shown in the table below.

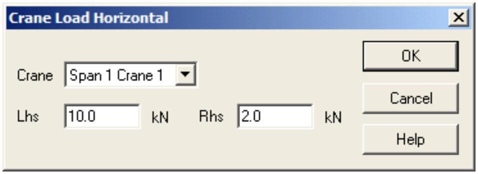


1. If the ***Crane*** page is not visible pick the ***Crane*** tab.



Tip *You can close the Frame ... Loadcase property sheet so that you can see the graphical display more clearly.*

2. Click **Horizontal...** to see the ***Horizontal Crane Load*** dialog.

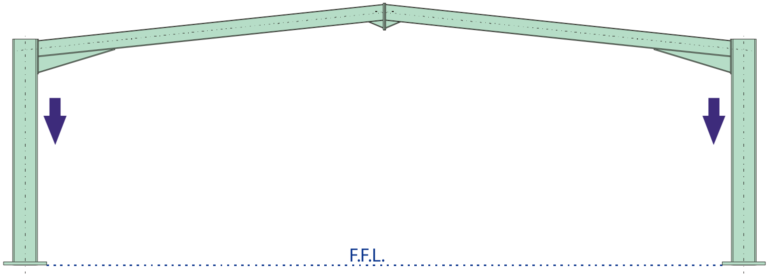


3. Select the **Crane** and enter the **Values** of the horizontal crane load at the left and right hand crane supports

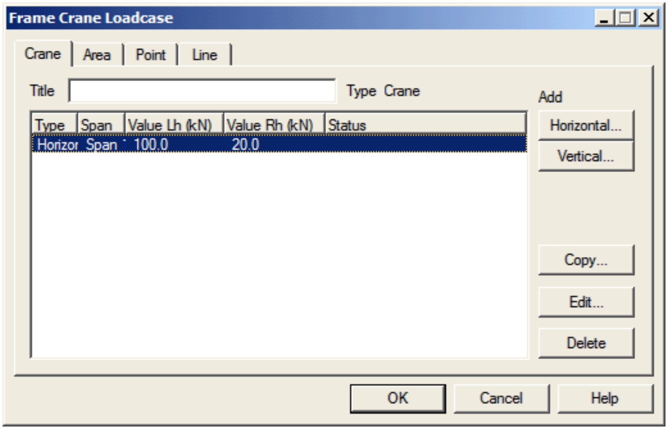
4. Once your horizontal crane load details are complete click **OK** to return to the ***Frame Loadcase*** property sheet.

To define vertical crane loads

The sign convention for vertical crane loads is shown in the table below.

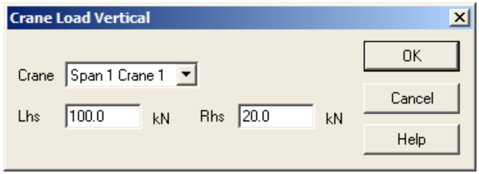


1. If the ***Crane*** page is not visible pick the ***Crane*** tab.



Tip You can close the *Frame ... Loadcase* property sheet so that you can see the graphical display more clearly.

2. Click **Vertical...** to see the ***Vertical Crane Load*** dialog.



3. Select the **Crane** and enter the **Values** of the vertical crane load at the left and right hand crane supports

To modify crane loads

1. Select the crane load that you want to modify and click **Edit...** to see the crane dialog for the selected crane load.

Note You can not change the type of load using this feature. You have to delete the load and then add a new one of the type you require.

Help For further information ***see***:

- *"To define horizontal crane loads"* on page 359,
- *"To define vertical crane loads"* on page 361,
- *"To delete crane loads"* on page 362.

2. Make your changes and click **OK** to return to the ***Frame Loadcase*** property sheet.

To delete crane loads

1. Select the crane load that you want to remove and click **Delete**.
2. The ***Frame Loadcase*** property sheet updates to remove the deleted load.

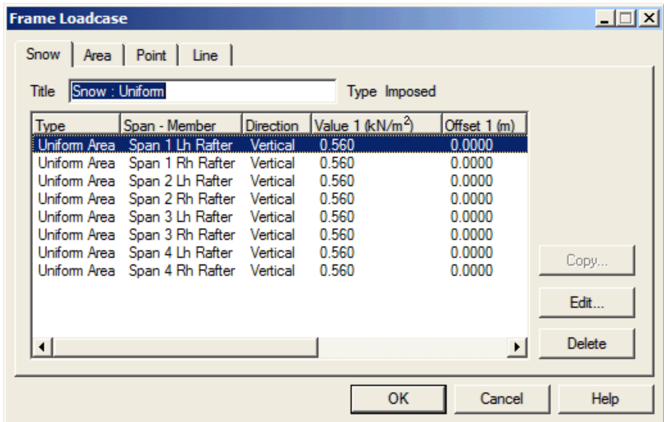
Understanding snow loads

When you have used the ***Snow Load Generator*** to calculate a snow loading on your frame, you can then change the details directly, without having to use the ***Snow Load Generator*** at all.

To edit snow loads directly

1. From the ***Frame Loading*** dialog, choose the snow loadcase whose details you want to change and then click **Edit**.

2. You will see the ***Frame Loadcase*** dialog which will show a ***Snow*** page containing the details for the snow load.



3. You can now edit each of the snow loads in turn by clicking on its line and clicking **Edit** and changing the details as for any other load.

4. Once you have made changes to all the loads you need to change click **OK** to return to the ***Frame Loading*** dialog.

Copying loadcases and loads

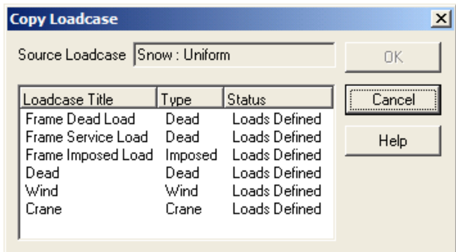
You can copy details from one loadcase to another or from one frame member to any other.

To copy a frame loadcase

1. Before you can copy the details from one ***Frame Loadcase*** to another you must first create the loadcase that is to receive the copied details. Use **Add Dead…**, **Add Imposed…**, **Add Wind…** or **Add Crane…**, give the new loadcase a title and then click **OK**.

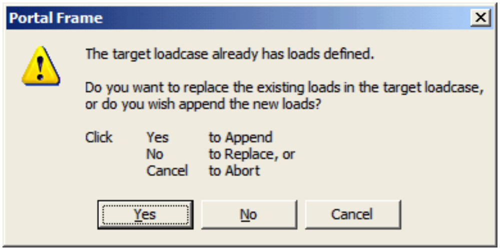
2. Select the source ***Frame Loadcase*** from the list of available loadcases.

3. Click **Copy…** to see the ***Copy Loadcase*** dialog which confirms the source loadcase and shows a list of target loadcases.



4. Select the target loadcase and click **OK**.

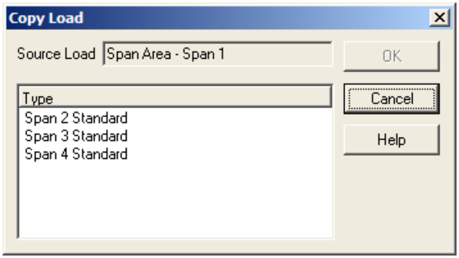
5. If your target loadcase contains loads, then ***Portal Frame*** asks if you want to add the copied loads to the existing ones in the loadcase, or if you want to replace the existing loads with the copied ones. You can also cancel the copy process if you have made an error.



To copy an individual load between members

Once you have defined a load on one member you can then copy its details to the other members of your frame.

1. Select the load in the ***Frame ... Loadcase*** property sheet and then click **Copy…** to see the ***Copy Load*** dialog. This shows a list of items to which you can copy the selected load.



2. Choose the targets for the copied load and then click **OK** to perform the copy.

Note You can select a range of items if necessary.

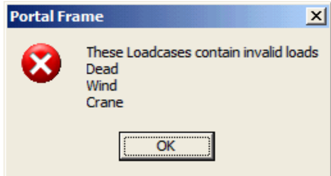
Caution If the load that you are copying requires distance information to position it along the member, and the target member is shorter than the source, then copy will always generate the load but may change the distances as follows:

- For a load that only requires one distance (e.g. a point load or a point moment) or for the end distance of a load that requires two distances (e.g. an area load or a line load) and where the copied distance is beyond the end of the member the distance will be reduced to the distance of the end of the member.
- For the start distance of a load that requires two distances (e.g. an area load or a line load) and where the copied distance would be beyond the end of the member the start distance will be set to zero.

Understanding loadcase validation

In the normal course of events you will find that you can not create loading which is in error in any way. However, if you change the layout of the frame after loads have been defined, then your changes might affect the correctness of the loading. Whenever you change the geometry of a frame in any way ***Portal Frame*** automatically checks all your loading for errors.

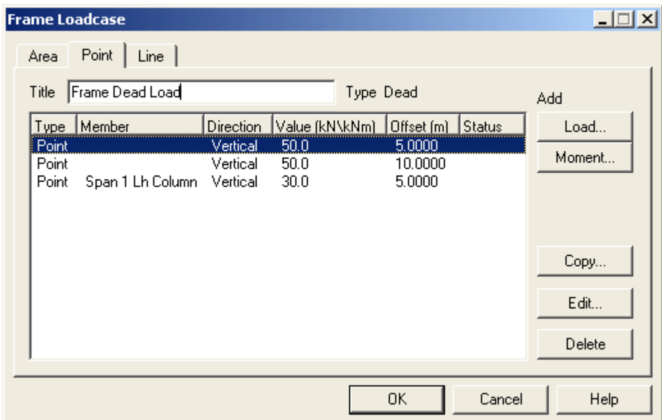
If erroneous information is found during this validation process, then you will be informed that a particular loadcase contains invalid loads.



To find out exactly which loads are invalid, you need to edit each of the invalid loadcases.

Help For further information *see* "To modify an existing frame loadcase" on page 297.

You can determine which loads in the loadcase are invalid since they will either have a **Status** of *Invalid*, or will not be associated with a member.



You must either delete the invalid loads, or edit them so that they are no longer invalid.

?

Help

For further information on editing and deleting loads ***see***:

- *"To modify area loads"* on page 313,
- *"To delete area loads"* on page 314,
- *"To modify point loads"* on page 320,
- *"To delete point loads"* on page 321,
- *"To modify line loads"* on page 330,
- *"To delete line loads"* on page 330,
- *"To modify wind loads"* on page 358,
- *"To delete wind loads"* on page 358,
- *"To modify crane loads"* on page 362,
- *"To delete crane loads"* on page 362.

22 Wind Load Generator[1]

The ***Wind Load Generator*** allows you to calculate the wind loading applied to your building in accordance with ***CP3 : Chapter V : Part 2 : September 1972*** or ***BS 6399 : Part 2 : 1997***.

Caution CP3 : Chapter V : Part 2 : 1972 has been withdrawn by the BSI. You should only use this code of practice for legacy designs.

Changing the wind code

You pick the wind code that you want to use as you use the ***Building Wizard*** to define your project.

Help For further information **see** *"Building Wizard"* on page 145.

You can change these details later if necessary, however if you change the design code, then you will have to re-define all your wind loading, since the current details will no longer be appropriate.

Help For further information **see** *"To change the design codes"* on page 159.

Changing the basic wind details

You define the basic wind details for the wind code that you pick as you use the ***Building Wizard*** to define your project.

Help For further information **see** *"Building Wizard"* on page 145.

1. *This is an additional plug-in module that you purchase separately to* ***Portal Frame****.*

You can change these basic details later if necessary and any wind loading you have created will be updated automatically.

Caution If you have modified any of the calculated loads, the values you have entered will be overwritten by the automatically recalculated ones.

Help For further information on the details for CP3 **see** *"To modify building wind load data"* on page 390 or for BS 6399 **see** *"To modify building wind load data"* on page 371.

Calculating the wind loading

You determine the wind conditions that are to be considered in conjunction with the other loading on the frame. From these conditions the ***Wind Load Generator*** calculates the actual wind load that the members of your frame have to carry, generating the appropriate loading automatically. Finally (if necessary) you can edit the load cases that have been generated to allow for any local effects that have not been considered by the ***Wind Load Generator***.

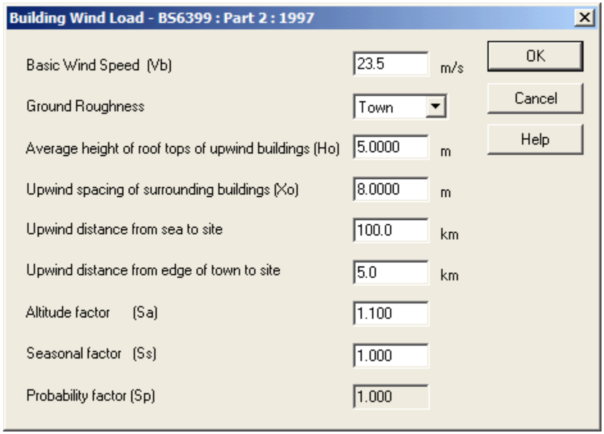
Wind Loading to BS 6399 Standard effective wind speeds Standard pressure coefficients

Calculation of wind loading to BS 6399 : Part 2 : 1997 is a two stage process:

- You define the building wind loading data (this is part of the ***Building Wizard***),
- You specify the details for the frame that you are defining.

To modify building wind load data

1. Select *Building/Wind Load…* to see the ***Building Wind Load*** dialog for BS 6399 Part 2 where you can define the building wind information.



2. Enter the value for the **Basic Wind Speed (Vb)**, taken from Figure 6 of BS 6399 : Part 2 : 1997 page 13.

3. Select the appropriate **Ground Roughness Category** for your site.

4. Enter the values for the **Average height of the roof tops of the upwind buildings (Ho)**, **Upwind spacing of surrounding buildings (X)**, **Upwind distance from sea to site**, **Upwind distance from edge of town to site**, **Altitude factor (Sa)**, **Seasonal factor (Ss)** and **Probability factor (Sp)**.

Note The calculated **Altitude factor** is correct for the altitude of the site in the *Building Definition* dialog ***provided that the topography is not considered significant***. For sites where ***topography is considered significant*** you will need to calculate the **Altitude factor** yourself and then enter the value directly.

Caution If you change the altitude of the site the value for the **Altitude Factor** is not changed automatically. You will need to enter any new value yourself.

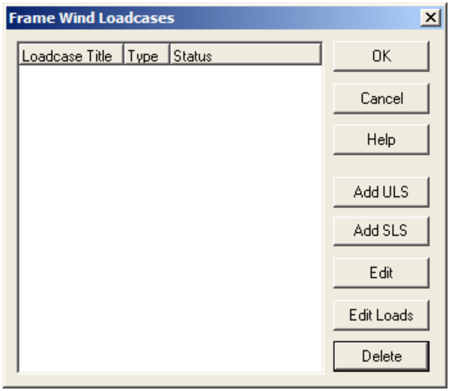
Help For further information on setting the building altitude using the *Building Definition* dialog ***see*** *“Building Wizard – Building Definition”* on page 147.

5. Once the information is correct click **OK** to define the wind load details.

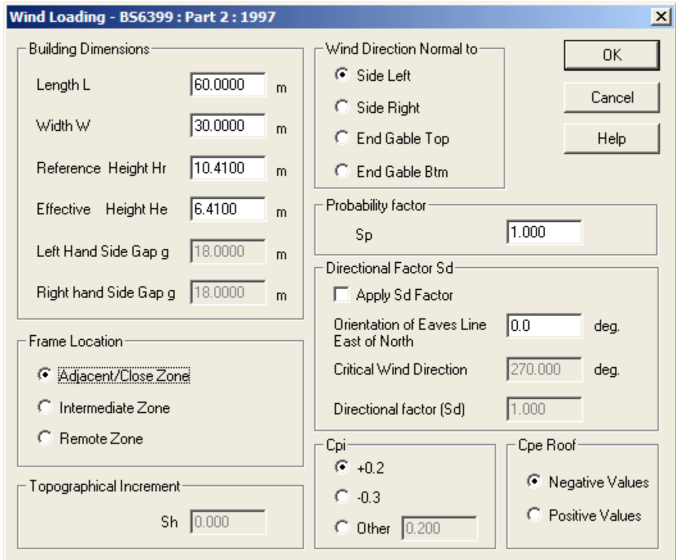
To define wind load details

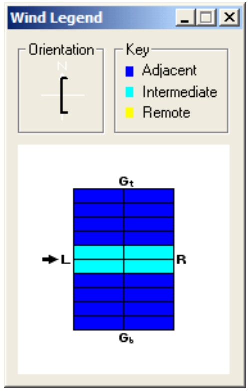
Once you have defined the basic wind details you can continue and define the particular wind cases appropriate to your building.

1. Select *Loading/Wind Loading…*



2. To create a new wind condition click **Add ULS** or **Add SLS** to add an ultimate limit state or serviceability limit state loadcase respectively. In either case you will see the ***BS 6399 Wind Loading*** and ***Wind Legend*** dialogs.





Note *The only difference between a ULS loadcase and a SLS one is in the factors which the* ***Wind Load Generator*** *uses to determine the pressure coefficients.*

Note *The building representation depends on the* **Wind Direction** *you pick.*

3. The basic **Building Dimensions** details are calculated from information that you have already defined. If the calculated values are not entirely accurate you can change them as necessary.

Note *These values are only used in the wind calculations and do not affect the geometry of your building.*

Note *The* ***Wind Load Generator*** *always takes the width of the building parallel to the direction of span of the frames. Conversely the length of the building is taken as the dimension perpendicular to this i.e. in the direction of the frame bay centres.*

4. If you are defining load on the gable ends of the building then once you have picked the load direction you will need to enter values for the **Left Hand Side Gap** and **Right Hand Side Gap** to allow the ***Wind Load Generator*** to take account of funnelling effects which increase the suction on either or both sides of the building when wind blows on the gables.

Note The initial values mean that no funnelling occurs. If you enter smaller values funnelling will increase to a maximum and then drop off as the gap becomes too narrow for turbulence.

Note The *Wind Load Generator*:

- always takes the width of the building parallel to the direction of span of the frames. Conversely the length of the building is taken as the dimension perpendicular to this i.e. in the direction of the frame bay centres.
- always takes the height of the frame as the maximum height of any apex or eaves point, including any parapets that you may have defined.

5. Pick the face to which the **Wind Direction** is **Normal** and pick the appropriate **Frame Location** as detailed below.

Wind blowing on side of building - For this case the **Frame Location** indicates whether or not the frame is close to or away from the leading corners of the building. *Adjacent/Close Zone* and *Remote Zone* indicate that the particular frame is near to a leading corner, *Intermediate Zone* indicates that it is not.

Wind blowing on end of building - For this condition the **Frame Location** indicates how close the current frame is to the windward gable of the building. *Adjacent/Close Zone* means that the frame is close to the gable end and will pick up the highest external pressure coefficients (usually 3).

Intermediate Zone indicates that the frame picks up the lower coefficients that apply to the next roof zone which is immediately behind the highest pressure zone. *Remote Zone* indicates that the frame picks up the lowest pressure coefficients that apply to any remaining area of roof beyond that covered by the *Intermediate Zone*.

Help For further information ***see***:

- *"Pressure zones for flat roofs"* on page 891,
- *"Pressure zones for monopitch roofs of rectangular clad buildings"* on page 892,
- *"Pressure zones for duopitch roofs"* on page 895,
- *"Pressure zones for hipped/flat top roofs"* on page 898,
- *"Pressure zones for Mansard portals"* on page 900,
- *"Pressure zones for Multi-bay portals"* on page 901.

6. If you know the orientation of your building, and want to take advantage of the **Sd** (basic wind speed adjustment by direction) factor, then tick **Apply Sd factor** and then specify the **Direction** of the eaves line as an angle east of north. The ***Wind Load Generator*** will calculate and use the appropriate factor.

7. Pick the ***Internal Pressure Coefficients*** that apply in this wind condition. You can either use one of the standard factors (which assume that there is a negligible probability that a dominant opening exists during a storm), or you can enter a factor of your own choosing.

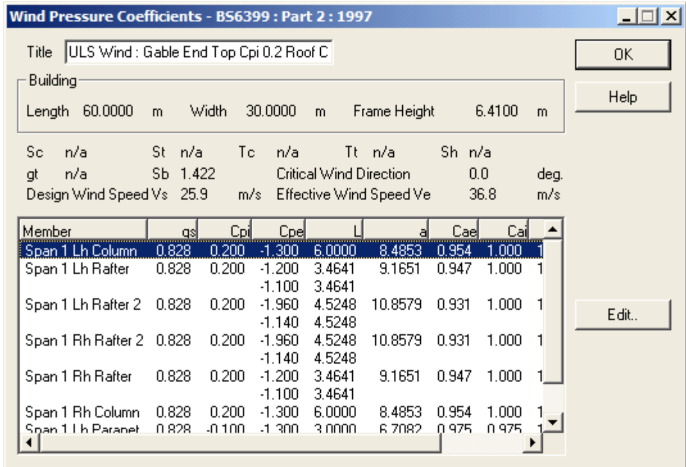
Note The **Cpi** value applies to the entire structure. The *Wind Load Generator* will not apply wind loads to internal walls. You can of course add these in yourself by manually editing the Wind loadcases.

8. For the majority of roof types there are some combinations of wind direction and roof pitch which cause the wind to cycle rapidly between pressure and suction over the whole or part of the roof. Both conditions must be investigated to determine that which is critical. Pick **Negative values** to investigate the negative (suction) values from the appropriate tables and wind directions, or **Positive values** to investigate the positive (pressure) effects.

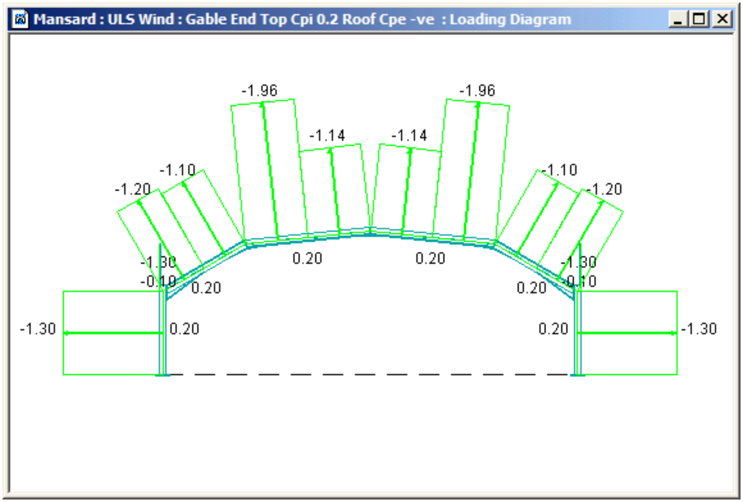
Note For some conditions there are ***only*** positive or negative values for a particular wind direction or roof area. In such cases the values in the table will be returned even though they are of the opposite sign to that which you selected. ***The choice that you make here is only used when there are two possible values of opposite sign***.

Caution Although both positive and negative values are given for most roof types and slopes there are some cases where particular values are not defined in the code (for instance a monopitch roof with a slope between 5° and just under 15° has no positive values). In all such cases the *Wind Load Generator* will interpolate between the lowest figure included in the table (in the case above the value for 15°) and zero. You will need either to check and ratify the use of these values, or to delete the automatically generated wind coefficients for such conditions based on your own engineering judgement.

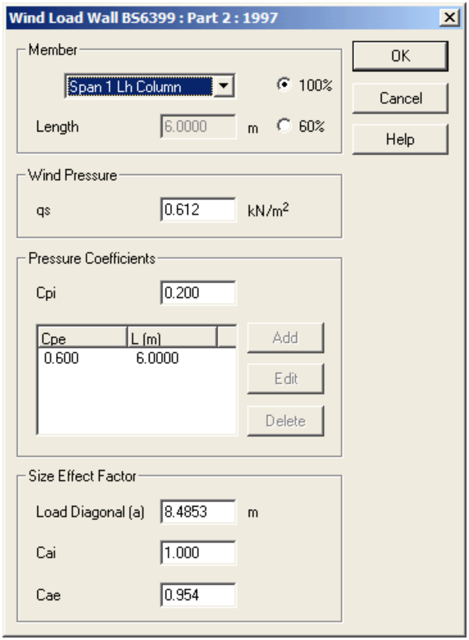
9. Once your settings are correct click **OK** to generate the coefficients and see the ***Wind Pressure Coefficients*** dialog.

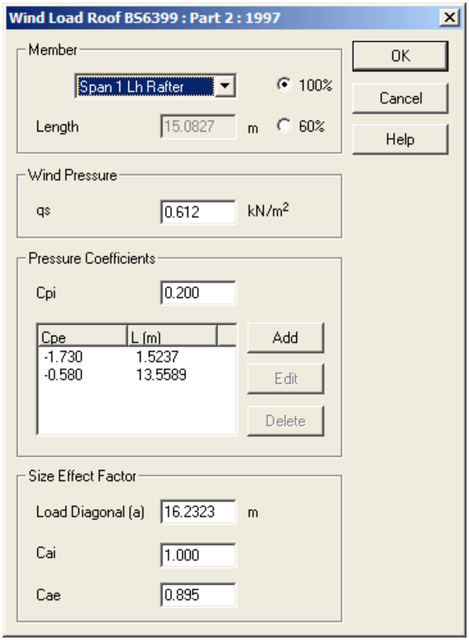


10. Once you click **OK** on this dialog you will see a graphical representation of the loading.



11. If the listed coefficients are satisfactory then click **OK** to complete your wind loadcase definition. If you need to modify any of the coefficients highlight any of the lines relating to the member whose coefficient you want to change and click **Edit…** to see the ***Wind Load*** dialog.





BS 6399 requires that you should make allowance for the effects of asymmetric loading (clause 2.1.3.7). Pick the appropriate **Percentage for asymmetric loads** for the current member.

Note BS 6399 Clause 2.1.3.7 stipulates that, for suction loads only, asymmetric wind loading must be considered. The asymmetric load is stipulated to be one member of the frame at a time taking 60% of the normal full suction load. The *Wind Load Generator* applies the 60% rule to the entire member, thus if you have a member which has both pressure and suction coefficients you will need to alter the coefficients yourself, rather than using the automatic reduction facility.

Note When you have set the 60% option for one member of your frame you will not be able to select it for any other member. You must first reset the original member to 100% and then set the 60% value for the new one.

If necessary enter a more exact value for the ***Wind Pressure***.

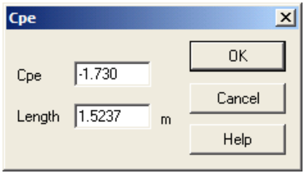
Note The *Wind Load Generator* automatically calculates the dynamic wind pressure from your building details, however it uses conservative values for some factors. If you perform your own calculations you can enter your calculated value here.

Enter the value for the **Internal pressure coefficient (Cpi)** if the value for this member is different to that for the entire building.

Enter the value for the **External pressure coefficients (Cpe) by zone** for each zone on your member.

Note The *Wind Load Generator* calculates zones of external pressure for each member of the frame. You can edit the details for any zone or add and delete zones to model the effects of wind on your structure in more detail.

Select the line relating to the zone whose C_{pe} value you want to change and then click **Edit** to see the ***Cpe*** dialog.



Enter the Cpe coefficient and length for this zone and then click **OK** to return to the ***Wind Load*** dialog.

The external diagonal dimension for a frame should be based on the effective width of the bay, and not on the overall length of the building. The value of ***a*** used by the ***Wind Load Generator*** is therefore calculated by the expression

$$a = \sqrt{\text{length of member}^2 + \text{effective width of panel}^2}$$

If you would prefer to use a different dimension, then enter the value for **Diagonal dimension a - (external)** directly.

Enter the value for the **Size effect factor (internal) Cai** if you have calculated your own value. The default value of 1.00 is normally conservative, however in some cases this might not be the case. You are advised to check this value.

E

Example *If you have suctions on the side of a building combined with internal pressure, then a calculated value of C_{AI} might give a more onerous result than the default of 1.00.*

If you have calculated your own value for the Size effect factor (external) then enter it. The default value is based on the external load diagonal detailed above.

Make any changes that are necessary and then click **OK** to return to the ***Wind Pressure Coefficients*** dialog.

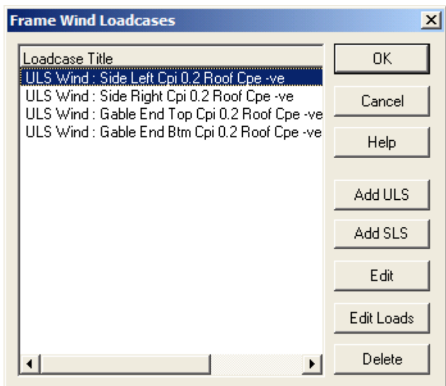
12. Make any other changes to the coefficients as necessary and then click **OK** to generate the wind loadcase.

Caution *If you edit the wind load using the Wind Load Generator, then any changed coefficients will be reset to their calculated values as part of the calculation process. To avoid this you should make modifications using Frame / Loading… rather than Frame / Wind Loading…*

To edit wind load details

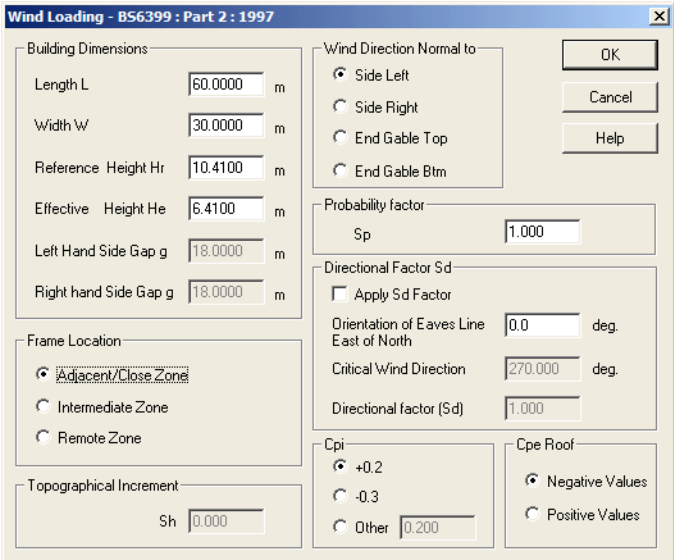
You can edit the wind loads that apply to the frame either by changing the details used to calculate the wind loading or by changing the calculated loads.

1. Select *Frame/Wind Loading…* to see the ***Frame Wind Loadcases*** dialog.

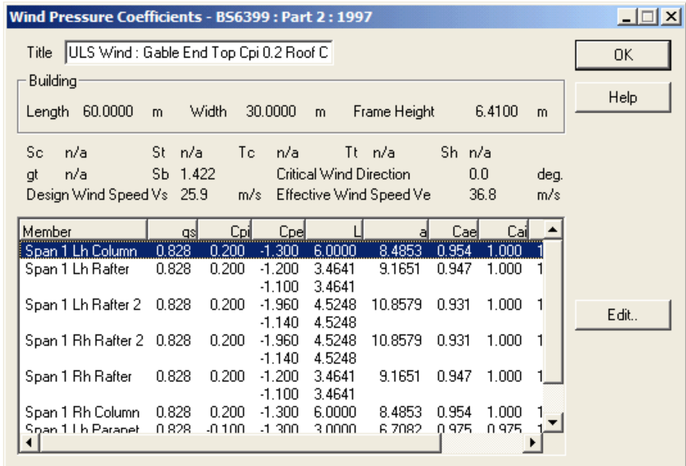


2. Pick the wind loadcase whose details you want to change.

3. If you want to change the details used to calculate the wind loads and calculate the loads again click **Edit** to see the ***Wind Loading*** dialog and make any changes following the procedure used to add a new wind load.



4. If you want to change the details for a specific wind pressure coefficient, then click **Edit Loads**. You will then see the pressure coefficients that have been calculated for the selected loadcase.



5. In either case make any changes that are necessary in the same way as you did when defining the original wind loadcase.

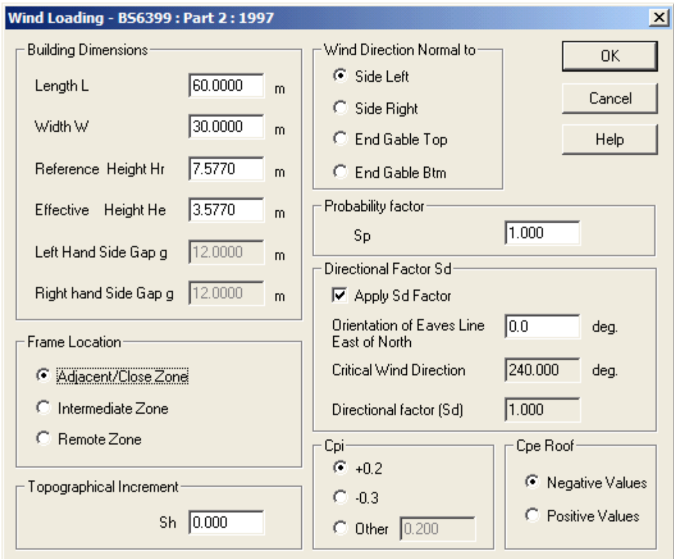
Help For details on editing a wind load using the same procedure as for defining a new wind load ***see*** *"To define wind load details"* on page 372

Wind Loading to BS 6399 Directional effective wind speeds Standard pressure coefficients

Calculation of wind loading to BS 6399 : Part 2 : 1997 with directional effective wind speeds and standard pressure coefficients is virtually identical to using the standard wind speeds. The differences are detailed in the following text.

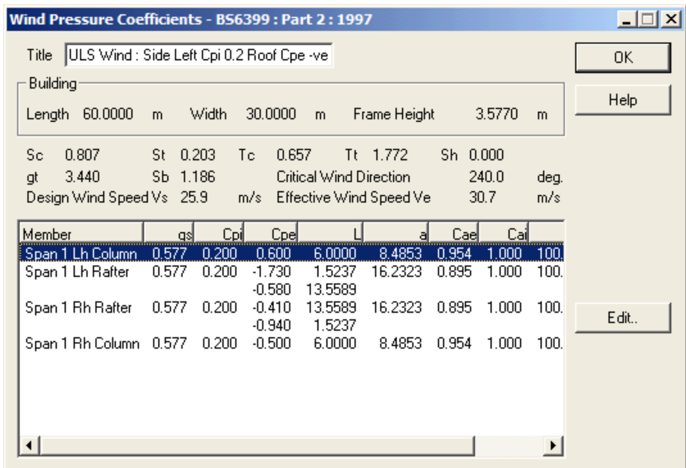
BS 6399 wind loading dialog

In order for the ***Wind Load Generator*** to calculate the directional wind speed and additional factor is required from you - the **Topographical Increment Sh**. ***The default value of 0.0 is not conservative***, you will need to provide calculations to justify this value. Since the ***Wind Load Generator*** does not have sufficient information to determine the appropriate value you must enter the appropriate value directly. The ***Wind Load Generator*** will use the value that you enter in the calculations for the directional effective wind speed.



The ***Wind Load Generator*** can calculate the other factors for the calculation of the effective wind speed from the known details for the frame. The fetch factor S_c and the turbulence factor S_t are obtained from Table 22, depending on the windward distance between the sea or other significant area of open water to your site.

If your current building is located within a town, then the fetch adjustment factor T_c and the turbulence adjustment factor T_t are obtained from Table 23. The gust peak factor g_t is obtained from Table 24. All these values are shown in the ***Wind Pressure Coefficients*** dialog for your convenience.



Wind Loading to CP3

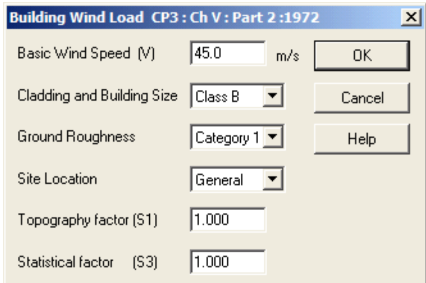
Calculation of wind loading to ***CP3 : Chapter V : Part 2 : September 1972*** is a two stage process:

- You define the building wind loading data (this is part of the ***Building Wizard***),
- You specify the details for the frame that you are defining.

Caution CP3 : Chapter V : Part 2 : 1972 has been withdrawn by the BSI. You should only use this code of practice for legacy designs.

To modify building wind load data

1. Select *Building/Wind Load…* to see the ***Building Wind Load*** dialog for CP3 where you can define the building wind information.



2. Enter the value for the **Basic Wind Speed (V)**, taken from the chart on CP3 : Chapter V page 8.

3. Select the appropriate **Cladding and Building Size Class**, **Ground Roughness Category** and **Site Location** from the lists. The tables below show the definitions given in CP3.

Class	*Description*
A	All units of cladding, glazing and roofing and their immediate fixings and individual members of unclad structures

Class	Description
B	All buildings and structures where neither the greatest horizontal dimension or greatest vertical dimension exceeds 50 m.
C	All buildings and structures whose greatest horizontal dimension or greatest vertical dimension exceeds 50 m.

Category	Description
1	Open country with no obstructions.
2	Open country with scattered windbreaks.
3	Country with many windbreaks; small towns; outskirts of large cities.
4	Surface with large and frequent obstructions, e.g. city centres.

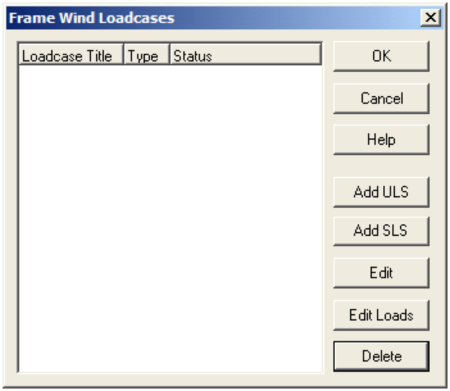
4. Enter the **Topography** and **Statistical** factors.

5. Once the information in the ***Building Wind Load*** dialog is correct click **OK** to define the wind load details.

To define wind load details

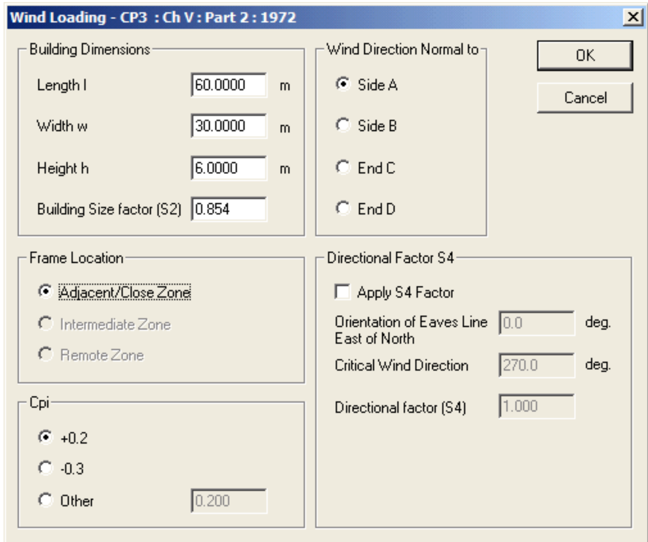
Once you have defined the basic wind details you can continue and define the particular wind cases appropriate to your building.

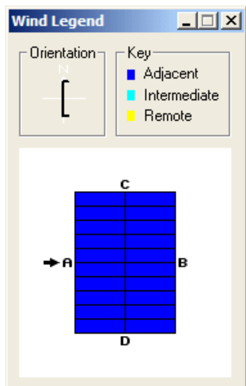
1. Select *Loading/Wind Loading…* to see the ***Frame Wind Loadcases*** dialog.



This dialog shows all the wind loadcases which you have defined using the ***Wind Load Generator***.

2. To create a new wind condition click **Add ULS** to see the ***CP3 Wind Loading*** and ***Wind Legend*** dialogs.





Note The building representation depends on the **Wind Direction** you pick.

3. The **Building Dimensions** details are calculated from information that you have already defined. If the calculated values are not entirely accurate you can change them as necessary.

Note These values are only used in the wind calculations and do not affect the geometry of your building.

Note The *Wind Load Generator* always takes the width of the building parallel to the direction of span of the frames. Conversely the length of the building is taken as the dimension perpendicular to this i.e. in the direction of the frame bay centres.

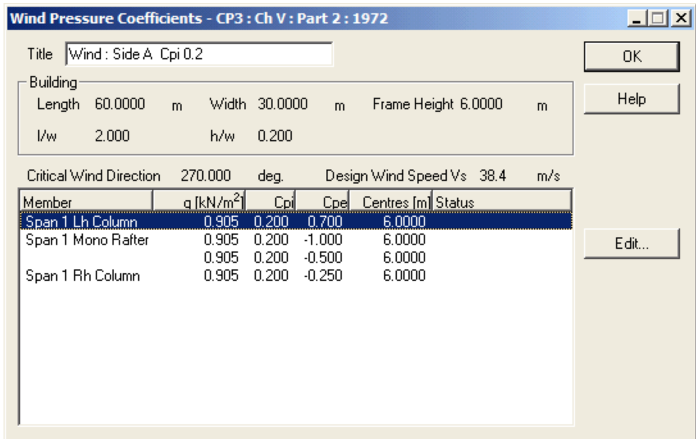
4. Pick the face to which the **Wind Direction** is **Normal**, and if the wind is blowing on the end of your structure pick the appropriate **Frame Location**. Pick **Adjacent/Close Zone** to use the higher coefficients or **Intermediate Zone** to use the lower ones. **Remote Zone** is not appropriate for CP3.

5. If you know the orientation of your building, and want to take advantage of the S_4 (basic wind speed adjustment by direction) factor, then tick **Apply S4 factor** and specify the **Direction** of the eaves line as an angle east of north. The *Wind Load Generator* will calculate and use the appropriate factor.

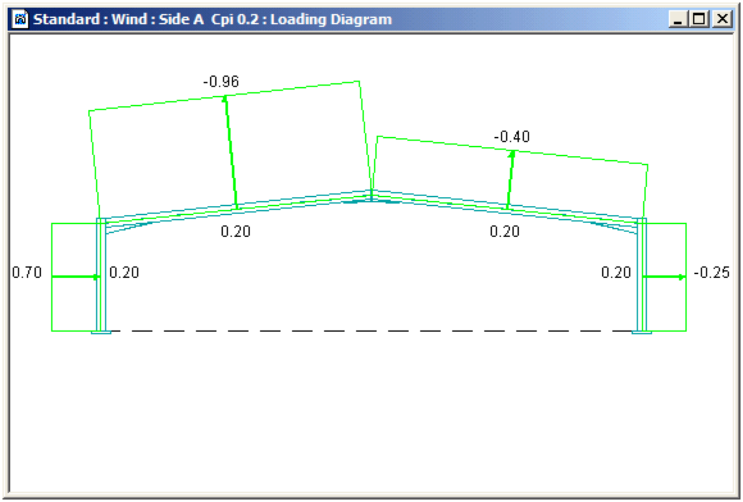
6. Finally define the *Internal Pressure Coefficients* for this wind condition. You can either use one of the standard factors (which assume that there is a negligible probability of a dominant opening during a storm) or enter a factor of your own choosing.

Note The **Cpi** value applies to the entire structure. The *Wind Load Generator* will not apply wind loads to internal walls. You can of course add these in yourself by manually editing the Wind loadcases.

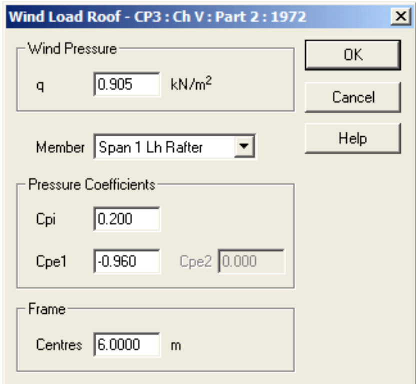
7. Once your settings are correct click **OK** to generate the coefficients and see the ***Wind Pressure Coefficients*** dialog.



When you click **OK** you will see the loads represented graphically.



8. If the listed coefficients are satisfactory click **OK** to complete your wind loadcase definition. If you need to modify any coefficients highlight the line relating to that coefficient and click **Edit…** to see the ***Wind Load*** dialog.



Make the changes and then click **OK**.

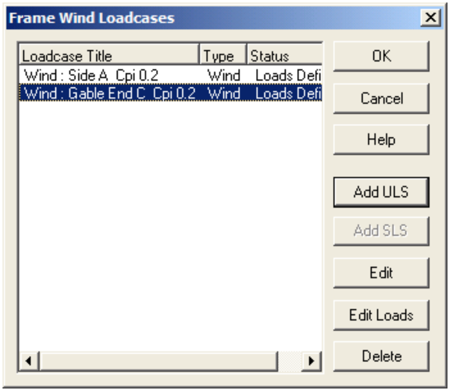
9. Once all the coefficients are correct click **OK** to generate the wind loadcase.

Caution If you edit the wind load using the *Wind Load Generator*, then any changed coefficients will be reset to their calculated values as part of the calculation process. To avoid this you should make modifications using *Frame / Loading…* rather than *Frame / Wind Loading…*

To edit wind load details

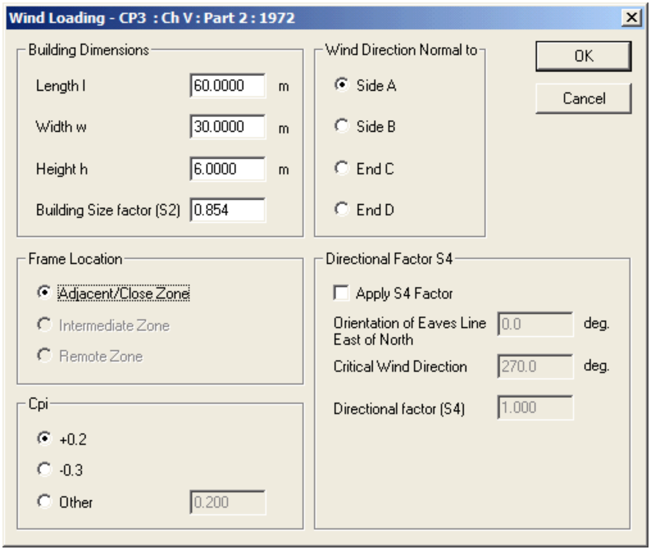
You can edit the wind loads that apply to the frame either by changing the details used to calculate the wind loading or by changing the calculated loads.

1. Select *Frame/Wind Loading…* to see the ***Frame Wind Loadcases*** dialog.

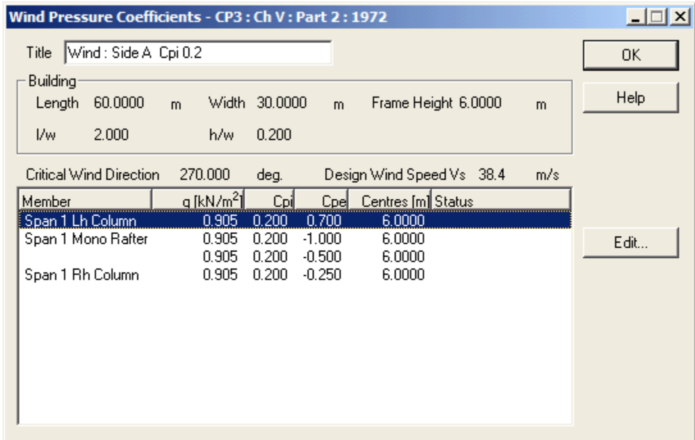


2. Pick the wind loadcase whose details you want to change.

3. If you want to change the details used to calculate the wind loads and calculate the loads again click **Edit** to see the ***Wind Loading*** dialog and make any changes following the procedure used to add a new wind load.



4. If you want to change the details for a specific wind pressure coefficient, then click **Edit Loads**. You will then see the pressure coefficients that have been calculated for the selected loadcase.



5. In either case make any changes that are necessary in the same way as you did when defining the original wind loadcase.

Help *For details on editing a wind load using the same procedure as for defining a new wind load* ***see*** *"To define wind load details" on page 391*

23 Snow Load Generator[1]

The ***Snow Load Generator*** allows you to calculate the snow loading applied to your building in accordance with ***BS 6399 : Part 3 : 1988***.

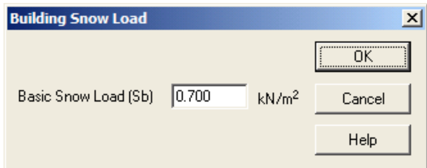
Snow Loading to BS 6399 : Part 3 : 1988

Calculation of snow loading to BS 6399 : Part 3 : 1988 is a two stage process:

- You define the building snow loading data (this is a part of the ***Building Wizard***,
- You specify the particular snow conditions for your frame.

To modify building snow load data

1. Select *Building/Snow Load…* to see the ***Building Snow Load*** dialog.



2. Enter the value for the **Basic Snow Load (Sb)**, taken from the chart on BS 6399 : Part 3 : 1988 page 5.

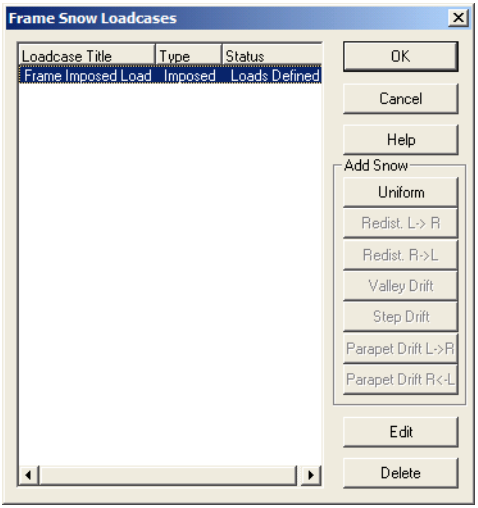
3. Once the information in the ***Building Snow Load*** dialog is correct click **OK** to define the snow load details.

To define snow load details

Once you have defined the basic snow load you can continue to define the particular conditions appropriate to your frame.

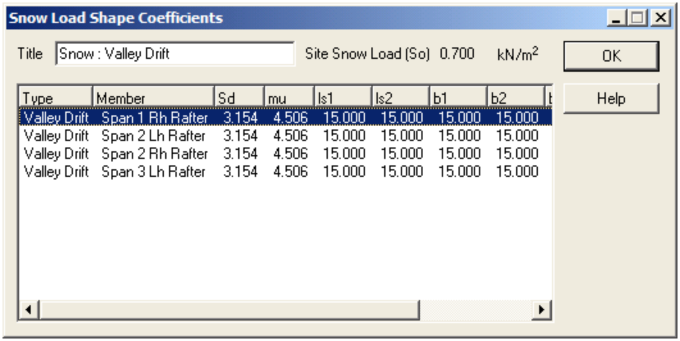
1. *This is an additional plug-in module that you purchase separately to* ***Portal Frame****.*

1. Select *Loading/Snow Loading…* to see the ***Frame Snow Loadcases*** dialog.

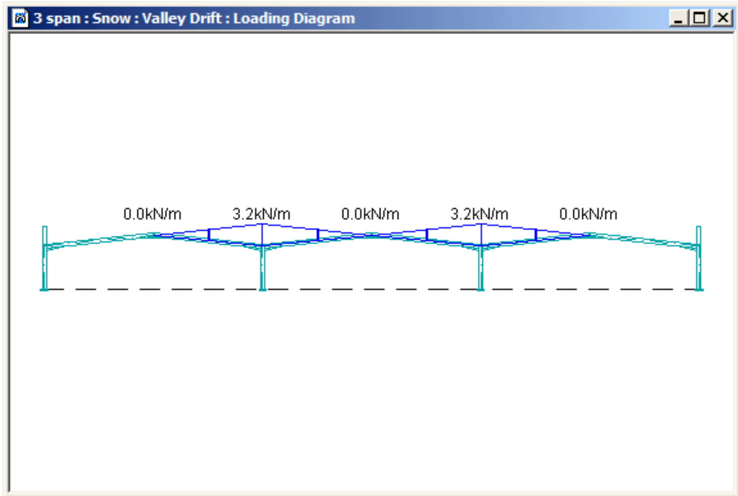


This dialog shows all the loadcases which you have defined using the ***Snow Load Generator***.

2. To create a new snow loading condition click **Uniform**, **Redist. L->R**, **Redist. R->L**, **Valley Drift**, **Step Drift**, **Parapet Drift L->R** or **Parapet Drift L->R** to create the type of snow load that you require and see the ***Snow Load Shape Coefficients*** dialog.



When you **OK** the list of coefficients you will see a diagram which gives you a graphical representation of the loading that has been calculated.



3. Once you have reviewed the coefficients click **OK** to generate the snow loadcase.

Note If you pick a snow load condition that involves a redistribution of the snow by drifting, then the *Snow Load Generator* automatically includes the partial safety factor of 1.05 stipulated by the code.

24 Design Combinations in Portal Frame

Once you have defined your loadcases you then assemble these into a series of combinations for design.

For each design combination you choose the loadcases which are to be included and the factors that are to be used. The loads and the factors are used to determine the loads that the beam must withstand at the ***Ultimate Limit State***. The same group of loadcases will be checked at the ***Serviceability Limit State*** to ensure that the beam behaves acceptably under working conditions.

?

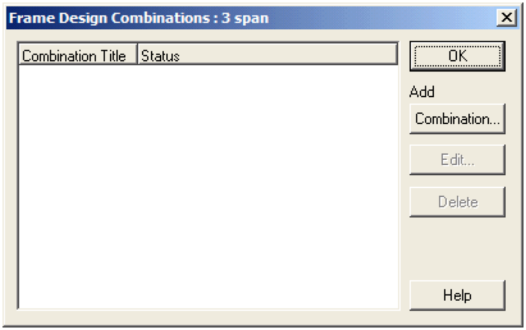
Help For further ***see*** *"Frame Loading in Portal Frame"* on page 290, *"Wind Load Generator"* on page 369 and *"Snow Load Generator"* on page 401.

Understanding design combinations

The ***Frame Design Combinations*** dialog allows you to specify those frame loadcases that you want to include in a particular combination, and the factors that are to be applied for the ultimate limit state and serviceability limit state conditions.

To define design combinations

1. Select *Loading/Combinations…*

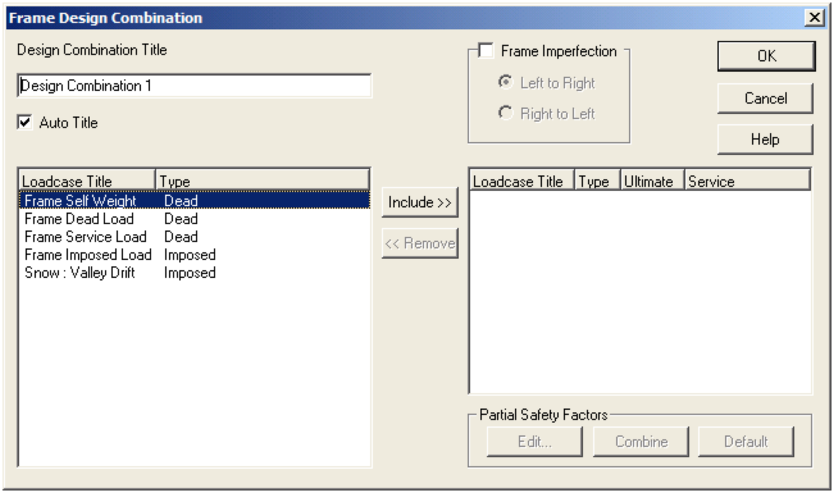


This dialog lists all current design combinations and allows you to handle them.

2. Once you have defined your design combinations click **OK** to return to the graphical display and proceed further.

To add a new frame design combination

1. Select **Add Combination…** to see the ***Frame Design Combination*** dialog.

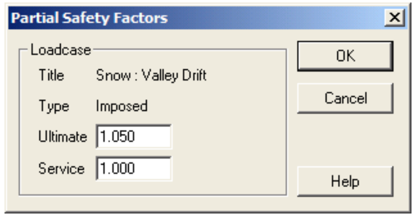


2. A numbered **Combination Title** is defaulted, but you can change this to a more meaningful description. However each combination title must be unique. You can also tick **Auto Title** to automatically generate a title as you add loadcases to the combination.

3. If you want to take account of frame imperfections in this combination ensure that **Frame Imperfection** is ticked and then pick the direction that you want to consider for the imperfection calculations.

4. Set the loadcases you want to include in this design combination. Select a loadcase that you want to include from the left hand list and then click **Include>>** to move it to the right hand one. You will see that the factors are automatically set depending on the types of loadcase in your combination.

5. If you want to change the factors then pick an included loadcase and then click **Edit…** to see the ***Partial Safety Factors*** dialog.



Enter the factors and click **OK** to return to the ***Frame Design Combination*** dialog.

Note *To set the factors to their default value for the loadcase types of load they contain (i.e. not taking account of the other loadcases in the combination) pick the line for an included loadcase and click* **Default**.

Note *To set the factors to the original default (taking into account the types of loadcase in the combination) pick the line for an included loadcase and then click* **Combine**.

6. Continue adding loadcases until your design combination is complete.

7. Once your design combination details are complete click **OK** to store it and return to the ***Frame Design Combinations*** dialog.

To edit a design combination

1. Select the design combination that you want to edit and click **Edit…** to see the details of that combination.

2. Make your changes and then click **OK** to return to the ***Frame Design Combinations*** dialog.

To delete an existing design combination

1. Select the design combination that you want to delete and click **Delete**. The combination will be deleted.

25 Manipulating frames

You can add new frames to your project, copying and modifying existing ones or creating new ones from scratch.

You can also copy details between frames, and choose the particular information that you want to copy. If your project contains frames that you no longer require, then you can delete these. You can also choose to design or check all the frames in a project as a single process rather than individually.

To create a new frame

1. Select *Frame/New* to see the ***Frame Definition*** dialog. You then proceed to define the new frame in the same way as for the first frame in the project.

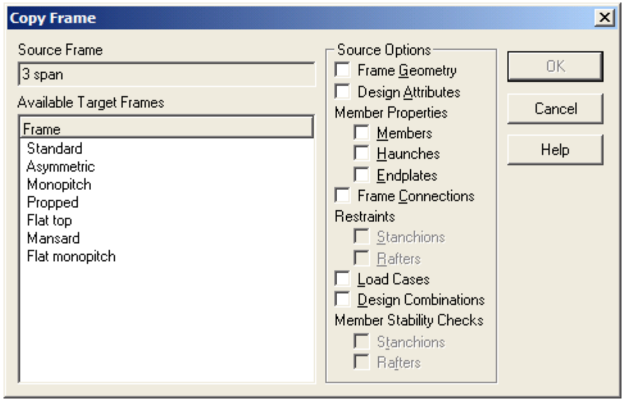
To copy an existing frame

1. First ensure that the frame that you want to copy is active and then select *Frame/Copy*. The frame is copied and will become the active one, ready for modification.

To copy information from one frame to another

When you have more than one frame in your project you can copy details between frames before making modifications and then designing or checking them.

1. First ensure that the frame that you want to copy is active and then select *Frame/Copy Attributes…* to see the **Copy Frame** dialog.

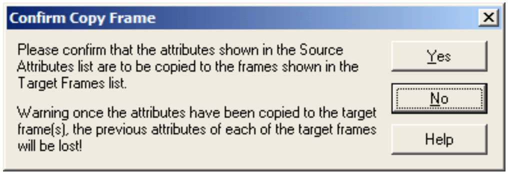


2. From the list of **Available Target Frames** pick the frames that are to receive details from the *Source Frame*.

3. Now set the information that you want to copy by ticking the appropriate boxes.

Note *If a box is dimmed then the source frame has no information of this type.*

4. Once your selections are complete click **OK** to perform the copy. You will see the ***Confirm Copy Frame*** dialog.

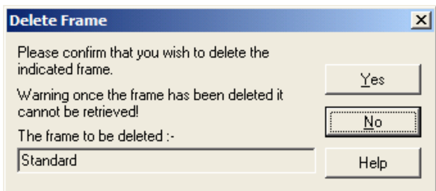


5. Take note of the warning. If you still want to perform the copy click **Yes**.

Caution If you click **Yes**, then the details for the target frames are replaced by those from the source frame and they cannot be recovered. ***Please take care when you use this feature.***

To delete a frame

1. First ensure that the frame that you want to delete is active and then select *Frame/Delete* to start to remove it. You will see the ***Delete Frame*** warning dialog.



2. Take note of the warning. If this is the frame that you want to delete click **Yes**.

Caution If you click **Yes**, then the frame is deleted permanently and its details cannot be recovered. ***Please take care when you use this feature***.

Note You can not delete the only frame in your project.

26 Portal Frame Design Wizard

Portal frame design is a complex process involving a range of different requirements which determine whether or not a particular design is acceptable. These requirements depend on such things as the design standards used in a particular organisation or the requirements for a particular site or building.

Portal Frame allows you a great degree of flexibility in the design process, so that you can tailor a design to meet particular requirements. The *Portal Frame* ***Design Wizard*** allows you to set the various factors that control the design.

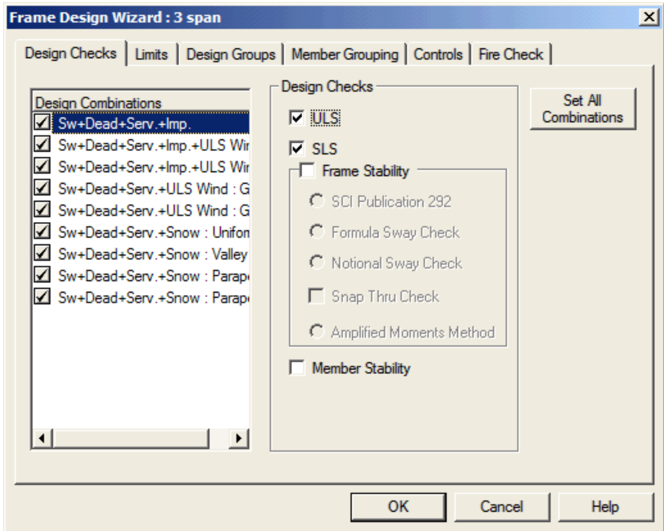
Understanding the design wizard

Using the design wizard you can choose:

- the design combinations and the ***design checks*** required for each combination,
- the ***limits*** on the deflections, rotations and plasticity of the frame,
- to set ***design groups*** and their properties i.e. the grade of steel, the sections that you want *Portal Frame* to use and any limits on the section sizes,
- to ***group*** members of the frame so that a single section size will be chosen for an entire group,
- to ***control*** which checks (other than strength) will result in an increase in section size,
- to control the ***fire checks*** that will be included for the frame.

To set design checks

1. Select *Design/Wizard…*



2. You will see a list of design combinations and area of design checks. If you do not want to perform any design checks for a combination then remove the tick to the left of the combination name. Otherwise select a particular design combination and pick the checks that you want to perform for that combination.

Tip **Set All Combinations** sets the current checks for all design combinations.

Help For further details of the checks ***see***:

- *"Member strength checks"* on page 850,
- *"Haunch strength checks"* on page 853,
- *"Frame stability checks"* on page 857,
- *"SCI publication P292"* on page 858,
- *"Notional Sway Check"* on page 860,
- *"Simplified Formula Sway Check"* on page 862,
- *"Snap-through stability checks"* on page 864,
- *"Amplified moments method check"* on page 864,
- *"In-plane buckling of individual members"* on page 865,
- *"Frame imperfections"* on page 866,
- *"Serviceability limit state"* on page 868,
- *"Member stability checks"* on page 871.

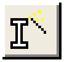
Connection Design Checks - Once you have completed the first ***Portal Frame*** design on your frame you will find that you can pick connections and automatically access ***Fastrak Moment Connections*** in order to design them.

Help For further details ***see*** *"Sharing details with Moment Connection Design"* on page 512,

Foundation Design Checks - Once you have completed the first ***Portal Frame*** design on your frame you will find that you can pick bases and automatically access ***Fastrak Bases*** in order to design them.

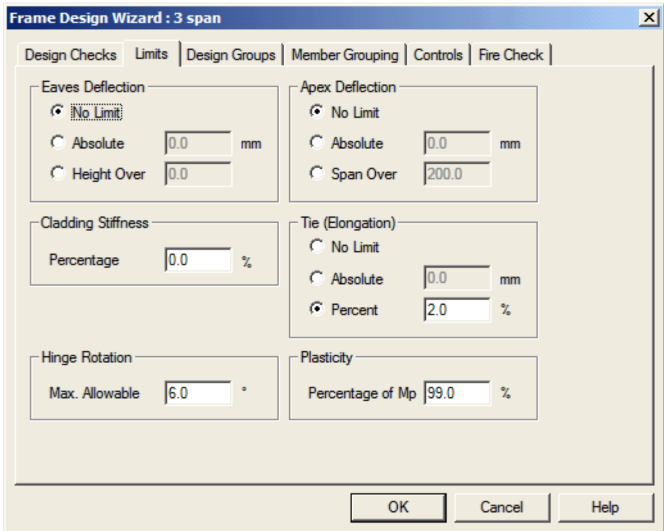
Help For further details ***see*** *"Sharing details with Column Base Design"* on page 516.

3. Pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.



To set limits

1. Select *Design/Wizard…* followed by the ***Limits*** tab.

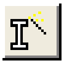


2. For the **Eaves Deflection**, **Apex Deflection**, **Sway Check** and **Tie Elongation** you can choose whether you want to apply a limit, and if so the type of limit that is to be applied. Make your choice and enter the appropriate value.

3. The **Hinge Rotation** is a required limit which, if exceeded, will stop the analysis. You should ensure that your steel sections can sustain this rotation and still act plastically.

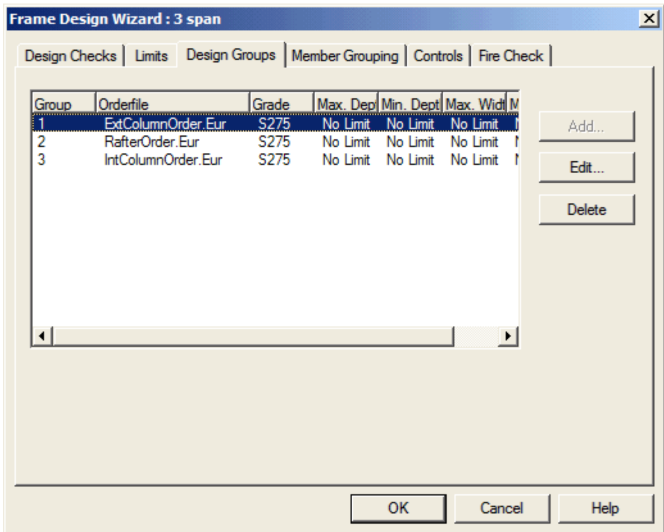
4. The **Percentage of Mp for Plasticity** is a required limit. It allows you to tell ***Portal Frame*** that points along members that reach this percentage of Mp are to be treated as though plasticity occurs at that point.

5. Pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.



To set design groups

1. Select *Design/Wizard...* followed by the ***Design Groups*** tab.



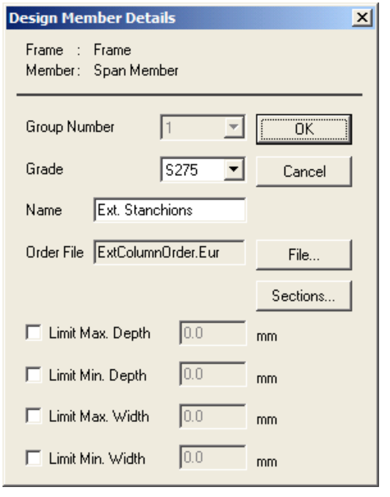
This shows all current groups that have been defined.

Note For a single span frame there are two automatically created groups one for the rafters and the other for the columns. For a multi-span frame an additional group is created for the internal columns.

Note *Grouping does not apply when you are checking a frame. Although this page is still shown it has no effect and you can not make any changes.*

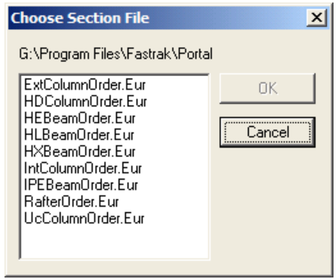
2. To add a new group click **Add…** The new group will take the next available number and you can use **Edit…** to set its details.

3. To change the details for a group select it and then click **Edit…** to see its ***Design Member Details*** dialog.



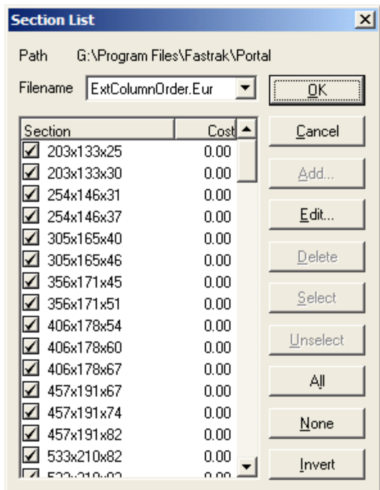
Select the **Grade** of steel that you want to use and enter the **Name** that you want to use for the group.

To change the order file that the group uses click **File…** to see a list of available files.



Select the file and click **OK**.

To change the sections within the order file that will be used for all designs click **Sections…**



If you remove the tick mark against a section, then that section will not be used during the design.

Caution The sections that are included/excluded are held for a particular order file. Thus if you exclude sections for this design they will remain excluded for all future designs for this and other frames until you include them again.

Once you have finalised the list of sections that you want to use click **OK** to register these.

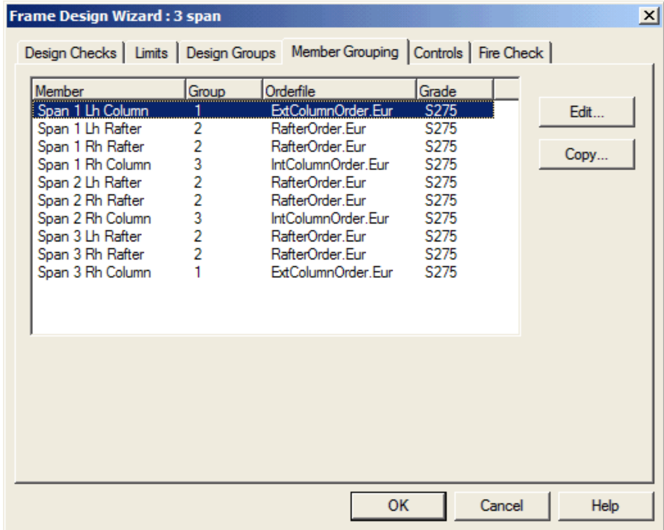
If you want to further limit the sections based on their physical size, then check that limit box and enter a value.

4. Once you have set the group details click **OK** to return to the ***Design Groups Page***.

5. Pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.

To set grouping

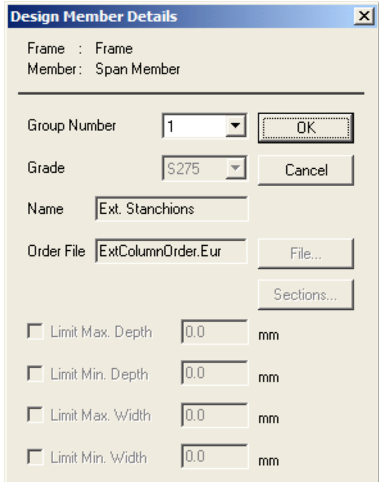
1. Select *Design/Wizard…* followed by the ***Member Grouping*** tab.



This shows all the members of the frame, their group and a brief resumé of some of the group data.

Note You must already have defined the groups that you want to use before you can allocate members to them.

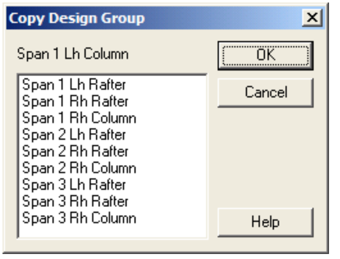
2. If you want to change a member's group then select it and click **Edit...** to see its ***Design Member Details*** dialog.



Note You can only set groups for members when you are designing the frame. If you are in check mode, then **Edit** and **Copy** on the main dialog are dimmed.

Select the group that you want to use from the **Group Number** list and click **OK** to return to the ***Member Grouping*** page.

3. To copy the group from one member to another select the member whose group you want to copy followed by **Copy…** to see the ***Copy Design Group*** dialog.

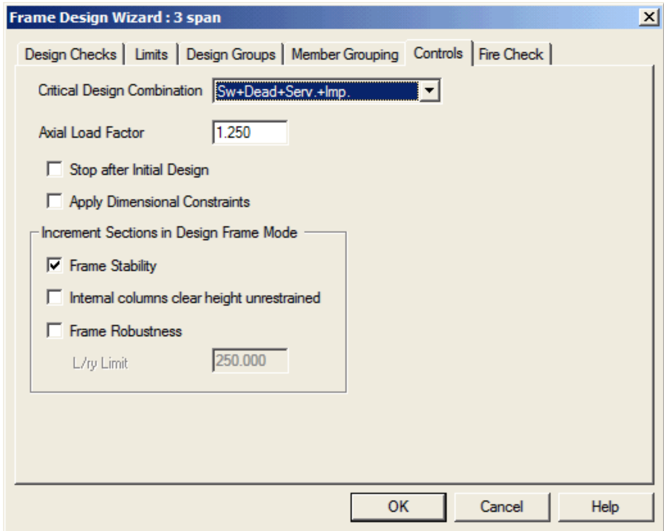


Select the target members followed by **OK** to perform the copy and return to the ***Member Grouping*** page.

4. Once you have set the correct groups for all members pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.

To set controls

1. Select *Design/Wizard…* followed by the ***Controls*** tab.



2. The ***Critical Design Combination*** is automatically set to the first one. If necessary choose an different one from the list.

Note You should choose the design combination that you think will control your frame's section sizes as the critical one. *Portal Frame* analyses and designs this design combination first and determines the sizes of the members needed to carry its forces and moments (this is known as the *Initial Design*). These sizes are then used to analyse and check all your other design combinations for your requested design checks.

Help For further information *see* *"To set design checks"* on page 415.

3. If you want to change the ***Axial Load Factor*** from the default value of 1.25, enter your own value.

Help For further information *see* *"Axial load factor"* on page 839.

4. If you want the design process to stop after the ***Initial Design*** so that you can confirm that the selected section sizes are acceptable then tick **Stop After Initial Design**. You can then tweak the section sizes to your satisfaction before checking out the remaining combinations.

5. If you have specified any ***dimensional constraints***, you can control whether these are to be applied or ignored. This allows you to see the effects of the constraints on the design solution.

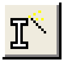
6. *Portal Frame* can ensure that the ***Frame Stability*** requirements in SCI Publication 292 are met. If you tick **Frame Stability SCI P292** *Portal Frame* will increase the member sizes to meet this requirement.

7. For the internal columns of multi-span frames the decisive factor in determining the section size is the need for member stability over the unrestrained length between the base and the underside of the eaves haunch. If you tick **Internal Column Clear Unrestrained Height** *Portal Frame* will ensure that your internal columns meet this criterion.

Help For further information *see* *"Member stability checks"* on page 871.

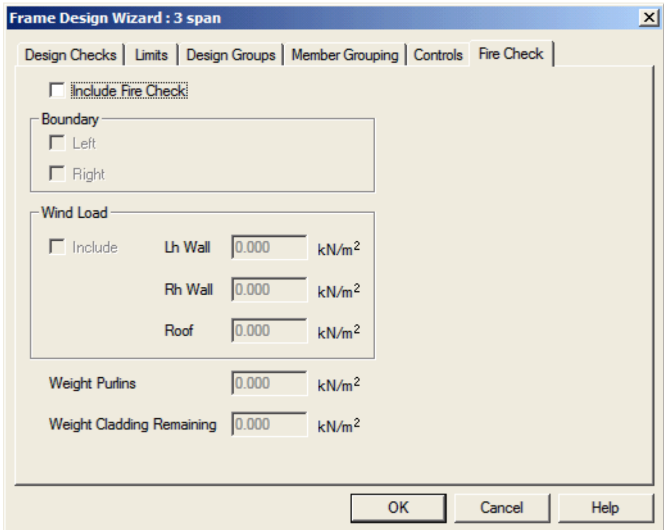
8. BS 5950-1: 2000 no longer requires that you meet minimum ***L/ry*** requirements. However in order to prevent ***Portal Frame*** choosing small sections for internal columns between main spans, or for the prop columns of propped spans you can tick **Frame Robustness** and then enter the ***L/ry*** limit that you want to use.

9. Once you have set the correct controls pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.



To set fire check requirements

1. Select *Design/Wizard...* followed by the ***Fire check*** tab.



2. Make the settings that you require and then pick another tab of the property sheet or click **OK** to save your ***Design Wizard*** settings.

27 Designing Frames in Portal Frame

Once you have defined your frame details you can proceed with the design.

Performing the design

The design time will depend on the number of design combinations and the design checks required for each combination.

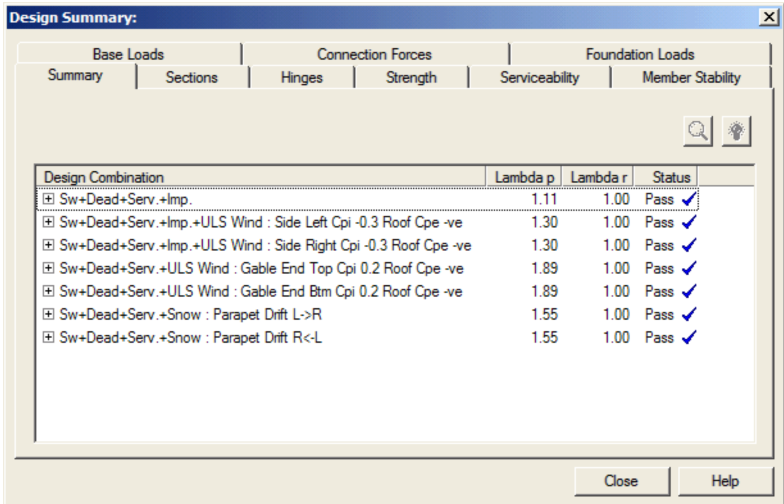
To perform the design

1. Select *Design/Frame*. This will validate the frame and start the design process.

Help For further information on *Frame Validation* **see** *"Understanding frame validation"* on page 430.

Note During this process you will see a series of dialogs indicating the part of the design that is being performed. This sequence may repeat several times which indicates that *Portal Frame* is changing the section sizes to achieve a satisfactory result.

Once the design is complete you will see a summary of the results. If you chose the ***Design Wizard*** option to pause the design process after the initial design, then the results will only be for the critical design combination, otherwise the results will be for all your design combinations.



?

Help For details on using the summary table ***see*** *"Design Results" on page 431.*

To design or check all frames in a project

1. Select *Building/Design all Frames…* Each frame in the building will be designed or checked in turn. The progress bar shows the current frame and the progress of the design.

Understanding frame validation

Whenever you perform a design ***Portal Frame*** first checks all the information that you have defined for the frame to ensure that it is complete and consistent. You cannot define a frame with inconsistencies, but if you subsequently make changes some problems may arise. If a problem is found, then you will see a warning message telling you what the problem is. Take note of the message, and then click **OK** to close the message box.

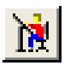
Return to the relevant ***Property Sheet*** and amend its details to remove the error. Once the error is corrected you can then proceed with the design of the frame.

28 Design Results

Portal Frame allows you to review the design results quickly, easily and intuitively by first showing you a table which summarises the design. You can identify and home in on particular areas of interest very quickly.

Viewing the design results

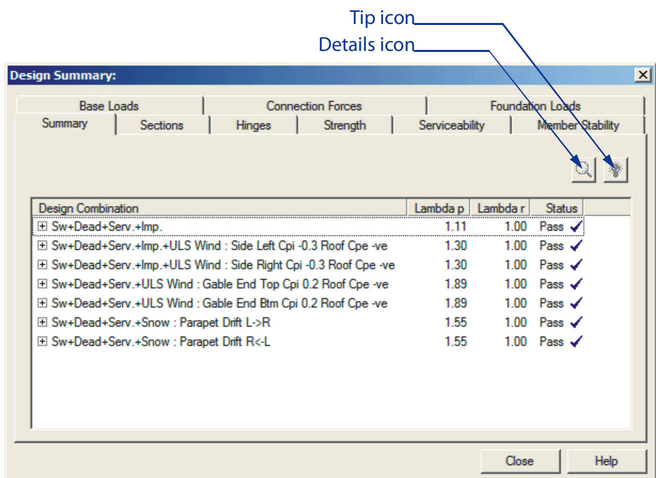
When you perform a design you will automatically see the design summary.



To view the design summary

1. Select *Design/Frame…*

   If your current design is valid you will see a summary immediately. Otherwise you might have to wait for a few moments while the design is performed.



2. The ***Summary*** page gives you an overview of the entire design. You can pick the other pages to view the results for a particular condition.

3. Once you have viewed the results that are of particular interest to yourself, click **Close** to return to the graphical display and to proceed with any remaining parts of your design.

To use the summaries effectively

To the left of many lines throughout the various pages of the summary you will see a plus icon. This indicates that there is more information available. Simply double click the line and a new dialog will open showing you the next level of detail that is available.

Tip You can also single click the line and then click the **Details** icon ( ).

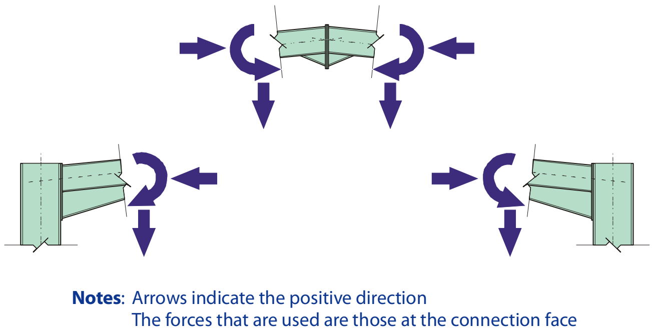
This next dialog might also have further information for some of its lines, therefore you can quickly delve down to the detail you need. Once you have reviewed the information you want simply close the dialogs until you return to the main summary page.

Additionally many checks in ***Portal Frame*** have a unique feature – the engineering tip. This gives you advice to help you modify your frame to achieve a satisfactory result. You can tell when an engineering tip is available since the line in the summary uses coloured text. To see the tip select that line and click the **Tip** icon ( ).

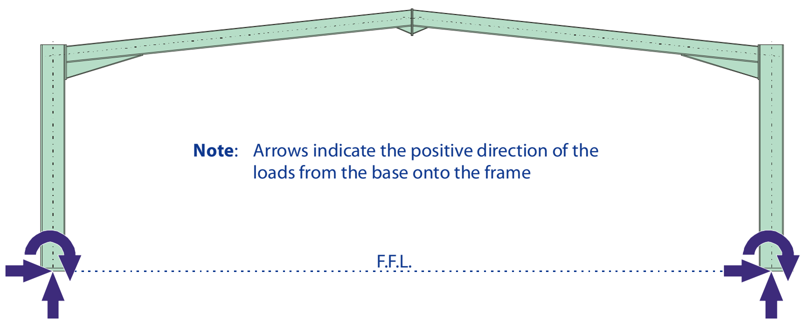
Results Sign Conventions

The following table shows the sign conventions for the results calculated by ***Portal Frame***.

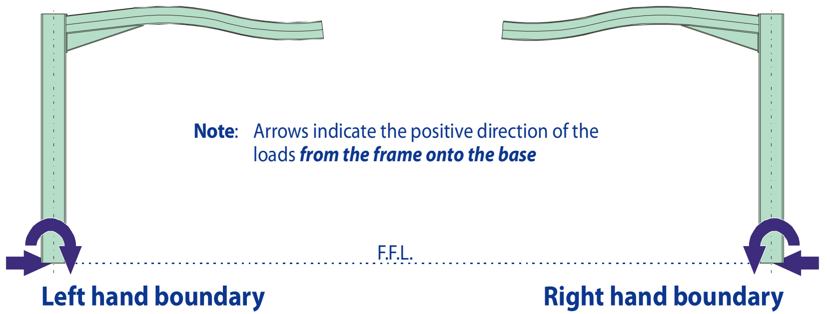
Sign conventions – Connection forces



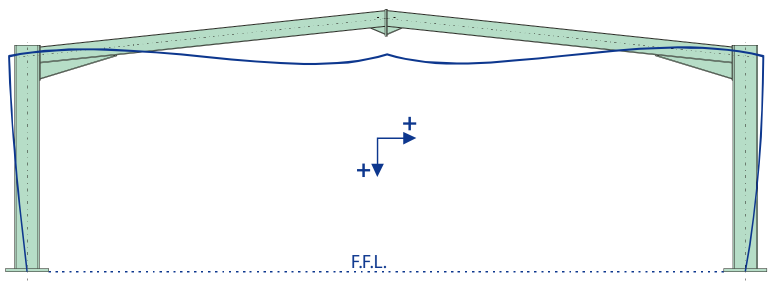
Sign conventions – Foundation loads



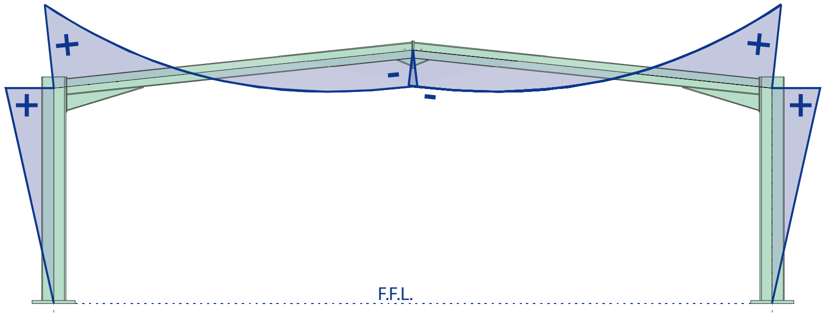
Sign conventions – Fire bases



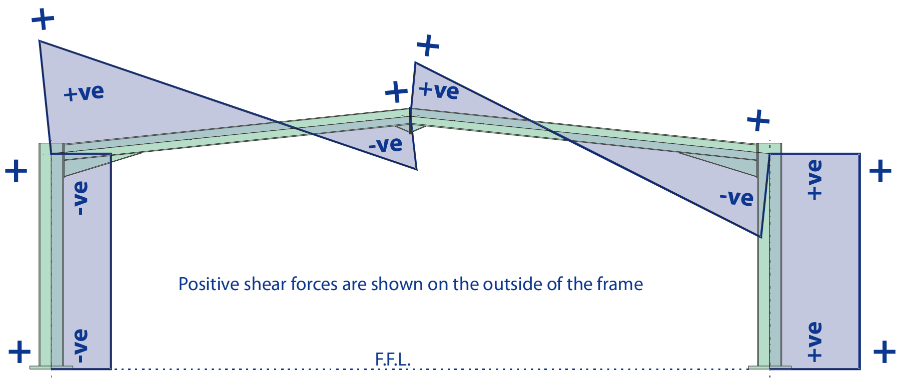
Sign conventions – Deflections



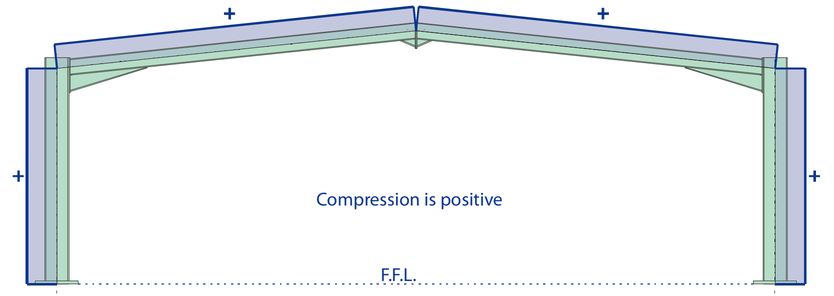
Sign conventions – Bending moments



Sign conventions – Shear forces



Sign conventions – Axial forces



29 Member Stability in Portal Frame

Once you have designed the members of your frame, you will need to check that they are locally stable. If your unrestrained member is not stable you will have to use the cladding rails and other ancillary steelwork to provide restraint in order to achieve stability.

In some cases the ancillary steelwork will provide restraint directly, without any modification. In other cases you will need to modify the ancillary steelwork to provide the type of restraint that is needed. In yet other cases you will have to add new steelwork to restrain the member adequately.

If a you need to restrain a particular member in many additional places in order for it to pass the member stability checks, then you might find that it is more economical to increase the member size thereby reducing the restraint requirements.

Checking member stability

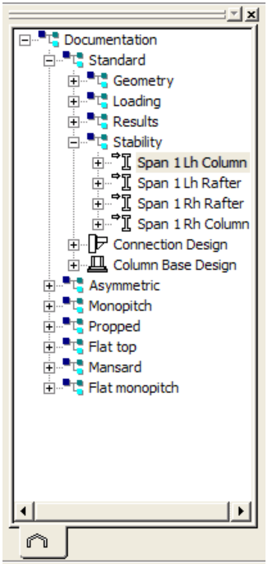
The usual procedure that you would adopt for undertaking member stability checks would be:

- start the member stability checking process by choosing the member you want to check from the ***Project Workspace***,
- choose the design combination under which you want to check the member by picking that combination from the ***Project Workspace***,
- add restraints to the member,
- define the checks that you want to perform along a particular length.
- if the member is not stable with these restraints and checks then modify the restraints and/or check types until the member is stable.

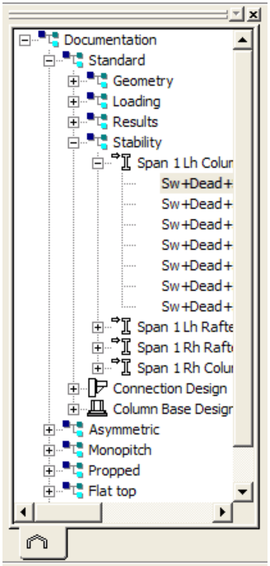
Each of these stages is covered in more detail in the following sections of this *User's Guide*.

To start checking member stability

1. Open the stability branch of the *Project Workspace* by clicking the plus sign to the side of the *Stability* item. This will show a list of all the members in the current frame which can be checked for local stability.



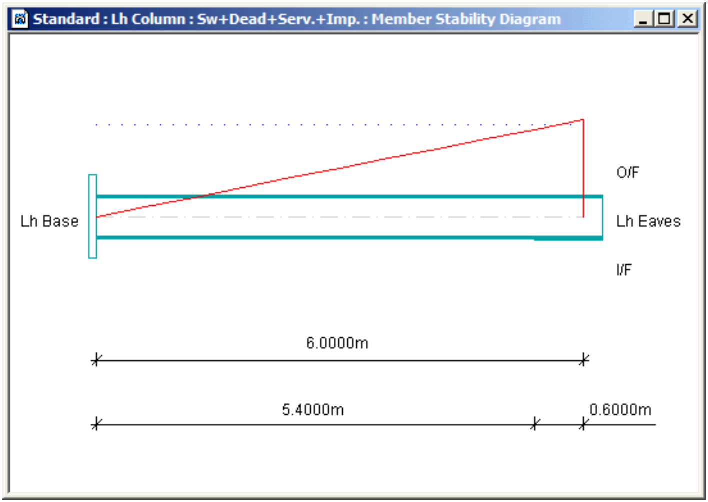
2. If you now click the plus sign to the left of the member you want to check, this will show a list of all the combinations under which the member can be checked.



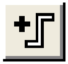
If you can not see the full title of the combination don't worry. You can

- scroll the ***Project Workspace*** to see more of the combination name,
- increase the width of the ***Project Workspace*** by dragging its resize bar,
- allow the pointer to rest over a particular combination whose details you can not see, and you will see a tool tip which gives the full combination name (Sw+Dead+Serv.+Imp.).

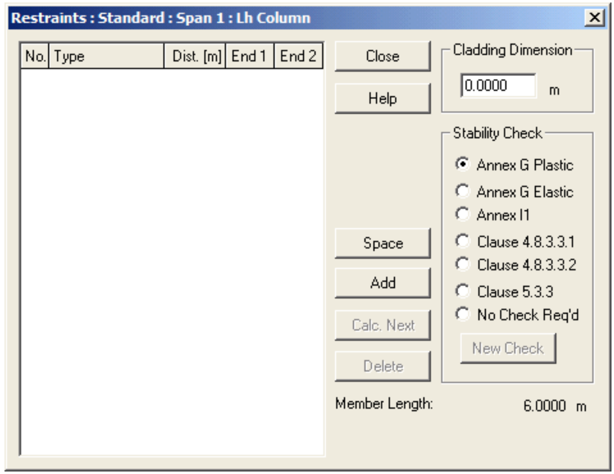
3. Double click the name of the combination under which you want to check member stability, and you will see a window which shows that member and combination.



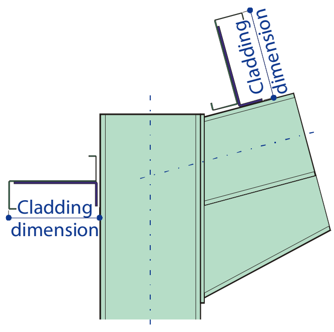
You will see that the *Design/Member Stability* menu options are now enabled as are the icons on the ***Stability*** toolbar are now activated.



4. Click **Member Restraints** or select *Design/Member Stability/Restraints…* to start adding restraints to the member.



5. Enter the **Cladding Dimension** – this is the distance from the face of the rafter/column to the inside face of the sheeting as indicated below.



Caution The **Cladding Dimension** that you specify here is used in the Annex G checks. The value used in these checks is set when you create the check. This allows you to change the value between checks if necessary. However you should note that you can not change this value in any existing calculations. If you need to do this you will have to delete and reinstate the affected checks.

Before you can define stability checks for a member you must define some restraints. You then check the stability of the member between these positions of restraint.

Automatically checking a member's stability

Portal Frame can automatically check the stability of a member with an array of restraints which you specify. ***Portal Frame*** applies your array of restraints to the member, and then proceeds to define the stability checks along the member based on the type of restraint and the bending moments. If a check passes, then ***Portal Frame*** moves on to check the next length of the member. If a particular check fails, then ***Portal Frame*** looks to see if an alternative check is applicable and if so applies this check to see if it passes. If ***Portal Frame*** exhausts the list of applicable checks without finding one that passes it shows the last check that it tried with a *Fail* status. this length as a fail length for the last check that was. Once the whole member has been checked ***Portal Frame*** shows the checks that have been applied and their status graphically.

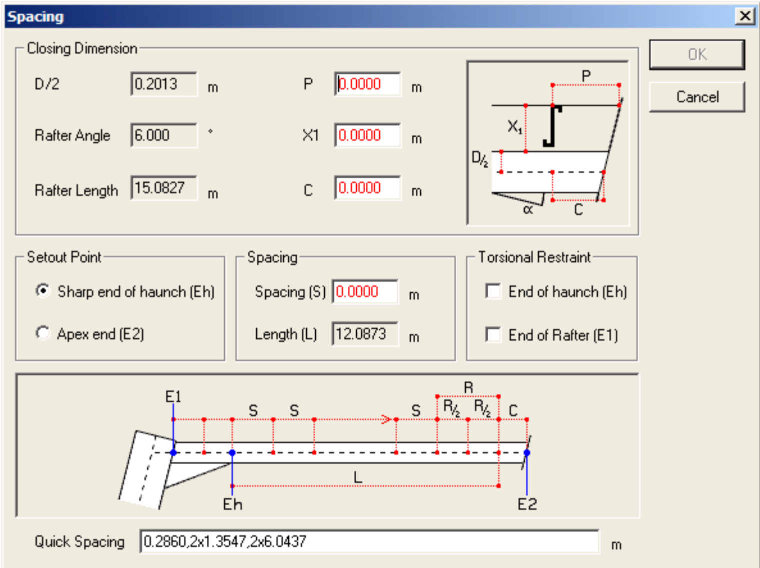
To automatically check a member's stability

1. Use the ***Project Workspace*** to choose the member whose stability you want to check automatically.

2. Click *Design/Member Stability/Automatic Member Stability* and you will see the appropriate restraint spacing dialog for the type of member.

Help For further information *see* *"Defining arrays of restraints"* on page 449.

3. Enter the details which are appropriate to the type of member that this is. The dialog below shows the ***Spacing*** dialog for a standard rafter.

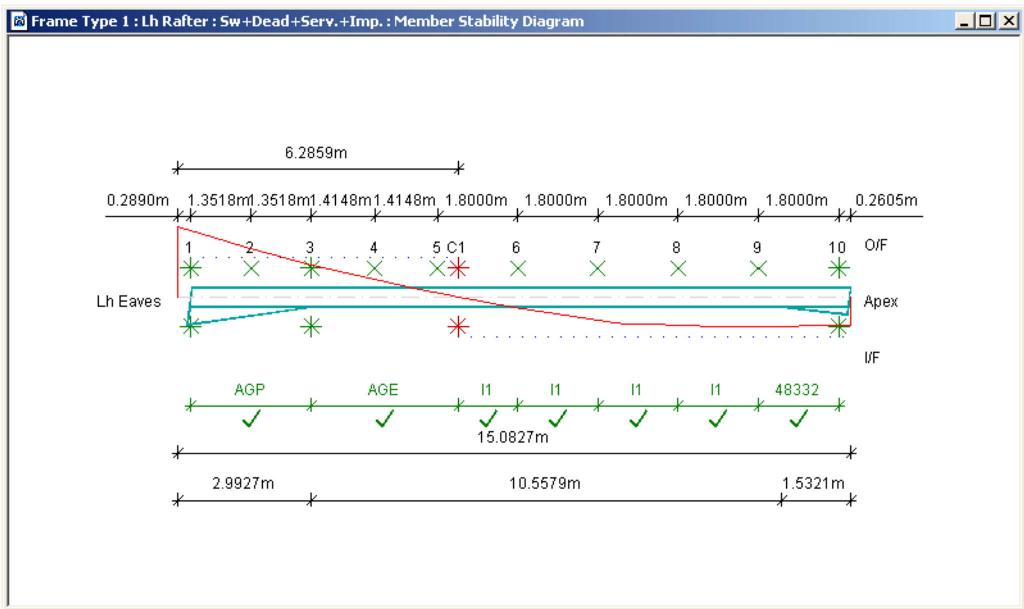


Caution The automatic routines require a torsional restraint at each end of the member. If there is a tick box on the dialog which allows you to control this (as in the representation above), then you should ensure that the box is ticked. If you don't do this, then *Portal Frame* will generate the restraints, but will not perform any member stability checks.

4. Once your details are correct click **OK** to create the restraints and automatically check the stability of the member.

> **Caution** *If you use this feature for a member with restraints, then Portal Frame deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints and performs new automatic stability checks.*

5. Once the checks are complete you will see the graphical display of the member's stability which shows which checks have been performed and the status of each of these.



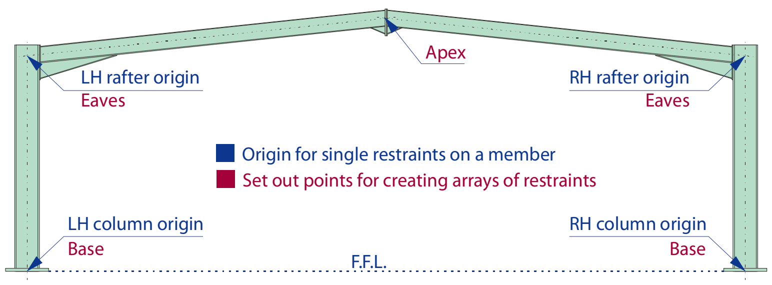
Defining restraints

You can define single restraint or an array of restraints on the current member using the ***Restraints*** dialog. Once you have defined restraints you can then add checks.

The ***origins*** for members of the different types of span are given in the following table when you define single restraints you give the distance from the origin. When you define arrays of restraints you can specify which ***set out point*** you want to use. Both origins and set out points are indicated clearly in the figures.

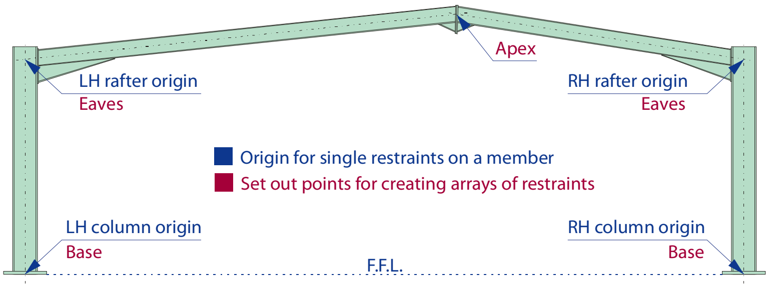
Member origins and set out points for restraints

Standard span

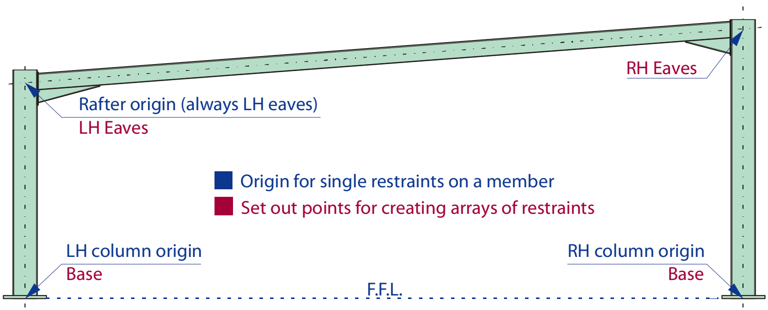


Member origins and set out points for restraints (Continued)

Asymmetric span

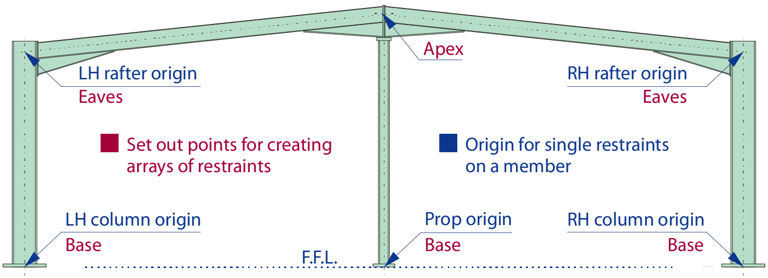


Monopitch span

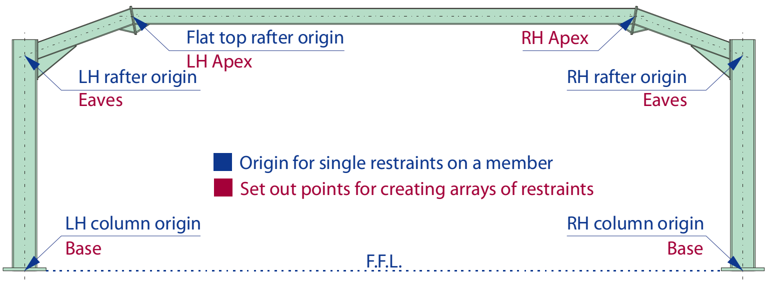


Member origins and set out points for restraints (Continued)

Propped span

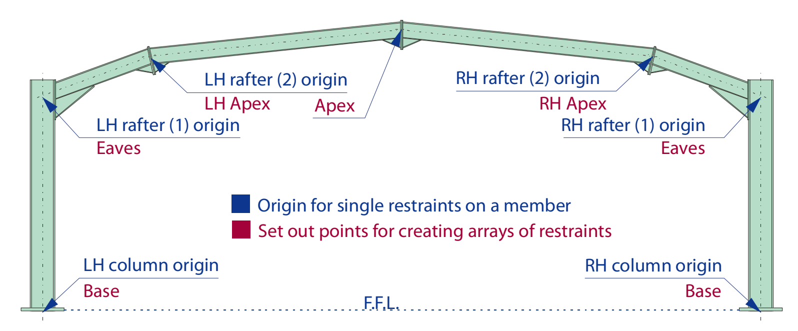


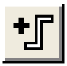
Flat top span



Member origins and set out points for restraints (Continued)

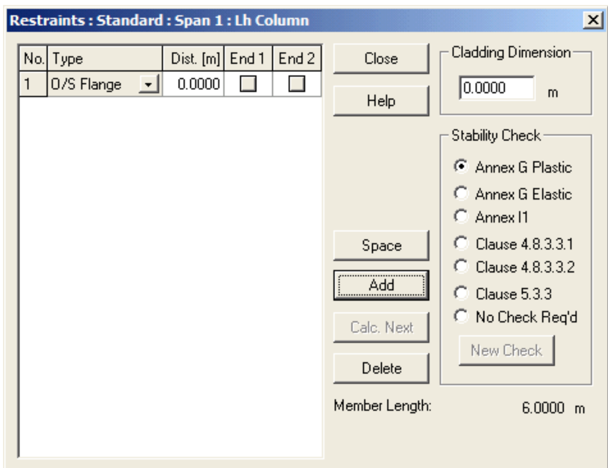
Mansard span





To define a single restraint

1. Click **Add** from the ***Restraints*** dialog. A new line will be added to the table of restraints.



2. Select the **Type** of restraint and enter its **Distance** from the origin of the member.

3. Continue to add restraints in this manner until you have defined all the restraints you require.

Tip If the remaining restraints are at equal spacing up the member you can use **Calc Next** to automatically generate these. The restraints will all be set to *Outside Flange* restraints.

Defining arrays of restraints

In many cases you will need to define several restraints on a member in order to get that member to pass the appropriate member stability checks. If you had to define each restraint individually, this would be a laborious and time-consuming operation. ***Portal Frame*** provides a means of defining all the restraints along a member in a single operation. If you subsequently need to alter the position, type of restraint, or even introduce new restraints, then you can do so using the single option as detailed above.

As there are several different types of member, there are different dialogs to allow you to define the restraints in a way appropriate to the member type. Each of these different dialogs is covered in the following text.

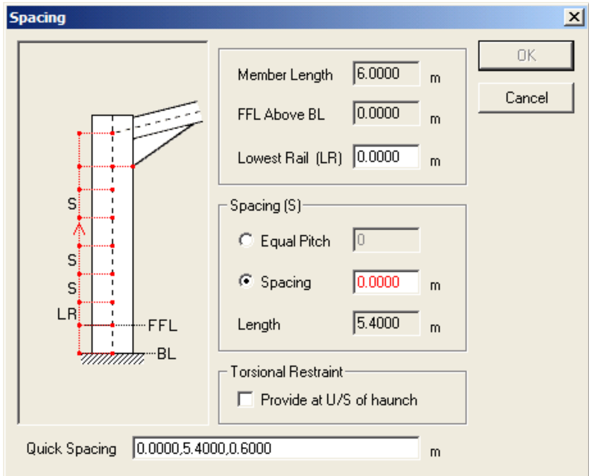
Help

For further information on the dialogs for the different frame types ***see***:

- *"To define an array of restraints for an external column"* on page 451,
- *"To define an array of restraints for an internal column equal eaves"* on page 452.
- *"To define an array of restraints for an internal column split eaves"* on page 454.
- *"To define an array of restraints for a standard rafter"* on page 456,
- *"To define an array of restraints for a monopitch rafter"* on page 458,
- *"To define an array of restraints for a flat rafter"* on page 459,
- *"To define an array of restraints for a Mansard second rafter"* on page 461.

To define an array of restraints for an external column

1. Click **Space...** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for a column.



Caution *If you use* ***Space*** *for a member with restraints, then* *Portal Frame* *deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.*

2. Enter the distance to the **Lowest Rail** from the member origin.

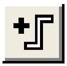
Help For further information **see** *"Member origins and set out points for restraints"* on page 445.

3. Now you need to choose how you want to space the rails. Either:
   - pick **Equal Pitch** and enter the number of rails you want to use, or
   - pick **Spacing** and enter the normal spacing between the rails.

4. *Portal Frame* will automatically place a torsional restraint at the base of the column and at the eaves. If you want the restraint at the underside of the haunch also to be a torsional restraint then tick **Torsional Restraint:Provide at U/S of haunch**.

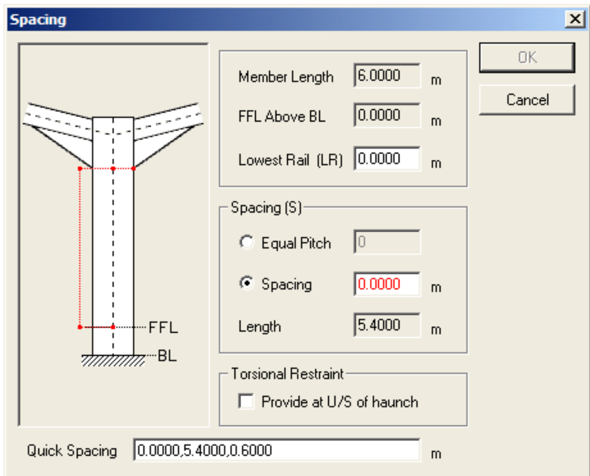
Note *Quick Spacing* shows the spacing that you will achieve and allows you to make any amendments you require.

5. Click **OK** to calculate the spacing of your rails and return to the *Restraints* dialog which will show these.



To define an array of restraints for an internal column equal eaves

1. Click **Space…** from the *Restraints* dialog – you will see the *Spacing* dialog for a column.



Caution If you use **Space** for a member with restraints, then *Portal Frame* deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.

Note Automatic spacing of restraints is not available for the internal column of a propped span.

2. Enter the distance to the **Lowest Rail** from the member origin.

Help For further information **see** *"Member origins and set out points for restraints"* on page 445.

3. Now you need to choose how you want to space the rails. Either:
   - pick **Equal Pitch** and enter the number of rails you want to use, or
   - pick **Spacing** and enter the normal spacing between the rails.

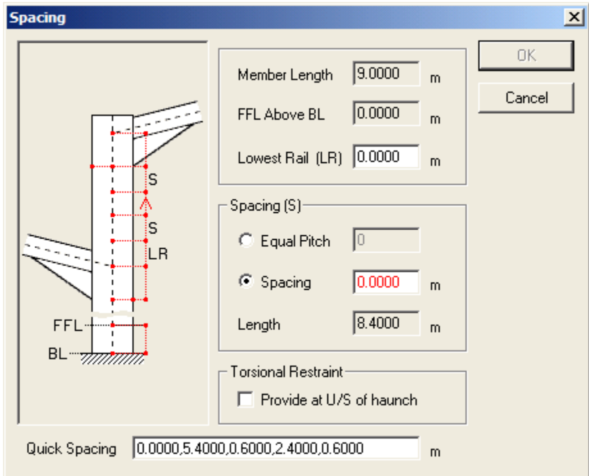
4. ***Portal Frame*** will automatically place a torsional restraint at the base of the column and at the eaves. If you want the restraint at the underside of the haunch also to be a torsional restraint then tick **Torsional Restraint:Provide at U/S of haunch**.

Note **Quick Spacing** shows the spacing that you will achieve and allows you to make any amendments you require.

5. Click **OK** to calculate the spacing of your rails and return to the ***Restraints*** dialog which will show these.

To define an array of restraints for an internal column split eaves

1. Click **Space…** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for a column.



Caution If you use **Space** for a member with restraints, then *Portal Frame* deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.

Note Automatic spacing of restraints is not available for the internal column of a propped span.

2. Enter the distance to the **Lowest Rail** from the member origin.

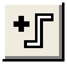
Help For further information **see** *"Member origins and set out points for restraints"* on page 445.

3. Now you need to choose how you want to space the rails. Either:
   - pick **Equal Pitch** and enter the number of rails you want to use, or
   - pick **Spacing** and enter the normal spacing between the rails.

4. *Portal Frame* will automatically place a torsional restraint at the base of the column and at the eaves. If you want the restraint at the underside of the haunch also to be a torsional restraint then tick **Torsional Restraint:Provide at U/S of haunch**.

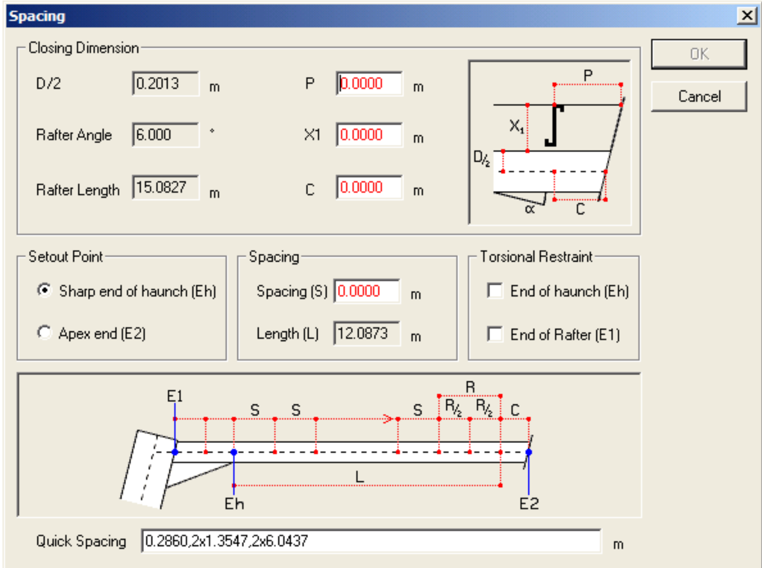
Note **Quick Spacing** shows the spacing that you will achieve and allows you to make any amendments you require.

5. Click **OK** to calculate the spacing of your rails and return to the *Restraints* dialog which will show these.



To define an array of restraints for a standard rafter

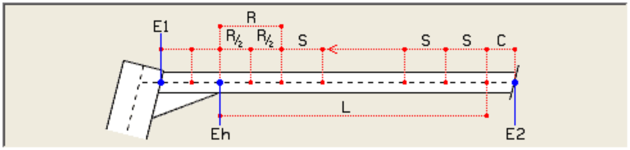
1. Click **Space…** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for a rafter.



Caution *If you use* ***Space*** *for a member with restraints, then* ***Portal Frame*** *deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.*

2. First pick whether you want to set the purlins out from the **Sharp end of the Haunch** or from the apex using the **Apex end (E2)**.

Note If you pick the latter option the diagram reconfigures to show the layout of purlins that you will achieve.



3. Enter the closing dimension at the underside of the sheeting (**P**).

4. Enter the distance from the face of the rafter/column to the inside face of the sheeting (**X1**).

Note This is the same distance as that in the *Restraints* dialog.

When you define this dimension *Portal Frame* automatically calculates the **Closing dimension (C)** for you.

5. If the **Closing dimension (C)** is not correct enter the value that you want to use.

Note When you define the **Closing Dimension (C)** *Portal Frame* automatically calculates the closing dimension at the underside of the sheeting (**P**) for you.

6. Enter the maximum **Spacing (S)** for your restraints.

7. ***Portal Frame*** does not create torsional restraints on a rafter automatically. If you want the restraints at the **End of haunch** and/or **End 1 of rafter** to be torsional restraints then tick the appropriate boxes. ***Portal Frame*** sets the other restraints to the outer flange.

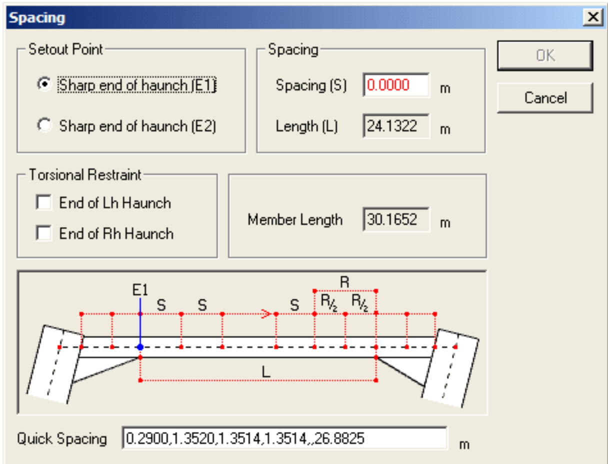
Note *Quick Spacing* shows the spacing that you will achieve and allows you to make any amendments you require.

8. Click **OK** to calculate the spacing of your purlins and return to the ***Restraints*** dialog which will show these.

To define an array of restraints for a monopitch rafter

This dialog is used to define an array of restraints for a monopitch rafter.

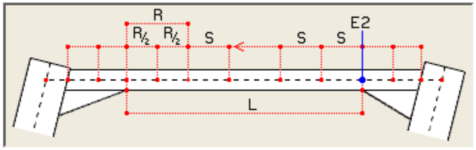
1. Click **Space…** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for a monopitch rafter.



Caution If you use *Space* for a member with restraints, then *Portal Frame* deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.

2. First pick whether you want to set the purlins out from the **Sharp end of Haunch (E1)** or from the **Sharp end of haunch (E2)**.

> **Note** — *If you pick the latter option the diagram reconfigures to show the layout of purlins that you will achieve.*



3. Enter the maximum **Spacing (S)** for your restraints.

> **Note** — ***Quick Spacing*** *shows the spacing that you will achieve and allows you to make any amendments you require.*

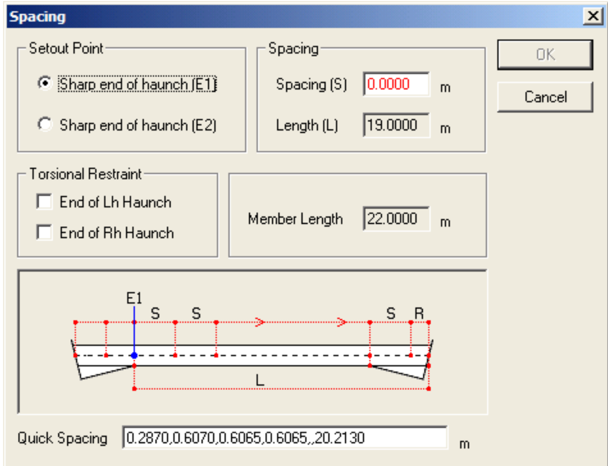
4. ***Portal Frame*** does not create torsional restraints on a rafter automatically. If you want the restraints at the **End of haunches** to be torsional restraints then tick the appropriate boxes. ***Portal Frame*** sets the other restraints to the outer flange.

5. Click **OK** to calculate the spacing of your purlins and return to the ***Restraints*** dialog which will show these.

To define an array of restraints for a flat rafter

This dialog is used to define an array of restraints for the rafter of a flat top portal.

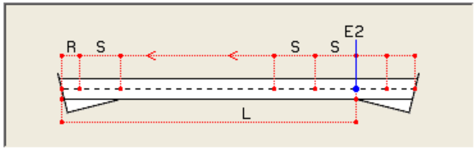
1. Click **Space…** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for a flat rafter.



Caution If you use **Space** for a member with restraints, then *Portal Frame* deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.

2. First pick whether you want to set the purlins out from the **Sharp end of Haunch (E1)** or from the **Sharp end of haunch (E2)**.

Note *If you pick the latter option the diagram reconfigures to show the layout of purlins that you will achieve.*



3. Enter the maximum **Spacing (S)** for your restraints.

Note ***Quick Spacing*** *shows the spacing that you will achieve and allows you to make any amendments you require.*

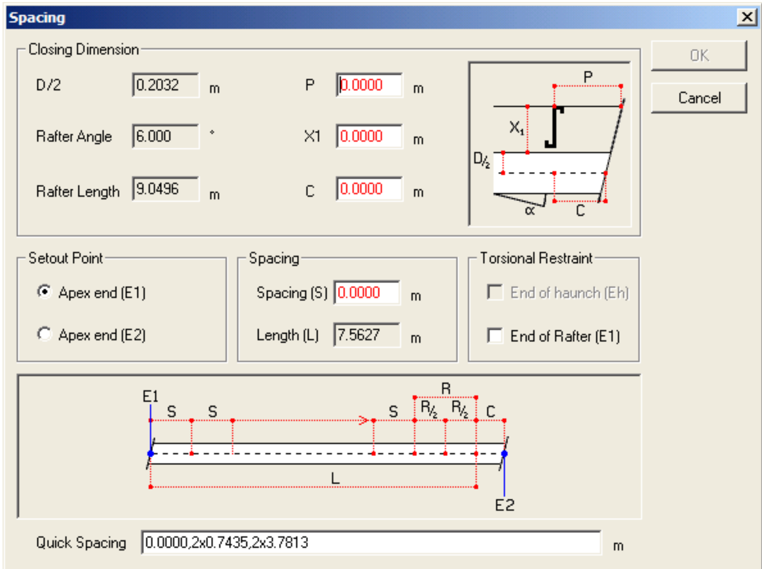
4. ***Portal Frame*** does not create torsional restraints on a rafter automatically. If you want the restraints at the **End of haunches** to be torsional restraints then tick the appropriate boxes. ***Portal Frame*** sets the other restraints to the outer flange.

5. Click **OK** to calculate the spacing of your purlins and return to the ***Restraints*** dialog which will show these.

To define an array of restraints for a Mansard second rafter

This dialog is used to define an array of restraints for the second rafter of a Mansard portal (the one that is not adjacent to the column).

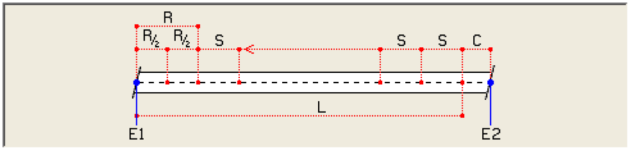
1. Click **Space…** from the ***Restraints*** dialog – you will see the ***Spacing*** dialog for the second Mansard rafter.



Caution *If you use* ***Space*** *for a member with restraints, then Portal Frame deletes the existing restraints (and any associated stability checks) before it creates the new array of restraints.*

2. First pick whether you want to set the purlins out from the **Apex end (E1)** or from the **Apex end (E2)**.

Note If you pick the latter option the diagram reconfigures to show the layout of purlins that you will achieve.



3. Enter the closing dimension at the underside of the sheeting (**P**).

4. Enter the distance from the face of the rafter/column to the inside face of the sheeting (**X1**).

Note This is the same distance as that in the *Restraints* dialog.

When you define this dimension *Portal Frame* automatically calculates the **Closing dimension (C)** for you.

5. If the **Closing dimension (C)** is not correct enter the value that you want to use.

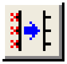
Note When you define the **Closing Dimension (C)** *Portal Frame* automatically calculates the closing dimension at the underside of the sheeting (**P**) for you.

6. Enter the maximum **Spacing (S)** for your restraints.

7. ***Portal Frame*** does not create torsional restraints on a rafter automatically. If you want the restraints at the **End of rafter (E1)** to be torsional restraints then tick the appropriate box. ***Portal Frame*** sets the other restraints to the outer flange.

Note *Quick Spacing shows the spacing that you will achieve and allows you to make any amendments you require.*

8. Click **OK** to calculate the spacing of your purlins and return to the ***Restraints*** dialog which will show these.



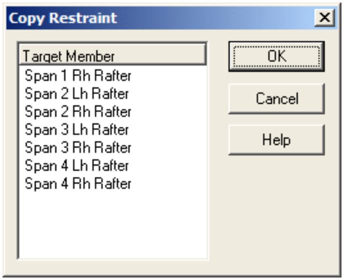
To copy restraints between frame members

You may find that the restraint details that you have given for one member are the same as those for other members in the frame. ***Portal Frame*** allows you to copy the restraint details for one member to all other allowable members within the frame.

1. Ensure that the member which you want to copy has the correct restraints defined, and that it is selected in the ***Stability*** toolbar.

Note *If the Stability toolbar is not active select Design / Member Stability / Check to activate it.*

2. Click **Copy Restraints to Members** from the ***Stability*** toolbar. You will then see the ***Copy Restraint*** dialog.



3. Select the members to which you want to copy the source member's restraint details. Once you have done this click **OK**. The details for the restraints on the original member will be copied to the other members that you have selected and the frame display will indicate these graphically.

Caution If you copy restraints to a member which already has them, then *Portal Frame* deletes the existing restraints (and any associated stability checks) before it copies the new restraints to that member.

Note If a member that you copy to is shorter than the one you are copying from, then only those restraints which are within the length of the target member will be copied.

Tip You can select as many members as you like when you do a copy thus generating restraints for many members quickly.

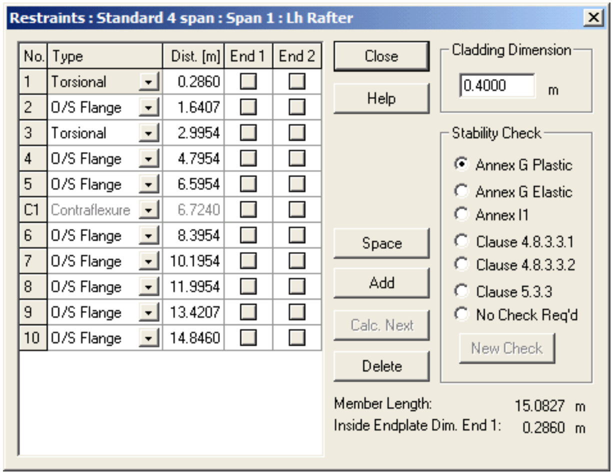
Tip If you select a member inadvertently don't worry - you can deselect it by clicking on it again.

To edit restraints using dialogs

If you need to change the position or type of restraint you can do so in ***Portal Frame***.

1. If the ***Stability*** window is not open select *Design/Member Stability/Check* to open it and activate the ***Stability*** toolbar.

2. Click **Member Restraints** to see the ***Restraints*** dialog.



3. Change the **Type** of restraint and its **Distance** from the member origin as necessary.

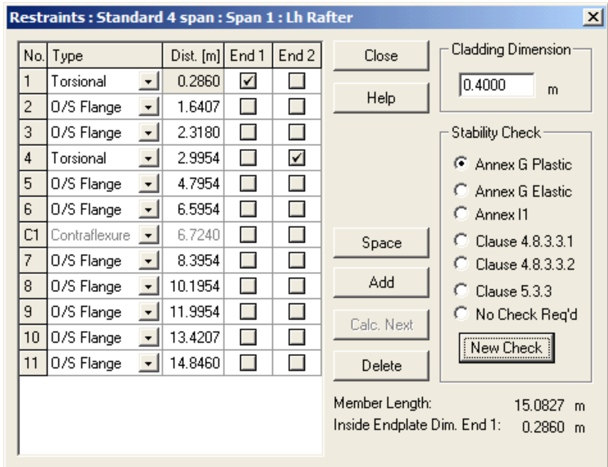
Help For further information *see "To define a single restraint" on page 448.*

Defining checks

When you have defined (at least two) restraints for your member you can define the stability checks with which you want to check the local stability of the member. You can either do so using the ***Restraints*** dialog, or graphically from the ***Member Stability*** window.

To define checks using the Restraints dialog

1. In the ***Restraints*** dialog click the tick boxes for the restraints which will be at **End 1** and **End 2** of your stability check.

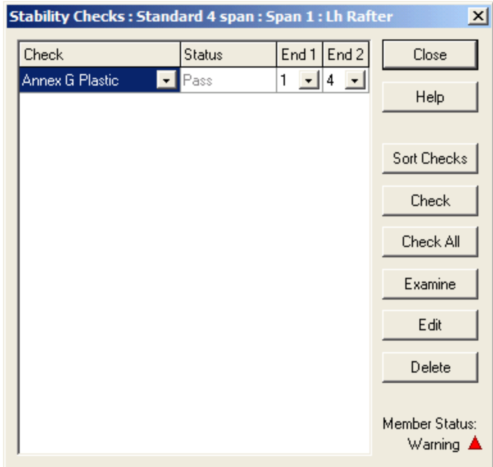


Note — If the *Restraints* dialog is not open select *Design / Member Stability / Define Member Restraints* to open it.

Note — *Portal Frame* automatically ticks the next appropriate restraint for you. In the capture above, since an Annex G Plastic check is picked *Portal Frame* automatically picks the next torsional restraint for you.

2. Pick the type of **Stability Check** that you want to perform between these two restraints.

3. Now click **New Check**. You will see the ***Stability Checks*** dialog which will include the check that you have just added.



4. Continue to add checks in this way until you are satisfied that the restrained member is stable or you have checked all the lengths for which you feel checking is necessary.

Note If you can not get a particular length to pass the checks, then you may need to add new restraints or to modify existing ones. You do this using the *Restraints* dialog. Any checks which are affected by your changes will be set to have the status *Unchecked*. You can then either select a particular check and click **Check**, or click **Check All** to perform the calculations for all *Unchecked* checks.

5. If the check has details that you can change you can do this by selecting the check and clicking **Edit**.

Note You can define the checks on your member in any order. If, at any time, you want to see your checks ordered in sequence up the member click **Sort Checks** to arrange them based on their start and end restraint numbers.

Help For further information on changing the details for the various check types ***see:***

- *"To set options for an Annex G Plastic or Annex G Elastic check"* on page 478,
- *"To set options for a clause 4.8.3.3.2, clause 4.8.3.3.1 or Annex I1 check"* on page 479.

To define checks graphically

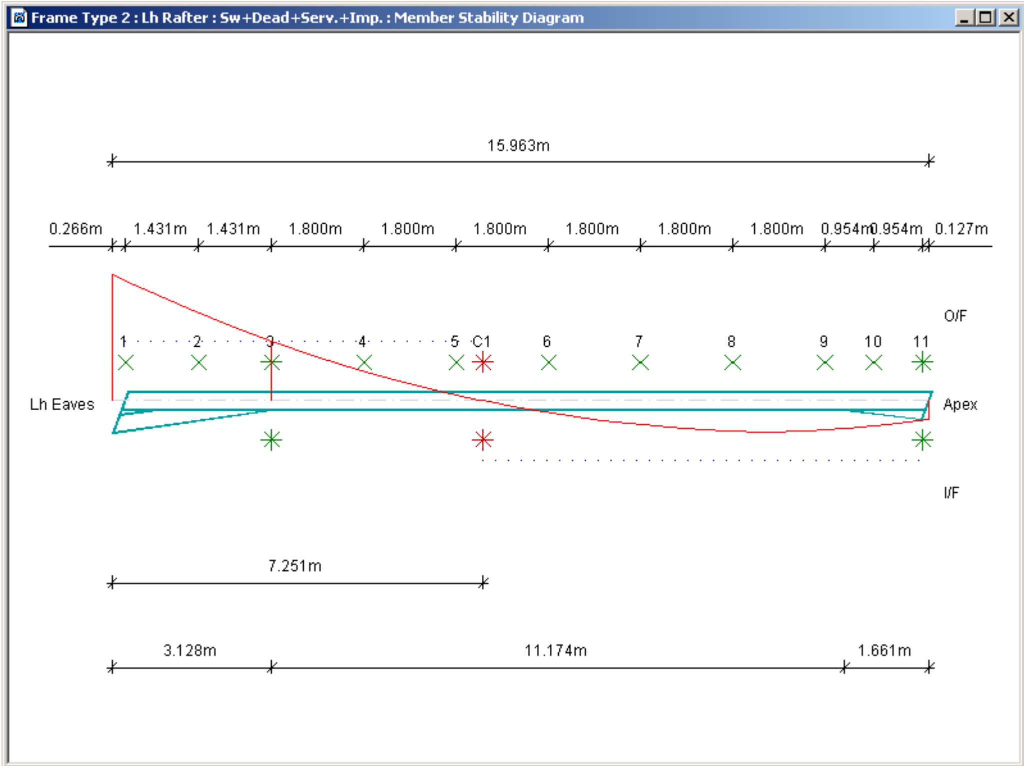
In order to use this method you must have defined the restraints on the member whose local stability you are checking.

Help

For further information on defining restraints ***see***:

- *"To define a single restraint"* on page 448,
- *"To define an array of restraints for an external column"* on page 451,
- *"To define an array of restraints for an internal column equal eaves"* on page 452,
- *"To define an array of restraints for an internal column split eaves"* on page 454,
- *"To define an array of restraints for a standard rafter"* on page 456,
- *"To define an array of restraints for a monopitch rafter"* on page 458,
- *"To define an array of restraints for a flat rafter"* on page 459,
- *"To define an array of restraints for a Mansard second rafter"* on page 461.

1. Ensure that the ***Member Stability Check*** window is active.

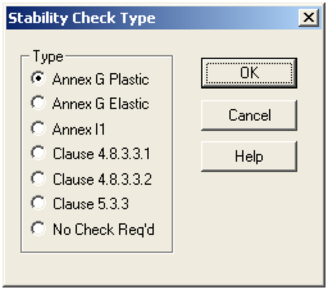


2. Click on the restraint symbol at the start of the length you want to check for stability.

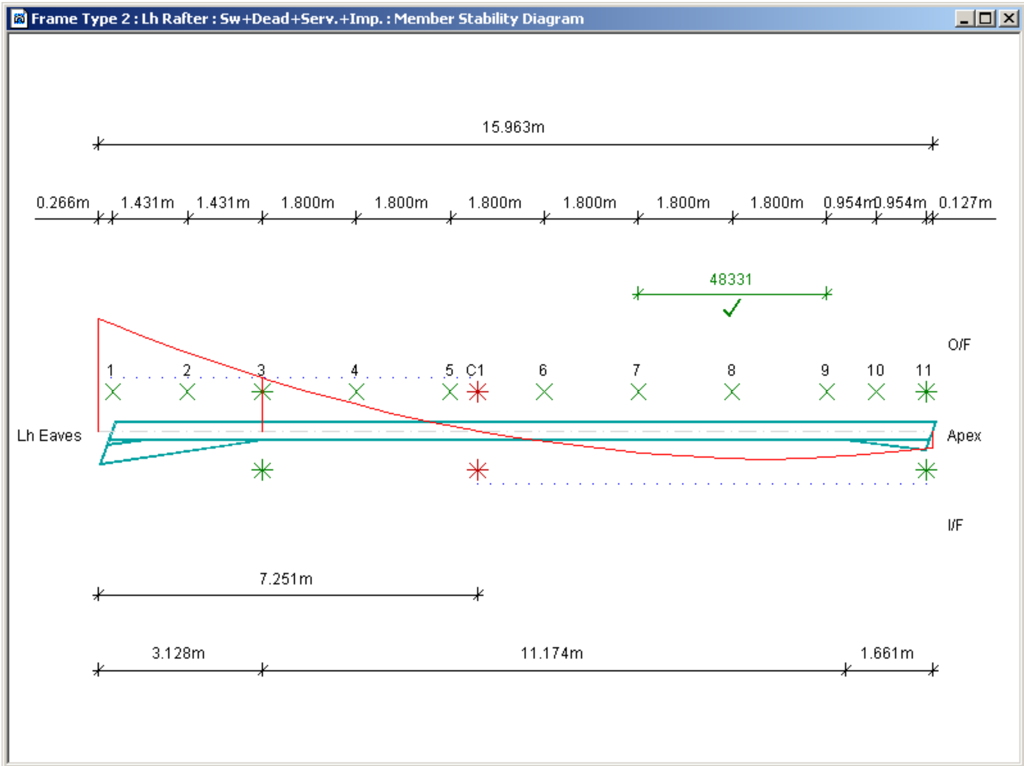
3. Move to the restraint symbol at the end of the length.

Note As you do this you will see a rubber line from the start point you picked. This helps you to see the exact length of your check.

4. You will see the ***Stability Check Type*** dialog which allows you to choose the type of check you want to perform.



5. Pick the type of **Stability Check** that you want to perform between these two restraint points and click **OK**. The check will be performed and you will see the result in the ***Member Stability Check*** window.



Modifying the start or end position of a check

As you modify your restraints *Portal Frame* automatically synchronises the stability checks with the restraints. If your changes mean that you need to change the restraints between which a particular check applies you can do this yourself.

To change the end restraints for a check

1. In the table of checks choose the new restraint position for the End of the check that you want to change.

2. As you make your changes the status of the check will be set to *Unchecked*. When you have the correct restraints for both ends of the check click **Check** to perform the calculations.

Deleting checks

If you want to delete checks that you have performed, because the length is covered by another check, or the check is no longer valid under the current design combination, this is easy in *Portal Frame*.

To delete a check

1. Select the check which you want to delete from the table of checks.

2. Click **Delete**. The selected check will be deleted.

Note If you delete restraints, then any checks which start or end at the deleted restraint will also be deleted.

Copying checks

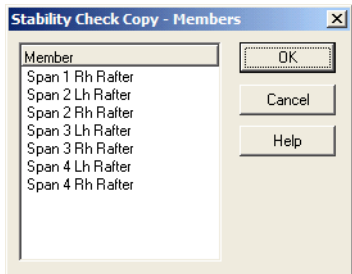
You may find that the checks that you have performed for a member and a particular design combination would be appropriate for the same design combination for another member, or would be appropriate for a different design combination for the current member. *Portal Frame* provides both of these copy options.

To copy stability checks to a different member

1. Ensure that the ***Project Workspace*** shows the design combination and frame member whose stability checks you want to copy.

   Note If the *Stability* toolbar is not active select *Design / Member Stability / Check* to activate it.

2. Select the *Design / Member Stability / Copy Stability Checks to Members* icon to see the ***Check Copy - Members*** dialog.



   The dialog only shows members to which you can copy stability checks.

3. Pick the members which are to receive the copied checks, then click **OK** to perform the copy.

   Note *Portal Frame* does not perform the copied checks automatically, you must pick the target member and perform the checks. Alternatively you can use **Check Selected Combination** or **Check All Combinations** from the *Stability* toolbar.

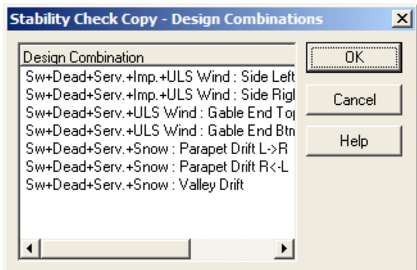
Caution If you copy stability checks then this will delete any existing restraint and stability check details for the target members.

Note If the target member is shorter than the one you copy from, then only those restraints and checks which are wholly within the length of the target member will be copied.

To copy stability checks to a different combination

1. Ensure that the ***Project Workspace*** shows the design combination and frame member whose details you want to copy.

2. Select *Design / Member Stability / Copy Stability Checks to Design Combinations* icon to see the ***Check Copy - Design Combinations*** dialog.



3. Pick the combinations which are to receive the copied checks, then click **OK** to perform the copy.

Note *Portal Frame* does not perform the copied checks automatically, you must pick the target combination and perform the checks. Alternatively you can use **Check Selected Combination** or **Check All Combinations** from the *Stability* toolbar.

Caution If you copy stability checks then this will delete any existing stability check details for the target design combinations.

To check the current combination

1. Ensure that the ***Project Workspace*** shows the design combination and frame member whose stability adequacy you want to check.

2. Select *Design / Member Stability / Check Selected Combinations* icon to perform the stability checks for the current combination.

To check all combinations

1. Ensure that the ***Project Workspace*** shows the frame member whose stability adequacy you want to check.

2. Select *Design / Member Stability / Check All Combinations* icon to perform the stability checks for all combinations.

Modifying check options

BS 5950 covers various conditions that can occur when considering local stability, for example:

- whether there is any loading applied to the length being considered,
- whether that loading is normal or destabilizing.

In practice for the design of portal frames the options are substantially reduced; for example the loading is usually applied to the frame through the sheeting rails, and thus there is no loading between points of restraint.

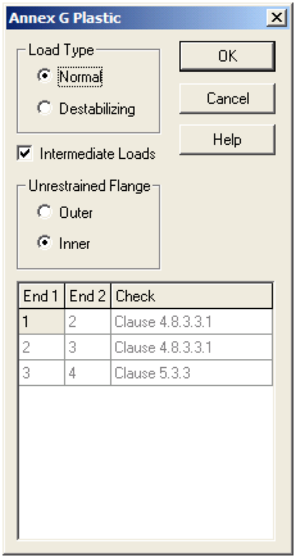
Portal Frame therefore adopts a set of values which will be correct in most cases. Full details of these are given in the ***Engineer's Handbook***.

Help For further information on the settings that are used ***see*** *"Member stability checks"* on page 871.

To set options for an Annex G Plastic or Annex G Elastic check

1. Pick the Annex G Plastic or Annex G Elastic check whose details you want to change.

2. Click **Edit** to see the options dialog for this check.



3. Pick the **Load Type** as **Normal** or **Destabilizing** for the intermediate loads.

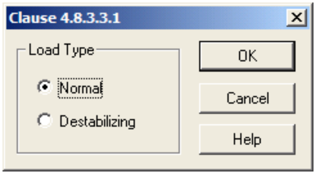
4. Pick the appropriate **Intermediate Loading** and the **Unrestrained Flange** settings that you want to consider for the intermediate length checks.

5. Once these options are correct click **OK** to return to the ***Stability Checks*** dialog, automatically performing the Annex G Plastic check with the changed options.

To set options for a clause 4.8.3.3.2, clause 4.8.3.3.1 or Annex I1 check

1. Ensure that the Clause 4.8.3.3.2, Clause 4.8.3.3.1 or Annex I1 check whose options you want to change is selected.

2. Click **Edit** to see the options dialog for this check.



3. Make the necessary selections for the **Load Type**.

4. If you are working on the prop column of a propped portal, then you can enter the effective length factors that you want use for the major (***xx***) and minor (***yy***) axes. In all other circumstances this information is not appropriate and is dimmed.

5. To return to the ***Stability Checks*** dialog, automatically performing the Clause 4.8.3.3.2 check with the changed options.

Viewing check results

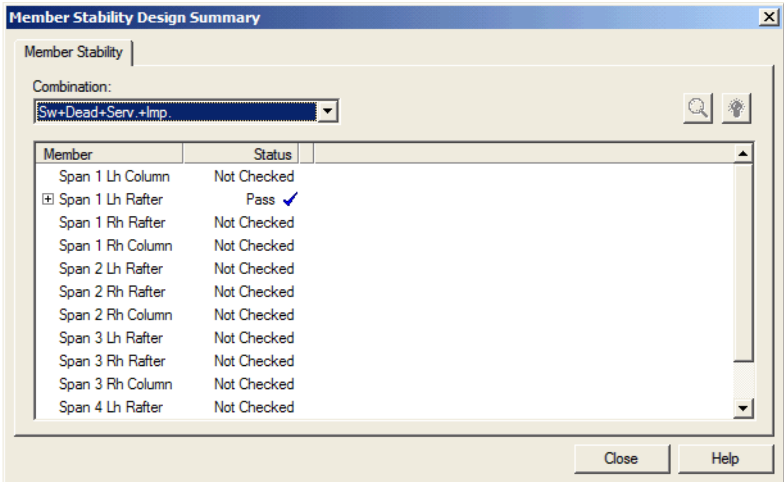
Whenever you **Add** a new check or **Edit** the options for an existing one the calculations for that check will be performed immediately. You will see the overall result for the check in the table of checks. The overall status of the check will also be shown on the member graphical display.

You may need to view the results for a particular member in order to refresh your memory, or you may need to view the details for a particular check in more detail. For example you might need to see how close a failed check is to passing before you take corrective measures by tweaking the restraints.

To view the results summary for a design combination

1. Ensure that the ***Stability*** toolbar shows the design combination of whose results you want a summary.

2. Select *Design / Member Stability / Check Status Summary* icon to see the ***Member Stability Design Summary*** dialog showing the results for the current combination.



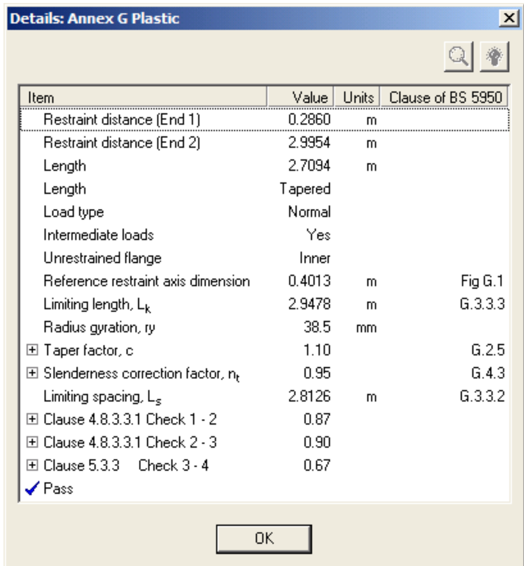
3. If you want to view the results for other design combinations select the one in which you are interested from the list. When you have finished viewing results click **Close** to shut the ***Member Stability Design Summary*** dialog.

To view the results for a check

1. Select the check whose results you want to view from the table of checks that you have performed.

Note If the list of checks is not open select ***Design/Member Stability/Define Stability Checks*** in order to see it. This option is only available when you have defined restraints on a member.

2. Click **Examine** to view a summary of the results for that check. You can see a typical example below.



If you have performed a check that requires the checking of other lengths within the main length you will see these listed at the end of the overall check. These checks will have a plus icon to their length. You may also see this icon to the left of other lines. This indicates that there is more information available. Simply double click the line and a new ***Details*** dialog will open showing you this additional information.

Tip You can also single click the line and then click the **Details** icon ( ).

This ***Details*** dialog might itself have further information for some of its lines. You can therefore delve down quickly to the detail you need. Once you have reviewed the pertinent information you can:

- click **OK** to close the individual dialogs until you return to the initial summary page and then click **Close**.
- Click **Close** on the initial summary page to close it and all the detail dialogs immediately.
- Select a different check in the ***Stability Checks*** dialog's table of checks to show the initial ***Check*** summary dialog for that check and close all open ***Details*** dialogs for the current check.

Additionally many checks in ***Portal Frame*** have a unique feature – the engineering tip. This gives you advice to help you modify your stability checks to achieve a satisfactory result. You can tell when an engineering tip is available since the line in the check Summary uses coloured text. To see the tip select that line and click the **Tip** icon ( ).

3. When you have finished viewing your stability check details click **Close** to shut the ***Check*** dialog.

30 Creating a Report

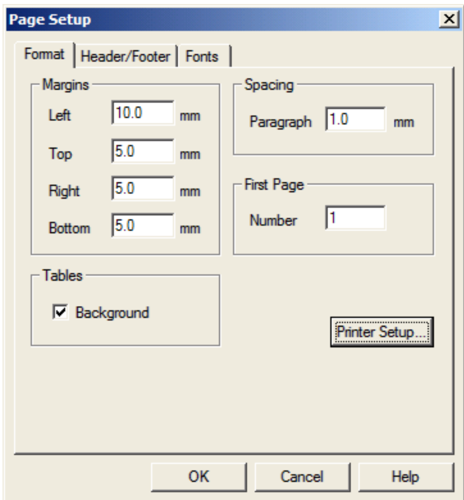
When you complete your portal frame design(s) you may need to create a report. Its scope could range from brief details (sufficient to jog your memory if the design is to be taken further) through to a complete submission (ready to send to a checking authority or to your client). *Portal Frame* has a sophisticated report generator which allows you to produce high quality reports that contain exactly the level of detail that you require.

Setting up the page details

The *Portal Frame Report Generator* lets you set the page layout – ***Margins***, ***Headers & Footers*** and ***Fonts*** – so that your calculations are aesthetically pleasing.

To set the page format details

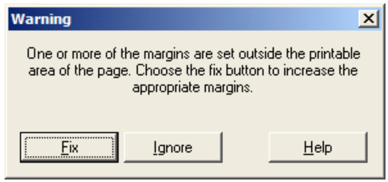
1. Select *File/Report Page Setup…*



2. Enter your preferred margins into the **Left**, **Top**, **Right** and **Bottom** boxes.

Fix margins dialog

Note If you enter a margin which is less than your printer's minimum and you try to leave the page, you will see the warning dialog.



If you **Fix** the problem any invalid margins will be set to the minimum value for your printer.

Caution If you **Ignore** the problem then your report may not print correctly.

3. **Paragraph Spacing** allows you to set the amount of white space that you want to see between the paragraphs in your report. A larger amount of space will make your report look less cluttered, but will increase the number of pages.

4. By default ***Portal Frame*** numbers all reports starting at page one. If you want to create a series of reports numbering them sequentially, then print each one in turn and set the **First Page Number** for the current report to one greater than the last page of the previous one.

5. Many of the reports include tables of information. If you want the headers of the tables shaded, then check **Table Background**.

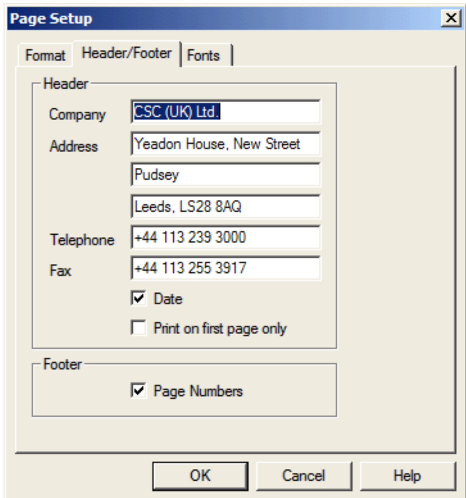
Caution *With some printers this may reduce the readability of the text. If you are going to fax your reports, we recommend that you do not check this option.*

6. If you want to check the system settings for your printer, then click **Printer Setup**. You will then see the ***System Print Setup*** dialog which will be system and printer dependent.

7. Pick another page of the ***Page Setup*** property sheet, or click **OK** to create your report.

To set the page header and footer details

1. Select *File/Report Page Setup…* and the ***Header/Footer*** tab.



2. Enter your **Company Name**, **Address**, **Telephone** and **Fax** details.

3. If you want you can date your report by checking the **Date** box.

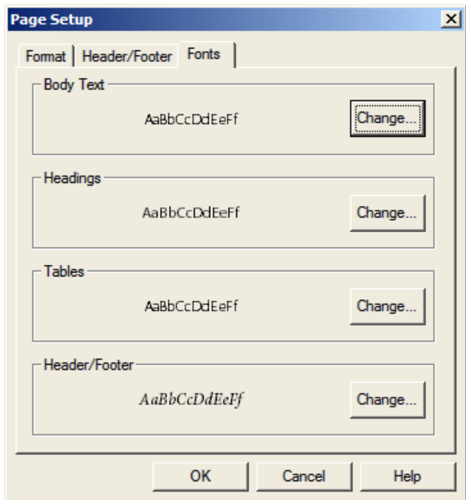
4. You can include the **Header** information (plus the project reference details) on just the first page or on all pages of the report by checking or unchecking the **Print on first page only** box.

5. If you want the report numbering check **Page Numbers**. The numbering will start at the **First Page Number** set on the ***Format*** page.

6. Pick another page of the ***Page Setup*** property sheet, or click **OK** to create your report.

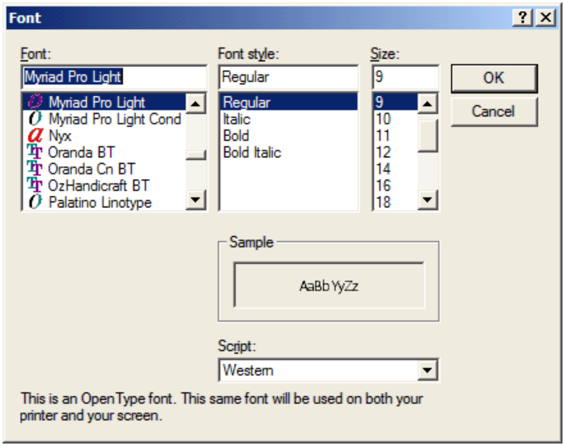
To set the font details

1. Select *File/Report Page Setup…* followed by the ***Fonts*** tab.



There are areas for the **Body Text**, **Headings**, **Tables** and **Header/Footer**. You will see samples of the current font for each of these. You can set each of these independently.

2. Click **Change…** for the text you want to change. You will see the ***Font*** dialog.



This is a standard ***Windows*** dialog which allows you to pick any installed, active font on your system.

When you have picked a font that you like click **OK** to return to the ***Fonts*** page.

3. Pick another page of the ***Page Setup*** property sheet, or click **OK** to create your report.

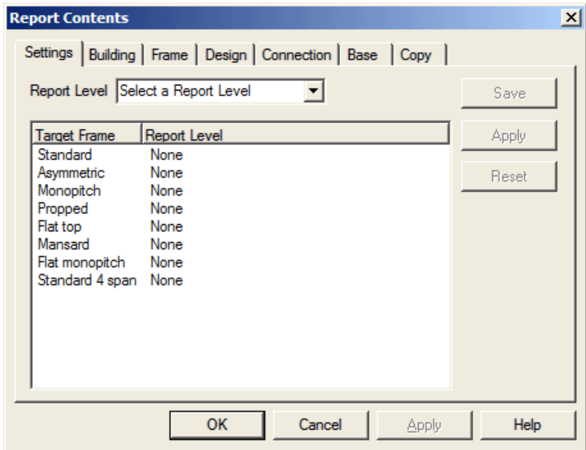
Specifying the content of the frame design report

Portal Frame provides powerful features to allow you to produce exactly the reports you require. When you install ***Portal Frame*** a series of standard report options are set. You can use these as they are, or adapt them to meet your requirements. If you do modify a report in this way, you can replace a standard setting with your changed one. You can then use these revised settings to set the same details for future projects and portal frames quickly.

To set the content of the frame design report

1. Select *File/Report Contents…*

2. You will then see the ***Report Contents*** property sheet.



You can use this page to:

- Pick the **Report Level** that you want to use for each portal frame in your project,
- Save the settings that you have made for one of your portal frames to one of the standard levels of output so that you can use these for other portal frames in this and future projects,
- Restore the default (as installed) settings for any level of output.

To set the level of output for a particular portal frame

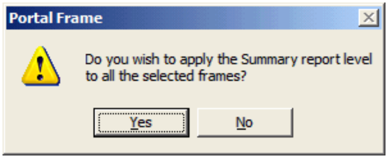
You can choose the level of output that you want to use for one or more portal frames.

1. Select the level of report that you require. You can select *Full*, *Reduced* or *Summary*.

2. From the list of *Target Frames* pick the portal frame(s) that you want to take this level of report.

Note *Apply is only enabled when you have completed steps 1 and 2 above.*

3. Click **Apply** and you will see a confirmation dialog.



Click **Yes** to set the level for all the portal frames you selected.

4. Pick another page of the *Report Contents* property sheet, or click **OK** to create your report.

To set the items that are included at a particular level

Portal Frame comes with several pre-configured standard report levels. You can tailor any of the standard reports to meet your particular requirements or preferences.

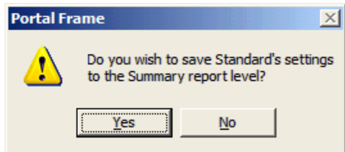
1. Select the level of report that you want to configure, *Full*, *Reduced* or *Summary*:

2. Pick a *Target Frame* that you will configure so that its output matches your requirements for this level of report.

3. Set the various pages of the ***Report Options*** property sheet to indicate the items you want to include for this level of report.

Help For further information on choosing the information to be included in a report ***see*** *"To set the content of the frame design report"* on page 488.

4. Once your settings are complete return to the ***Settings*** page and click **Save**. You are asked to confirm the save.



Click **Yes** and the selected items are stored away ready to be applied when you select this level of report in future.

5. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

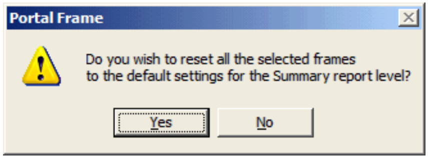
To use the default items for a particular level

Portal Frame allows you to reset any of your portal frames to use the standard (as supplied) report contents, rather than your amended ones.

1. From the list of **Report Levels** select the level which initially contained the items you want to include.

2. From the list of *Target Frames* choose the portal frame(s) that you want to reset to use the default items.

3. Click **Reset**. You are asked to confirm the reset.



Click **Yes** and the selected portal frames will take the default (as installed) settings for that level.

Tip *If you want to restore the default items for future use for this level then click **Save** immediately after clicking **Reset**.*

4. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

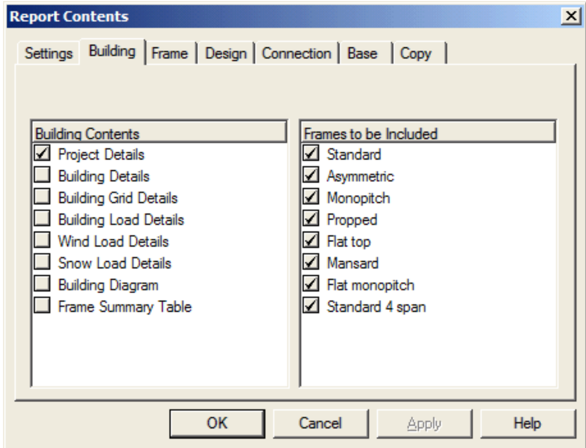
Controlling the information contained in the frame design report

You can control the information that is included in the report at various levels:

- ***Building level*** – you can control the building details that are included in the report.
- ***Frame level*** – you choose the portal frames and the major items (input, design and diagram) for each portal frame,
- ***Local level*** – for each portal frame you choose which design combinations you want to include and the checks for each design combination.

To set building contents and the frames to be included

1. Select *File/Report Content* followed by the ***Building*** tab.



2. In the **Building Contents** list tick the ***items*** you want to include in the report.

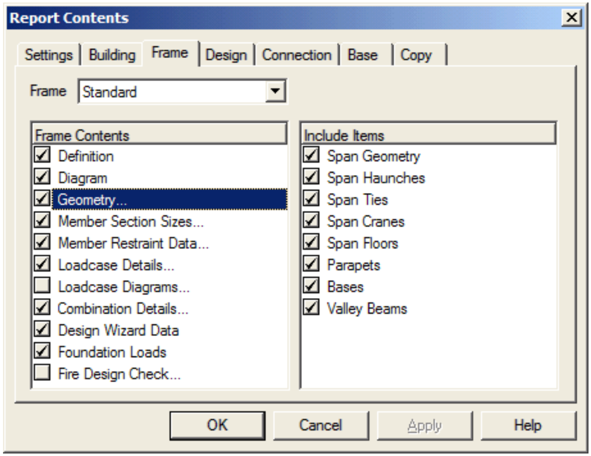
3. In the **Frames to be Included** list tick the frames you want to include in the report.

Caution — *If you remove the tick to the side of a particular frame then* ***no details*** *will be included in the report irrespective of the settings on the other pages of the property sheet.*

4. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

To set frame contents

1. Select *File/Report Content* followed by the ***Frame*** tab.



Pick the **Frame** whose report contents you want to define. The items shown in the *Frame Contents* and *Include Items* lists are set on a frame by frame basis.

Some items in the *Frame Contents* list have additional sub-items which are shown in the *Include* Items list.

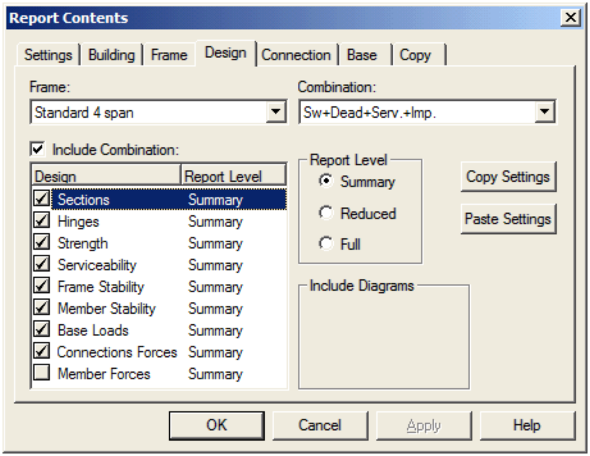
2. Tick the **Frame Contents** items that you want to include in your report, together with any associated ***Sub-items*** from the **Include Items** list.

Caution *If a **Frame Contents** item is not ticked, then none of the sub-Items from its **Include Items** list will be included in the report irrespective of whether they are ticked or not.*

3. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

To set design contents

1. Select *File/Report Content* followed by the ***Design*** tab.



Pick the **Frame** and **Combination** whose report contents you want to define. The remaining details on this page are set on a **Frame / Combination** basis.

2. If you want to include the design results for this combination tick **Include Combination**.

Caution *If you do not tick **Include Combination** then no details will be included irrespective of the other settings you make on this page.*

3. In the **Design** list tick the items that you want to include and pick the appropriate level for each item.

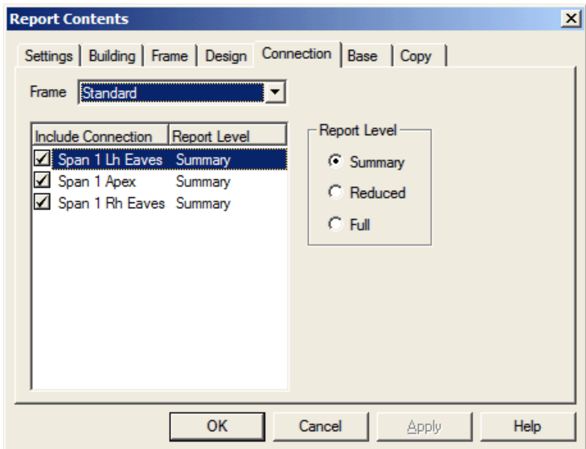
Caution If you do not tick an item in the **Design** list then no details will be included for the item irrespective of the **Report Level** and **Include Diagrams** settings you make.

Note Diagrams are available for some items. When you pick such an item you will see check boxes for the diagrams. Tick those you want to include in your report.

4. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

To set connection contents

1. Select *File/Report Content* followed by the ***Connection*** tab.



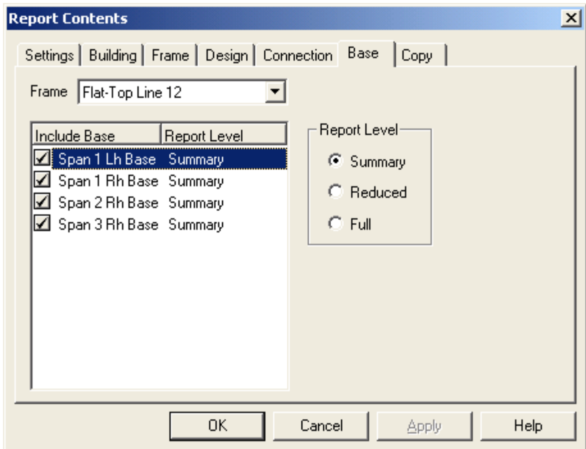
In this case you will see a list of all the connections you have designed.

2. In the **Include Connection** list tick the connections you want to include in your report and set the level of output that you want to use for that connection.

3. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

To include foundation contents

1. Select *File/Report Content* followed by the ***Base*** tab.



2. In the **Include Base** list tick the bases you want to include in your report and set the level of output that you want to use for that base.

3. Pick another page of the ***Report Contents*** property sheet, or click **OK** to create your report.

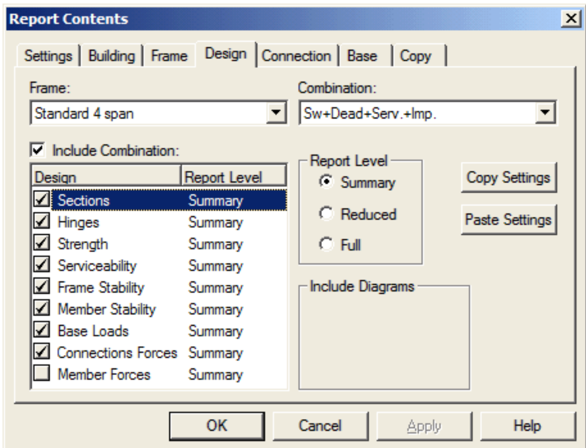
Note *The design results for the bases will take the preferences which you set in Fastrak Bases, and not those which are current in Portal Frame. To avoid inconsistencies you should check that your preferences are identical between applications.*

Repeating frame design report settings across design combinations

You can copy report settings from one design combination to any other design combinations with a couple of mouse clicks.

To copy design combination settings

1. Select *File/Report Content* followed by the ***Design*** tab.



2. Pick a **Frame** and **Combination** from the lists of frames and combinations in your project.

3. Make the settings that you require for this condition and then click **Copy Settings**.

4. Now select the **Frame** and **Combination** that is to receive the copied settings.

Tip You can change the frame if necessary. You might prefer to use the *Copy* page to copy settings from one frame to another.

Help For further information on copying between frames **see** *"Repeating frame design report settings between frames"* on page 498.

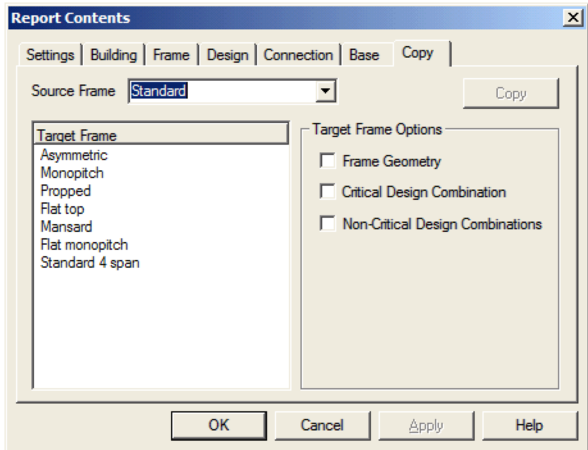
5. Finally click **Paste Settings**, and the copied settings will be applied to this design combination.

Repeating frame design report settings between frames

When you have set your report content for one frame you can copy this to any others. This gives your reports consistency and makes them easy to follow and understand.

To copy report settings between frames

1. Select *File/Report Contents…* followed by the ***Copy*** tab.



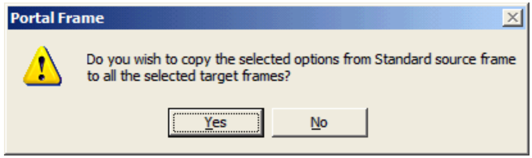
2. From the *Source Frame* list pick the frame whose report settings you want to copy. When you do this the list of *Target Frames* will show only the other portal frames in the project.

3. Select the frame(s) whose options you want to set.

4. From the list of *Copy options* pick the particular options to copy:
   - **Frame Geometry** – will copy the ***Frame*** page settings to all the target portal frames.
   - **Critical Design Combination** – will copy the ***Design*** page settings for the critical design combination to the critical design combination of the target portal frames.

- **Non Critical Design Combination** – will copy the ***Design*** page settings for the first non critical design combination to all the non critical design combinations of the target portal frames.

5. Click **Copy** and you will see a confirmation dialog.



Click **Yes** to copy your selected source frame options to all selected target frames.

Note *Copy is only enabled when you have completed steps 2 to 4 above.*

6. Pick another page of the ***Report Contents*** property sheet, or click **OK** to register your settings.

Viewing the frame design report

If you have already defined and created a report but have closed it down, ***Portal Frame*** will open the current report. If there is no current report, then ***Portal Frame*** will create it for you.

To view a frame design report

1. Select *File/Report/Frame Design*.

Viewing the materials list report

If you have already defined and created a materials list report but have closed it down, ***Portal Frame*** will open the current report. If there is no current report, then ***Portal Frame*** will create it for you.

To view a materials list report

1. Select *File/Report/Material Listing*.

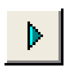
Using the report window

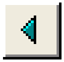
The ***Report Window*** shows you an exact preview of the printed report. You can move through the report and choose how the information is presented on the screen. Once you are satisfied that the report meets your needs you can:

- print it,
- export it to ***TEDDS***,
- export it to ***Word***.

To move through the report

You use the icons on the ***View*** toolbar and the scroll bars on the ***Report Window*** to move around your report. You can also use the following keyboard short-cuts:

Icon	*Short-Cut*	*Action – move*
	HOME	to top of current page
	END	to bottom of current page
	UP ARROW	up current page
	DOWN ARROW	down current page
	PAGE DOWN	to top of next page

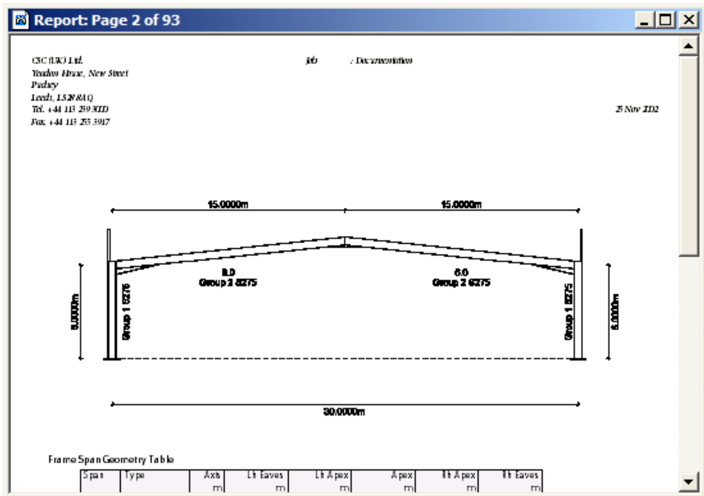
Icon	*Short-Cut*	*Action – move*
	PAGE UP	to top of previous page
	CTRL+HOME	back to start of report
	CTRL+END	to end of report

To control the view of the report

You can see the report in page width, full page and double page layouts.

To view in page width format

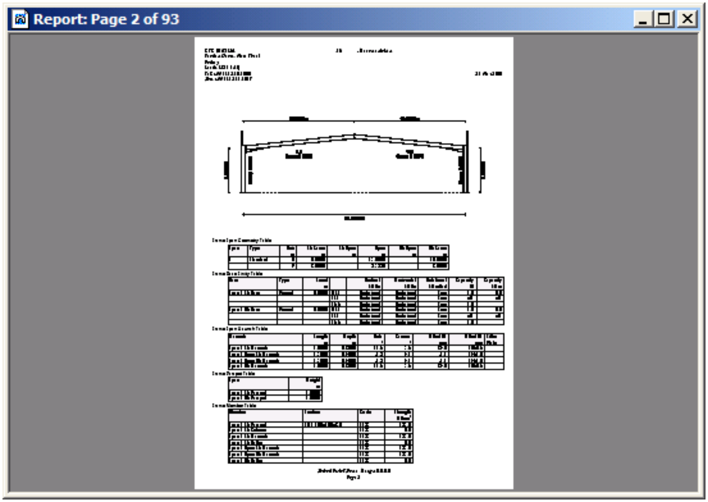
1. Click **Page Width** from the ***Toolbar***.



Mouse *Click anywhere over a report view with the right mouse button. Select* ***Page Width*** *from the context menu.*

To view in full page format

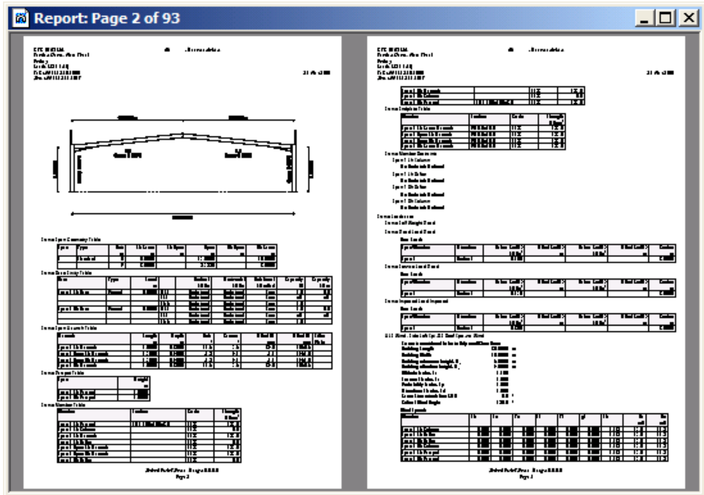
1. Click **Full Page** from the ***Toolbar***.



Mouse *Click anywhere over a report view with the right mouse button. Select* ***Full Page*** *from the context menu.*

To view in double page format

1. Click **Double Page** from the *Toolbar*.

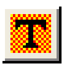


Mouse — Click anywhere over a report view with the right mouse button. Select *Double Page* from the context menu.

To print the report

1. Select *File/Print*

   You will see the *Windows Printer* dialog which depends on your version of *Windows* and your printer. Make your settings and then click **OK** to print the report.



To transfer the report to TEDDS

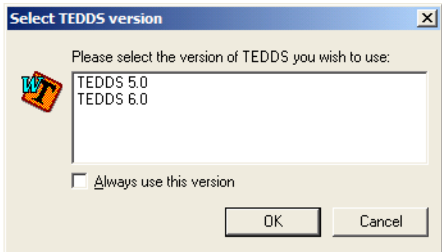
1. Set the content of your report and create it.

2. With the *Report Window* active select *File/Export Report to TEDDS…*

Mouse Click anywhere over a report view with the right mouse button. Select *Export Report to TEDDS...* from the context menu.

Note If the *Report Window* is not active, then these menu options will not be available.

If you have several versions of ***TEDDS*** on your computer you will see a dialog which asks which version of ***TEDDS*** you want to use.



3. If you always want to use this version of ***TEDDS*** click **Always use this version**.

4. If this version of ***TEDDS*** is not running, then it will start. After a short delay you will see your exported report in ***TEDDS***.

To transfer the report to Microsoft Word

1. Set the content of your report and create it.

2. With the ***Report Window*** active select *File/Export Report to Word...*

Mouse Click anywhere over a report view with the right mouse button. Select *Export Report to Word...* from the context menu.

Note If the *Report Window* is not active, then these menu options will not be available.

3. If ***Word*** is not open, then it will be started. Your report will be transferred to ***Word*** via the ***Windows*** clipboard.

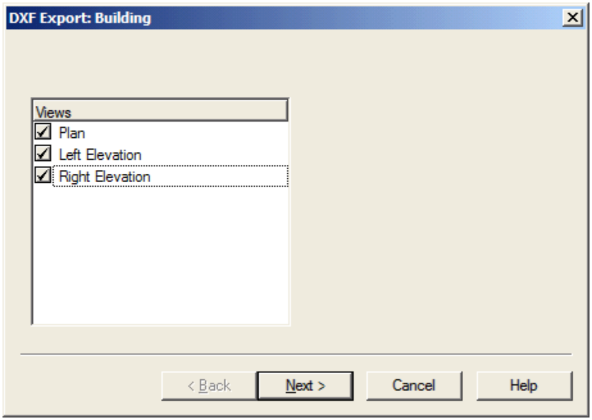
To transfer drawings to a CAD system

You can create ***.dxf*** file in ***Portal Frame*** that you can subsequently use in any CAD system that is capable of opening or importing this format.

Tip Complete your design before creating *.dxf* files, since this maximises the amount of data that is available for transfer.

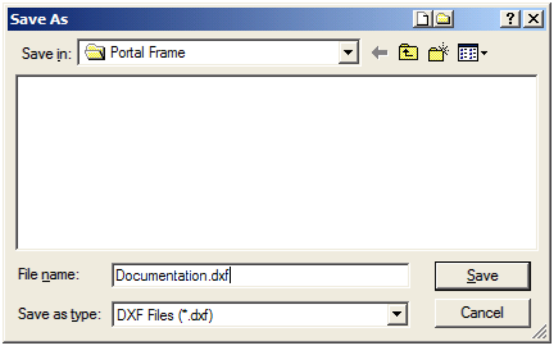
1. Ensure that the window whose details you want to put into the dxf file is active.

2. Select *File/Export to DXF Files…*

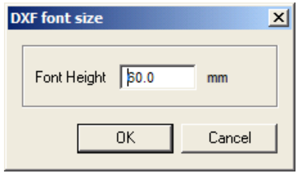


This dialog allows you to set the details that you want in the .dxf files. The options will depend on the type of window which is active.

3. Pick the views that you want to create by ticking the appropriate boxes. Once you have done this click **Next >**.



4. Enter the name of the dxf file you want to create and then click **Save**.



5. Enter the appropriate height. This depends on the scale at which you expect to print the dxf file once it is created.

6. Click **OK** to create the file. You can now import it into your CAD system.

31 The Project Workspace

The ***Project Workspace*** gives you a complete overview of the entire project.

You can choose whether or not you want to display the ***Project Workspace***. You can also choose the position of the ***Project Workspace***, docking it to a side of the ***Portal Frame*** window, or placing it anywhere on your ***Windows*** desktop.

Help For further information on showing and hiding the *Project Workspace* **see** *"To remove the Project Workspace"* on page 102 and *"To reinstate the Project Workspace"* on page 102.

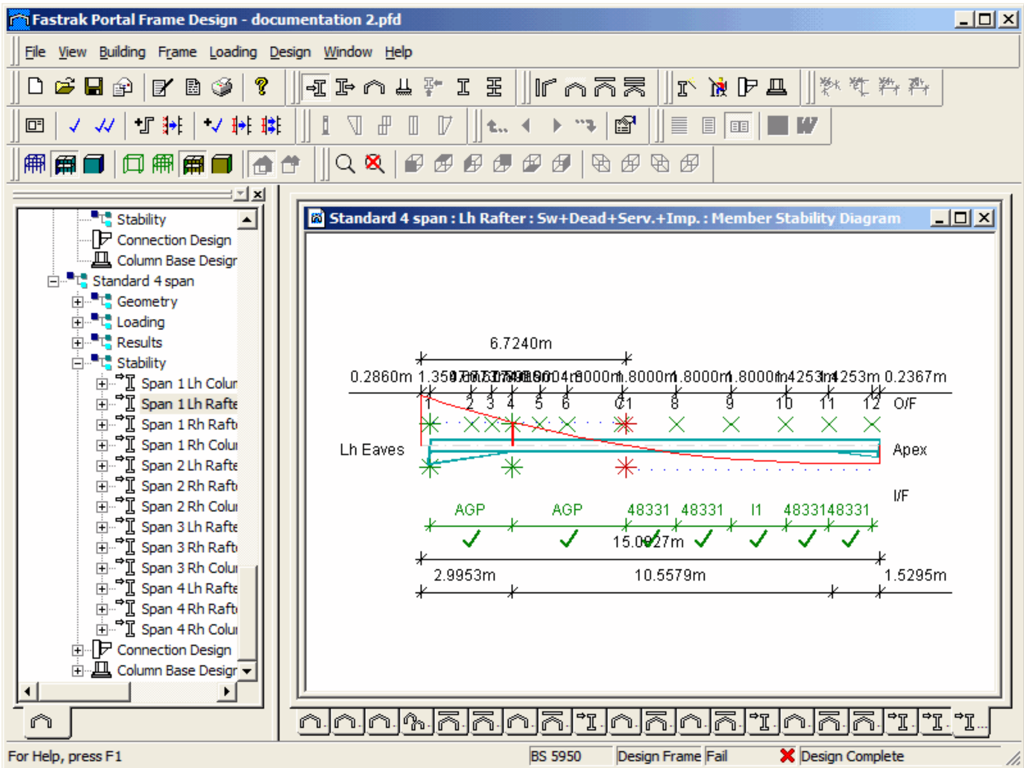
Help For further information on docking the *Project Workspace* **see** *"To choose a docked or a floating Project Workspace"* on page 102.

Using the project workspace

The ***Project Workspace*** allows you to choose a particular frame to work to view its details, to copy or delete it quickly and easily.

To choose the frame on which to work

When you are working on a project which contains a large number of frames you might not be able to identify a particular frame from the workbook tabs, much less the individual window you require. If the windows for the frame have been closed, you can not select the frame using the options of the *Window* menu. You can always select any window using the ***Project Workspace***.



1. Simply open the various items in the Project Workspace until you can see the line for the information you require, then double click that line to open the window.

32 Sharing details with Moment Connection Design

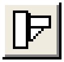
It is simple to transfer the details for one or more connections from ***Portal Frame*** into ***Moment Connection***. You can also copy the details for one connection (bolt layout, stiffeners, welds etc.) to another similar connection and then check that the connection is adequate to carry its own forces and moments.

To check a connection directly

1. Once your frame has been designed you will see that all connections on the graphical display are shown with a blue box around them and with blue text. This indicates that the direct link with moment connection design is active and available. If you click anywhere in the blue box, then you will be instantly transferred to ***Moment Connection*** which will already have all the appropriate details for that connection.

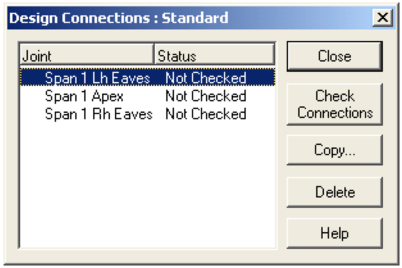
2. Define and check the connection in the usual manner. Once your design is complete When you have achieved a satisfactory solution select *Connection/ Return to Portal Frame* to return the connection details in their entirety to ***Portal Frame***. You will see that the ***Design Connections*** dialog shows the current status of the design as the information in ***Portal Frame*** is updated. You can then include the connection in your ***Portal Frame*** report.

Note *Moment Connection* asks if you want to save the connection. This is to allow you to create a copy that you can open without running *Portal Frame*. If you do not want such a file, then you should click **No** at this dialog.



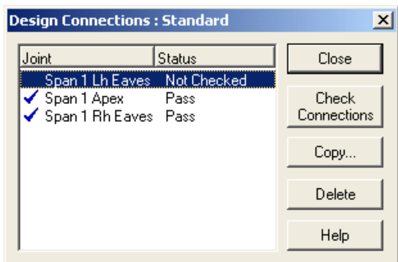
To select connections to check

1. Select *Design/Check Connections…* You will see the ***Design Connections*** dialog.



2. Select the connections whose designs you want to check and then click **Check Connections**. You will be instantly transferred to ***Moment Connection*** which will have all the details for the selected connections.

3. Define and check the connections in ***Moment Connection*** in the usual manner. When you have achieved a satisfactory solution for a connection select *Connection/Return to Portal Frame* to return the details for this connection to ***Portal Frame***. You will see that the ***Design Connections*** dialog shows the current status of the design as the information in ***Portal Frame*** is updated.

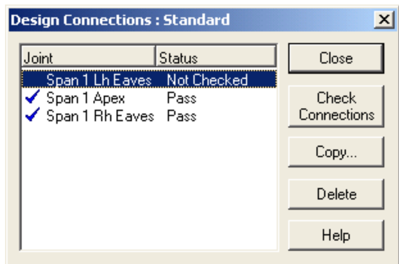


4. Once you have checked the connections close ***Moment Connection.*** All the connections whose details you have chosen to return to ***Portal Frame*** are available for you to include in your ***Portal Frame*** report.

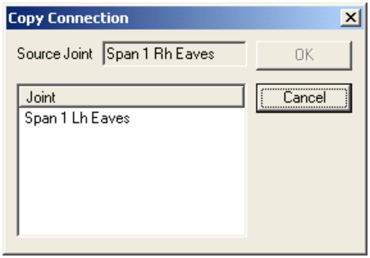
Note *Moment Connection* asks if you want to save the connections. This is to allow you to create a copy that you can open without running *Portal Frame*. If you do not want such a file, then you should click **No** at this dialog.

To copy details from one connection to another

1. Select *Design/Check Connections…* You will see the ***Design Connections*** dialog.



2. Select the connection whose details you want to copy and then click **Copy…** to see the ***Copy Connection*** dialog which will show the source connection and those connections which can receive the copied data.



Note You can copy the details from the **Source Connection** to any or all of the connections in the list.

Note With this version of *Portal Frame* you can only copy between like connections:

- External eaves to external eaves,
- Apex to apex (all rafter/rafter connections for flat top and mansard portals),
- Internal eaves to internal eaves.

You cannot copy valley connections with this release of *Portal Frame*.

3. Click **OK** to copy the data from the source to the target connections. Once the copy is complete you will be returned to the ***Design Connections*** dialog from where you can launch ***Moment Connection Design*** on each of the copied connections.

33 Sharing details with Column Base Design

It is simple to transfer the details for one or more bases from ***Portal Frame*** into ***Column Base***. You can also copy the details for one base (bolt layout, stiffeners, welds etc.) to another similar base and then check that the base is adequate to carry its own forces and moments.

To check a base directly

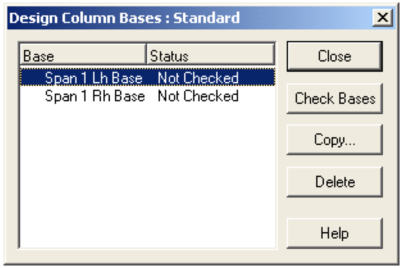
1. Once your frame has been designed you will see that all bases on the graphical display are shown with a blue box around them and with blue text. This indicates that the direct link with column base design is active and available. If you click anywhere in the blue box, then you will be instantly transferred to ***Column Base*** which will already have all the appropriate details for that base.

2. Define and check the base in the usual manner. Once your design is complete close ***Column Base***. When you have achieved a satisfactory solution select *Column Base/Return to Portal Frame* to return the base details in their entirety to ***Portal Frame***. You will see that the ***Design Bases*** dialog shows the current status of the design as the information in ***Portal Frame*** is updated. You can then include the base in your ***Portal Frame*** report.

Note *Column Base* asks if you want to save the base. This is to allow you to create a copy that you can open without running *Portal Frame*. If you do not want such a file, then you should click **No** at this dialog.

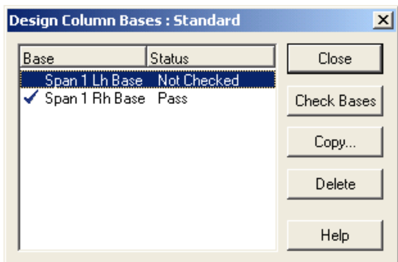
To select bases to check

1. Select *Design/Check Bases…* You will see the ***Design Bases*** dialog.



2. Select the bases whose designs you want to check and then click **Check Bases**. You will be instantly transferred to ***Column Base*** which will have all the details for the selected bases.

3. Define and check the bases in the usual manner. When you have achieved a satisfactory solution for a base select *Column Base/Return to Portal Frame* to return the details for this base to ***Portal Frame***. You will see that the ***Design Bases*** dialog shows the current status of the design as the information in ***Portal Frame*** is updated.

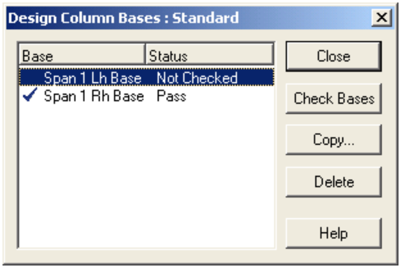


4. Once you have checked the bases close ***Column Base.*** All the bases whose details you have chosen to return to ***Portal Frame*** are available for you to include in your ***Portal Frame*** report.

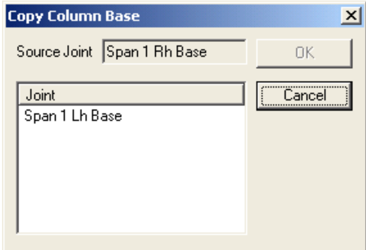
Note *Column Base asks if you want to save the bases. This is to allow you to create a copy that you can open without running Portal Frame. If you do not want such a file, then you should click **No** at this dialog.*

To copy details from one base to another

1. Select *Design/Check Bases…* You will see the ***Design Bases*** dialog.



2. Select the base whose details you want to copy and then click **Copy...** to see the ***Copy Base*** dialog which will show the source base and those bases which can receive the copied data.



Note *You can copy the details from the* **Source Base** *to any or all of the bases in the list.*

Note *You can only copy between bases which have the same column size.*

3. Click **OK** to copy the data from the source to the target bases. Once the copy is complete you will be returned to the ***Design Bases*** dialog from where you can launch ***Column Base Design*** on each of the copied bases.

34 Other Features

The previous chapters of this ***User's Guide*** have covered the aspects of ***Portal Frame*** that you will use to define and design or check a portal frame. This chapter deals with other features that you might find useful to tailor ***Portal Frame*** even further.

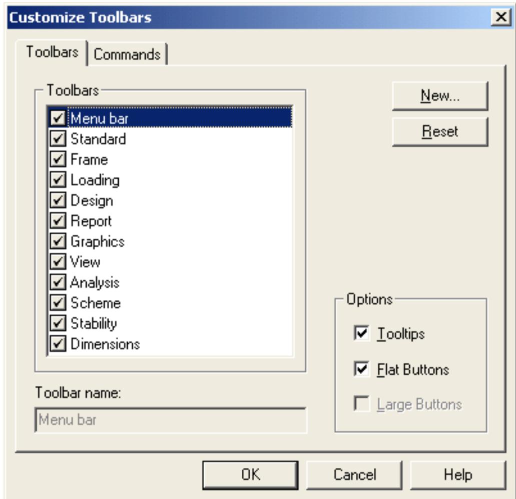
Customizing the menu and toolbars

The ***Portal Frame*** menu and toolbars allow you to access all its features. You control the look of the toolbars and you can tailor them and the menu to suit your particular requirements; for instance you might:

- include only those features that you normally use,
- add new toolbars including particular groupings of those icons and/or menus that compliment each other etc. … …

To set menu and toolbar options

1. Select *View/Toolbar* and then select the *Customize…* option from the cascading menu that appears to see the ***Customize Toolbars*** property sheet.



Caution Whenever this property sheet is open you can manipulate any *Portal Frame* toolbar or menu bar. **Please take care to ensure that your changes are intentional**.

2. Set the options you require:

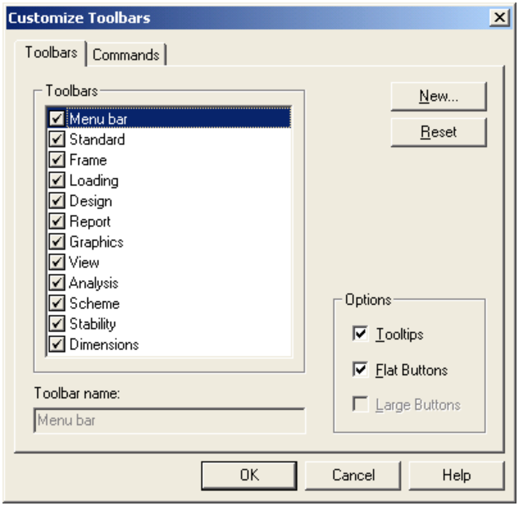
   ***Tooltips* -** Check this option to see a tip indicating the effect of any toolbar icon when you allow the pointer to rest over it.

   ***Flat Buttons* -** Check this option to see the edges of an icon only when the pointer is over it or uncheck it to see the edges at all times.

3. Once your options are set correctly click **OK** to register these and return to the main ***Portal Frame*** window.

To customize the menu bar and toolbars

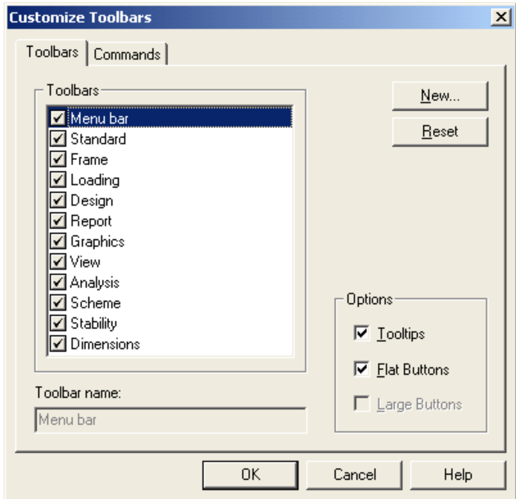
1. Select *View/Toolbar* and then select the *Customize…* option from the cascading menu that appears.



You can now pick the toolbars that are to be displayed, remove icons from a displayed toolbar, move icons between toolbars, create new toolbars, add icons to toolbars. Each of these options is covered in more detail below.

To choose the toolbars that are displayed

1. Select *View/Toolbar* and then select the *Customize…* option from the cascading menu that appears.



The ***Toolbars*** page contains a list of all the current toolbars.

2. To the left of the list of **Toolbars** will see a tick box associated with each toolbar name. Tick the box to display the toolbar or remove the tick to hide it.

To remove icons from a toolbar

1. Ensure that you can see the toolbar containing the icon that you want to delete.

2. Drag and drop the icon from the toolbar anywhere on the screen, but not over another toolbar or the menu bar.

To move icons between toolbars

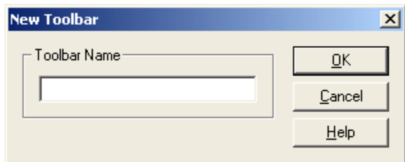
1. Ensure that you can see both toolbars.
2. Drag and drop the icon from one toolbar to the correct position in the other.
3. The icon will be moved from one toolbar to the other.

Note The icon appears in the toolbar as near as possible to the place where you drop it. The other icons are moved to create space if necessary.

Tip You can change the order of icons within a toolbar by dragging and dropping an icon within the toolbar.

To create a new toolbar

1. Click **New…**



2. Enter the **Toolbar Name**, and then click **OK**. A new empty toolbar with this name is created towards the top left of the ***Portal Frame*** window.

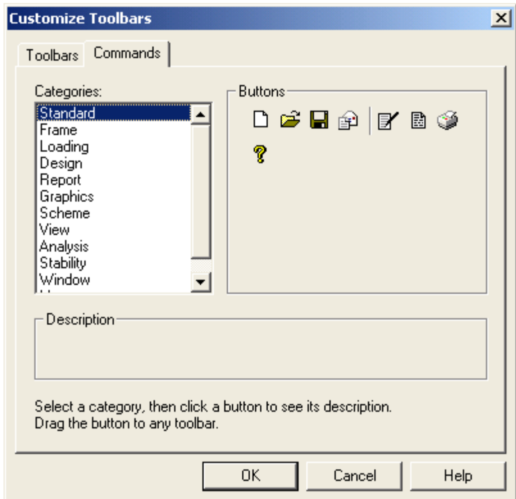
Tip Ensure that the name identifies the toolbar commands otherwise you will need to show the toolbar to identify its commands.

Note If you do not give the toolbar a name, then it will appear as (noname) in the list of toolbars.

Note You can change the name of a toolbar you have created whenever you want by typing over the existing **Toolbar Name**.

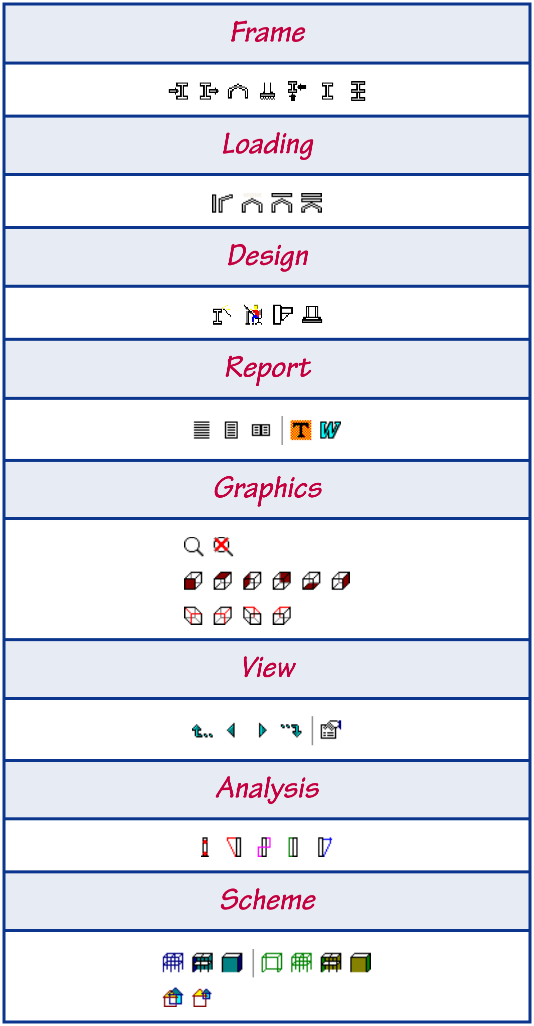
To add new icons to a toolbar

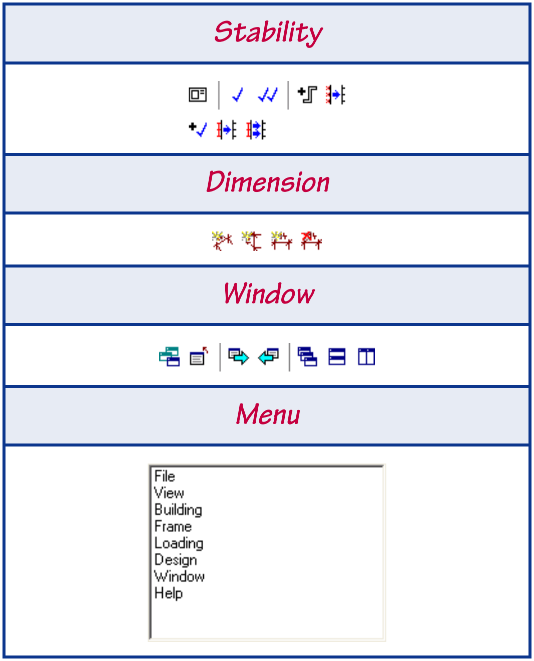
1. Select *View/Toolbar* and then select the *Customize…* option from the cascading menu that appears. Now pick ***Commands*** to see that page.



The left hand list shows the available **Categories**, while the right hand list shows the icons or items that are available for each category. These are detailed below:

Standard



Frame
Loading
Design
Report
Graphics
View
Analysis
Scheme



2. Pick the appropriate category and then drag and drop the icon or menu item of your choice to the toolbar or menu where you want to add it.

3. Once your toolbar or menu layout is complete close the ***Customize Toolbars*** dialog and enjoy using your own personalised toolbar.

Tip *If you drag an existing icon in a toolbar slightly to the right, then this will create a separator just to the left of the moved icon. The dragged icon and any icons to its right will be moved to create room for the separator.*

To delete a custom toolbar

1. Pick the toolbar that you want to delete from the list of toolbars.

2. Click **Delete** and the toolbar will be permanently deleted.

Caution *Once a toolbar has been deleted it cannot be recovered. You will have to repeat the process that you used to create it.* ***Please take due care when using this facility.***

Note *If you pick one of the standard toolbars provided with Portal Frame, then* **Delete** *will not be available. Standard toolbars cannot be deleted, they can only be reset to their initial configuration.*

To reset a standard toolbar

1. Pick the toolbar that you want to reset from the list of toolbars.

2. Click **Reset** and the toolbar will be reset to the layout that was used when *Portal Frame* was installed.

Caution *Once a toolbar has been reset any customizing cannot be recovered. You will have to repeat the process that you used to customize it in the first place.* ***Please take due care when using this facility.***

Note *If you pick a created toolbar* **Reset** *is not available. You can not reset a created toolbar, you can only delete it.*

A Property Files and Order Files

For maximum flexibility ***Portal Frame*** uses pairs of files to control the sections that it chooses. ***Property Files*** contain the details for the sections themselves while ***Order Files*** control which sections you want to use and the sequence in which they will be tried during the design.

Property Files

The property files that are available for use in ***Portal Frame*** are:

- United Kingdom and European sections - Euro.dls
    - Universal Beams
    - Universal Columns
    - Rolled Steel Joists
    - Rolled Steel Channels
    - Rectangular Hollow Sections
    - Square Hollow Sections
    - Circular Hollow Sections
    - Structural Tee from UB
    - Structural Tee from UC
    - Equal Angles
    - Unequal Angles
    - Flat Bars
    - Parallel Faced Flange Beams
    - Wide/Very Wide Flanged Beams
    - Wide Flanged Columns

- Australian sections - Aussie.dls
  - Universal Beams
  - Universal Columns
  - Rolled Steel Joists
  - Rolled Steel Channels
  - Rectangular Hollow Sections
  - Square Hollow Sections
  - Circular Hollow Sections
  - Structural Tee from UB
  - Structural Tee from UC
  - Equal Angles
  - Unequal Angles
- Japanese sections - Japan.dls
  - Universal Beams
  - Universal Columns
  - Rolled Steel Joists
  - Rolled Steel Channels
  - Rectangular Hollow Sections
  - Square Hollow Sections
  - Circular Hollow Sections
  - Equal Angles
  - Unequal Angles
- United States sections - aiscsc.dls
  - W & M

- H
- S
- WT & MT
- Channels
- Misc. Channels
- Angles
- Double Angles (Equal)
- Double Angles (long legs back to back)
- Double Angles (short legs back to back)
- Square Tubes
- Rectangular Tubes
- Pipes

Order files

The names of the order files that are available are:

- United Kingdom sections
- IntColumnOrder.Eur order file for internal columns
- ExtColumnOrder.Eur order file for external columns
- RafterOrder.Eur order file for rafters
- European sections
- HDColumnOrder.Eur order file for HD sections
- HEBeamOrder.Eur order file for HE sections
- HLBeamOrder.Eur order file for HL sections
- HXBeamOrder.Eur order file for HX sections
- IPEBeamOrder.Eur order file for IPE sections

- Australian sections
- IntColumnOrder.Aus   order file for internal columns
- ExtColumnOrder.Aus   order file for external columns
- RafterOrder.Aus   order file for rafters
- Japanese sections
- IntColumnOrder.Jpn   order file for internal columns
- ExtColumnOrder.Jpn   order file for external columns
- RafterOrder.Jpn   order file for rafters
- United States sections
- IntColumnOrder.Usa   order file for internal columns
- ExtColumnOrder.Usa   order file for external columns
- RafterOrder.Usa   order file for rafters

Help For further information on order files **see** *“To set design groups”* on page 418.

Caution The sections that are included / excluded are remembered for a particular order file. Thus if you exclude sections in an order file they will remain excluded for all designs until you decide to include them again.

Portal Modeller

Portal Frame Design
Release 4.0

1 The Portal Modeller[1]

The *Portal Modeller* allows you to define the ancillary steelwork for your portal structure including the:

- gable posts,
- roof bracing,
- side bracing,
- gable bracing,
- gable rails,
- standard cold-rolled sections,
- hip purlins,
- floor joists, and
- eaves ties,

which make up your total portal model. In this document each of these member types is referred to as an element.

The *Portal Modeller* also includes wizards which allow you to define:

- hips (including the hip raker and jack rafters),
- floors, and
- standard cold-rolled sections,

quickly and easily.

1. *This is an additional plug-in module that you purchase separately to* ***Portal Frame****.*

Floor wizard overview

The floor wizard allows you to create a floor over an area of your building which you choose. You can define the details for:

- the level of the floor and its reference,
- the area the floor covers,
- the section size details for the floor beams,
- the section size details for the floor joists (which run between the stanchions of your main frame and between the tops of any floor props, tying them together,
- the section size details of any floor props you define.

? **Help** *see* "Using the Floor Wizard" on page 541.

Hip wizard overview

The hip wizard caters for:

- structures where there are no intermediate frames between that where the hip starts and that where it ends, and
- structures where there are intermediate frames between that where the hip starts and that where it ends. In this case the wizard automatically changes the intermediate frames to flat-top portals with the appropriate span details.

In either case the hip wizard automatically converts the frame where the hip ends to a monopitches.

The hip wizard also allows you to define:

- the size of the hip rakers[1],
- the number and size of the jack rafters[2] and

1. *Since the hip wizard performs all the necessary calculations for you it is the only way of defining hip rakers.*
2. *Since the hip wizard performs all the necessary calculations for you it is the only way of defining jack rafters.*

- the number and size of the gable posts. These can have their major axis rotated to be at 90° to that of the hip end frame's main sections.

? **Help** *see* *"Using the Hip Wizard"* on page 562.

Cold-rolled wizard overview

The cold-rolled sections wizard allows you to convert the restraints which you have defined for the members of your portal frames into true cold rolled sections. Although the display of the true cold-rolled section shapes is currently beyond the scope of ***Portal Frame***, when you export details either to a `.dxf` file, or to ***3D+*** the true section information will then be transferred. You can define the details for:

- the purlins,
- the side rails,
- the eaves beams

which your building is to contain. The wizard then generates runs of purlins, side rails, gable rails and eaves beams. Where necessary the wizard also creates any hip purlins that your structure requires.

? **Help** *see* *"Using the Cold Rolled Sections Wizard"* on page 576.

Working with elements

You use the ***Select*** toolbar to control the element with which you want to work, the action that you want to perform and the way in which you want to select these elements.

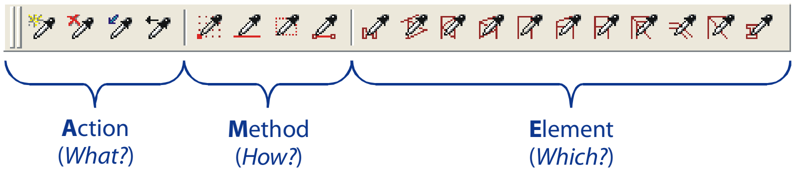
? **Help** For a full review of the *Select* toolbar *see* *"The Select toolbar"* on page 57. A brief summary follows.

You place and work with all ***Portal Frame*** elements in a similar manner, following the simple procedure below. Once you are familiar with this you will find that it is an intuitive way of working.

- ***A*** – ***A***ction, what you want to do,
- ***M*** – ***M***ethod, how you want to do it,
- ***E*** – ***E***lement, on which type of element you want to do it.

You might find it helpful to remember the mnemonic ***AME*** (***A***ction, ***M***ethod, ***E***lement).

You set each of these by choosing icons from the appropriate area of the ***Select*** toolbar. These are shown below:



Note In some cases you may have to choose the ***Element*** type with which you want to work in order to be able to set the ***Action*** and ***Method*** since not all ***Actions*** and ***Methods*** are appropriate for all ***Element*** types.

For your convenience all the available buttons are included in the table below, with a brief description of their effect:

Type	*Icon*	*Icon Effect*
Action		**Create** - Creates members of the current element type which you choose using the current method.

Type	Icon	Icon Effect
Action		**Delete** - Deletes elements of the current element type which you choose using the current method.
Action		**Set Attributes** - Resets the properties of the current element type which you choose using the current method.
Action		**Move** - Moves objects of the current element type which you choose using the current method.
Method		**Grid Points** - Determine which elements are to be actioned by picking grid points in your structure.
Method		**Grid Line** - Determine which elements are to be actioned by picking a grid line in your structure.
Method		**Area** - Determine which elements are to be actioned by dragging across an area in your structure.
Method		**Single** - Determine which elements are to be actioned by picking a single element/ area.
Element		**Gable Post** - Perform the action using the current method on the gable posts in your structure.

Type	Icon	Icon Effect
Element		**Roof Bracing** - Perform the action using the current method on the roof bracing in your structure.
Element		**Side Bracing** - Perform the action using the current method on the side bracing in your structure.
Element		**Gable Bracing** - Perform the action using the current method on the gable bracing in your structure.
Element		**Standard Purlin** - Perform the action using the current method on the cold-rolled purlins in your structure.
Element		**Standard Rail** - Perform the action using the current method on the cold-rolled side rails in your structure.
Element		**Hip Purlin** - Perform the action using the current method on the hip purlins in your structure.
Element		**Gable Rail** - Perform the action using the current method on the gable rails in your structure.
Element		**Eaves Beam** - Perform the action using the current method on the eaves beams in your structure.

Type	Icon	Icon Effect
Element		**Jack Rafter** - Perform the action using the current method on the jack rafters in your structure.
Element		**Hip Raker** - Perform the action using the current method on the hip rakers in your structure.
Element		**Floor Joist** - Perform the action using the current method on the connecting floor beams in your structure.
Element		**Floor Area** - Perform the action using the current method on the floor areas in your structure.

2 Using the Floor Wizard

The ***Floor Wizard*** allows you to add a floor to your model. This is a two stage process:

- add the floor into your structure,
- enter the floored area's details:
  - the extent of the floor,
  - the details of the:
    - floor beams,
    - joists, and
    - props

  within the floored area.

Caution **In this release of** *Portal Frame* **the floors are not involved in the design of the portal frames**. Floors are merely a way of adding ancillary steelwork into your model.

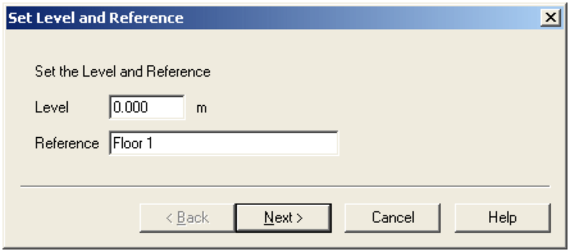
To use the floor wizard

Create the frames in your structure in the usual manner, and use the ***Building Grid*** dialog (*Building/Grid…*) to position these. You do not need to make any allowance for the floor in your structure when you do this.

You must use the ***Structure*** window to define the floors in your structure when using the ***Floor Wizard***[1].

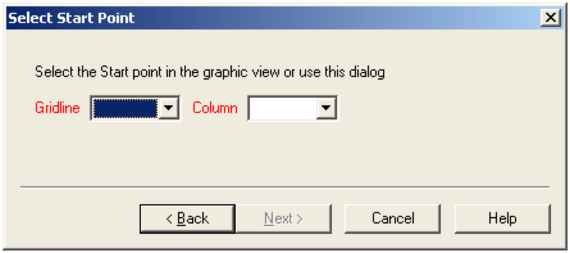
1. *The* **Floor Wizard** *icon is only available if you have purchased, installed and unlocked the* ***Portal Modeller****. If you have not done so, then the* ***Structure*** *window is not available and the* **Floor Wizard** *icon is dimmed.*

1. Click the ***Floor Wizard*** icon to start the definition of your floor. You will see the ***Floor Level*** dialog.

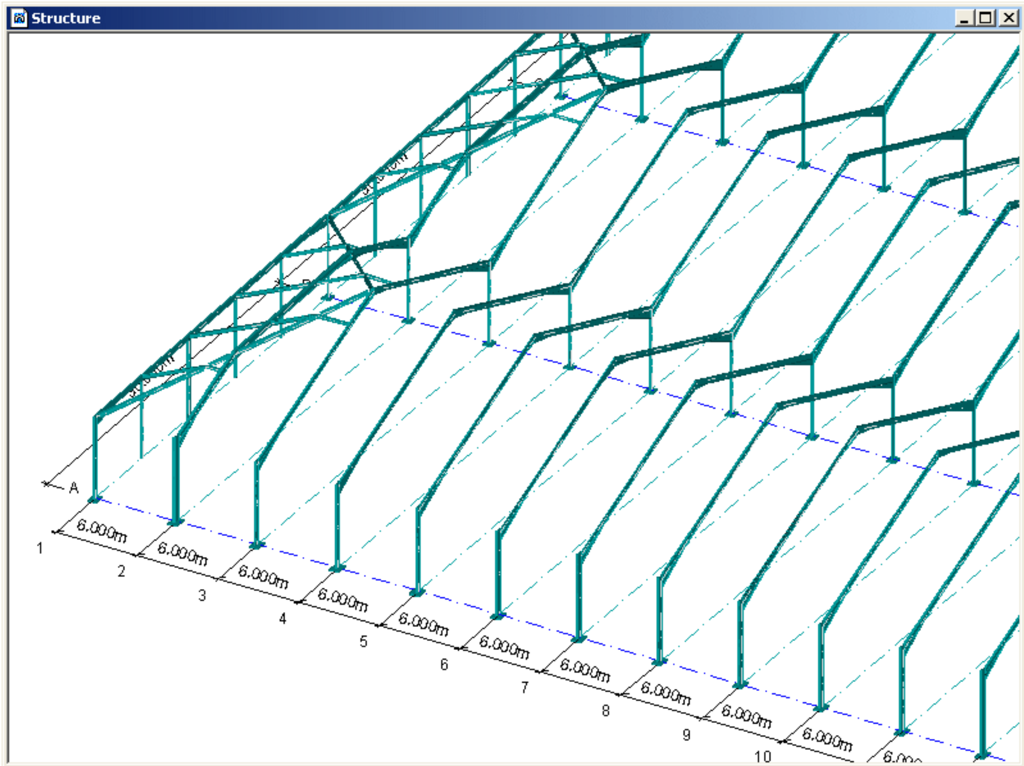


2. Enter the **Level** of your floor and give it a **Reference** for your convenience.

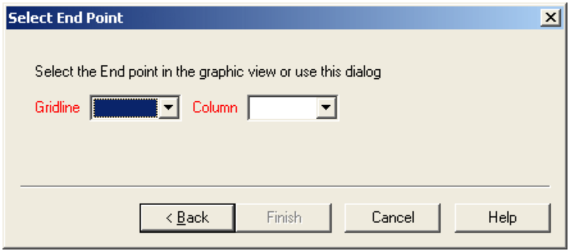
3. Once these details are correct click **OK**. You will see the ***Select Start Point*** dialog which allows you to define the reference of the grid point where your floor starts.



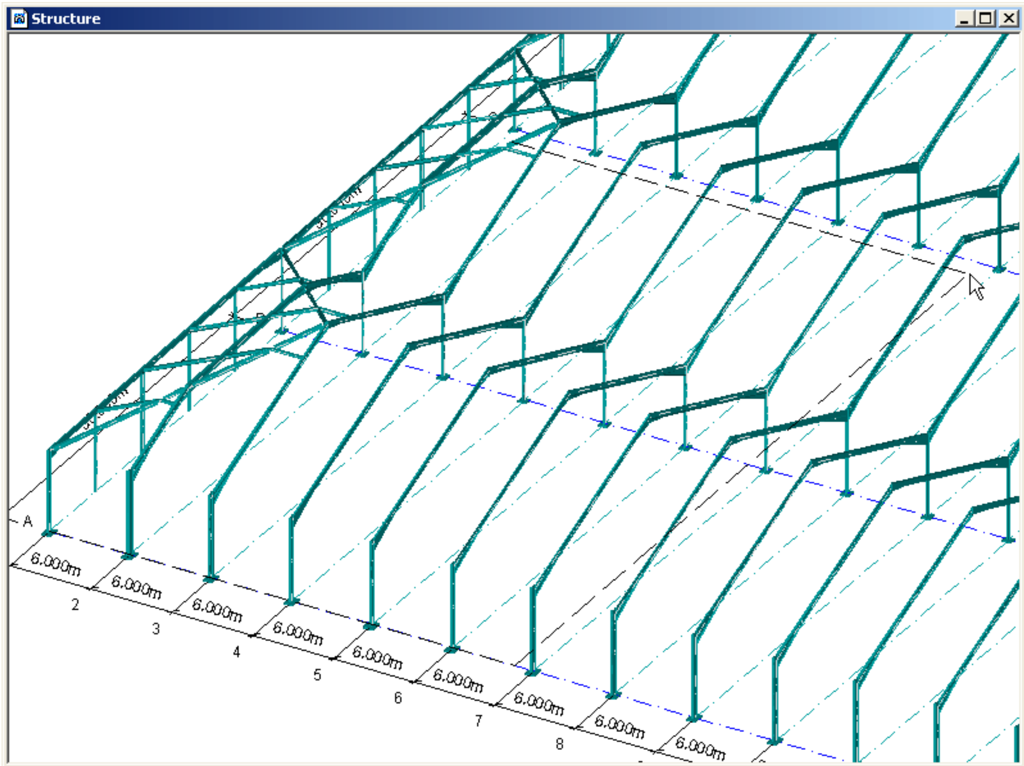
You can either use the dialog to choose this location by picking the appropriate references from the list and then clicking **Next>**, or alternatively you can pick the intersection point directly from the graphical display.



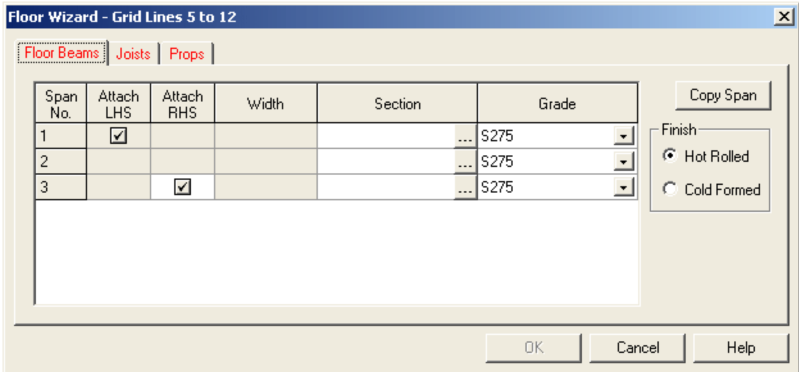
4. Once you have used either method you will see the ***Select End Point*** dialog which allows you to define the reference of the grid point where your floor ends.



Again you can either use the dialog to choose this location by picking the appropriate references from the list and then clicking **Next>**, or alternatively you can pick the intersection point directly from the graphical display. If you use the latter option, the you will see a rectangle which indicates the area over which the floor will be created.



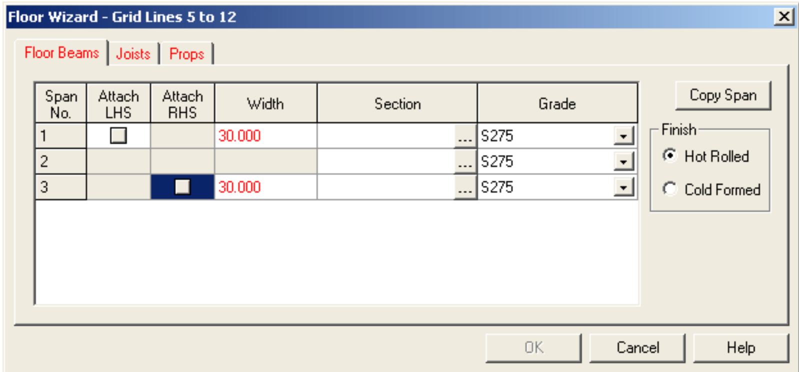
5. Once you have used either method you will see ***Floor Wizard*** property sheet which allow you to define the details of the members which make up the floor.



Comment The capture above does not relate to the area shown previously (which placed the floor over two spans). The above capture is for a floor over three spans.

Note The floor beams run in the same direction as the span of your portal frame, the joists run between your portal stanchions and the props lie underneath, and support, the floor beams.

6. If the floor extends all the way across the area you have indicated, then leave the ticks against the **Attach LHS** and **Attach RHS** columns, otherwise clear the ticks and the dialog will reconfigure to allow you to specify the width of the floored area.



Note *If you are defining a floored area which extends over more than one span, then if you leave the* **Attach LHS** *button ticked the floor will span across the entire width of the first floored span. Similarly if you leave the* **Attach RHS** *button ticked the floor will span across the entire width of the last floored span. If you remove the ticks against* **Attach LHS** *or* **Attach RHS** *then the width you specify is that from the first internal column in the span outward towards the edge of the frame.*

Example If you have a 3 span frame the columns will lie on column lines A, B, C and D. If you define a floor across the entire width of the frame, then this will run between grid lines A and D. If you then remove the tick against both the **Attach LHS** and **Attach RHS** buttons the width which you specify for **Span No.** 1 is measured from column line B towards column line A, and the width you specify for **Span No.** 3 is measured from grid line C towards grid line D.

Note If your floor area is only over a single span, then either **Attach LHS** or **Attach RHS** must be ticked, otherwise you would have a completely free standing floor with no connection whatever to your portal structure.

7. If necessary enter the **Width** for each edge of the floor as appropriate.

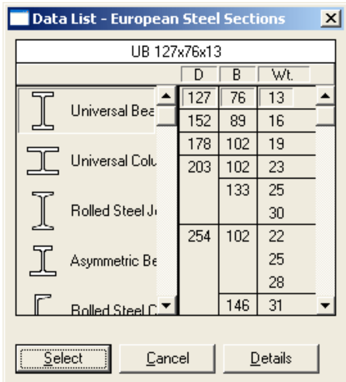
Floor beam details

8. Choose the **Finish** of the floor beams you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

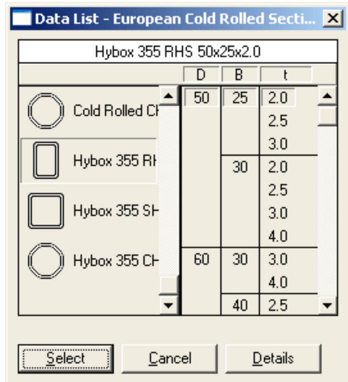
Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

9. To enter the size of your floor beams click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections



Cold Formed Sections



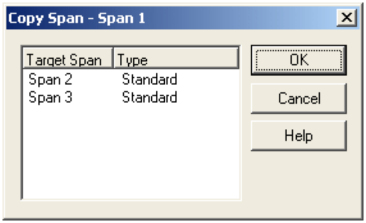
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

10. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the* ***Grade*** *box will show* ***S355****, and you can not change this since these sections are only available in this grade.*

11. If you want to define different floor beams in the other spans covered by your floor, then you need to repeat steps 9 to 10 for each span.

12. Alternatively if you want to use the same details for the floor beams in one or more of the other spans covered by your floor click **Copy Span**.



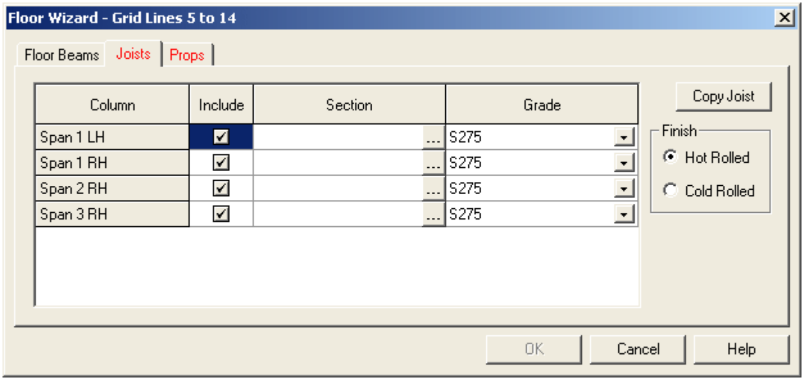
13. Pick the span(s) to which you want to copy the details of your current span.

14. Click **OK** to return to the ***Floor Wizard*** sheet which will show the details copied from the current span in the target span(s).

15. Pick another tab of the ***Floor Wizard*** sheet in order to define the details for that element type. Once the details for all your floor steelwork is correct click **OK** to return to the graphical display of the structure which will be updated to show the floor which you have defined.

Joist details

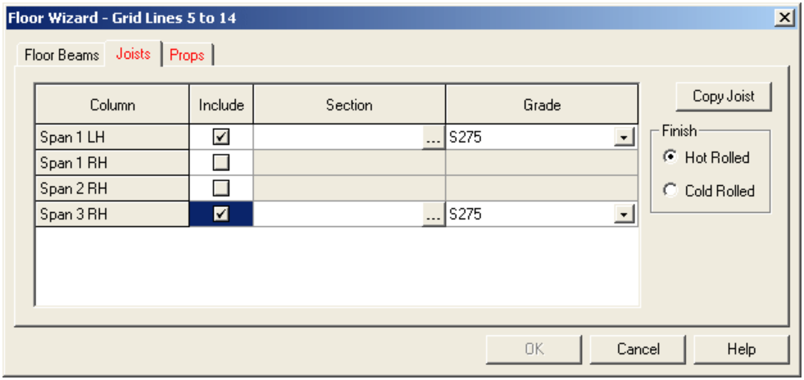
16. Click the ***Joists*** tab of the ***Floor Wizard*** sheet.



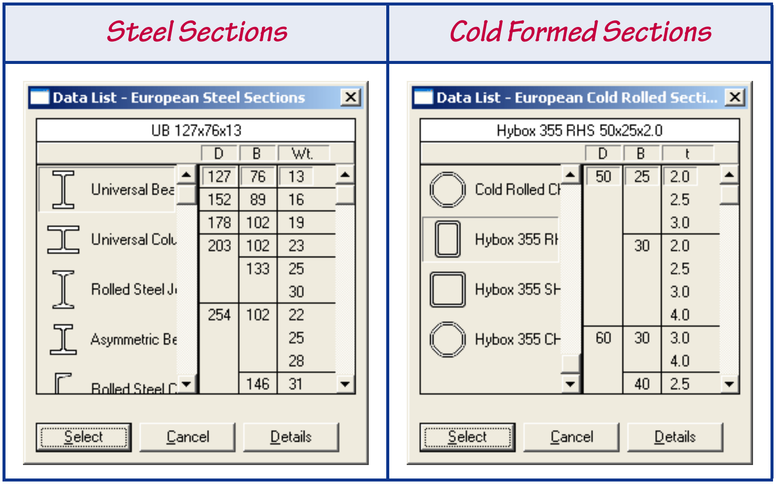
17. Choose the **Finish** of the joists you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

18. You can specify which stanchions are to have joists running between them by ticking the appropriate **Include** boxes. The ***Floor Wizard*** sheet will reconfigure to allow you to define the details of the joists in these locations.



19. To enter the size of your joists click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



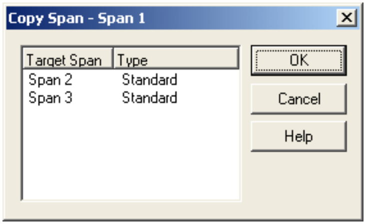
20. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

21. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

22. If you want to define different joists between the stanchions in the other locations covered by your floor, then you need to repeat steps 19 to 21 for each location.

23. Alternatively if you want to use the same details for the joists in one or more of the other locations covered by your floor click **Copy Span**.



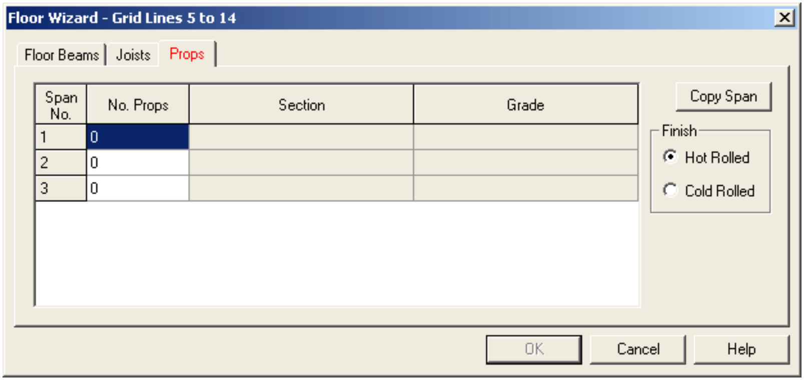
24. Pick the span(s) to which you want to copy the details of your current span.

25. Click **OK** to return to the ***Floor Wizard*** sheet which will show the details copied from the current location in the target location(s).

26. Pick another tab of the ***Floor Wizard*** sheet in order to define the details for that element type. Once the details for all your floor steelwork is correct click **OK** to return to the graphical display of the structure which will be updated to show the floor which you have defined.

Prop details

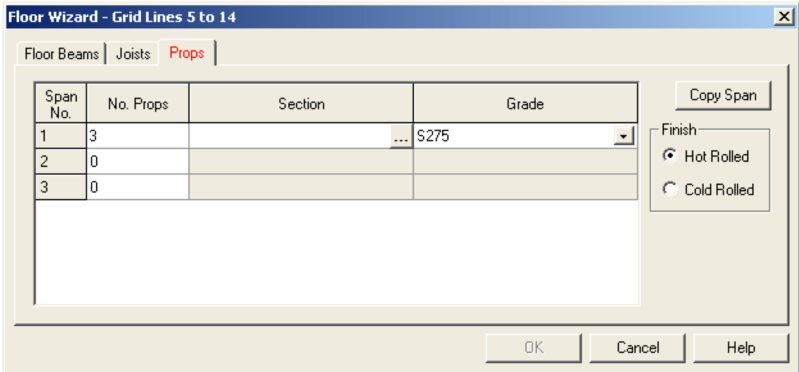
27. Click the ***Props*** tab of the ***Floor Wizard*** sheet.



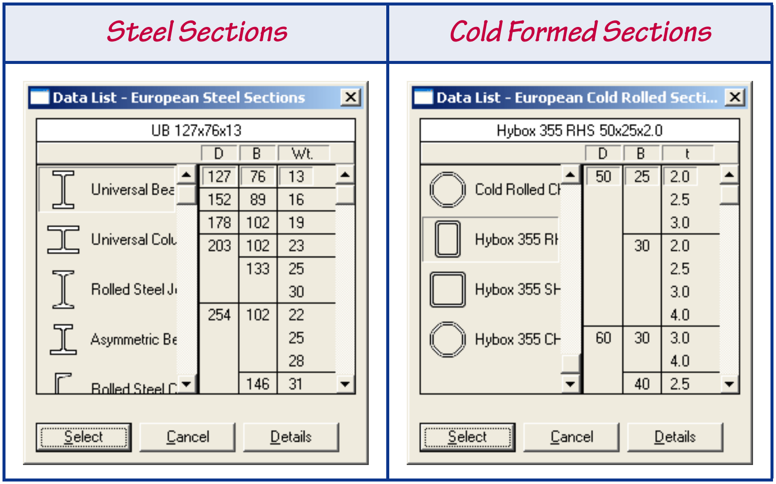
28. Choose the **Finish** of the props you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

29. For each span, covered totally or partially by your floor you can specify the number of props you require by entering a non-zero value for the **No. Props**. The ***Floor Wizard*** sheet will reconfigure to allow you to define the details of the props in these locations.



30. To enter the size of your props click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



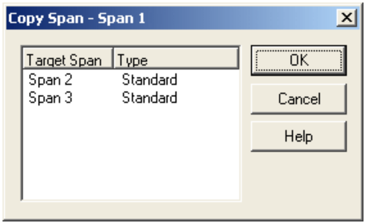
31. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

32. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

33. If you want to define different props between the stanchions in the other locations covered by your floor, then you need to repeat steps 30 to 32 for each location.

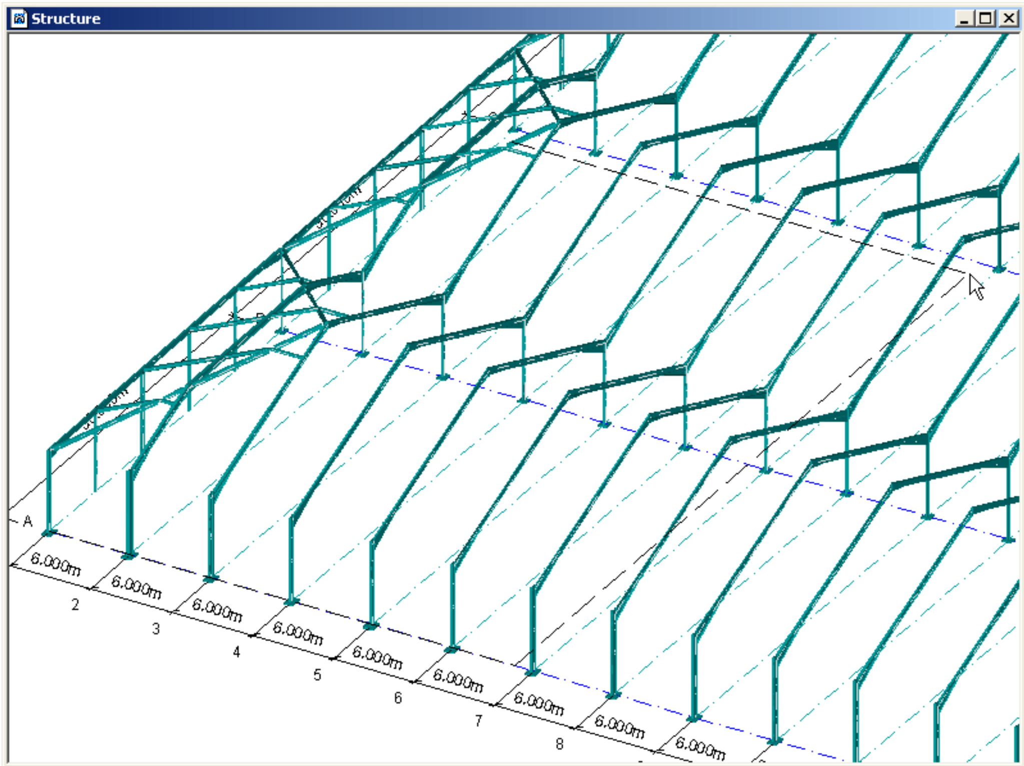
34. Alternatively if you want to use the same details for the props in one or more of the other locations covered by your floor click **Copy Span**.



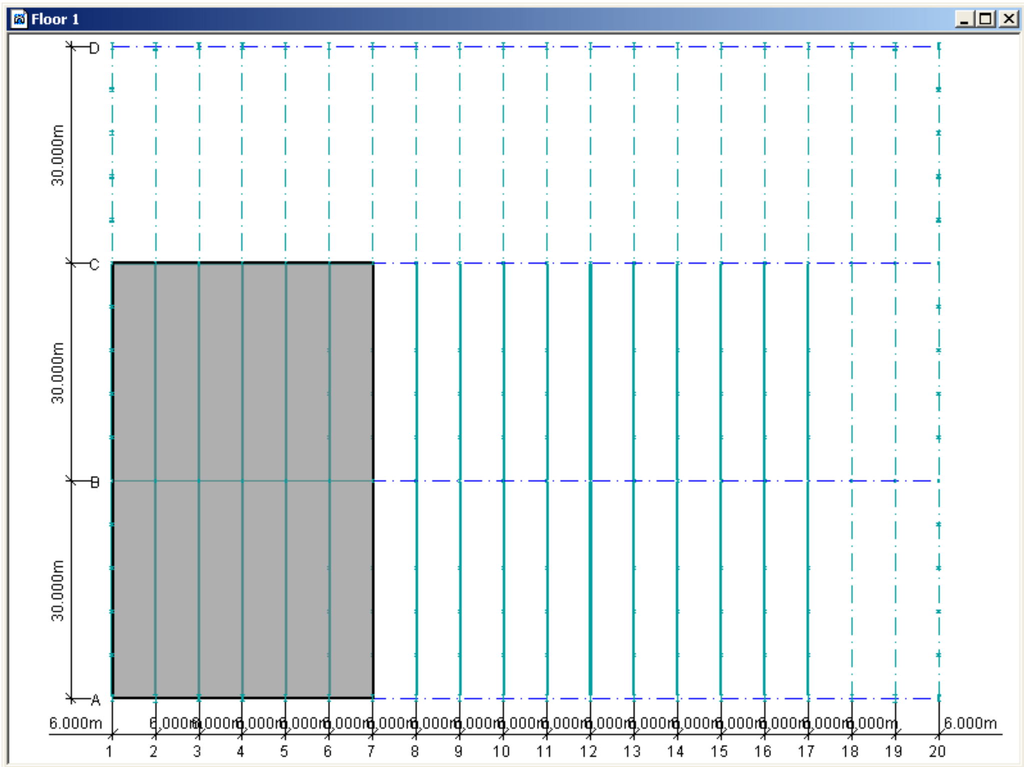
35. Pick the span(s) to which you want to copy the details of your current span.

36. Click **OK** to return to the ***Floor Wizard*** sheet which will show the details copied from the current location in the target location(s).

37. Pick another tab of the ***Floor Wizard*** sheet in order to define the details for that element type. Once the details for all your floor steelwork is correct click **OK** to return to the graphical display. The ***Structure*** window will be updated to show the floor which you have just defined.



You will also see that a new window is added to the display. This is a plan view of the floor which you have defined.



You can use this view to add:

- further areas to your floor using **Floor Area** icon
- further joists to an existing area using the **Floor Joist** icon

both these buttons are on the ***Select*** toolbar.

?

Help

see:

- *"Working with Floor Areas"* on page 810.
- *"Working with Floor Joists"* on page 796.

3 Using the Hip Wizard

The ***Hip Wizard*** allows you to add a hipped area to your model. You simply tell the ***Hip Wizard***:

- where the hip is to start,
- where the hip is to end,

Note The maximum number of intermediate frames between the hip's start and end is 2.

- the section size you want to use for the hip raker,
- the number and size of the jack rafters, and
- the number, size and orientation of the gable posts.

The ***Hip Wizard*** automatically:

- creates a new frame type for the frame where the hip starts (that furthest from the appropriate gable), since it will be subject to a differing load pattern than any other frames of the same type which already exist in your structure,
- converts the frame where the hip ends so that it contains only monopitch spans,
- furthermore if there are intermediate frames between the start and end of the hip, then the ***Hip Wizard*** changes these into flat-top spans and calculates the appropriate geometry.

Any new frames which the ***Hip Wizard*** generates during hip creation are automatically introduced into your model at the correct locations.

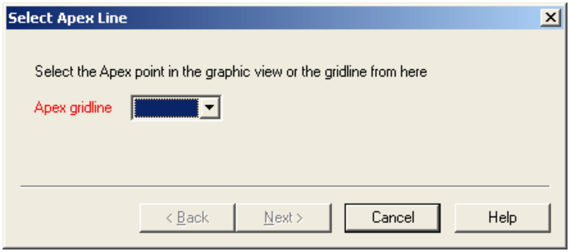
To use the hip wizard

Create the frames in your structure in the usual manner, and use the ***Building Grid*** dialog (*Building/Grid…*) to position these. You do not need to make any allowance for the gable or hipped frames in your structure.

1. Click on, or open the ***Structure*** window[1] for your model.

2. If necessary adjust the view until you can see the area where you want to create your hip clearly.

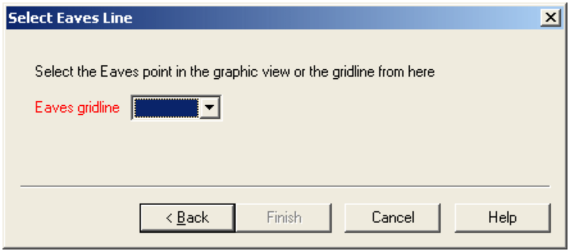
3. Click the ***Hip Wizard*** icon, and you will see the ***Select Apex Line*** dialog which allows you to define the apex point at which the hipped area starts.



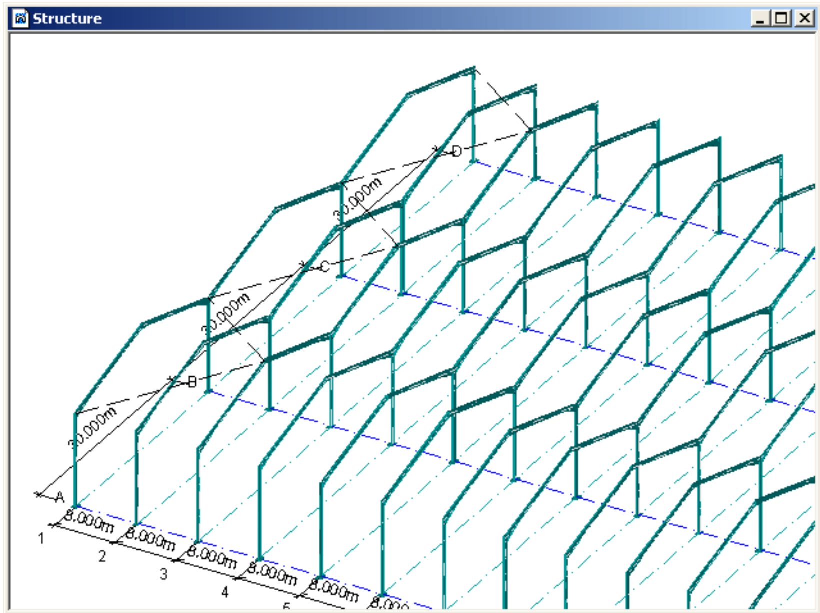
4. You can either use the dialog to choose this location by picking the appropriate reference from the list and then clicking **Next>**, or alternatively you can pick the apex point directly from the graphical display.

1. *The **Structure** window is only available if you have purchased, installed and unlocked the **Portal Modeller**.*

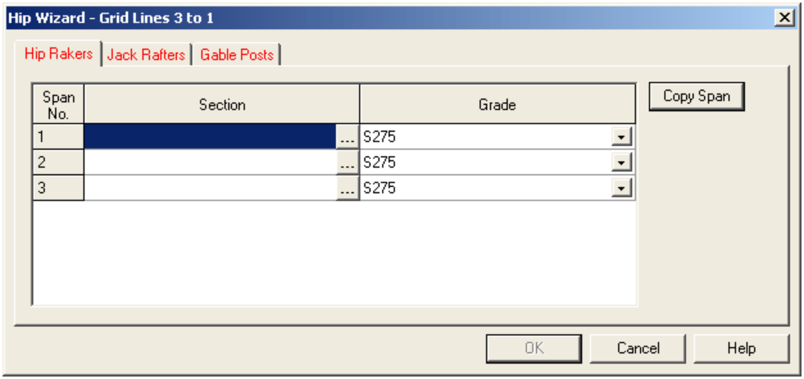
5. Once you have used either method you will see the ***Select Eaves Line*** dialog which allows you to define the eaves point at which the hipped area ends.



Again you can either use the dialog to choose this location by picking the appropriate references from the list and then clicking **Next>**, or alternatively you can pick the intersection point directly from the graphical display. If you use the latter option, the you will see dotted line which represent where the the hip rakers will run with this selection to help you pick the correct point.



6. Once you have defined the area which the hip covers you will see the ***Hip Wizard*** dialog which allows you to define the details of the hip's members.

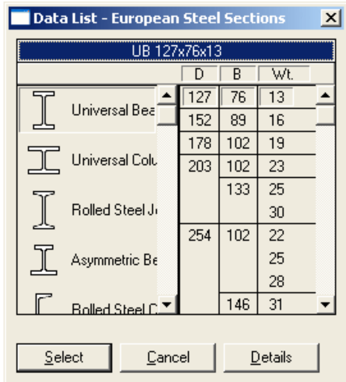


You will see that the ***Hip Wizard*** shows a line for each span in your frame.

Note For clarity we show only one span in the screen captures of our frame.

Hip raker size

7. To enter the size of your hip rakers click the … button to the right of the **Section** column whose raker size you want to set. You will see the ***Steel Section Data List*** for the country which is currently set in your ***Preferences***.

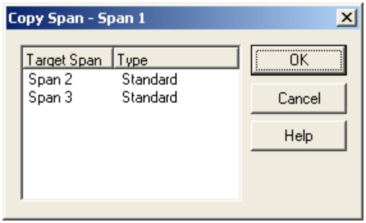


Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

9. If you want to define different hip rakers in the other spans of your frame, then you need to repeat steps 7 to 8 for each span.

10. Alternatively if you want to use the same details for one or more of the other spans in your frame click **Copy Span**.



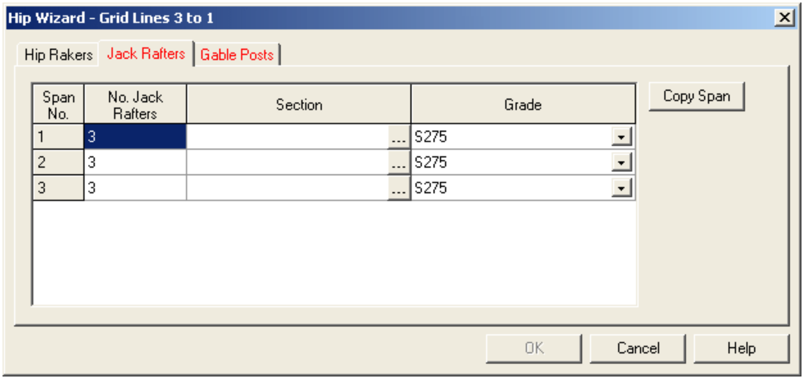
11. Pick the span(s) to which you want to copy the details of your current span.

12. Click **OK** to return to the ***Hip Wizard*** sheet which will show the details copied from the current span in the target span(s).

13. Pick another tab of the ***Hip Wizard*** sheet in order to define the details for that element type. Once the details for all your hip steelwork is correct click **OK** to return to the graphical display of the structure which will be updated to show the hip which you have defined.

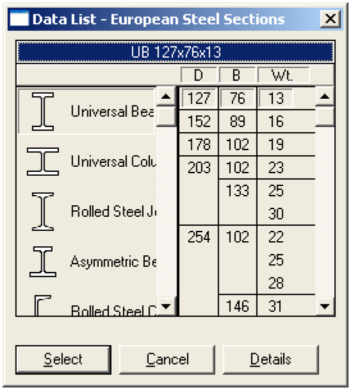
Jack rafter number and size

14. If you can not see the ***Jack Rafters*** page, then click the ***Jack Rafters*** tab to see its details.



15. The number of jack rafters which you need will depend on the span of your frames, simply enter the **No. Jack Rafters** that you need.

16. To enter the size of your jack rafters click the … button to the right of the **Section** column whose jack rafter size you want to set. You will see the ***Steel Section Data List*** for the country which is currently set in your ***Preferences***.

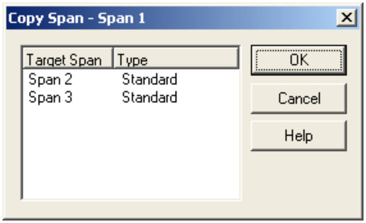


Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Wizard*** dialog which will show your selected size.

17. Pick the **Grade** of material from the list of available grades.

18. If you want to define different jack rafters in the other spans of your frame, then you need to repeat steps 15 to 17 for each span.

19. Alternatively if you want to use the same details for one or more of the other spans in your frame click **Copy Span**.



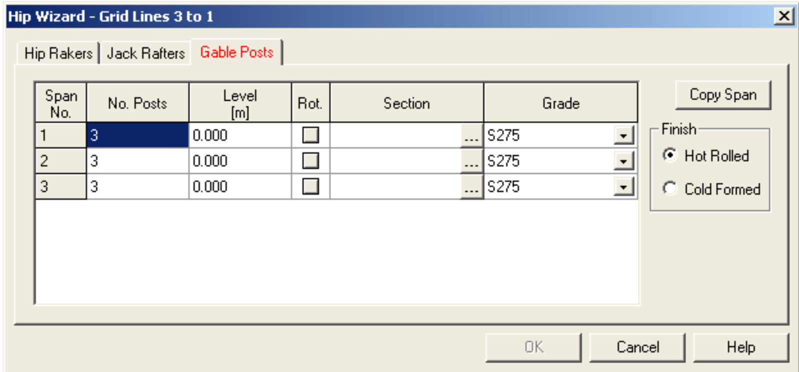
20. Pick the span(s) to which you want to copy the details of your current span.

21. Click **OK** to return to the ***Hip Wizard*** sheet which will show the details copied from the current span in the target span(s).

22. Pick another tab of the ***Hip Wizard*** sheet in order to define the details for that element type. Once the details for all your hip steelwork is correct click **OK** to return to the graphical display of the structure which will be updated to show the hip which you have defined.

Gable post number and size

23. If you can not see the ***Gable Posts*** page, then click the ***Gable Posts*** tab to see its details.



24. The number of gable posts which you need will depend on the span of your frames, simply enter the **No. Posts** that you need.

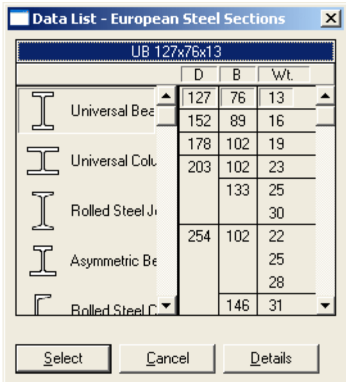
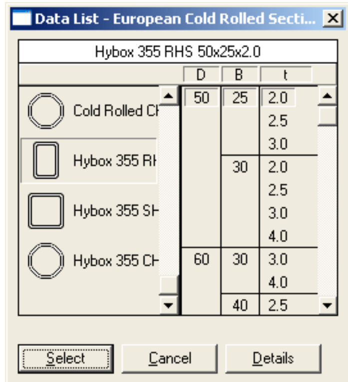
25. Enter the **Level** at which the bases of your gable posts are to lie.

26. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable posts so that their webs are parallel to those of the main frame members, then tick the **Rot.** (rotation) box.

27. Pick the **Finish** that you want your gable posts to use. You can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.*

28. To enter the size of your gable posts click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Rolled Sections*
	

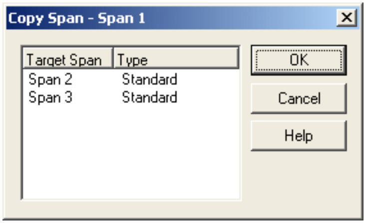
Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Wizard*** dialog which will show your selected size.

29. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.*

30. If you want to define different gable posts in the other spans of your frame, then you need to repeat steps 24 to 29 for each span.

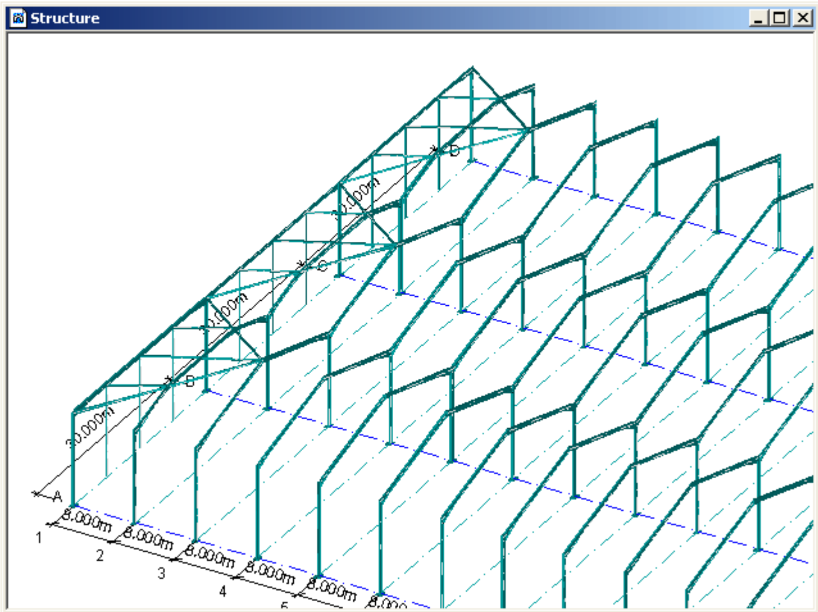
31. Alternatively if you want to use the same details for one or more of the other spans in your frame click **Copy Span**.



32. Pick the span(s) to which you want to copy the details of your current span.

33. Click **OK** to return to the ***Hip Wizard*** sheet which will show the details copied from the current span in the target span(s).

34. Pick another tab of the ***Hip Wizard*** sheet in order to define the details for that element type. Once the details for all your hip steelwork is correct click **OK** to return to the graphical display of the structure which will be updated to show the hip which you have defined.



Note Not only have the hip raker, jack rafter and gable post elements been created, but the frame where the hip ends has been converted to a monopitch portal.

4 Using the Cold Rolled Sections Wizard

The ***Cold Rolled Sections Wizard*** allows you to define the size and type of purlin, side rail and eaves beam which you want to use for your structure[1]. Although the true graphical display of such members is beyond the scope of ***Portal Frame***, which shows a single line, when you transfer the structure to ***3D+*** you will find that all the information relating to the section size, shape and position has been transferred correctly.

When you use the ***Cold Rolled Sections Wizard*** it maps the existing layout of restraints to your frame, overlaying these with the purlin, side rail and eaves beam details you specify. You therefore need to have defined some restraints in your structure before you invoke the ***Cold Rolled Sections Wizard***.

Note This chapter topic deals with the *Cold Rolled Sections Wizard*. You can also add cold rolled sections to your structure in a more interactive manner which gives you more control on the final result. You can also use the interactive methods to alter the cold rolled section details which you have created using the *Cold Rolled Sections Wizard.*

Help cross reference

Understanding the cold rolled sections wizard

Before you use the ***Cold Rolled Sections Wizard*** you should have defined a consistent set of restraints to all members in your structure. The ***Cold Rolled Sections Wizard*** then applies a series of simple rules to create consistent runs of

1. *These details are immaterial to* ***Portal Frame****, which only needs details of the types of restraint and their position in order to determine the out-of-plane stability of the individual frame members.*

purlins, side rails and eaves beams. If you are to get the most out of the wizard, then you need to understand these rules and their effect, these are illustrated in the figures below:

Purlins

The ***Cold Rolled Sections Wizard*** determines its runs of purlin using the rules below. At any point where you have specified a torsional restraint the ***Cold Rolled Sections Wizard*** adds stays to the inner flange of the member[1].

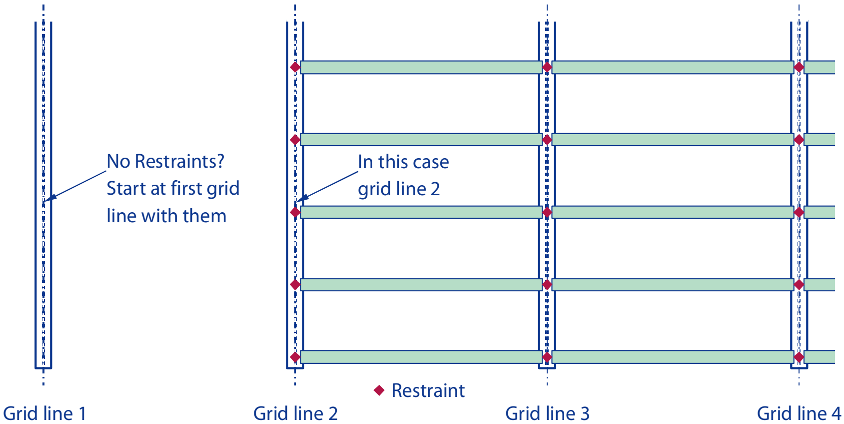
1. The ***Cold Rolled Sections Wizard*** looks at each rafter on grid line 1 in turn:

Help If your structure has hips, then the *Cold Rolled Sections Wizard* handles these areas differently. For further information ***see***:

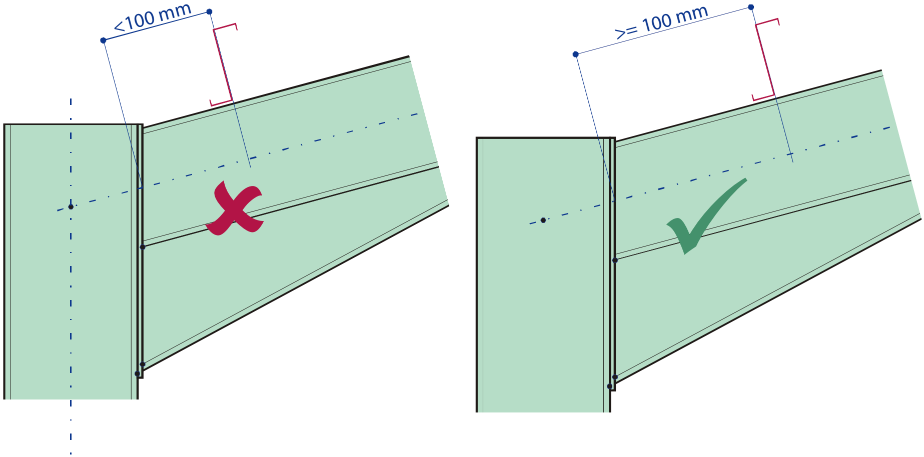
- *"Hips towards grid line 1"* on page 588,
- *"Hips away from grid line 1"* on page 588.

1. *The purlins are transferred to* ***3D+*** *as true sections, the stays are transferred as simple lines.*

a. If there are no restraints on the rafter, then the ***Cold-Rolled Sections Wizard*** skips to the next grid line and repeats this process until a restraint is found.

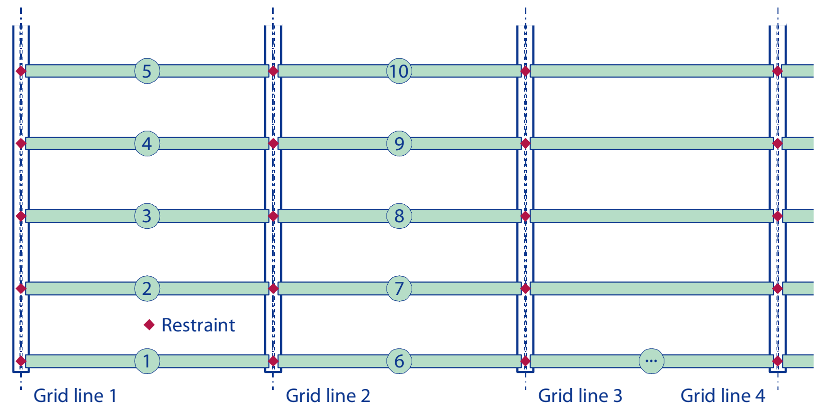


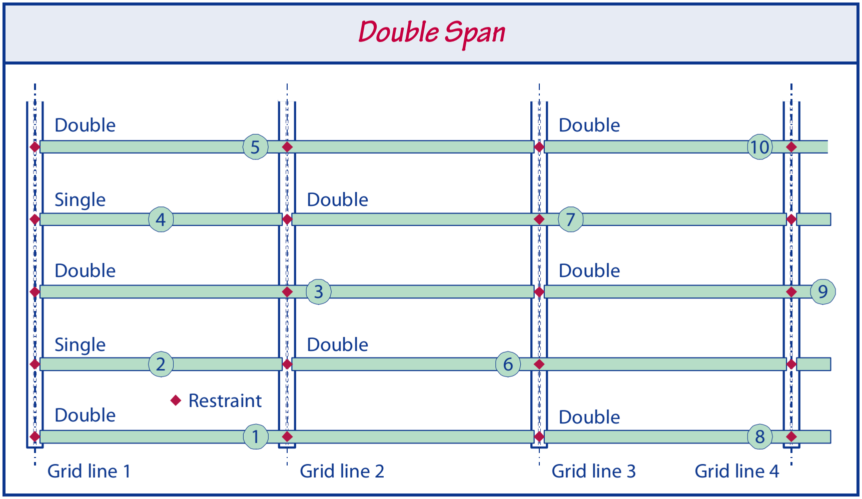
b. If the restraint is beyond the physical end of the rafter or within 100 mm of the end, then the ***Cold-Rolled Sections Wizard*** skips to the next restraint up the rafter without creating a purlin in this location.



c. The ***Cold-Rolled Sections Wizard*** thus finds the position of the first valid restraint up the rafter, then (assuming that none of the conditions detailed below occurs) creates the purlins in the layout you choose. The various layouts are shown below.

Single Span

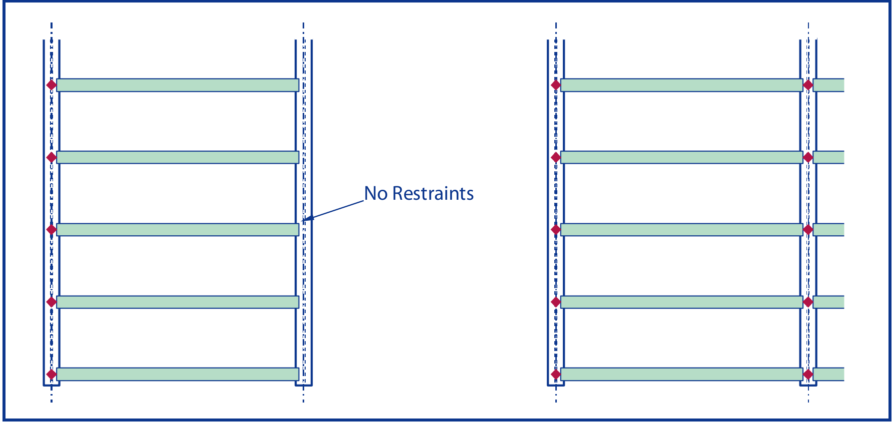


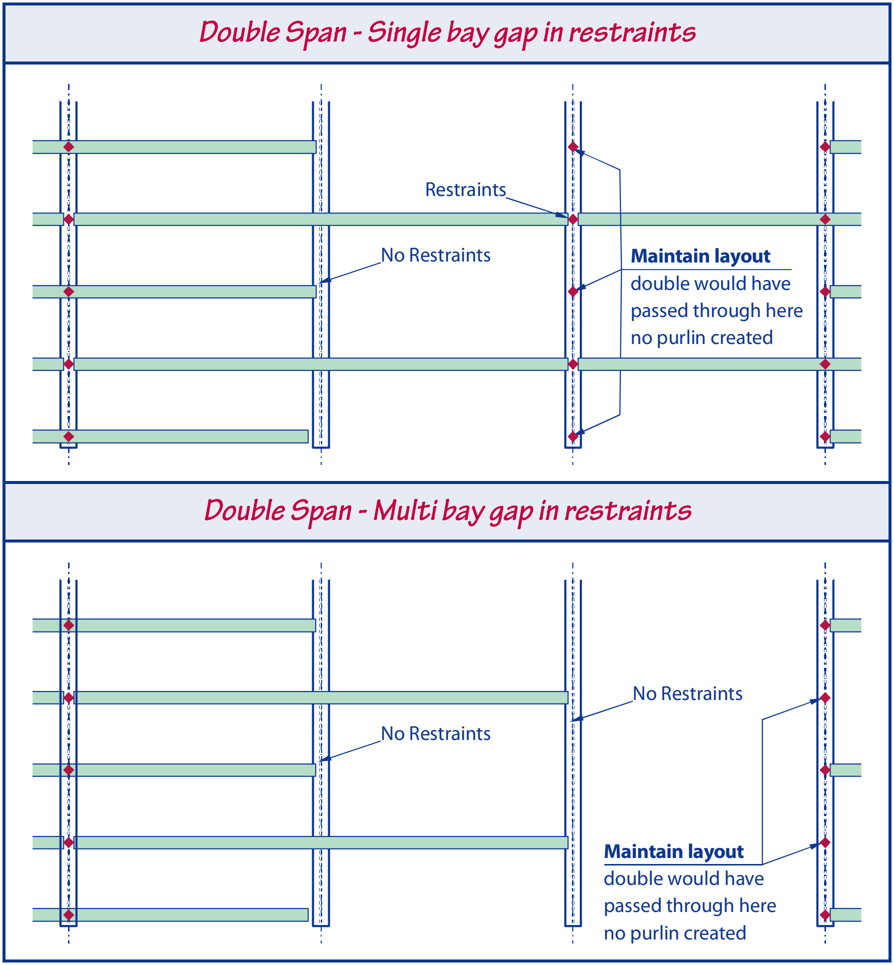

Double Span
Double
Single
Double
Single
Double
Double
Double
Double
Double
Double
Restraint
1
2
3
4
5
6
7
8
9
10
Grid line 1
Grid line 2
Grid line 3
Grid line 4

Handling a gap

If a rafter has no restraints, then the ***Cold Rolled Sections Wizard*** will not create any new purlins at that point. For a double span option, any double span purlins created from the last rafter will pass across the unrestrained rafter. The various options are illustrated below:

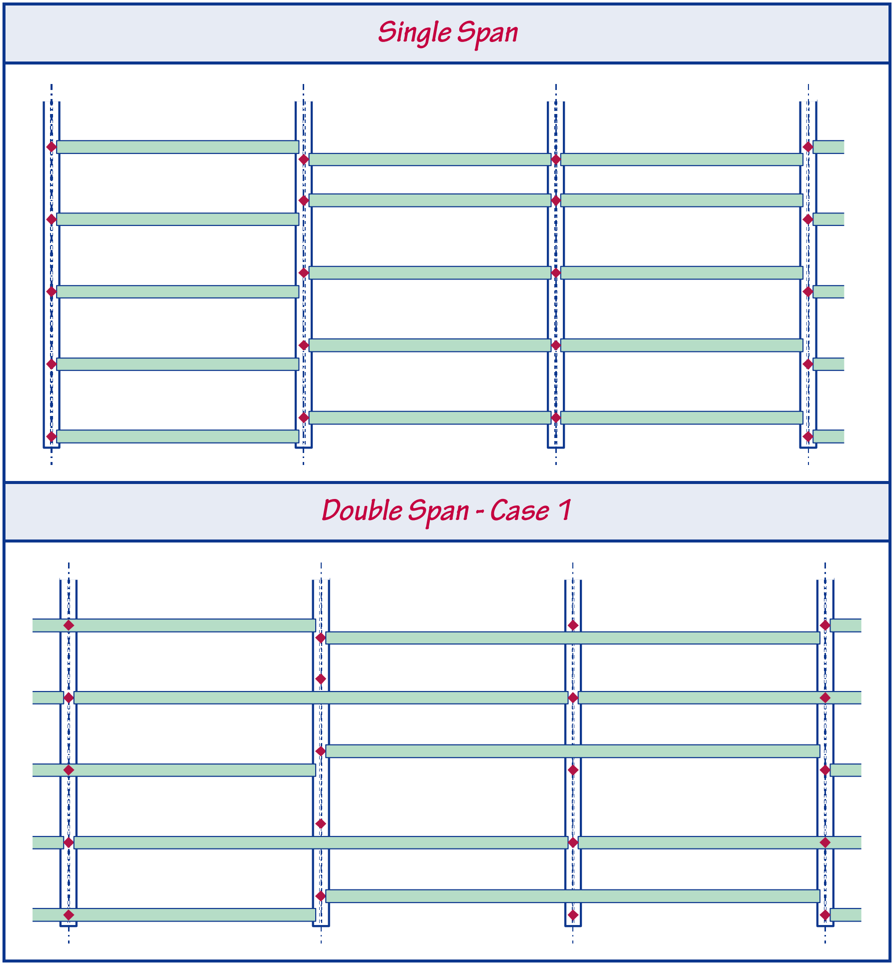
Single Span - gap in restraints



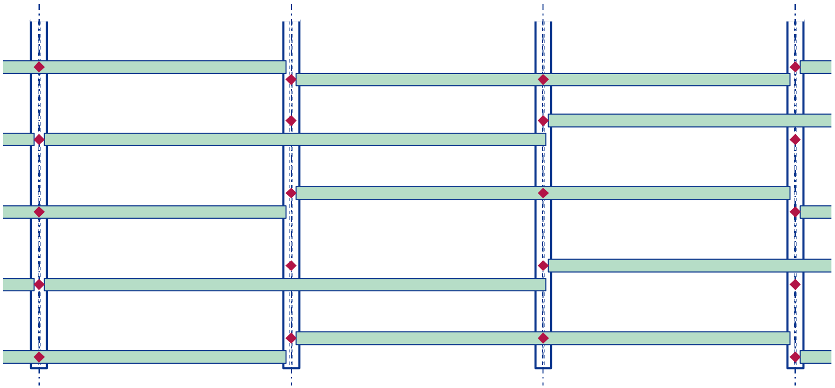


Handling out of line restraints

If a rafter along the side of your building has the same number of restraints as the other rafters on this roof slope, then the ***Cold Rolled Sections Wizard*** will start to use the new restraint positions as it creates new purlins past that point. Any double spanning purlins that come from a restraint at a previous rafter will not be at the new location. The various options are illustrated below:



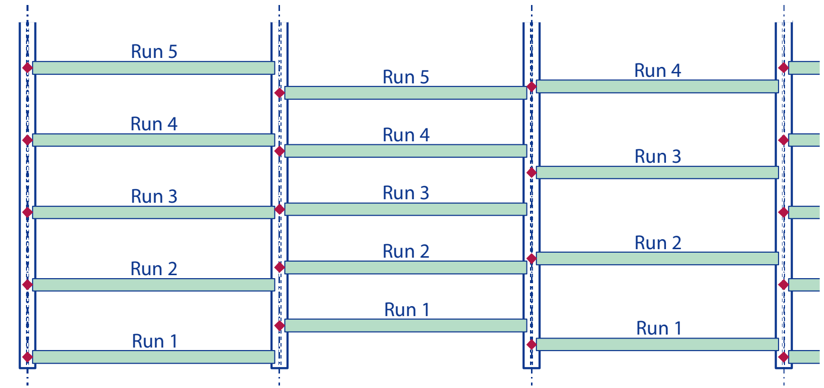
Double Span - Case 2



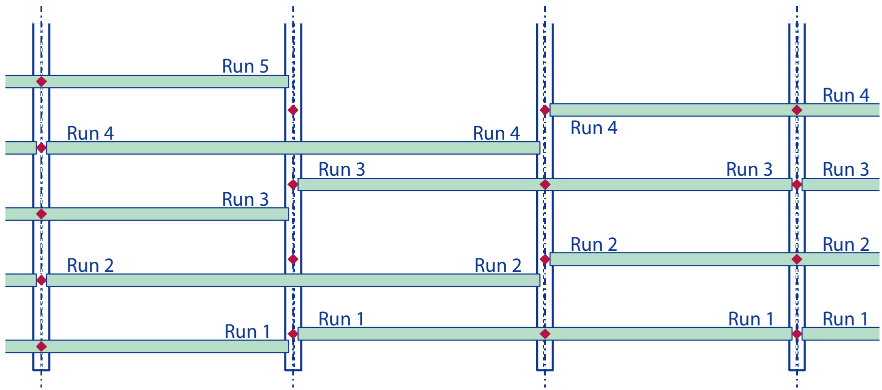
Handling different numbers of restraints

If you have different numbers of restraints on the rafters along a particular roof slope, then the new purlin position is used immediately for single span purlins. For double span purlins the ***Cold Rolled Sections Wizard*** maintains the existing pattern based on the number of the particular restraint up a rafter. The options are illustrated below:

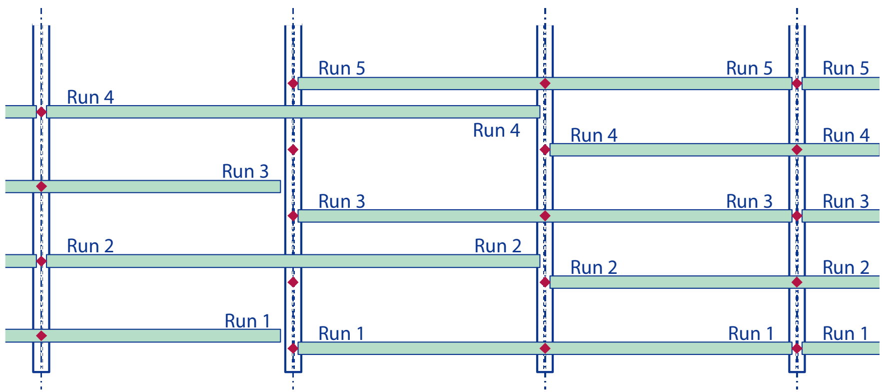
Single Span

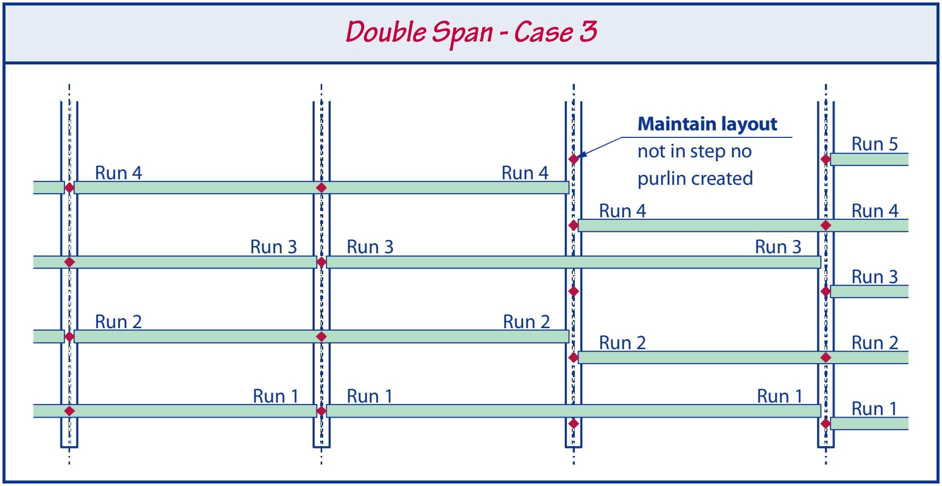


Double Span - Case 1



Double Span Case 2





Hips towards grid line 1

For hips where the monopitch end of the hip is nearer to grid line 1 than the apex of the hip the ***Cold Rolled Sections Wizard*** applies the rules for purlins above as though no hip was present. This gives a consistent set of purlin runs down the entire length of the structure. The ***Cold Rolled Sections Wizard*** then checks to see where the runs of purlin impinge on the hip raker. Any purlins entirely to the grid line 1 side of the hip raker are removed, any purlins which cross the hip raker are truncated at that raker. You will need to manually check and handle any small purlin fragments which are created by this curtailment process.

At the hip locations where the main purlin runs impinge on the hip raker, a purlin is created parallel to the gable and positioned such that the top edges of the two purlins are congruent, taking into account any difference in slope between the main roof and that of the hip.

Hips away from grid line 1

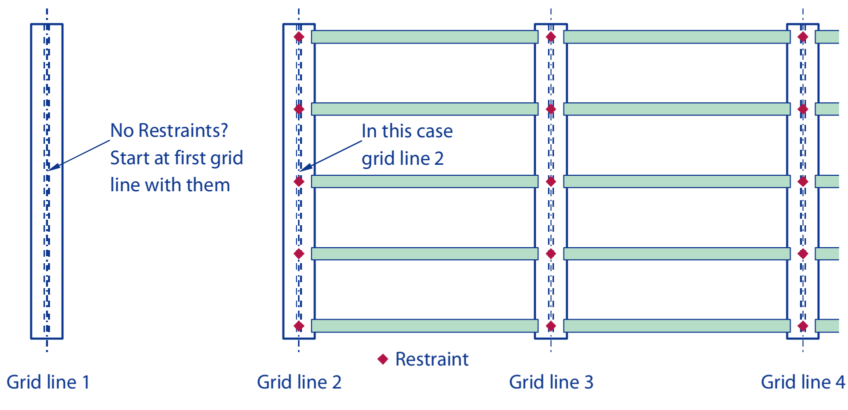
For hips where the monopitch end of the hip is further from grid line 1 than the apex of the hip the ***Cold Rolled Sections Wizard*** applies the rules for purlins above until the hip is reached. The ***Cold Rolled Sections Wizard*** then checks to see where the runs of purlin impinge on the hip raker. Any purlins entirely to the side of the hip raker which is farther from grid line 1 are removed, any purlins which cross the hip raker are truncated at that raker.

At the hip locations where the main purlin runs impinge on the hip raker, a purlin is created parallel to the gable and positioned such that the top edges of the two purlins are congruent, taking into account any difference in slope between the main roof and that of the hip.

Sheeting rails

The ***Cold Rolled Sections Wizard*** determines its runs of side rail using the rules below. At any point where you have specified a torsional restraint the ***Cold Rolled Sections Wizard*** adds stays to the inner flange of the member[1]. The details are described below.

1. The ***Cold Rolled Sections Wizard*** looks at each column on grid line 1 in turn:
   a. If there are no restraints on the column then the ***Cold-Rolled Sections Wizard*** skips to the next grid line. The ***Cold-Rolled Sections Wizard*** repeats this process until a restraint is found.

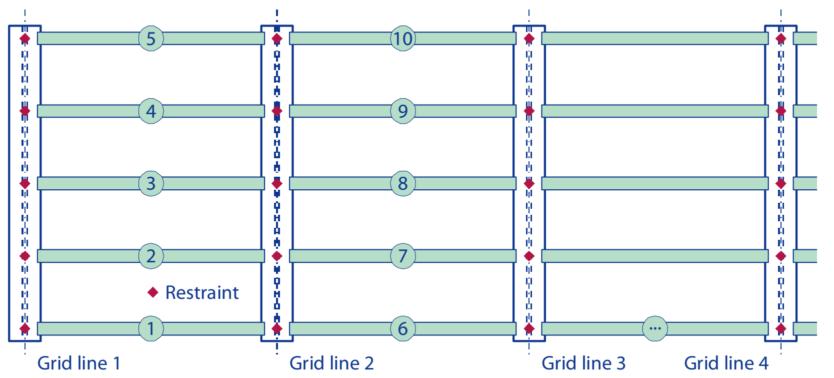


1. *The side rails are transferred to **3D+** as true sections, the stays are transferred as simple lines.*

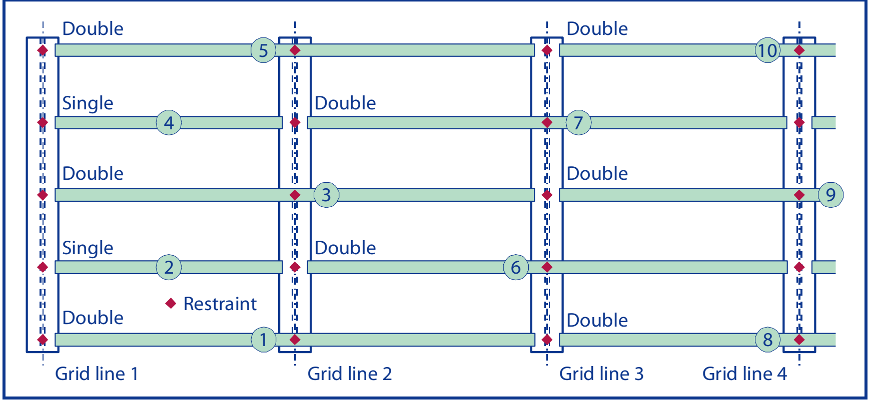
b. If the restraint is within 100 mm of the bottom of the column, then the ***Cold-Rolled Sections Wizard*** skips to the next restraint up the column without creating a purlin in this location.

c. The ***Cold Rolled Sections Wizard*** thus finds the position of the first restraint up the column, then (assuming that none of the conditions detailed below occurs) creates the sheeting rails in the layout you choose. The various layouts are shown below.

Single Span

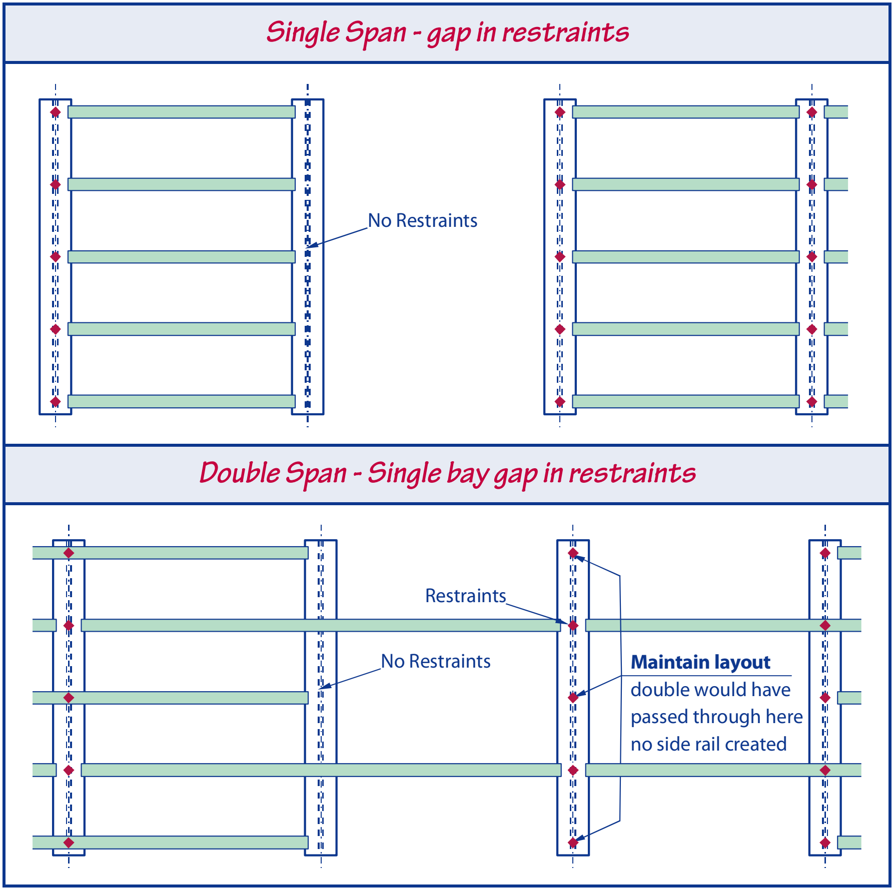


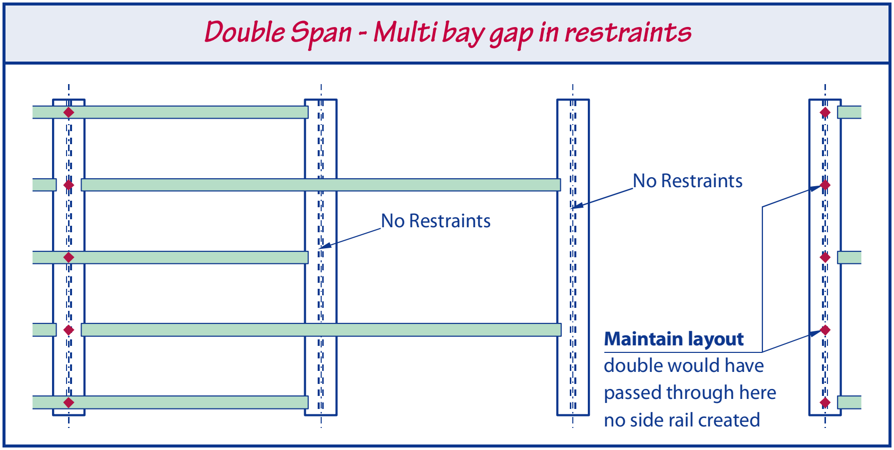
Double Span


Double
Single
Double
Single
Double
Double
Double
Double
Double
Double
5
4
3
2
1
10
7
9
6
8
Restraint
Grid line 1
Grid line 2
Grid line 3
Grid line 4

Handling a gap

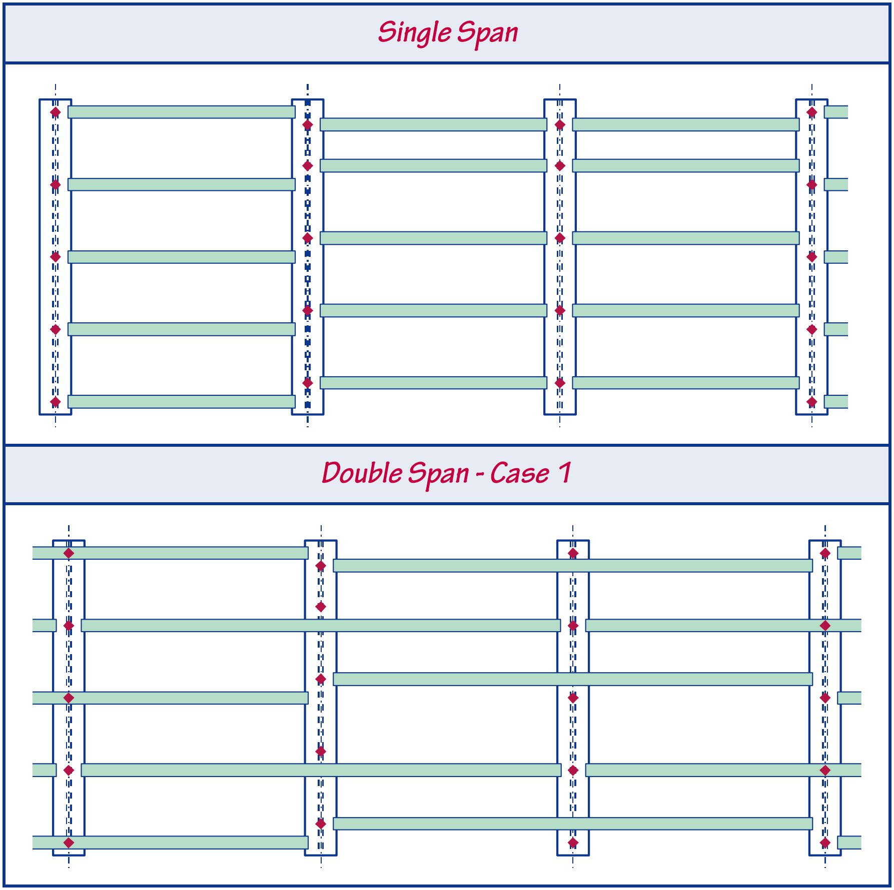
If a column along the side of your building has no restraints, then the ***Cold Rolled Sections Wizard*** will not create any new sheeting rails at that point. For a double span option, any double span rails created from the last column will pass across the unrestrained column. The various options are illustrated below:



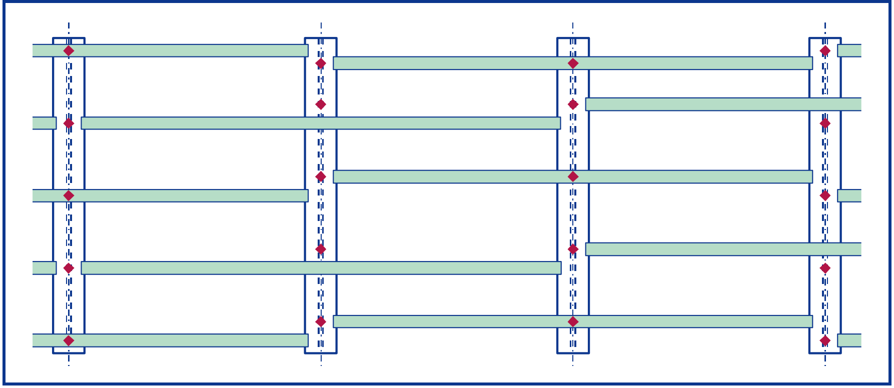


Handling out of line restraints

If a column along the side of your building has the same number of restraints as the other stanchions in this line of stanchions, then the ***Cold Rolled Sections Wizard*** will start to use the new restraint positions as it creates new side rails past that point. Any double spanning side rails that come from a restraint at a previous column will not be at the new location. The various options are illustrated below:

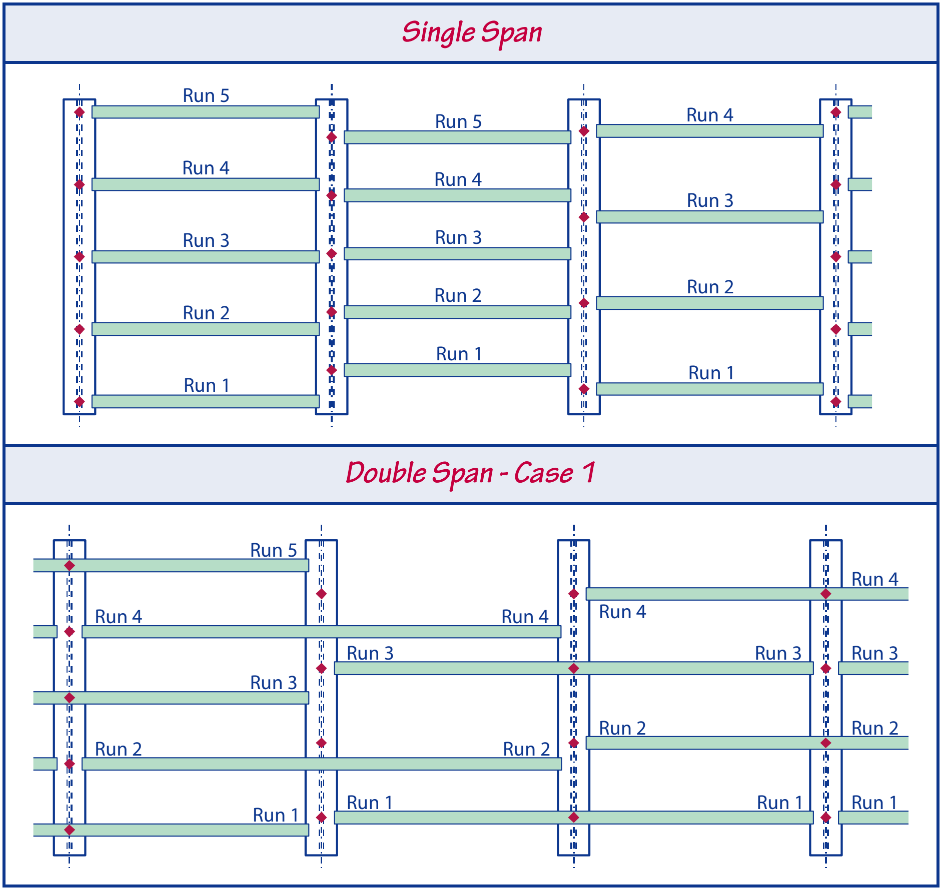


Double Span - Case 2

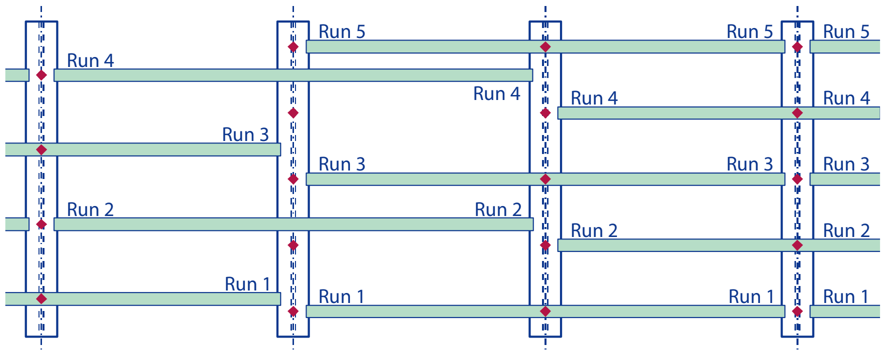


Handling different numbers of restraints

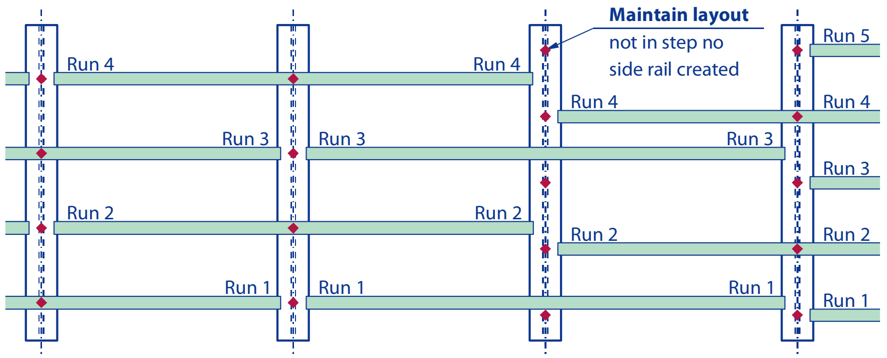
If you have different numbers of restraints on the stanchions along a line, then the new number and position of restraints is applied immediately for single span side rails. For double span side rails the ***Cold Rolled Sections Wizard*** maintains the existing pattern based on the number of the particular restraint up a column. The options are illustrated below:



Double Span Case 2



Double Span - Case 3



Eaves beams

Eaves beams are simpler than either sheeting rails or purlins, since there is only one run of eaves beam along any column line in your structure. The rules which the *Cold Rolled Sections Wizard* applies are simpler and are described below. But first a quick word about ***intersection points***.

Intersection points

If both the column and rafter at a particular location (***column / rafter pair***) have restraints, then the ***Cold Rolled Section Wizard*** uses the cladding dimensions which you have specified for these as the dimensions from the face of the column/rafter to the inner face of the sheeting. The ***Cold Rolled Section Wizard*** then calculates the ***intersection point*** of these two faces and places the eaves beam appropriately.

If the rafter at a particular location has restraints, while the column does not, then the ***Cold Rolled Section Wizard*** will calculate the intersection point between the inner face of the roof sheeting and the outside flange of the column.

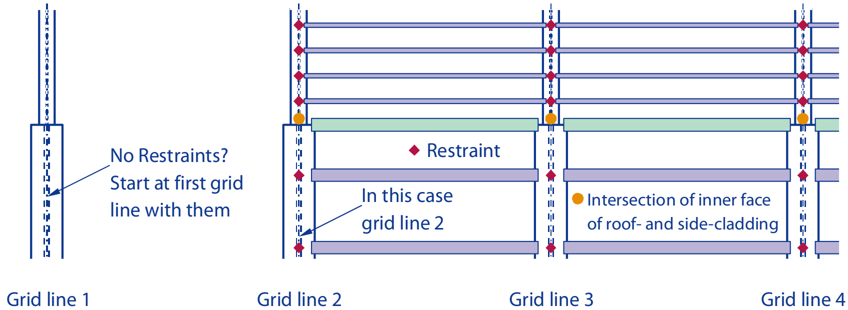
If the column at a particular location has restraints, while the rafter does not, then the ***Cold Rolled Sections Wizard*** will not determine an intersection point at that location. It will therefore not start an eaves beam at that location and will handle an ongoing run of eaves beam using the gap rules detailed below.

If both the column and rafter pair at a particular location do not have any restraints, then the ***Cold Rolled Sections Wizard*** will not determine an intersection point at that location. It will therefore not start an eaves beam at that location and will handle an ongoing run of eaves beam using the gap rules detailed below.

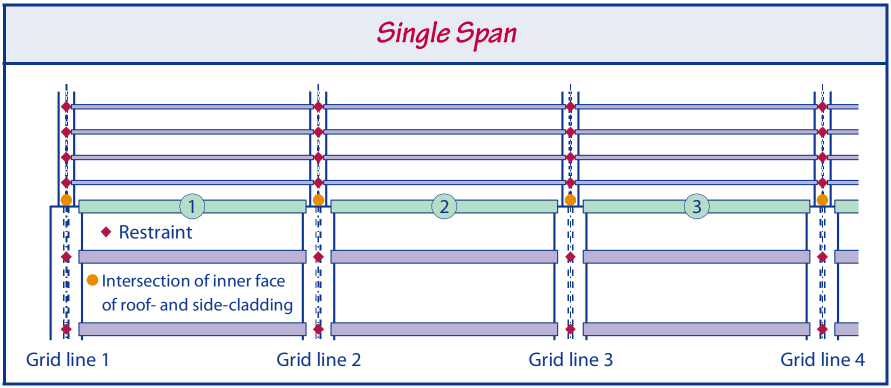
Eaves beam creation

1. The ***Cold Rolled Sections Wizard*** looks at each intersection point on grid line 1 in turn:

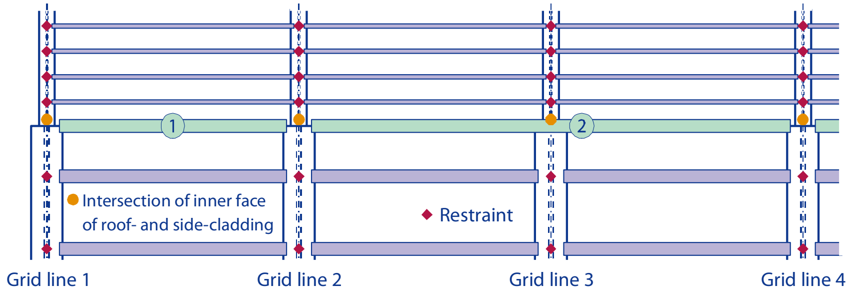
   a. If there is no intersection point on grid line 1, skip to the next grid line. Repeat until an intersection point is found.



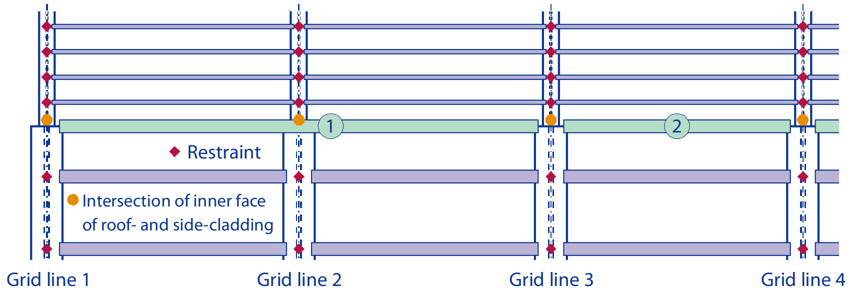
   b. If the ***Cold Rolled Sections Wizard*** finds an intersection point then, (assuming that none of the conditions detailed below occur), creates the eaves beam placing it so that the appropriate point on the eaves beam coincides with the intersection point. The various layouts are shown below.



Double Span

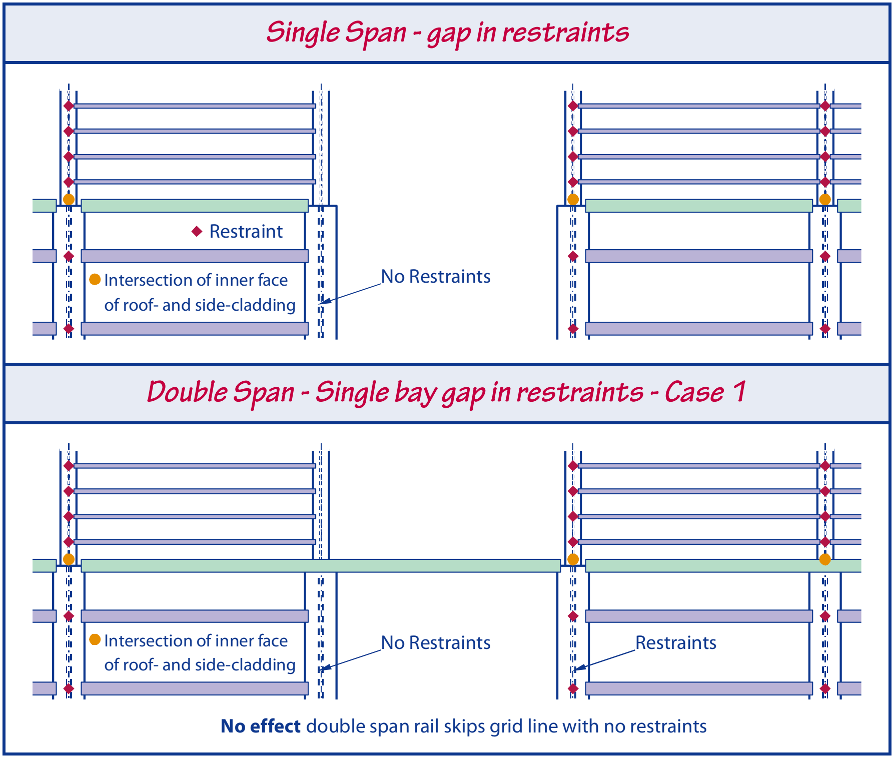


Double Span + Start with Double

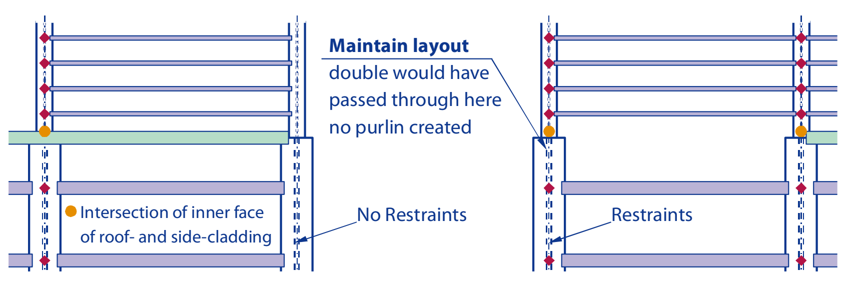


Handling a gap

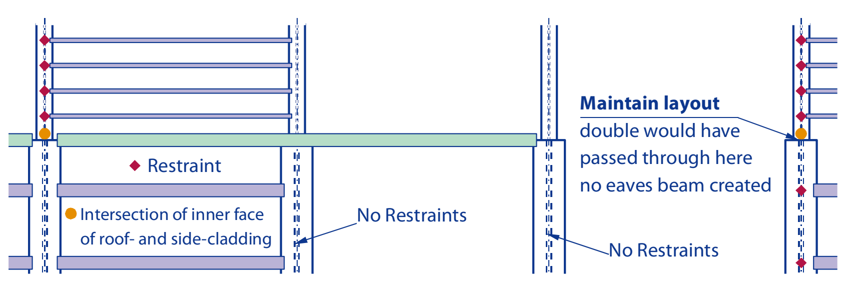
If there is no intersection point on a particular grid line, then the ***Cold Rolled Sections Wizard*** will not create any new eaves beam at that point. For a double span option, any double span eaves beam created from the previous intersection point will pass across the unrestrained column and rafter combination. The various options are illustrated below:



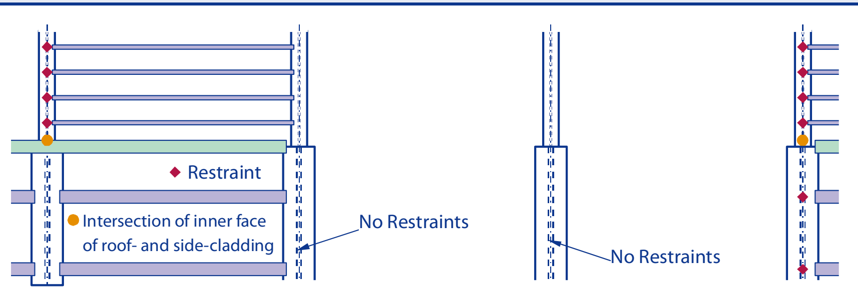
Double Span - Single bay gap in restraints - Case 2



Double Span - Multi bay gap in restraints - Case 1

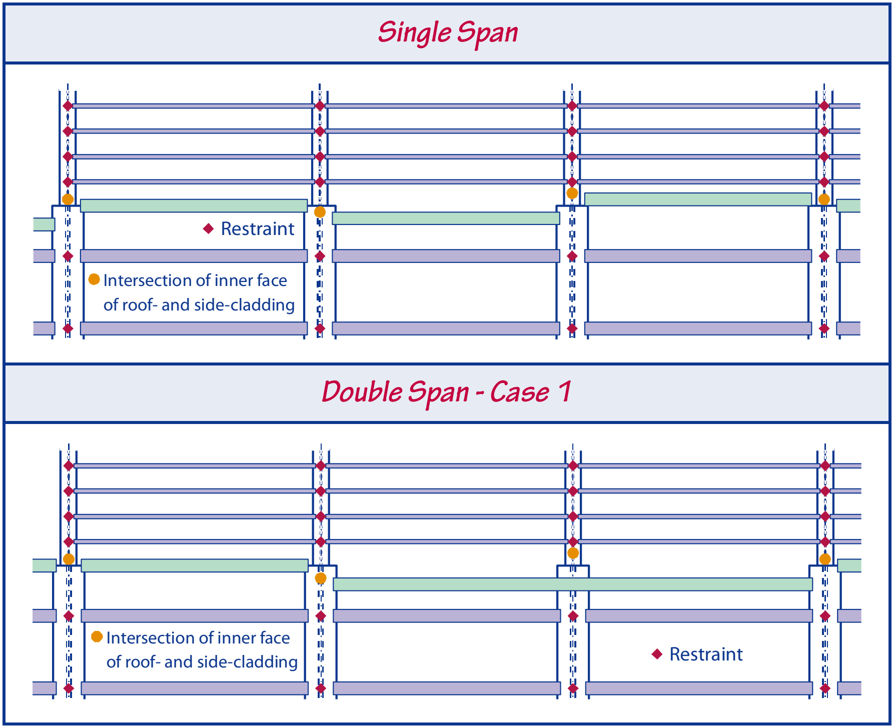


Double Span - Multi bay gap in restraints - Case 1

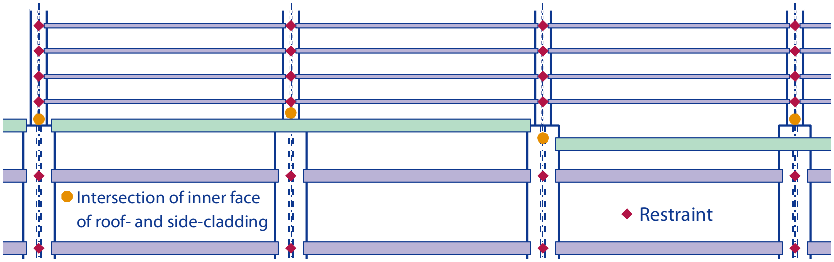


Handling out of line restraints

If the position of the intersection point varies along the side of your building, then the ***Cold Rolled Sections Wizard*** will start to use the new position if it creates a new eaves beams from that point. Any double spanning eaves beam that comes from a previous intersection point will therefore not be at the new intersection point's location. The various options are illustrated below:



Double Span - Case 2



To use the cold rolled sections wizard

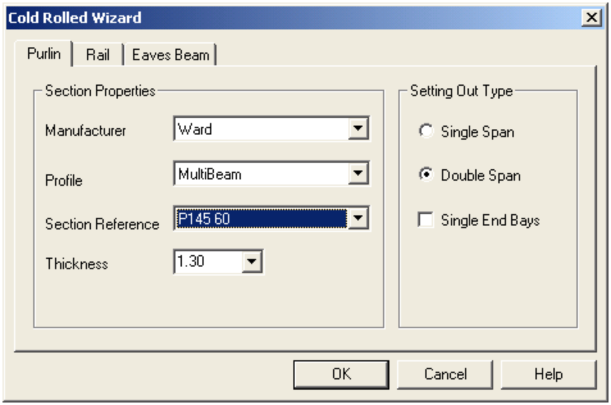
Create the frames in your structure in the usual manner, and use the ***Building Grid*** dialog (*Building/Grid…*) to position these. You should create a consistent layout of restraints for these frames if the ***Cold Rolled Sections Wizard*** is to have much chance of creating the layout which you require.

1. Click on, or open the ***Structure*** window[1] for your model.

1. *The **Structure** window is only available if you have purchased, installed and unlocked the **Portal Modeller**.*

Purlins

2. Click the ***Cold-Rolled Sections Wizard*** icon to see the ***Purlin*** page of the ***Cold Rolled Sections Wizard***.



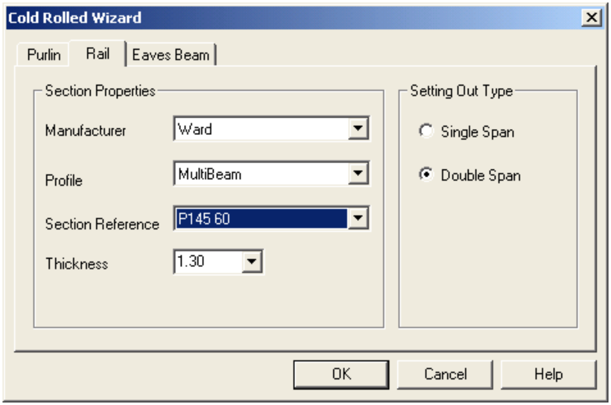
3. Pick the purlin **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Now choose the layout details for the purlins you want the ***Cold Rolled Sections Wizard*** to add. If you choose the **Double Span** option, then indicate whether you want the ***Cold Rolled Sections Wizard*** to add single span purlins to the end bay of your structure if necessary.

5. Pick another page of the property sheet, or click **OK** to create the cold rolled sections using the current information.

Side Rails

6. Click the ***Cold Rolled Sections Wizard*** icon followed by the ***Rail*** tab to see the ***Rail*** page of the ***Cold Rolled Sections Wizard***.



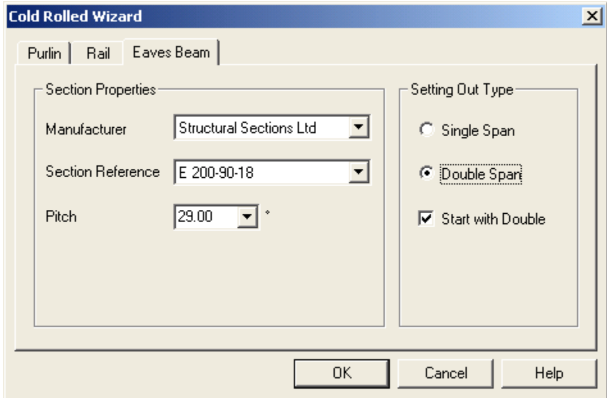
7. Pick the rail **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

8. Now choose the layout details for the side rails you want the ***Cold Rolled Sections Wizard*** to add.

9. Pick another page of the property sheet, or click **OK** to create the cold rolled sections using the current information.

Eaves Beams

10. Click the ***Cold Rolled Sections Wizard*** icon followed by the ***Eaves Beam*** tab to see the ***Eaves Beam*** page of the ***Cold Rolled Sections Wizard***.



11. Pick the rail **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

Note *The available pitches will depend on the manufacturer of the eaves beam.*

12. Now choose the layout details for the side rails you want the ***Cold Rolled Sections Wizard*** to add:
    - Again if you choose the **Double Span** option, then indicate whether the first eaves beam placed (starting from grid line 1, and assuming an even number of grids) is to be single or double spanning. If you don't tick **Start with Double**, then the first eaves beam placed will be single spanning, then double span eaves beams will be used to the end of the column line. If you do tick **Start with Double**, then the first eaves beam placed will be double spanning.

13. Pick another page of the property sheet, or click **OK** to create the cold rolled sections using the current information.

5 Working with Gable Posts

You create, delete or change the properties of gable posts graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with gable posts.

Creating gable posts

You can create gable posts by:

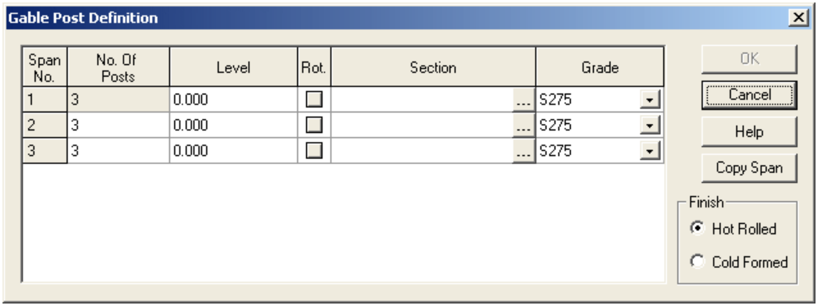
Method	*Details*
Grid Line	the gable posts are created on the grid line you choose. You can specify the number of gable posts that you want to create in each separate span of the frame.
Single	this option allows you to add new gable posts into a particular span.

To create gable posts by grid line

You must use the ***Structure*** window to define the gable posts in your structure.

1. Click the **Create**, **Grid Line** and **Gable Post** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Gable Post* menu options).

2. Select the grid line from the display on which you want to create the gable posts. (This must be a grid line which runs across your building – one on which your portal frames lie.) You will see the ***Gable Post Definition*** dialog which will include a line for each span of your frame.



Note If you pick a grid line on which gable posts are already defined, then the dialog will show the number of gable posts that each span contains and the size of the first gable post in the span.

Caution If you have modified a span's gable posts, by amending their details, position or both, and you **OK** this dialog, you will:

- position the gable posts at equally spaced centres across the span,
- change the **Section** and **Grade** of all gable posts in the span to those shown in this dialog.

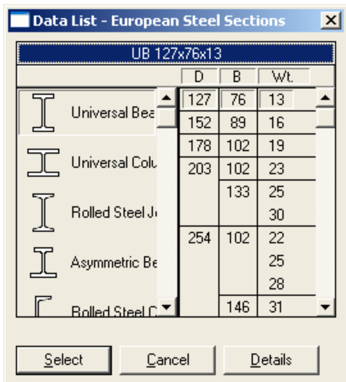
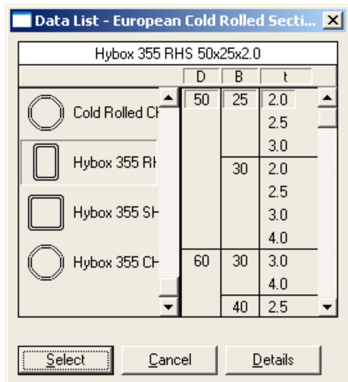
3. Enter the **No. of Posts** that you want to create and the **Level** at which the bases are to lie.

4. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable posts so that their webs are parallel to those of the main frame members, then tick the **Rot.** (rotation) box.

5. Choose the **Finish** of the gable posts you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

> ***Caution*** *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

6. To enter the size of your gable posts click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

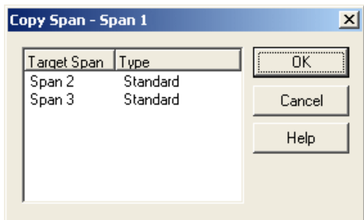
7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the* ***Grade*** *box will show S355, and you can not change this since these sections are only available in this grade.*

9. If you want to define different gable posts in the other spans of your frame, then you need to repeat steps 3 to 8 for each span.

10. Alternatively if you want to use the same details for one or more of the other spans in your frame click **Copy Span**.



11. Pick the span(s) to which you want to copy the details of your current span.

12. Click **OK** to return to the ***Gable Post Definition*** dialog which will show the details copied from the current span in the target span(s).

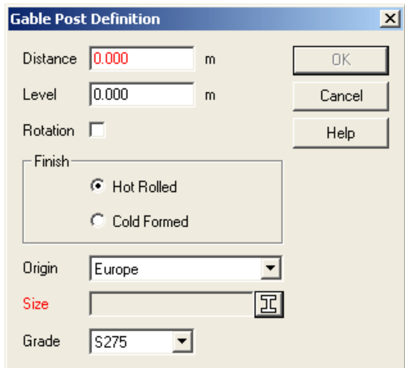
13. Once the details for all your gable posts are correct click **OK** to return to the graphical display of the structure which will be updated to show the gable posts which you have defined.

To create gable posts singly

You must use the *Structure* window to define the gable posts in your structure.

1. Click the **Create**, **Single** and **Gable Post** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Gable Post* menu options).

2. Click the intersection point between the grid line and column line which lies nearest to column line A for the new gable post's span.



3. Enter the **Distance** to the new gable post from the intersection point you picked in step 2 above.

4. Enter the **Level** at which the base of the new gable post is to lie.

5. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable posts so that their webs are parallel to those of the main frame members, then tick the **Rotation** box.

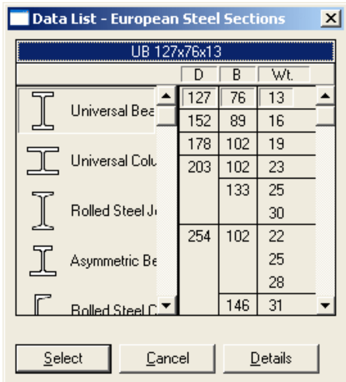
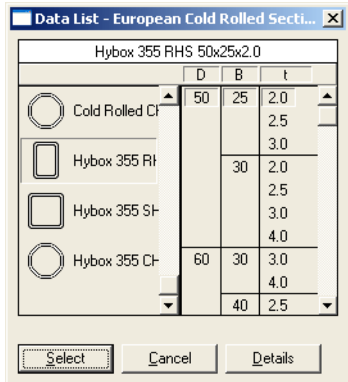
6. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. Choose the **Finish** of the gable post you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

8. To enter the size of your gable post click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

9. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

10. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the* ***Grade*** *box will show S355, and you can not change this since these sections are only available in this grade.*

11. Once the details for your gable post is correct click **OK** to return to the graphical display of the structure which will be updated to show the new gable post.

Deleting gable posts

You can delete gable posts by:

Method	*Details*
Grid Line	all gable posts on the selected grid line are deleted
Area	all gable posts which lie wholly within the selected area are deleted
Single	the gable post you pick is deleted

To delete gable posts by grid line

You must use the ***Structure*** window to delete the gable posts in your structure.

1. Click the **Delete**, **Grid Line** and **Gable Post** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Gable Post* menu options).
2. Click the grid line along which the gable post(s) you want to delete lie. All gable posts along this line are deleted.
3. Continue to identify grid lines until you have deleted all the gable posts that you require.

To delete gable posts by area

You must use the ***Structure*** window to delete the gable posts in your structure.

1. Click the **Delete**, **Area** and **Gable Post** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Gable Post* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable posts which lie entirely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the gable posts that you require.

To delete gable posts singly

You must use the ***Structure*** window to delete the gable posts in your structure.

1. Click the **Delete**, **Single** and **Gable Post** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Gable Post* menu options).

2. Select the gable post that you wish to delete.

3. Continue to identify gable posts until you have deleted all the gable posts that you require.

Changing gable post attributes

When you create gable posts you give the appropriate details for those gable posts. If you need to make changes you can do so without deleting and re-creating these gable posts.

You can modify gable posts details by:

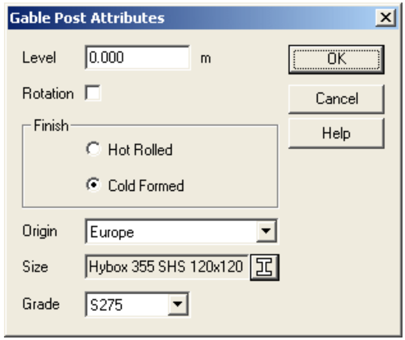
Method	*Details*
Grid Line	all gable posts on the selected grid line are modified to take the details you specify
Area	all gable posts which lie wholly within the area you select are modified to take the details you specify
Single	the gable post you pick is modified to take the details you specify

To modify gable post attributes by grid line

You must use the ***Structure*** window to modify the properties of the gable posts in your structure.

1. Click the **Attributes**, **Grid Line** and **Gable Post** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Gable Post* menu options).

2. Click the grid line along which the gable post(s) whose details you want to modify lie. You will see the ***Gable Post Attributes*** dialog which allows you to set the attributes that all gable posts along this grid line are to take.



Note *The initial details in this dialog are those for the gable post nearest to column line A on the line you select.*

3. Enter the **Level** at which the bases are to lie.

4. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable posts so that their webs are parallel to those of the main frame members, then tick the **Rotation** box.

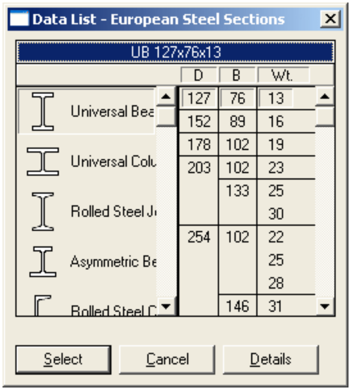
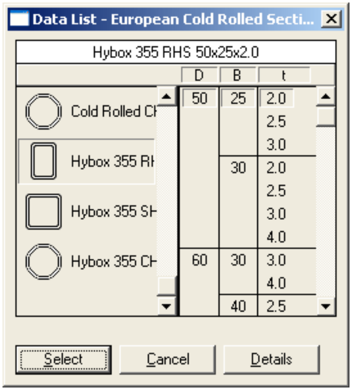
5. Choose the **Finish** of the gable posts you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.*

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note — The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your gable posts click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

10. Once the details for your gable posts are correct click **OK** to return to the graphical display of the structure. All the gable posts on the grid line will be updated to the attributes which you have just defined.

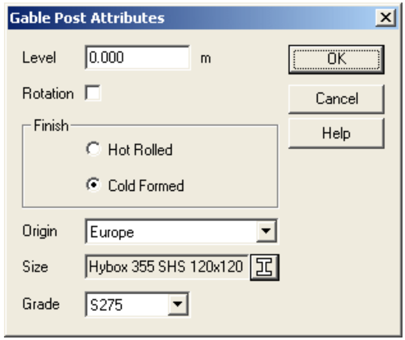
To modify gable post attributes by area

You must use the ***Structure*** window to modify the properties of the gable posts in your structure.

1. Click the **Attributes**, **Area** and **Gable Post** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Gable Post* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable posts which lie totally within this area will be selected.

3. You will see the ***Gable Post Attributes*** dialog which allows you to set the attributes that all the selected gable posts are to take.



Note *The initial details in this dialog are those for the gable post in your selection which is nearest to column line A.*

4. Enter the **Level** at which the bases are to lie.

5. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable posts so that their webs are parallel to those of the main frame members, then tick the **Rotation** box.

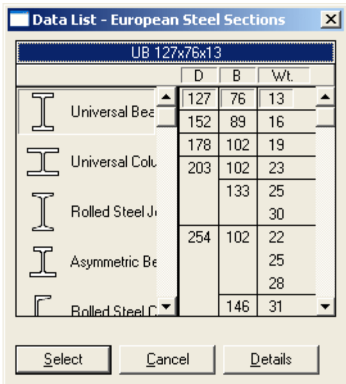
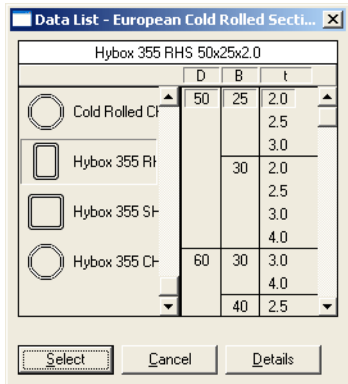
6. Choose the **Finish** of the gable posts you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.*

7. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your Preferences.*

8. To enter the size of your gable posts click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

9. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

10. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

11. Once the details for your gable posts are correct click **OK** to return to the graphical display of the structure. All the selected gable posts will be updated to the attributes which you have just defined.

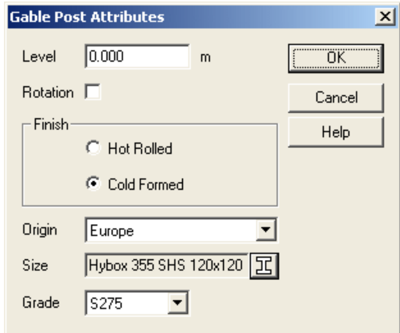
12. Continue to identify areas until you have modified the properties of all the gable posts that you require.

To modify gable post attributes singly

You must use the ***Structure*** window to modify the properties of the gable posts in your structure.

1. Click the **Attributes**, **Single** and **Gable Post** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Gable Post* menu options).

2. Select a gable post whose properties you wish to modify. You will see the ***Gable Post Attributes*** dialog which allows you to set the attributes that this gable posts is to take.



3. Enter the **Level** at which the base is to lie.

4. By default the gable posts are oriented with their webs perpendicular to those of the main frame members. If you want to orient your gable post so that its web is parallel to those of the main frame members, then tick the **Rotation** box.

5. Choose the **Finish** of the gable post you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

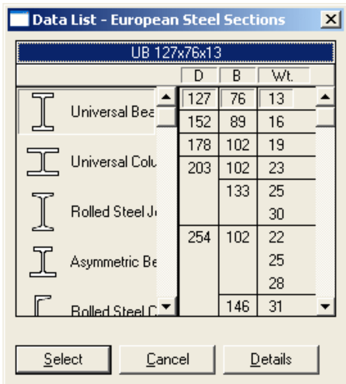
Caution *If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.*

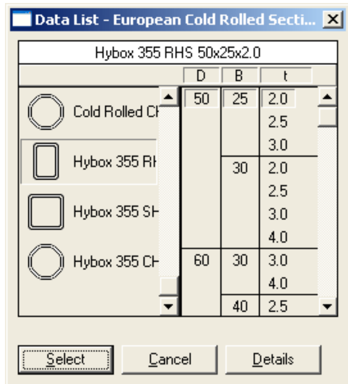
6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your Preferences.*

7. To enter the size of your gable post click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*





8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.*

10. Once the details for your gable post are correct click **OK** to return to the graphical display of the structure. The gable post will be updated to the attributes which you have just defined.

11. Continue to identify gable posts until you have modified the properties of all the gable posts that you require.

Changing gable post positions

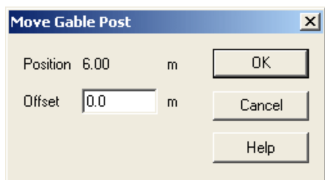
When you create gable posts on a grid line, the gable posts are equally spaced across the span (thus if you define 3 gable posts ***Portal Frame*** will position these at the quarter, half and three-quarter points across your span). If this positioning is that which you require, then you need take no further action. If, however, you need a different spacing, then you can move a particular gable post to the required position.

To move gable posts singly

You must use the ***Structure*** window to modify the positions of the gable posts in your structure.

1. Click the **Move**, **Single** and **Gable Post** buttons (or choose the *Select / Move*, *Select / Single* and *Select / Gable Post* menu options).

2. Select the gable post which you wish to move. You will see the ***Move Gable Post*** dialog which allows you to specify the distance by which the gable post is to move.



3. Enter the **Offset** distance by which the gable post is to move from its current location and then click **OK**.

Note The **Offset** you specify can be positive (to place the new post farther from column line A) or negative (to place the new post closer to column line A). If there are other gable posts in the span the **Offset** you specify cannot cause the new post to lie beyond any existing post. If you specify an **Offset** which breaks this requirement then the **Offset** you have defined will show in error and **OK** will be dimmed.

Note The **Position** information is updated to show the location at which the gable post will be placed when you click **OK**.

4. Continue to identify gable posts and move them until you have the layout that you require.

6 Working with Roof Bracings

You create, delete or change the properties of roof bracings graphically. As with all ancillary steelwork you must use the *Structure* window to work with roof bracings.

Creating roof bracings

You can create roof bracings by:

Method	*Details*
Grid Line	the roof bracings are created between the grid line you choose and the grid line next farther away from grid line 1. You can specify the number of roof bracings that you want to create in each separate span of the frame, the pattern of bracing that you require and the details of that pattern.
Single	this option allows you to add new roof bracings into a particular span.

To create roof bracings by grid line

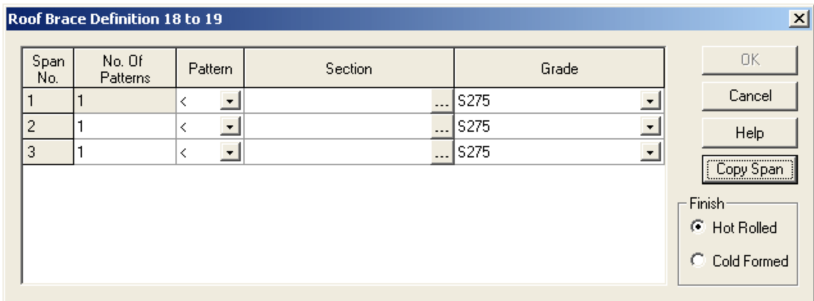
You must use the *Structure* window to define the roof bracings in your structure.

1. Click the **Create**, **Grid Line** and **Roof Bracing** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Roof Bracing* menu options).

2. Select the grid line from the display which you want use to create the roof bracings. (This must be a grid line which runs across your building – one on which your portal frames lie.) The roof bracings will be created between this grid line and the grid line next farther away from grid line 1.

Note *You need to click on the grid line and not on the frame members.*

You will see the ***Roof Bracing Definition*** dialog which will include a line for each span of your frame, and will also indicate between which two grid lines the roof bracings will lie.



Note *If you pick a grid line on which roof bracings are already defined, then the dialog will show the number of roof bracings that each span contains, and the details of the first roof bracing in the span.*

Caution If you have modified a span's roof bracings, by amending the **Pattern**, **Section** or **Grade** details, or if you have moved roof bracings, and you **OK** this dialog, you will:

- change the **Pattern**, **Section** and **Grade** of all roof bracings in the span to those shown in this dialog,
- create the **No. of Patterns** specified in this dialog, these will be equally spaced across the span.

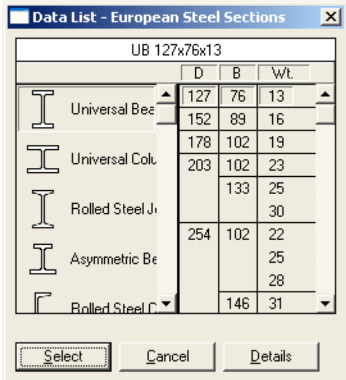
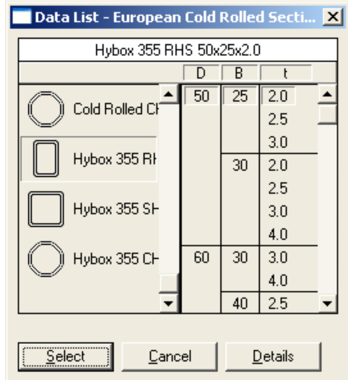
3. Enter the **No. of Patterns** that you want to create in the span.

4. For each span which is to contain roof bracings choose the type of roof bracing you want to use by selecting the appropriate **Pattern** from the list of available patterns.

5. Choose the **Finish** of the roof bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. To enter the size of your roof bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

Note All the roof bracings in the span take the same size at this stage. If you want the braces to be different sizes you must change these individually once you have created them.

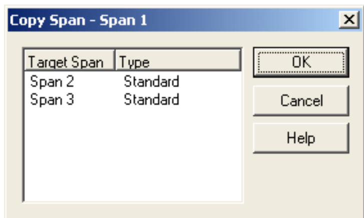
Caution If you choose a roof bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.

9. If you want to define different numbers of roof bracings in the other spans of your frame, then you need to repeat steps 3 to 8 for each span.

10. Alternatively if you want to use the same details for one or more of the other spans in your frame click **Copy Span**.



11. Pick the span(s) to which you want to copy the details of your current span.

12. Click **OK** to return to the ***Roof Bracing Definition*** dialog which will show the details copied from the current span in the target span(s).

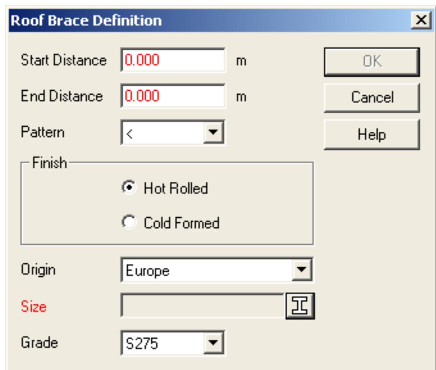
13. Once the details for all your roof bracings are correct click **OK** to return to the graphical display of the structure which will be updated to show the roof bracing layout which you have defined.

To create roof bracings singly

You must use the ***Structure*** window to define the roof bracings in your structure.

1. Click the **Create**, **Single** and **Roof Bracing** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Roof Bracing* menu options).

2. Click one of the rafters in the span into which you want to add the new roof bracing.



Note You need to click the rafter on the grid line which is nearest to grid line 1 in order to create the roof bracing between this rafter and that on the line next further away from grid line 1.

3. Enter the **Start Distance** and **End Distance** for the new roof bracing. These dimensions are measured on plan from the column line at the start of the span (that nearest column line A) into which you are adding the roof bracing.

Note If you add a new roof bracing to a span which already contains other roof bracings, then *Portal Frame* will automatically adjust the start and/or end distances to those bracings to take account of the new one.

4. Choose the **Pattern** of the new roof bracing from the list of available patterns.

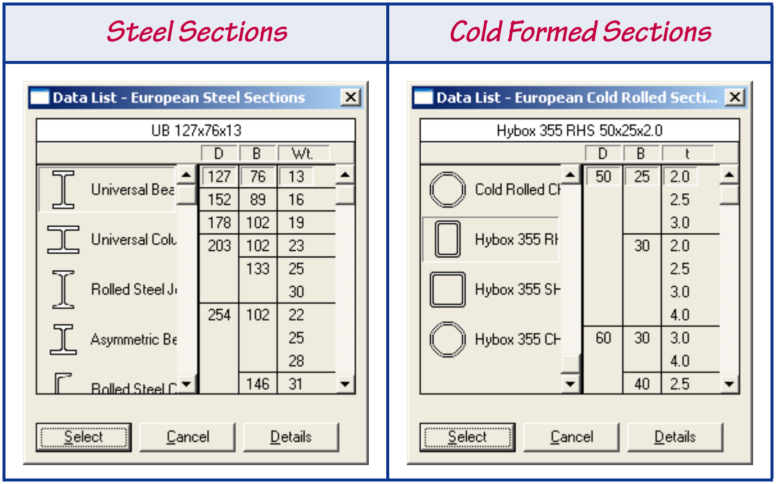
5. Choose the **Finish** of the roof bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your roof bracing click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

10. Once the details for your roof bracing is correct click **OK** to return to the graphical display of the structure which will be updated to show the new roof bracing.

Deleting roof bracings

You can delete roof bracings by:

Method	*Details*
Grid Line	all roof bracings which lie between the grid line you choose and that next farther away from grid line 1 are deleted
Area	all roof bracing patterns which lie wholly within the selected area are deleted
Single	the roof bracing you pick is deleted

To delete roof bracings by grid line

You must use the ***Structure*** window to delete the roof bracings in your structure.

1. Click the **Delete**, **Grid Line** and **Roof Bracing** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Roof Bracing* menu options).

2. Click the grid line which you initially used to create the roof bracings (the one nearest grid line 1). The roof bracings between this grid line and that next farther away from grid line 1 are deleted.

3. Continue to identify grid lines until you have deleted all the roof bracings that you require.

To delete roof bracings by area

You must use the ***Structure*** window to delete the roof bracings in your structure.

1. Click the **Delete**, **Area** and **Roof Bracing** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Roof Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All roof bracings of which lie entirely within the area which you select are deleted.

   Note *If you have picked a type of roof bracing which consists of more than one member, then that pattern will not be deleted unless all members making up that pattern are within the area you selected.*

3. Continue to identify areas until you have deleted all the roof bracings that you require.

To delete roof bracings singly

You must use the ***Structure*** window to delete the roof bracings in your structure.

1. Click the **Delete**, **Single** and **Roof Bracing** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Roof Bracing* menu options).

2. Select the roof bracing that you wish to delete.

   Note *If you have picked a type of roof bracing which consists of more than one member, then all members making up that pattern of bracing will be deleted.*

3. Continue to identify roof bracings until you have deleted all the roof bracings that you require.

Changing roof bracing attributes

When you create roof bracings by grid line you give the appropriate details for the roof bracings on a span by span basis. All the roof bracings in the same span take the same details (pattern, section size and grade). If you need to make changes you can do so without deleting and re-creating the entire roof bracing system, whichever option you used to create it.

You can modify roof bracing details by:

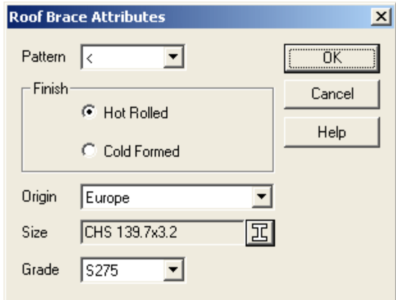
Method	*Details*
Grid Line	all roof bracings which lie between the grid line you choose and that next farther away from grid line 1 are modified to take the details you specify
Area	all roof bracing patterns which lie wholly within the area you select are modified to take the details you specify
Single	the roof bracing you pick is modified to take the details you specify

To modify roof bracing attributes by grid line

You must use the ***Structure*** window to modify the properties of the roof bracings in your structure.

1. Click the **Attributes**, **Grid Line** and **Roof Bracing** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Roof Bracing* menu options).

2. Click the grid line which you initially used to create the roof bracings (the one nearest grid line 1). You will see the ***Roof Bracing Attributes*** dialog which allows you to set the attributes that all roof bracings between this and the next grid line are to take.



Note *The initial details in this dialog are those for the roof bracing nearest to column line A on the line you select.*

3. Pick the **Pattern** which you want to use from the list of available patterns.

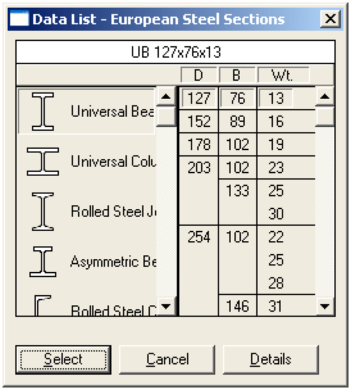
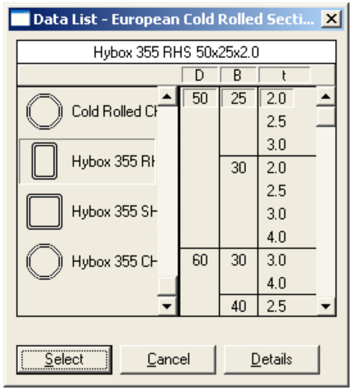
4. Choose the **Finish** of the roof bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

5. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial* **Origin** *setting will be that of the country which is set in your* *Preferences.*

6. To enter the size of your roof bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the* **Grade** *box will show* *S355**, and you can not change this since these sections are only available in this grade.*

9. Once the details for your roof bracings are correct click **OK** to return to the graphical display of the structure. All the roof bracings between the two grid lines will be updated to the attributes which you have just defined.

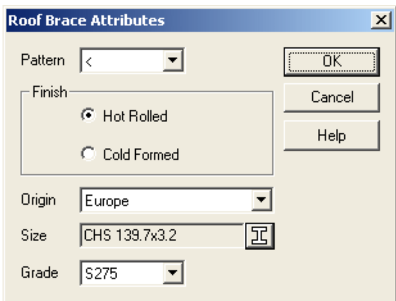
To modify roof bracing attributes by area

You must use the ***Structure*** window to modify the properties of the roof bracings in your structure.

1. Click the **Attributes**, **Area** and **Roof Bracing** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Roof Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All roof bracings which lie totally within this area will be selected.

3. You will see the ***Roof Bracing Attributes*** dialog which allows you to set the attributes that all the selected roof bracings are to take.



Note The initial details in this dialog are those for the roof bracing in your selection which is nearest to column line A.

4. Pick the **Pattern** which you want to use from the list of available patterns.

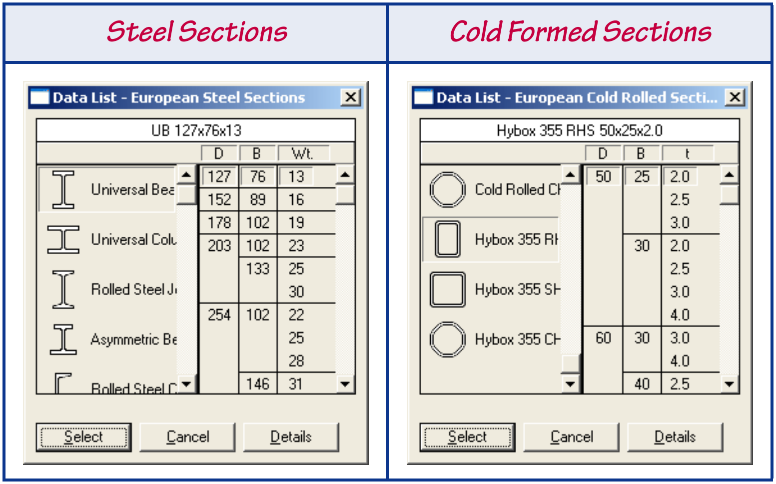
5. Choose the **Finish** of the roof bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your roof bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

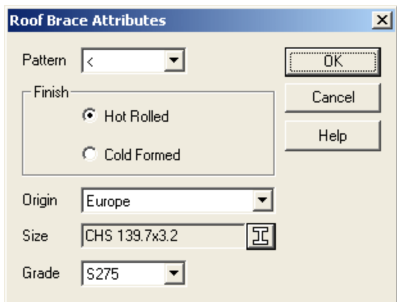
10. Once the details for your roof bracings are correct click **OK** to return to the graphical display of the structure. All the selected roof bracings will be updated to the attributes which you have just defined.

To modify roof bracing attributes singly

You must use the ***Structure*** window to modify the properties of the roof bracings in your structure.

1. Click the **Attributes**, **Single** and **Roof Bracing** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Roof Bracing* menu options).

2. Select a roof bracing whose properties you wish to modify. You will see the ***Roof Bracing Attributes*** dialog which allows you to set the attributes that this roof bracing is to take.



Note The initial details in this dialog are those for the roof bracing which you select.

Caution If you choose a roof bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

3. Pick the **Pattern** which you want to use from the list of available patterns.

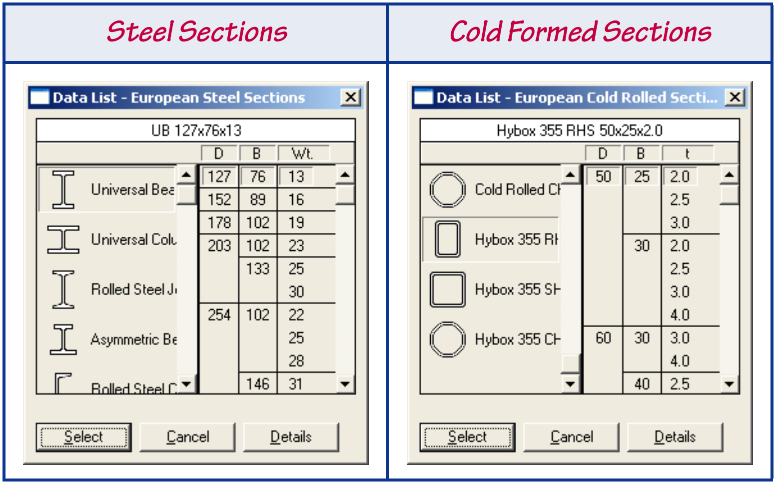
4. Choose the **Finish** of the roof bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

5. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

6. To enter the size of your roof bracing click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. Once the details for your roof bracing are correct click **OK** to return to the graphical display of the structure. The selected roof bracing will be updated to the attributes which you have just defined.

10. Continue to identify roof bracings until you have modified the properties of all those that you require.

Changing roof bracing positions

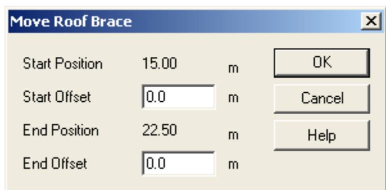
When you create roof bracings on a grid line, the patterns are equally spaced across the span. If this positioning is that which you require, then you need take no further action. If, however, you need a different spacing, then you can move a particular roof bracing to the required position, the other patterns in the span will be modified to suit your new positioning.

To move roof bracings singly

You must use the ***Structure*** window to modify the positions of the roof bracings in your structure.

1. Click the **Move**, **Single** and **Roof Bracing** buttons (or choose the *Select / Move*, *Select / Single* and *Select / Roof Bracing* menu options).

2. Select the roof bracing which you wish to move. You will see the ***Move Roof Bracing*** dialog which allows you to specify the distance by which the ends of the roof bracing are to move.



3. Enter the **Start Offset** distance by which the start point of the roof bracing (that nearest column line A) is to move from its current location.

4. Enter the **End Offset** distance by which the end point of the roof bracing (that farthest from column line A) is to move from its current location.

Note If you move a roof bracing which contains two members (< or >), then any modifications you make will apply to the entire pattern and the point of the pattern will lie halfway between the moved Start and/or End of the pattern.

Note The **Position** information is updated to show the location at which the roof bracing will be placed when you click **OK**.

5. Once the offset details are correct click **OK** to move the pattern. Any adjacent patterns will adjust automatically to accommodate the changes.

Note The **Offset** you specify can be either positive (to move the appropriate point of the pattern farther across the span) or negative (to reduce the distance from the start of the span to the point). The minimum distance between the start and end of a pattern is 0.75 m. If you enter offsets which violate this constraint both the **Start Offset** and **End Offset** will show in error and **OK** will be dimmed.

6. Continue to identify roof bracings and move them until you have the layout that you require.

7 Working with Side Bracings

You create, delete or change the properties of side bracings graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with side bracings.

Creating side bracings

You can create side bracings by:

Method	*Details*
Grid Line	the side bracings are created between the grid line you choose and that next farther away from grid line 1. You can specify the number of side bracings that you want to create at each separate span boundary of the frame, and the details of that pattern.
Single	this option allows you to add the side bracings you require in a single bay of your structure.

To create side bracings by grid line

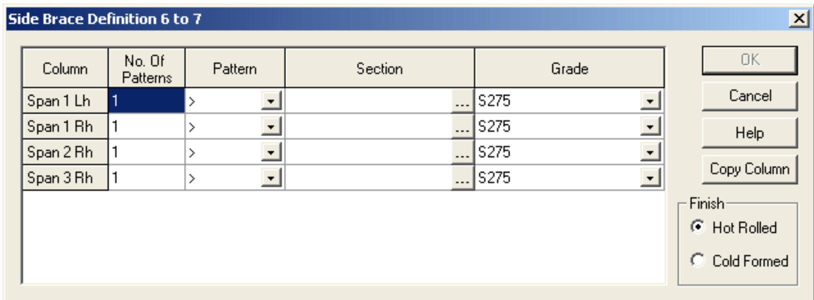
You must use the ***Structure*** window to define the side bracings in your structure.

1. Click the **Create**, **Grid Line** and **Side Bracing** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Side Bracing* menu options).

2. Select the grid line from the display which you want use to create the side bracings. (This must be a grid line which runs across your building – one on which your portal frames lie.) The side bracings will be created:
    - in the bays between the grid line which you select and the next grid line farther away from grid line 1,
    - across each span of the frame (at your choosing).

Note You need to click on the grid line and not on the frame members.

You will see the ***Side Bracing Definition*** dialog which will include a line for each column line along the grid line you picked. The title of the dialog indicates between which two grid lines the side bracings will lie.



Note If you pick a grid line on which side bracings are already defined, then the dialog will show the number of side bracings that you defined for each span of the frame, and the details of the first side bracing in that location.

Caution If you have modified a side bracings, by amending the **Pattern**, **Section** or **Grade** details, or if you have moved side bracings, and you **OK** this dialog, you will:

- change the **Pattern**, **Section** and **Grade** of all side bracings in the span to those shown in this dialog,
- create the **No. of Patterns** specified in this dialog, these will be equally spaced up the bay.

Note If you have a valley condition to either side of the bay where a side bracing would lie, then you can not place a side bracing in that location. If you try to specify a non-zero **No. of Patterns** then this generates an error condition and **OK** is dimmed.

3. Enter the **No. of Patterns** that you want to create at each **Column** location, a single pattern is created across the bay, and the number of patterns you specify is created up the bay.

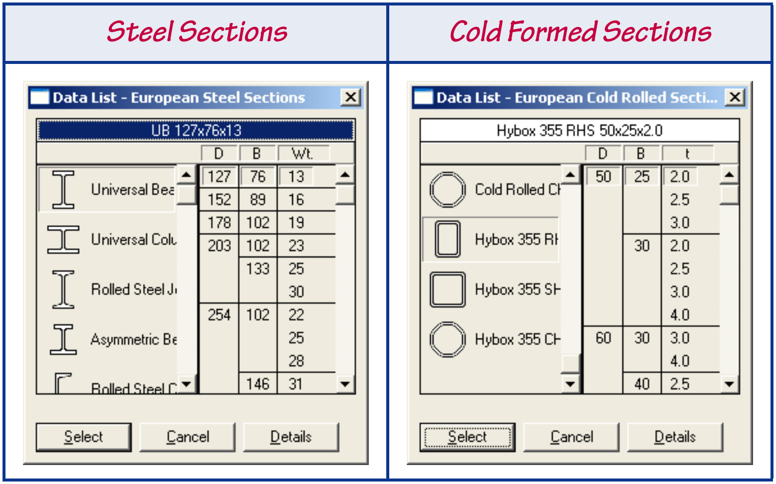
4. For each span boundary which is to contain a side bracing choose the type of side bracing you want to use by selecting the **Pattern** from the list of available patterns.

Note At this stage all the patterns up the bay are identical, you can change this later if necessary.

5. Choose the **Finish** of the side bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. To enter the size of your side bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

Note All the side bracings in the span take the same size at this stage. If you want the braces to be different sizes you must change these individually once you have created them.

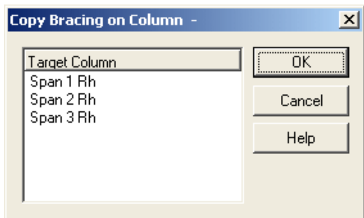
Caution If you choose a side bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. If you want to define different numbers of side bracings for the other span boundary (boundaries) of your frame, then you need to repeat steps 3 to 8 for each such boundary.

10. Alternatively if you want to use the same details for the other span boundary (boundaries) in your frame click **Copy Column**.



11. Pick the span boundary location(s) to which you want to copy the details from the current location.

12. Click **OK** to return to the ***Side Bracing Definition*** dialog which will show the details copied from the current boundary in the target boundary (boundaries).

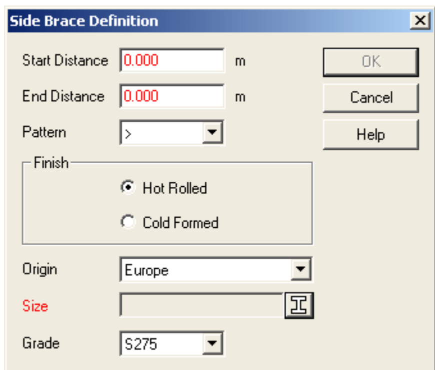
13. Once the details for all your side bracings are correct click **OK** to return to the graphical display of the structure which will be updated to show the side bracing layout which you have defined.

To create side bracings singly

You must use the ***Structure*** window to define the side bracings in your structure.

1. Click the **Create**, **Single** and **Side Bracing** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Side Bracing* menu options).

2. Click the base of the column at the side of the bay where you want to create your new side bracing.



Note You need to click the column base on the grid line which is nearest to grid line 1 in order to create the side bracing between this column base and that on the line next further away from grid line 1.

Note If you have a valley condition to either side of the bay which you select you will not be allowed to define any details, In order for the side bracing to be valid there must be a column at both sides of the bay.

3. Enter the **Start Distance** and **End Distance** for the new side bracing. These dimensions are measured from datum zero and not from the column base.

Note If you add a new side bracing to a span which already contains other side bracings, then *Portal Frame* will automatically adjust the start and/or end distances to those bracings to take account of the new one.

4. Select the appropriate **Pattern** from the list of available patterns.

Note At this stage all the patterns up the bay are identical, you can change this later if necessary.

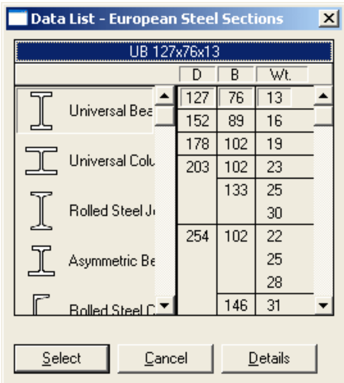
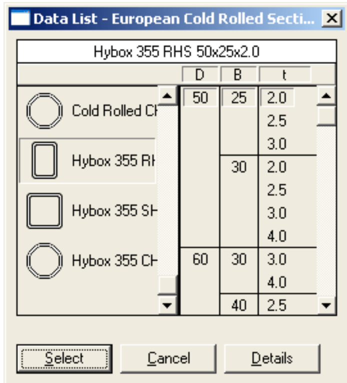
5. Choose the **Finish** of the side bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your Preferences.*

7. To enter the size of your side bracing click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

Caution *If you choose a side bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.*

9. Pick the **Grade** of material from the list of available grades.

> **Note** *If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.*

10. Once the details for your side bracing is correct click **OK** to return to the graphical display of the structure which will be updated to show the side bracing layout which you have defined.

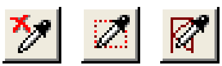
Deleting side bracings

You can delete side bracings by:

Method	*Details*
Area	all side bracings which lie totally within the selected area are deleted
Single	the side bracing you pick is deleted

To delete side bracings by area

You must use the ***Structure*** window to delete the side bracings in your structure.



1. Click the **Delete**, **Area** and **Side Bracing** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Side Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All side bracings which lie totally within the area which you select are deleted.

Note *If you have picked a type of side bracing which consists of more than one member, then that pattern will not be deleted unless all members making up that pattern are within the area you selected.*

3. Continue to identify areas until you have deleted all the side bracings that you require.

To delete side bracings singly

You must use the ***Structure*** window to delete the side bracings in your structure.

1. Click the **Delete**, **Single** and **Side Bracing** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Side Bracing* menu options).

2. Select the side bracing that you wish to delete.

3. Continue to identify side bracings until you have deleted all the side bracings that you require.

Changing side bracing attributes

When you create side bracings you give the appropriate details on a bay by bay basis. All the side bracings up the same bay take the same details (pattern, section size and grade). If you need to make changes you can do so without deleting and re-creating the entire side bracing system in any bay.

You can modify side bracing details by:

Method	*Details*
Area	all side bracings which lie totally within the area you select are modified to take the details you specify

Method	Details
Single	the side bracing you pick is modified to take the details you specify

To modify side bracing attributes by area

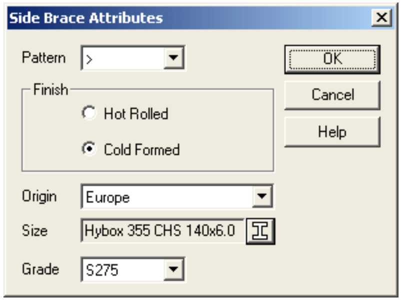
You must use the ***Structure*** window to modify the properties of the side bracings in your structure.

1. Click the **Attributes**, **Area** and **Side Bracing** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Side Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All side bracings which lie totally within this area will be selected.

Note If you have picked a type of side bracing which consists of more than one member, then that pattern will not be selected unless all members making up that pattern are within the area you selected.

3. You will see the ***Side Bracing Attributes*** dialog which allows you to set the attributes that all the selected side bracings are to take.



Note *The initial details in this dialog are those for the side bracing in your selection which is nearest to column line A.*

4. Pick the **Pattern** which you want to use from the list of available patterns.

5. Choose the **Finish** of the side bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

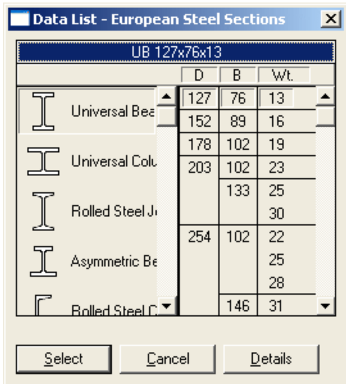
Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

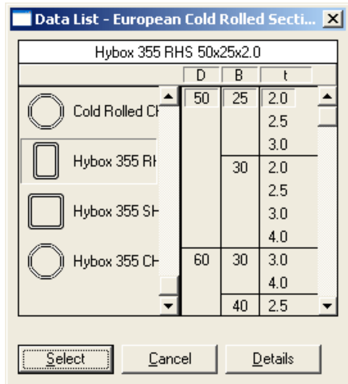
6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial* **Origin** *setting will be that of the country which is set in your* ***Preferences****.*

7. To enter the size of your side bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*





8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

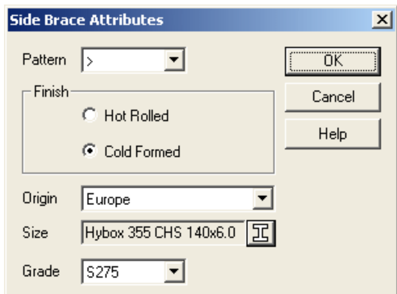
10. Once the details for your side bracings are correct click **OK** to return to the graphical display of the structure. All the selected side bracings will be updated to the attributes which you have just defined.

To modify side bracing attributes singly

You must use the ***Structure*** window to modify the properties of the side bracings in your structure.

1. Click the **Attributes**, **Single** and **Side Bracing** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Side Bracing* menu options).

2. Select a side bracing whose properties you wish to modify. You will see the ***Side Bracing Attributes*** dialog which allows you to set the attributes that this side bracing is to take.



Note *The initial details in this dialog are those for the side bracing which you select.*

Caution If you choose a side bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

3. Pick the **Pattern** which you want to use from the list of available patterns.

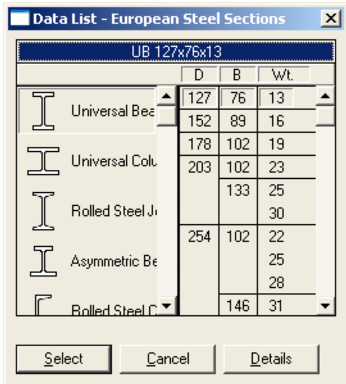
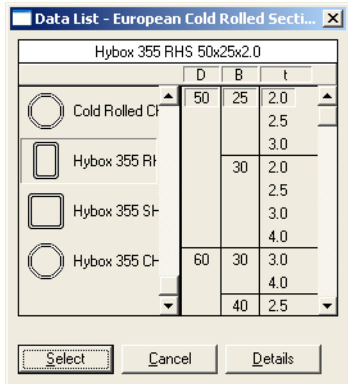
4. Choose the **Finish** of the side bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

5. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

6. To enter the size of your side bracing click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. Once the details for your side bracing are correct click **OK** to return to the graphical display of the structure. The selected side bracing will be updated to the attributes which you have just defined.

10. Continue to identify side bracings until you have modified the properties of all those that you require.

Changing side bracing positions

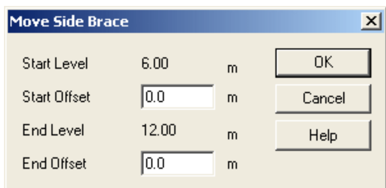
When you create side bracings in a bay, the patterns are equally spaced up the bay. If this positioning is that which you require, then you need take no further action. If, however, you need a different spacing, then you can move a particular pattern in your side bracing to the required position, the other patterns up the bay will be modified to suit your new positioning.

To move side bracings singly

You must use the ***Structure*** window to modify the positions of the side bracings in your structure.

1. Click the **Move**, **Single** and **Side Bracing** buttons (or choose the *Select / Move*, *Select / Single* and *Select / Side Bracing* menu options).

2. Select the side bracing which you wish to move. You will see the ***Move Side Bracing*** dialog which allows you to specify the distance by which the ends of the side bracing are to move.



3. Enter the **Start Offset** distance by which the start point of the side bracing (that nearest the base) is to move from its current location.

4. Enter the **End Offset** distance by which the end point of the side bracing (that farthest from the base) is to move from its current location.

Note The **Offset** you specify can be either positive (to move the pattern point farther from the base) or negative (to move the pattern point nearer to the base). The minimum distance between the start and end of a pattern is 0.75 m. If you enter offsets which violate this constraint both the **Start Offset** and **End Offset** will show in error and **OK** will be dimmed.

Note If you move a side bracing which contains two members (< or >), then any modifications you make will apply to the entire pattern and the point of the pattern will lie halfway between the moved Start and/or End of the pattern.

Note The **Position** information is updated to show the location at which the side bracing will be placed when you click **OK**.

5. Once the offset details are correct click **OK** to move the pattern. Any adjacent patterns will adjust automatically to accommodate the changes.

6. Continue to identify side bracings and move them until you have the layout that you require.

8 Working with Gable Bracings

You create, delete or change the properties of gable bracings graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with gable bracings. #####

Creating gable bracings

You can create gable bracings by:

Method	*Details*
Single	this option allows you to add a gable bracing at a particular location in your structure.

To create gable bracings singly

You must use the ***Structure*** window to define the gable bracings in your structure.

You can only create gable bracing in a span which contains gable posts. If there are no gable posts in the span, then you will see the message.

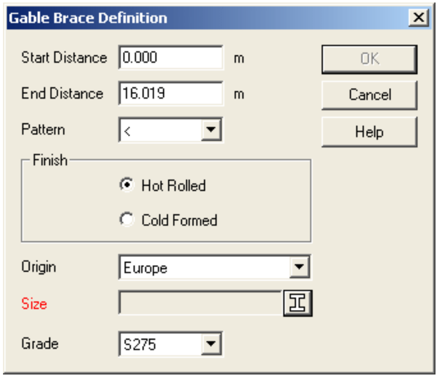
In this case you must define your gable posts before creating your gable bracing.

1. Click the **Create**, **Single** and **Gable Bracing** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Gable Bracing* menu options).

2. Click the column or gable post at that side of the region where you want to create your new gable bracing which is nearest grid line 1. The gable bracing will be created in that bay adjacent to the column or gable post you select which is farther away from column line A. The gable bracing will lie along the gable line containing the column or gable post you select.

Note For this option you need to click on the column or gable post itself and not the grid line.

You will see the ***Gable Bracing Definition*** dialog which allows you to define the gable bracing in this region.



3. Enter the **Start Distance** and **End Distance** for the new gable bracing. These dimensions are measured from datum zero and not from the column or gable post base.

Note If you add a new gable bracing to a span which already contains other gable bracings, then *Portal Frame* will automatically adjust the start and/or end distances to those bracings to take account of the new one.

4. Select the appropriate **Pattern** from the list of available patterns.

Note At this stage all the patterns up the bay are identical, you can change this later if necessary.

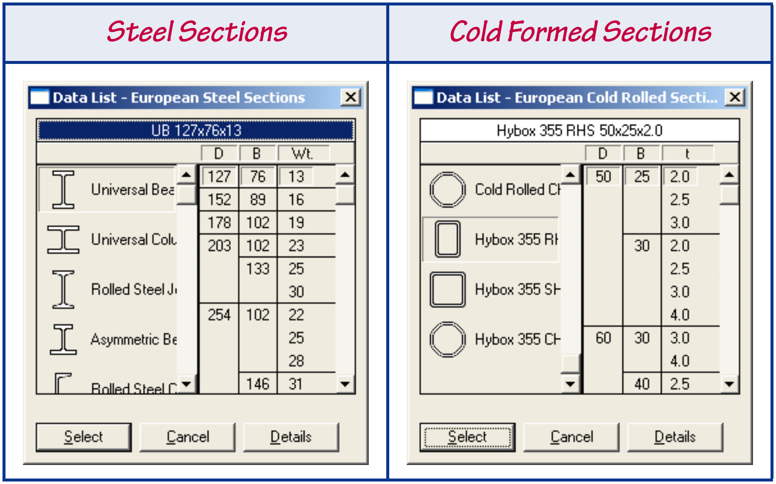
5. Choose the **Finish** of the gable bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your gable bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

Note *All the gable bracings in this location take the same size at this stage. If you want the braces to be different sizes you must change these individually once you have created them.*

Caution If you choose a gable bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.

10. Once the details for your gable bracing is correct click **OK** to return to the graphical display of the structure which will be updated to show the gable bracing layout which you have defined.

Deleting gable bracings

You can delete gable bracings by:

Method	*Details*
Area	all gable bracings which lie totally within the selected area are deleted
Single	the gable bracing you pick is deleted

To delete gable bracings by area

You must use the ***Structure*** window to delete the gable bracings in your structure.

1. Click the **Delete**, **Area** and **Gable Bracing** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Gable Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable bracings which lie totally within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the gable bracings that you require.

To delete gable bracings singly

You must use the ***Structure*** window to delete the gable bracings in your structure.

1. Click the **Delete**, **Single** and **Gable Bracing** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Gable Bracing* menu options).

2. Select the gable bracing that you wish to delete.

3. Continue to identify gable bracings until you have deleted all the gable bracings that you require.

Changing gable bracing attributes

When you create gable bracings you give the appropriate details. If you need to make changes you can do so, either working with a range of gable bracings which you select or with individual gable bracings.

You can modify gable bracing details by:

Method	*Details*
Area	all gable bracings which lie totally within the area you select are modified to take the details you specify

Method	*Details*
Single	the gable bracing you pick is modified to take the details you specify

To modify gable bracing attributes by area

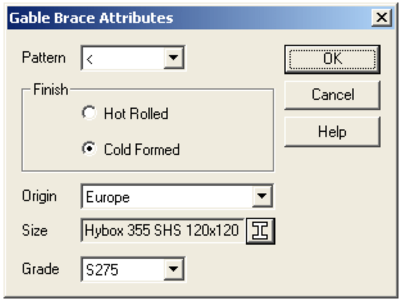
You must use the *Structure* window to modify the properties of the gable bracings in your structure.

1. Click the **Attributes**, **Area** and **Gable Bracing** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Gable Bracing* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable bracings which lie totally within this area will be selected.

Note If you have picked a type of gable bracing which consists of more than one member, then that pattern will not be selected unless all members making up that pattern are within the area you selected.

3. You will see the ***Gable Bracing Attributes*** dialog which allows you to set the attributes that all the selected gable bracings are to take.



Note The initial details in this dialog are those for the last gable bracing which has been selected.

4. Pick the **Pattern** which you want to use from the list of available patterns.

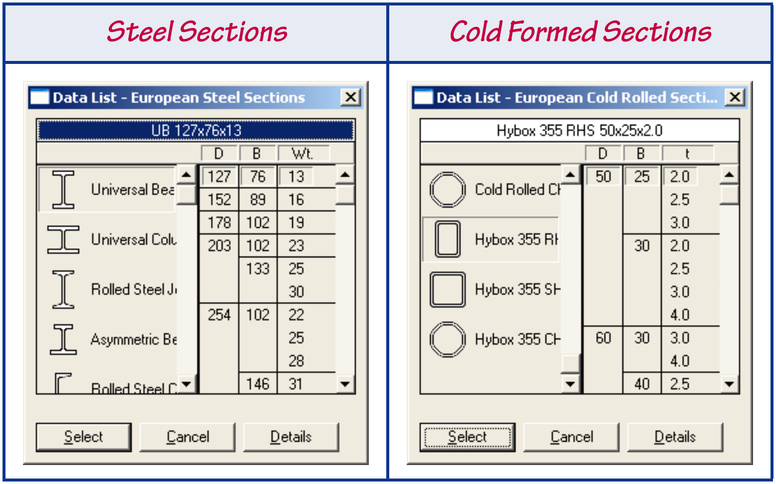
5. Choose the **Finish** of the gable bracings you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your gable bracings click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



8. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

9. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

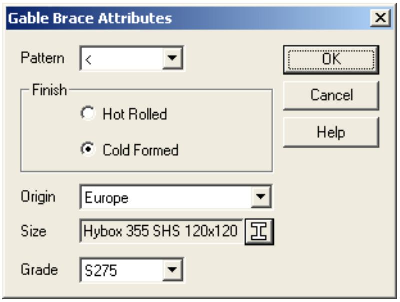
10. Once the details for your gable bracings are correct click **OK** to return to the graphical display of the structure. All the selected gable bracings will be updated to the attributes which you have just defined.

To modify gable bracing attributes singly

You must use the ***Structure*** window to modify the properties of the gable bracings in your structure.

1. Click the **Attributes**, **Single** and **Gable Bracing** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Gable Bracing* menu options).

2. Select a gable bracing whose properties you wish to modify. You will see the ***Gable Bracing Attributes*** dialog which allows you to set the attributes that this gable bracing is to take.



Note *The initial details in this dialog are those for the gable bracing which you select.*

Caution If you choose a gable bracing which comprises two members (< or >), then any modifications you make will apply to both members of the pattern. If you want different sizes for each member, then you will need to use an alternate pattern (\ or /), and amend alternate members to be of the opposite type.

3. Pick the **Pattern** which you want to use from the list of available patterns.

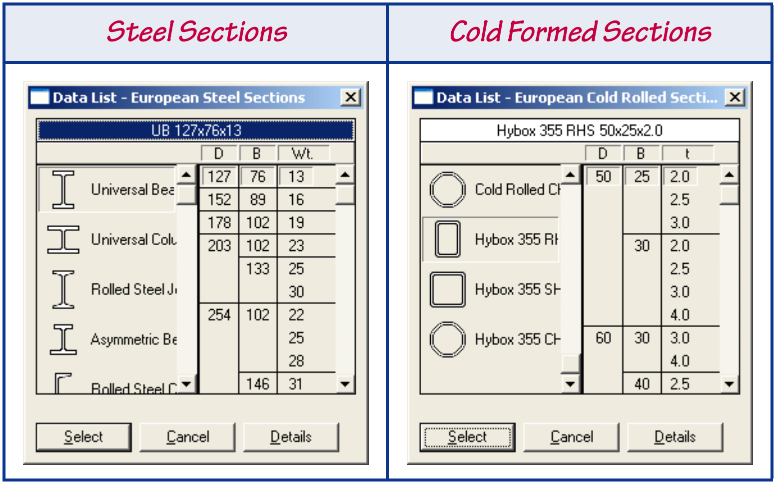
4. Choose the **Finish** of the gable bracing you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

5. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

6. To enter the size of your gable bracing click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. Once the details for your gable bracing are correct click **OK** to return to the graphical display of the structure. The selected gable bracing will be updated to the attributes which you have just defined.

10. Continue to identify gable bracings until you have modified the properties of all those that you require.

Changing gable bracing positions

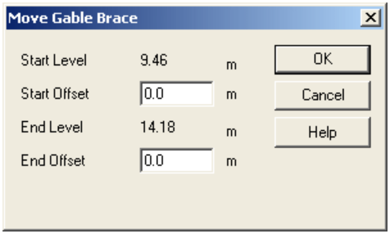
When you create gable bracings in a region, the patterns are spaced equally up the region taking the length of the longer or shorter gable post as appropriate for the pattern which you are defining. If this positioning is that which you require, then you need take no further action. If, however, you need a different spacing, then you can move a particular pattern in your gable bracing to the required position, the other patterns up the region will be modified to suit your new positioning.

To move gable bracings singly

You must use the ***Structure*** window to modify the positions of the gable bracings in your structure.

1. Click the **Move**, **Single** and **Gable Bracing** buttons (or choose the *Select / Move*, *Select / Single* and *Select / Gable Bracing* menu options).

2. Select the gable bracing which you wish to move. You will see the ***Move Gable Bracing*** dialog which allows you to specify the distance by which the ends of the gable bracing are to move.



3. Enter the **Start Offset** distance by which the start point of the gable bracing (that nearest the base) is to move from its current location.

4. Enter the **End Offset** distance by which the end point of the gable bracing (that farthest from the base) is to move from its current location.

Note The **Offset** you specify can be either positive (to move the pattern point farther from the base) or negative (to move the pattern point nearer to the base). The minimum distance between the start and end of a pattern is 0.75 m. If you enter offsets which violate this constraint both the **Start Offset** and **End Offset** will show in error and **OK** will be dimmed.

Note If you move a gable bracing which contains two members (< or >), then any modifications you make will apply to the entire pattern and the point of the pattern will lie halfway between the moved Start and/or End of the pattern.

Note The **Position** information is updated to show the location at which the gable bracing will be placed when you click **OK**.

5. Once the offset details are correct click **OK** to move the pattern. Any adjacent patterns will adjust automatically to accommodate the changes.

6. Continue to identify gable bracings and move them until you have the layout that you require.

9 Working with Eaves Ties

You create, delete or change the properties of eaves ties graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with eaves ties.

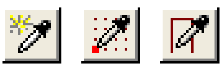
Creating eaves ties

You can create eaves ties by:

Method	*Details*
Grid Point	the eaves ties are created between the two grid points that you choose. These must be on the same column line
Single	this option allows you to add a eaves tie at a particular location in your structure.

To create eaves ties by grid points

You must use the ***Structure*** window to define the eaves ties in your structure.

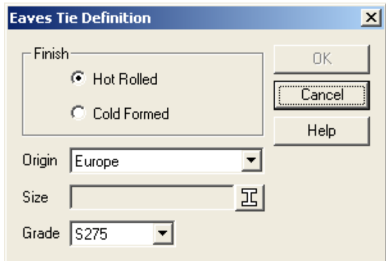


1. Click the **Create**, **Grid Points** and **EavesTie** buttons (or choose the *Select / Create*, *Select / Grid Points* and *Select / Eaves Tie* menu options).

2. Click the grid point at the base of the column where you want your eaves tie to start.

Note *You can click anywhere along the column.*

3. Now move the cursor to the grid point at the base of the column where you want your eaves tie to end. You will see a rubber line which follows the cursor. This indicates the extent of your eaves tie. When you are over the correct grid point click again and you will see the ***Eaves Tie Definition*** dialog which allows you to define the eaves tie's details.



Note Again you can click anywhere along the column.

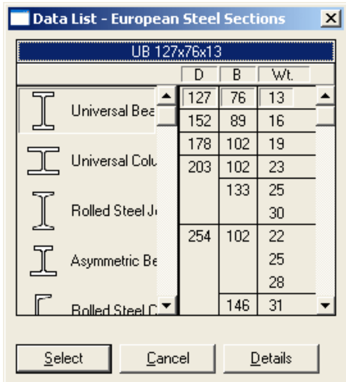
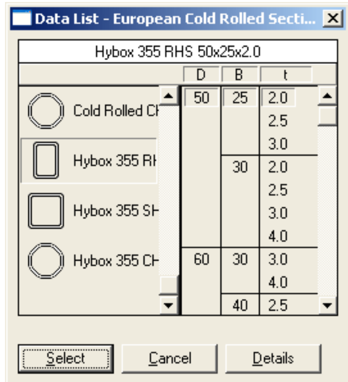
Note If you already have an eaves tie between the grid points which you select, then the dialog will show the details of the first eaves tie within the length which you specify.

4. Choose the **Finish** of the eaves ties you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

5. By default when you define your eaves ties the **Origin** is that for the country which is currently set in your ***Preferences***. If you want to use sections from a different country, then choose the country from the list of available countries.

6. To enter the size of your eaves ties click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. Once the details for your eaves tie are correct click **OK** to return to the graphical display of the structure which will be updated to show the eaves tie which you have defined.

To create eaves ties singly

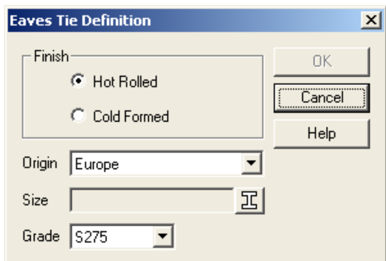
You must use the ***Structure*** window to define the eaves ties in your structure.

1. Click the **Create**, **Single** and **EavesTie** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Eaves Tie* menu options).

2. Click the column to that side of the bay in which you want to create your new eaves tie nearest grid line 1. The eaves tie will be created in the bay adjacent to the column which is farther away from grid line 1. The eaves tie will lie along the grid line containing the column you select.

Note For this option you need to click on the column itself and not the grid line.

You will see the ***Eaves Tie Definition*** dialog which allows you to define the eaves tie in this region.



Note If you pick a location for which an eaves tie is already defined, then the dialog will show the details of that eaves tie.

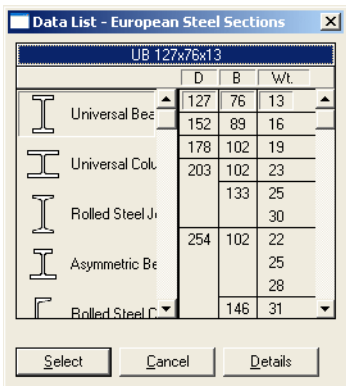
3. Choose the **Finish** of the eaves tie you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

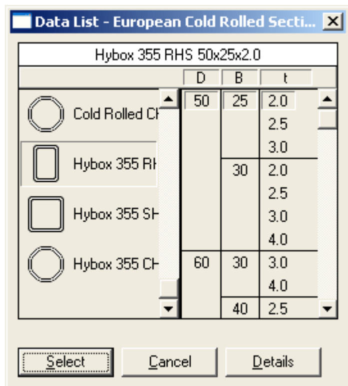
   Caution — *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

4. By default when you define your eaves ties the **Origin** is that for the country which is currently set in your ***Preferences***. If you want to use sections from a different country, then choose the country from the list of available countries.

5. To enter the size of your eaves tie click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*





6. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

7. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.*

8. Once the details for your eaves tie are correct click **OK** to return to the graphical display of the structure which will be updated to show the eaves tie which you have defined.

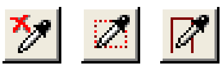
Deleting eaves ties

You can delete eaves ties by:

Method	*Details*
Area	all eaves ties which lie totally within the selected area are deleted
Single	the eaves tie you pick is deleted

To delete eaves ties by area

You must use the ***Structure*** window to delete the eaves ties in your structure.



1. Click the **Delete**, **Area** and **EavesTie** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Eaves Tie* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All eaves ties which lie totally within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the eaves ties that you require.

To delete eaves ties singly

You must use the *Structure* window to delete the eaves ties in your structure.

1. Click the **Delete**, **Single** and **EavesTie** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Eaves Tie* menu options).

2. Select the eaves tie that you wish to delete.

3. Continue to identify eaves ties until you have deleted all the eaves ties that you require.

Changing eaves tie attributes

When you create eaves ties you give the appropriate details on a bay by bay basis. If you need to make changes you can do so without deleting and re-creating the entire eaves tie system.

You can modify eaves tie details by:

Method	*Details*
Area	all eaves ties which lie totally within the area you select are modified to take the details you specify
Single	the eaves tie you pick is modified to take the details you specify

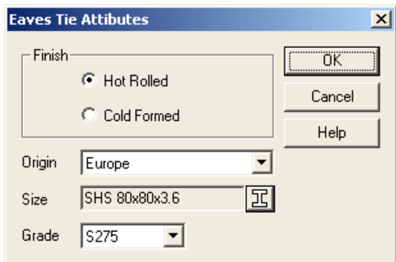
To modify eaves tie attributes by area

You must use the ***Structure*** window to modify the properties of the eaves ties in your structure.

1. Click the **Attributes**, **Area** and **EavesTie** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Eaves Tie* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All eaves ties which lie totally within this area will be selected.

3. You will see the ***Eaves Tie Attributes*** dialog which allows you to set the attributes that all the selected eaves ties are to take.



Note *The initial details in this dialog are those for the eaves tie in your selection which is nearest to grid line 1 and column line A.*

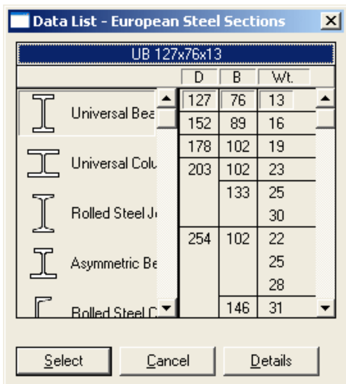
4. Choose the **Finish** of the eaves ties you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

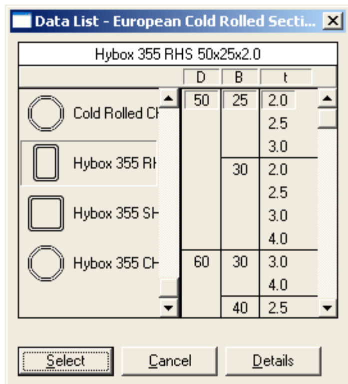
5. If you want to use sections from a different country, then choose the country of **Origin** from the list of available countries.

6. To enter the size of your eaves ties click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections



Cold Formed Sections



7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the* **Grade** *box will show* *S355**, and you can not change this since these sections are only available in this grade.*

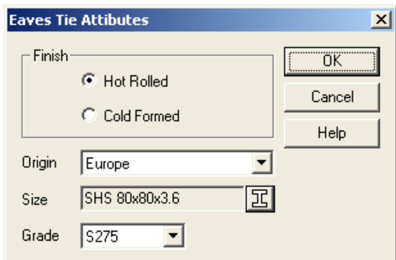
9. Once the details for your eaves ties are correct click **OK** to return to the graphical display of the structure. All the selected eaves ties will be updated to the attributes which you have just defined.

To modify eaves tie attributes singly

You must use the ***Structure*** window to modify the properties of the eaves ties in your structure.

1. Click the **Attributes**, **Single** and **EavesTie** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Eaves Tie* menu options).

2. Select a eaves tie whose properties you wish to modify. You will see the ***Eaves Tie Attributes*** dialog which allows you to set the attributes that this eaves tie is to take.



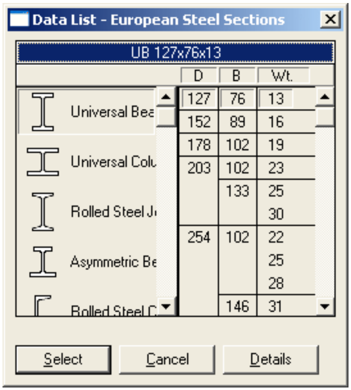
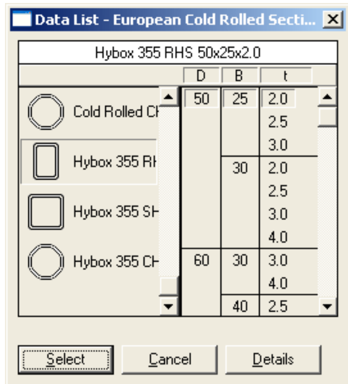
Note *The initial details in this dialog are those for the eaves tie which you select.*

3. Choose the **Finish** of the eaves ties you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

4. If you want to use sections from a different country, then choose the country of **Origin** from the list of available countries.

5. To enter the size of your eaves ties click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

6. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

7. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

8. Once the details for your eaves tie are correct click **OK** to return to the graphical display of the structure. The selected eaves tie will be updated to the attributes which you have just defined.

9. Continue to identify eaves ties until you have modified the properties of all those that you require.

10 Working with Cold-Rolled Purlins

You create, delete or change the properties of cold-rolled purlins graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with cold-rolled purlins.

Creating cold-rolled purlins

You can create cold-rolled purlins by:

Method	*Details*
Grid Line	the cold-rolled purlins are created on the grid line you choose. The position of the cold-rolled purlins which start in a particular bay is determined by the layout of restraints on the rafter nearest to grid line 1. If there are no restraints for this rafter, then no cold-rolled purlins will start here. For double span cold-rolled purlins which pass across the current rafter the layout depends on the restraints positions on the previous rafter. If there are no restraints on this rafter, then no double span cold-rolled purlins will cross the current rafter.
Single	this option allows you to add cold-rolled purlins to a particular frame member.

To create cold-rolled purlins by grid line

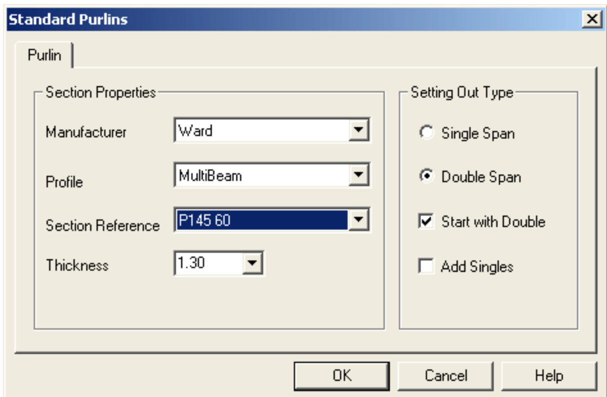
You must use the ***Structure*** window to define the cold-rolled purlins in your structure.

1. Click the **Create**, **Grid Line** and **Standard Purlin** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Standard Purlin* menu options).

2. Select the grid line from the display on which you want to create the cold-rolled purlins.

Note *Cold-Rolled Purlins are created based on the restraints defined for a member, on a member by member basis. If there are no restraints for a particular member on this grid line, then no cold-rolled purlins will be start here.*

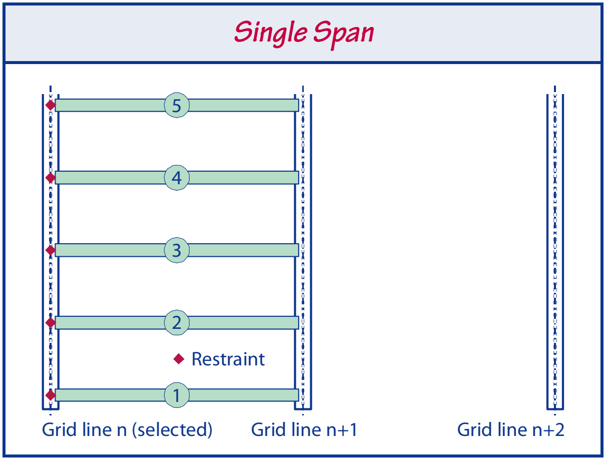
You will see the ***Standard Purlin*** dialog which allows you to define the purlin's details.

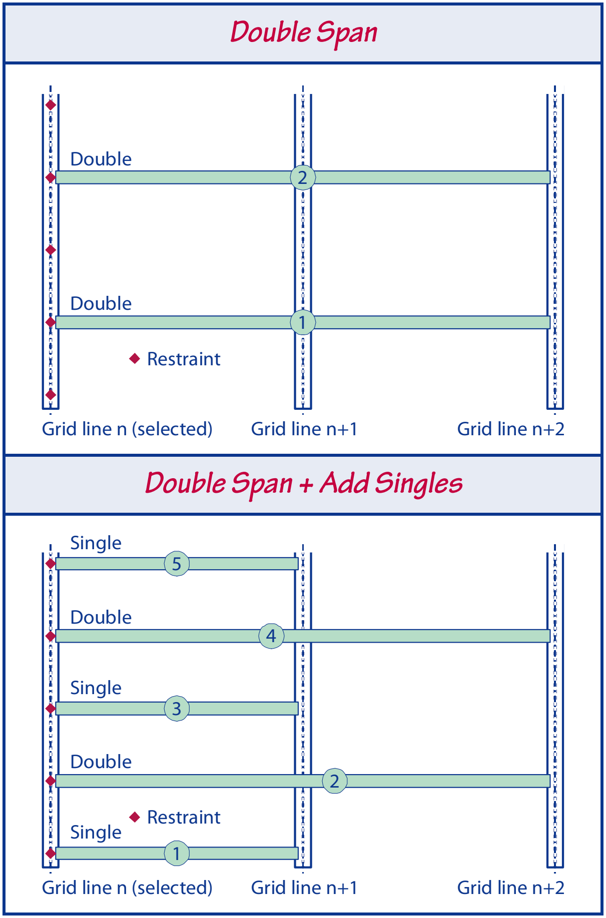


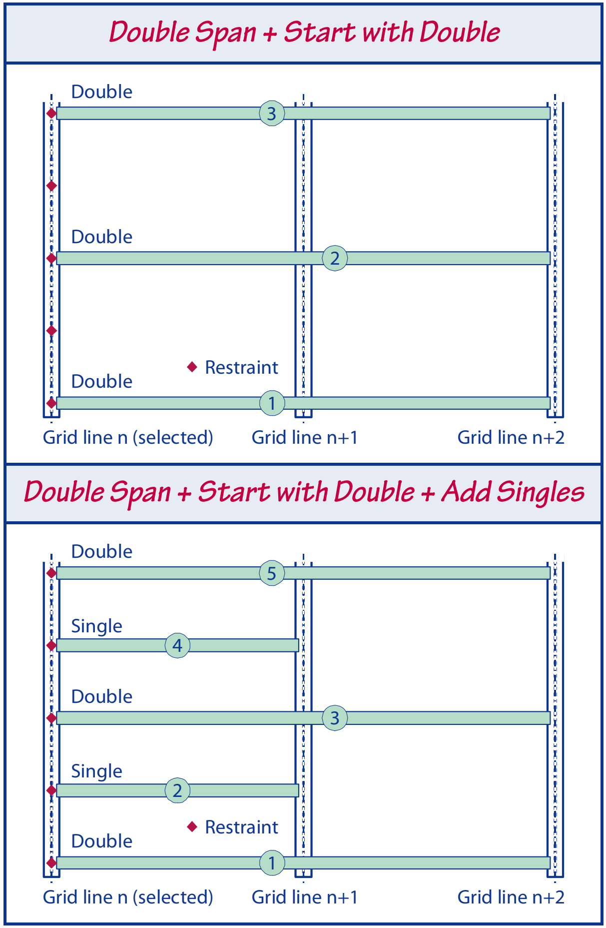
3. Pick the purlin **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Now choose the layout details for the purlins you want to add. If you choose the **Double Span** option, then:
   - tick **Start with Double** if you want the first purlin (that at the restraint nearest to the eaves) to be double spanning. If you don't tick this option, then the first purlin will be set to be single spanning and will only be created if **Add Singles** is ticked.
   - tick **Add Singles** if you want to add single span purlins at this grid line,

   The various layouts are shown below.




Double Span
Double
2
Double
1
Restraint
Grid line n (selected)
Grid line n+1
Grid line n+2
Double Span + Add Singles
Single
5
Double
4
Single
3
Double
2
Restraint
Single
1
Grid line n (selected)
Grid line n+1
Grid line n+2



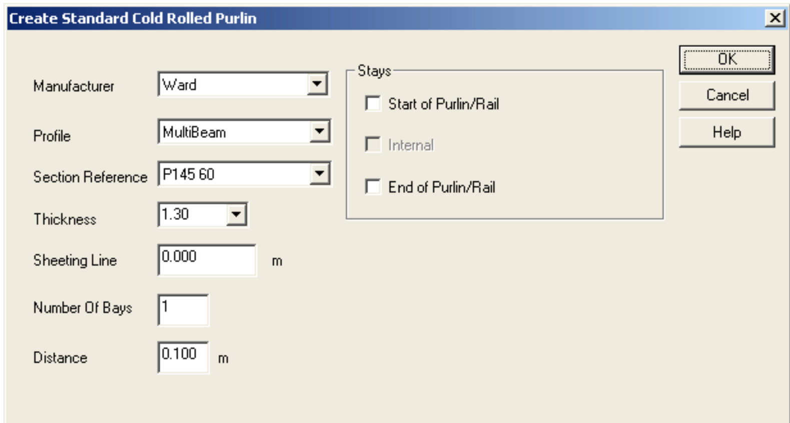
5. Once your settings are correct click **OK** to create the cold-rolled purlins using the current information.

To create cold-rolled purlins singly

You must use the *Structure* window to define the cold-rolled purlins in your structure.

1. Click the **Create**, **Single** and **Standard Purlin** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Standard Purlin* menu options).

2. Click the rafter on which you want to create your new cold-rolled purlin.



3. Pick the purlin **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Enter the **Sheeting Line** distance for the purlin - this is the distance from the centre-line of the rafter which you picked to the inner face of the sheeting.

5. Specify the **Number of Bays** which your purlin is to cross. *Portal Frame* will not allow you to specify a number of bays greater than that which is available.

Example *If there are 3 frames to the right of the rafter on which you are creating purlins, the maximum number of bays would be 3.*

6. If you want to generate stays from your purlin to the rafters, then ensure that the appropriate **Stays** boxes are ticked.

Note *You can only define internal stays when your purlin spans two or more bays.*

7. Enter the **Distance** to the purlin from the origin of your selected rafter and then click **OK** to generate a purlin with these settings.

Deleting cold-rolled purlins

You can delete cold-rolled purlins by:

Method	*Details*
Grid Line	all cold-rolled purlins on the selected grid line are deleted
Area	all cold-rolled purlins which lie wholly within the selected area are deleted
Single	the cold-rolled purlin you pick is deleted

To delete cold-rolled purlins by grid line

You must use the ***Structure*** window to delete the cold-rolled purlins in your structure.

1. Click the **Delete**, **Grid Line** and **Standard Purlin** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Standard Purlin* menu options).

2. Click the grid line along which the cold-rolled purlin(s) you want to delete lie. All cold-rolled purlins which start on this grid line are deleted.

3. Continue to identify grid lines until you have deleted all the cold-rolled purlins that you require.

To delete cold-rolled purlins by area

You must use the ***Structure*** window to delete the cold-rolled purlins in your structure.

1. Click the **Delete**, **Area** and **Standard Purlin** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Standard Purlin* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All cold-rolled purlins which lie completely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the cold-rolled purlins that you require.

To delete cold-rolled purlins singly

You must use the ***Structure*** window to delete the cold-rolled purlins in your structure.

1. Click the **Delete**, **Single** and **Standard Purlin** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Standard Purlin* menu options).

2. Select the cold-rolled purlin that you wish to delete.

3. Continue to identify cold-rolled purlins until you have deleted all the cold-rolled purlins that you require.

Changing cold-rolled purlin attributes

When you create cold-rolled purlins you give the appropriate details for those cold-rolled purlins. If you need to make changes you can do so without deleting and re-creating these cold-rolled purlins.

You can modify cold-rolled purlins details by:

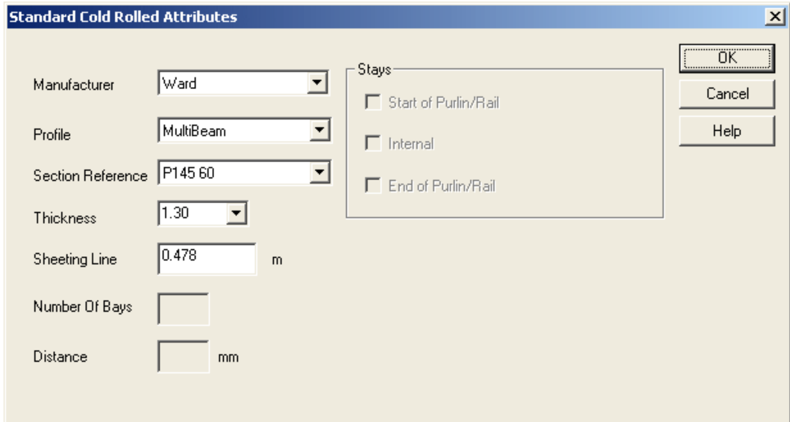
Method	*Details*
Grid Line	all cold-rolled purlins on the selected grid line are modified to take the details you specify
Area	all cold-rolled purlins which lie totally within the area you select are modified to take the details you specify
Single	the cold-rolled purlin you pick is modified to take the details you specify

To modify cold-rolled purlin attributes by grid line

You must use the ***Structure*** window to modify the properties of the cold-rolled purlins in your structure.

1. Click the **Attributes**, **Grid Line** and **Standard Purlin** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Standard Purlin* menu options).

2. Click the grid line along which the cold-rolled purlin(s) whose details you want to modify lie. ***Portal Frame*** will highlight all the cold-rolled purlins on this grid line and you will see the ***Cold-Rolled Purlin Attributes*** dialog which allows you to set the attributes that all cold-rolled purlins along this grid line are to take.



Note *The initial details in this dialog are those for the cold-rolled purlin nearest to column line A on the line you select.*

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line** from the face of the gable posts. The default value is the distance from the centre line of the rafter with which the first cold-rolled purlin is associated to the inside face of the sheeting and is used to place the sheeting rails exactly when they are transferred to ***3D+***.

Since you are working with all the cold-rolled purlins on a particular line, you cannot define the number of spans each cold-rolled purlin covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

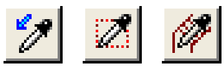
Help *see "To modify cold-rolled purlins attributes singly" on page 707.*

5. Once the details for your cold-rolled purlins are correct click **OK** to return to the graphical display of the structure. All the highlighted cold-rolled purlins will be updated to the attributes which you have just defined.

6. Continue to identify lines until you have modified the properties of all the cold-rolled purlins that you require.

To modify cold-rolled purlin attributes by area

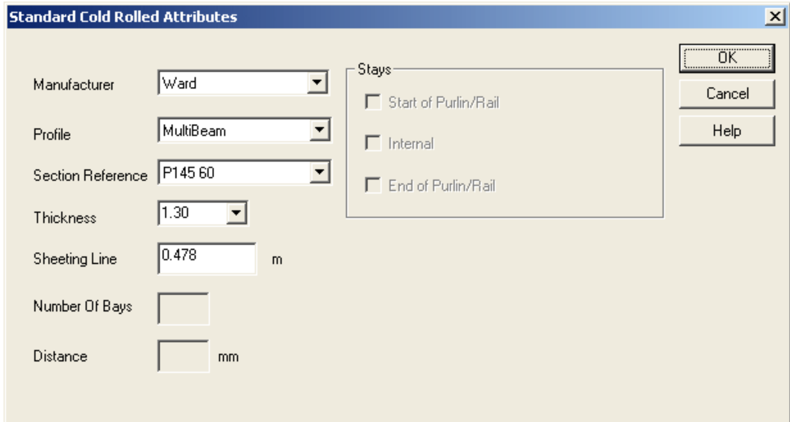
You must use the ***Structure*** window to modify the properties of the cold-rolled purlins in your structure.



1. Click the **Attributes**, **Area** and **Standard Purlin** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Standard Purlin* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All cold-rolled

purlins which lie entirely within this area will be selected. You will see the ***Cold-Rolled Purlin Attributes*** dialog which allows you to set the attributes that all the selected cold-rolled purlins are to take.



Note *The initial details in this dialog are those for the cold-rolled purlin nearest to grid line 1 and column line A in the area you select.*

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line** from the face of the gable posts. The default value is the distance from the centre line of the rafter with which the first cold-rolled purlin is associated to the inside face of the sheeting and is used to place the sheeting rails exactly when they are transferred to ***3D+***.

Since you are working with all the cold-rolled purlins in a particular area, you cannot define the number of spans each cold-rolled purlin covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

Help *see "To modify cold-rolled purlins attributes singly" on page 707.*

5. Once the details for your cold-rolled purlins are correct click **OK** to return to the graphical display of the structure. All the highlighted cold-rolled purlins will be updated to the attributes which you have just defined.

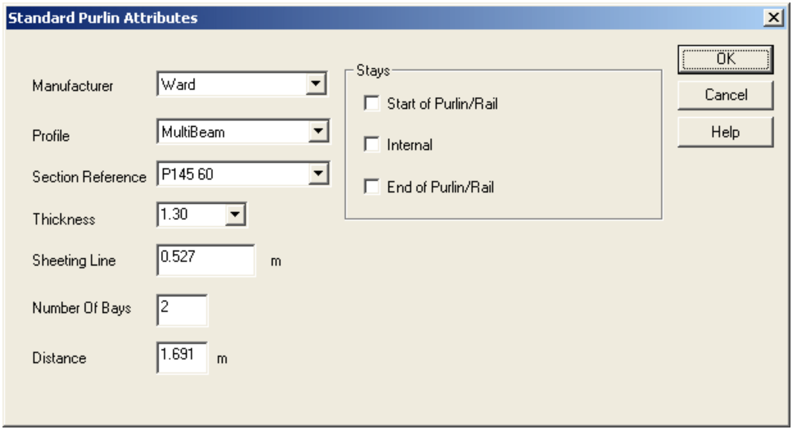
6. Continue to identify areas until you have modified the properties of all the cold-rolled purlins that you require.

To modify cold-rolled purlins attributes singly

You must use the ***Structure*** window to modify the properties of the cold-rolled purlins in your structure.

1. Click the **Attributes**, **Single** and **Standard Purlin** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Standard Purlin* menu options).

2. Select a cold-rolled purlin whose properties you wish to modify. You will see the ***Cold-Rolled Purlin Attributes*** dialog which allows you to set the attributes that this cold-rolled purlin is to take.



3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**. This is the distance from the centre-line of the rafter to the inner face of the sheeting and is used to place the cold-rolled purlin exactly when it is transferred to ***3D+***.

5. Specify the **Number of Bays** which your purlin is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

E

Example *If there are 3 frames to the right of the rafter on which you are creating purlins, the maximum number of bays would be 3.*

Caution If you increase the number of bays a particular purlin covers, and this causes it to overlap other purlins, then you will need to delete any unwanted purlins yourself since *Portal Frame* will not do this automatically.

6. Enter the **Distance** from the origin of the rafter to the purlin, measured along the centre-line of the rafter.

7. If you need to provide stays to the inner flange of the rafter, then choose the positions where restraints are to be provided. You can choose any or all of **Start of Purlin/Rail**, **Internal** and **End of Purlin/Rail**. These settings will be applied to the current cold-rolled purlin.

Note The Internal option only has an effect for purlins which span 2 or more bays.

8. Once the details for your cold-rolled purlin are correct click **OK** to return to the graphical display of the structure. The cold-rolled purlin will be updated to the attributes which you have just defined.

9. Continue to identify cold-rolled purlins until you have modified the properties of all those that you require.

11 Working with Cold Rolled Side Rails

You create, delete or change the properties of cold rolled side rails graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with cold rolled side rails.

Creating cold rolled side rails

You can create cold rolled side rails by:

Method	*Details*
Grid Point	the cold rolled side rails are created between the two grid points that you choose. These must be on the same column line
Grid Line	the cold rolled side rails are created on the grid line you choose. The position of the cold rolled side rails which start in a particular bay is determined by the layout of restraints on the column nearest to grid line 1. If there are no restraints for this column, then no cold rolled side rails will start here. For double span cold rolled side rails which pass across the current column the layout depends on the restraints positions on the previous column. If there are no restraints on this column, then no double span cold rolled side rails will cross the current rafter.
Single	this option allows you to add cold rolled side rails to a particular frame member.

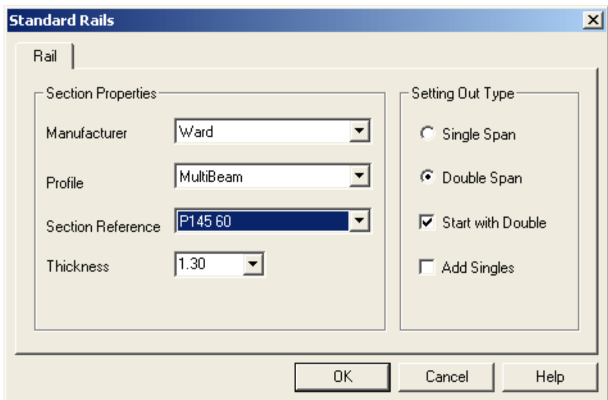
To create cold rolled side rails by grid points

You must use the *Structure* window to define the cold rolled side rails in your structure.

1. Click the **Create**, **Grid Points** and **Standard Rail** buttons (or choose the *Select / Create*, *Select / Grid Points* and *Select / Standard Rail* menu options).

2. Click the grid point at the base of the column where you want your run of rails to start.

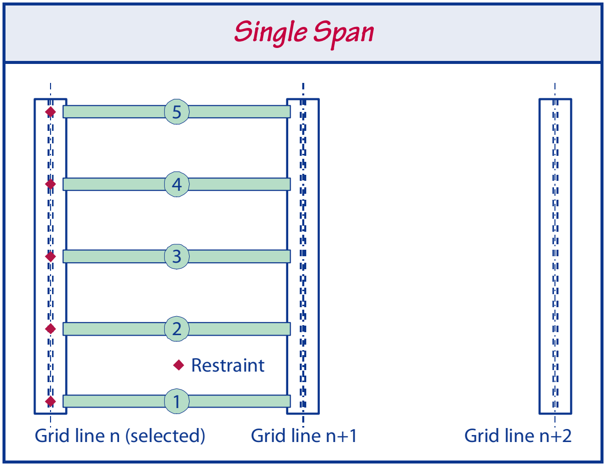
3. Now move the cursor to the grid point at the base of the column where you want your run of rails to end. You will see a rubber line which follows the cursor. This indicates the extent of your run of rails. When you are over the correct grid point click again and you will see the *Standard Rails* dialog which allows you to define the cold rolled side rail's details.



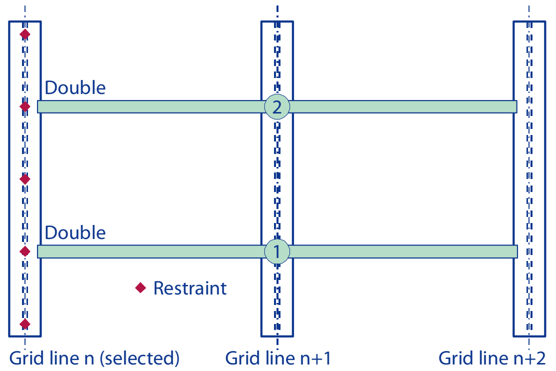
4. Pick the rail **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

5. Now choose the layout details for the rails you want to add. If you choose the **Double Span** option, then:
   - tick **Start with Double** if you want the first rail (that at the restraint nearest to the eaves) to be double spanning. If you don't tick this option, then the first rail will be set to be single spanning and will only be created if **Add Singles** is ticked.
   - tick **Add Singles** if you want to add single span rails at this grid line,

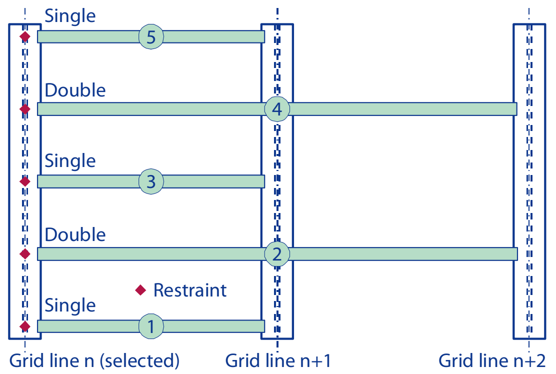
   The various layouts are shown below.



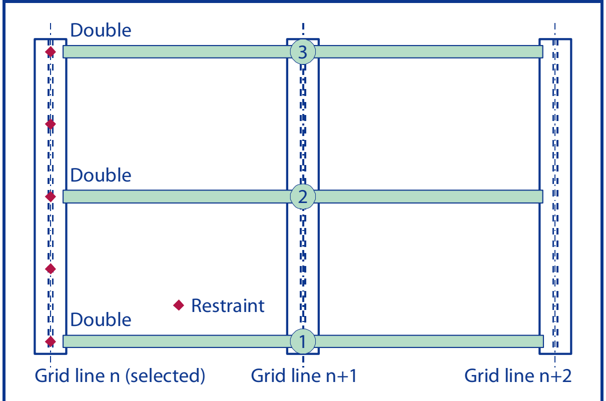
Double Span



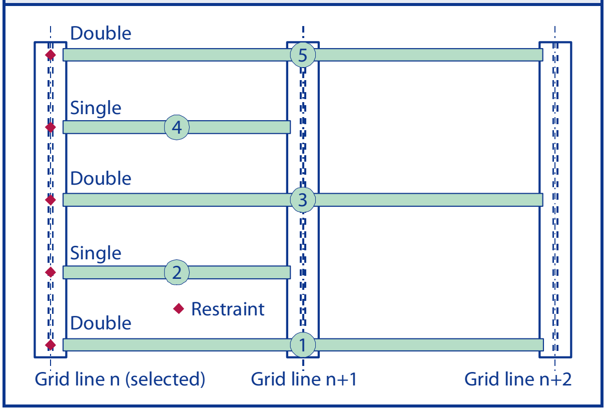
Double Span + Add Singles



Double Span + Start with Double



Double Span + Start with Double + Add Singles



6. Once your settings are correct click **OK** to create the cold rolled side rails using the current information.

To create cold rolled side rails by grid line

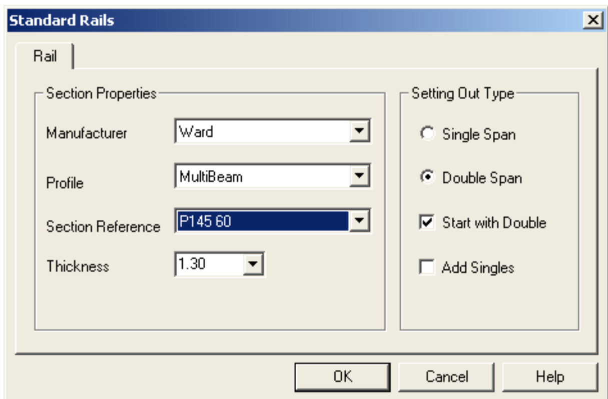
You must use the ***Structure*** window to define the cold rolled side rails in your structure.

1. Click the **Create**, **Grid Line** and **Standard Rail** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Standard Rail* menu options).

2. Select the grid line from the display on which you want to create the cold rolled side rails.

Note *Cold Rolled Side Rails are created based on the restraints defined for a member, on a member by member basis. If there are no restraints for a particular member on this grid line, then no cold rolled side rails will be created.*

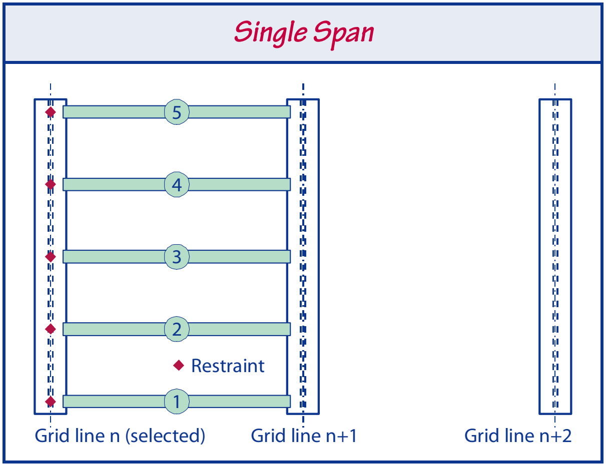
You will see the ***Standard Rails*** dialog which allows you to define the rail's details.

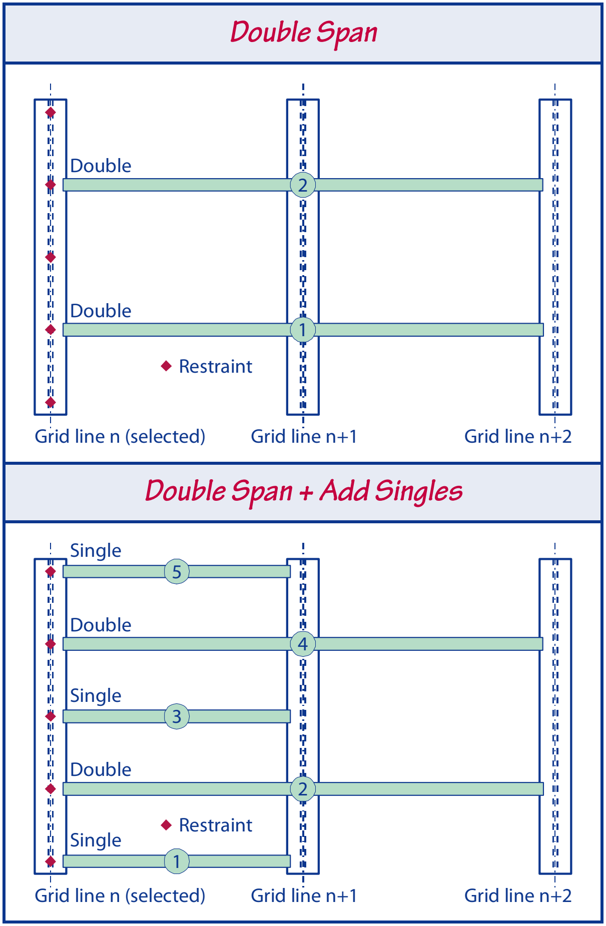


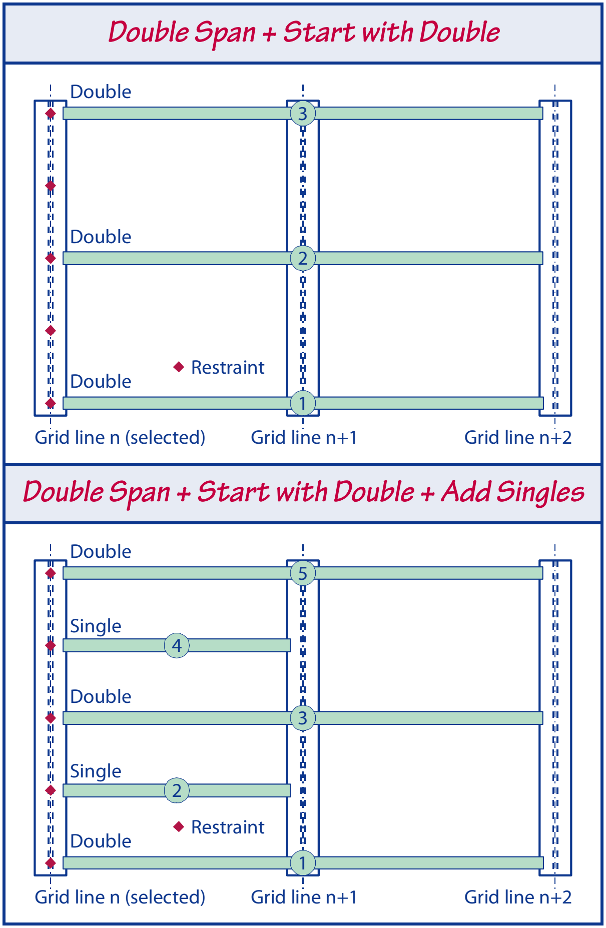
3. Pick the rail **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Now choose the layout details for the rails you want to add. If you choose the **Double Span** option, then:
   - tick **Start with Double** if you want the first rail (that at the restraint nearest to the eaves) to be double spanning. If you don't tick this option, then the first rail will be set to be single spanning and will only be created if **Add Singles** is ticked.
   - tick **Add Singles** if you want to add single span rails at this grid line,

The various layouts are shown below.




Double Span
Double
2
Double
1
Restraint
Grid line n (selected)
Grid line n+1
Grid line n+2
Double Span + Add Singles
Single
5
Double
4
Single
3
Double
2
Restraint
Single
1
Grid line n (selected)
Grid line n+1
Grid line n+2



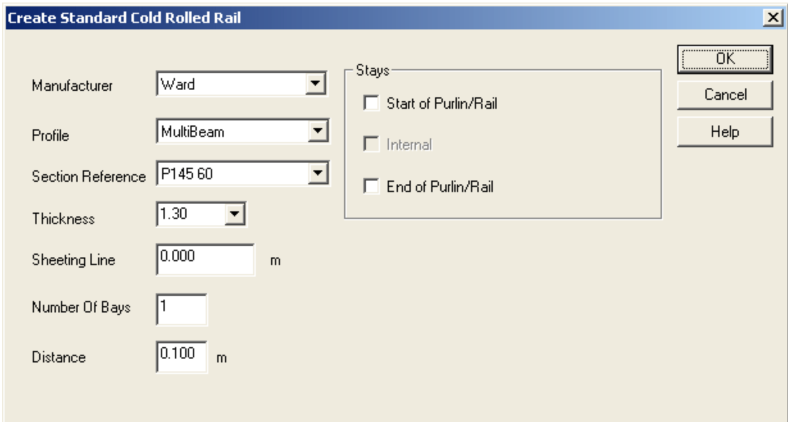
5. Once your settings are correct click **OK** to create the cold rolled side rails using the current information.

To create cold rolled side rails singly

You must use the ***Structure*** window to define the cold rolled side rails in your structure.

1. Click the **Create**, **Single** and **Standard Rail** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Standard Rail* menu options).

2. Click the column on which you want to create your new cold rolled side rail.



3. Pick the rail **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Enter the **Sheeting Line** distance for the rail - this is the distance from the centre-line of the column which you picked to the inner face of the sheeting.

5. Specify the **Number of Bays** which your rail is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

Example *If there are 3 frames to the right of the column on which you are creating rails, the maximum number of bays would be 3.*

6. If you want to generate stays from your rail to the stanchions, then ensure that the appropriate **Stays** boxes are ticked.

Note *You can only define internal stays when your rail spans two or more bays.*

7. Enter the distance to the rail from the origin of your selected column and then click **OK** to generate a rail with these settings.

Deleting cold rolled side rails

You can delete cold rolled side rails by:

Method	*Details*
Grid Line	all cold rolled side rails on the selected grid line are deleted
Area	all cold rolled side rails which lie wholly within the selected area are deleted
Single	the cold rolled side rail you pick is deleted

To delete cold rolled side rails by grid line

You must use the ***Structure*** window to delete the cold rolled side rails in your structure.

1. Click the **Delete**, **Grid Line** and **Standard Rail** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Standard Rail* menu options).

2. Click the grid line along which the cold rolled side rail(s) you want to delete lie. All cold rolled side rails along this line are deleted.

3. Continue to identify grid lines until you have deleted all the cold rolled side rails that you require.

To delete cold rolled side rails by area

You must use the ***Structure*** window to delete the cold rolled side rails in your structure.

1. Click the **Delete**, **Area** and **Standard Rail** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Standard Rail* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All cold rolled side rails which lie completely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the cold rolled side rails that you require.

To delete cold rolled side rails singly

You must use the ***Structure*** window to delete the cold rolled side rails in your structure.

1. Click the **Delete**, **Single** and **Standard Rail** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Standard Rail* menu options).

2. Select the cold rolled side rail that you wish to delete.

3. Continue to identify cold rolled side rails until you have deleted all the cold rolled side rails that you require.

Changing cold rolled side rail attributes

When you create cold rolled side rails you give the appropriate details for those cold rolled side rails. If you need to make changes you can do so without deleting and re-creating these cold rolled side rails.

You can modify cold rolled side rails details by:

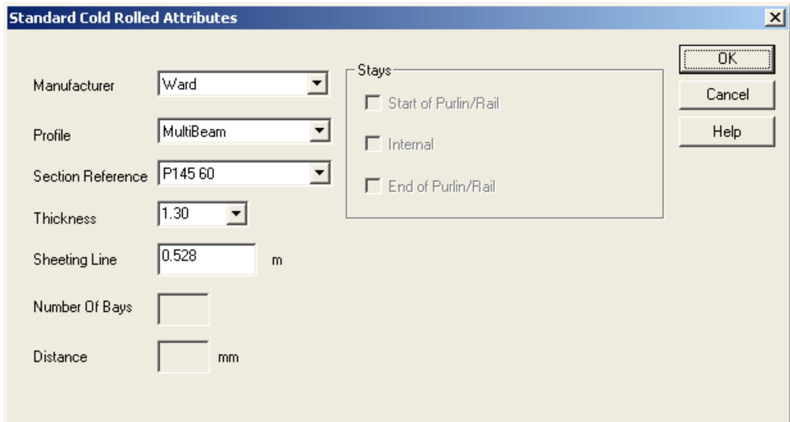
Method	*Details*
Grid Line	all cold rolled side rails on the selected grid line are modified to take the details you specify
Area	all cold rolled side rails which lie totally within the area you select are modified to take the details you specify
Single	the cold rolled side rail you pick is modified to take the details you specify

To modify cold rolled side rail attributes by grid line

You must use the ***Structure*** window to modify the properties of the cold rolled side rails in your structure.

1. Click the **Attributes**, **Grid Line** and **Standard Rail** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Standard Rail* menu options).

2. Click the grid line along which the cold rolled side rail(s) whose details you want to modify lie. ***Portal Frame*** will highlight all the cold rolled side rails on this grid line and you will see the ***Cold Rolled Side Rail Attributes*** dialog which allows you to set the attributes that all cold rolled side rails along this grid line are to take.



Note *The initial details in this dialog are those for the cold rolled side rail nearest to column line A on the line you select.*

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line** from the face of the gable posts. The default value is the distance from the centre line of the column with which the first cold rolled side rail is associated to the inside face of the sheeting and is used to place the sheeting rails exactly when they are transferred to ***3D+***.

Since you are working with all the cold rolled side rails on a particular line, you cannot define the number of spans each cold rolled side rail covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

Help *see "To modify cold rolled side rails attributes singly" on page 726.*

5. Once the details for your cold rolled side rails are correct click **OK** to return to the graphical display of the structure. All the highlighted cold rolled side rails will be updated to the attributes which you have just defined.

6. Continue to identify lines until you have modified the properties of all the cold rolled side rails that you require.

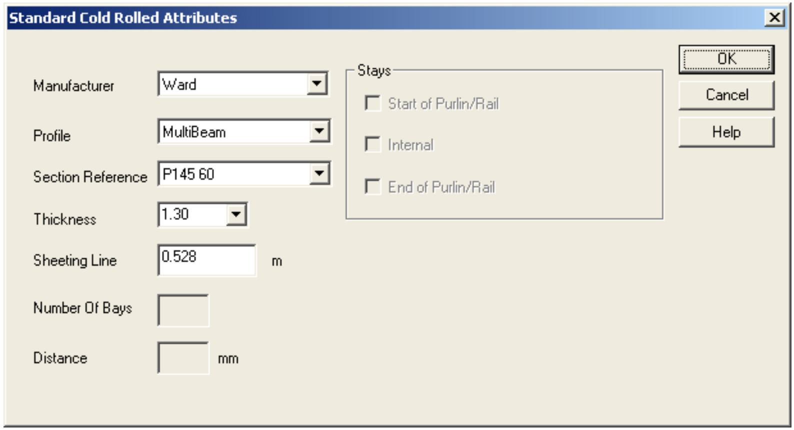
To modify cold rolled side rail attributes by area

You must use the ***Structure*** window to modify the properties of the cold rolled side rails in your structure.

1. Click the **Attributes**, **Area** and **Standard Rail** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Standard Rail* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All cold rolled side

rails which lie entirely within this area will be selected. You will see the ***Cold Rolled Side Rail Attributes*** dialog which allows you to set the attributes that all the selected cold rolled side rails are to take.



Note *The initial details in this dialog are those for the cold rolled side rail nearest to grid line 1 and column line A in the area you select.*

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line** from the face of the gable posts. The default value is the distance from the centre line of the column with which the first cold rolled side rail is associated to the inside face of the sheeting and is used to place the sheeting rails exactly when they are transferred to ***3D+***.

Since you are working with all the cold rolled side rails in a particular area, you cannot define the number of spans each cold rolled side rail covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

?

Help *see* *"To modify cold rolled side rails attributes singly"* on page 726.

5. Once the details for your cold rolled side rails are correct click **OK** to return to the graphical display of the structure. All the highlighted cold rolled side rails will be updated to the attributes which you have just defined.

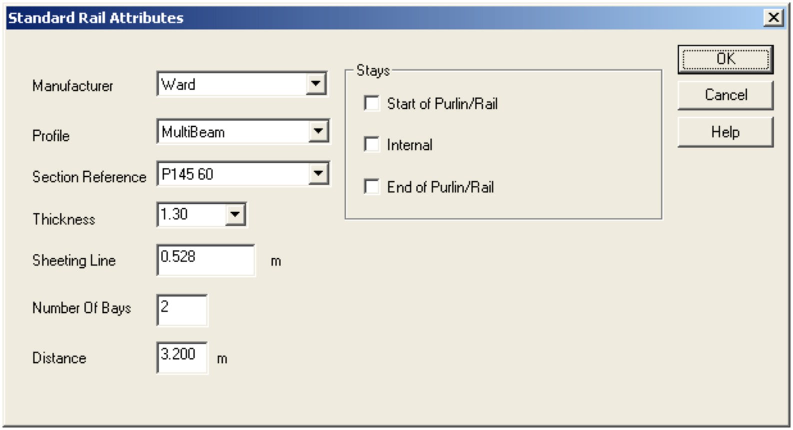
6. Continue to identify areas until you have modified the properties of all the cold rolled side rails that you require.

To modify cold rolled side rails attributes singly

You must use the ***Structure*** window to modify the properties of the cold rolled side rails in your structure.

1. Click the **Attributes**, **Single** and **Standard Rail** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Standard Rail* menu options).

2. Select a cold rolled side rail whose properties you wish to modify. You will see the ***Cold Rolled Side Rail Attributes*** dialog which allows you to set the attributes that this cold rolled side rail is to take.



3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**. This is the distance from the centre-line of the column to the inner face of the sheeting and is used to place the cold rolled side rail exactly when it is transferred to ***3D+***.

5. Specify the **Number of Bays** which your rail is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

E

Example *If there are 3 frames to the right of the column on which you are creating rails, the maximum number of bays would be 3.*

Caution If you increase the number of bays a particular rail covers, and this causes it to overlap other rails, then you will need to delete any unwanted rails yourself since *Portal Frame* will not do this automatically.

6. Enter the **Distance** from the origin of the column to the rail, measured along the centre-line of the column.

7. If you need to provide stays to the inner flange of the column, then choose the positions where restraints are to be provided. You can choose any or all of **Start of Purlin/Rail**, **Internal** and **End of Purlin/Rail**. These settings will be applied to the current cold rolled side rail.

Note The Internal option only has an effect for rails which span 2 or more bays.

8. Once the details for your cold rolled side rail are correct click **OK** to return to the graphical display of the structure. The cold rolled side rail will be updated to the attributes which you have just defined.

9. Continue to identify cold rolled side rails until you have modified the properties of all those that you require.

12 Working with Hip Purlins

You create, delete or change the properties of hip purlins graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with hip purlins.

Creating hip purlins

You can create hip purlins by:

Method	*Details*
Grid Line	the hip purlins are created in the hip area associated with the monopitch grid line you choose. The positions of the hip purlins are predicated by the points at which the main runs of purlin along the left-hand side of each span in your building intersect with the hip raker. You must therefore have created runs of purlin before using this option, otherwise no hip purlins will be created.
Single	this option allows you to add hip purlins into a particular span. You can specify the details of the hip purlin, its exact position and the number of spans it covers.

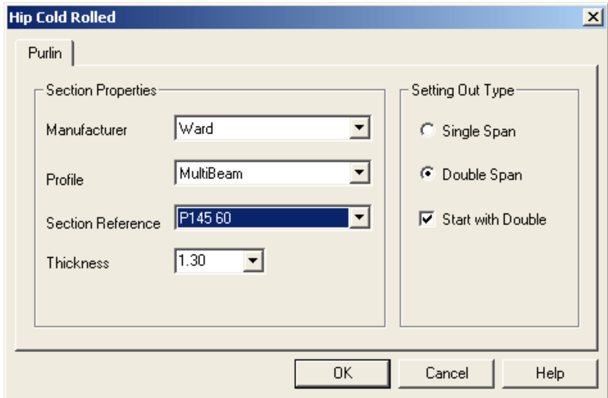
To create hip purlins by grid line

You must use the ***Structure*** window to define the hip purlins in your structure.

1. Click the **Create**, **Grid Line** and **Hip Purlin** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Hip Purlin* menu options).

2. From the display select the hip region's associated grid line (this is the monopitch grid line which you selected for the eaves of the hip).

   You will see the ***Hip Purlin Definition*** dialog.



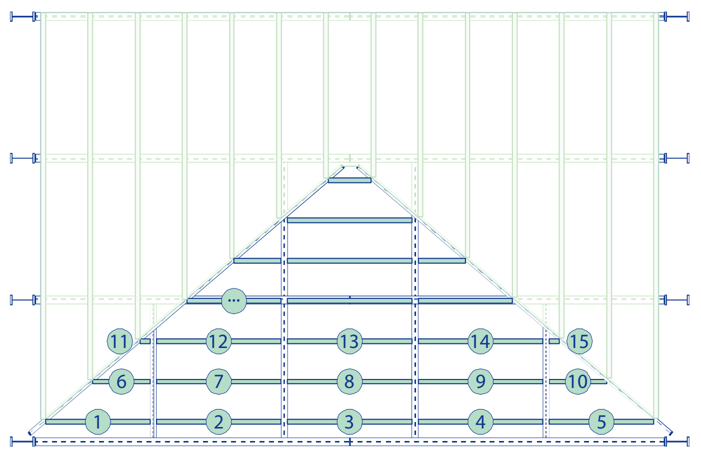
Caution If you have modified a span's hip purlins, by amending their details, position or both, and you **OK** this dialog, you will:

- position the hip purlins to their initial settings (based on the points where the main runs of purlin along your building intersect with the hip raker,
- change the **Manufacturer**, **Profile**, **Section Reference**, and **Setting Out Type** details of all hip purlins along this grid line to those shown in this dialog.

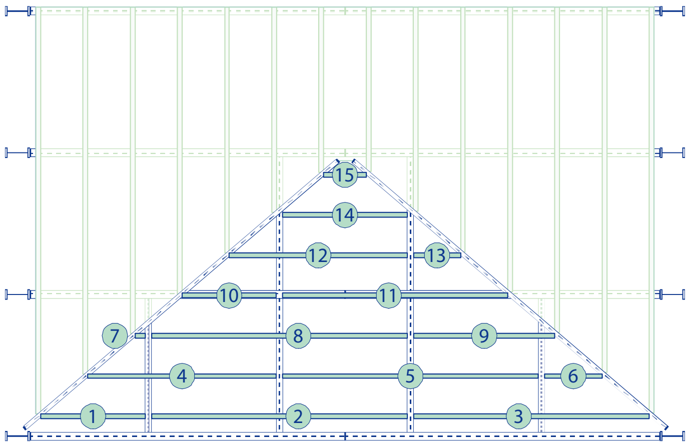
3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Pick the **Setting Out Type** details that you want to use. The various options are covered in the following diagrams. These have been rotated by 90° clockwise for clarity.

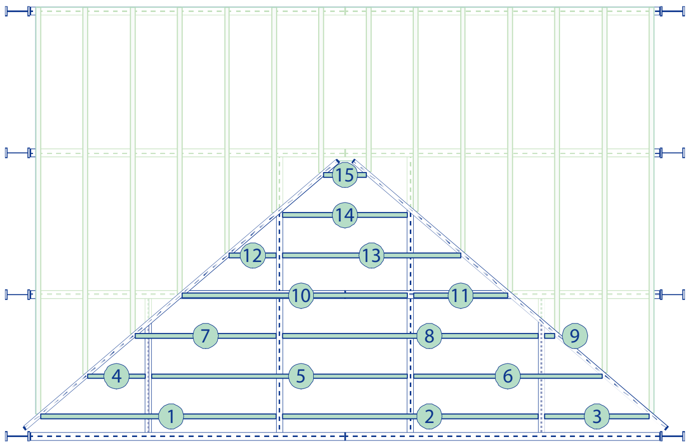
Single Span



Double Span



Double Span + Start with Double



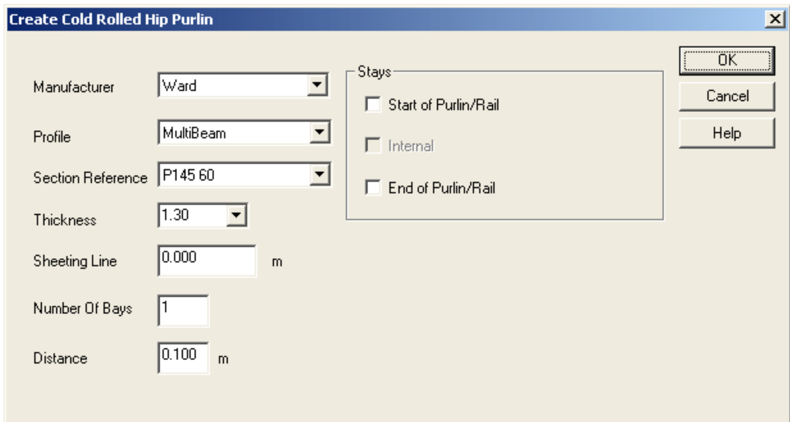
5. Once your details are correct click **OK** to return to the graphical display of the structure which will be updated to show the hip purlins which you have defined.

To create hip purlins singly

You must use the ***Structure*** window to define the hip purlins in your structure.

1. Click the **Create**, **Single** and **Hip Purlin** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Hip Purlin* menu options).

2. Click the hip raker or jack rafter next to which you want to create your new hip purlin (this must be the one nearest to column line A, and the hip purlin will run from this hip raker or jack rafter away from column line A). You will see the ***Hip Purlin Definition*** dialog.



3. Pick the hip purlin **Manufacturer**, **Profile**, **Section Reference** and **Thickness** from the appropriate lists.

4. Enter the **Sheeting Line** distance for the hip purlin - this is the distance from the centre-line of the hip raker or jack rafter which you picked to the inner face of the sheeting.

5. Specify the **Number of Bays** which your hip purlin is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

E

Example If there are 3 jack rafters to the right of the hip raker or jack rafter on which you are creating hip purlins, the maximum number of bays would be 3.

6. If you want to generate stays from your hip purlin to the rafters, then ensure that the appropriate **Stays** boxes are ticked.

Note You can only define internal stays when your hip purlin spans two or more bays.

7. Enter the **Distance** to the hip purlin from the origin of your selected hip raker or jack rafter and then click **OK** to generate a hip purlin with these settings.

Deleting hip purlins

You can delete hip purlins by:

Method	*Details*
Grid Line	all hip purlins on the selected grid line are deleted
Area	all hip purlins which lie wholly within the selected area are deleted

Method	*Details*
Single	the hip purlin you pick is deleted

To delete hip purlins by grid line

You must use the ***Structure*** window to delete the hip purlins in your structure.

1. Click the **Delete**, **Grid Line** and **Hip Purlin** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Hip Purlin* menu options).

2. Click the grid line along which the hip purlin(s) you want to delete lie. All hip purlins along this line are deleted.

3. Continue to identify grid lines until you have deleted all the hip purlins that you require.

To delete hip purlins by area

You must use the ***Structure*** window to delete the hip purlins in your structure.

1. Click the **Delete**, **Area** and **Hip Purlin** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Hip Purlin* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All hip purlins which lie totally within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the hip purlins that you require.

To delete hip purlins singly

You must use the *Structure* window to delete the hip purlins in your structure.

1. Click the **Delete**, **Single** and **Hip Purlin** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Hip Purlin* menu options).
2. Select the hip purlin that you wish to delete.
3. Continue to identify hip purlins until you have deleted all the hip purlins that you require.

Changing hip purlin attributes

When you create hip purlins you give the appropriate details for those hip purlins. If you need to make changes you can do so without deleting and re-creating these hip purlins.

You can modify hip purlins details by:

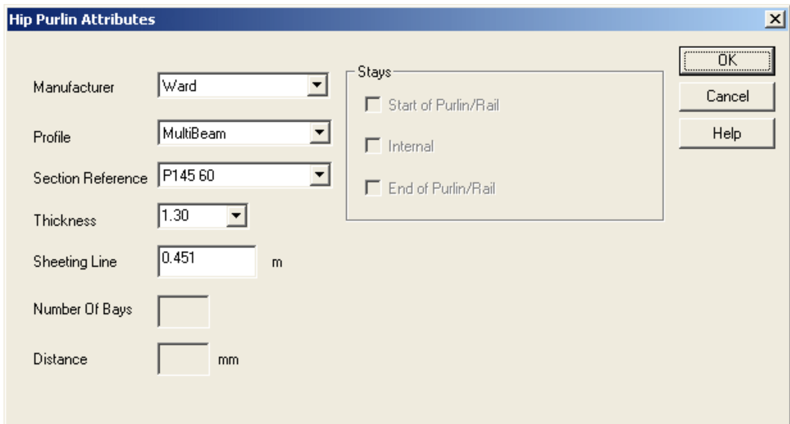
Method	*Details*
Grid Line	all hip purlins on the selected grid line are modified to take the details you specify
Area	all hip purlins which lie totally within the area you select are modified to take the details you specify
Single	the hip purlin you pick is modified to take the details you specify

To modify hip purlin attributes by grid line

You must use the ***Structure*** window to modify the properties of the hip purlins in your structure.

1. Click the **Attributes**, **Grid Line** and **Hip Purlin** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Hip Purlin* menu options).

2. Click the grid line along which the hip purlin(s) whose details you want to modify lie. You will see the ***Hip Purlin Attributes*** dialog which allows you to set the attributes that all hip purlins along this grid line are to take.



Note The initial details in this dialog are those for the bottom hip purlin in the bay which is both nearest to column line A and which contains hip purlins on the line you select.

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the initial hip raker or jack rafter[1] to the inner face of the sheeting.

   Since you are working with all the hip purlins on a particular line, you cannot define the number of spans each hip purlin covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

Help *see* *"To modify hip purlin attributes singly"* on page 740.

5. Once the details for your hip purlins are correct click **OK** to return to the graphical display of the structure. All the hip purlins on the grid line will be updated to the attributes which you have just defined.

To modify hip purlin attributes by area

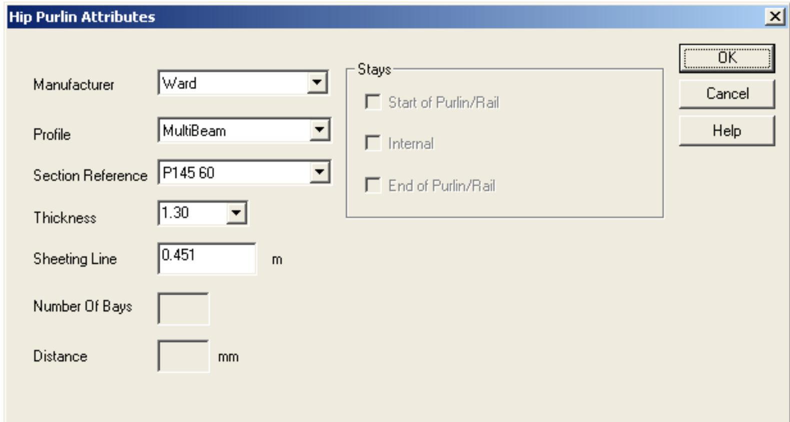
You must use the ***Structure*** window to modify the properties of the hip purlins in your structure.

1. Click the **Attributes**, **Area** and **Hip Purlin** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Hip Purlin* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All hip purlins

1. *That to which the initial hip purlin is related.*

which lie entirely within this area will be selected. You will see the ***Hip Purlin Attributes*** dialog which allows you to set the attributes that all the selected hip purlins are to take.



Note The initial details in this dialog are those for the bottom hip purlin in the bay which is both nearest to column line A and which contains hip purlins on the line you select.

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the initial hip raker or jack rafter[1] to the inner face of the sheeting.

1. *That to which the initial hip purlin is related.*

Since you are working with all the hip purlins in a particular area, you cannot define the number of spans each hip purlin covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

? **Help** *see "To modify hip purlin attributes singly" on page 740.*

5. Once the details for your hip purlins are correct click **OK** to return to the graphical display of the structure. All the selected hip purlins will be updated to the attributes which you have just defined.

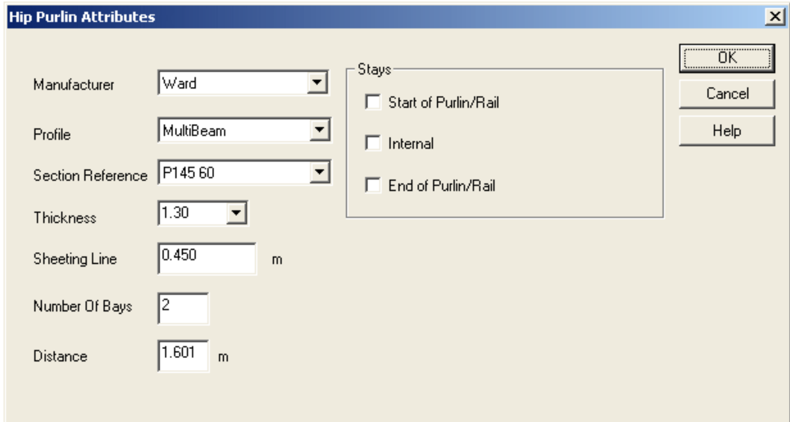
6. Continue to identify areas until you have modified the properties of all the hip purlins that you require.

To modify hip purlin attributes singly

You must use the ***Structure*** window to modify the properties of the hip purlins in your structure.

1. Click the **Attributes**, **Single** and **Hip Purlin** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Hip Purlin* menu options).

2. Select a hip purlin whose properties you wish to modify. You will see the ***Hip Purlin Attributes*** dialog which allows you to set the attributes that this hip purlin is to take.



3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the hip raker or jack rafter with which this single hip purlin is related to the inner face of the sheeting. This is used to place the sheeting rails exactly when they are transferred to ***3D+***.

5. Since you are working with a single hip purlin, you can define the number of spans each hip purlin covers here. Simply enter the requisite **Number of Bays**.

Caution If you increase the number of bays a particular hip purlin covers, and this causes it to overlap other hip purlins, then you will need to delete any unwanted hip purlins yourself since *Portal Frame* will not do this automatically.

6. Enter the **Distance** to the hip purlin purlin from the origin of the hip raker or jack rafter with which it is associated.

7. If you need to provide stays to the inner flange of the hip raker or jack rafter, then choose the positions where restraints are to be provided. You can choose any or all of **Start of Purlin/Rail**, **Internal** and **End of Purlin/Rail**. These settings will be applied to all the selected hip purlins.

8. Once the details for your hip purlin are correct click **OK** to return to the graphical display of the structure. The hip purlin will be updated to the attributes which you have just defined.

9. Continue to identify hip purlins until you have modified the properties of all the hip purlins that you require.

13 Working with Gable Rails

You create, delete or change the properties of gable rails graphically. As with all ancillary steelwork you must use the *Structure* window to work with gable rails.

Creating gable rails

You can create gable rails by:

Method	*Details*
Grid Line	the gable rails are created on the grid line you choose. The position of the gable rails is determined by the layout of restraints on the left-hand column of each span. If there are no restraints for an intermediate column, then the gable rails will be placed at the same locations as those in the previous span.
Single	this option allows you to add gable rails into a particular span. You can specify the details of the rail, its exact position and the number of spans it covers.

To create gable rails by grid line

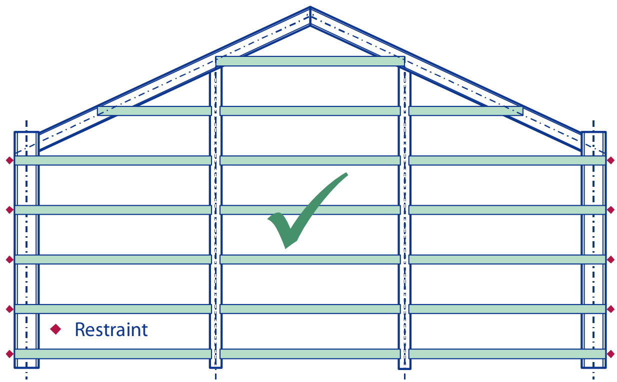
You must use the *Structure* window to define the gable rails in your structure.

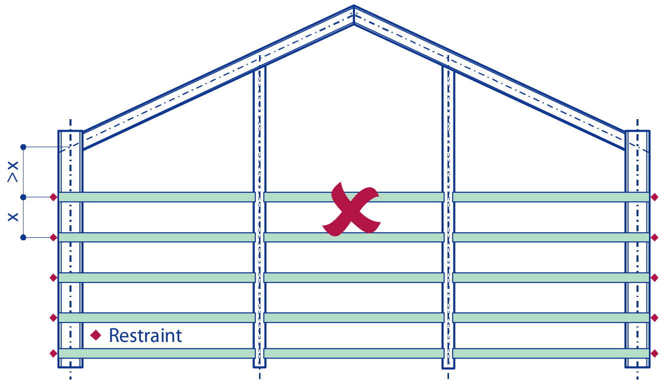
1. Click the **Create**, **Grid Line** and **Gable Rail** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Gable Rail* menu options).

2. Select the grid line from the display on which you want to create the gable rails. This must satisfy the following conditions:
   - it must be a grid line (that is not a column line),
   - you must have already created the gable posts along the grid line,
   - you must have defined restraints for at least one column along the grid line since the position of the gable rails depends on the spacing of the column restraints as follows:

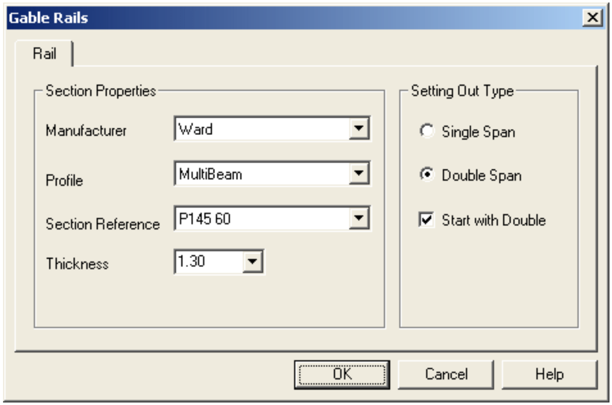
Rules for positioning gable rails

     - **Portal Frame** starts with the left-hand-most column of the current span (that nearest column line A). If this column has restraints then a gable rail is placed at the position of each such restraint,
     - if there are no restraints on this left-hand-most column then **Portal Frame** moves to the next column and looks for restraints there. If this column has no restraints, then **Portal Frame** moves to the left-hand column of the next span and so on until all left-hand stanchions are exhausted. In this case there are no restraints on the line, and so no gable rails are created,
     - if **Portal Frame** has started to create gable rails, and the left-hand column of the current span has no restraints, then the layout of gable rails from the previous span is continued in this one,
     - if the left-hand column of the current span does have restraints, then **Portal Frame** uses these positions for any new rails it creates from this point.
   - furthermore, if you want the gable rails to continue up to the apex of the frame the distance between the top two restraints to the column must be greater than the distance from the top restraint to the intersection of the rafter and column centre-lines as illustrated in the figures below.


Restraint


>X
X
Restraint

You will see the ***Gable Rail Definition*** dialog.

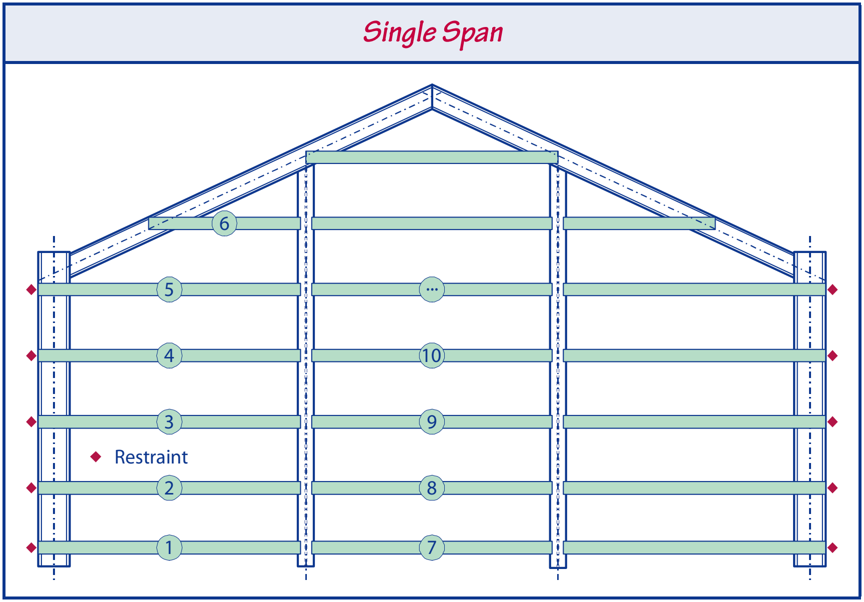


Caution If you have modified a span's gable rails, by amending their details, position or both, and you **OK** this dialog, you will:

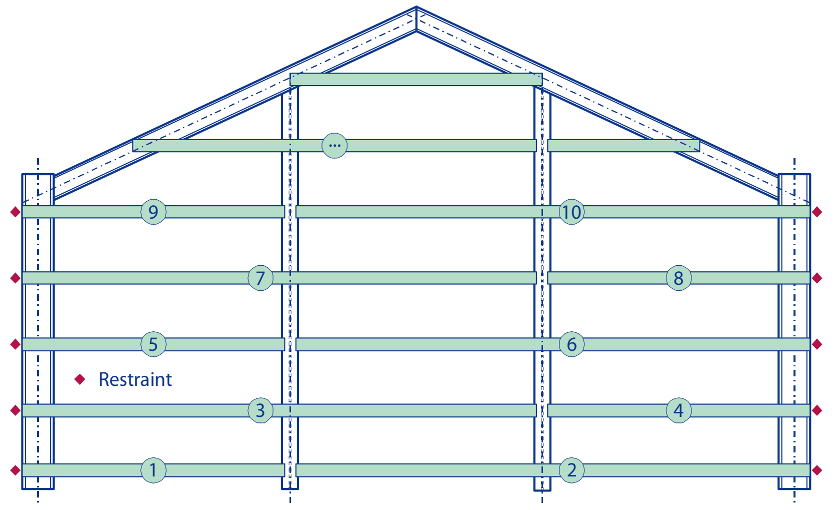
- position the gable rails to their initial settings (based on the restraint layout of the column,
- change the **Manufacturer**, **Profile**, **Section Reference**, and **Setting Out Type** details of all gable rails along this grid line to those shown in this dialog.

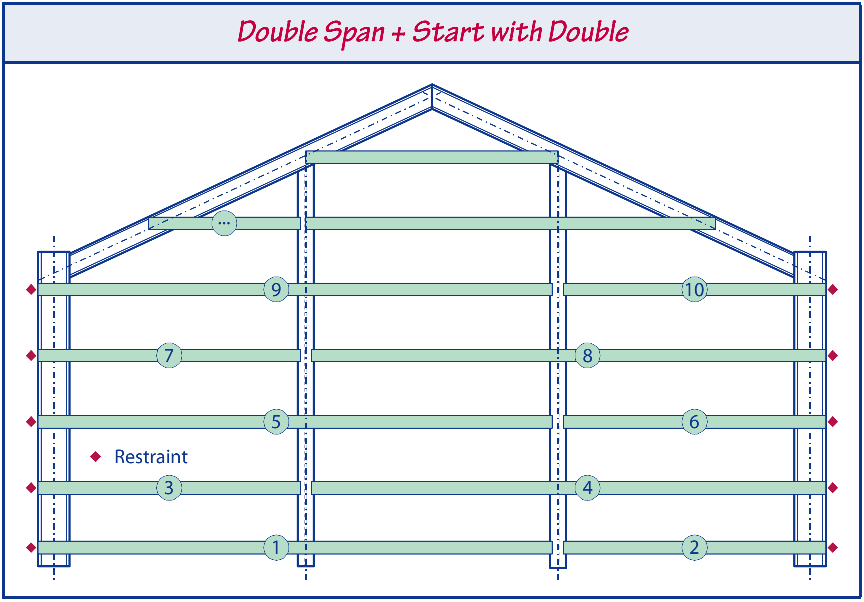
3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Pick the **Setting Out Type** details that you want to use. The various options are covered in the following diagrams.



Double Span





5. Once your details are correct click **OK** to return to the graphical display of the structure which will be updated to show the gable rails which you have defined.

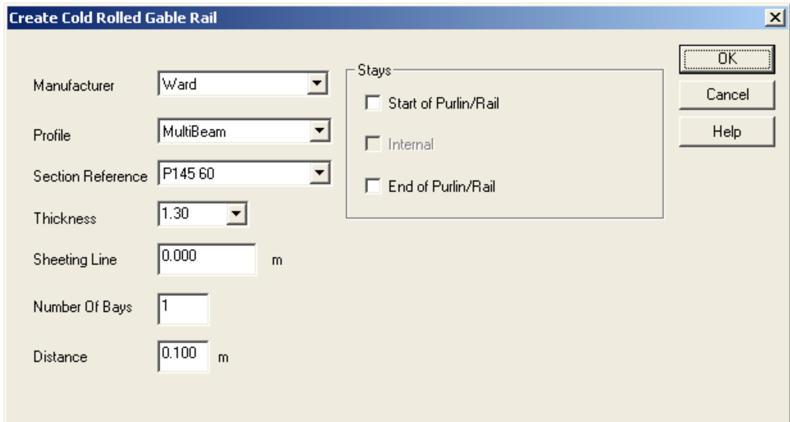
To create gable rails singly

You must use the *Structure* window to define the gable rails in your structure.

Although this option is a create one, it is effectively limited to adding a new gable rail into a bay which already contains at least one gable rail. If you want to add gable rails into a bay where there are none, then you need to use the option to define gable rails by grid line.

1. Click the **Create**, **Single** and **Gable Rail** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Gable Rail* menu options).

2. Click the column or gable post which lies at the left hand end (that nearest column line A) of the bay(s) to which you want to add your gable rail. ***Portal Frame*** highlights this column or rail to indicate that the gable rail will start here. You will see the ***Create Cold-Rolled Gable Rail*** dialog which allows you to define the details for this gable rail.

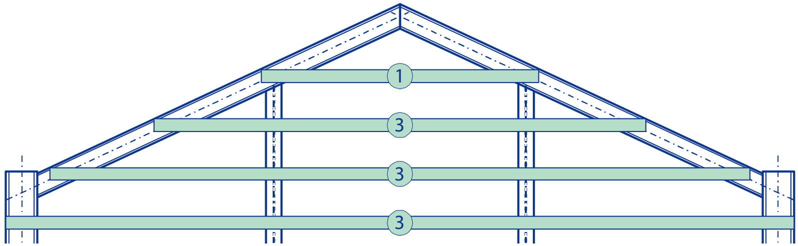


3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the **Sheeting Line** distance for the rail - this is the distance from the centre-line of the column or gable rail which you picked to the inner face of the sheeting.

Note *If you are working with the gable on line 1, or you want to create gable rails on the grid-line-1-ward face of any other frame, then you need to specify a negative* **Sheeting Line** *distance.*

5. Specify the **Number of Bays** which your gable rail is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available. The number of bays will decrease when you are creating gable rails within the gable peak as illustrated below.



In no case can you define a gable rail which is above the top of both the member you selected and that next farther across the frame.

6. If you want to generate stays from your gable rail to the stanchions or gable posts, then ensure that the appropriate **Stays** boxes are ticked.

Note *You can only define internal stays when your gable rail spans two or more bays.*

7. Enter the distance to the gable rail from the bottom of the member you selected initially and then click **OK** to generate the gable rail to these settings.

Deleting gable rails

You can delete gable rails by:

Method	*Details*
Grid Line	all gable rails on the selected grid line are deleted

Method	Details
Area	all gable rails which lie totally within the selected area are deleted
Single	the gable rail you pick is deleted

To delete gable rails by grid line

You must use the ***Structure*** window to delete the gable rails in your structure.

1. Click the **Delete**, **Grid Line** and **Gable Rail** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Gable Rail* menu options).
2. Click the grid line along which the gable rail(s) you want to delete lie. All gable rails along this line are deleted.
3. Continue to identify grid lines until you have deleted all the gable rails that you require.

To delete gable rails by area

You must use the ***Structure*** window to delete the gable rails in your structure.

1. Click the **Delete**, **Area** and **Gable Rail** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Gable Rail* menu options).
2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable rails which lie totally within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the gable rails that you require.

To delete gable rails singly

You must use the *Structure* window to delete the gable rails in your structure.

1. Click the **Delete**, **Single** and **Gable Rail** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Gable Rail* menu options).

2. Select the gable rail that you wish to delete.

3. Continue to identify gable rails until you have deleted all the gable rails that you require.

Changing gable rail attributes

When you create gable rails you give the appropriate details for those gable rails. If you need to make changes you can do so without deleting and re-creating these gable rails.

You can modify gable rails details by:

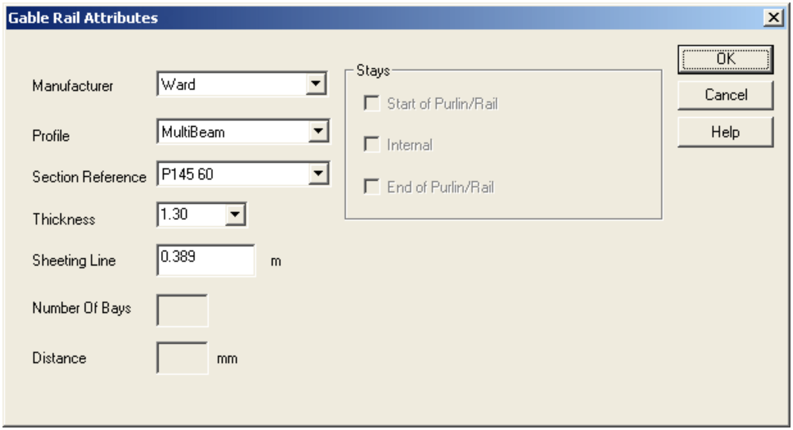
Method	*Details*
Grid Line	all gable rails on the selected grid line are modified to take the details you specify
Area	all gable rails which lie totally within the area you select are modified to take the details you specify
Single	the gable rail you pick is modified to take the details you specify

To modify gable rail attributes by grid line

You must use the ***Structure*** window to modify the properties of the gable rails in your structure.

1. Click the **Attributes**, **Grid Line** and **Gable Rail** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Gable Rail* menu options).

2. Click the grid line along which the gable rail(s) whose details you want to modify lie. You will see the ***Gable Rail Attributes*** dialog which allows you to set the attributes that all gable rails along this grid line are to take.



Note The initial details in this dialog are those for the bottom gable rail in the bay which is both nearest to column line A and which contains gable rails on the line you select.

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the initial column or gable post[1] to the inner face of the sheeting.

Note *If you are working with the gable on line 1, or you want to create gable rails on the grid-line-1-ward face of any other frame, then you need to specify a negative Sheeting Line distance.*

Since you are working with all the gable rails on a particular line, you cannot define the number of spans each gable rail covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

Help *see "To modify gable rail attributes singly" on page 757.*

5. Once the details for your gable rails are correct click **OK** to return to the graphical display of the structure. All the gable rails on the grid line will be updated to the attributes which you have just defined.

To modify gable rail attributes by area

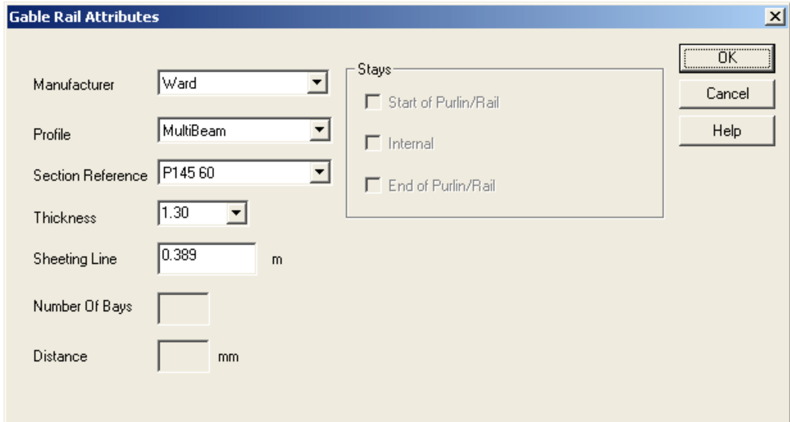
You must use the ***Structure*** window to modify the properties of the gable rails in your structure.

1. Click the **Attributes**, **Area** and **Gable Rail** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Gable Rail* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All gable rails which

1. *That to which the initial gable rail is related.*

lie entirely within this area will be selected. You will see the ***Gable Rail Attributes*** dialog which allows you to set the attributes that all the selected gable rails are to take.



Note *The initial details in this dialog are those for the bottom gable rail in the bay which is both nearest to column line A and which contains gable rails in the area you select.*

3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the initial column or gable post[1] to the inner face of the sheeting.

1. *That to which the initial gable rail is related.*

Note If you are working with the gable on line 1, or you want to create gable rails on the grid-line-1-ward face of any other frame, then you need to specify a negative Sheeting Line distance.

Since you are working with all the gable rails in a particular area, you cannot define the number of spans each gable rail covers here or details of any stays. If you want define this information then you will need to use the **Single** option.

Help *see* *"To modify gable rail attributes singly"* on page 757.

5. Once the details for your gable rails are correct click **OK** to return to the graphical display of the structure. All the selected gable rails will be updated to the attributes which you have just defined.

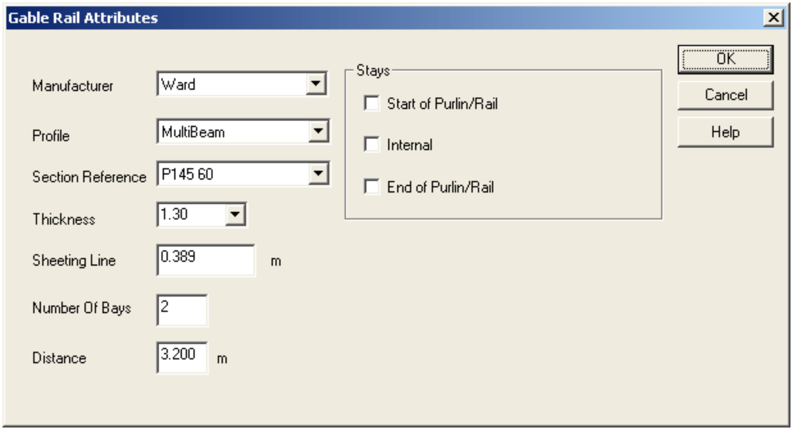
6. Continue to identify areas until you have modified the properties of all the gable rails that you require.

To modify gable rail attributes singly

You must use the ***Structure*** window to modify the properties of the gable rails in your structure.

1. Click the **Attributes**, **Single** and **Gable Rail** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Gable Rail* menu options).

2. Select a gable rail whose properties you wish to modify. You will see the ***Gable Rail Attributes*** dialog which allows you to set the attributes that this gable rail is to take.



3. Pick the **Manufacturer**, **Profile**, **Section Reference** and **Thickness** details from the appropriate lists.

4. Enter the details of the distance to the **Sheeting Line**, this is the distance from the centre-line of the column or gable post with which this single gable rail is related to the inner face of the sheeting. This is used to place the sheeting rails exactly when they are transferred to ***3D+***.

Note If you are working with the gable on line 1, or you want to create gable rails on the grid-line-1-ward face of any other frame, then you need to specify a negative Sheeting Line distance.

5. Since you are working with a single gable rail, you can define the number of spans each gable rail covers here. Simply enter the requisite **Number of Bays**.

> **Caution** If you increase the number of bays here, then length of the current gable rail will be increased. If this causes the gable rail to overlay other rails, these will not be deleted automatically you will need to identify and delete these as necessary.

6. Enter the **Distance** for the new gable rail. These dimensions are measured from datum zero and not from the column base.

7. If you need to provide stays to the inner flange of the column or gable post which this rail crossed, then choose the positions where restraints are to be provided. You can choose any or all of **Start of Purlin/Rail**, **Internal** (for a rail which crosses more than one bay) and **End of Purlin/Rail**. These settings will be applied to the current gable rail.

8. Once the details for your gable rail are correct click **OK** to return to the graphical display of the structure. The gable rail will be updated to the attributes which you have just defined.

9. Continue to identify gable rails until you have modified the properties of all the gable rails that you require.

14 Working with Eaves Beams

You create, delete or change the properties of eaves beams graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with eaves beams.

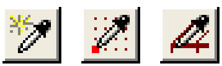
Creating eaves beams

You can create eaves beams by:

Method	*Details*
Grid Point	the eaves beams are created between the two grid points that you choose. These must be on the same column line
Grid Line	the eaves beams are created on the grid line you choose.
Single	this option allows you to add new eaves beams into a particular bay.

To create eaves beams by grid points

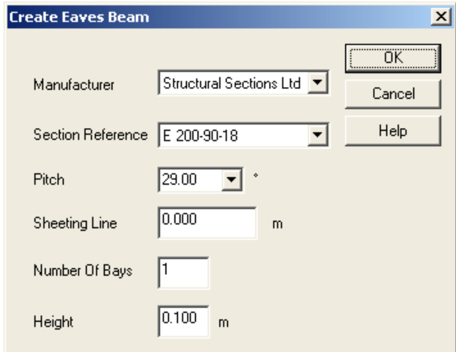
You must use the ***Structure*** window to define the eaves beams in your structure.



1. Click the **Create**, **Grid Points** and **Eaves Beam** buttons (or choose the *Select / Create*, *Select / Grid Points* and *Select / Eaves Beam* menu options).

2. Click the grid point at the base of the column where you want your run of beams to start.

3. Now move the cursor to the grid point at the base of the column where you want your run of beams to end. You will see a rubber line which follows the cursor. This indicates the extent of your run of beams. When you are over the correct grid point click again and you will see the ***Eaves Beam Definition*** dialog which allows you to define the eaves beam's details.



Note If you pick grid points which encompass existing eaves beams, then the dialog will show the details for the first such eaves beam.

Caution If you have modified such eaves beams, by amending their details, position or both, and you **OK** this dialog, you will reset the details for these encompassed eaves beams to those you set using this option.

4. Pick the beam **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

5. Enter the **Sheeting Line** distance for the beam - this is the distance from the centre-line of the first column between which you are creating eaves beams to the inner face of the sheeting.

Note *Since you are defining the details for a number of eaves beams, you cannot set the* **Number of Bays** *and* **Height** *details here. You must instead use the Single definition method*

Help *see "To create eaves beams singly" on page 764.*

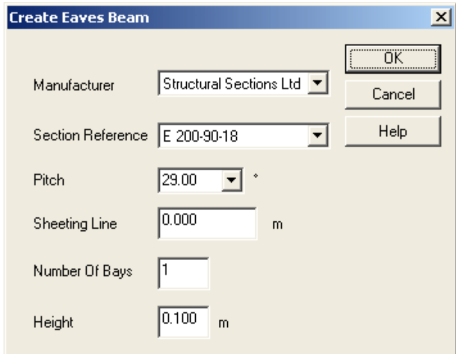
6. Once your settings are correct click **OK** to create the eaves beams using the current information.

To create eaves beams by grid line

You must use the ***Structure*** window to define the eaves beams in your structure.

1. Click the **Create**, **Grid Line** and **Eaves Beam** buttons (or choose the *Select / Create*, *Select / Grid Line* and *Select / Eaves Beam* menu options).

Select the grid line from the display on which you want to create the eaves beams. You will see the ***Eaves Beam Definition*** dialog which allows you to define the eaves beam's details.



Note If you pick grid points which encompass existing eaves beams, then the dialog will show the details for the first such eaves beam.

Caution If you have modified such eaves beams, by amending their details, position or both, and you **OK** this dialog, you will reset the details for these encompassed eaves beams to those you set using this option.

2. Pick the beam **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

3. Enter the **Sheeting Line** distance for the beam - this is the distance from the centre-line of the first column between which you are creating eaves beams to the inner face of the sheeting.

Note Since you are defining the details for a number of eaves beams, you cannot set the **Number of Bays** and **Height** details here. You must instead use the Single definition method

Help see *"To create eaves beams singly"* on page 764.

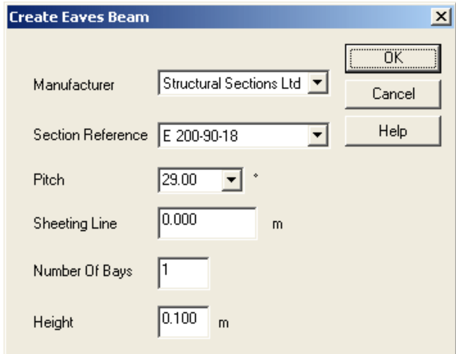
4. Once your settings are correct click **OK** to create the eaves beams using the current information. Once the details for all your eaves beams are correct click **OK** to return to the graphical display of the structure which will be updated to show the eaves beams which you have defined.

To create eaves beams singly

You must use the ***Structure*** window to define the eaves beams in your structure.

1. Click the **Create**, **Single** and **Eaves Beam** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Eaves Beam* menu options).

2. Click the grid line and column line intersection point which lies nearest to grid line 1 to indicate the bay in which you want to create your new eaves beam.



3. Pick the beam **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

4. Enter the **Sheeting Line** distance for the beam - this is the distance from the centre-line of the first column between which you are creating eaves beams to the inner face of the sheeting.

5. Specify the **Number of Bays** which your beam is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

E

Example *If there are 3 frames to the right of the grid point on which you are creating beams, the maximum number of bays would be 3.*

6. Enter the **Height** to the new eaves beam from the base of the column on the grid point you picked in step 2 above.

7. Once your settings are correct click **OK** to create the eaves beam using the current information. Click **OK** to return to the graphical display of the structure which will be updated to show the eaves beam which you have defined.

Deleting eaves beams

You can delete eaves beams by:

Method	*Details*
Grid Line	all eaves beams on the selected grid line are deleted
Area	all eaves beams which lie wholly within the selected area are deleted
Single	the eaves beam you pick is deleted

To delete eaves beams by grid line

You must use the ***Structure*** window to delete the eaves beams in your structure.

1. Click the **Delete**, **Grid Line** and **Eaves Beam** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Eaves Beam* menu options).

2. Click the grid line along which the eaves beam(s) you want to delete lie. All eaves beams along this line are deleted.

3. Continue to identify grid lines until you have deleted all the eaves beams that you require.

To delete eaves beams by area

You must use the ***Structure*** window to delete the eaves beams in your structure.

1. Click the **Delete**, **Area** and **Eaves Beam** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Eaves Beam* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All eaves beams which lie entirely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the eaves beams that you require.

To delete eaves beams singly

You must use the ***Structure*** window to delete the eaves beams in your structure.

1. Click the **Delete**, **Single** and **Eaves Beam** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Eaves Beam* menu options).

2. Select the eaves beam that you wish to delete.

3. Continue to identify eaves beams until you have deleted all the eaves beams that you require.

Changing eaves beam attributes

When you create eaves beams you give the appropriate details for those eaves beams. If you need to make changes you can do so without deleting and re-creating these eaves beams.

You can modify eaves beams details by:

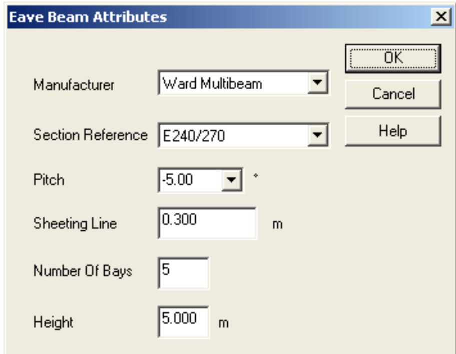
Method	*Details*
Grid Line	all eaves beams on the selected grid line are modified to take the details you specify
Area	all eaves beams which lie wholly within the area you select are modified to take the details you specify
Single	the eaves beam you pick is modified to take the details you specify

To modify eaves beam attributes by grid line

You must use the ***Structure*** window to modify the properties of the eaves beams in your structure.

1. Click the **Attributes**, **Grid Line** and **Eaves Beam** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Eaves Beam* menu options).

2. Click the column line along which the eaves beam(s) whose details you want to modify lie. You will see the ***Eaves Beam Attributes*** dialog which allows you to set the attributes that all eaves beams along this column line are to take.



Note The initial details in this dialog are those for the eaves beam nearest to grid line 1 on the column line you select.

Caution If you have modified eaves beams, by amending their details, position or both, and you **OK** this dialog, you will reset the details for all the eaves beams on this line to those shown in this dialog.

3. Pick the beam **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

4. Enter the **Sheeting Line** distance for the beam - this is the distance from the centre-line of the first column between which you are creating eaves beams to the inner face of the sheeting.

Note Since you are amending the details for a number of eaves beams, you cannot set the **Number of Bays** and **Height** details here. You must instead use the **Single** definition method

Help *see* "To modify eaves beam attributes singly" on page 772.

5. Once your settings are correct click **OK** to modify the attributes of the eaves beams to the values shown in the dialog.

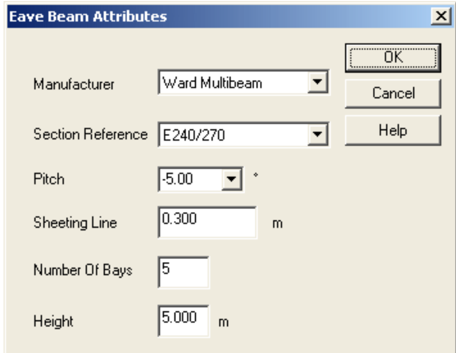
To modify eaves beam attributes by area

You must use the ***Structure*** window to modify the properties of the eaves beams in your structure.

1. Click the **Attributes**, **Area** and **Eaves Beam** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Eaves Beam* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All eaves beams which lie totally within this area will be selected.

3. You will see the ***Eaves Beam Attributes*** dialog which allows you to set the attributes that all eaves beams along this column line are to take.



Note The initial details in this dialog are those for the eaves beam nearest to grid line 1 on the column line you select.

Caution If you have modified eaves beams, by amending their details, position or both, and you **OK** this dialog, you will reset the details for all the selected eaves beams to those shown in this dialog.

4. Pick the beam **Manufacturer**, **Section Reference** and **Pitch** from the appropriate lists.

5. Enter the **Sheeting Line** distance for the beam - this is the distance from the centre-line of the first column between which you are creating eaves beams to the inner face of the sheeting.

Note Since you are amending the details for a number of eaves beams, you cannot set the **Number of Bays** and **Height** details here. You must instead use the **Single** definition method

Help *see* "To modify eaves beam attributes singly" on page 772.

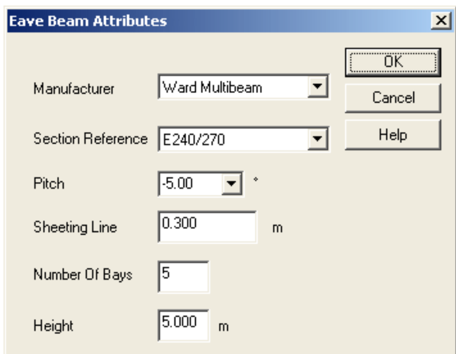
6. Once your settings are correct click **OK** to modify the attributes of the eaves beams to the values shown in the dialog.

To modify eaves beam attributes singly

You must use the ***Structure*** window to modify the properties of the eaves beams in your structure.

1. Click the **Attributes**, **Single** and **Eaves Beam** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Eaves Beam* menu options).

2. Select a eaves beam whose properties you wish to modify. You will see the ***Eaves Beam Attributes*** dialog which allows you to set the attributes that this eaves beams is to take.



3. Specify the **Number of Bays** which your beam is to cross. ***Portal Frame*** will not allow you to specify a number of bays greater than that which is available.

E

Example If there are 3 frames to the right of the grid point on which you are creating beams, the maximum number of bays would be 3.

4. Enter the **Height** to the new eaves beam from the base of the column on the grid point you picked in step 2 above.

5. Once your settings are correct click **OK** to modify the attributes of the eaves beam to the values shown in the dialog.

15 Working with Jack Rafters

You can delete, change the properties of or move jack rafters graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with jack rafters.

Creating jack rafters

You can create jack rafters by:

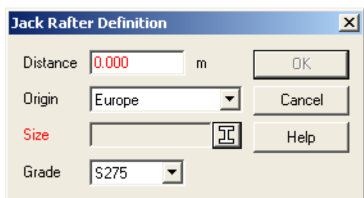
Method	*Details*
Single	this option allows you to add a new jack rafter into a particular span.

To create jack rafters singly

You must use the ***Structure*** window to define the jack rafters in your structure.

1. Click the **Create**, **Single** and **Jack Rafter** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Jack Rafter* menu options).

2. Click the monopitch rafter for the span which is to contain your new jack rafter. You will see the ***Jack Rafter Definition*** dialog.



3. Enter the **Distance** to the new jack rafter from the intersection point you picked in step 2 above.

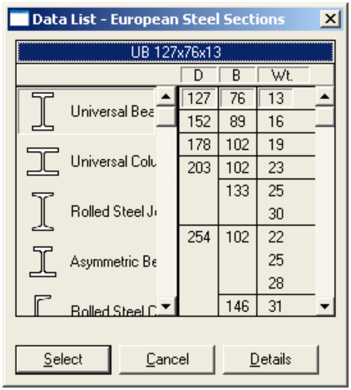
4. Enter the **Level** at which the base of the new jack rafter is to lie.

5. By default the jack rafters are oriented with their webs perpendicular to those of the main frame members. If you want to orient your jack rafters so that their webs are parallel to those of the main frame members, then tick the **Rotation** box.

6. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

7. To enter the size of your jack rafters click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Jack Rafter Definition*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

9. Once the details for your jack rafter is correct click **OK** to return to the graphical display of the structure which will be updated to show the new jack rafter.

Deleting jack rafters

You can delete jack rafters by:

Method	*Details*
Grid Line	all jack rafters associated with the selected grid line are deleted
Area	all jack rafters which lie wholly within the selected area are deleted
Single	the jack rafter you pick is deleted

To delete jack rafters by grid line

You must use the ***Structure*** window to delete the jack rafters in your structure.

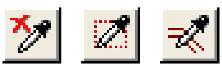
1. Click the **Delete**, **Grid Line** and **Jack Rafter** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Jack Rafter* menu options).

2. Click the grid line associated with the hipped area[1] whose jack rafter(s) you want to delete lie. All jack rafters associated with this grid line are deleted.

3. Continue to identify grid lines until you have deleted all the jack rafters that you require.

To delete jack rafters by area

You must use the ***Structure*** window to delete the jack rafters in your structure.



1. Click the **Delete**, **Area** and **Jack Rafter** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Jack Rafter* menu options).

1. *This is the grid line at the monopitch end of the hipped area.*

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All jack rafters which lie entirely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the jack rafters that you require.

To delete jack rafters singly

You must use the ***Structure*** window to delete the jack rafters in your structure.

1. Click the **Delete**, **Single** and **Jack Rafter** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Jack Rafter* menu options).

2. Select the jack rafter that you wish to delete.

3. Continue to identify jack rafters until you have deleted all the jack rafters that you require.

Changing jack rafter attributes

When you create jack rafters you give the appropriate details for then. If you need to make changes you can do so without deleting and re-creating these jack rafters.

You can modify jack rafters details by:

Method	*Details*
Grid Line	all jack rafters on the selected grid line are modified to take the details you specify
Area	all jack rafters which lie wholly within the area you select are modified to take the details you specify

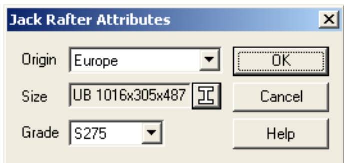
Method	*Details*
Single	the jack rafter you pick is modified to take the details you specify

To modify jack rafter attributes by grid line

You must use the ***Structure*** window to modify the properties of the jack rafters in your structure.

1. Click the **Attributes**, **Grid Line** and **Jack Rafter** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Jack Rafter* menu options).

2. Click the grid line along associated with the hipped area whose jack rafter details you want to modify. You will see the ***Jack Rafter Attributes*** dialog which allows you to set the attributes that all jack rafters along this grid line are to take.

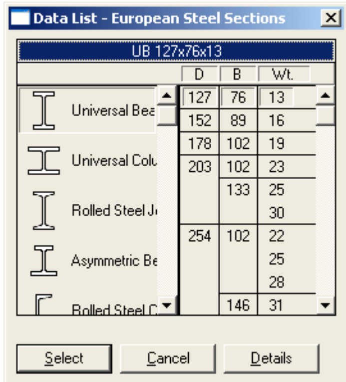


Note *The initial details in this dialog are those for the jack rafter nearest to column line A on the line you select.*

3. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial *Origin* setting will be that of the country which is set in your *Preferences*.

4. To enter the size of your jack rafters click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



   Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Jack Rafter Attributes*** dialog which will show your selected size.

5. Pick the **Grade** of material from the list of available grades.

6. Once the details for your jack rafters are correct click **OK** to return to the graphical display of the structure. All the jack rafters associated with the grid line will be updated to the attributes which you have just defined.

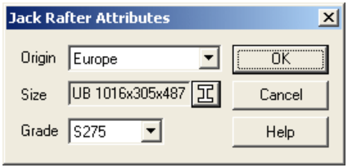
To modify jack rafter attributes by area

You must use the ***Structure*** window to modify the properties of the jack rafters in your structure.

1. Click the **Attributes**, **Area** and **Jack Rafter** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Jack Rafter* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All jack rafters which lie totally within this area will be selected.

3. You will see the ***Jack Rafter Attributes*** dialog which allows you to set the attributes that all the selected jack rafters are to take.

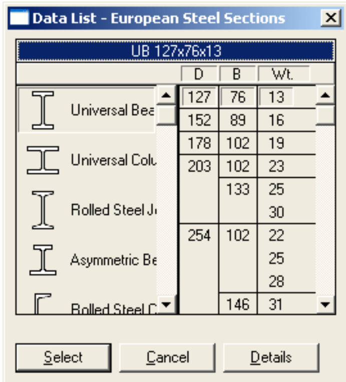


Note *The initial details in this dialog are those for the jack rafter in your selection which is nearest to grid line 1 and column line A.*

4. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your **Preferences**.*

5. To enter the size of your jack rafters click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Jack Rafter Attributes*** dialog which will show your selected size.

6. Pick the **Grade** of material from the list of available grades.

7. Once the details for your jack rafters are correct click **OK** to return to the graphical display of the structure. All the selected jack rafters will be updated to the attributes which you have just defined.

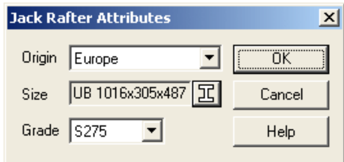
8. Continue to identify areas until you have modified the properties of all the jack rafters that you require.

To modify jack rafter attributes singly

You must use the ***Structure*** window to modify the properties of the jack rafters in your structure.

1. Click the **Attributes**, **Single** and **Jack Rafter** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Jack Rafter* menu options).

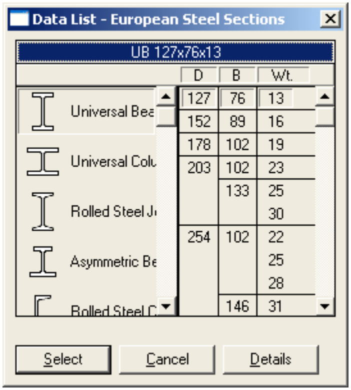
2. Select a jack rafter whose properties you wish to modify. You will see the ***Jack Rafter Attributes*** dialog which allows you to set the attributes that this jack rafters is to take.



3. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your **Preferences**.*

4. To enter the size of your jack rafters click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Jack Rafter Attributes*** dialog which will show your selected size.

5. Pick the **Grade** of material from the list of available grades.

6. Once the details for your jack rafter are correct click **OK** to return to the graphical display of the structure. The jack rafter will be updated to the attributes which you have just defined.

7. Continue to identify jack rafters until you have modified the properties of all the jack rafters that you require.

Changing jack rafter positions

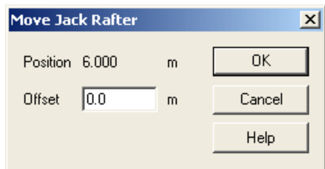
When you create jack rafters on a grid line, the jack rafters are equally spaced across the span (thus if you define 3 jack rafters ***Portal Frame*** will position these at the quarter, half and three-quarter points across your span). If this positioning is that which you require, then you need take no further action. If, however, you need a different spacing, then you can move a particular jack rafter to the required position.

To move jack rafters singly

You must use the ***Structure*** window to modify the positions of the jack rafters in your structure.

1. Click the **Move**, **Single** and **Jack Rafter** buttons (or choose the *Select / Move*, *Select / Single* and *Select / Jack Rafter* menu options).

2. Select the jack rafter which you wish to move. You will see the ***Move Jack Rafter*** dialog which allows you to specify the distance by which the jack rafter is to move.



3. Enter the **Offset** distance by which the jack rafter is to move from its current location and then click **OK**.

Note The **Offset** you specify can be positive (to place the new post farther from column line A) or negative (to place the new post closer to column line A). If there are other jack rafters in the span the **Offset** you specify cannot cause the new post to lie beyond any existing post. If you specify an **Offset** which breaks this requirement then the **Offset** you have defined will show in error and **OK** will be dimmed.

Note The **Position** information is updated to show the location at which the jack rafter will be placed when you click **OK**.

4. Continue to identify jack rafters and move them until you have the layout that you require.

16 Working with Hip Rakers

You can delete or change the properties of hip rakers graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with hip rakers.

Note *You can not create hip rakers in this way, instead you must use the Hip Wizard.*

Deleting hip rakers

You can delete hip rakers by:

Method	*Details*
Grid Line	all hip rakers associated with the selected grid line are deleted
Area	all hip rakers which lie wholly within the selected area are deleted
Single	the hip raker you pick is deleted

To delete hip rakers by grid line

You must use the ***Structure*** window to delete the hip rakers in your structure.

1. Click the **Delete**, **Grid Line** and **Hip Raker** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Hip Raker* menu options).

2. Click the grid line associated with the hipped area[1] whose hip rakers you want to delete. All hip rakers associated with this grid line are deleted.

1. *This is the grid line at the monopitch end of the hipped area.*

Caution If you delete hip rakers, then all details associated with those hip rakers will also be removed. This includes all jack rafters, all hip purlins and all decomposed loads. ***Please use this feature with care***.

3. Continue to identify grid lines until you have deleted all the hip rakers that you require.

To delete hip rakers by area

You must use the ***Structure*** window to delete the hip rakers in your structure.

1. Click the **Delete**, **Area** and **Hip Raker** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Hip Raker* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All hip rakers which lie entirely within the area which you select are deleted.

Caution If you delete hip rakers, then all details associated with those hip rakers will also be removed. This includes all jack rafters, all hip purlins and all decomposed loads. ***Please use this feature with care***.

3. Continue to identify areas until you have deleted all the hip rakers that you require.

To delete hip rakers singly

You must use the ***Structure*** window to delete the hip rakers in your structure.

1. Click the **Delete**, **Single** and **Hip Raker** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Hip Raker* menu options).

2. Select the hip raker that you wish to delete.

Caution If you delete a hip raker, then all details associated with th1t hip raker will also be removed. This includes all jack rafters, all hip purlins and all decomposed loads. ***Please use this feature with care***.

3. Continue to identify hip rakers until you have deleted all the hip rakers that you require.

Changing hip raker attributes

When you create hip rakers you give the appropriate details for those hip rakers. If you need to make changes you can do so without deleting and re-creating these hip rakers.

You can modify hip rakers details by:

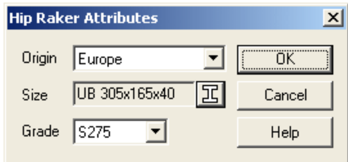
Method	*Details*
Grid Line	all hip rakers on the selected grid line are modified to take the details you specify
Area	all hip rakers which lie wholly within the area you select are modified to take the details you specify
Single	the hip raker you pick is modified to take the details you specify

To modify hip raker attributes by grid line

You must use the ***Structure*** window to modify the properties of the hip rakers in your structure.

1. Click the **Attributes**, **Grid Line** and **Hip Raker** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Hip Raker* menu options).

2. Click the grid line along which the hip raker(s) whose details you want to modify lie. You will see the ***Hip Raker Attributes*** dialog which allows you to set the attributes that all hip rakers along this grid line are to take.

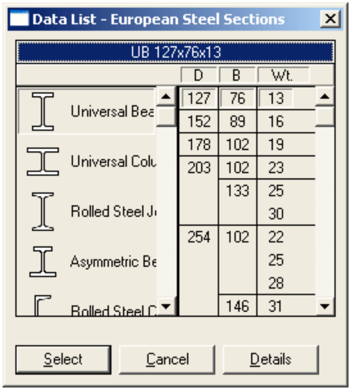


Note *The initial details in this dialog are those for the hip raker nearest to column line A on the grid line you select.*

3. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial* **Origin** *setting will be that of the country which is set in your* ***Preferences***.

4. To enter the size of your hip rakers click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Raker Attributes*** dialog which will show your selected size.

5. Pick the **Grade** of material from the list of available grades.

6. Once the details for your hip rakers are correct click **OK** to return to the graphical display of the structure. All the hip rakers on the grid line will be updated to the attributes which you have just defined.

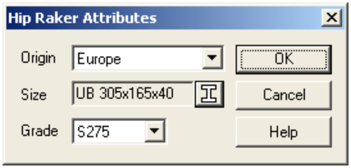
To modify hip raker attributes by area

You must use the ***Structure*** window to modify the properties of the hip rakers in your structure.

1. Click the **Attributes**, **Area** and **Hip Raker** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Hip Raker* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All hip rakers which lie totally within this area will be selected.

3. You will see the ***Hip Raker Attributes*** dialog which allows you to set the attributes that all the selected hip rakers are to take.

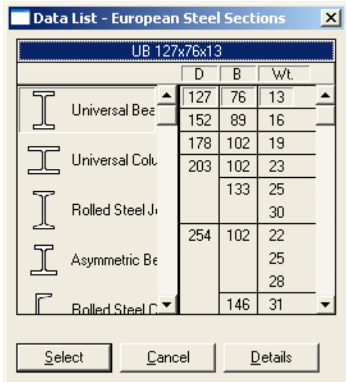


Note The initial details in this dialog are those for the hip raker in your selection which is nearest to column line A and grid line 1.

4. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

5. To enter the size of your hip rakers click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Raker Attributes*** dialog which will show your selected size.

6. Pick the **Grade** of material from the list of available grades.

7. Once the details for your hip rakers are correct click **OK** to return to the graphical display of the structure. All the selected hip rakers will be updated to the attributes which you have just defined.

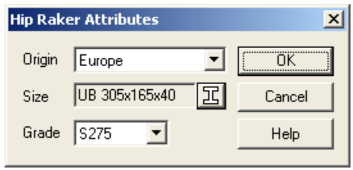
8. Continue to identify areas until you have modified the properties of all the hip rakers that you require.

To modify hip raker attributes singly

You must use the ***Structure*** window to modify the properties of the hip rakers in your structure.

1. Click the **Attributes**, **Single** and **Hip Raker** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Hip Raker* menu options).

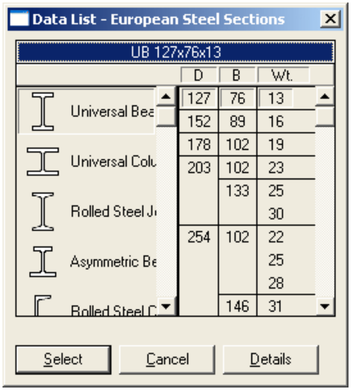
2. Select a hip raker whose properties you wish to modify. You will see the ***Hip Raker Attributes*** dialog which allows you to set the attributes that this hip rakers is to take.



3. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your Preferences.*

4. To enter the size of your hip rakers click the **Section** icon to the right of the **Size** box. You will see the ***Steel Section Data List*** for the region you selected above.



Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Hip Raker Attributes*** dialog which will show your selected size.

5. Pick the **Grade** of material from the list of available grades.

6. Once the details for your hip raker are correct click **OK** to return to the graphical display of the structure. The hip raker will be updated to the attributes which you have just defined.

7. Continue to identify hip rakers until you have modified the properties of all the hip rakers that you require.

17 Working with Floor Joists

You create, delete or change the properties of floor joists graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with floor joists.

Creating floor joists

You can create floor joists by:

Method	*Details*
Single	this option allows you to add new floor joists into a particular span.

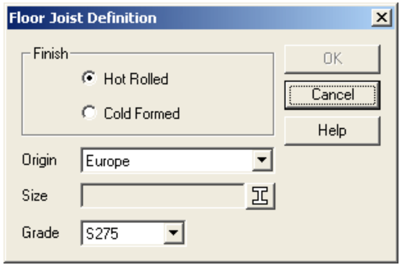
Caution **In this release of** *Portal Frame* **the floors are not involved in the design of the portal frames**. Floors are merely a way of adding ancillary steelwork into your model.

To create floor joists singly

You must use a ***Floor*** window to define the floor joists in your structure.

1. Click the **Create**, **Single** and **Floor Joist** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Floor Joist* menu options).

2. Click the column line or the edge of an area where you want to create the new floor joist.



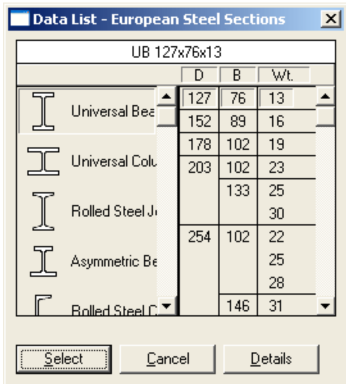
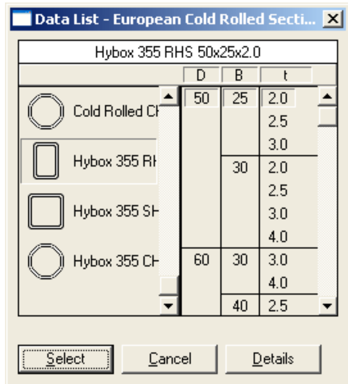
3. Choose the **Finish** of the floor joist you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.*

4. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial **Origin** setting will be that of the country which is set in your Preferences.*

5. To enter the size of your floor joist click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

6. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

7. Pick the **Grade** of material from the list of available grades.

Note *If you choose Hybox sections, then the **Grade** box will show S355, and you can not change this since these sections are only available in this grade.*

8. Once the details for your floor joist is correct click **OK** to return to the graphical display of the structure which will be updated to show the new floor joist.

Deleting floor joists

You can delete floor joists by:

Method	*Details*
Grid Line	all floor joists on the selected grid line are deleted
Area	all floor joists which lie wholly within the selected area are deleted
Single	the floor joist you pick is deleted

To delete floor joists by grid line

You must use a ***Floor*** window to delete the floor joists in your structure.

1. Click the **Delete**, **Grid Line** and **Floor Joist** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Floor Joist* menu options).

2. Click the or grid line along which the floor joist(s) you want to delete lie. All floor joists between this grid line and the next are deleted.

3. Continue to identify grid lines until you have deleted all the floor joists that you require.

To delete floor joists by area

You must use a ***Floor*** window to delete the floor joists in your structure.

1. Click the **Delete**, **Area** and **Floor Joist** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Floor Joist* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All floor joists which lie entirely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the floor joists that you require.

To delete floor joists singly

You must use a ***Floor*** window to delete the floor joists in your structure.

1. Click the **Delete**, **Single** and **Floor Joist** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Floor Joist* menu options).

2. Select the floor joist that you wish to delete.

3. Continue to identify floor joists until you have deleted all the floor joists that you require.

Changing floor joist attributes

When you create floor joists you give the appropriate details for those floor joists. If you need to make changes you can do so without deleting and re-creating these floor joists.

You can modify floor joists details by:

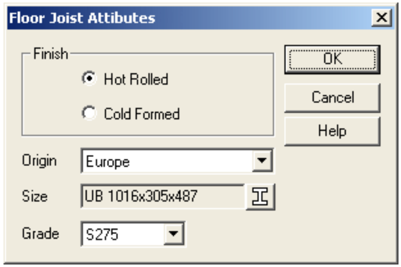
Method	*Details*
Grid Line	all floor joists on the selected grid line are modified to take the details you specify
Area	all floor joists which lie wholly within the area you select are modified to take the details you specify
Single	the floor joist you pick is modified to take the details you specify

To modify floor joist attributes by grid line

You must use a *Floor* window to modify the properties of the floor joists in your structure.

1. Click the **Attributes**, **Grid Line** and **Floor Joist** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Floor Joist* menu options).

2. Click the grid line along which the floor joist(s) whose details you want to modify lie. You will see the ***Floor Joist Attributes*** dialog which allows you to set the attributes that all floor joists along this grid line are to take.



Note *The initial details in this dialog are those for the floor joist nearest to column line A on the line you select.*

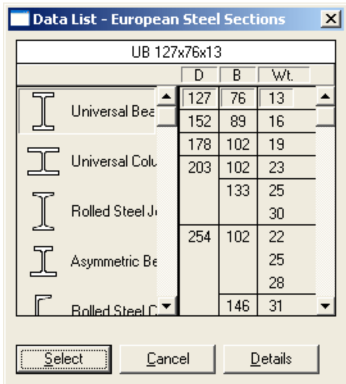
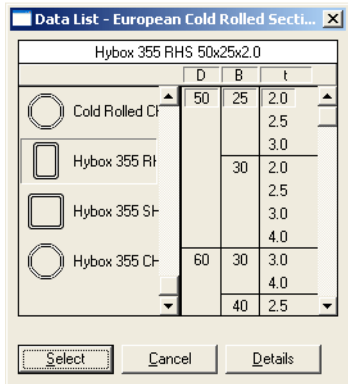
3. Choose the **Finish** of the floor joists you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

4. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

Note *The initial* **Origin** *setting will be that of the country which is set in your* *Preferences*.

5. To enter the size of your floor joists click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections	*Cold Formed Sections*
	

6. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

7. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

8. Once the details for your floor joists are correct click **OK** to return to the graphical display of the structure. All the floor joists on the grid line will be updated to the attributes which you have just defined.

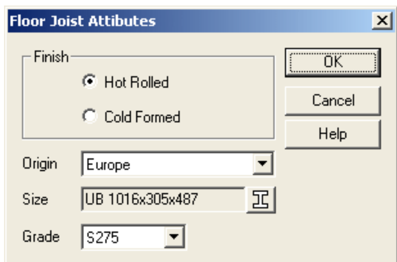
To modify floor joist attributes by area

You must use a ***Floor*** window to modify the properties of the floor joists in your structure.

1. Click the **Attributes**, **Area** and **Floor Joist** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Floor Joist* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All floor joists which lie totally within this area will be selected.

3. You will see the ***Floor Joist Attributes*** dialog which allows you to set the attributes that all the selected floor joists are to take.



Note *The initial details in this dialog are those for the floor joist in your selection which is nearest to column line A and grid line 1.*

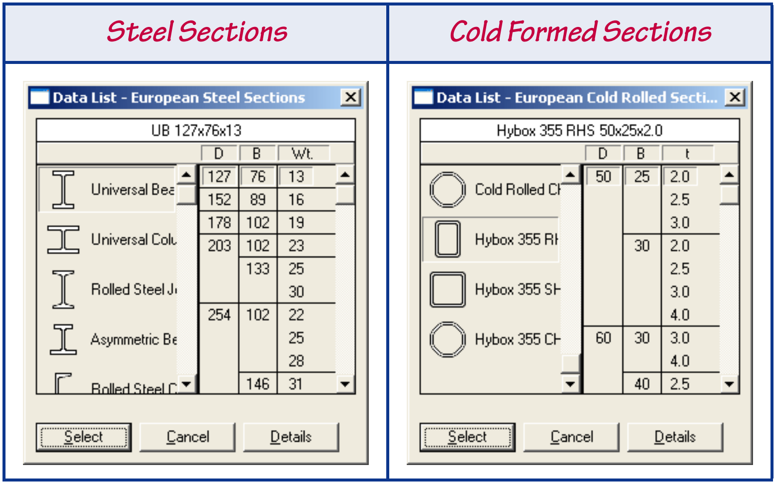
4. Choose the **Finish** of the floor joists you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

Caution If you have chosen section sizes and you subsequently change the **Finish** the existing details will be permanently removed you will have to choose new ones.

5. If you want to use steel from a country other than that which is set in your *Preferences*, then choose the country of **Origin** from the list of available countries.

Note The initial **Origin** setting will be that of the country which is set in your *Preferences*.

6. To enter the size of your floor joists click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.



7. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

8. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

9. Once the details for your floor joists are correct click **OK** to return to the graphical display of the structure. All the selected floor joists will be updated to the attributes which you have just defined.

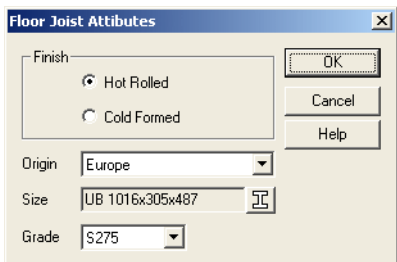
10. Continue to identify areas until you have modified the properties of all the floor joists that you require.

To modify floor joist attributes singly

You must use a *Floor* window to modify the properties of the floor joists in your structure.

1. Click the **Attributes**, **Single** and **Floor Joist** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Floor Joist* menu options).

2. Select a floor joist whose properties you wish to modify. You will see the ***Floor Joist Attributes*** dialog which allows you to set the attributes that this floor joists is to take.



3. Choose the **Finish** of the floor joist you want to use - you can use either **Hot Rolled** or **Cold Formed** sections.

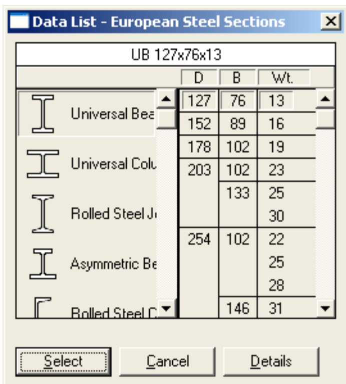
Caution *If you have chosen section sizes and you subsequently change the* **Finish** *the existing details will be permanently removed you will have to choose new ones.*

4. If you want to use steel from a country other than that which is set in your ***Preferences***, then choose the country of **Origin** from the list of available countries.

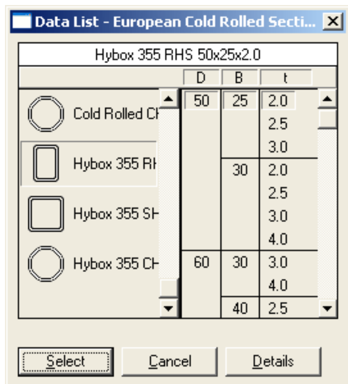
Note *The initial* **Origin** *setting will be that of the country which is set in your* *Preferences.*

5. To enter the size of your floor joist click the … button to the right of the **Section** column whose size you want to set. You will either see the ***Steel Section Data List*** or the ***Cold Rolled Section Data List*** for the country which is currently set in your ***Preferences***.

Steel Sections



Cold Formed Sections



6. Pick the ***Type*** of section that you want to use from the left hand list and the specific ***Section*** from the right. Once you have made your selection click **Select** to return to the ***Floor Wizard*** dialog which will show your selected size.

7. Pick the **Grade** of material from the list of available grades.

Note If you choose Hybox sections, then the **Grade** box will show *S355*, and you can not change this since these sections are only available in this grade.

8. Once the details for your floor joist are correct click **OK** to return to the graphical display of the structure. The floor joist will be updated to the attributes which you have just defined.

9. Continue to identify floor joists until you have modified the properties of all the floor joists that you require.

18 Working with Floor Areas

You create, delete or change the properties of floor areas graphically. As with all ancillary steelwork you must use the ***Structure*** window to work with floor areas.

Creating floor areas

You can create floor areas by:

Method	*Details*
Single	this option allows you to add new floor areas into a particular location.

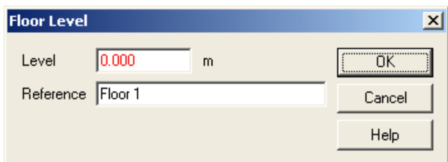
Caution **In this release of *Portal Frame* the floors are not involved in the design of the portal frames**. Floors are merely a way of adding ancillary steelwork into your model.

To create a floor, and its associated ***Floor*** window you can either use the ***Floor Wizard***, or you can create a floor directly using the ***Project Workspace***.

To create a floor using the Project Workspace

1. Right click over the **Floor** icon ( ) in the ***Project Workspace***.

2. From the context menu that appears click *New Floor…*



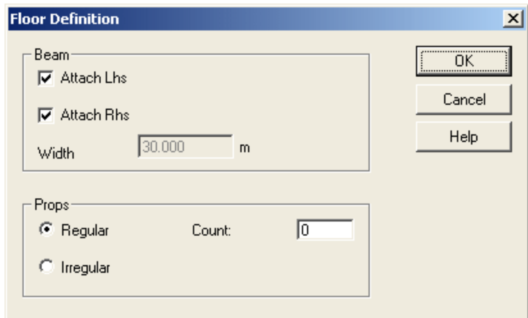
3. Enter the **Level** and **Reference** of the floor and click **OK**. The new floor will be added to your structure, and a ***Floor*** window will be opened for it. You can use this window to define the various floored areas you require.

To create floor areas singly

You must use a ***Floor*** window to define the floor areas in your structure. You can either use the ***Floor Wizard*** to create a floor, or you can create one using the ***Project Workspace***.

1. Click the **Create**, **Single** and **Floor Area** buttons (or choose the *Select / Create*, *Select / Single* and *Select / Floor Area* menu options).

2. Click the bay in which you want to create the new floor area.



3. If the floor extends all the way across the area you have picked, then leave the ticks against the **Attach LHS** and **Attach RHS** columns, otherwise clear the ticks against the side which is unattached and the dialog will reconfigure to allow you to specify the width of the floored area.

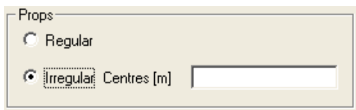
Note With this option you are defining a floored area which extends over a single span, you can therefore not remove the ticks against both **Attach LHS** and **Attach RHS** otherwise you would have a completely free standing floor with no connection whatever to your portal structure.

4. If necessary enter the **Width** for the floor.

5. If you want to space your floor props at equal centres across the span, then click **Regular** and enter the number of props you want to create and ***Portal Frame*** will calculate the appropriate centres for you.

Example If you have a 30 m span frame and you specify a **Count** of 4 then *Portal Frame* will place these props at 6m, 12 m, 18 m, and 24 m effectively splitting the floor beam into 5 equal bays of 6 m.

6. If instead you want to create floor props at unequal centres, then click **Irregular** and the dialog reconfigures to allow you to define the centres you require between successive floor props.



Enter the distances between the props, separating them with commas.

E

Example If you enter centres of *4.5,6,6,4.5* you will create props at *4.5, 10.5, 16.5*, and *21* metres.

You can enter successive, identical values as a multiple separated by an asterisk, *. So, for example, *6,6,6,6* is the same as *4*6*, so you could enter *4.5,2*6, 4.5* to create frames at the same centres as those above.

7. Once the details for your floor areas are correct click **OK** to return to the graphical display of the structure. All the floor areas on the grid line will be updated to the attributes which you have just defined.

Deleting floor areas

You can delete floor areas by:

Method	*Details*
Grid Line	all floor areas on the selected grid line are deleted
Area	all floor areas which lie wholly within the selected area are deleted
Single	the floor area you pick is deleted

Note These options only delete particular floor areas, they do not delete the floor itself. If you want to delete the entire floor, then you must do so using the *Project Workspace*.

Help *see* *"To delete an entire floor"* on page 815.

To delete floor areas by grid line

You must use the ***Structure*** window to delete the floor areas in your structure.

1. Click the **Delete**, **Grid Line** and **Floor Area** buttons (or choose the *Select / Delete*, *Select / Grid Line* and *Select / Floor Area* menu options).

2. Click the grid line along which the floor area(s) you want to delete lie. All floor areas along this line are deleted.

3. Continue to identify grid lines until you have deleted all the floor areas that you require.

To delete floor areas by area

You must use the ***Structure*** window to delete the floor areas in your structure.

1. Click the **Delete**, **Area** and **Floor Area** buttons (or choose the *Select / Delete*, *Select / Area* and *Select / Floor Area* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All floor areas which lie entirely within the area which you select are deleted.

3. Continue to identify areas until you have deleted all the floor areas that you require.

To delete floor areas singly

You must use the ***Structure*** window to delete the floor areas in your structure.

1. Click the **Delete**, **Single** and **Floor Area** buttons (or choose the *Select / Delete*, *Select / Single* and *Select / Floor Area* menu options).

2. Select the floor area that you wish to delete.

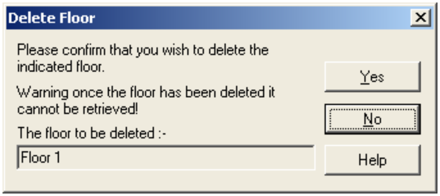
3. Continue to identify floor areas until you have deleted all the floor areas that you require.

To delete an entire floor

In order to delete an entire floor you must use the *Project Workspace*.

1. Right click over the floor reference in the *Project Workspace*.

2. Pick *Delete Floor* from the context menu which appears. You will see the *Delete Floor* dialog, which asks you to confirm the deletion.



3. After you have confirmed that this is the floor you intended to delete click **Yes**. The entire floor will be removed, including all its floored areas, floor beams, floor joists and floor props. This information cannot be recovered, so please use this facility with care.

Changing floor area attributes

When you create floor areas you give the appropriate details for those floor areas. If you need to make changes you can do so without deleting and re-creating these floor areas.

You can modify floor areas details by:

Method	*Details*
Grid Line	all floor areas on the selected grid line are modified to take the details you specify
Area	all floor areas which lie wholly within the area you select are modified to take the details you specify
Single	the floor area you pick is modified to take the details you specify

Note These options only deal with particular floor areas, they do not change the details of the floor itself. If you want to change the Level or Reference of the floor, then you must do so using the *Project Workspace*.

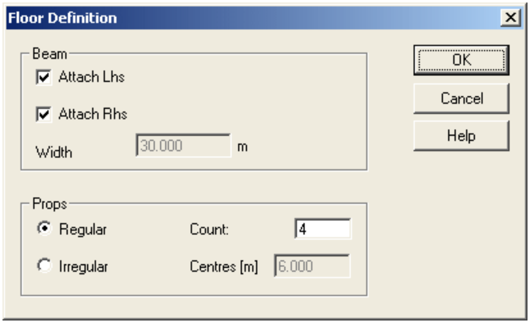
Help *see* *"To delete an entire floor"* on page 815.

To modify floor area attributes by grid line

You must use the ***Structure*** window to modify the properties of the floor areas in your structure.

1. Click the **Attributes**, **Grid Line** and **Floor Area** buttons (or choose the *Select / Attributes*, *Select / Grid Line* and *Select / Floor Area* menu options).

2. Click the grid line to the grid line 1-ward side of the floor area you want to modify. The floor area you are working with will be highlighted. You will see the ***Floor Area Attributes*** dialog which allows you to set the attributes that all floor areas along this grid line are to take.



Note — The initial details in this dialog are those for the floor area nearest to column line A on the line you select.

3. If the floor extends all the way across the area you have indicated, then leave the ticks against the **Attach LHS** and **Attach RHS** columns, otherwise clear the ticks and the dialog will reconfigure to allow you to specify the width of the floored area.

Note *If you are defining a floored area which extends over more than one span, then if you leave the* **Attach LHS** *button ticked the floor will span across the entire width of the first floored span. Similarly if you leave the* **Attach RHS** *button ticked the floor will span across the entire width of the last floored span. If you remove the ticks against* **Attach LHS** *or* **Attach RHS** *then the width you specify is that from the first internal column in the span outward towards the edge of the frame.*

Example If you have a 3 span frame the columns will lie on column lines A, B, C and D. If you define a floor across the entire width of the frame, then this will run between grid lines A and D. If you then remove the tick against both the **Attach LHS** and **Attach RHS** buttons the width which you specify for **Span No.** 1 is measured from column line B towards column line A, and the width you specify for **Span No.** 3 is measured from grid line C towards grid line D.

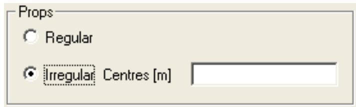
Note *If your floor area is only over a single span, then either* **Attach LHS** *or* **Attach RHS** *must be ticked, otherwise you would have a completely free standing floor with no connection whatever to your portal structure.*

4. If necessary enter the **Width** for each edge of the floor as appropriate.

5. If you want to space your floor props at equal centres across the span, then click **Regular** and enter the number of props you want to create and ***Portal Frame*** will calculate the appropriate centres for you.

Example If you have a 30 m span frame and you specify a **Count** of 4 then *Portal Frame* will place these props at 6m, 12 m, 18 m, and 24 m effectively splitting the floor beam into 5 equal bays of 6 m.

6. If instead you want to create floor props at unequal centres, then click **Irregular** and the dialog reconfigures to allow you to define the centres you require between successive floor props.



Enter the distances between the props, separating them with commas.

Example If you enter centres of *4.5,6,6,4.5* you will create props at *4.5, 10.5, 16.5*, and *21* metres.

You can enter successive, identical values as a multiple separated by an asterisk, *. So, for example, *6,6,6,6* is the same as *4*6*, so you could enter *4.5,2*6, 4.5* to create frames at the same centres as those above.

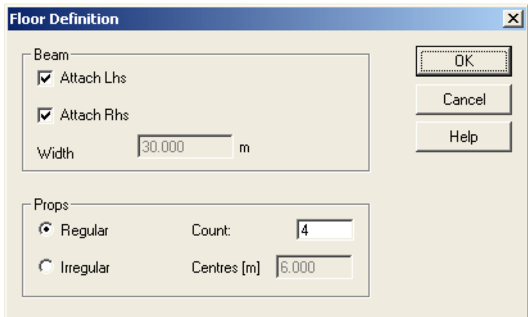
7. Once the details for your floor areas are correct click **OK** to return to the graphical display of the structure. All the floor areas on the grid line will be updated to the attributes which you have just defined.

To modify floor area attributes by area

You must use the ***Structure*** window to modify the properties of the floor areas in your structure.

1. Click the **Attributes**, **Area** and **Floor Area** buttons (or choose the *Select / Attributes*, *Select / Area* and *Select / Floor Area* menu options).

2. Position the cursor at one corner of the rectangle that will define your area. Drag the cursor to the opposite corner. When you are happy with the rectangle that has been drawn, release the mouse button. All floor areas which lie totally within this area will be selected. You will see the ***Floor Area Attributes*** dialog which allows you to set the attributes that all the selected floor areas are to take.



Note *The initial details in this dialog are those for the floor area nearest to column line A and grid line 1 in the area you select.*

3. If the floor extends all the way across the area you have indicated, then leave the ticks against the **Attach LHS** and **Attach RHS** columns, otherwise clear the ticks and the dialog will reconfigure to allow you to specify the width of the floored area.

Note If you are defining a floored area which extends over more than one span, then if you leave the **Attach LHS** button ticked the floor will span across the entire width of the first floored span. Similarly if you leave the **Attach RHS** button ticked the floor will span across the entire width of the last floored span. If you remove the ticks against **Attach LHS** or **Attach RHS** then the width you specify is that from the first internal column in the span outward towards the edge of the frame.

Example If you have a 3 span frame the columns will lie on column lines A, B, C and D. If you define a floor across the entire width of the frame, then this will run between grid lines A and D. If you then remove the tick against both the **Attach LHS** and **Attach RHS** buttons the width which you specify for **Span No.** 1 is measured from column line B towards column line A, and the width you specify for **Span No.** 3 is measured from grid line C towards grid line D.

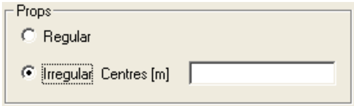
Note If your floor area is only over a single span, then either **Attach LHS** or **Attach RHS** must be ticked, otherwise you would have a completely free standing floor with no connection whatever to your portal structure.

4. If necessary enter the **Width** for each edge of the floor as appropriate.

5. If you want to space your floor props at equal centres across the span, then click **Regular** and enter the number of props you want to create and ***Portal Frame*** will calculate the appropriate centres for you.

Example If you have a 30 m span frame and you specify a **Count** of 4 then *Portal Frame* will place these props at 6m, 12 m, 18 m, and 24 m effectively splitting the floor beam into 5 equal bays of 6 m.

6. If instead you want to create floor props at unequal centres, then click **Irregular** and the dialog reconfigures to allow you to define the centres you require between successive floor props.



Enter the distances between the props, separating them with commas.

Example If you enter centres of *4.5,6,6,4.5* you will create props at *4.5, 10.5, 16.5*, and *21* metres.

You can enter successive, identical values as a multiple separated by an asterisk, *. So, for example, *6,6,6,6* is the same as *4*6*, so you could enter *4.5,2*6, 4.5* to create frames at the same centres as those above.

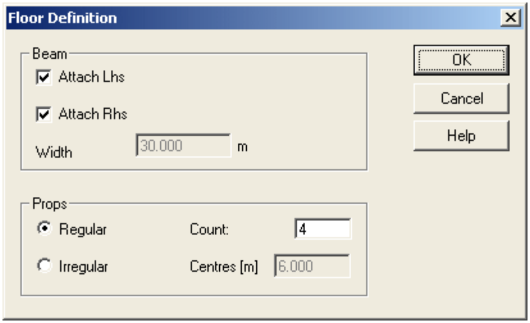
7. Continue to identify areas until you have modified the properties of all the floor areas that you require.

To modify floor area attributes singly

You must use the ***Structure*** window to modify the properties of the floor areas in your structure.

1. Click the **Attributes**, **Single** and **Floor Area** buttons (or choose the *Select / Attributes*, *Select / Single* and *Select / Floor Area* menu options).

2. Select a floor area whose properties you wish to modify. You will see the ***Floor Area Attributes*** dialog which allows you to set the attributes that this floor area is to take.



3. If the floor extends all the way across the area you have indicated, then leave the ticks against the **Attach LHS** and **Attach RHS** columns, otherwise clear the ticks and the dialog will reconfigure to allow you to specify the width of the floored area.

Note *If you are defining a floored area which extends over more than one span, then if you leave the* **Attach LHS** *button ticked the floor will span across the entire width of the first floored span. Similarly if you leave the* **Attach RHS** *button ticked the floor will span across the entire width of the last floored span. If you remove the ticks against* **Attach LHS** *or* **Attach RHS** *then the width you specify is that from the first internal column in the span outward towards the edge of the frame.*

E

Example If you have a 3 span frame the columns will lie on column lines A, B, C and D. If you define a floor across the entire width of the frame, then this will run between grid lines A and D. If you then remove the tick against both the **Attach LHS** and **Attach RHS** buttons the width which you specify for **Span No.** 1 is measured from column line B towards column line A, and the width you specify for **Span No.** 3 is measured from grid line C towards grid line D.

Note If your floor area is only over a single span, then either **Attach LHS** or **Attach RHS** must be ticked, otherwise you would have a completely free standing floor with no connection whatever to your portal structure.

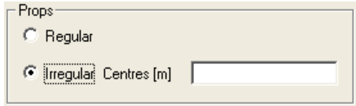
4. If necessary enter the **Width** for each edge of the floor as appropriate.

5. If you want to space your floor props at equal centres across the span, then click **Regular** and enter the number of props you want to create and ***Portal Frame*** will calculate the appropriate centres for you.

E

Example If you have a 30 m span frame and you specify a **Count** of 4 then *Portal Frame* will place these props at 6m, 12 m, 18 m, and 24 m effectively splitting the floor beam into 5 equal bays of 6 m.

6. If instead you want to create floor props at unequal centres, then click `Irregular` and the dialog reconfigures to allow you to define the centres you require between successive floor props.



Enter the distances between the props, separating them with commas.

Example If you enter centres of *4.5,6,6,4.5* you will create props at *4.5, 10.5*, *16.5*, and *21* metres.

You can enter successive, identical values as a multiple separated by an asterisk, *. So, for example, *6,6,6,6* is the same as *4*6*, so you could enter *4.5,2*6, 4.5* to create frames at the same centres as those above.

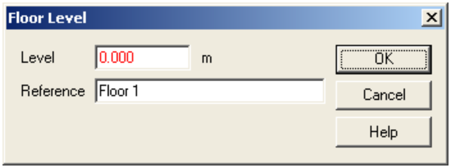
7. Continue to identify floor areas until you have modified the properties of all the floor areas that you require.

To modify an entire floor's details

In order to edit a floor's **Level** or **Reference** you must use the ***Project Workspace***.

1. Right click over the floor reference in the ***Project Workspace***.

2. Pick *Edit Floor* from the context menu which appears.



3. Amend the **Level** and **Reference** of the floor and click **OK**. The floor will take your changed details.

Engineer's Handbook

Portal Frame Design
Release 4.0

1 Scope

This chapter summarizes the scope of ***Portal Frame*** and covers the types of:

- portal span,
- section,
- bases and valley beams,
- haunches,
- additional steelwork,
- loadcases,
- loads,
- design combination,
- member restraints,
- stability check.

Each of these is considered in turn in the following sections of this ***Engineer's Handbook***.

Types of span

Portal Frame can design or check multi-span portal frames. You make up your frames by incorporating spans of the following types:

- symmetrical standard pitched portals,
- asymmetrical standard pitched portals,
- monopitch portals,
- propped portals,
- flat top portals,
- mansard portals.

Types of section

Portal Frame allows you to use:

- *I* and *H* shaped sections for any part of a portal span.
- Any type of section for a tie.

Types of base

You can define bases of the following types:

- pinned,
- fixed,
- spring - horizontal, vertical and rotational.

Valley beams

You can add a valley beam at any eaves point around your frame, specifying the sections making up the valley and its horizontal, vertical and rotational properties.

Types of haunch

You can add a haunch to a span at any eaves, knuckle or apex. these can be:

- cut from an I-section
- built up from plate

Types of additional steelwork

You can include additional steelwork in any span of your frame:

- cranes,
- ties,
- parapets.

Types of loadcase

The loads that you define can be included in loadcases of the following types:

- dead,
- imposed,
- wind,

- crane (providing a crane has been added to a span).

Types of load

You can define loads with the following types:

- full area,
- sheeting,
- area uniform distributed load,
- area varying distributed load,
- line uniform distributed load,
- line varying distributed load,
- point load,
- moment,
- wind,
- crane.

Design combinations

You define your applied loads in loadcases, you then combine these loadcases into a series of design combinations. You can apply the ultimate and serviceability limit state partial safety factors which you want to use for each loadcase in the design combination.

Design

You can either use *Portal Frame* to find acceptable sections for the geometry and loading that you have defined (design frame mode), or you can use *Portal Frame* to check whether a set of sections that you have specified are adequate (check frame mode).

Automatic design

When *Portal Frame* finds section sizes for you it takes the section properties from *Property Files*. The property files that are used depend on the country that you have selected in your preferences. The sections in these files are sorted by serial size, and then by weight. This order is probably not the best for design. ***Portal Frame*** therefore uses another file which lists the sections in the order in which they are to be considered during the design (**Order Files**). If the first section selected from the order file is found to be inadequate for a particular member, then the next section size in the order file is taken and checked. The actual order files and section property files will depend on the country that you have selected in your preferences.

Help For further information on the *Property Files* and *Order Files* ***see***:

- *"Property Files"* on page 529,
- *"Order files"* on page 531.

In the order files the sections are listed in order of increasing weight and increasing M_p value with certain less desirable sections moved to the end of the order files.

Check design

For a check design, you use the property files directly, selecting the sections that you want to check. The actual sections that are available will depend on the country that you have chosen in your preferences.

Help For further information on the *Property Files* ***see*** *"Property Files"* on page 529.

Design checks performed

Portal Frame performs strength checks for the columns, rafters and haunches in accordance with:

- Clause 4.2.3 of BS 5950-1:2000 for shear capacity,
- Clause 4.2.5 or Clause 4.2.6 of BS 5950-1:2000 for moment capacity,
- Clause 4.8.3.2 of BS 5950-1:2000 for cross-section capacity.

As you define each combination you can choose to include the effects of frame imperfections. You can also choose whether you want to perform the ultimate limit state checks and/or the serviceability limit state checks.

Again, on a combination by combination basis, you can opt to perform frame stability checks in accordance with:

- SCI Publication 292,
- Clause 5.5.4.2 of BS 5950-1:2000 for formula sway[1],
- Clause 5.5.4.2 of BS 5950-1:2000 for notional sway[1],
- Clause 5.5.4.4 for the amplified moments method.

You can also check all frame members (whether tapered, uniform or cranked) for out-of-plane stability in accordance with:

- Annex G Elastic,
- Annex G Plastic,
- Clause 5.3.3,
- Annex I.1
- Clause 4.8.3.3.2,
- Clause 4.8.3.3.1.

Note When the program calculates the compressive strength of a member, the approach given in *Annex C of BS 5950* is adopted.

Note When checking members for lateral torsional buckling, the approach in *Annex B of BS 5950* is used.

1. *When the frame has 3 or more spans a Clause 5.5.4.3 snap through check is included for you.*

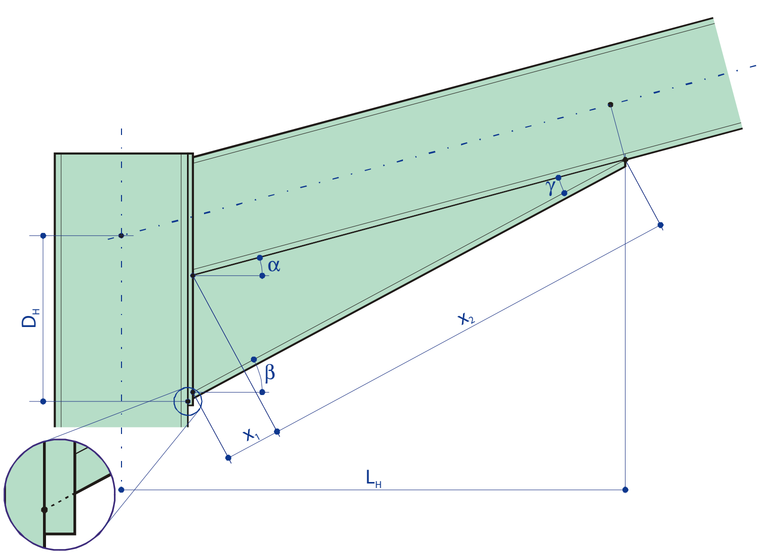
If you have defined restraints and out-of-plane stability checks for members you can choose to automatically perform these as part of the overall design process.

2 Setting-out Details

The following figures show the dimensions that you must give in order to define eaves and apex connections. They also show those symbols that are referred to in the calculations.

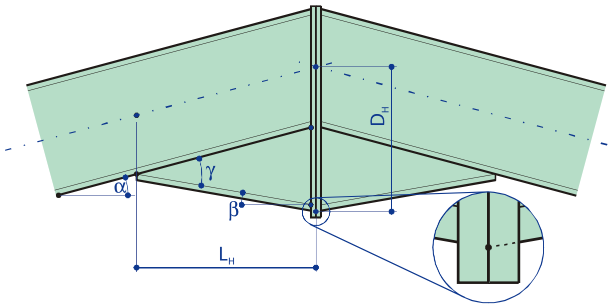
Setting out dimensions

Eaves haunch



Setting out dimensions (Continued)

Apex haunch



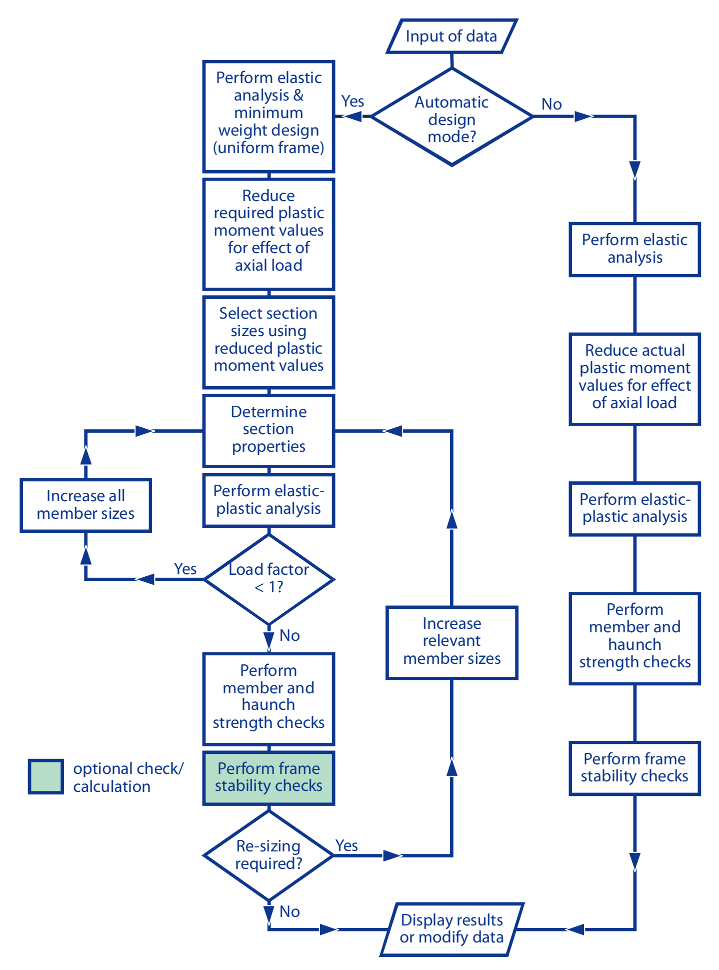
3 Theory and Assumptions

This section describes the theory used in the development of ***Portal Frame*** and the major assumptions that have been made, particularly with respect to interpretation of BS 5950-1:2000[1]. A basic knowledge of analysis and design methods for portal frames is assumed.

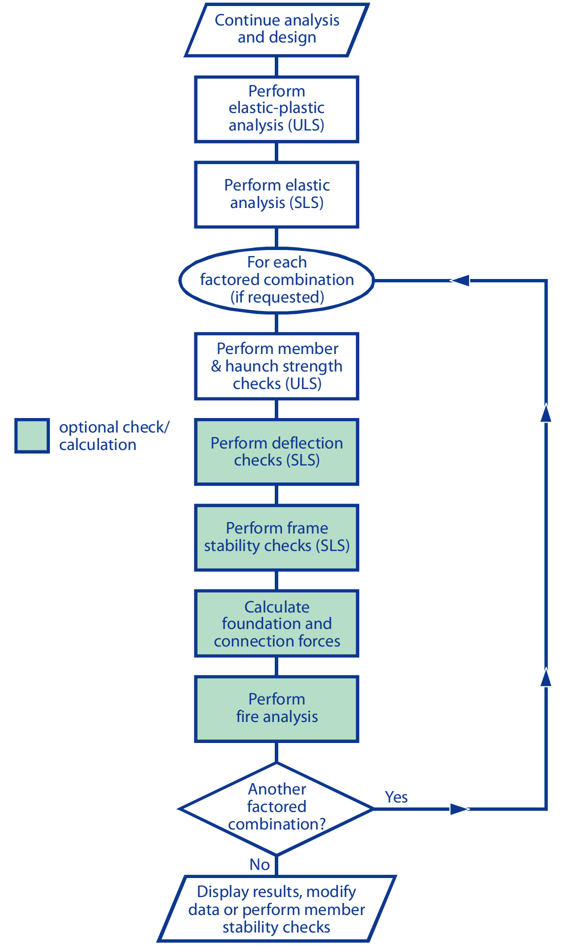
It should be noted that the figures used to illustrate the theory and assumptions in this section are essentially diagrammatic; they are not supposed to represent practical frame designs.

An overview of ***Portal Frame's*** engineering processes is given in the table below.

Engineering processes critical design combination



Engineering processes all other design combinations



Definitions

In this section, terms which have a special technical meaning in the context of *Portal Frame* are explained.

Axial load factor

In certain stages of the analysis method, reduced plastic moment capacity (M_{pr}) values are calculated allowing for the effect of axial load. The loads are taken from an initial elastic analysis of the frame and the M_{pr} values are then used in the elastic-plastic analysis. Plastic re-distribution will alter the member forces between those established for the elastic analysis and those for the elastic-plastic analysis. For this reason, a factor is applied to the initial axial loads prior to calculating the M_{pr} values. *Portal Frame's* default value is based on experience at CSC.

?

Help For information on altering the default axial load factor ***see*** *"To set controls"* on page 425.

Critical section

When the analysis model is generated by *Portal Frame*, it involves the geometry and loading of the frame and points at which plastic hinges might occur. These points are known as ***Critical Sections***. Each critical section will not necessarily be a point at which a plastic hinge forms. However, each plastic hinge will form at a critical section, e.g. at a base or the underside of a haunch.

Critical sections are positioned by *Portal Frame* according to the geometry of the frame. You have no control over the location of the critical sections. There are some complex rules which *Portal Frame* follows in order to locate the critical sections.

There are two types of critical section - those which are fixed to a particular position on a member, i.e. ***Stationary***, and those which *Portal Frame* can move to points of maximum moment along members, i.e. ***Travelling***.

The travelling critical sections are moved during the elastic-plastic analysis of the frame when plastic hinges would form at those points of maximum moment. A point of maximum moment is determined assuming a parabolic distribution which only strictly applies to uniformly distributed loading but is a good approximation in most other cases. This can be checked by viewing the bending moment distribution at a particular load factor. If it is then clear that the approximation is not sufficiently accurate and a significant change in the bending moment distribution exists away from the position of the travelling critical section, then the particular frame must be considered beyond the current scope of *Portal Frame*.

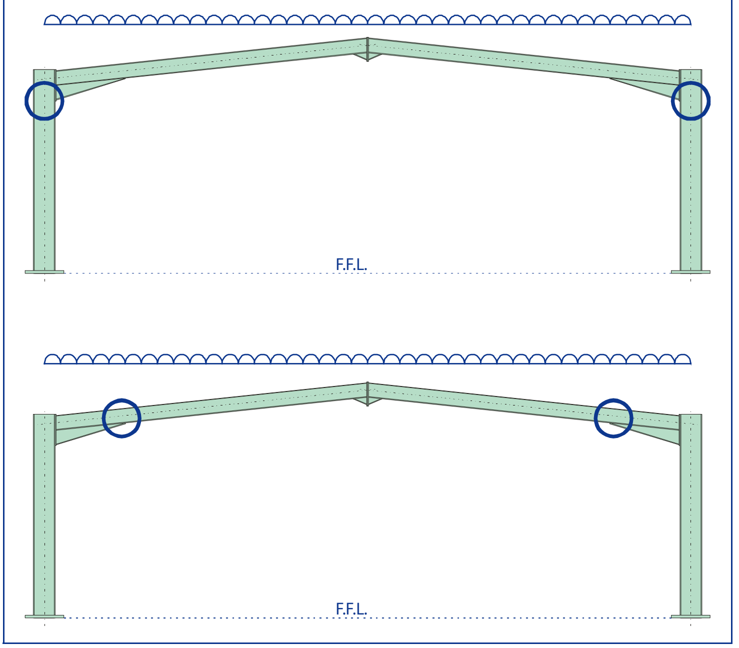
Note *Once a travelling critical section has arrived at a point of hinge formation, it becomes Stationary. It can then* **not** *be moved by Portal Frame, even if the point of maximum moment were to move as a result of redistribution of moments. Such effects are usually small and make very little difference to the final collapse.*

False or `spurious' mechanisms

In certain situations, a frame may contain enough hinges to form a mechanism and yet will not collapse under the particular configuration of applied loads. This is because at least one of the rotations required for the mechanism to cause collapse is in the opposite sense to the bending moment at the hinge position. Such a mechanism is termed ***False*** or ***Spurious***.

The problem of false mechanisms in elastic-plastic analysis was highlighted in an article by Professor J M Davies[4]; an example is shown in the figure below.

False or 'spurious' mechanisms



For either of the mechanisms shown above, the only mode of failure would be in sway and there is no theoretical reason for sway to occur under the loading applied. However, if sway should occur in practice, this would reduce the moment at one of the hinges and so would effectively stop the failure.

CSC has developed routines which overcome this problem. When a possible mechanism has been detected, *Portal Frame* checks that a rotation applied at the final hinge position causes rotations at other hinges that have the same sign as the corresponding bending moments.

Hinge reversal

When any hinge forms, all other current hinges are checked in order to ensure that the applied bending moments are still equal to the reduced plastic moment capacities. If the applied moment has reduced at a given section, then the hinge is considered to have ***reversed***, i.e. the member has regained its stiffness but has a permanent plastic rotation. Subsequently, the member is treated as if no hinge is present. ***The onus is on you to determine the restraint requirements for such a hinge***.

This check is carried out ***before*** testing for a possible collapse mechanism. If formation of the ‘Final’ hinge causes another hinge to reverse, the analysis can continue since a valid collapse mechanism has not been reached.

Maximum plastic hinge rotation

Portal Frame checks against an upper limit on the rotation at any plastic hinge. If any rotation exceeds the limit, the analysis is halted and the frame is deemed to have collapsed, although a mechanism might not have occurred. Provided that the limit is not exceeded at Ultimate Limit State (1oad factor 1.0), then the design is valid. Otherwise, *Portal Frame* will give you an error message.

Research at *Bradford University* has indicated that well-restrained hinges in sections classified as *Plastic* to *BS 5950* cannot sustain rotations exceeding 6° typically. This value has been adopted as a default by *Portal Frame*, although you can change it if you feel that such a change is justifiable.

? **Help** For information on altering the hinge rotation that you deem acceptable **see** “To set limits” on page 417.

Percentage of M_p for plasticity

In a symmetric portal frame, positions other than those hinge positions identified by *Portal Frame* might be very close to collapse. These other positions would not be shown on the bending moment diagram if the percentage of M_p for plasticity were very close to 100. By entering a lower value, you can have points where the moment is further below M_p shown as plastic hinge positions. Also, the stability checks will use this percentage value when determining the acceptability of the various types of check.

? **Help** For information on altering the percentage of Mp at which plasticity will be indicated *see* "To set limits" on page 417.

Travelling critical section

see ***"Critical section"*** on page 839.

Design method

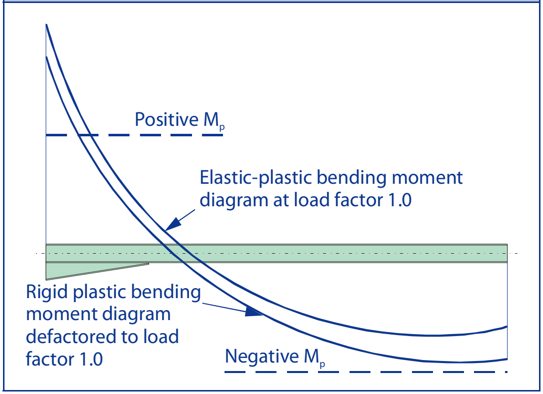
Portal Frame uses the ***Elastic-plastic*** design method rather than the more common ***Rigid Plastic*** method. In addition to determining the final collapse mechanism for a structure, the elastic-plastic method gives information about the redistribution process which takes place prior to collapse.

This method finds the order in which the hinges form, calculates the load factor associated with each hinge formation and identifies how the bending moments in the frame vary between each hinge formation.

The main advantage of the elastic-plastic design method is the ability to establish the state of the frame at any load factor and not only at collapse. This allows a more accurate determination of the bending moment diagram at the design load factor.

A comparison of the bending moment diagrams associated with the elastic-plastic and rigid plastic design methods is given in the figure below. It may be noted that the position of the point of contraflexure can vary throughout the analysis process.

Comparison of bending moment diagrams



Analysis for the critical design combination

The analysis for the critical design combination depends on whether you are checking sections that you have specified, or are designing the frame when the section sizes will not be known.

Manual design

First stage

When performing manual design (i.e. specified section sizes are checked for adequacy), the first stage is to determine the reduced value of plastic moment capacity M_{pr} for each member group which allows for the presence of axial load.

Axial loads in all members are found from a linear elastic analysis of the frame, which uses a standard stiffness matrix solution. A reduced plastic modulus (and hence the M_{pr} value) may be calculated using equations given in the SCI Guide to BS 5950[3]. As plastic re-distribution in the frame will alter the member forces, a factor is applied to the axial loads prior to calculating the M_{pr} values. You can alter this value when necessary.

As an alternative, you may specify M_{pr} values for some or all of the members as part of the input data. This avoids the problem of the reduced capacity for a group being based either upon the maximum axial load in any member in the group, or upon an axial load that is much higher than that at a hinge position. Specifying M_{pr} values in this way allows you to reduce pessimism. If you have specified reduced capacities for all members, then the results of the analysis and calculations described above are not taken into account; the elastic-plastic analysis is performed using your specified values.

Second stage

The second stage of the manual design process for the critical design combination consists of an elastic-plastic analysis. This performs a linear elastic analysis on the frame with additional terms in the stiffness matrix that represent the application of unit rotations at critical sections (possible hinge positions).

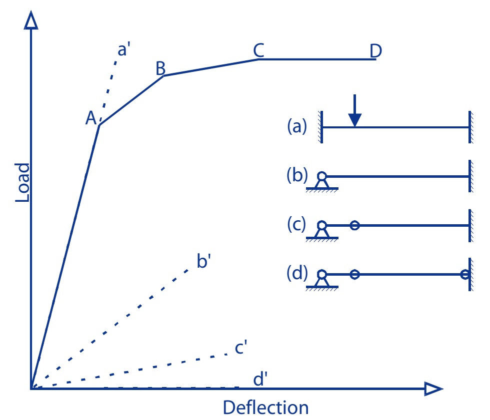
Initially, the elastic behaviour of the frame is examined. The applied moment M_{app} at every critical section is compared to the M_{pr} value and a hinge is set at the position with a maximum ratio of M_{app} to M_{pr}. The load factor corresponding to formation of this hinge is then calculated.

An arbitrary increment of load above this load factor is then applied in combination with the effects of plastic rotation at the hinge. By examining the rate of change of bending moment at other critical sections, the next hinge position may be determined. Factoring of the applied bending moment at that position so that it is equal to the relevant value of M_{pr} will then give the load factor corresponding to the formation of the second hinge.

The above procedure is repeated until a failure mechanism is found. This approach corresponds to a conventional incremental approach to elastic-plastic analysis, but, because the elastic analysis is not re-performed each time a hinge is formed, it is considerably quicker.

It is important to note that the structure behaves elastically between hinge positions. This is shown diagrammatically in the figure below which indicates the changing state of a single mechanism through four stages of hinge formation.

Elastic behaviour between hinge positions



Automatic design

The automatic design route uses a minimum weight approach to determine initial section sizes, then performs an elastic-plastic analysis identical to that described in the section on Manual Design. If the frame or any of its members is not adequate, the size is increased and the analysis is performed again.

You can control the checks that will force ***Portal Frame*** to increment the sizes of the members of your frame above those needed to meet the basic strength requirements.

?

Help For information ***see*** *"To set controls"* on page 425.

First stage

The first stage of automatic design commences with a linear elastic analysis for the critical design combination (including an allowance for the frame self-weight). This analysis is performed assuming uniform arbitrary section properties.

The ***critical*** design combination is the one which ***Portal Frame*** uses to govern the selection of sections; you specify which is the critical combination during input.

The results are used in order to determine a required plastic moment of resistance M_p value for each group of members that will give a minimum overall weight for the frame. This approach follows that described by Professor J M Davies in his paper ***Approximate minimum weight design of steel frames***[5].

The approach gives required M_p values (for each member group) which are a close approximation to the 'true' minimum weight values for most frame geometries. Once the required M_p values are established, ***Portal Frame*** selects steel sections with capacities greater than those required. In some instances, it might be possible to improve upon the selected sections by performing a manual design.

A point of note is that the weight of a member is assumed to be proportional to its plastic moment capacity – this is a good approximation for most rolled sections.

Second stage

The second stage of automatic design consists of finding a suitable section size for each member group, based upon the required M_{pr} values determined by the minimum weight design as well as the required BS 5950-1:2000 section classification – ***Plastic*** or ***Compact***. The former requirement is satisfied by using a method similar to that carried out in manual design (see above), calculating a reduced plastic moment capacity M_{pr} which allows for the presence of axial load.

Once the section sizes are known, the properties of all members and haunches are found in preparation for an elastic-plastic analysis. The self-weight loading for the current member sizes is also determined at this stage.

(Third stage)

An elastic-plastic analysis is now performed using the trial section sizes. The analysis is carried out in a similar way to that for manual design.

It is possible (although very unlikely) that the resulting load factor for the frame would be less than 1.0. In such a case, each member group would be increased to the next size that satisfies the requirements of the second stage of the automatic design in the relevant design order file. The section properties would then be re-calculated and the elastic-plastic analysis repeated.

Once the frame has a load factor greater than or equal to 1.0, the member and haunch strength checks and the frame stability checks are performed (see below). If any member (other than a haunch) fails these checks, all the members in its group are increased to the next size that satisfies the requirements of the second stage of the automatic design in the relevant design order file. The section properties are then recalculated and the elastic-plastic analysis is repeated.

Additional controls on the design process

In addition to the checks that are required by BS 5950-1:2000 ***Portal Frame*** allows some other checks that can affect the sizes that it picks for the members of a frame.

These checks are controlled from the ***Controls*** page of the ***Design Wizard***.

Help For further information ***see*** *"To set controls"* on page 425.

Slenderness and stability of internal columns

For internal columns in symmetrical multi-span frames that are subject to vertical loading only, the bending moments will be very small. This might cause the initial designed member size to be unrealistically small where the choice of member size depends upon bending moment alone.

Overall buckling of the internal column can be checked using the equation given in Clause 4.8.3.3.2 of BS 5950-1:2000. In carrying out this check, a value of ***1.0*** is used for ***m*** and the unrestrained length is assumed to be the clear height to the underside of the eaves. For more information about this check ***see*** *"Clause 4.8.3.3.2 check"* on page 872

You can specify a slenderness ratio limit (defaulted to 250) for all internal columns. The effective length will be taken as 1.0 times the clear height and the resulting minor axis slenderness is checked against the limit you have set. This is simply a robustness requirement (and not a code check) so that you can ensure that you have a reasonably sized, robust column at that position.

The overall buckling check is carried out by ***Portal Frame*** on completion of the elastic-plastic analysis since it is only at this stage that details of the forces and moments are known. In the event that this check fails, an additional pass through the elastic-plastic analysis is carried out.

Member strength checks

Member strength checks are performed at ten points on the column (from the base to the underside of the haunch), at twenty points on the rafter (from the sharp end of the haunch to the apex) and at five points in the eaves haunch and apex haunch.

Additionally, any other points of interest (such as the start and end positions for distributed loads and the positions of application for point loads or point couples) are checked. The results at the most severe positions for moment, for shear and for combined axial force and moment are displayed for your convenience.

The checks on section classification, shear force, bending moment and combined axial force and bending moment are performed in accordance with BS 5950-1:2000 unless noted otherwise in the following sections.

Section classification

The classification of the basic (two flanged) cross-section is determined using Table 11 and Clause 3.5 of BS 5950-1:2000.

Rafters, columns and haunches can be classified as:

- Plastic (Class 1)
- Compact (Class 2)
- Semi-compact (Class 3)

Slender (Class 4) sections are not allowed.

The following restrictions are applied as to when a particular classification is acceptable:

- If a plastic hinge exists at a particular cross-section at a load factor less than 1.0 (the ultimate limit state load factor), then the section classification must be plastic (Class 1).

- If the frame is plastic (that is at least one hinge forms before a load factor of 1.0 is reached) and there is:
  - no hinge at the particular cross-section at any load factor,
  - a hinge at the particular cross-section which only occurs at a load factor greater than 1.0.

  then the minimum section classification is compact (Class 2).
- If the frame is elastic (that is no hinges occur below a load factor of 1.0), then the section must have the minimum classification of semi-compact (Class 3).

The flanges and the web are classified separately and the overall classification of the section is the worse of these.

Shear capacity

The member shear capacity is determined in accordance with Clause 4.2.3 of BS 5950-1:2000. Where the applied force exceeds 60% of the capacity, the high shear condition applies to the bending capacity checks (see the following section).

All I and H sections have depth-to-thickness ratios less than the limiting value of $70 \mathbf{x} \varepsilon$, so that the shear buckling checks of Clause 4.4.5 have not been included in ***Portal Frame***.

Bending moment capacity

The bending moment capacity for the member is calculated using the equations given in Clause 4.2.5 of BS 5950-1:2000 for plastic, compact and semi-compact sections. The level of shear (low or high) in the section under consideration governs which sub-clause is used.

Axial capacity

The axial capacity (tension and compression) for the member is calculated to guard against the possibility of not identifying a failure when the moment is zero and the ‘alternative’ formula for cross-section capacity in Clause 4.8.2.3 is used. The capacity is based on the gross section area and does not include for the effect of any holes.

Note The compression resistance of the member is a buckling check and is covered in the member stability checks.

Help For further information *see* "Frame stability checks" on page 857.

Cross-section capacity

The cross-section capacity check covers the interaction between axial force and bending moment in accordance with Clause 4.8.2 and Clause 4.8.3.2. For portal frames there is no minor axis bending, furthermore ***Portal Frame*** uses the absolute values of the force and moment which allows the formulae to be simplified and used irrespective of the sense of the load.

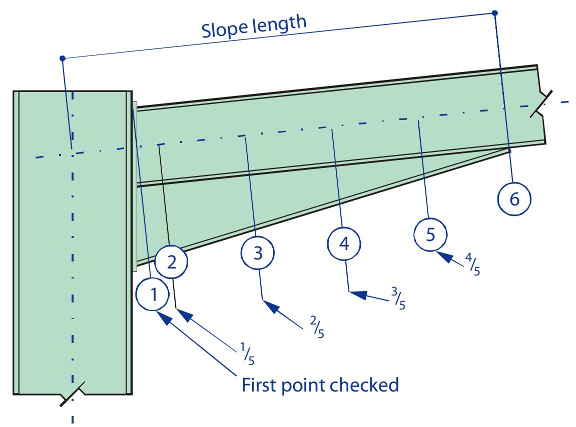
For plastic (Class 1) and compact (Class 2) unhaunched sections in the low shear condition the cross-section capacity is calculated in accordance with Clause 4.8.2.3. For semi-compact (Class 3) sections the simplified method in Clause 4.8.3.2(a) is used.

The high shear condition is generally rare in portal frames and therefore the requirements of Annex H.3 have not been implemented. This condition is deemed beyond the scope of the current version of the program and will yield a ***Beyond Scope*** status for this check.

Haunch strength checks

The capacity of haunched members is checked at sections 1 to 5 inclusive as shown in the figure below. The length between sections 5 and 6 forms the transition between the haunched portion and the uniform portion of the rafter.

Positions of haunch checks



Additionally, any other points of interest (such as the start and end positions for distributed loads and the positions of application for point loads or point couples) between sections 1 and 5 are checked. Strength checks are carried out for both eaves and apex haunches.

The checks are similar to those performed for the member checks (*see* "Member strength checks" on page 850) but differ in detail because of the approach to classification for haunched members (see below).

Haunch classification

Firstly the haunch is idealised into a three flange section without root radii.

Note This introduces a small amount of conservatism into the classification since the depth of the web when calculating the *d / t* ratio will be slightly larger than that for the rolled section of the rafter.

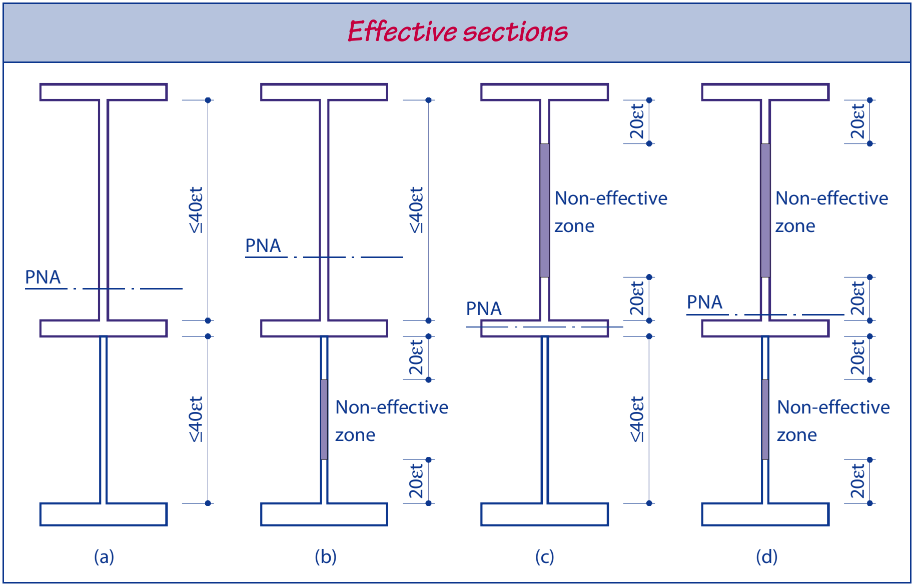
The flanges and webs of the haunch sections are classified separately.

For a three flanged section the classification of the flanges is not independent of the load. For positive moments the flanges of both the rafter and the haunch sections are classified since it is not certain whether the middle flange is in tension or compression. For negative moments the rafter flange only is classified since it is extremely unlikely that the haunch flange will be in compression in this bending condition.

For the webs the ***d/t*** ratios of the section and the haunch are determined. If both of these are $\leq 40\,\varepsilon$, then the webs will be stable even if they are at full p_y throughout. Hence under any combination of axial load and bending the webs are plastic.

If the ***d/t*** ratio for either web is $> 40\,\varepsilon$, then the r_2 value for both the rafter and the haunch section is calculated in accordance with Clause 3.5.5(b) of BS 5950-1:2000. These values are checked against the semi-compact/slender limit. If either of these are found to be slender, then the section is failed since slender sections are beyond the scope of this version of ***Portal Frame***.

Provided that the webs are not slender and the flanges are classified either as plastic or compact, then the effective section is determined in accordance with the requirements of Clause 3.5.5(b) of BS 5950-1:2000. This effective section varies depending in whether the depth of one or both of the webs is > 40 ε as shown in the figure below.



The section is then deemed to be Effective Compact (Class 2) and its plastic modulus is defined as S_{eff}.

If any flange is neither Plastic or Compact, then the web classification is taken as semi-compact (Class 3).

The overall classification of the section is the worst of the web and flange classifications.

Shear capacity

This is determined using the shear capacity equation in Clause 4.2.3 of BS 5950-1:2000. The depth of the section is calculated using one of the following:

- the total depth of the haunched member for a rolled section haunch,
- the total depth less the bottom flange thickness for a built-up haunch.

The shear area uses this depth and multiplies it by the minimum web thickness.

Where the applied force exceeds 60% of the capacity, the high shear condition applies to the bending capacity checks (*see* "Bending moment capacity" on page 856)

The check on the limiting depth-to-thickness ratio needed to avoid shear buckling is performed on both the rafter web and the haunch web. This assumes that the middle flange prevents buckling over the full depth of the haunch. For the rafter component and for haunches fabricated from a section cutting, the limit is taken as 70ε. For a haunch which is fabricated from plates (a built-up haunch) a limit of 62ε is applied.

No shear buckling calculations are performed, but a warning is given if the above limits are exceeded.

Bending moment capacity

The bending moment capacity for the member is calculated using the equations given in Clause 4.2.5 of BS 5950-1:2000 for plastic, compact or semi-compact sections. As for the member checks, the level of shear (low or high) in the section under consideration governs which sub-clause is used.

Axial capacity

The axial capacity (tension and compression) for the haunch is calculated for completeness only. The capacity is based on the gross section area and does not include for the effect of any holes.

Note The compression resistance of the member is a buckling check and is covered in the member stability checks.

Help For further information *see* "Frame stability checks" on page 857.

Interaction between axial force and bending moment

The interaction between axial force and bending moment is checked for plastic, compact or semi-compact sections using the equation given in Clause 4.8.3.2(a) of BS 5950-1:2000. The effective area for cases with axial tension is assumed to be equal to the gross area, so that the expression equates to that given in Clause 4.8.2. The form of the equation means that the interaction check will fail if the axial capacity of the member cross-section is exceeded. Nevertheless, a separate check is carried out for axial capacity.

If the high shear condition occurs, the action taken will be identical to that for the main member strength checks (*see* "Cross-section capacity" on page 852)

Frame stability checks

Four main checks are available:

- SCI Publication P292,
- Formula sway to Clause 5.5.4.2,
- Notional sway to Clause 5.5.4.2,
- Amplified moments method to Clause 5.5.4.4.

Both the formula and notional sway checks include a snap through check when this is appropriate.

Certain individual members are also checked for in-plane buckling.

Any of the sway stability and snap-through checks can be used as check conditions for any design combination. Then, failure will simply be reported to you using the normal results process.

SCI publication P292

In this method the load factor at failure, λ_f, is calculated from the first order collapse load factor, λ_p. It uses the principle of Conservation of Energy in the deflected structure. This makes it a more general method that can readily be applied to rigid-plastic or elastic-plastic analysis.

When considering a rigid-plastic analysis, a hand method can be developed that uses the Virtual Work of the collapse mechanism – this is mathematically identical to applying the principle of Conservation of Energy.

The theory behind both the hand method for rigid-plastic analysis and the method suitable for implementation within elastic-plastic analysis software is described in the Steel Construction Institute publication, P-292, "***In-plane stability calculations for portal frames***"[1]. Full details with examples of the hand method are contained in that publication.

For both methods the Conservation of Energy can be expressed in terms of the increment of energy for an infinitesimal increment in deflection. Thus the method calculates the load factor at failure from the equation:

$$dU_{L1f} + dU_{aL2f} = dU_{M1}$$

where

dU_{L1f} = the increment of energy released by the loads at failure

dU_{aL2f} = the additional increment of energy released by the loads due to second-order effects

dU_{M1} = the increment of energy absorbed by the frame in first-order behaviour in both elastic curvature and plastic hinge rotation

1. *C.S.C. (UK) Ltd. are pleased to have been able to collaborate very closely with the Steel Construction Institute, and in particular Mr. C. M. King, in the development of the methods contained in this publication.*

The failure load factor, λ_f, can now be introduced in the above equation which, with some rearrangement, will give a direct determination of λ_f.

$$\lambda_f = \lambda_p \times (1 - U_{aL2f} / dU_{M1})$$

The method (Conservation of Energy) for calculating the load factor at failure, λ_f, relies on determining the increment of energy released by the load effects and that absorbed by the frame. The increment should be as small as possible. This requires that a prediction be made of the forces and deflections around the frame at a load factor that is very close (in this case 99% of λ_p) to the collapse factor, λ_p. This load factor is called λ_p'.

Energy released

The increment of energy released by the loads due to second order effects, dU_{aL2f}, has two components:

$$dU_{aL2f} = \Sigma P_2 \times d\delta_{rb} + \Sigma \int P_2 \times ds$$

where

P_2 = the member axial forces from the first order analysis at collapse enhanced to allow for second-order effects

$d\delta_{rb}$ = the increment of the rigid body movement between λ_p' and λ_p in the line of action of the axial force in the member, P_2

ds = the increment in the shortening of the member due to curvature between λ_p' and λ_p

The first term is called the "rigid body movement term" and the second term is called the "shortening due to curvature term".

Energy absorbed

The increment of energy absorbed by the frame in bending, dU_{M1}, has two components:

$$dU_{M1} = \int M \times dk + \Sigma M_p \times d\theta$$

where

M = the values of moment around the frame

dk = the increment of curvature associated with the moments, M, around the frame between λ_p' and λ_p

M_p = the reduced plastic moment of resistance of the members at the hinge position

$d\theta$ = the increment of rotation at each of the hinge positions between λ_p' and λ_p. This can be zero if the hinge is the last one to form or has a "fixed rotation" (reversed hinge)

The first item is called the "moment curvature term" and the second term is called the "hinge rotation term".

Sway check methods

These are performed in accordance with Clause 5.5.4.2 of BS 5950-1:2000. The two checks are described below.

Notional Sway Check

The Notional Horizontal Forces Sway Check has a different formulation for those Design Combinations containing gravity loads only and for those containing horizontal loads. In this context gravity loads are taken to mean Dead and Imposed loadcase types (even though horizontal loads can be included in such loadcases) as well as Crane load types when these are vertical and are not acting in conjunction with horizontal crane loads. On the other hand, any design combination containing the Wind loadcase type or horizontal crane loadcase type is considered as having horizontal loads.

Certain restrictions are applied to the use of the Sway Check Method (in both the notional loads and formula guises). These are geometrical checks based on the dimensions of each span. The check is valid if:

- the span divided by the mean height of the columns (measured from base to eaves) is less than or equal to 5.0. That is $L/(h_1 + h_2)/2 \leq 5.0$
- the height of the apex above the tops of the stanchions is limited to a proportion of the span (see Figure 18 in BS 5950-1:2000).

One further restriction is the allowance for the stiffening effects of the cladding. This is not permitted when considering gravity loads only, but is allowed when the design combination includes wind loads.

The notional sway check can not be used for tied portals.

Gravity loads

In this check notional horizontal loads are applied to the frame. The resulting horizontal deflection at the top of each column is checked against the base-to-eaves distance divided by the factor that you specified on the ***Limits*** page of the ***Design Wizard*** (default 1000). If the deflection for any column exceeds the limit, then the frame has failed the check.

Note As well as an ***height over*** limit, you can set an absolute limit in mm, or you can specify that no limit is to be applied.

Note For gravity loading no allowance should be made for the restraining effect of cladding.

In order to perform this check, you must have selected this option for a particular design combination in the ***Design Wizard***. If a column has two eaves levels, the horizontal load is split between each level and the check is also performed at each level. Any point loads applied to the column (e.g. crane loads) are considered to act at their point of application, not at the eaves.

Horizontal loads

For design combinations containing wind loadcases the approach is different. First, an estimate of the critical buckling load factor for sway modes is made for each column using the formula:

$$\lambda_{sc} = h_i/(200 \times \delta_i)$$

Note BS 5950-1:2000 allows you to take account of bracing and/or sheeting to reduce δ_i.

Note If $\lambda_{sc} < 5.0$, then the frame is not suitable for treatment in such a simple fashion under this loading regime. The status of the check will be set to Beyond Scope and the check will fail.

The minimum value of the elastic critical buckling load factor for sway modes considering all the stanchions in the frame is then used to determine the required load factor (at collapse), λ_r.

$$\lambda_r = \lambda_{sc}/(\lambda_{sc} - 1)$$

Simplified Formula Sway Check

The Formula Sway Check has a different formulation for those design combinations containing gravity loads only and for those containing horizontal loads. In this context gravity loads are taken to mean dead and imposed loadcase types (even though horizontal loads can be included in such loadcases) whilst any design combination containing a wind loadcase type is considered as having horizontal loads. Design combinations containing the crane loadcase type are specifically excluded from the Formula Sway Check.

Certain restrictions are applied to the use of the Sway Check Method (in both the formula and notional loads guises). These are geometrical checks based on the dimensions of each span. The check is valid if:

- the span divided by the mean height of the columns (measured from base to eaves) is less than or equal to 5.0. That is $L/(h_1 + h_2)/2 \leq 5.0$

- the height of the apex above the tops of the stanchions is limited to a proportion of the span (see Figure 18 in BS 5950-1:2000).

In addition you can not choose the Formula Sway Check in the following conditions:

- the design combination contains a crane loadcase type,
- the frame has one or more valley bases.

Caution If the frame is subject to significant concentrated loads from valley beams or other sources, then this check is not appropriate. However it is not possible for *Portal Frame* to determine what constitutes a significant concentrated load. This is a matter for your judgment, you should only use this check if you deem that there are no significant concentrated loads in this design combination.

The Formula Sway Check can not be used for tied portals.

Gravity loads

In the calculation of the arching ratio (omega – Ω) the haunches are included because plastic failure of the rafter can occur only over the un-haunched length.

This check cannot be performed for any span which has:

- one or more of the columns omitted (e.g. for valley bases),
- two rafters of a different size,

as the equation is then meaningless.

Horizontal loads

For design combinations containing wind loadcases the approach is different. First, an estimate of the elastic critical buckling load factor, λ_{sc}, for sway modes is made for each span using the formula in the code.

Note If $\lambda_{sc} < 5.0$, then the frame is not suitable for treatment in such a simple fashion under this loading regime. The status of the check will be set to Beyond Scope and the check will fail.

The minimum value of the elastic critical buckling load factor for sway modes considering all the spans in the frame is then used to determine the required load factor (at collapse), λ_r.

$$\lambda_r = \lambda_{sc}/(\lambda_{sc} - 1)$$

Snap-through stability checks

A check on the snap-through stability of the frame can be performed when there are more than two spans. The check is applied using the equation given in Clause 5.5.4.3 of BS 5950-1:2000.

Instability could occur in a given span through spreading of the columns and inversion of the rafters causing the beneficial effect of axial thrusts from adjacent spans to be lost. As a consequence, this check is not applicable to monopitch spans.

Furthermore, the check is not carried out if columns have been omitted from the particular span, as the equation is then meaningless. Another constraint is that the average rafter slope must lie between 0° and 45°, as the equation is not suitable for a value of 0° and snap-through is very unlikely for slopes greater than 45°.

Amplified moments method check

In this method the required load factor (λ_r) is determined from Clause 5.5.4.4. This is the Ultimate Limit State load factor at which the forces and moments around the frame are determined using the results of the first order elastic-plastic analysis for the individual design combination.

This method requires the determination of the elastic critical buckling load factor, λ_{crit}. This is determined directly from an elastic buckling analysis of the frame for each design combination.

If λ_{crit} is greater than 10 then λ_r is taken as 1.0. Otherwise λ_r is determined from,

$$\lambda_r = 0.9 \times \lambda_{crit} / (\lambda_{crit} - 1)$$

If λ_{crit} is less than 4.6, the amplified moments method is not suitable for this frame and design combination. ***Portal Frame*** sets a design status for the design combination to *Invalid*.

In-plane buckling of individual members

For most structures, all the members resisting axial compression must be checked to ensure adequate resistance to buckling about both the major and minor axes.

For portal frames checked for in-plane stability using one of the methods in Clause 5.5.4 of BS 5950-1:2000, in-plane buckling is not the critical design case for most members. These members include those in which both:

- axial compressive loads are relatively low, and
- relatively large bending moments occur away from the maximum strut action moments.

For such members the strut action moment is so low relative to the maximum moments that separate checks for in-plane buckling are not required. Exceptions to this are:

- internal columns where no significant ‘step’ exists,
- pinned props,
- rafters in tied portals.

For such members a Clause 4.8.3.3.2(a) check is performed.

Analysis for other design combinations

All selected design combinations, other than the ***Critical*** and the ***Fire*** condition (if specified), are analysed in a similar way to that for manual design for the critical combination (***see*** *"Manual design"* on page 844). Thus, when in automatic design mode, re-sizing of members and re-analysis of the frame is ***not*** performed if any design checks fail in design combinations other than the critical.

It is important to note that the values for reduced plastic moment capacity M_{pr} are not re-calculated for each combination, i.e. the values for the critical combination are used throughout.

For details of the analysis performed for the Fire condition ***see*** *"Fire analysis"* on page 868

Frame imperfections

Clause 2.4.2.3 of BS 5950-1:2000 states:

> "To provide a practical level of robustness against the effects of incidental loading, all structures, including portions between expansion joints, should have adequate resistance to horizontal forces."

For gravity load design combinations (those including only dead and imposed loadcases) this is achieved by applying the notional horizontal forces given in Clause 2.4.2.4. For other design combinations (those including wind and crane loadcases) there will generally be sufficient horizontal load present to ensure this level of robustness. For design combinations that include crane loadcases this is deemed always to be the case. For those containing wind loadcases the code requires that the wind loads should be not less than 1.0% of the dead load applied horizontally. ***Portal Frame*** assumes this to be the case, and so for design combinations containing wind loadcases the inclusion of notional horizontal loads is deemed unnecessary.

Determination of notional horizontal forces

Notional horizontal forces are applied to allow for the effects of practical imperfections in the structure, for example the lack of verticality. They are taken as 0.5% of the factored vertical dead and imposed loads applied at the same level.

You can set up design combinations, which, although they contain only dead and imposed loadcase types, do not need to include for the effect of frame imperfections. An example of such a design combination would be one which contains a loadcase relating to snow drift loads. In this case the design combination does only include dead and imposed loadcases. However the asymmetric drift loading introduces sufficient asymmetry (the tendency to deflect horizontally) that this is an adequate substitute for the notional horizontal forces. In any case snow drift loads could be interpreted as ***pattern loading*** which the Code specifically excludes from being combined with notional horizontal forces.

Thus *Portal Frame* gives you control on whether to include the effects of notional horizontal forces in each design combination. Furthermore, the notional horizontal forces should be applied only in one direction at a time. Except in the simplest of frames it is impossible to determine the most onerous condition for any given design combination. *Portal Frame* allows you to specify in which direction the notional horizontal forces are to act – left-to-right or right-to-left. Once you have specified that a design combination is to include notional horizontal forces and their sense *Portal Frame* adds the appropriate loads for you automatically.

Application of notional horizontal forces

Notional horizontal forces from the vertical rafter loads are applied at the eaves. Any specific axial loads in the stanchions (for example crane loads) are applied at the same position as the original load. Equal and opposite forces are applied at the bases to form a ***closed system*** such that they do not contribute to the total horizontal loading on the frame. Thus, for example, at a split eaves if notional horizontal forces of 0.7 kN and 0.9 kN are applied left-to-right at the lower and upper eaves respectively, then a horizontal load of 1.6 kN right-to-left must be applied at the column base.

In design frame mode the notional horizontal forces are excluded from the analysis model for the approximate minimum weight design solution. Their effect is nearly always relatively small and, as such, should not influence the choice of section size. They are included in the subsequent elastic-plastic analyses.

Serviceability limit state

Deflections at the serviceability limit state can be checked using the results from a linear elastic analysis of the frame. The design combinations that you have specified during input for this purpose are assumed to have suitable load factors and plasticity is not expected at any point within the frame.

You may specify deflection limits for vertical movement at apices or horizontal movement at eaves, either as a proportion of the relevant dimension or as an absolute value.

In order for the checks to be performed, you must have selected at least one design combination for serviceability checking. If the actual deflection exceeds the specified limit at any of the apex or eaves positions, the check is considered to have failed. Re-sizing of members and re-analysis of the frame is ***not*** performed if this occurs.

Fire analysis

The fire analysis performed by ***Portal Frame*** is based on BS 5950: Part 8[7].

The term ***boundary*** is taken as that described and defined in Clause 14.4 and Appendix E of Approved Document B of the ***Building Regulations 2000***[6]. The basic performance requirement of the Building Regulations is that fire should not be able to spread to adjacent properties.

For portal frames in which the external walls and their supporting structure (columns) are required to be fire-resisting due to boundary conditions, the Steel Construction Institute document[5] provides a method for calculating the overturning moment which is applied to the column and the base as a result of the collapse of an unprotected rafter member. ***Portal Frame*** calculates this overturning moment.

You can then check the capacity of the column bases using the ***Fastrak Column Base*** program or by hand and thus demonstrate that the fire-resisting elements remain stable. Consequently, the spread of fire across the boundary would be prevented and the performance requirements of the Building Regulations would be satisfied.

Position of boundaries

Portal Frame assumes that any external column is at a boundary, so that both boundaries are checked for a single-span frame. The left-hand or right-hand or both boundaries can be checked for a multi-span frame.

If only one side of a single-span frame is adjacent to a site boundary, then the results produced by ***Portal Frame*** for the other side are for information only and need not be used for designing the column base see Section 8.2 of the Steel Construction Institute document[5].

Design overturning moment

The overturning moment at any boundary is taken as either the calculated over-turning moment or 10% of the plastic moment of resistance of the column, whichever is the greater.

The Steel Construction Institute document[5] recommends that a column in a monopitch portal frame should be designed for 25% of the plastic moment of resistance (implying that an overturning moment based upon loads from the collapsing rafters need not be calculated). This recommendation would give an abrupt jump in the design moment between a standard portal frame with a small rise and a monopitch, whereas there is likely to be a smooth transition in reality. For these reasons, the recommendation is not implemented in ***Portal Frame***.

Internal supports

Internal supports (e.g. columns) are assumed to remain in place during a fire because of :–

- an adequate level of fire resistance,
- stability being provided by other members.

The Steel Construction Institute document[5] justifies this assumption by stating that the internal column will generally undergo only a partial collapse and this will not significantly increase overturning moments on external columns in most cases.

The Steel Construction Institute document[5] recommends that unprotected props should be ignored and hence the full span from main column to main column be assumed in the calculation of the overturning moment. For a propped portal, ***Portal Frame*** therefore gives a status of 'beyond scope' and does not perform the fire check. If you model the frame using two monopitch portals back-to-back ***Portal Frame*** assumes that the 'prop' is fire protected and hence bases the overturning moment on the shorter span from the boundary (main) column to the internal column (prop).

Frames with spring bases

The Steel Construction Institute document[5] does not cover portal frames with spring bases.

Caution If this condition arises *Portal Frame* will issue a warning and will treat any columns with spring bases as if they had a pinned base when performing the fire analysis. You must take responsibility for this action.

Valley bases

The case where a span contains a valley base at an external column is outside the scope of ***Portal Frame*** and so the fire check will not be performed if requested for such a span.

If a valley base is specified for any internal column in a multi-span frame, the assumptions made in the Steel Construction Institute document[5] might be invalidated and so a warning message is issued by ***Portal Frame***.

Fixed bases

The Steel Construction Institute document[5] states that frames with fixed bases need not be checked for the fire condition. if you specify a boundary at a column having a fixed base, ***Portal Frame*** will issue a warning and will treat the column as if it had a pinned base when performing the fire analysis.

Member stability checks

You can select the stability checks for each member. This will depend upon the state of stress (i.e. elastic or plastic) and the type of restraint that is present (i.e. lateral restraint to inner or outer flange, or torsional restraint). ***Portal Frame*** identifies whether a restraint lies within ***D/2*** of a hinge, where ***D/2*** is a distance equal to half the depth of the rafter or column in which the hinge occurs.

BS 5950-1:2000 clause 5.5.5 indicates that a point of contraflexure may be taken as a torsional restraint. However, you should be aware that the points of contraflexure on a given frame can be subject to a change of position at each hinge formation. The point of contraflexure shown on the ***Member Stability*** screen is that given at the ultimate limit state (1oad factor 1.0).

The checks that can be performed by ***Portal Frame*** are described below.

Clause 5.3.3 check

The Clause 5.3.3 check is a limiting length check based upon the member section properties and the maximum axial load within the checked length. It can be conservative, and, with one exception, must be applied to the segment adjacent to a plastic hinge.

The exception is for the segment that extends into the eaves haunch when a hinge exists at the sharp end. Providing the haunch remains elastic for its entire length, then a 5.3.3 check is not essential and an alternative check can be used (see Clause 5.3.5.1).

You may request the check to be carried out between two specified positions which are restrained either torsionally or laterally (to the compression flange). The maximum axial load between the two points will be used in the check. The length between the restraints can be of uniform or tapered section. The axial stress, if tensile, is set to zero for this check.

For tapered sections, the various terms in the equation given in Clause 5.3.3 are chosen so as to minimise the allowable distance, e.g. the minimum value for radius of gyration in the length between restraints is used.

An allowance is made for moment gradient in a uniform section, but subject to other limiting criteria (see Clause 5.3.3(b)) this can improve the limiting length.

Clause 4.8.3.3.2 check

An overall buckling check on a length between two specified compression flange restraint positions may be carried out in accordance with Clause 4.8.3.3.2(a) of BS 5950-1:2000. Only the second formula for out-of-plane buckling is used since in-plane buckling is taken into account using the methods for sway stability described earlier. The check can be used for uniform sections of any valid classification.

When used in the apex haunch area the ***improvement*** in the section properties provided by the haunch is ignored. That is the section is assumed uniform.

This check also ensures that the length will not undergo lateral torsional buckling.

The axial force, if tensile, is set to zero for the check.

Clause 4.8.3.3.1 check

An overall buckling check on a length between two specified compression flange restraint positions may be carried out in accordance with Clause 4.8.3.3.1(a) of BS 5950-1:2000. Only the second formula for out-of-plane buckling is used since in-plane buckling is taken into account using the methods for sway stability described earlier. The check can be used for tapered sections of any valid classification.

When used in the apex haunch area the ***improvement*** in the section properties provided by the haunch is included. For all tapered sections the value of the uniform moment factor m_{LT} is taken as 1.0.

This check also ensures that the length will not undergo lateral torsional buckling.

The axial force, if tensile, is set to zero for the check. For tapered lengths, the minimum values of radius of gyration and gross area are used in the determination of the compression capacity, even though the values may occur at opposite ends of a length. Sections are considered to be welded (for the purpose of calculating the compressive strength) only if a built-up haunch is present.

Annex B.2.5 of BS 5950-1:2000 does not give any guidance for the calculation of slenderness correction factor when the flange area ratio R_f is less than 0.2. In such a case, *Portal Frame* will issue a warning and terminate the check.

Annex I.1 check

This annex provides alternative calculations to the approach in Clause 4.8.3.3.2 for the overall buckling resistance of a segment. This annex can only be used for doubly symmetric cross-sections in members which are of uniform section and which are plastic (Class 1) or compact (Class 2). In portal frames there are no minor axis moments and so the simplified formulae given in Clause I.1(a) are used. As with the Clause 4.8.3.3.2 check detailed previously only the out-of-plane buckling need be checked.

This check is carried out for a segment between compression flange restraints and as part of an Annex G check.

Note If $\lambda_r \geq 85.8 \times \varepsilon$, then the Annex I.1 check will yield the same results as a Clause 4.8.3.3.2 check.

For this check tensile forces are taken as zero.

Annex G checks

The checks described in Annex G of BS 5950-1:2000 are applicable to a length between torsional restraints which has intermediate lateral restraints to the tension flange. Their implementation within ***Portal Frame*** is described below.

After the Annex G checks have been performed, ***Portal Frame*** also examines all portions of the length that lie between intermediate restraints and applies an appropriate check to each. There must be at lease one intermediate restraint specified.

A typical use for these checks would be with a length which had failed a Clause 4.8.3.3.2 check due to lack of restraint to the compression flange but which was stabilised by purlins connected to the tension (top) flange.

Annex G Plastic Check

Either uniform or tapered members may be checked, using G.3 of BS 5950-1:2000. The checks can be performed even if plasticity does not occur within the un-haunched section of the length under consideration. The axial force, if tensile, is set to zero for either check.

A warning is issued by ***Portal Frame*** if the calculated value of slenderness correction factor n_t or any of its constituent terms ***R*** exceeds 1.0, although the check is not terminated.

It should be noted that the S_x values used in the calculation of ***R*** are determined at each section for haunched members.

The equation for limiting length L_k given in G.3.3.3 of BS 5950-1:2000 contains a discontinuity for certain combinations of yield stress, modulus of elasticity and torsional index ***x*** In order to solve this problem, the value of the bracketed term in the denominator of the equation is limited to a minimum value of ***0.05*** and the value of ***x*** is re-calculated accordingly.

This revised value is also used when calculating ***c*** and avoids the possible discontinuity in the relevant equation. It is important to note that the revised value of ***x*** is used for the Annex G checks only.

Annex G Elastic Check

Either uniform or tapered members can be checked, using G.2 of BS 5950-1:2000. The checks will not be allowed if plasticity occurs within the length under consideration. The axial force, if tensile, is set to zero for either check.

The checks will be terminated if the slenderness correction factor n_t or any of its constituent terms ***R*** exceeds 1.0, as this is beyond the scope of BS 5950-1:2000. It should be noted that the Z_{xc} values used in the calculation of ***R*** are determined at each section for haunched members.

For tapered members, the lateral torsional buckling resistance is calculated using the section modulus at the point under consideration, but with one value of the lateral torsional buckling strength, p_b, for the whole segment.

The equation for the term ***c*** given in G.2.5 of BS 5950-1:2000 contains a discontinuity at a torsional index ***x*** value of 9. Another discontinuity occurs in the equation for limiting length L_k in G.3.3.3. The solution adopted for the latter problem automatically avoids the discontinuity when calculating ***c***.

For tapered sections, the torsional index of the haunch is given in the code as that of the original I-section from which the section forming the haunch is made. The principles embodied in this approach are adopted for haunches built-up from plates. Thus the torsional index of a built-up haunch is calculated assuming an equivalent section ***twice*** the size of the built-up haunch.

Intermediate length checks

Checks between intermediate restraints as part of an Annex G check are automatically carried out. The appropriate check is selected as follows:

- for intermediate lengths of uniform section and which are either class 1 or class 2, an Annex I.1 check is performed,
- for intermediate lengths of uniform section and which are class 3, the formula in Clause 4.8.3.3.2(a) for out-of-plane stability is used,
- for intermediate lengths of tapered section and which are class 1, class 2 or class 3, the second formula in Clause 4.8.3.3.1 is used,
- for intermediate lengths which are of either uniform or tapered section, but which are adjacent to a plastic hinge a Clause 5.3.3 check is used unless the particular intermediate length is wholly or partially in the haunch and the whole of the haunch remains elastic, in which case a Clause 4.8.3.3.1 (second formula only) check is performed.

Ties

Tie members can be introduced into portal frames to achieve three effects:

- to control deflections - spread at eaves,
- to reduce section sizes,
- a combination of I and 2.

Within ***Portal Frame*** ties are assumed (and in most cases constrained) to be horizontal. They are given pinned ends such that they attract no moment.

The introduction of even weak ties has a significant effect on the performance of the frame. This, along with several unusual design considerations, means that care should be exercised when using ties particularly to reduce section sizes.

Portal frames without ties have sagging moments in the top portion of the rafter when subject to gravity loads. Tied portals, on the other hand, can have hogging moments at the apex and sagging moments in the central portion of the rafter. This can cause several effects:

- large axial loads are created in the rafter due to the tying action. This can have a significant destabilizing effect on the frame. This will be reflected in a large reduction in the (first order) collapse load factor when checking frame stability to the Steel Construction Publication P292,
- the connection design moment for the apex can be the reverse of that normally expected.
- the maximum deflection of the rafter can occur at a significant distance away from the apex; up to the mid-length position of the un-haunched portion of the rafter. The deflection at the mid-length position can be checked from the ***Serviceability*** page of the ***Frame Design Summary Property Sheet***.

Performance of yielding ties

A Yielding Tie is likely to be in reality a light member such as a tube, a rod or even a wire which is deemed to have no strength in compression (over the sort of length required to tie portal frames). During the elastic-plastic analysis not only can this type of tie sustain elastic strains but, when the force in the member reaches its capacity or that of its connection it undergoes plastic strain i.e. it yields. Furthermore, since it has no compression capacity, if the member is predicted to have a compressive force at some stage during the analysis then it ***buckles*** (capacity set to zero) and plays no further part in the behaviour of the frame. Unless, at a later stage the hinge formations and deformation of the structure make the force in the tie become tension again in which case it recovers its full capacity (and could, later still, yield in tension or buckle again). This is only possible with an elastic-plastic analysis approach as used by ***Portal Frame***.

If the tie yields this will be reported in the hinge history and indicated on the hinge history graphics in the same manner as a true hinge. If the tie goes into compression and buckles then this also is indicated on the hinge history. A warning to this effect is also included in the design results.

Buckling of ties is treated on a loadcase by loadcase basis i.e. a tie which has buckled in one loadcase is inserted at its full capacity in other loadcases (although subsequently may buckle or yield in one or more of those loadcases). It is essential to treat compressive forces in Yielding Ties in this manner as they are deemed to be ***tension only*** members. One potential drawback is that suddenly setting its capacity to zero (buckling) may cause a significant drop in stiffness of the frame and consequently the analysis could become ill- conditioned. This could result in a valid collapse not being found. If this occurs then you can increase or decrease the strength or area of the tie to force the frame through a slightly different hinge history. Alternatively, change the Yielding Tie to a Tie/ Strut and allow for the resulting compression force in your design.

Note For the conditions under which tie members are treated as either Yielding Ties or Tie/Struts *see* "Analysis" on page 879.

Performance of tie/struts

Unlike Yielding Ties a Tie/Strut is an elastic member which can sustain both tension or compression. Thus there is no concept of the tie yielding during the elastic-plastic analysis; hence there is no requirement to specify a capacity at the input stage. The area of the Tie/Strut is of course required to contribute to the elastic stiffness of the frame.

Note For the conditions under which tie members are treated as either Yielding Ties or Tie/Struts *see* "Analysis" on page 879.

Analysis

There are various analyses carried out when using ***Portal Frame***. There follows a description of how each of these treat tie members.

(Approximate) minimum weight design

This analysis mode is used to determine initial section sizes when in Automatic Design mode. Tie members (Yielding Ties and Tie/Struts) are not included within this model since the program can not determine whether you wish to introduce a tie to reduce section sizes or to control deflections. Hence the subsequent elastic-plastic analysis of Ultimate Limit State Design Combinations will result in load factors significantly greater than 1.0. Subsequent elastic analyses for the Serviceability Limit State will show whether this has achieved the level of deflection control you were seeking. On the other hand if you wish to reduce your section sizes then you will have to change to Manual mode and select the sections you desire.

It is worth noting that since the tie is not included in the Minimum Weight Design the axial force in the rafter will be higher in the subsequent analyses. This may be sufficient to either alter the classification of the section or to fail it due to the interaction of axial load and moment. In this case you may find it useful to alter the Axial Load Factor on the Analysis Attributes screen to something slightly larger than 1.25, say, to 1.3.

Initial elastic analysis

This analysis mode is used to establish initials values for the axial load distribution based on the sections you have specified when in Manual Design mode. These axial loads are then enhanced by the Axial Load Factor on the ***Controls*** page of the ***Design Wizard*** to arrive at values for the Reduced Plastic Moment of Resistance, M_{pr}. The elastic analysis is carried out at ULS load factors and hence the force in the tie is likely to be greater than that at load factor 1.0 from the subsequent elastic-plastic analysis. Consequently the axial force in the rafter will be lower in the elastic-plastic analysis at load factor 1.0 than in the initial elastic analysis. This may be sufficient to cause an unnecessary level of conservatism in the design at ULS for interaction of axial load and moment. In this case you may find it useful to alter the Axial Load Factor on the Analysis Attributes screen to something slightly smaller than 1.25, say, to 1.2. Since this is an elastic analysis there can be no concept of tie members yielding. Therefore both Yielding Ties and Tie/Struts are entered into the analysis model as Tie/Struts.

Elastic-plastic analysis

This is the type used for the analysis of Ultimate Limit State Design Combinations. This the only analysis mode in which tie members can be treated as Yielding Ties. Obviously if you have specified the tie member to be a Tie/Strut then it is treated as such i.e. no yielding of the tie takes place.

Elastic analysis

This is the type used for the analysis of Serviceability Limit State Design Combinations and the Notional Sway Combinations. Since this an elastic analysis there can be no concept of tie members yielding. Therefore both Yielding Ties and Tie/Struts are entered into the analysis model as Tie/Struts.

Design

The program does not design the tie members but simply reports in the results the force in the tie and its elongation. It is up to the designer to provide the necessary calculations to justify the performance of the tie. For Yielding Ties which yield prior to ULS (L/F 1.0) the force in the tie will be equal to its capacity. This is perfectly acceptable providing you judge that the total strains (elastic and plastic) are within acceptable limits bearing in mind that if the connection is the weak link then in general these can not sustain as large an elongation as the tie member itself. Some other points you may need to consider in your design are as follows:

- for Yielding Ties, there is a possibility that the tie force at Serviceability Limit State is greater than its capacity - the program will warn you if this is the case. This infers plastic strains at working loads which in itself for tie members is not unacceptable but does infer that the analysis should have proceeded in a different manner once the tie had yielded. Since all SLS Design Combinations are subject to elastic analysis only, then the correct load response history can not be determined and the elastic deflections will be incorrect to some degree.
- you need to decide whether the area you enter during input is the gross area or net area allowing for holes.
- the capacity required for yielding ties can be that of the connection or the tie member itself. Obviously whichever type of tie is specified both the tie and its connection need to be checked for the resulting force. Bear in mind the comment at the start of this section with regard to the strain capacity of connections.
- Tie/Struts which go into compression will need to be checked for major and minor axis strut buckling depending upon the position and direction of any intermediate restraints. Compression in the Tie/Strut may only occur due to wind loads.

- there can be no applied loading to tie members in the program and with pinned ends no induced moments. Hence there are only self weight bending moments which, depending upon the size, weight and span of the tie member, might be ignored.

Yielding ties

It is important to note that, for Yielding Ties, the state of the tie at various stages during the elastic-plastic analysis process can be included in the Hinge History. If the Yielding Tie has taken part in the formation of the collapse mechanism then it will appear in the table (and associated graphical displays) of hinge history with one of three states:

- Yielded - the force in the tie has reach its capacity and it will strain plastically during any further hinge formations
- Buckled - at one step in the analysis the tie has been detected as going into compression and hence has been allowed to buckle (capacity set to zero) prior to making that step.
- Reset:
  - either the force in a tie which has Yielded has dropped below its capacity and is therefore acting elastically again,
  - or the force in a tie which has Buckled (gone into compression) has reversed and is now tension again. (The capacity of the Yielding Tie will be reset to its original full value.)

Floors

Although you can define floors in your structure using the ***Portal Modeller***, these floors are currently not considered in the design of the frame.

References

1. British Standards Institution. ***BS 5950-1:2000: Structural use of steelwork in building; Part 1: Code of practice for design in simple and continuous construction: hot rolled sections***. BSI, 2001.

2. Steel Construction Institute. ***Guide to BS 5950-1:2000. Volume 1, Section properties, Member capacities***. SCI, 2001.

3. Davies, J.M. 'False mechanisms in elastic-plastic analysis' in ***The Structural Engineer. Volume 66, Number 16, 268***. The Institution of Structural Engineers, 16th August 1988.

4. Davies, J.M. ***Approximate minimum weight design of steel frames. Proceedings of the International Symposium on Computer-Aided Structural Design***, University of Warwick, July 1972: Peter Peregrinus, 1973.

5. Newman G M; ***The Behaviour of Steel Portal Frames in Fire Boundary Conditions***; SCI.

6. Department of the Environment. ***The Building Regulations 2000***. HMSO, 2000.

7. BS 5950: Part 8: ***Code of Practice for the Design of Fire Protection for Structural Steelwork***: 1990; BSI

Bibliography

- Morris, L.J., and Randall, Al. ***Plastic Design***. Constrado and British Constructional Steelwork Association, 1979.
- Davies, J.M. ***Frame instability and strain hardening in plastic theory***. Journal of the Structural Division; Proceedings of the American Society of Civil Engineers. Volume 92, Number ST3, 1–15. ASCE, June 1966.
- Davies, J.M. ***Collapse and shakedown loads of plane frames***. Journal of the Structural Division; Proceedings of the American Society of Civil Engineers. Volume 93, Number ST3, 35–50. ASCE, June 1967.

- Davies, J.M. ***A new formulation of the plastic design problem for plane frames***. International Journal for Numerical Methods in Engineering. Volume 5, 185–192. John Wiley & Sons, 1972.
- Davies, J.M. ***The contribution of cladding and second-order effects***. Proceedings of the One Day Symposium on Plastic Design of Steel Structures, Garforth, Leeds, 26th October 1982. Institution of Structural Engineers and British Constructional Steelwork Association.
- Woolcock, S.T. and Kitipomchai, S. 'Deflection limits for portal frames' in ***Steel Construction***. Volume 20, Number 3, 2–10. Australian Institute of Steel Construction, August 1986.
- Davies, Professor J M; ***In-plane stability in Portal Frames***; The Structural Engineer (Volume 68, No. 8) 17th April 1990.
- Davies J M and Brown B A; Plastic Design to BS 5950; SCI.

4 Wind Load Generator[1]

The ***Fastrak Portal Frame Plus Wind Load Generator*** allows you to calculate the wind loading applied to your building either in accordance with ***CP3 : Chapter V : Part 2 : September 1972*** or ***BS 6399 : Part 2 : 1997***.

The following text indicates any limitations of the ***Wind Load Generator***, and particular interpretations of the codes that have been used in its implementation.

BS 6399 : Part 2 : 1997

Scope - The current release of the ***Wind Load Generator*** allows you to calculate wind loads in accordance with the standard method given in BS 6399: Part 2: 1997. You can also use the hybrid method (returning at 3.4.2) to calculate the directional effective wind speed.

As yet you cannot use the ***Wind Load Generator*** to calculate and use directional pressure coefficients.

Standard effective wind speed

Dynamic Augmentation Factor/Overall Loads - The requirements of BS 6399: Part 2: 1997 clause 2.1.3.6 specifically refer to horizontal loading applied to the entire building.

The ***Wind Load Generator*** deals with all loads (horizontal and vertical) that apply to a single frame.

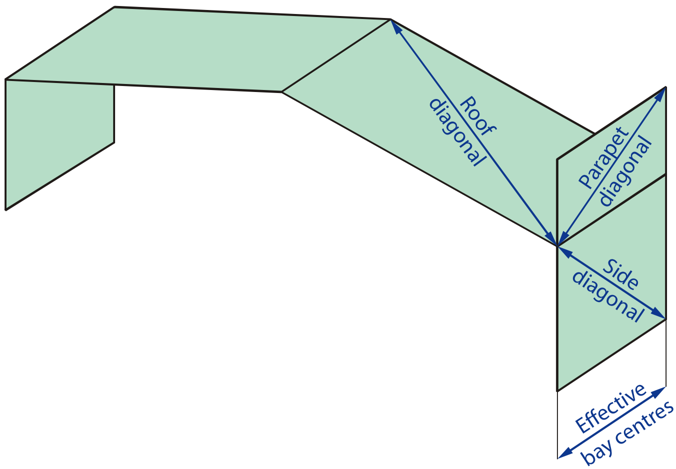
As a consequence the reductions that apply to the building loads would not appear to apply to a single frame. This being the case there is no need to calculate the Dynamic Augmentation Factor since it is only used in clause 2.1.3.6.

1. *This is an additional plug-in module that you purchase separately to **Portal Frame**.*

The ***Wind Load Generator*** allows you to calculate the loading for the wind blowing in orthogonal directions on the frame. You can, if engineering judgement warrants it, model the maximum stresses on a corner column as the sum or 80% of the loads arising from each orthogonal case. To do this you will either have to add additional loads into one or other loadcase, or include both load cases and modify the factors used in the design combination.

Asymmetric loads - The ***Wind Load Generator*** allows you to easily consider the effects of asymmetric loads. When the appropriate pressure coefficients have been calculated, you change the percentage of load applied to any one member from the default of 100% to the reduced value of 60% stipulated by the code.

Diagonal of loaded areas - For portal structures the design should be considered on a frame by frame basis, rather than for the entire building. For external pressure coefficients the ***Wind Load Generator*** uses the loaded diagonals for side wind shown in the figure below.



For the internal pressure coefficients the loaded diagonal is determined from the volume of the storey as detailed in clause 2.6.1. This information is not available for the *Wind Load Generator*, and depends on many factors. A value of unity is therefore defaulted. You can calculate an alternative value and enter it directly if you so desire.

Basic wind speed **-** The basic wind speed for any location can be taken directly from the map shown in Figure 6. The map shows a series of major towns, for your convenience these towns are given in a list. When you select one of these towns the basic wind speed appropriate to that town will be returned automatically for you.

Altitude factor, S_a **-** The *Wind Load Generator* takes account of the level of the site based on the **Altitude** that you specify in the *Building Definition*. The calculated factor is based on the condition where topography is not considered significant. If topography is significant for your site, then you will need to calculate the appropriate factor and enter it directly.

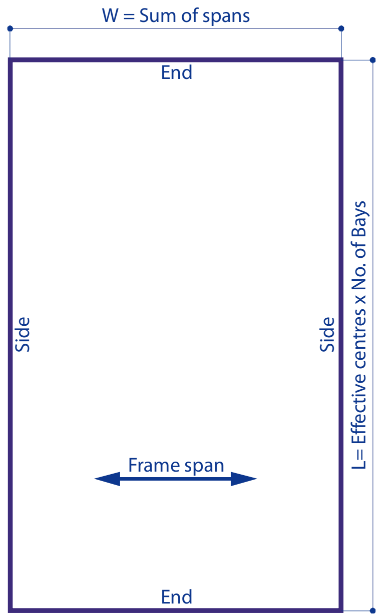
Direction factor, S_d **-** If you do not pick the option to **Apply Sd factors** then a value of unity will be used as stipulated in the code.

If you do choose to **Apply Sd factors** then the *Wind Load Generator* uses a supplemented version of Table 3 for the calculation of S_d. This supplemented version has values of S_d for every 5° round the compass. When you specify an orientation the *Wind Load Generator* looks in the table for values of S_d in 5° increments within the range ±45° of the direction normal to the face that is facing the wind and uses the most onerous value.

Seasonal factor, S_s **-** A seasonal factor of unity is used by the *Wind Load Generator*. If you are checking a condition which only occurs during construction, then you might want to take advantage of the reduced factors given in Annex D; Table D1 entering this directly.

***Probability factor, S_p* -** Again a factor of unity is used. If you want to change this, then you should enter the value directly.

***Building width* -** The *Wind Load Generator* always takes the width of the building parallel to the direction of span of the frames. Conversely the length of the building is taken as the dimension perpendicular to this i.e. in the direction of the frame bay centres.



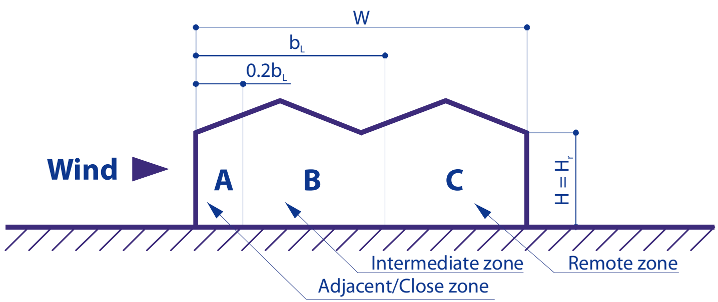
Note The *Wind Load Generator* always assumes that frames span left to right as shown above irrespective of the overall dimensions of the building.

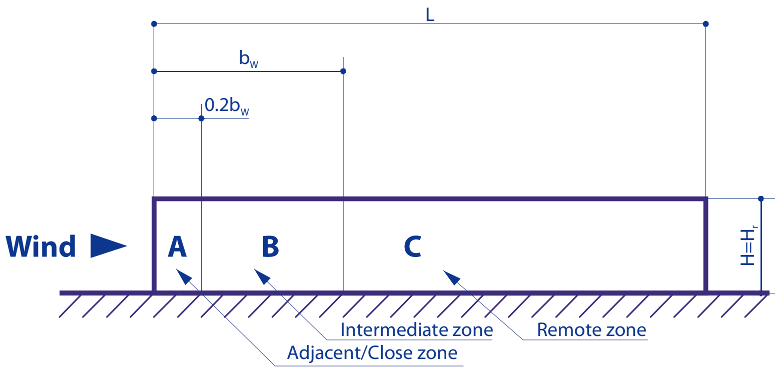
Height of building - For the walls of the building and for flat roofs only the *Wind Load Generator* uses the height of each wall or wall plus parapet (if a parapet exists) in the calculation of the external pressure coefficients.

For other roofs the *Wind Load Generator* takes the height of the building as the height of the highest eaves or apex in the current frame. This height excludes any parapets that have been defined for the building. If your building has parapets whose tops are higher than the height determined by the *Wind Load Generator*, then you will need to use engineering judgement and increase the height of your building if you feel that this is necessary.

The height defined above is used in conjunction with the length or width of the building (depending on the wind direction) to determine the extent of the various roof pressure zones.

Pressure coefficients for the walls of rectangular clad buildings - The figures below show you the location of the various zones of wind pressure when the wind is blowing on the sides and ends of the building.



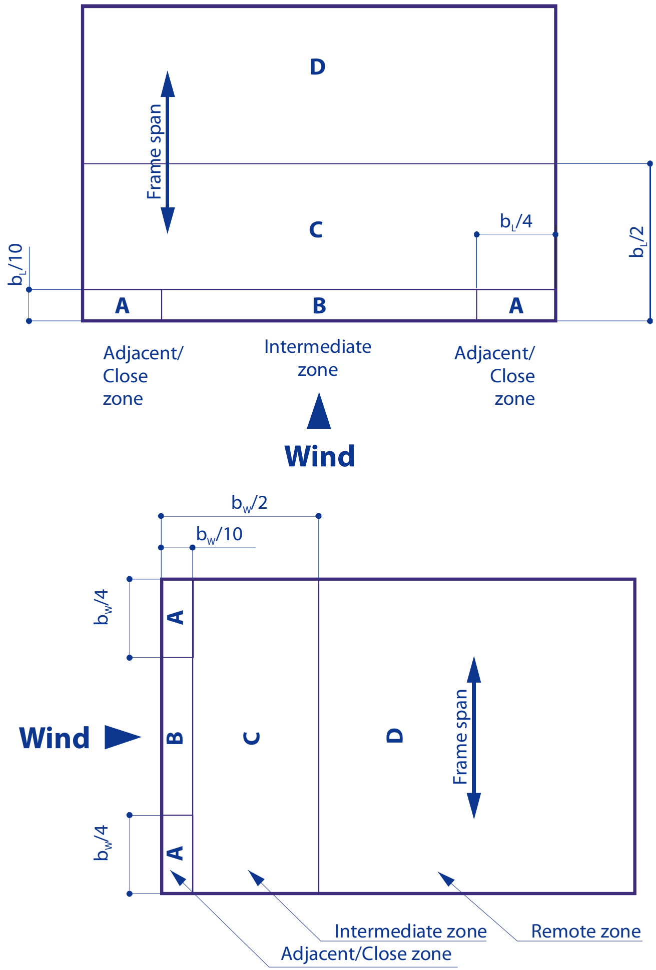


Caution Short buildings may not have sufficient wind depth for all the zones indicated above to exist. It is your responsibility to ensure that the correct zones are included in your design.

Note The extent of the various zones for the walls of the building may well be different from the extent of the zones for the roof.

Note The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

Pressure zones for flat roofs **-** The coefficients and the zones where they apply are detailed in the figures below.



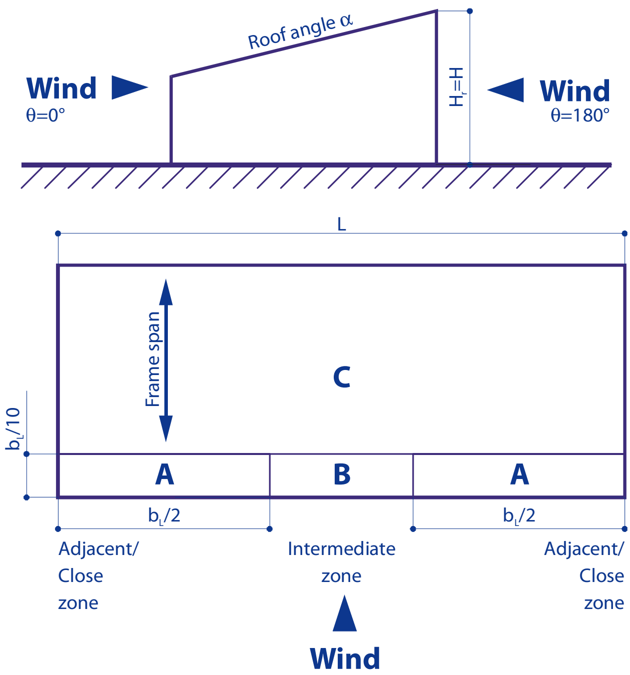
Caution Short buildings may not have sufficient wind depth for all the zones indicated above to exist. It is your responsibility to ensure that the correct zones are included in your design.

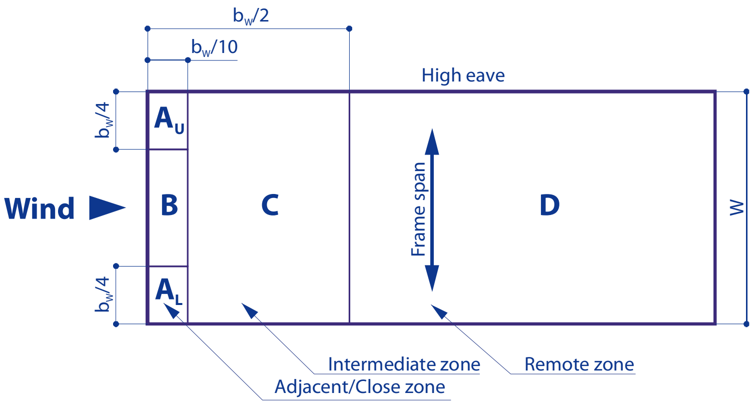
Note Monopitch and duopitch roofs which have pitches in the range -5° to +5° are considered to be flat and their external pressure coefficients are taken from Table 8. In all other cases the values for the external pressure coefficients are taken from the table appropriate to type of roof. The option to compare suction coefficients with those from the flat roof table and then use the least negative value (see Table 8 Note 5) is not implemented.

Note The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

Pressure zones for monopitch roofs of rectangular clad buildings - If the height to width ratio of your building does not comply with the constraints imposed for this table, then zero values will be returned for the ***Cpe*** values. You will then need to determine appropriate coefficients and enter them before you attempt to perform a design. If you leave the zero values unchanged, then this will generate an invalid loadcase error and prevent the design of the frame.

The coefficients and the zones for monopitch roofs are detailed in the figures below.





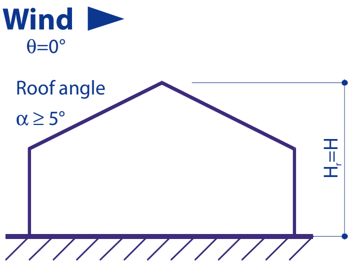
Caution Short buildings may not have sufficient wind depth for all the zones indicated above to exist. It is your responsibility to ensure that the correct zones are included in your design.

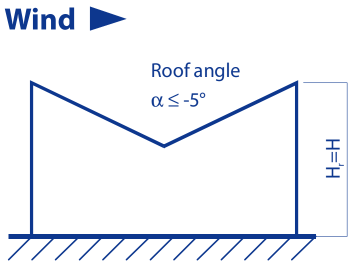
Caution The *Wind Load Generator* always treats monopitches as such. For example, if you define a frame with two equal monopitches back to back then you will get the results for two monopitches and not those for a duopitch portal. (In this case a propped portal will give the results for the portal rafters treated as such, rather than monopitches and will use one span less).

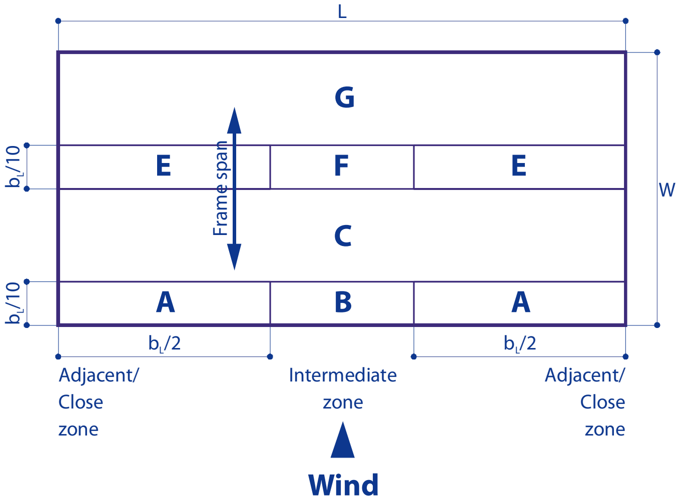
Note Monopitch roofs which have pitches in the range -5° to +5° are considered to be flat and their external pressure coefficients are taken from Table 8. In all other cases the values for the external pressure coefficients are taken from table 9. The option to compare suction coefficients with those from table 8 (flat roofs) and then use the least negative value (see Table 8 Note 5) is not implemented.

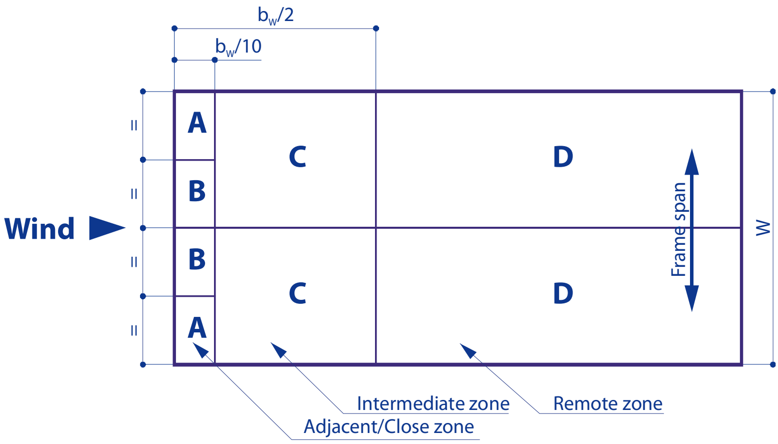
Note The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

Pressure zones for duopitch roofs - The coefficients and the zones for duopitch roofs are detailed in the figures below.






L
G
E
F
E
C
A
B
A
Frame span
W
$b_L/10$
$b_L/10$
$b_L/2$
$b_L/2$
Adjacent/ Close zone
Intermediate zone
Adjacent/ Close zone
Wind

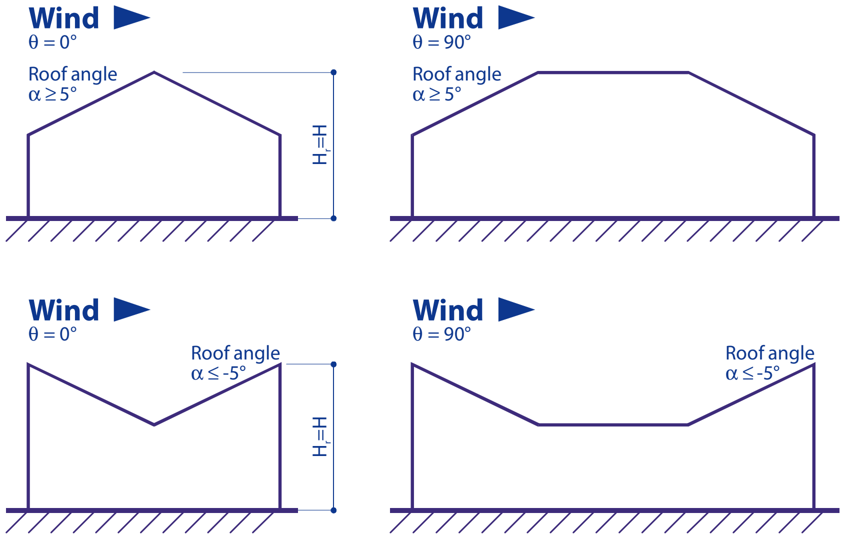

$b_W/2$
$b_W/10$
A
B
B
A
C
C
D
D
Frame span
W
Wind
Intermediate zone
Remote zone
Adjacent/Close zone

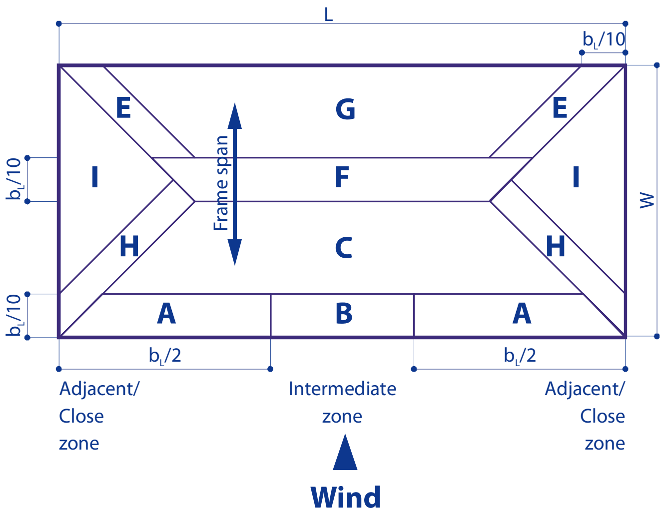
Caution Short buildings may not have sufficient wind depth for all the zones indicated above to exist. It is your responsibility to ensure that the correct zones are included in your design.

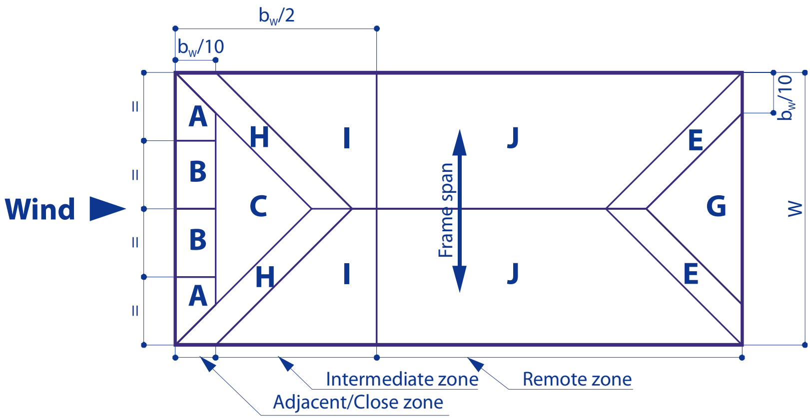
Note Duopitch roofs which have pitches in the range -5° to +5° are considered to be flat and their external pressure coefficients are taken from Table 8. In all other cases the values for the external pressure coefficients are taken from table 10. The option to compare suction coefficients with those from table 8 (flat roofs) and then use the least negative value (see Table 8 Note 5) is not implemented.

Note The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

Pressure zones for hipped/flat top roofs - The coefficients and the zones for hipped and flat top roofs are detailed in the figures below.







Caution *Short buildings may not have sufficient wind depth for all the zones indicated above to exist. It is your responsibility to ensure that the correct zones are included in your design.*

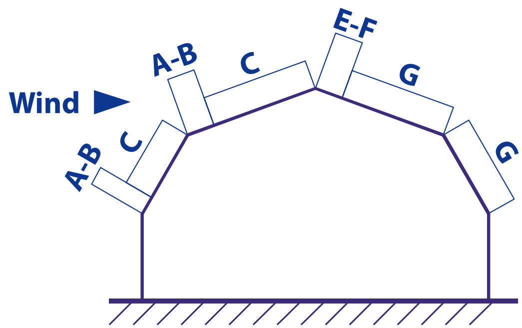
Note

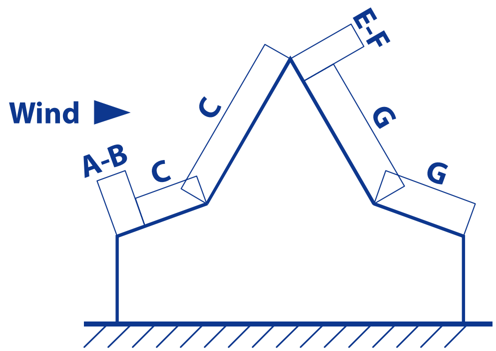
Hipped roofs which have pitches in the range -5° to +5° are considered to be flat and their external pressure coefficients are taken from Table 8. In all other cases the values for the external pressure coefficients are taken from table 11. The option to compare suction coefficients with those from table 8 (flat roofs) and then use the least negative value (see Table 8 Note 5) is not implemented.

Note

The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

Pressure zones for Mansard portals - For these cases a maximum of two coefficients are returned for each slope with the appropriate lengths for the zone. Both cases shown below are allowed.

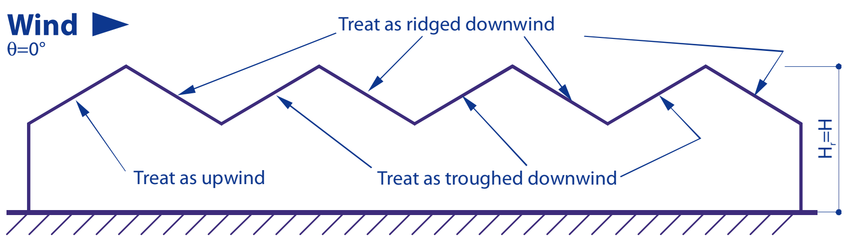


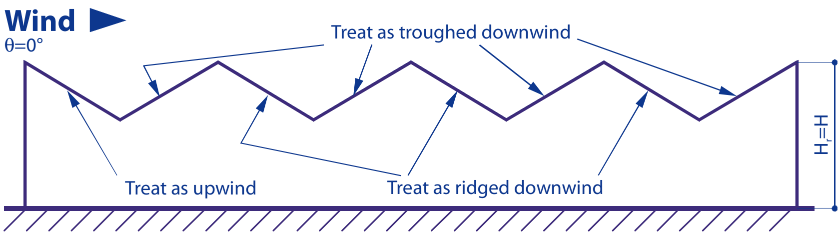


In this case the requirement (for duopitch roofs) that the upwind and downwind pitch angles are within 5° of each other is ignored.

Note The *Wind Load Generator* only gives the loads that are applied to the zone that you specify. If a particular frame carries only partial loads from a zone, or loads from more than one zone, then you will have to calculate and enter the details yourself.

***Pressure zones for Multi-bay portals* -** The **Wind Load Generator** handles multi-bay portals by considering the repeat roof geometry of the building (rather than the repeat span geometry. Typical examples are shown below.





Caution For multi-bay portals where the roof pitch for any span lies in the range -5° < α < 5° the code would allow the entire roof to be considered as a flat roof. This option is not considered by the *Wind Load Generator*. If you want to model the building in this way you would need to calculate and enter the appropriate values directly.

Wind Loads on internal columns **-** Wind loads are only applied to the external portions of columns by the ***Wind Load Generator***, however you can add further loads by selecting *Frame/Loading…* and then editing the Wind loadcases as appropriate.

Parapets **-** The ***Wind Load Generator*** uses 1.2 for the net pressure coefficients for all parapets. It is felt that this value is used because a solidity factor of 0.8 is considered to be appropriate for portal construction.

These coefficients are used for both windward and leeward parapets. For wind blowing on the gables a net suction coefficient of -1.2 is used.

Canopies **-** These are not catered for by this release of the ***Wind Load Generator***. If you try to model canopies using monopitches, then the values of ***Cpe*** that are generated will be incorrect as they will be taken from the table which relates to monopitches rather than that which relates to canopies.

Directional effective wind speed

The calculations for the directional effective wind speeds are performed identical to those for the standard effective wind speed except that the effective wind speed is calculated in accordance with clause 3.2.3.

Topographic increment, S_h **-** This increment depends on details of your site that are not available to the *Wind Load Generator*. A ***non-conservative*** default value of zero is used, you must calculate and enter an alternate value directly.

CP3 : Chapter V : Part 2 : September 1972

Building width **-** The *Wind Load Generator* always takes the width of the building parallel to the direction of span of the frames. Conversely the length of the building is taken as the dimension perpendicular to this i.e. in the direction of the frame bay centres.

Height of building **-** The *Wind Load Generator* takes the height of the building as that of the highest eaves or parapet in the current frame.

This building height is used to determine the **Ground roughness, building size and height above ground, factor S2** and also to determine the building shape ratio **h/w** used in tables 7, 8 and 9.

Table 7. Pressure coefficients for the walls of rectangular clad buildings **-** The *Wind Load Generator* allows a building which is square on plan (*l/w = 1*) even though this is slightly beyond a strict interpretation of the conditions allowed by the table.

Pressure coefficients C_{pe} for monopitch roofs of rectangular clad buildings with h/w < 2 **-** If the height to width ratio of your building does not comply with the constraints imposed for this table, then zero values will be returned for the *Cpe* values. You will then need to determine appropriate coefficients and enter them before you attempt to perform a design. If you leave the zero values unchanged, then this will generate an invalid loadcase error and prevent the design of the frame.

Caution The *Wind Load Generator* always treats monopitches as such. For example, if you define a frame with two equal monopitches back to back then you will get the results for two monopitches and not those for a duopitch portal. (In this case a propped portal will give the results for the portal rafters treated as such, rather than monopitches and will use one span less).

Local coefficients - For tables 7, 8 and 9 the ***Wind Load Generator*** only determines the overall pressure coefficients and not the local ones. For tables 7 and 8 only one coefficient is returned per slope. For table 9 two coefficients are returned, one for each half of the slope.

Flat top and Mansard portals - For these cases two coefficients are returned, one for each half of each slope. The assumption is made that a Mansard portal behaves as a series of monopitches.

Frame Location - If the wind is blowing on the end of the structure, then you can specify where the frame is in relation to the windward end of the structure. Pick **Adjacent/Close Zone** if the higher coefficients are to be used where appropriate, alternatively pick **Intermediate Zone** if the lower coefficients apply to this frame.

Wind Loads on internal columns - Wind loads are only applied to the external portions of columns by the ***Wind Load Generator***, however you can add further loads by selecting *Frame/Loading…* and then editing the Wind loadcases as appropriate.

***Canopies* -** These are not catered for by this release of the *Wind Load Generator*. If you try to model canopies using monopitches, then the values of *Cpe* that are generated will be incorrect as they will be taken from Table 9 which relates to monopitches and not from Table 13 which relates to canopies.

Limitations

When the roof pitch for the windward rafter is 30°, the value that is returned from the data table for the external pressure coefficient *Cpe* is zero. However the same value is also returned when the data table contains no information for a particular condition. Therefore the *Wind Load Generator* has been configured to flag a zero value as invalid (the line for that pressure coefficient on the screen is denoted with red text). For the above case therefore, you must adjust the value of *Cpe* slightly (so that it is no longer zero e.g. *0.001*).

When dealing with an asymmetric portal where the right hand rafter continues to rise from the apex to the right hand eaves (or the mirror image of this), then the external pressure coefficient *Cpe* for the right hand rafter is returned as zero (left hand rafter for the mirror image case). Again these are treated as invalid by the *Wind Load Generator* (as indeed they are). You will need to calculate and enter your own value directly.

5 Snow Load Generator[1]

The ***Fastrak Portal Frame Plus Snow Load Generator*** allows you to calculate the snow loading applied to your building in accordance with ***BS 6399 : Part 2 : 1997***.

The following text indicates any limitations of the ***Snow Load Generator***, and particular interpretations of the code that have been used in its implementation.

BS 6399: Part 2: 1997

Site Snow Load - The site snow load is always calculated using the equation:

$$s_o = s_b + s_{alt} \cdot ((A - 100)/100)$$

where s_{alt} is taken from Table 1 of BS 6399 : Part 2 : 1997. This approach is allowed by the code and gives reductions in the site snow load for altitudes less than 100 m.

If your site is at an altitude of more than 500m, then you cannot use the ***Snow Load Generator*** to calculate the loading that applies to that frame. To ensure this you will be prevented from entering an altitude greater than 500m in your ***Building Definition***.

Roof shapes - The ***Snow Load Generator*** deals with all the shapes of portal that you can define using ***Portal Frame***. whether or not they include parapets, valleys or steps along the length of the frame.

1. *This is an additional plug-in module that you purchase separately to **Portal Frame**.*

The ***Snow Load Generator*** only deals with geometry in the plane of the frame that you are defining. It does not consider any effects resulting from any other geometry, be they changes in height along the length of the building, changes in direction of the structure, additional features behind which snow can drift etc.

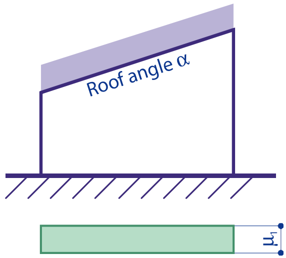
If such features do affect your structure, then you can use the ***Snow Load Generator*** to generate the basic snow loading for your frame, and then modify the snow coefficients and/or add new loads that you have calculated yourself to model such effects.

The ***Snow Load Generator*** considers only natural patterns of snow fall and redistribution. If manual or mechanical methods of snow removal are used, then you will need to calculate and enter your own details for the snow load directly, rather than using the ***Snow Load Generator***.

Note If you choose a snow load condition that involves a redistribution of the snow by drifting, then the *Snow Load Generator* automatically includes the partial safety factor of 1.05 stipulated by the code.

The following figures indicate the nomenclature that is used, and the types of snow load that you can define.

Monopitch portals - The ***Snow Load Generator*** treats flat top and monopitch portals identically the figure below indicates the pattern of snow load that is covered.

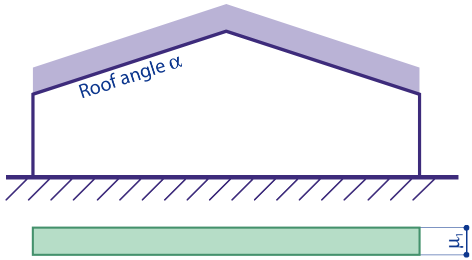


The amount of snow that gathers depends on the slope of the roof. The snow load coefficients are calculated using the formulae below:

Roof pitch	*Shape coefficient*
$0° \leq \alpha \leq 30°$	$\mu_1 = 0.8$
$30° < \alpha < 60°$	$\mu_1 = 0.8 \cdot \left[\frac{60 - \alpha}{30}\right]$
$60° \leq \alpha$	$\mu_1 = 0$

Caution The *Snow Load Generator* always treats monopitches as such. For example, if you define a frame with two equal monopitches back to back then you will get the results for two monopitches and not those for a duopitch portal. (In this case a propped portal will give the results for the portal rafters treated as such, rather than monopitches and will use one span less).

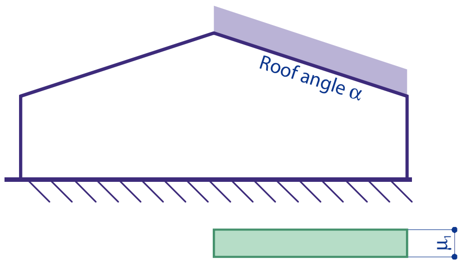
Duopitch portals with symmetric load - The **Snow Load Generator** allows you to define both symmetric and asymmetric snow patterns on pitched roofs where the pitch is greater than 15°. If the pitch is less than this, then only the symmetric snow pattern is allowed as indicated in the figure below.

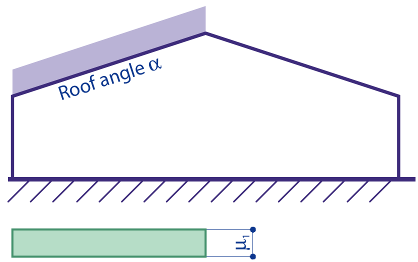


The amount of snow that gathers depends on the slope of the roof. The snow load coefficients are calculated using the formulae below:

Roof pitch	Shape coefficient
$0° \leq \alpha \leq 30°$	$\mu_1 = 0.8$
$30° < \alpha < 60°$	$\mu_1 = 0.8 \cdot \left[\frac{60 - \alpha}{30}\right]$
$60° \leq \alpha$	$\mu_1 = 0$

Duopitch portals with asymmetric load - The ***Snow Load Generator*** allows you to define asymmetric snow patterns on pitched roofs where the pitch is greater than 15°. These patterns are referred to as **Redistributed Left to Right** and **Redistributed Right to Left** since the snow is blown from one slope to lie on the other. The two patterns are shown in the figures below.



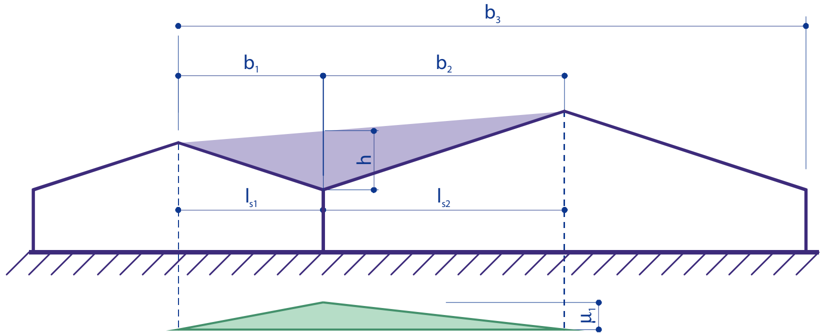


The amount of snow that gathers depends on the slope of the roof. The snow load coefficients are calculated using the formulae below:

Roof pitch	*Shape coefficient*
$0° \le \alpha \le 15°$	$\mu_1 = 0$

Roof pitch	*Shape coefficient*
$15° < \alpha < 30°$	$\mu_1 = 0.8 + 0.4 \cdot \left[\frac{\alpha - 15}{15}\right]$
$30° < \alpha < 60°$	$\mu_1 = 1.2 \cdot \left[\frac{60 - \alpha}{30}\right]$
$60° \leq \alpha$	$\mu_1 = 0$

Valley snow **-** The **Snow Load Generator** automatically deals with valley snow conditions in accordance with the flowchart given in Figure 5 of BS 6399: Part 3: 1988. The nomenclature for this is shown below for reference.

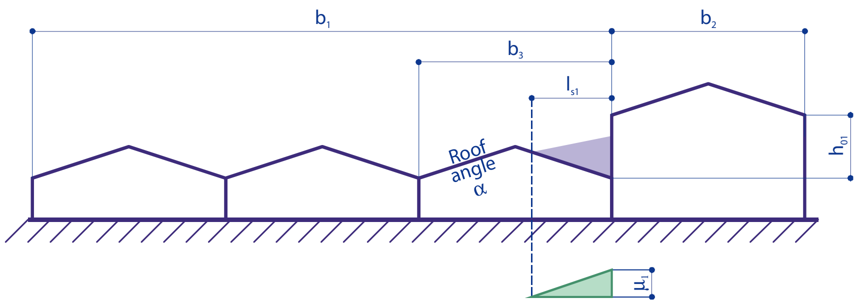


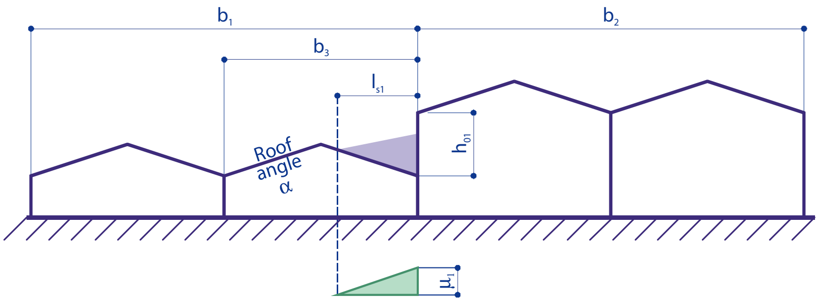
Note The following comments are pertinent to the calculation of valley loads:

- The calculation of b_3 is calculated as the span of the frame with the highest apex plus the distance to the apex of the adjacent span.

- If the spans are symmetrical, then the option to calculate b_3 as 1.5 x frame span is not implemented.
- The *Snow Load Generator* calculates valley drift loads at all valleys in the frame. If you decide that this is not justified for the geometry of a particular building, then you can edit the loadcase through the *Loadcase* dialog and remove any unnecessary loads.
- The valley drift length for a Mansard or flat top portal is limited to the end of the first slope out of the valley.
- If the step at a valley is greater than 1.0m then no valley drift will occur at that valley, instead you should use the **Step** option to determine the step drift load at that location.

Step snow* -** The ***Snow Load Generator automatically deals with snow drifting at locations where there is an abrupt change of roof height, in the plane of the portal span. The calculations are in accordance with the flowchart given in Figure 6 of BS 6399: Part 3: 1988. The nomenclature for this is shown below for reference.



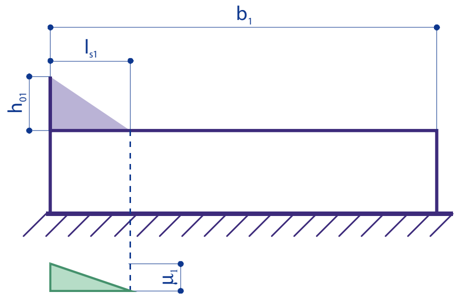


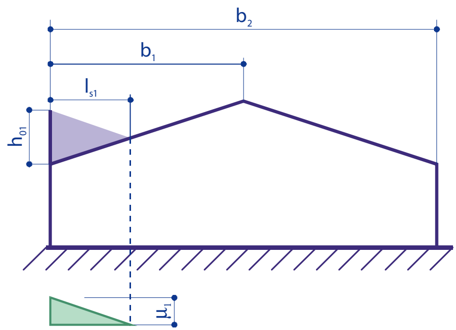
Note The following comments are appropriate for Step drift loads:

- If the step at a particular location is less than 1.0m, then a step drift load will not be calculated, a valley drift load should be used instead.
- If a step load is calculated, then no loading is considered on the upper rafter. If, in your judgement, such loading would arise you would need to calculate the appropriate details and enter the loading directly through the normal *Loading* dialog, rather than by using the *Snow Load Generator*.
- The *Snow Load Generator* does not allow the length of snow l_{s1} to extend beyond the apex of a span. If such a condition would occur, then you would need to calculate the appropriate details and enter the loading directly through the normal *Loading* dialog, rather than by using the *Snow Load Generator*.
- The condition limiting l_{s1} to $b_3/2$ when $b_1 = b_3$ and the roof slope is greater than 60° is not implemented within the *Snow Load Generator*. If such a condition would occur, then you would need to calculate the appropriate details

and enter the loading directly through the normal *Loading* dialog, rather than by using the *Snow Load Generator*.

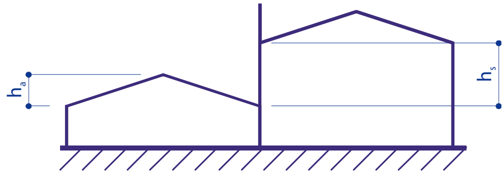
Parapet snow - The ***Snow Load Generator*** will calculate the snow load shape coefficients for the conditions shown in the following diagrams. The calculations are performed in accordance with the flowchart given in Figure 9 of BS 6399: Part 3: 1988.





Note b_2 should be used in the calculation of the snow load shape coefficient.

For the condition where there is a parapet on the high eaves of a stepped building, you must exercise engineering judgment to determine the correctness of the approach adopted by the ***Snow Load Generator***.



If the height of the step ***hs*** (excluding the parapet) is 1.0 m or greater, then the ***Snow Load Generator*** treats this as a step drift condition and uses the total height as the height of the step plus the height of the parapet.

If the height of the step ***hs*** (excluding the parapet) is less than 1.0 m then the ***Snow Load Generator*** treats this as a valley drift condition and uses the difference in height between the valley and the apex ***ha***. If you judge that the parapet height is significant to the snow loading then you will need to calculate the values that you want to use and enter these directly.

Other conditions such as snow drifting against obstructions are not covered and you will need to perform your own calculations and then augment the loading created by the ***Snow Load Generator*** to include these.

Canopies - The ***Snow Load Generator*** will not calculate the snow load on canopies. Neither can other geometry be used to represent canopies without the ***Snow Load Generator*** giving erroneous results.

Index

Portal Frame Design
Release 4.0

Numerics

A

B

C

H

I

J

L

M

N

O

P

T

U

V