

Part 2 :

PowerConnect Reference Manual

© 2008, BuildSoft NV

All rights reserved. No part of this document may be reproduced or transmitted in any form or by any means, electronic or manual, for any purpose whatsoever, without the prior written consent from BuildSoft.

The programs described in this manual are subject to copyright by BuildSoft. They may only be used by the licensee and may only be copied for the purpose of creating a security copy. It is prohibited by law to copy them for any other purpose than the licensee's own use.

Although BuildSoft has tested the programs described in this manual and has reviewed this manual, they are delivered 'As Is', without any warranty as to their quality, performance, merchantability or fitness for any particular purpose. The entire risk as to the results and performance of the programs, and as to the information contained in the manual, lies with the end-user.

Table of Contents

1	INTRODUCTION	1
2	WORKFLOW	2
2.1	DEFINING A NEW CONNECTION MODEL	2
2.2	COMPLETING THE CONNECTION MODEL DEFINITION	5
2.2.1	Loads	5
2.2.2	Changing individual connection elements & adding stiffeners	6
2.3	RUNNING THE CONNECTION DESIGN ANALYSIS	6
2.4	INTERPRETING DESIGN ANALYSIS RESULTS	6
2.5	REPORTING	8
3	WORKING ENVIRONMENT	9
3.1	THE ICON TOOLBAR	9
3.2	THE MAIN WINDOWS	12
3.2.1	The 'Geometry' window	12
3.2.1.1	Rotating the model	13
3.2.1.2	Scaling the model	13
3.2.1.3	Moving the model	13
3.2.1.4	Making the model visible	13
3.2.1.5	Changing elements of the model	14
3.2.1.6	Adding stiffeners to elements of the model	14
3.2.1.7	Changing color and perspective	15
3.2.2	The 'Loads' window	17
3.2.2.1	Defining loads combinations	17
3.2.2.2	Defining loads	18
3.2.3	The 'Data' window	19
3.2.4	The 'Plots' window	21
3.2.4.1	The rigidity graph	21
3.2.4.2	Limit loading diagram	24
3.2.5	The 'Results' window	26
4	LIBRARIES	28
4.1	MATERIAL LIBRARY	28
4.2	SECTION LIBRARY	30
4.3	BOLTS AND ANCHOR BOLTS LIBRARY	32
5	CONNECTION ELEMENTS	34
5.1	ELEMENTS VS. COMPONENTS	34
5.2	WORKING WITH ELEMENTS	34
5.2.1	Defining elements	34
5.2.2	Adding elements	35
5.2.3	Removing elements	35
5.3	DEFINITION OF ELEMENTS	36
5.3.1	General information	36
5.3.1.1	Default values for elements	36
5.3.1.2	Symmetric elements	36
5.3.1.3	Leveling	37

5.3.2	Bar elements.....	37
5.3.2.1	H- or I- sections.....	37
5.3.2.2	Hollow structural sections	41
5.3.3	Connecting elements	43
5.3.3.1	Bolts.....	43
5.3.3.2	Anchor bolts	45
5.3.3.3	Welds.....	47
5.3.3.4	Bolted end plates (moment connections).....	48
5.3.3.5	Bolted angle cleats.....	52
5.3.3.6	Column base plates	59
5.3.3.7	Bolted end plates (shear connections).....	63
5.3.3.8	Bolted fin plates	66
5.3.3.9	Bolted flange plates	69
5.3.3.10	Bolted web plate	72
5.3.4	Stiffening elements	75
5.3.4.1	Stiffening elements for columns	75
5.3.4.2	Haunches.....	80
5.3.4.3	Gusset plates	82
5.3.4.4	Stiffening elements for beams.....	84
5.3.4.5	Column base stiffeners	87
5.3.5	Other elements	89
5.3.5.1	Concrete base.....	89
5.3.5.2	Cramps	90
5.4	DEFINITION OF ELEMENT DEFAULTS	92
5.4.1	Specification of default values for individual element types	92
5.4.2	Use of element default values.....	93
6	GLOBAL FUNCTIONS AND OPTIONS	95
6.1	FILE MANAGEMENT.....	95
6.2	CHARACTERISTIC DISTANCES	95
6.3	BOLT DISTANCES	97
6.4	ANALYSIS OPTIONS	98
6.5	UNITS AND DECIMALS	99
7	REPORTING	100
7.1	PAGE SETUP	100
7.2	REPORT CREATION	101
7.2.1	Previewing	102
7.2.2	Printing.....	103
7.2.3	Printing to RTF.....	103
7.3	REPORT CONFIGURATION	103
7.3.1	Tab page 'General'	103
7.3.1.1	Print project data.....	103
7.3.1.2	Summary drawings	104
7.3.2	Tab page 'Loads'	105
7.3.3	Tab page 'Results'	106
7.3.4	Tab page 'Element info'	107
7.3.5	Tab page 'Connection view'	107
7.3.6	Advanced parameters.....	108
8	PLOTTING	109

8.1	2D CONNECTION DRAWINGS.....	109
8.2	EXPORT OF ELEMENT DRAWINGS TO DXF.....	109

1 Introduction

The objective of this second part of the PowerConnect manual is to provide a comprehensive answer to the questions that may arise during the use of the PowerConnect software. Special attention is given to the workflow and working environment to make sure that the user has easy access to all information that is needed for an efficient use of PowerConnect. Users that have got acquainted with PowerConnect using 'Part 1: Getting Started with PowerConnect', but need more background information as to how PowerConnect operates will therefore benefit the most from this reference manual.

This reference manual does not discuss the supported analysis methods and their theoretical background. Those are covered in separate sections (for example, part 3 of the PowerConnect manual in case of EUROCODE 3), which may prove to be quite useful for the more experienced users. A correct interpretation of the results provided by PowerConnect, which is essential for an efficient and successful use of the product, requires the user to be well informed on those analysis methods and their hypotheses.

In spite of the care devoted to the elaboration of this and other manuals, some readers may find that specific functions or capabilities have not been explained in sufficient detail. If this is the case, do not hesitate to contact the BuildSoft team and to communicate any suggestion(s) for improving the quality of this manual.


2 Workflow

This section briefly informs the PowerConnect user on the program's workflow. Without going into any detail, the different steps of setting up a PowerConnect model, running the analysis and generating the report are summarized. As a result, the user will be made familiar with the general concepts of the software, the typical procedures and the global navigation in the PowerConnect working environment.

Too much detail is avoided in this section, so as to remain focused on the global working principles of the software. Instead, the user is referenced to other parts of this reference manual where such detail information is included.

2.1 Defining a new connection model

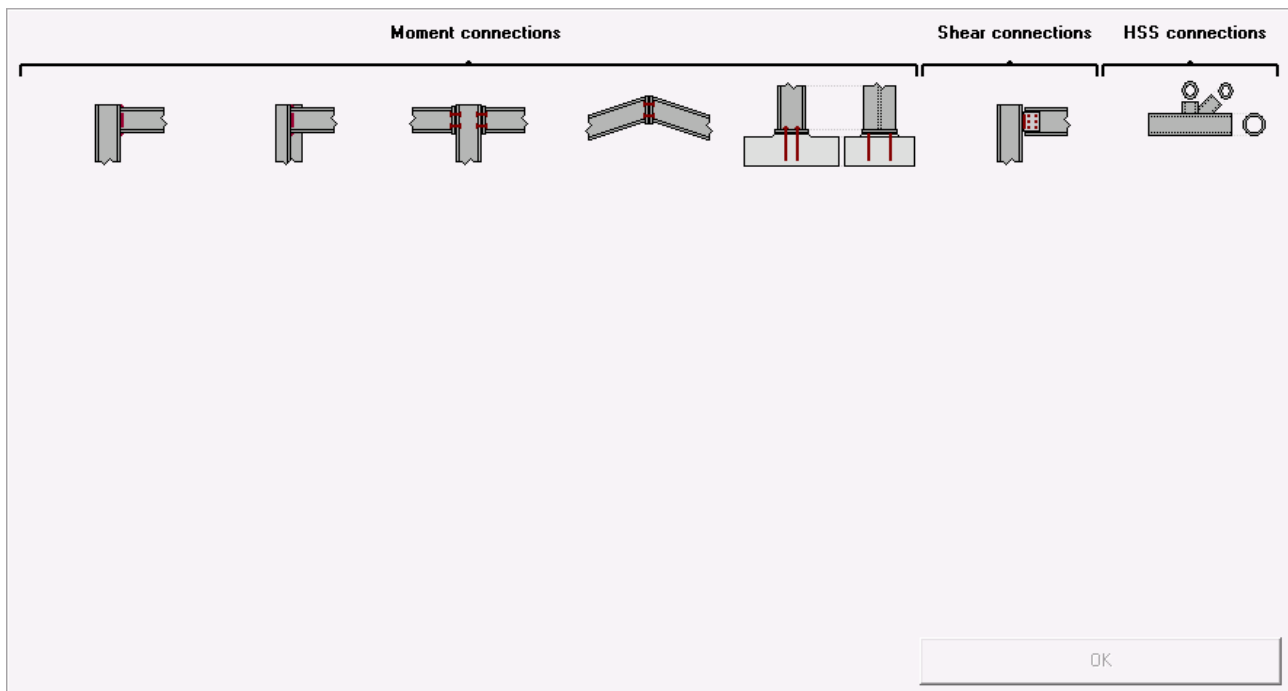
To define a new connection model,

- either use the menu function 'File' – 'New',
- or click on the  icon of the icon toolbar.

This will launch the so-called navigation window in which the user can easily select the type of connection to be analyzed. The selection is made by navigating through an intelligent tree structure that gradually rolls out depending on the initial choices made by the user.

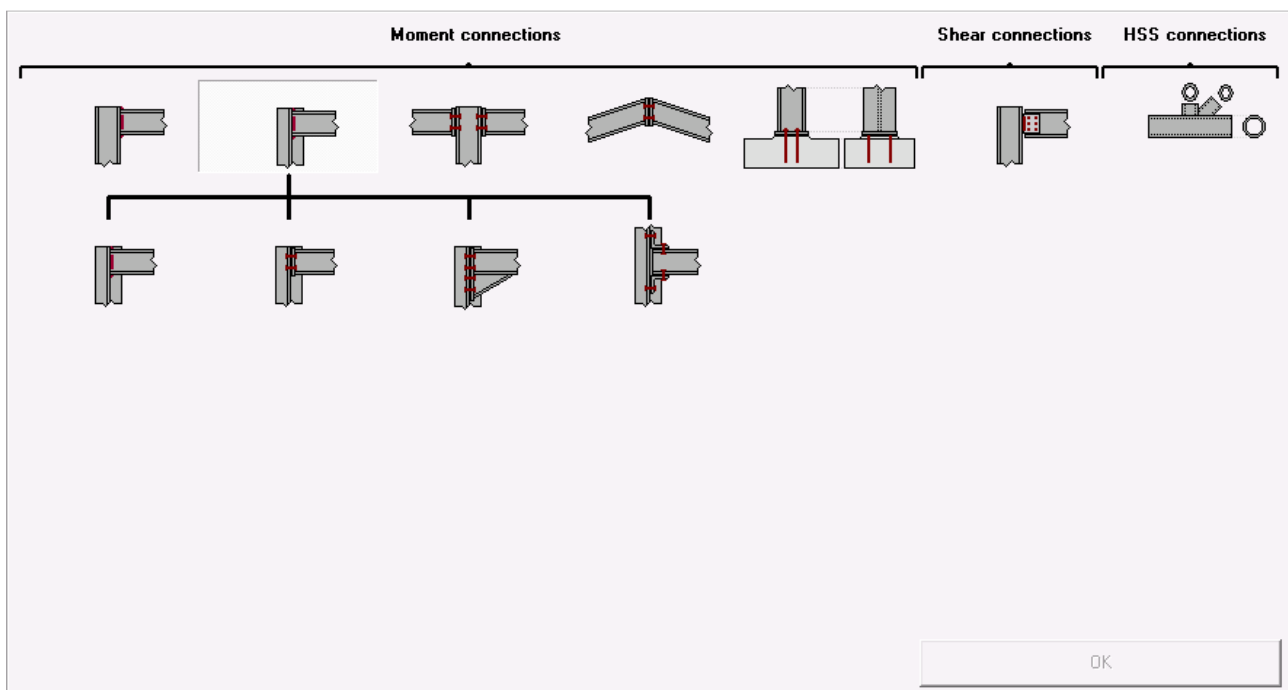
Initially, the user has the choice between following connection types:

- moment connections, subdivided in
 - single-sided beam to column flange
 - single-sided beam to column web
 - double-side beam to column flange
 - beam to beam
 - column base plate
- shear connections
- HSS connections (hollow structural section connections)



Choosing one of the available options, the navigation tree will start to roll out. The user is thus invited to make the next choice in the definition process. As an example, the single-sided beam to column web connection will be selected. The user will then have the following options:

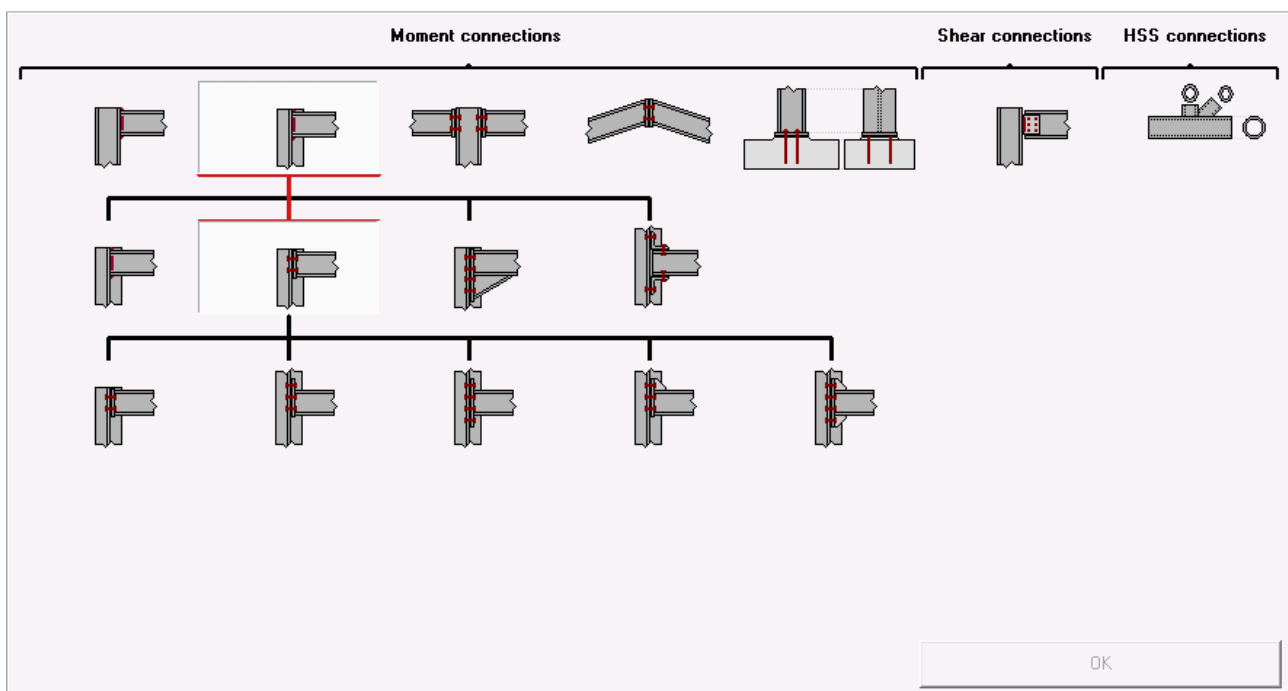
- beam & haunched beam welded
- bolted moment end plate
- haunched beam end plate
- bolted angle cleats



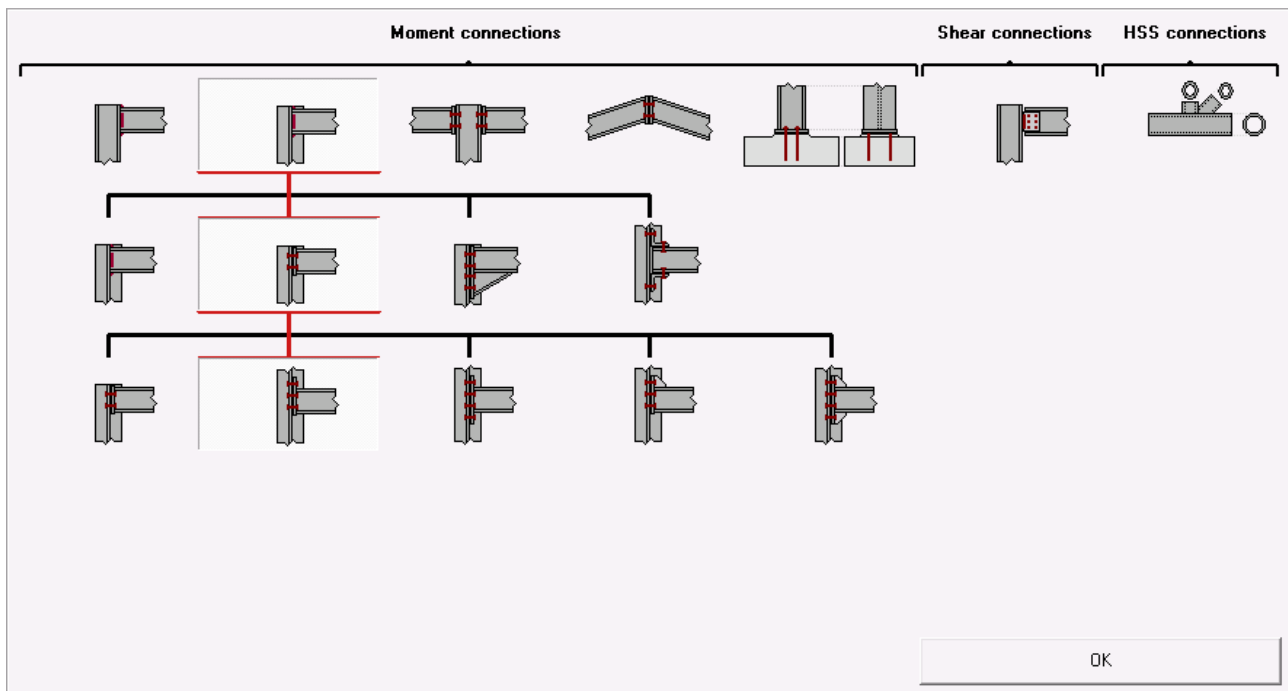
Choosing now for the bolted moment end plate, the navigation tree will further roll out to present following options:

- flush moment end plate
- extended moment end plate (1 side)
- extended moment end plate (2 sides)
- extended moment end plate (1 side) with stiffener
- extended moment end plate (2 sides) with stiffener

as can be seen from the following view of the navigation window.



Making the next choice, no further options will be presented for this particular type of connection and the user will confirm the current selection by means of the 'OK'-button in the navigation window.



As soon as the final selection has been made and has been confirmed by the user, a new dialogue window will be presented in which the user is invited to confirm or modify the characteristics of the different connection elements, e.g.

- beam and column section
- dimensions of end plate
- bolt grade


As soon as a confirmation is given, the user will enter into the 'Geometry' window which presents a 3D view of the connection. More information on the 'Geometry' window is given in section 3.2.1 of this reference manual.

2.2 Completing the connection model definition

As soon as the user has arrived in the 'Geometry' window, it is possible to complete the connection model with additional attributes.

2.2.1 Loads

Load cases and related loads can be defined in the 'Loads' window. Users can switch from the 'Geometry' window to the 'Loads' window

- either by using the menu function 'Windows' – 'Loads',
- or by clicking on the  icon of the icon toolbar.

More information on the 'Loads' window can be found in section 3.2.2 of this reference manual.


2.2.2 Changing individual connection elements & adding stiffeners


The properties of individual connection elements (be it the column or beam, an end plate, bolts or welds) can be modified anytime by double-clicking with the left-hand mouse button on the element that needs modification. Refer to section 3.2.1.5 of this reference manual for more information on this topic.

Stiffeners can be added to the beam or column elements by simply selecting this element and by using the right-hand mouse button to make a dialog window pop up which presents all available stiffener types for the selected element. Refer to section 3.2.1.6 of this reference manual for more information on this topic.

2.3 Running the connection design analysis


Once the definition of the connection has been completed (including applied loads & definition of stiffeners), the design analysis can be launched

- either by using the menu function 'Analysis' – 'Analysis',
- or by clicking on the  icon of the icon toolbar,
- or by using the F9 shortcut of your keyboard.

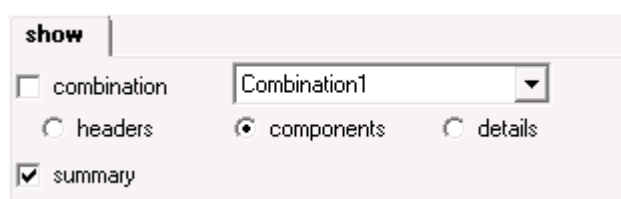
Once the design analysis has been completed, PowerConnect automatically switches to the 'Results' window in order to present a summary of the results or a more detailed results report. The user can always switch to the 'Results' window by means of the  icon of the icon toolbar.

2.4 Interpreting design analysis results


To allow for a comprehensive interpretation of the design analysis results, PowerConnect offers 3 windows to the user:

- the 'Results' window, which presents a summary of the analysis results or a more detailed results report, if requested. The user can at any time switch to the 'Results' window
 - either by using the menu function 'Windows' – 'Results',
 - or by clicking on the  icon of the icon toolbar.


Within the 'Results' window, the user can switch between summary & detail reports by clicking with the left mouse button on the label 'Results Preferences' at the right-hand bottom corner of the 'Results' window. This makes a dialogue appear, in which the level of reporting can be specified by the user.



Without going in any further detail at this time, it should be noted that the detail reporting on the individual combinations gives an insight into the critical failure mechanisms within the connection. More information can be found in the section 3.2.5 of this Reference Manual.

- the 'Limit loading diagram' window, which presents a colour-coded representation of the loading level of each individual part of the connection. Parts which are close to or even at exhaustion level, are colored red, where-as parts which experience low loading levels are colored blue. The user can at any time switch to the 'Limit loading diagram' window
 - either by using the menu function 'Windows' – 'Limit loading diagram',
 - or by clicking on the  icon of the icon toolbar.




More information on the use of this window can be found in the section 3.2.4.2 of this Reference Manual.

- the 'Rigidity graph' window, which presents the load-dependent bending stiffness of the connection (of course, for moment connections only). The user can at any time switch to the 'Rigidity graph' window
 - either by using the menu function 'Windows' – 'Rigidity graph',
 - or by clicking on the  icon of the icon toolbar.

More information on the use of this window can be found in the section 3.2.4.1 of this Reference Manual.

2.5 Reporting

Once the analysis of the connection has been completed with satisfactory results, the user has access to a number of reporting tools:

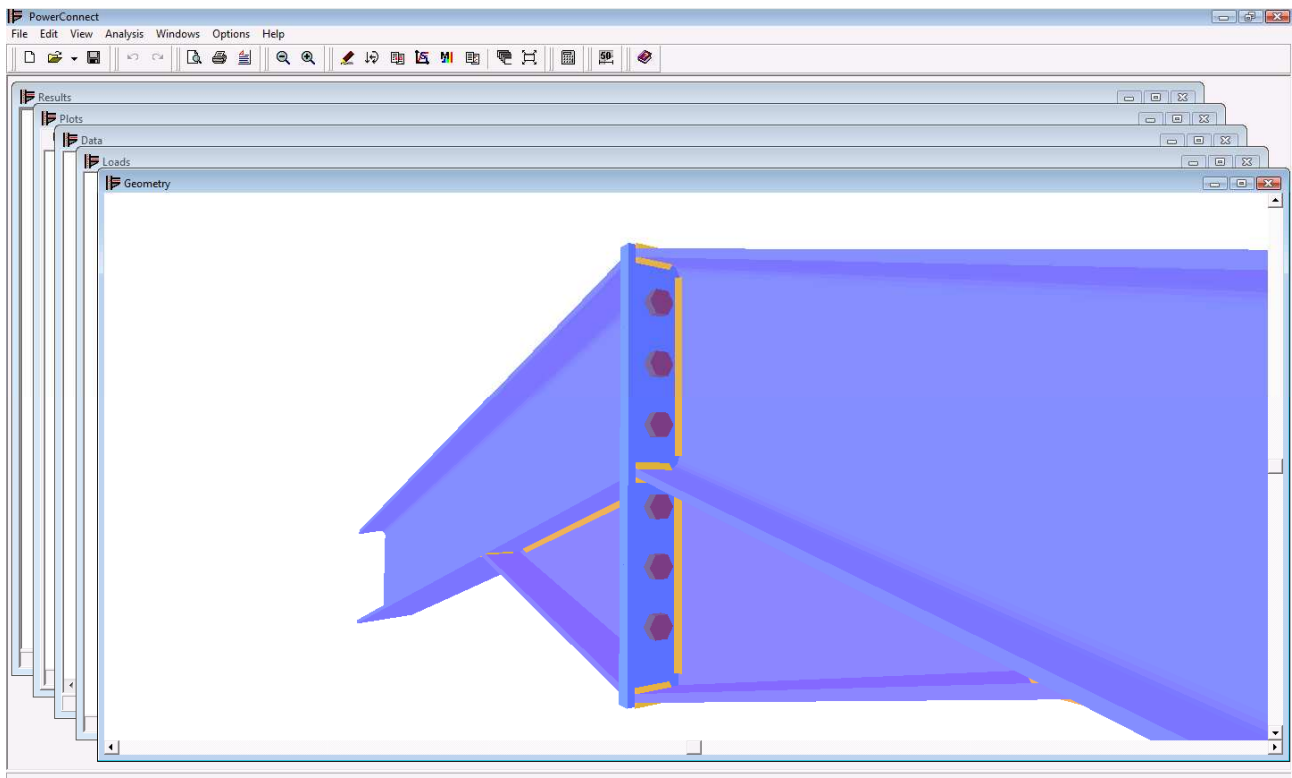
- a 'Print Preview' function, which can be activated
 - either by using the menu function 'File' – 'Preview',
 - or by clicking on the  icon of the icon toolbar.
- a 'Print Report' function, which can be activated
 - either by using the menu function 'File' – 'Print Report',
 - or by clicking on the  icon of the icon toolbar.
- a 'Print Report to RTF' function, which can be activated
 - either by using the menu function 'File' – 'Print Report to RTF',
 - or by clicking on the  icon of the icon toolbar.

All reporting tools provide the capability to specify the contents of the report that should be generated. Those capabilities are presented in more detail in the section 7 of this Reference Manual.

As an aside, it can be mentioned that the report lay-out can be specified using the function 'File' – 'Page Setup'.

3 Working environment.

Once a new connection has been defined through the navigation window or once an existing PowerConnect file has been opened, the user will enter into the PowerConnect working environment. By default, PowerConnect will present a 3D representation of the connection model in the 'Geometry'-window (as shown below).



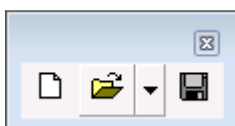
The current section of the PowerConnect reference manual will describe the main components of the working environment. More in particular, the focus will be on

- the PowerConnect icon toolbar,
- the five main windows.




3.1 The icon toolbar




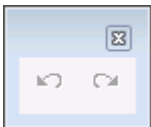
The icon toolbar consists of a series of icons which are logically grouped into units that can freely be moved or removed by the user.



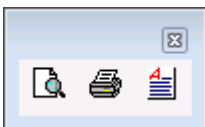
This icon set includes a number of icons related to PowerConnect file management:

-  : open a new project,
-  : open an existing project,
-  : save the active project to file.




PowerConnect keeps track of the last 4 projects that have been opened. Through the small arrow at the right hand side of the  icon, the user can access the related project files.

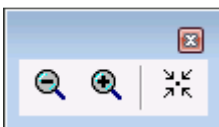





PowerConnect always schedules a back-up of the model prior to any modification. It is thus always possible to undo (or redo) the last operation, through the icons illustrated in the adjacent icon set.

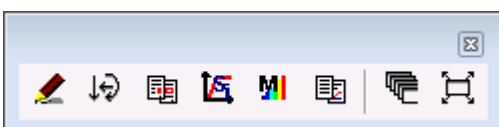


This icon set includes icons which give access to PowerConnect's reporting capabilities:

-  : previewing a report
-  : printing a report,
-  : saving a report to a Rich Text Format (RTF) file. The RTF file can be edited by means of a word processor.



All icons of the current icon set are related to the visualization of the 3D geometry model – more in particular to the ZOOM IN () and ZOOM OUT () functions. The icon  will reposition the connection model within the working window if it has been moved outside inadvertently.



The six icons to the left allow the user to access the PowerConnect main windows:

 : go to 'Geometry' window,



: go to 'Loads' window,



: go to 'Data' window,



: go to 'Rigidity Graph' in the 'Plots' window,



: go to 'Limit Loading Diagram' in the 'Plots' window,



: go to 'Results' window.

The two icons to the right allow to define the window layout within the PowerConnect environment :



: stack all windows on top of each other,



: maximize size of active window to size of PowerConnect working window.



Through the use of the icon shown to the left, PowerConnect is instructed to launch the connection design analysis. Prior to the actual analysis, PowerConnect will perform a number of verifications with respect to any restrictions on bolt distances, bolt positions, component dimensions, ... that are specified by the user or imposed by the selected standard.



The icon to the left launches a separate window in which a number of plan views and elevation views of the entire connection can be combined. It should be noted that those views can be exported to a DXF file through the appropriate export function accessible from the PowerConnect main menu.



The last icon in the toolbar enables the user to access the on-line help function.

3.2 The main windows

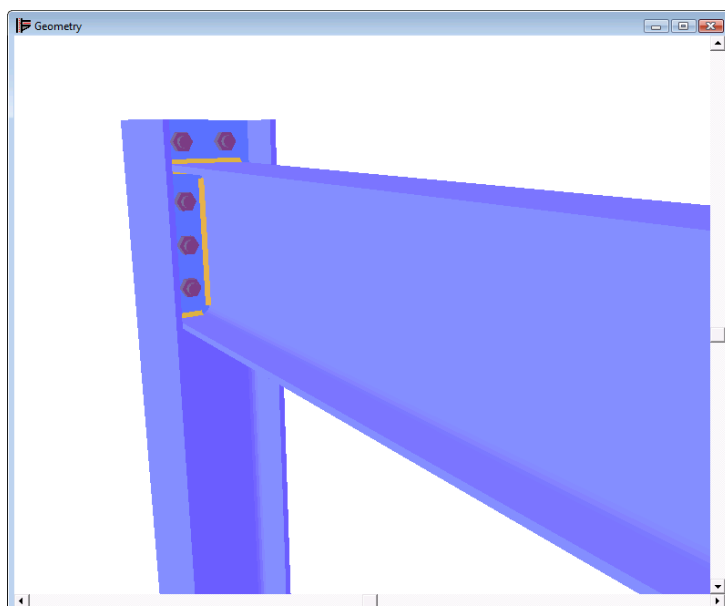
Each of the 5 main windows is dedicated to a specific task of the modeling & design analysis process:

- the '*Geometry*' window: defining & modifying the geometry of the connection model,
- the '*Loads*' window: defining & modifying the loads applied on the connection model,
- the '*Data*' window: presenting the properties of all elements which define the connection,
- the '*Plots*' window: presenting the connection's rigidity graph and its limit loading diagram,
- the '*Results*' window: presenting all design analysis results, either in a summary view or in a fully detailed report.

The functionalities associated to each of the main windows will now be discussed in more detail.

3.2.1 The '*Geometry*' window

The '*Geometry*' window contains a 3D representation of the connection geometry, with a number of associated capabilities.





3.2.1.1 Rotating the model

By means of the sliders on the right-hand and bottom side of the screen, the 3D model representation of the connection model can be rotated in order to define the most optimal viewing point:

- moving the right-hand slider rotates the model around a horizontal axis
- moving the bottom slider rotates the model around a vertical axis


3.2.1.2 Scaling the model

To enhance visibility of specific connection elements, the 3D model can be scaled step-wise using the ZOOM IN () & ZOOM OUT () functions of the icon toolbar.

3.2.1.3 Moving the model

The model PAN function is activated by clicking with the left mouse button on any point within the 'Geometry' window. Then move the mouse to freely move the model within the window.

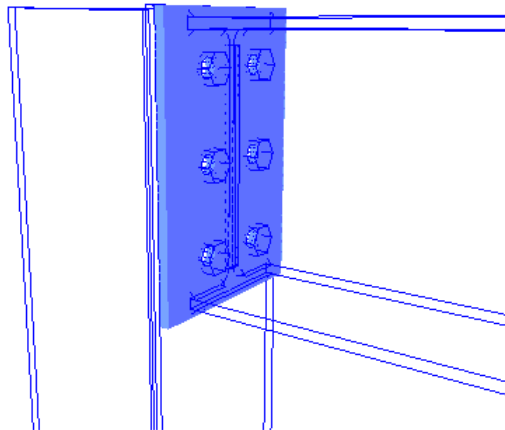
3.2.1.4 Making the model visible

Whenever the model is invisible, use the function  to reposition the connection model within the working window if it has been moved outside inadvertently.

3.2.1.5 Changing elements of the model

Any modifications to existing elements of a PowerConnect model are consistently defined using the following procedure:

- select by means of the left-hand mouse button the connection element for which a modification is to be specified.
- as a result, only the selected element will be rendered. All other elements will be presented with full transparency.

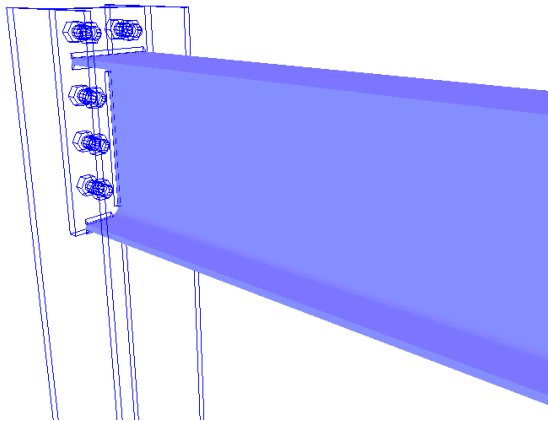


To actually define the modification to the selected element, click again on the element using the left-hand button of the mouse. The appropriate dialogue window will appear, allowing to specify the proposed modifications.

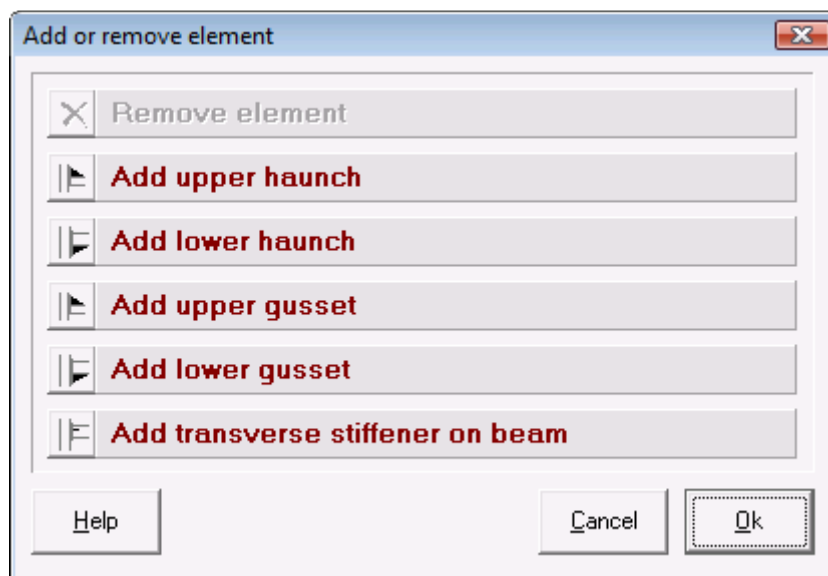
3.2.1.6 Adding stiffeners to elements of the model

Adding stiffeners to existing elements of a PowerConnect model is consistently accomplished using the following procedure:

- select by means of the left-hand mouse button the connection element for which a stiffener is to be added.
- as a result, only the selected element will be rendered. All other elements will be presented with full transparency.



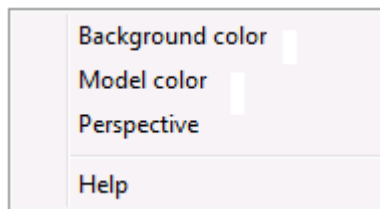
Then use the right hand mouse button to make a new window pop up which presents a list of possible stiffeners that can be added to the selected element of the connection. It is sufficient to select the appropriate stiffener from the list, and to select this choice by means of the 'OK' button.



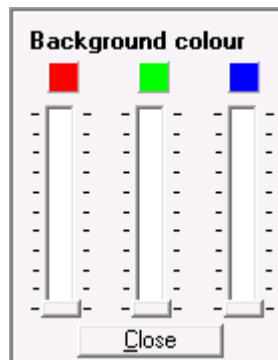
The contents of the above window will of course depend on the type of connection element that has been selected. This will be discussed in more detail in a dedicated section of this reference manual.

3.2.1.7 Changing color and perspective

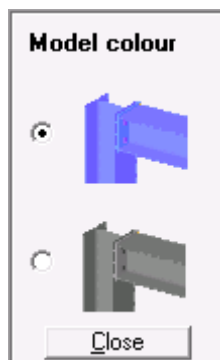
By means of the Geometry window's floating menu, which can be made visible by clicking the right-hand mouse button within the window, it is possible to change the window background color, the color which is used to render the geometry model and the perspective view.



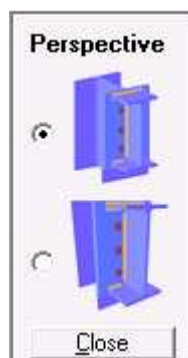
To change the window background color, select the option 'Background color'. A window with RGB-cursors will appear, in which the user can specify to what extent the 3 color Red / Green / Blue should be mixed to define the window background color.



To change the color used to render the 3D geometry model, select the option 'Model color' of the floating menu. A choice can be made between blue and grey rendering colors.

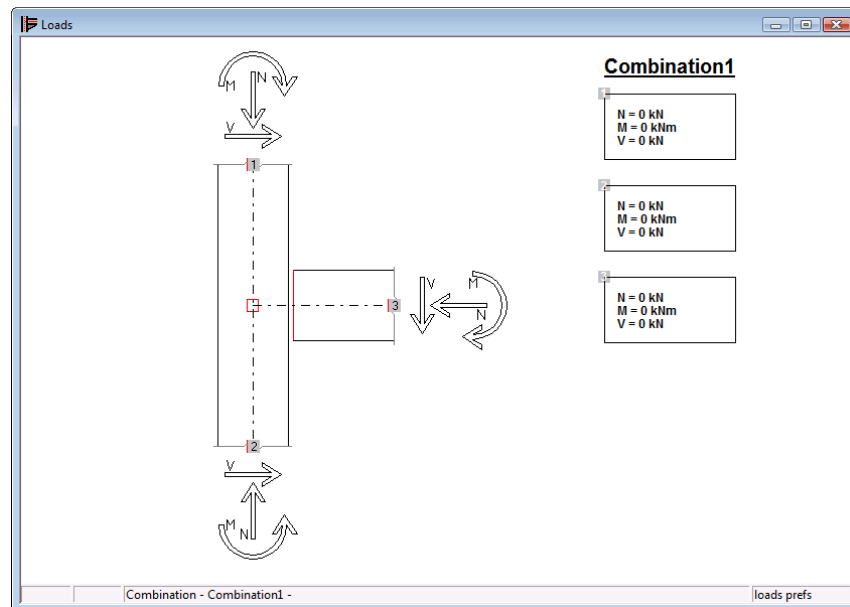


To change the view, select the option 'Perspective' of the floating menu. A choice can be made between an axionometric view and a true perspective.



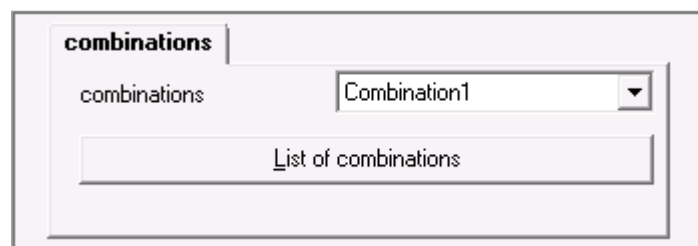
3.2.2 The 'Loads' window

The 'Loads' window contains a 2D representation of the connection geometry, along with the applied loads for the active loads combination. The window contains a number of functions to define extra loads combinations, and to define the contents of each loads combination.



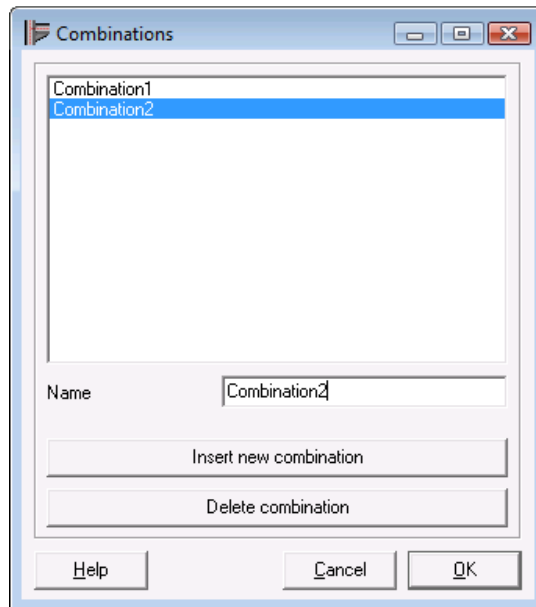
3.2.2.1 Defining loads combinations

To get an overview of the existing loads combinations within the active PowerConnect project, click with the left-hand mouse button on the label "Loads Preferences" at the right-hand bottom corner of the 'Loads' window. This action will make the following dialogue window appear:



The pull-down menu contains the loads combinations that have already been defined within the active PowerConnect project. By scrolling through this list, the user can switch between existing loads combinations.

To add a new loads combination to the list, use the button "List of combinations" in the above window, to get an overview of the existing combinations.



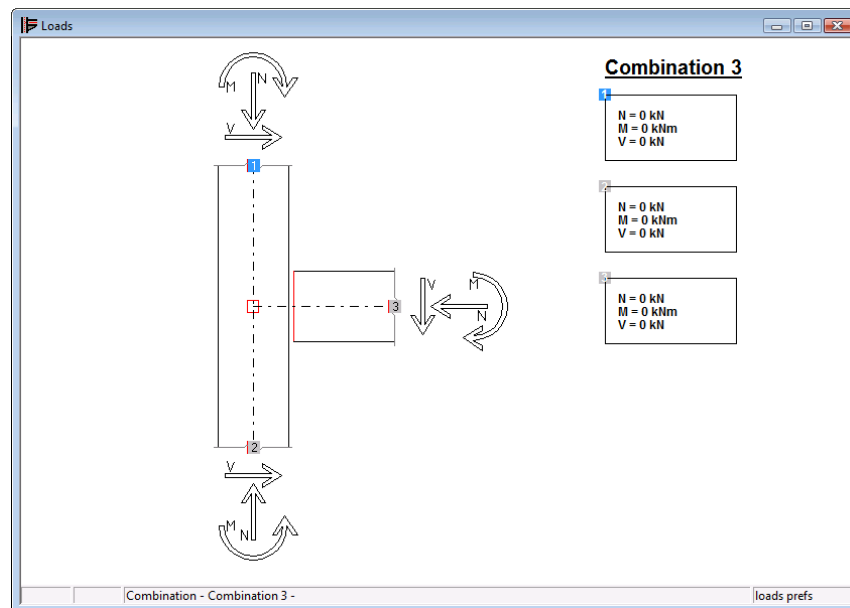
Insert a new loads combination by means of the button 'Insert new combination'. Then change the name of the newly defined combination. If the name of an already existing loads combination needs to be modified, first select this combination from the list. Then just edit the name of the loads combination.

Existing loads combinations can be removed from the list by selecting the appropriate combination and by then using the button 'Delete combination'.

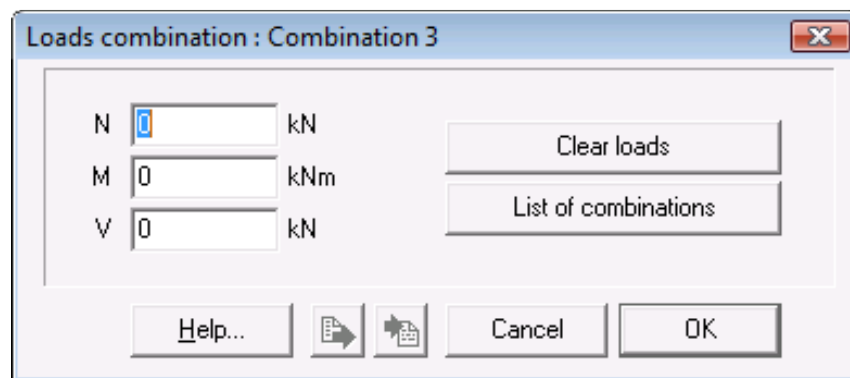
Any changes that are defined within this window still need to be confirmed by means of the 'OK' button when the window is left. If this is not done, all changes will be considered as temporary changes and will be canceled upon leaving the window.

3.2.2.2 Defining loads

To apply loads on an element of the connection, click with the left-hand mouse button on the appropriate square label at the end of a beam or column.



This will make a dialogue window pop up in which the corresponding loads can be entered for the active loads combination:



Enter the appropriate values, or reset any existing values to ZERO by means of the 'Clear Loads' button.

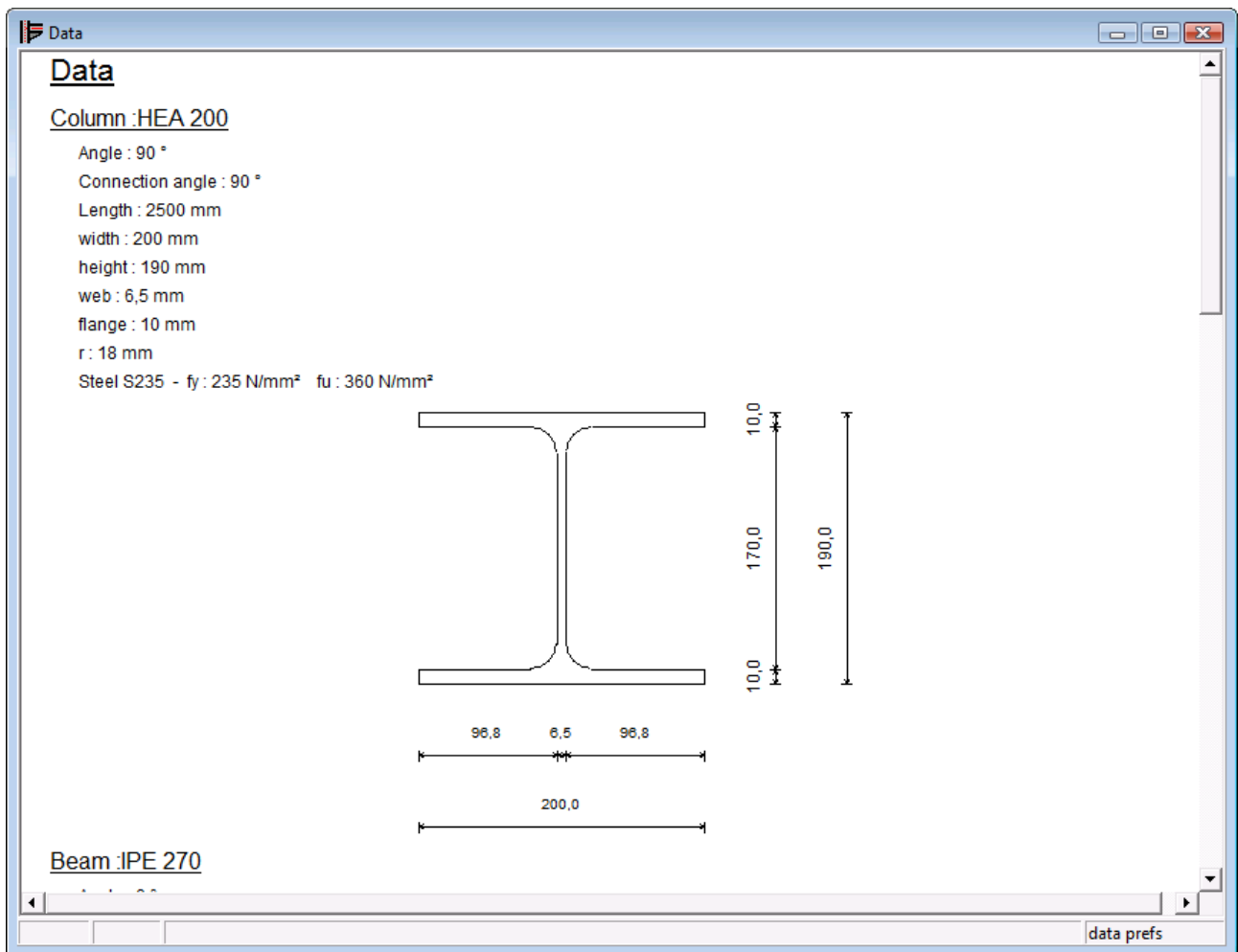
New combinations can also be added from this dialogue by means of the "List of combinations" button.

3.2.3 The 'Data' window

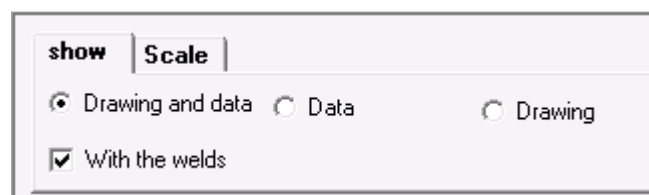
The 'Data' window contains an overview of all individual elements of the connection. For each element, following information can be presented:

- description of geometric and material properties
- graphic representation of element, along with dimensions

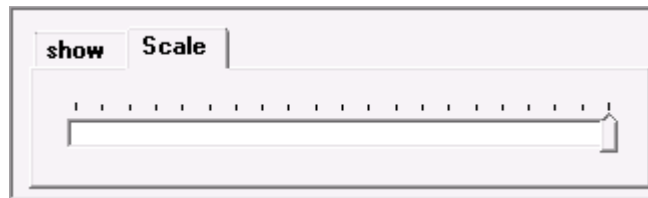
if requested by the user.



To specify which type of information should be presented in the 'Data' window, click with the left-hand mouse button on the label 'Data Preferences' at the right-hand bottom of the window. Select one of the available options, and select the option "With welds" if the representation of welds on the geometry is requested.



By switching to the 'Scale' tab page in the above window, the user can also control the scale which is used for the graphical representation of the connection elements. Use the slider to manually adjust the scale.



Remark : in case the user has not selected any particular connection element in the 'Geometry' window, the 'Data' window will include a representation of all connection elements. If it is the intention to verify the data for a single element only, select this element in the 'Geometry' window prior to opening the 'Data' window.

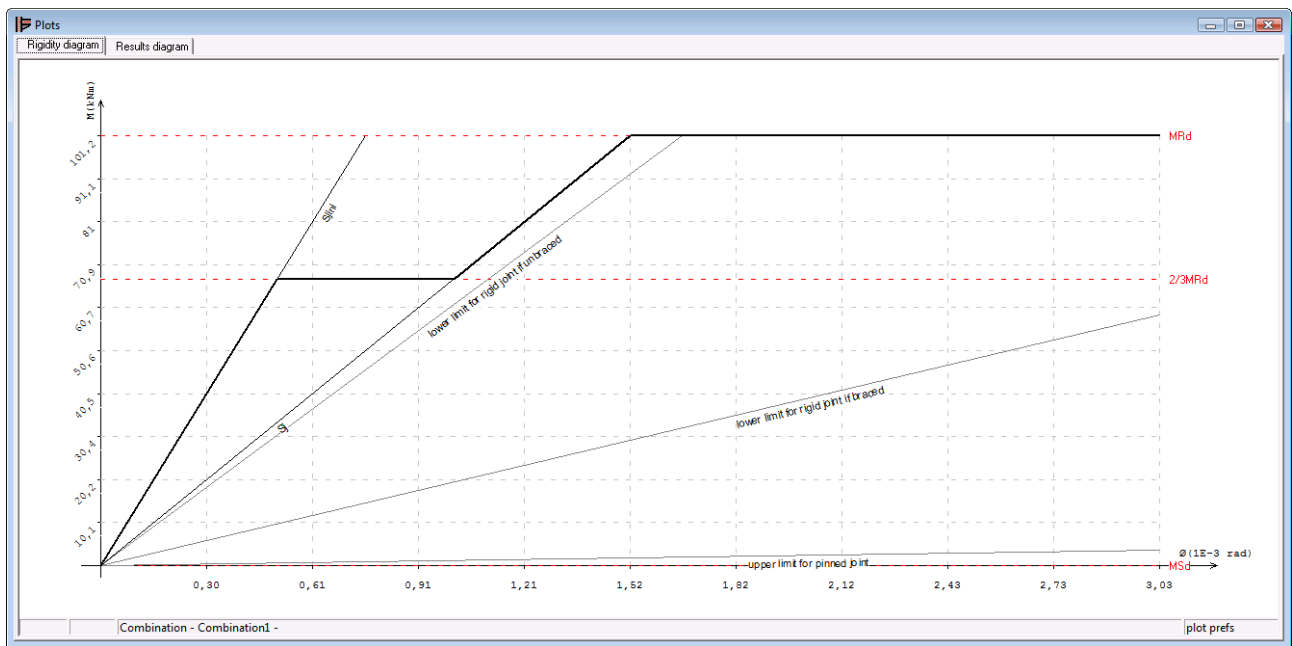
3.2.4 The 'Plots' window

The 'Plots' window includes 2 tab pages:

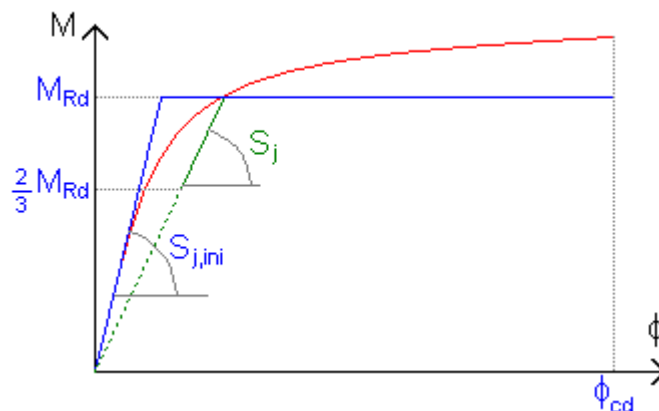
- "Rigidity graph", documenting the load-dependent bending stiffness of the connection (if applicable to the type of connection being analyzed).
- "Limit loading diagram", documenting the load level of the individual connection elements (for a moment load applied to the connection).

3.2.4.1 The rigidity graph

The rigidity graph represents the connection's angular rotation as a function of the applied bending moment. The actual connection stiffness corresponds to the slope of a straight line connecting the origin with a particular point on the curve corresponding with a specific combination of bending moment and angular rotation.



The above stiffness diagram is of course an idealized representation of the actual connection stiffness. The actual connection stiffness will normally correspond to a continuous curve very similar to the red curve in the diagram below.



As the calculation of such a stiffness diagram seriously complicates the analysis, the EC3 standard proposes to replace the red curve by a bilinear or trilinear diagram. PowerConnect will always propose a bilinear diagram. With such a diagram, the connection will have a constant stiffness value independent of the bending moment which is applied to the connection. Once the connection's bending moment resistance has been reached, the stiffness will be reduced to zero. Nevertheless, PowerConnect will make a distinction between the stiffness S_j and the initial stiffness $S_{j,ini}$. It is indeed observed that in case a connection is subjected to a bending moment, the connection will initially be characterized by a linear elastic behavior. For the lower values of the applied bending moment, the connection's angular

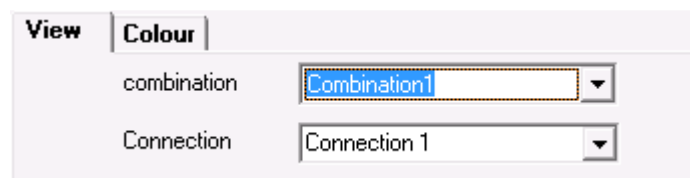
rotation will increase proportionally with the applied bending moment. It can thus reasonably be assumed that the connection stiffness is constant. Drawing the tangent line to the real stiffness curve of the connection will deliver the corresponding constant stiffness value, the so-called initial stiffness of the connection.

When the applied bending moment is increased step by step, the linear elastic behavior will no longer be observed as soon as higher load levels are achieved. It will indeed be seen that connection stiffness will decrease with applied loading levels. The EC3 standard specifies that the connection's initial stiffness can be used for bending moments up to 2/3 of the connection's bending resistance. For bending moments above this threshold, a reduced stiffness should be considered as stiffness will now decrease progressively with increasing bending load.

This complex stiffness characteristic will normally be described by one reduced stiffness value (shown in green on the graph above). This stiffness value corresponds to the initial connection stiffness reduced by a factor depending on the type of connection.

Next to performing the calculation of connection stiffness, PowerConnect will also classify the connection as either RIGID, SEMI-RIGID or PINNED. PowerConnect will indeed indicate the stiffness ranges in which the connection can be considered RIGID or PINNED. Depending on those ranges, PowerConnect will then classify the connection by comparing its initial stiffness value ($S_{j,ini}$) with the afore mentioned ranges, also in case the applied bending moment exceeds 2/3 of the connection's bending resistance.

PowerConnect will present a rigidity graph for each connection (in case of a two-sided connection) and for each loads combination. To show the corresponding diagram, click with the left-hand mouse button on the label 'Plot preferences' to make the following window appear in which the appropriate choices can be made.

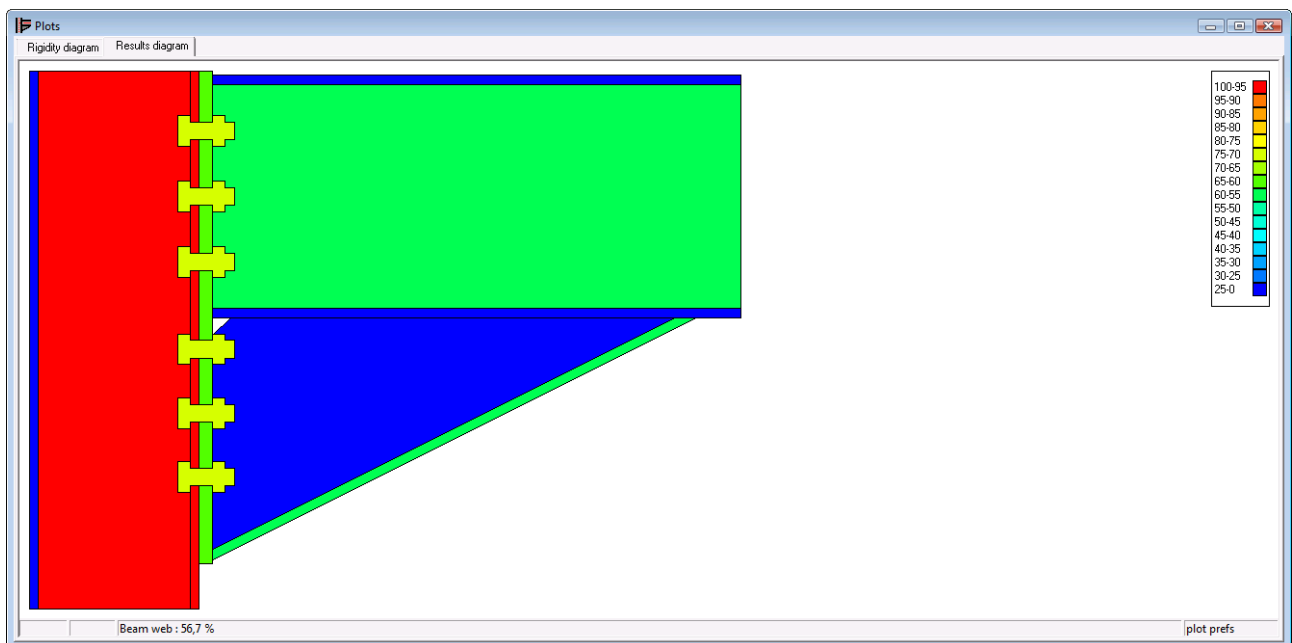


The second tab page allows to modify the background color of the rigidity graph window.



3.2.4.2 Limit loading diagram

The limit loading diagram offers a convenient tool to optimize connection resistance. Indeed, this diagram shows to which extent each individual element is loaded compared to its maximum loading capacity (expressed as a percentage of the element's maximum loading capacity, visualized by means of a color scale).



As mentioned before, the color coding of each element is done on the basis of its current loading level compared to its maximum loading capacity. For moment connections, maximum loading capacity is determined on the basis of bending resistance. For shear connections, maximum loading capacity is calculated bases on shear resistance only.

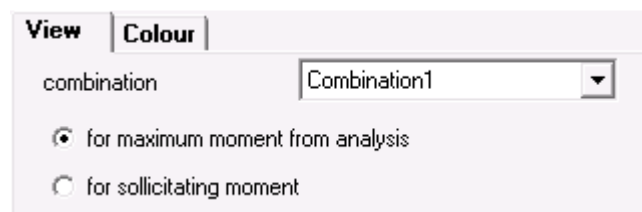
The major advantage of the limit loading diagram is that critically loaded elements can easily be identified thanks to their (near-to) red color, such that the user is guided towards the most effective connection design changes. Stiffening those elements or changing the connection in such a way that those elements are unloaded, will be the most effective solutions to increase the connection's loading capacity.

Using the same principles, elements with a (near-to) green color in the diagram can be downsized or even be eliminated, as they contribute very little to the connection's overall design resistance.

The limit loading diagram can be visualized for each loads combination that has been defined. Furthermore, the user has the option to show results on the diagram

- either as a function of maximum loading capacity (moment resistance or shear resistance),
- or as a function of the actually applied loads.

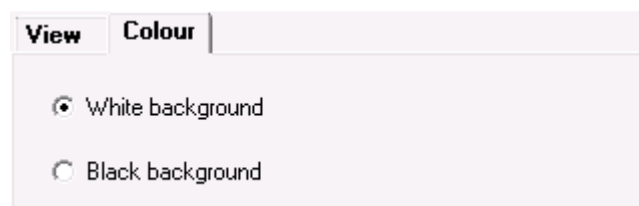
To switch between both options, click with the left mouse button at the right-hand bottom corner of the window on the label 'Plot Preferences', and select the appropriate option.



Remarks :

- *In case the limit loading diagram is drawn on the basis of the connection's maximum loading capacity, it will always include at least one element is colored red. This doesn't necessarily imply that the connection has been undersized, but it does indicate the parts of the connection with the largest potential for increasing resistance at minimal cost.*
- *In case the limit loading diagram is drawn on the basis of the loads applied to the connection, one or more elements of the connection may be colored red. In this case, the connection clearly has insufficient strength and must be changed to meet all related requirements to arrive at a limit loading diagram in which all values are below 100%.*

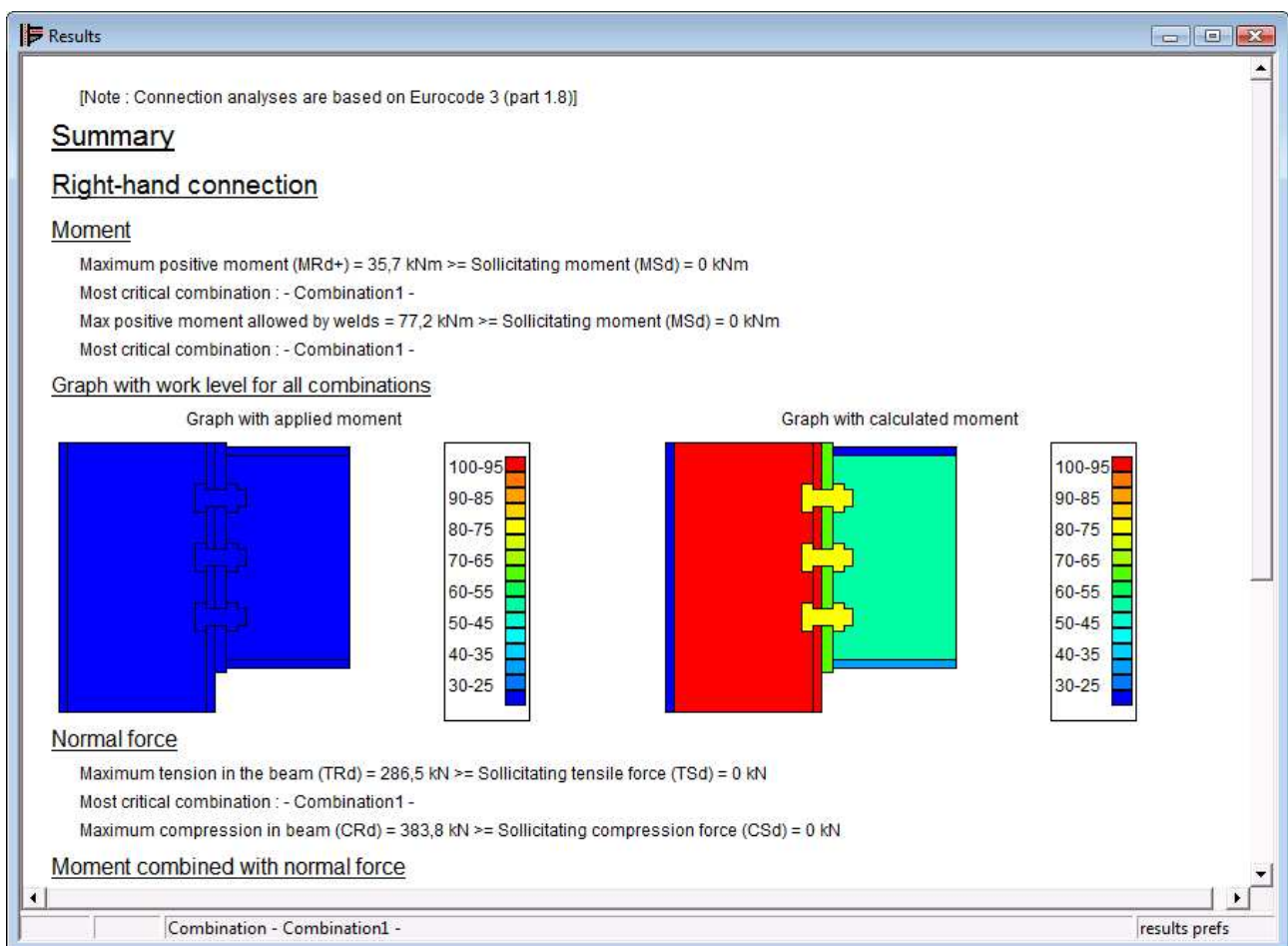
It can finally be remarked that the background color of the window used to show the limit loading diagram can be changed from white to black. The user should select the 'Color' tab page in the above window to switch to the appropriate dialogue.



3.2.5 The 'Results' window

PowerConnect automatically switches to the 'Results' window as soon as a connection design analysis has been completed. By default, PowerConnect will present a summary report as shown below, including the specification of the most critical loads combination.

In case the connection's design resistance is insufficient compared to the applied loads, a warning will be issued in red to attract the attention of the user.



The user can however switch very easily to a more detailed results report by clicking with the left mouse button on the label 'Results Preferences' at the right-hand bottom corner of the window. Then the field 'combination' should be selected to switch to the detail report for a specific loads combination. The loads combination itself (e.g. the most critical loads combination as included in the summary report above) should be selected from the pull-down menu.

show

☐ combination Combination1

☐ headers ☒ components ☐ details

☒ summary

For each loads combination, results reporting can be performed on 3 detail levels:

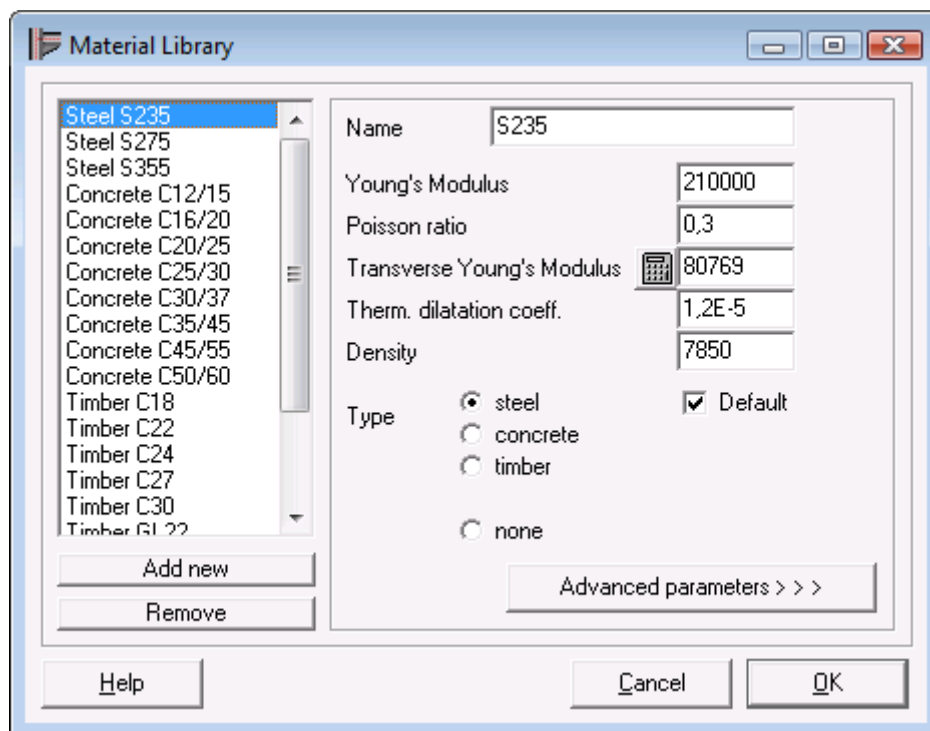
- **'headers'**-level : only the major analysis results will be reported globally,
- **'components'**-level : analysis results will be reported for each individual component,
- **'details'**-level : analysis results will be reported for each individual component, including the details of how those results are obtained.

4 Libraries

A typical use scenario for PowerConnect will always start with the definition of a geometry model, and will mostly include also a number of modifications applied to the geometry model. Those tasks are significantly enhanced through the availability of a number of libraries within the PowerConnect environment. Those libraries will be discussed in more detail within the current chapter of the reference manual.

4.1 Material library

PowerConnect includes a material library with the most common concrete and steel grades. The library itself can be opened for editing and modifications through the menu 'Edit – Material Library...', which will open the following dialogue window.




Next to the elastic material properties, the library also contains a number of strength characteristics that are typically needed for design verifications with respect to national or international design standards. Those characteristics can be accessed through the button Advanced parameters >>> in the above window.

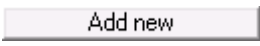
The first column presents a list of all materials currently available within the library. To edit the properties of an existing material, just select the material

in the list and edit the elastic properties available on the right-hand side of the window.


Following elastic properties are available within the PowerConnect material library:


- Young's modulus (E),
- Poisson ratio (ν),
- shear modulus (G),
- thermal expansion coefficient (α),
- material density (ρ)

For a material which behaves according to an elastic law, the first 3 properties are not mutually independent. Therefore, a dedicated icon button  allows to calculate the shear modulus from Young's modulus and Poisson ratio.

In case a new material needs to be added to the current list, click on the button  just below the list of materials. Now edit the name of the new material in the edit field just above the list of elastic material properties. Make sure to fill out those characteristics as well. Do not forget to specify the type of material (steel or concrete), as this choice will determine how the material can be used in PowerConnect.

For both concrete and steel, one material grade can be defined to be the default grade. Make sure to select this option for the desired default grade. As a consequence, all (concrete and steel) elements that will be defined in PowerConnect will automatically have the selected default grade.

A material can be removed from the current list by means of the appropriate  button, after having selected the material that should be removed.

By using the button  in the above dialogue window, it is also possible to edit the strength characteristics of the selected material. On the left hand side, all design codes supported by PowerConnect are listed. On the right-hand side, the corresponding strength characteristics are documented for the select design code.

In case a particular steel grade has been selected as the active material, following characteristics are available:

- yield strength and ultimate tensile strength as a function of plate thickness

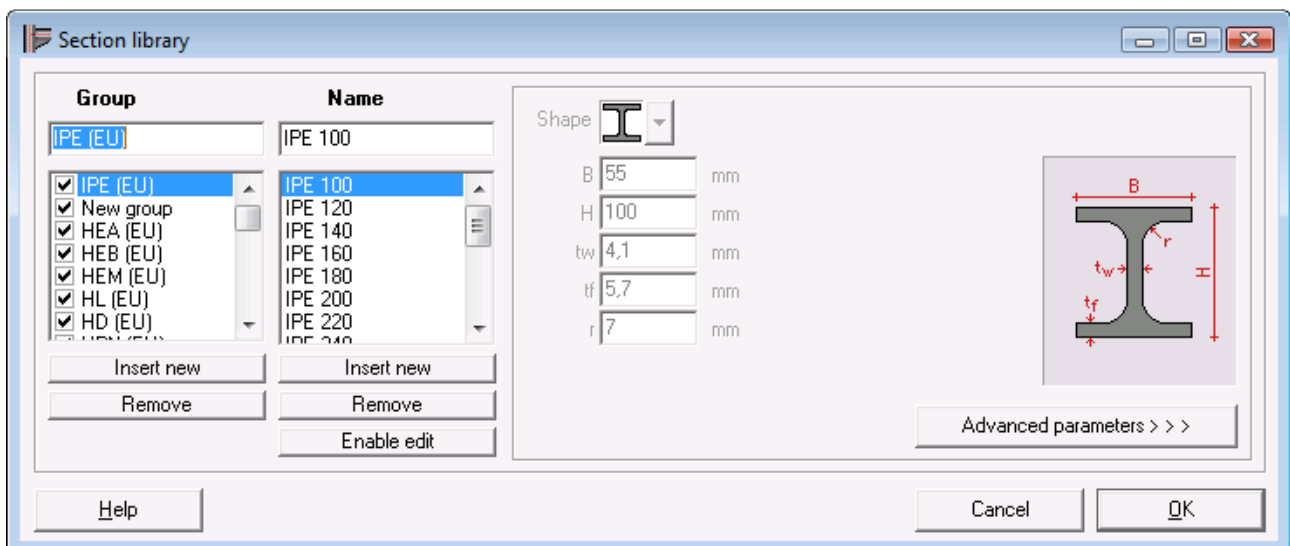
- partial safety factors

Those parameters depend of course on the type of standard that is selected. For more information, reference is made to the appropriate standard.

In case a particular concrete grade has been selected as the active material, a wide range of characteristics are available. However, within the context of a PowerConnect design analysis only the compressive strength will intervene.

4.2 Section library

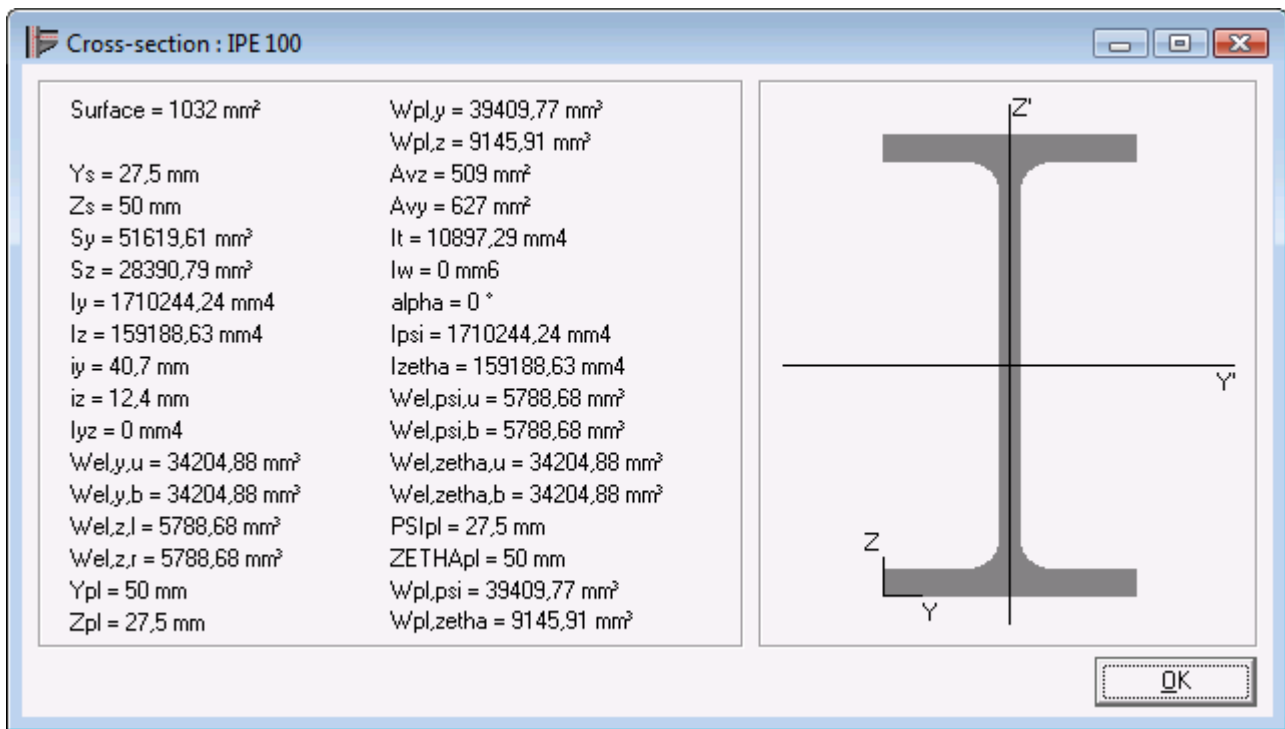
PowerConnect includes a section library with the most common European, English, American & Indian steel sections. The library itself can be opened for editing and modifications through the menu 'Edit – Section Library...'.



The first column presents a list of all section groups currently available within the library. The second column presents a list of all sections which are part of selected group.

At the right-hand side of the window, the selected section is represented graphically. More detailed information on section properties (moments of inertia, resistance, ...) is obtained by means of the button

Advanced parameters >>>



In case a new section group needs to be added to the current list, click on the button **Insert new** just below the group list. Now edit the name of the new group in edit field just on top of the Group list.

To add a new section within the active group, click on the button **Insert new** just below the list of sections for the current group. Again, edit the section name in the edit field just on top of the Section list. Then start defining the section properties in the right-hand side of the window:

- first choose the section type from the pull-down menu next to the label “Shape”,
- then fill out the requested section dimensions.

Those actions should be repeated as many times as the number of new section that need to be added to the current group. Any new sections defined within the active group will be added just below the currently selected section within the group.

A section group or a section can be removed by means of the appropriate **Remove** button, after having selected the group or the section that should be removed.

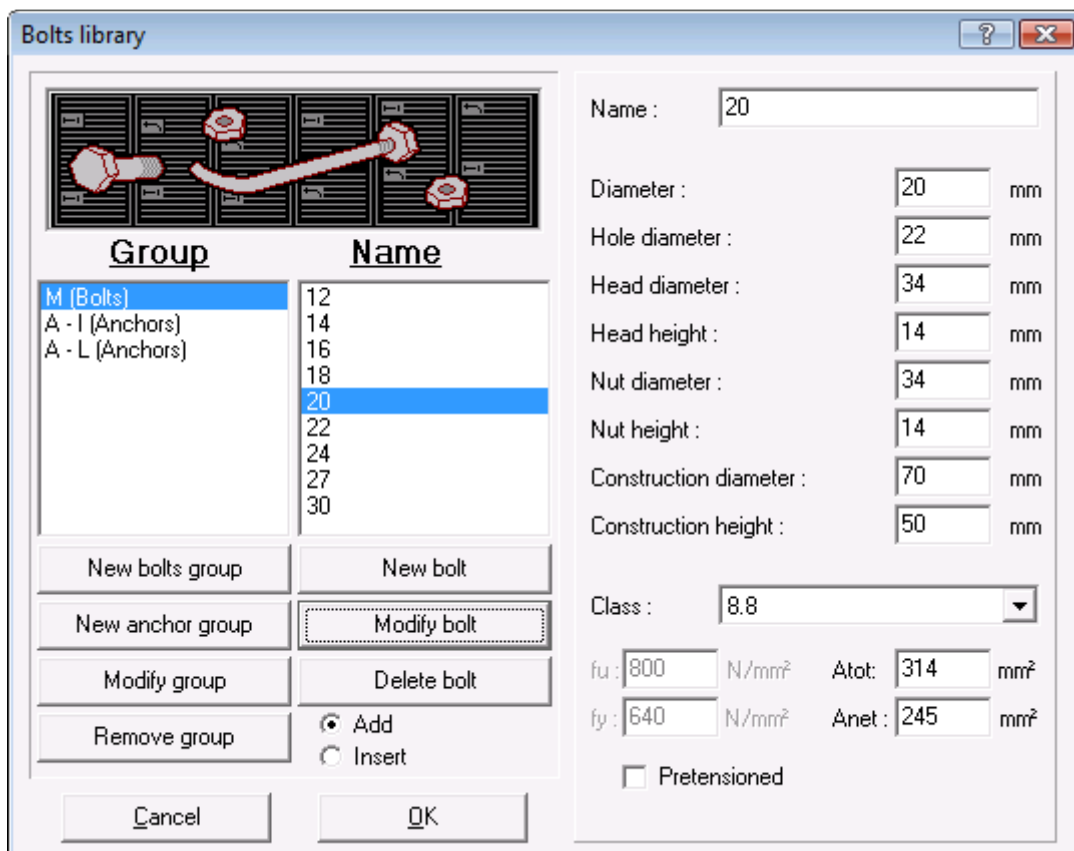
In case the dimensions of a particular section should be edited, first select the section in the list and then use the button **Enable edit** to gain access to the section’s dimensions. It should be noted that section properties (such as moments of inertia, etc...) depend on those dimensions and will therefore

automatically be recalculated as a function of the newly specified dimensions. It is not possible to modify any of those automatically calculated properties.

As already stated, the section library includes by default a number of European, English, American and Indian sections. It may occur however that the user is not necessarily interested in all those sections, depending on his or her geographic region. In that case, it is interesting to know that section groups can be made active or inactive by selecting or unselecting them in the group list. Note that de-activating a group will not remove the sections from the section library, it will just make those sections invisible in the list whenever the user wants to assign a section to a selected bar element. It is thus a very convenient tool to shorten the list of available sections down to the daily needs and practice of each individual user.

4.3 Bolts and anchor bolts library

PowerConnect finally includes a bolts (& anchor bolts) library with the most common (anchor) bolt types. The library itself can be opened for editing and modifications through the menu 'Edit – Bolts Library...', which will bring up the following dialogue window.

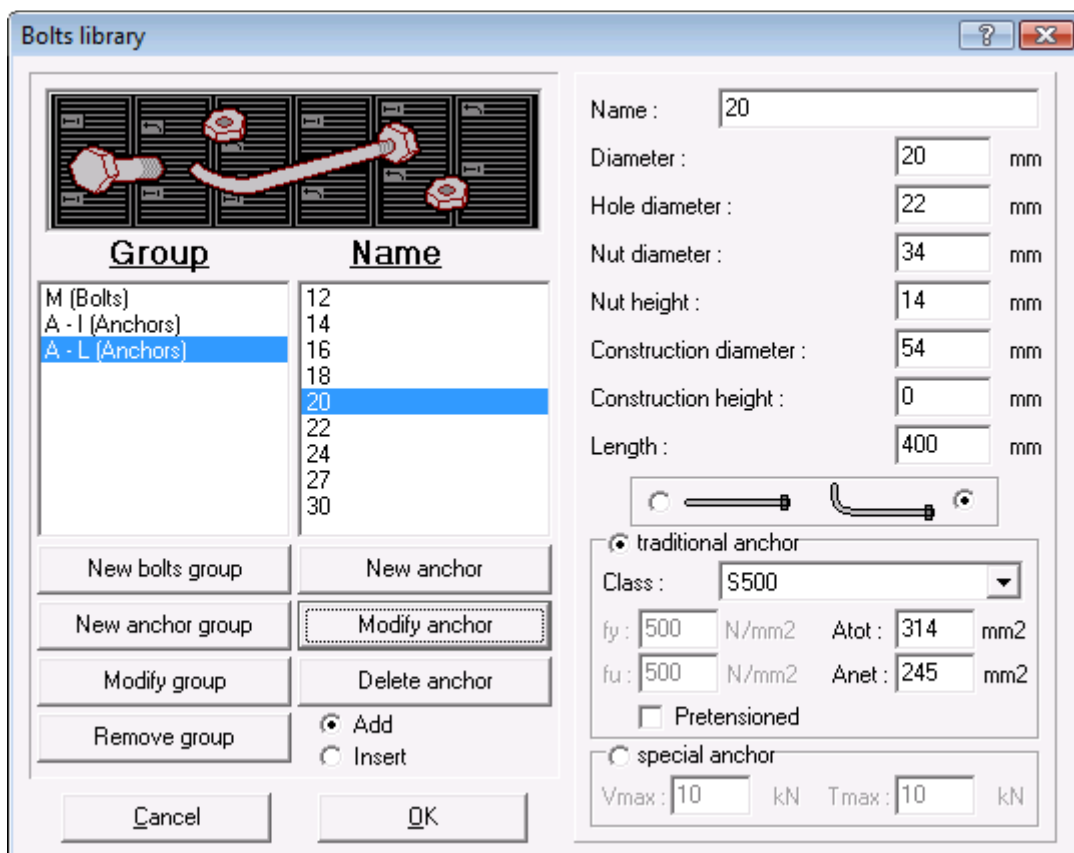


PowerConnect includes by default a bolt group labeled 'M' and two anchor bolt groups labeled 'A - I' and 'A - L'. The bolt group includes a series of

bolts grade 8.8, while the anchor bolt groups contain a series of anchor bolts grade S500. The anchor group labeled 'A-I' includes straight anchor bolts, while the group labeled 'A-L' includes bent anchor bolts.

The user can add as many bolt groups as he or she wishes to do, or remove them (as long as at least one group remains available). To this purpose, the buttons below the left column can be used.

Each group either contains bolts or anchor bolts. Within each group, (anchor) bolts can be added, edited or removed by means of the buttons below the list of (anchor) bolts. To edit or delete a (anchor) bolt, make sure to select it from the list. This selection will extend the dialogue window to the right, to include the (anchor) bolt properties.



To edit those properties, use the button 'Modify anchor' to gain access to the currently defined values.

5 Connection elements

5.1 Elements vs. components

To spell out things as clearly as possible, this section starts with the definition of what are considered **elements** and **components** by PowerConnect.

- an **element** is a physical part of a connection, such as for example
 - a bar that is connected to another bar or to a concrete base,
 - a bolt or anchor bolt,
 - a weld,
 - an end plate,
 - an angle cleat,
 - a fin plate,
 - a stiffener,
 - ...

The term “element” is therefore always associated with the geometry model of the connection.

- a **component** is not a physical part of a connection, but rather a analysis model representation of a part of the connection that takes into consideration its elastic-plastic failure mechanisms and its deformation characteristics.

The term “component” is therefore always associated with an elasto-plastic design analysis process, and the method used by this process is called the “component method”.

5.2 Working with elements

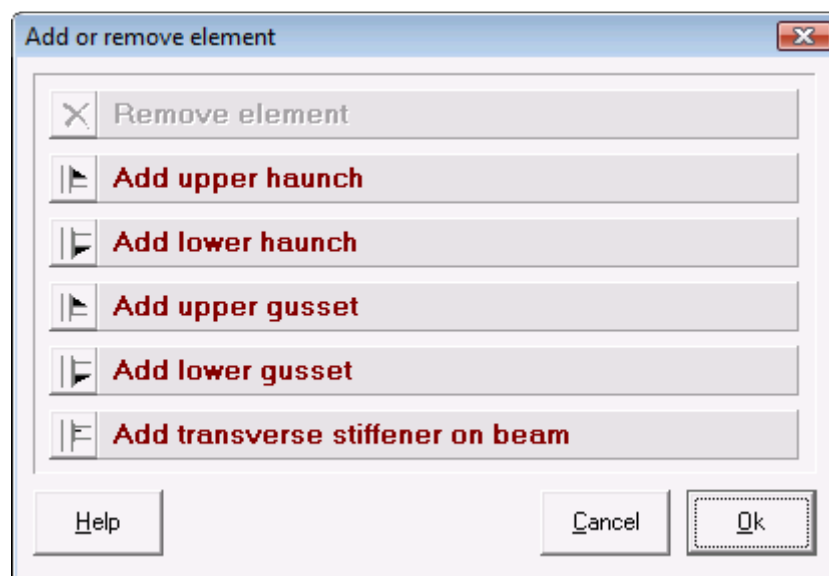
5.2.1 Defining elements

Element properties can be specified as part of the initial definition phase (using the navigation window) or as soon as the element has been added to the PowerConnect model. Elements can be added at any time using the method described in section 5.2.2. The definition of element properties for all supported element types is described in detail in section 5.3 of this reference manual.

5.2.2 Adding elements

Adding elements to a PowerConnect model is done by clicking with the right-hand mouse button on an already selected connection element. To add a haunch to a column-beam connection, for example, click with the right-hand mouse button on the beam element after it has been selected. To add web stiffening plates, click with the right-hand mouse button on the web that needs stiffening. In all cases, this operation will launch a window in which a selection can be made between all elements that can meaningfully be added to the selected element.

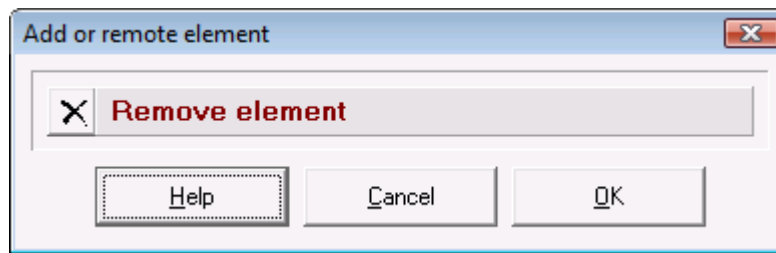
Of course, the list of available elements which can be added to a selected component will depend on the type of component that was selected. As a consequence, the selection window can also be quite different as a function of the selected component. One typical example is given just below.



To add a particular element to the selected element, click on the button corresponding to the element that must be added. Confirm this choice by means of the 'OK' button.

5.2.3 Removing elements

Removing elements from a PowerConnect model is done by selecting first the element with the left-hand mouse button and by then clicking with the right-hand mouse button on the element, which will make the following window pop up. Then use the 'Remove element' icon, and confirm by means of the 'OK' button.





5.3 Definition of elements

5.3.1 General information

Before all relevant dialogue windows for all PowerConnect element types will be discussed in detail, attention will be given to a number of items that are common for a wide range of windows.

5.3.1.1 Default values for elements

Next to the 'OK', 'Cancel' and 'Help'-buttons, each dialogue window includes 2 icon buttons:  . Those icons allow to manage the default settings for the element that can be defined in the active window. Default settings for all element types supported by PowerConnect are accessible at any time through the menu 'Edit' – 'Default values'. For each of the supported element types, a dialogue window can be opened in which default values can be defined or changed.

What is then the purpose of both icons above?

- the first icon will read the default values that have previously been defined for the selected type of element, and it will apply them to the selected element.
- the second icon will save the values that have just been defined for the selected element as the default values for the corresponding element type.

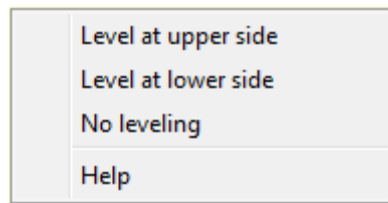
5.3.1.2 Symmetric elements

Within some dialogue windows, the option 'Symmetric element' may be visible in the lower left corner of the window. This option will appear only in case of double-sided connections (as eg. a beam-colum-beam connection). By selecting this option, any modification defined for one side of the connection will automatically be applied also to its symmetric counterpart.

5.3.1.3 Leveling

In case of 'dubble-sided beam to column flange' connections and 'beam to beam' connections with different beam cross-sections, the user is able to level the connected elements.

By means of the Geometry window's floating menu, which can be made visible by clicking the right-hand mouse button within the window, it is possible to choose one of the following three options:



The first option levels the upper beam flanges, while the second option levels the lower beam flanges.

5.3.2 Bar elements

5.3.2.1 H- or I- sections

Dialogue windows related to the definition of bar elements appear whenever the user double-clicks on a bar element with H- or I-section. Those dialogue windows automatically adapt to the type of connection and to the type of bar that has been selected. In particular, the graphical representations which are included in the dialogue window depend on the type of connection.

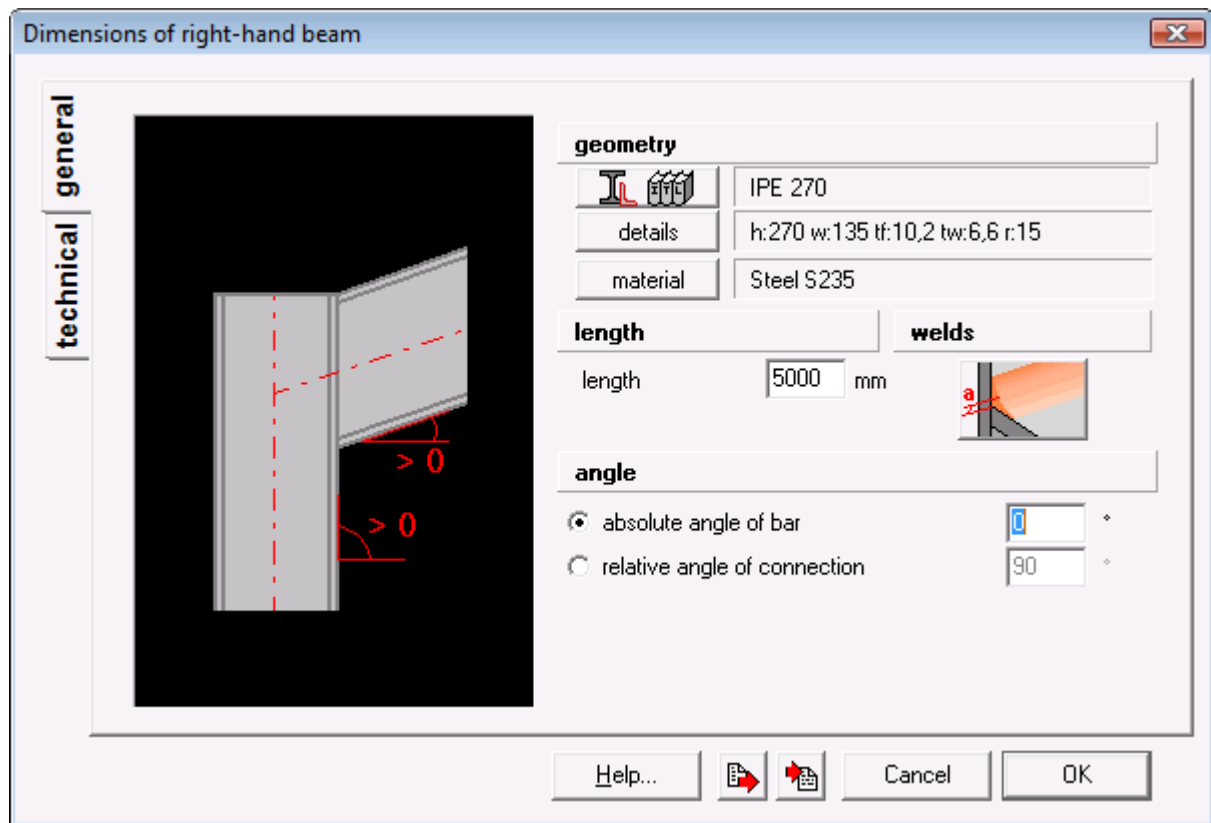
In general, those dialogue window include up to 2 tab pages:

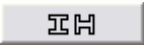
- General
- Technical or Details

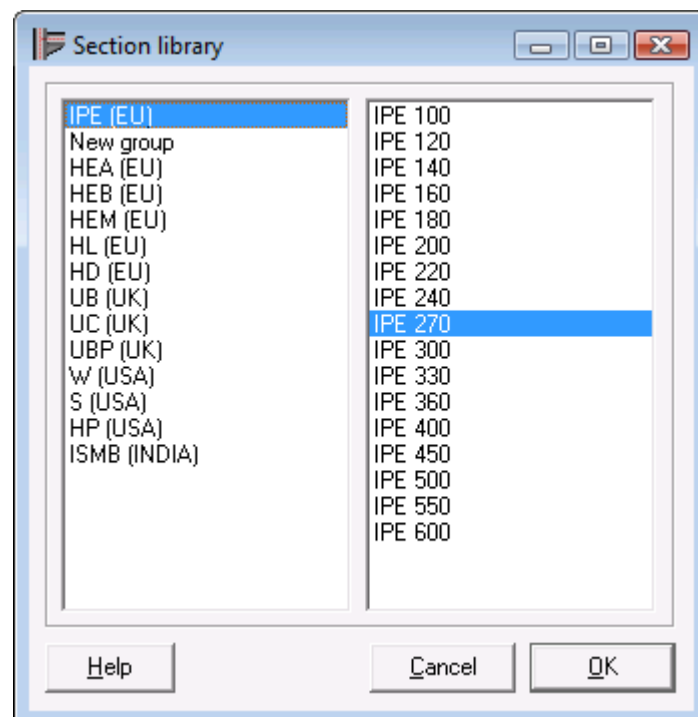
The functions included with those pages are presented in detail below. The tab page "General" will always be available, while the presence of the other tabs depends on the type of connection that has been selected. Both tabs Technical/Details will never be present simultaneously.

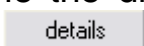
5.3.2.1.1 Tab page "General"


This tab page will always be the active tab page whenever the dialogue window is opened, but its contents may change slightly with the type of connection being selected. The most common functions are documented below.



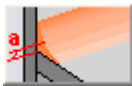
Use the button  to define a section from the PowerConnect section library, through the following selector window which pops up.



In case the user wants or needs to modify or refine the dimensions of the currently selected section, rather use the button . It will then be possible to edit all dimensions individually and change section name, to create a new section independent of the PowerConnect section library.

Finally, the button  allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

Next, the length of the bar element can be defined. In case the selected bar meets the flanges of an adjoining bar at one of its ends, only 1 length can be specified (being of course the total length of the bar). In all other cases, it is also possible to define an additional length, being the length of the bar at the other side of the node where both bars meet. This additional length may have an impact on the calculated bending resistance.

In case the bar is eg. welded to an end plate, an icon  is available to specify detailed information on those welds. More information can be found in the section 5.3.3.3 of this reference manual, which is dedicated to this topic.

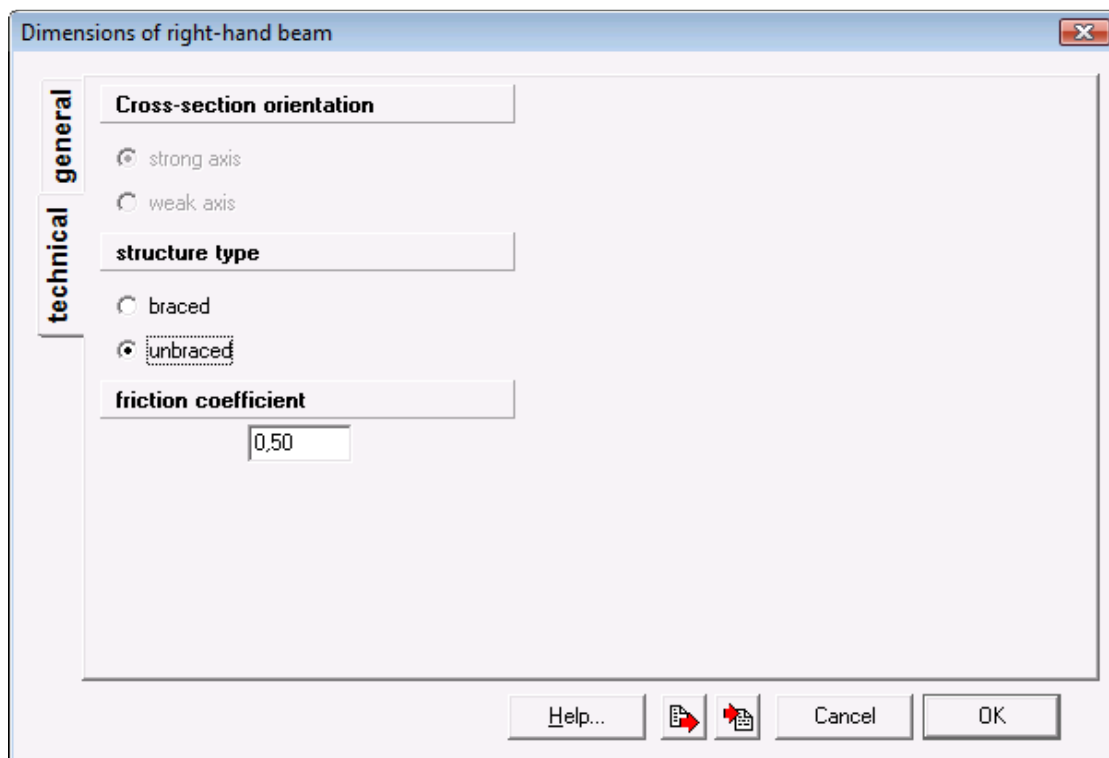
In case it is possible to define a slope angle for a selected bar, up to 2 scenarios may be available to define this slope:

- either through the absolute angle of the bar element with respect to a horizontal axis,
- or through the relative angle between both bar elements which are connected at the node.

In some cases, it may be possible to define the position of a bar element by means of an eccentricity or intermediate distance between connected bar element.

5.3.2.1.2 Tab page “Technical”

In case the selected bar element is a column, it is possible to define the section orientation.

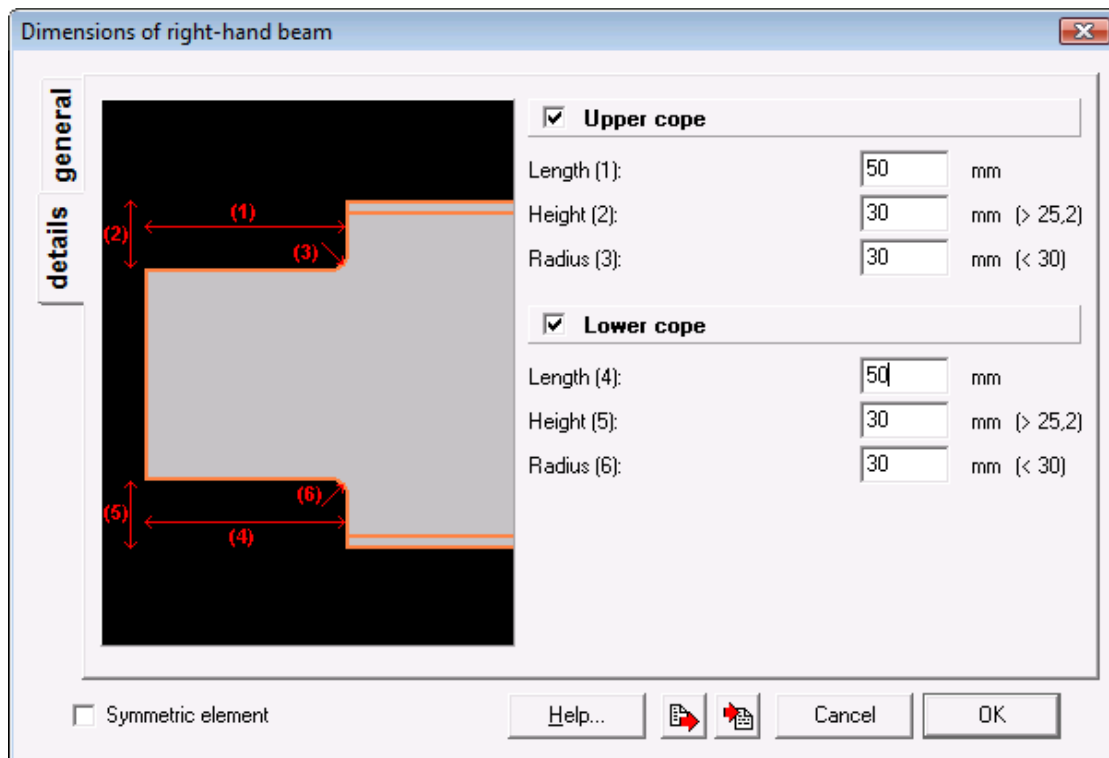


The specification braced vs. unbraced is an important parameter used to determine whether the connection should be classified as rigid or semi-rigid.

Finally, a friction coefficient can be defined to take into consideration during the evaluation of maximum shear force.

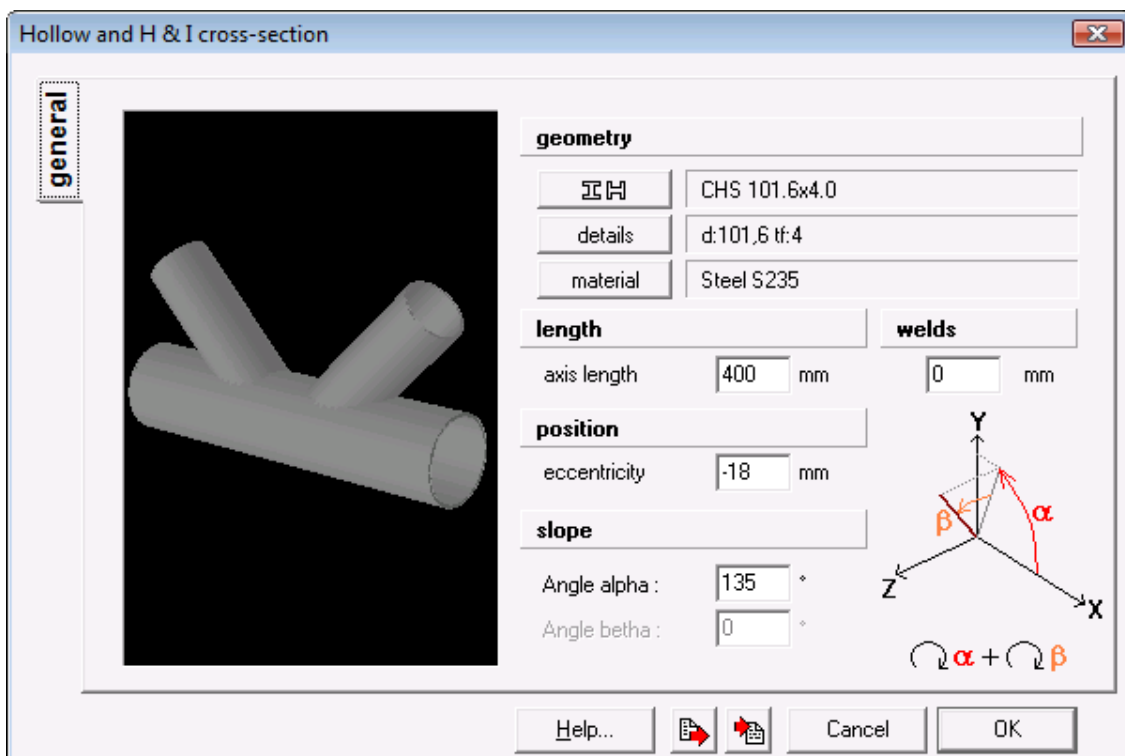
5.3.2.1.3 Tab page “Details”

This tab page is entirely dedicated to the definition of a number of dimensions in case bars are coped to fit in between the flanges of the other bar element. The meaning of all dimension parameters is clearly documented on the adjoining graph. At the same time, PowerConnect will propose minimum or maximum values to be respected, to ensure the dimensions of both connected bars allow for a proper fit.



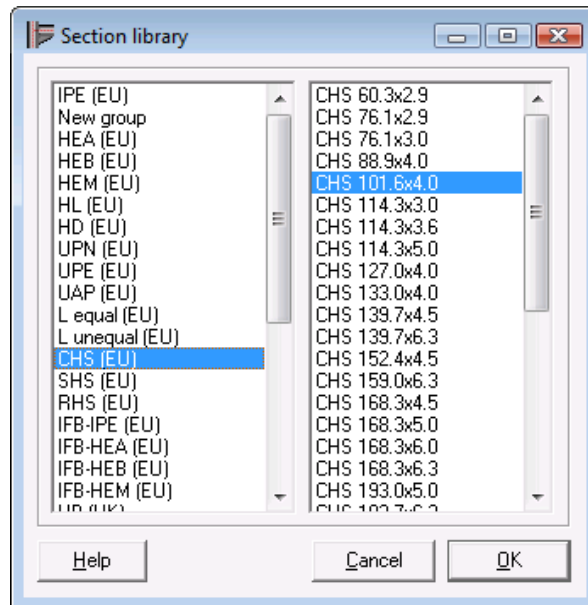
5.3.2.2 Hollow structural sections


Dialogue windows related to the definition of hollow structural sections appear whenever the user double-clicks on a bar element with such a type of section. The contents of those dialogue windows may be a bit different, depending on the type of connection and on the type of bar that has been selected. In particular, the graphical representations which are included in the dialogue window depend on the type of connection.

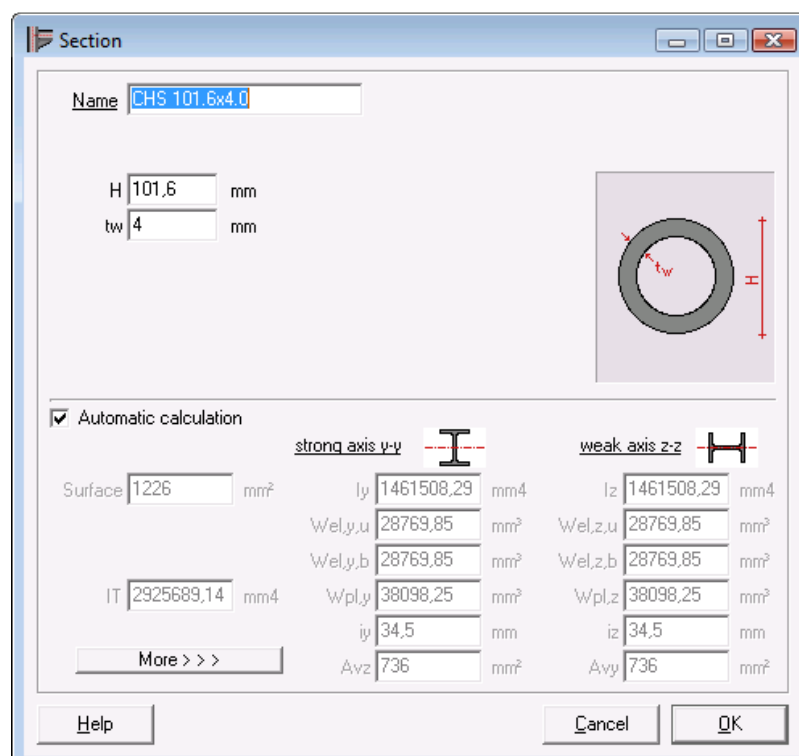



The dialogue window offers 2 function buttons to define or modify section properties.

First, the  button, gives access to the PowerConnect section library.



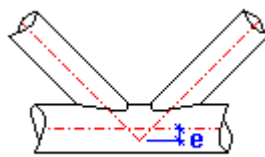
The second button  allows to modify or refine the dimensions of the currently selected section. It is possible to edit all dimensions individually and change section name, to create a new section independent of the PowerConnect section library.



Finally, the button  allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

Next to the above section properties, the dialogue window allows to specify following values:

- bar length, which is purely for documentation purposes as this parameter does not play any role in the connection design analysis
- thickness of welds
- eccentricity of principal bar (a positive value corresponds to a downward shift of the bar)

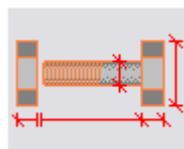


Finally, two input fields remain available to define the spatial orientation of the hollow section member by means of the appropriate orientation angles α and β . This definition is supported by a graphical representation which includes the definition of both angles. For 2D connection, the angle β remains equal to zero.

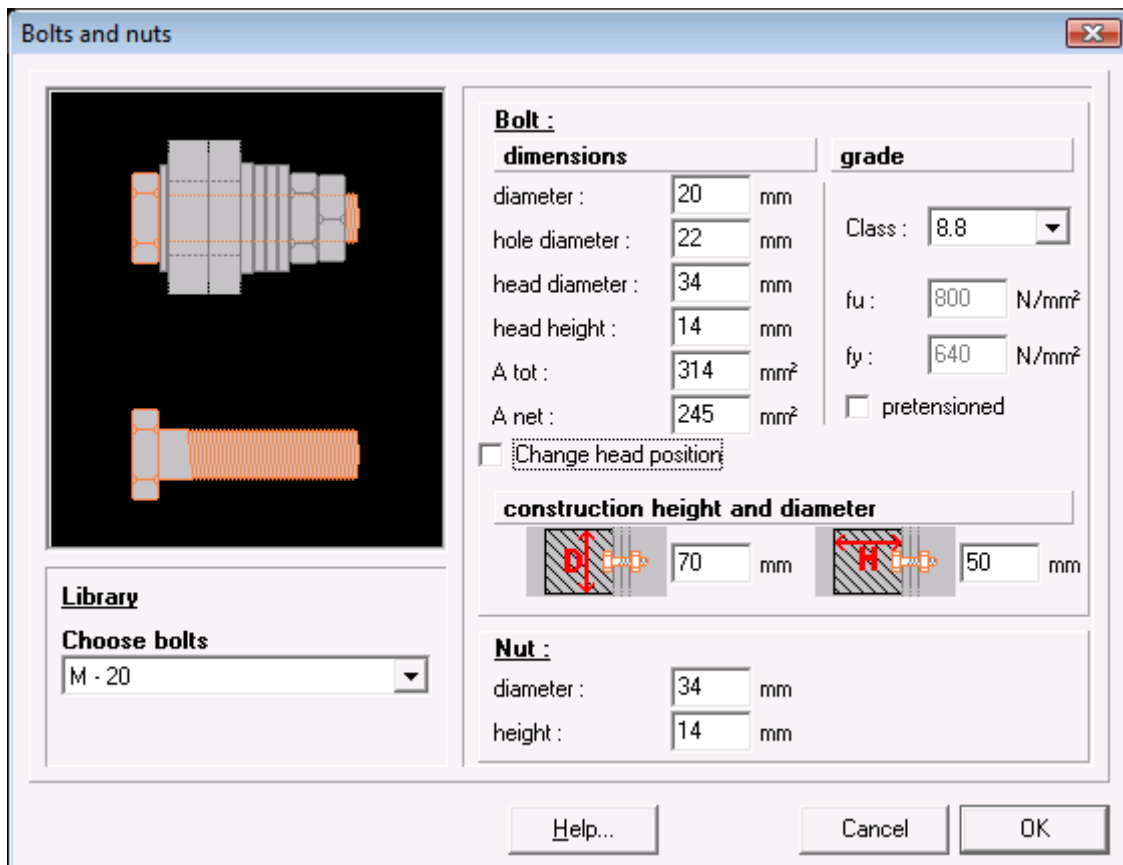
5.3.3 Connecting elements

5.3.3.1 Bolts

Each dialogue window that allows to define bolt row lay-out includes the following button related to the specification of bolt properties. (It should be noted that the dialogue window for bolt row lay-out definition will pop up whenever the user double-clicks on a bolt in the 'Geometry' window. This dialogue window will be discussed in more detail in the remainder of this chapter).



Using the above button will present the following window:



To start with, the user can select a bolt type from the PowerConnect bolts library by means of the pull-down menu at the bottom left-hand side.

As it may occur that the user wants to modify some of the parameters that are assigned to the selected bolt type in the bolts library, it is possible to edit all relevant bolt parameters in the right-hand part of the above dialogue window.

It can be remarked that not all parameters will necessarily intervene in the design analysis. In principle, only total and net bolt section, bolt grade and pretensioning will have a direct influence on the analysis results. All other bolt dimensions are mainly used for the graphical representation of the connection. The option 'Change head position' needs no further explanation.

As PowerConnect's bolt row lay-out functions will perform all necessary checks on minimal bolt distances, the bolt definition includes a number of parameters to define the free space which is needed around the bolt head to enable a correct positioning of the bolt. Those parameters are referred to as the construction height & diameter H & D, and are defined in the input fields next to the following icons:

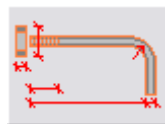


The values which are shown by PowerConnect are taken from the bolts library, and correspond to the minimum values which are usually specified by suppliers. In the current dialogue, the user can change those parameters in the current dialogue window (if the user hasn't done this already in the bolts library directly) if there is a need to deviate from the default supplier specifications.

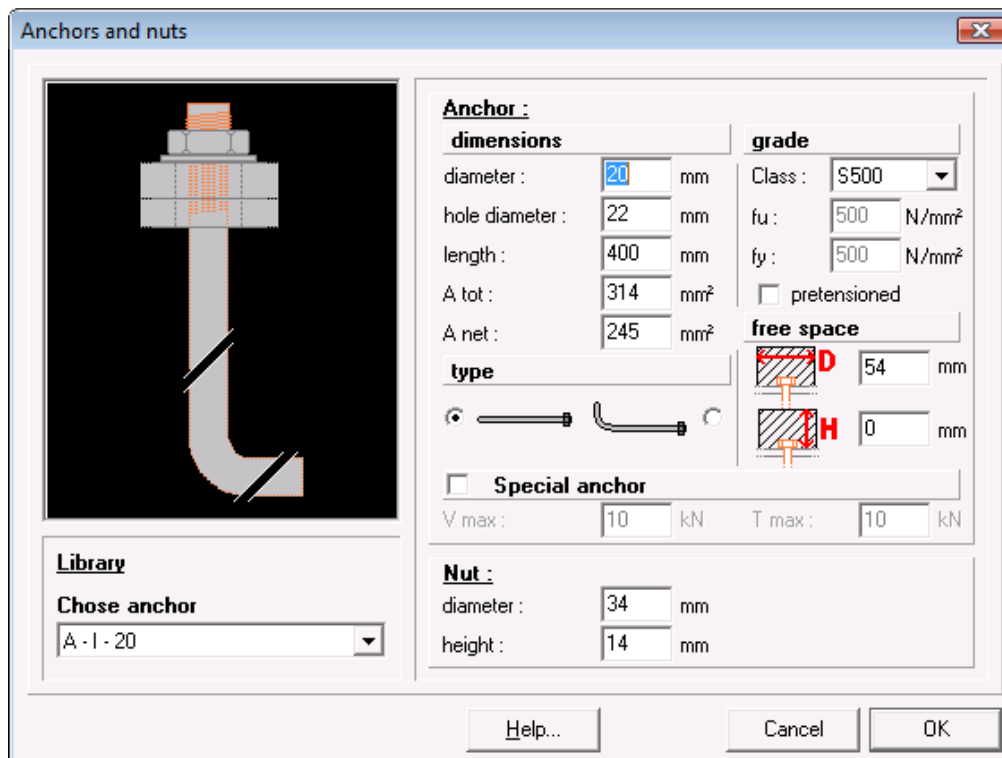
Finally, the bolt grade can be defined by selecting the appropriate grade from the pull-down menu on the right-hand side. Depending on the selected grade, the yield strength f_y and the ultimate tensile strength f_u will automatically be displayed. It is not possible to edit those values, unless the user selects a grade 'Others' from the pull-down menu.

5.3.3.2 Anchor bolts

Each dialogue window that allows to define anchor bolt row lay-out includes the following button related to the specification of anchor bolt properties. (It should be noted that the dialogue window for anchor bolt row lay-out definition will pop up whenever the user double-clicks on an anchor bolt in the 'Geometry' window. This dialogue window will be discussed in more detail in the remainder of this chapter).



Using the above button will present the following window:



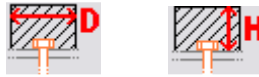
To start with, the user can select an anchor bolt type from the PowerConnect anchor bolts library by means of the pull-down menu at the bottom left-hand side.

The PowerConnect anchor bolt library includes 2 different types of anchor bolts: straight anchor bolts ("I") and bent anchor bolts ("L"). In the case of bent anchor bolts, the length specified in the above dialogue is the length of the straight part.

As it may occur that the user wants to modify some of the parameters that are assigned to the selected anchor bolt type in the anchor bolts library, it is possible to edit all relevant anchor bolt parameters in the right-hand part of the above dialogue window.

It can be remarked that not all parameters will necessarily intervene in the design analysis. In principle, only total and net anchor bolt section, anchor bolt grade and pretensioning will have a direct influence on the analysis results. All other anchor bolt dimensions are mainly used for the graphical representation of the connection.

As PowerConnect's anchor bolt row lay-out functions will perform all necessary checks on minimal anchor bolt distances, the anchor bolt definition includes a number of parameters to define the free space which is needed around the bolt head to enable a correct positioning of the anchor bolt. Those parameters are referred to as the construction height & diameter H & D, and are defined in the input fields next to the following icons:



The values which are shown by PowerConnect are taken from the anchor bolts library, and correspond to the minimum values which are usually specified by suppliers. In the current dialogue, the user can change those parameters in the current dialogue window (if the user hasn't done this already in the anchor bolts library directly) if there is a need to deviate from the default supplier specifications.

The anchor bolt grade can be defined by selecting the appropriate grade from the pull-down menu on the right-hand side. Depending on the selected grade, the yield strength f_y and the ultimate tensile strength f_u will automatically be displayed. It is not possible to edit those values, unless the user selects a grade 'Others' from the pull-down menu.

Next to the straight and bent anchor bolt types, the user can also define special anchor bolt types directly through the specification of maximum shear force and maximum tensile force. This function allows to use any type of anchor bolt system in the PowerConnect modeling process (as eg. chemical anchor systems).

5.3.3.3 Welds

Dialogue windows that allow to define the properties of welds may differ as a function of the type of connection being analyzed. A dialogue window for the definition of welds will pop up whenever the user double-clicks on a weld in the 'Geometry' window. In all cases, the dialogue will allow full editing of all detail properties.

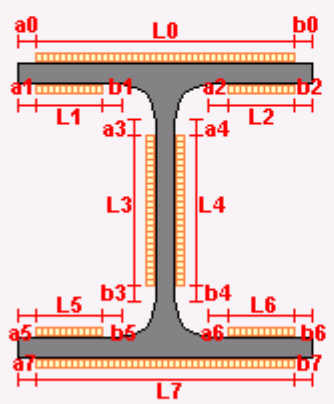
Throat thicknesses for welds on a flange or a web can be defined independently.

Welds

throat thickness (flange) : mm

throat thickness (web) : mm

☒ Automatic calculation of welding lengths



welds in mm

a0	<input type="text" value="0"/>	L0	<input type="text" value="135"/>	b0	<input type="text" value="0"/>
a1	<input type="text" value="0"/>	L1	<input type="text" value="49,2"/>	b1	<input type="text" value="0"/>
a2	<input type="text" value="0"/>	L2	<input type="text" value="49,2"/>	b2	<input type="text" value="0"/>
a3	<input type="text" value="0"/>	L3	<input type="text" value="219,6"/>	b3	<input type="text" value="0"/>
a4	<input type="text" value="0"/>	L4	<input type="text" value="219,6"/>	b4	<input type="text" value="0"/>
a5	<input type="text" value="0"/>	L5	<input type="text" value="49,2"/>	b5	<input type="text" value="0"/>
a6	<input type="text" value="0"/>	L6	<input type="text" value="49,2"/>	b6	<input type="text" value="0"/>
a7	<input type="text" value="0"/>	L7	<input type="text" value="135"/>	b7	<input type="text" value="0"/>

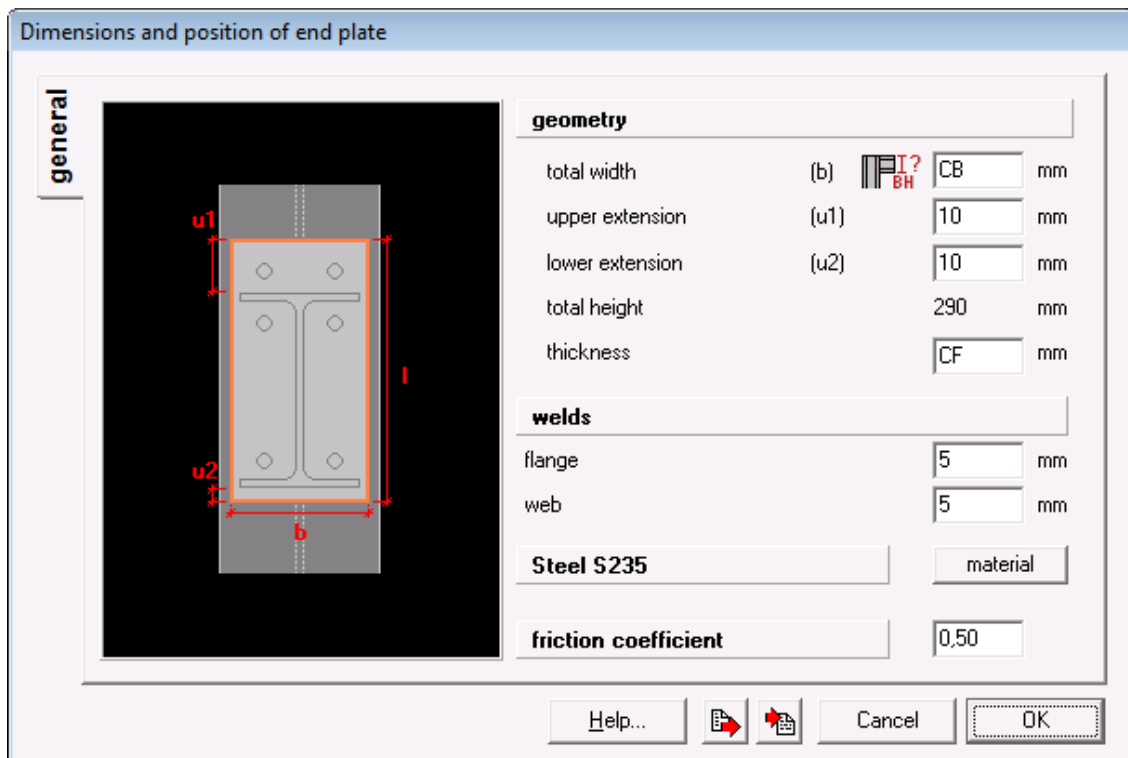
Help Cancel OK

In case the option 'Automatic calculation of welding lengths' is selected, PowerConnect will automatically calculate the lengths of all welds. If, on the other hand, this option is not selected, then the user will have the opportunity to edit all lengths individually in the right-hand part of the dialogue window. To remove a specific weld, it is sufficient to specify a zero value for the corresponding length (L0...L7).

5.3.3.4 Bolted end plates (moment connections)

5.3.3.4.1 End plate


The dialogue window that allows to define the properties of a bolted end plate will pop up whenever the user double-clicks on the end plate in the 'Geometry' window.



Only a limited set of parameters are available for editing: total width, length of upper & lower extension, and plate thickness. Total height is calculated by PowerConnect taking into consideration the specific configuration of the connection (in particular, the dimensions of the connected bars) and the imposed extension lengths.

The throat thickness of the welds on flange and web can be specified manually in the fields below, while the steel grade used for the end plate can be defined through the button (giving access to all steel grades available in the material library).

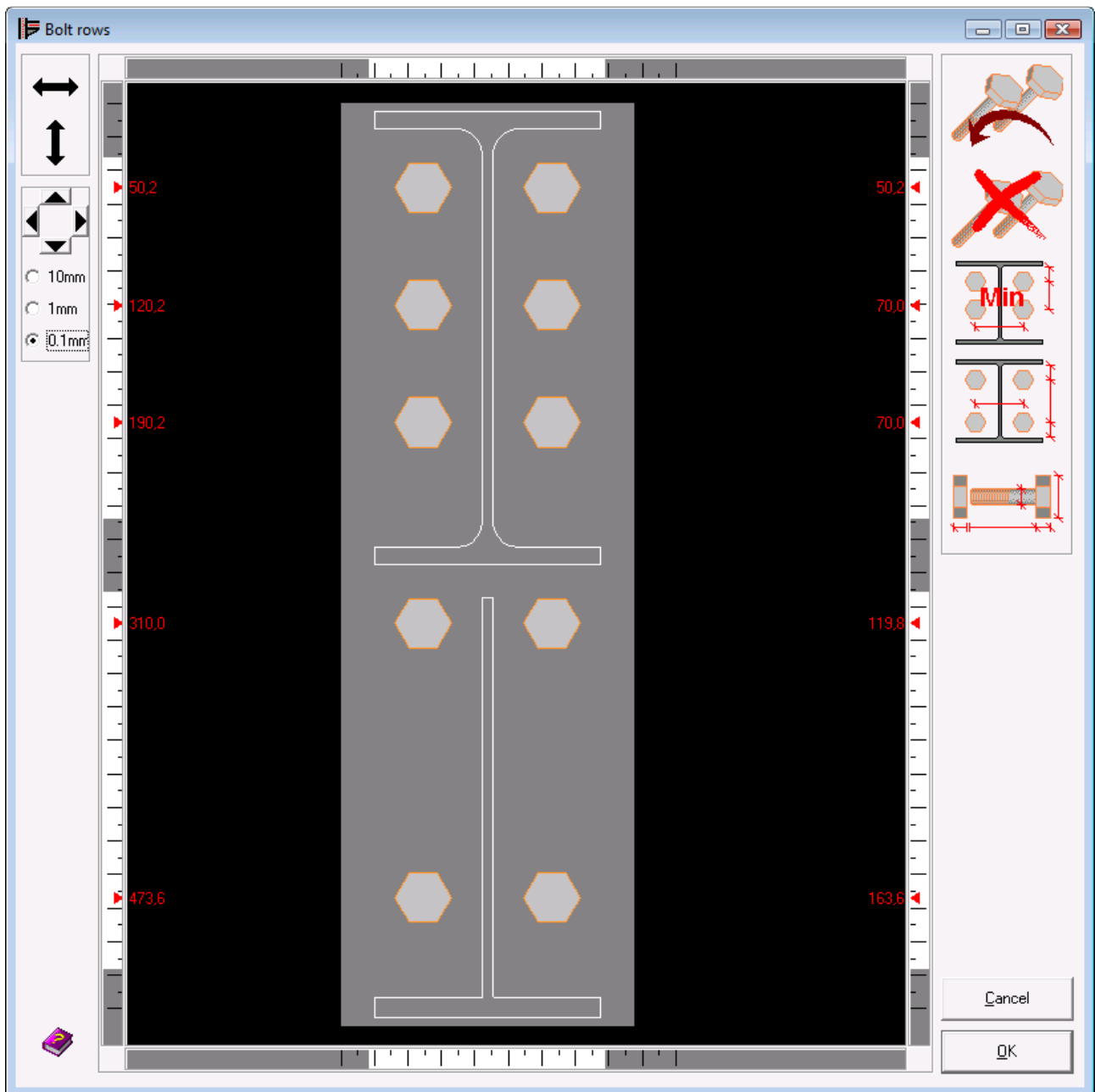
The friction coefficient specified in the above dialogue window will intervene in the calculation of shear resistance.



It is observed that some input fields contain parameters rather than numbers. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the  icon in the above dialogue window.

The previously described parameters are referred to as “characteristic distances” by PowerConnect. More information on characteristic distances is found in section 6.2 of this reference manual.

5.3.3.4.2 Bolt configuration

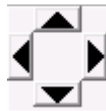
The dialogue window that allows to define the bolt configuration will pop up whenever the user double-clicks on a bolt in the ‘Geometry’ window.



To modify the position of a specific bolt row, click on one of the bolts belonging to this row, and then initiate either the horizontal  or vertical  displacement of the selected bolts by means of the appropriate buttons

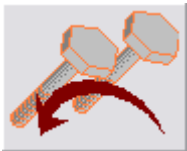
in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window



to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows shown above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

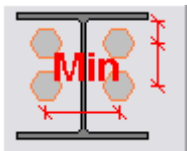
The bolt row dialogue window contains 5 other buttons with dedicated functionality.



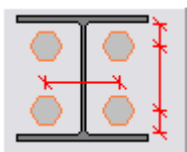
Use this button to add a new bolt row below all existing rows.



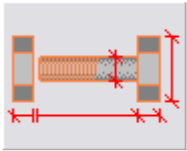
Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



Use this button to optimize bolt row positions fully compliant with the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user. During this repositioning, PowerConnect will take into account the presense of stiffeners, haunches, ...



Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user, but at the same time maximizing the distance in between bolt rows. During this repositioning, PowerConnect will take into account the presense of stiffeners, haunches, ...



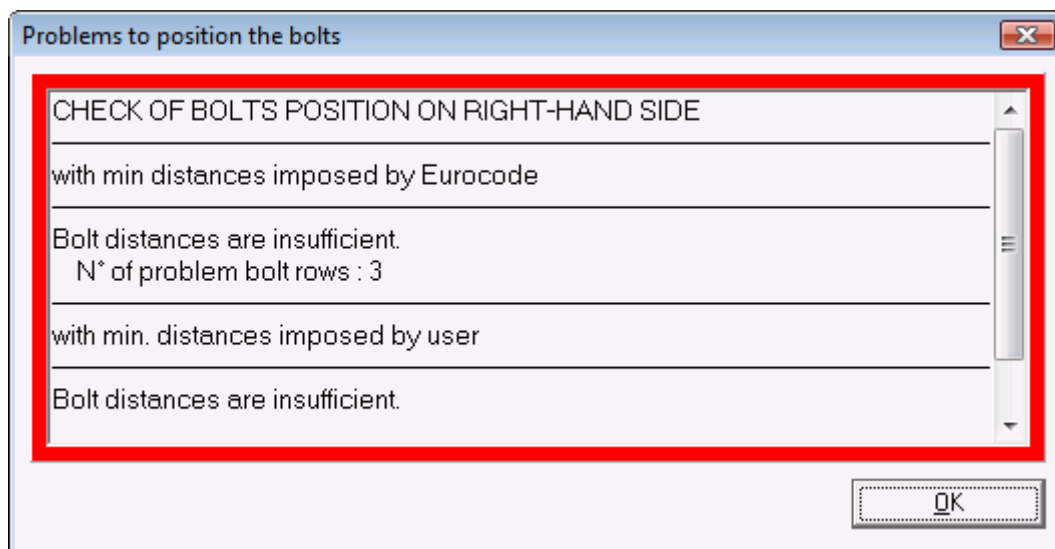
Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

The strips to the left / to the right / below / on top of the central part of the dialogue window are partly colored white. Those parts of the strips correspond to valid bolt positions (considering distances with respect to flanges or web that must be respected to allow for proper bolt positioning). No bolts can be added or positioned in the areas corresponding to the grey parts of the strips.

Once all necessary operations have been performed in the current dialogue window, PowerConnect will automatically perform a number of verifications on bolt positions. More in particular, following checks are performed:

- first, it is verified whether the positions comply with minimum distance requirements imposed by the current design standard (eg. Eurocode 3) are respected,
- then, it is verified whether the positions comply with minimum distance requirements imposed by the user are respected.

If at least one of those verifications is not successful, this will be reported to the user as can be seen below.



5.3.3.5 Bolted angle cleats

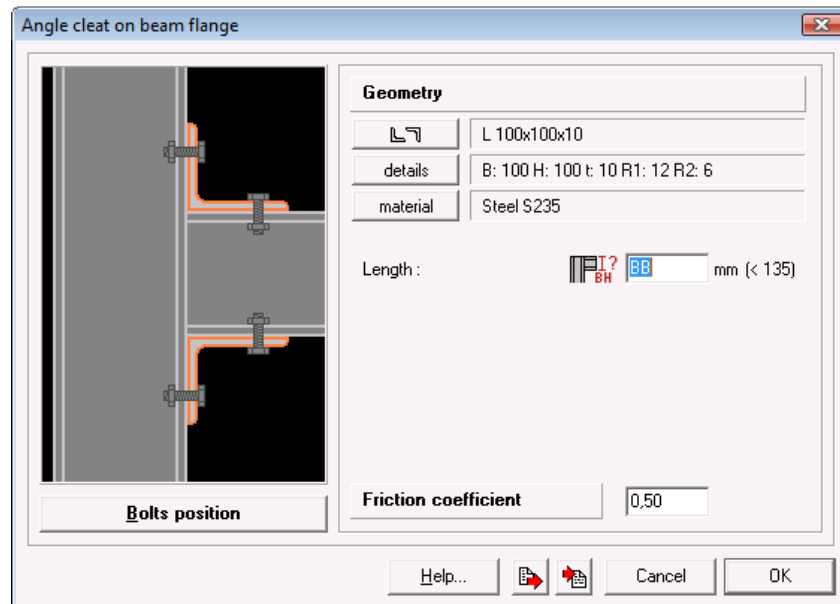
Two types of angle cleats can be distinguished :

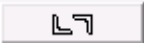

- those connected to the flanges of a H- or I-section;
- those connected to the web of a H- or I-section.

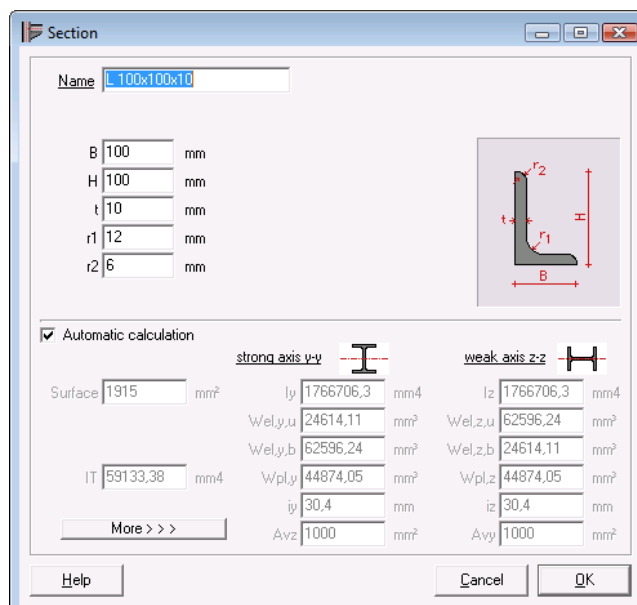
5.3.3.5.1 Angle cleats bolted to beam flange


5.3.3.5.1.1 Cleats

The dialogue window that allows to define the properties of bolted cleats will pop up whenever the user double-clicks on a cleat bolted to a flange in the 'Geometry' window.



Use the button  to define a section from the PowerConnect section library. In case the user wants or needs to modify or refine the dimensions of the currently selected section, rather use the button . It will then be possible to edit all dimensions individually and change section name, to create a new section independent of the PowerConnect section library.



The  button allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

The input field labeled 'Length' corresponds to the total length of the cleat. This length should not exceed the maximum value indicated at the right hand side of the input field.

It is observed that this input field may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

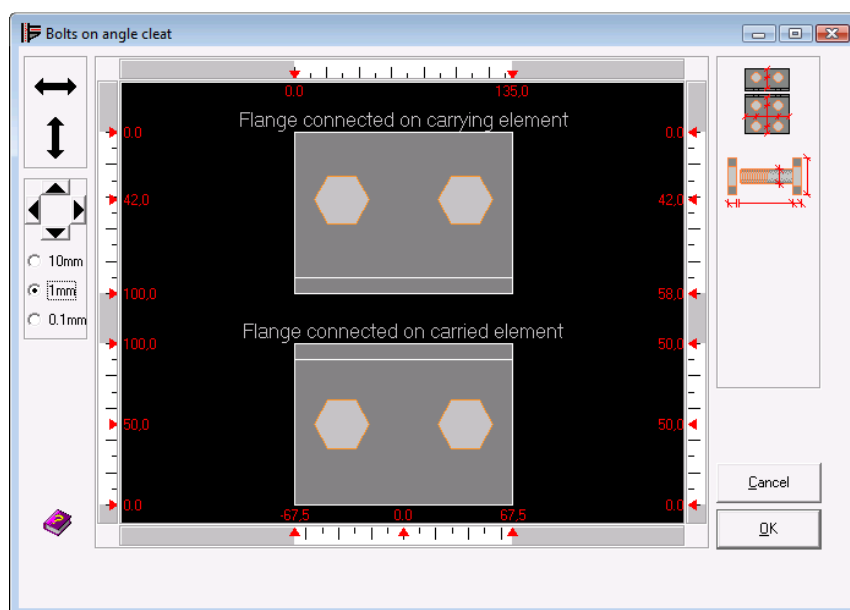



icon in the above dialogue window.

Finally, a friction coefficient can be defined to take into consideration during the evaluation of maximum shear force.

In case there is a need for immediate repositioning of the bolts, use the button 'Bolts position' just below the graphical part of the dialogue window. Of course, bolt can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.

5.3.3.5.1.2 Bolt configuration

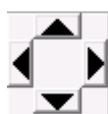


To modify the position of a specific bolt row, click on one of the bolts belonging to this row, and then initiate either the horizontal  or vertical



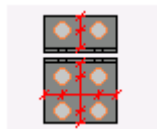
displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window

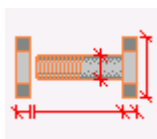


to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows shown above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

The bolt row dialogue window contains 2 other buttons with dedicated functionality.



Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user.



Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

The strips to the left / to the right / below / on top of the central part of the dialogue window are partly colored white. Those parts of the strips correspond to valid bolt positions (considering distances with respect to flanges that must be respected to allow for proper bolt positioning). No bolts can be added or positioned in the areas corresponding to the grey parts of the strips.

Once all necessary operations have been performed in the current dialogue window, PowerConnect will automatically perform a number of verifications on bolt positions. More in particular, following checks are performed:

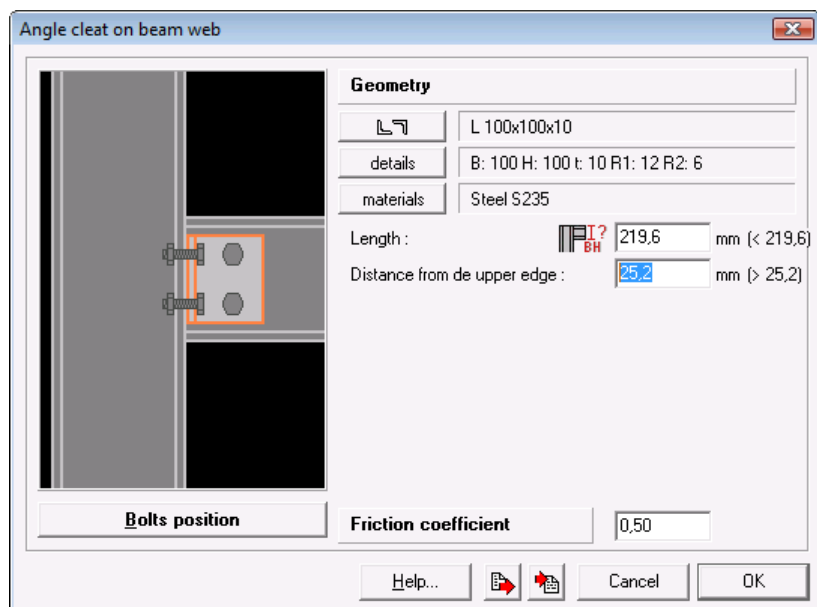
- first, it is verified whether the positions comply with minimum distance requirements imposed by the current design standard (eg. Eurocode 3) are respected,
- then, it is verified whether the positions comply with minimum distance requirements imposed by the user are respected.

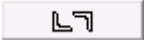
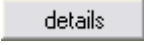
If at least one of those verifications is not successful, this will be reported clearly to the user.


5.3.3.5.2 Angle cleats bolted to beam web

5.3.3.5.2.1 Cleats

The dialogue window that allows to define the properties of bolted cleats will pop up whenever the user double-clicks on an angle cleat bolted to a beam web in the 'Geometry' window.



Use the button  to define a section from the PowerConnect section library. In case the user wants or needs to modify or refine the dimensions of the currently selected section, rather use the button . It will then be possible to edit all dimensions individually and change section name, to create a new section independent of the PowerConnect section library.

The  button allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

The input field labeled 'Length' corresponds to the total length of the cleat. This length should not exceed the maximum value indicated at the right hand side of the input field. This maximum length also takes into account the

distance between the outer edge of the angle cleat and the edge of the beam web. This distance may be edited by the user in the corresponding input field.

It is observed that both input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

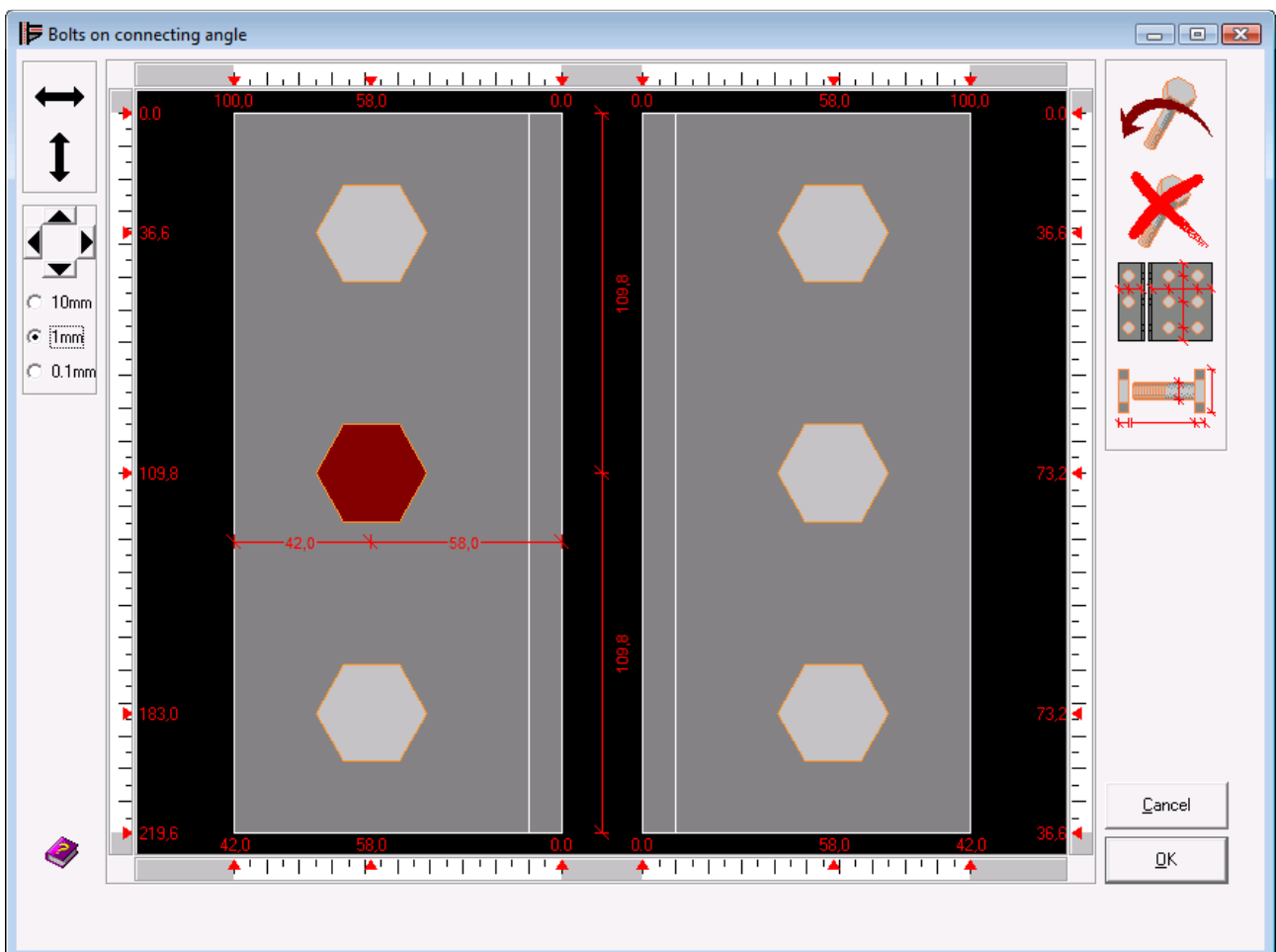


icon in the above dialogue window.

Finally, a friction coefficient can be defined to take into consideration during the evaluation of maximum shear force when pretensioned bolts are used.

In case there is a need for immediate repositioning of the bolts, use the button 'Bolts position' just below the graphical part of the dialogue window. Of course, bolt can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.

5.3.3.5.2.2 Bolt configuration

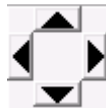


To modify the position of a specific bolt row, click on one of the bolts belonging to this row, and then initiate either the horizontal



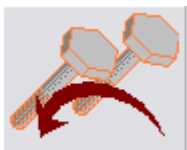
or vertical displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window



to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows shown above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

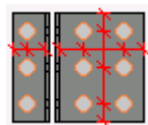
The bolt row dialogue window contains 4 other buttons with dedicated functionality.



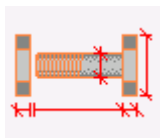
Use this button to add a new bolt row below all existing rows.



Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user.



Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

The strips to the left / to the right / below / on top of the central part of the dialogue window are partly colored white. Those parts of the strips correspond to valid bolt positions (considering distances with respect to flanges that must be respected to allow for proper bolt positioning). No bolts can be added or positioned in the areas corresponding to the grey parts of the strips.

Once all necessary operations have been performed in the current dialogue window, PowerConnect will automatically perform a number of verifications on bolt positions. More in particular, following checks are performed:

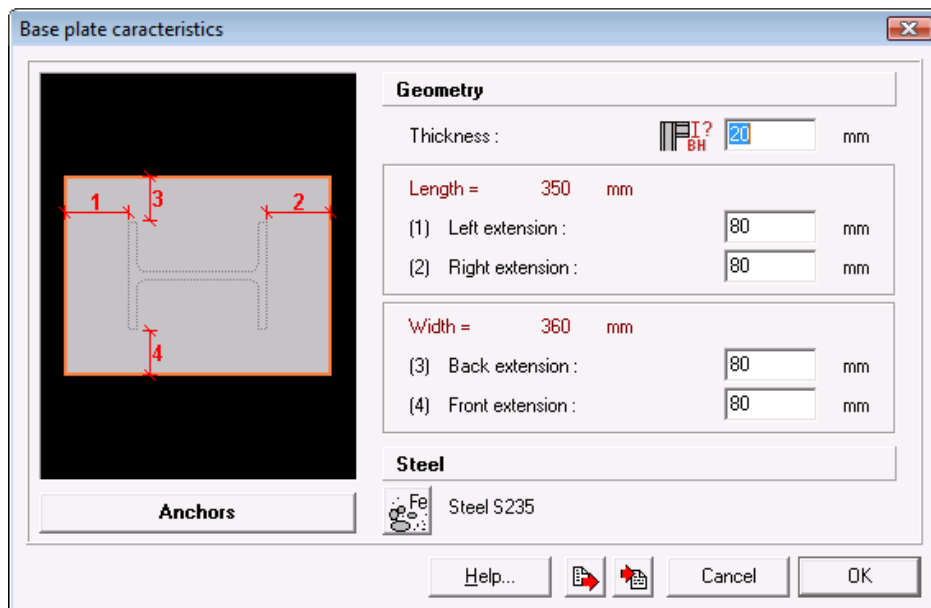
- first, it is verified whether the positions comply with minimum distance requirements imposed by the current design standard (eg. Eurocode 3) are respected,
- then, it is verified whether the positions comply with minimum distance requirements imposed by the user are respected.

If at least one of those verifications is not successful, this will be reported clearly to the user.

5.3.3.6 Column base plates

5.3.3.6.1 Base plate

The dialogue window that allows to define the properties of a base plate will pop up whenever the user double-clicks on the base plate in the 'Geometry' window.



Only a limited set of parameters are available for editing: plate thickness and the length of left/right & upper/lower extensions. Total length and width in the above dialogue are automatically calculated by PowerConnect depending on the dimensions of the column section, and cannot be edited directly by the user.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

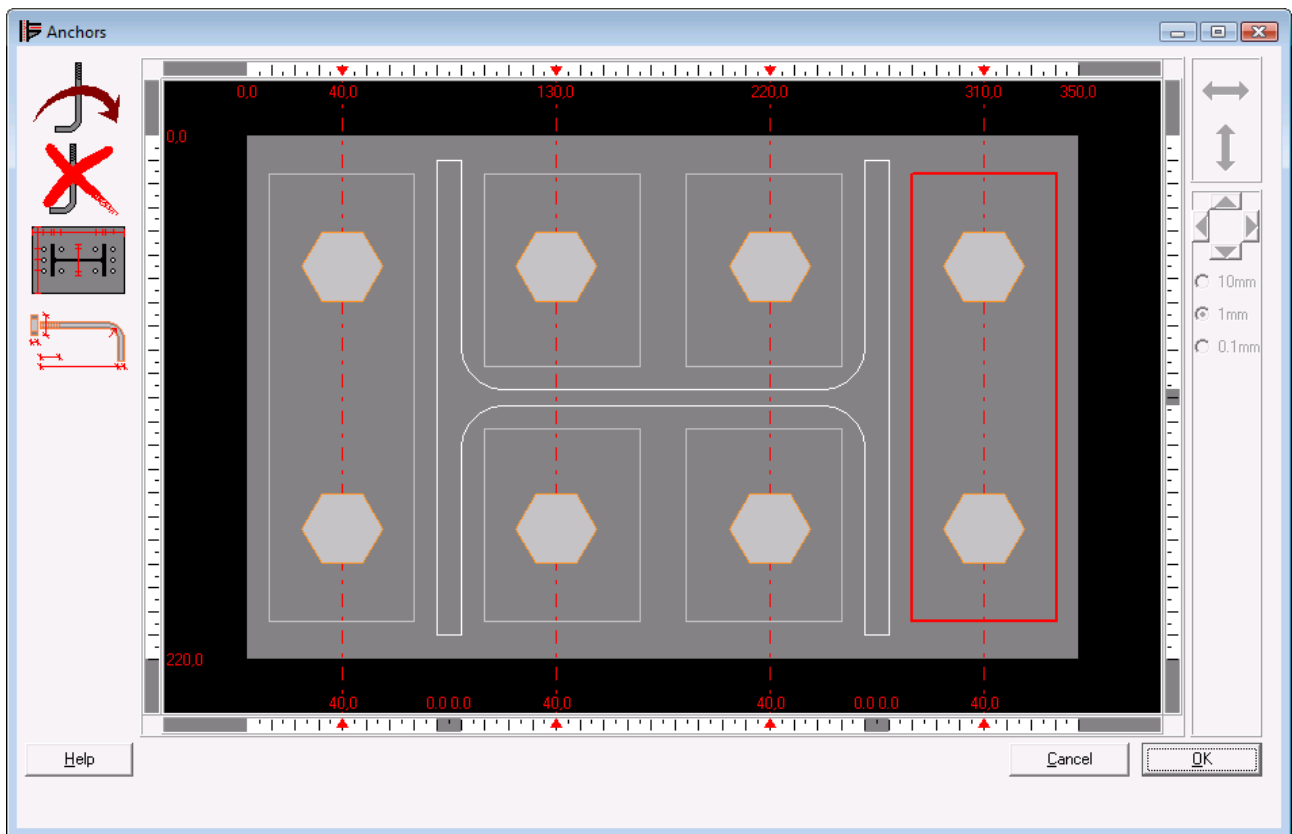


icon in the above dialogue window.

The steel grade used for the base plate can be defined through the material icon button (giving access to all steel grades available in the material library).

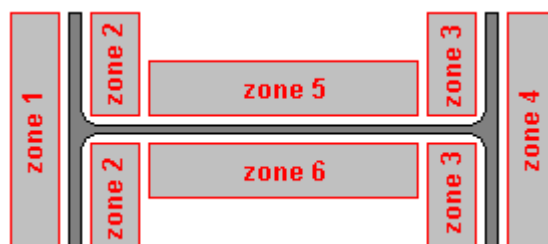
In case there is a need for immediate repositioning of the anchor bolts, use the button 'Anchors' just below the graphical part of the dialogue window. Of course, anchor bolts can also be repositioned at any time by double-clicking on one of the anchor bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.

5.3.3.6.2 Anchor bolt configuration





This dialogue window is very similar to the one to define bolt positioning. Nevertheless, a number of functions are specifically related to the definition of an anchor bolt lay-out.

The figure below shows a number of well-defined zones, in which a number of anchor bolts can be defined. In total, six zones can be distinguished – one zone at each side of the flanges and the web of the column section.

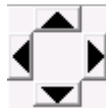


Each zone is represented by a rectangle (or possibly by 2 rectangles). Select a particular zone by just clicking on the zone with the mouse. In that case, the zone's circumference will color red.

It should be remarked that not necessarily all of the above 6 zones are displayed in the graphical part of the above dialogue window. PowerConnect will automatically eliminate zones in which no anchor bolts can be defined (in case there is insufficient space to allow for the bolts to be placed in practice).

To modify the position of a specific anchor bolt row, click on one of the anchor bolts belonging to this row, and then initiate either the horizontal  or vertical  displacement of the selected anchor bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of anchor bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected anchor bolt row, read the dimension annotations which are shown next to the graphical representation of the anchor bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune anchor bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window



to move the selected anchor bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows shown above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

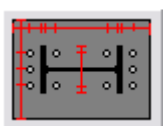
The anchor bolt row dialogue window contains 4 other buttons with dedicated functionality.



Use this button to add a new anchor bolt row below all existing rows.

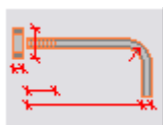


Use this button to remove an anchor bolt, after having selected the anchor bolt row that needs to be eliminated. In case a zone is selected, rather than an anchor bolt, this function will remove the entire zone.



Use this button to optimize anchor bolt row positions respecting the minimum anchor bolt distances imposed by the design standard (eg. Eurocode 3) or by the user. When this button is being used, PowerConnect will always propose a symmetric anchor bolt configuration when possible. The

anchor bolt positioning will take into account the presence of base plate stiffeners.



Use this icon to access the dialogue window which allows to define anchor bolt properties (refer to section 5.3.3.2 for more information).

The strips to the left / to the right / below / on top of the central part of the dialogue window are partly colored white. Those parts of the strips correspond to valid anchor bolt positions (considering distances with respect to flanges that must be respected to allow for proper anchor bolt positioning). No anchor bolts can be added or positioned in the areas corresponding to the grey parts of the strips.

Once all necessary operations have been performed in the current dialogue window, PowerConnect will automatically perform a number of verifications on bolt positions. More in particular, following checks are performed:

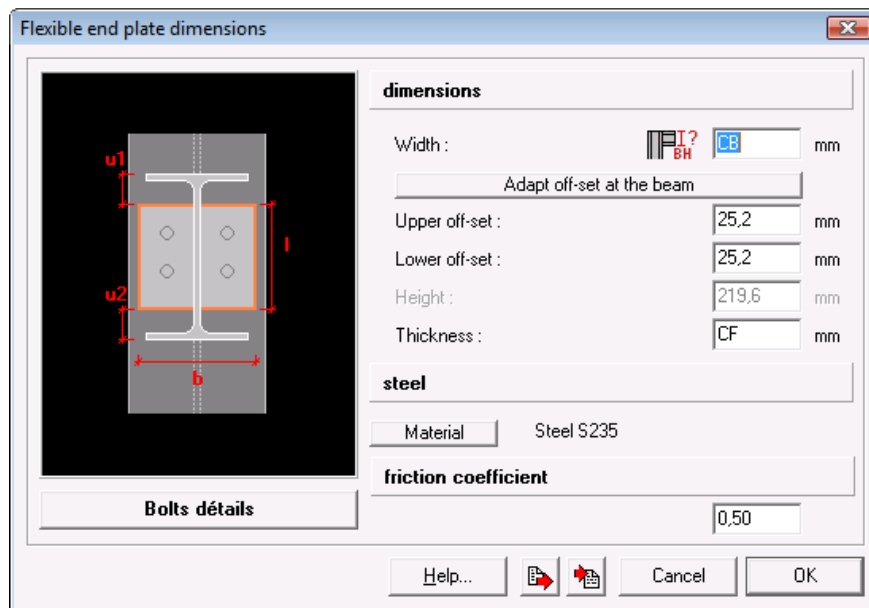
- first, it is verified whether the positions comply with minimum distance requirements imposed by the current design standard (eg. Eurocode 3) are respected,
- then, it is verified whether the positions comply with minimum distance requirements imposed by the user are respected.

If at least one of those verifications is not successful, this will be reported clearly to the user.

5.3.3.7 Bolted end plates (shear connections)

5.3.3.7.1 End plate

The dialogue window that allows to define the properties of bolted end plates for shear connections will pop up whenever the user double-clicks on such an end plate in the 'Geometry' window.




Prior to a more detailed presentation of the above dialogue window, the major difference between end plates for rigid (or semi-rigid) connections and for shear connections will be discussed. For rigid (or semi-rigid) connections, the end plate will have a length which is at least equal to the total height of the beam section, and the end plate will normally extend beyond both beam flanges. For shear connections on the contrary, the end plate will have a limited height and will always be in between both beam flanges, such that the required rotation capacity can be guaranteed.

Only a limited set of parameters are available for editing: total plate width, plate thickness and lower/upper off-set of the end plate with respect to both beam flanges. Based on the known height of the beam element, the total height of the end plate will be calculated considering the previously defined parameters.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



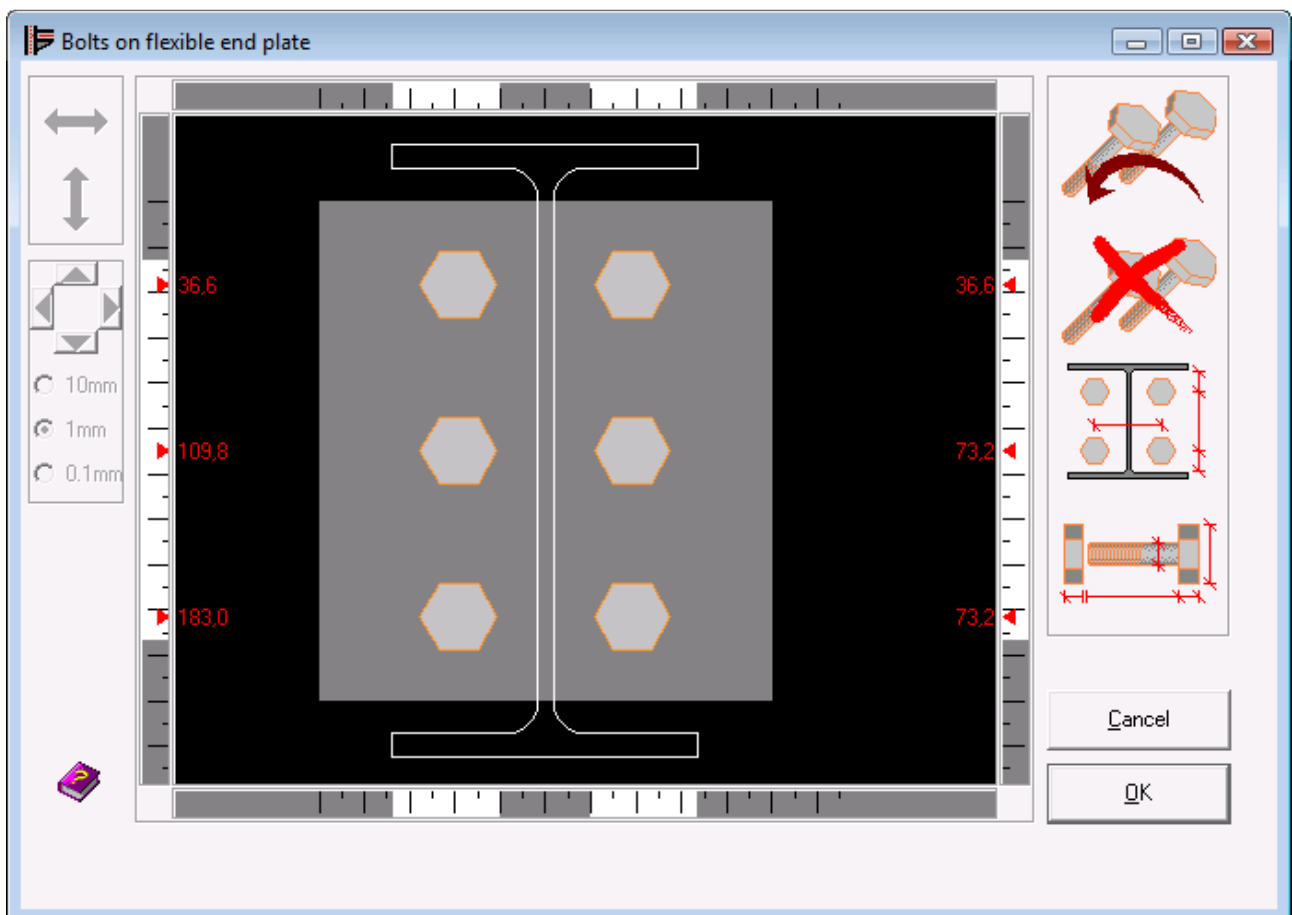
icon in the above dialogue window.

The steel grade used for the base plate can be defined through the  button (giving access to all steel grades available in the material library).

Finally, a friction coefficient can be defined to take into consideration during the evaluation of maximum shear force when pretensioned bolts are used.


In case there is a need for immediate repositioning of the bolts, use the button 'Bolt details' just below the graphical part of the dialogue window. Of course, bolts can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.

5.3.3.7.2 Bolt configuration

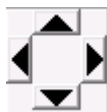


To modify the position of a specific bolt row, click on one of the bolts

belonging to this row, and then initiate either the horizontal  or vertical

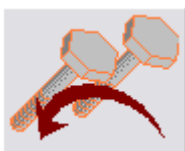
 displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window



to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows show above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

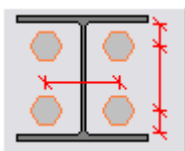
The bolt row dialogue window contains 4 other buttons with dedicated functionality.



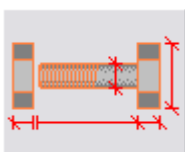
Use this button to add a new bolt row below all existing rows.



Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user, but with maximum intermediate distances between bolt rows.

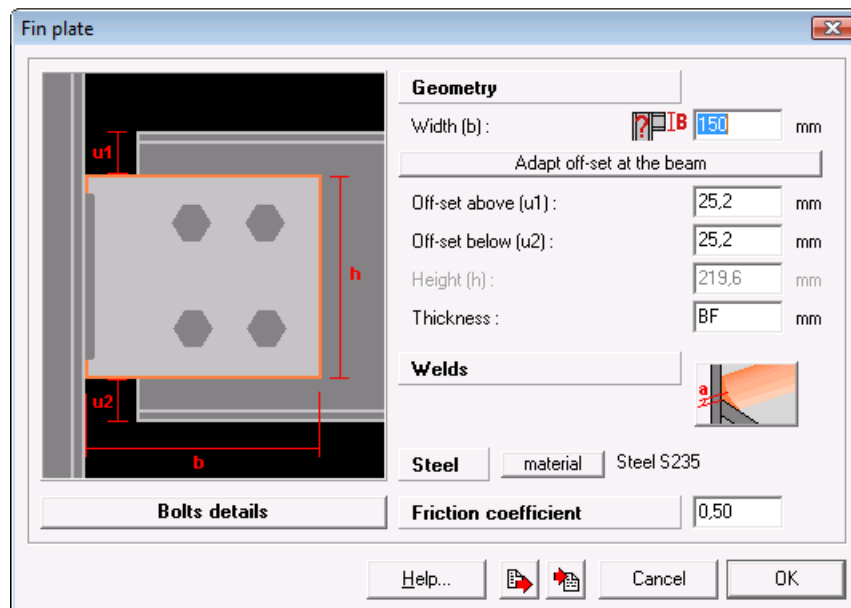


Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

5.3.3.8 Bolted fin plates

5.3.3.8.1 Fin plate

Fin plates are used only as part of shear connections, as no significant bending moment can be developed in this case. The dialogue window that allows to define the properties of bolted fin plates will pop up whenever the user double-clicks on such a fin plate in the 'Geometry' window.



First, the total width of the fin plate can be defined.

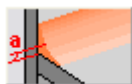
Next, the upper and lower off-set of the fin plate with respect to the outer faces of the beam flanges should be specified. The entry of this data is facilitated by the information included in the graphical part of the dialogue window. As can be seen from the window, the off-set values should take into account the flange thickness and the curvature between beam flange and web. By means of the function 'Adapt off-set', PowerConnect will automatically calculate the minimum off-set values and will derive the corresponding fin plate height (based of course also on the beam height).

Then, the thickness of the fin plate can be specified.


It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.



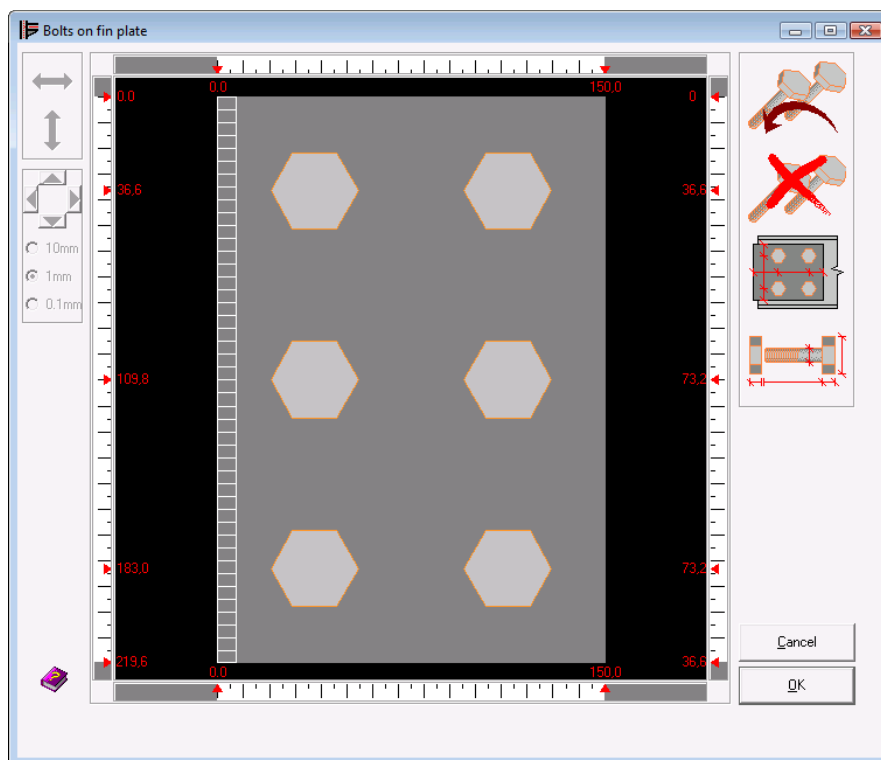
The icon is available to specify detailed information on the welds by which the fin plate is connected to the column flange. More information can be found in the section 5.3.3.3 of this reference manual, which is dedicated to this topic. By default, welds are assumed to extend over the entire height of the fin plate.

The  button allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

Finally, a friction coefficient can be defined to take into consideration during the evaluation of maximum shear force.


In case there is a need for immediate repositioning of the bolts, use the button 'Bolts details' just below the graphical part of the dialogue window. Of course, bolts can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.


5.3.3.8.2 Bolt configuration



A fin plate can contain a number of horizontal bolt rows, but there should at least be 2 rows. Moreover, each bolt row may include one or several bolts.

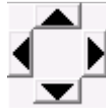
To modify the position of a specific bolt row, click on one of the bolts

belonging to this row, and then initiate either the horizontal  or vertical

 displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row,

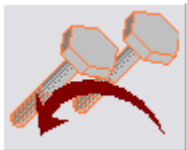
read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window

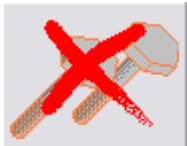


to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows show above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

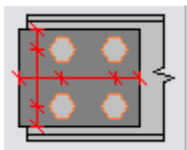
The bolt row dialogue window contains 4 other buttons with dedicated functionality.



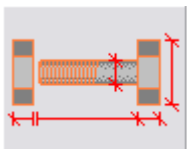
Use this button to add a new bolt row below all existing rows.



Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



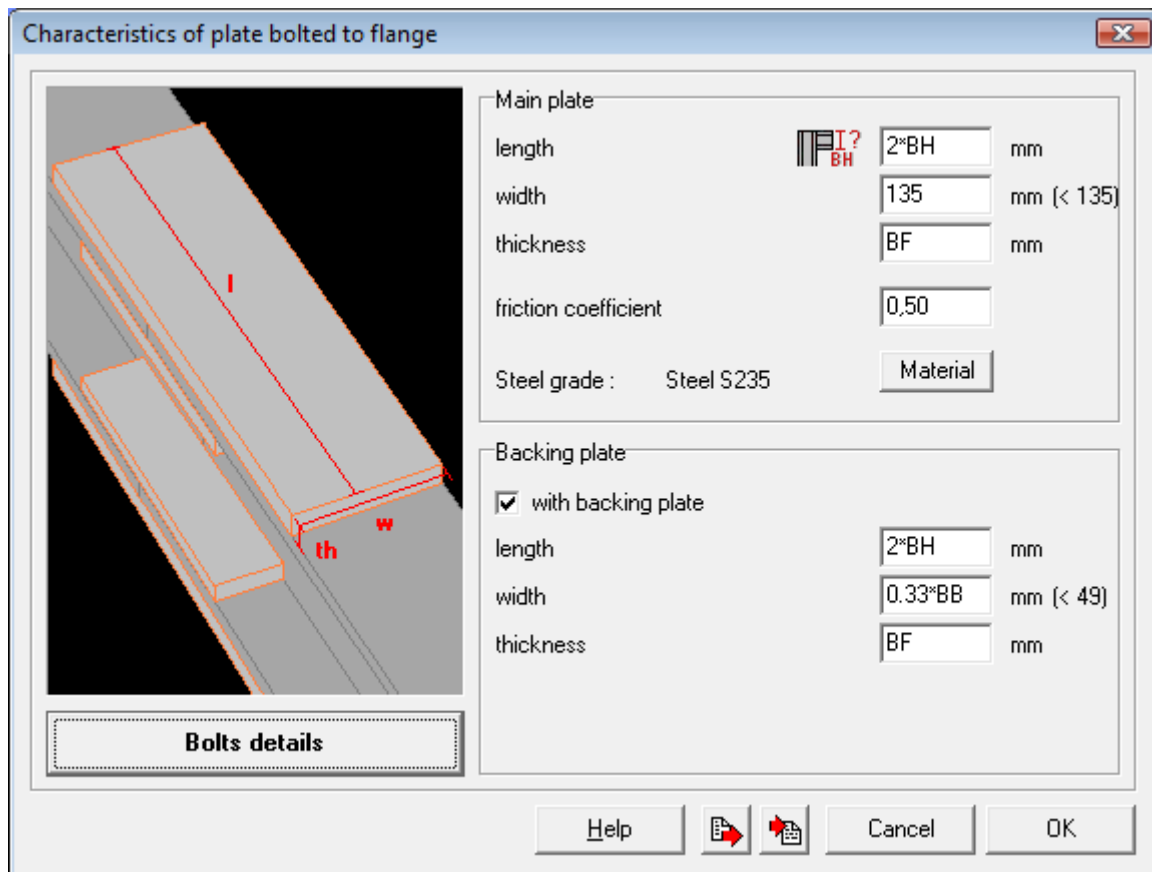
Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user, but with maximum intermediate distances between bolt rows.



Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

5.3.3.9 Bolted flange plates

5.3.3.9.1 Flange plate



Bolted flange plates are typically used with beam splices. Flange plates are necessarily arranged symmetrically with respect to the center line of the splice.


The first 3 input fields are used to enter length, width and thickness of the upper & lower main flange plates. The maximum width is specified as a constraint to assist the user.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.

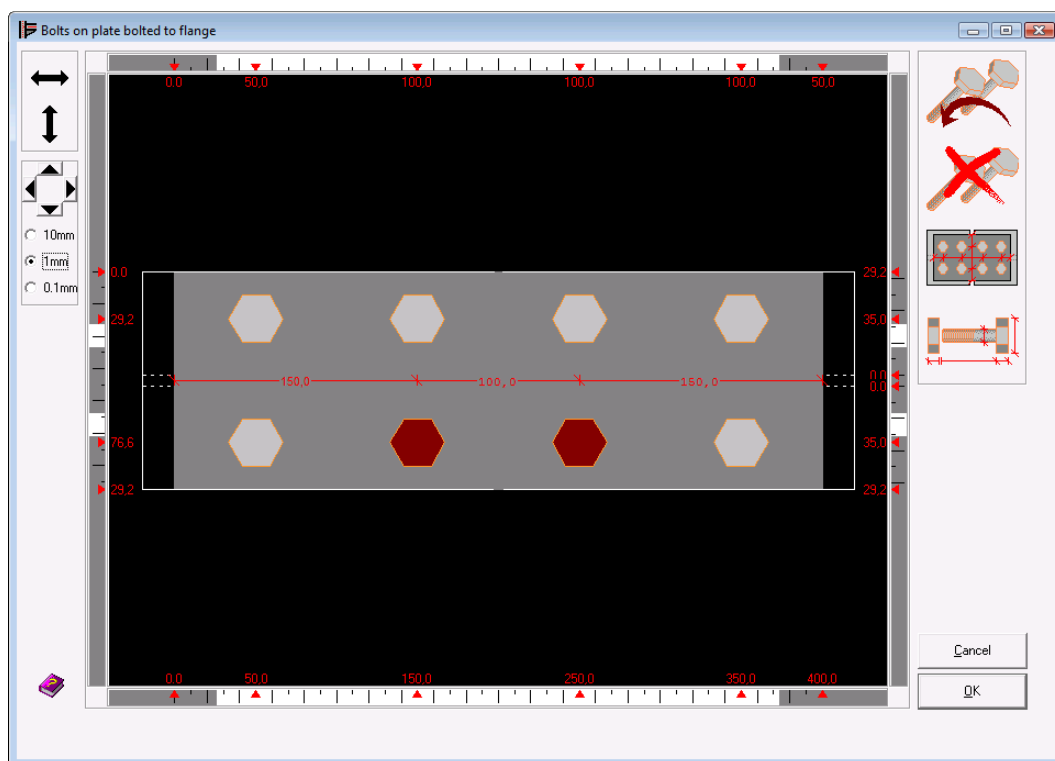
Next, a friction coefficient can be defined to take into the effect of pretensioning the bolts.

The  button allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

By selecting the option 'with backing plate' a backing plate will be added to the inner side of the beam flanges. Use the next 3 input fields to specify the length, width and thickness of this element.


In case there is a need for immediate repositioning of the bolts, use the button 'Bolts details' just below the graphical part of the dialogue window. Of course, bolts can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.


5.3.3.9.2 Bolt configuration



Bolted splices are always considered to be symmetric by PowerConnect. Symmetry also applies to the arrangement of the bolts.

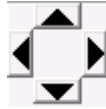
To modify the position of a specific bolt row, click on one of the bolts

belonging to this row, and then initiate either the horizontal  or vertical

 displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may

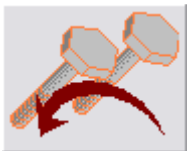
decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window

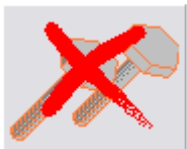


to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows show above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

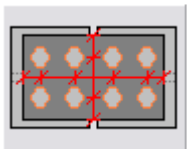
The bolt row dialogue window contains 4 other buttons with dedicated functionality.



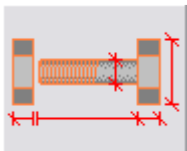
Use this button to add a new bolt row below all existing rows.



Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



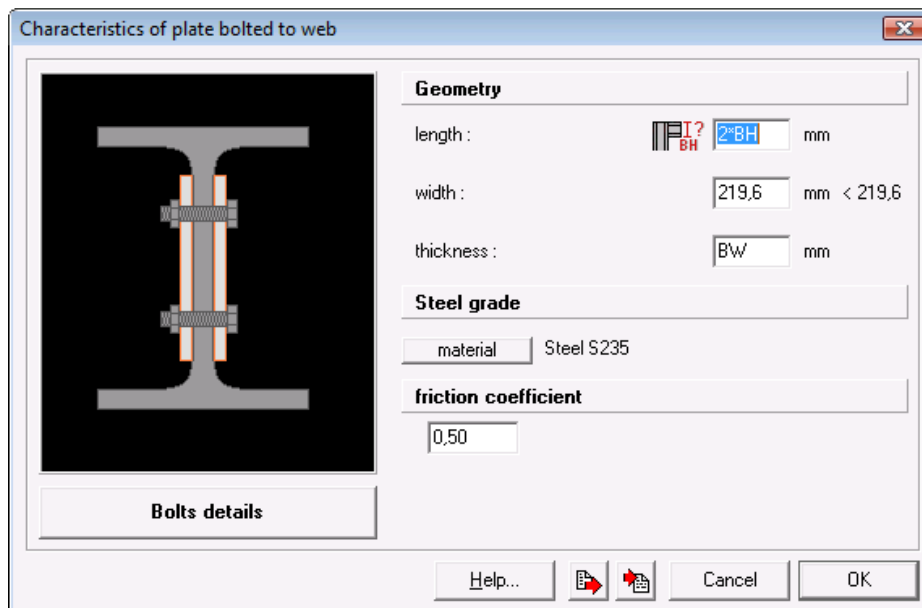
Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user, but with maximum intermediate distances between bolt rows.



Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

5.3.3.10 Bolted web plate

5.3.3.10.1 Web plate




Bolted web plates are typically used with beam splices. Web plates are necessarily arranged symmetrically with respect to the center line of the splice.

The first 3 input fields are used to enter length, width and thickness of the upper & lower flange plates. The maximum width is specified as a constraint to assist the user.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



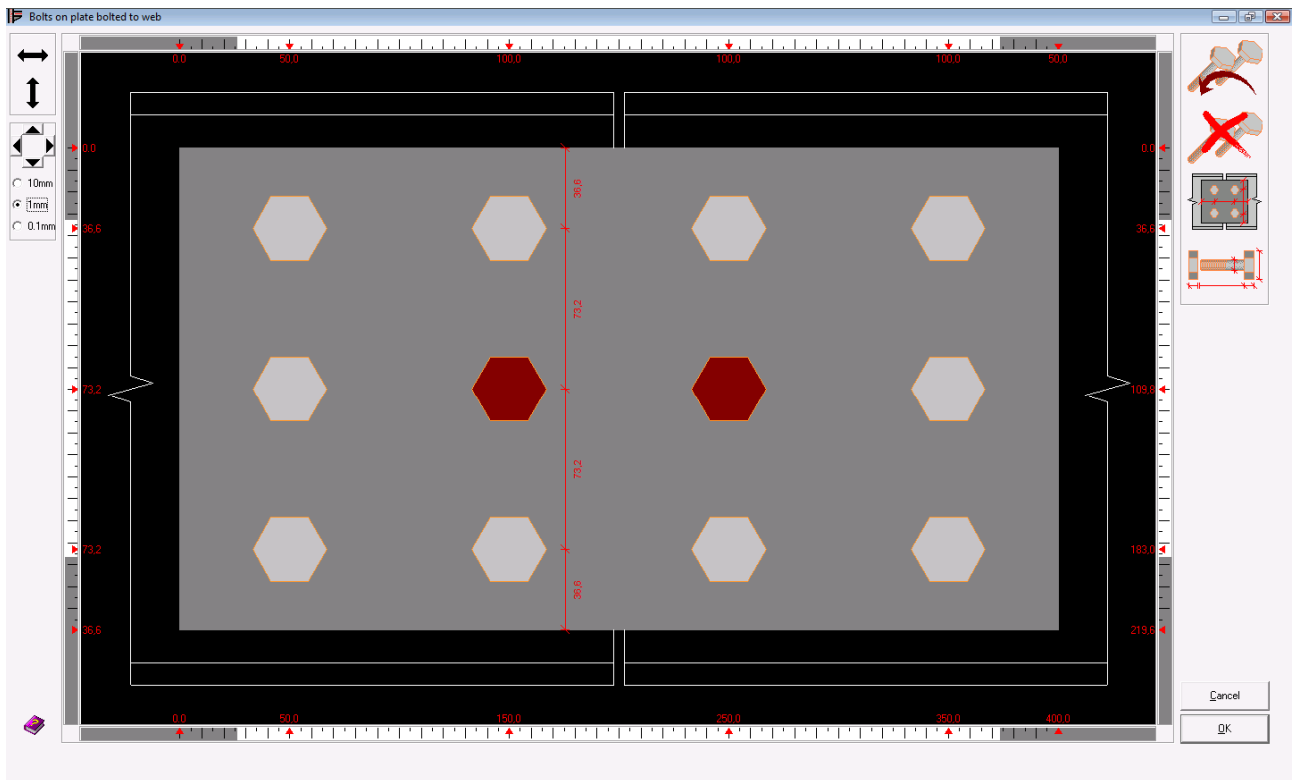
icon in the above dialogue window.



The  button allows to change steel grade by selecting another entry from the list of steel grades currently available in the material library.

Finally, a friction coefficient can be defined to take into the effect of pretensioning the bolts.

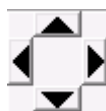
In case there is a need for immediate repositioning of the bolts, use the button 'Bolts details' just below the graphical part of the dialogue window. Of course, bolts can also be repositioned at any time by double-clicking on one of the bolts in the 'Geometry' window. In both cases, the dialogue window presented in the next section will pop up.

5.3.3.10.2 Bolt configuration



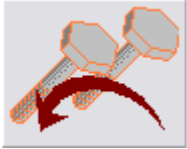
To modify the position of a specific bolt row, click on one of the bolts belonging to this row, and then initiate either the horizontal  or vertical  displacement of the selected bolts by means of the appropriate buttons in the upper right corner of the dialogue window. This function allows for a quick repositioning of bolts, but may not always provide the necessary accuracy. To actually read out the current position of the selected bolt row, read the dimension annotations which are shown next to the graphical representation of the bolt rows. Depending on this information, the user may decide to accept the quick repositioning as it is, and to further fine-tune bolt positions with a number of complementary functions.

It is indeed also possible to use the following icon in the upper left part of the dialogue window

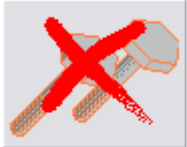


to move the selected bolts step-wise. The step-size can be chosen between the available values of 0.1mm, 1mm or 10mm by selecting the appropriate option. As an alternative to the use of the arrows show above, the user may also use the arrows on the keyboard. The step-size can also be adapted by means of the 'TAB'-button on the keyboard.

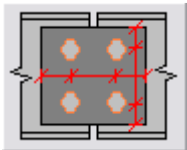
The bolt row dialogue window contains 4 other buttons with dedicated functionality.



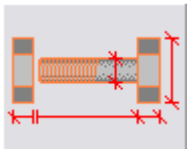
Use this button to add a new bolt row below all existing rows.



Use this button to remove an entire bolt row, after having selected the bolt row that needs to be eliminated.



Use this button to optimize bolt row positions respecting the minimum bolt distances imposed by the design standard (eg. Eurocode 3) or by the user, but with maximum intermediate distances between bolt rows.



Use this icon to access the dialogue window which allows to define bolt properties (refer to section 5.3.3.1 for more information).

5.3.4 Stiffening elements

5.3.4.1 Stiffening elements for columns

5.3.4.1.1 Transverse stiffeners

4 types of transverse column stiffeners can be distinguished:

- an upper stiffener,
- a lower stiffener,
- a diagonal stiffener,
- or an other type of transverse stiffener.

All stiffener types can be defined through the same dialogue window. The stiffener which is actually selected is clearly indicated in colour on the graphical representation, while its type is also selected in the type list to the right of the graphical representation. In some cases, the type of stiffener may still be changed by selecting an other entry from the list.

It should be remarked that for any of the 4 stiffener types listed above, PowerConnect will automatically position 2 stiffener elements at both sides of the column web.

- the corner gap width & length, which do not intervene in the actual connection design analysis but which can be defined to avoid potential problems related to the curved transition between the web and flanges of the column.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.

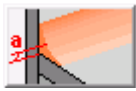

In case of a diagonal stiffener, one extra parameter is shown in the dialogue window:

- the angle of the stiffener with respect to a horizontal reference line,

This parameter cannot be edited directly, but is controlled from within the graphical part of the dialogue window. Through the arrows on the left-hand and the right-hand side of the graphical representation, the vertical position of both ends of a diagonal stiffener can be controlled independently. The step size can be controlled below the graphical representation.

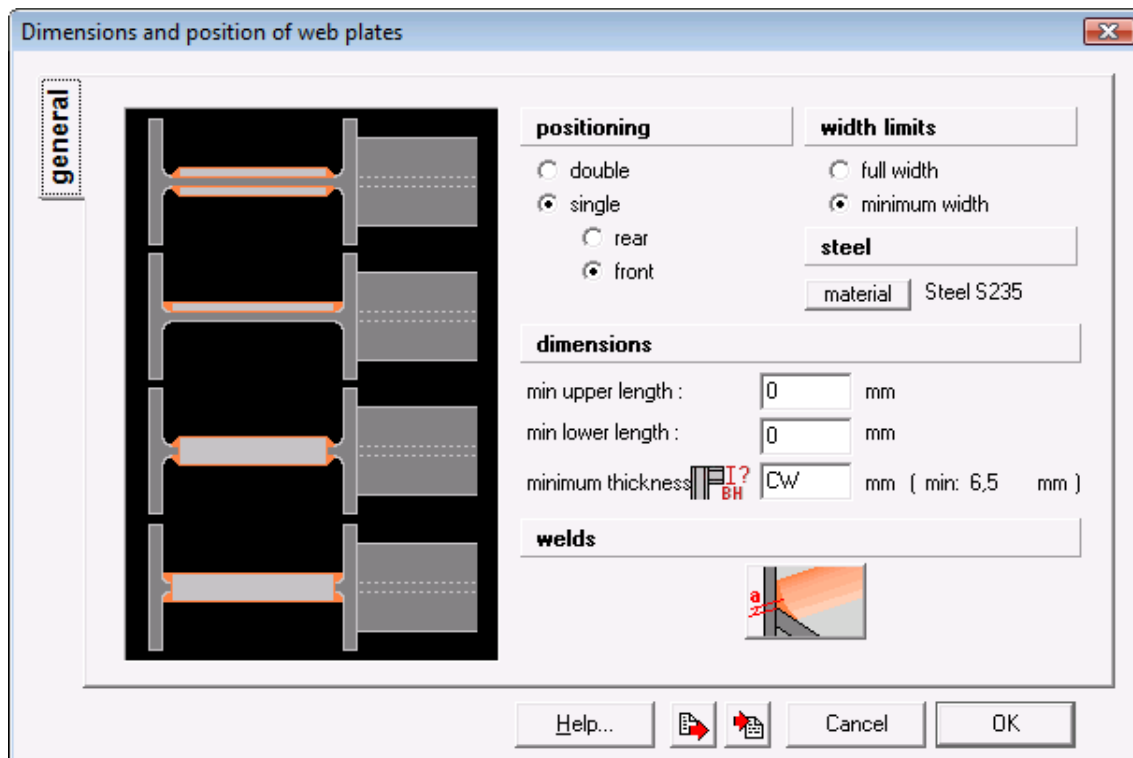
With transverse stiffeners other than lower, upper or diagonal stiffeners, the vertical position of the stiffener can also be controlled graphically.

Finally, two more parameters are available for the definition of all stiffener types:

- the properties of the welds, accessible through the  icon button,
- the steel grade being used for the stiffener, accessible through the  icon button.

5.3.4.1.2 Web plates

Web plates are used whenever the shear force which develops in the column web becomes too large. The use of web plates allows to increase the local stiffness of the column web at the position where the column is connected to a beam element.



Web plate dimensions are mostly determined by the strict rules specified by the Eurocodes. Plate width depends on how the plates are welded to the web (see graphical part of the above dialogue window. In principle, web plates extend over the entire height of the beam-column connection. If needed, an minimum overlength can be specified both at the upper and the lower side of the web plate.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



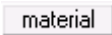
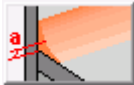
icon in the above dialogue window.

Web plates can be used according to 3 possible schemes:

- either, web plates are used at both the rear and the front side of the column web (double web plate),
- or, a single web plate is used at the front side of the column web only,
- or, a single web plate is used at the rear side of the column web only.

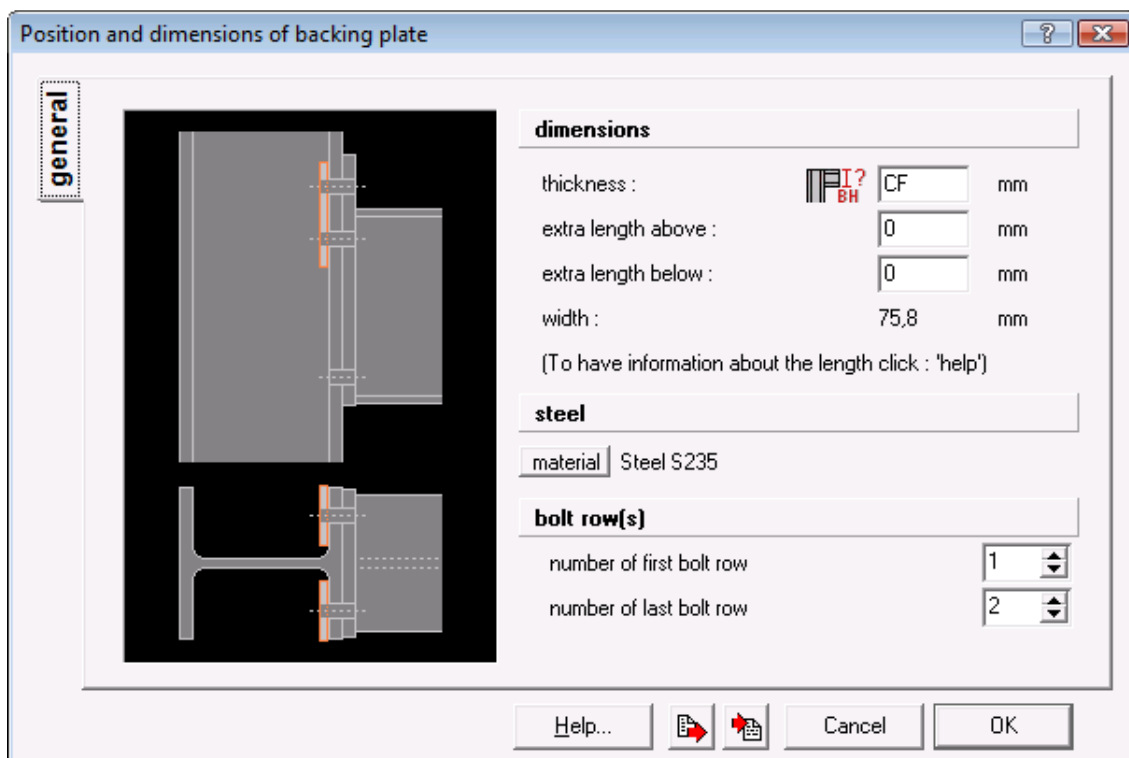
In the case of double web plates, the additional design resistance which is delivered by the web plates is not considered to be 2 times the additional design resistance delivered by a single plate, but only 1.5 times.

Finally, following parameters can still be specified to complete the definition of the web plate(s):

- the steel grade being used for the web plate, accessible through the  button.
- the properties of the welds, accessible through the  icon button,
- the thickness of the web plate. Eurocode imposes a minimum thickness equal to the thickness of the column web. Therefore, lower thickness values are not accepted by PowerConnect.

5.3.4.1.3 Backing plates

Backing plates can be foreseen at the rear side of bolted column flanges. Backing plates introduced into a PowerConnect model comply with Eurocode 3 regulations.



The length of backing plates should at least be equal to the distance in between the first and that last bolt row corresponding to this stiffening plate, increased with twice the bolt diameter at both the lower and the upper side of the backing plate. Of course, the user can introduce an extra length at both the upper and the lower side of the plate through the appropriate fields in the dialogue window.


The backing plate width depends on the column section, and is calculated automatically by PowerConnect.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.

Finally, following parameters can still be specified to complete the definition of the web plate(s):



- the steel grade being used for the web plate, accessible through the  button.
- the row numbers of the first and the last bolt row which connect the backing plate to the column flange. In case the number of the last bolt row is larger than the number of available bolt rows, PowerConnect will automatically halt the backing plate just below the last available bolt row. If further bolt rows are added, the backing plate will automatically be extended as long as the total number of actual bolt rows is lower than the row number of the last bolt row specified in the above dialogue window.


5.3.4.2 Haunches

Both lower and upper haunches can be used as part of a steel connection. Both of them are defined in exactly the same way. The two principal dimensions of a haunch are its height at the column side and its length along the beam element. Those dimensions are measured with respect to the intersection of the beam and column center lines. They are not affected by the presence of gaps.

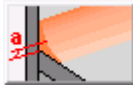
Haunches are fully defined by a set of parameters which are entered through the tab pages 'General' and 'Technical' in the dialogue window below.

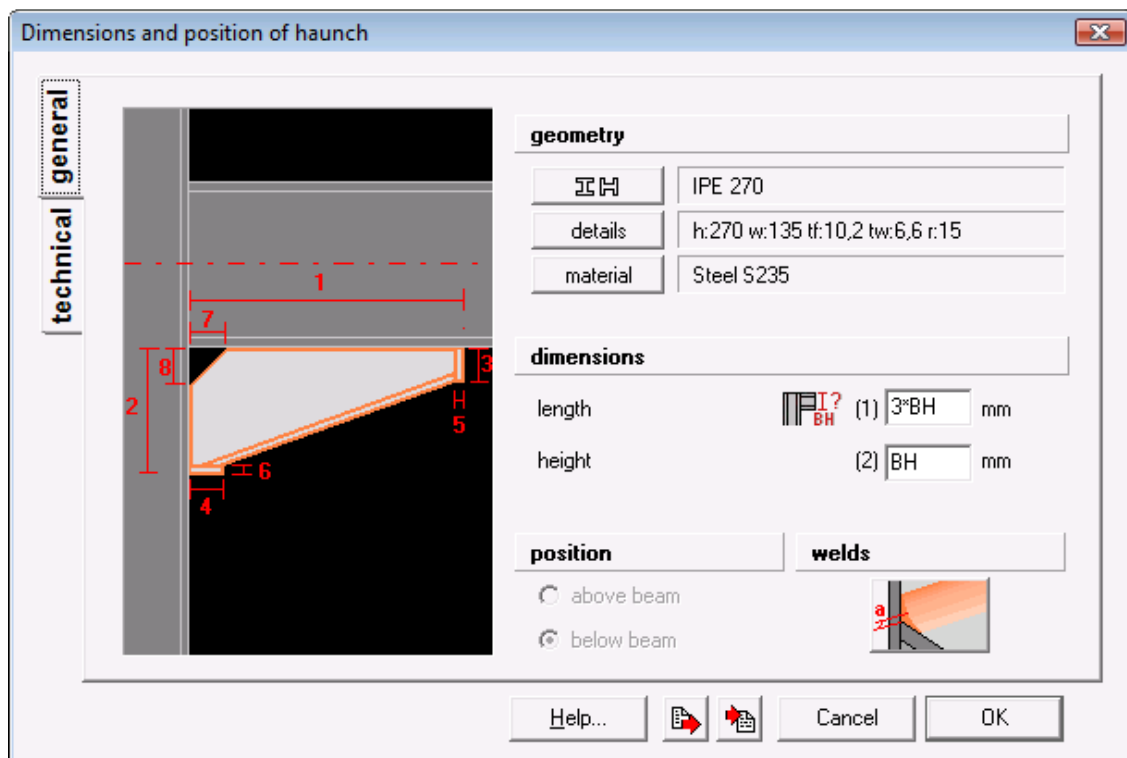
5.3.4.2.1 Tab page 'General'

A haunch is mostly fabricated using standard steel sections. The original section used to make the haunch can be selected from the PowerConnect section library using the  icon button or through the  button (in this case, the user has access to individual dimensions parameters, allowing to define a section which is different from any section available within

the PowerConnect section library). Finally, the  button allows to complete the base definition of the haunch.

Following parameters are left to finalize the haunch definition:

- the properties of the welds, accessible through the  icon button,
- the length and height of the haunch.



It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



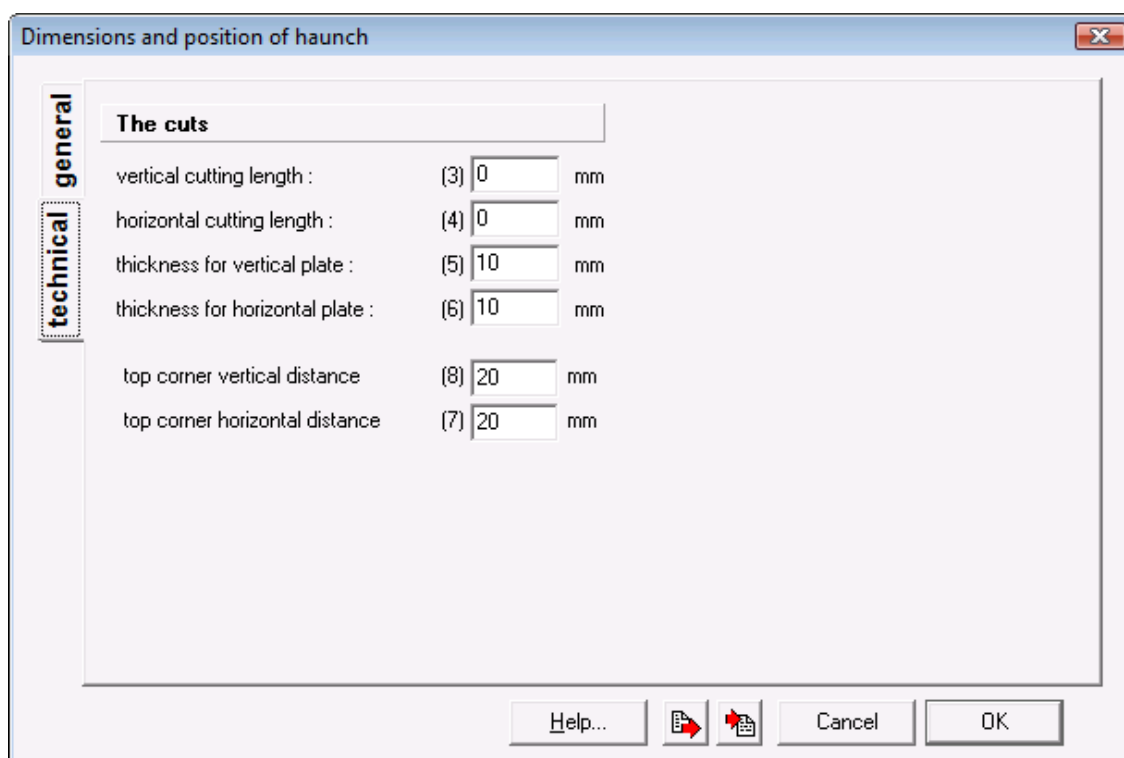
icon in the above dialogue window.

5.3.4.2.2 Tab page 'Technical'

This tab page allows for the specification of a range of detail parameters related to

- the definition of the corner gap
- the definition of vertical and horizontal plates foreseen at both ends of the haunch. In case such plates are not needed or wanted, leave the corresponding values equal to zero. In case positive values are

entered, PowerConnect will perform all checks which are necessary to confirm whether those end plates are possible or not. If this is not the case, PowerConnect will not further consider them and will not include them in the geometry model.



5.3.4.3 Gusset plates

This type of element is quite similar to the haunches which were previously discussed, but in contrast to a haunch, a gusset plate consists of a single steel plate only. A gusset plate can not be added as a stiffener if a haunch has already been defined at the same place.

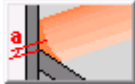
The two principal dimensions of a gusset plate are its height at the column side and its length along the beam element. Those dimensions are measured with respect to the intersection of the beam and column center lines. They are not affected by the presence of gaps.

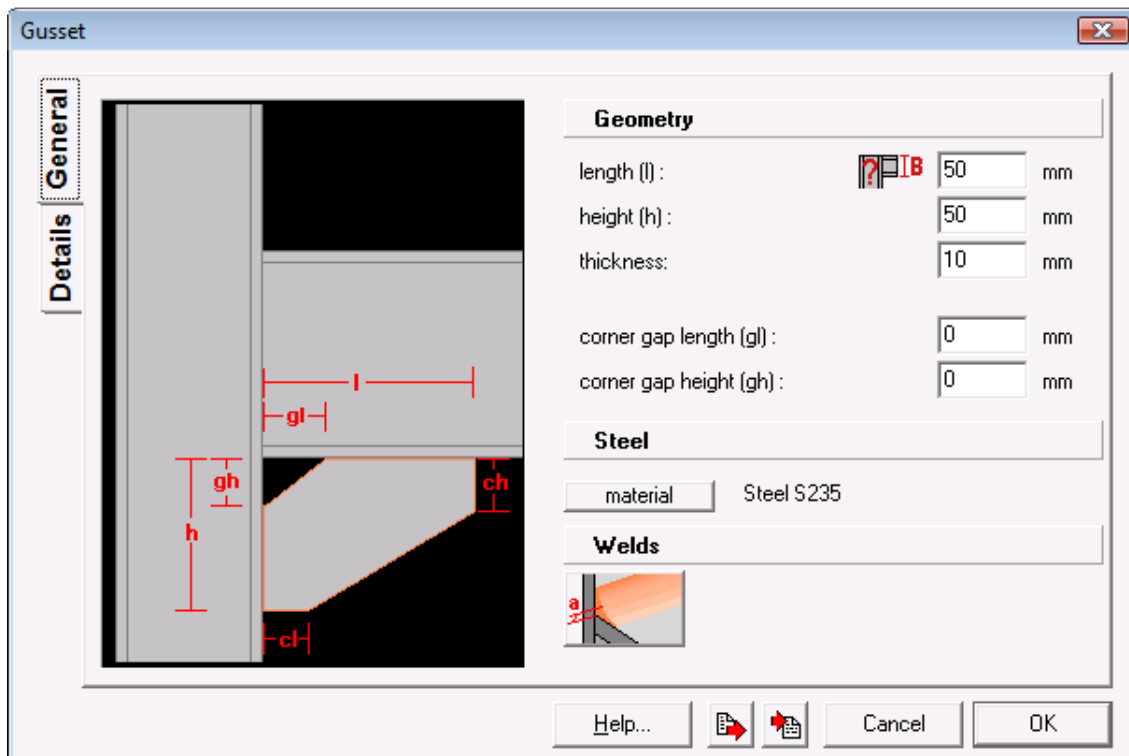
It will not be surprising to the user that quite a number of analogies exist between the dialogue windows for haunches and gusset plates.

5.3.4.3.1 Tab page 'General'

Following parameters are used for the gusset plate definition:

- the length and height of the gusset plate,
- the steel grade used for the gusset plate, by means of the material button,

- the properties of the welds, accessible through the  icon button,
- the corner gap length & height



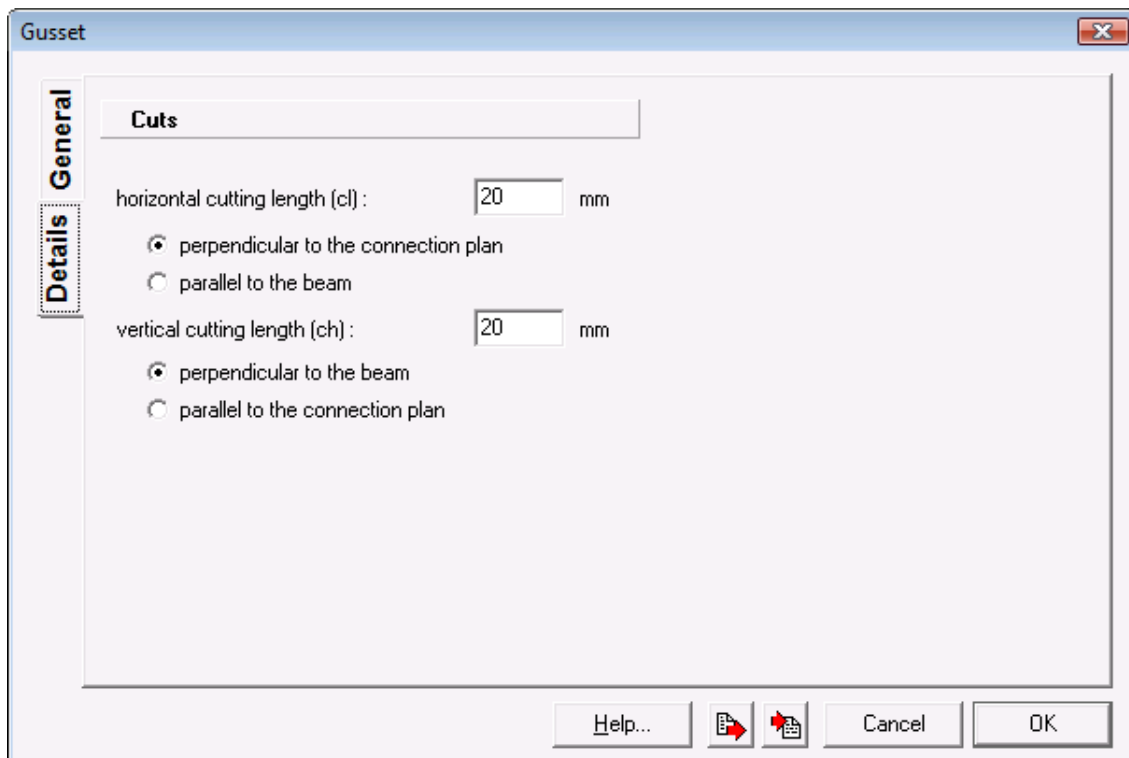
It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.

5.3.4.3.2 Tab page 'Details'

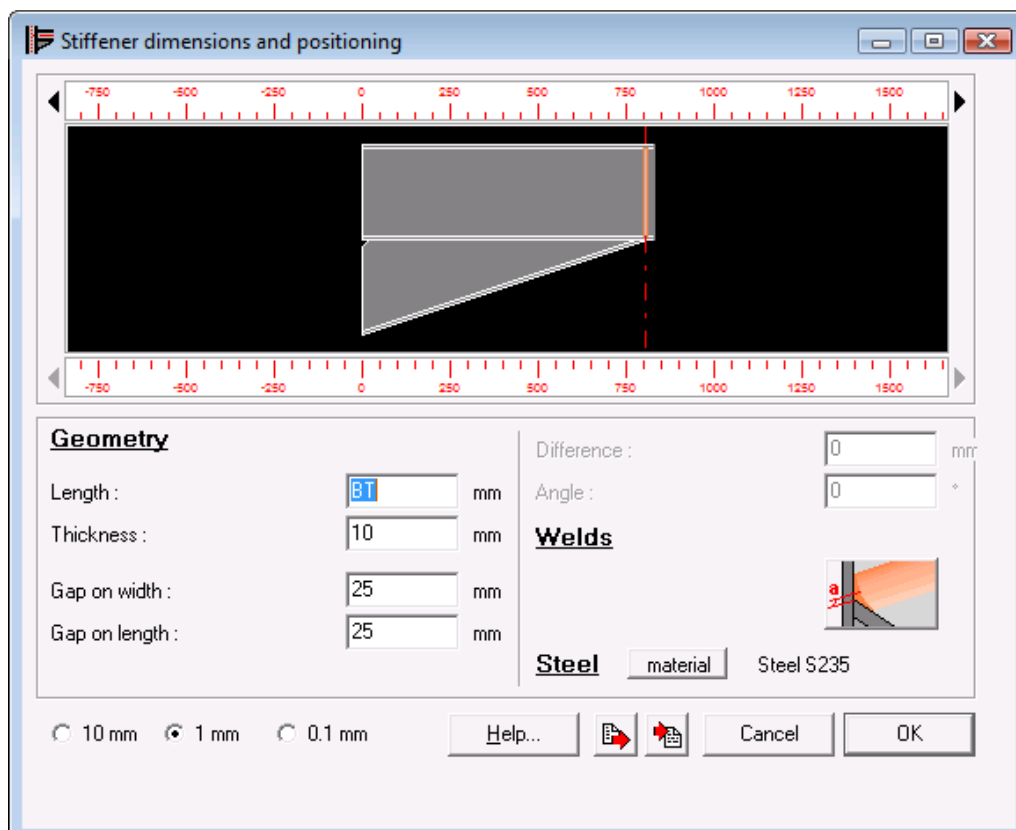
This tab page allows for the specification of a range of detail parameters related to the definition of the cuts foreseen at both ends of the gusset plate.



5.3.4.4 Stiffening elements for beams

5.3.4.4.1 Flange stiffeners

Flange stiffeners can only be used in case a haunch has been added to the connection.




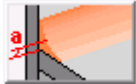
Three types of flange stiffeners can be distinguished:

- a flange stiffener as an extension of a lower haunch;
- a flange stiffener as an extension of an upper haunch;
- a flange stiffener in between a lower and an upper haunch.

In the first 2 cases, the flange stiffener need not necessarily be perpendicular to the beam axis. In case of a flange stiffener in between an upper and a lower haunch, the orientation of the stiffener is imposed by the characteristics of both haunches. It will thus not be possible to edit the related parameters in the above dialogue window for this type of flange stiffener.

By means of the arrows on top of and below the graphical area of the above dialogue window, the position of the flange stiffener can freely be modified. The user should select the appropriate step size (either 0.1mm, 1mm or 10mm) by which the position of one or both stiffener ends should be changed.

Following parameters are further used for the flange stiffener definition:

- the length and thickness of the flange stiffener,
- the steel grade used for the flange stiffener, by means of the  button,
- the properties of the welds, accessible through the  icon button,
- the corner gap width & height

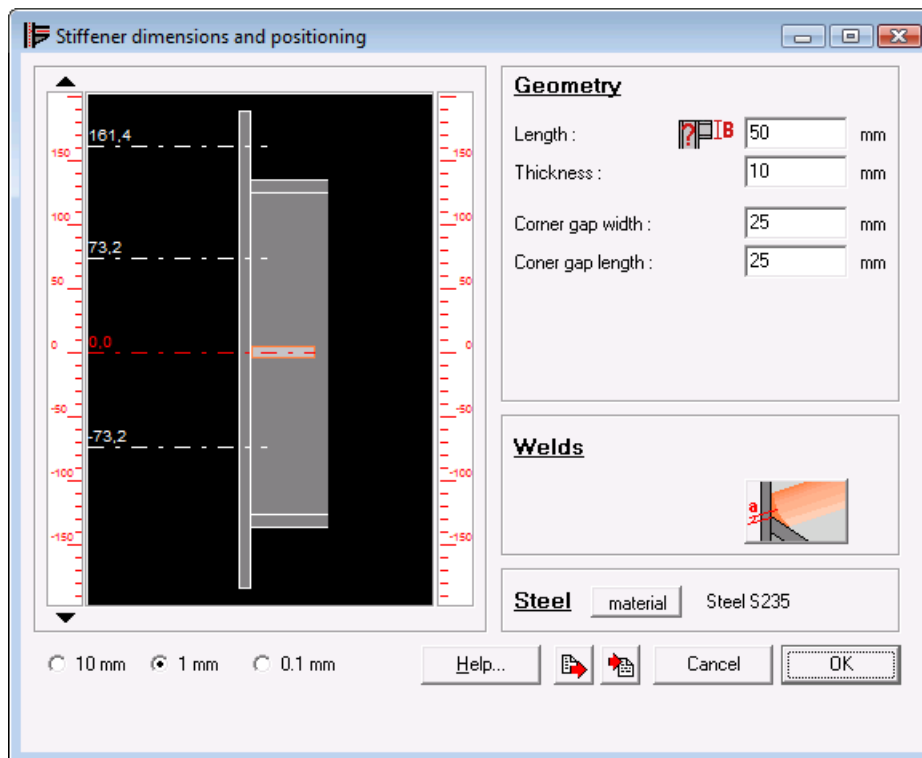
It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.


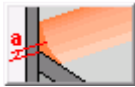
5.3.4.4.2 Transverse stiffeners

Transverse stiffeners can only be used to stiffen an end plate which has already been added to the connection.



By means of the arrows on both sides of the graphical area of the above dialogue window, the position of the transverse stiffener can freely be modified. The user should select the appropriate step size (either 0.1mm, 1mm or 10mm) by which the position of one or both stiffener ends should be changed. The position of the transverse stiffeners is always specified relative to bolt row positions.

Following parameters are further used for the transverse stiffener definition:

- the length and thickness of the transverse stiffener,
- the steel grade used for the transverse stiffener, by means of the  button,
- the properties of the welds, accessible through the  icon button,
- the corner gap width & height

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

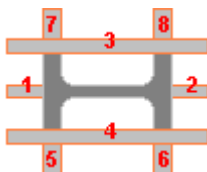


icon in the above dialogue window.

5.3.4.5 Column base stiffeners

The dialogue window for column base stiffeners also contains two tab pages. It is important to observe that the actual stiffener element that is being defined, is shown with red alphanumeric labels on the 'General' tab page.


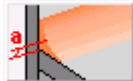
PowerConnect allows to define up to 8 different column base stiffeners.



The same dialogue window is used for the 8 column base stiffener elements. The contents of the window will however automatically adapt themselves to the type of element that has been selected in the 'Geometry' window. It is indeed not possible to select an other stiffener element within this dialogue window. Stiffening elements can be selected only by double-clicking on the element itself within the 'Geometry' window, after it has been added to the column base model.

5.3.4.5.1 Tab page 'General'

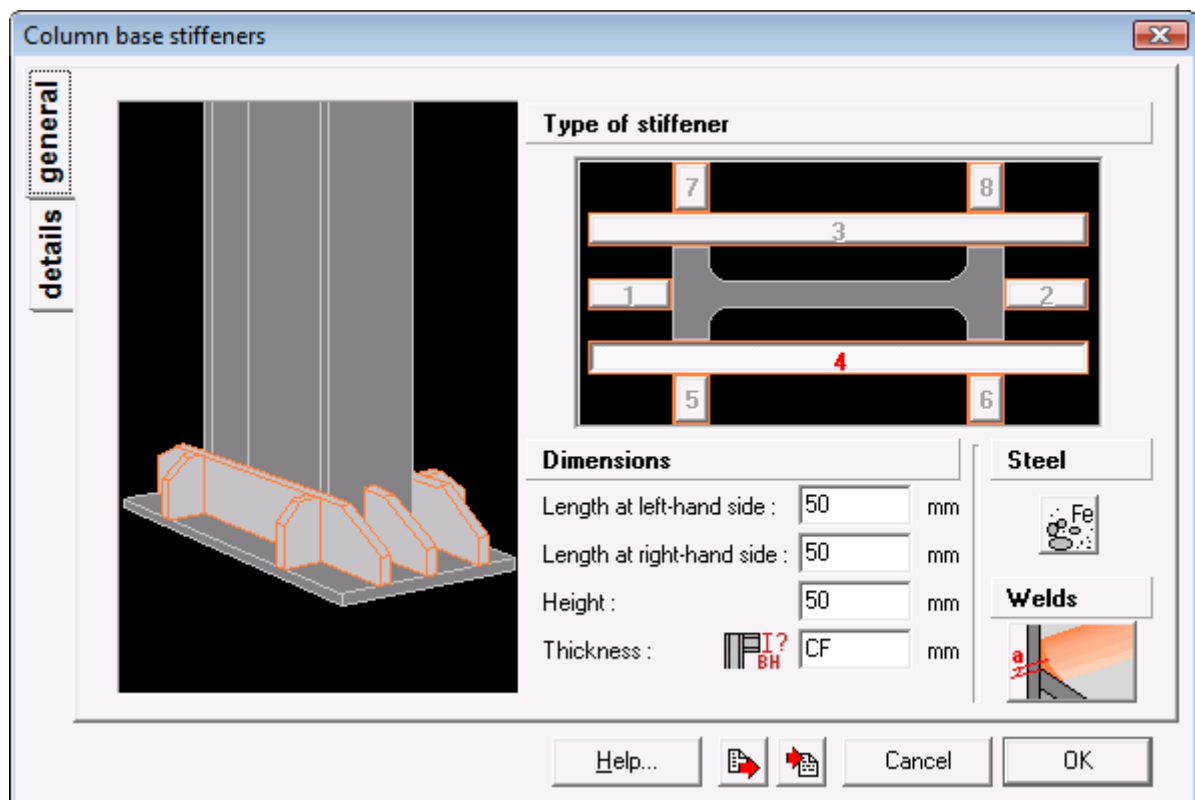
Following parameters are used for the definition of a column base stiffener:

- Length, height and thickness of the stiffener,
- the steel grade used for the stiffener, by means of the  icon button,
- the properties of the welds, accessible through the  icon button.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

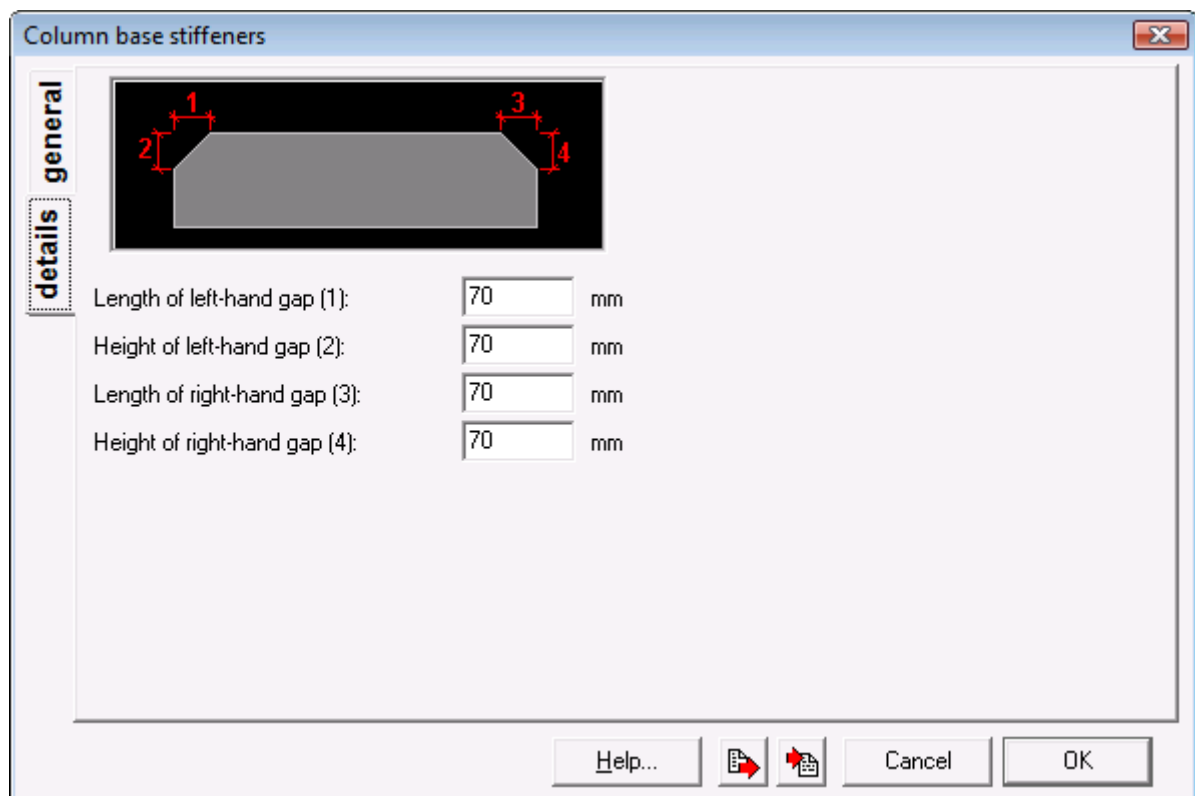


icon in the above dialogue window.



5.3.4.5.2 Tab page 'Details'

The second tab page allows to define in more detail the contours of the column base stiffeners. All parameters are explained in the graphical part of the window.




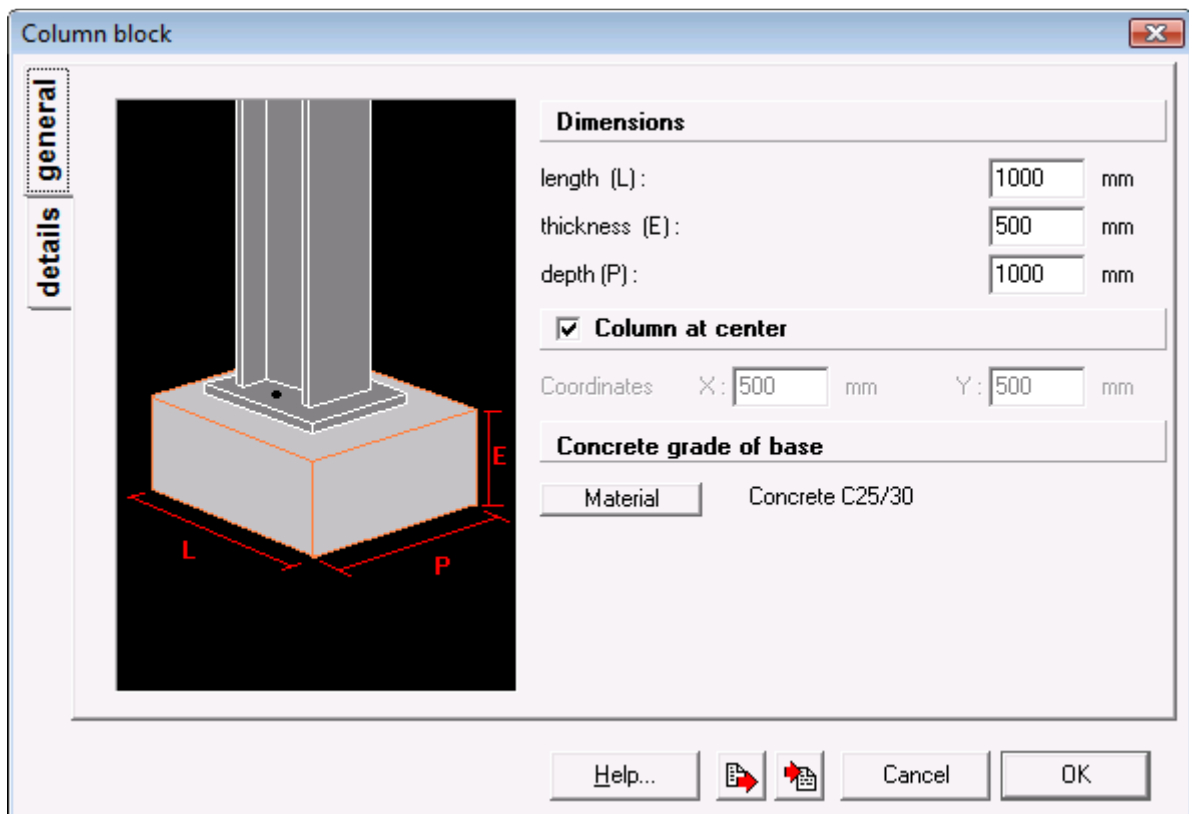
5.3.5 Other elements

5.3.5.1 Concrete base

5.3.5.1.1 Tab page 'General'

Following parameters are used for the definition of a concrete base:

- length, thickness and depth of the column block,
- position of the center line of the column. In case the option “Column at center” is selected, no further definition of coordinates is required and the column will automatically be positioned at the centre of the base. If this option is not selected, then the (X,Y)-coordinates of the center line position are entered with respect to the upper left corner of the column block (in plan view). Those values are necessarily positive values, and should of course not exceed the length and depth of the column block.
- concrete grade used for the block, by means of the  icon button. This allows to select any of the concrete grades available in the PowerConnect material library.



It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication.

The actual meaning of any parameter can simply be obtained by means of the



icon in the above dialogue window.

5.3.5.1.2 Tab page 'Details'

The end plate of the column base connection is not supported directly by the concrete block. Instead, a grout layer is being used to improve contact between both the end plate and the upper surface of the concrete block. Of course, the design analysis will have to take into account the grade of grout being used. The thickness of the grout layer and grout grade can be defined on the tab page 'Details'. Attention is drawn to the remarks shown on the right-hand side of this tab page.

Column block

general

Grout

grout thickness : mm

details

Grout grade

characteristic compressive strength (fk): N/mm²

friction coefficient between plate and grout (Cfd):

Grout thickness should not exceed 0.2 times minimum width of steel base plate


Characteristic strength of grout must at least be 0.2 of characteristic strength of concrete block.

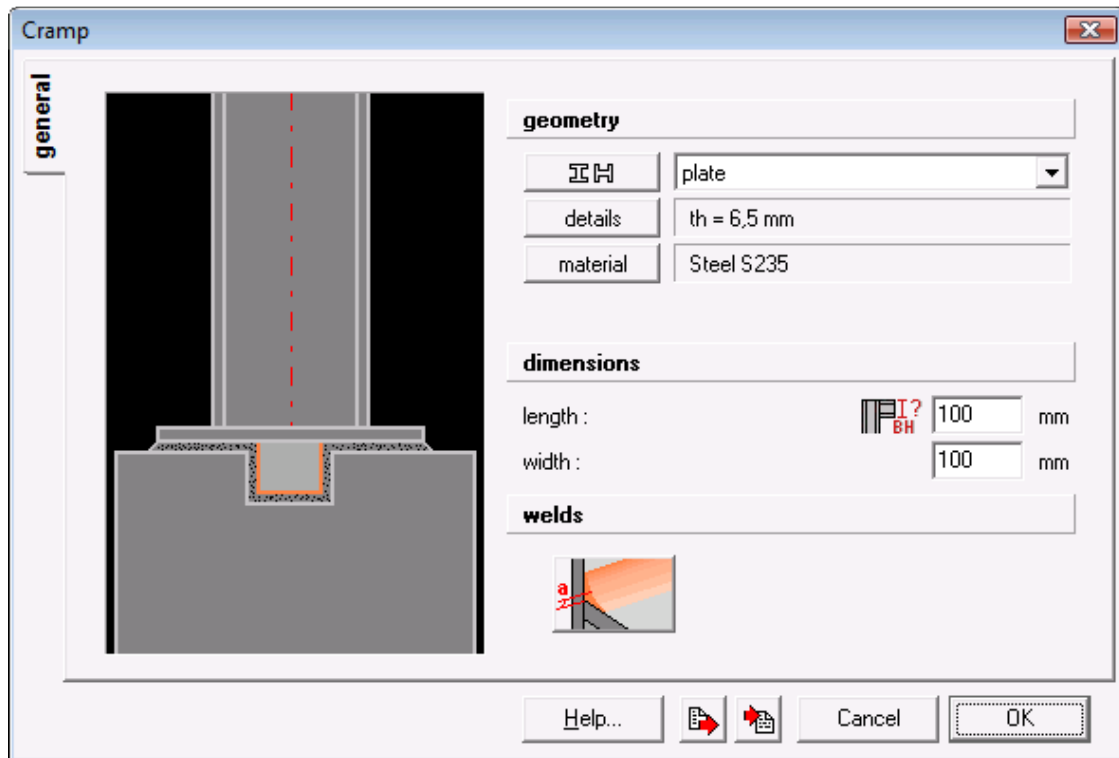
Help... Cancel OK


Finally, PowerConnect allows for the definition of the friction coefficient between the end plate and the grout layer. PowerConnect will consider this friction coefficient during the evaluation of shear resistance.

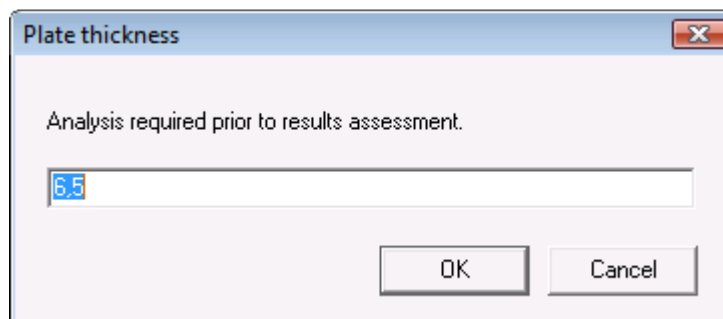
5.3.5.2 Cramps

It may however occur that the sollicitating shear force is too high compared to the shear resistance that can be delivered by the anchor bolts and the friction between end plate and grout layer. In that case, cramps may be used to increase the column base shear resistance up to a level which exceeds the sollicitating shear force.

A cramp may either be a simple steel plate or a steel section which is welded to the lower side of the end plate. In case a steel section is being used, select the option 'cross-section' from the pull-down menu in the dialogue window below. Access will then be provided to a range of sections available within the PowerConnect section library. The button  can be used to modify individual section properties, to enable the use of section that are not necessarily stored in the PowerConnect section library.


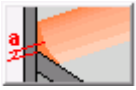


In case a simple steel plate is being used for the cramp, select the option 'plate' from the pull-down menu in the dialogue window above. The button  can directly be used to modify plate thickness.



Finally, following parameters are used to complete the definition of a cramp:

- length and width of cramp (width only in case a steel plate is being used),

- the steel grade used for the cramp, by means of the  button,
- the properties of the welds used to connect the cramp to the end plate, accessible through the  icon button.

It is observed that some input fields may contain a parameter rather than a number. In this case, the corresponding dimension is linked to the dimension of another element of the connection. PowerConnect indeed includes a number of coded parameters which allow to be used as a reference as part of the definition of element dimensions. It is interesting to note that it is possible to use those parameters as part of arithmetic expressions involving addition or multiplication. The actual meaning of any parameter can simply be obtained by means of the

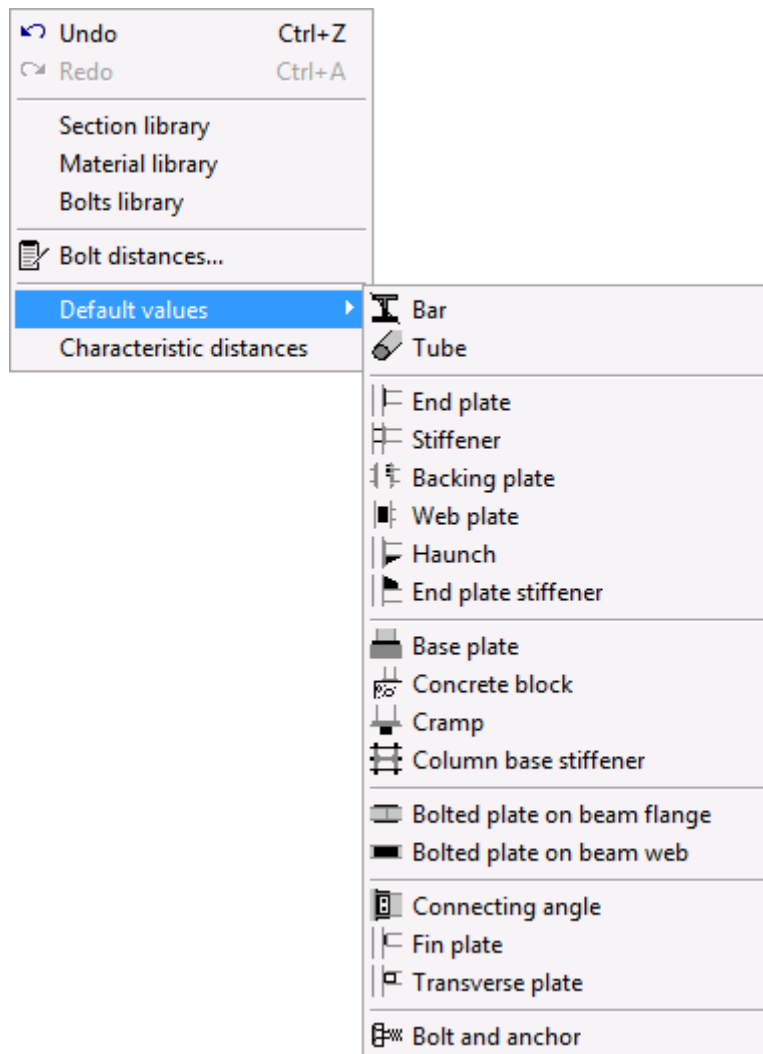


icon in the above dialogue window.

5.4 Definition of element defaults

5.4.1 Specification of default values for individual element types



Default values can be specified for all parameters which define the element types supported by PowerConnect (and which are described in this Chapter 5 of the reference manual). To do so, use the menu 'Edit – Default values' and select the appropriate element type from the list of available types (see below).





Selecting a particular type will make a dialogue window pop up, in which the default settings for the selected component type can be modified and specified. Those dialogue windows are quite similar to the ones which have been discussed in sections 5.3.2, 5.3.3, 5.3.4 and 5.3.5 of this reference manual. As they are furthermore self-documented with all graphics required for a good understanding of all involved parameters, those dialogue windows will not be further discussed in this reference manual.

5.4.2 Use of element default values


To complete the information on element default values, the information contained in section 5.3.1.1 of this reference manual is briefly recalled.


Any PowerConnect dialogue window that allows for the definition of bar elements, connection elements, stiffening elements or any other element, includes 2 icon buttons   next to the 'OK', 'Cancel' and 'Help'-buttons. Those icons allow to manage the default settings for the active element:

- the  icon will read the default values that have previously been defined for the selected element type, and will apply them to the selected element.
- the  icon will save the values that have just been defined for the selected element as the default values for the corresponding element type.

6 Global functions and options

6.1 File management

PowerConnect v5.x files have a file extension *.bpc. Existing project files (which have previously been saved to hard disk) can be opened through the menu 'File – Open' or through the  icon of the icon toolbar. The 4 files which have most recently been saved to hard disk are automatically remembered by PowerConnect, and will be included in a list of recent files accessible through the arrow in the previously mentioned icon.


To save the current PowerConnect project to hard disc, the menu commands 'File – Save' or 'File – Save as...' are available, along with the  icon of the icon toolbar.

A PowerConnect .bpc file can also be opened by double-clicking on its name or icon in the Windows Explorer or on your desktop.

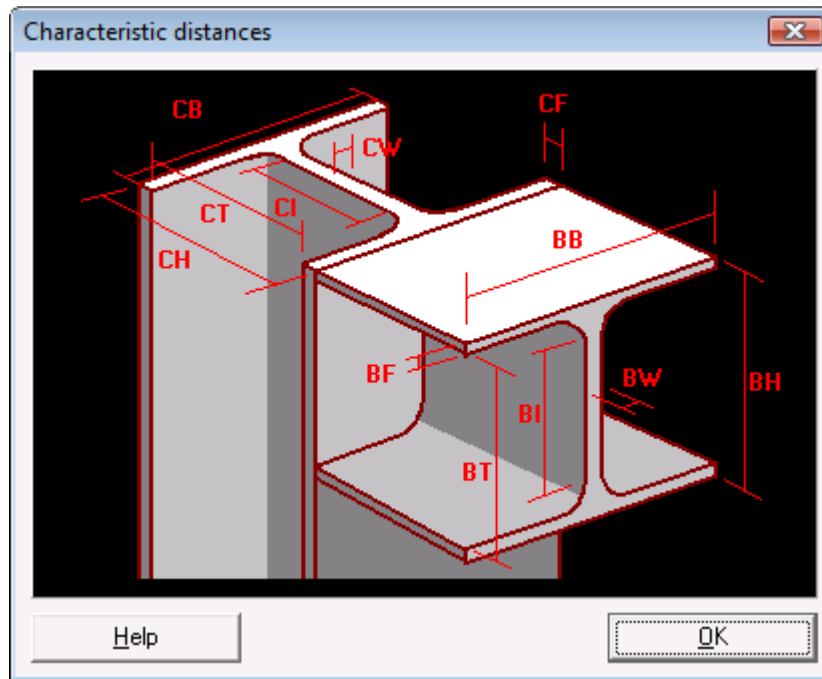
With previous versions of PowerConnect, project files were saved to a different file format. In particular, *.pc3 and *.pco files were supported by PowerConnect v3.x/v4.x. To open such files and to convert them into *.bpc files, use the menu command 'File – Open' and select the appropriate file type from the pull-down list of supported file types.

6.2 Characteristic distances

PowerConnect allows for the specification of frequently used dimensions through a series of parameters. Those parameters are referred to as "Characteristic distances" – 12 such parameters are available in total. They mostly relate to typical dimensions of beams and columns which are part of the connection. The symbolic annotation used to describe those parameters, along with their meaning, is graphically documented in the PowerConnect software. This information is accessible

- either through the menu command 'Edit – Characteristic distances'
- either directly from the dialogue windows for the supported element types, whenever it is allowed to use those characteristic distances as input parameters. In this case, the  icon allows to access the detail information on characteristic distances.

In both cases, the user will be presented the following window:



Following 12 parameters are available:

- for column elements

CH: height of column

CB: width of column

CW: thickness of column web

CF: thickness of column flanges

CI: height of the straight part of the column web, in between both column flanges

CT: height of column web in between both column flanges

- for beam elements

BH: height of beam

BB: width of beam

BW: thickness of beam web

BF: thickness of beam flanges

BI: height of the straight part of the beam web, in between both beam flanges

BT: height of beam web in between both beam flanges

The use of characteristic distances will be described through a number of examples.

In case an end plate is to be defined such that its thickness should be equal to 1.2 times the thickness of the column flanges, it will be sufficient to enter following values:

- either $1.2 \cdot CF$,
- or $CF \cdot 1.2$.

If for some reason the column section is changed, then its flange thickness CF will normally also change. What is important, is that the thickness of the end plate will change in exactly the same way as the thickness of the column flange, such that a fixed ratio of 1.2 will still exist between both thickness.

It can be seen from the above example that characteristic distances do not necessarily have to be used just by themselves. Instead, they can be included in arithmetic expressions involving addition or multiplication.

A few examples (in which %% corresponds to the notion of a characteristic distance):

Description	Example
[number]	15
%%	CF
%%*[number]	$CF \cdot 1.2$
[number]*%%	$1.2 \cdot CF$
%%+[number]	$CF + 2$
[number]+%%	$2 + CF$

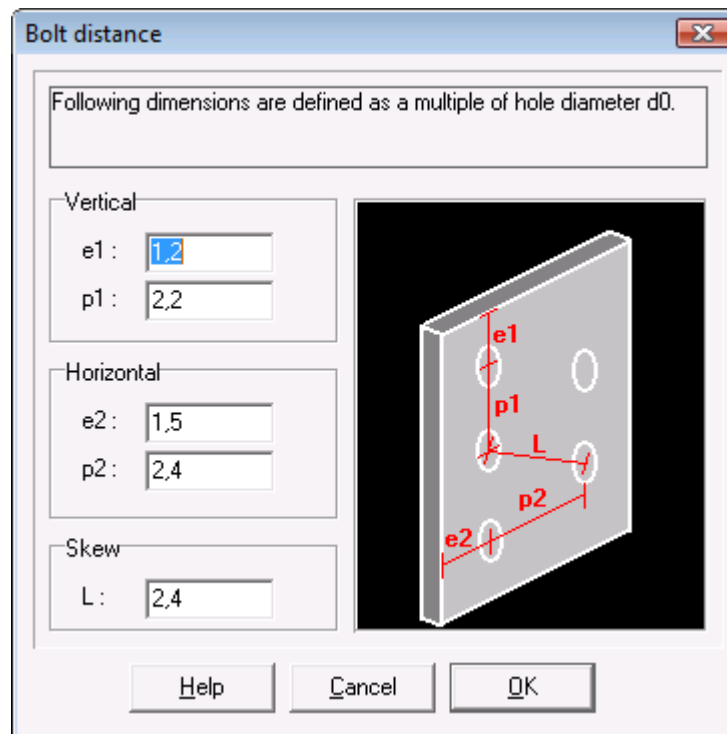
If it is observed that the input field is colored red when input data are provided, then PowerConnect cannot interpret correctly the entered data. This may either be explained by an erroneous input formula (so the user should carefully inspect the syntax of what has been entered), or by the fact that the arithmetic expression yields a result which is not compatible with any maximum or minimum limits for the selected input field.

An example: if the thickness of an end plate is defined as $0.01 \cdot CF$ with $CF = 10\text{mm}$, PowerConnect will assign a red color the input field as a minimum thickness of 3mm should be respected for the end plate.

6.3 Bolt distances

Eurocode 3 imposes minimum distances between bolt center Lines. Furthermore, the user can specify minimum bolt distances by himself, to allow for an optimal bolt lay-out with PowerConnect.

The default values considered by the software can be changed at any time by the user through the menu command 'Edit – Bolt distances...', which will issue the following dialogue window.

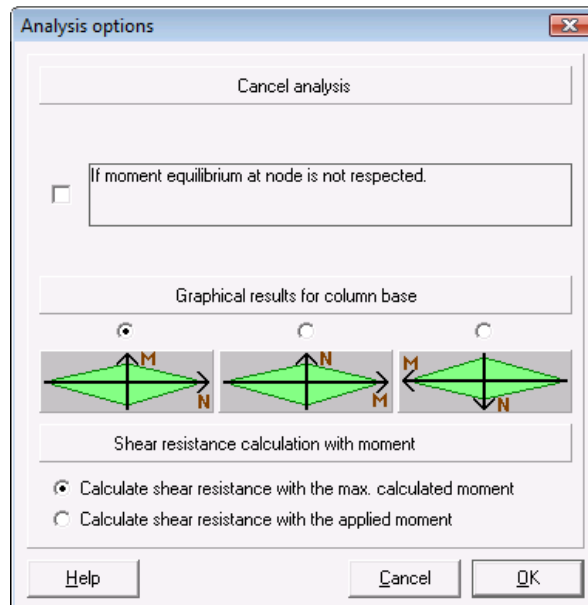


Values in this dialogue window are defined as a multiple of hole diameter.

6.4 Analysis options

Through the menu command 'Analysis – Analysis options', a dialogue window is opened which allows to define a series of analysis parameters:

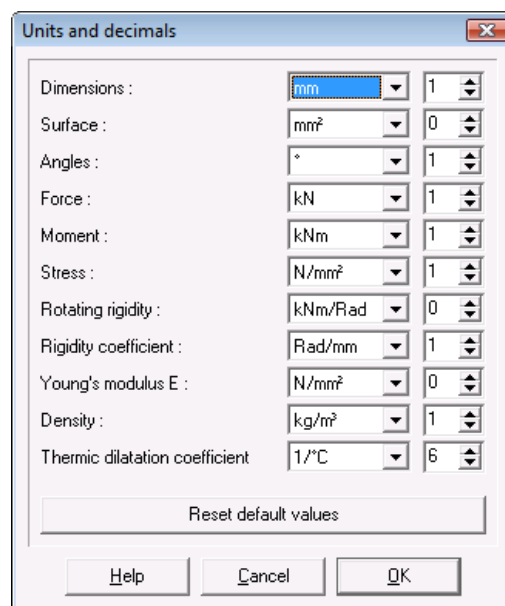
- the first option, if selected, will cancel the analysis in case the moment equilibrium of the node is not respected. If this option is not selected, then the analysis will always be performed even in case the applied loads do not respect moment equilibrium conditions.
- next, a selection can be made between different representations for the diagram describing valid (M,N) combinations for a column base.
- finally, it can be specified how shear resistance should be calculated in case a bending moment is applied to the connection. A choice can be made between both following options:
 - consider the bending moment resistance (MRd) of the connection to evaluate what shear resistance is left available,
 - consider the bending moment applied to the connection (MSd) to evaluate what shear resistance is left available.



6.5 Units and decimals

Through the menu instruction 'Options – Units and decimals', the units and the accuracy for all supported data types can be specified. It should be noted that changing those parameters does not affect the analysis accuracy, as internally PowerConnect will always use the same consistent set of units. The units and decimals specifications provided by the user only affect the representation of data and results in graphs and reports.

In the first column, an appropriate unit can be chosen from a pull-down list for the selected unit type (this list includes both metric and other units). In the second column, the number of decimals can be specified in the range between 0 and 5.



7 Reporting

7.1 Page setup

Through the menu command 'File – Page setup', a number of global page settings can be specified for use with any report to be created by PowerConnect. All specifications are given in the dialogue window below, which is issued by the afore mentioned menu command.

Page setup

Margin

10 mm
15 mm
10 mm
10 mm

Font
Arial

Font size

	normal	header	footer	Title 1	Title 2	Title 3	Title 4	Title 5
	9	9	9	16	14	12	11	10

Frame ☒ Use a frame for report

logo (bmp) C:\Program Files\BuildSoft\PowerConnect\LogoBu ...

Header

left
middle
right

Footer

left
middle
right


Advanced

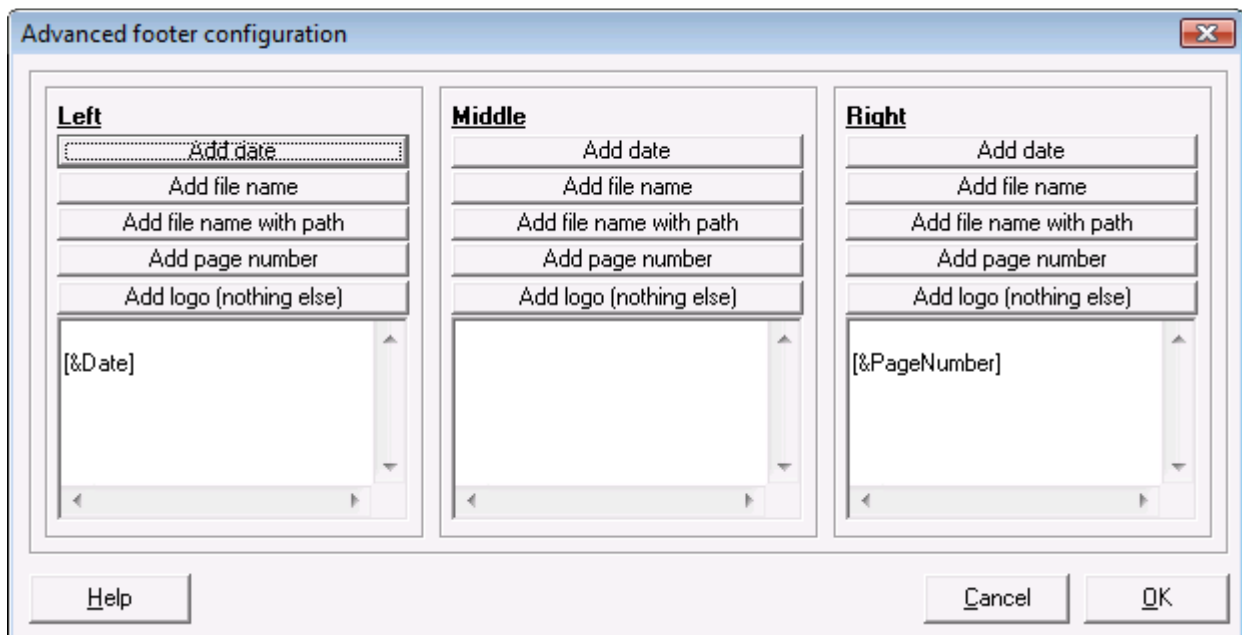
☒ Use
Set-up

Help Cancel OK

Apart from the straightforward parameters as margin, font & font size, following parameters may need some extra clarification:

- **FRAME:** if the option 'Use frame for report' is selected, both the header, body and footer of the report will be framed.

- LOGO(BMP): allows to specify a bitmap-file (with the company logo). This file can then be used as part of the report header or footer.
- HEADER/FOOTER
 - If the 'Advanced' option is not selected, use the left/middle/right pull-down menus to define the contents of the header & footer fields. At the bottom of the pull-down menu, the field named 'Logo' refers to the bitmap file defined in the previous step.
 - If the 'Advanced' option is selected, use the  button to enter in a new dialogue window allowing for a more customized definition of header and footer.



Next to the available standard fields, like 'date' – 'file name' – 'page number' – 'logo' - ..., text can freely be entered by the user in the edit boxes for the left/middle/right part of header & footer.

Any changes that have been made in the above dialogue windows, and which have been properly confirmed by means of the 'OK'-buttons, will be active until the next changes are defined. Until then, all changes will be used for any new report that is created by PowerConnect.

7.2 Report creation



Once the actual contents of a report have been defined by the user, the report can be created. The content specification will be discussed in the next section, but at this time it should be noted already that 3 creation modes are available with the PowerConnect reporting function:

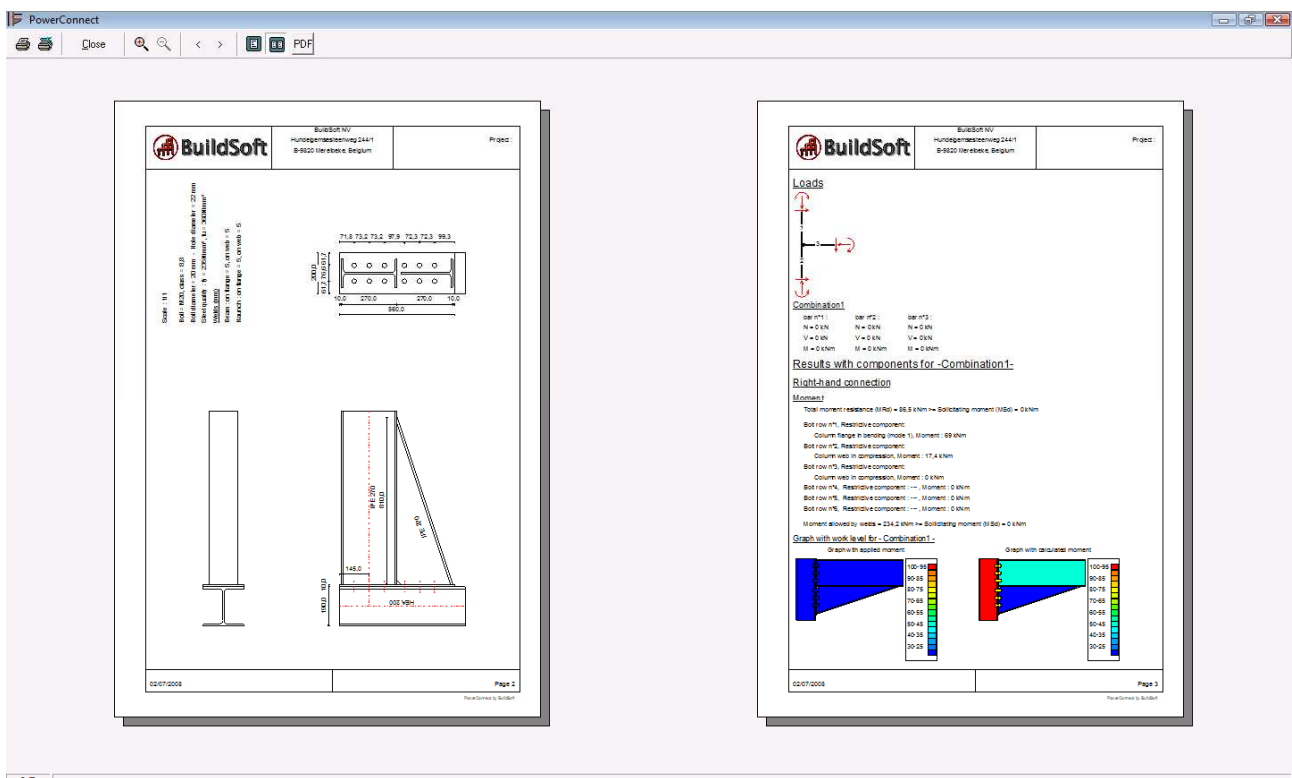
- **Previewing**, to visually check the report before it is actually printed.

- **Printing**, to send the report to one of the printers installed on the workstation where PowerConnect is being used.
- **Printing to RTF**, to export the report to a Rich Text Format file. Such a file can be opened by a word processor, although it is usually more appropriate to save it to a format which is proprietary to the word processor being used. RTF files can easily be opened by different word processors, but are not necessarily optimized in terms of disk space requirements. Saving to a proprietary format may easily compact the file by a factor of 10 (or higher).


Each of the afore mentioned creation modes is now discussed in more detail.







7.2.1 Previewing

Use the menu command 'File – Preview' or the  icon of the icon toolbar to launch the 'Preview' dialogue window. Refer to section 7.3 for more information on how the report contents should be defined. If this has been done, use the  button.





The 'Preview' window contains following functions:



-  - Print : opens the standard MS Windows print dialogue, from which the appropriate printer can be selected, and from which a print command can be issued,

-  - Printer Setup : opens a standard MS Windows dialogue from which the desired printer can be selected, and from which its detail settings can be controlled,
-  : no further explanation is needed,
-  : magnifying glasses to either zoom in or zoom out on the report preview,
-  : arrows, allowing to scroll forward & backward through the report preview,
-  : buttons, allowing to switch between a 1-page & 2-page preview,
-  : creates a PDF report consistent with the current report preview. The user will be prompted to specify where and with which name the PDF file should be saved to hard disk.

7.2.2 Printing

Use the menu command 'File – Print report' or the  icon of the icon toolbar to launch the 'Print Report' dialogue window. Refer to section 7.3 for more information on how the report contents should be defined. If this has been done, use the  button.

7.2.3 Printing to RTF

Use the menu command 'File – Print report to RTF' or the  icon of the icon toolbar to launch the 'Print Report to RTF' dialogue window. Refer to section 7.3 for more information on how the report contents should be defined. If this has been done, use the  button.

7.3 Report configuration

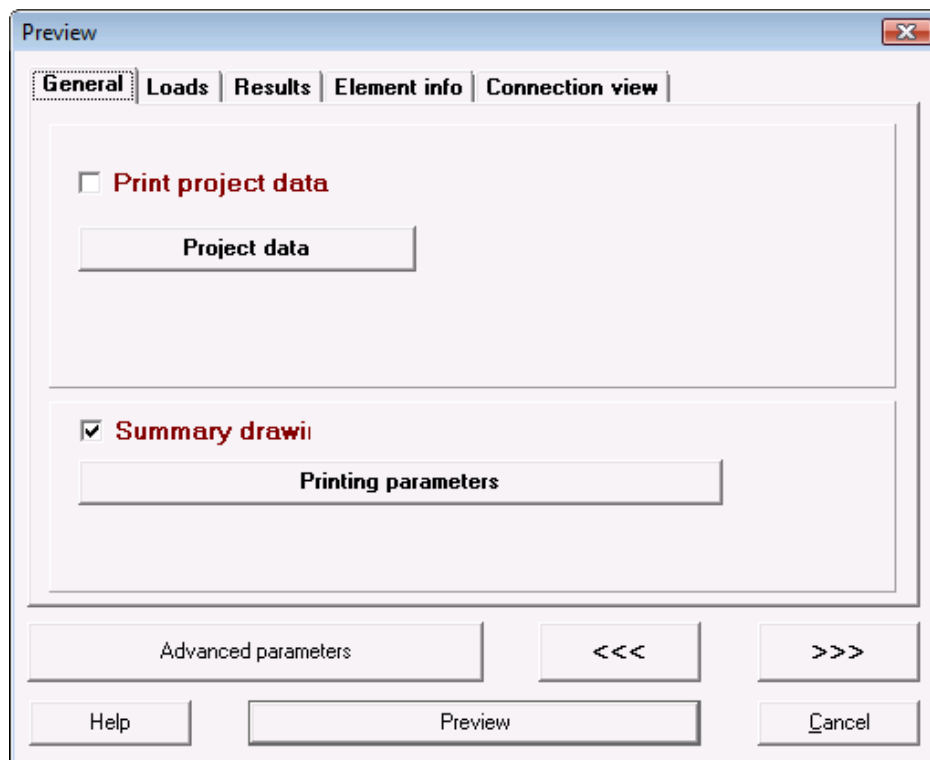
Report contents are configured through 5 different tab pages, corresponding to 'General' items, 'Loads', 'Results', 'Element info' and 'Connection view'.

7.3.1 Tab page 'General'

7.3.1.1 Print project data

Select the option 'Print project data' to include the data which have defined through the menu command 'File – Project data'. In case those data still

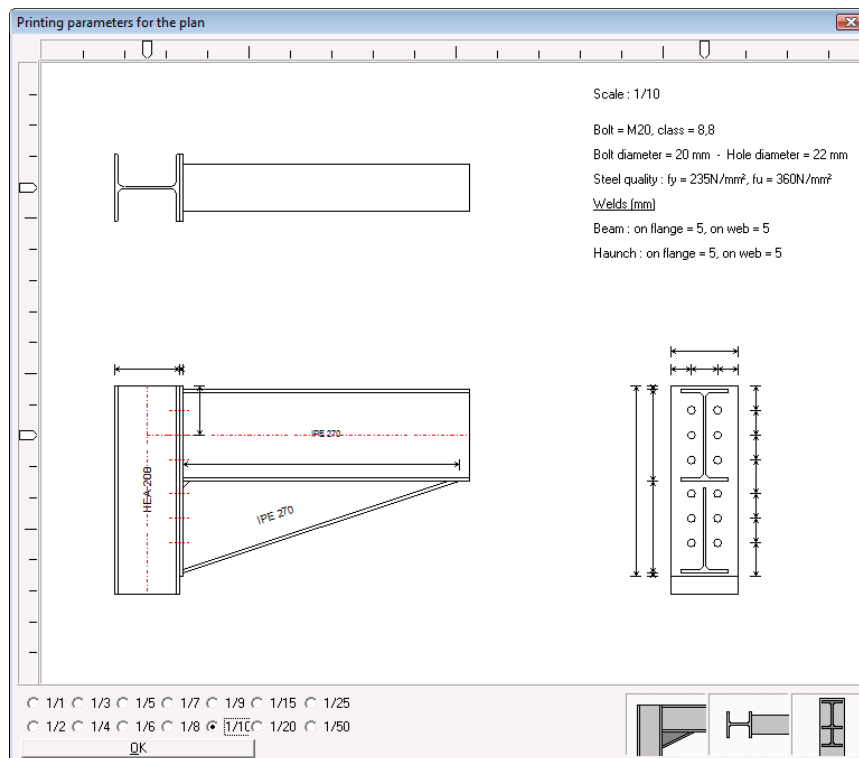
need further elaboration, this can be done through the button labeled 'Project data' in the dialogue window below.



7.3.1.2 Summary drawings

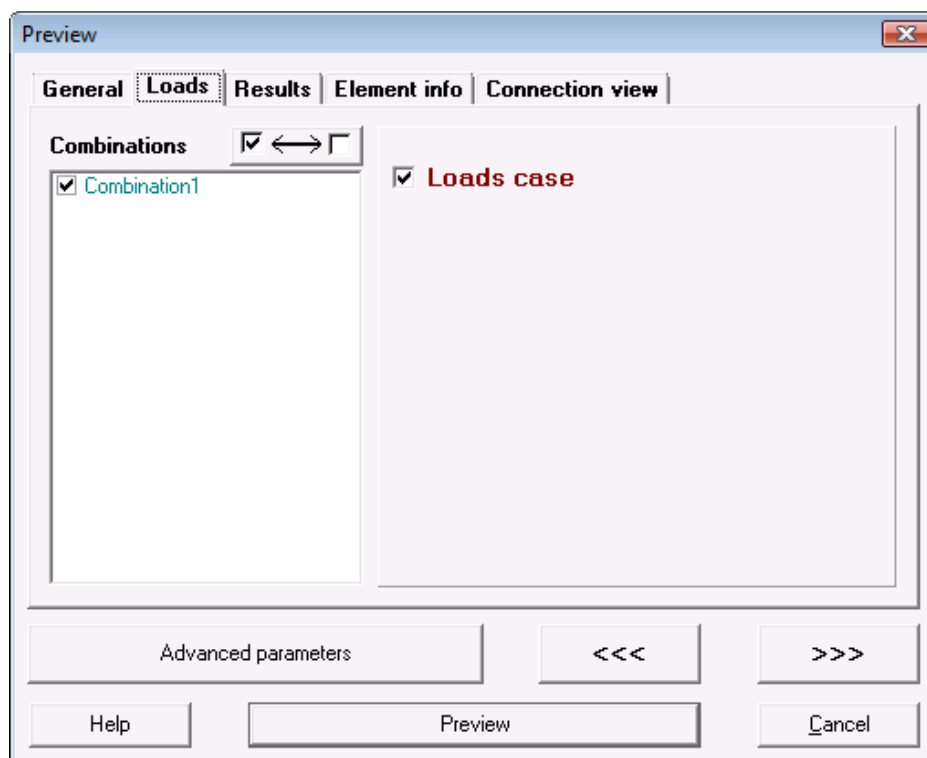
Select the option 'Summary drawings' to include a global connection drawing in the report. How those drawings are presented can be specified through the button labeled 'Printing parameters', which will open a new dialogue window with following controls:

- a series of buttons at the right bottom corner which enable the user to specify which drawings are to be included,
- horizontal and vertical cursors which allow to control the position of the different drawings
- a range of values to define the scale of drawings.



7.3.2 Tab page 'Loads'

Select the option 'Loads combinations' to ensure that loads information will be incorporated into the report. Then, individual combinations can be selected or deselected in the left part of the dialogue window.



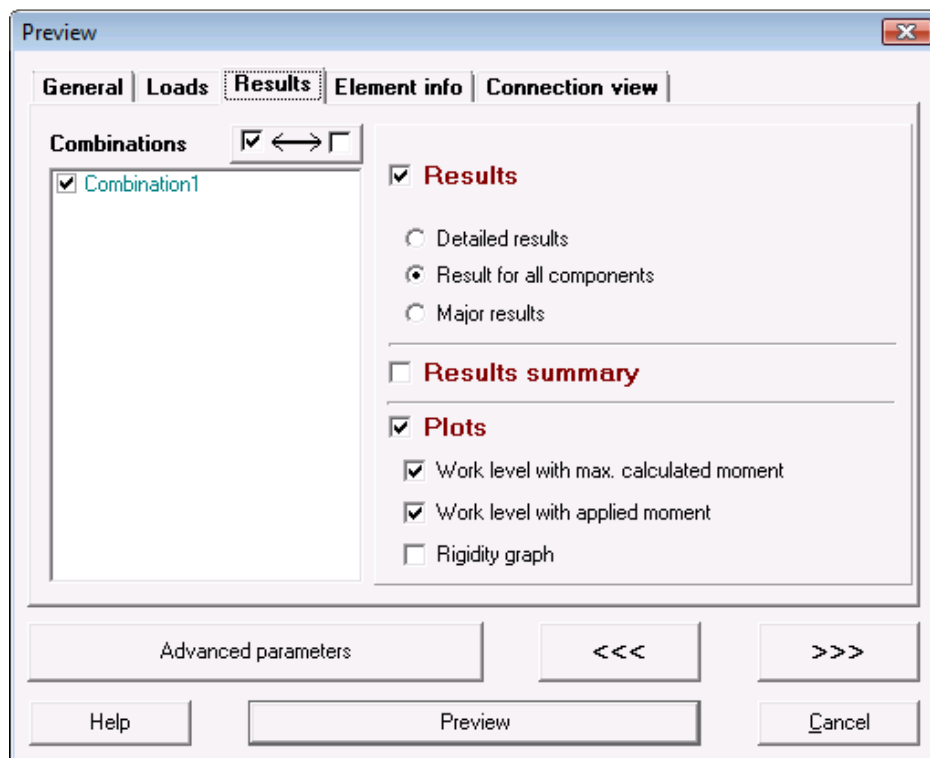
7.3.3 Tab page 'Results'

To the left, all loads combinations which are available with the current PowerConnect project are listed. Make sure to select only those combinations for which design analysis results are to be included in the report.

To the right, further specifications can be given with respect to

- **detail level of results reporting**, giving a choice between 3 levels:
 - detailed results;
 - detailed results for all components;
 - major results only.

In case no detailed results reporting is required, select the 'Results summary' option.



- **types of plots to be reported**, giving a choice between 3 types:
 - work level with maximum calculated moment;
 - work level with applied moment;
 - rigidity graph.

7.3.4 Tab page 'Element info'

On this tab page, the user will find a list of all elements that are part of the connection being analyzed. Next to the name of each element, 3 columns are available:

- in column 1, it is specified whether or not element data are to be included in the report. The term element data refers to an alphanumeric description of the major element characteristics.
- in column 2, it is specified whether or not an element drawing is to be generated as part of the report. The scale used to create such a drawing can be defined in column 3. In case the drawings would not fit on the selected page format with the selected scale, then PowerConnect will automatically modify the scale factor of the drawing to ensure it does fit on the selected page format.

If data and/or drawings should be created for all elements, then just use the 'V yes' buttons on top of column 1 and/or 2. By means of the pull-down menu on top of column 3, a uniform scale factor can be specified for all drawings.

Preview

General | Loads | Results | **Element info** | Connection view

Right-hand connection V no V yes V no V yes 1/5

Element name	Data	Drawing	Scale
Column	no	no	1/5
Beam	no	no	1/5
Lower haunch on right-hand beam	no	no	1/5
End plate on right-hand beam	no	no	1/5

☒ Drawing with welds

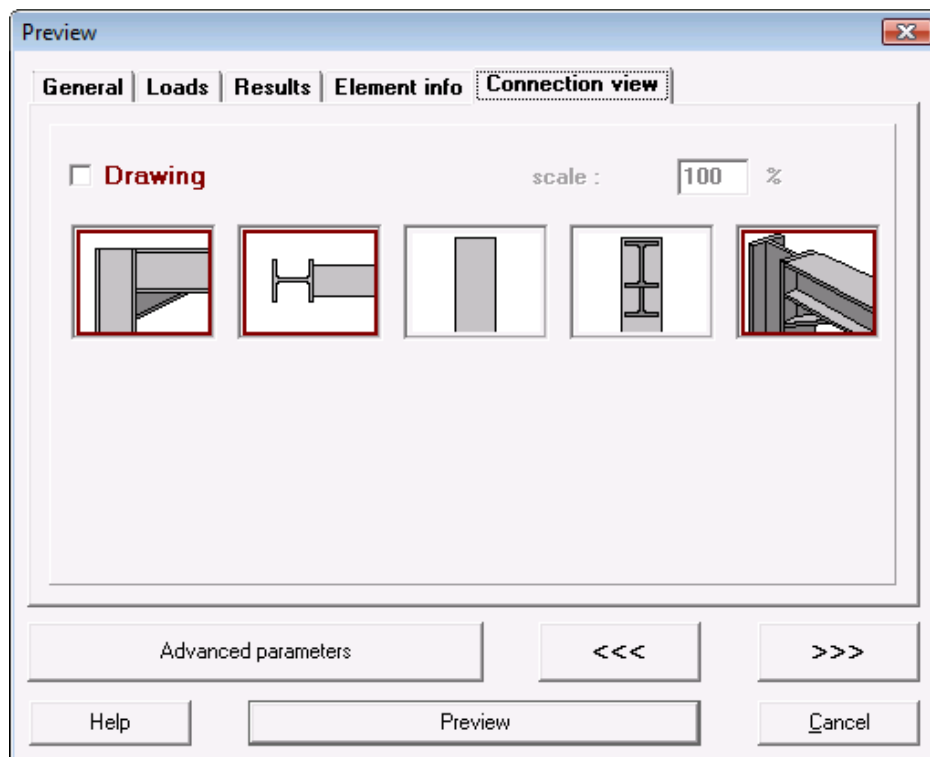
Advanced parameters <<< >>>

Help Preview Cancel

7.3.5 Tab page 'Connection view'

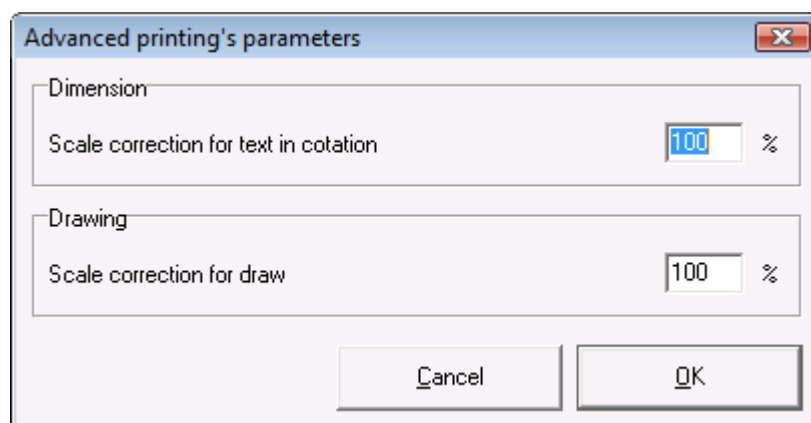
Select the option 'Connection view' to include one or more of the available connection views in the report. It should be noted that those views are intended only for qualitative reporting, giving an indication about connection

geometry. In no way are those views to be compared with the drawings as described in sections 7.3.1.2 and 7.3.4.



7.3.6 Advanced parameters


Finally, some information related to the dialogue window issued from the button 'Advanced parameters' with the Preview/Printing/Printing to RTF functions. This dialogue allows to modify the scale used for dimension annotation on drawings and the scale used for the drawings themselves.

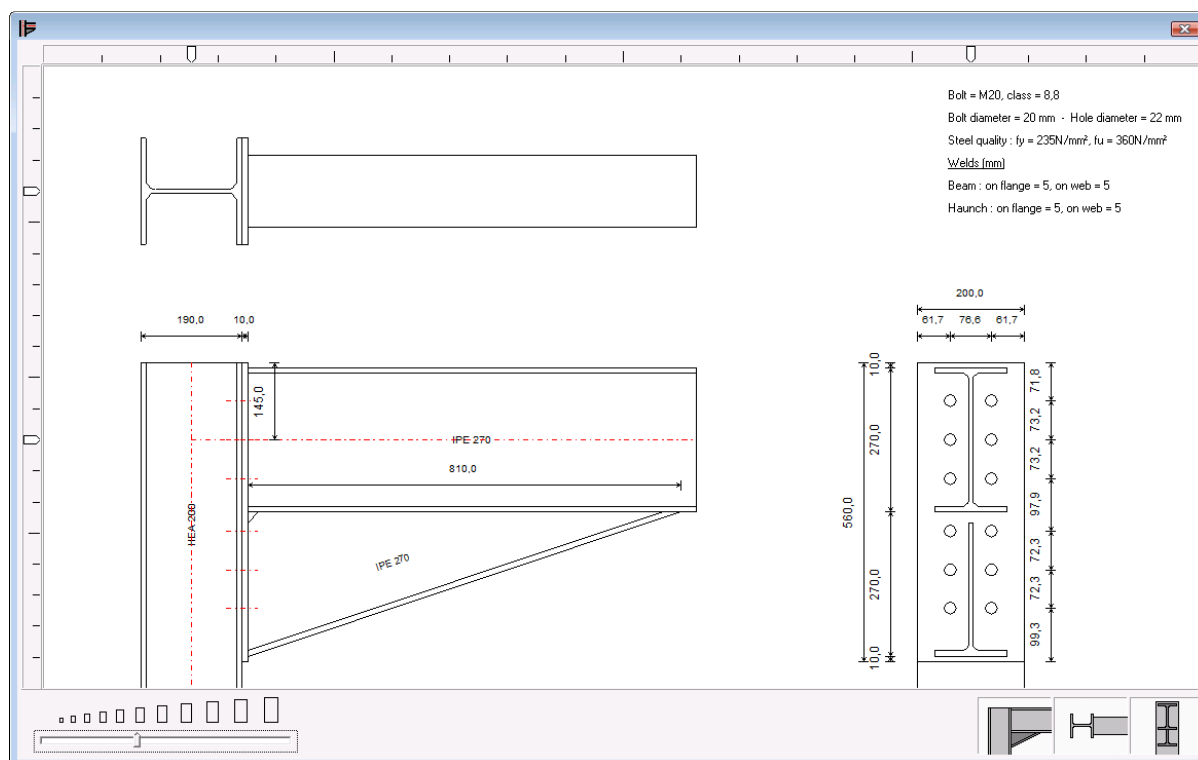


Depending on the type of printer being used, it may or may not be needed to change those parameters (which have a default value of 100%). It remains up to the user to check this out.

8 Plotting

8.1 2D connection drawings

Use the icon  of the Icon Toolbar to launch the dialogue window below which allows to specify how 2D connection drawings should be generated for the connection currently being analyzed.



This dialogue window offers following controls:

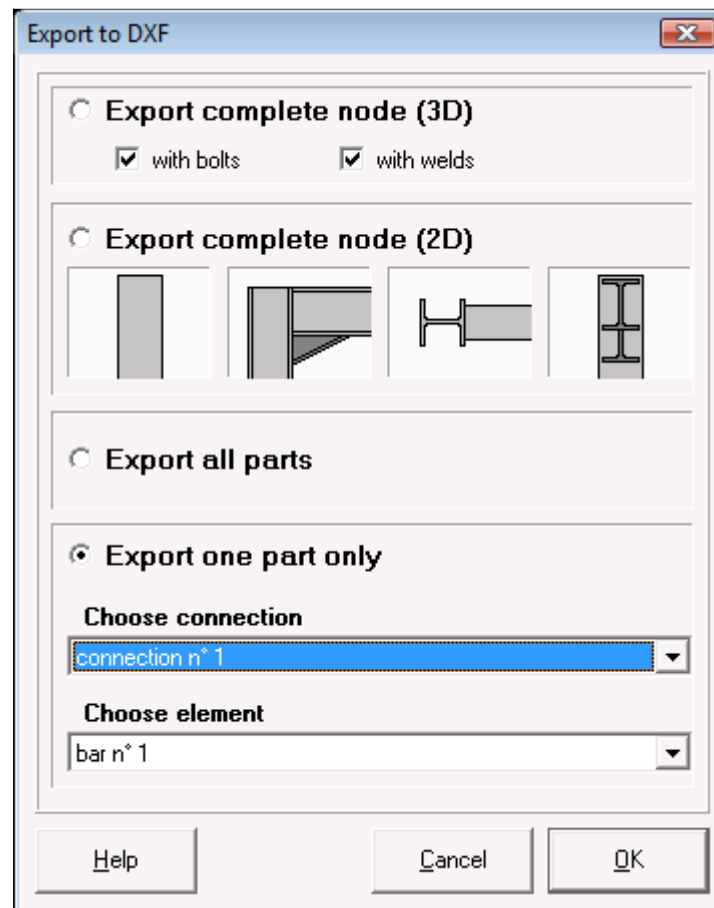
- a series of buttons at the right bottom corner which enable the user to specify which drawings are to be included,
- horizontal and vertical cursors which allow to control the position of the different drawings
- a slider to define the scale of drawings.

8.2 Export of element drawings to DXF

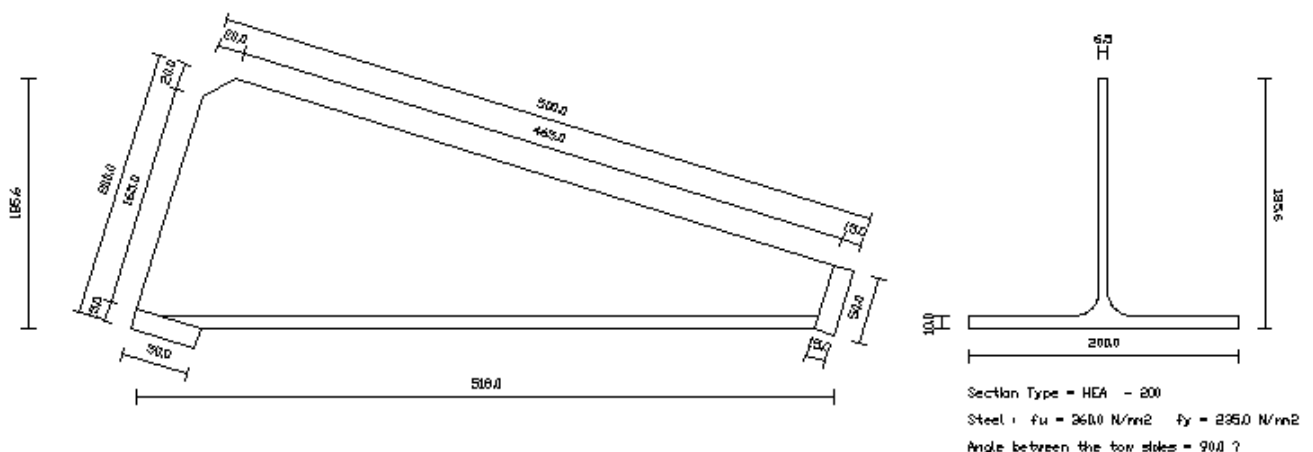
All elements which are part of a connection, as well as the connection itself, can be exported to a DXF file for further elaboration in a CAD environment. To do so, use the menu 'File – Export – Drawing to DXF'.

In the dialogue window below, make a choice between the 4 available options. In case only one element is to be exported to DXF, select the name of the element from the pull-down menu. In case the connection is a double connection (eg. beam-column-beam connection), the user should also make sure to specify which part of the connection is to be considered for export.

After confirmation by means of the 'OK'-button, PowerConnect will prompt where to save the DXF-file and what the name of the file should be.



Below, an example of a haunch which has been exported to DXF and which has then been imported into a CAD software.



Remark : the drawings which are exported to DXF always have a unit system with millimeter as length unit.