

# **Part 1:**

## **Getting Started with PowerConnect**

© 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

<b>1</b>	<b>INTRODUCTION .....</b>	<b>1</b>
1.1	INSTALLING POWERCONNECT .....	1
1.1.1	<i>Hard- &amp; software requirements.....</i>	<i>1</i>
1.1.2	<i>Installation procedure.....</i>	<i>1</i>
1.1.3	<i>Demo- and evaluation copies of the software.....</i>	<i>1</i>
1.2	WHY POWERCONNECT?.....	2
1.3	POWERCONNECT BENEFITS .....	2
1.3.1	<i>Design analysis according to EUROCODE 3 .....</i>	<i>3</i>
<b>2</b>	<b>POWERCONNECT TUTORIALS .....</b>	<b>6</b>
2.1	TUTORIAL 1: BOLTED BEAM TO COLUMN FLANGE (EC3) .....	7
2.1.1	<i>Setting up the model.....</i>	<i>7</i>
2.1.2	<i>Completing the geometry model .....</i>	<i>9</i>
2.1.3	<i>Defining the loads .....</i>	<i>10</i>
2.1.4	<i>Running a first connection design analysis .....</i>	<i>12</i>
2.1.5	<i>Optimizing the connection design.....</i>	<i>14</i>
2.1.6	<i>Re-running the connection design analysis on the optimized connection .</i>	<i>17</i>
2.1.7	<i>Some comments on the previous design analysis results.....</i>	<i>19</i>
2.2	TUTORIAL 2 : BOLTED BEAM TO BEAM WITH MOMENT END PLATE (EC3) .....	21
2.2.1	<i>Setting up the model.....</i>	<i>21</i>
2.2.2	<i>Defining the loads .....</i>	<i>25</i>
2.2.3	<i>Running a first connection design analysis .....</i>	<i>26</i>
2.2.4	<i>Improving the connection design .....</i>	<i>27</i>
2.2.5	<i>Re-running the connection design analysis on the modified connection...</i>	<i>28</i>
2.3	TUTORIAL 3 : COLUMN BASE WITH EXTENDED END PLATE (EC3) .....	31
2.3.1	<i>Setting up the model.....</i>	<i>31</i>
2.3.2	<i>Running the connection design analysis .....</i>	<i>37</i>
2.4	TUTORIAL 4 : BOLTED BEAM TO COLUMN WEB (EC3) .....	39
2.4.1	<i>Setting up the model.....</i>	<i>39</i>
2.4.2	<i>Running the connection design analysis .....</i>	<i>42</i>
2.4.3	<i>Improving the connection design .....</i>	<i>43</i>
2.4.4	<i>Re-running the connection design analysis on the modified connection...</i>	<i>44</i>
2.5	TUTORIAL 5: BOLTED SPLICE (EC3) .....	45
2.5.1	<i>Defining the loads .....</i>	<i>47</i>
2.5.2	<i>Running the connection design analysis .....</i>	<i>48</i>
2.6	TUTORIAL 6 : SHEAR CONNECTION – BEAM TO COLUMN FLANGE WITH FIN PLATE (EC3).....	50
2.6.1	<i>Setting up the model.....</i>	<i>50</i>
2.6.2	<i>Verifying the geometry model .....</i>	<i>52</i>
2.6.3	<i>Defining the loads .....</i>	<i>54</i>
2.6.4	<i>Running the connection design analysis .....</i>	<i>54</i>
2.6.5	<i>Improving the connection design .....</i>	<i>56</i>

2.6.6	<i>Re-running the connection design analysis on the modified connection...</i>	57
2.7	TUTORIAL 7 : SHEAR CONNECTION – BEAM TO BEAM WEB WITH BOLTED ANGLE CLEATS (EC3).....	58
2.7.1	<i>Setting up the model</i> .....	58
2.7.2	<i>Verifying the connection elements</i> .....	60
2.7.3	<i>Running the connection design analysis</i> .....	62
2.8	TUTORIAL 8 : HSS CONNECTION (CIRCULAR MEMBERS) .....	63
2.8.1	<i>Setting up the model</i> .....	63
2.8.2	<i>Defining the loads</i> .....	64
2.8.3	<i>Running the design analysis</i> .....	65
2.9	TUTORIAL 9 : HSS CONNECTIONS (RECTANGULAR MEMBERS) .....	66
2.9.1	<i>Setting up the model</i> .....	66
2.9.2	<i>Defining the loads</i> .....	67
2.9.3	<i>Running the connection design analysis</i> .....	68
<b>3</b>	<b>POWERCONNECT REPORTING TUTORIALS .....</b>	<b>69</b>
3.1	TUTORIAL 1 : BOLTED BEAM TO COLUMN FLANGE (EC3) .....	69
3.1.1	<i>Page setup</i> .....	69
3.1.2	<i>Report configuration</i> .....	71
3.2	TUTORIAL 2 : BOLTED SPLICE (EC3) .....	75
3.2.1	<i>Page setup</i> .....	75
3.2.2	<i>Report configuration</i> .....	75
3.3	TUTORIAL 3 : SHEAR CONNECTION – BEAM TO COLUMN FLANGE WITH FIN PLATE (EC3).....	80
3.3.1	<i>Page setup</i> .....	80
3.3.2	<i>Report configuration</i> .....	80

# 1 Introduction

## 1.1 Installing PowerConnect

### 1.1.1 Hard- & software requirements

PowerConnect is a 32-bit software program for the MS Windows operating system. Although it may be possible to run PowerConnect on previous versions of MS Windows (without any warranty from BuildSoft, however), it is highly recommended to use it on **MS Windows XP** or **MS Windows Vista**.

For a smooth operation of the software program, following requirements should be met:

- a minimum of 512 MB of RAM.
- A graphical card (by preference nVidia or ATI) with minimum 32MB RAM on-board, and supporting OpenGL

### 1.1.2 Installation procedure

The PowerConnect software can be installed from CD-ROM or over the internet. In this case, you should have a valid LOGIN and PASSWORD to access the “Customer Care” section of the BuildSoft web site <http://www.buildsoft.eu>. BuildSoft customers receive access to this protected section as part of their maintenance or lease service contract. Other people interested in evaluating PowerConnect receive valid access data for a 30-days period provided they register at the BuildSoft web site.

In case all defaults are accepted as proposed by the PowerConnect installation procedure, the software will be installed on the directory “C:\Program Files\Buildsoft\PowerConnect”.

### 1.1.3 Demo- and evaluation copies of the software

In case PowerConnect is installed for the first time on a MS Windows workstation, the user will be granted 30 days free access to the program’s full functionality. Once this period has expired, a valid license on a USB dongle is required to have continued access to PowerConnect. If no such valid license is available, the program will only deliver a demo license. Existing models can be opened with a PowerConnect demo license, but it will be impossible to define any changes to cross sections of connection members.

## 1.2 Why PowerConnect?

PowerConnect is an exceptionally easy to use software program. Connection design analyses that would require hours when done by hand, can be performed in a very limited time frame when PowerConnect is being used. At the same time, PowerConnect will offer a significantly higher results accuracy because of the more refined analysis methods that have been implemented.

PowerConnect's user interface has been designed to enable the design engineer to define as easily as possible modifications to existing connection designs and to test in the shortest possible time frame the impact of various design changes on the connection strength & stiffness. As a consequence, optimal connection design becomes feasible. Throughout this process, the user will be supported by well-documented dialogue windows, thus easing the design task and minimising error risk.

Although the underlying design analysis methods are quite complex in nature, the user will not be hindered by this complexity during the design analysis process. As the PowerConnect analysis engine is quite fast, results will almost immediately be available so that the impact of various design modifications can truly be tested to gain more insight into the effect of various design parameters.

Each part of every connection can be documented in full detail. The graphics are an excellent aid to visually control all connection design analyses.

At the end of the process, a clear and concise analysis report can be produced. Drawings of connection elements (along with the appropriate dimensions) can directly be included in such a report and/or can be exported to various CAD programs for further exploitation.

## 1.3 PowerConnect benefits

The PowerConnect software allows for design analysis of various steel connection types, with or without a wide range of stiffening elements.

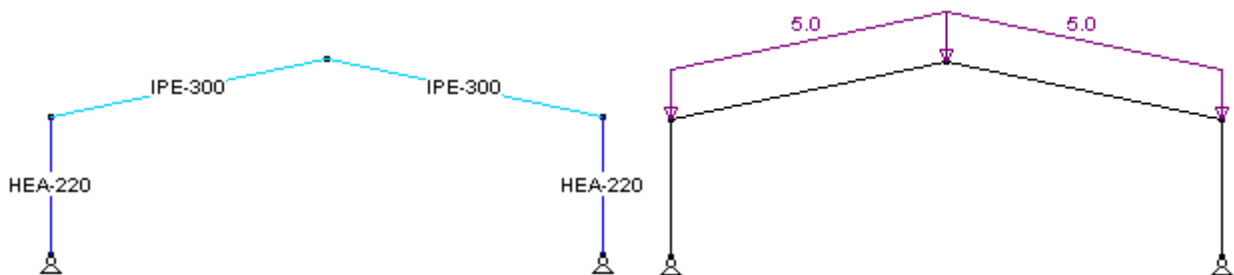
PowerConnect can be used as a stand-alone program, in which case all geometry and loads data are entered manually by the user. PowerConnect is also integrated within BuildSoft's **PowerFrame** program for 3D frame analysis. The PowerFrame Master license allows for an automated transfer of geometry and loads data from the 3D frame model to the PowerConnect environment for detailed steel connection analysis. During the transfer

procedure, the user can apply filter criteria in order to automate the selection of relevant load cases.

### 1.3.1 Design analysis according to EUROCODE 3

The EUROCODE 3 standard (prEN 1993 – 1-8: 2003, part 1.8) covers the design analysis of structural joints connecting H- or I-sections and hollow structural section joints. The major advantage of this standard is that it has been based upon the so-called component method. This implies that every connection is analysed such that all composing elements are calculated in detail. As a result of those analyses, under- or oversized elements can easily be identified within the connection.

The traditional approach in which a connection is either perfectly rigid or pinned, is therefore no longer used. The fact that such a traditional approach offers a rather conventional approach with less sense of reality can easily be illustrated by means of the following example. A 3D frame structure is subsequently analysed in PowerFrame using rigid and semi-rigid nodes. The impact of nodal stiffness will be evident from the example.



Scenario 1	Scenario 2
Calculation of bending moments and vertical displacements for a 3D frame with perfectly rigid nodes	Calculation of bending moments and vertical displacements for a 3D frame with semi-rigid nodes
bending moment	Bending moment
vertical displacement	vertical displacement

It is obvious that bending moments are redistributed when using semi-rigid nodes in the analysis. More rigid nodes will attract higher bending moments.



The use of semi-rigid nodes in frame analysis models will usually have following consequences:

- semi-rigid connections are simpler to make than rigid connections;
- bending moments will be redistributed throughout the frame structure, usually allowing for a more economical section choice;
- deflections will increase with semi-rigid connections.

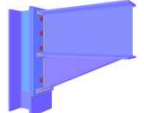
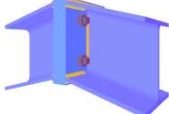
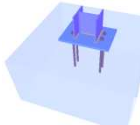
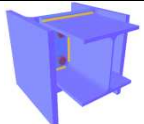
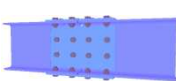

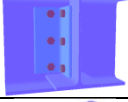


Thanks to PowerConnect, optimal connection design for a given set of loads truly becomes possible within a matter of minutes.

## 2 PowerConnect tutorials

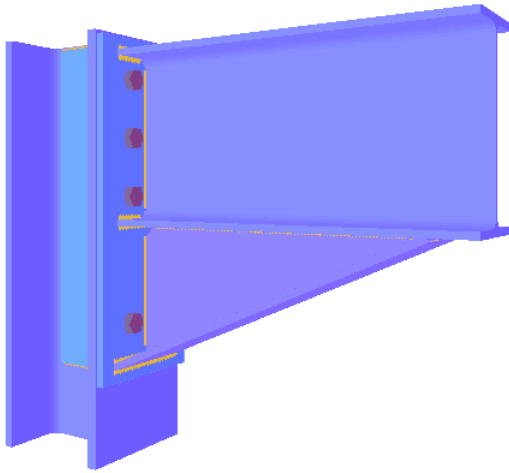
The best way to get acquainted with the PowerConnect software is to explore the product's functionality through a number of examples which highlight the various connection types supported by PowerConnect.

Those examples are covered in the tutorials below. The objective of the tutorials is not to provide a detailed and complete overview of the PowerConnect capabilities, but rather to concentrate on the information that is needed to get started with PowerConnect in a short time frame. The more detailed documentation of all product functions are covered in a separate reference manual.


### Inventory of tutorials:

Page	Section	Tutorial contents	Design code	Connection
page 7	§ 2.1	Tutorial 1: bolted beam to column flange	EC3	
page 21	§ 2.2	Tutorial 2 : bolted beam to beam with moment end plate	EC3	
page 31	§ 2.3	Tutorial 3 : column base with extended end plate	EC3	
page 39	§ 2.4	Tutorial 4 : bolted beam to column web	EC3	
page 45	§ 2.5	Tutorial 5: bolted splice	EC3	
page 50	§ 2.6	Tutorial 6 : shear connection – beam to column flange with fin plate	EC3	
page 58	§ 2.7	Tutorial 7 : shear connection – beam to beam web with bolted angle cleats	EC3	
page 63	§ 2.8	Tutorial 8 : HSS connection (circular members)	EC3	
page 66	§ 2.9	Tutorial 9 : HSS connections (rectangular members)	EC3	

## 2.1 Tutorial 1: bolted beam to column flange (EC3)



### 2.1.1 Setting up the model

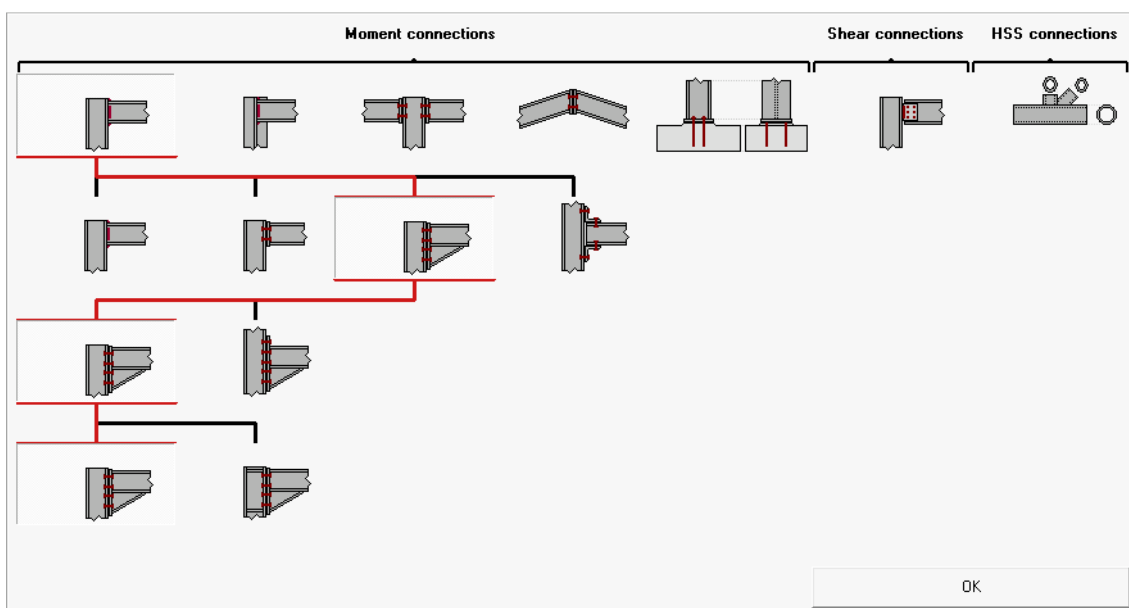
Launch PowerConnect and use the “File – New” menu instruction or the  icon. A window will appear (the so-called navigation window) in which you can select the type of connection that you want to design. Among the available moment connection types, choose the one labeled “Single-sided beam to column flange” (note: the labeling is done through the use of

tooltips that appear when you move the mouse over the available icons). You will then initiate a tree structure through which you can navigate to select a specific connection type.

Choose the following entries

- haunched beam end plate
- flush moment end plate

to define a haunched connection without any further stiffening elements. Confirm your final choice using the ‘OK’-button. Please check that the navigation window appears as illustrated below.

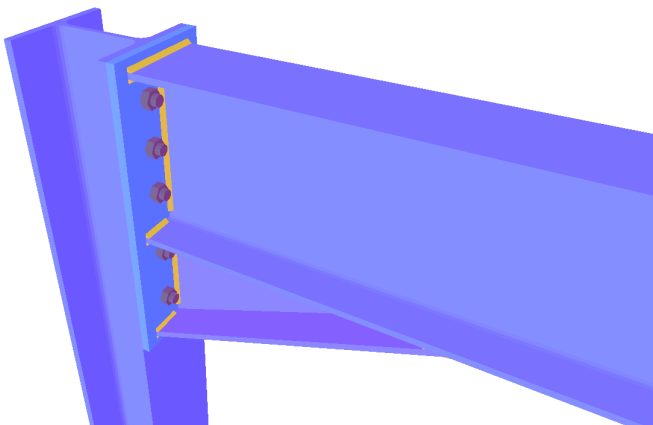



By means of the dialogue window that appears next, the connection definition can quickly be completed. While the proposed default values for length and slope of connected members will be accepted, beam and column sections are modified as follows:

- HEB 200 for the column member,
- IPE 300 for the beam member.

Verify all other values as proposed by the program and change them, if needed, as shown in the dialogue window below. It should be realized that it remains possibly to modify them at any time through a straightforward interaction with the geometry model just by double-clicking on an element that needs modification.



**Remark:** it is possible that some of the input fields in the above dialogue contain parameters rather than numbers (or even arithmetic expressions, rather than parameters by themselves). In this case, the corresponding dimension is linked to the dimension of another element of the connection. The actual meaning of the parameters can be clarified by means of the icon



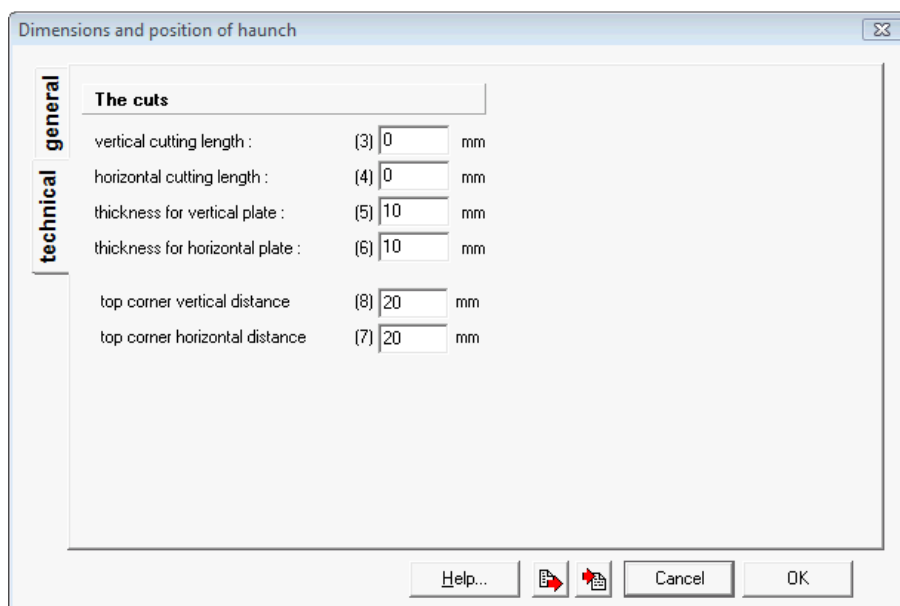
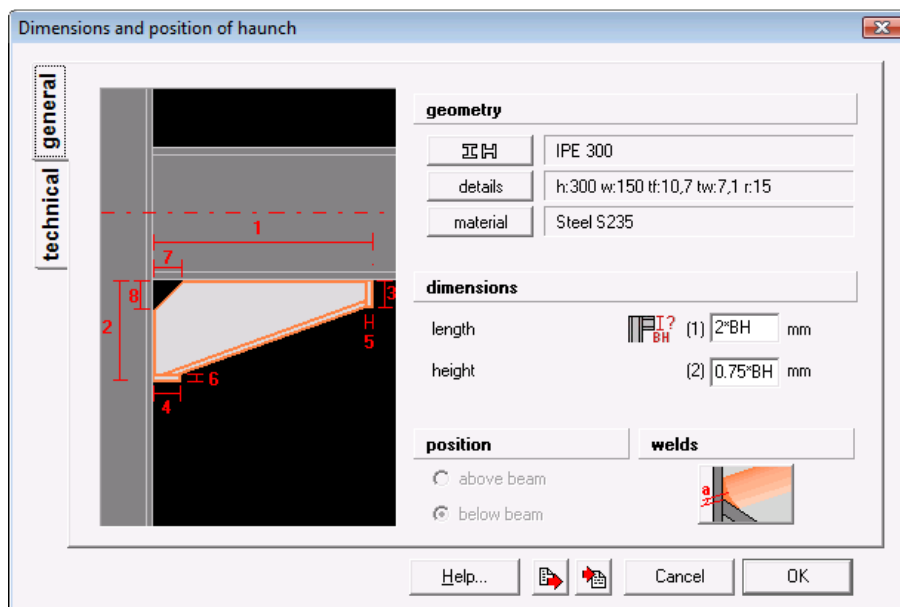
button  in the dialogue window. For more information on this topic, consult the PowerConnect reference manual.


This definition is finally confirmed through the 'OK'-button, and as a consequence a 3D geometry model will be created and visualized on the screen.

## 2.1.2 Completing the geometry model

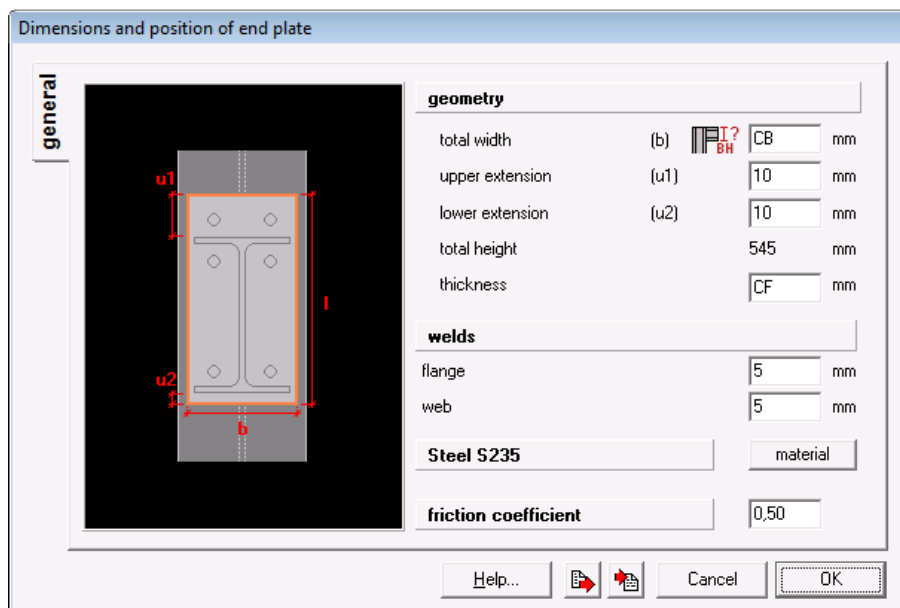
The scale which is used for model representation can be modified using the 'Zoom in' (  ) and 'Zoom out' (  ) icons of the icon toolbar or by using the scrollbar of the mouse. The geometry model can furthermore be moved across the screen by holding down the left-hand mouse button while moving the mouse across the 'Geometry' window (take care not to position the mouse such that any of the connection elements is selected, it should really be positioned on the drawing canvas of the 'Geometry' window).

The dimensions of various elements of the connection that has just been created will now be further verified and modified, if needed. Double-click for instance with the left mouse button on the haunch and verify if the fields of the dialogue window are as shown below:




It should be noted that by default, the haunch is based on a IPE300 section, which is the section used for the beam element. If needed, the haunch can be based on a different section defined through the icon  on the 'general' tab sheet.

The dimensions and the position of the end plate should also be verified. Again, double click on the end plate in the 3D 'Geometry' window to open the dialogue window below and fill out the parameters as shown hereafter.



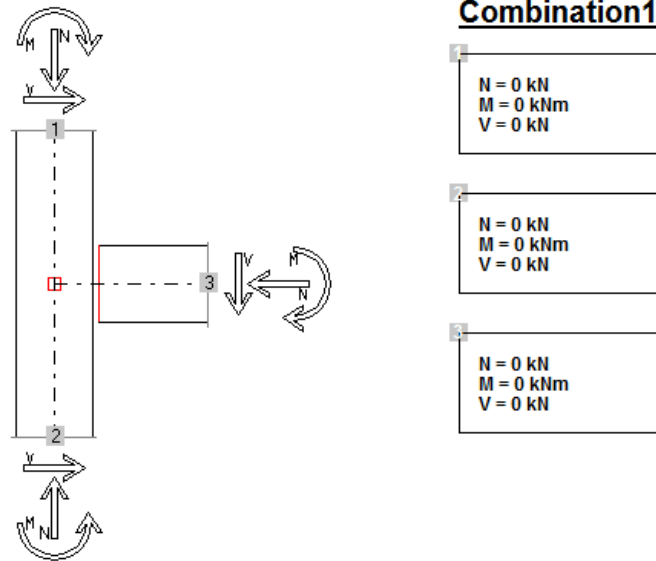
Each time a set of dimensions is confirmed for a selected element, PowerConnect will automatically verify the position and alignment of individual bolts. In case a problem is identified (either with respect to minimum distances imposed by standards or with respect to minimum distances specified by the user), PowerConnect will issue a warning on the matter.

### 2.1.3 Defining the loads

Now that the basic geometry definition has been completed, loads can be assigned to the connection. Switch to the "Loads" window by clicking on the -icon that is part of the 'Windows'-toolbar:



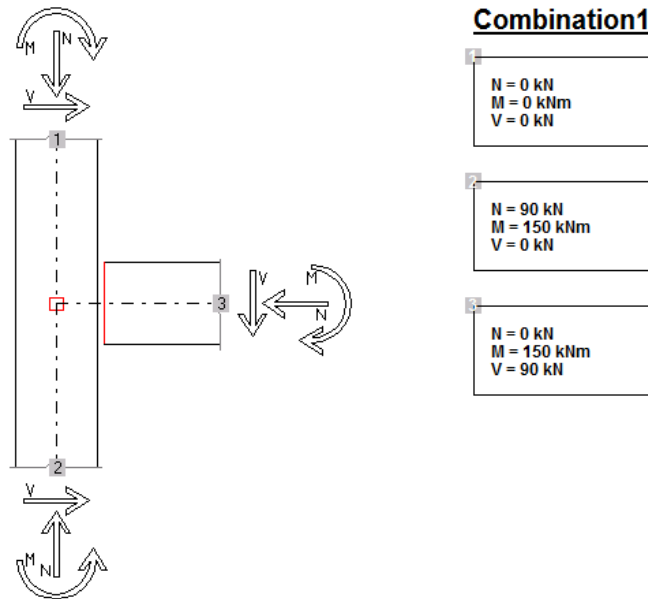
The following information will appear in the 'Loads'-window.



Following loads should be applied:


- at the lower end of the column, a bending moment of 150 kNm and a normal force of 90 kN. Activate the appropriate dialogue by clicking with the mouse on the label “2” at the lower end of the column and by filling out the values in the dialogue which pops up;
- at the right end of the beam, a bending moment of 150 kNm and a shear force of 90 kN. Activate the appropriate dialogue by clicking with the mouse on the label “3” at the right end of the beam and by filling out the values in the dialogue which pops up.

As a result, the information in the ‘Loads’-window should now appear as follows:

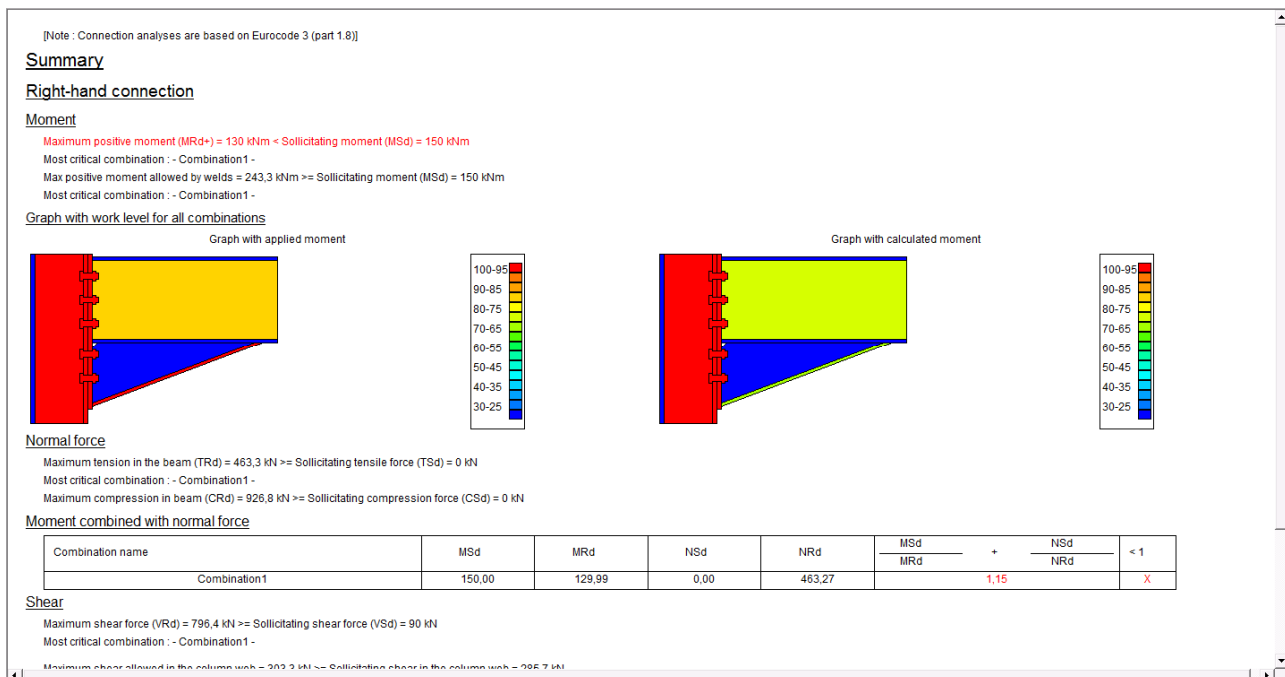


## 2.1.4 Running a first connection design analysis

The connection design analysis can be initiated in 3 different ways:

- through the menu command 'Analysis – Analysis',
- through the shortcut F9,
- through the  icon in the Analysis toolbar.

The results of the design analysis are presented below.



A more in-depth analysis of the above results already provides following feedback:

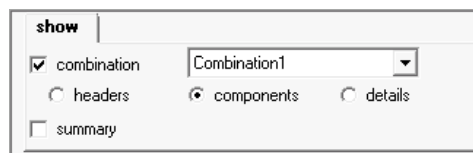


- the current connection design doesn't fulfill the imposed strength requirements, as the ratio of applied loading to resistance achieves a value of 1.15 (looking at the combination of **bending moment and normal force**),
- from the results available for **moments** only, it can be derived that the welds are OK
- from the **work level graph** valid for **moments** only, it shows that the column web, column flange, bolts and haunch flange are loaded up to their maximum capacity,
- from the results available for **shear** only, it can be concluded that the connection does provide sufficient shear strength

For a more detailed interpretation of analysis results, it is possible to switch to a more advanced reporting lay-out. To this purpose, click with the mouse on the field 'Results preferences' at the right-hand bottom of the PowerConnect window.



In the dialog which appears now, ask for the results related to Combination 1, rather than to look at summary results. For the time being, it is sufficient to ask for all components results without any further details (as shown below).



This will provide the information shown below (only the first part of the screen report is shown; the remaining information can be screened by using the scrollbar on the right hand side of the PowerConnect window), from which following conclusions can be drawn:

- only bolt rows 1 & 2 contribute to the connection's resistance in bending
- the most critical components of the connection are the end plate (failure in bending) and the column web (shear failure)

This information, consistent with the information provided by the work level graphs shown before, allows to modify the connection to achieve the desired level of resistance.

**Results with components for -Combination1-**

**Right-hand connection**

**Moment**

Total moment resistance (MRd) = 130 kNm < Sollicitating moment (MSd) = 150 kNm  
Sollicitation must be reduced or connection must be adapted

Bolt row n°1, Restrictive component:  
End plate in bending (mode 2) (mode 3), Moment : 106,3 kNm

Bolt row n°2, Restrictive component:  
Column web in shear, Moment : 23,7 kNm

Bolt row n°3, Restrictive component:  
Column web in shear, Moment : 0 kNm

Bolt row n°4, Restrictive component: —, Moment : 0 kNm

Bolt row n°5, Restrictive component: —, Moment : 0 kNm

Moment allowed by welds = 243,3 kNm >= Sollicitating moment (MSd) = 150 kNm

**Components**

**Bolt row**

level arm and tensile force in bolt-row

n° bolt-row	1	2	3	4	5
level arm (mm)	446,4	363,6	280,7	171	86
BFRd(kN)	141,1	141,1	141,1	141,1	141,1

**Column web in shear**

Shear resistance of column web = 303,3 kN  
Beta = 1

**Column web in compression**

Compression resistance of web = 308,9 kN

**Haunch flange in compression**

Compression resistance of flange = 353,2 kN

**Column flange in bending**

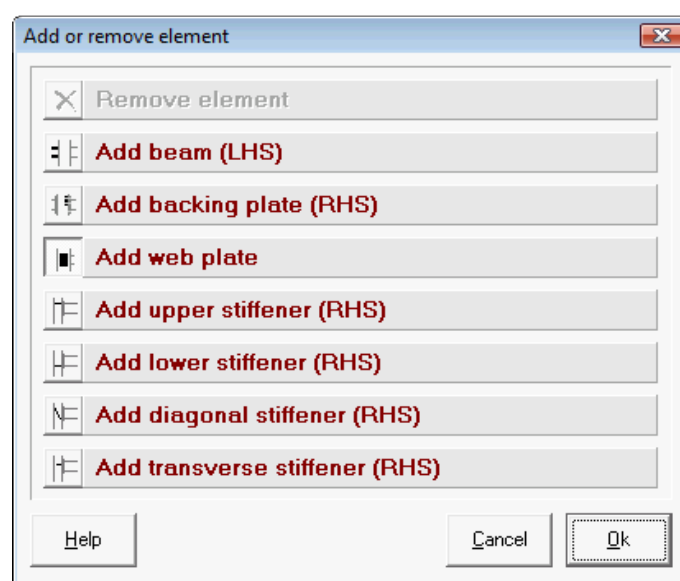
FitxRd (kN)

(1): 250,4	(2+1): 457,4	(3+...+1): 664,4	(4+...+1): 887,6	(5+...+1): 1095,9
------------	--------------	------------------	------------------	-------------------

## 2.1.5 Optimizing the connection design

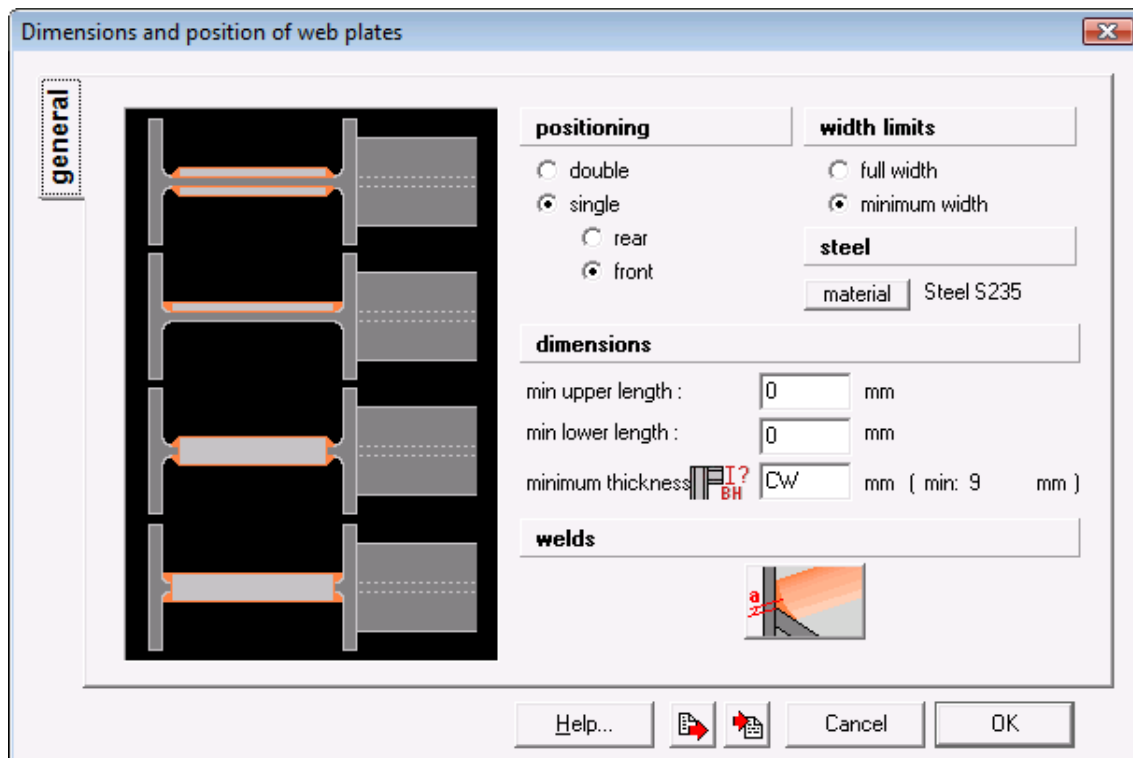
Based on the results provided by the first design analysis, a number of modifications will now be defined to the existing connection such that its resistance is increased by focusing on the most critical component information.

In a first step, it is decided to add a web plate to counteract the column web shear failure. Therefore, select the column web in the 'Geometry' window and click on the right-hand mouse button to make the following window appear on the screen.



Choose the option "Add web plate", and confirm this choice by means of the 'OK'-button.

In order to verify and to possibly modify the web plate dimensions, double-click on the web plate that is now visible on the geometry model. A new window will appear providing full access to all geometry details of the web plate:



Make sure all parameters match with the ones presented above.

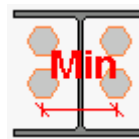
Finally, double-click on one of the bolts that is part of the 3D geometry model to have access to the definition of bolts & bolts lay-out. As only 2 bolt rows have an active contribution to the connection's bending resistance, a number of bolt rows will now be removed as follows:

- select one of the bolts belonging to bolt row 3 and then use the icon

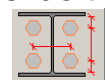


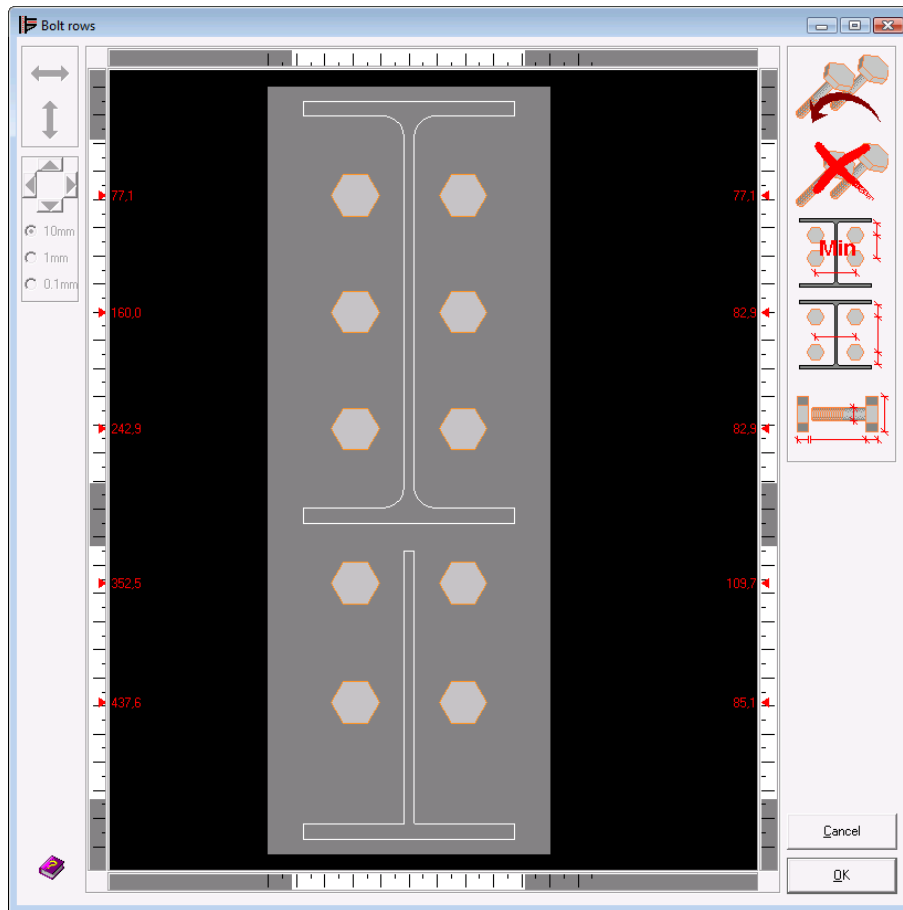
to remove this bolt row,

- out of the remaining 4 bolt rows, again select bolt row 3 to remove it using the same icon,





- then use the icon 'Minimum optimisation'

positions with minimum intermediate distance (the other icon  will optimize bolt row positions with maximum intermediate distance)

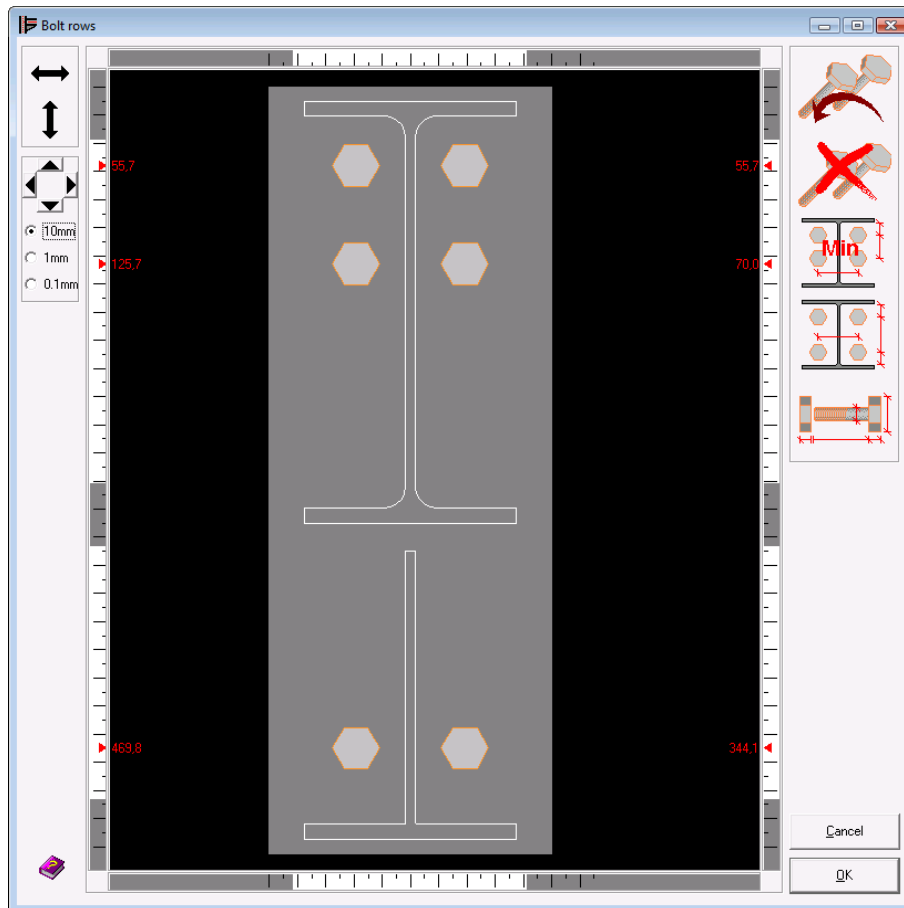


In case the user wants to further fine-tune bolt row positions, this can be easily achieved as follows:

- select the bolt row which needs re-positioning
- activate the vertical displacement function  and move the selected bolt row as required
- alternatively, the arrows  can be used to move the selected bolt row according to the selected precision level;



Currently, it is assumed that no such modifications are implemented to the lay-out that was obtained using the minimum optimization function. The bolt row lay-out should thus look like



## 2.1.6 Re-running the connection design analysis on the optimized connection

Running the connection design analysis will now produce following summary results:

[Note : Connection analyses are based on Eurocode 3 : EN 1993-1-8:2005]

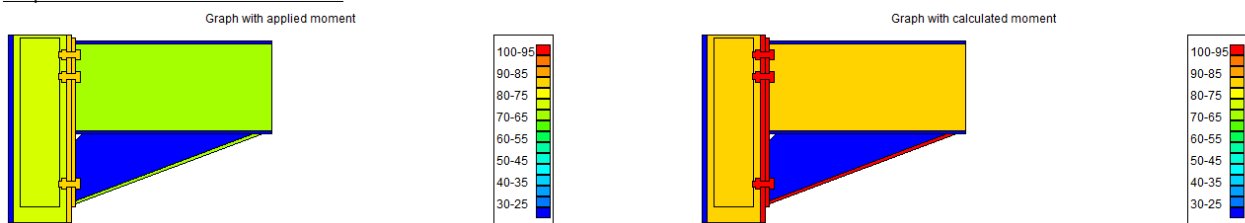
### Summary

#### Right-hand connection

##### Moment

Maximum positive moment (MRd+) = 178.1 kNm >= Sollicitating moment (MSd) = 150 kNm  
 Most critical combination : - combinatie 1 -  
 Max positive moment allowed by welds = 243.3 kNm >= Sollicitating moment (MSd) = 150 kNm  
 Most critical combination : - combinatie 1 -

##### Graph with work level for all combinations



##### Normal force

Maximum tension in the beam (TRd) = 632.5 kN >= Sollicitating tensile force (TSd) = 0 kN  
 Most critical combination : - combinatie 1 -  
 Maximum compression in beam (CRd) = 1317.1 kN >= Sollicitating compression force (CSd) = 0 kN

##### Moment combined with normal force

Combination name	MSd	MRd	NSd	NRd	$\frac{MSd}{MRd} + \frac{NSd}{NRd}$	< 1
combinatie 1	150.00	178.14	0.00	632.46	0.84	V

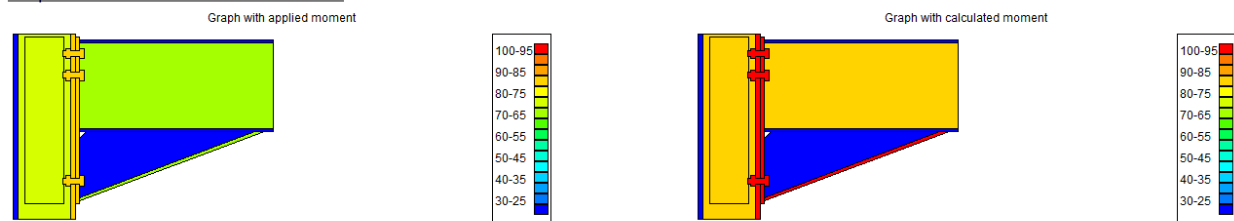
##### Shear

Maximum shear force (VRd) = 371.4 kN >= Sollicitating shear force (VSd) = 90 kN

Those results confirm that all strength requirements are met by the connection, after the described changes have been introduced into the geometry model. Just like before, it remains possible to look at more detailed analysis results by clicking with the mouse on the field 'Results preferences' at the right-hand bottom of the PowerConnect window and by asking for the appropriate detail level for results reporting.

For the time being, remain with the Summary report but use the scrollbar on the right-hand side of the PowerConnect window to go to the end of the summary report (as shown below).

#### Graph with work level for all combinations



#### Normal force

Maximum tension in the beam (TRd) = 632.5 kN >= Sollicitating tensile force (TSd) = 0 kN  
 Most critical combination : - combinatie 1 -  
 Maximum compression in beam (CRd) = 1317.1 kN >= Sollicitating compression force (CSd) = 0 kN

#### Moment combined with normal force

Combination name	MSd	MRd	NSd	NRd	$\frac{MSd}{MRd} + \frac{NSd}{NRd}$	< 1
combinatie 1	150.00	178.14	0.00	632.46	0.84	V

#### Shear

Maximum shear force (VRd) = 371.4 kN >= Sollicitating shear force (VSd) = 90 kN  
 Most critical combination : - combinatie 1 -  
 Maximum shear allowed in the column web = 439.5 kN >= Sollicitating shear in the column web = 285.7 kN  
 Most critical combination : - combinatie 1 -

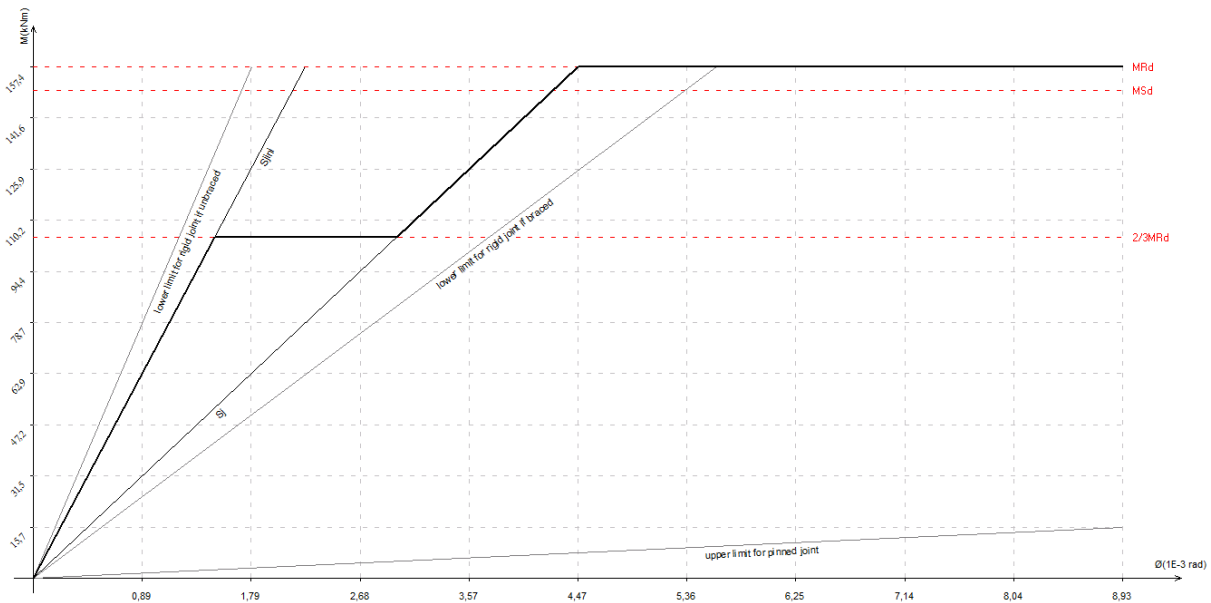
#### Stiffness

##### For a positive moment

Sjini = 70484 kNm/Rad  
 Sj = 35242 kNm/Rad  
 The connection is Semi-Rigid.  
 Most critical combination : - combinatie 1 -

It will now be visible that the PowerConnect design analysis has not only calculated strength characteristics, but stiffness characteristics as well. Those stiffness results can also be represented graphically by means of the

 icon of the 'Windows' toolbar:



Up to 2/3 of the connection's resistance in bending, an initial stiffness will be assigned to the connection ( $S_{j,ini}$ ). For higher loading levels, a reduced stiffness will be used.

## 2.1.7 Some comments on the previous design analysis results

A more detailed analysis of the previous calculation results will show that the bending moment resistance of the connection is actually determined by 2 connection elements only, being the end plate and the haunch. Changing one of those elements will have a significant impact on connection resistance.

On the other hand, the 3<sup>rd</sup> and the 4<sup>th</sup> bolt row have no contribution at all to connection resistance. Removing those bolt rows will not decrease strength characteristics (except for shear loading).

*Remark: the contribution of the first bolt row is limited by bending failure of the end plate and by tension failure of the bolts. For each bolt row, 3 failure modes are considered. The first failure mode will consider plastic failure of the plate while the bolts remain fully elastic. The second failure mode will consider joint failure of the plate in bending and the bolts in tension, whereas with the third failure mode only the bolts will be involved in the failure mechanism. In case failure mode 3 is reported as the critical failure mode, it will be necessary to change the bolts in order to increase bending moment resistance. In case failure mode 2 is reported as the critical failure mode, both end plate and bolts need to be changed. In case failure mode 1 is reported as the critical failure mode, it is sufficient to change the end plate only.*

As far as welds are concerned, Eurocode 3 assumes that shear resistance is delivered by welds on the web where-as bending moment resistance is delivered by welds on the flanges. As PowerConnect allows for different throat thicknesses for different types of welds, welds can really be customized to the needs of the user. In most cases, the bending moment resistance offered by the welds exceeds the bending moment resistance calculated from the bolts. If this is not the case, it will be needed to increase the welds' throat thickness.

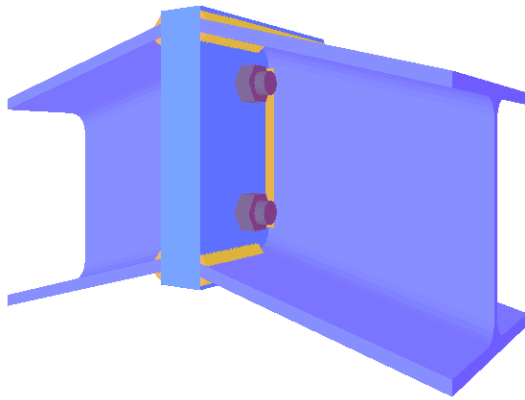
In the example that was considered, shear resistance is significantly higher than shear loading. In case shear loading would have been much higher, more or stronger bolts would be needed. If on the other the problem would be related to the column web, one or two (one on each side) web stiffening plates would be required. In case even such a solution proves to be unsatisfactory, the column section itself may need to be adapted.

Let us also consider the results related to normal forces. PowerConnect calculates maximum compressive and tensile forces that can be resisted by the connection. Combined with the calculated bending moment resistance, PowerConnect will thus be able to verify if following combined condition can be met for the applied loading:


$$\frac{M_{Sd}}{M_{Rd}} + \frac{N_{Sd}}{N_{Rd}} \leq 1$$



## 2.2 Tutorial 2 : bolted beam to beam with moment end plate (EC3)



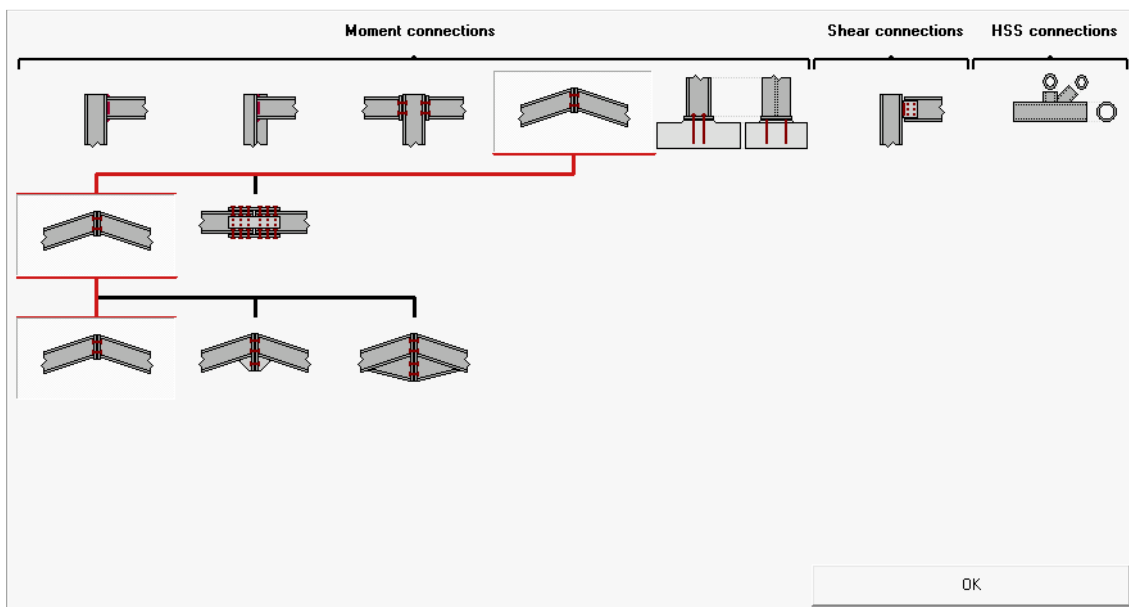
### 2.2.1 Setting up the model

To define the model for this second tutorial, click on the 'New' icon  of the icon toolbar. Among the connection types which are available in the navigation window, select the one labeled "Beam to beam" (remember: the labeling is done through the use of tooltips that appear when you move the mouse over the

available icons). Then further navigate through the tree structure by choosing the following entry

- bolted moment end plate

to finally select a beam to beam connection without any further stiffening elements. Confirm this final choice using the 'OK'-button, checking that the navigation window appears as illustrated below.



In the window that appears next, modify the data as follows:

- beam section : HEA 200
- beam length : 3000 mm

- slope of beam :  $-15^\circ$

to arrive at the input data as shown below.

**Bolted beam - beam connection**

☐ Braced

**Beam**

length: 3000 mm

slope:  $-15^\circ$

welds: 5 mm

Profile: HEA 200

**End plate**

thickness: CF mm

width: CB mm

**Bolts**

type: M 20

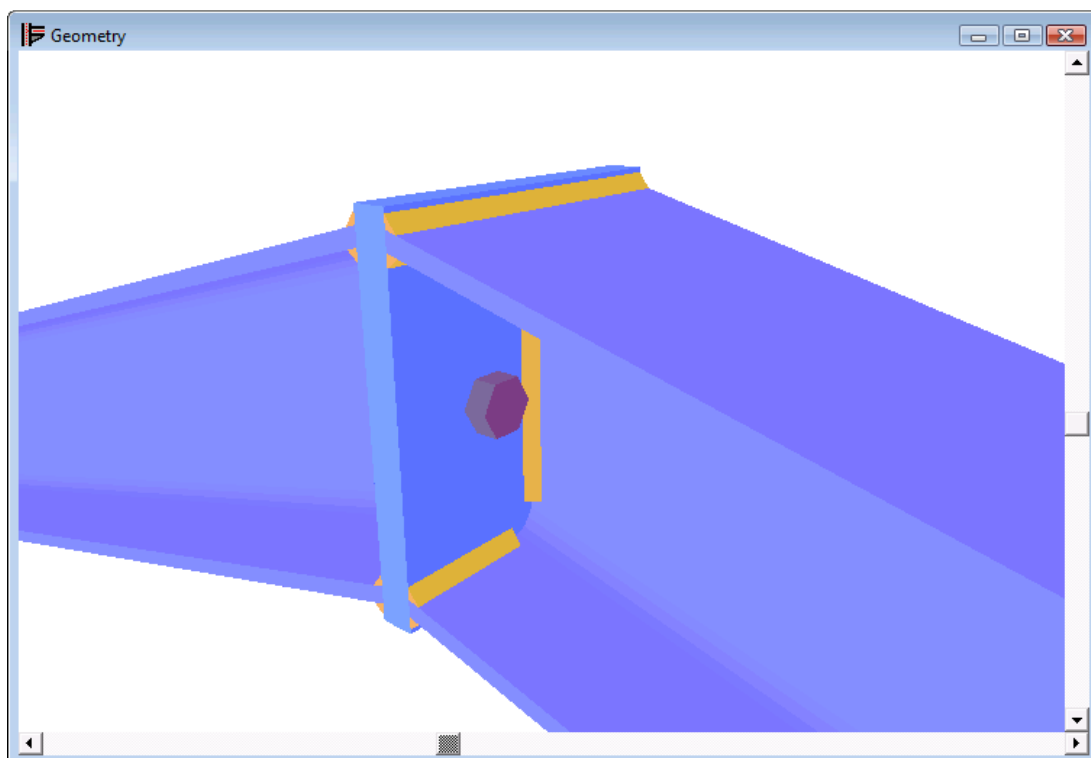
class: 8.8

min. vertical distance: 70 mm

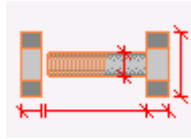
horizontal distance: 76.6 mm

Buttons: Help, Cancel, OK

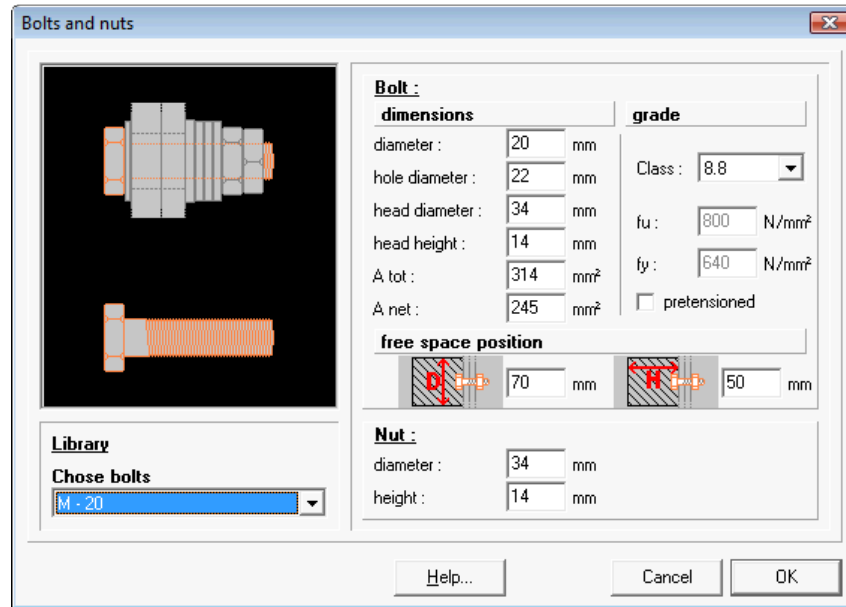
Confirm by means of the 'OK'-button to make following geometry model appear.



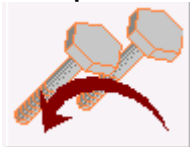
To add a bolt row to the single one that is proposed by PowerConnect, double-click on one of the bolts to make the bolt definition dialogue appear.



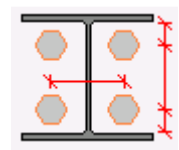
Use the icon 'Bolt details' of this dialogue window to gain access to the more detailed bolts specification. On the left hand side of the dialogue window, a bolt type can be selected from the bolts library. If needed, individual parameters as net surface, free spacing, ... can still be adapted.



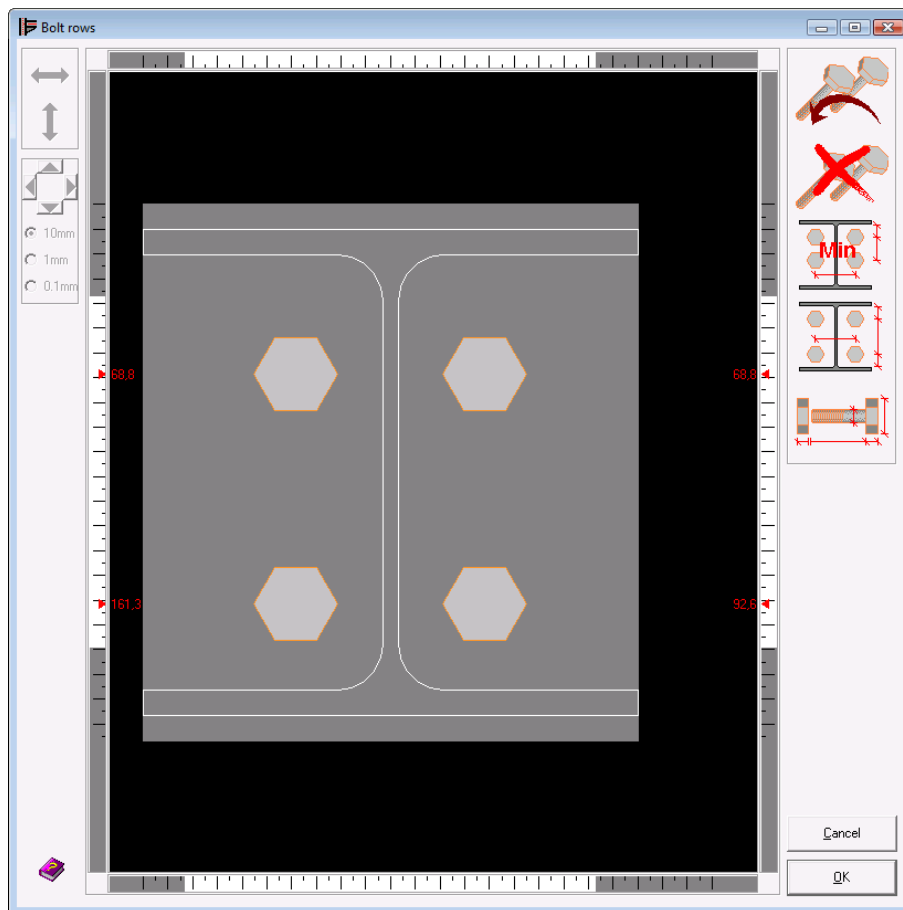
For the time being, do not implement any changes to the values proposed by the software, but simply use the 'OK'-button to confirm the current selection of bolt parameters. Having returned to the bolt definition window, now use the



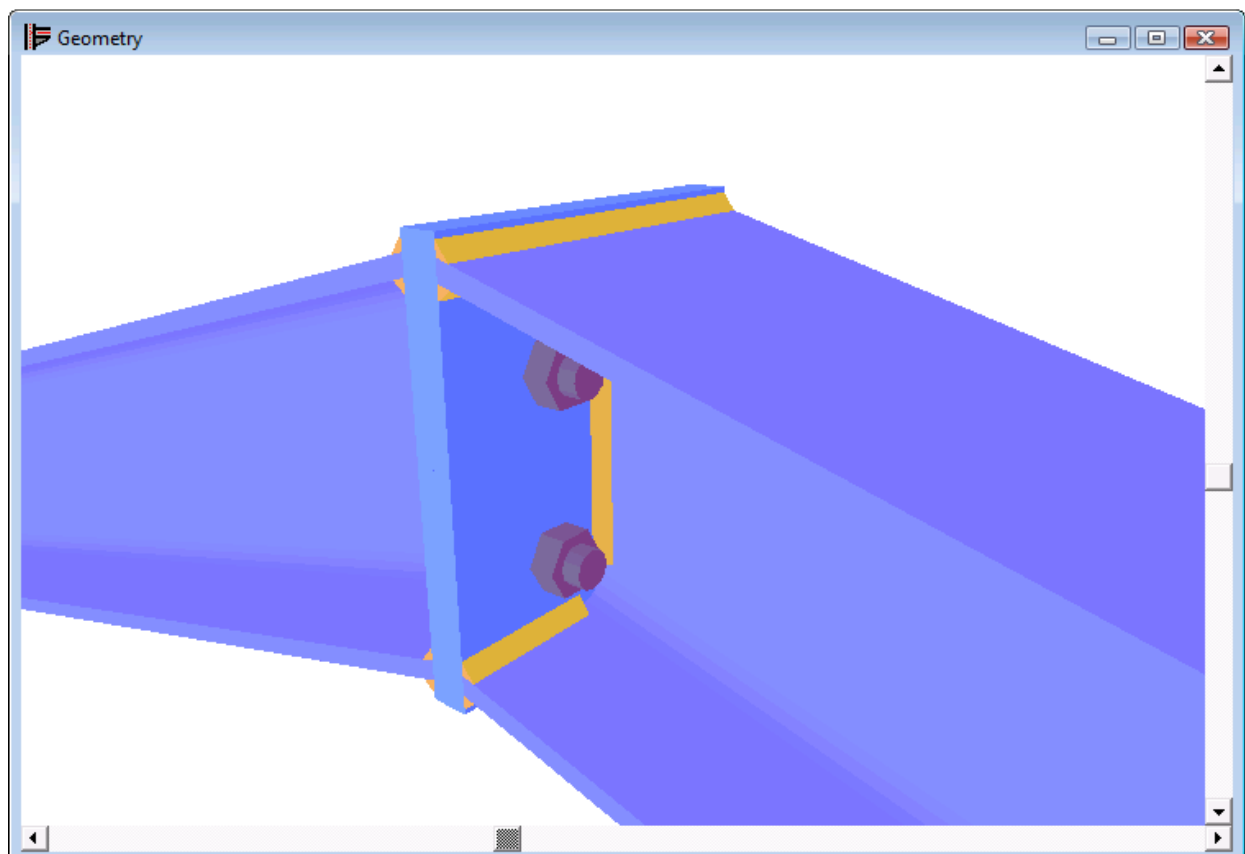
icon to add a second bolt row.




Then optimize bolt row lay-out by means of the 'Optimization' icon such that the following lay-out is obtained:



As a result, the 3D geometry model will look as follows in the 'Geometry' window:

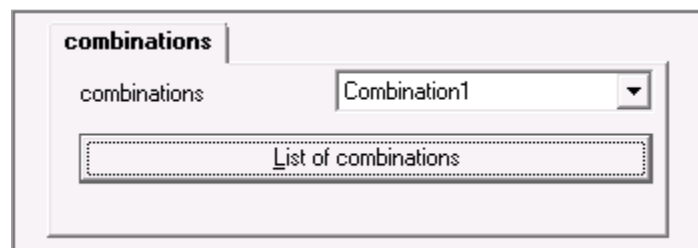


## 2.2.2 Defining the loads

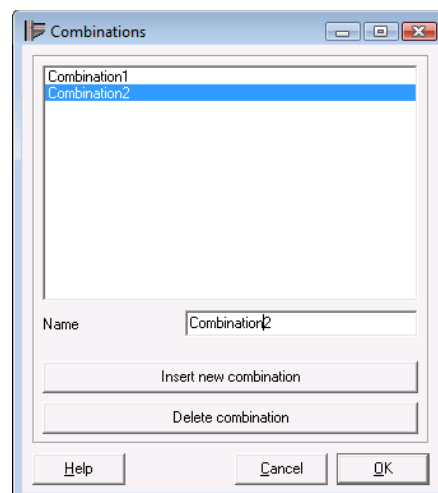
Now switch to the 'Loads'-window by means of the  icon. 2 loads combinations will be defined:

- in a first combination, a bending moment of 80 kNm will be applied at both sides of the connection
- in the second combination, a bending moment of 60 kNm and a compressive force of 150 kN will be applied at both sides of the connection

By default, PowerConnect presents only 1 loads combination. Before the values for this first combination will actually be filled out, the second combination that is needed with this model will first be created. To do so, click on the label 'Loads preferences' at the right hand bottom of the 'Loads'-window. Then use the button "List of combinations" to open the appropriate definition dialogue.



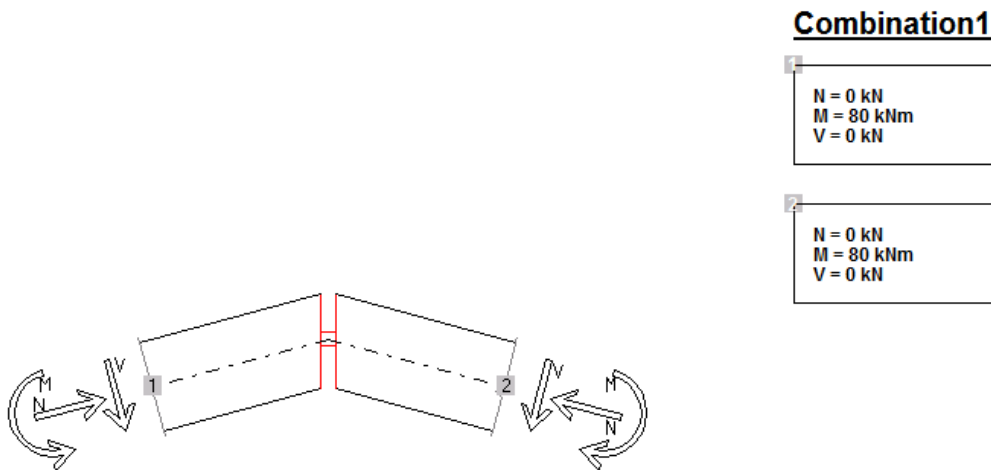
Use the 'Insert new combination'-button and specify the name "Combination2" for the new combination that has just been created. Now click 'OK' to confirm those definitions and to quit the window



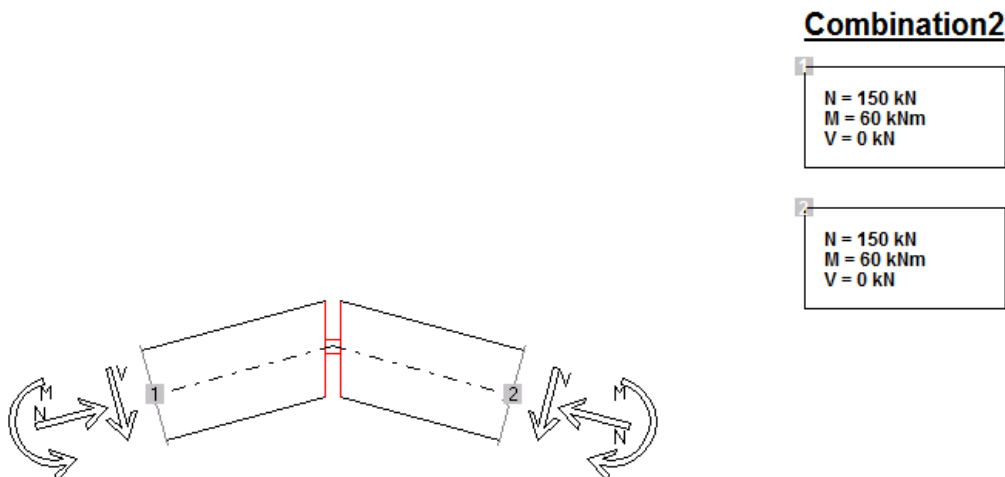
Make sure to leave the combination list visible at the right hand bottom of the 'Loads'-window and check if 'Combination1' is the active combination. Then

enter the appropriate load values for this combination, by clicking on the small labeled numbers which are visible on the geometry representation in the 'Loads'-window.

This should deliver the following result :

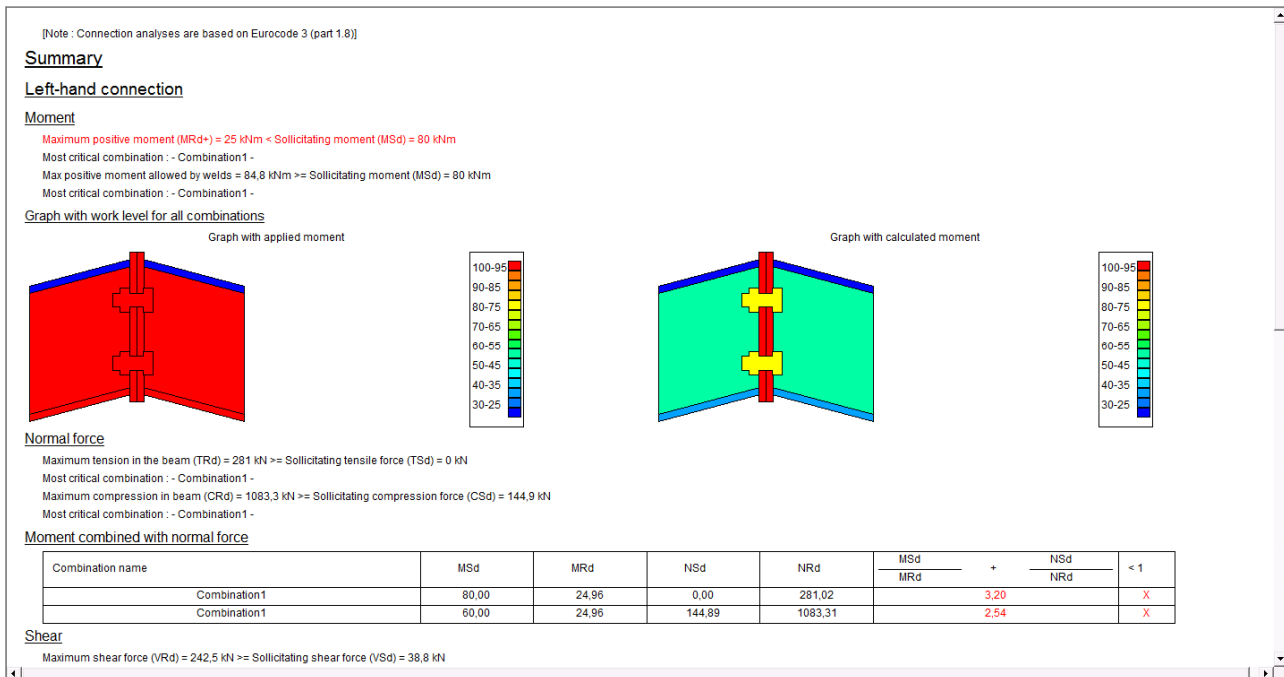


Now make sure 'Combination2' becomes the active loads combination (combination list at the right hand bottom of the 'Loads'-window !) and enter the loads data as follows:



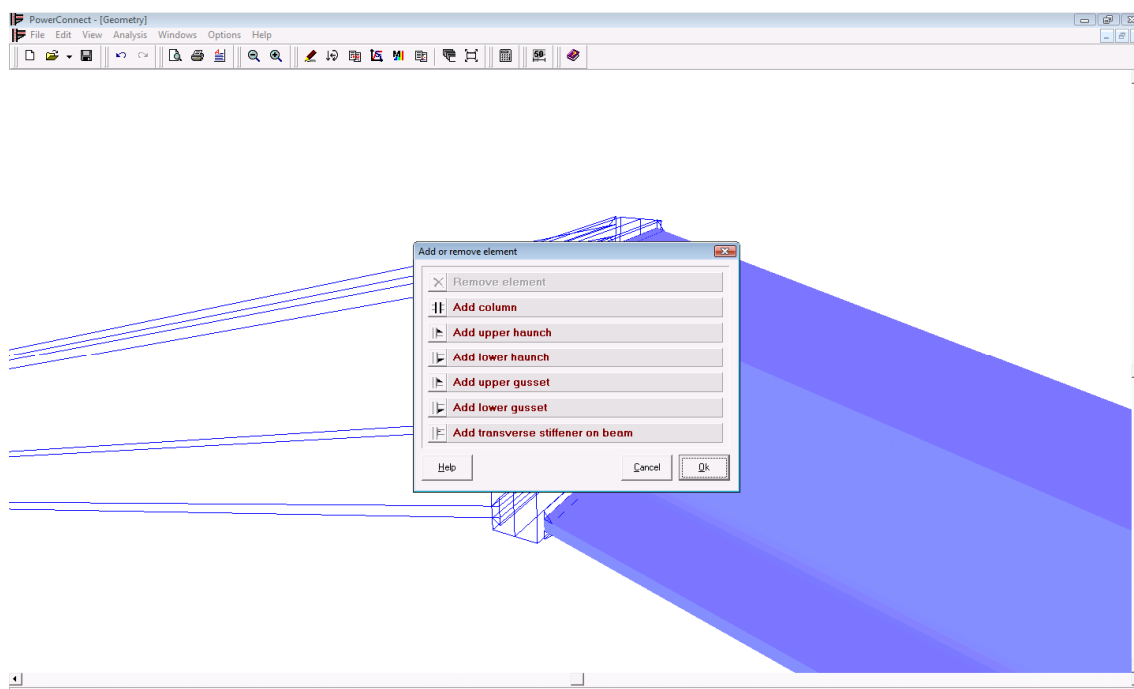
## 2.2.3 Running a first connection design analysis

As the connection is perfectly symmetric, it is of course sufficient to inspect the analysis results only for one side of the connection. For the time being, this inspection will be limited to the results summary (so the option "Summary" should be active when opening the window by clicking on the "Results preferences" label at the right hand bottom of the 'Results' window).



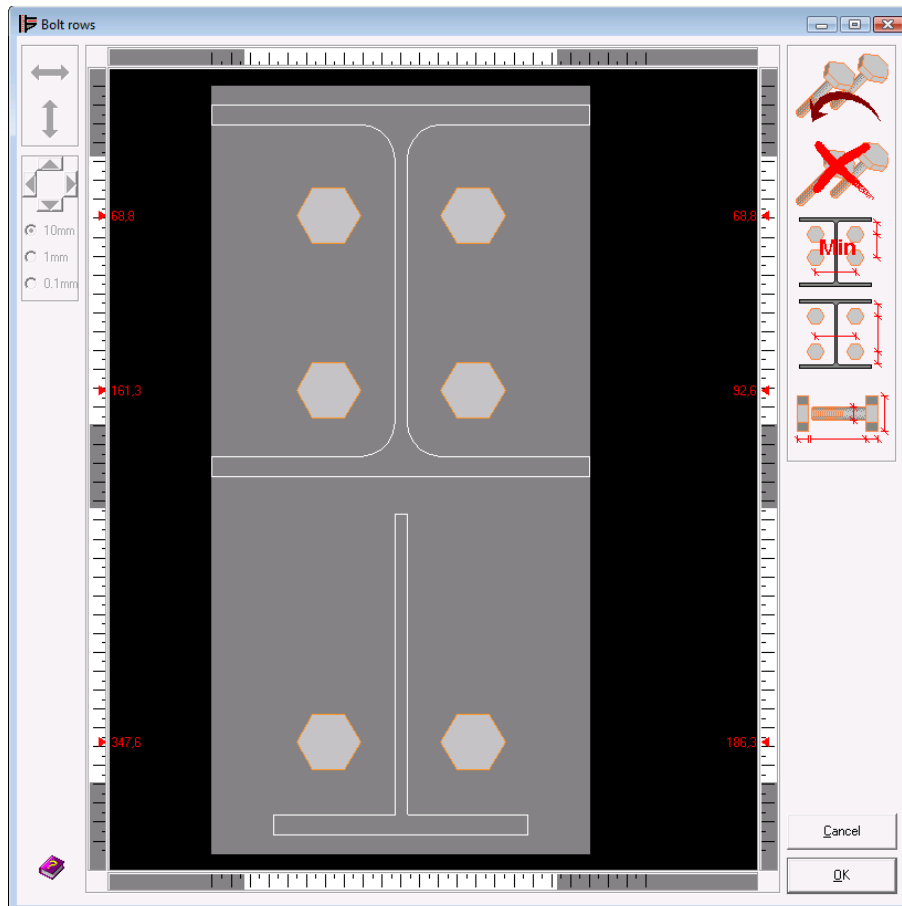
## 2.2.4 Improving the connection design

From the results summary, it is clear that the applied bending moment is too high as compared to the connection's moment resistance (for both loads combinations). To solve such type of problem, a haunch can for instance be added below the connected beams. To do this, select one of the beams in the 'Geometry'-window and make the "Add or remove element" dialogue appear by pressing the right-hand button of the mouse. Select the "Add lower haunch" option.



Now repeat this step for the other beam elements to create a symmetrically haunched connection.

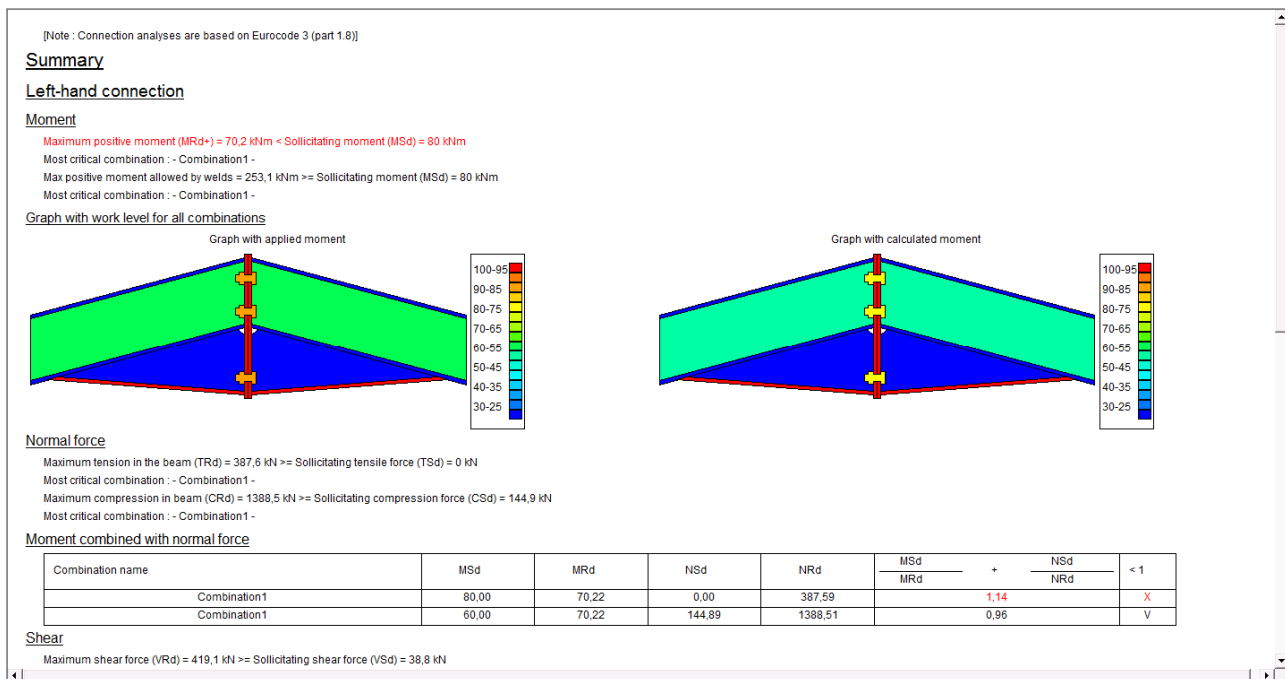
Double-click on one of the bolts to enter into the dialogue window where an extra bolt row can be added and where the bolt row lay-out can be optimized (same procedure as before).



## 2.2.5 Re-running the connection design analysis on the modified connection

Running the connection design analysis again will now produce following summary results:





As can be seen from the results summary, the connection still has insufficient strength with respect to the applied loads combinations. Failure will occur due yielding of the end plate, which brings us to increasing the thickness of both end plates from CF (=10mm) to 12mm in the dialogue window which is obtained by double-clicking on the end plates. After rerunning the analysis, it will be seen that the connection now has sufficient strength.

### Dimensions and position of end plate

**general**

**geometry**

total width (b)  mm

upper extension (u1)  mm

lower extension (u2)  mm

total height  mm

thickness  mm

**welds**

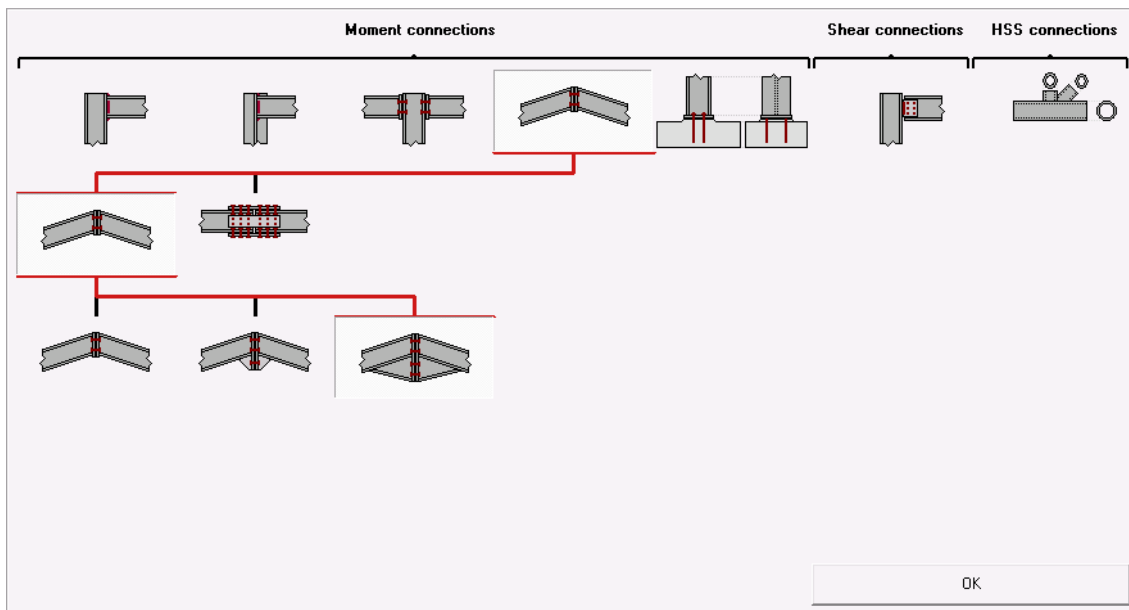
flange  mm

web  mm

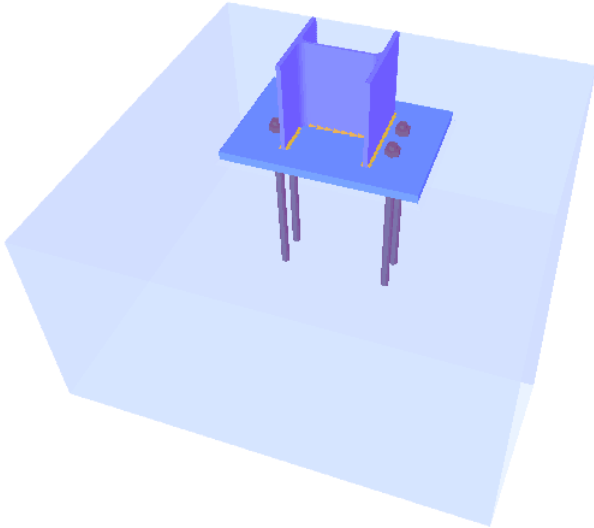
**Steel S235**

**friction coefficient**


The haunched beam to beam connection that has been obtained as the final result, could also have been created directly by making the appropriate choice from the navigation window (see below).



## 2.3 Tutorial 3 : column base with extended end plate (EC3)

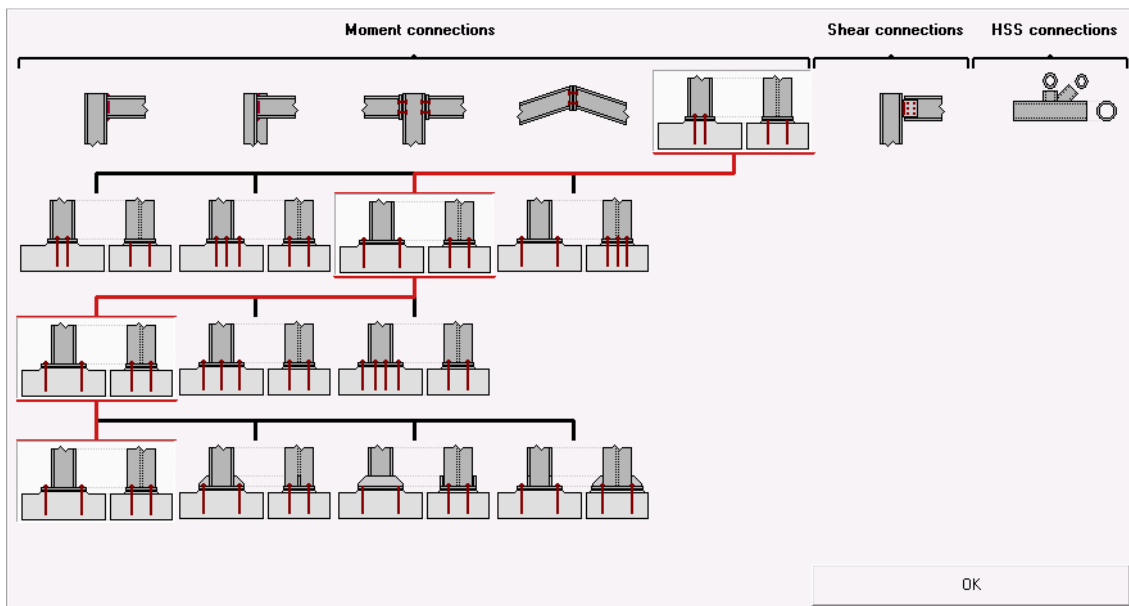


### 2.3.1 Setting up the model

Use the icon  to define a new project. In the navigation window, select the connection type labeled “Column base plate”. Then further navigate through the window by choosing the entries

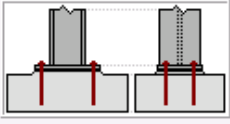
- 2 anchor bolt rows
- extended base plate (2 sides - 2 anchor bolts on each row)

to arrive at a column base connection without any particular stiffener elements.




To actually create connection geometry, fill out all parameters as shown in the dialogue below.

**Column base connection**


☐ Braced

**Column**



length  mm

welds  mm

**Concrete bloc**

height  mm

length  mm

width  mm

**Base plate**

thickness  mm

left-right extension  mm

front-back extension  mm

☐ with cramp

**Anchors**

type

class

Help

Cancel

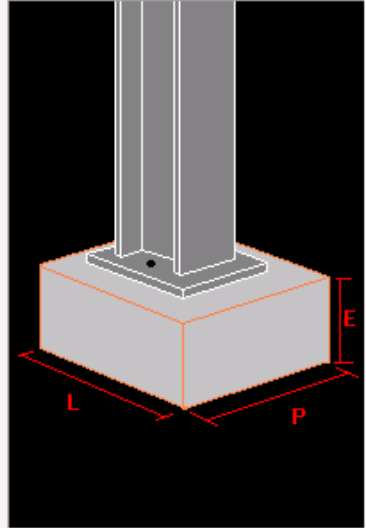
OK

Now double-click on the concrete block to make a dialogue appear in which further details of the concrete base can be verified or modified. Check e.g. if the concrete grade of the base on the “General” tab page is C25/30, and modify (if necessary) by using the “Material”-button.

**Column block**

**general**

**details**



**Dimensions**

length (L) :  mm

thickness (E) :  mm

depth (P) :  mm



☒ Column at center

Coordinates X :  mm Y :  mm

**Concrete grade of base**

Material

Help...

Cancel

OK

Switch to the ‘Details’-tab for further data on grout thickness & grade.

**Column block**

**general**

**Grout**

grout thickness :  mm

**details**



**Grout grade**

characteristic compressive strength (fk):  
 N/mm<sup>2</sup>

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

Double-click on the base plate to access its properties.

**Base plate characteristics**

**Geometry**

Thickness :  mm

Length = 430 mm

(1) Left extension :  mm


(2) Right extension :  mm

Width = 300 mm



(3) Back extension :  mm

(4) Front extension :  mm

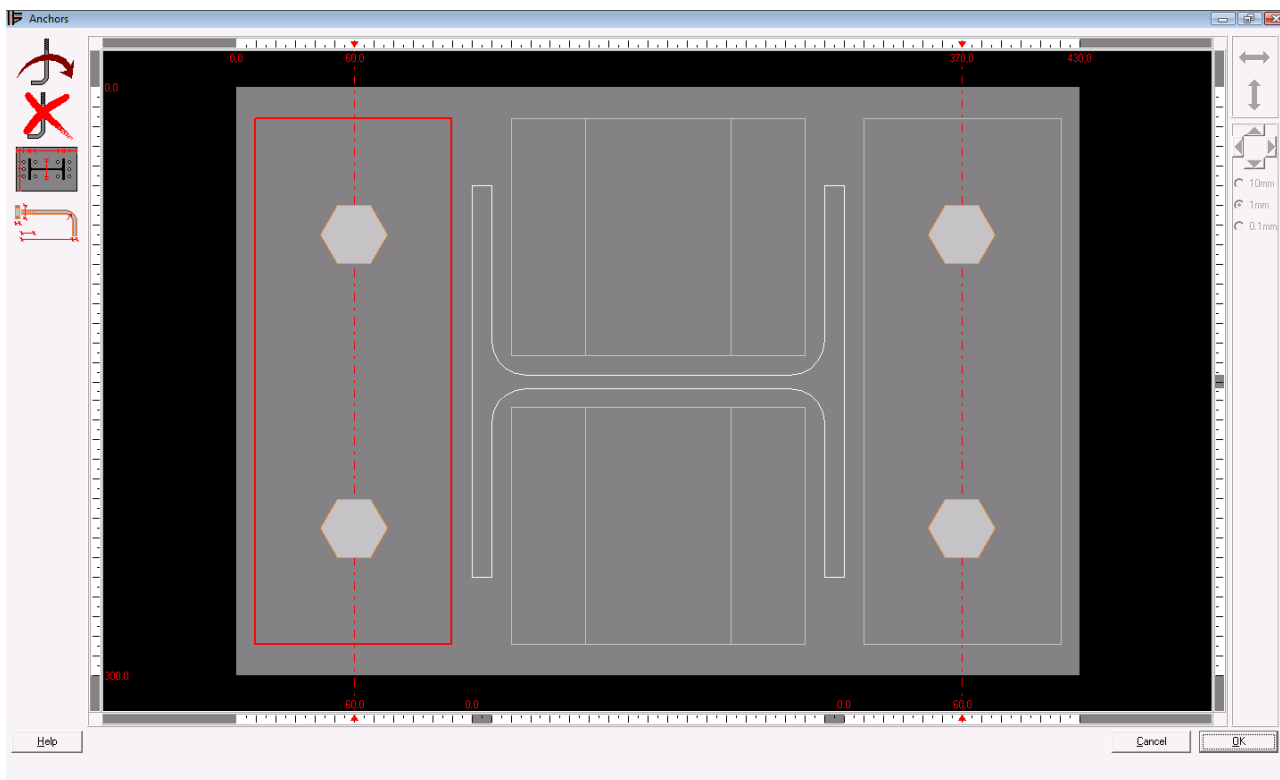
**Steel**

 Steel S235

**Anchors**

Help...   Cancel OK

Use the 'Anchors'-function to gain access to anchor bolts details. If needed, maximize the window size to get a good view on anchor bolt lay-out & details.



The window not only shows base plate dimensions and anchor bolts positions, but also includes a number of anchor zones that can be used to define the anchor bolts configuration. In case an anchor bolt row needs to be added, the appropriate anchor zone must first be selected using the mouse. The borders of a selected zone will be highlighted in red. Next, the icon

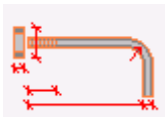


should be used to add a bolt row to the selected zone.

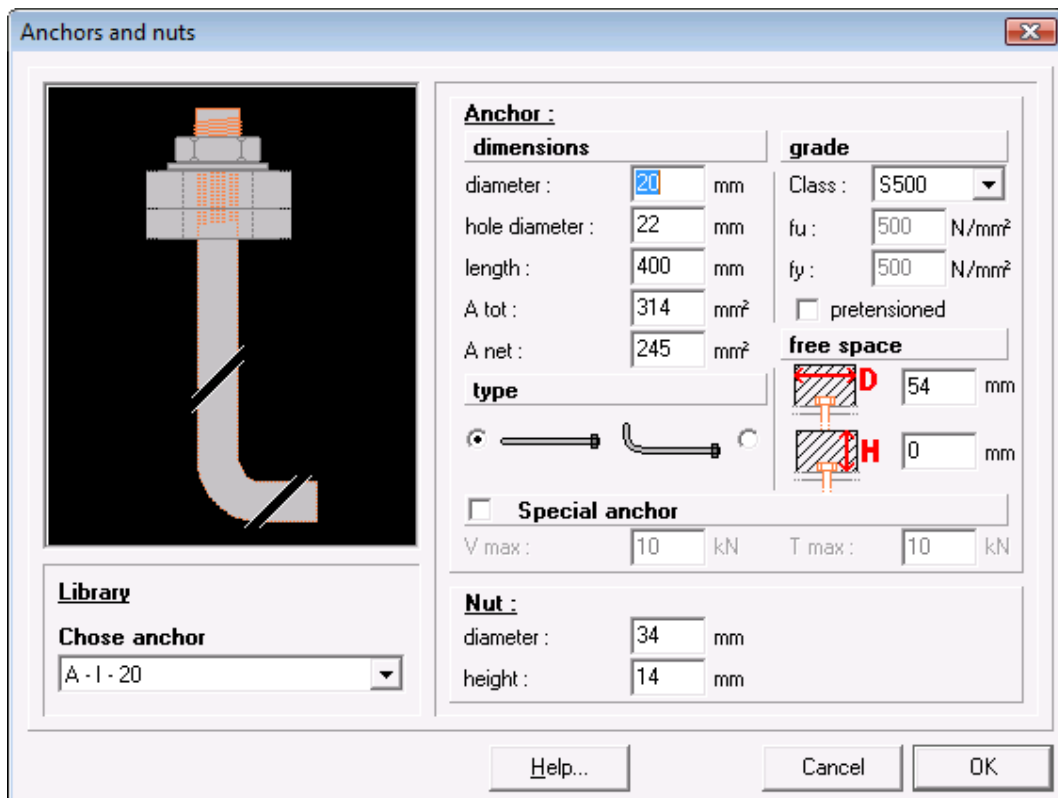
In general, the central part of the base plate can contain up to 4 anchor zones (depending on the available space):

- 2 zones will serve for anchor bolts parallel to the column flanges,
- 2 zones will serve for anchor bolts parallel to the column web. Anchor bolts parallel to the column web will not contribute to the base plate connection's moment resistance, unless no anchor bolts are present in any of the other zones.

The type of anchor bolt should of course also be specified. Use the “Anchor



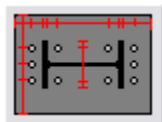
details” icon to this purpose, which will make the following dialogue window appear.



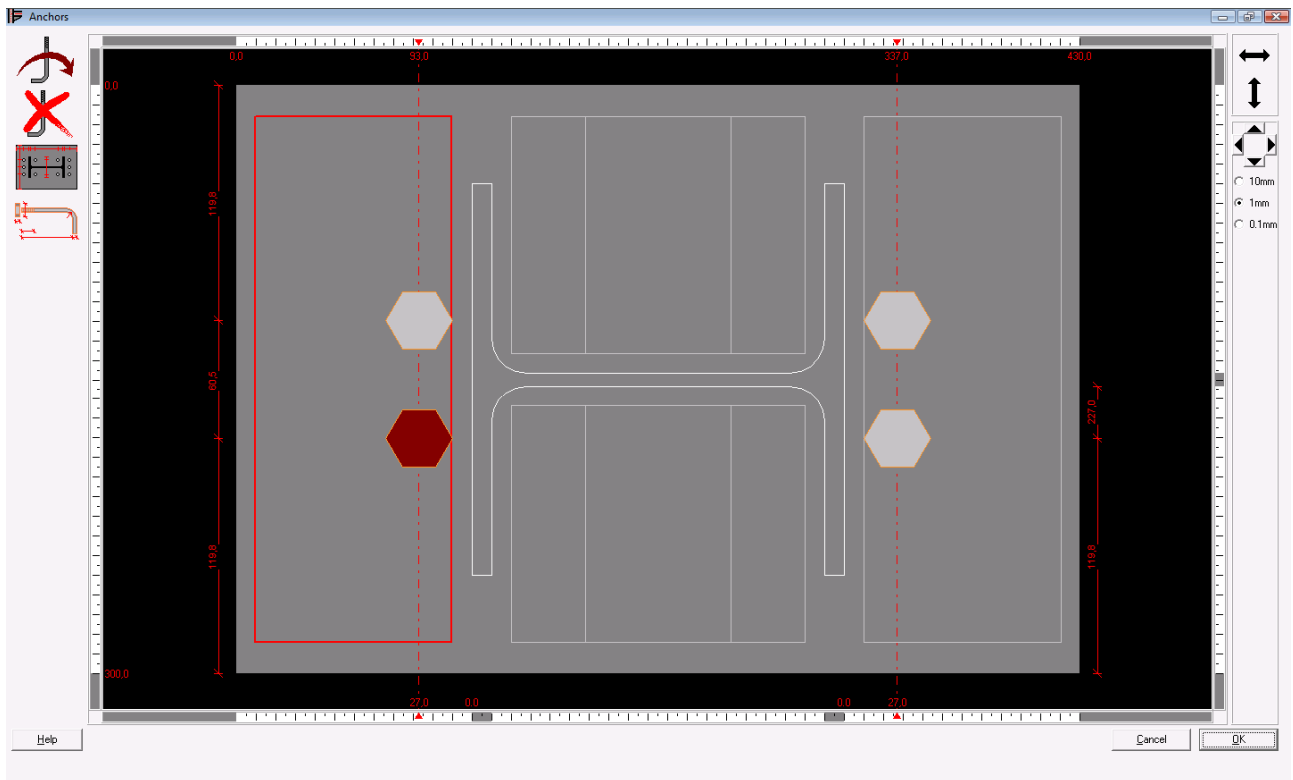
On the left hand side, a specific anchor bolt can be selected from PowerConnect's library of bolts & anchor bolts. For the time being, keep the default 'A-I-20' proposed by the program. 'A-I-20' corresponds to straight anchor bolts with a diameter of 20mm.

As no modifications must be defined as far as anchor bolt choice is concerned, the "Cancel"-button can be used to return to the anchor bolt lay-

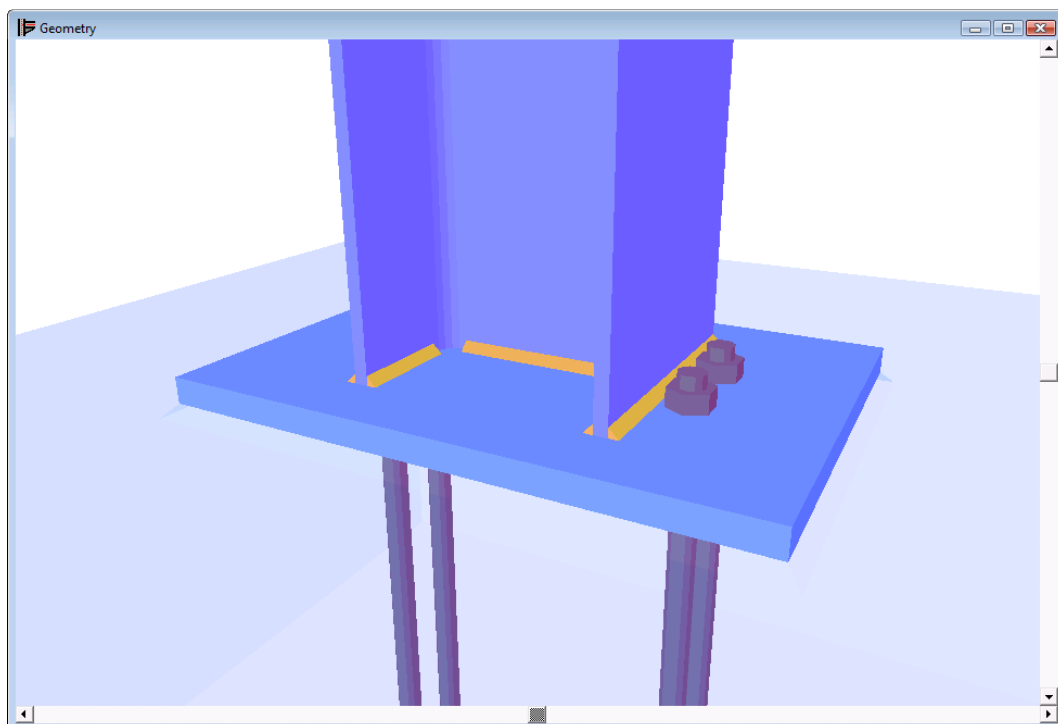
out window. Use the icon



to optimize anchor bolt positions, to obtain the following lay-out:



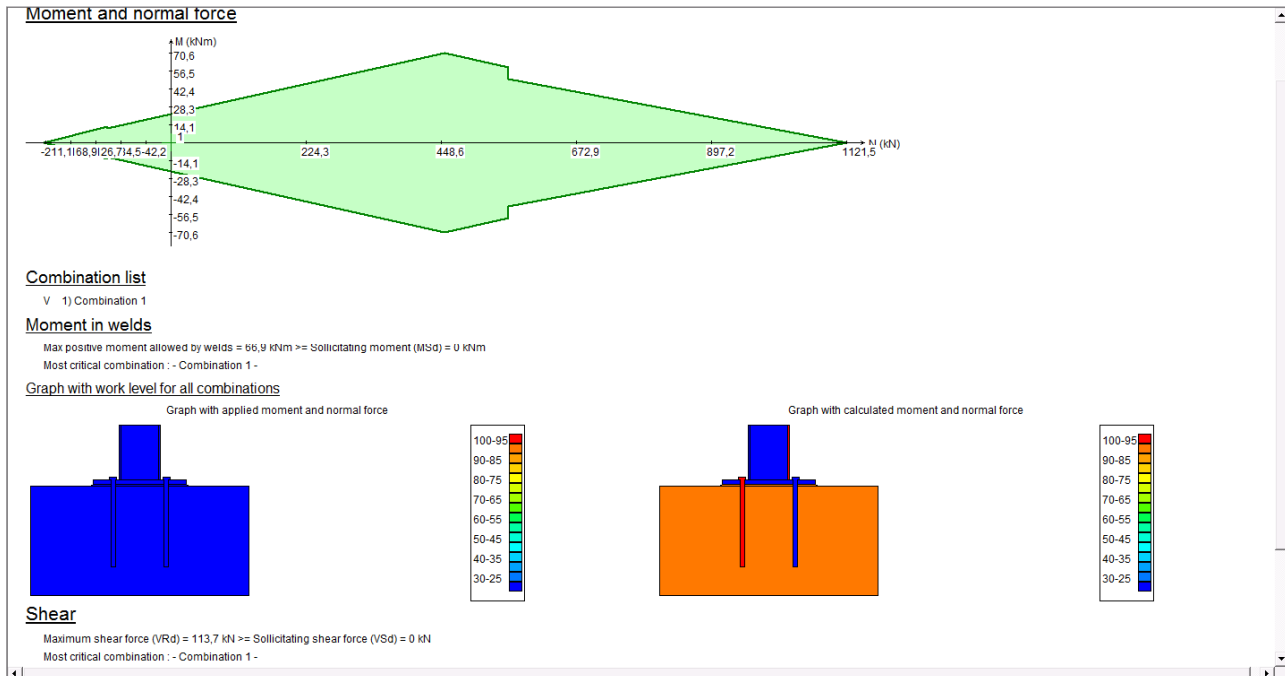
Click 'OK' to complete the base plate connection definition. In the 'Geometry' window, the following model will now be presented:





## 2.3.2 Running the connection design analysis

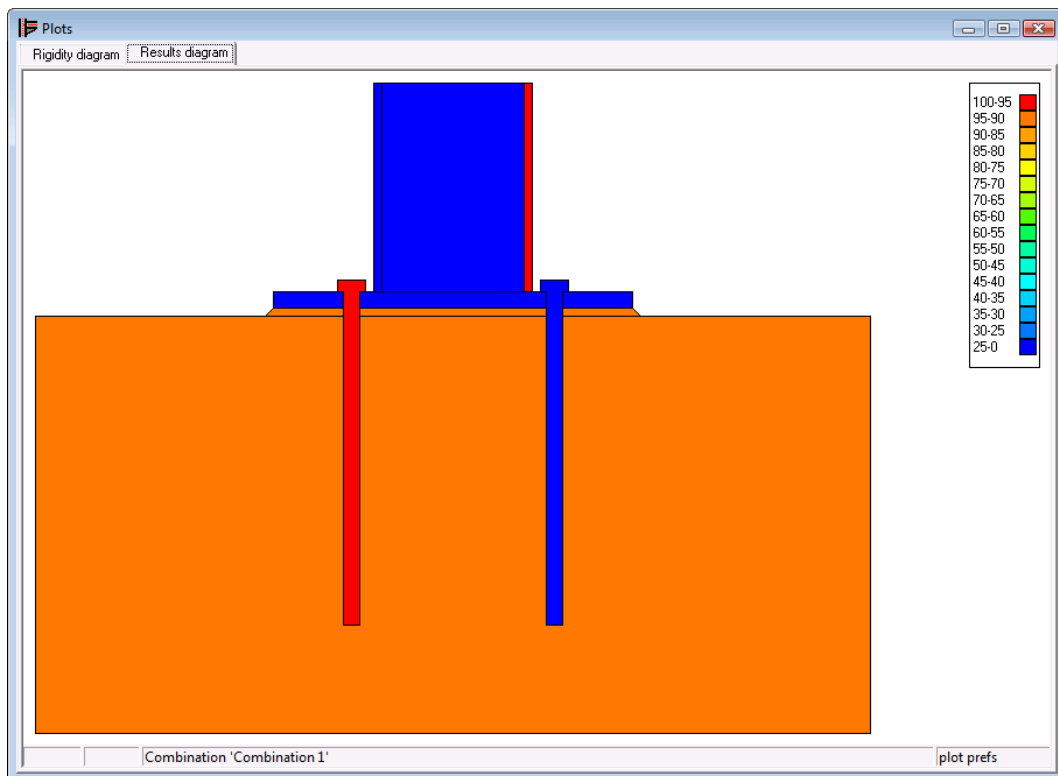
For the current analysis, no specific external loads will be applied. As a consequence, the design analysis will evaluate the connection's resistance, independent of any loading. The following result will be obtained:



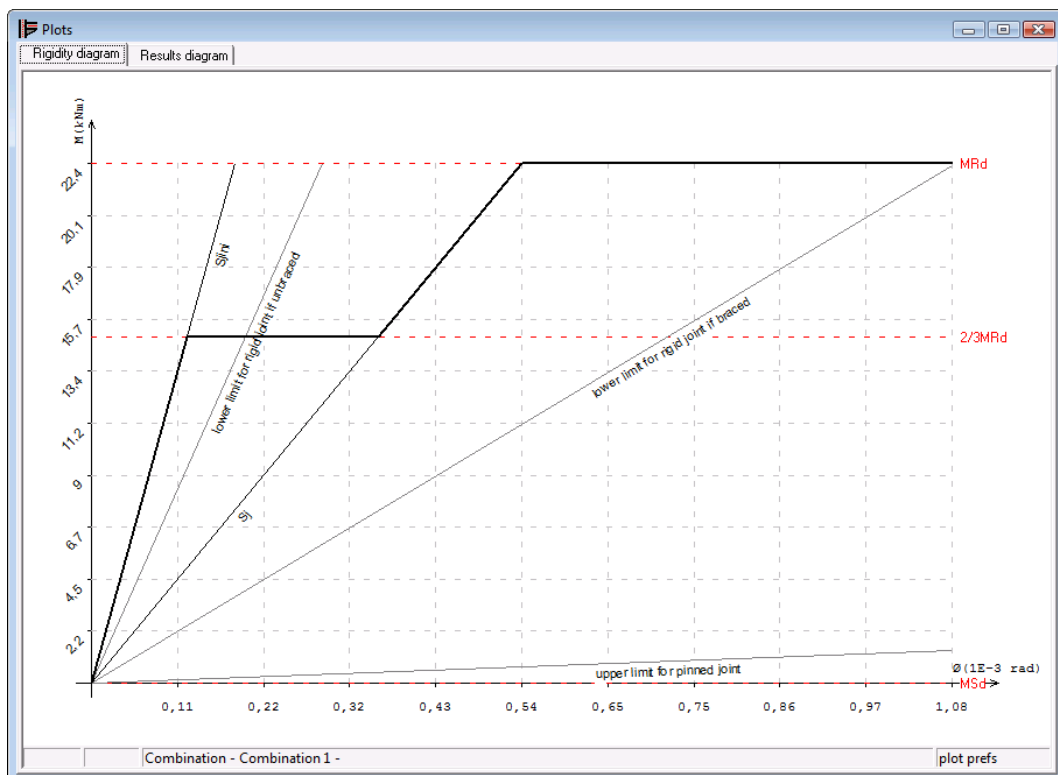
With this particular user scenario, there is no such thing as a “bad” or “good” connection. The resistance that has been calculated should be compared to a specific set of loads to which this type of connection will be subjected to enable that kind of judgment.

Depending on the failure capacity of all parts of the base plate connection, PowerConnect will show a diagram which represents all allowable combinations of bending moment & normal force (compressive forces are positive).

All loads combinations which fall within the green area of this diagram, correspond to loads which can be sustained. In case a particular loads combination falls outside this area, the base plate connection should be modified to resist the applied loading. The more detailed design analysis results that are made available by PowerConnect can be used to better understand the critical connection components and to optimize connection design. Critical components can be identified from the ‘Results diagram’ as shown below, and from the more detailed results reporting.

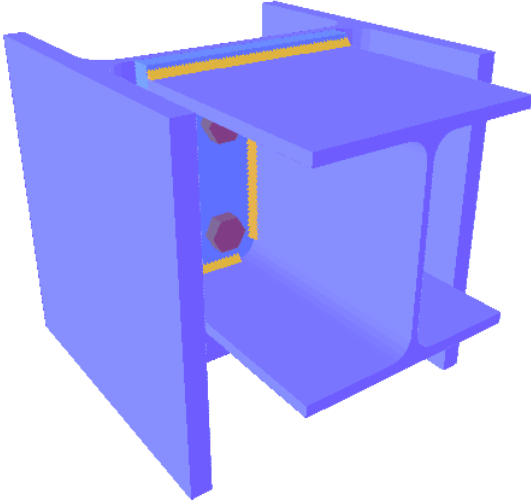


Next to connection resistance, the analysis will also evaluate connection stiffness and present it in a bi-linear diagram as shown in the graph below. This rigidity graph only represents the stiffness provided by the connection itself, it does not consider any possible relative displacements between the concrete block and the underlying soil layers.



## 2.4

## Tutorial 4 : bolted beam to column web (EC3)



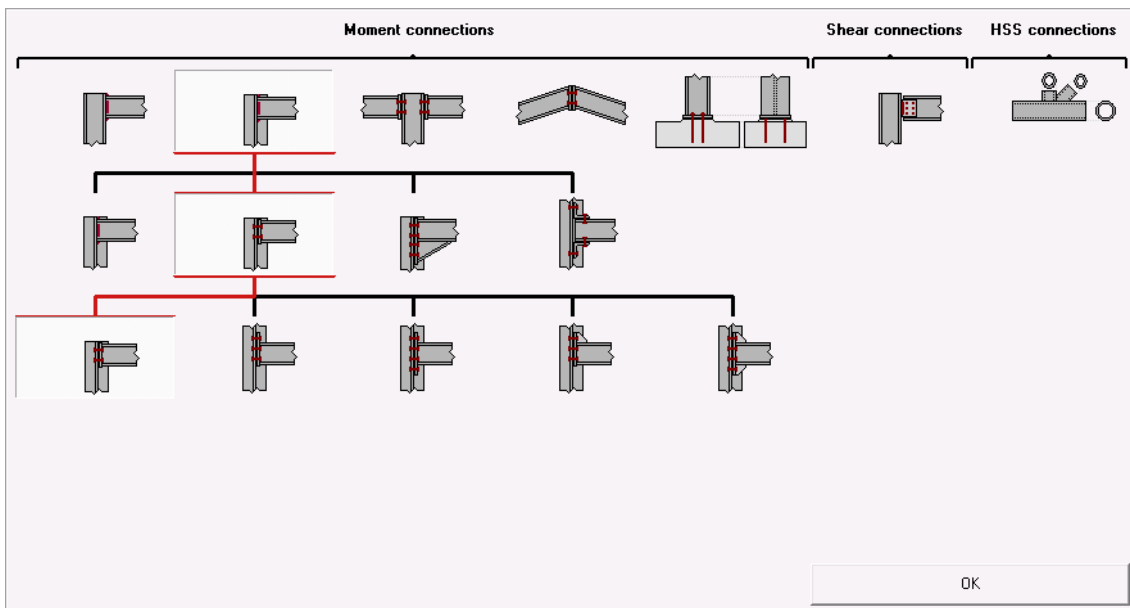
### 2.4.1

## Setting up the model

A beam with HEA 200 section is connected to the web of a column with HEB 300 section by means of a bolted moment end plate (2 rows of M20 bolts). No particular stiffener elements are added.

No loads will be applied on the connection, so that the design analysis will be limited to the evaluation of the connection's resistance in bending & shear.

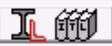
Use the navigation window to arrive at a single-sided beam to column web connection using a flush end plate:




In the next step, make sure that the beam and column sections are defined correctly. Choose other sections from PowerConnect's section library, if needed.

**Bolted column - beam connection**

☐ Braced

**Column**  HEB 300

**Beam**  HEA 200

length  mm

slope  °

welds  mm

**End plate**

thickness  mm

width  mm


**Bolts**

type

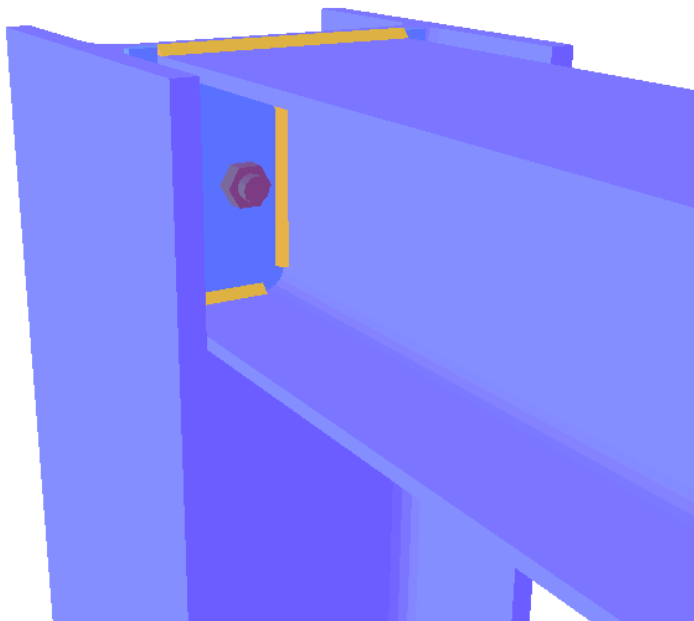
class

min. vertical distance  mm

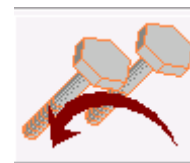
horizontal distance  mm

[Help](#)  [Cancel](#) [OK](#)

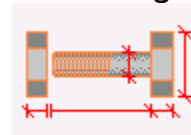
When confirming the above parameters, the following 3D geometry model will be presented.



If needed, end plate & bolt characteristics may still be modified. In particular, a second bolt row should be added in this particular application. Double-click on one of the bolts to open the appropriate dialogue window.

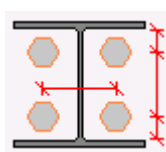


Use the icon to add an extra bolt row to the current configuration. The

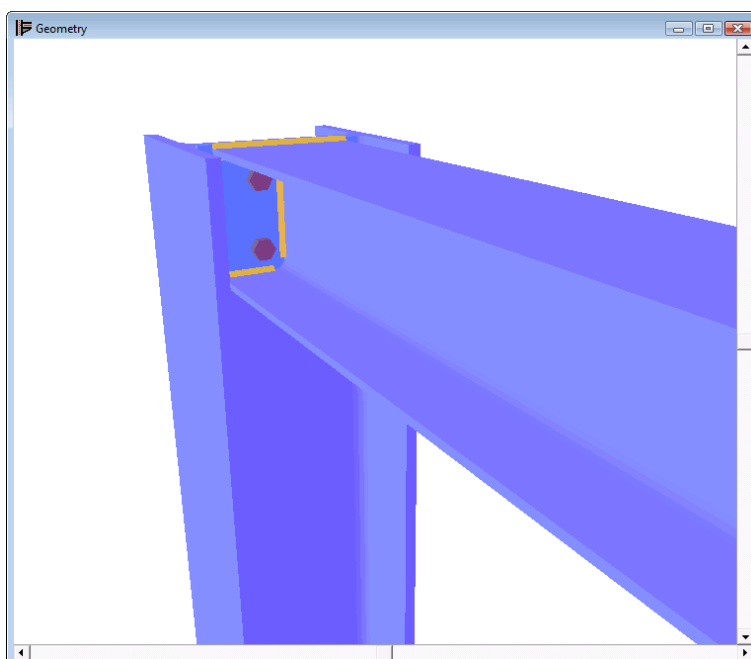
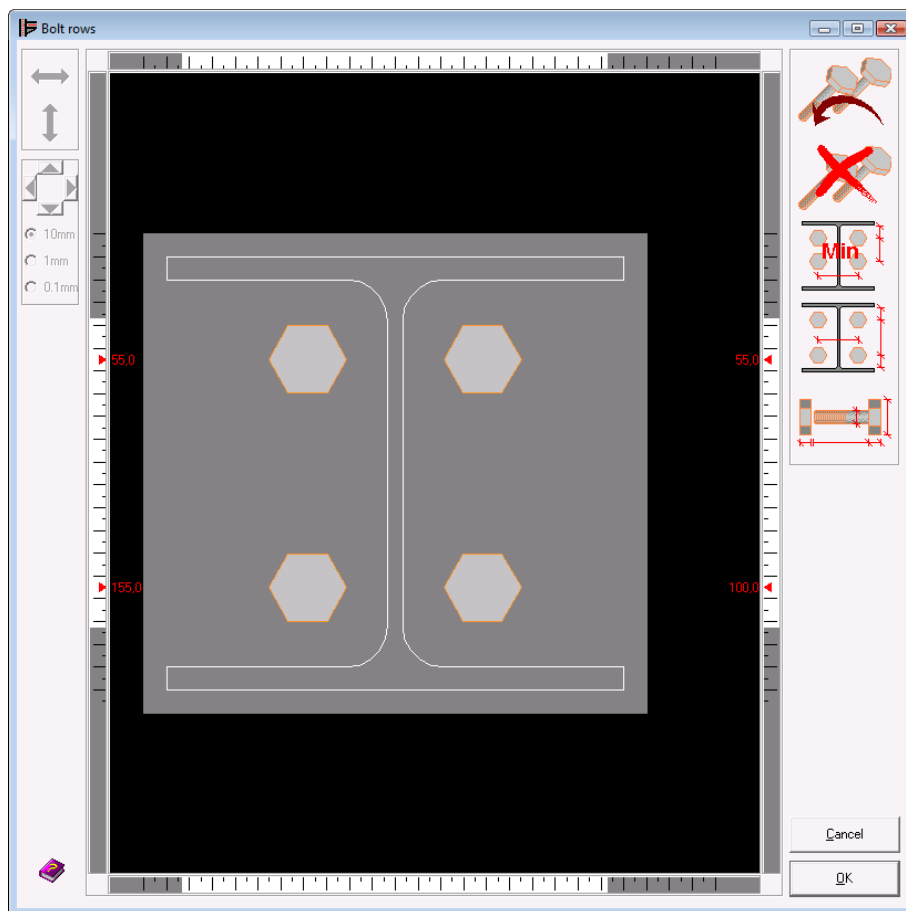


icon can be used

to verify whether the currently selected bolts are of type M20.



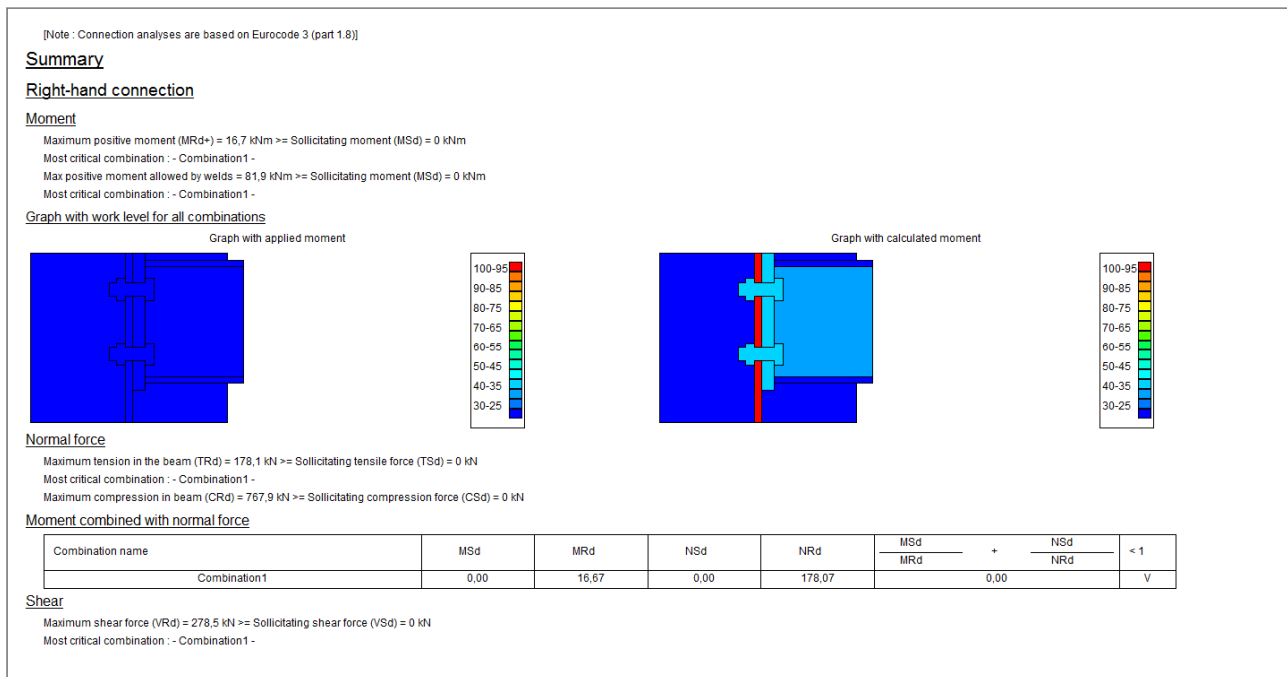
Finally, use the icon to optimize bolt row positions, to arrive at the following lay-out:



All parameters have now been defined so that now the connection design analysis can be launched.

## 2.4.2 Running the connection design analysis

Launch the design analysis to evaluate connection resistance. Following results are obtained.



In case a beam is directly connected to the column web (albeit through the use of an end plate), it may happen that the column web does not provide a sufficiently high resistance. Both local or global failure of the column web may occur. A local failure mechanism occurs when e.g. a bolt row that is subjected to tension, fails.

Three types of local failure mechanisms are possible on the column web: bending, punching or a combination of bending & punching. PowerConnect will screen all possible local & global failure mechanisms and will present detailed analysis results for all of them. In the current case, the connection will fail globally.

**Moment**

Total moment resistance (MRd) = 16,7 kNm >= Sollicitating moment (MSd) = 0 kNm

Bolt row n°1, Restrictive component:  
Column web global failure (weak axis orient.), Moment : 15,5 kNm

Bolt row n°2, Restrictive component:  
Column web global failure (weak axis orient.), Moment : 1,2 kNm

Moment allowed by welds = 81,9 kNm >= Sollicitating moment (MSd) = 0 kNm

**Components**

**Bolt row**

level arm and tensile force in bolt-row

n° bolt-row	1	2
level arm (mm)	140	40
BRd(kN)	141,1	141,1

**Compression in beam flange and web**

Compression resistance of flange = 560,8 kN

**Column web bending (weak axis orient.)**

Column web bending with compression (weak axis orient.) = 1745,5 kN

tensile forces for each bolt group Ft(x)Rd (kN)

(1): 131,3	(2+1): 178,1
	(2): 131,3

**Column web punching (weak axis orient.)**

Column web punching with compression (weak axis orient.) = 789,4 kN

tensile forces for each bolt group Ft(x)Rd (kN)

(1): 142,4	(2+1): 1182,6
	(2): 142,4

**Column web punching and bending (weak axis orient.)**

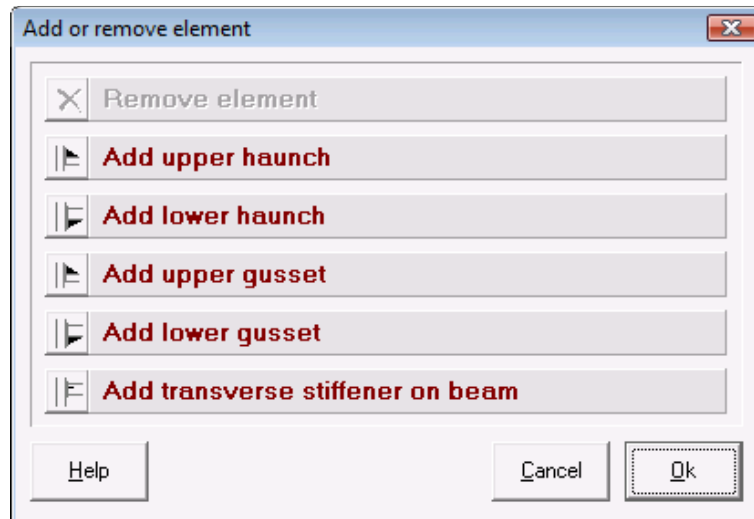
Column web punching and bending with compression (weak axis orient.) = 439,4 kN

tensile forces for each bolt group Ft(x)Rd (kN)

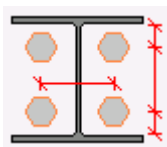
(1): 131,1	(2+1): 178,1
	(2): 131,1

## 2.4.3 Improving the connection design

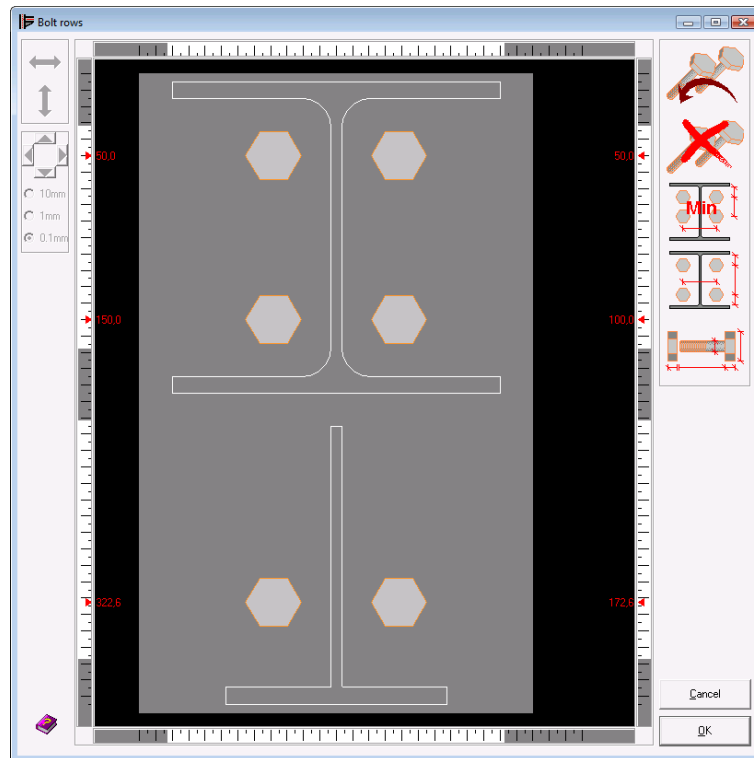
Now return to the geometry window and select the beam element with the mouse. Using the right-hand button of the mouse, make all available stiffening elements appear, and select the lower haunch.



Now double-click on one of the bolts and then add an extra bolt row. Use the

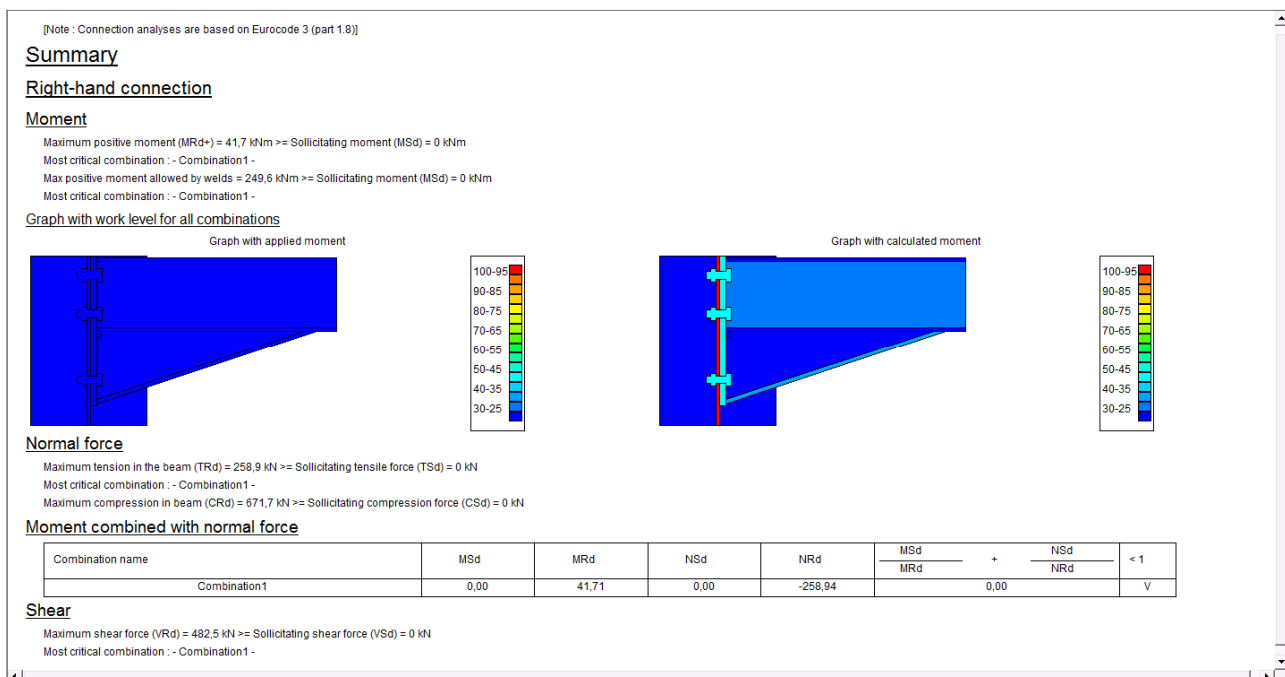


icon to optimize bolt row positions, to arrive at the following lay-out:



## 2.4.4 Re-running the connection design analysis on the modified connection

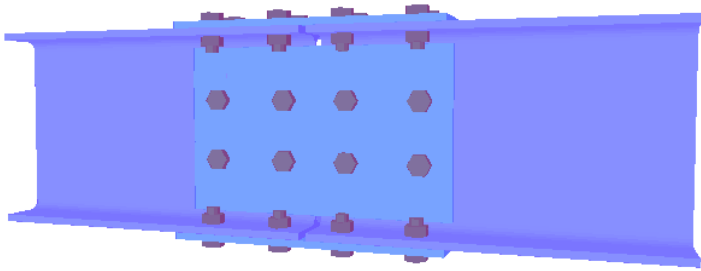
Running the connection design analysis on the modified connection will produce following summary results:



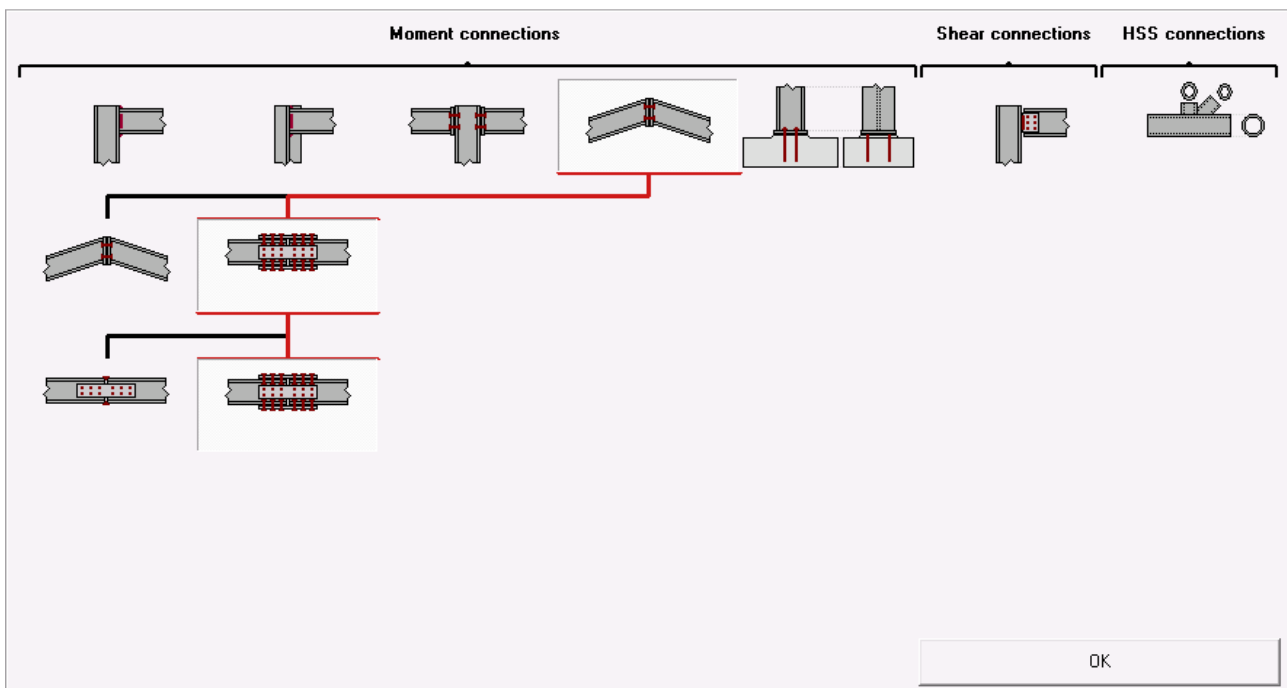
As a result, the connection's bending resistance has been increased from 16.7 kNm to 41.7 kNm.



## 2.5 Tutorial 5: bolted splice (EC3)



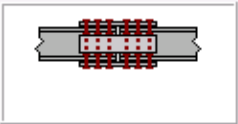
Define a new connection and make the proper choice in the navigation window shown below, to define a bolted beam splice.



Upon confirmation of the above choice, a new dialogue window is presented to complete the definition. In particular, remember to enter the following parameters:

- IPE 300 section for the beam element
- bolts M20 – grade 8.8 for web & flange plates
- minimum intermediate distances for web plate bolts 100 mm in both directions, to ensure that only 2 bolt rows are used over the height of the web plate


**Bolted splice**



**Beam**

length 500 mm

welds 5 mm



IPE 300

**Web plate**

thickness BW mm

length 2\*BH mm

**Bolts on plate bolted to web**

type M 20

class 8.8

min. vertical distance 100 mm

min. horizontal distance 100 mm

**Flange plates**

thickness BF mm

length 2\*BH mm

**Bolts on plates bolted to flange**


type M 20

class 8.8

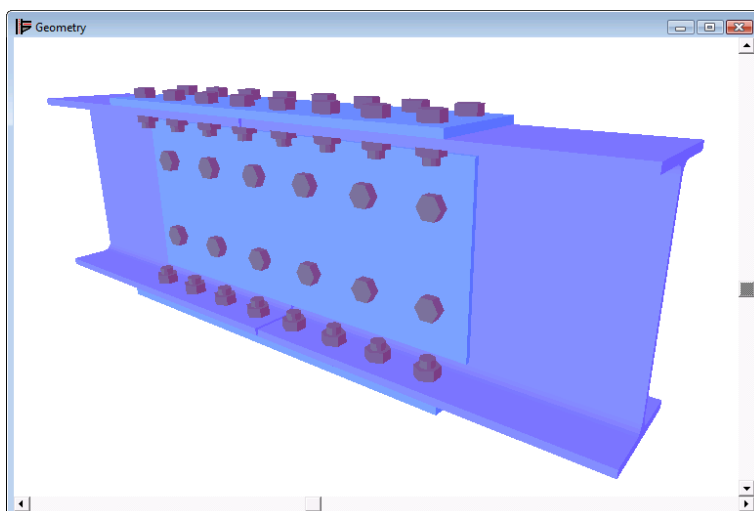
min longitudinal distance 70 mm

min. perpendicular distance 76,6 mm

Help



Cancel OK



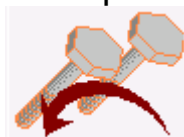
As a result, the beam splice as illustrated in the 3D representation is obtained.

Bolted beam splices are always considered to be symmetrical by PowerConnect. As a consequence, the left half of the connection is identical to the right half of the connection, and any

modification that is specified for any of the flanges will automatically be applied also to the other flanges.

During the previous steps, the bolt type and the number of bolt rows for the web plate have been defined. The lay-out of the different bolt rows will now be screened in more detail and bolt positions will be optimized when needed.

Just double-click on any of the bolts on the web plate to enter the appropriate dialogue window. The number of bolts in a vertical row can freely be chosen by the user. Adding a bolt to a specific row is done by selecting a bolt from

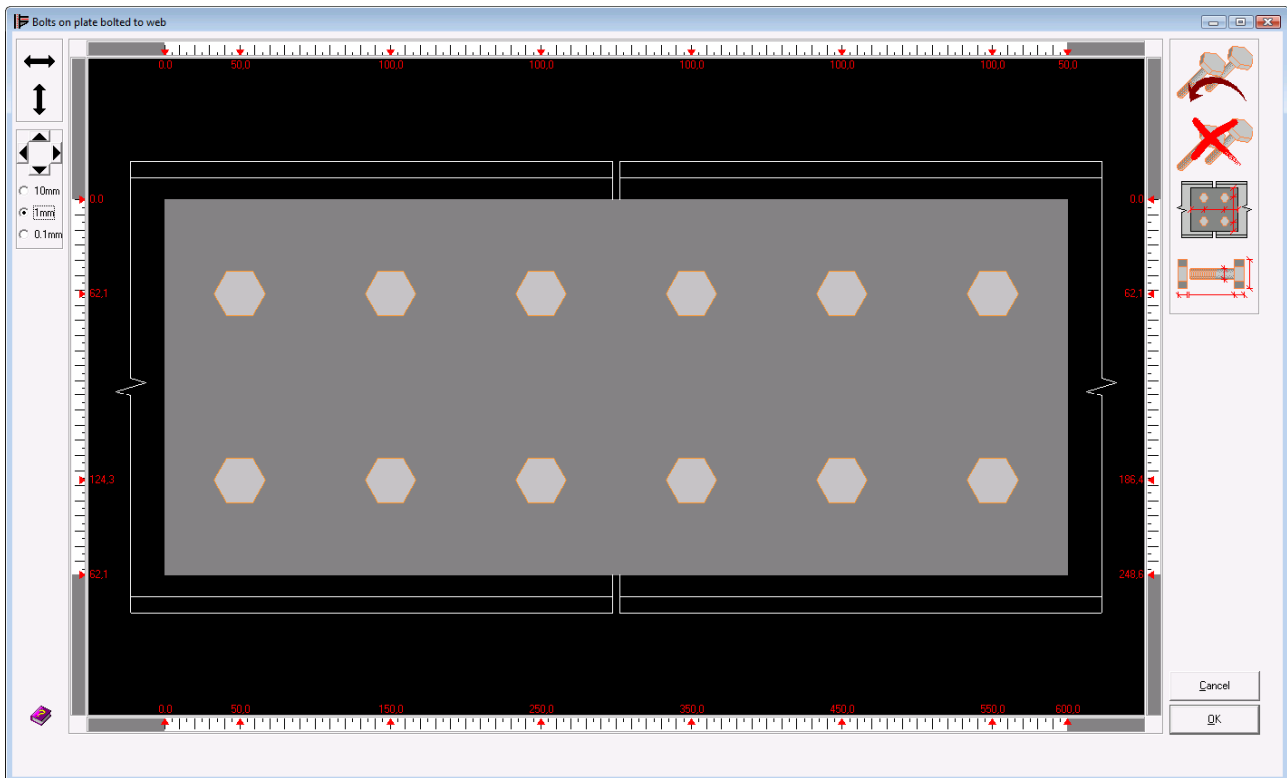


that row and then use the icon to actually add a row. Adding a complete bolt row is done by means of the same icon, but ensuring no single

bolt row is currently selected. To remove a bolt row, selected one of its bolts




and use the icon.



The same procedure can be used for the bolts of the flanges plates. It is sufficient to define the required modifications for one of the flange plates. Because of symmetry conditions, those changes will automatically be propagated to the other flange plate.

For this particular tutorial, it is not required to define any changes. The values that were proposed by PowerConnect after the initial definition steps can be accepted.

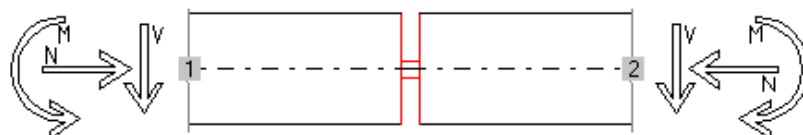
## 2.5.1 Defining the loads

Switch to the 'Loads'-window by means of the  icon, and apply a tensile load of 500 kN at both sides. Remember: click on the small squares labeled "1" and "2" to actually assign those values to the node and to end up with the situation illustrated below.

## Combination1

N = -500 kN  
M = 0 kNm  
V = 0 kN

N = -500 kN  
M = 0 kNm  
V = 0 kN



## 2.5.2 Running the connection design analysis

The connection design analysis will deliver following results, using the previously defined parameters:

[Note : Connection analyses are based on Eurocode 3 : EN 1993-1-8:2005]

### Summary

#### Moment

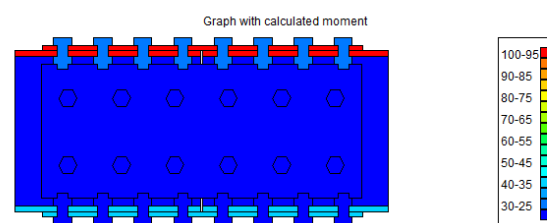
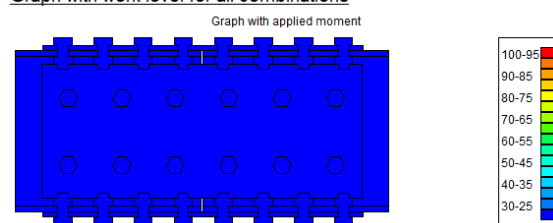
Maximum positive moment (MRd+) = 54.3 kNm >= Sollicitating moment (MSd) = 0 kNm

Most critical combination : - combinatie 1 -

Maximum negative moment (MRd-) = -54.3 kNm <= Sollicitating moment (MSd) = 0 kNm

Most critical combination : - combinatie 1 -

#### Graph with work level for all combinations



#### Shear

Maximum shear force (VRd) = 104.8 kN >= Sollicitating shear force (VSd) = 0 kN

Most critical combination : - combinatie 1 -

#### Normal force

Maximum compression force (CRd) = 1219.2 kN >= Sollicitating compression force (CSd) = 0 kN

Most critical combination : - combinatie 1 -

Maximum tensile force (TRd) = -1019.7 kN <= Sollicitating tensile force (TSd) = -500 kN

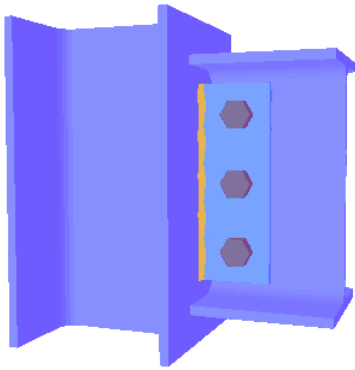
Most critical combination : - combinatie 1 -

For this type of connection, PowerConnect will report the ultimate bending moments (both positive and negative). Those bending moments are calculated taking into account the shear force and normal force that have previously been applied to the connection ( $V = 0$  kN &  $N = -500$  kN).

The analysis results in terms of maximum normal force or shear force do not consider however the presence of a bending moment that is applied to the connection. In case the connection is loaded by a bending moment, maximum shear & normal force values should be reduced.

Further details on the analysis results can be obtained by clicking with the mouse on “Results Preferences” field in the right hand bottom of the PowerConnect window. Choose the appropriate option to obtain the required level of reporting detail.

## 2.6 Tutorial 6 : shear connection – beam to column flange with fin plate (EC3)



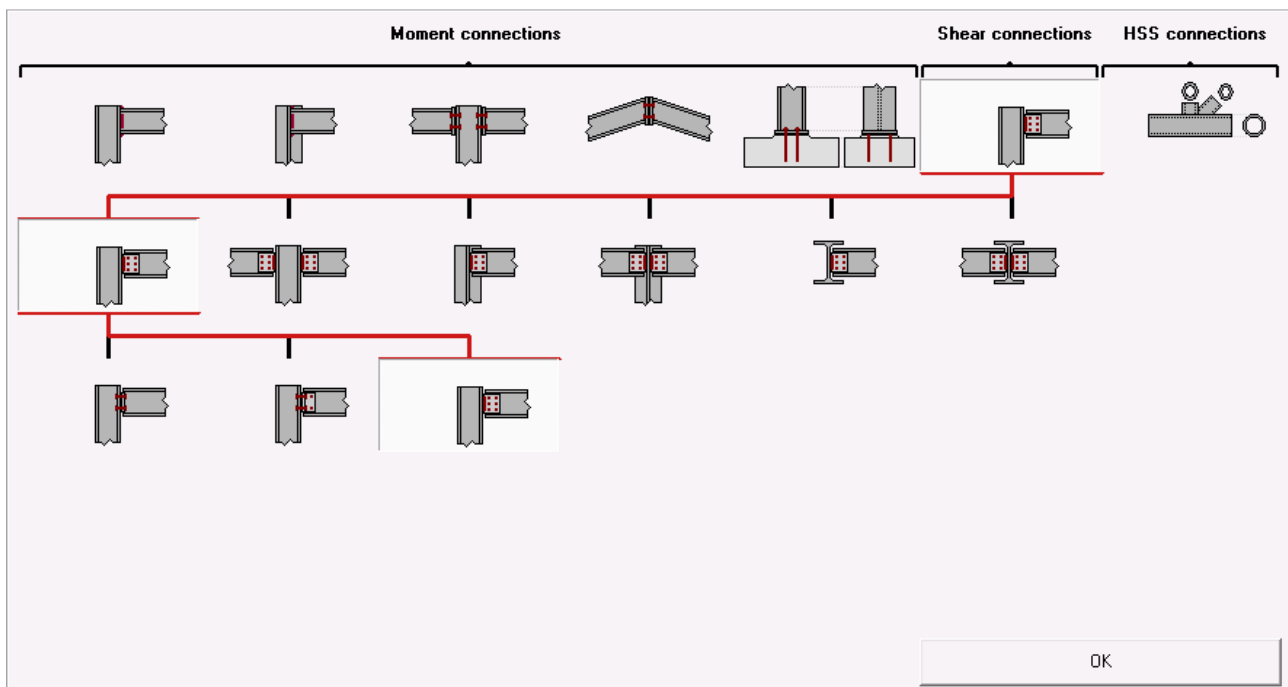
In this first tutorial on shear connection design, the focus will be on a beam to column flange connection through fin plates.

### 2.6.1 Setting up the model

In the PowerConnect navigation window, first select the **Shear Connection** icon to start the navigation. Then choose following specifications to complete the connection type selection:

- Single-sided beam to column flange
- Fin plate

to end up with a selection tree as below:



Upon confirmation of this choice through the “OK”-button, a new dialogue window appears in which further details on the connection elements can be entered.

**Shear connection with fin plate**

**Column** HEA 200

**Beam** IPE 270

**Fin plate**

thickness BF mm

width 80 mm

upper off-set 25,2 mm

lower off-set 25,2 mm

welds 5 mm

**Bolts**

type M 20

class 8.8

min. vertical distance 70 mm

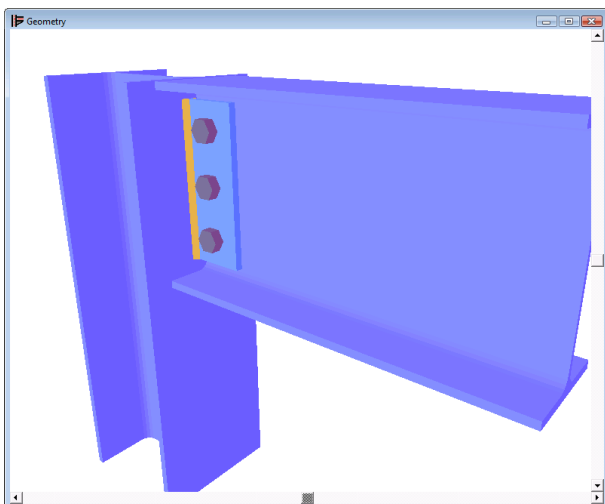
horizontal distance 70 mm

Help Cancel OK

In particular, following parameters should be verified:

- section of column : HEA 200
- section of beam : IPE 270
- width of fin plate : 80 mm

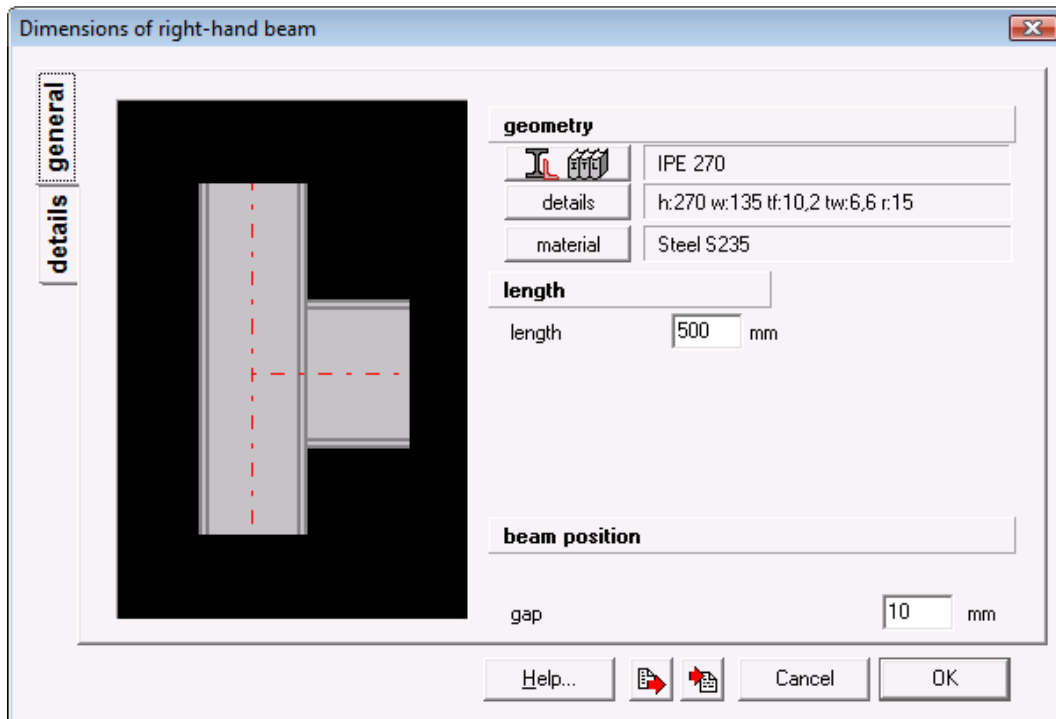
Column and beam length have no impact on the analysis results whatsoever, as all verifications during the analysis are related to shear force only. As this type of connection is verified only for shear force, no bending stiffness is evaluated and the connection is thus assumed to be a pinned connection.



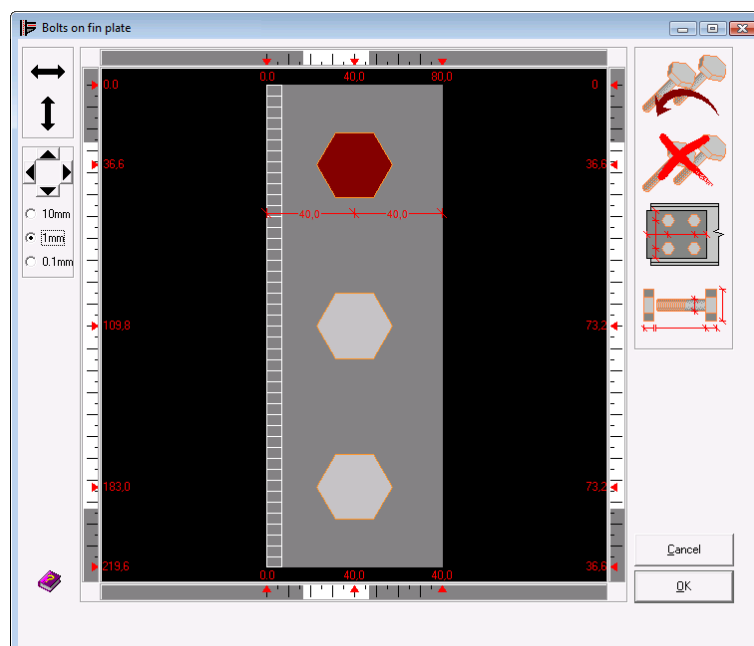
Confirm any modifications to defaults values by means of the 'OK'-button, to arrive at the connection as illustrated alongside.

## 2.6.2 Verifying the geometry model

Further details on specific connection element scan be obtained by double-clicking on any element with the mouse. For example, double-click on the beam to verify that a gap of 10mm has been specified between the beam and column, and modify if needed.



The position of the bolts will also be verified. Double-click on any of the bolts to see that all bolts are positioned centrally with respect to the fin plate.







Assume now that all bolts should be moved horizontally to the right over a distance of 5mm. PowerConnect has a number of tools, next to the optimized bolt positioning functions, to define bolt positions manually. To start with, it is important to understand that those tools operate on horizontal bolt rows, and that each tool requires the selection of a single horizontal bolt row. As in the current example each bolt row contains exactly 1 bolt, bolts need to be selected individually in this example and then the requested horizontal shift can be specified.

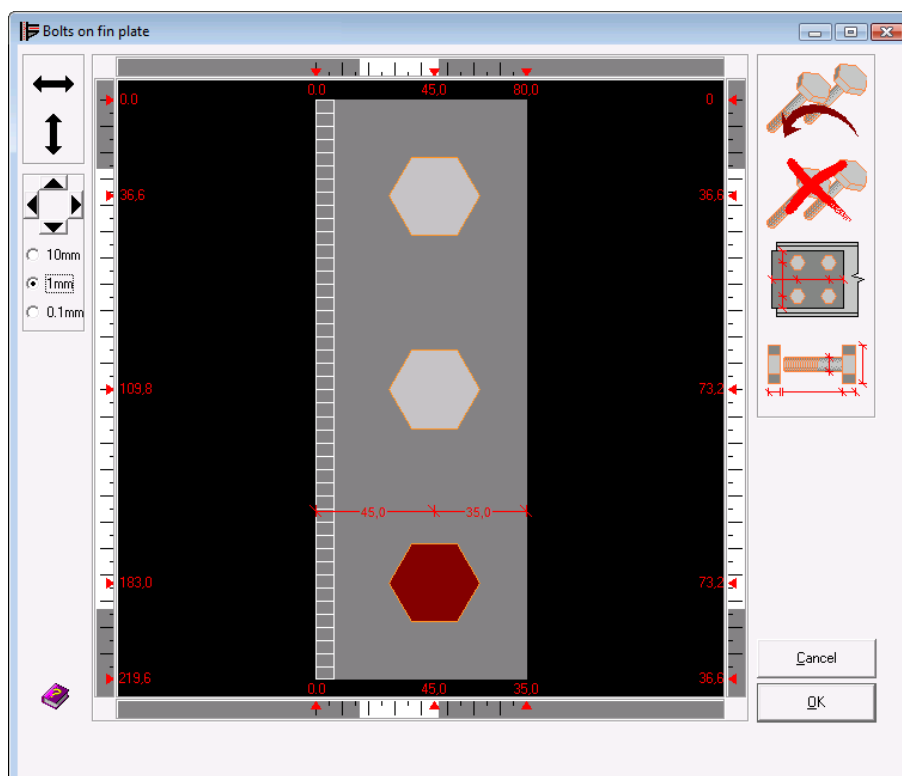
Select the first bolt, and a number of items on the left hand side of the window will be activated



Use the icon and the color will switch to red . This indicates that the selected bolt can easily be moved horizontally by sliding it with the mouse. Use this tool only for roughly repositioning selected bolts.




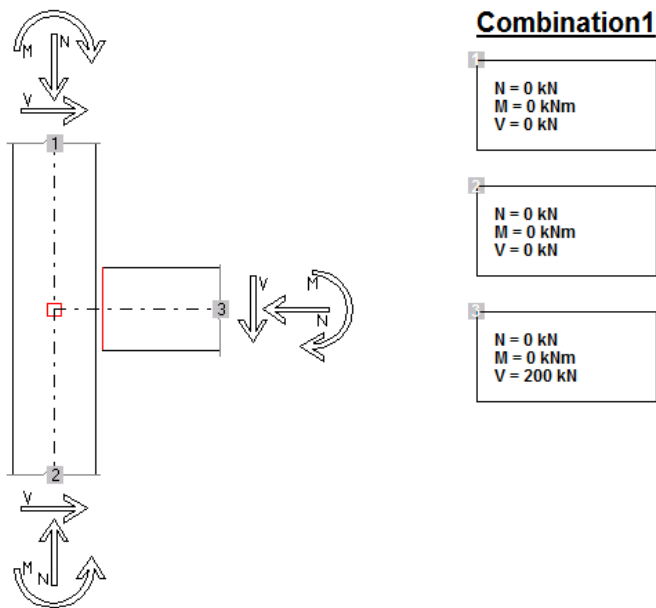
This function can be used in case an exact repositioning of bolts is required. First select the requested precision (0.1, 1 or 10mm – select 1mm for the current application. Then press the  icon five times to move the selected bolted by 5mm to the right. Repeat this operation for the other bolts, to arrive at the status illustrated below.




Confirm the new lay-out using the 'OK'-button to return to the 3D 'Geometry'-window.

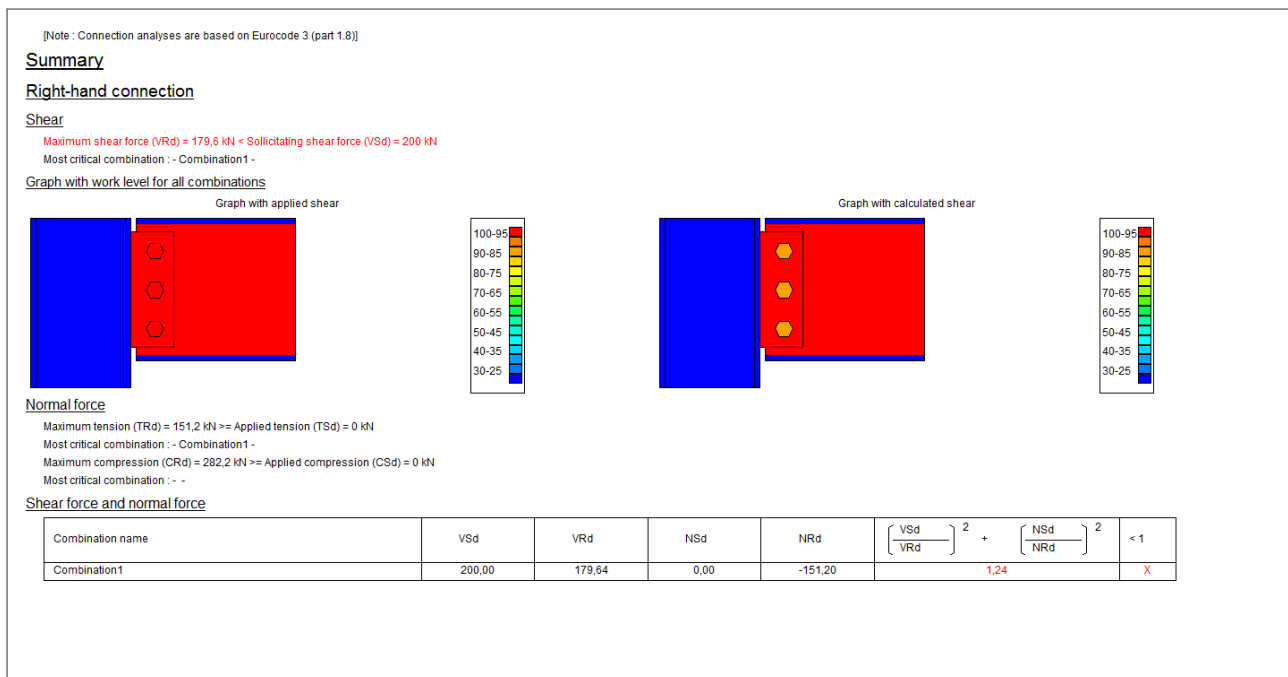
## 2.6.3 Defining the loads

Switch to the 'Loads'-window using the  icon, click with the mouse on the label "3" shown on the 2D geometry representation to enter a shear force of 200 kN. As a result, the contents of the 'Loads'-window should look as follows:

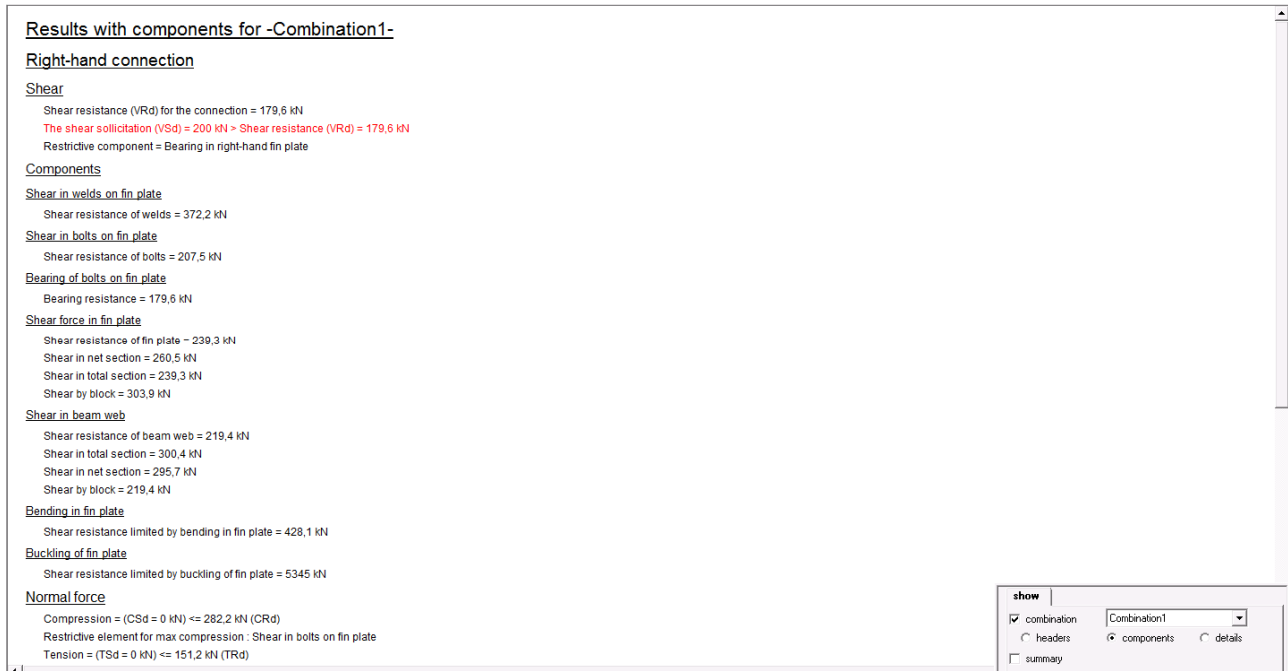


## 2.6.4 Running the connection design analysis

Use the  icon to launch the design analysis. This analysis will present following summary, from which it can be concluded that the connection needs modification in order to be able to resist to the applied shear force.



The color graphs above indicate that the fin plate and the beam web are the most critical elements of the connection. To better understand the connection's actual failure mechanism, switch to more detailed reporting by clicking with the mouse on the 'Results Preferences' field at the right hand bottom of the PowerConnect window.

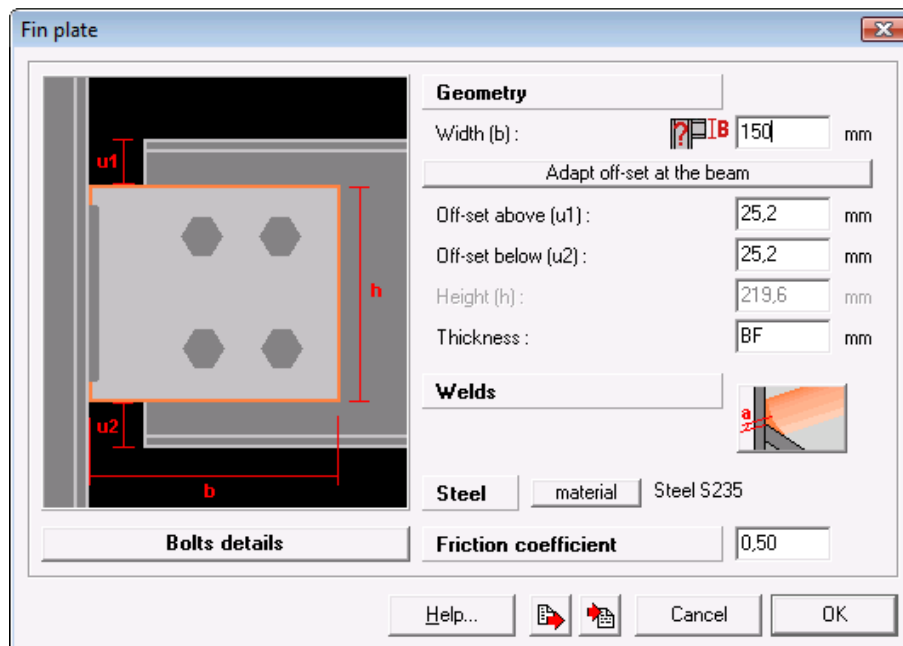


This information confirms that the connection's resistance is limited by bearing of bolts on the fin plate. Furthermore, it is clear that maximum shear in fin plate and shear web are quite comparable – and both are higher than the applied shear load. This is important information, as it indicates that increasing the thickness of the fin plate will not significantly affect the

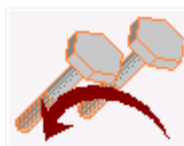
connection's resistance. A solution should rather be provided by increasing the number of bolts.

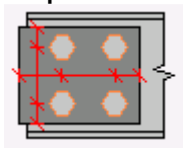
## 2.6.5 Improving the connection design

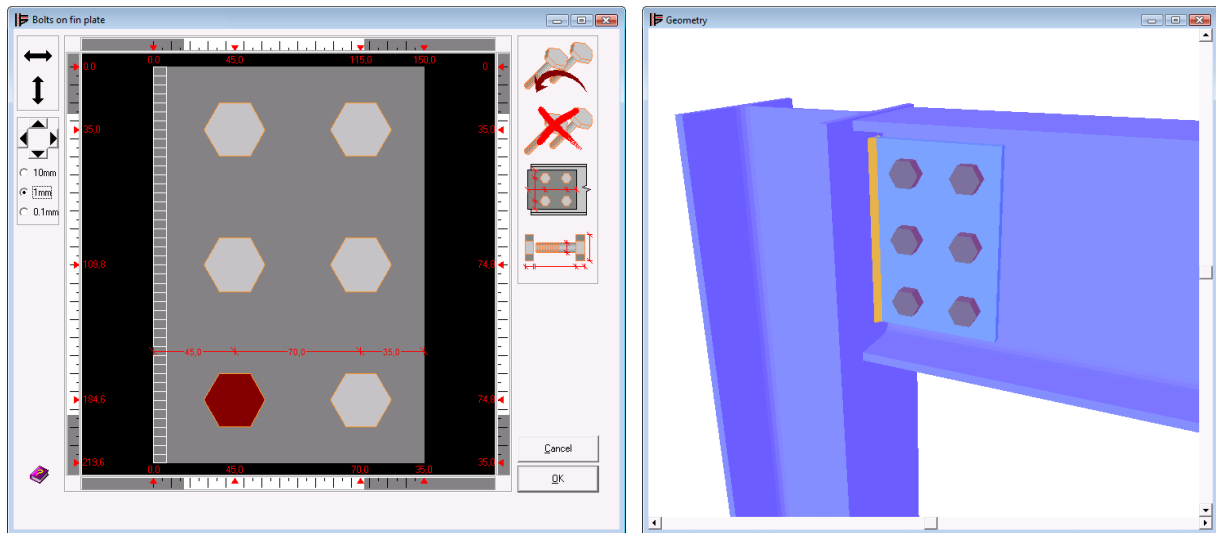
Considering previously specified minimum distances between bolts, this can only be achieved by increasing the fin plate width from 80mm to 150mm. Just double-click on the fin plate and adapt the width in the dialogue window that appears (see below).



Next, a bolt will be added on each bolt row. Proceed as follows:

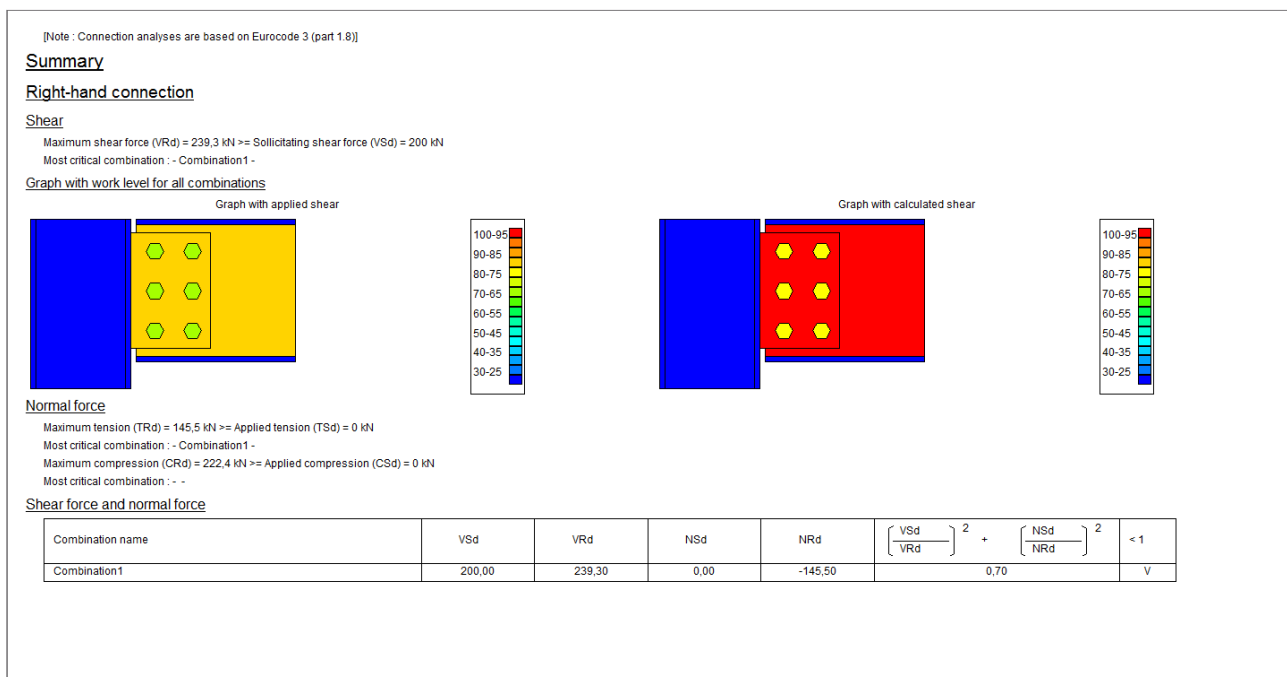
- select the bolt on the first row and use the  icon to add a bolt on the same row
- repeat the previous step for the second and the third bolt row

- use the  icon to automatically reposition the bolts



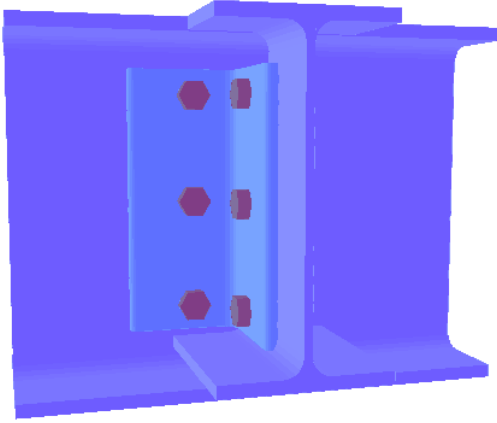
## 2.6.6 Re-running the connection design analysis on the modified connection

Running the connection design analysis on the modified connection will produce following results summary:




Connection shear resistance has been increased from 179.6 kN to 239.3 kN, which is sufficient to resist the applied shear load of 200 kN.

## 2.7 Tutorial 7 : shear connection – beam to beam web with bolted angle cleats (EC3)



As a second example for shear connections, a beam to beam connection, using bolted angle cleats will be considered.

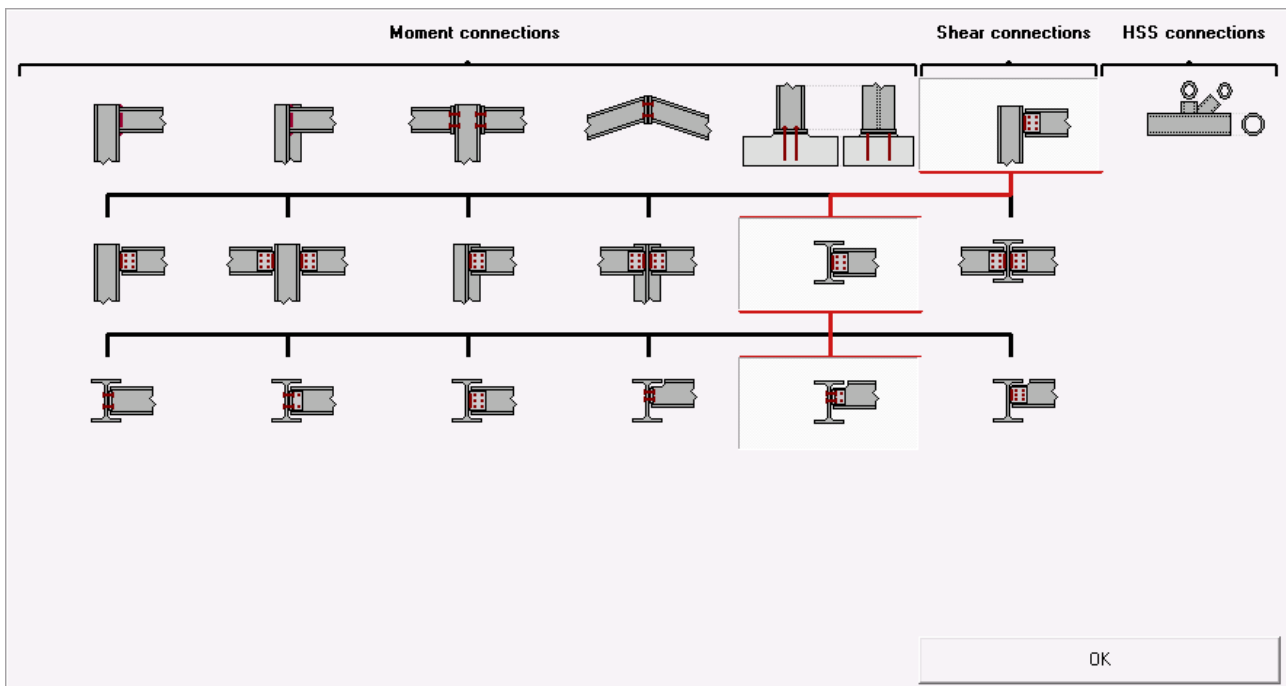
### 2.7.1 Setting up the model

Create a new PowerConnect project using the  icon. In the PowerConnect navigation window, first select the Shear Connection icon to start the navigation.

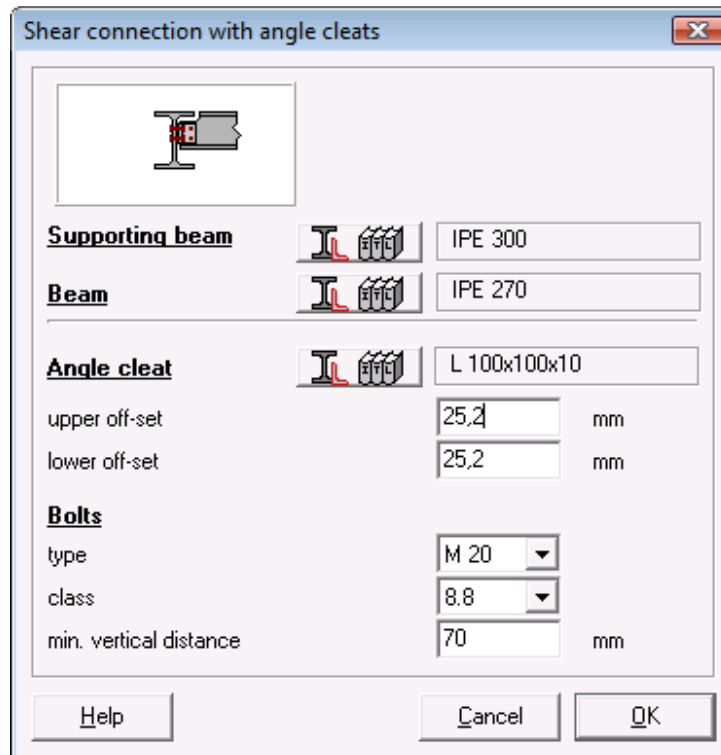
Then choose following specifications to complete the connection type selection:

- Single-sided beam to web
- Coped beam to bolted angle cleat

to end up with a selection tree as below:



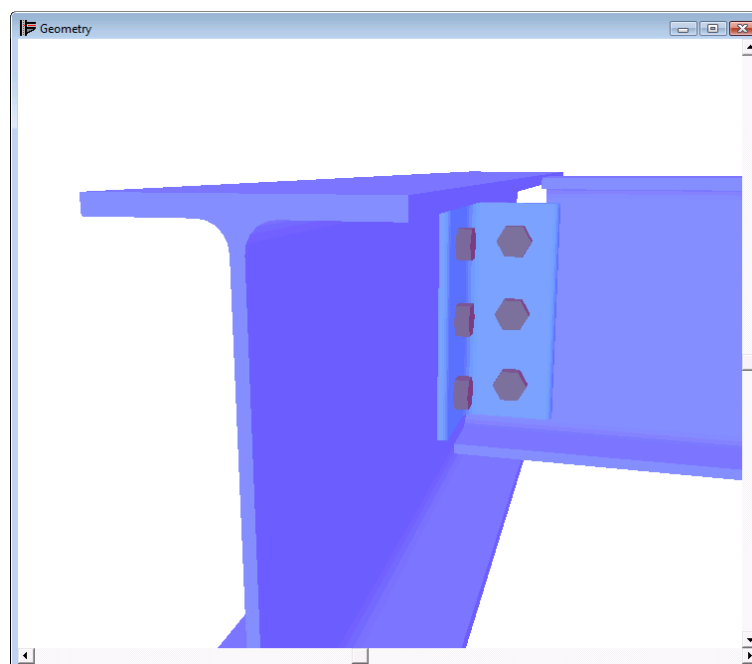
Confirm this choice with the 'OK'-button and then define further details in the following dialogue window.



In particular, following parameters should be verified:

- section of supporting beam : IPE 300
- section of beam : IPE 270
- angle cleat : L 100x100x10

Confirm with the 'OK'-button to end up with the connection shown below.

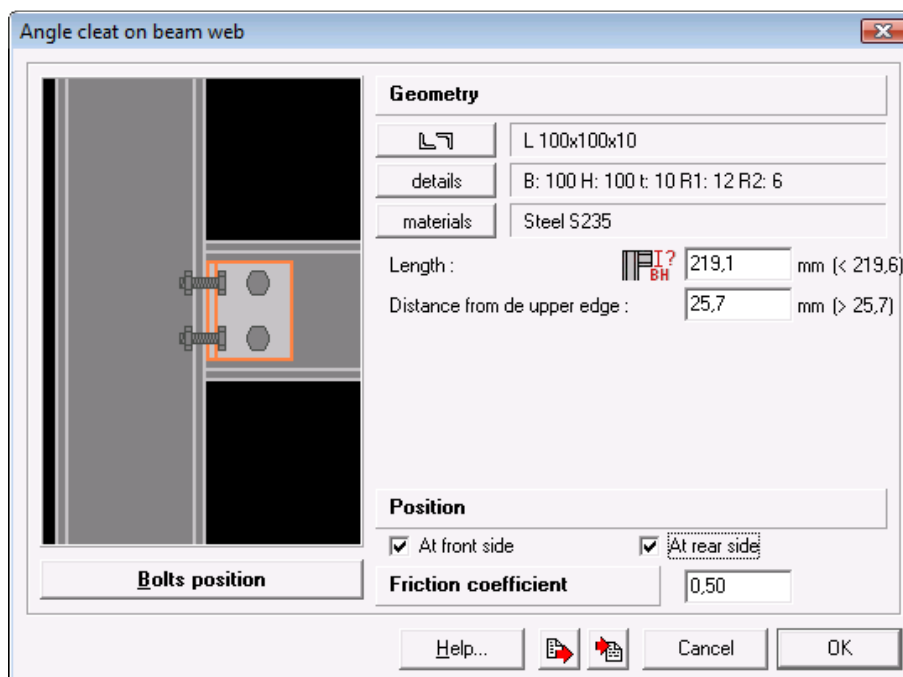


With this type of connection, the upper surfaces of both beams are automatically aligned. In case the lower surfaces of the beams should be aligned, click with the right-hand button on the PowerConnect 'Geometry'-window and choose the appropriate entry in the floating which allows to align the bottom surfaces of both beams. With this particular example, remain with the upper surfaces aligned.

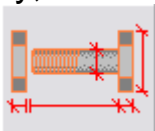
## 2.7.2 Verifying the connection elements

The connection which has just been created can directly be used for design analysis. It remains however possible to manually change the characteristics of any connection element (or at least to verify them) by double-clicking on the element. This can be done for the supporting beam and for the other beam, but keep the values as they are proposed by default.

Now double-click on the angle cleat, and verify if angle cleats have been foreseen at both sides of the beam web. If not, make sure both the "Front side" and "Rear side" options have been checked.



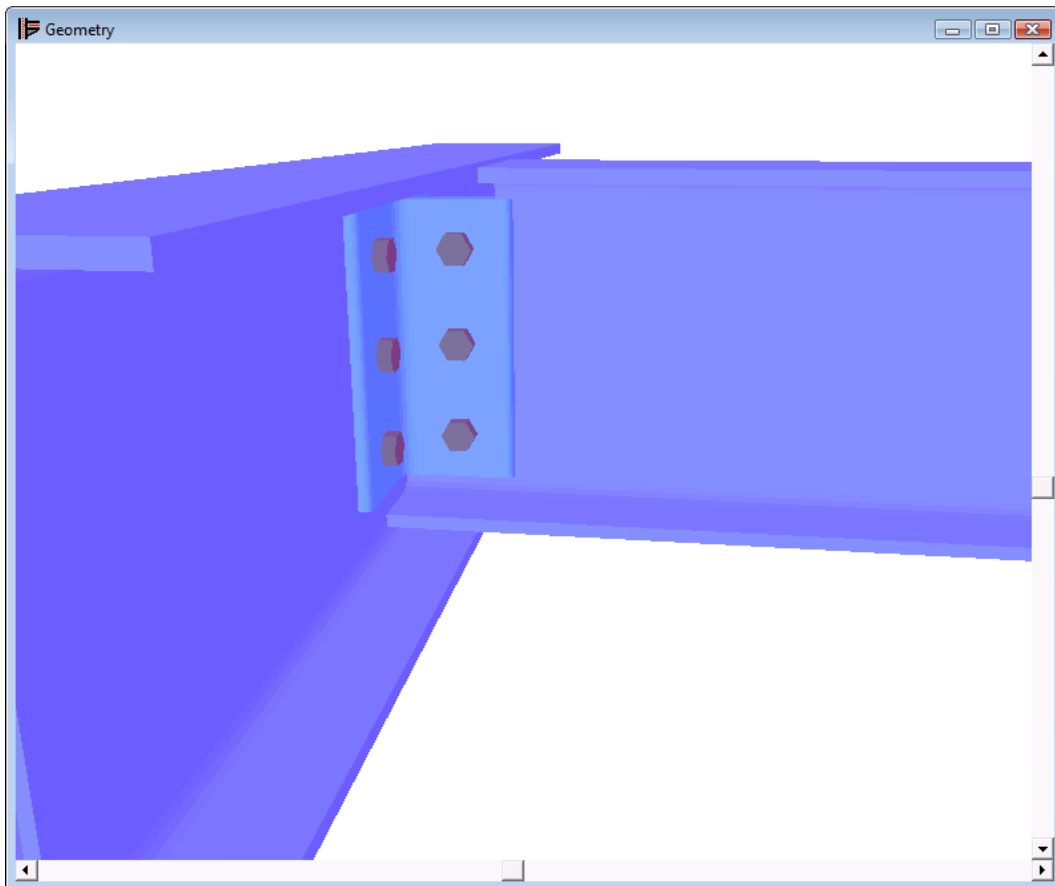
Finally, have a closer look at the bolts. Double-click on one of them, and use



the icon to change the bolt type from M20 (default bolt type) to M16 (8.8 grade).

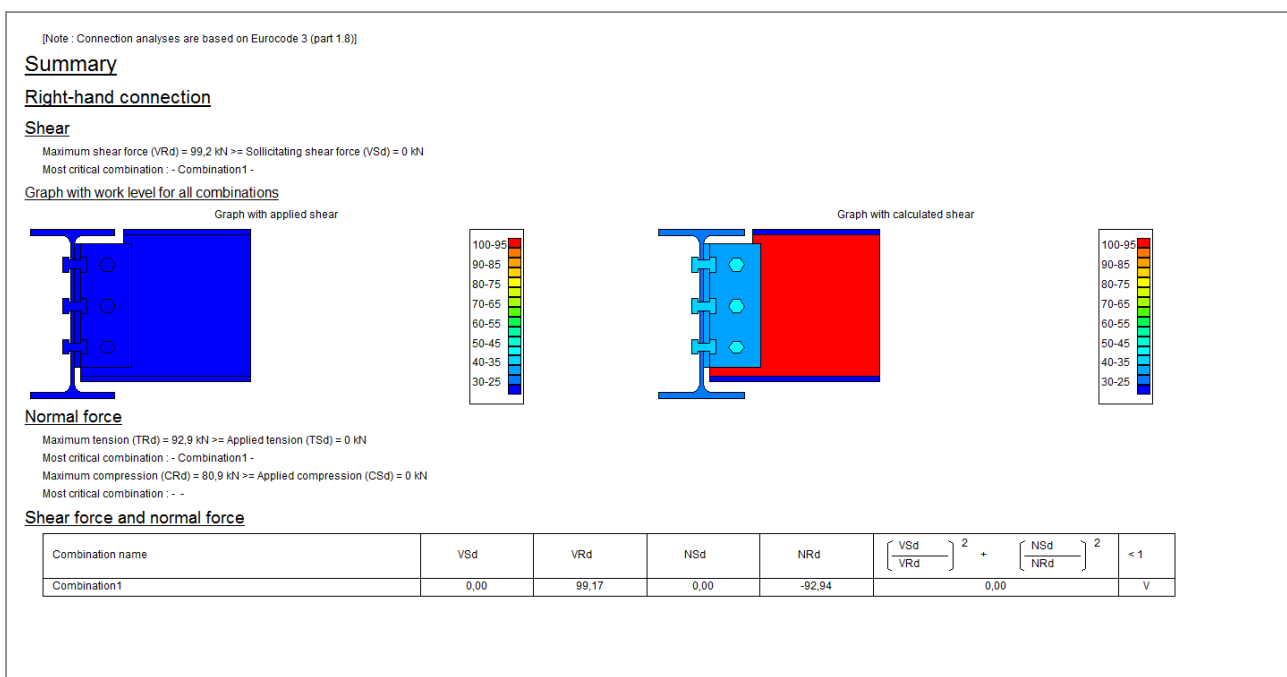




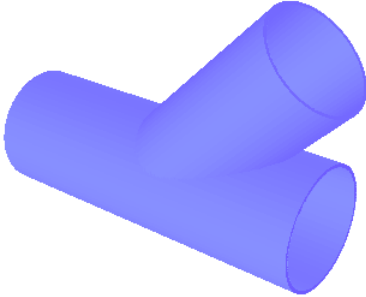


## 2.7.3 Running the connection design analysis


No loads are applied on the connection, as the only interest is to evaluate maximum shear resistance.



## 2.8 Tutorial 8 : HSS connection (circular members)

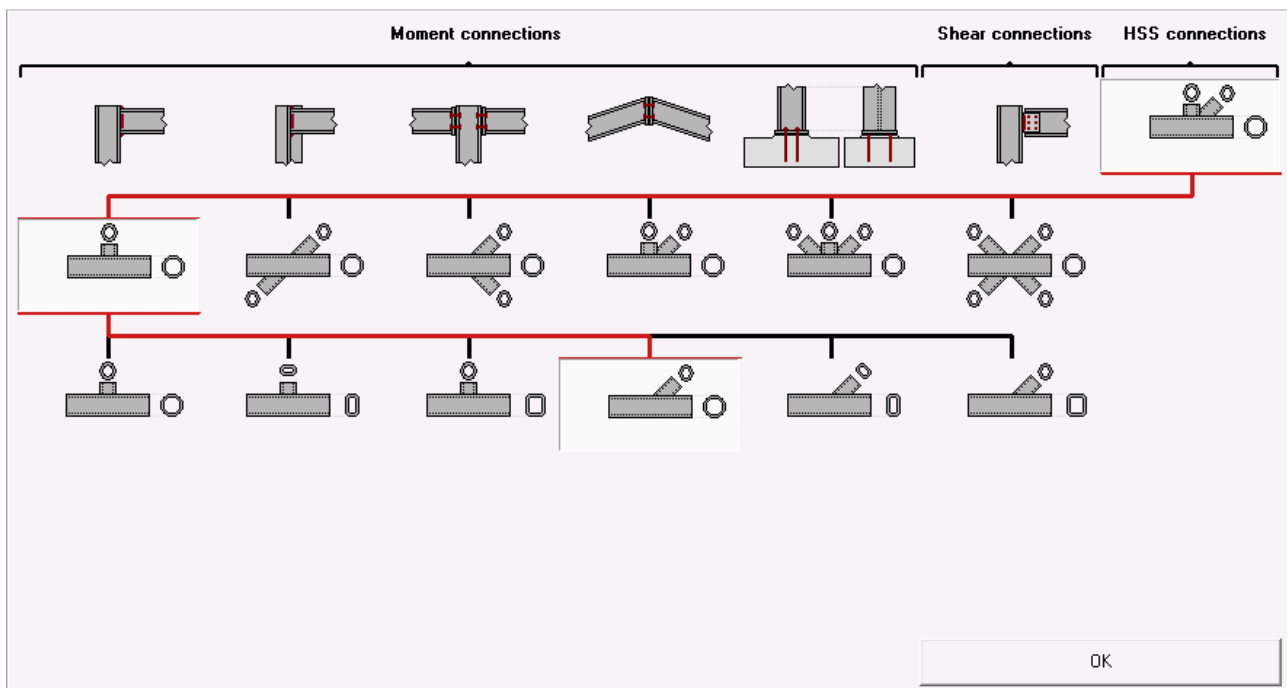


### 2.8.1 Setting up the model

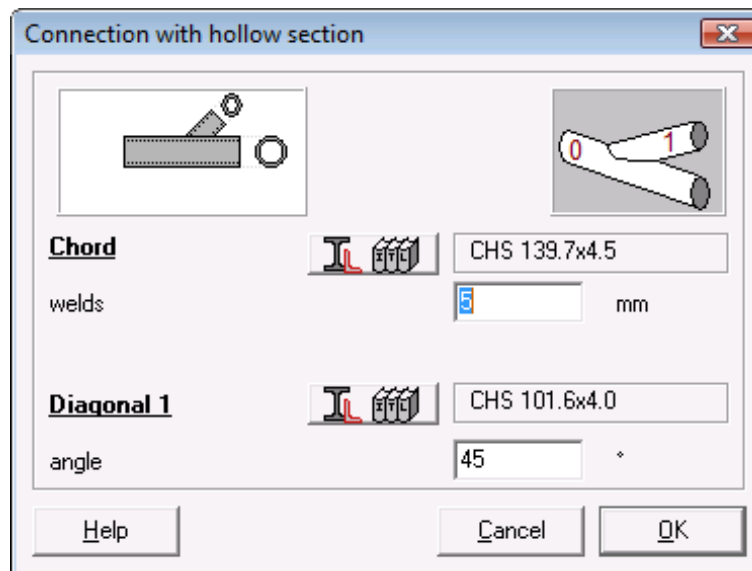
Click on the icon  to start a new PowerConnect project and select the **HSS Connections** icon from the navigation window. Choose following specifications to complete the connection type selection:

- Y/T connections
- Y (circular)

corresponding to the selection tree shown below:




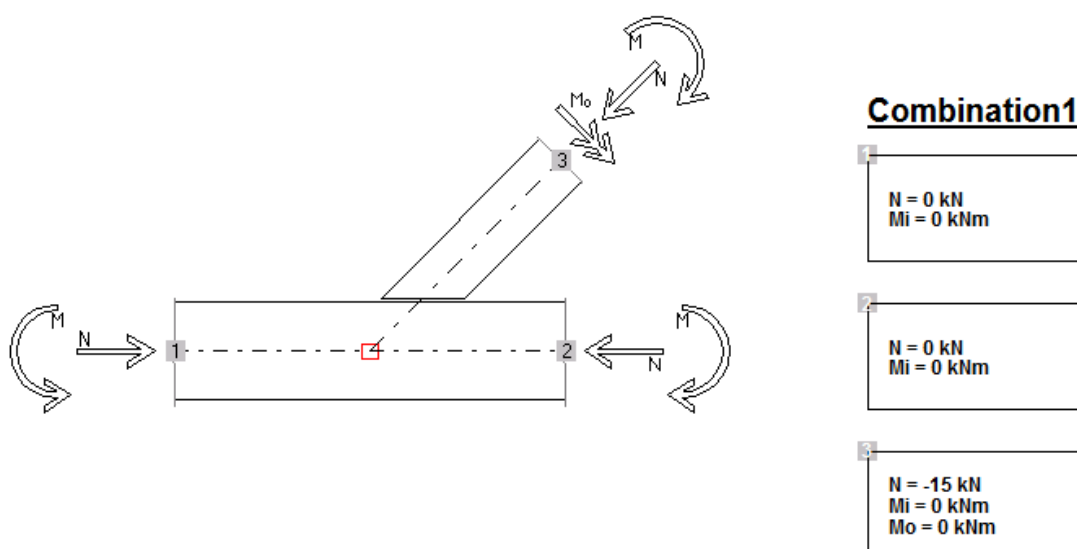
In a next dialogue window, further details can be provided on hollow structural sections (including their relative orientation) and welds. Accept the defaults as proposed by PowerConnect.




If wanted, further details on the hollow structural sections can be obtained (or modifications can be defined) by double-clicking on the corresponding members in the 3D visualization of the connection geometry.

## 2.8.2 Defining the loads

To apply a tensile load of 15kN to the diagonal chord, switch to the 'Loads'-window by means of the  icon. Click with the mouse on the label "3" of the 2D geometry representation in this 'Loads'-window to impose the load (use a value of -15kN to ensure a tensile load is defined).



## 2.8.3 Running the design analysis

Launch the design analysis . At this time, it should be remarked that the analysis method used for HSS connections is entirely different than the one used for the connections previously discussed in this tutorial. The method that is currently used should in future normally be replaced by the component method which is currently used by PowerConnect for all other connection types.

At this time, EUROCODE proposes a number of formula published by CIDECT, allowing for the analysis of a limited type of HSS connection configurations. This limits somewhat the number of connection types that can currently be analyzed by PowerConnect. Limitations are mostly related to the type of loads that can be considered during the analysis.

For the HSS connection that has been defined, the analysis will deliver following results.

Results with components for -Combination1-

Normal force

Maximum tension (NRd) = 120,1 kN >= Applied tension (NSd) = 15 kN

Tensile force resistance with the chord face failure = 120,1 kN

Tensile force resistance with the punching shear failure = 235,2 kN

Tensile force resistance with the brace cross-section = 288,1 kN

In-plane moment

Maximum moment (MRd) = 6,8 kNm >= Applied moment (MSd) = 0 kNm

Moment resistance with the chord face failure = 9,5 kNm

Moment resistance with the punching shear failure = 9,8 kNm

Moment resistance with the brace cross-section = 6,8 kNm

Out-of-plane moment

Maximum moment (MoRd) = 4,5 kNm >= Applied moment (MSd) = 0 kNm

Moment resistance with the chord face failure = 4,5 kNm

Moment resistance with the punching shear failure = 11,7 kNm

Moment resistance with the brace cross-section = 6,8 kNm

Moments and normal force

$$\frac{NSd}{NRd} + \left( \frac{MSd}{MRd} \right)^2 + \frac{MSd}{MoRd} \quad (= 0,125) < 1$$

show

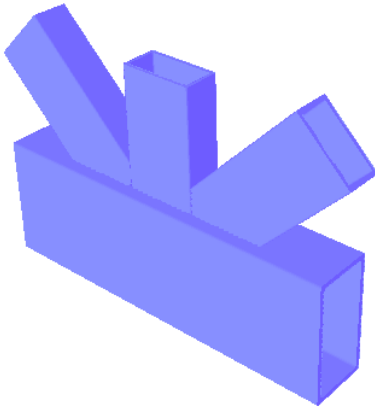
☒ combination ☐ headers ☒ components ☐ details

☐ summary


PowerConnect evaluates the maximum values for different types of loads that can be applied on the connection, in particular normal forces and in-plane & out-of-plane bending moments.

PowerConnect will furthermore use those values to summarize the overall “loading level” of the connection, by comparing applied loads to maximum resistance using a dedicated combination formula. As such, judgment of connection strength really becomes a straightforward matter.

## 2.9 Tutorial 9 : HSS connections (rectangular members)

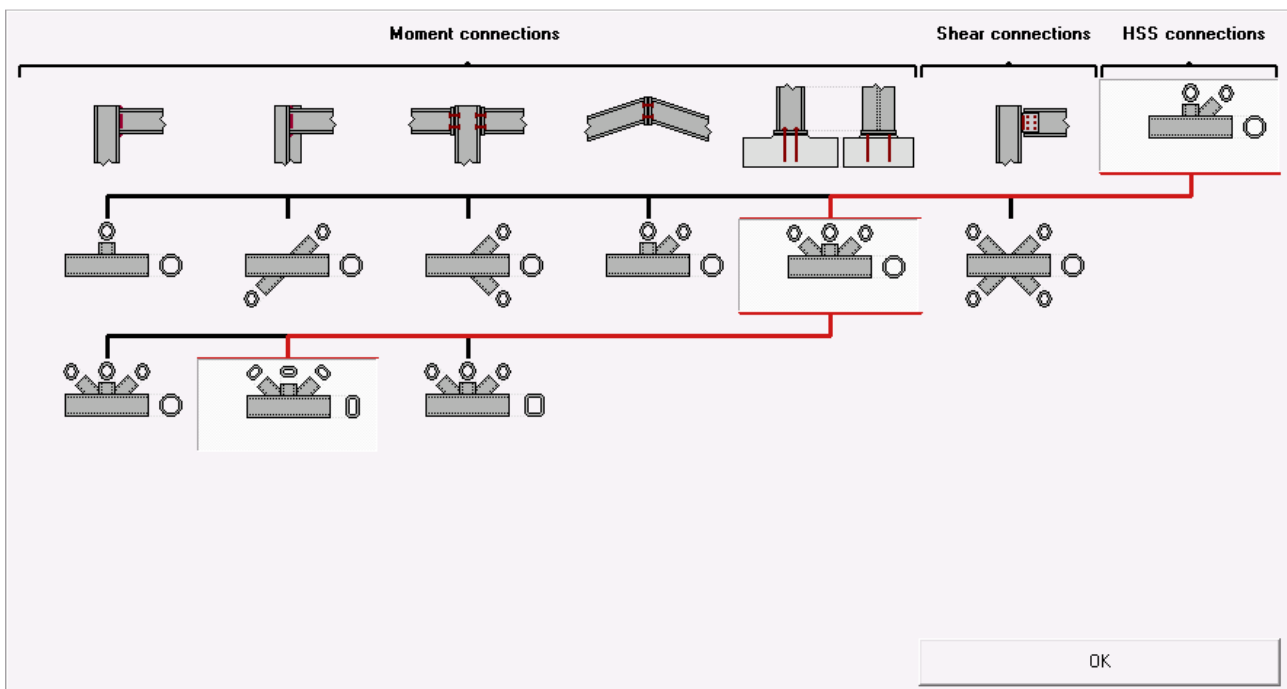


### 2.9.1 Setting up the model

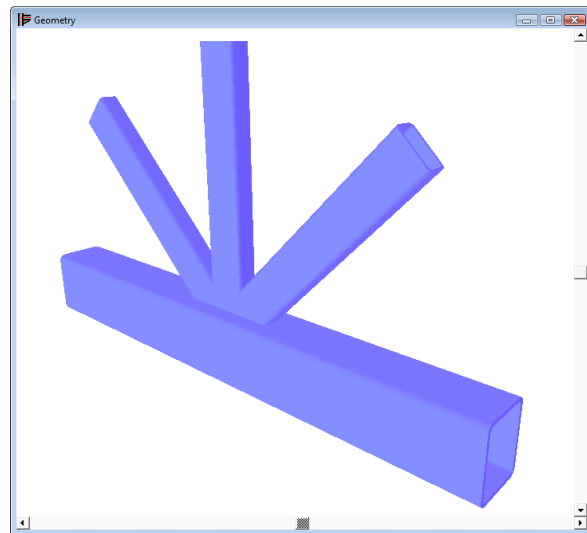
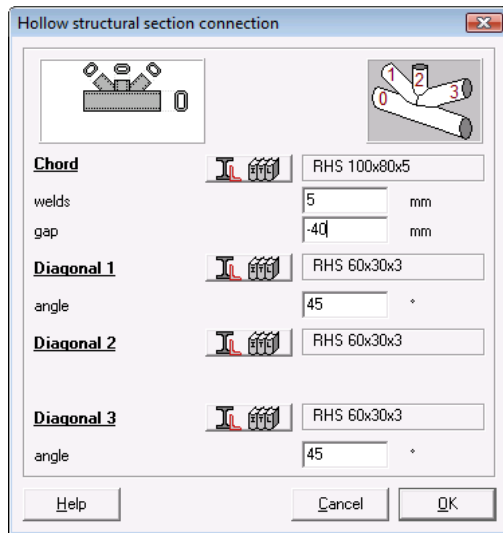
Click on the icon  to start a new PowerConnect project and select the **HSS Connections** icon from the navigation window. Choose following specifications to complete the connection type selection:

- KT connections
- KT (rectangular)

corresponding to the selection tree shown below:

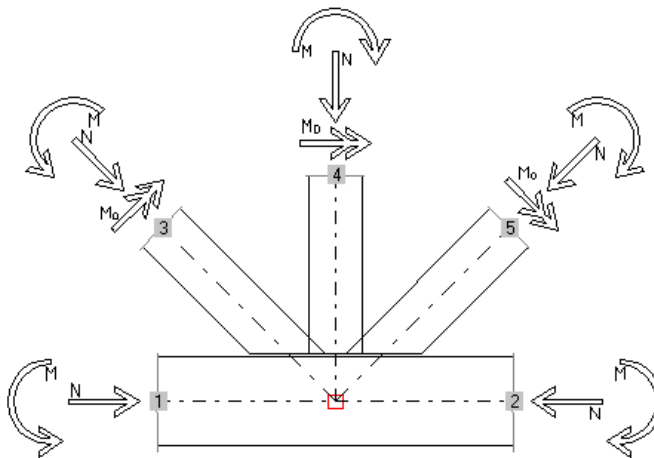


For the main chord, a 100\*80\*5 section will be used, whereas for a 60\*30\*3 section will be used for all diagonals. Accept all other values as proposed by default by PowerConnect.



## 2.9.2 Defining the loads

Now switch to the 'Loads'-windows and apply the loads as illustrated below, by clicking with the mouse on the appropriate labels with the 2D geometry representation.



### Combination1

1  
N = 0 kN  
Mi = 0 kNm

2  
N = 0 kN  
Mi = 0 kNm

3  
N = 25 kN  
Mi = 0 kNm  
Mo = 0 kNm

4  
N = 20 kN  
Mi = 0 kNm  
Mo = 0 kNm

5  
N = -25 kN  
Mi = 0 kNm  
Mo = 0 kNm

## 2.9.3 Running the connection design analysis

The design analysis will present following results, and thus confirm that the connection can withstand the specified loads.

### Summary

#### Left-hand hollow section

##### Moments and normal force

NSd	NRd	MSd	MRd	MoSd	MoRd	$\frac{NSd}{NRd} + \frac{MSd}{MRd} + \frac{MoSd}{MoRd}$
25 kN	90,5 kN	-0,3 kNm	1,6 kNm	0 kNm	0 kNm	0,470

NSd = Sollicitating normal force in secondary tube

NRd = Normal force resistance of secondary tube

MSd = Sollicitating in-plane moment on secondary tube

MRd = In-plane moment resistance of secondary tube

MoSd = Sollicitating out-of-plane moment on secondary tube

MoRd = Out-of-plane moment resistance of secondary tube

The most critical combination is : Combination1

#### Middle hollow section

##### Moments and normal force

NSd	NRd	MSd	MRd	MoSd	MoRd	$\frac{NSd}{NRd} + \frac{MSd}{MRd} + \frac{MoSd}{MoRd}$
20 kN	55,7 kN	0 kNm	1,6 kNm	0 kNm	0 kNm	0,359

NSd = Sollicitating normal force in secondary tube

NRd = Normal force resistance of secondary tube

MSd = Sollicitating in-plane moment on secondary tube

MRd = In-plane moment resistance of secondary tube

MoSd = Sollicitating out-of-plane moment on secondary tube

MoRd = Out-of-plane moment resistance of secondary tube

The most critical combination is : Combination1

#### Right-hand hollow section

##### Moments and normal force

NSd	NRd	MSd	MRd	MoSd	MoRd	$\frac{NSd}{NRd} + \frac{MSd}{MRd} + \frac{MoSd}{MoRd}$
25 kN	90,5 kN	-0,3 kNm	1,6 kNm	0 kNm	0 kNm	0,470

NSd = Sollicitating normal force in secondary tube

NRd = Normal force resistance of secondary tube

MSd = Sollicitating in-plane moment on secondary tube

MRd = In-plane moment resistance of secondary tube

MoSd = Sollicitating out-of-plane moment on secondary tube

MoRd = Out-of-plane moment resistance of secondary tube

The most critical combination is : Combination1

#### All upper hollow sections

##### In-plane moment

Combination1 :  $|MSd(3) + MSd(2) - MSd(1)|$  (=0 kNm) <= MRd (= 4,8 kNm)

##### Out-of-plane moment

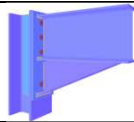
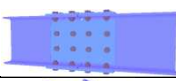
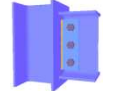
Combination1 :  $|MoSd(1) + MoSd(2) + MoSd(3)|$  (=0 kNm) <= MoRd (= 0 kNm)



# 3 PowerConnect reporting tutorials

Using some of the tutorial examples described in section 2 of this manual, the current section will focus on the information that is needed to get started with PowerConnect reporting in a short time frame. The reporting capabilities will not be described in detail, as the documentation related to this subject is covered in a separate reference manual.

## Inventory of reporting tutorials:

Page	Section	Tutorial contents	Design code	Connection
page 69	§ 3.1	Tutorial 1 : bolted beam to column flange (EC3)	EC3	
page 75	§ 3.2	Tutorial 2 : bolted splice	EC3	
page 80	§ 3.3	Tutorial 3 : shear connection – beam to column flange with fin plate	EC3	

## 3.1 Tutorial 1 : bolted beam to column flange (EC3)

This reporting tutorial is based on the model created in section 2.1 of this manual.

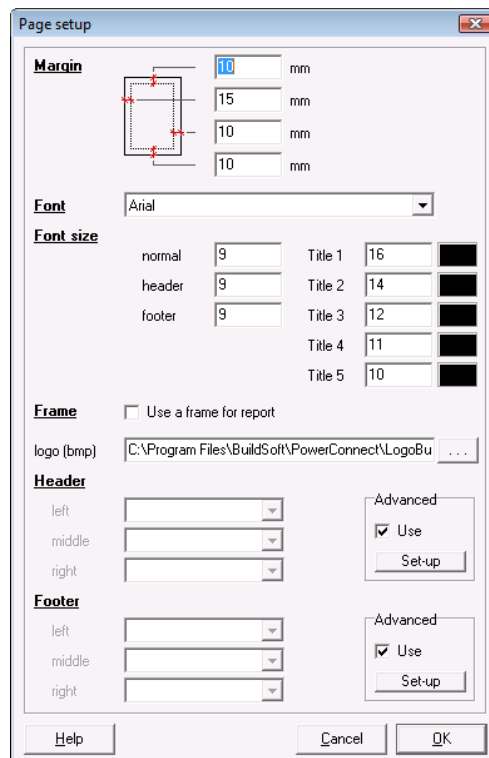
### 3.1.1 Page setup

Before the actual report contents will be defined, the report page setup should be specified. Any page setup definition performed by the user will be remembered by PowerConnect, and will therefore be used also to create any subsequent report until new modification to the current page setup will be specified. For all reporting tutorials in this manual, we will stick with the page setup definition as specified in this section.

To start the page setup, use the menu command 'File – Page setup', so that the dialogue window below will be launched.

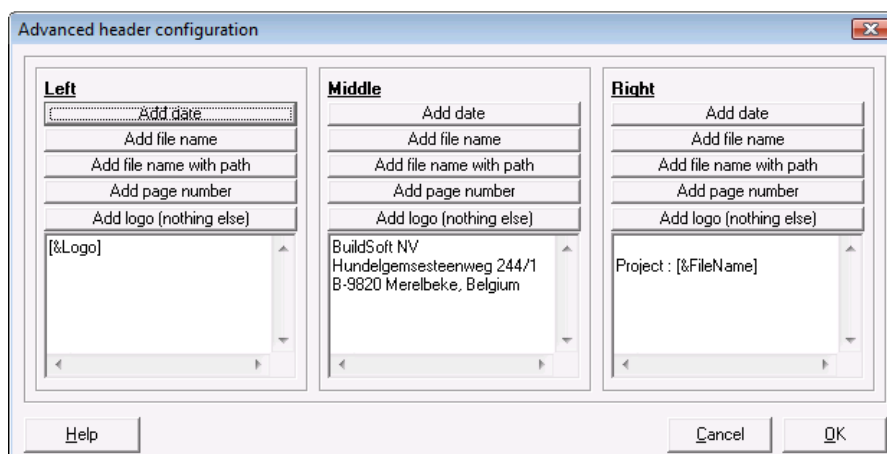
Apart from the rather obvious options related to definition of margins, fonts and font size, this dialogue allows to define the location of a bitmap file that

should be used to include a logo in the report. If needed, the user can also request the text and graphics of the report to be included in a frame by selecting the option 'Use a frame for report'. This option is not activated with the current reporting tutorial.

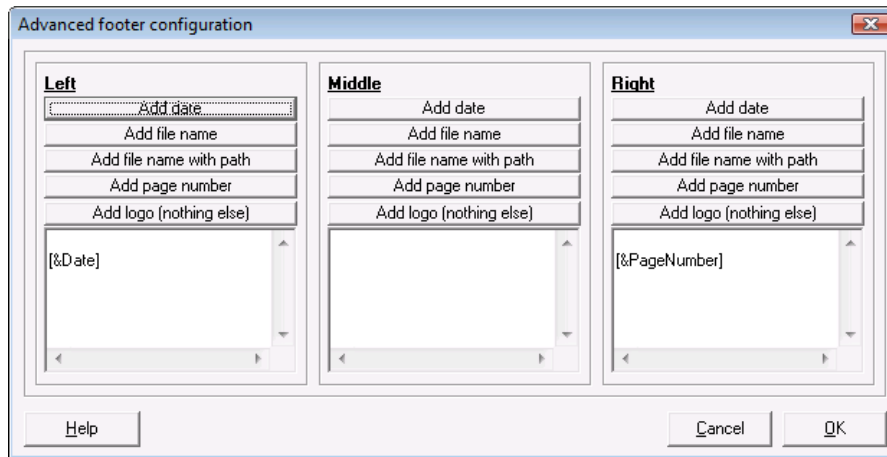


To define report header & footer, the user can either use a number of preconfigured fields or enter into an advanced definition mode. Switch to this advanced definition mode, and enter the data as shown below:

- for the report header:






- for the report footer:



### 3.1.2 Report configuration

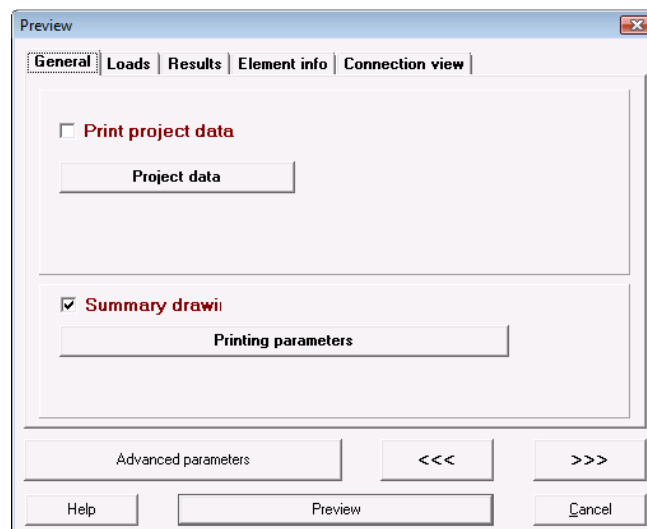
The report can be configured in either one of the following three modes:

- **Preview** (initiated by the  icon of the icon toolbar),
- **Print report** (initiated by the  icon of the icon toolbar),
- **Print report to RTF** (initiated by the  icon of the icon toolbar).

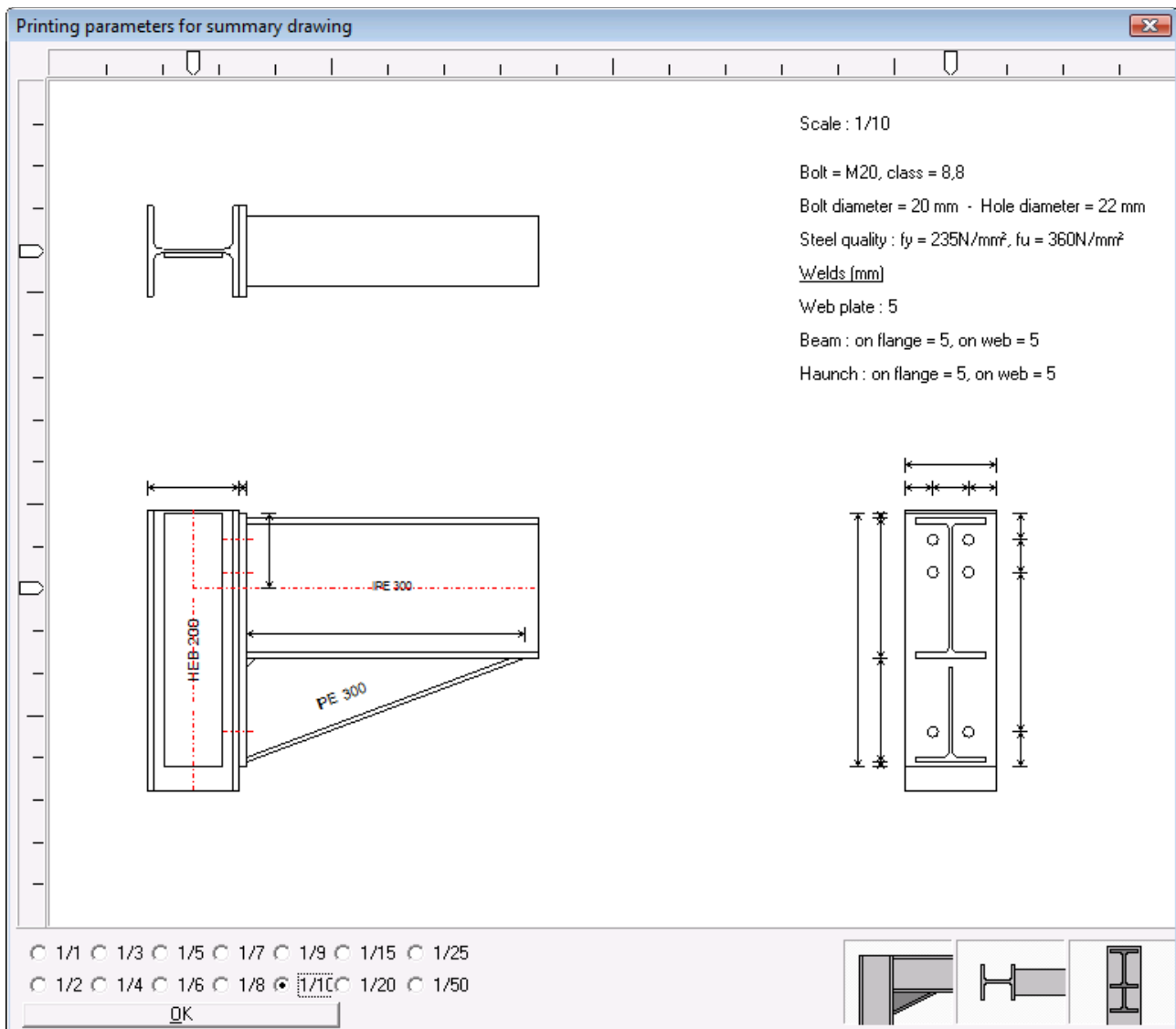
Irrespective of the chosen configuration mode, the working procedure is the same. In the scope of this tutorial, we will use the **Preview** mode.

In Preview or another configuration mode, 5 tab pages are available in the dialogue window used to define report contents. In this tutorial, only the first 3 tab pages will be used.

Start by selecting the option 'Summary drawing' on the first tab page. By doing so, the report will start with a summary overview of the entire connection geometry. How this summary overview should look like, can be defined by using the button 'Printing parameters', which opens a canvas window in which the summary overview definition can be completed.

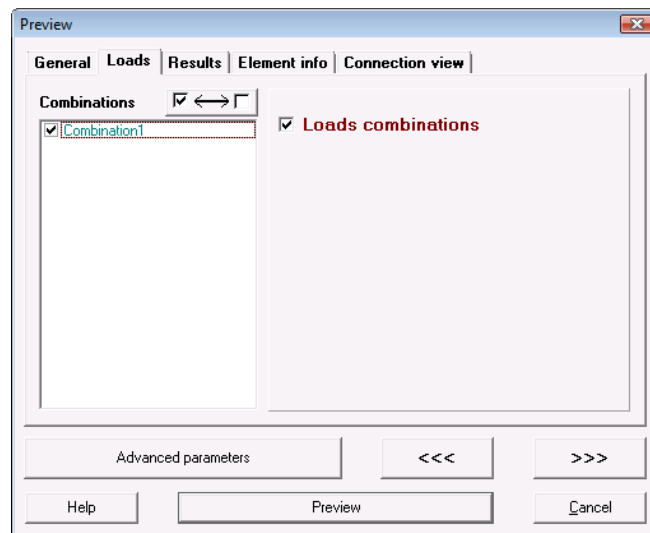


To create the kind of summary drawing shown below, make sure to select all connection views shown at the right-hand bottom part of the canvas window. Use the cursor icons to position the selected drawings in the appropriate positions, and also make sure the proper scale is selected (the selected scale of 1/10 is appropriate for a report on A4 paper format).

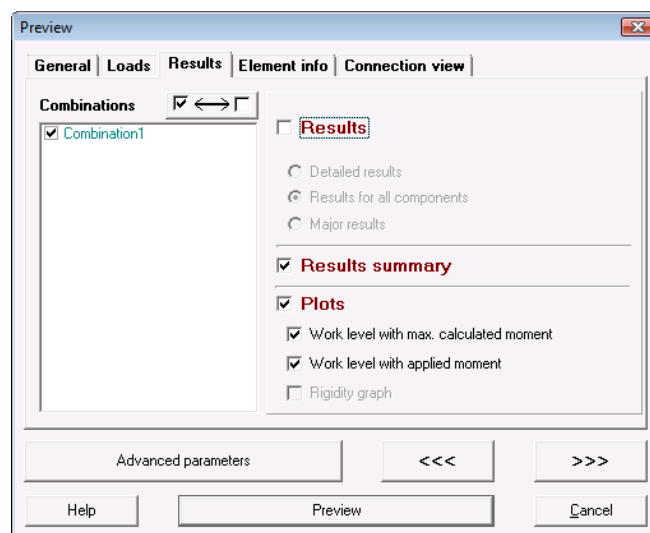


As soon as a proper lay-out has been defined, confirm by means of the 'OK'-button to return to the definition dialogue window.

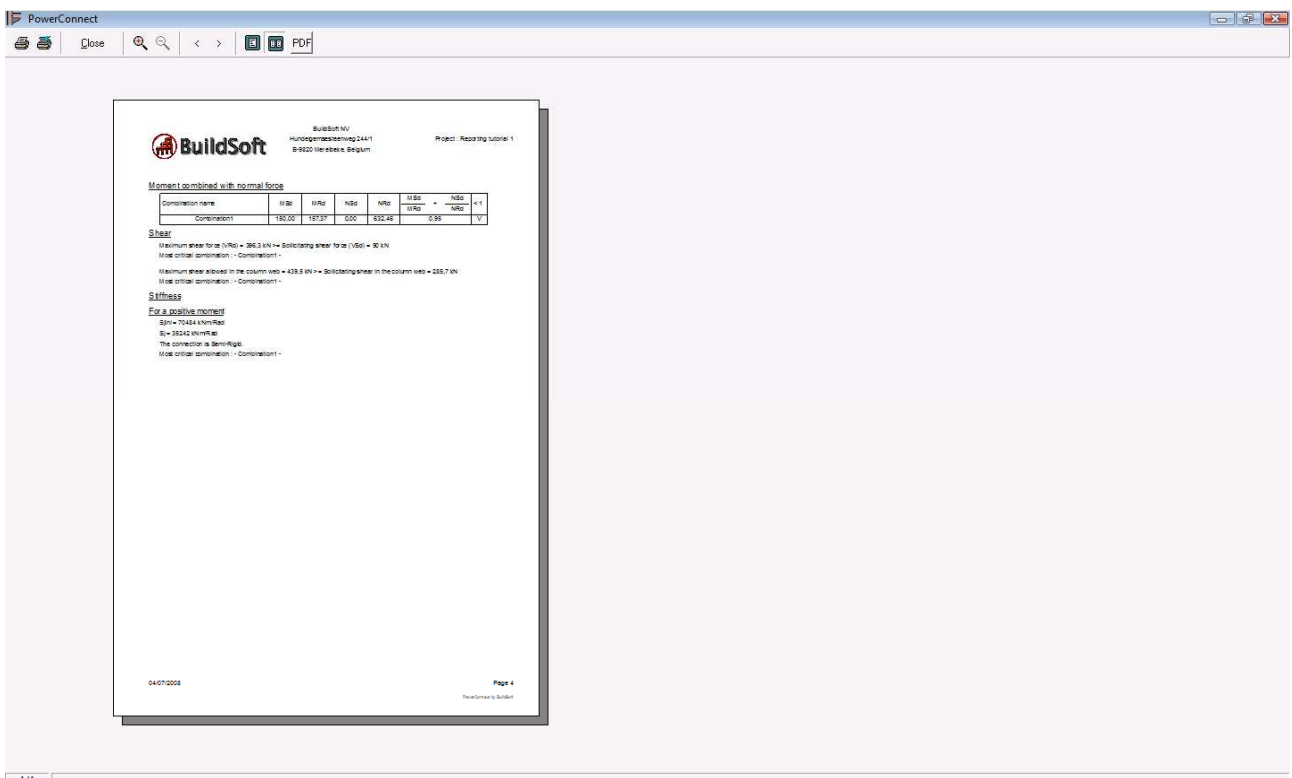
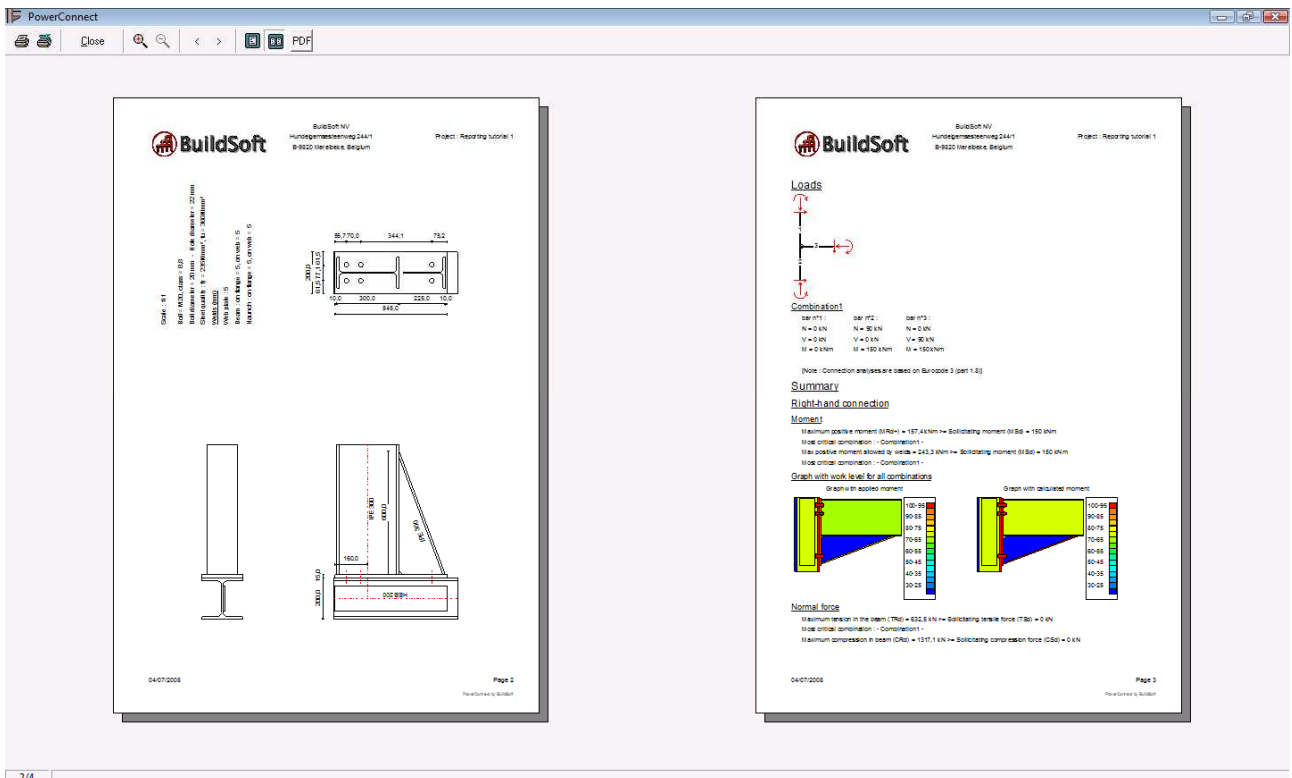
Now switch to the second tab page, and make sure that the option 'Loads combinations' is selected. This will ensure that the report includes an overview of all defined loads combinations. In case more than 1 loads combination is present, you should also make sure to select the loads combinations to be used for reporting in the list on the left-hand side. In the current tutorial example, 'Combination1' should be selected as indicated below.





Now switch to the third tab page to specify how analysis results should be reported. Within this first reporting tutorial, the entire focus is on the creation of a concise analysis report. To do so, select the options 'Results summary' and 'Plots' as indicated below.



The report configuration can now be considered to be complete, and the report itself can be visualized by means of the 'Preview' button at the bottom of the dialogue window. The outcome will be a 3 page report as shown below.



In case the current report should be available in PDF-format, use the appropriate  button. If not, use the 'Close'-button to return to the PowerConnect working environment. To actually print the report that has just been configured, use the icon  to enter in **Print report** mode, and use the 'Print' button to send the report to the selected printer.


## 3.2 Tutorial 2 : bolted splice (EC3)

This reporting tutorial is based on the model created in section 2.5 of this manual.

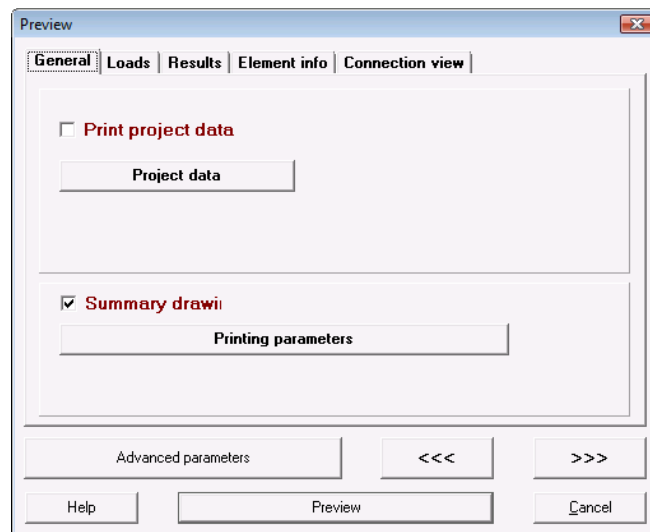
### 3.2.1 Page setup

No changes will be made to the page setup defined in section 3.1.1 of this manual. The user should return to this section for more information on the current page setup.

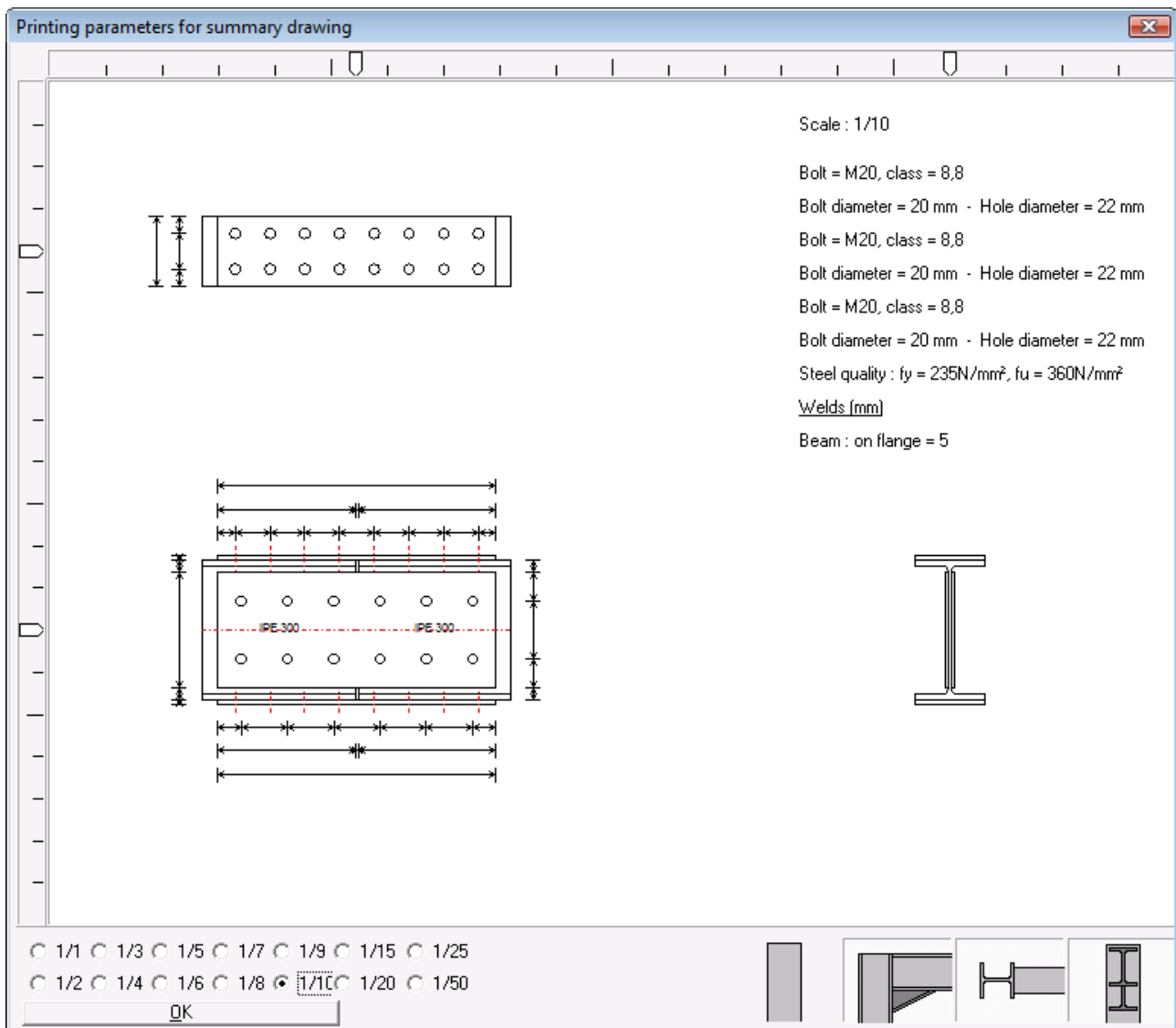
### 3.2.2 Report configuration

Go to **Preview** mode by means of the  icon of the icon toolbar.

Again, start by selecting the option 'Summary drawing' on the first tab page and define how this summary overview should look like by using the button 'Printing parameters', which opens a canvas window to complete the summary overview definition.



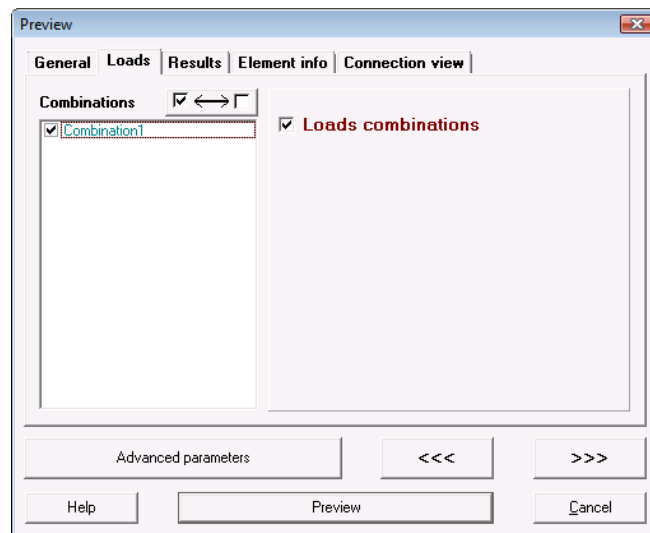
To create the kind of summary drawing shown below, make sure to select all connection views shown at the right-hand bottom part of the canvas window. Use the cursor icons to position the selected drawings in the appropriate positions, and also make sure the proper scale is selected (the selected scale of 1/10 is appropriate for a report on A4 paper format).



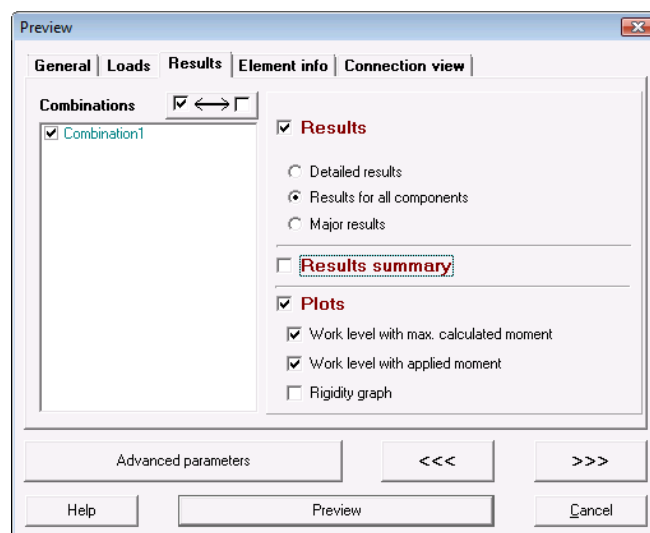
As soon as a proper lay-out has been defined, confirm by means of the 'OK'-button to return to the definition dialogue window.

Now switch to the second tab page, and make sure that the option 'Loads combinations' is selected. This will ensure that the report includes an overview of the loads combination that is part of this project, at least if 'Combination1' is selected as indicated below.



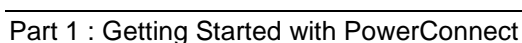
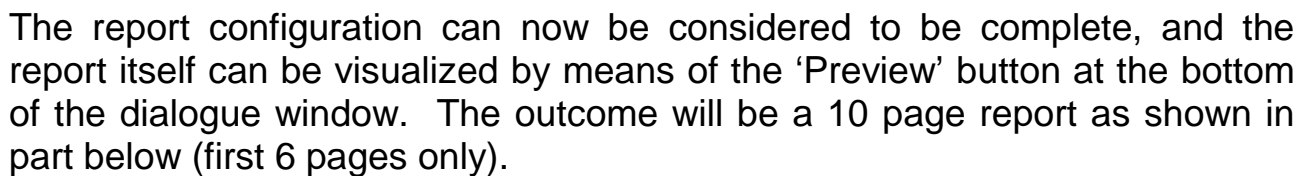


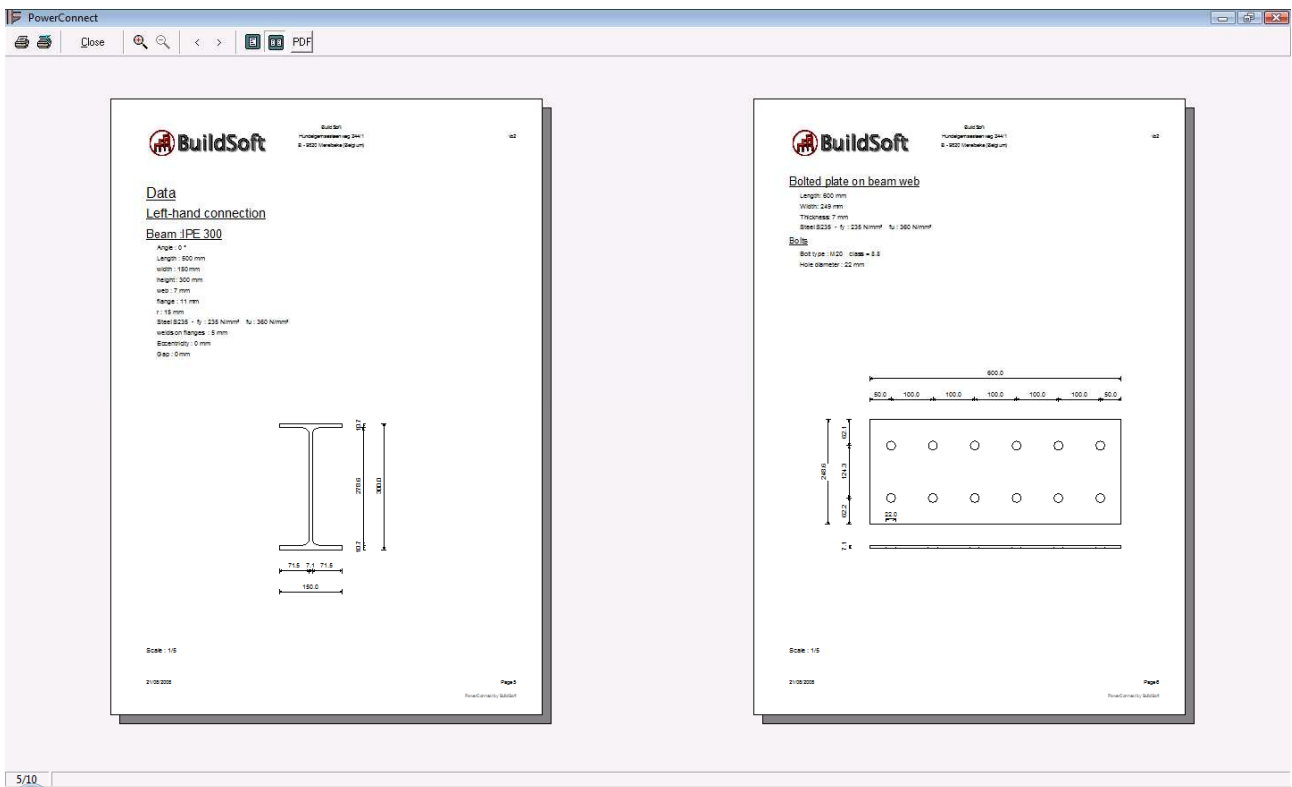
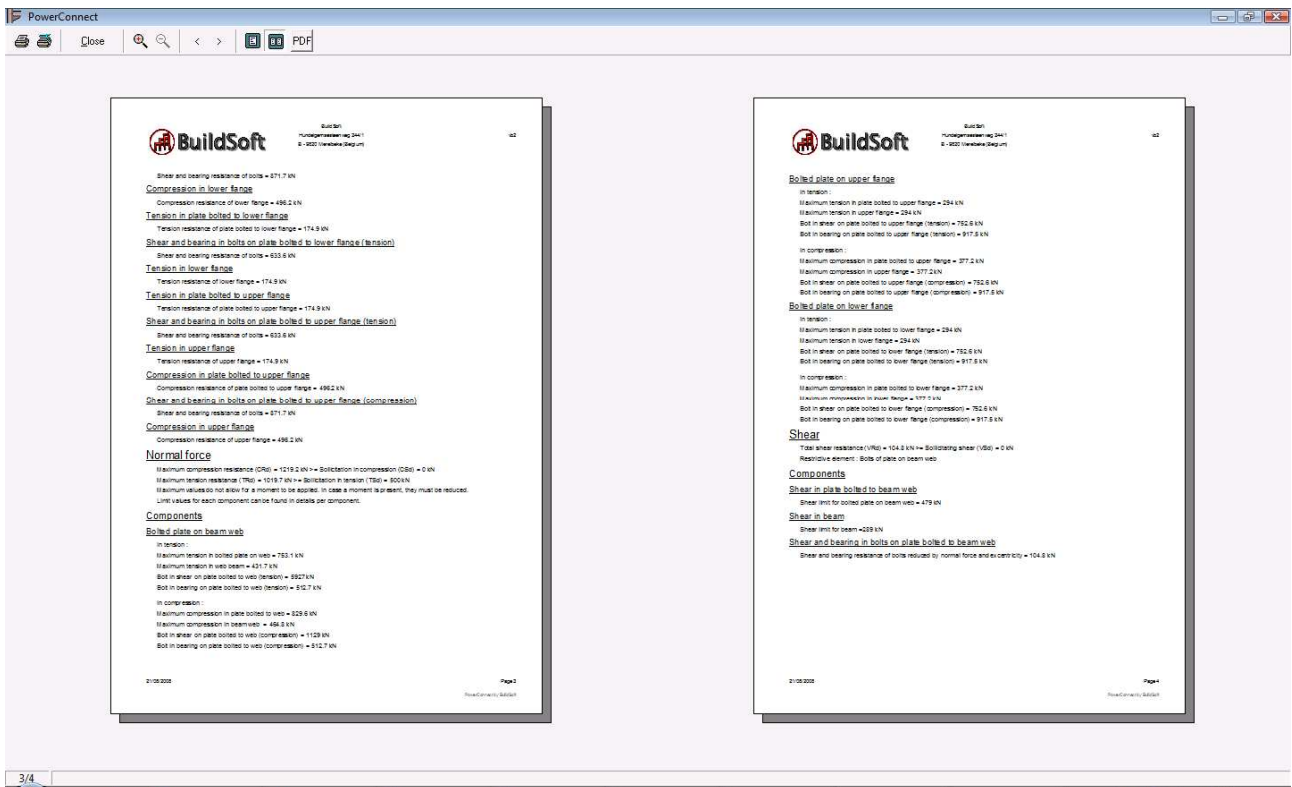
Now switch to the third tab page to specify how analysis results should be reported. Within this second reporting tutorial, the focus is on the creation of a more detailed analysis report. To do so, select the options 'Results – Results for all components' and 'Plots' as indicated below.





Now switch to the fourth tab page to specify that detailed drawings and data of all individual connection elements are to included into the report. For all elements, make sure to specify 'Yes' for both the 'Data' and 'Drawing' columns. This can be done for each entry individually, by selecting 'Yes' from the available pull-down menus. Alternatively, the buttons 'V Yes' on top of the columns can be used to convert globally to this setting for all elements.

In the 'Scale' column, a proper scale can be defined to be used for the individual element drawings. Again, this scale can be defined globally by choosing the proper scale factor through the pull-down menu on top of the column. It should be remarked that the drawing will always be rescaled automatically by PowerConnect in case it will not fit on the selected page format. If it does fit, the scale as defined by the user will not be modified.





In case the current report should be available in PDF-format, use the appropriate  button. If not, use the 'Close'-button to return to the PowerConnect working environment. To actually print the report that has just been configured, use the icon  to enter in **Print report** mode, and use the 'Print' button to send the report to the selected printer.


## 3.3 Tutorial 3 : shear connection – beam to column flange with fin plate (EC3)

This reporting tutorial is based on the model created in section 2.6 of this manual.

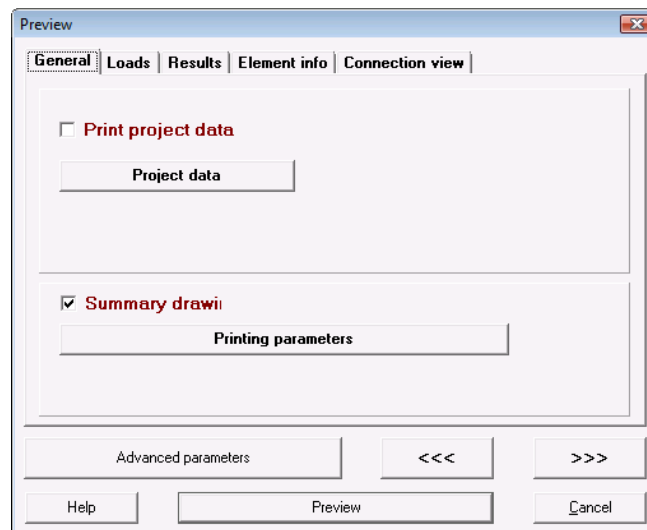
### 3.3.1 Page setup

No changes will be made to the page setup defined in section 3.1.1 of this manual. The user should return to this section for more information on the current page setup.

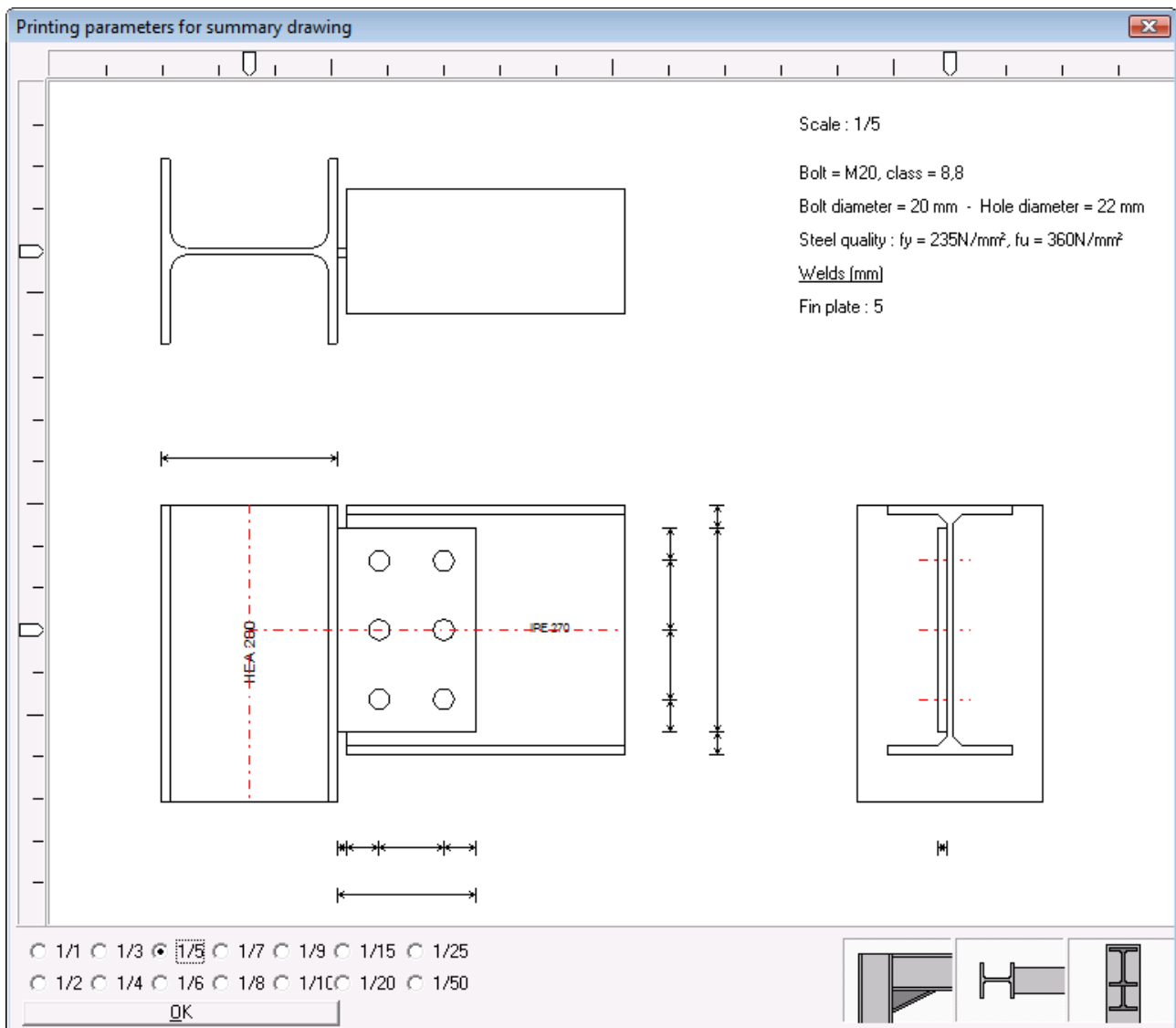
### 3.3.2 Report configuration

Go to **Preview** mode by means of the  icon of the icon toolbar.

Again, start by selecting the option ‘Summary drawing’ on the first tab page and define how this summary overview should look like by using the button ‘Printing parameters’, which opens a canvas window to complete the summary overview definition.

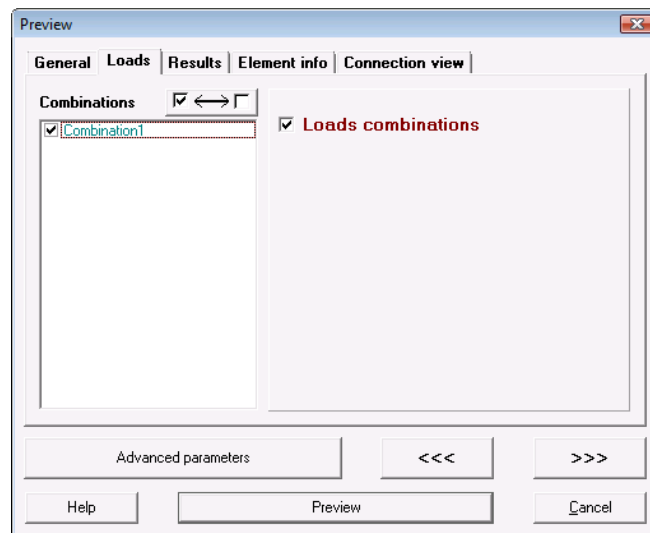


To create the kind of summary drawing shown below, make sure to select all connection views shown at the right-hand bottom part of the canvas window. Use the cursor icons to position the selected drawings in the appropriate positions, and also make sure the proper scale is selected (the selected scale of 1/5 is appropriate for a report on A4 paper format).

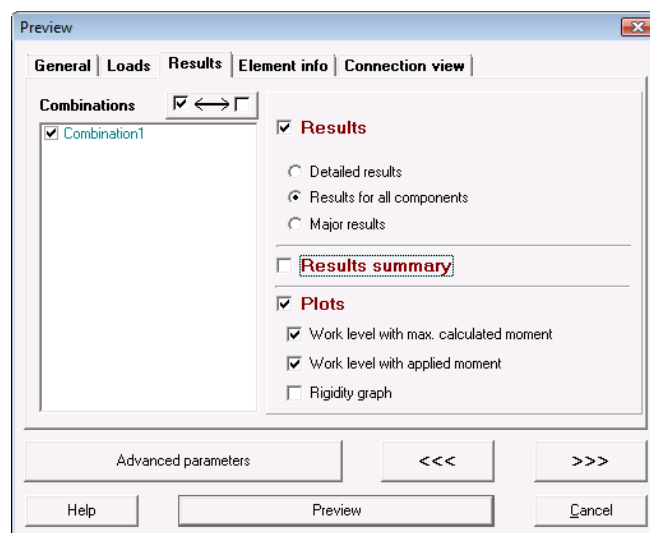


As soon as a proper lay-out has been defined, confirm by means of the 'OK'-button to return to the definition dialogue window.

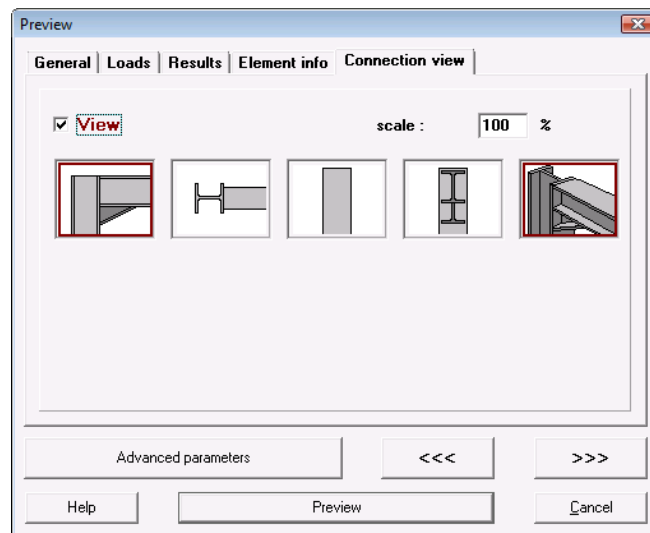
Now switch to the second tab page, and make sure that the option 'Loads combinations' is selected. This will ensure that the report includes an overview of the loads combination that is part of this project, at least if 'Combination1' is selected as indicated below.



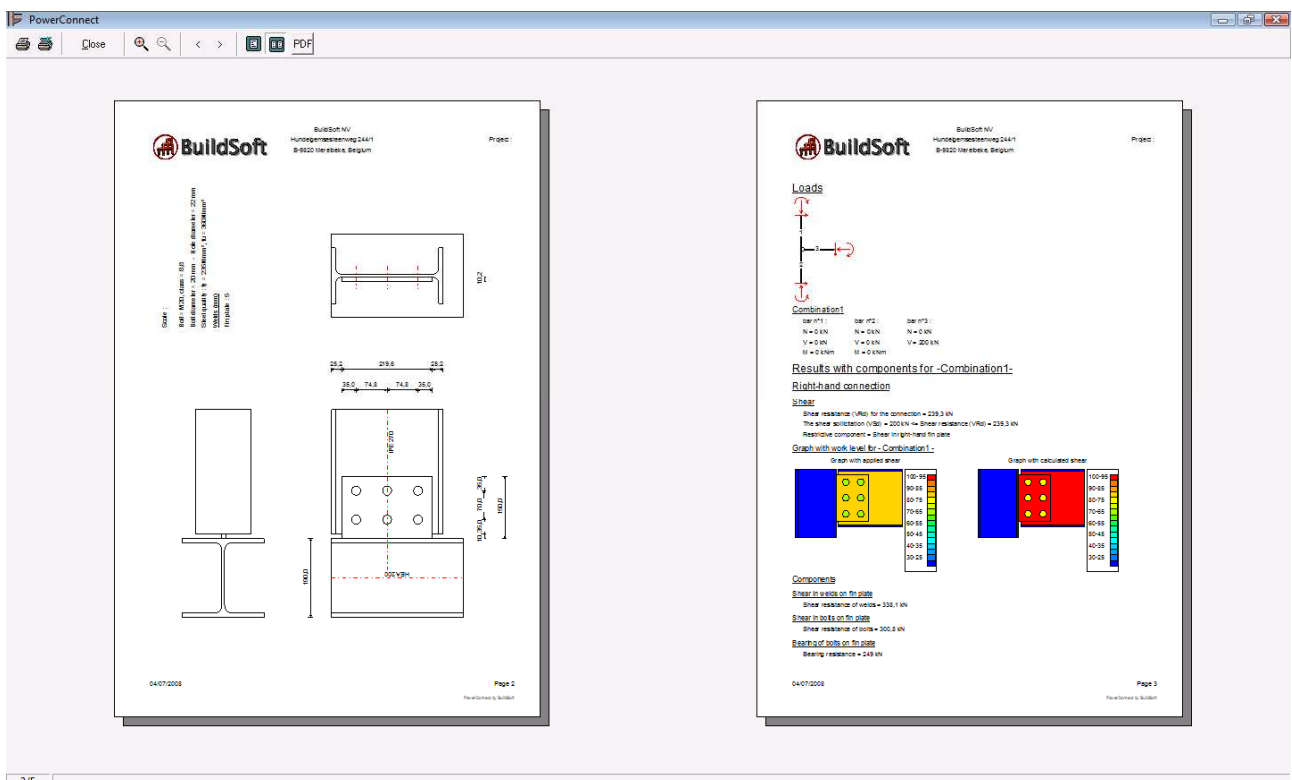
Now switch to the third tab page to specify how analysis results should be reported. Within this second reporting tutorial, the focus is on the creation of a more detailed analysis report. To do so, select the options 'Results – Results for all components' and 'Plots' as indicated below.

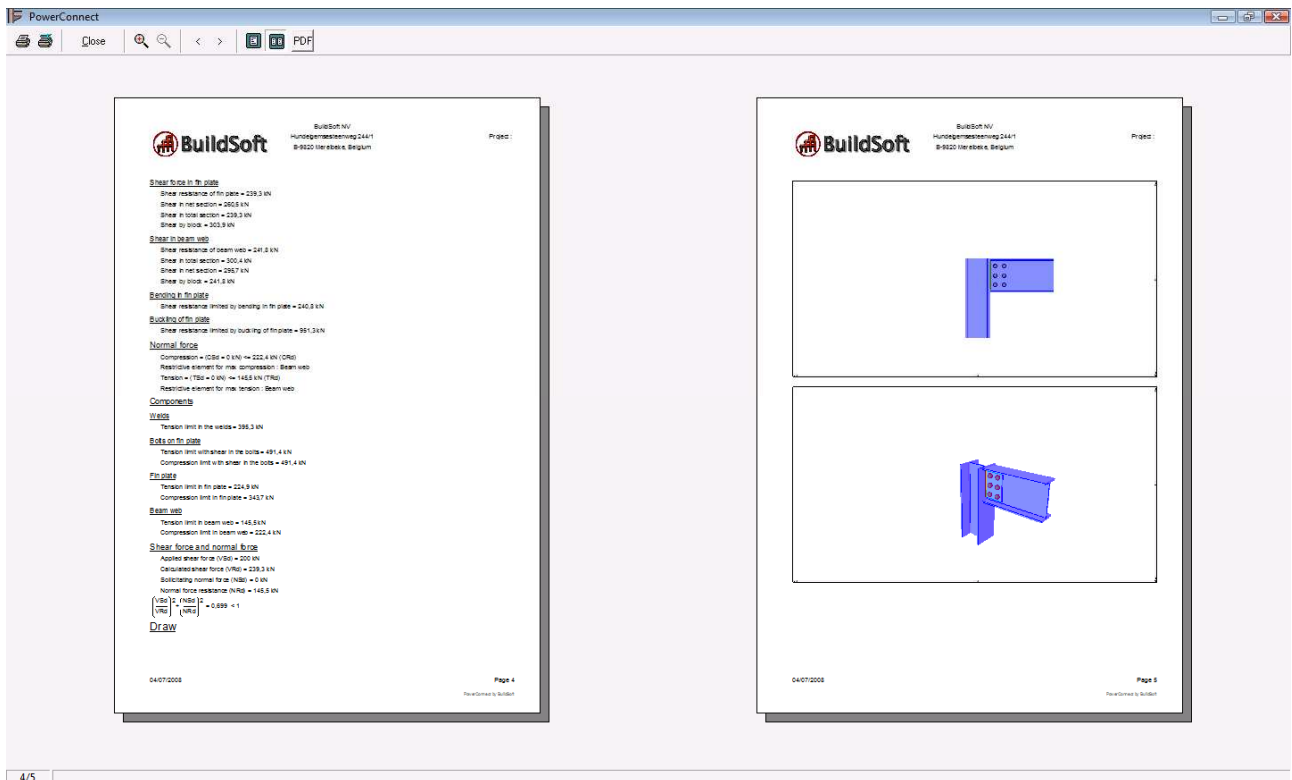




Now switch to the fifth tab page to specify that rendered views of the entire connection are to included into the report. Select the option 'View' and make sure to select both the 3D and the front view icons.



The report configuration can now be considered to be complete, and the report itself can be visualized by means of the 'Preview' button at the bottom of the dialogue window. The outcome will be a 4 page report as shown below.





In case the current report should be available in PDF-format, use the appropriate  button. If not, use the 'Close'-button to return to the PowerConnect working environment. To actually print the report that has just been configured, use the icon  to enter in **Print report** mode, and use the 'Print' button to send the report to the selected printer.