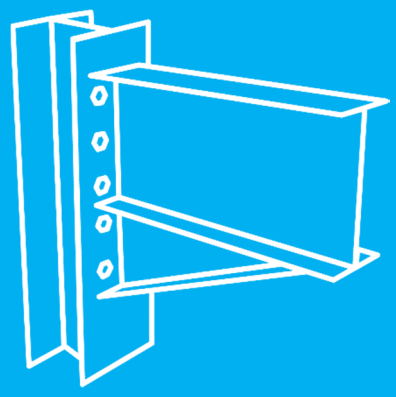
lateral buckling restraint - attaches - steel check - creep - charges climatiques - dynamic analysis - lateral buckling - brandweerstandsanalyse - timber - 1st order - verstijvers - buisverbinding - diseño de planos de armaduras - pandeo lateral - verbindingen - shear connection - verificación - armatures longitudinales - pórtico - unión base columna - voorontwerp - unión tubular - haunch - connexion moment - cimbras - vérification acier - unity check - Eurocode 2 - mesh - retaining wall - raidisseur - Eurocode 3 - longitudes de pandeo - connections - ACI 138 - acero - 2nd ordre - portal frame - Eurocode 8 - andamios - kip - dwarskrachtverbinding - BS 8110 - dalle de fondation - seismische analyse - armaduras longitudinales - BIM - gelaste verbinding - 2de orde - buckling - funderingszool - poutre sur plusieurs appuis - maillage - malla - uniones - 2D raamwerken - fire resistance analysis - voiles - cracked deformation - gescheurde doorbuiging - longueurs de flambement - pandeo - reinforcement - unity check - cantonera - dynamische analyse - hout - ossatures 3D - koudgevormde profielen - placa de extreme - 1er orden - continuous beam - connexion soudée - momentverbinding - praktische wapening - renforts au déversement - fluencia - estribos - déformation fissurée - EHE - beugels - Eurocódigo 3 - platine de bout - análisis dinámico - column base plate - kruip - rigid link - welded connection - charpente métallique - moment connections - estructuras 2D - kniestuk - assemblage métallique - 3D raamwerken - second ordre - beam grid - cargas climáticas - Eurocode 2 - Eurocode 5 - wall - deformación fisurada - lien rigide - enlace rígido - 2D frames - estructuras 3D - éléments finis - vloerplaat - steel connection - scheurvorming - integrated connection design - armatures pratiques - analyse sismique - nieve y viento - practical reinforcement - charges mobiles - dalle - wapening - perfiles conformados en frío - Eurocode 3 - connexion tubulaire - unión a momento - 3D frames - treillis de poutres - roof truss - practical reinforcement design - portique - kipsteunen - análisis sísmico - Eurocode 8 - seismic analysis - B.A.E.L 91 - uniones atornilladas - bolts - ossatures 2D - eindige elementen - losa de cimentación - restricciones para el pandeo lateral - optimisation - wand - kniklengtes - end plate - dakspanten - kolomvoetverbinding - stirrups - acier - staalcontrole - cálculo de uniones integrado - paroi - dessin du plan de ferraillage - stiffeners - mobiele lasten - Eurocódigo 8 - Eurocódigo 5 - longitudinal reinorcement - doorlopende liggers - rigidizador - beton armé - fluage - CTE - connexion pied de poteau - langswapening - connexions - hormigón - neige et vent - elementos finitos -



Getting started

PowerConnect

armaduras - cold formed steel - jarret - uittekenen wapening - puente grúa - analyse dynamique - flambement - keerwanden - optimisation - steel - cercha - 2º orden - slab on grade foundation - entramado de vigas - Eurocode 5 - prédimensionnement - multi span beam - bouten - armatures - floor slab - poutre continue - pared - staal - 1er ordre - NEN 6770-6771 - connexion cisaillement - losa - déversement - viga continua - predimensionering - 1ste orde - unión metálica - CM 66 - madera - análisis resistencia al fuego - verbindingen - 2nd order - bois - Eurocode 2 - profilés formés à froid - verificación acero - predesign - unión soldada - fisuración - beton - muro de contención - optimalisatie - foundation pads - fissuration - concrete - AISC-LRFD - HCSS - assemblage métallique - Eurocode 3 - viga con varios apoyos - armaduras prácticas - balkenroosters - unión a cortante - buckling length - boulons - cracking - Eurocode 8 - knik - Eurocode 2 - radier - eindplaat - Eurocódigo 2 - FEM - tornillos - NEN 6720 - moving loads - balk op meerdere steunpunten - cargas - brandweerstandsanalyse - timber - 1st order - verstijvers - buisverbinding - diseño de planos de armaduras - pandeo lateral - losa

Table of Contents

1 Introduction

1.1 What does this manual has to offer?

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

This reference manual does not discuss the supported analysis methods and their theoretical background. A correct interpretation of the results provided by PowerConnect, which is essential for an efficient and successful use of the product, requires the user to be well informed on those analysis methods and their hypotheses.

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

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 minimizing 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.

1.2.1 The advantages of PowerConnect

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 **Diamonds** program for 3D frame analysis. The license 'Steel connection design' allows 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.

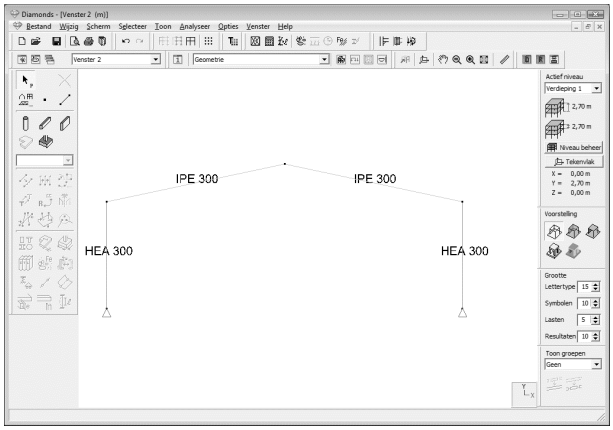
1.2.2 Design according to EUROCODE 3

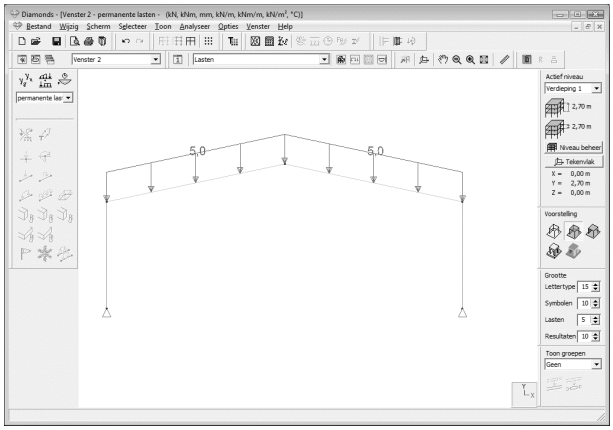
EN 1993-1-8: 2005 covers the design of structural joints connecting H- or I-sections and hollow structural section joints. The major advantage of this standard is that it has been based upon the so-called component method. This implies that every connection is analyzed such that all composing elements 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

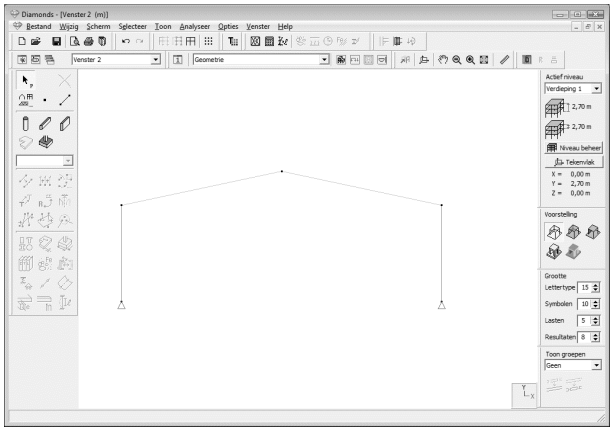
subsequently analysed in Diamonds using rigid and semi-rigid nodes. The impact of nodal stiffness will be evident from the example.

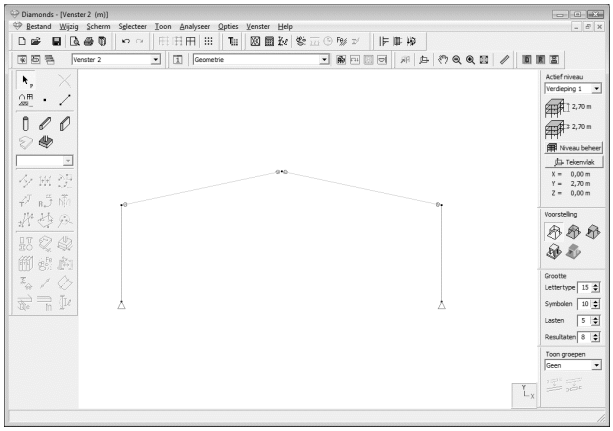
Consider the following example:

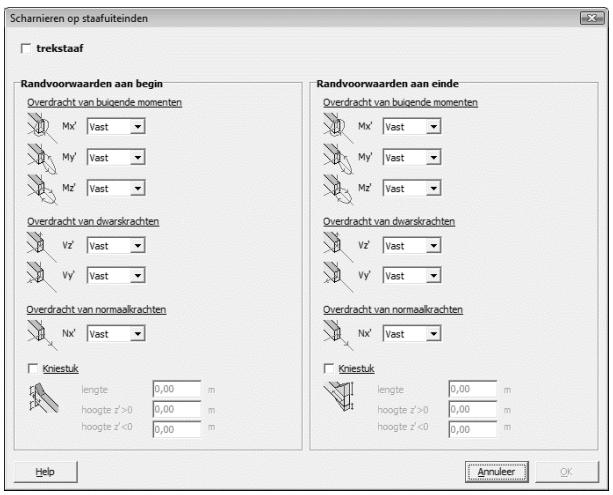


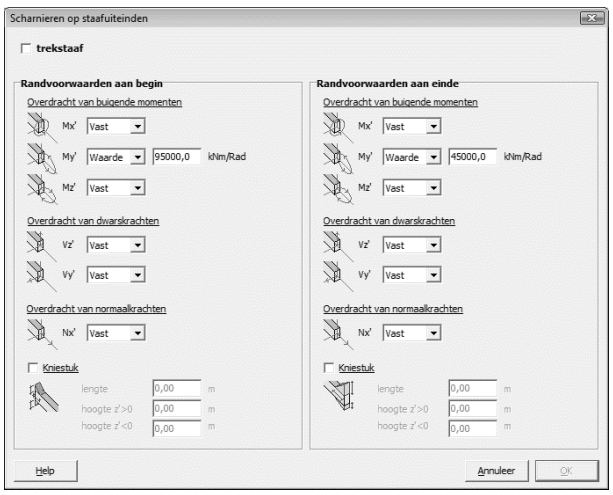


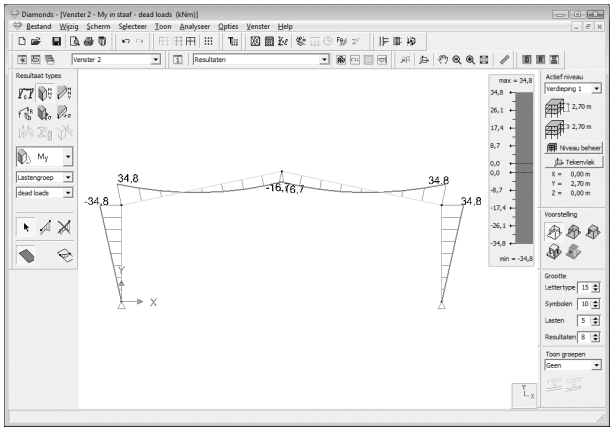
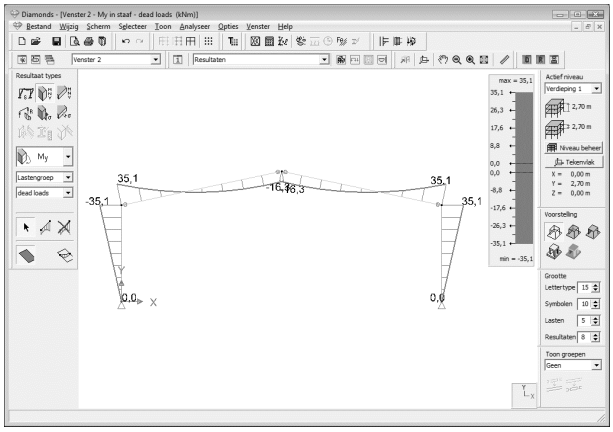
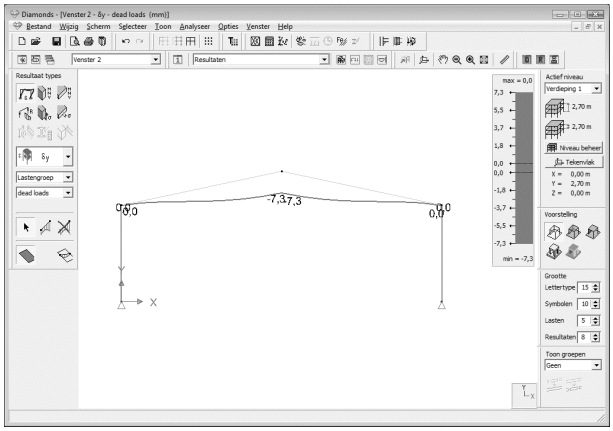
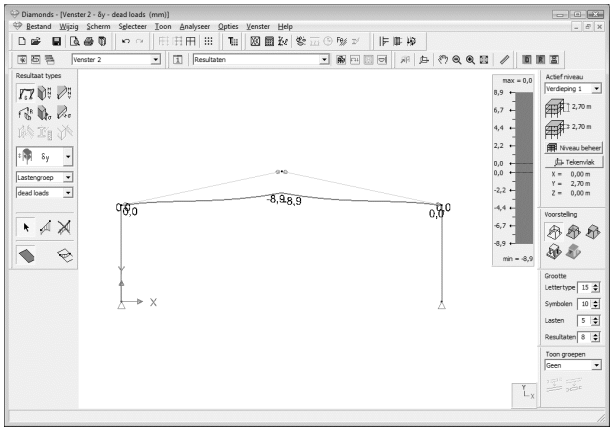
Scenario 1	Scenario 2
Calculation of bending moments and vertical displacements for a 3D frame with perfectly rigid nodes	Calculation of bending moments and vertical displacements for a 3D frame with semi-rigid nodes









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.

1.3 Preliminary information

Before going further, you need to be familiar with elementary commands of your MS Windows *operating system* and with the use of windows and icons, selected features and using a mouse. An overview:

Icon	Graphic representation of a program or part of a program.
Clicking with the mouse	Pointing a given element or given area and clicking 1 time with the mouse button.
Selection	Clicking once on an icon or element. You may also select several elements with the use of a grid: click the top left corner of a rectangle which is to comprise the chosen selection – hold the mouse button and drag it towards the opposite right corner, then release the mouse button. You may enlarge the selection by following the above procedure with pressed SHIFT key.
Double-clicking	Quickly click your mouse 2 times. This feature is used to start a program or a part of a program.
Dragging	Drag a given element by selecting it and moving the mouse while keeping the button pressed.

1.4 Overview of the shortcuts

A number of shortcut keys enable to work quicker and more efficiently in PowerConnect. Below you will find a list of shortcuts available for the most common features:

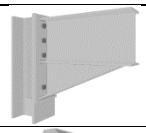
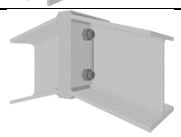
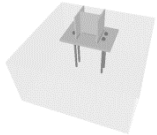
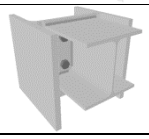
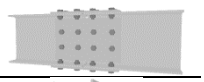
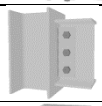
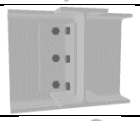
– CTRL + N	New file
– CTRL + O	Open file
– CTRL + P	Print file
– CTRL + Q	Close PowerConnect
– CTRL + S	Save file
– CTRL + Z	Undo
– SHIFT + CTRL + Z	Redo
– F1	Open PowerConnect Help
– F9	Elastic analysis
– F10	Maximize
– F11	Minimize
– F12	Show all
– SCROLL	• Push the wheel and move the mouse: drag (Pan)Scrollen: in/uitzoomen • Scroll: zoom in or out
– SHIFT + SCROLL pushed in	Rotate (3D orbit)

2 PowerConnect tutorials

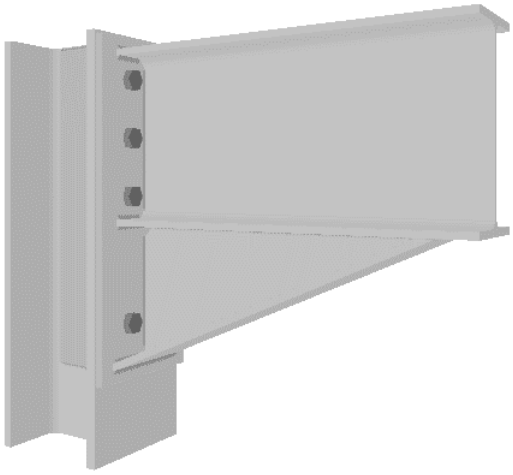
The best way to get acquainted with the PowerConnect software is to explore the product's functionality through a number of examples which highlight the various connection types supported by PowerConnect.

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:

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

2.1 Tutorial 1: bolted beam to column flange



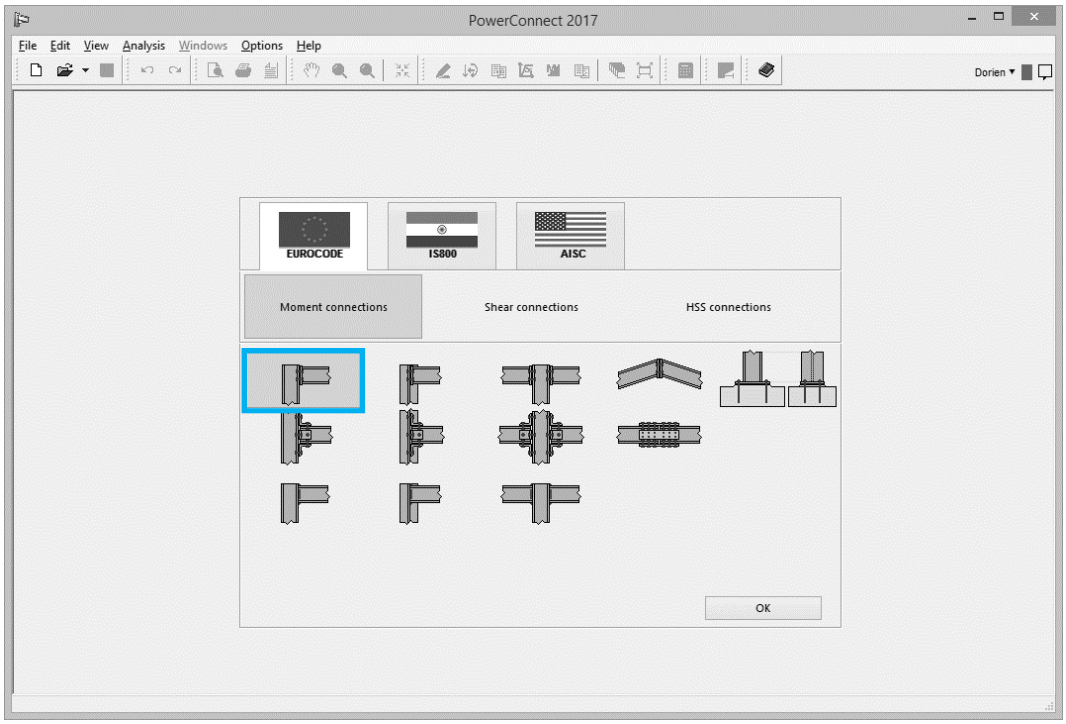
2.1.1 Setting up the model

Launch PowerConnect. In the navigation window appears. In this window you select the desired standard and the type connection type.

If you have already an open project, then select “File – New” or the  icon.

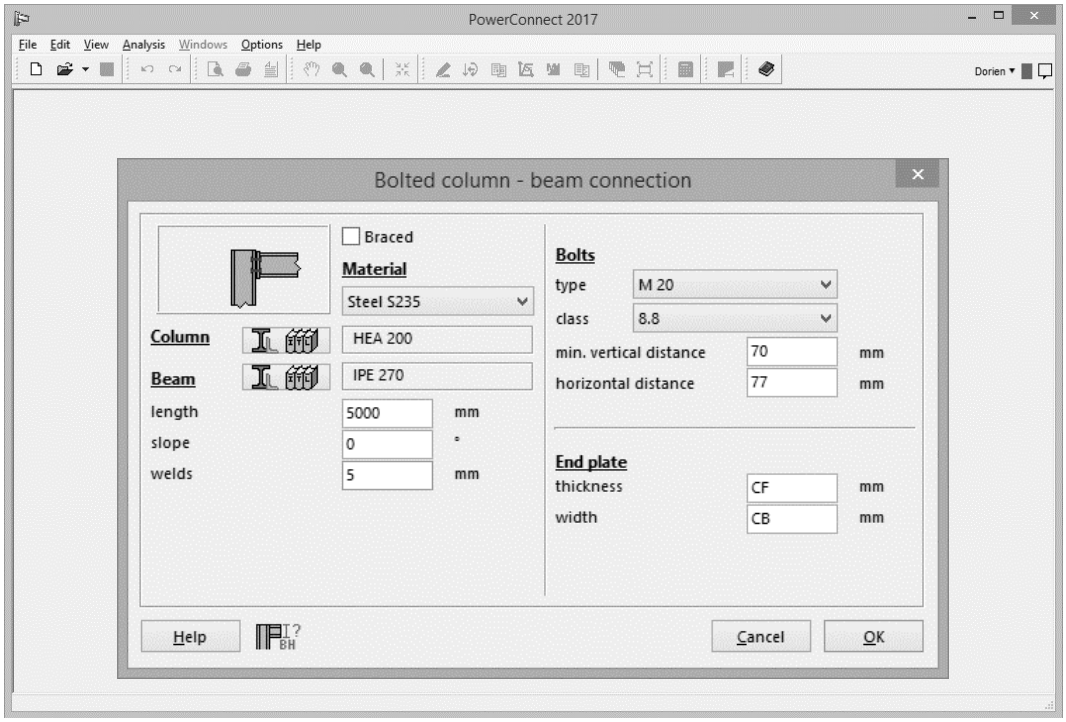
Select the standard ‘Eurocode’.

Among the available moment connection types, choose the one labelled “Single-sided beam to column flange”. The labelling will appear when you move the mouse over the available connections.



Confirm your choice using the ‘OK’-button.

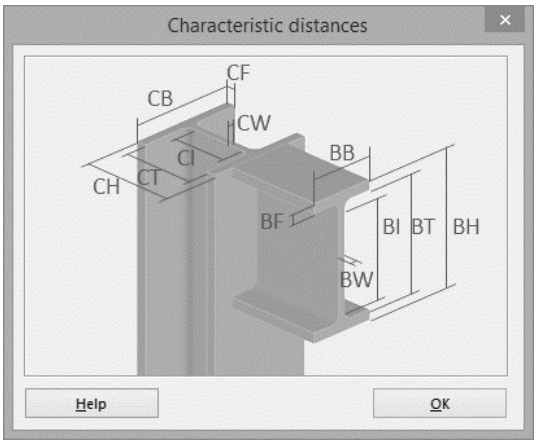
In the dialog box that appears complete the definition of the model.



Beam and column sections are modified as follows:

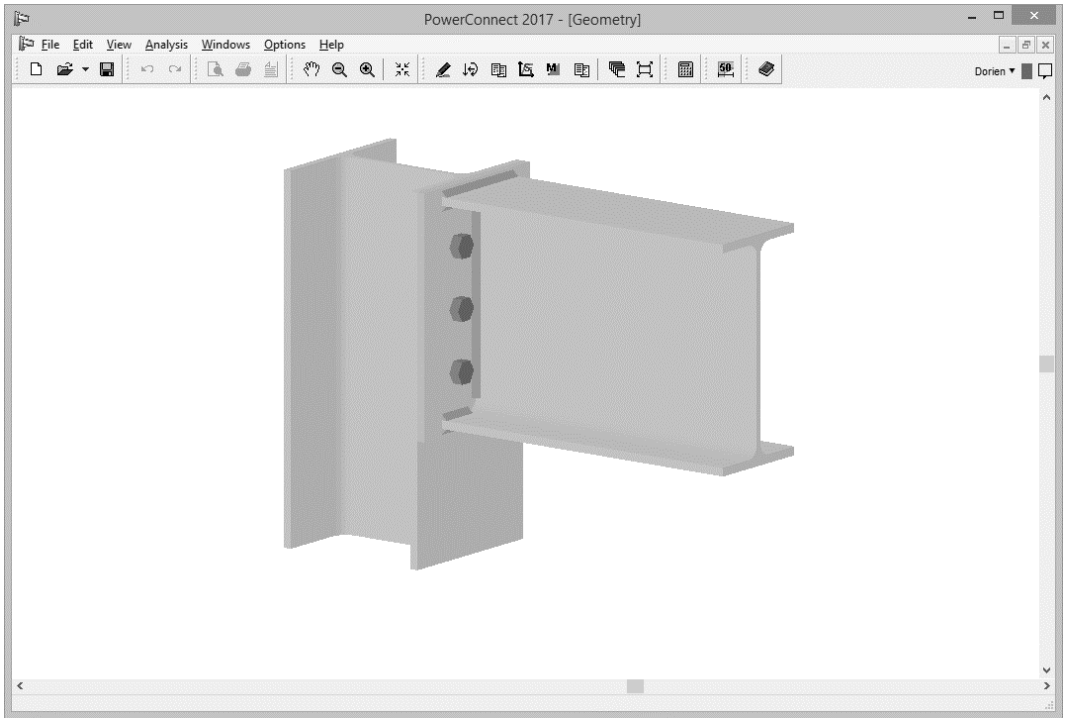
- HEA 200 for the column member,
- IPE 270 for the beam member.

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 option, 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.

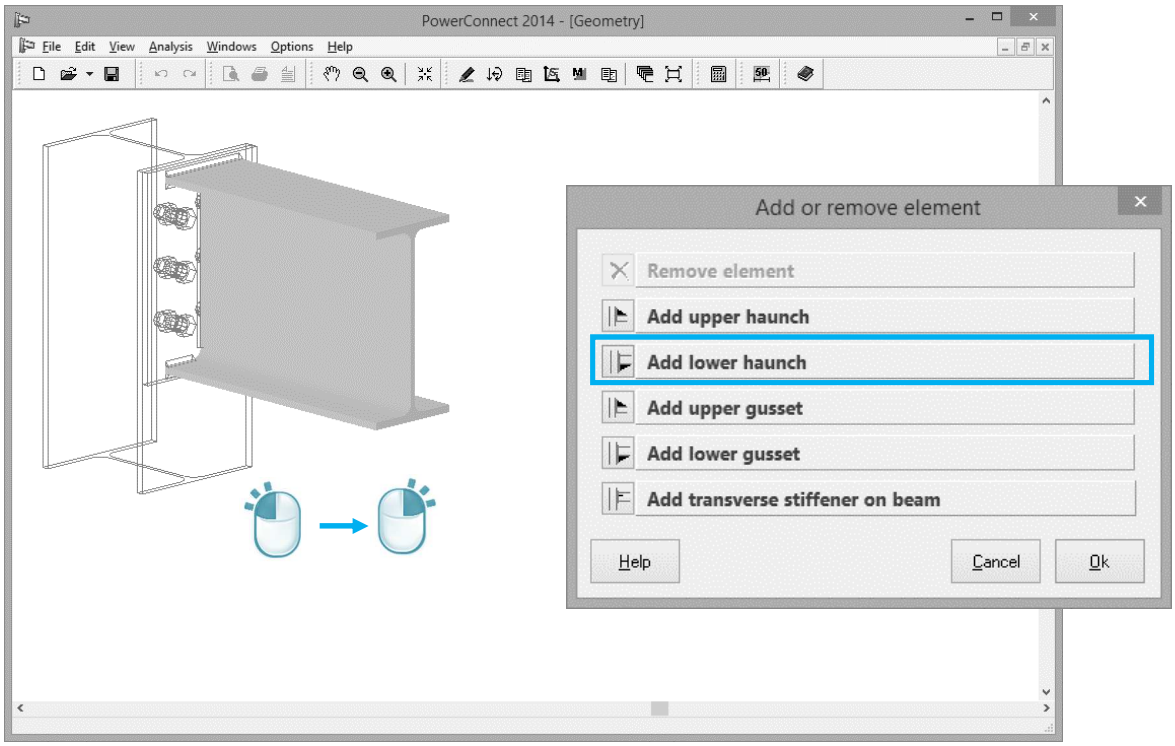


Note**: if you don't see the figure, close PowerConnect and follow the steps in §Fout! Verwijzingsbron niet gevonden.**.*

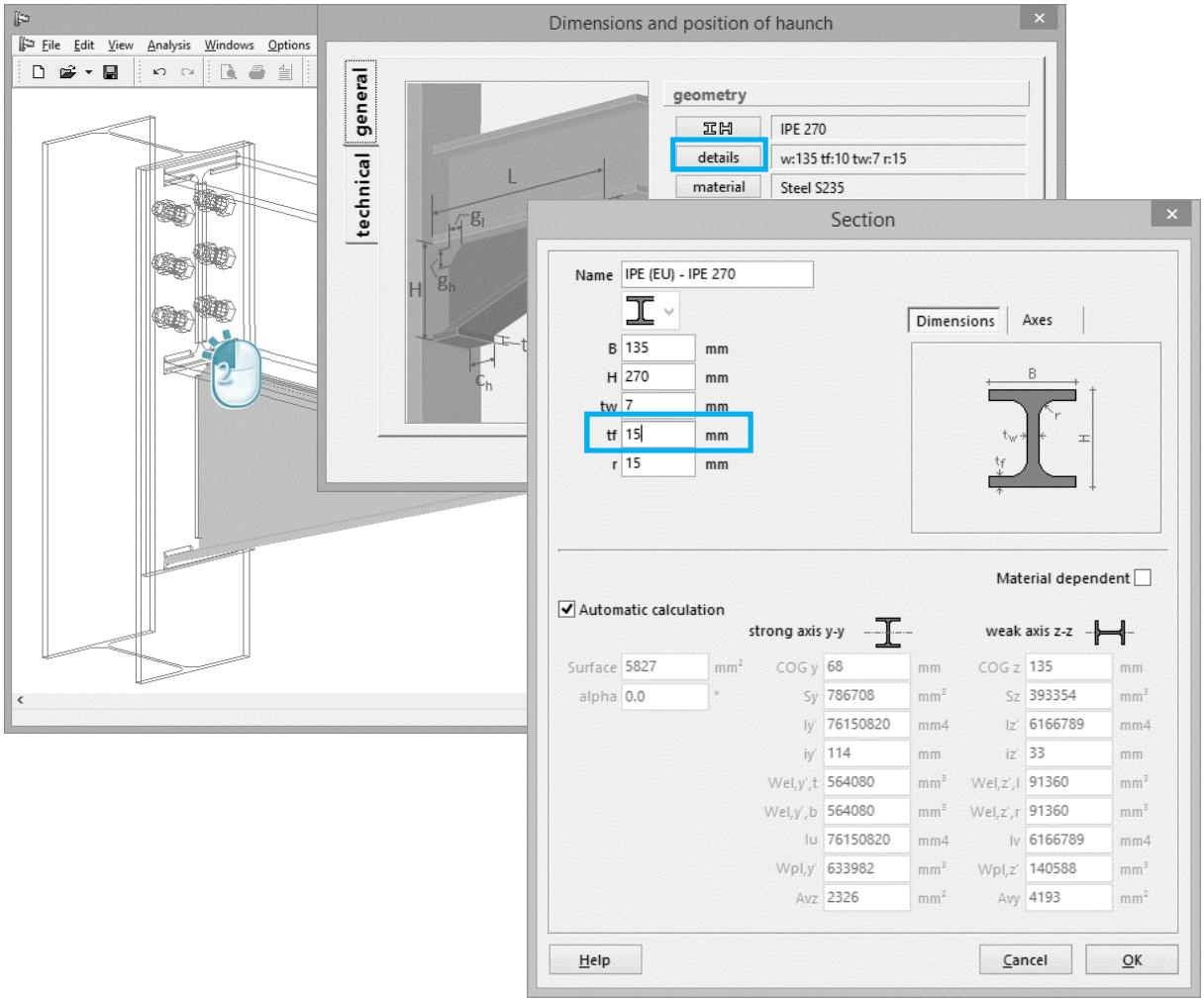
2.1.2 Completing the geometry model

The scale which is used for model representation can be modified using the icons  and  or by using the scrollbar of the mouse. The geometry model can furthermore be moved across the screen using the icon  or by holding down the left-hand mouse button while moving the mouse across the 'Geometry' window.

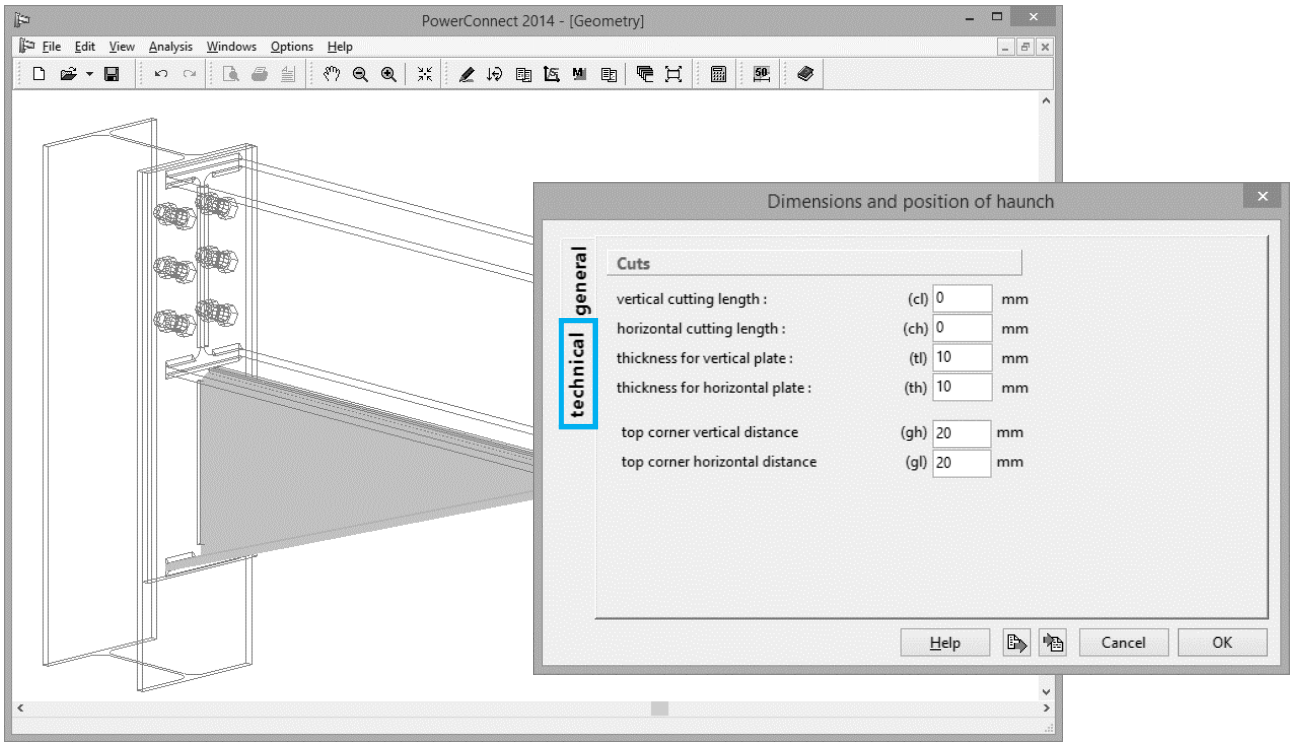
Now add a haunch under the beam. Select the beam and click once with the right mouse button on the beam. You'll see the following window:



Now double click the haunch with the left mouse button to see/adjust its properties.

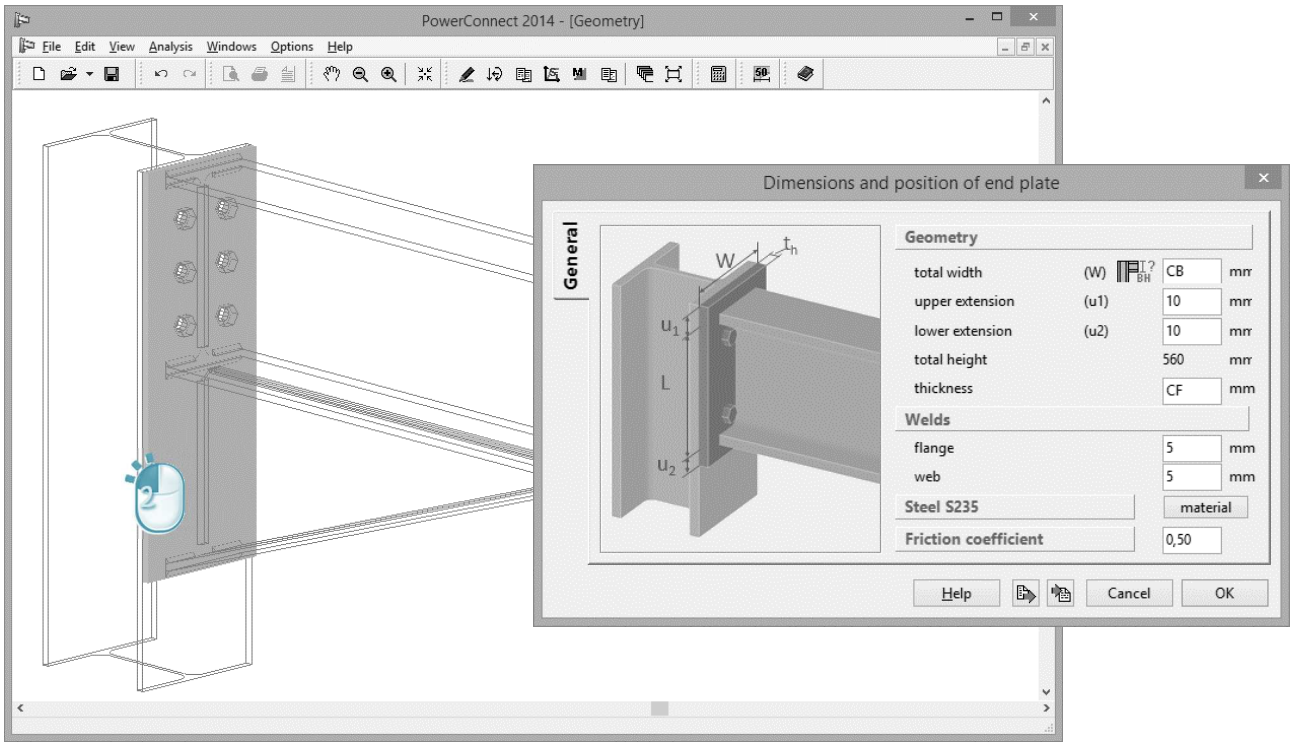


The haunch is based on a IPE270 section, which is the default section used for the beam. If needed, the haunch can be based on a different section defined through the icon  on the ‘general’ tab page. In the tab page ‘Technical’ you can enter the dimensions of any cut outs.

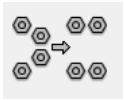


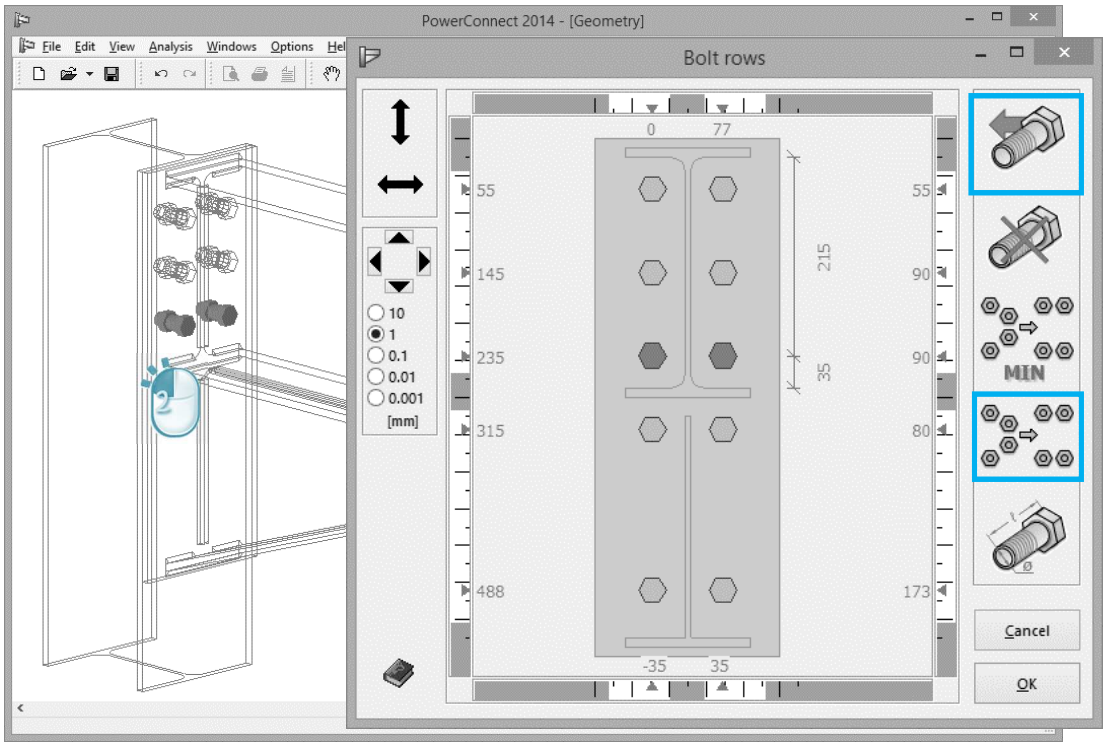
Click ‘OK’ to close this window.

We verify the dimensions and the position of the end plate. 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.



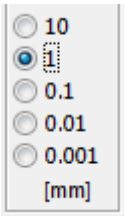
Click ‘OK’ to close this window.

Now double click on the bolts. Add 2 bolt rows using the button  and optimize their position  .



If you wish the reposition the bolts manually, follow these steps:

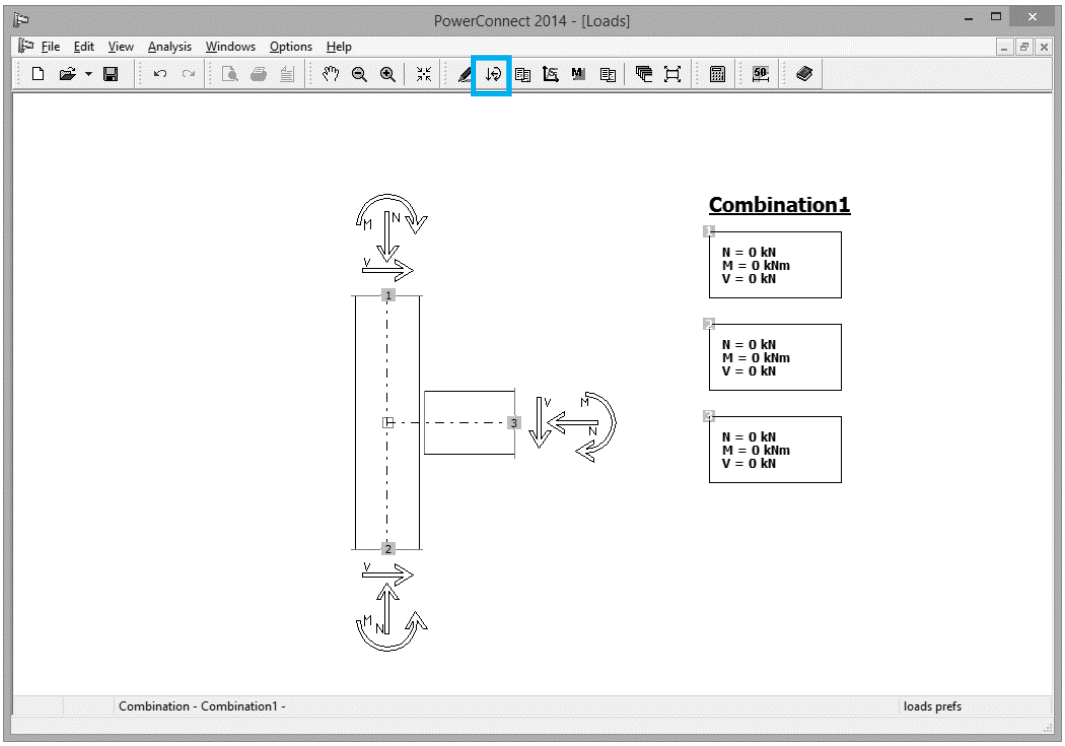
- Select the bolt row you want to reposition
- Activate the function for vertical displacement  and move the row to its desired position.
- or, use the arrows  to move the selected bolt row and this according to the selected accuracy.



PowerConnect will automatically verify the position and alignment of individual bolts each time you calculate the connection. 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.

2.1.3 Defining the loads

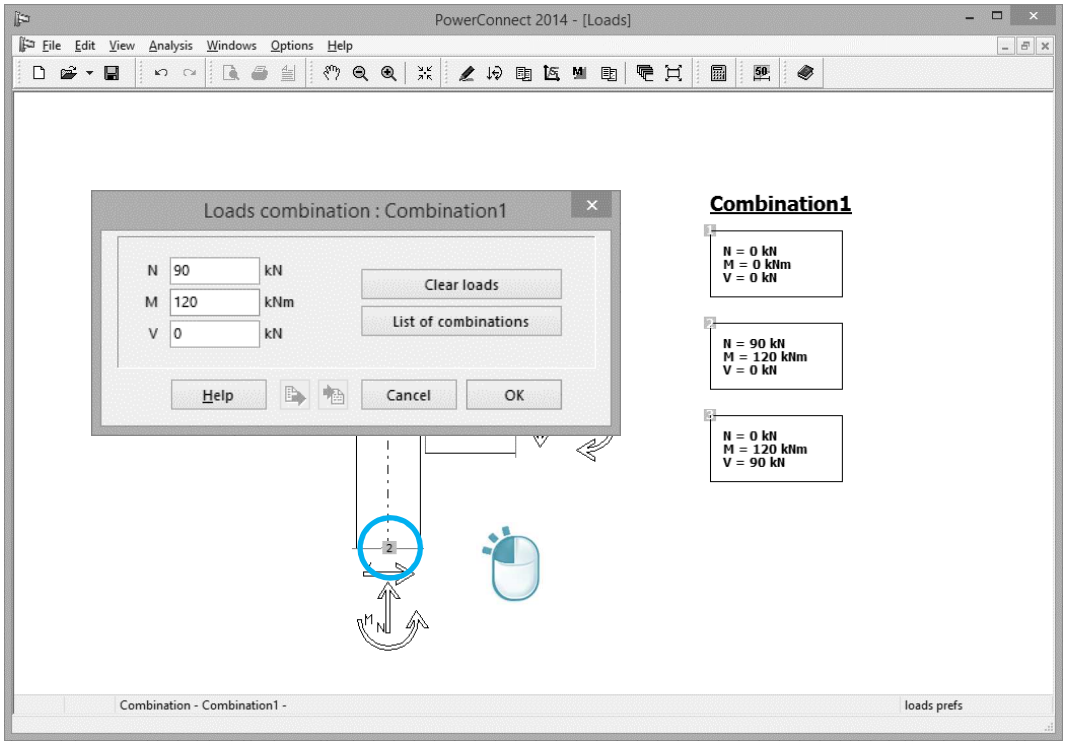
Now that the basic geometry definition has been completed, loads can be assigned to the connection. Switch to the “Loads” window by clicking on the  -icon in the toolbar. The following information will appear in the ‘Loads’-window.



Following loads should be applied:

- at the lower end of the column, a bending moment of 120kNm and a normal force of 90kN. 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 120kNm and a shear force of 90kN. 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:

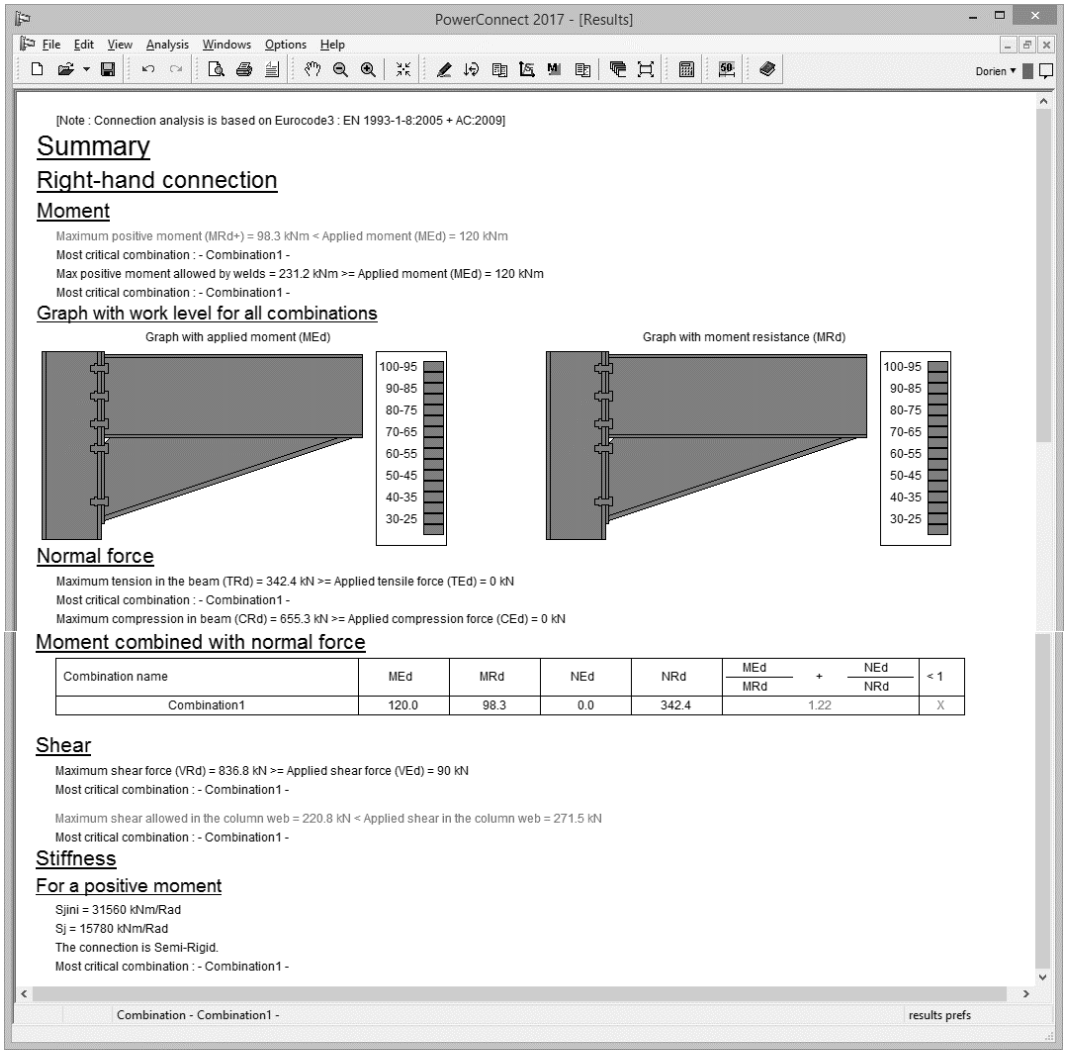


2.1.4 Running a first connection design analysis

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

- through the menu command ‘Analysis – Analysis’
- through the shortkey F9
- through the  icon in the Analysis toolbar.

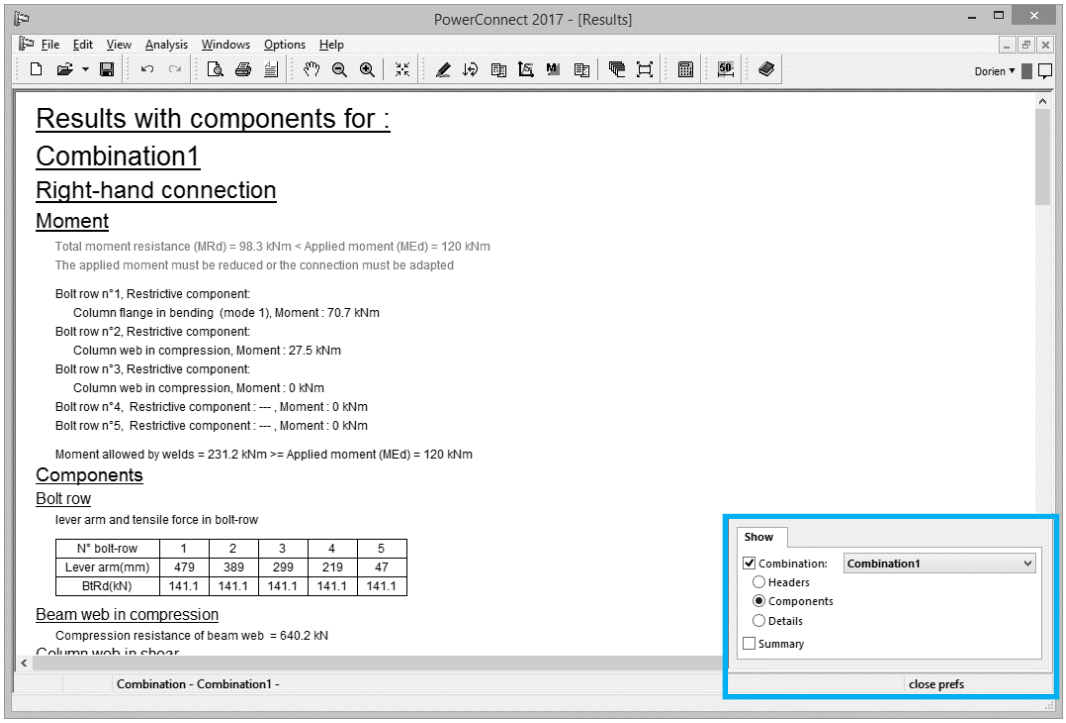
The results of the design analysis are presented below.



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

- From the results available for **moments** only, it can be derived that the welds are OK. The resistant moment is bigger than the applied moment.
- The **work level graph** (valid for **moments** only) shows that the column web, column flange on the right and the end plate are loaded up to their maximum capacity.
- The current connection design doesn't fulfil the imposed strength requirements, as the ratio of applied loading to resistance achieves a value of 1,22 (looking at the combination of **bending moment and normal force**).
- From the results available for **shear** only, it can be concluded that the connection does not provide sufficient shear strength.

For a more detailed interpretation of analysis results, it is possible to switch to a more advanced reporting lay-out. 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.

This will provide the information shown above. 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:

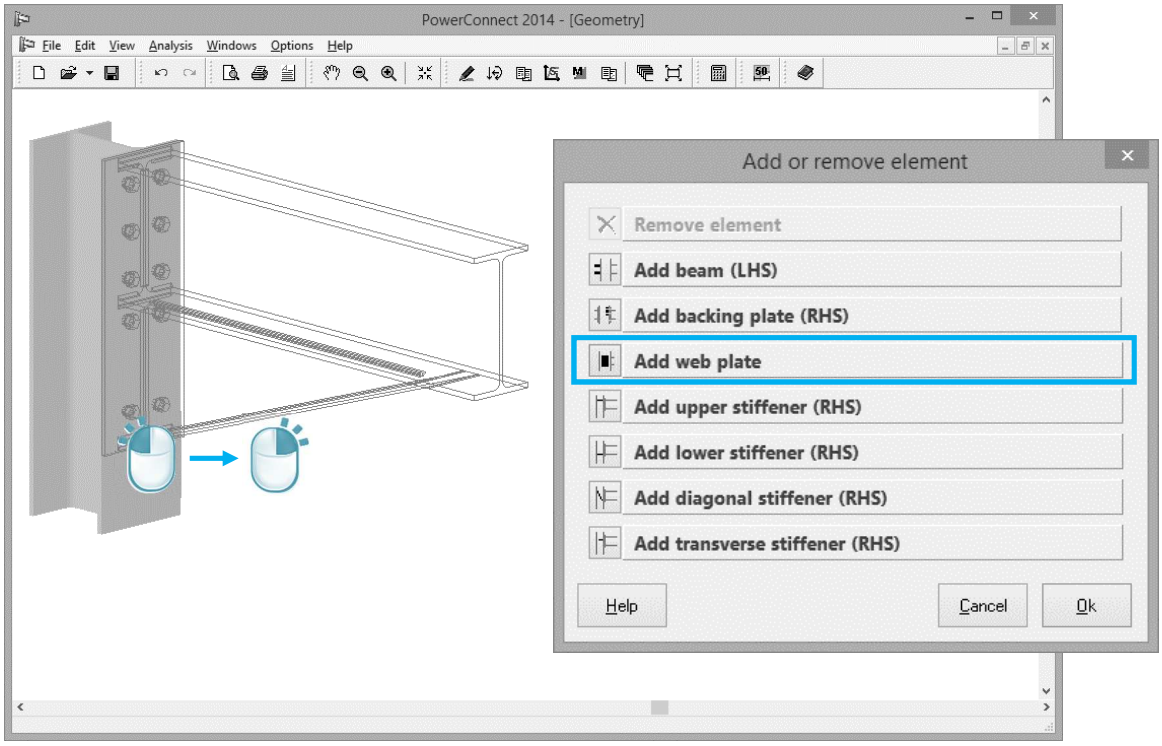
- only bolt rows 1 & 2 contribute to the connection's resistance in bending
- the most critical (limiting) components of the connection are
    - the column flange end plate (failure in bending)
    - the column web (shear failure)

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.

2.1.5 Optimizing the connection design

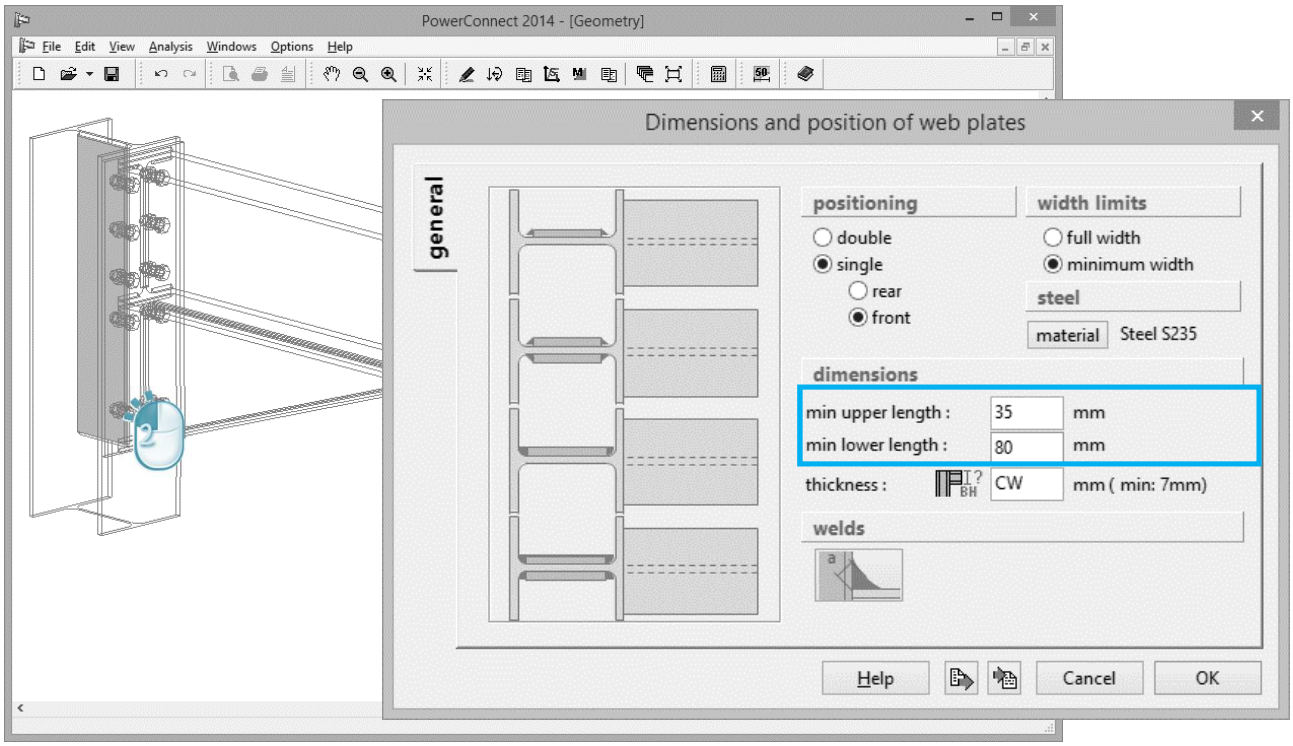
Based on the results provided by the first design analysis, a number of modifications will now be defined to the existing connection such that its resistance is increased by focusing on the most critical component information.

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 mouse button to make the following window appear on the screen.



Choose the option “Add web plate”, and confirm by clicking ‘OK’-button.

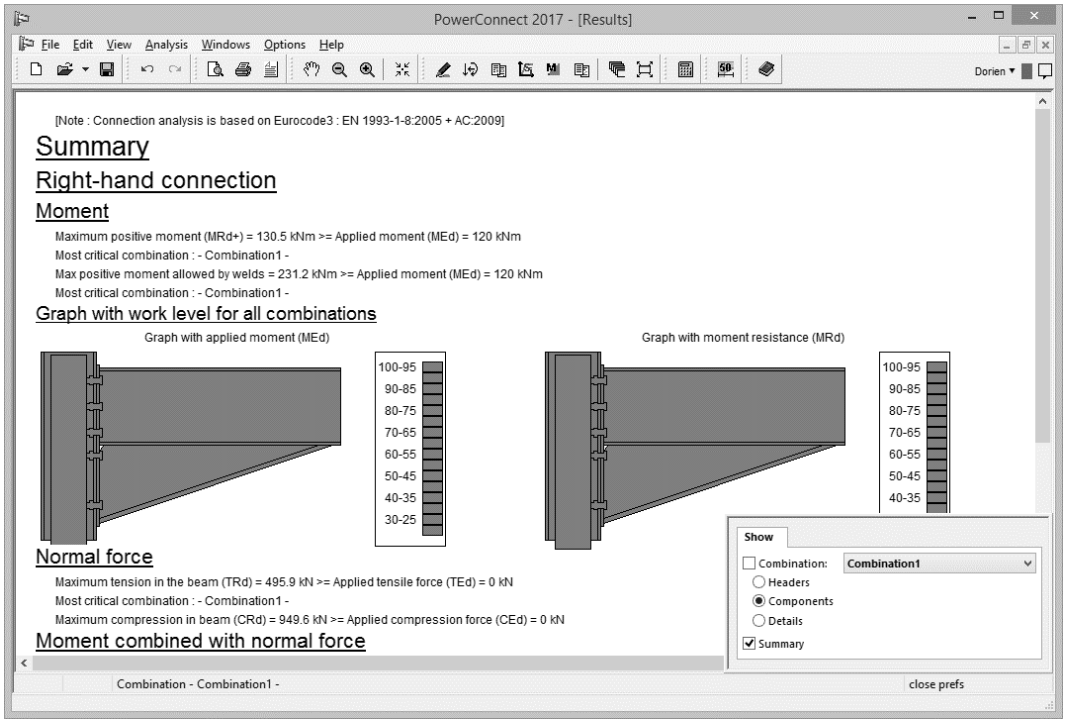
In order to verify and to possibly modify the web plate dimensions, double-click on the web plate that is now visible on the geometry model. A new window will appear providing full access to all geometry details of the web plate:



Make sure all parameters match with the ones presented above.

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

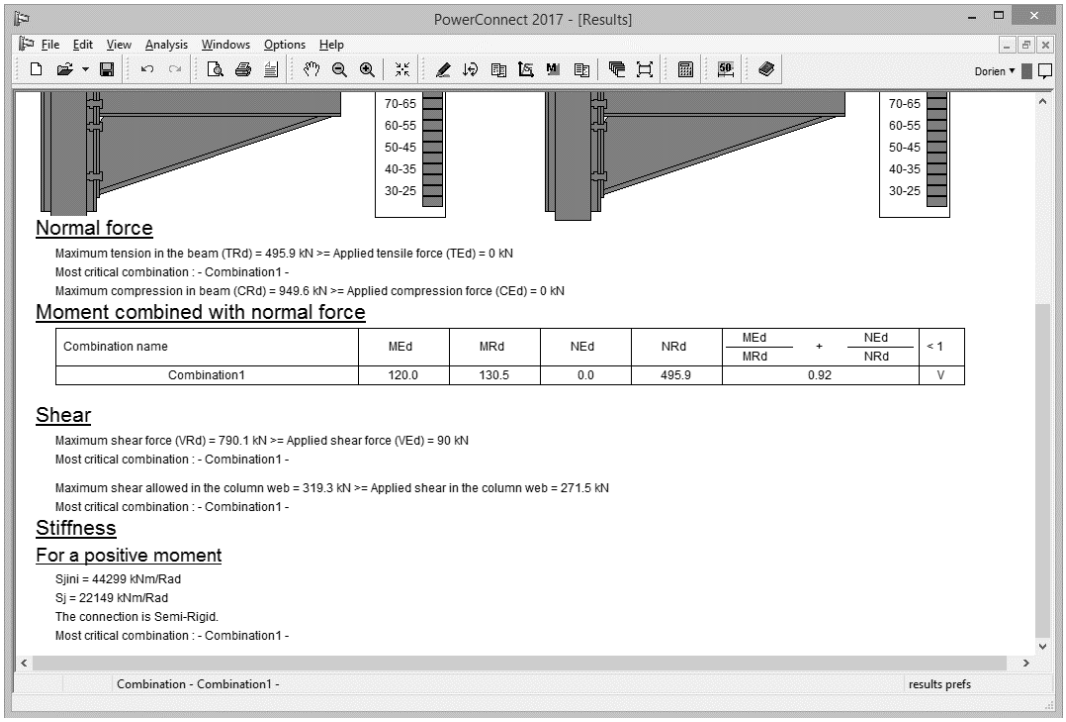
Running the connection design analysis will now produce following summary results:



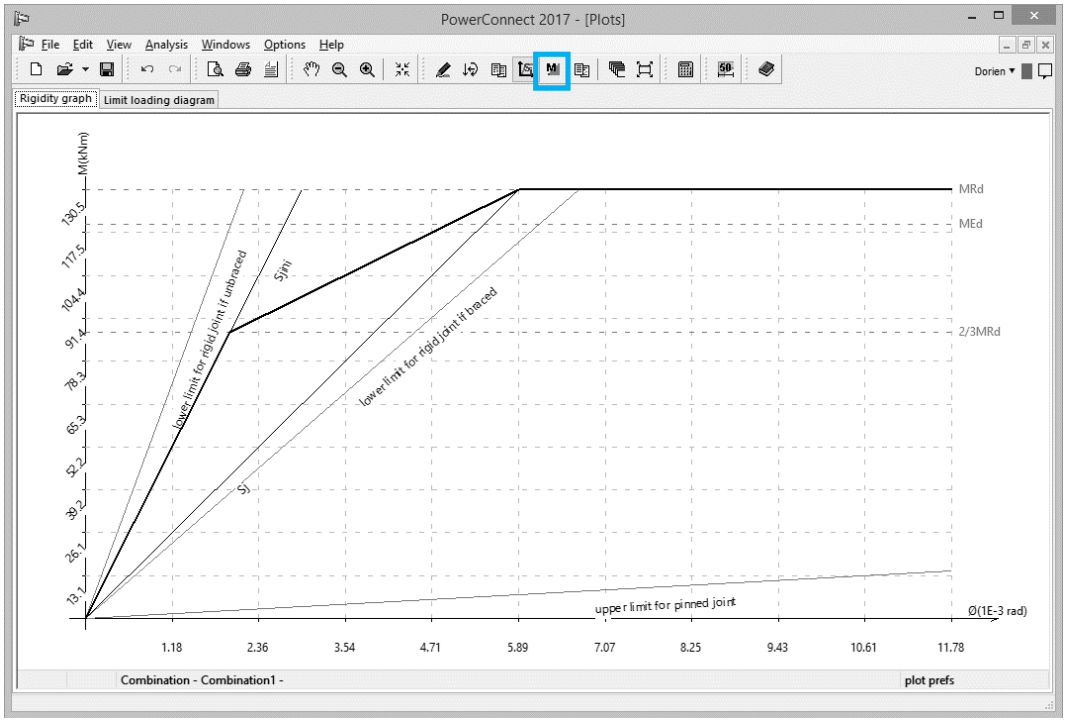
Just like before, it remains possible to look at more detailed analysis results by clicking with the mouse on the field ‘Results preferences” at the right bottom of the PowerConnect window and by asking for the appropriate detail level for results reporting. Choose for ‘Summary’.

The results above confirm that all strength requirements are met by the connection, after the described changes have been introduced into the geometry model.

Use the scrollbar on the right-hand side of the PowerConnect window to go to the end of the summary report (as shown below).



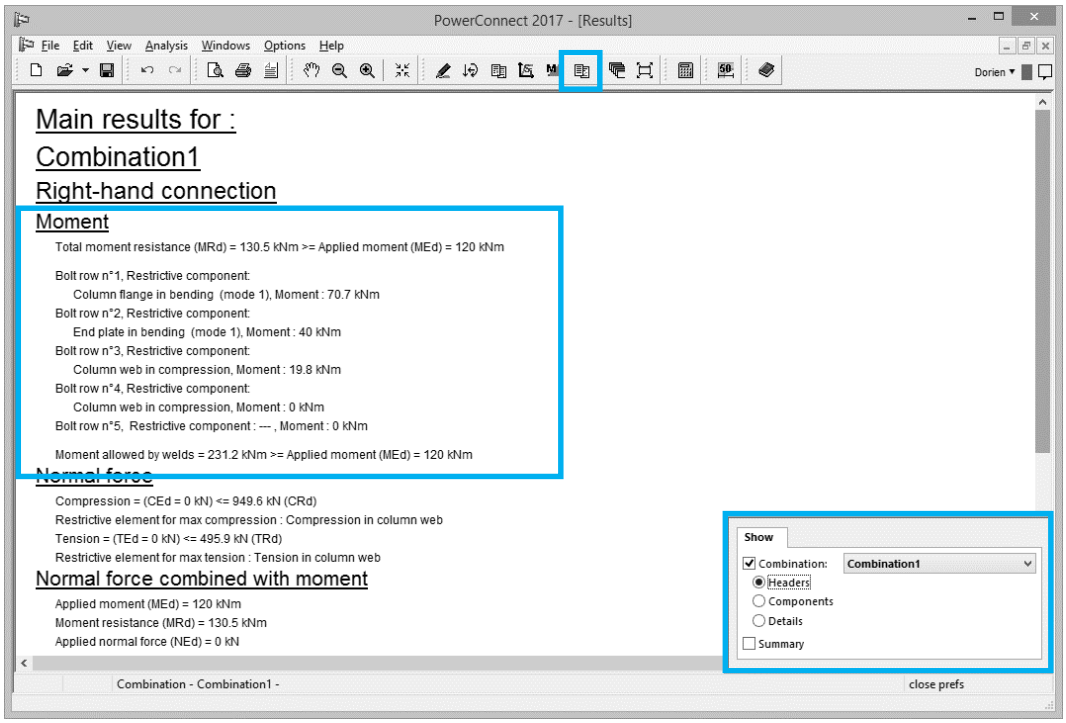
It will now be visible that the PowerConnect design analysis has not only calculated strength characteristics, but stiffness characteristics as well. Those stiffness results can also be represented graphically by means of the  icon of the 'Windows' toolbar:



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 column flange and the end plate Changing one of those elements will have a significant impact on connection resistance M_{Rd}.



On the other hand, the 4th and the 5th bolt row have no contribution at all to connection resistance. Removing those bolt rows will not decrease strength characteristics (except for shear loading).

Remark: 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.

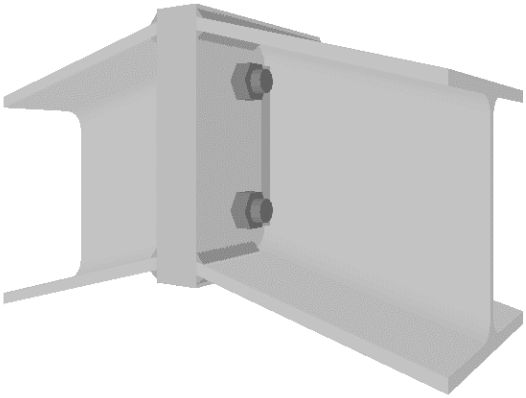
In the example that was considered, shear resistance is significantly higher than shear loading. In case shear loading would have been much higher, more or stronger bolts would be needed. If on the other the problem would be related to the column web, one or two (one on each side) web stiffening plates 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:

$$\frac{N_{Ed}}{N_{Rd}} + \frac{M_{Ed}}{M_{Rd}} < 1$$

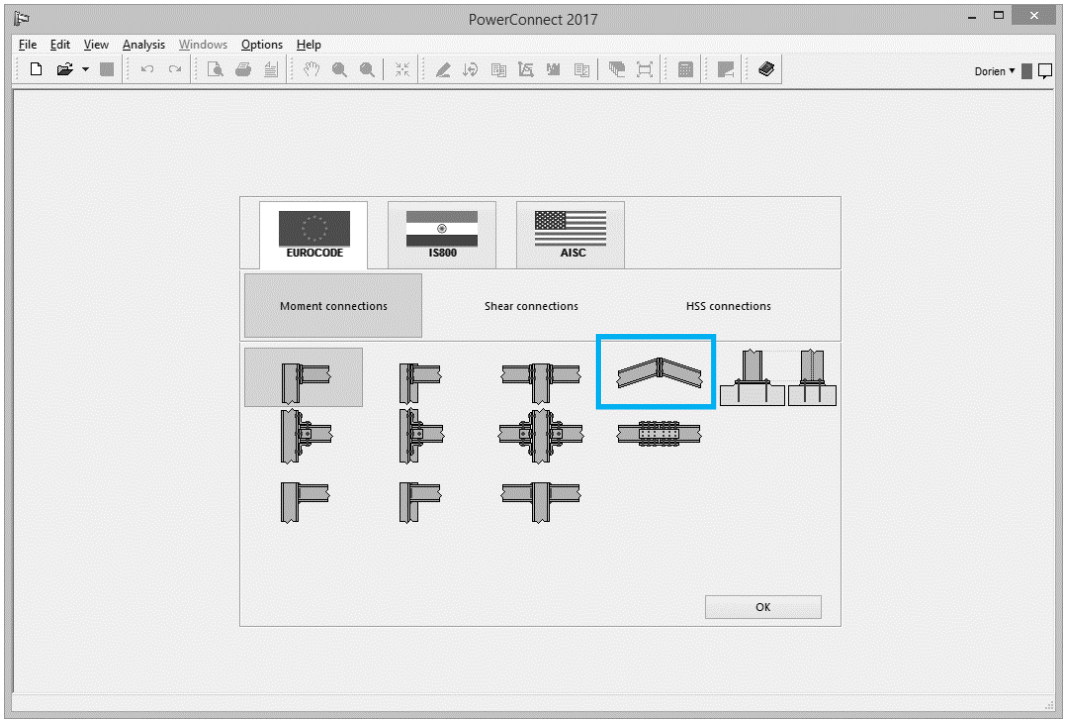
In §3.1 you can read how to make de report for this connection.

2.2 Tutorial 2: bolted beam to beam with moment end plate



2.2.1 Setting up the model

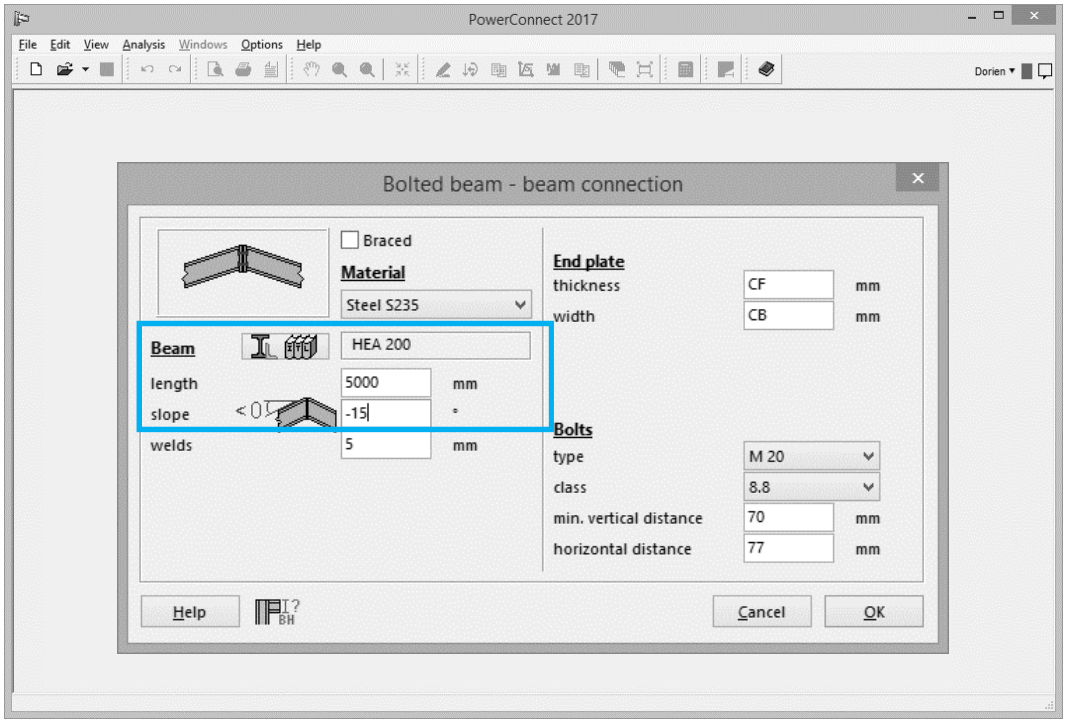
To define the model for this second tutorial, click on the 'New' icon of the icon toolbar. Among the available connection types in the navigation window, select the one labelled "Beam to beam" (remember: the labelling is done through the use of tooltips that appear when you move the mouse over the available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



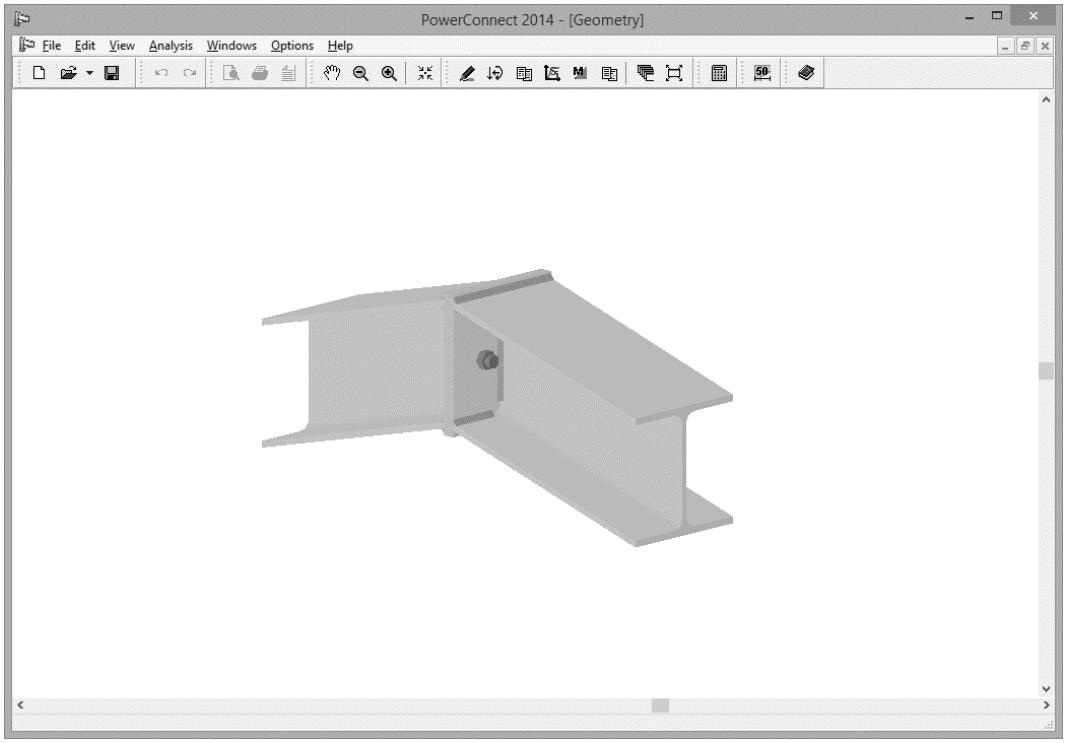
Confirm this final choice using the 'OK'-button.

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

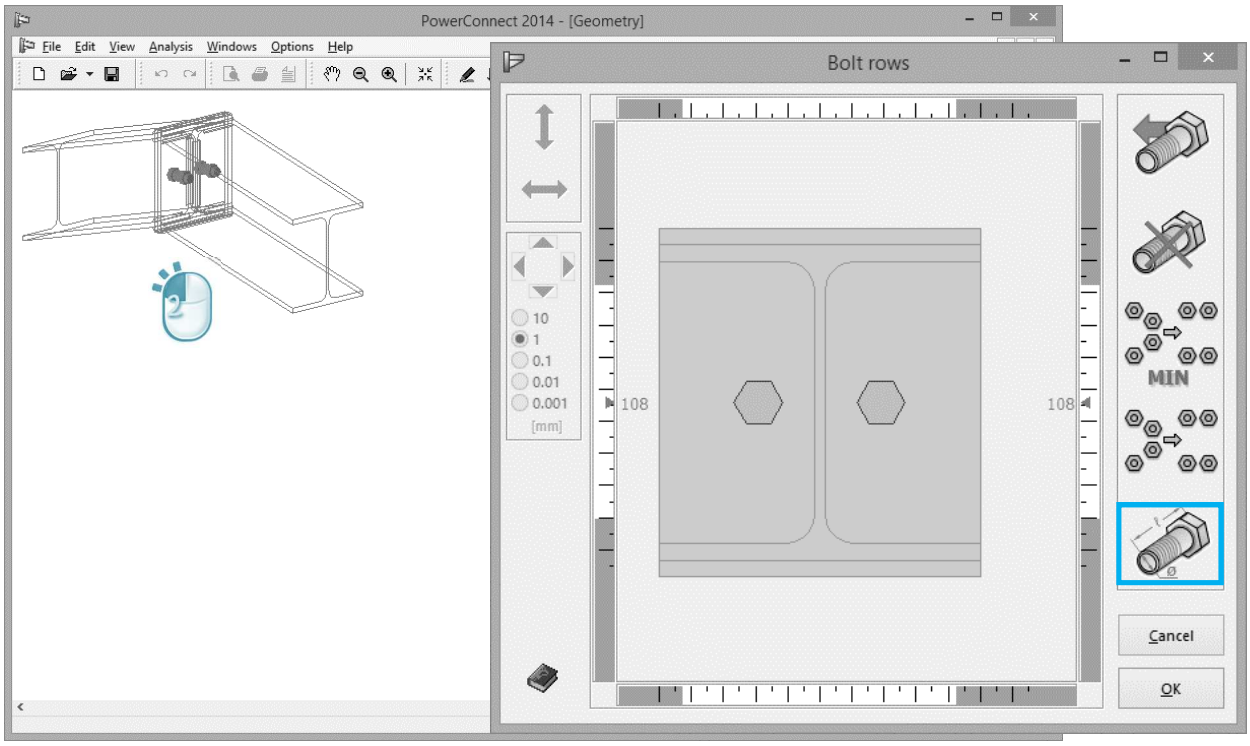
- beam section : HEA 200
- beam length : 3000 mm
- slope of beam : -15°



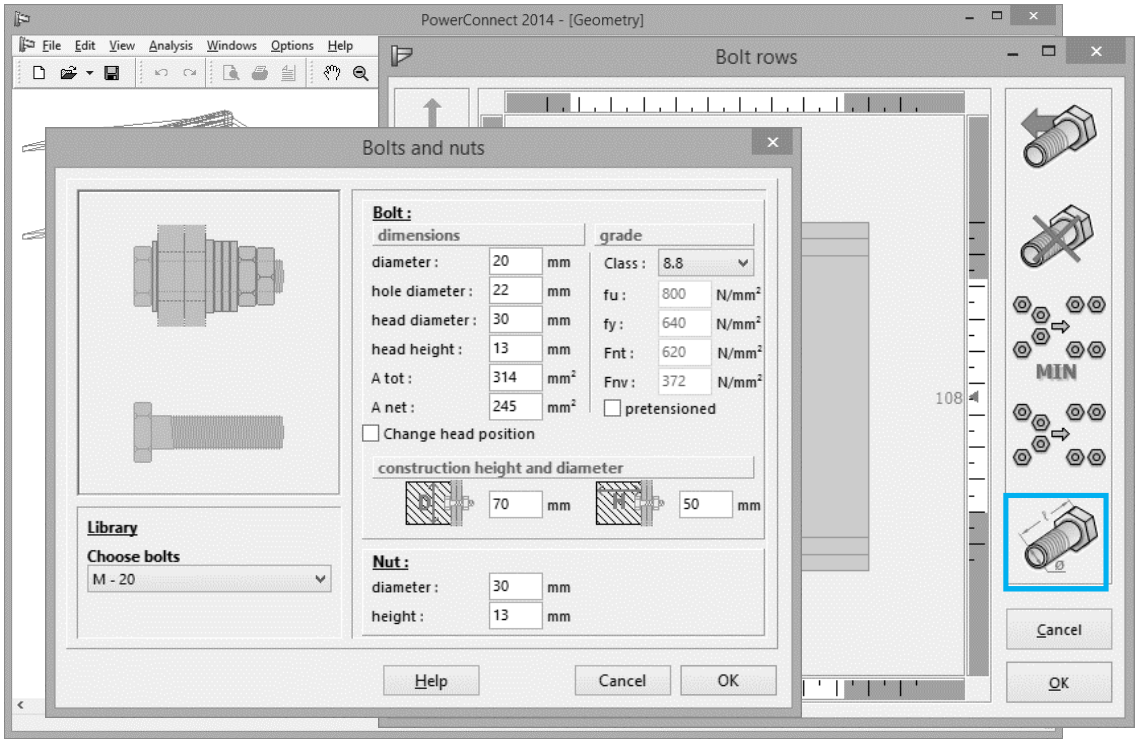
Confirm with the ‘OK’-button to make following geometry model appear.



We opt to use 2 bolt rows instead of 1. Double-click on one of the bolts to make the bolt definition dialogue appear.

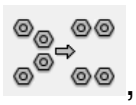


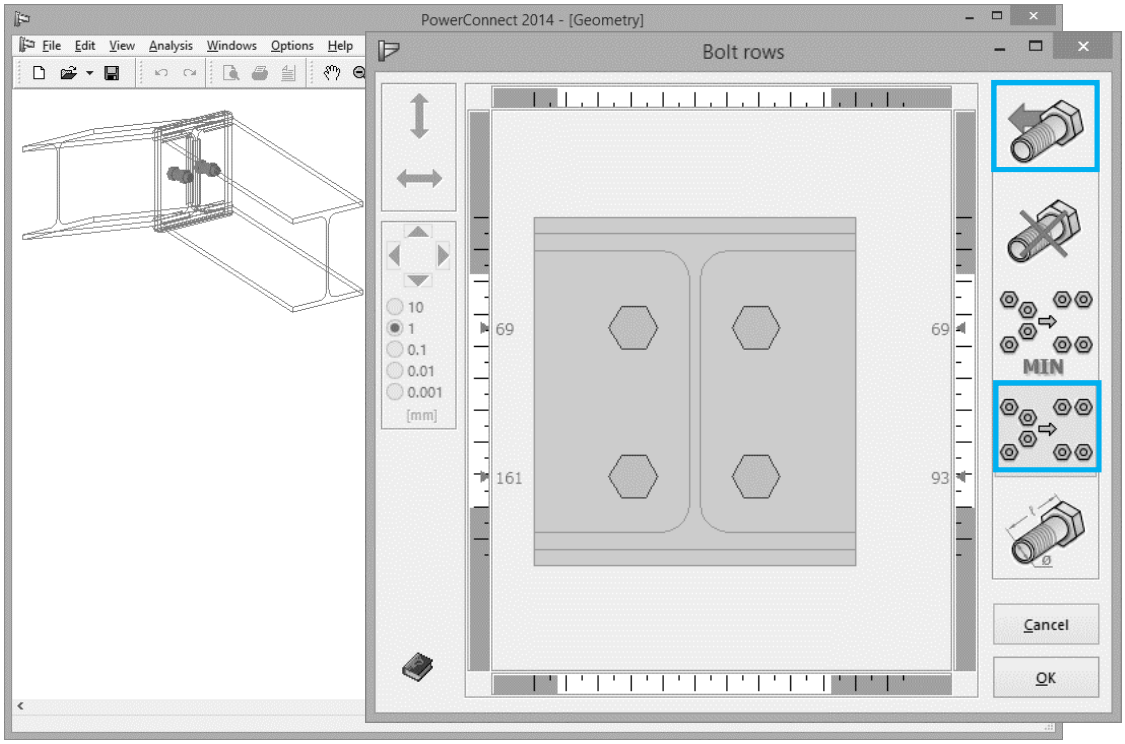
Use the icon ‘Bolt details’  of this dialogue window to gain access to the more detailed bolts specification. On the left hand side of the dialogue window, a bolt type can be selected from the bolts library. If needed, individual parameters as net surface, free spacing,… can still be adapted.



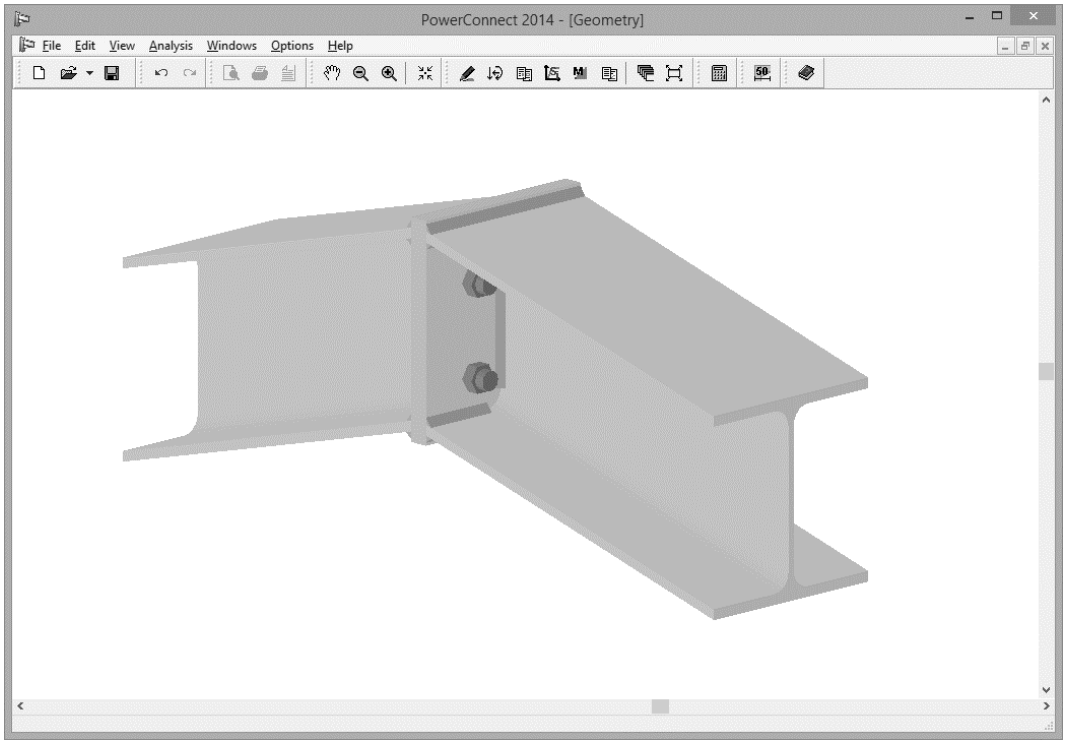
Accept the default values and click ‘OK’.

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:

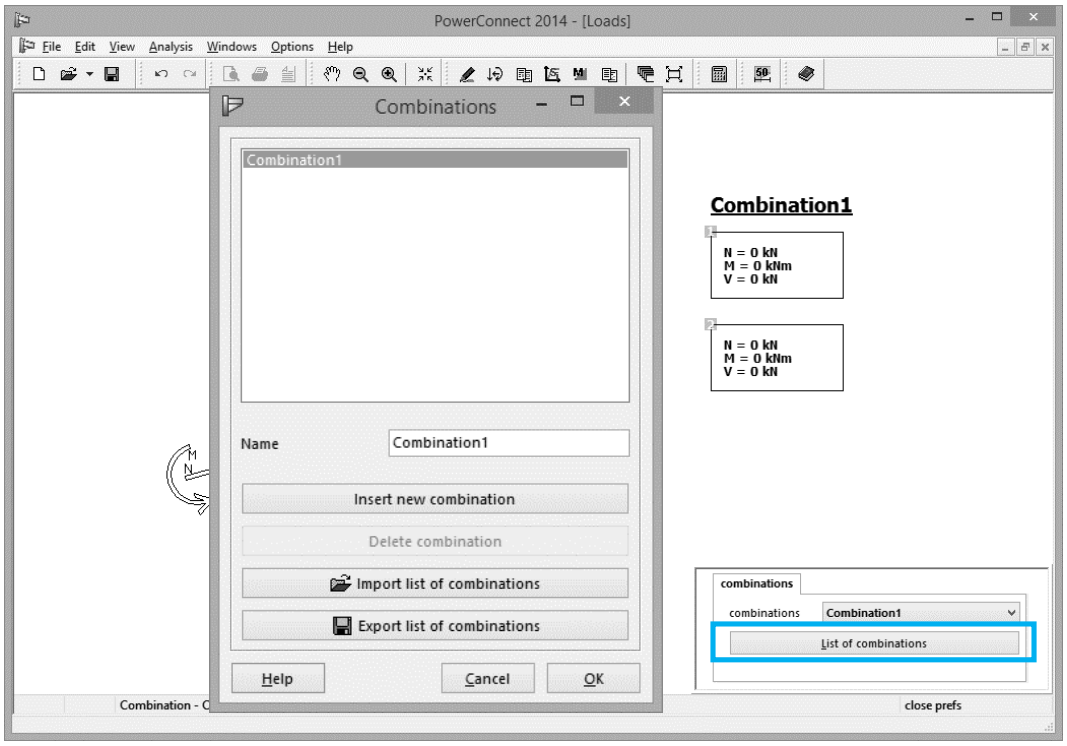


2.2.2 Defining the loads

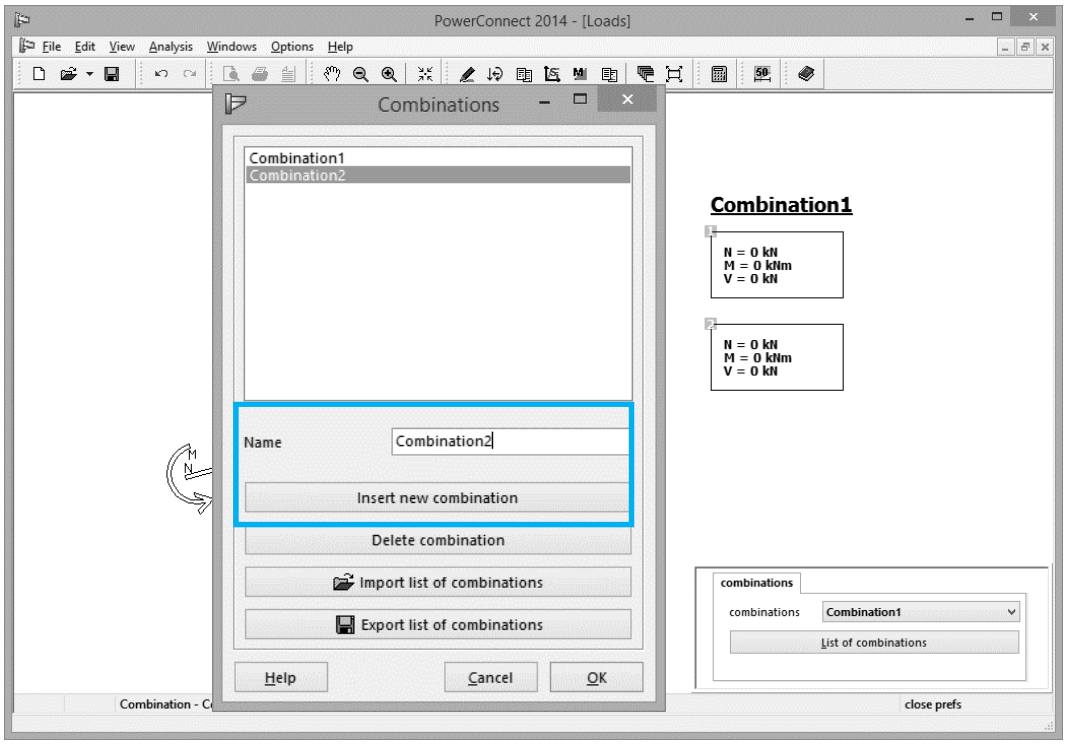
Now switch to the 'Loads'-window by means of the  icon. 2 loads combinations will be defined:

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

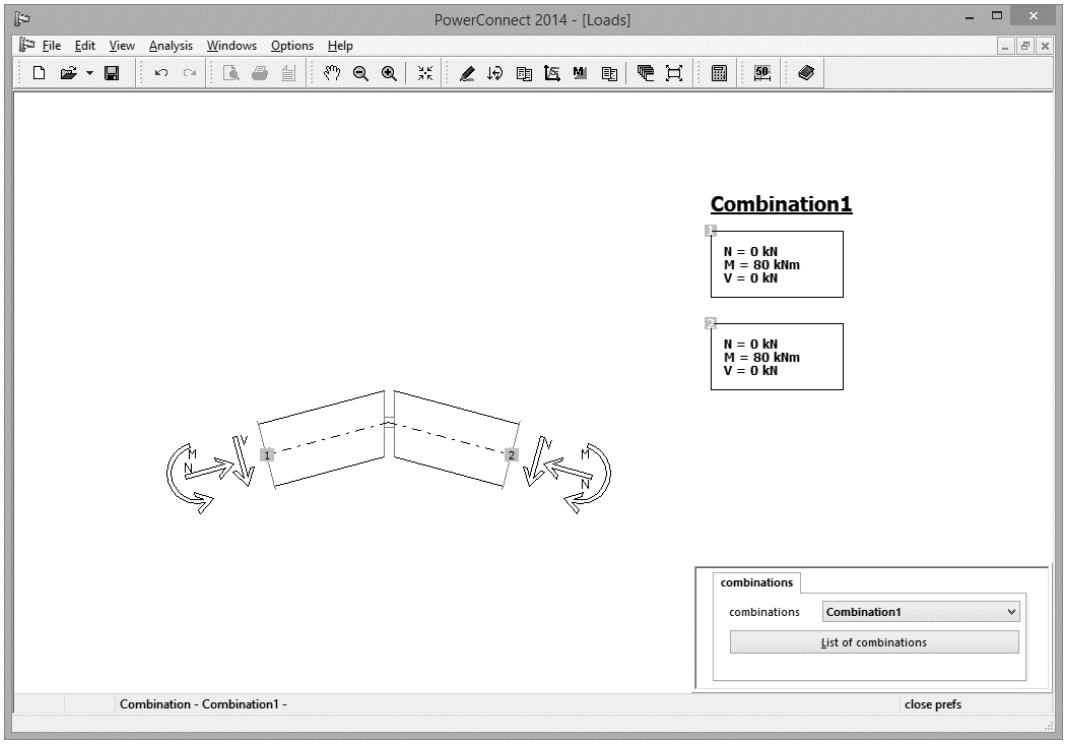


Use  and specify the name “Combination2” for the new combination that has just been created.

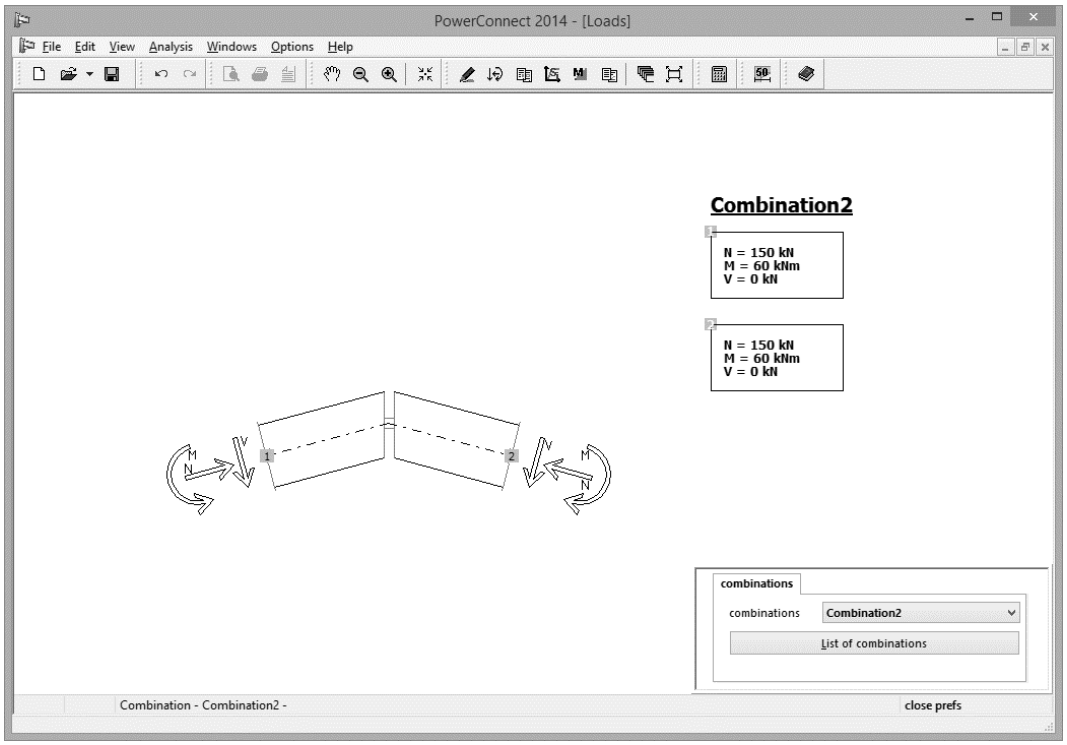


Make sure to leave the combination list visible at the right hand bottom of the ‘Loads’-window and check if ‘Combination1’ is the active combination. Then enter the appropriate load values for this combination, by clicking on the small

labeled numbers which are visible on the geometry representation in the ‘Loads’-window.
This should deliver the following result:

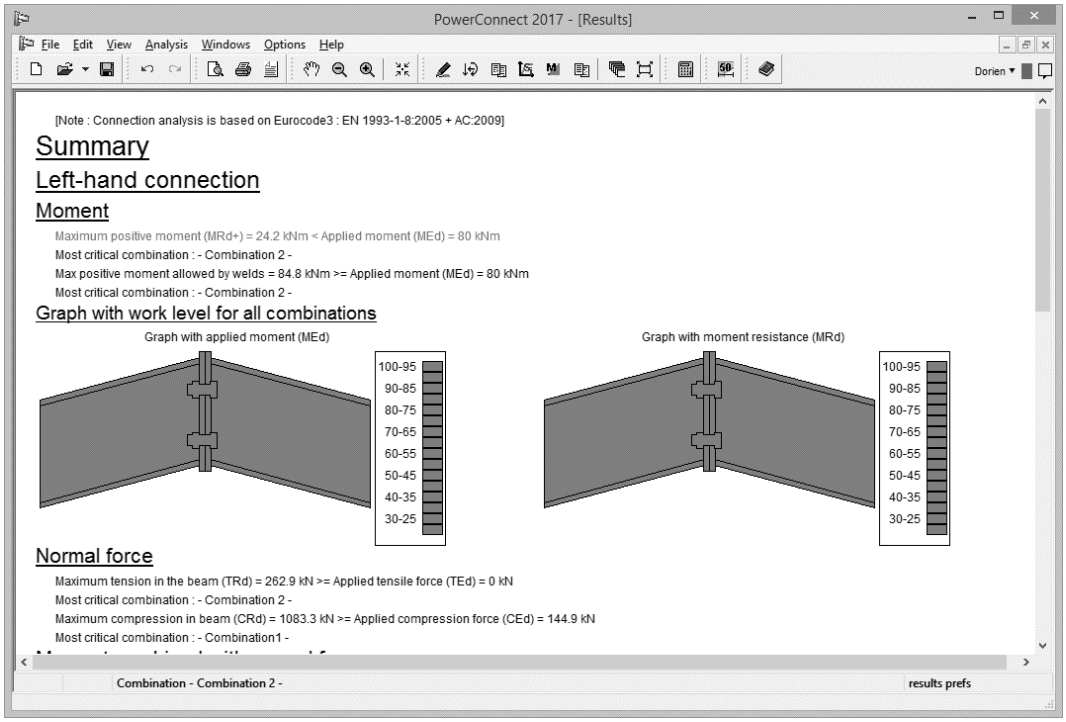


Now make sure ‘Combination2’ becomes the active loads combination (combination list at the right bottom of the ‘Loads’-window !) and enter the loads data as follows:



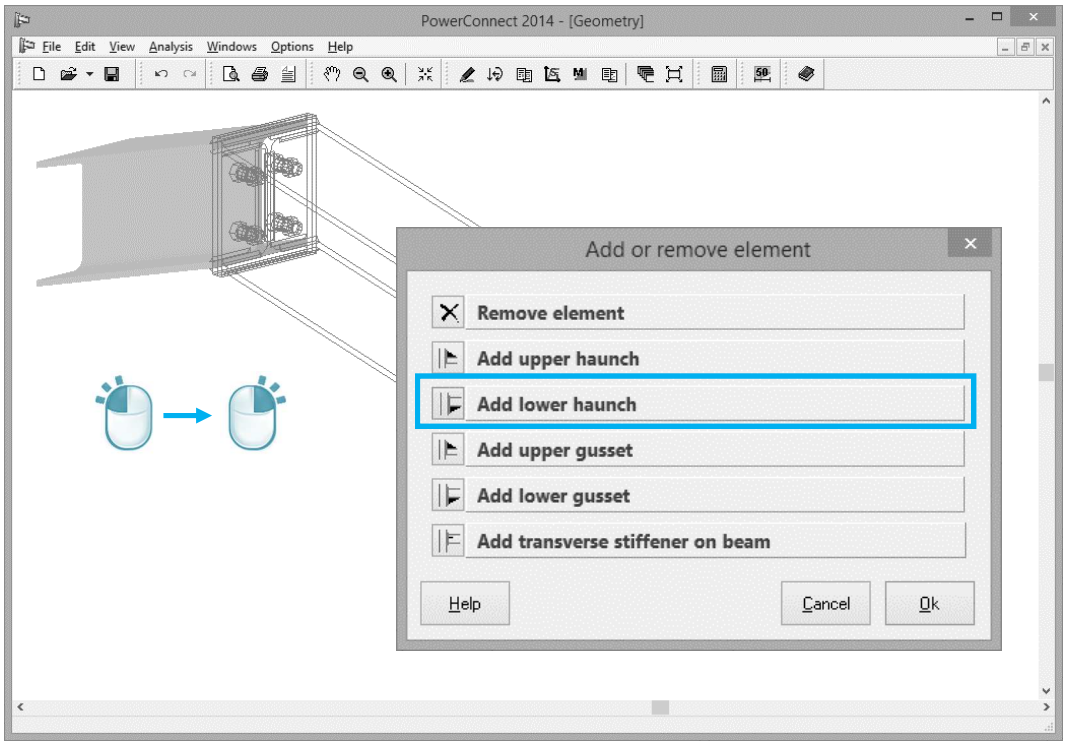
2.2.3 Running a first connection design analysis

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).

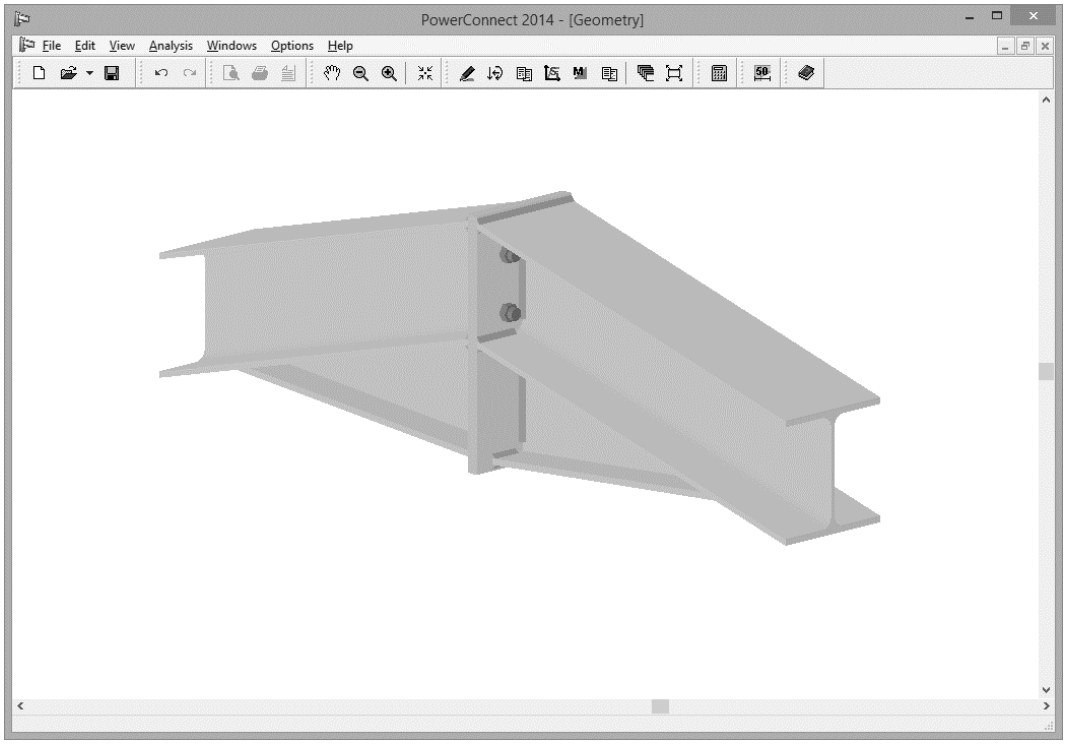


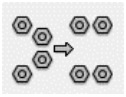
2.2.4 Improving the connection design

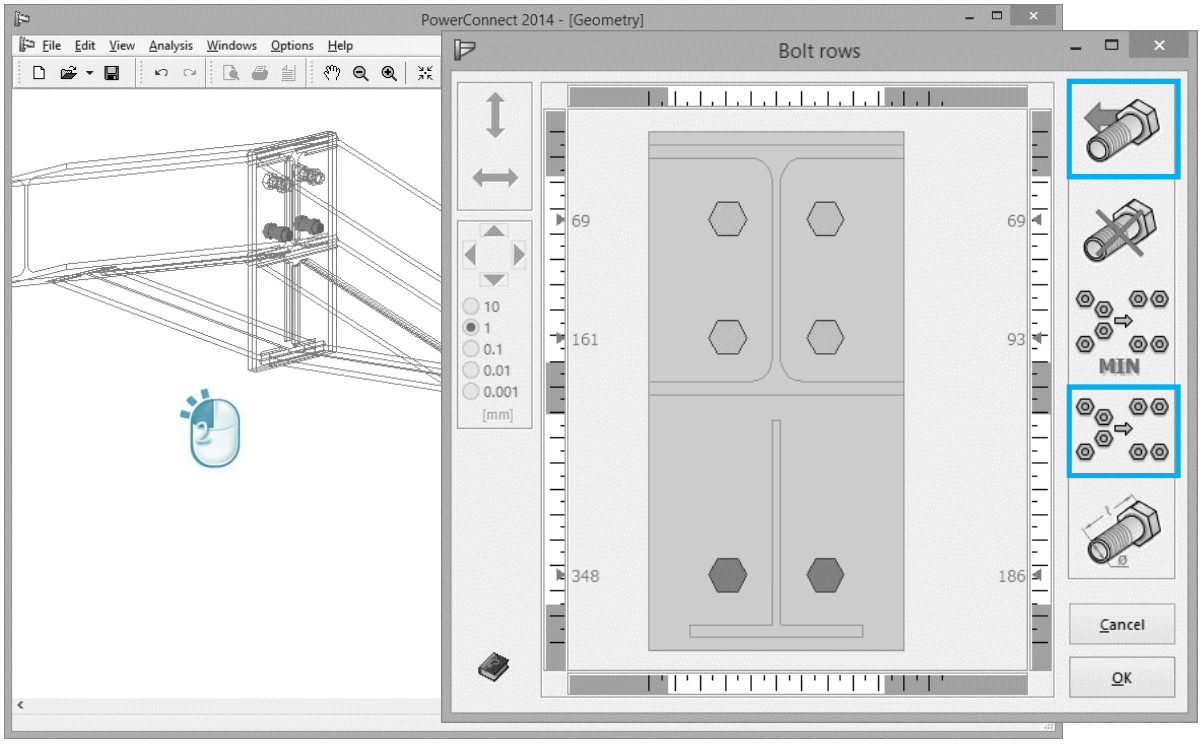
From the results summary, it is clear that the applied bending moment is too high as compared to the connection’s moment resistance (for both loads combinations). To solve such type of problem, a haunch can for instance be added below the connected beams. To do this, select one of the beams in the ‘Geometry’-window 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.

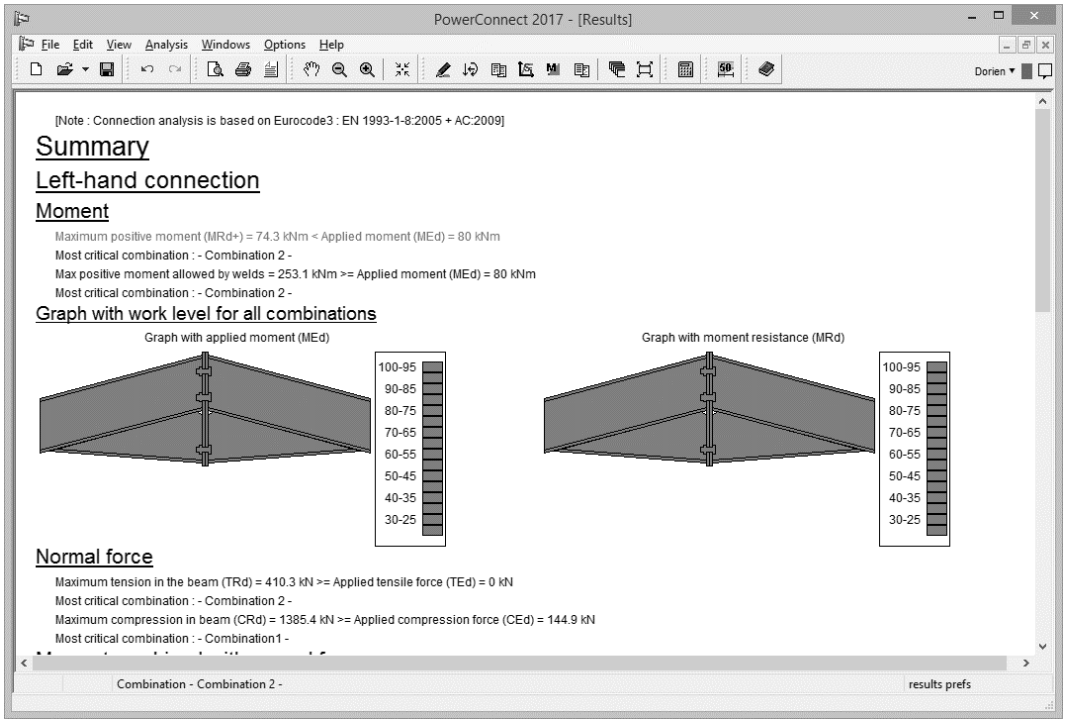


Double-click on of the bolts to enter into the dialogue window where an extra bolt row can be added  and where the bolt row lay-out can be optimized  .

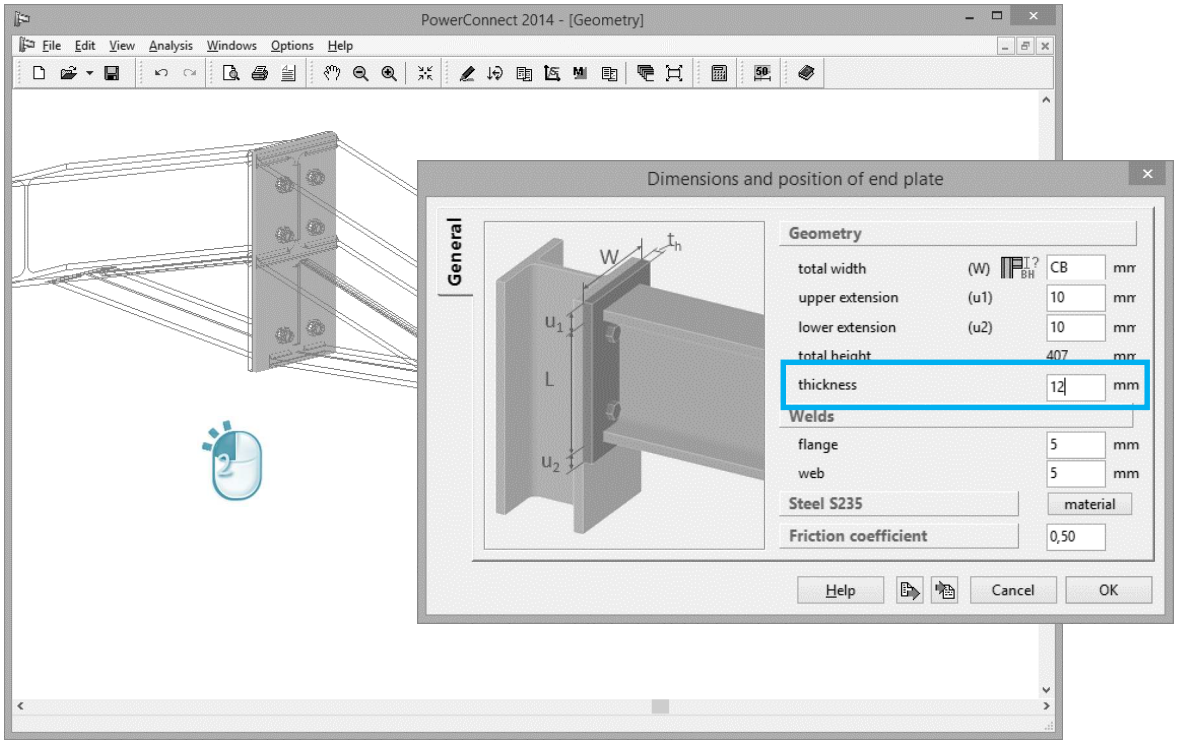


2.2.5 Re-running the connection design analysis on the modified connection

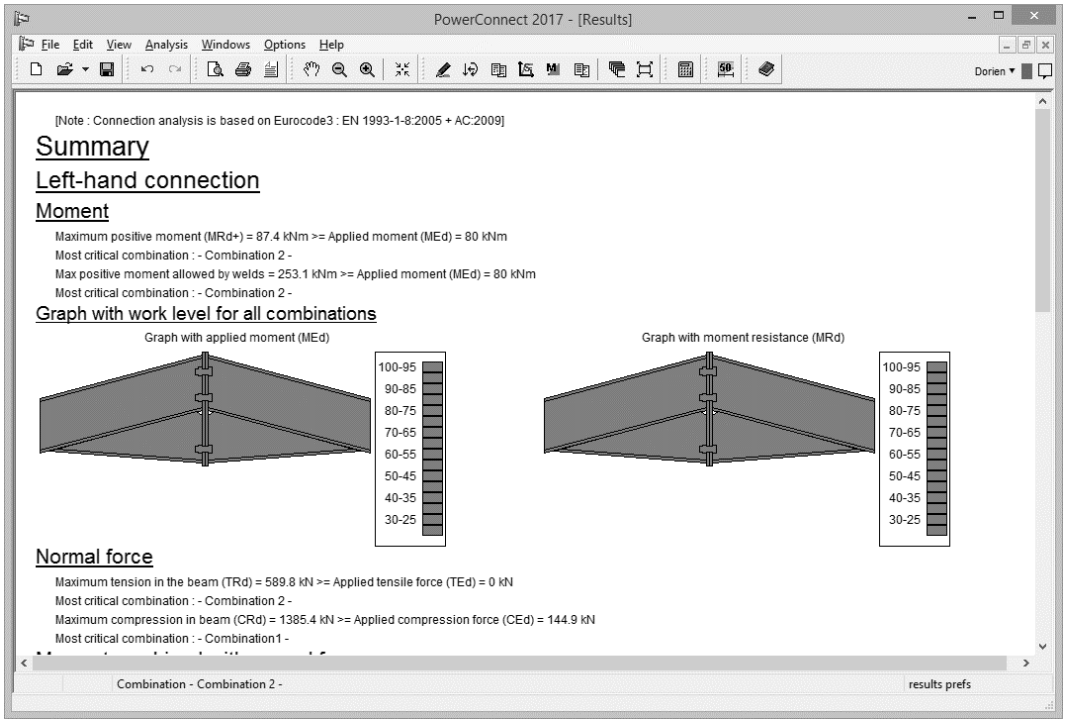
Running the connection design analysis again will now produce following summary results:



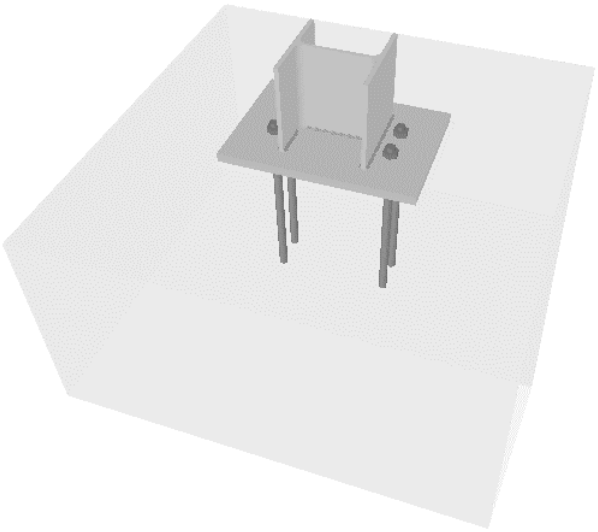
As can be seen from the results summary, the connection still has insufficient strength with respect to the applied loads combinations. Failure will occur due yielding of the end plate, which brings us to increasing the thickness of both end plates from CF (=10mm) to 12mm in the dialogue window which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



After rerunning the analysis, the connection now has sufficient strength.

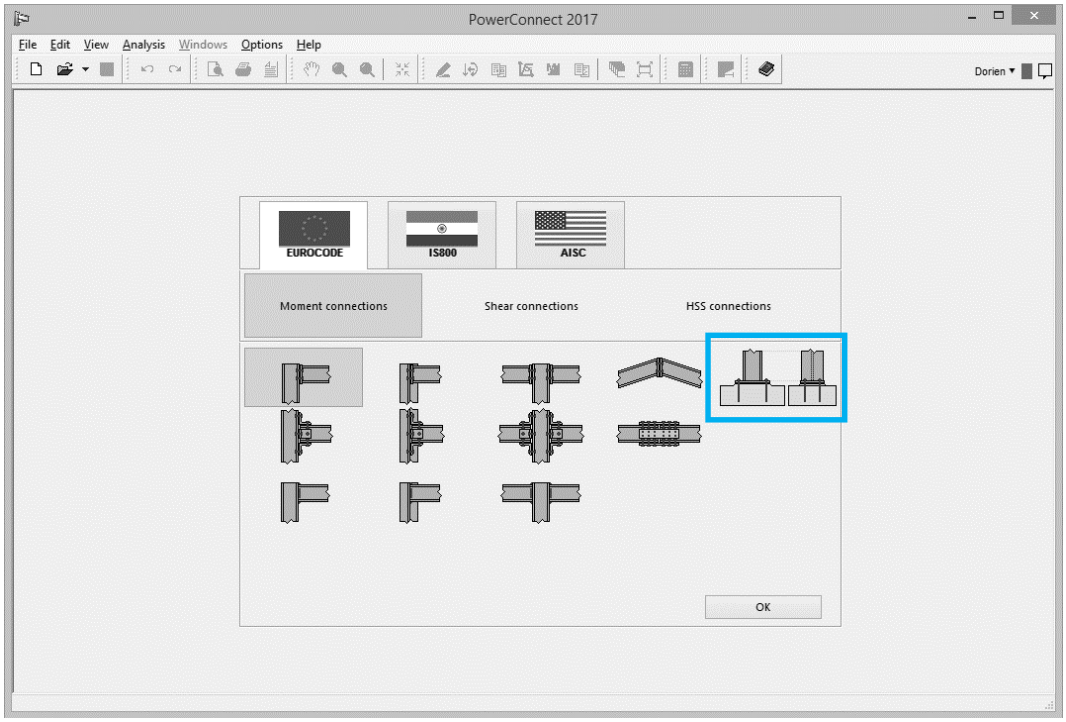

PowerConnect 2017 - [Results]
File Edit View Analysis Windows Options Help
Dorien
[Note : Connection analysis is based on Eurocode3 : EN 1993-1-8:2005 + AC:2009]
Summary
Left-hand connection
Moment
Maximum positive moment (MRd+) = 87.4 kNm >= Applied moment (MEd) = 80 kNm
Most critical combination : - Combination 2 -
Max positive moment allowed by welds = 253.1 kNm >= Applied moment (MEd) = 80 kNm
Most critical combination : - Combination 2 -
Graph with work level for all combinations
Graph with applied moment (MEd)
Graph with moment resistance (MRd)
100-95
90-85
80-75
70-65
60-55
50-45
40-35
30-25
Normal force
Maximum tension in the beam (TRd) = 589.8 kN >= Applied tensile force (TEd) = 0 kN
Most critical combination : - Combination 2 -
Maximum compression in beam (CRd) = 1385.4 kN >= Applied compression force (CEd) = 144.9 kN
Most critical combination : - Combination1 -
Combination - Combination 2 -
results prefs

2.3 Tutorial 3: column base with extended end plate

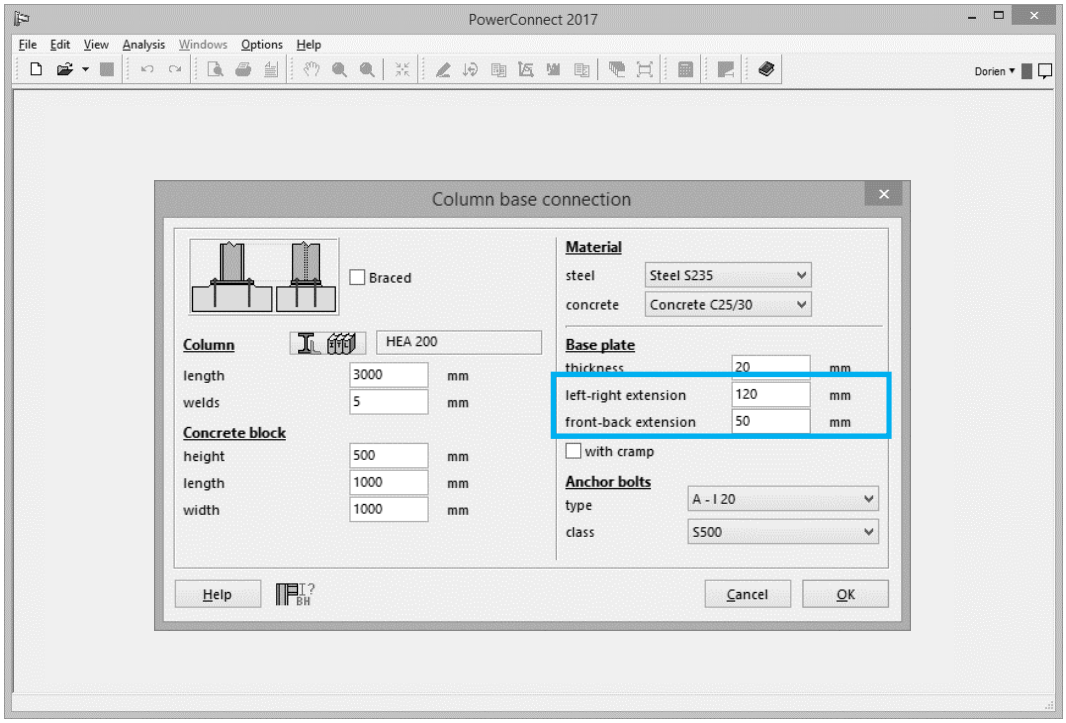


2.3.1 Setting up the model

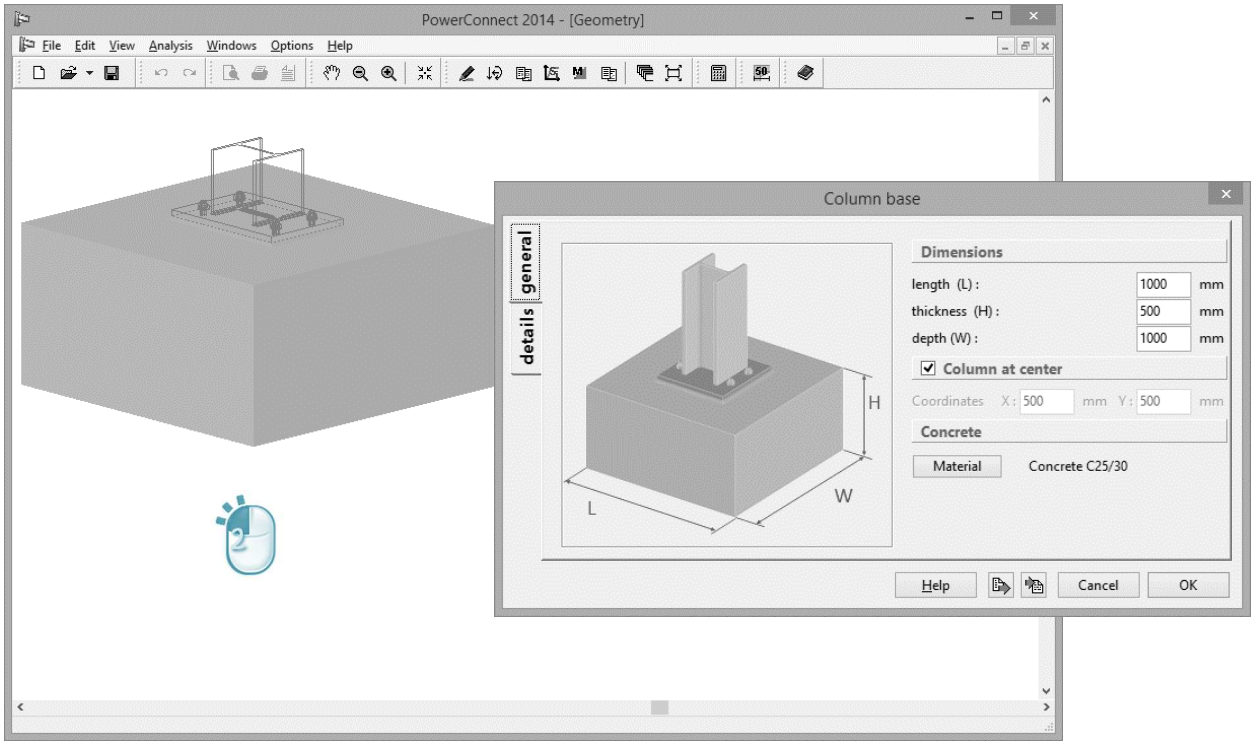
Use the icon  to define a new project. In the navigation window, select the connection type labelled “Column base plate’.



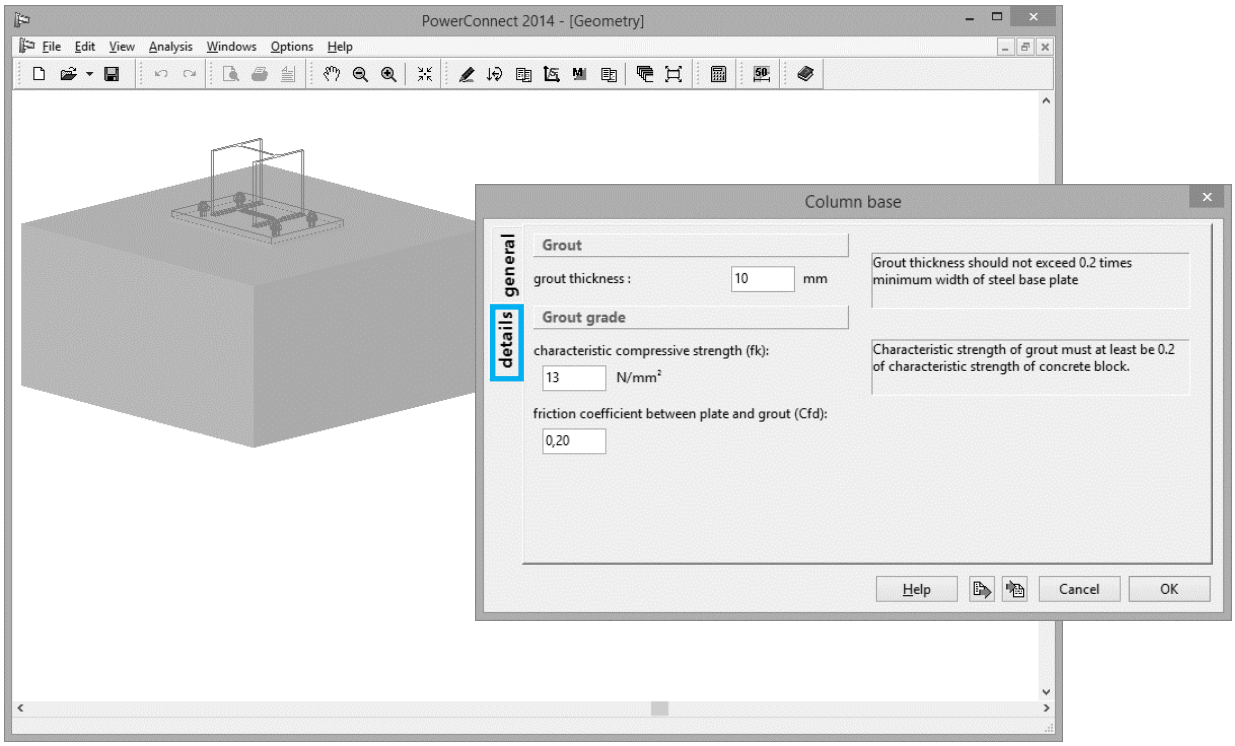
To actually create connection geometry, fill out all parameters as shown in the dialogue below.



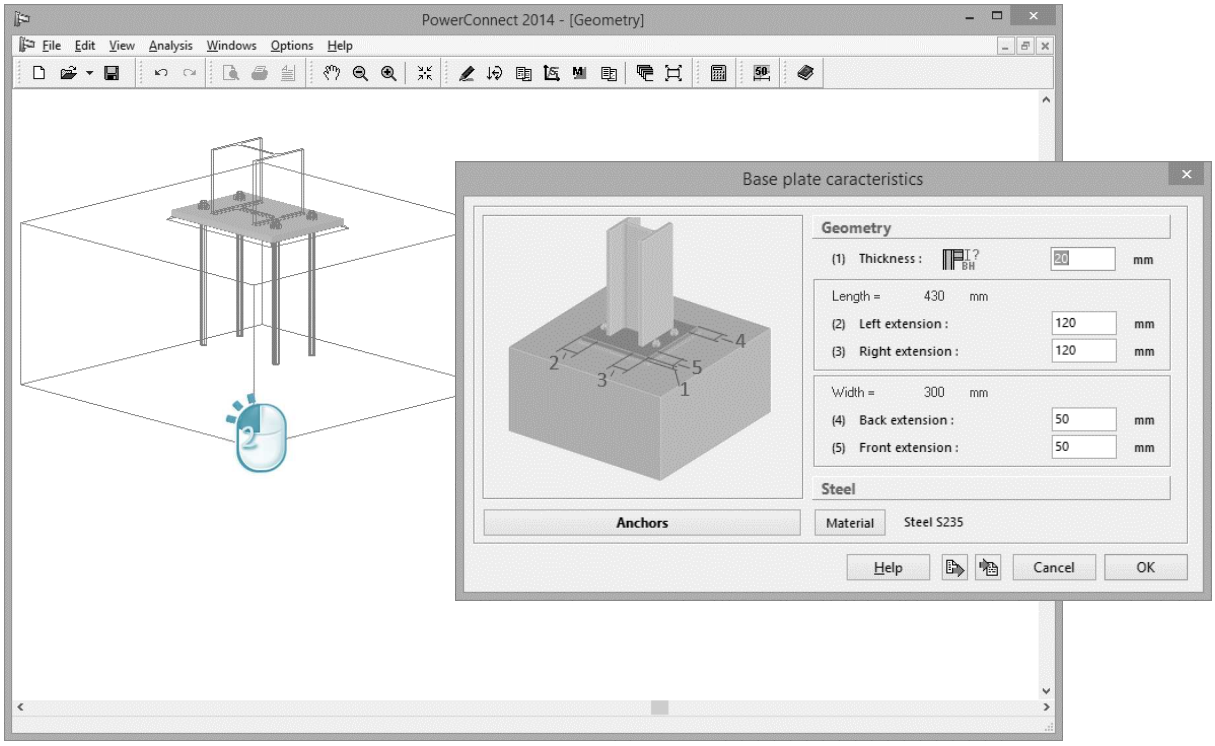
Now double-click on the concrete block to make a dialogue appear in which further details of the concrete base can be verified or modified. Check e.g. if the concrete grade of the base on the “General” tab page is C25/30, and modify (if necessary) by using the “Material”-button.



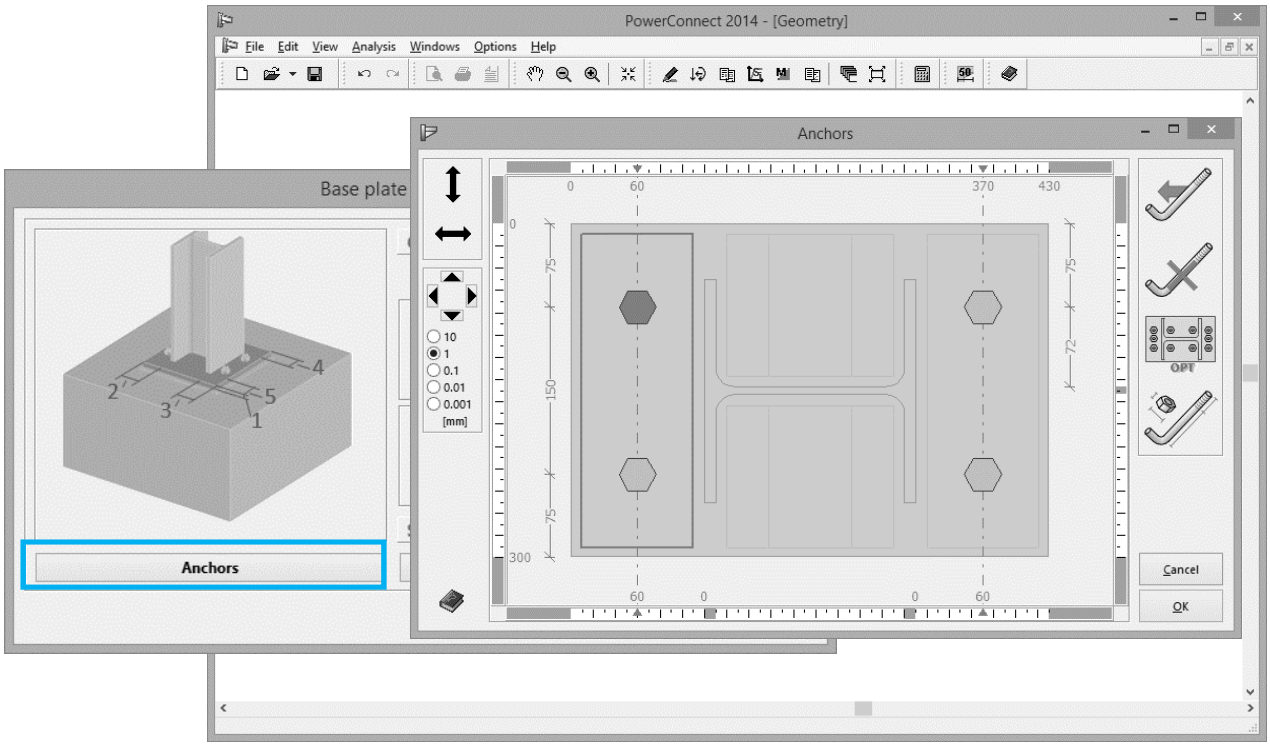
Switch to the ‘Details’-tab for further data on grout thickness & grade.



Close this window by clicking ‘OK’.
Double-click on the base plate to access its properties.



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

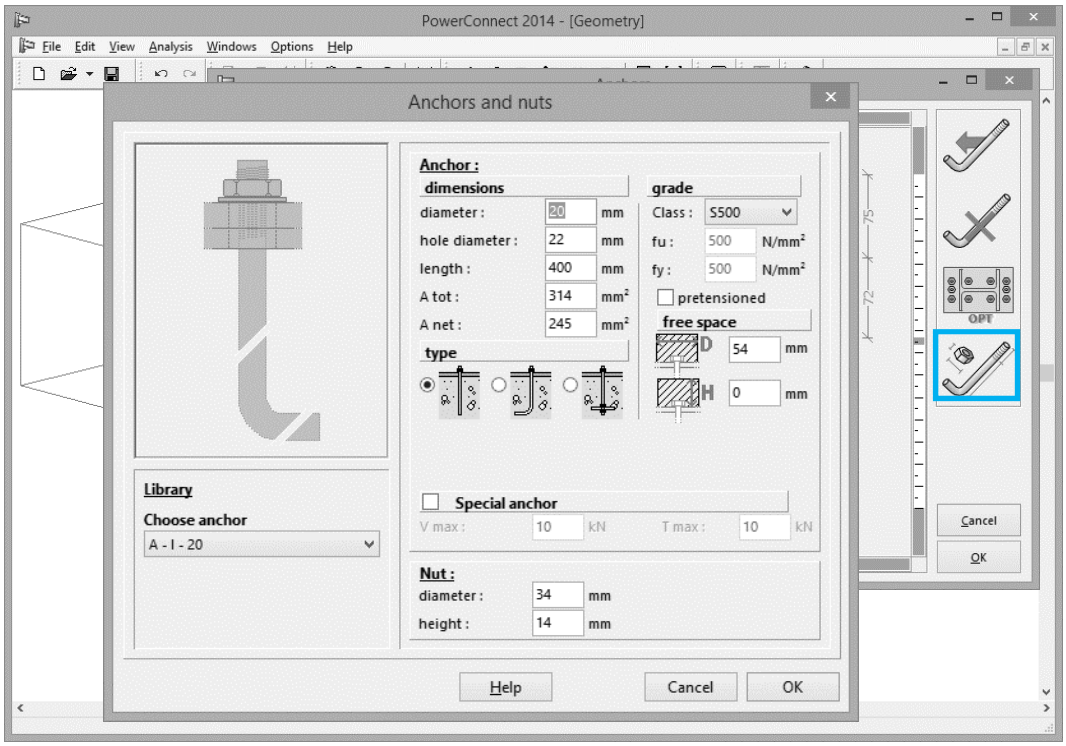


The window not only shows base plate dimensions and anchor bolts positions, but also includes a number of anchor zones that can be used to define the anchor bolts configuration. In case an anchor bolt row needs to be added, the appropriate anchor zone must first be selected using the mouse. The borders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

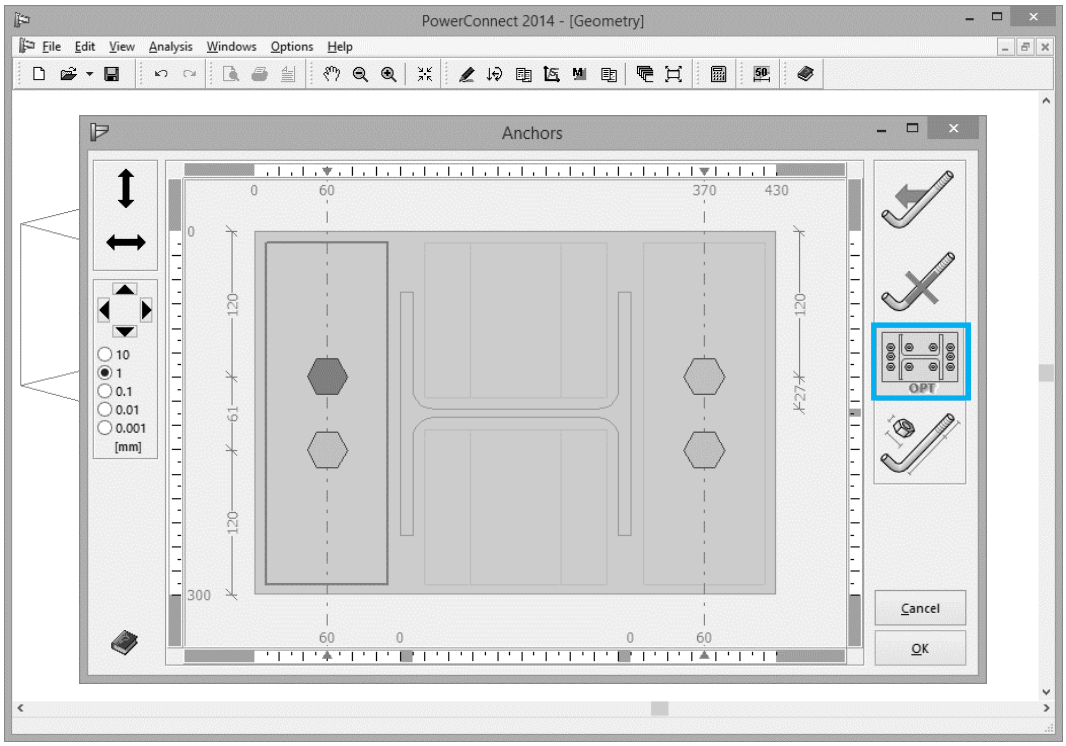


On the left hand side, a specific anchor bolt can be selected from PowerConnect's library of bolts & anchor bolts. For the time being, keep the default 'A-I-20' proposed by the program. 'A-I-20' corresponds to straight anchor bolts with a diameter of 20mm.

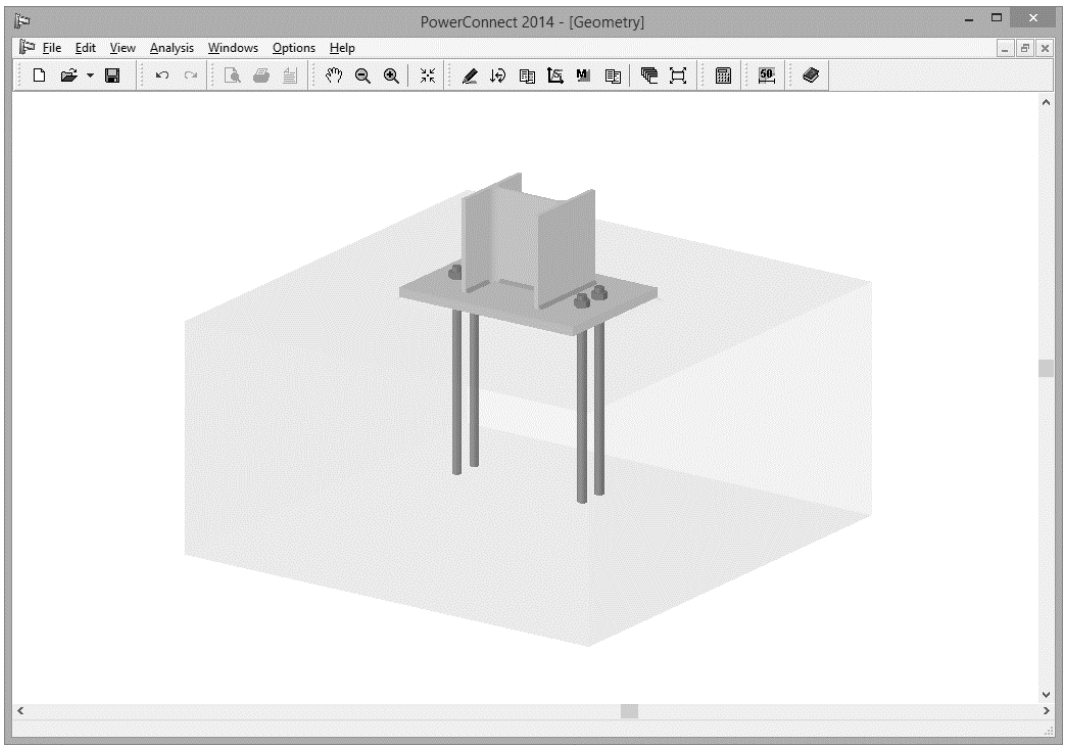
As no modifications must be defined as far as anchor bolt choice is concerned, the "Cancel"-button can be used to return to the anchor bolt lay-out window.

Use the icon 

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

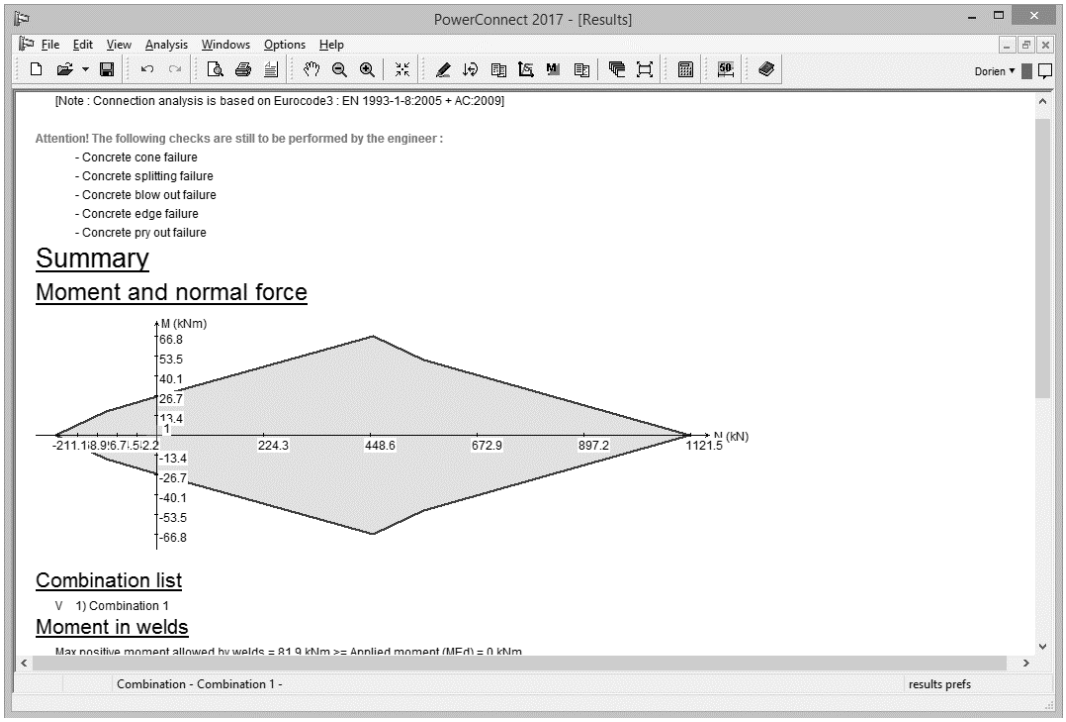


Click ‘OK’ to complete the base plate connection definition. In the ‘Geometry’-window, the following model will now be presented:



2.3.2 Running the connection design analysis

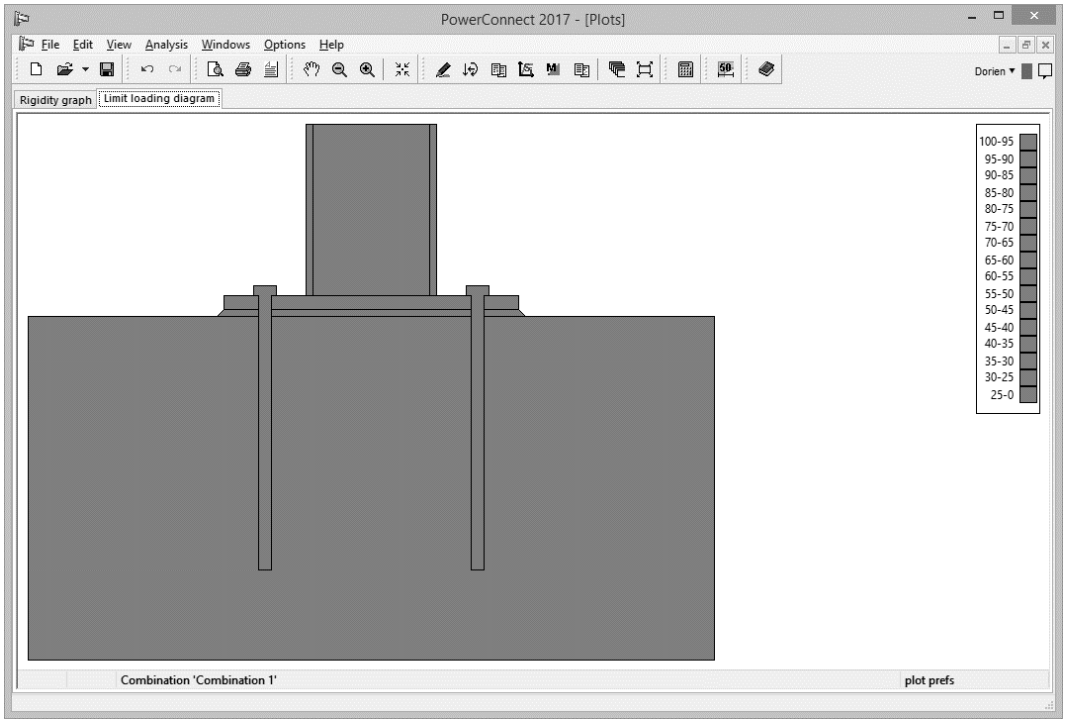
For the current analysis, no specific external loads will be applied. As a consequence, the design analysis will evaluate the connection's resistance, independent of any loading. The following result will be obtained:



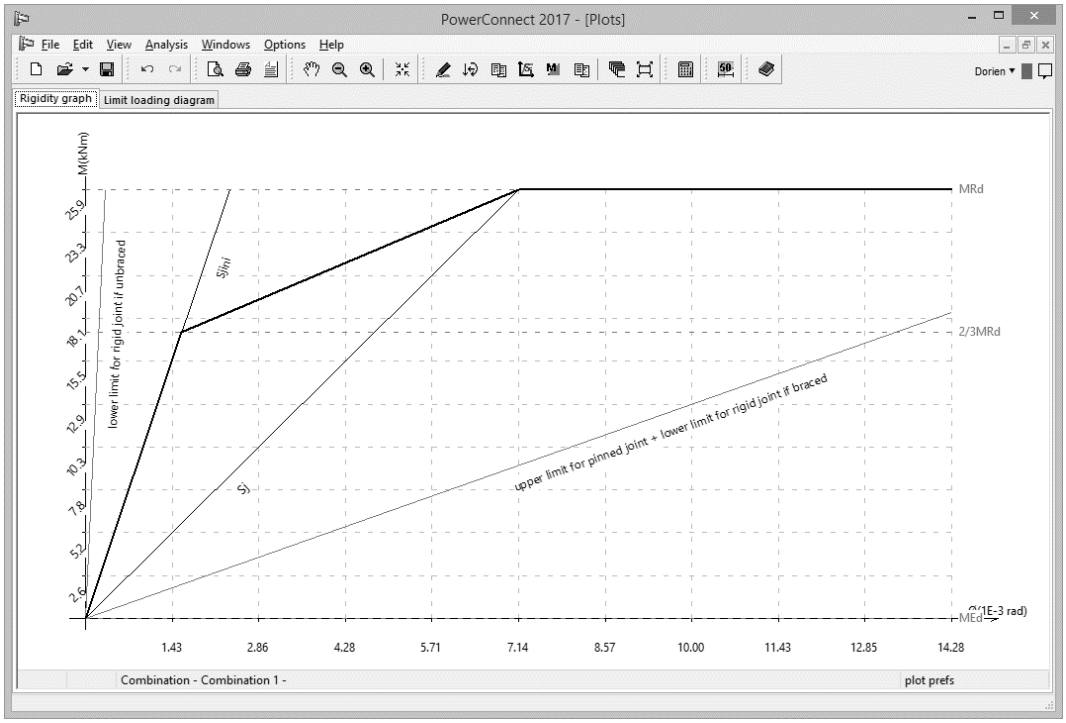
With this particular user scenario, there is no such thing as a "bad" or "good" connection. The resistance that has been calculated should be compared to a specific set of loads to which this type of connection will be subjected to enable that kind of judgment.

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).

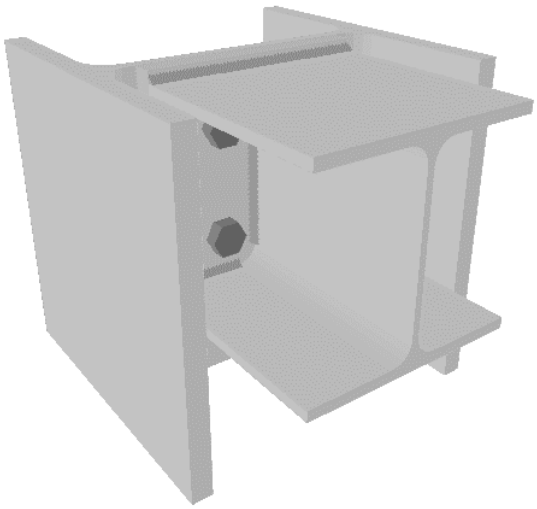
All loads combinations which fall within the green area of this diagram, correspond to loads which can be sustained. In case a particular loads combination falls outside this area, the base plate connection should be modified to resist the applied loading. The more detailed design analysis results that 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)

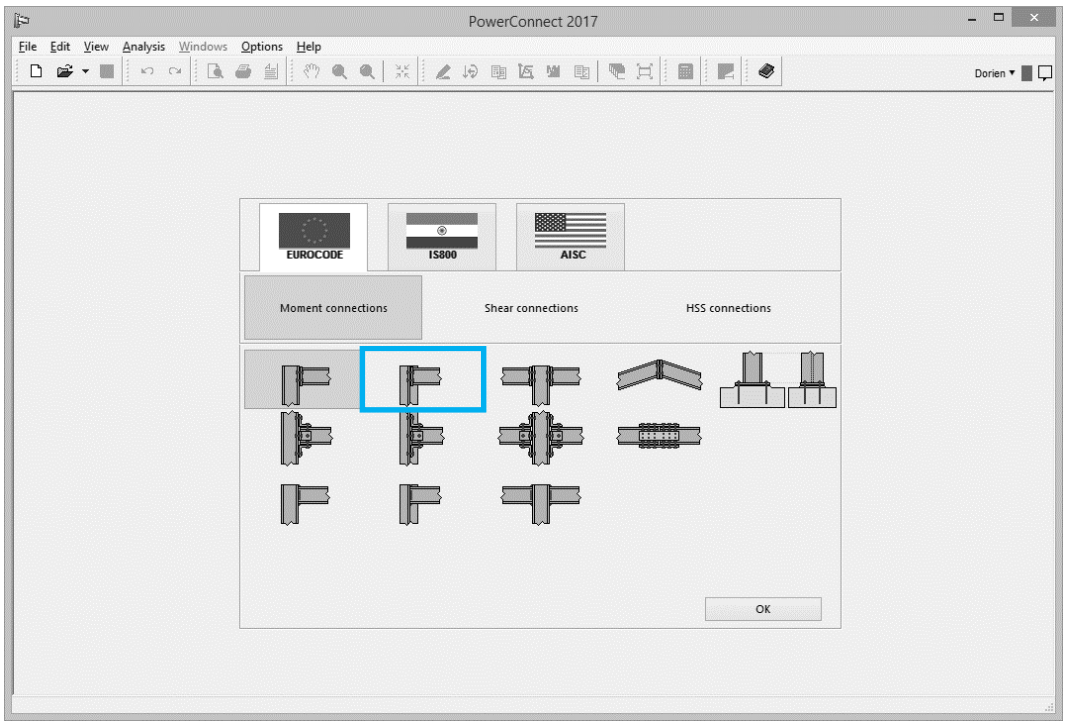


2.4.1 Setting up the model

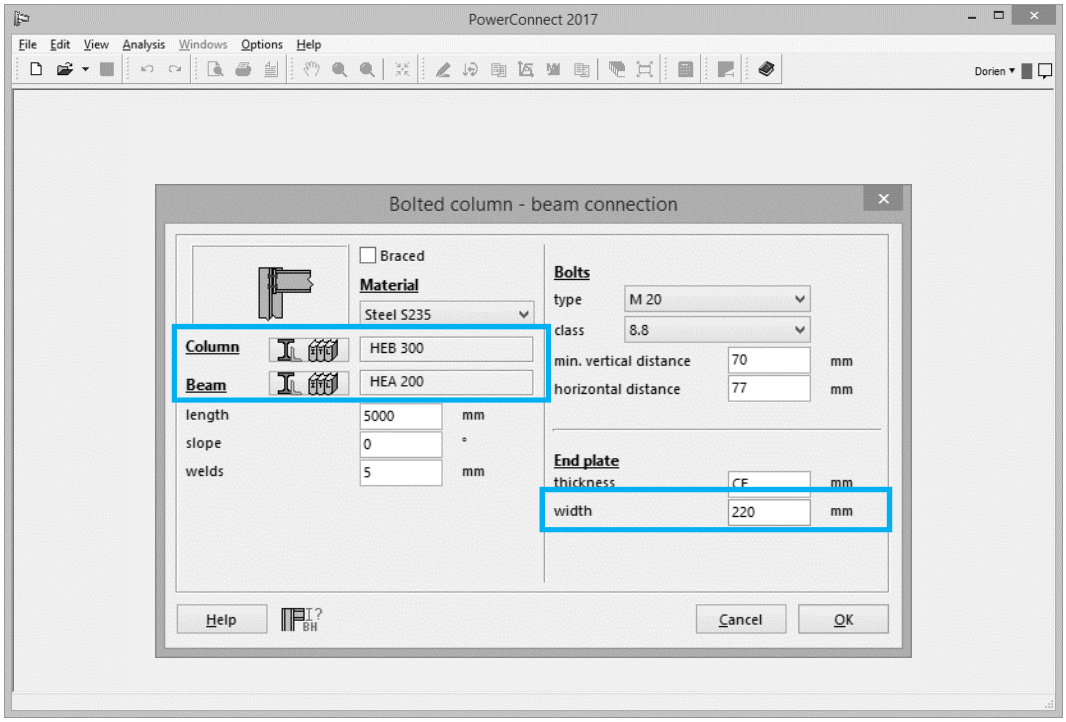
A beam with HEA 200 section is connected to the web of a column with **HEB** 300 section by means of a bolted moment end plate (2 rows of M20 bolts). No particular stiffener elements are added.

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.

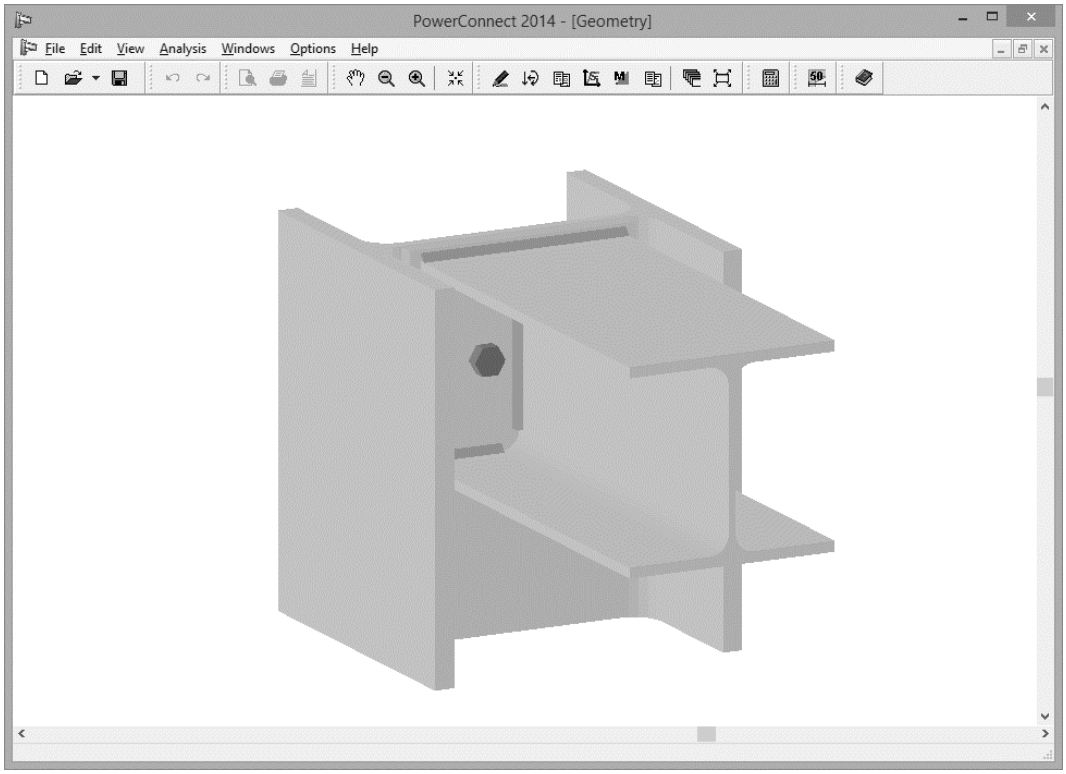
In the navigation window, select the connection type labelled "Bolted moment end plate connection (weak axis)'.



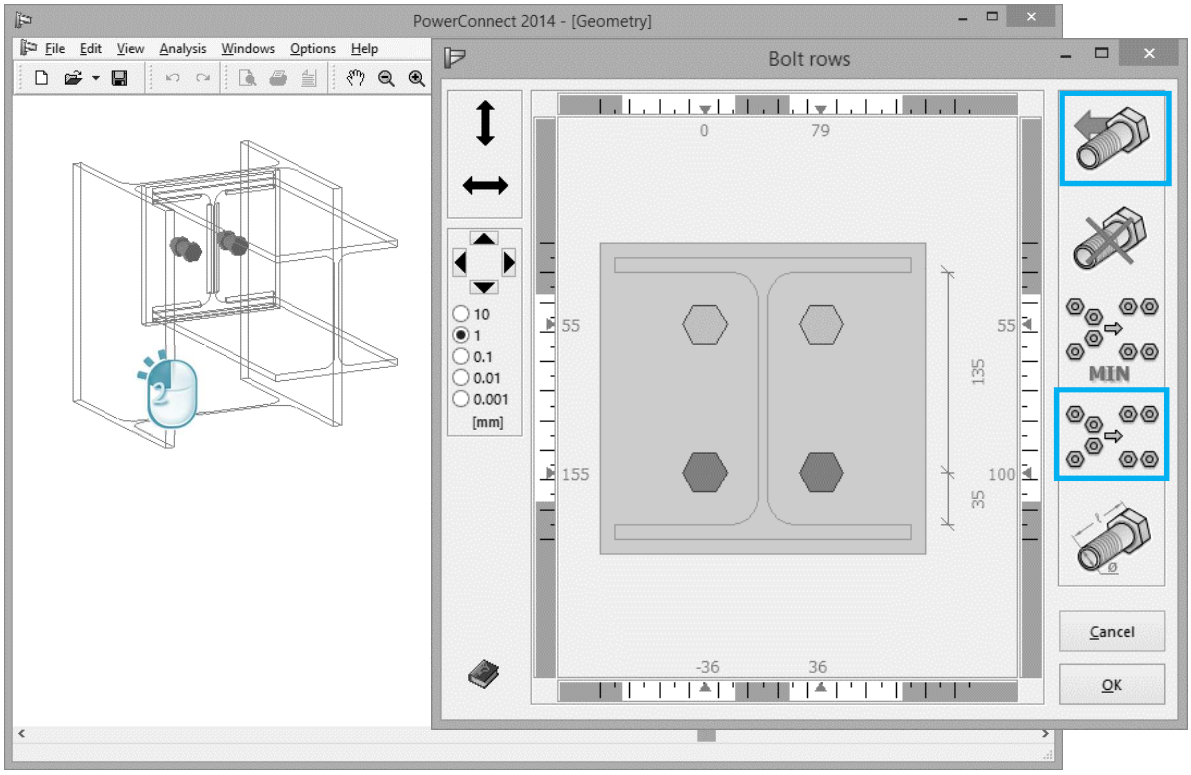
In the next step, make sure that the beam and column sections are defined correctly. Choose other sections from PowerConnect's section library, if needed.

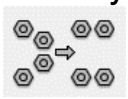


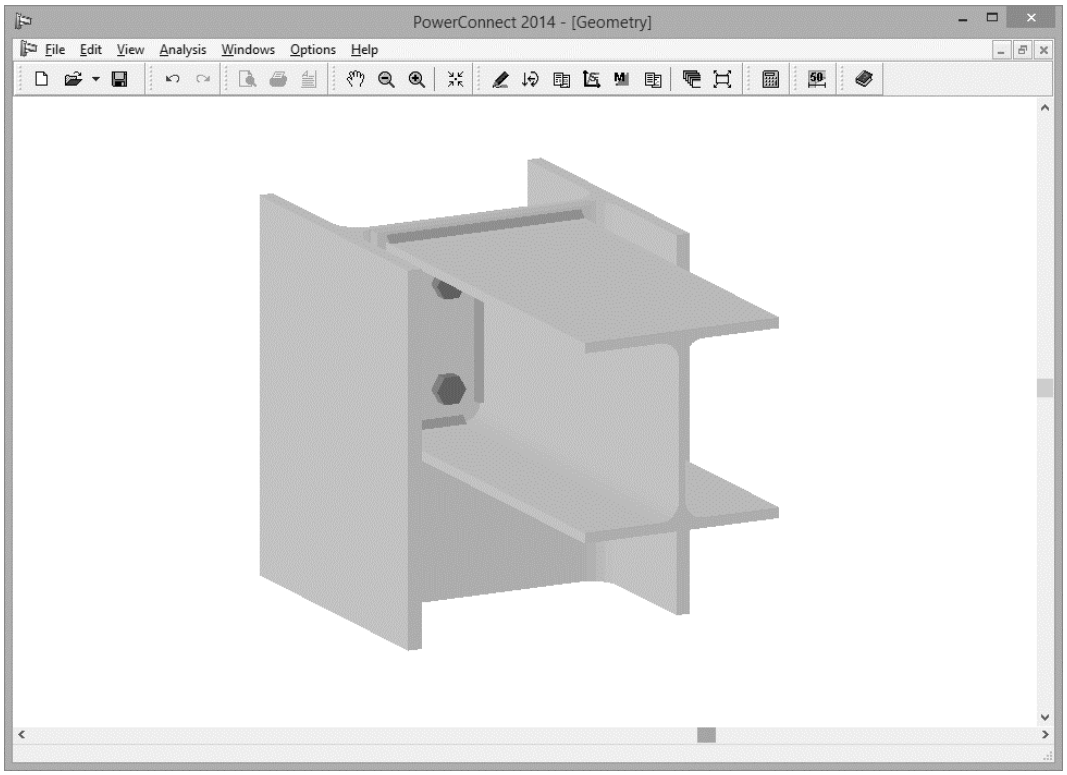
When confirming the above parameters, the following 3D geometry model will be presented.



If needed, end plate and 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.



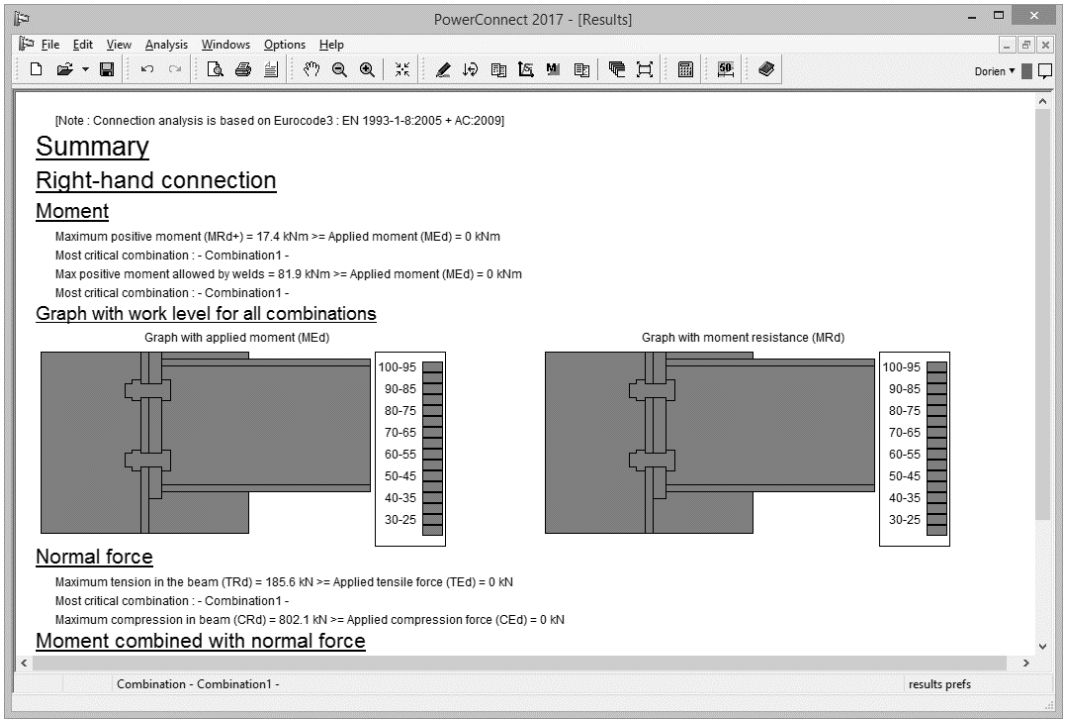
Use the  icon to add an extra bolt row to the current configuration. The icon  can be used to verify whether the currently selected bolts are of type M20. Finally, use the  icon to optimize bolt row positions, to arrive at the following lay-out:



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, following results are obtained:

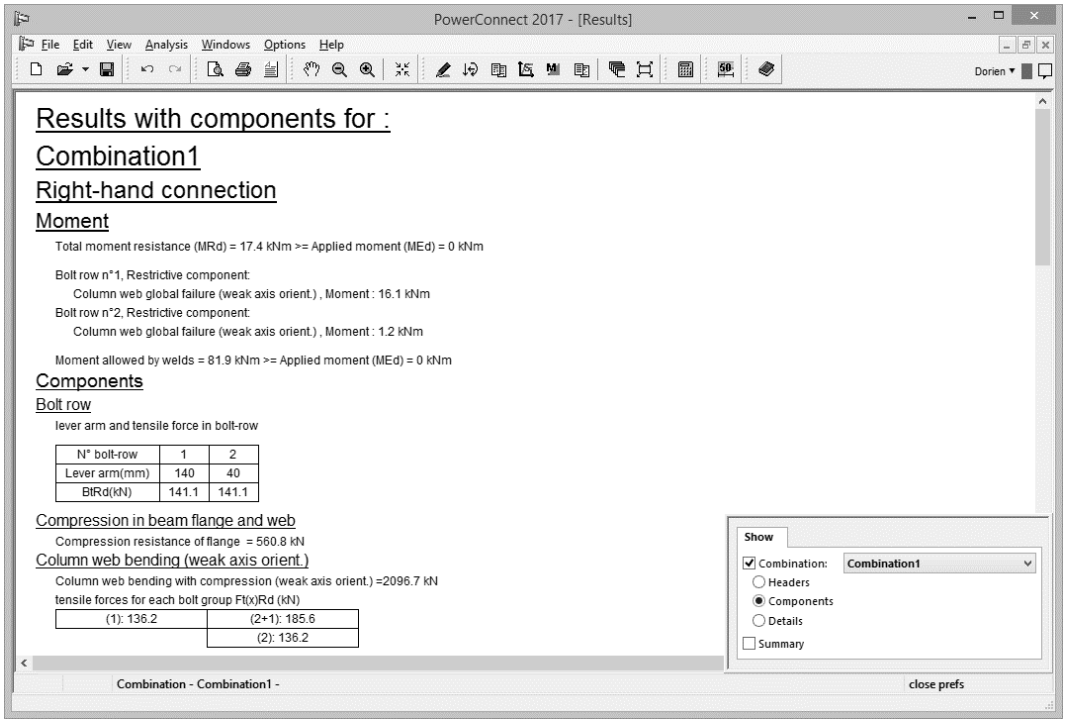


In case a beam is directly connected to the column web (even through the use of an end plate), it may happen that the column web does not provide a sufficiently high resistance. Both local and global failure of the column web may occur. A local failure mechanism occurs when e.g. a bolt row that is subjected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

Three types of local failure mechanisms are possible on the column web:

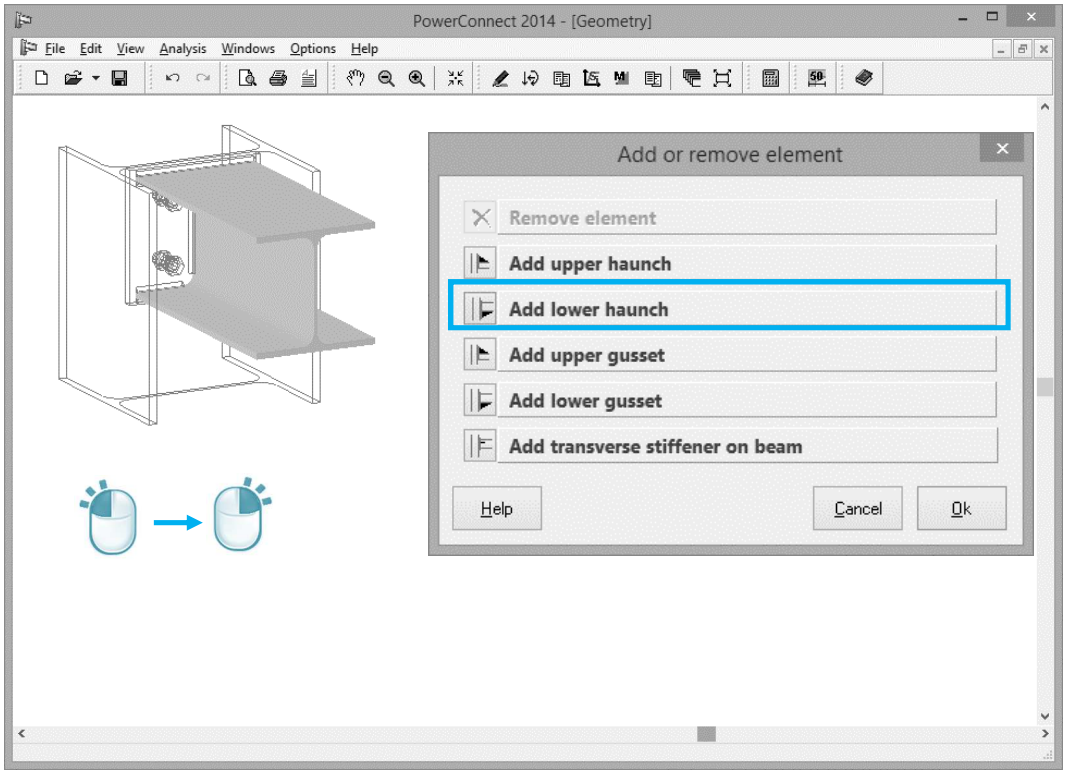
- bending
- shear
- a combination of the previous two.

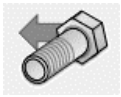
PowerConnect will screen all possible local & global failure mechanisms and will present detailed analysis results for all of them. In the current case, the connection will fail globally.

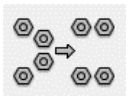


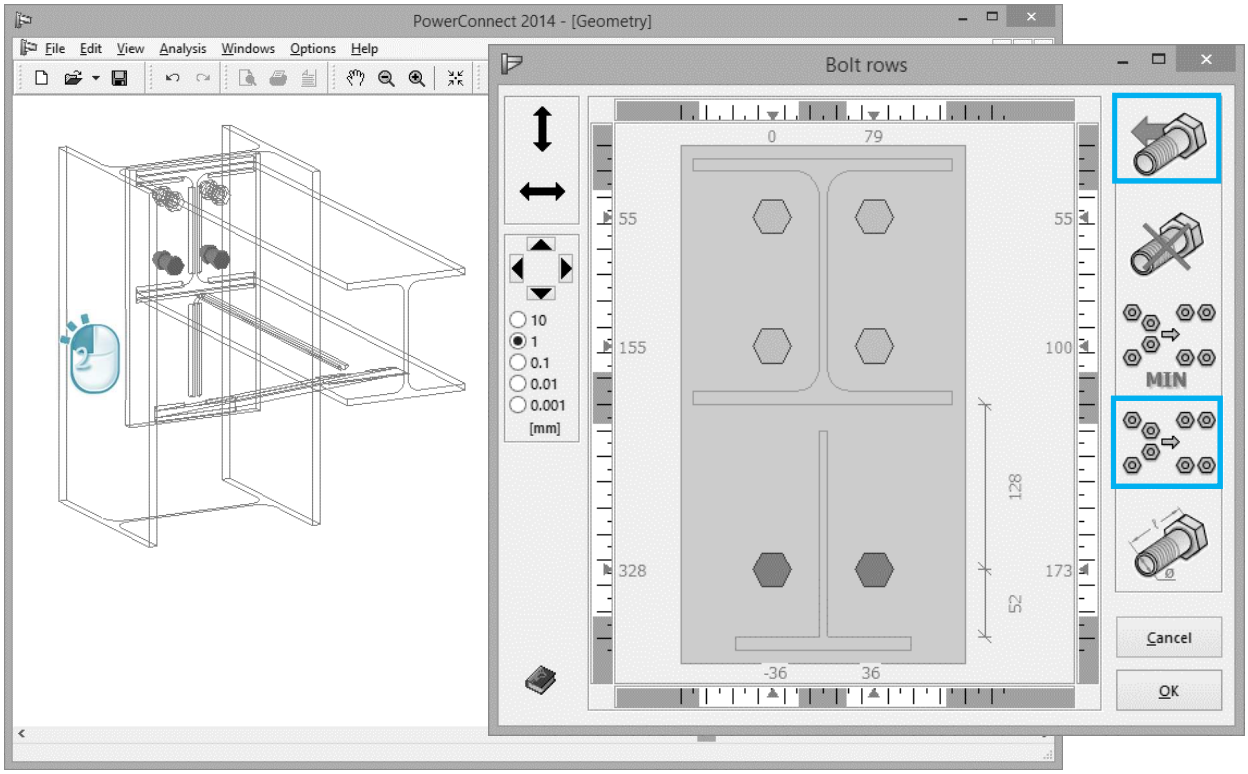
2.4.3 Improving the connection design

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



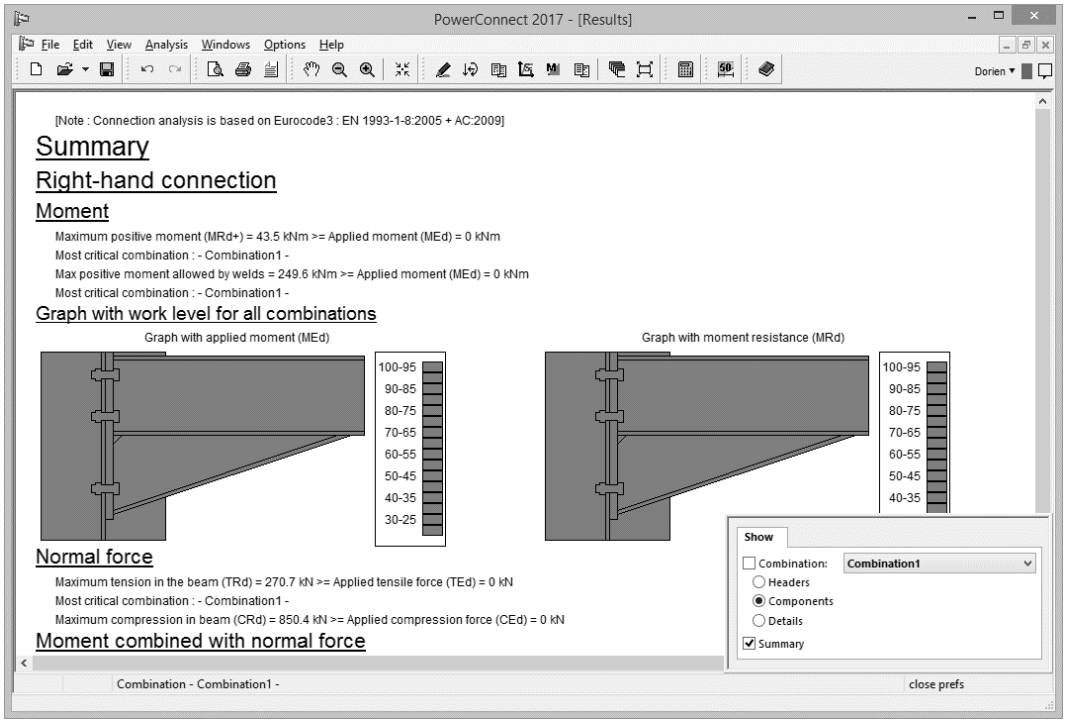
Now double-click on one of the bolts and then add an extra bolt row  .

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



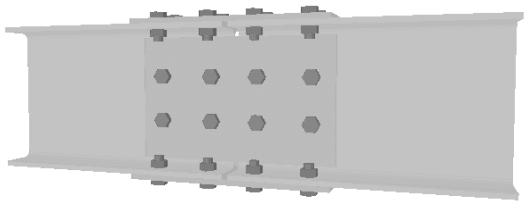
2.4.4 Re-running the connection design analysis on the modified connection

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



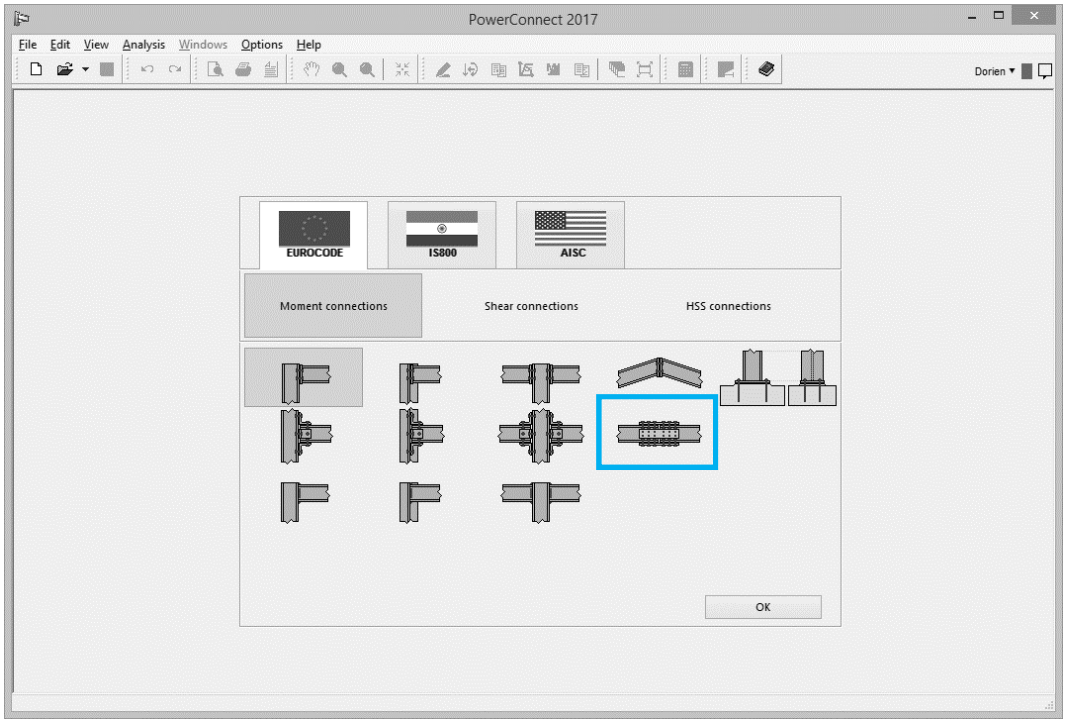
As a result, the connection's bending resistance has been increased from 17,4kNm to 43,5kNm.

2.5 Tutorial 5: bolted splice

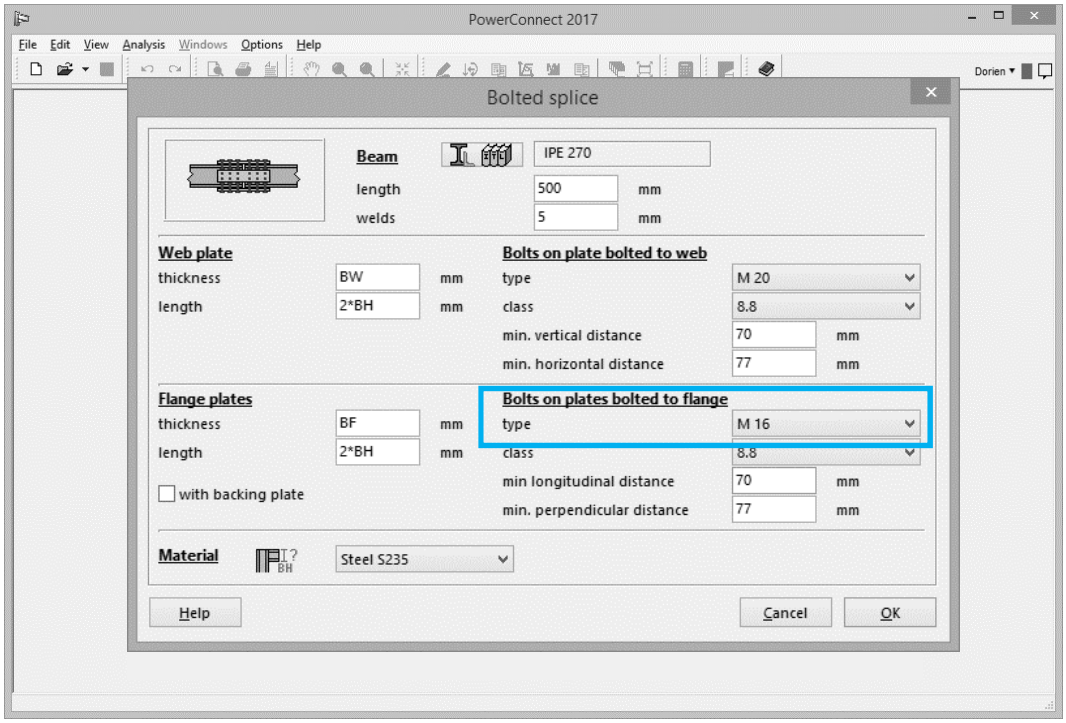


2.5.1 Setting up the model

Define a new connection . Among the available connection types in the navigation window, select the one labelled “Bolted splice”.



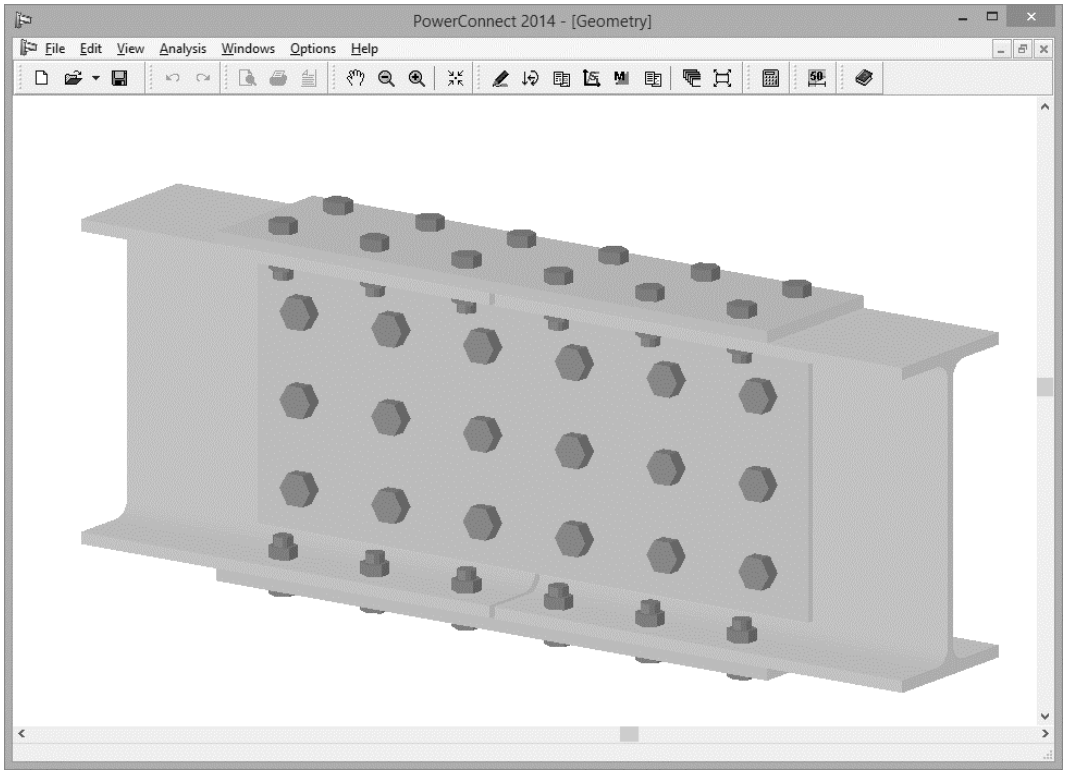
After confirmation of the connections above, a new dialogue window is presented to complete the definition.



In particular, remember to enter the following parameters:

- IPE 270 section for the beam element
- bolts M16 – grade 8.8 for the flange plates

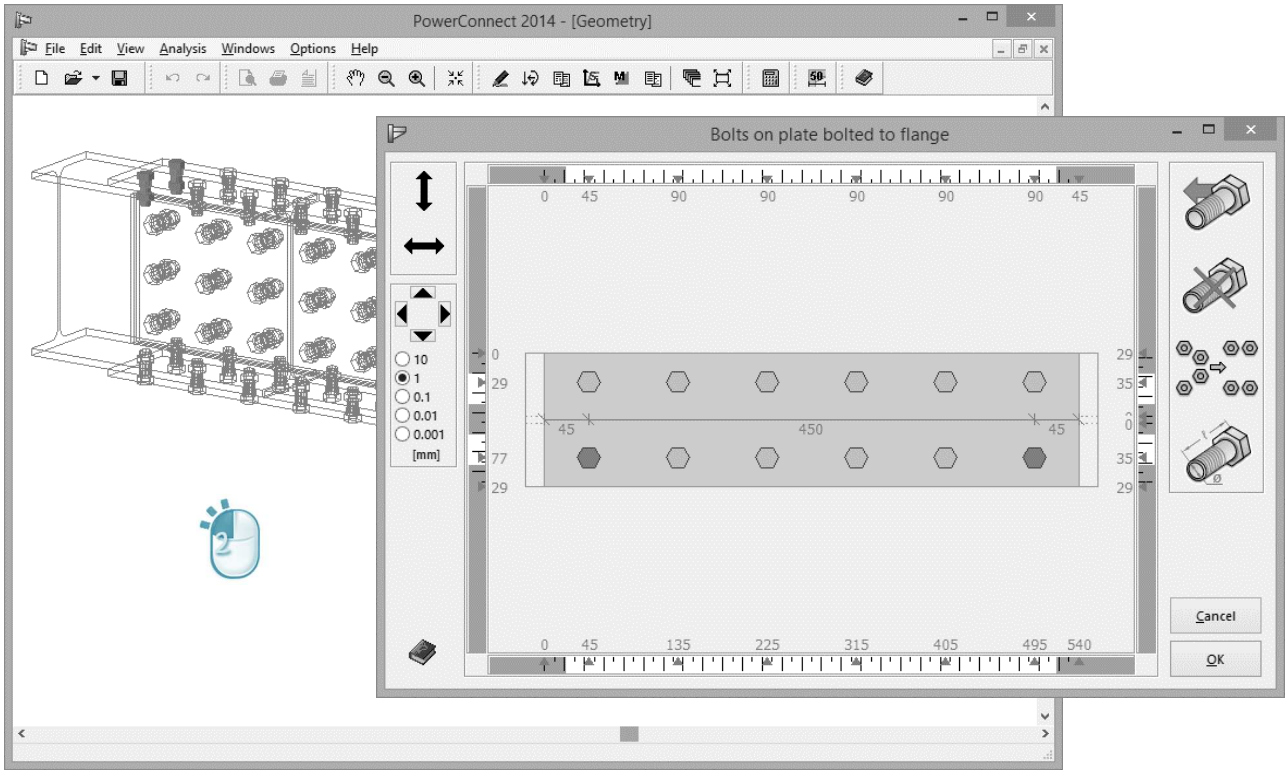
As a result, the beam splice as illustrated in the 3D representation is obtained.



Bolted beam splices are always symmetrical in 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.

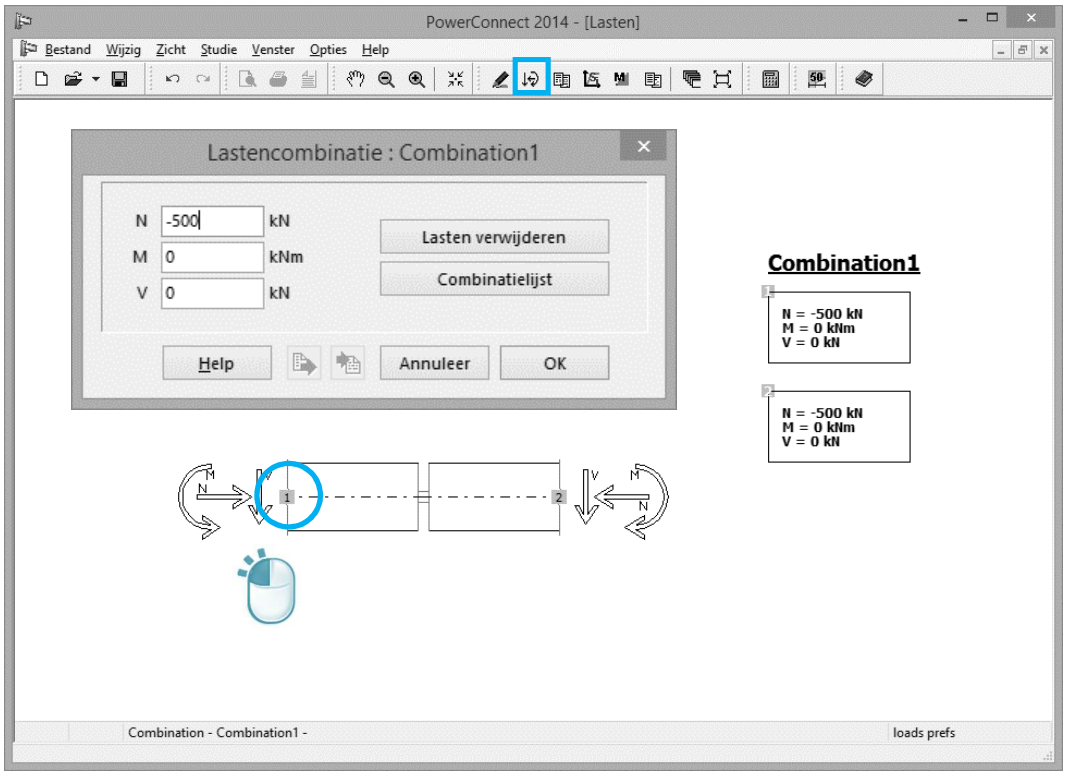
To remove a bolt row, selected one of its bolts and use the  icon.

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

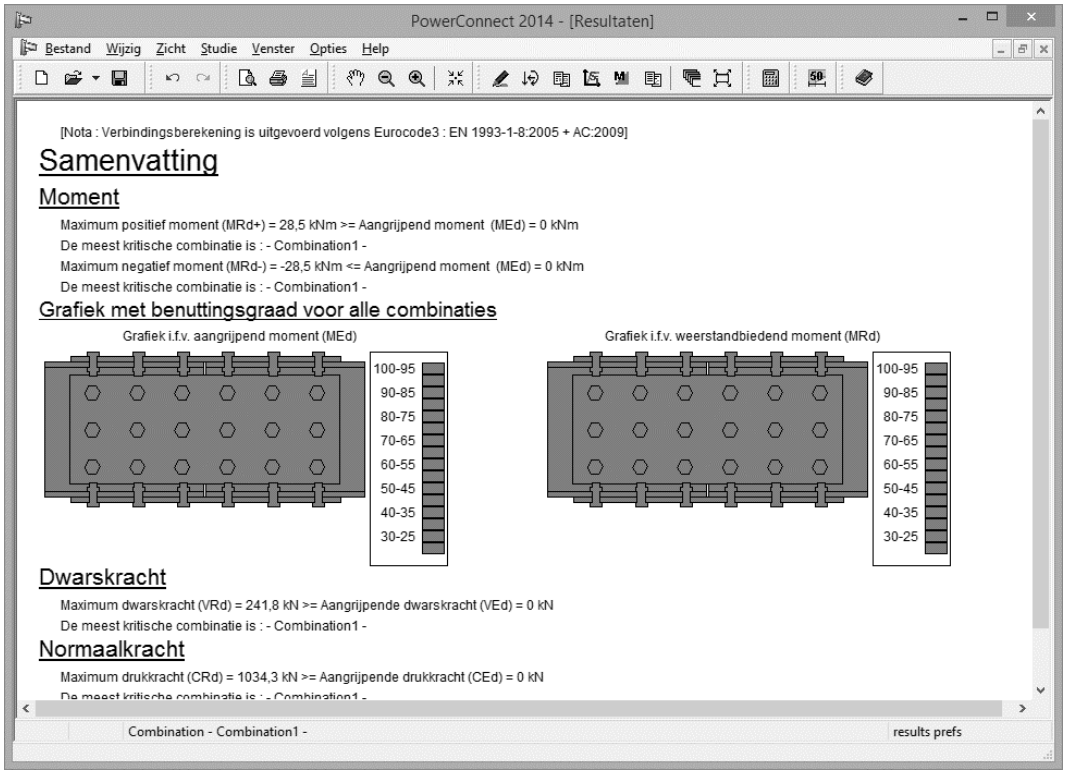
2.5.2 Defining the loads

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



2.5.3 Running the connection design analysis

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



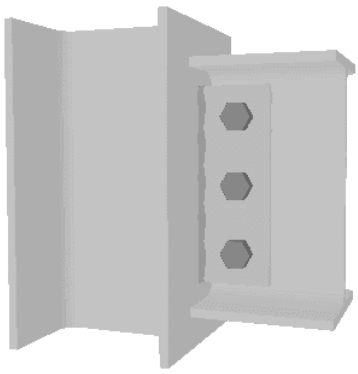
For this type of connection, PowerConnect will report the ultimate bending moments (both positive and negative). Those bending moments are calculated taking into account the shear force and normal force that have previously been applied to the connection ($V = 0kN$ and $N = -500kN$).

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.

In §3.2 you can read how to make de report for this connection.

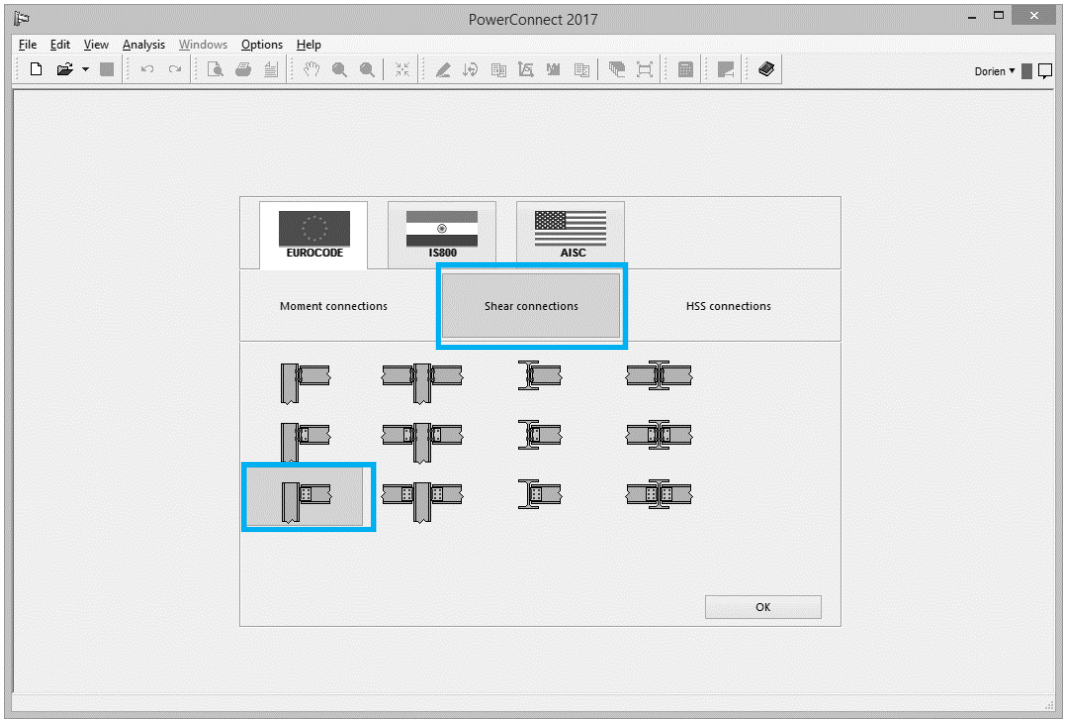
2.6 Tutorial 6: shear connection – beam to column flange with fin plate



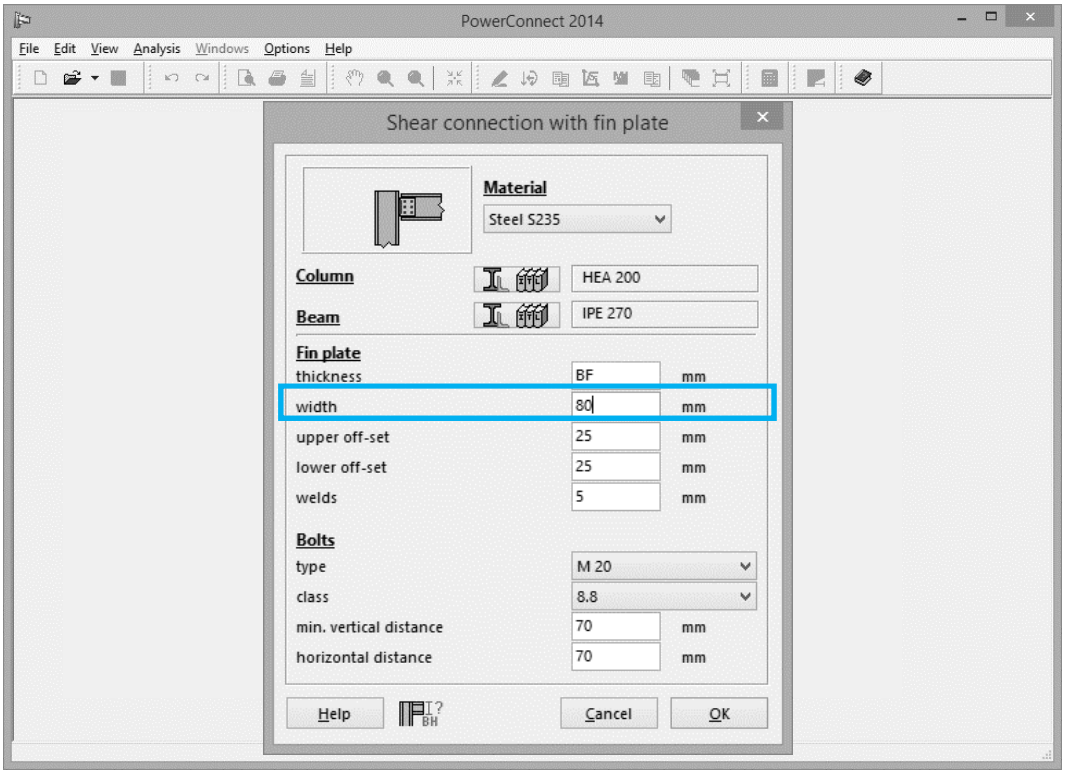
In this first tutorial on shear connection design, the focus will be on a beam to column flange connection through fin plates.

2.6.1 Setting up the model

Use the icon  to define a new project. In the navigation window, select the connection type labelled “Fin plate connection”.



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.

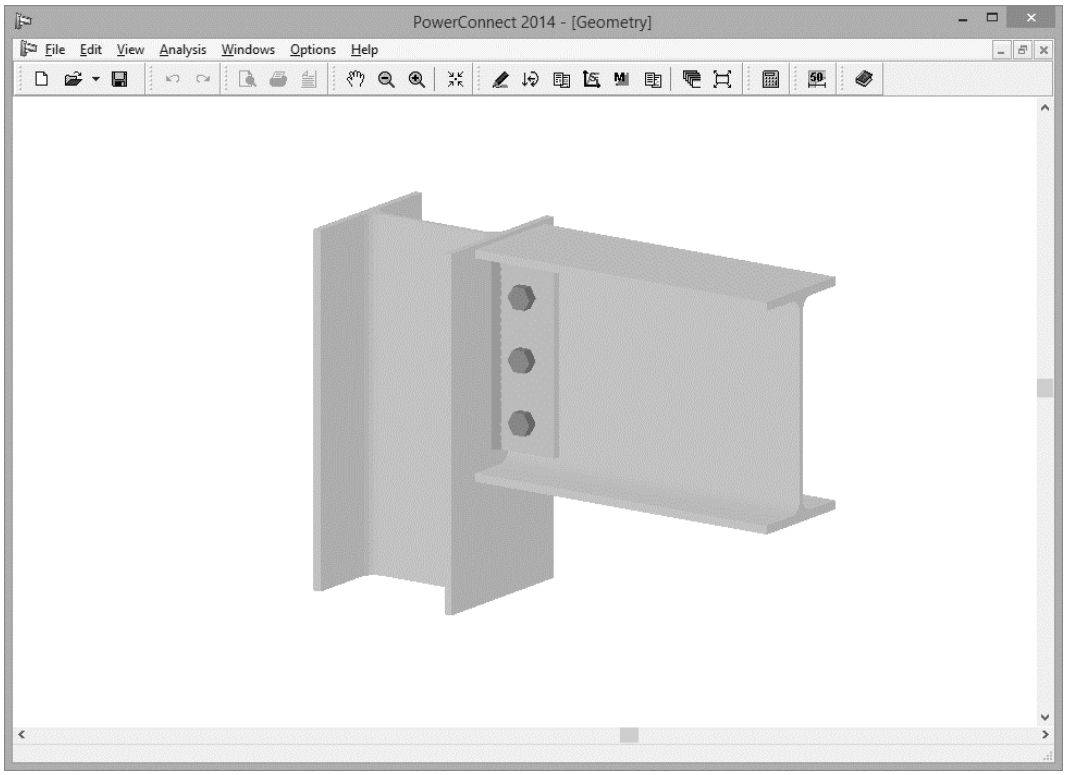


In particular, following parameters should be verified:

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

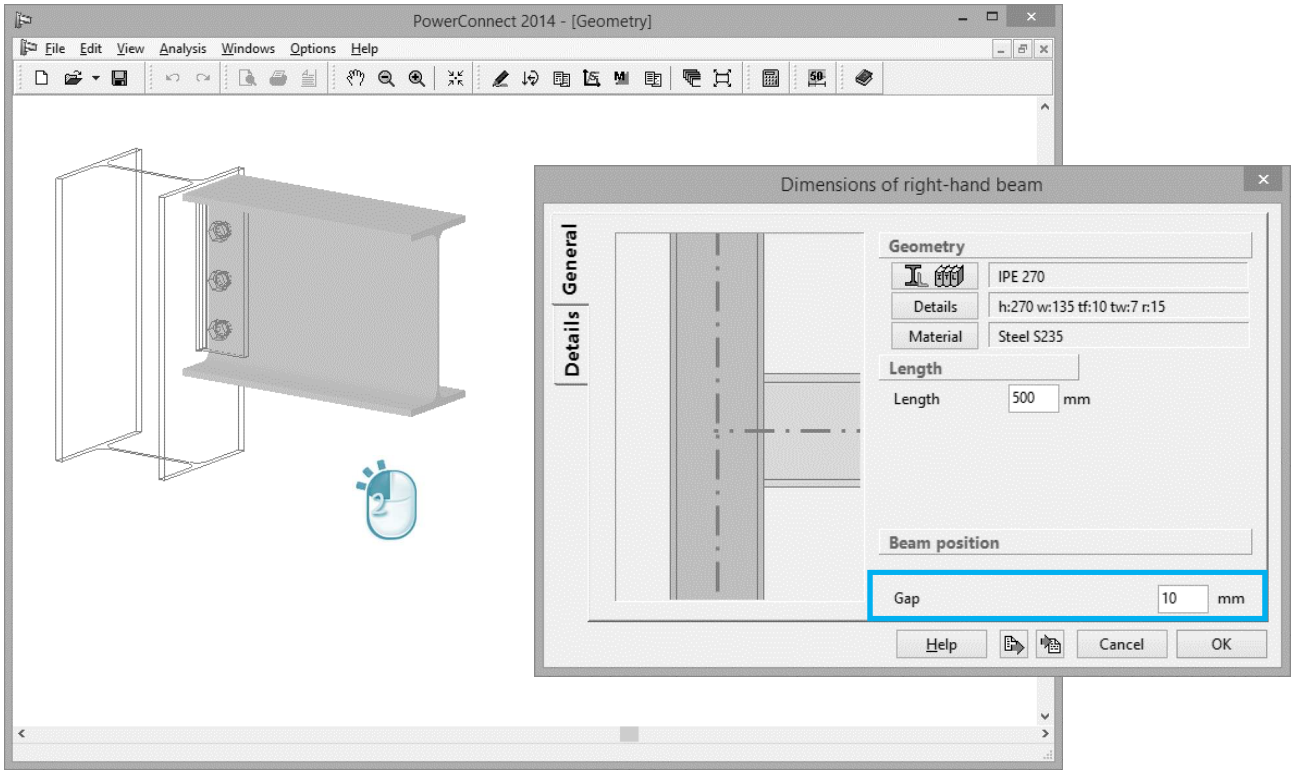
Column and beam length have no impact on the analysis results whatsoever, as all verifications during the analysis are related to shear force only. As this type of connection is verified only for shear force, no bending stiffness is evaluated and the connection is thus assumed to be a pinned connection.

Confirm any modifications to defaults values by means of the ‘OK’-button, to arrive at the connection.

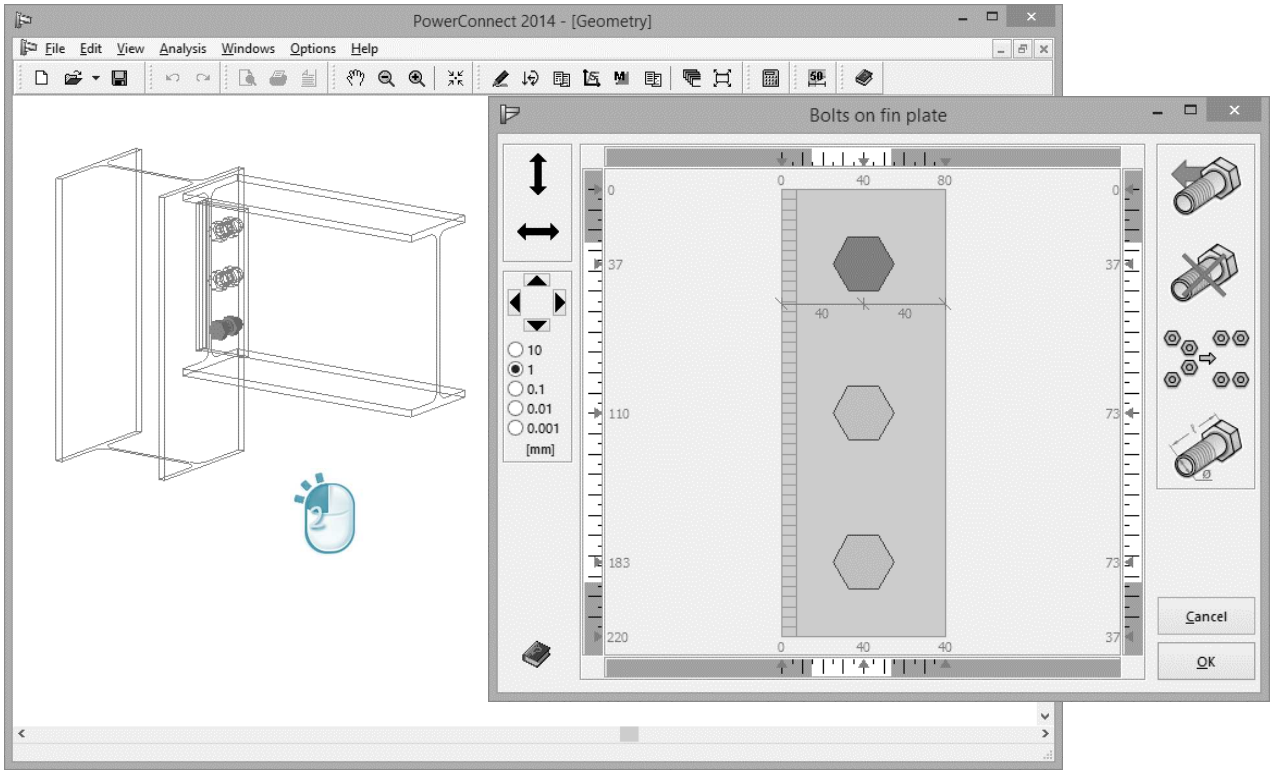


2.6.2 Verifying the geometry model

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.



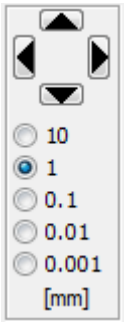
The position of the bolts will also be verified. Double-click on any of the bolts to see that all bolts are positioned centrally with respect to the fin plate.



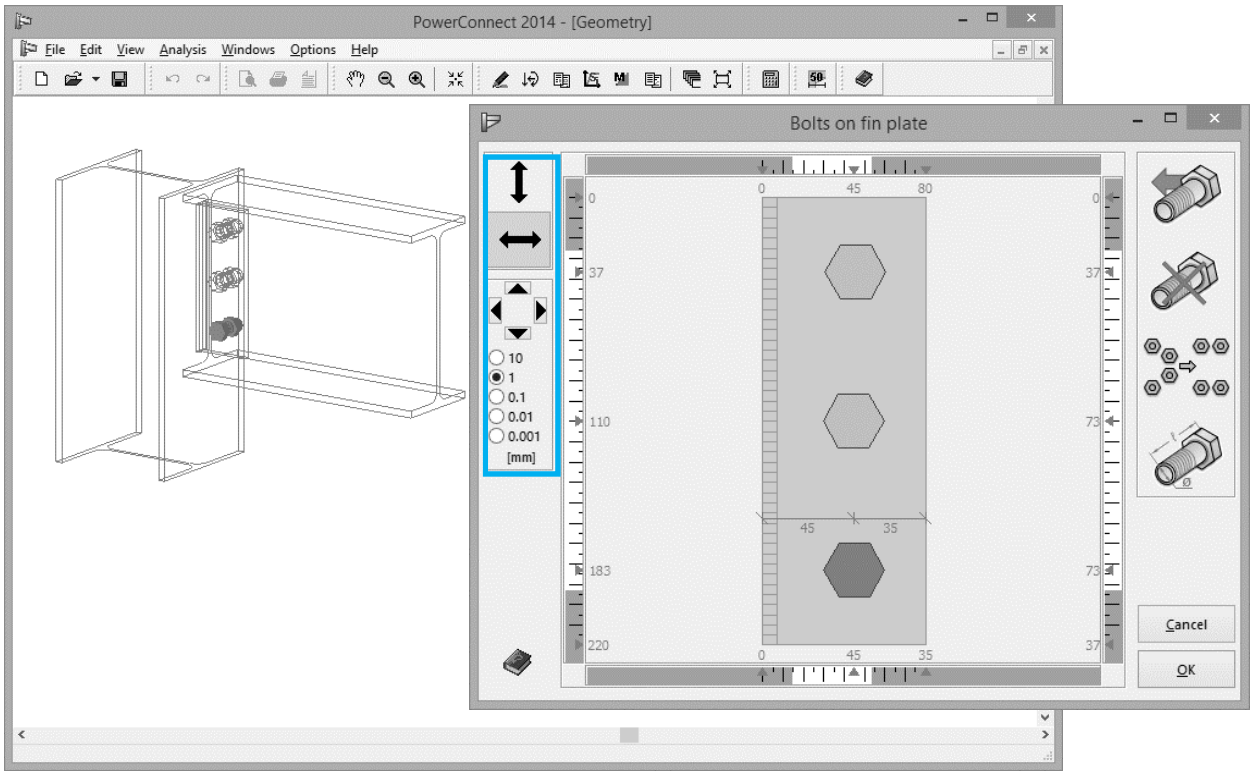
Assume now that all bolts should be moved horizontally to the right over a distance of 5mm. PowerConnect has a number of tools, next to the optimized bolt positioning functions, to define bolt positions manually. To start with, it is important to understand that those tools operate on horizontal bolt 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

These icons allows you to move the selected bolt(s) with the mouse. Use this tool only for roughly repositioning selected bolts.



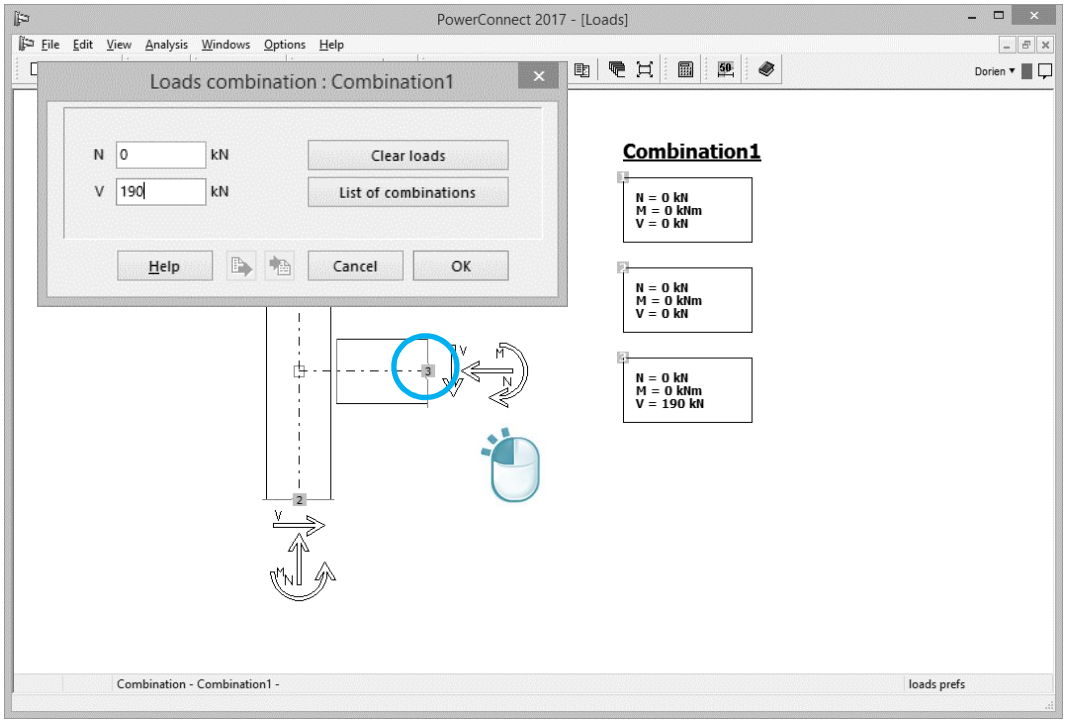
This function can be used in case an exact repositioning of bolts is required. First select the requested precision (10, 1.0, 0.1, 0.01 or 0.001 mm – select 1mm for the current application. Then press the icon five times to move the selected bolted by 5mm to the right. Repeat this operation for the other bolts, to arrive at the status illustrated below.



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

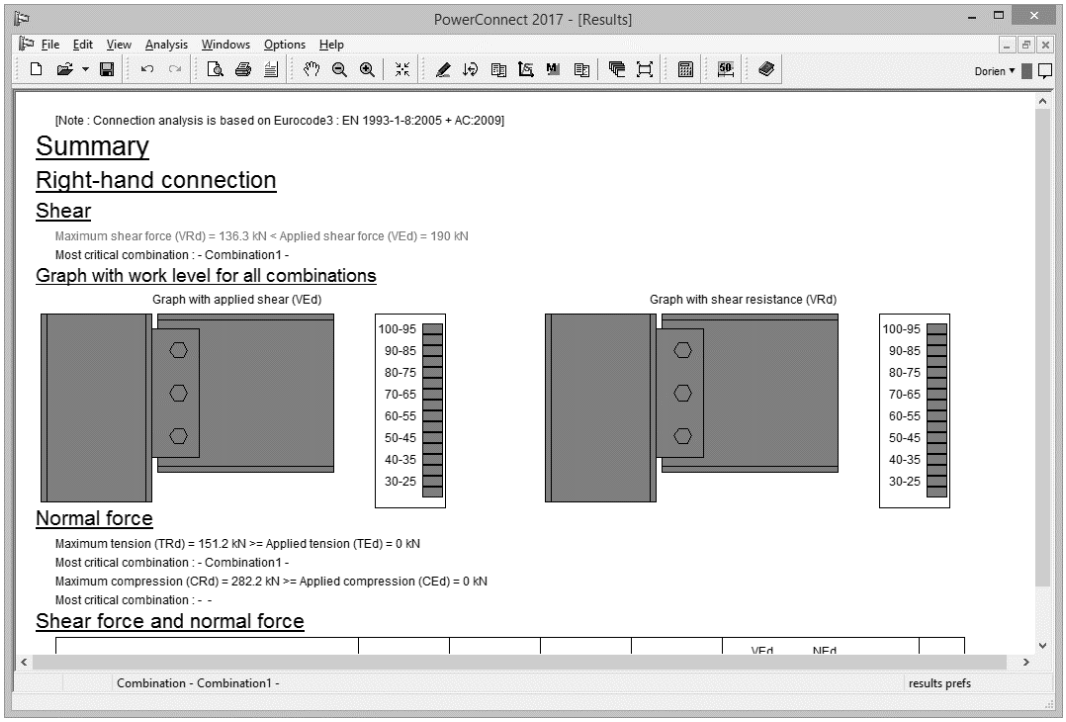
2.6.3 Defining the loads

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 190kN. As a result, the contents of the ‘Loads’-window should look as follows:



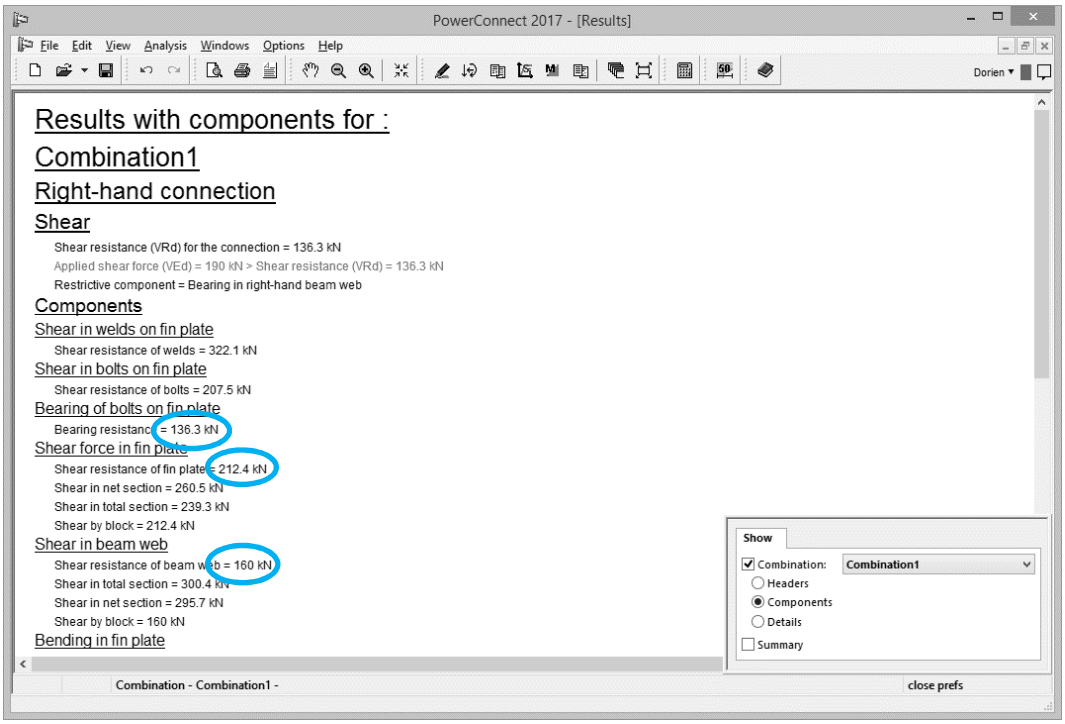
2.6.4 Running the connection design analysis

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 graph on the lift indicates that the bolts, the fin plate and the beam web are the most critical elements of the connection.

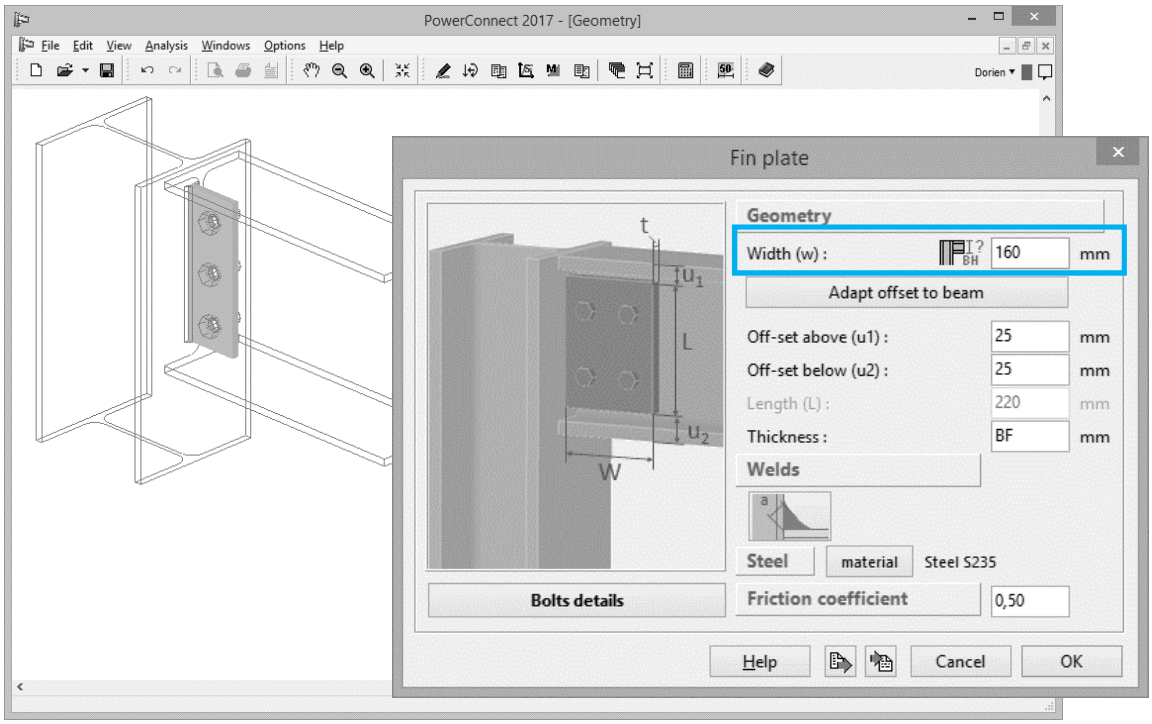
To better understand the connection's actual failure mechanism, switch to more detailed reporting by clicking with the mouse on the 'Results Preferences' field at the right hand bottom of the PowerConnect window. Choose for 'Combination 1' results 'with components'.



This information confirms that the connection's resistance is limited by the bearing resistance of the bolts (136,3kN). Secondly, the shear resistance of the beam web is limiting (160kN).
Furthermore, it is clear that maximum shear resistance of the fin plate (212,4kN) is 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.

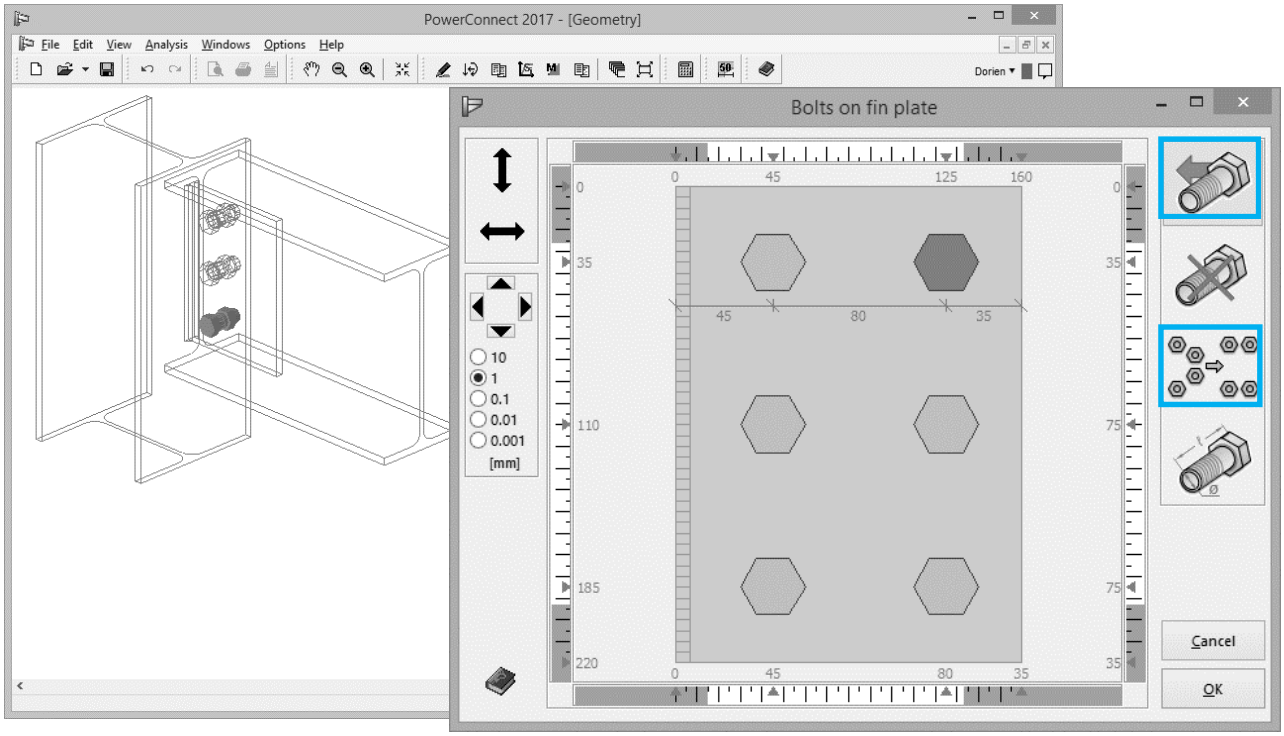
2.6.5 Improving the connection design

Considering previously specified minimum distances between bolts, this can only be achieved by increasing the fin plate width from 80mm to 160mm. Just double-click on the fin plate and adapt the width in the dialogue window that appears:



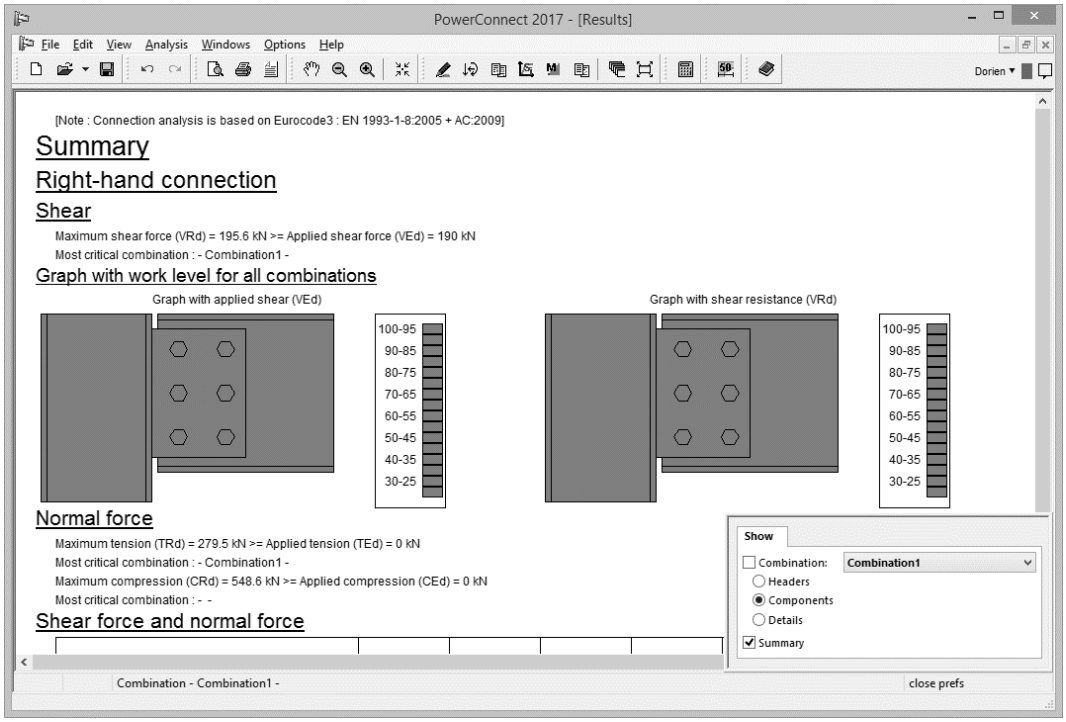
Next, a bolt will be added on each bolt row. Proceed as follows:

- Double click one of the bolts.
- 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

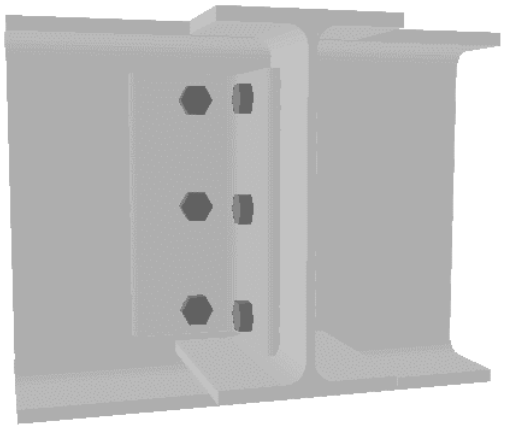
Running the connection design analysis on the modified connection will produce following results summary:



Connection shear resistance has been increased from 136,3kN to 195,66kN, which is sufficient to resist the applied shear load of 200kN.

In §3.3 you can read how to make de report for this connection.

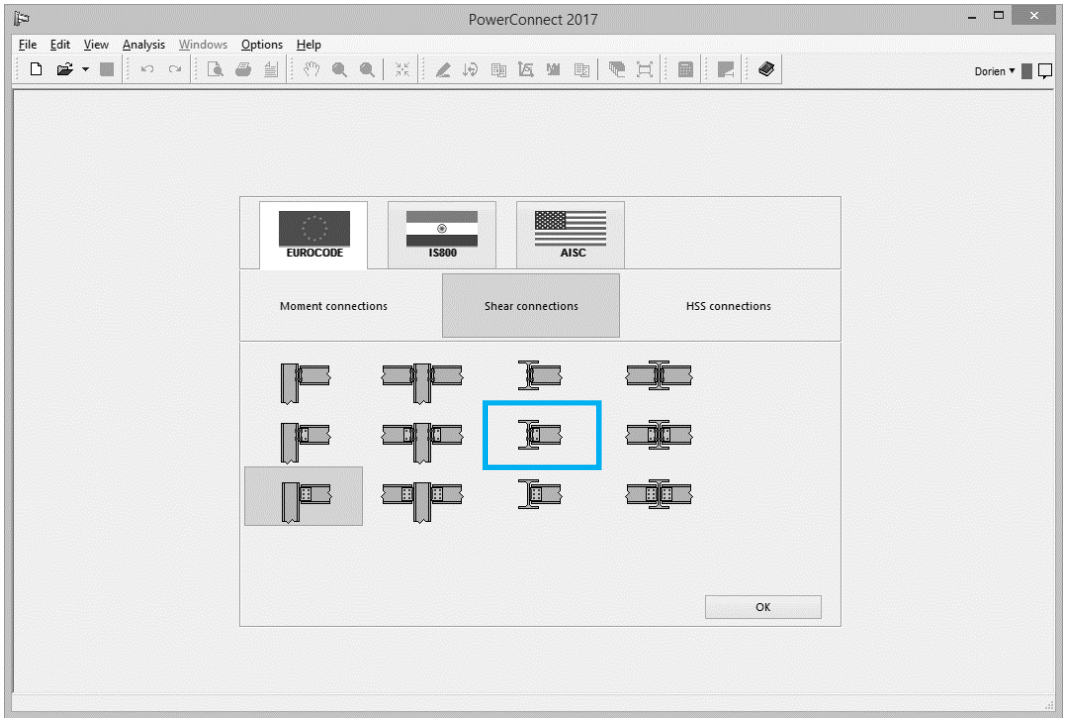
2.7 Tutorial 7: shear connection – beam to beam web with bolted angle cleats



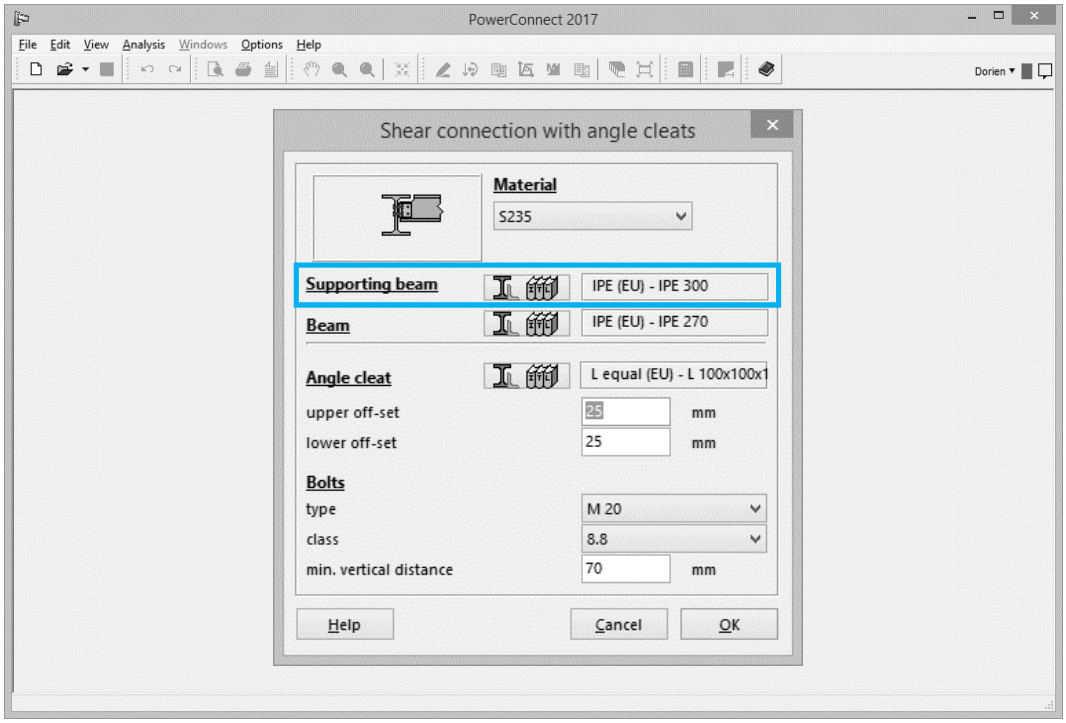
As a second example for shear connections, a beam to beam connection, using bolted angle cleats will be considered.

2.7.1 Setting up the model

Use the icon  to define a new project. In the navigation window, select the connection type labelled “Angle cleat bolted to beam web”.



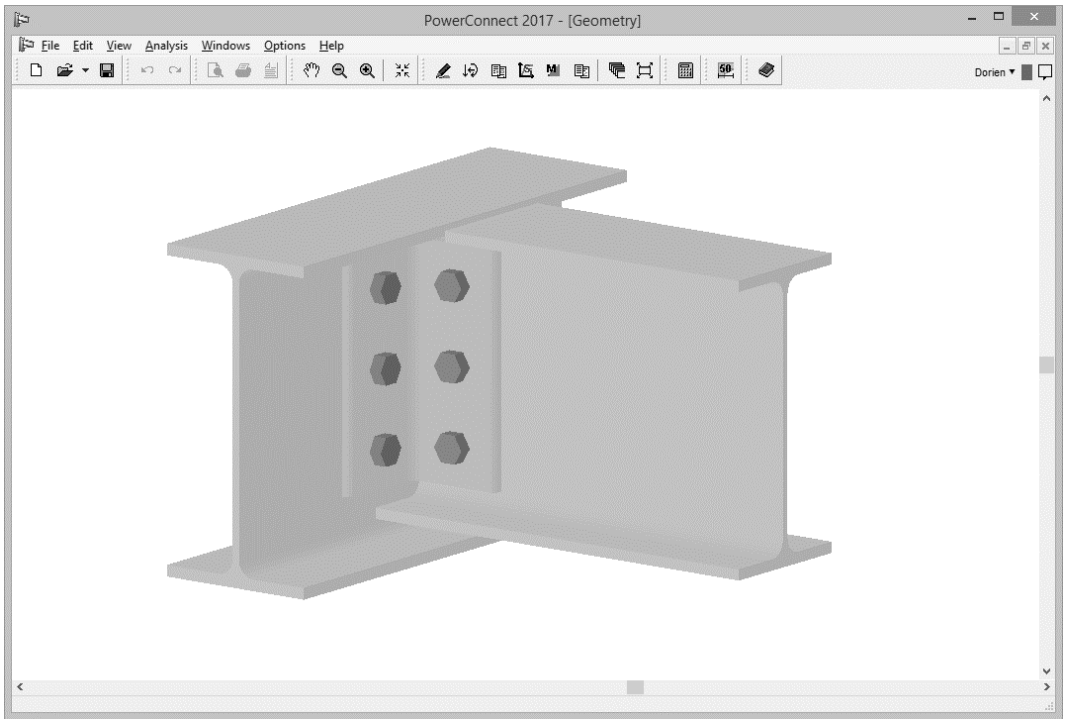
Confirm this choice with the ‘OK’-button and then define further details in the following dialogue window.



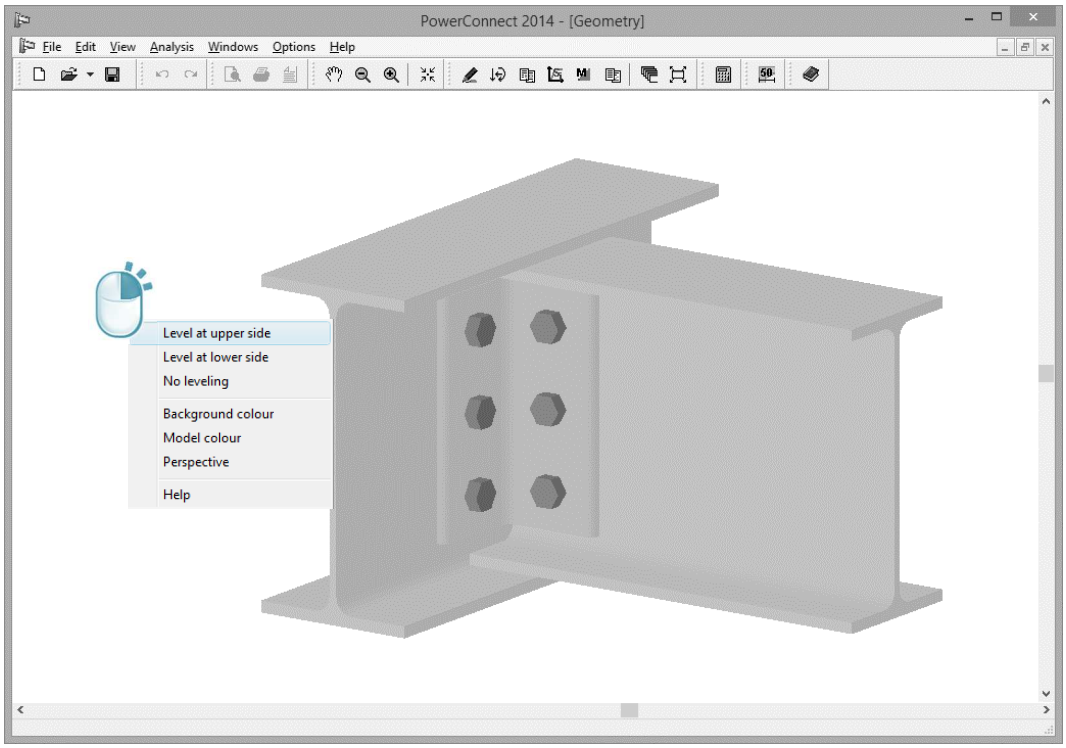
In particular, following parameters should be verified:

- 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.

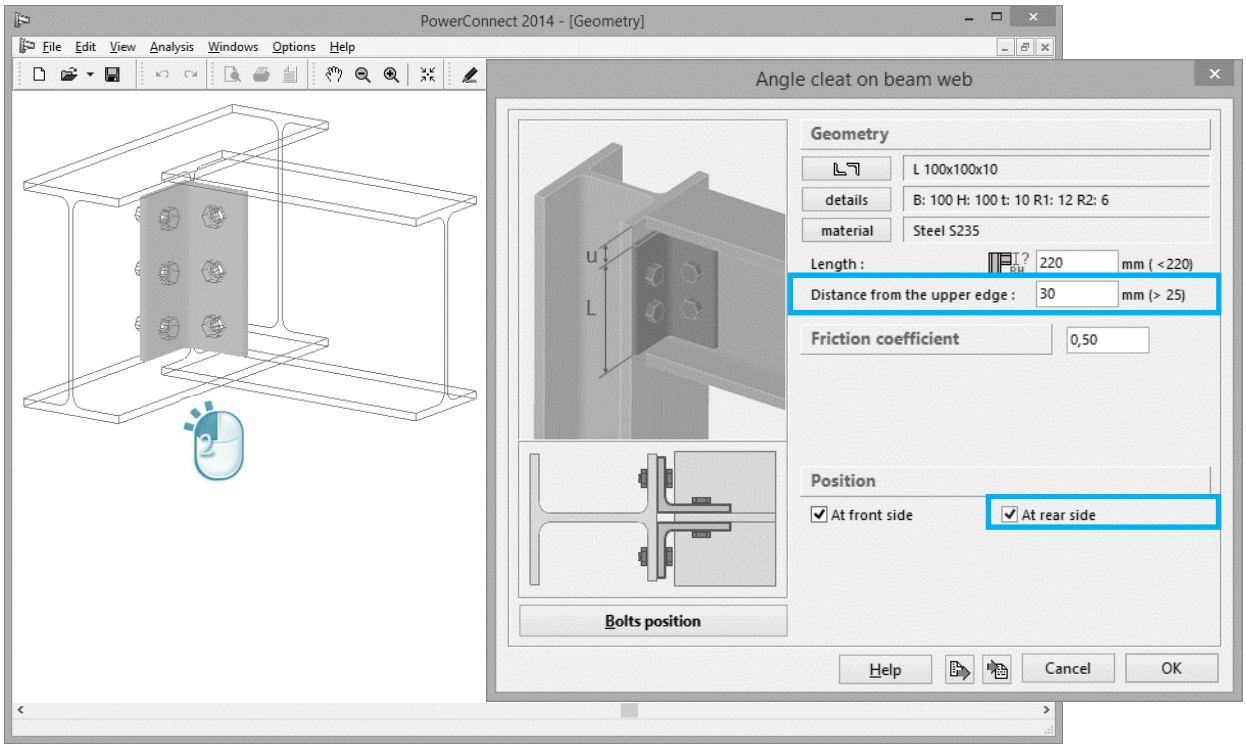


With this particular example, remain with the upper surfaces aligned.

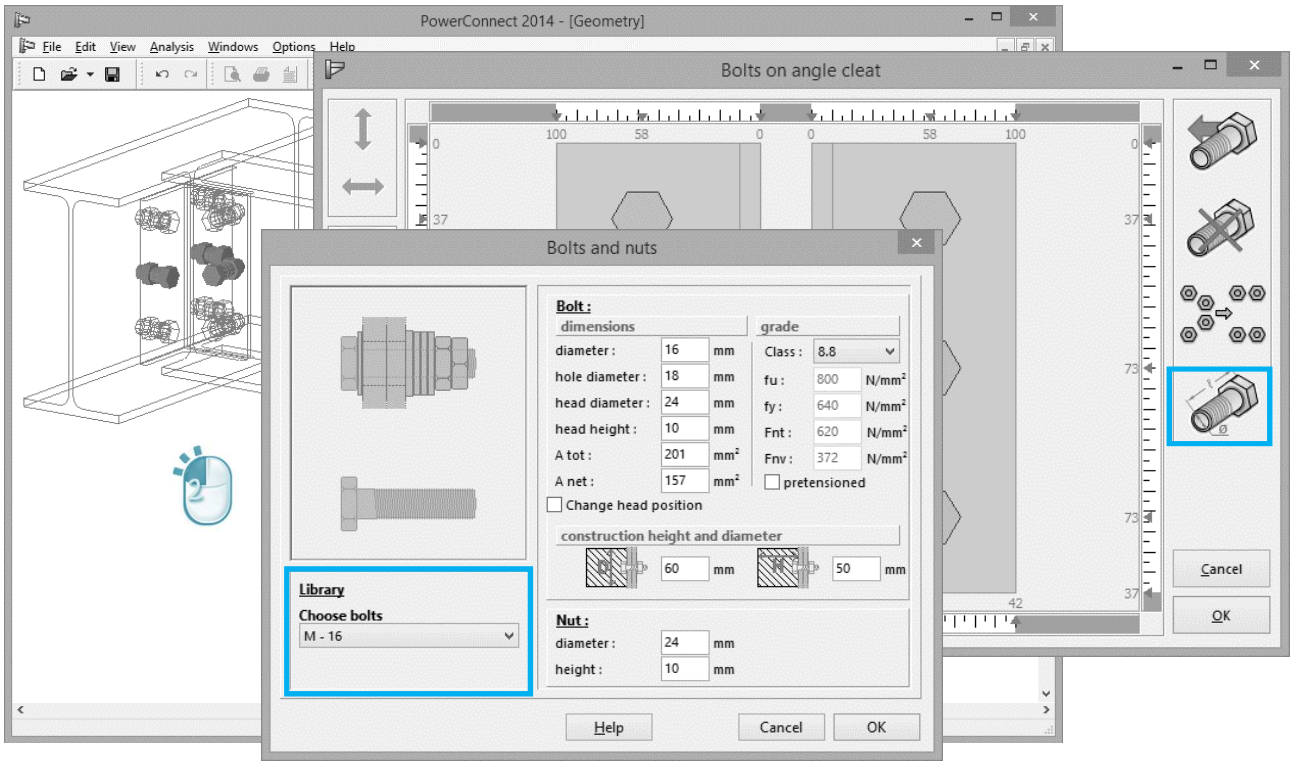
2.7.2 Verifying the connection elements

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.

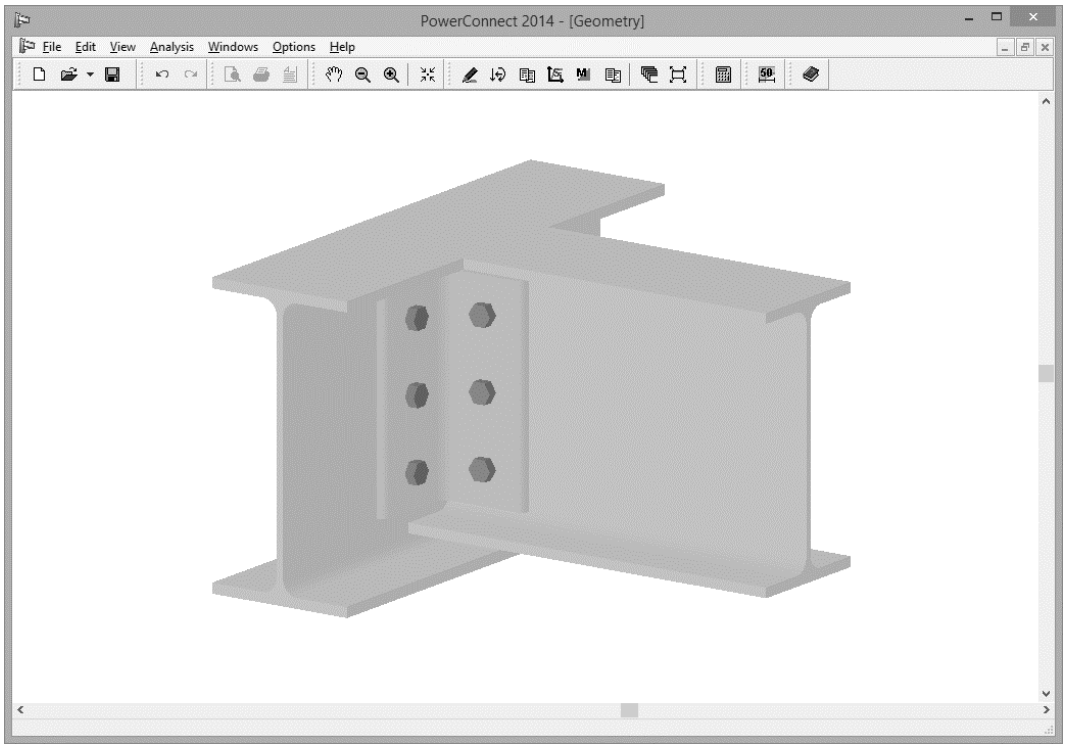
Now double-click on the angle cleat, and verify if angle cleats have been foreseen at both sides of the beam web. If not, make sure both the "Front side" and "Rear side" options have been checked.



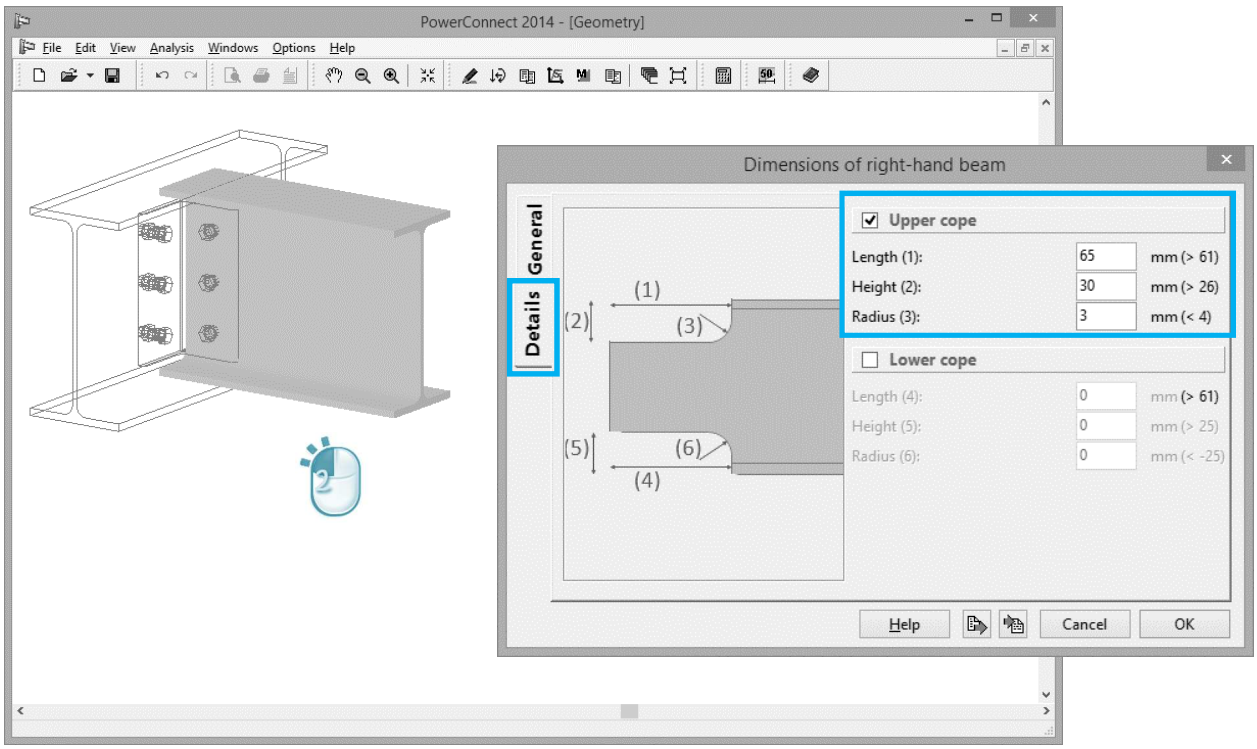
Finally, have a closer look at the bolts. Double-click on one of them, and use the  icon to change the bolt type from M20 (default bolt type) to M16 (8.8 grade).



Confirm this change to arrive at the following bolt lay-out:

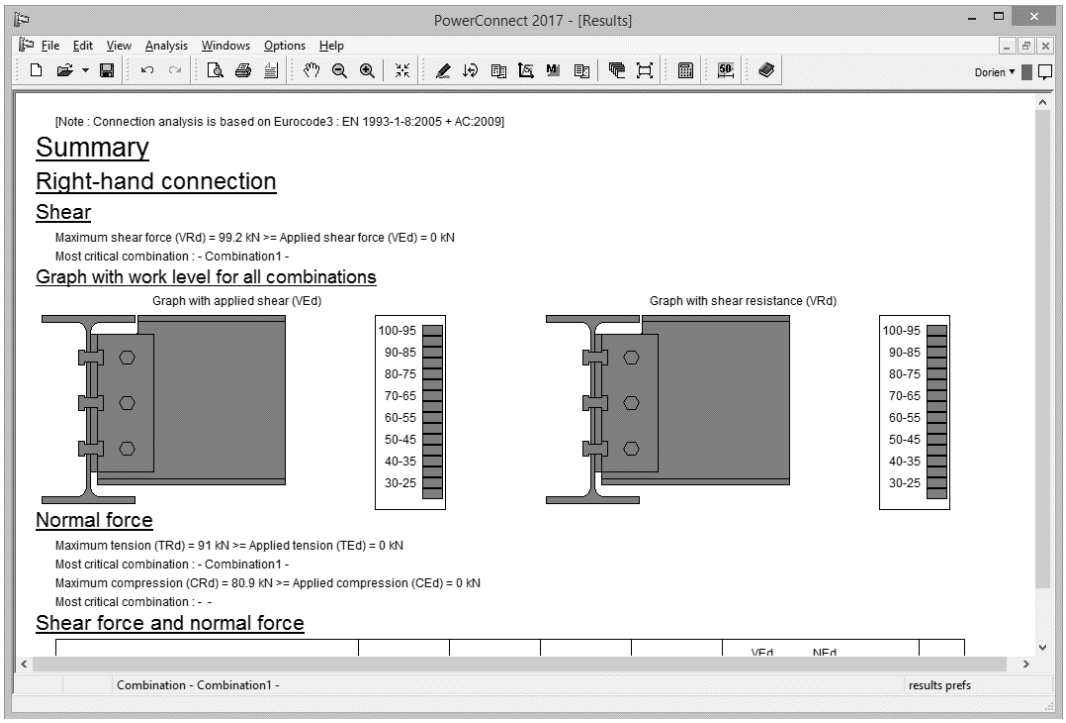


Provide a cope at the top of the beam. Double click the beam and complete the tab page ‘Details’ as follows:

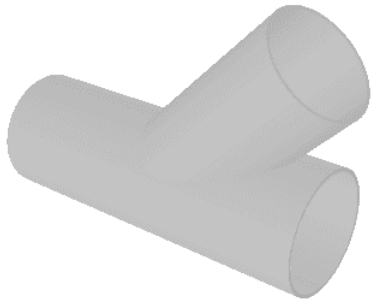


2.7.3 Running the connection design analysis

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

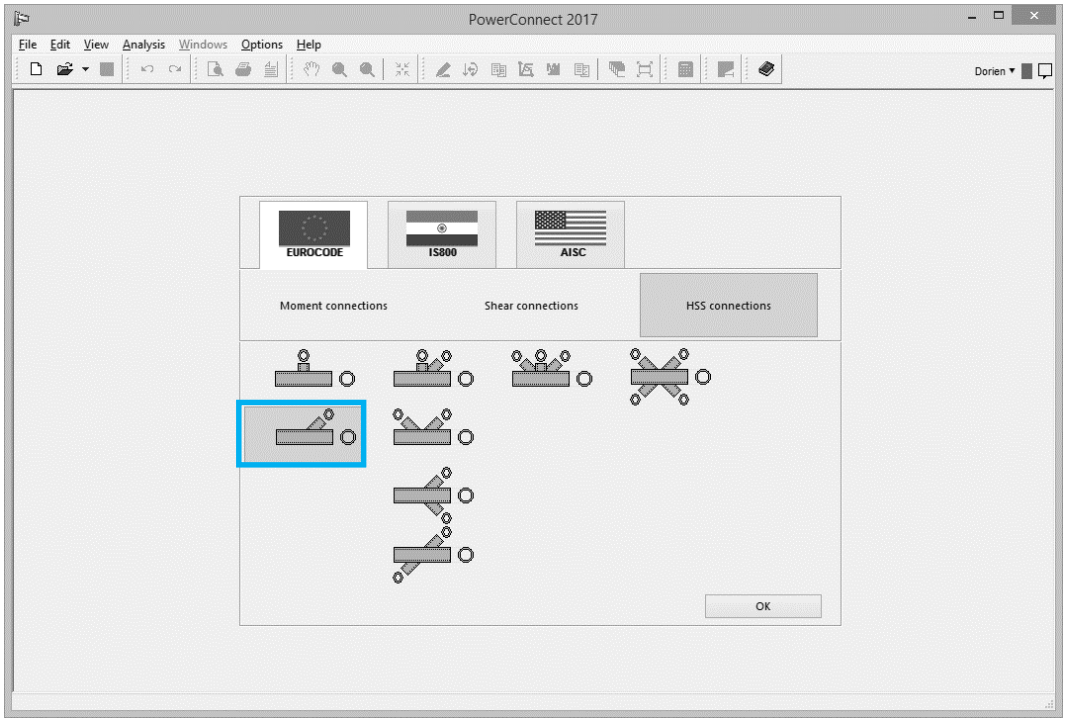


2.8 Tutorial 8: HSS connection (circular members)

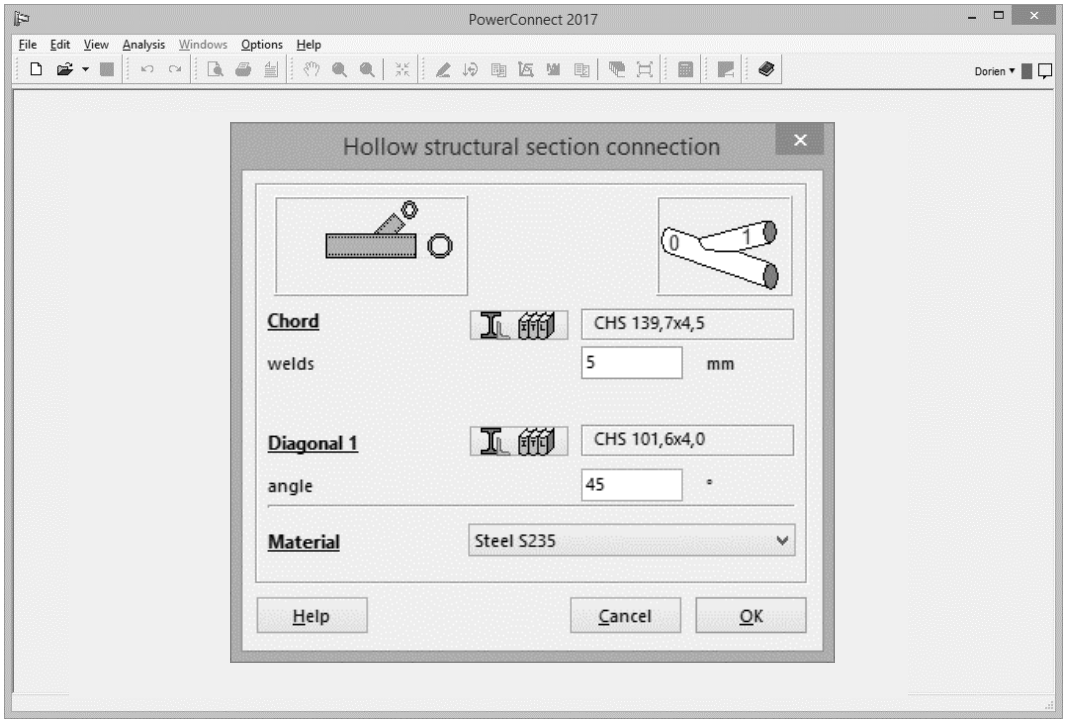


2.8.1 Setting up the model

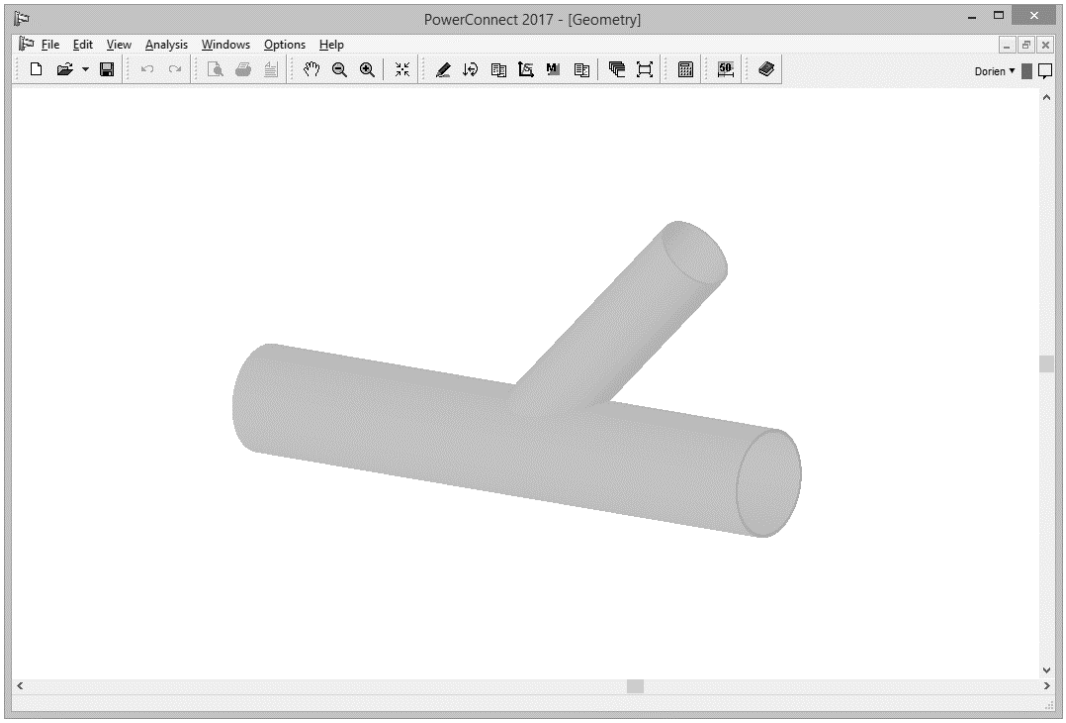
Click on the icon  to start a new PowerConnect project and select the **HSS Connections** and a 'Y-connection' from the navigation window.



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.



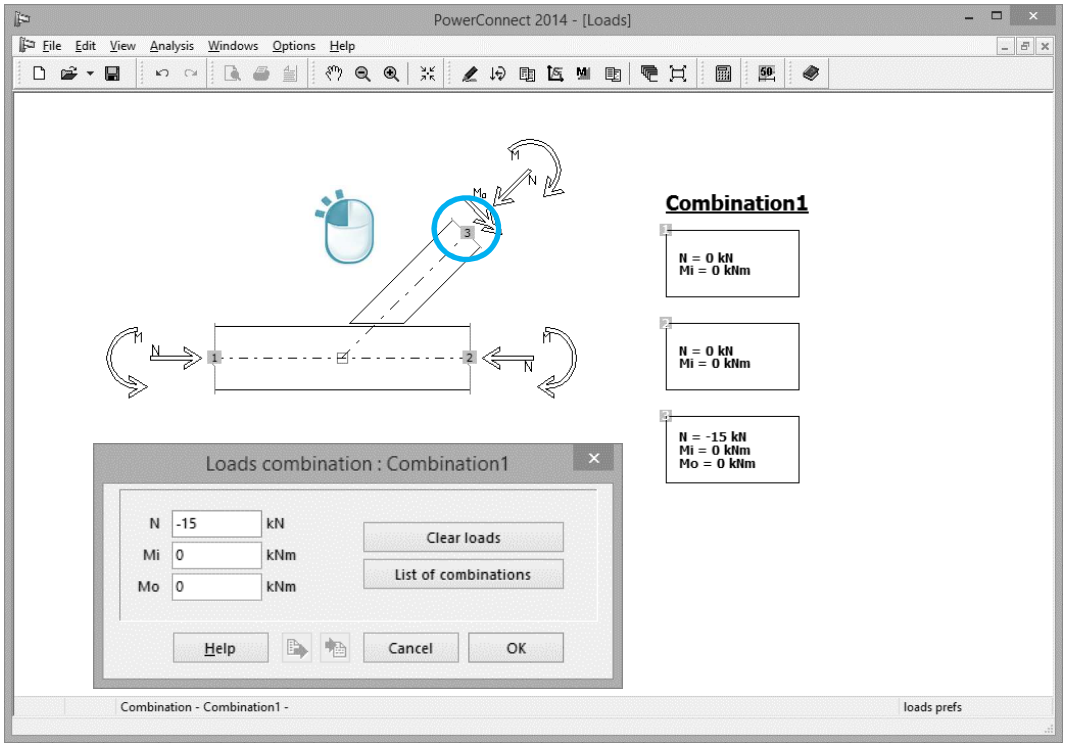
If wanted, further details on the hollow structural sections can be obtained (or modifications can be defined) by double-clicking on the corresponding members in the 3D visualization of the connection geometry.



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).

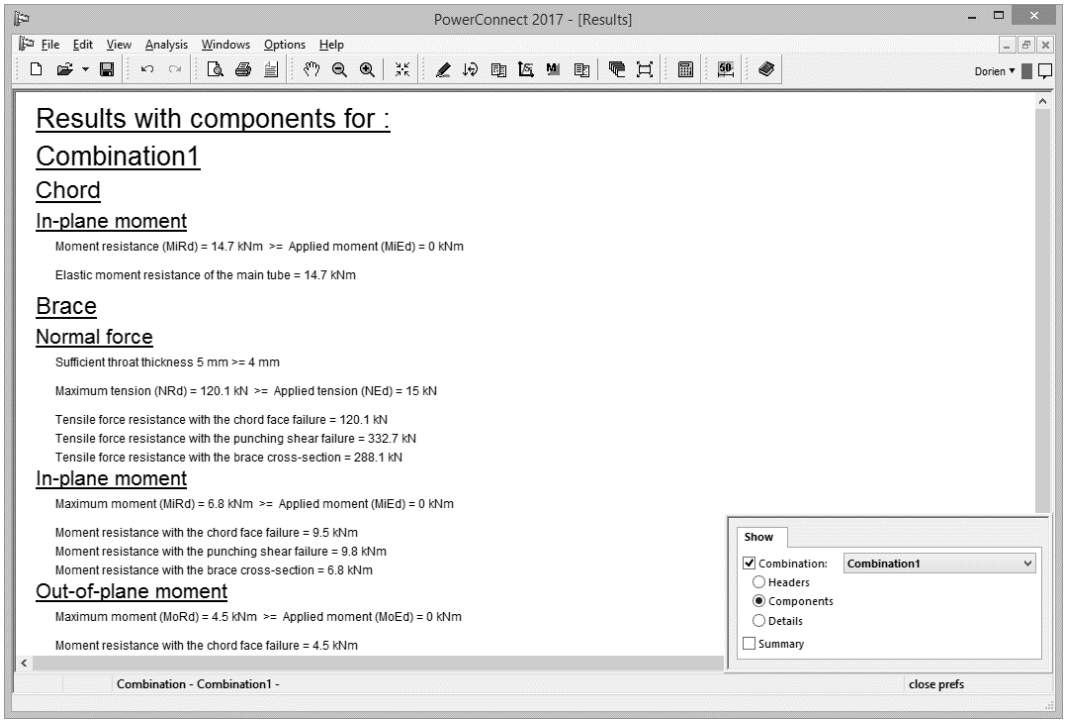


2.8.3 Running the design analysis

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 analysed by PowerConnect. Limitations are mostly related to the type of loads that can be considered during the analysis.

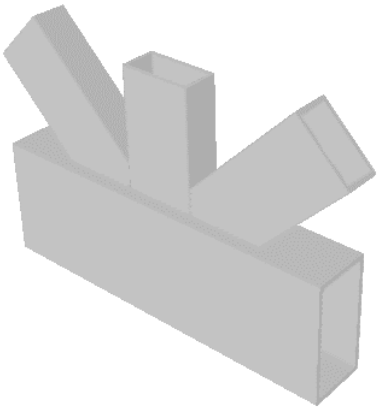
For the HSS connection that has been defined, the analysis will deliver following results.



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.

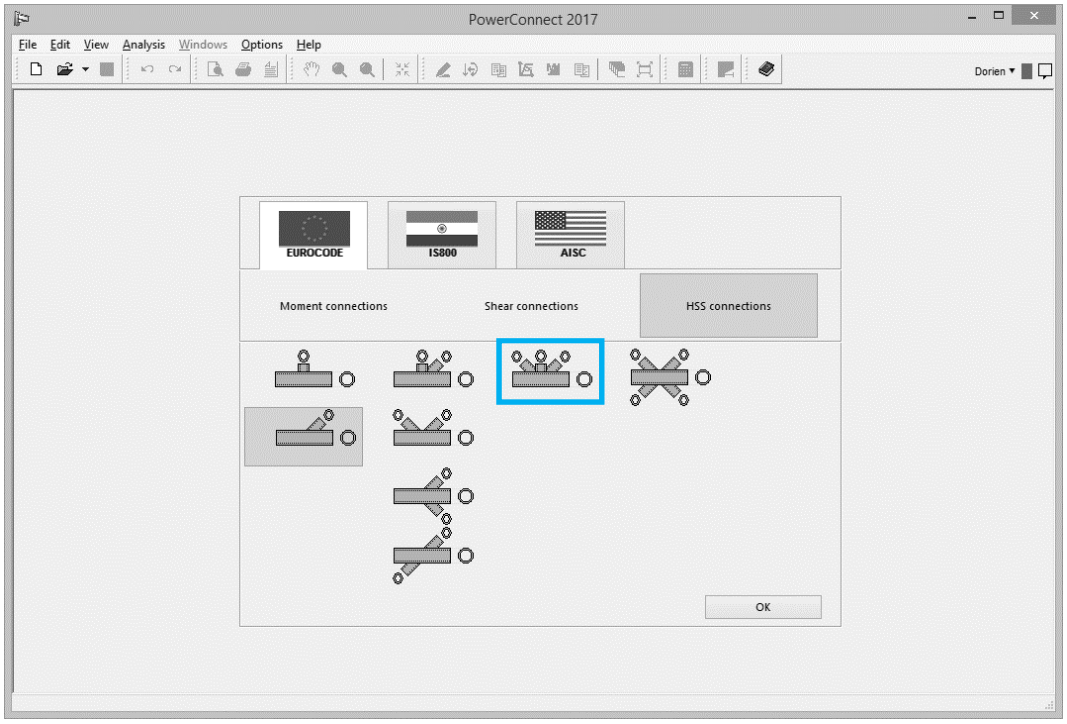
PowerConnect will furthermore use those values to summarize the overall “loading level” of the connection, by comparing applied loads to maximum resistance using a dedicated combination formula. As such, judgment of connection strength really becomes a straightforward matter.

2.9 Tutorial 9: HSS connections (rectangular members)

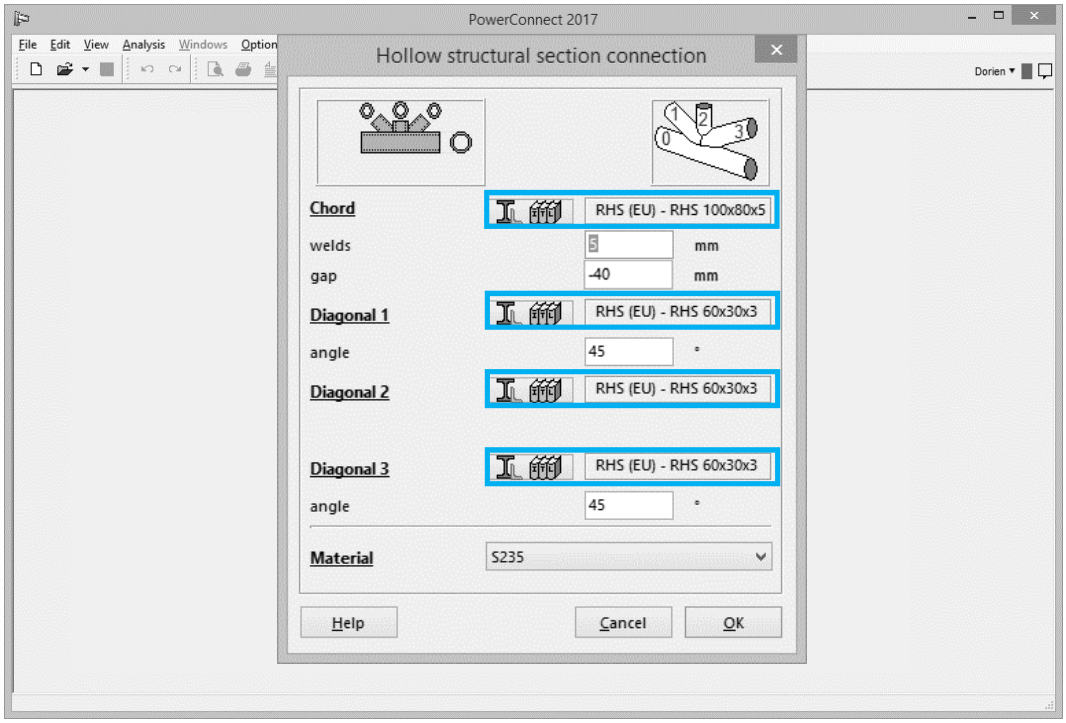


2.9.1 Setting up the model

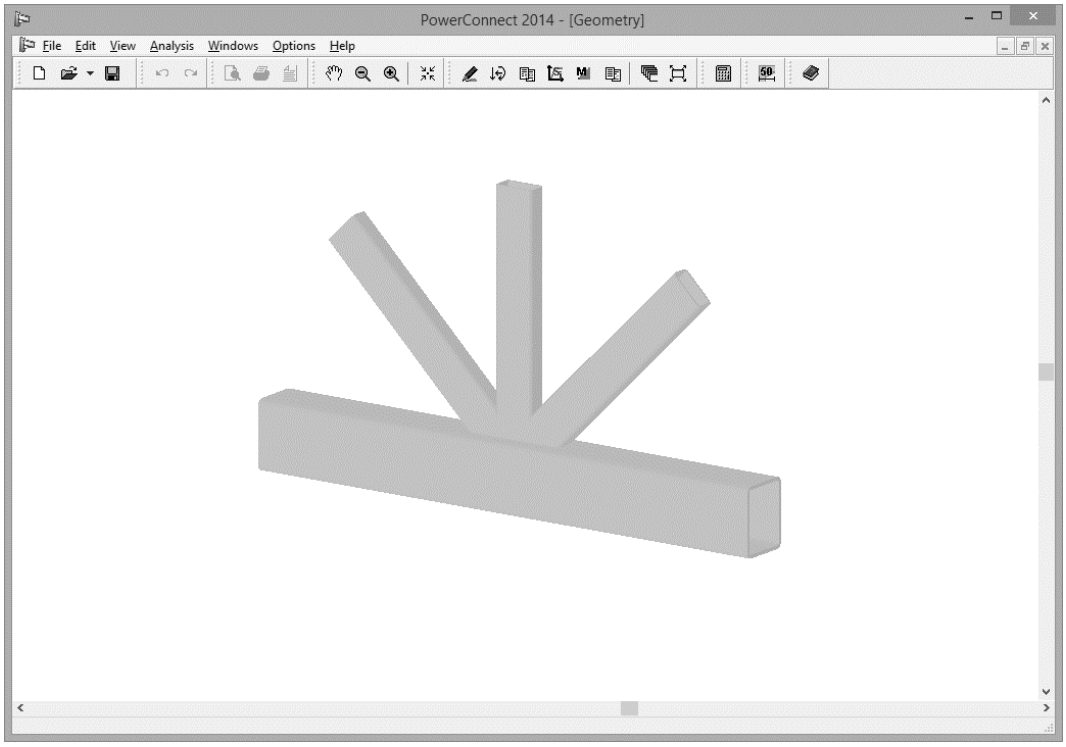
Click on the icon  to start a new PowerConnect project and select the **HSS Connections** and a 'KT-connection' from the navigation window.



For the main chord, a RHS100x80x5 section will be used, whereas for a RHS60x30x3 section will be used for all diagonals. Accept all other values as proposed by default by PowerConnect.

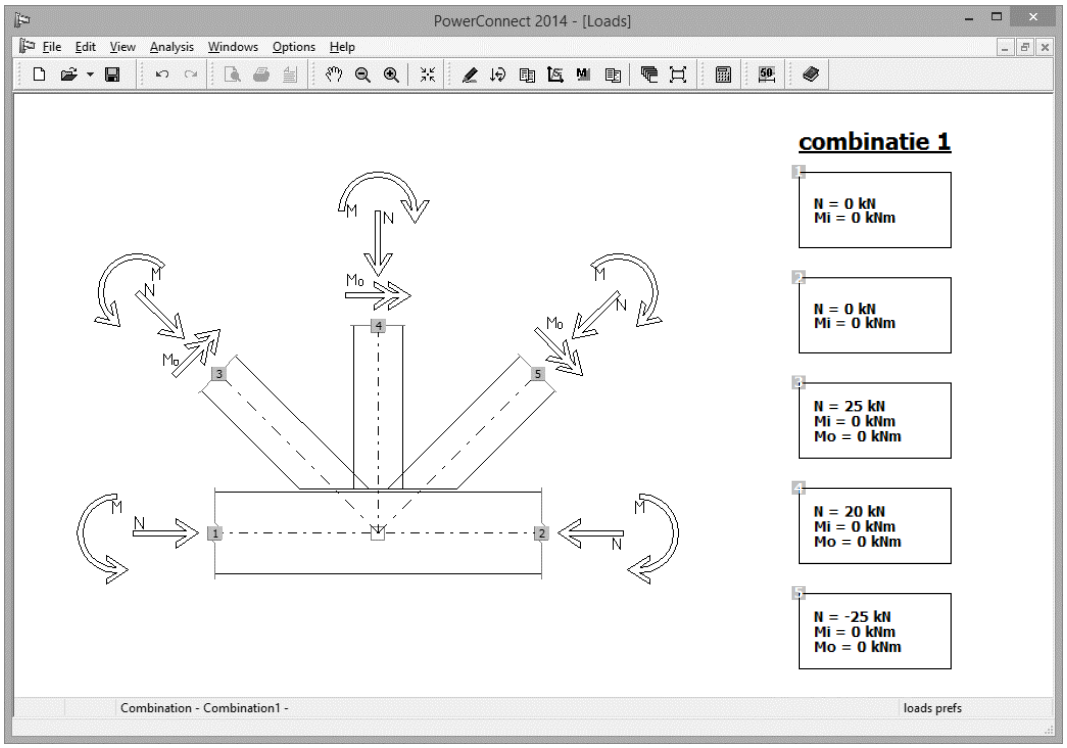


Result:



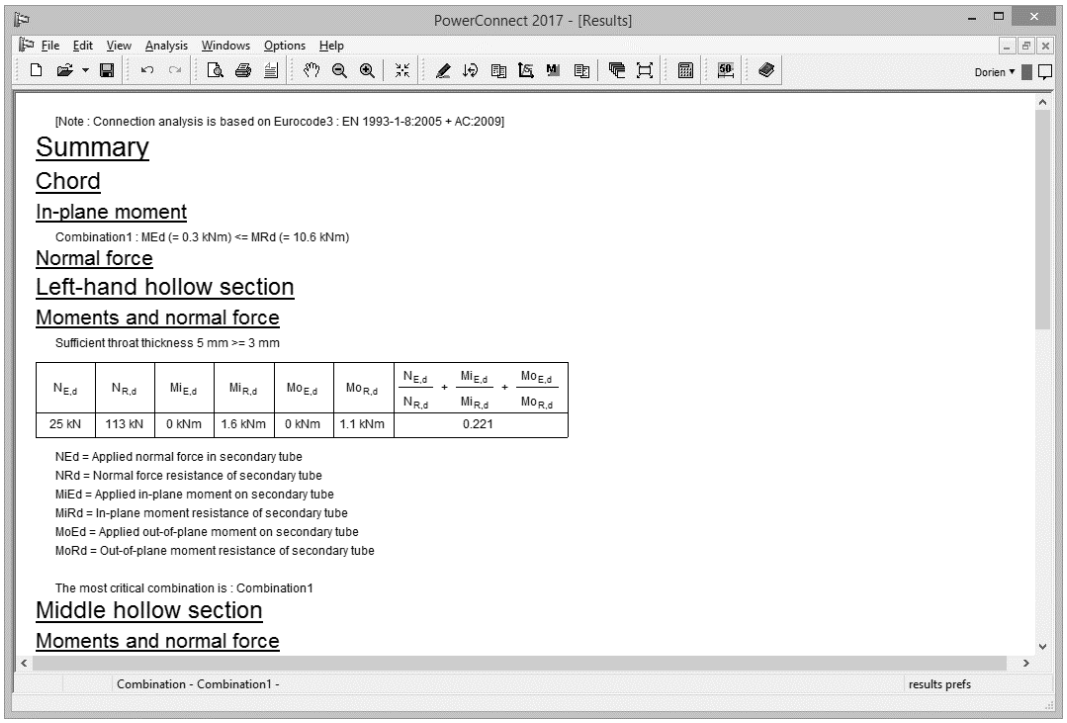
2.9.2 Defining the loads

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.



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.

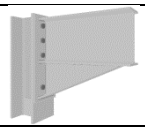
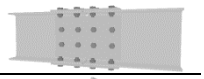
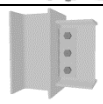


$N_{E,d}$	$N_{R,d}$	$Mi_{E,d}$	$Mi_{R,d}$	$Mo_{E,d}$	$Mo_{R,d}$	$\frac{N_{E,d}}{N_{R,d}} + \frac{Mi_{E,d}}{Mi_{R,d}} + \frac{Mo_{E,d}}{Mo_{R,d}}$
25 kN	113 kN	0 kNm	1.6 kNm	0 kNm	1.1 kNm	0.221

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:

Section	Tutorial contents	Design code	Connection
§3.1	Tutorial 1: bolted beam to column flange	EC3	
§3.2	Tutorial 2: bolted splice	EC3	
§3.3	Tutorial 3: shear connection – beam to column flange with fin plate	EC3	

3.1 Tutorial 1: bolted beam to column flange

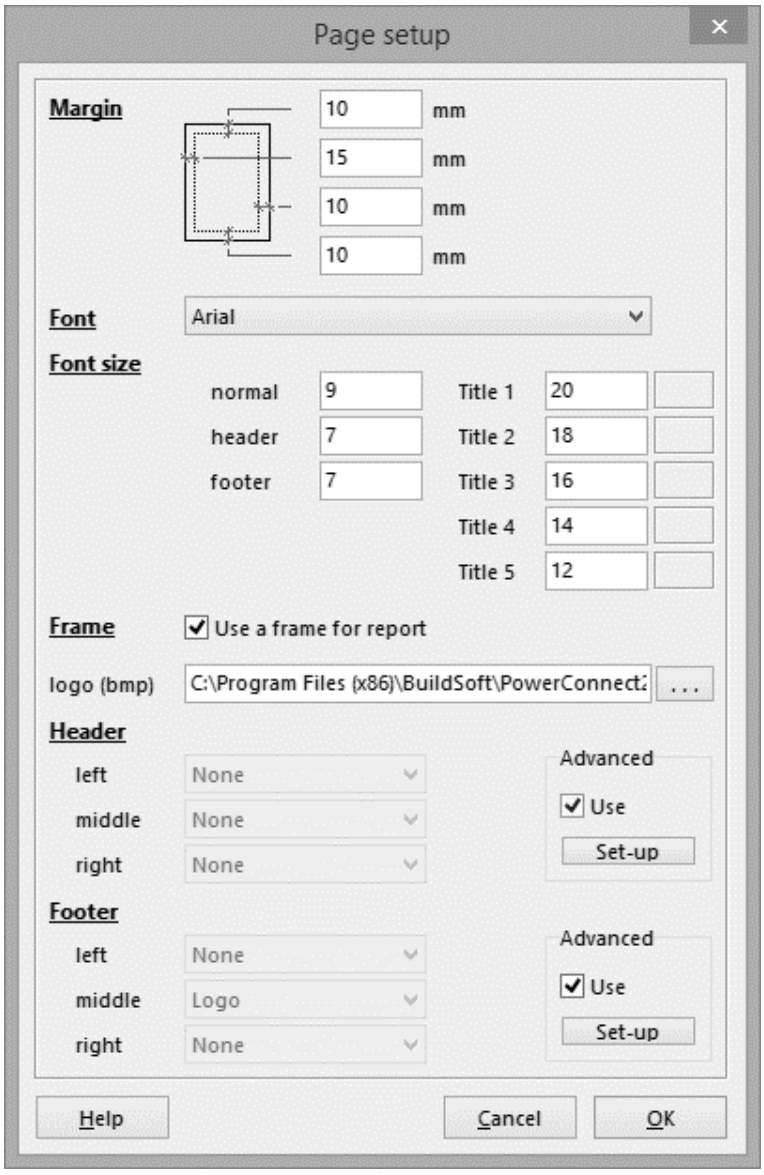
This reporting tutorial is based on the model created in §2.1.

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.

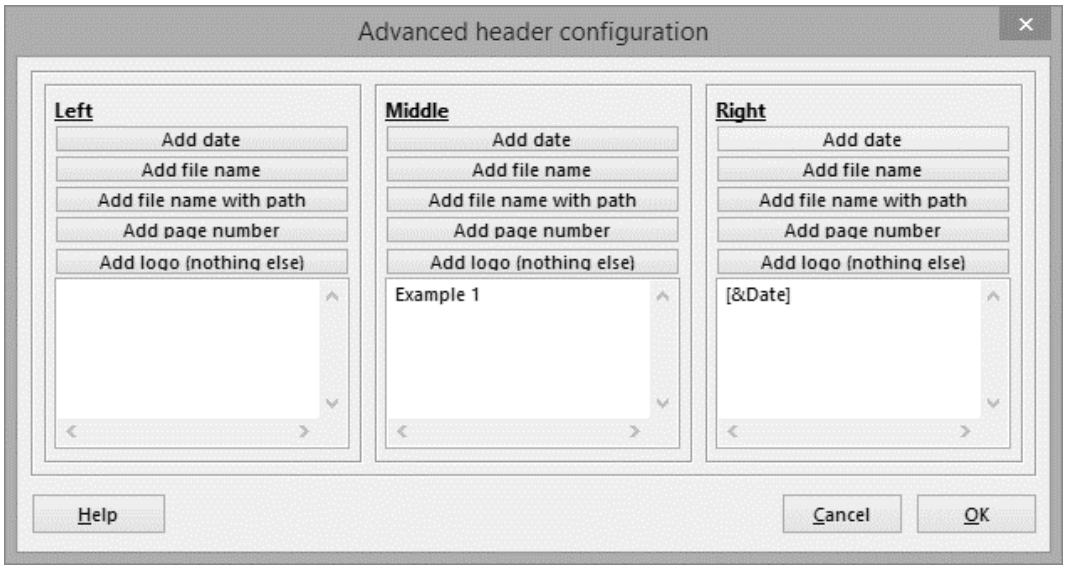
To start the page setup, use the menu command ‘File – Page setup’, so that the dialogue window below will be launched.

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.

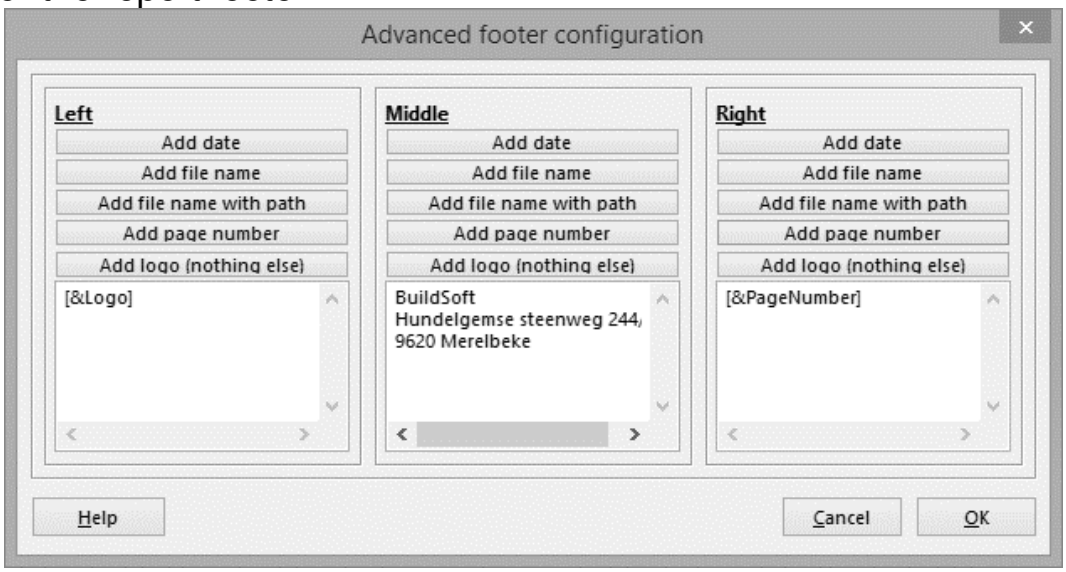


To define report header & footer, the user can either use a number of preconfigured fields or enter into an advanced definition mode. Switch to this advanced definition mode using the button ‘Set-up’, and enter the data as shown below:

- for the report header:



- for the report footer:



3.1.2 Report configuration

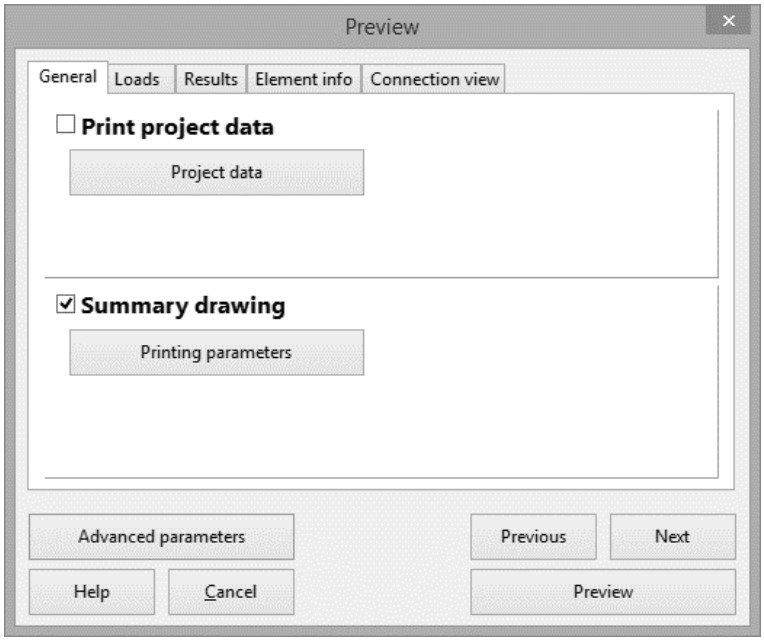
The report can be configured in either one of the following three modes:

- **Preview** (initiated by the icon)
- **Print report** (initiated by the icon)
- **Print report to R**TF (initiated by the).

Irrespective of the chosen configuration mode (preview, print report or print to RTF), the working procedure is the same. In the scope of this tutorial, we will use the **Preview** mode.

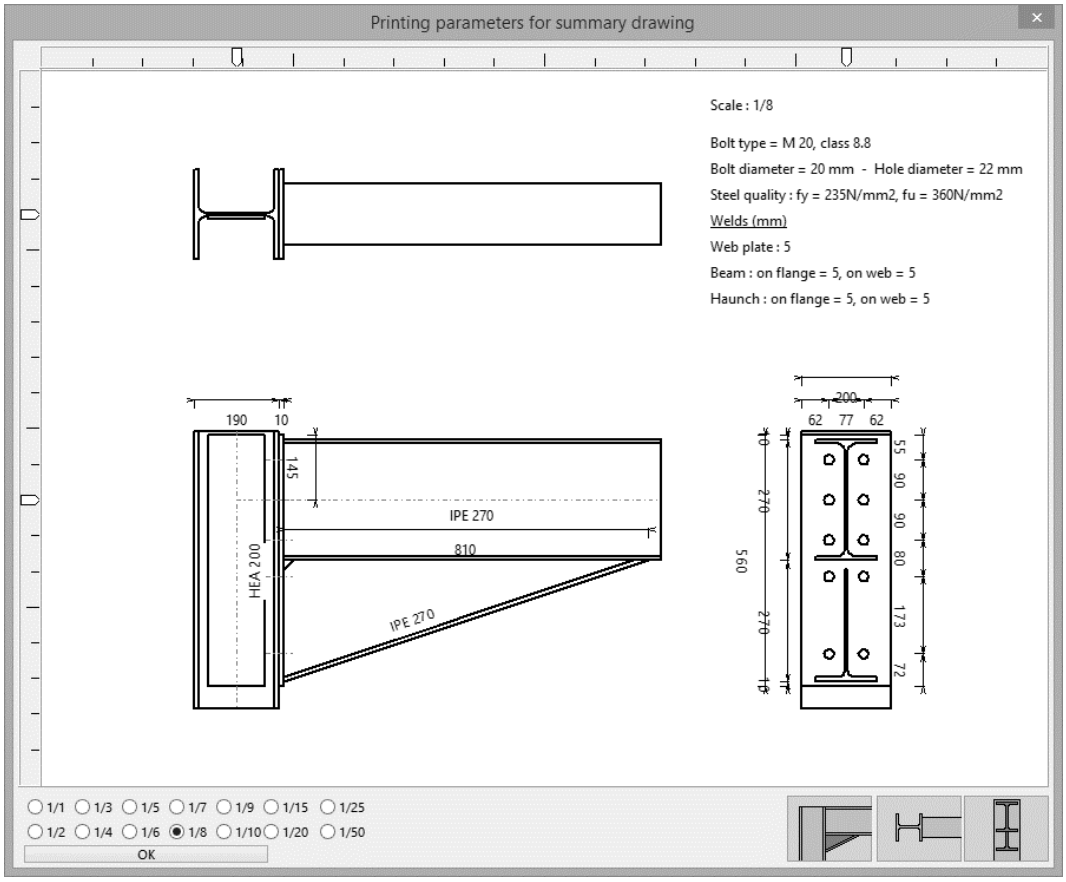
In preview or another configuration mode, 5 tab pages are available in the dialogue window used to define report contents. In this tutorial, only the first 3 tab pages will be used.

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 completed.



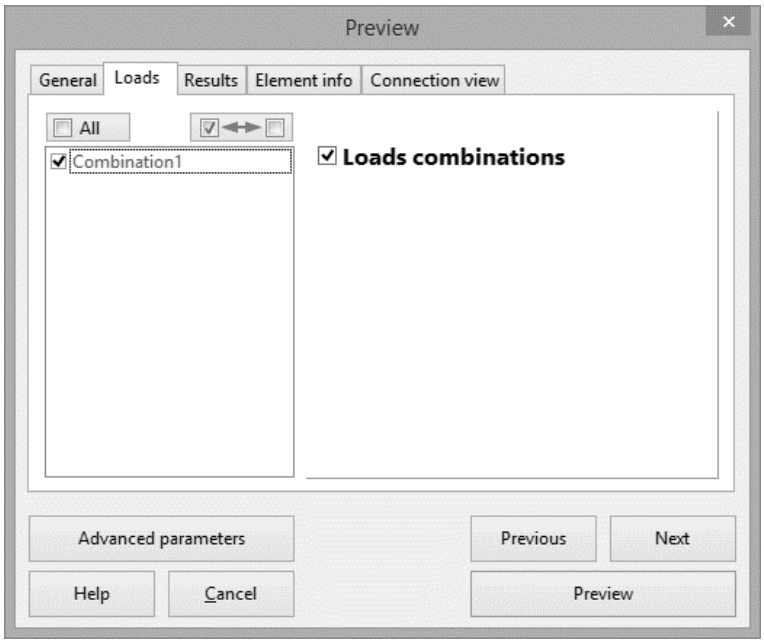
To create the kind of summary drawing shown below

- Select all connection views shown at the right bottom part of the canvas window.
- Use the cursor icons to position the selected drawings in the appropriate positions.
- Select a proper scale (the selected scale of 1/8 or 1/10 is appropriate for a report on A4 paper format).

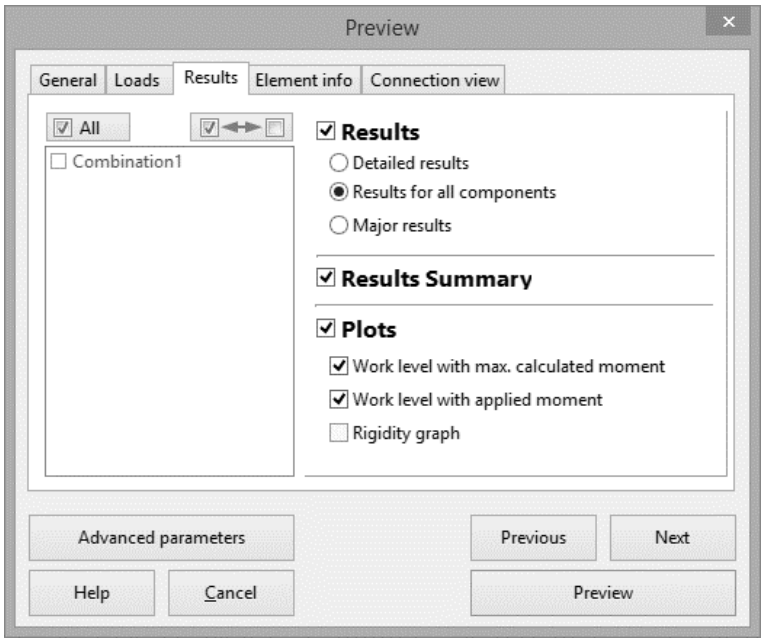


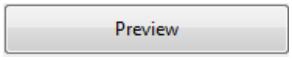
As soon as a proper lay-out has been defined, confirm by means of the ‘OK’-button to return to the definition dialogue window.

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.

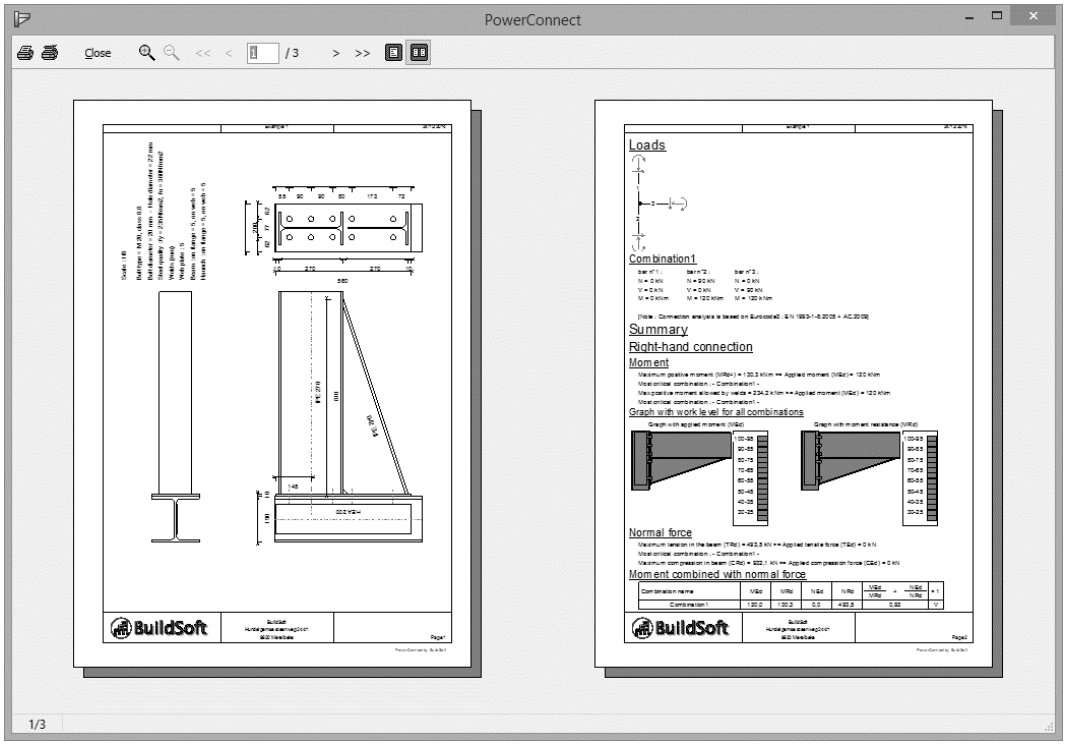


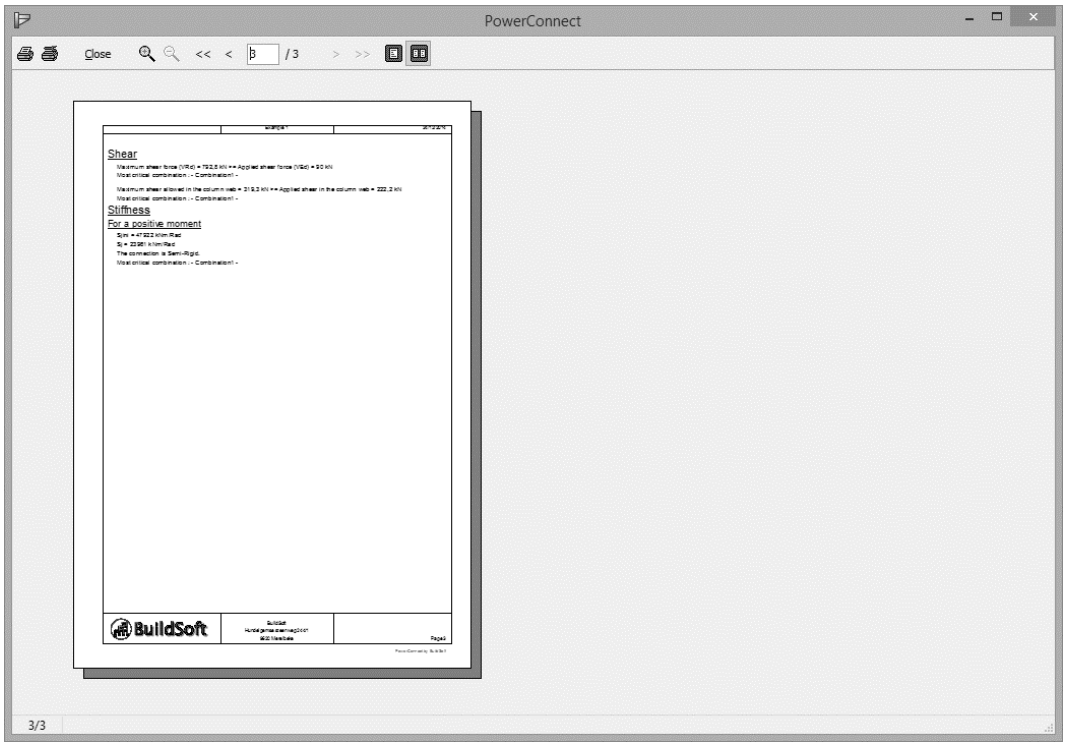
Now switch to the third tab page ‘Results’ to specify how analysis results should be reported. Within this first reporting tutorial, the entire focus is on the creation of a concise analysis report. To do so, select the options ‘Results summary’ and ‘Plots’ as indicated below.



The report configuration can now be considered to be complete, and the report itself can be visualized by means of the 

button at the bottom of the dialogue window. The outcome will be a three page report as shown below.





To actually print the report, use the icon  to send the report to the selected printer. If not, use the 'Close'-button to return to the PowerConnect working environment.

3.2 Tutorial 2: bolted splice

This reporting tutorial is based on the model created in §2.5.

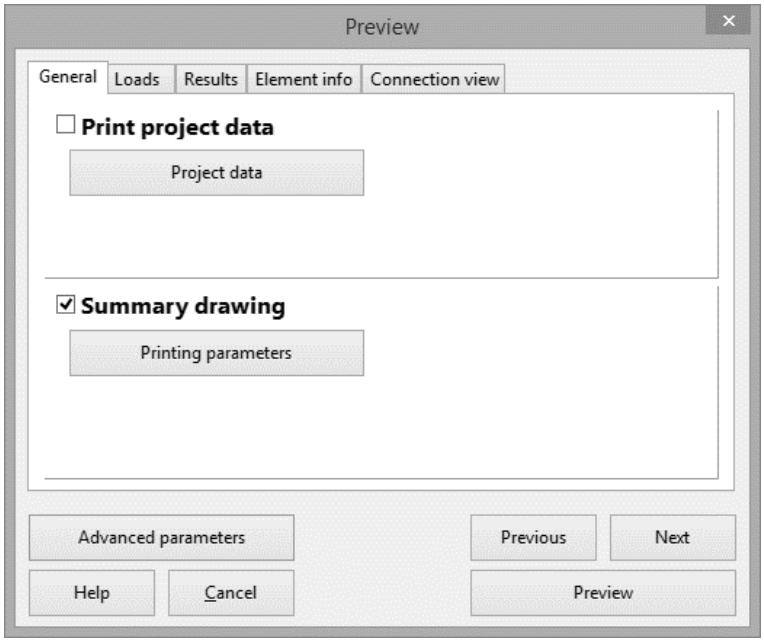
3.2.1 Page setup

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

3.2.2 Report configuration

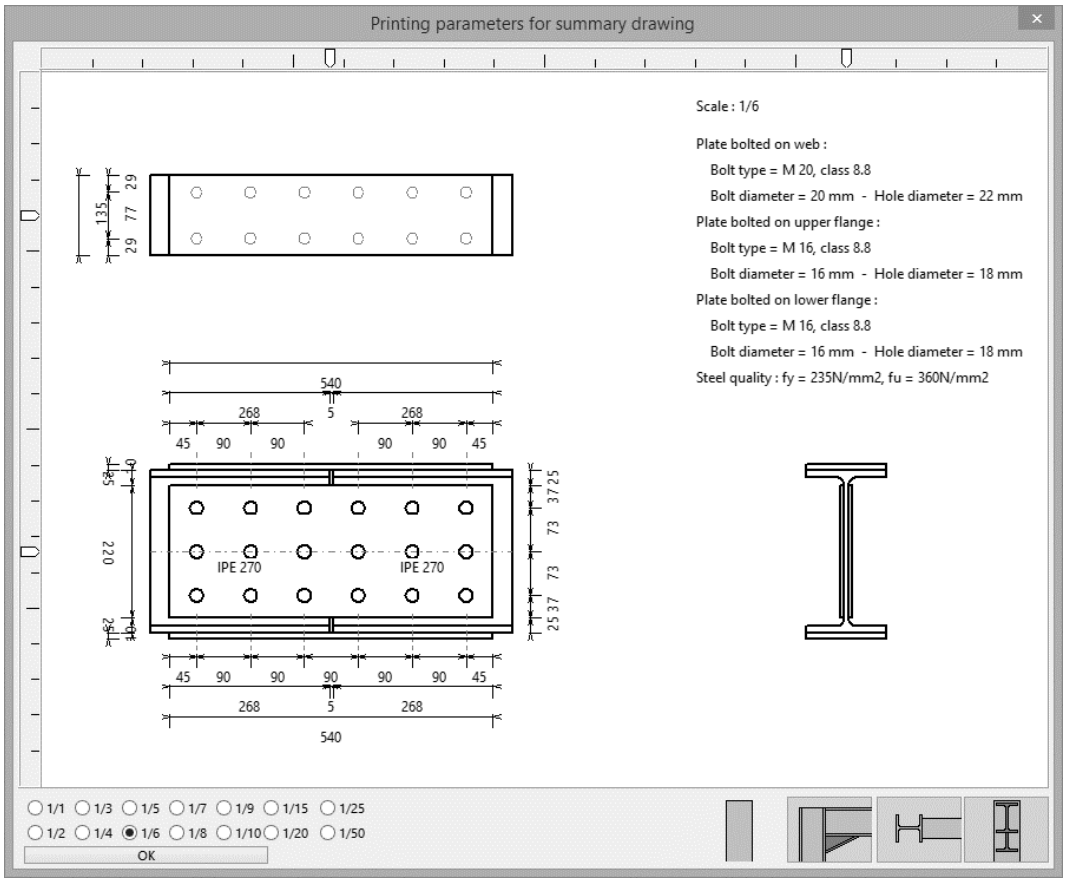
Go to **Preview** mode by means of the  icon of the icon toolbar.

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.



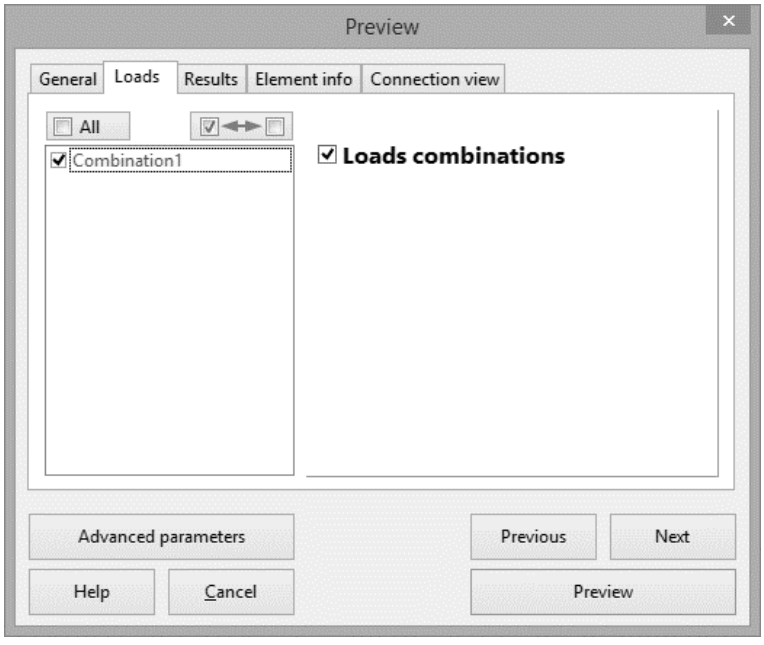
To create the kind of summary drawing shown below

- Select all connection views shown at the right bottom part of the canvas window.
- Use the cursor icons to position the selected drawings in the appropriate positions.
- Select a proper scale (here we chose 1/6)

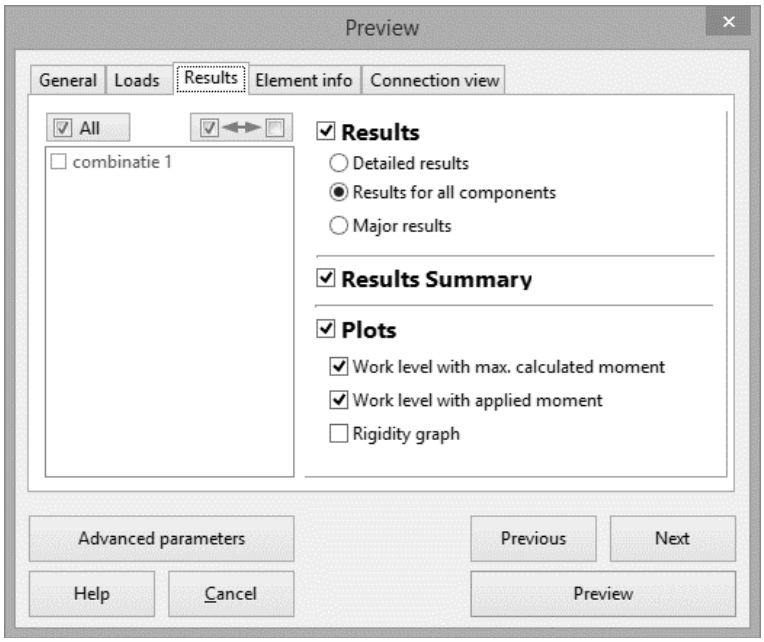


As soon as a proper lay-out has been defined, confirm by means of the 'OK'-button to return to the definition dialogue window.

Now switch to the second tab page 'Loads', 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.

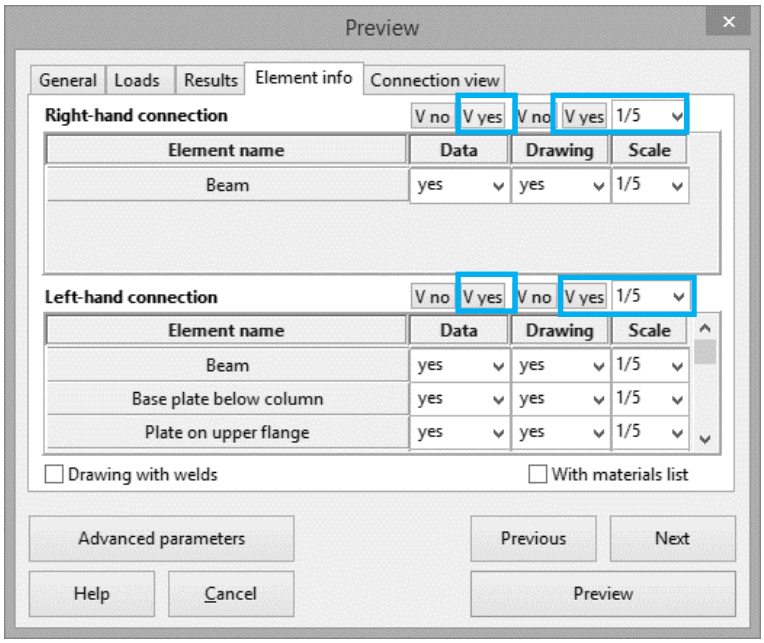


Now switch to the third tab page to specify how analysis results should be reported. Within this second reporting tutorial, the focus is on the creation of a more detailed analysis report. To do so, select the options ‘Results – Results for all components’ and ‘Plots’ as indicated below.

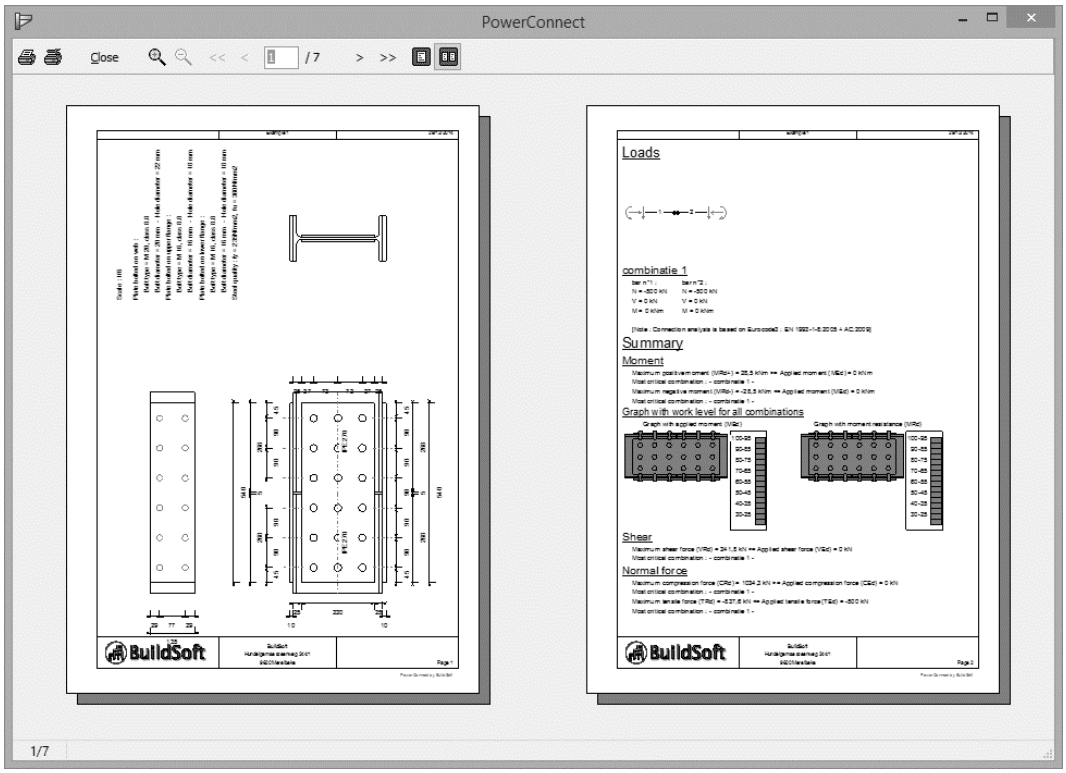


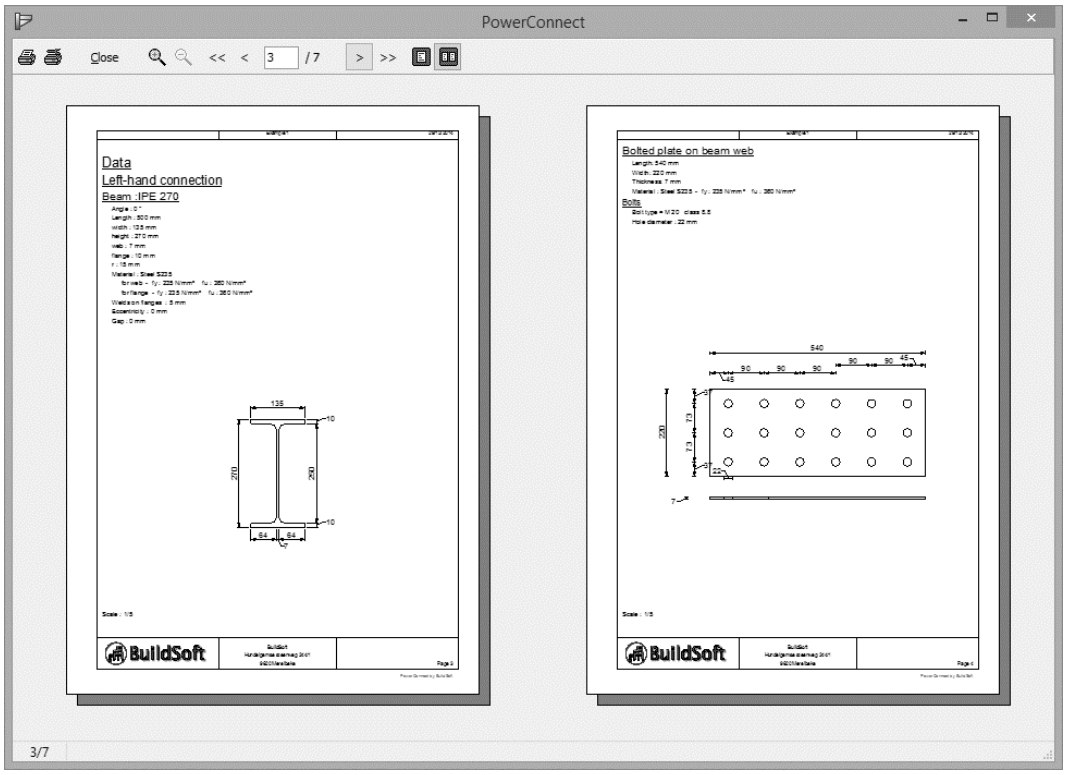
Now switch to the fourth tab page to specify that detailed drawings and data of all individual connection elements to be included into the report. For all elements, make sure to specify ‘Yes’ for both the ‘Data’ and ‘Drawing’ columns. This can be done for each entry individually, by selecting ‘Yes’ from 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.



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 7 page report as shown in part below (4 pages only).





To actually print the report, use the icon  to send the report to the selected printer. If not, use the 'Close'-button to return to the PowerConnect working environment.

3.3 Tutorial 3: shear connection – beam to column flange with fin plate

This reporting tutorial is based on the model created in §2.6.

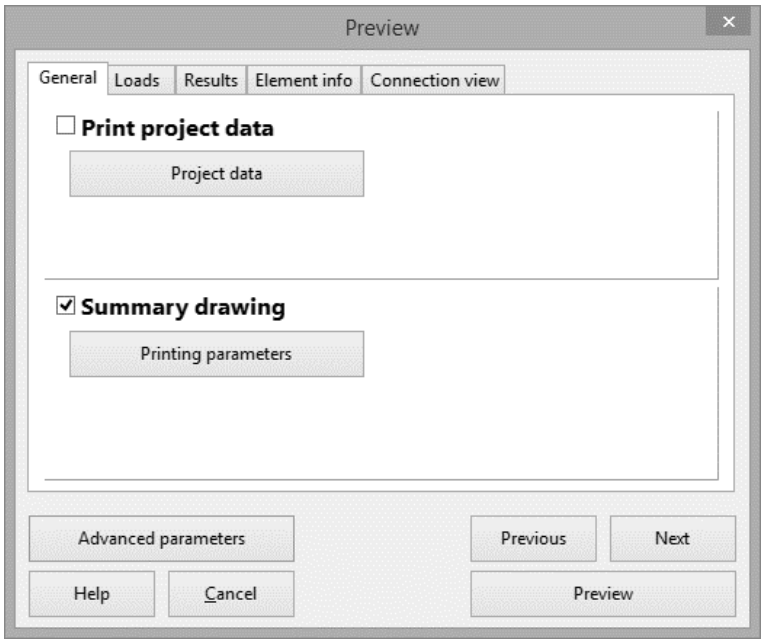
3.3.1 Page setup

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

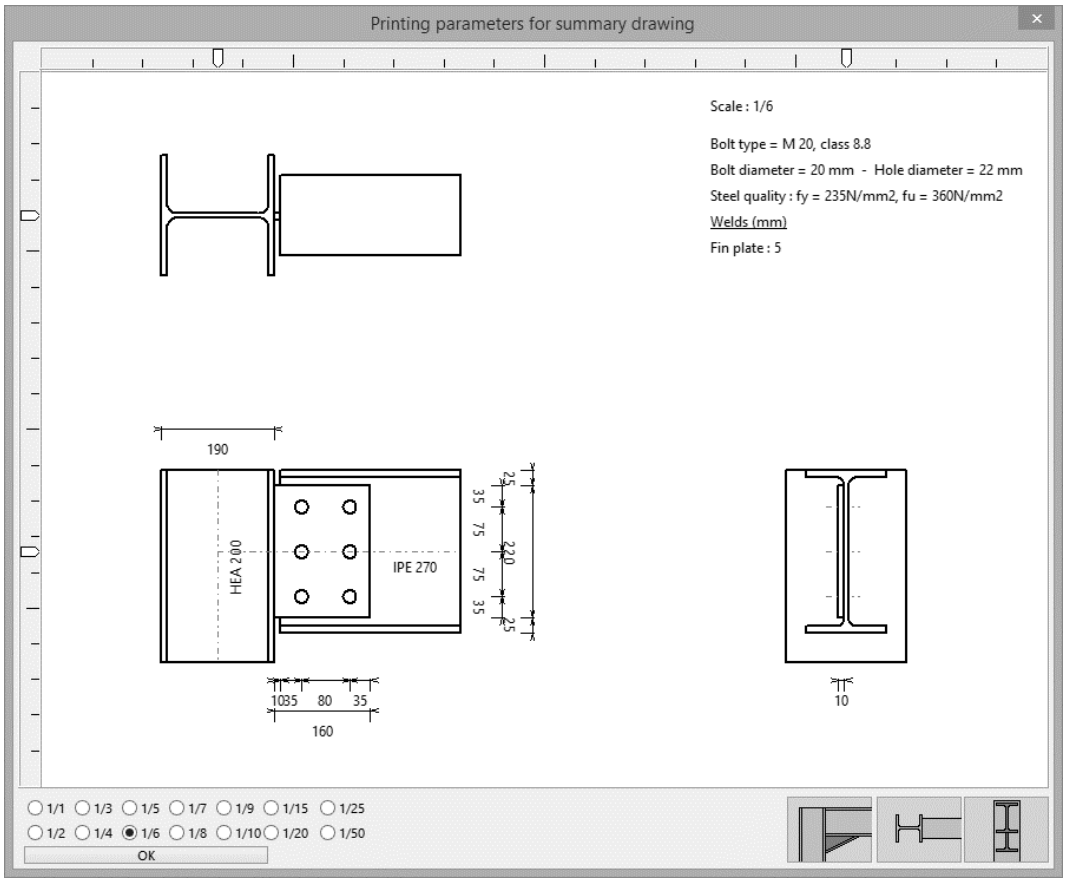
3.3.2 Report configuration

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.

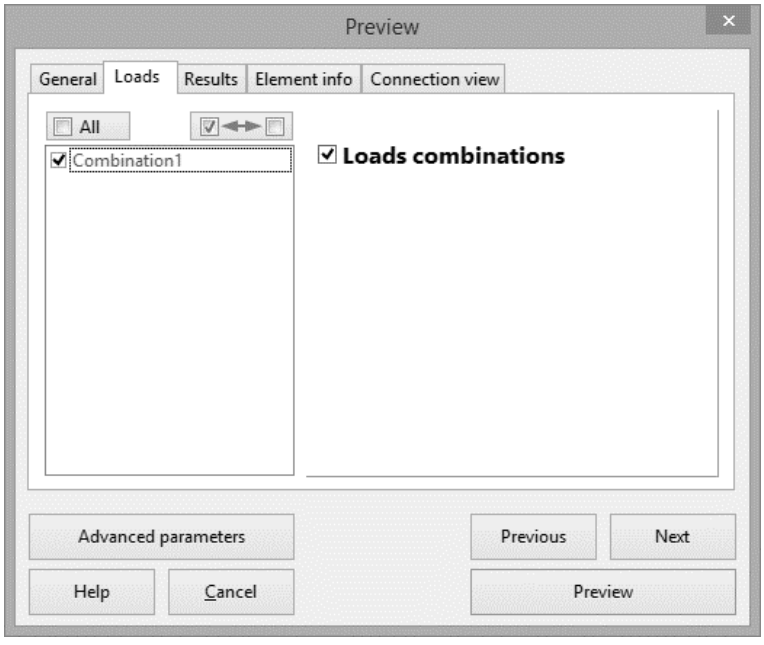


To create the kind of summary drawing shown below, make sure to select all connection views shown at the right-hand bottom part of the canvas window. Use the cursor icons to position the selected drawings in the appropriate positions, and also make sure the proper scale is selected (the selected scale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

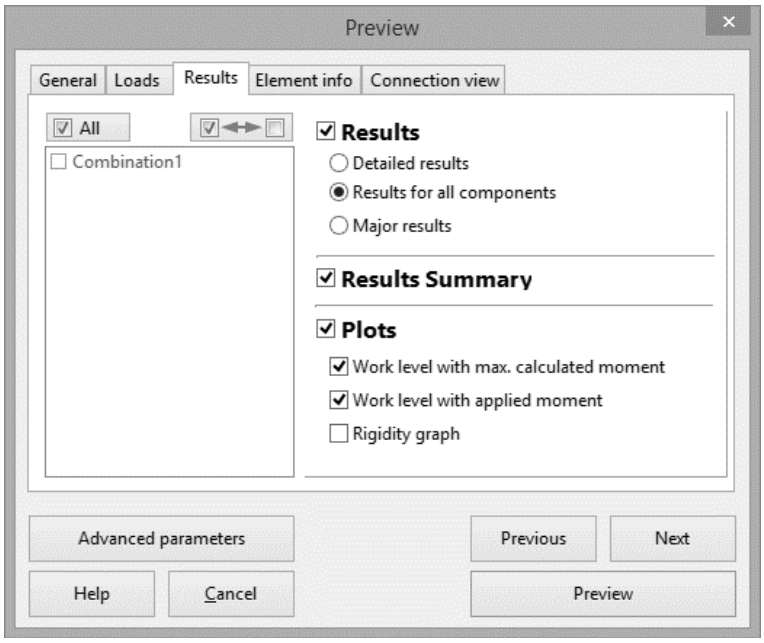


As soon as a proper lay-out has been defined, confirm by means of the 'OK'-button to return to the definition dialogue window.

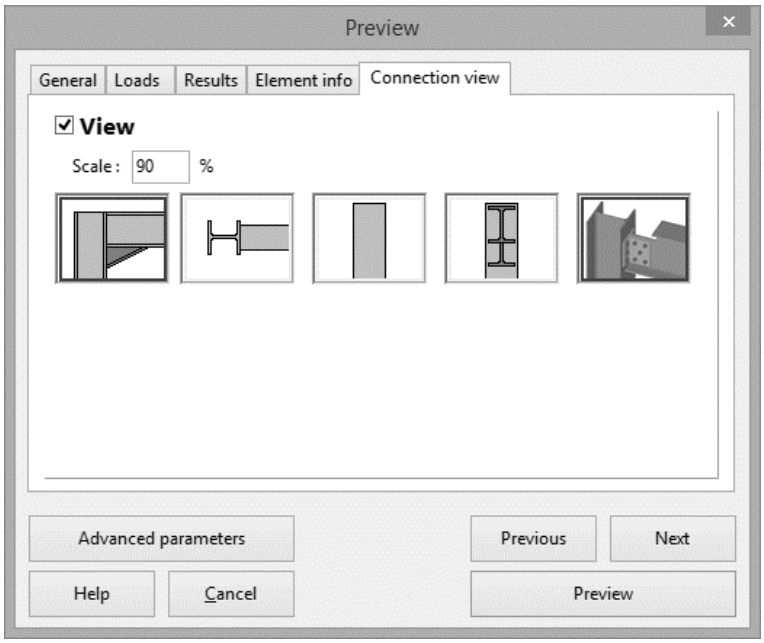
Now switch to the second tab page 'Loads', and make sure that the option 'Loads combinations' is selected. This will ensure that the report includes an overview of the loads combination that is part of this project, at least if 'Combination1' is selected as indicated below.



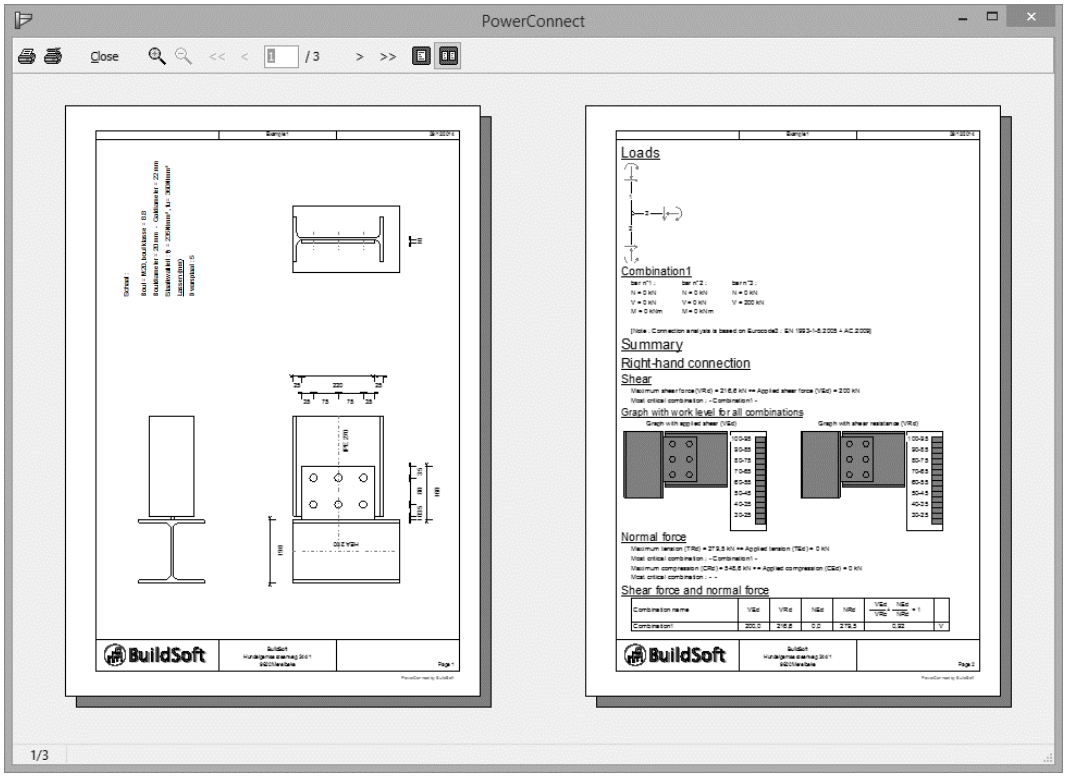
Now switch to the third tab page ‘Results’ to specify how analysis results should be reported. Within this second reporting tutorial, the focus is on the creation of a more detailed analysis report. To do so, select the options ‘Results – Results for all components’ and ‘Plots’ as indicated below.

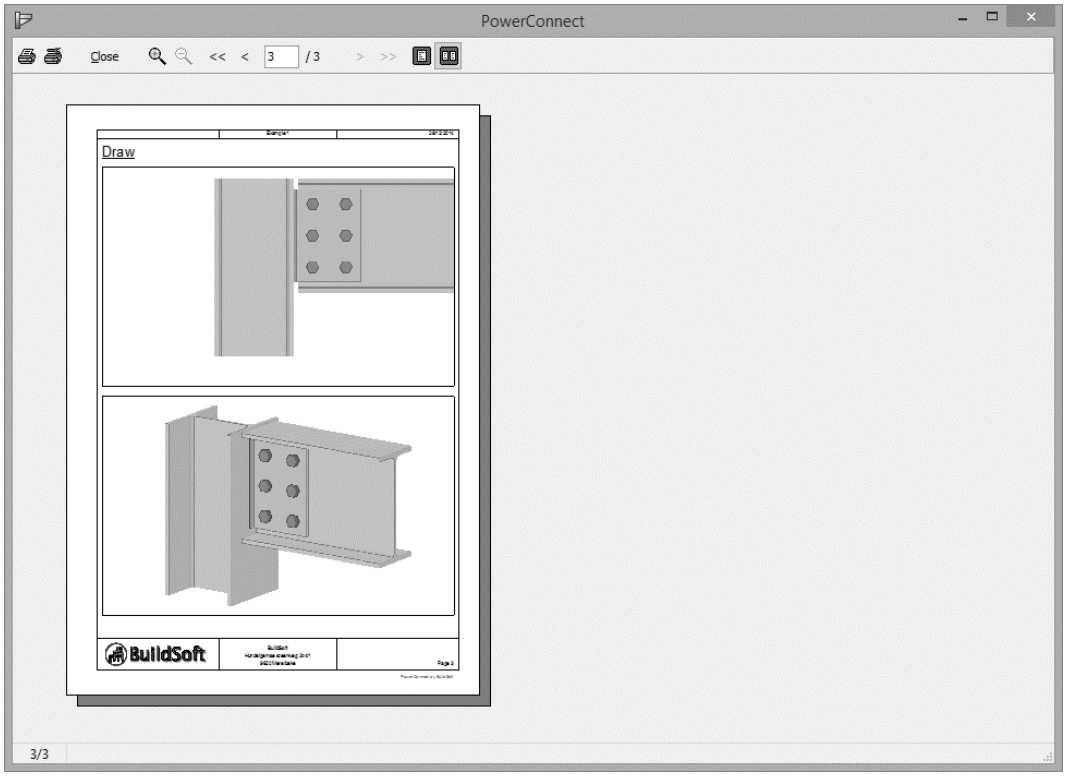


Now switch to the fifth tab page ‘Connection view’ to specify that rendered views of the entire connection are to included into the report. Select the option ‘View’ and make sure to select both the 3D and the front view icons.



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 3 page report as shown below.





To actually print the report, use the icon to send the report to the selected printer. If not, use the 'Close'-button to return to the PowerConnect working environment.