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 - armaduras - cold

Release informatie

Diamonds 2017

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 móviles - funderingsplaat - étriers - analyse resistance au feu - cercha - globale knikfactor - dynamische analyse - wapening - maillage - malla - uniones - radier

Table of contents

1. Before getting started with Diamonds 2017

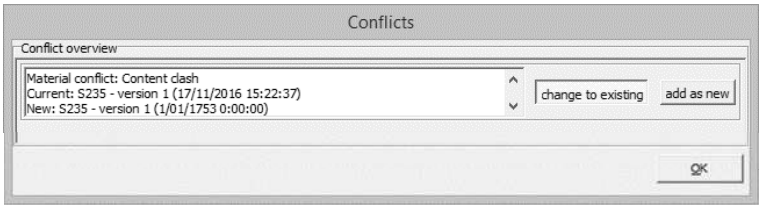
IMPORTANT: please read the installtation guide thoroughly, before getting started!

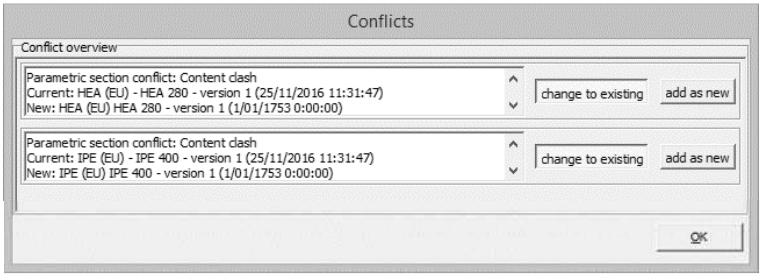
The section and material library are now database oriented. Consequently, there is no backwards compatibility between Diamonds 2017 and older versions of Diamonds.

A model that is made or saved with Diamonds 2017 can not be used in older version of Diamonds nor PowerFrame.

A model made with Diamonds 2015 r06 or older, will give a notification concerning the sections and materials when opening it in Diamonds 2017.

- If you select 'Change to existing', Diamonds 2017 will find the correspondeing material (or section) in the new library and assign this to the geometry.
- If you select 'Add as new', Diamonds 2017 will copy the old material properties (or section properties) to a new material in the current library. This material will be assigne to the geometry.

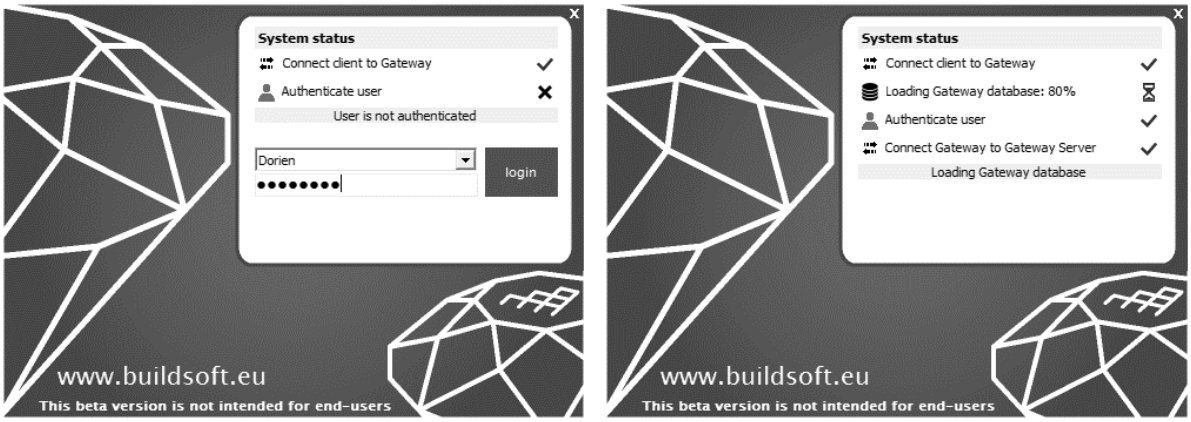




2. Starting up the first time

The Diamonds 2017 requires a one-time identification on start up. The user name and password to fill out are identical to your Windows credentials. Diamonds will present you a list with known Windows users. Select the appropriate user and enter the password.

This identification is necessary to load the material and section library.

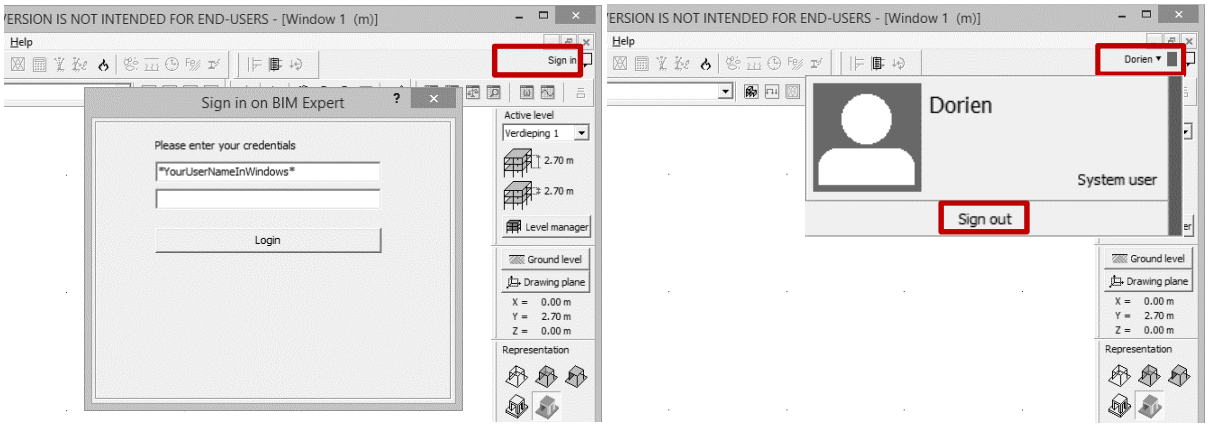


3. Work environment

3.1. Interface

After logging in, Diamonds 2017 will show your username in the right upper corner. If you are not signed in – instead of your user name, you see 'Sign in' – you can sign in overhere.

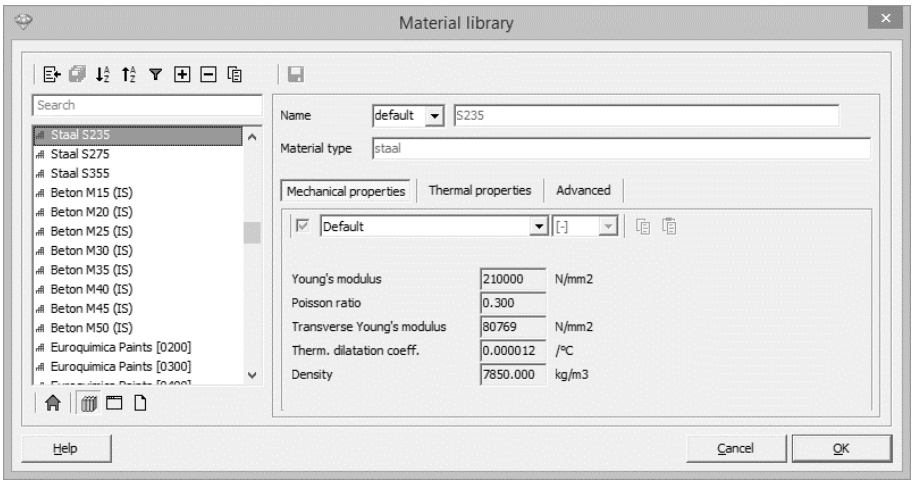
To sign out by clicking the arrow next to your username and choose 'Log out'.



Diamonds 2017 has additional features for storing one central managed material and section library, allowing each user to work with this library with the same properties. An administrator can manage materials and sections. When the administratior updates materials and sections while other users are running Diamonds, they will be notified of the update by means of the icon .

3.2. Material library

The material library is reworked in Diamonds 2017. It is now fully database oriented. At any time, you can adjust or complete this library by selecting the option 'Edit – Material library …'. The following dialogue box will appear:



On the left, you will find a list of all the defined materials and, on the right, the corresponding properties.

Materials preceded by the icon are standard materials. Materials preceded by the icon are user defined materials.

It is not possible to edit standard materials. However, you can copy a standard material using . This new material is fully editable by the user.

To add a new material click . Define at least the mechanical properties. The button deletes the selected user-defined material from the list on the left.

To search a material in the library, type the name of the material in the cell 'Search'. Use the button to filter on material type (steel, concrete, timber …).

The buttons and sort the list in alphabetical order descending or ascending. If you prefer a different sorting order, you can select a material with the cursor and drag it to the location of your choice.

The button allows you to import an external material library.

If a material is used in the current model, the button will light up when this material is selected. If you want a user defined material to be available during your whole Diamonds session (until you close the software), click the button . Use the button , to add a user defined material to the library.

Use the button or the right mouse button to set the currect select material as the default steel, timber or concrete material (only one of each possible).

Click to save all changes. Use to save only the modifications to the selected material.

The material properties are divided into 3 tabpages:

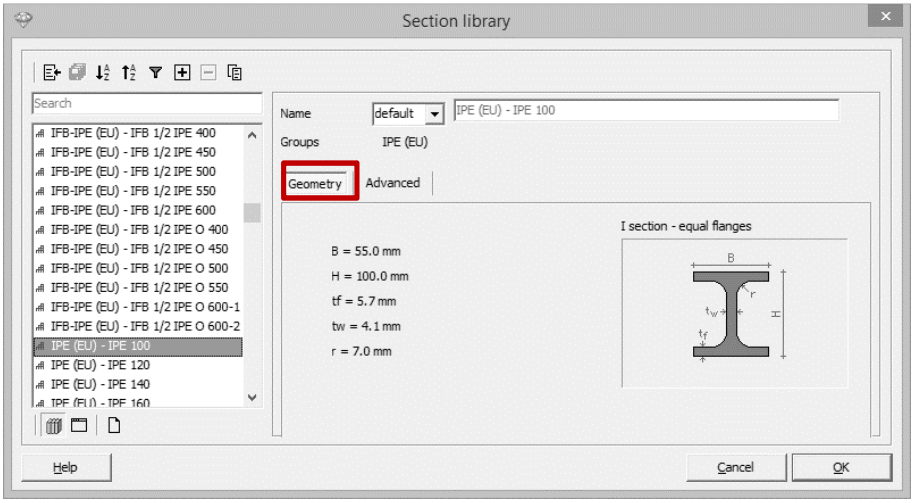
- Mechanical (or elastic) properties
- Thermal properties

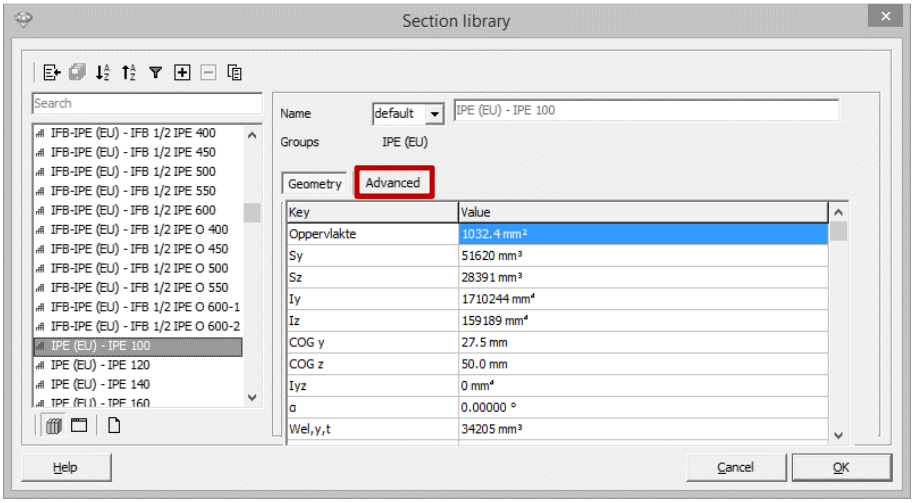
- Advanced properties (design code settings)

The contents and functionality of these tabpages have not changed.

3.3. Section library

Diamonds 2017 comes with a library containing most-used standard steel sections. You can open the library with the use of this menu: 'Edit – Section library…'.



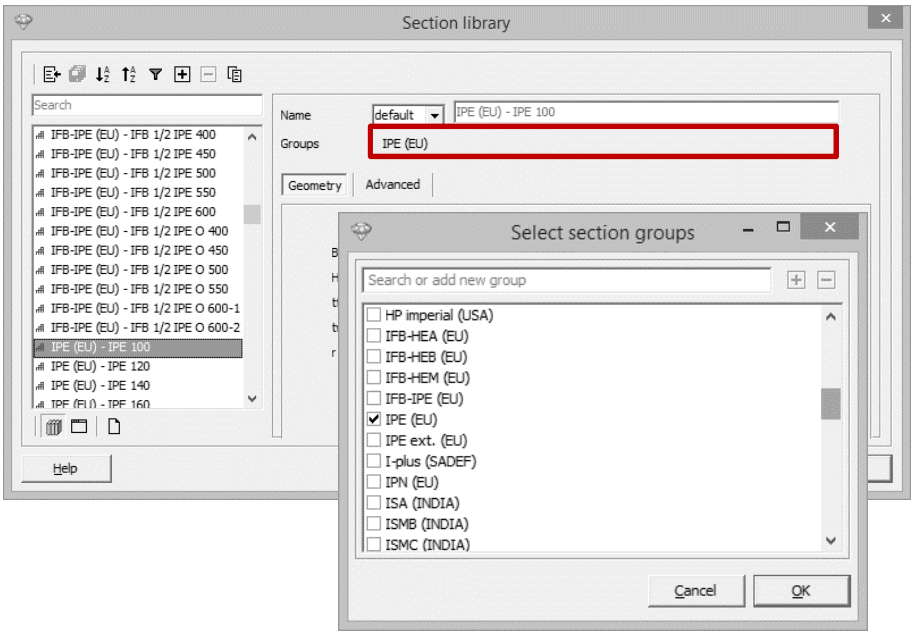


On the left, you will find a list of all steel sections and, on the right, the corresponding dimensions and properties:

Sections preceded by the icon are standard sections. Sections preceded by the icon are user defined sections. It is not possible to edit standard sections. However, you can copy a standard section using . This new section is fully editable by the user.

To add a new section, click . The button deletes the selected user-defined section from the list on the left.

Click on the cell 'Groups' to assign the section to multiple section groups.



To search a section in the library, type the name of the material in the cell 'Search'. Use the button to filter on section type.

Use the buttons and to sort the section list in alphabetical order descending or ascending. If you prefer a different sorting order, you can select a section with the cursor and drag it to the location of your choice.

The button allows you to import an external section library.

If a section is used in the current model, the button will light up when this section is selected. If you want a user defined section to be available during your whole Diamonds session (until you close the software), click the button . Use the button , to add a user defined section to the library.

Click to save all changes. Use to save only the modifications to the selected cross-section.

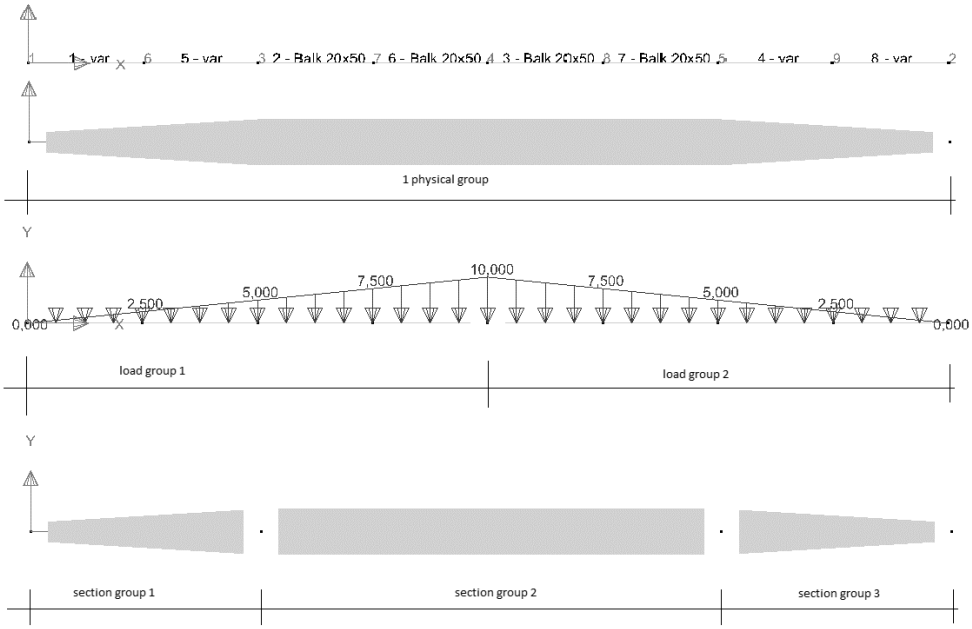
3.4. Groups

It was already possible to group bars for buckling and as a physical group. But in Diamonds 2017 it is also possible to define groups for 'sections' and 'loads':

- Section groups must be used for defining cross sections with variable height extending over multiple bar elements
- Load groups must be used for defining distributed loads extending over multiple bar elements

These extra groups enlarges the possibilities of defining cross-sections and loads in Diamonds.

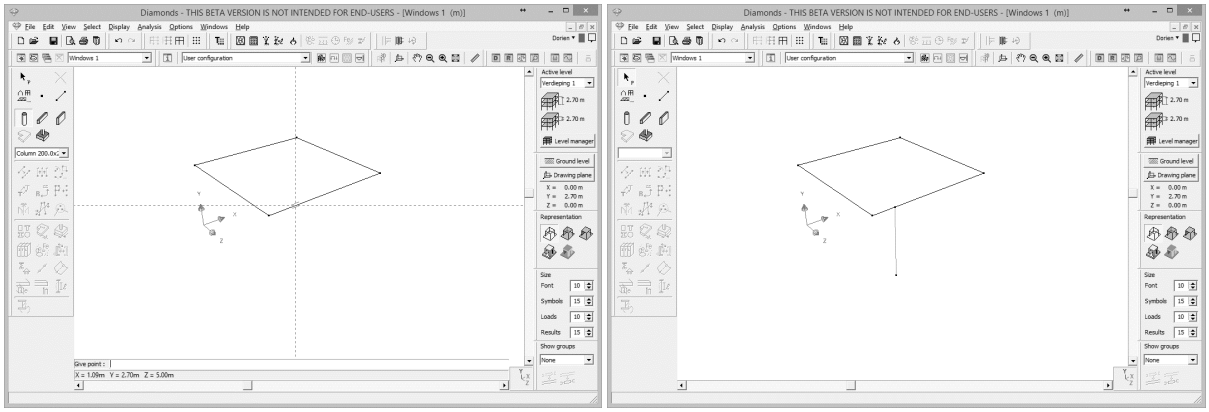
The bars grouped in 'physical' aren't necessarily the same as the bars grouped to define sections or loads. So in Diamonds 2017 the following situation is possible:



4. Geometry

4.1. Drawing

Draw additional nodes or columns in a 3D view on existing lines using the buttons and .



4.2. Extension of the plate types

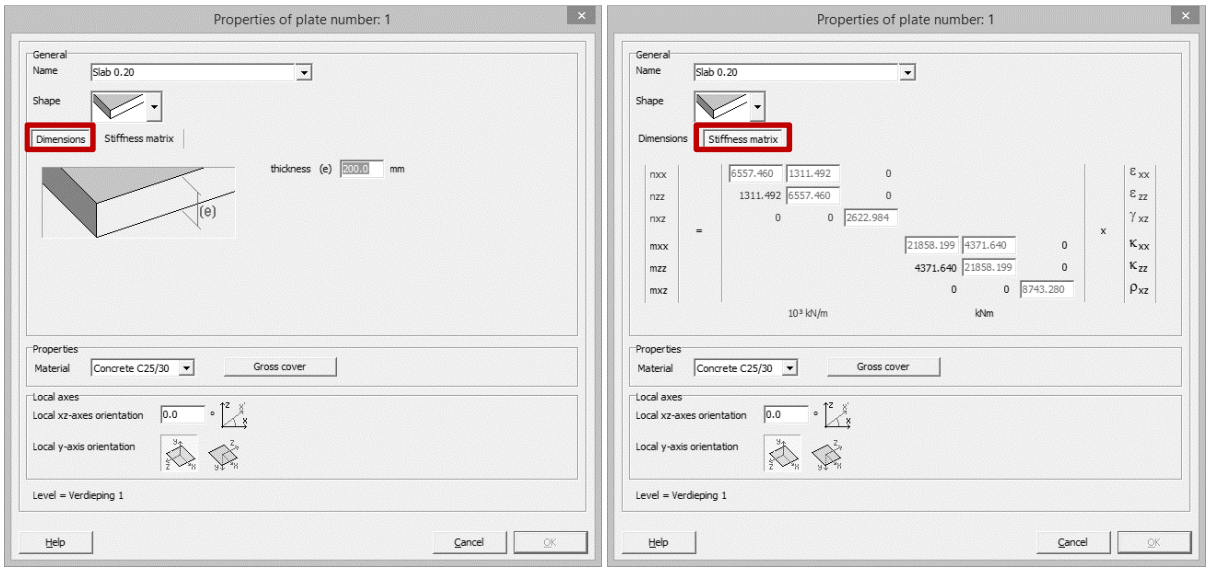
Diamonds 2017 has 3 additional plate types:

- Diaphragm (§4.2.1),
- Hollow core slabs (§4.2.1),
- Arbitrary plates with stiffness matrix definition (§4.2.3).

The definition and analysis of diaphragms does not require any plate or slab license. The analysis of all other plate types requires at least the 2D Slabs license.

4.2.1. Diaphragm

A **diaphragm** is a plate with no bending stiffness, only capable of transferring normal forces. This type of plate is used to simulate the membrane action.



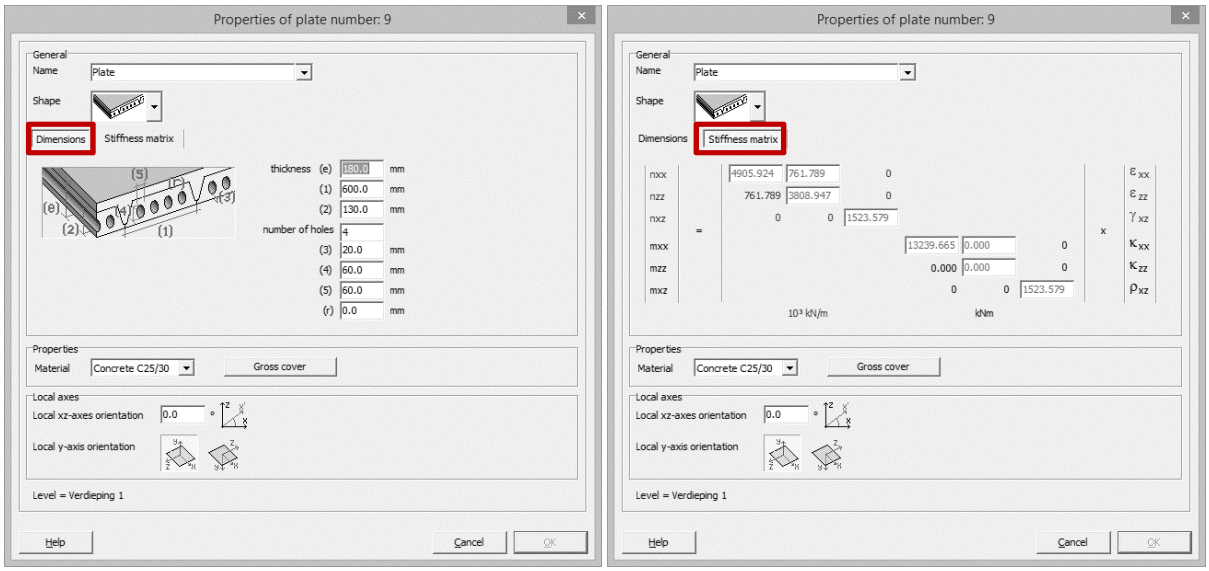
Diaphragms require a thickness and material so that the correct self weight can be calculated and taken into account.

Due to the fact that this plate has no bending stiffness, you should not load this plate perpendicular to the surface. If you choose to do this anyway, you will experience very large out-of-proportion deformations.

4.2.2. Hollow core slab

A **hollow core slab** is a plate bearing in one direction, with an optional cast-in place layer of concrete, bearing in two directions. This slab is characterized by the weight-saving wholes. Diamonds assumes the distances between the wholes being constant.

This type of plate can be used in the elastic calculation, but it is not yet possible to determine the reinforcement. However, this is possible with the software ‘ConCrete HCS’.



4.2.3. An arbitrary plate with stiffness matrix definition

Diamonds is a **Finite Element Method (FEM)** based software program. The FEM needs to structure to be chopped in a finite number of elements, which will then be logically linked to each other. There are a number of requirements to these links, depending on the type of element (beam, column, plate, wall, …). In any case, it is required that the nodes of the elements displace together.

This method allows to approximate the behavior of a complex structure by solving a matrix equation. The matrix equation expresses the relation between the elements in the structure. In case of a linear static analysis, the matrix equation writes as follows:

$$\underbrace{[\boldsymbol{Q}]}_{\substack{node\\loads}} = \overbrace{[\boldsymbol{K}]}^{\substack{stiffness\\matrix}} \cdot \underbrace{[\boldsymbol{U}]}_{\substack{node\\displacements}}$$

The matrix equation containts a stiffness matrix $[\boldsymbol{K}]$, a matrix with node displacements $[\boldsymbol{U}]$ and a matrix with node loads $[\boldsymbol{Q}]$.

The stiffness matrix $[\boldsymbol{K}]$ will condition the behavior of the elements (beam, column, plate, wall, …). Without going further into detail on the derivation, we assume that the stiffness matrix $[\boldsymbol{K}]$ for a slab or plate looks as follows:

$$[\boldsymbol{K}] = \begin{bmatrix} L_{xx} & L_{\upsilon} & 0 & 0 & 0 & 0 \\ L_{\upsilon} & L_{zz} & 0 & 0 & 0 & 0 \\ 0 & 0 & L_{xz} & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{xx} & D_{\upsilon} & 0 \\ 0 & 0 & 0 & D_{\upsilon} & D_{zz} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{xz} \end{bmatrix}$$

The factors with 'L' describe the behavior for membrane action (= normal forces, the factor with 'D' describe the behavior for bending. The factors for shear are not mentioned here. The table below gives an overview of these factors for a number of plate types:

	Isotropic plate	Anisotropic plate – bearing in one direction	Diafragma
L_{xx}	$\frac{Ee}{(1-\upsilon^2)}$	$\frac{Ee}{(1-\upsilon^2)}$	$\frac{Ee}{(1-\upsilon^2)}$
L_{zz}	L_{xx}	≈ 0	L_{xx}
L_{υ}	$\upsilon \cdot L_{xx}$	≈ 0	$\upsilon \cdot L_{xx}$
L_{xz}	$G \cdot e$	≈ 0	$G \cdot e$
D_{xx}	$\frac{Ee^3}{12(1-\upsilon^2)}$	$\frac{Ee^3}{12(1-\upsilon^2)}$	0
D_{zz}	D_{xx}	0	0
D_{υ}	$\upsilon \cdot D_{xx}$	0	0
D_{xz}	$D_{xx} \cdot 0.5\ (1-\upsilon)$	$\tau \cdot D_{xx} \cdot 0.5\ (1-\upsilon)$	0

More information about FEM and stiffness matrices can be found in:

- O.C.Zienkiewicz, The Finite Element Method, McGraw-Hill book Company, 1977
- J. Blauwendraad, Plates and FEM, Surprices and pitfalls, ISBN 978-90-481-3595-0

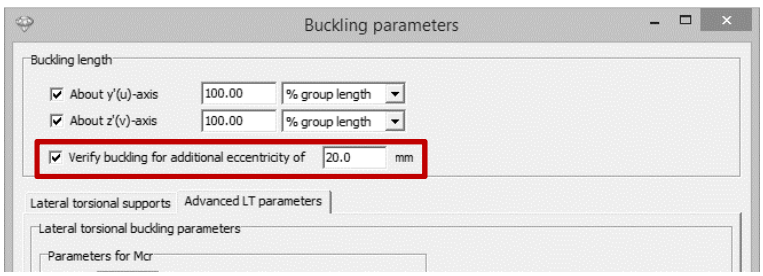
4.3. Buckling parameters

EN 1992-1-1 § 5.1.4 poses that for concrete, second order effects should be taken into account:

- Or by performing a complete second order analysis (F9 in Diamonds + 'second order analysis')
- Or by using a simplified method. Diamonds uses the nominal curvature method (EN 1992-1-1 §5.8.8).

Up until Diamonds 2015 r06 you could disable or enable this method in the material library (option 'verify buckling for additional eccentricity').

In Diamonds 2017, this option is now located in the buckling lengths window  . That way you can easily activate or deactivate this option for a number of bars at once.



5. Loads

5.1. Wind and snow

The Polish (PL) and Norwegian (NO) annex to EN 1991-1-3 for snow and EN 1991-1-4 for wind are added in Diamonds 2017. The British (UK) national annex to EN 1991-1-3 and EN 1991-1-4 were already added in Diamonds 2015r06.

5.2. Seismic

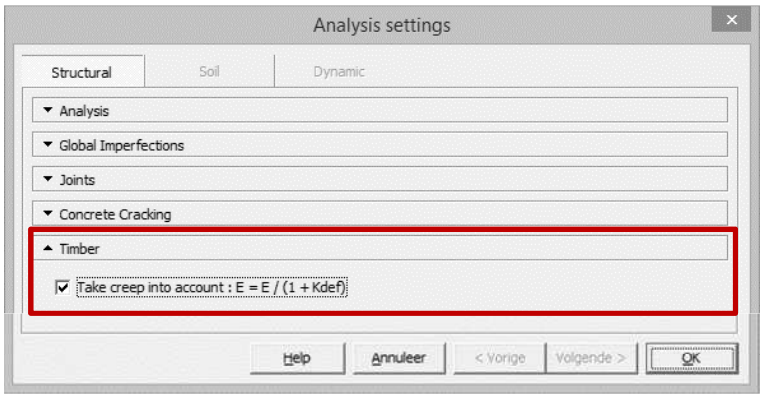
The Belgian (BE), French (FR) and Luxemburg (LU) annex to EN 1998-1 for seismic design, as well as the Dutch guideline NPR 9998:2015 are included in Diamonds 2017.

6. Analysis

6.1. Tab page 'Structural'

As of Diamonds 2017 you take the creep effect for timber structures into account. The creep effect is taken into account by correcting the Young's Modulus with a factor k_{def} according to EN 1995-1-1 §2.3.2.2. (1).

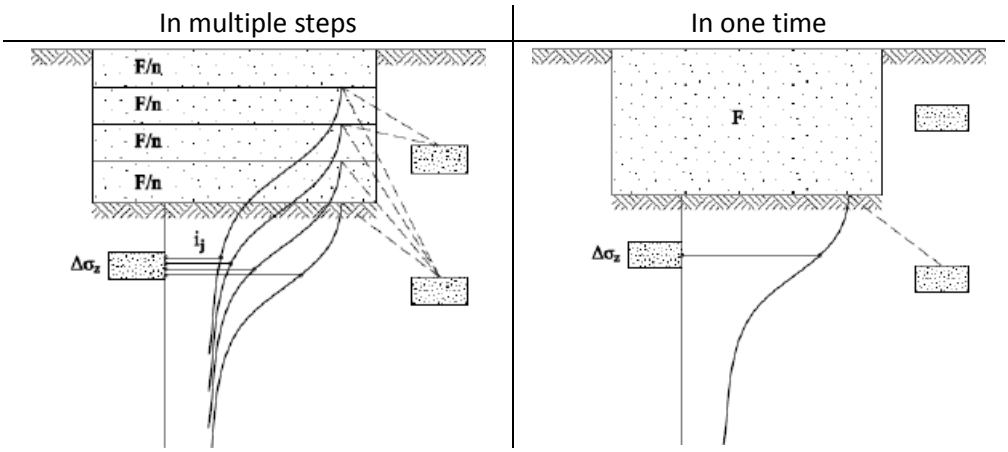
The k_{def} factor can have values from 0,6 to 2,0. Mark that a k_{def} factor of 2,0 will lead to deformations being three times bigger compared to the ones calculated in previous version of Diamonds!

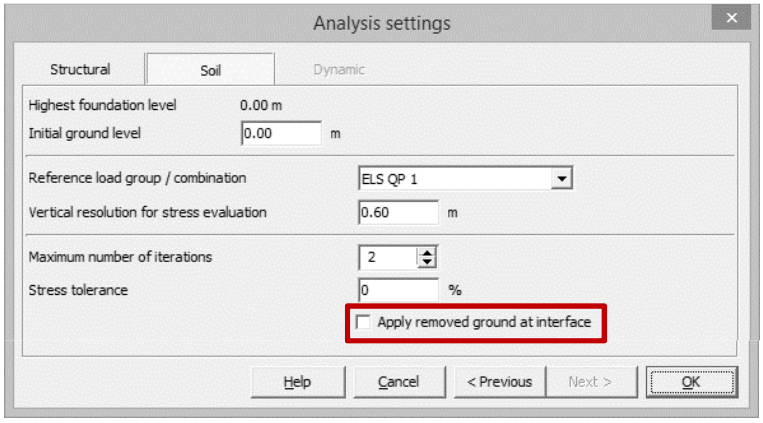


6.2. Tab page 'Soil'

The option 'apply removed ground at interface' handles how the excavation should be taken into account:

- The option is ticked off, then Diamonds assumes that the ground will be removed in different steps, leading to a reduction of stress, under as well as next to the excavation.
- The option is ticked on, then Diamonds assumes that all removed ground is taken away at once, leading to a larger prestress at the excavation level.





6.3. Fire for concrete section

To perform a fire analysis, follow these steps (the order is important):

- Draw the geometry of the structure as usual
- Define the fire curves (see Reference manual §2.3.1.30)
- Calculate the response
    - Thermal response (see Reference manual §4.6)
    - Elastic response
        - Generate the combinations, don't forget to make the fire combinations ULS FI (see Reference manual §2.4.2.2)
        - Global elastic analysis (or F9)
- Reinforcement calculation (or F2)

Notes:

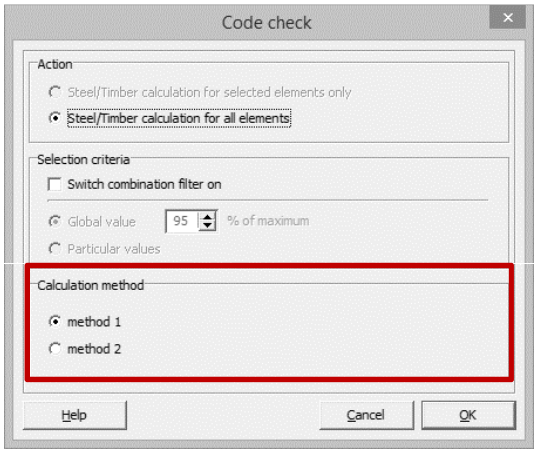
- More information about the thermal calculations can be found in the document about EN 199x-1-2 on the BuildSoft website → Support → About Eurocodes.
- When importing a PowerFrame model it is recommended to redefine the fire load.

7. Design

7.1. Steel

The Polish (PL) and Norwegian (NO) annex to EN 1993-1-1 for steel are added in Diamonds 2017. Also French code existing before Eurocode, CM 66 and CM 66 + add.80 are added. The British (UK) national annex to EN 1993-1-1 was already added in Diamonds 2015r06.

The choice between 'method 1' and 'method 2' (if allowed by the national annex) for the calculation of interaction (buckling + LTB) is moved to the window for the steel check.



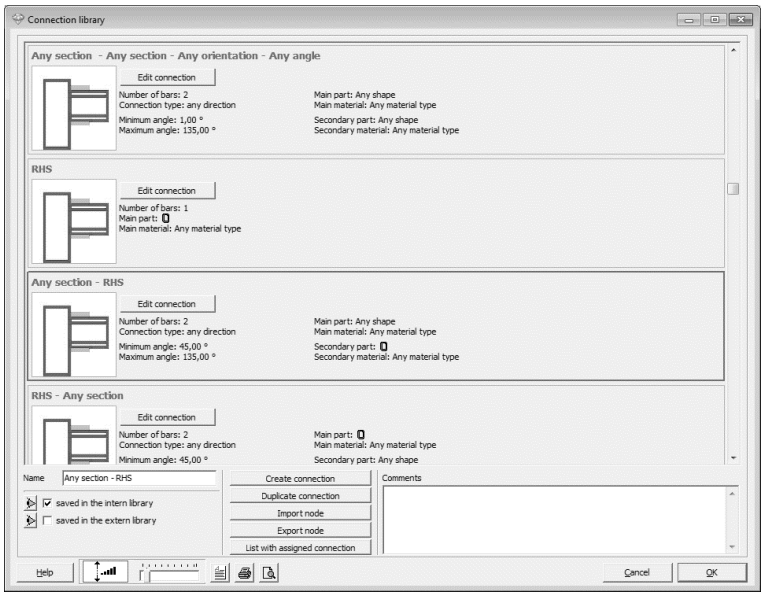
7.2. Concrete

The Polish (PL) and Norwegian (NO) annex to EN 1992-1-1 for concrete are added in Diamonds 2017. The British (UK) national annex to EN 1992-1-3 was already added in Diamonds 2015r06.

7.3. Connection library

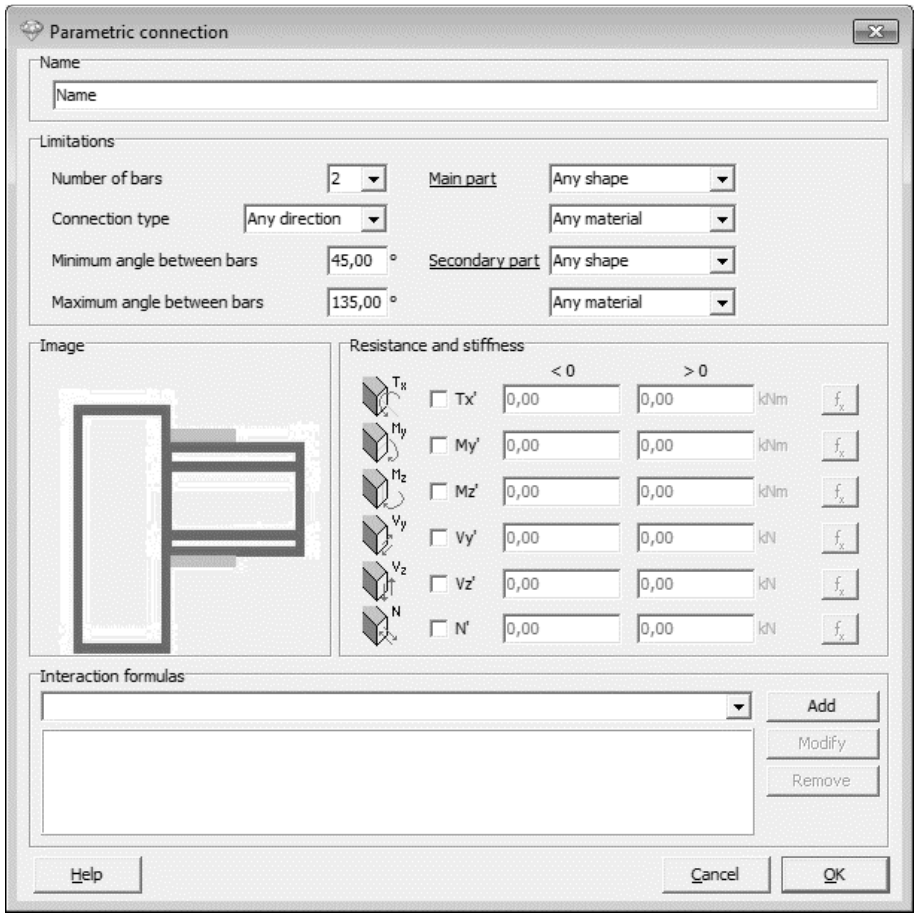
Diamonds has a connection library, used to store connections designed with PowerConnect. As from Diamonds 2017 this library can be extended with parametric connections, having (known) design resistance and stiffness values. The connection library has a built-in tool to create parametric connections. These connections can be used and verified in the global analysis with user-definable interaction checks.

To open the Connection library, click  in the Connections toolbar.



7.3.1. Creating a new custom connection

In the bottom half of the library overview, click ‘Create connection’ to build a new custom connection.



Each connection can be given a name – which does not have to be unique. Next you set the general limitations:

- The connection can consist of 1 or 2 bars. In case of 2 bars, the connection has is a 'main bar' and a 'secondary bar'. The stiffness function will be assigned on the secondary element. In case of just 1 bar, the stiffness function is assigned to the main element.
- The connection type can be along the strong or weak axis (of the main part)
- In case of 2 bars, you can limit the angle between the bars with the minimum and maximum angle.
- For both main an secondary element, you can allow
    - For the cross-section
        - Any shape
        - A specific type of shapes {list}
        - One specific cross-section from the section library
    - For the material
        - Any material
        - A specific type of material (steel, concrete, timber, etc. ...)
        - One specific material from the material library

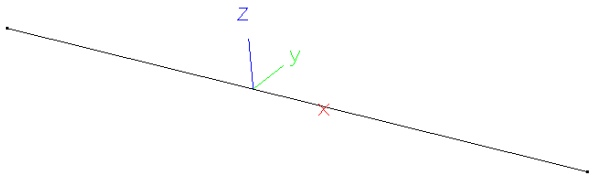
Each connection can have an image – below is the default image shown.

Six components can be taken into account in the calculation, if the corresponding checkbox is ticked:

- Torsion Tx'
- Bending moment My'
- Bending moment Mz'
- Shear Vy'
- Shear Vz'
- Normal force N'

The prime (') refers to the local axis system of the main element. This can be visualized on screen in the Window Configuration  :

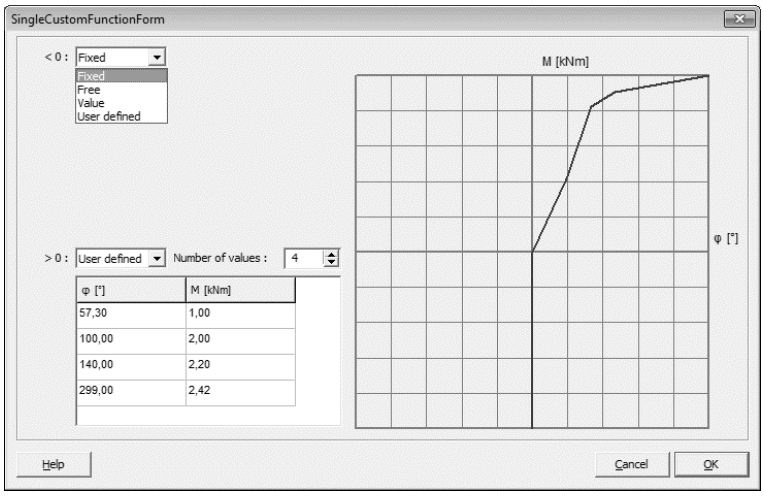
Default the local axis are as following, but:



For the global analysis, the stiffness is relevant. The connection verification takes the resistance and interaction formulas into account.

7.3.1.1. Stiffness

The default stiffness for each component is fixed. You can modify this by clicking the function button  at the end.

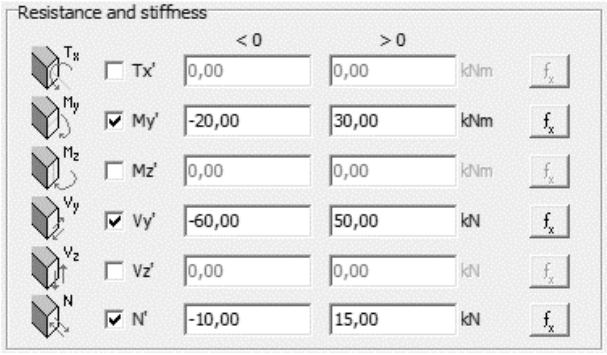


Here you set the stiffness behaviour for the component being negative (< 0, top part) and positive (>0, bottom part). Either you select one of the predefined states – fixed, free or value (value required) or you add a user-defined diagram by entering multiple sets of values for

- (φ, M) for moment components
- (u , F) for shear or normal force components

7.3.1.2. Resistance

For connection verification purposes, you can enter a negative (< 0, first value) and positive (>0, second value) resistance.



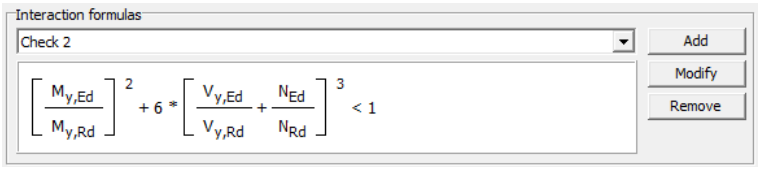
Diamonds will check the resistance of each component individually, by comparing the acting force F_{Ed} to the resisting force F_{Rd}. The result should be less than 1:

$$\frac{F_{Ed}}{F_{Rd}} < 1$$

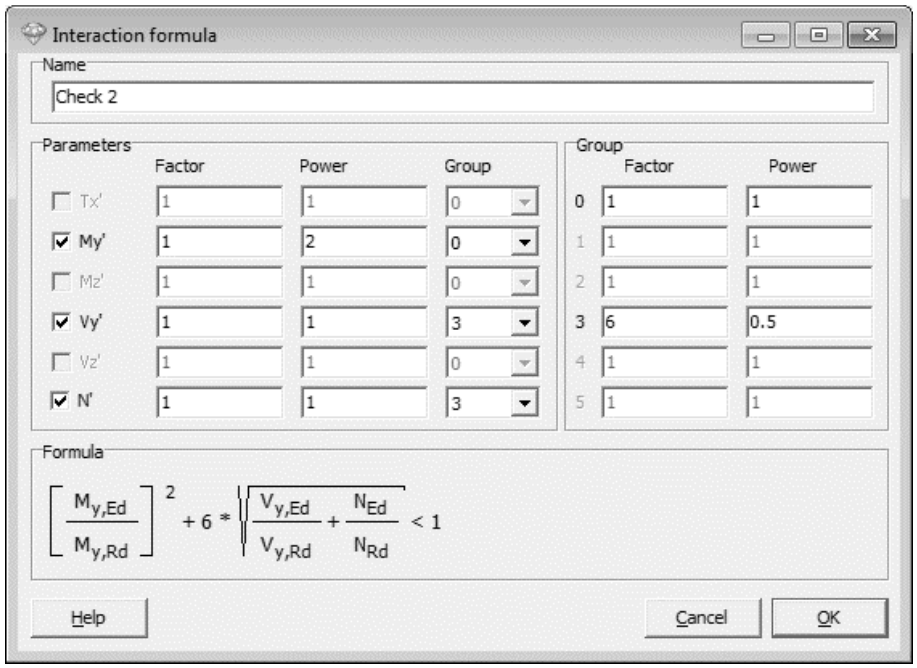
F_{Rd} corresponds to the entered value in the image above.

7.3.1.3. Interaction

For connection verification purposes, you can enter one or multiple interaction formulas for the component resistance. Already entered formulas accessible through the pull-down list. As Diamonds automatically checks the resistances for the components individually, it is unnecessary to add them here.



You can add, modify and remove formulas. On clicking the Add button, following dialog window appears:



You can enter a name for the interaction formula.

In the previous window ticked components are displayed in black and available for use in the interaction formulas. Unticked components are greyed out and not available.

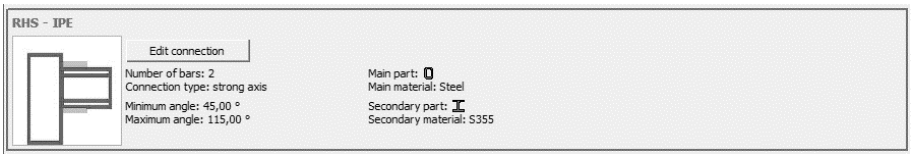
To include a component to the formula, you thick the check box. Components will be added as the acting force divided by the resisting force $\frac{F_{Ed}}{F_{Rd}}$. This ratio can be multiplied with a factor and put to a power– in the 'Parameters' section. It is possible to assign the component to a group. A group consists of one or multiple components put 'between brackets'. This group can also have a multiplication factor and a power. Note that the power is not limited to integers, you can enter for example a square root with a 0.5 value.

At the bottom of the window, you immediately see the formula appearing, allowing easy visual verification and modification. The entire formula should result in a value less than 1, in order to have an acceptable connection verification.

Click 'OK' to save the interaction formula or made modifications to it.

7.3.2. Listing in library

After you have defined the new connection, it is added to the internal & external library.

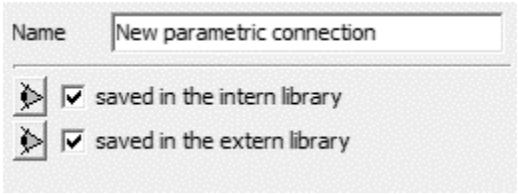


Following information is displayed:

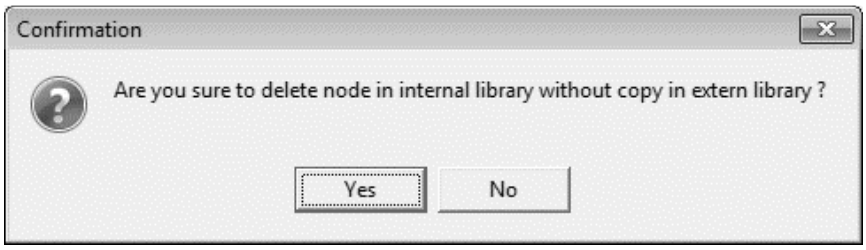
- Name
- Connection type (any/strong axis/weak axis)
- Minimum angle
- Maximum angle
- Main Bar: section and material
- Secondary: section and material

7.3.3. Saving and deleting a parametric connection

A parametric connection is automatically saved in the internal (in-file usage) and external (cross-file usage) connection library. Thanks to the external library you can use the connection in every Diamonds file.



You can modify by ticking the appropriate check boxes. If you untick them both, the connection will be deleted.



On clicking 'Yes' to the above warning, the connection is completely deleted.

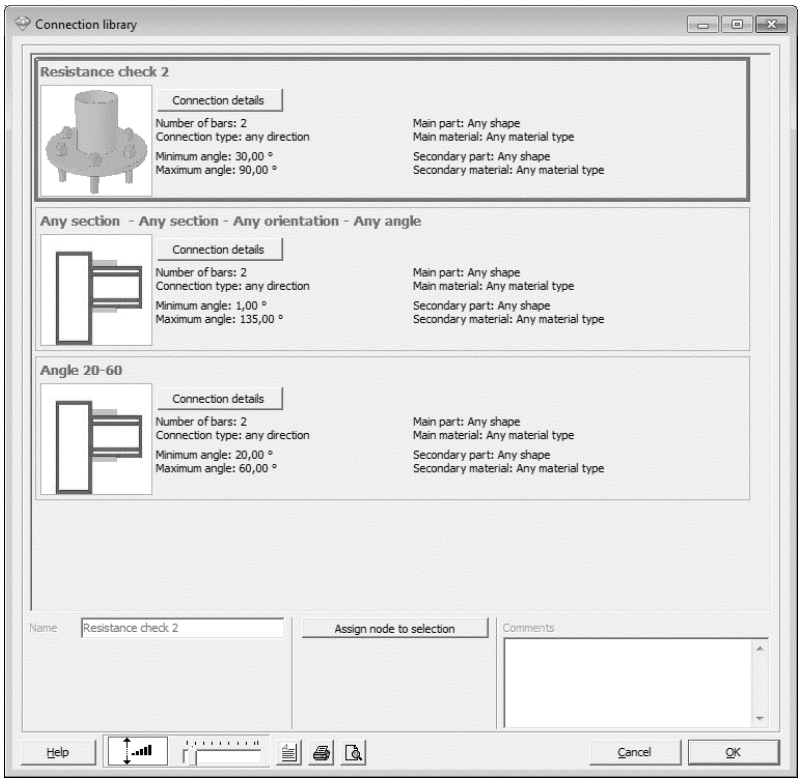
7.3.4. Duplicating a parametric connection

Parametric connections can be copied in just one mouse click, allowing you to quickly build up a complete library of (similar) connections.

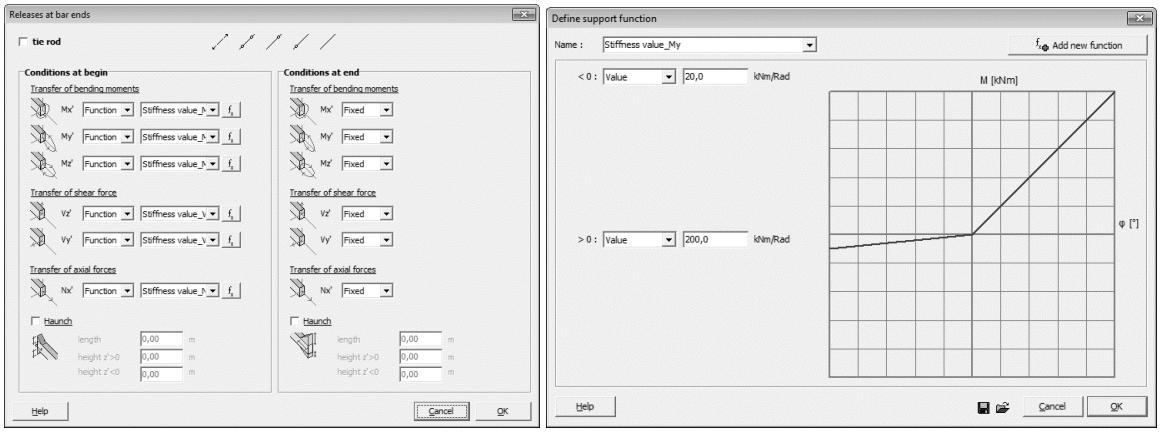
Select the connection you want to copy and click the 'Duplicate connection' button. The 'Parametric connection' input window pops up with all the settings of the selected connection. Modify the parameters of your choice and click 'OK' to save this new connection to the library.

7.3.5. Assigning a parametric connection

Once you have created a parametric connection, you can assign it to one or more bar elements in the model. On selecting 1 or 2 bar elements and clicking on the connection library icon , you will get a list of all suitable connections.



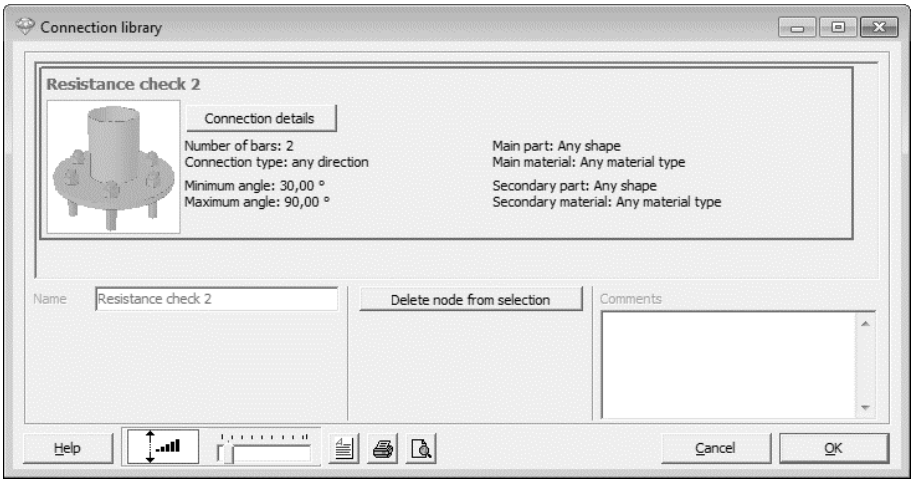
Select the connection you want to assign, click the ‘Assign node to selection’ at the bottom of the window. The assigned connection will now have a blue and red selection frame around it. Click ‘OK’ to confirm. The name of the connection should now appear next to the node. The appearance of the connection name can be set in the Window Configuration . The global analysis will take into account the stiffness of the assigned parametric connection and perform a resistance and interaction verification. The assigned stiffness function can be viewed in the ’Releases at bar ends’ dialog .



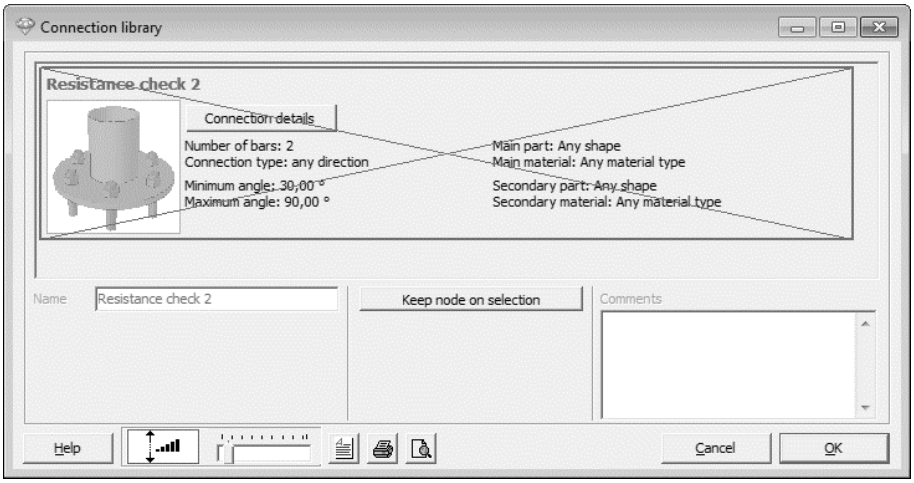
If there is no suitable connection found in the library, you will get a library overview of all connections. The ‘Assign node to selection’ will not be available.

7.3.6. Unassigning a parametric connection

To unassign a parametric connection, you select the 2 bars and go to the connection library  .



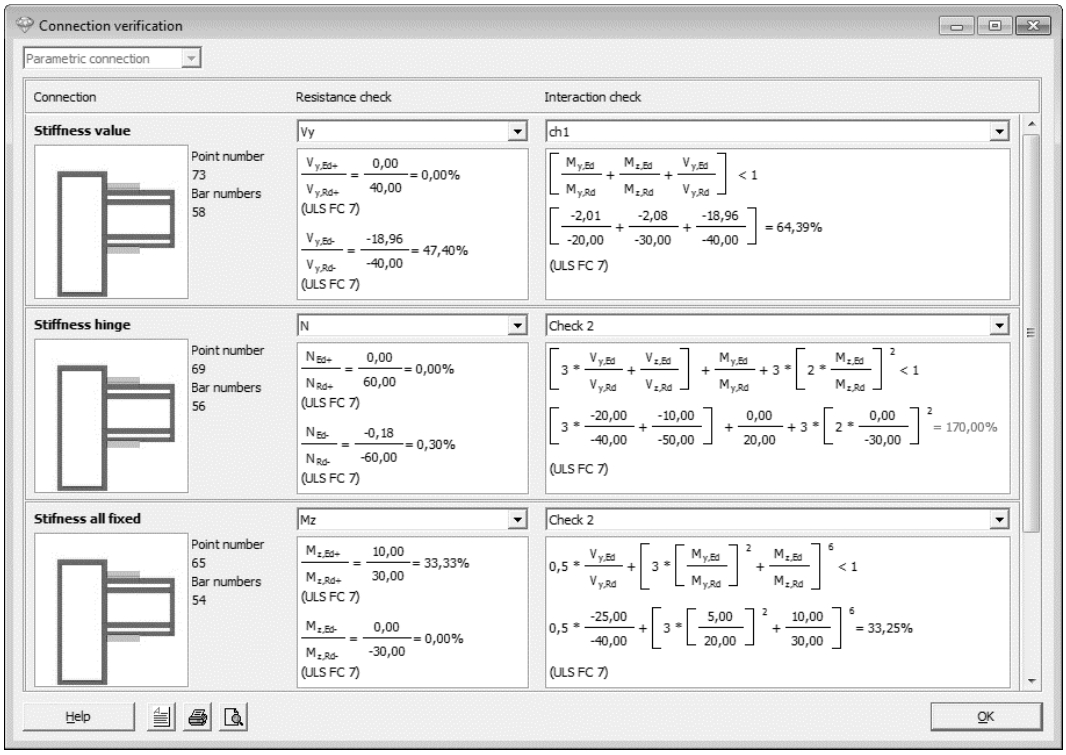
By means of the ‘Delete node from selection’ you remove the connection. If you have used this button by mistake, you can use ‘Keep node on selection’. If you do want to delete it, click the ‘OK’ button to confirm.



Important: the stiffness function at bar ends is NOT deleted. You should remove them manually in ’Releases at bar ends’ dialog  .

7.3.7. Verification results

The results of the resistance and interaction checks are available in the Connection Verification window, accessible with  .



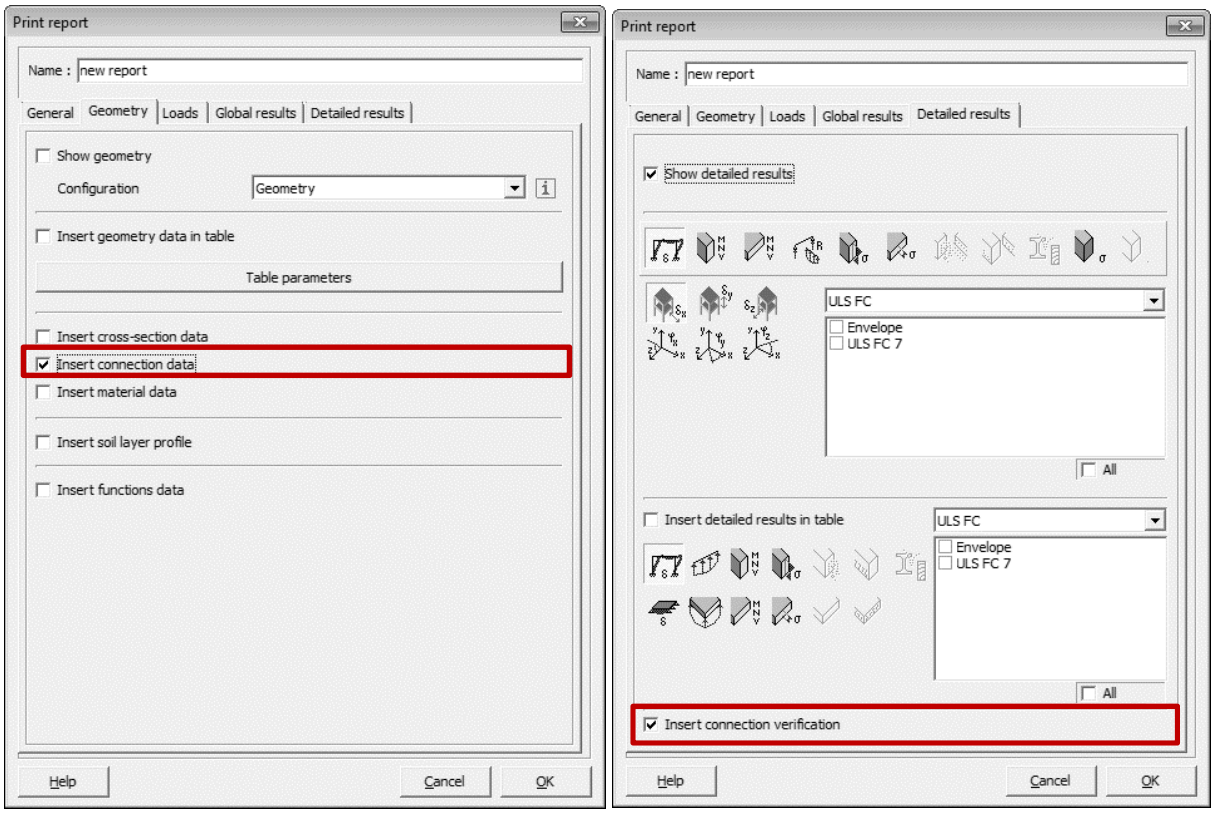
The resistance is checked for each component individually, with the drop down list you switch between the components resistance results. The interaction is checked for the defined interaction formulas, with the drop down list you switch between the different interaction results. The determining ULS combination is shown between brackets each time.

Checks that do not comply with the "< 1" requirement are displayed in red.

7.3.8. Reporting parametric connections

Parametric connections can be added to the report:

- In the 'Geometry' tab: 'Insert connection data'. This will report the geometry data of the all the connections in the internal library
- In the 'Detailed results' tab: 'Insert connection verification'. This will report the resistance and interaction checks of all assigned connections.



8. Import and export to BIM Expert

8.1. From Diamonds to BIM Expert

To export a Diamonds model to BIM Expert, do the following:

- Start Diamonds and log in (if necessary). Open the project you'd like to export.
- Start BIM Expert.
- In Diamonds go to 'File' → 'Export to' → 'Export to BIM Expert' or hit F8.

You'll need the license 'BIM Expert Basic' and 'BIM Expert Diamonds plug-in'.

In BIM Expert you can:

- Either send this model to a colleague. You'll need the license 'BIM Expert Server'.
- Either send this model to Tekla. You'll need the license 'BIM Expert Tekla structures Plug-in'.

Consult the BIM Expert manual for more information.

8.2. From BIM Expert to Diamonds

To send a model from BIM Expert to Diamonds, consult the the BIM Expert manual.

When the model is received by Diamonds, you'll get a notification with  . You than have the option to load the model or remove it.