RELEASE NOTES

Release notes IDEA StatiCa 24.1

Oct 23, 2024

Table of Contents

Version 24.1 is focused on engineers' everyday tasks no matter if you are steel connection designer, concrete engineer or precast specialist.

Concrete Design

Detail 3D (Eurocode only)

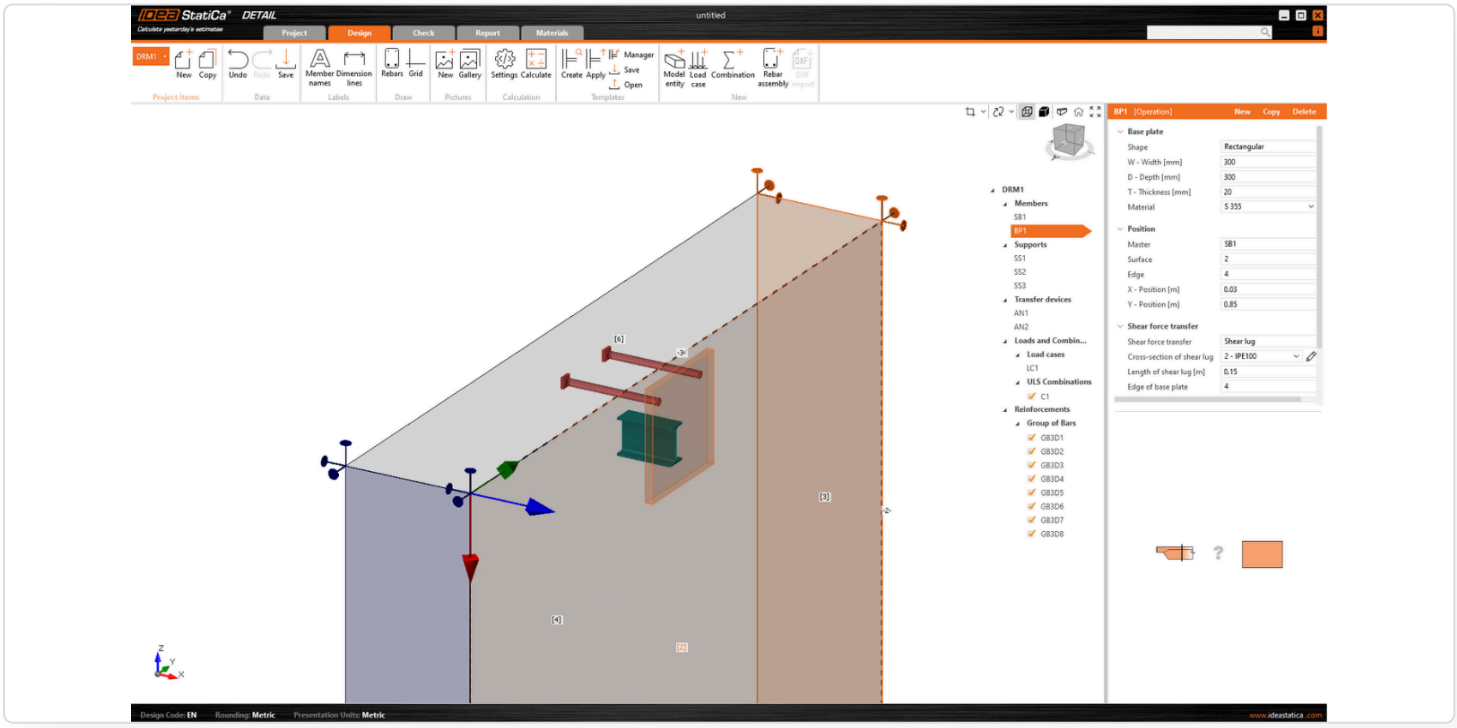
3D Detail out of Beta

Detail 3D is fully functional and verified for all types of Anchoring. Verifications have been added, and functionalities have been expanded from modeling to results. In the following article, you will find all the possibilities and limitations.
IDEA StatiCa is a tool for solving complex 3D tasks and is fully verified for anchoring in concrete blocks. This solution allows you to perform designs without oversimplifications and provides checks based on the Ultimate Limit State (ULS). Thus, we have a tool to capture all types of concrete failures for footing. Together with IDEA Connection for anchorage verification, we offer a comprehensive package for everyone dealing with steel-to-concrete connections.

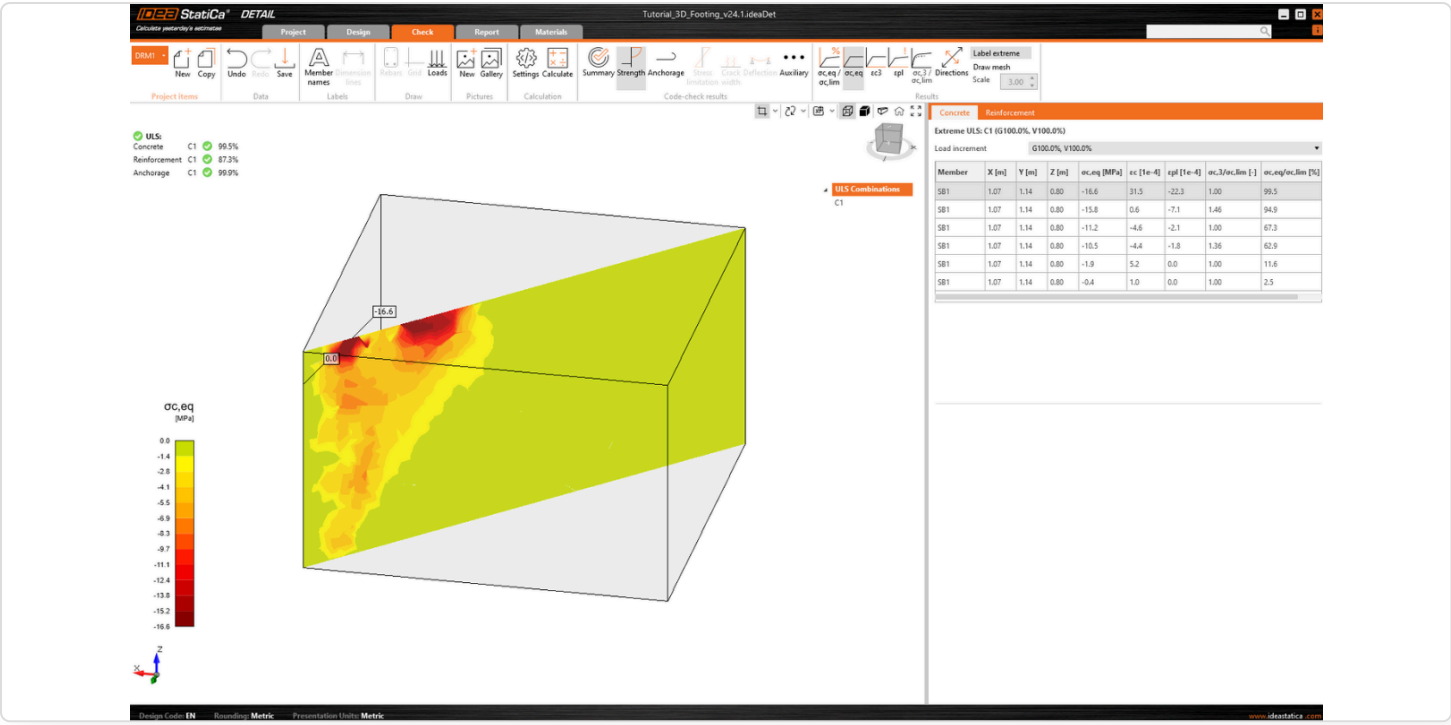
Enhanced shear transfer, improved results, and simplified integration between **IDEA StatiCa Connection** and **IDEA StatiCa Detail**, along with the verifications, means the solution is now fully capable of handling real, practical examples of any kind of anchoring.

What are some key improvements compared to the BETA version?

- **Shear transferring devices:** Fundamental entities that enable the general design of all types of anchoring **(Anchors, Shear lugs, and Friction)**. There are all types corresponding to the options in IDEA StatiCa Connection, allowing smooth import. Read the separate release note dedicated to Shear transferring devices.



- **Result Sections**: As part of the improvements for a better understanding of the results, Detail provides the opportunity to see the actual behavior of the structure. Read the separate release note: Results interpretation improvement



- **Full Connection export with new entities:** The import is possible, including all information about materials, types of anchors, and their end treatments. Read the separate release note: Import of anchoring from Connection to Detail
- **Working Grid Plane**: There is a new grid to help surface identification during modeling, reinforcement, creating sections, etc. Read the separate release note: Modelling improvement - Grids and Self-weight
- **Self-weight:** The application includes another load type, self-weight. It is automatically calculated based on the dimensions and the selected material.
- **Refined mesh around anchors:** The mesh around the anchors has been locally refined for accurate results. This setting cannot be changed. The mesh is generated automatically.

Comprehensive functionality description of 3D Detail.

Note: Currently for Eurocode (EN) only.

Known limitations for Detail 3D

Since Detail is just a tool that cannot replace engineering judgment, a safe understanding of its functions, benefits, and limitations is necessary. Read the limitations that must be taken into account:

- The solution is suitable only **for reinforced concrete.**
- The application provides **ULS checks** according to EN.
- **Only one concrete block** is supported in Detail.
- In Detail, the anchors are only checked for tensile strength. It is necessary to **use Connection for shear and interaction checks**.
- Only models **anchored via the base plate** and **only Direct contact** can be imported to Detail (from Connection).
- **Imported loads and user-input loads cannot be combined within one model.**

For a full list of limitations with further explanation, see the article: Known Limitations for 3D Detail

Verifications

We emphasize once again that although Detail 3D is a general solution for modeling any detail, we verify examples step by step. The current focus is mainly on verifying functionalities related to anchoring. See the overview of what are possible use cases and plans for further development below:

Use-case	Functionality ready	Verification provided	Verification in version
Footings of steel columns	YES	YES	24.1.0
General anchorings of steel members (beams, bracings, lifting lugs...)	YES	YES	24.1.0
General use of the Detail 3D for other cases (Pile caps, Pier caps, etc.)	YES	NO	in development
Wall model type	NO	NO	in development

We continuously update and add new verifications for 3D Detail, where we verify the functionality and the assumptions. For a deeper understanding of the method, read the comprehensive Theoretical Background, which includes the main assumptions, a description of the material model, and more.

Released in IDEA StatiCa version 24.1

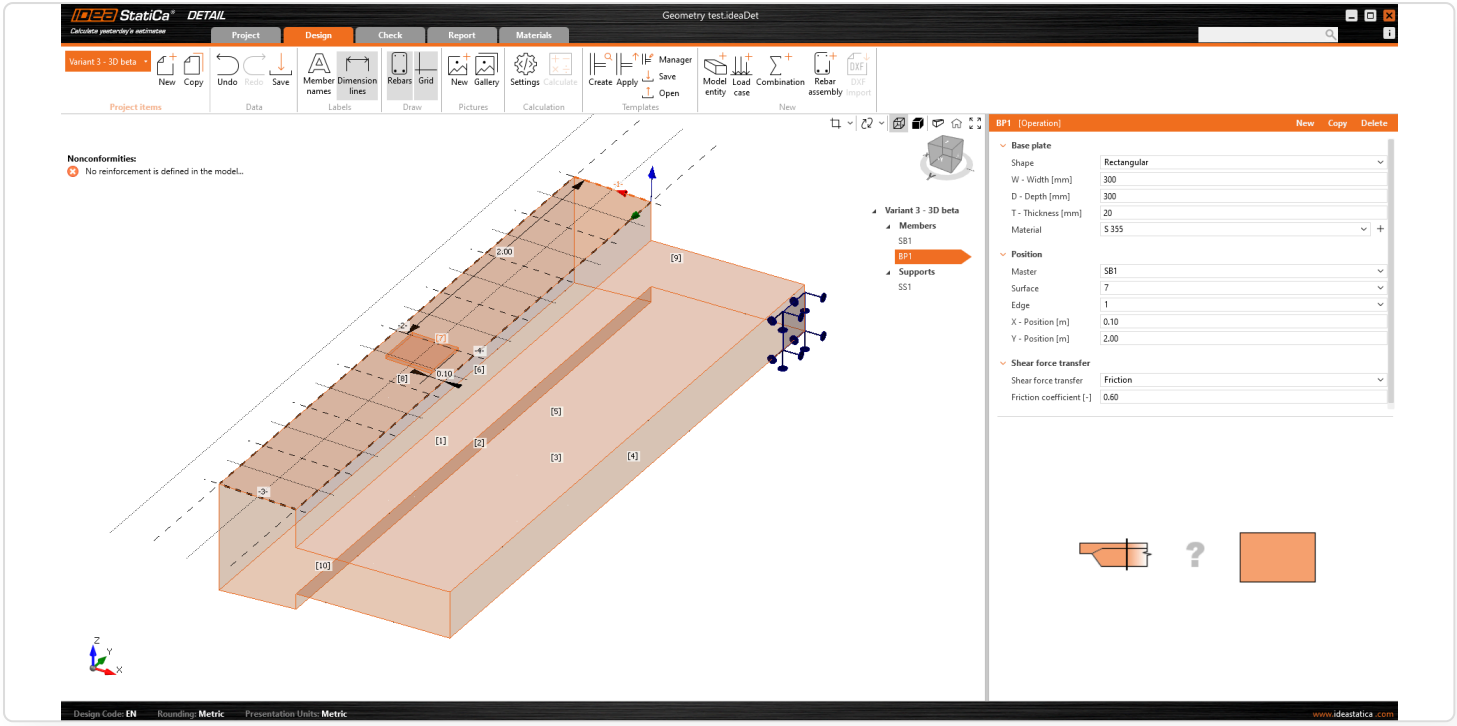
Shear transfer through anchors, shear lugs, and friction

In IDEA StatiCa Detail, various types of devices transferring shear to the concrete are implemented to ensure that all cases can be included. Besides the possibility of considering pure friction between the base plate and concrete, shear transfer through anchors or shear lug is also available.
See the following article for a detailed description of all options and their input:

Load transferring devices in 3D Detail

Load transferring devices

Load transferring devices contain two entities the base plate and single anchor. Let's start with the Base plate. To specify the position, a reference surface and edge must be selected. These define the origin of the coordinates from which the X and Y distances are measured. There are two shape definition options, Rectangular and Polygon.



The base plate is connected to the concrete element by a contact that transfers compressive stresses and, if the user chooses, can also transmit shear stresses. There are three shear transfer mechanisms that can be selected:

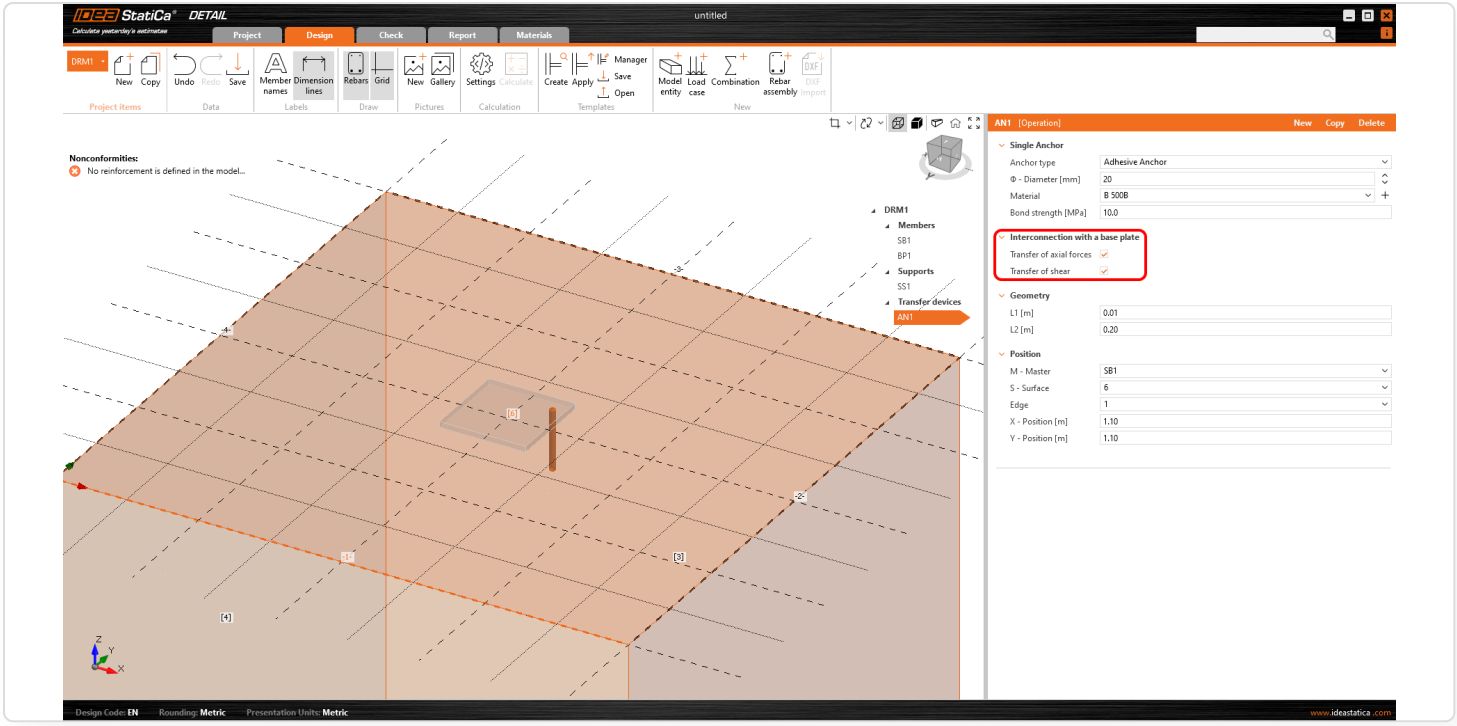
- **by friction**
- **by anchors**
- **by shear lug**

The software does not allow you to combine these shear transfer mechanisms.

For the option by friction, the design value of the friction coefficient needs to be entered. For the option by shear lug, the steel profile, including geometry and position, needs to be inputted.

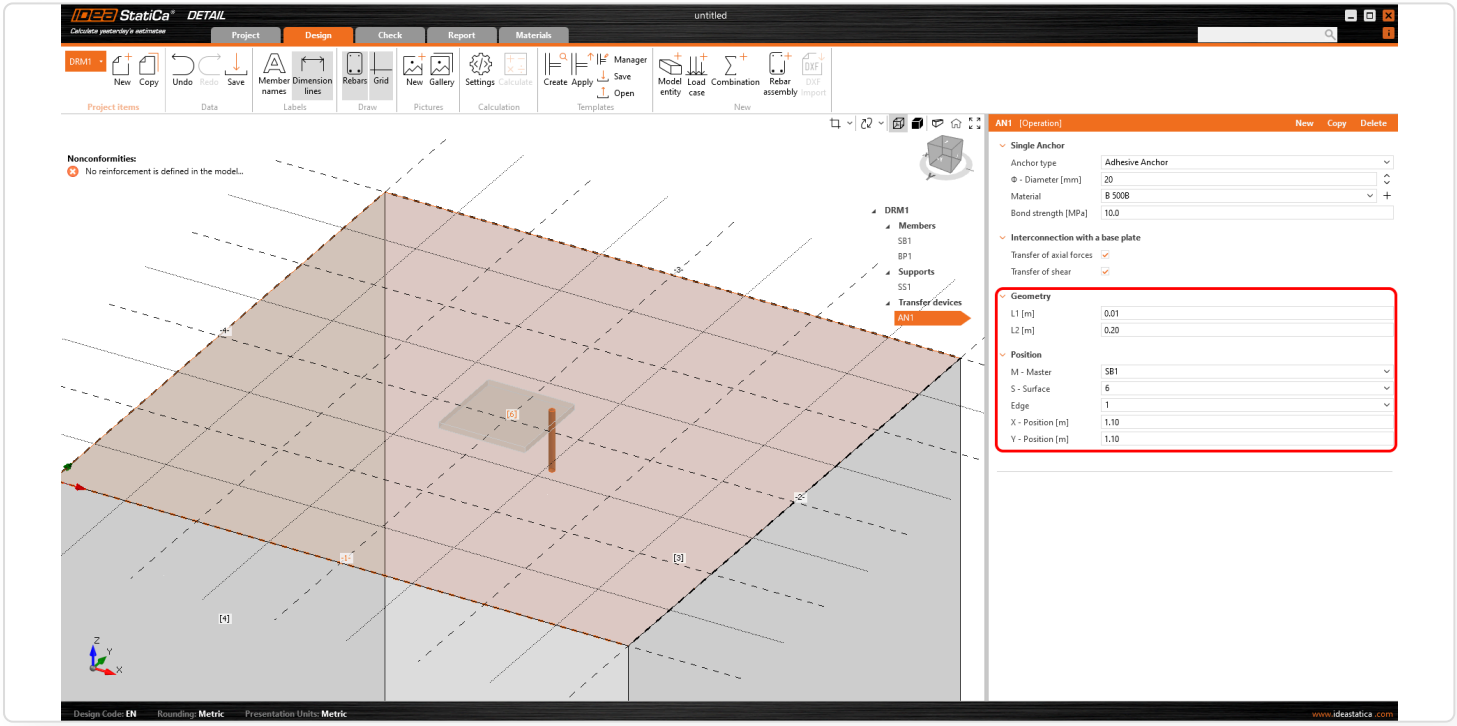
The base plate can transmit either a point load or a group of forces. For a point load, the model can be loaded with six internal forces (Fx, Fy, Fz, Mx, My, and Mz) at any position on the base plate. For a group of forces, users can input the forces' positions, intensities, and directions into a table, allowing for a general positioning on the base plate.

A second load transfer device, the single anchor, can be added and interconnected with the base plate to create, for example, a base plate of the column anchored with four anchors (see the figure below). It is also possible to model separate anchors without a base plate.



More information about the interconnection with the base plate can be found in the Theoretical background.

In terms of position and geometry, the anchors are referenced to the surface and edge of the block, including the determination of the relative position as with the base plate. Of course, it is possible to specify the length of the anchor in the concrete and the length above the concrete surface.

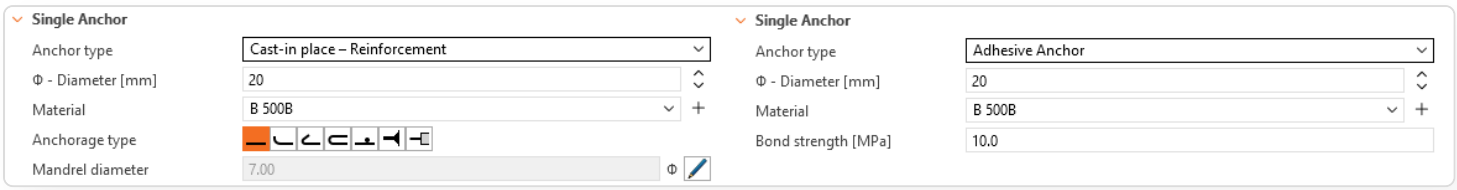


The anchors are implemented in two variants:

- Cast-in-place - Reinforcement
- Adhesive anchors

For the Cast-in-place Reinforcement, the Bond strength and is used, according to EN 1992-1-1 chap. 8.4.2. In addition, it is possible to specify the Anchorage type for this type of anchor as for conventional reinforcement.

For Adhesive anchors, it is possible to directly input the bond strength, which the user can find out from the technical data sheet of the applied adhesive mortar. Note that **it is necessary to input the design value of the bond strength**.



A thorough description of the behavior of the interconnection between the anchor and base plate is described in the Theoretical background.

Known limitations

Since Detail is just a tool and cannot replace engineering judgment, a safe understanding of its functions, benefits, and limitations is necessary. Read the following limitations, which must be taken into account:

- In Detail, the anchors are only checked for tensile strength. It is necessary to **use Connection for shear and interaction checks**.
- Only models **anchored via the base plate** and **only Direct contact** can be imported to Detail (from Connection).

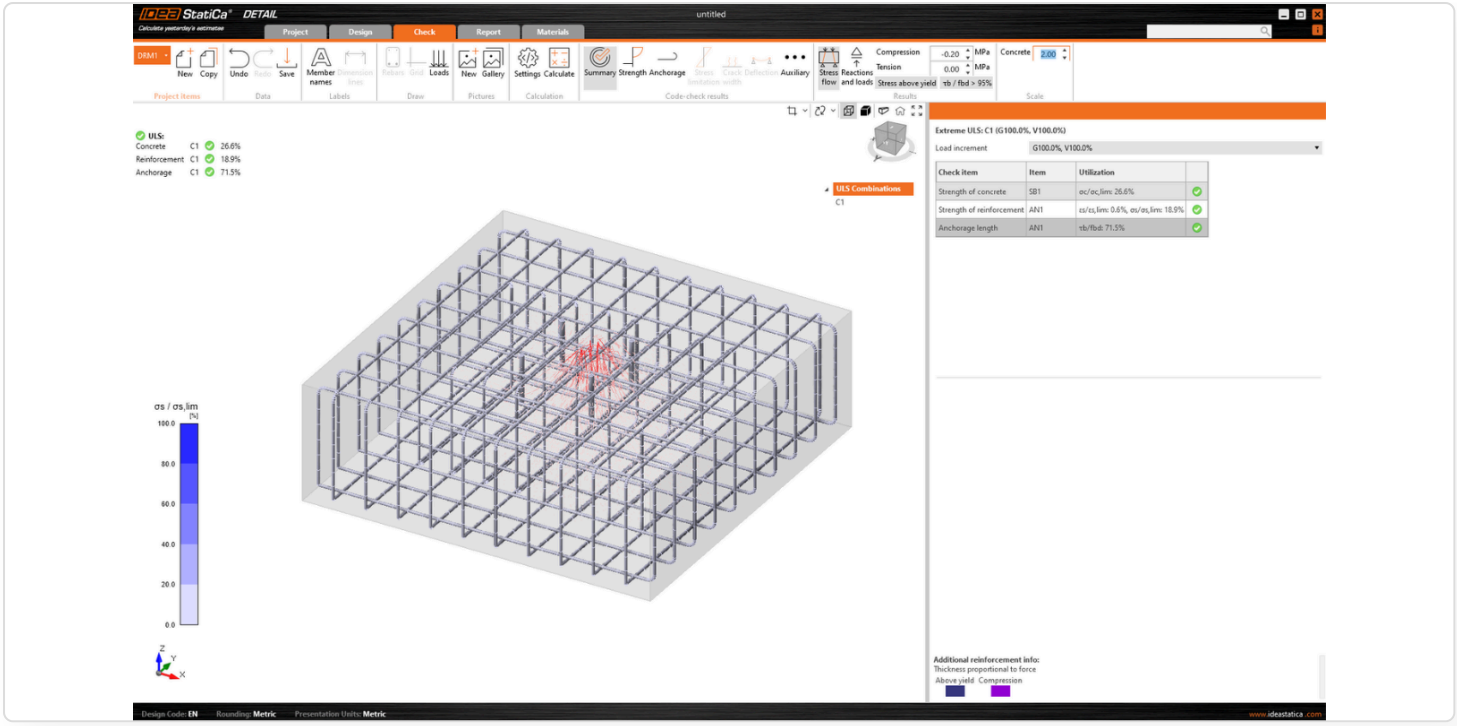
For a full list of limitations with further explanation, see the article: Known limitations for 3D Detail

Released in IDEA StatiCa version 24.1

Import of anchoring from Connection to Detail

Is anchoring into concrete blocks the biggest challenge for you? Our two innovative applications work together to provide you with a better workflow. Learn more about the link between IDEA StatiCa Connection and Detail.

Anchoring in a plain concrete block can be modeled and code-checked in IDEA StatiCa Connection. Sometimes, it could be useful or necessary to reinforce the concrete block. Although, this capability isn't available within the Connection app, we have 3D Detail. 3D Detail is focused on solving anchoring into concrete blocks and analysis of both the anchoring elements and the concrete block itself. Moreover, a direct link is implemented between the Connection and Detail applications to simplify the process.



Connection users who design **anchoring according to Eurocode** can **import their model from Connection to the advanced 3D Detail by one button click**.

How does it work?

- Import is allowed just for anchoring. If there is no concrete block in the Connection model, the export to Detail button is disabled.
- The model in Connection has to be calculated. If results are not available, the export icon is disabled with the tooltip: "Results unavailable, calculate the model first"
- When the anchoring model has been calculated, the import button is enabled.
- Only one concrete block for the import/export is allowed.

For a full list of limitations with further explanation, see the article Known Limitations for 3D Detail

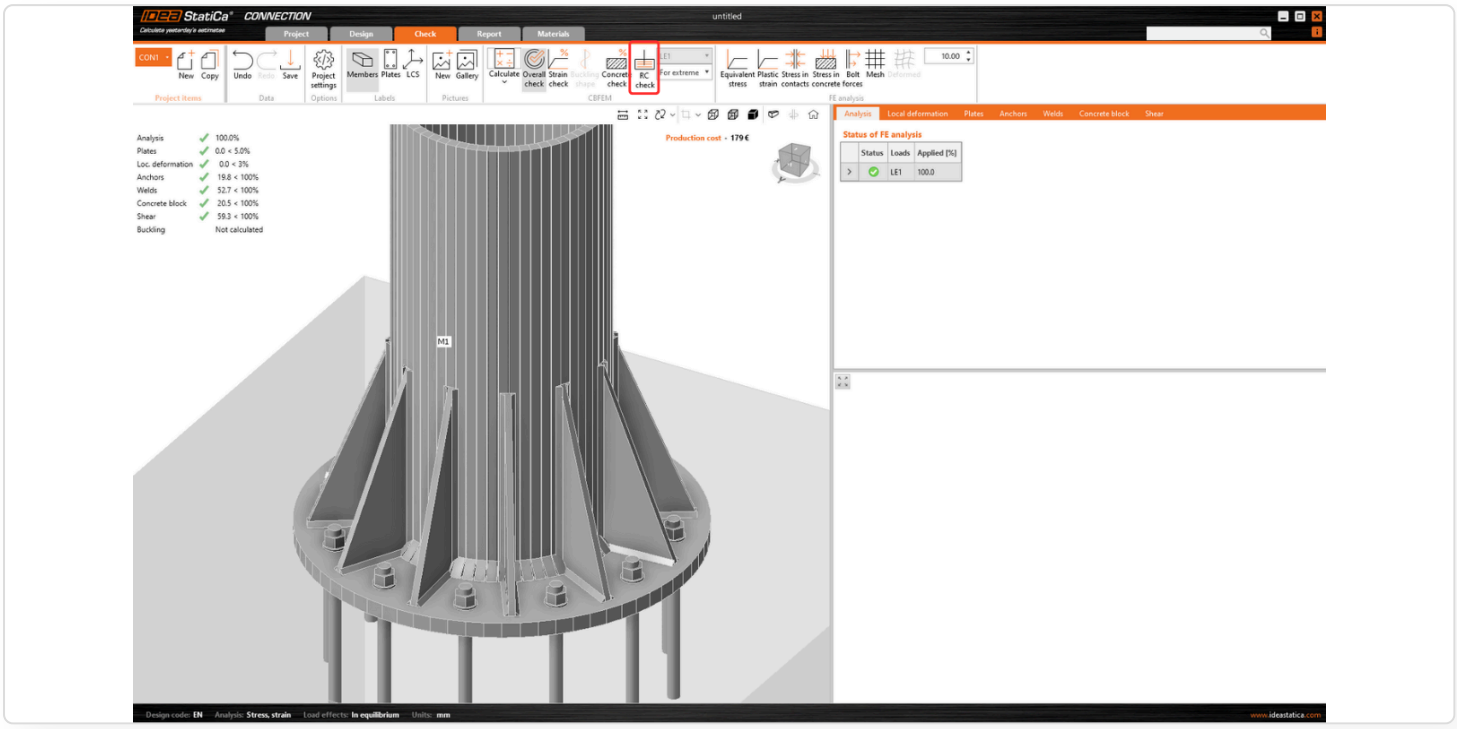
The connection is imported, including

- The model
- Reinforcement
- The base plate
- Loads
- Shear transfer (through Anchors, Shear lugs, and Friction) according to setting in the Connection*
- Material*
- Anchorage Type: Adhesive/Cast-in place*
- Anchorage type at the end: Washer/Straight/Hook*
- Friction coefficient*

How to export anchoring from Connection to Detail

First, create a model of anchoring in Connection according to Eurocode and click the Calculate button.

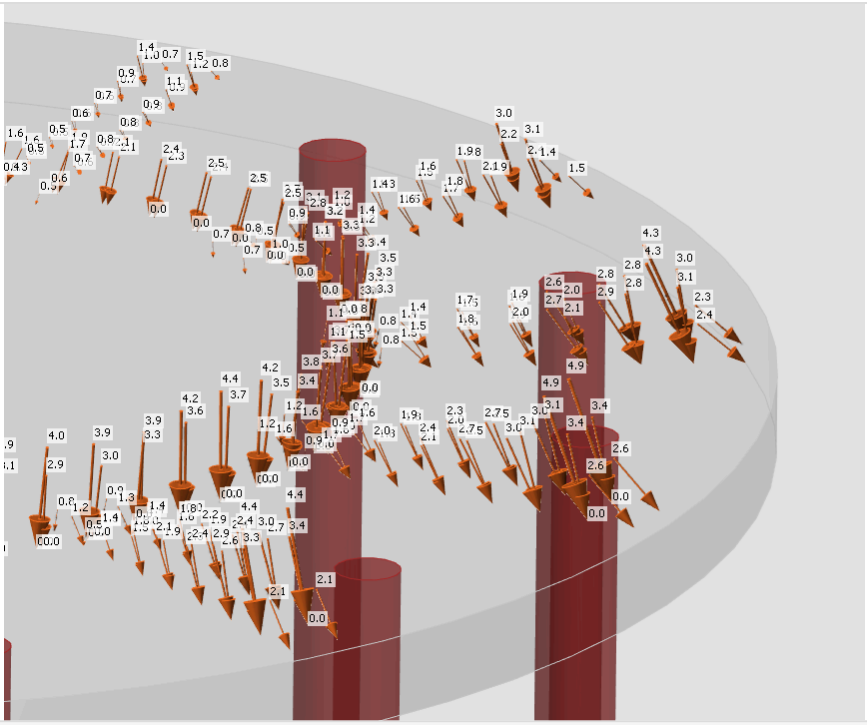
When results exist, export of footing is enabled. By clicking the button "**RC Check**" in the ribbon, a dialog asking for the location and the name of the newly created Detail file appears.



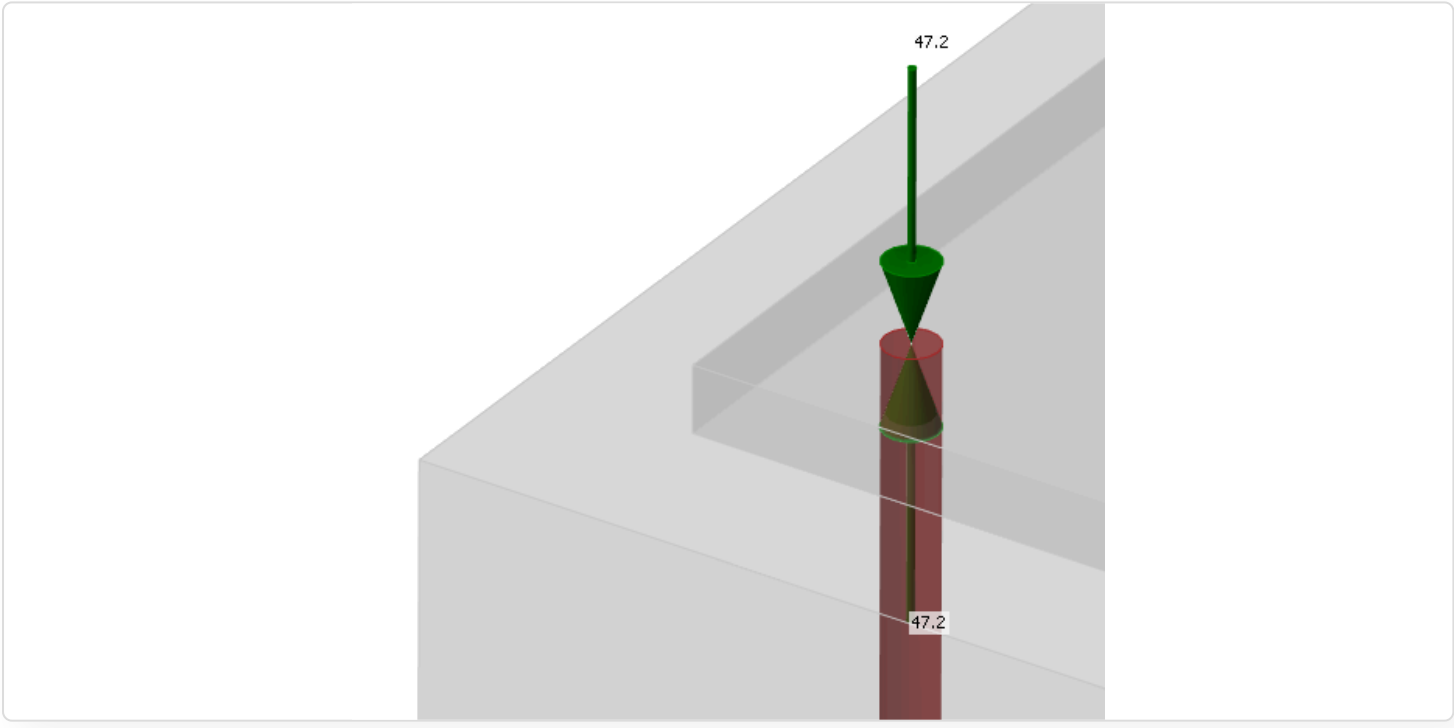
After a successful export, the project in Detail is created. The geometry of the concrete block and the base plate, the position and properties of anchors, and the load are automatically transferred to Detail. Surface support placed at the bottom surface of the concrete block is automatically created.

The most tricky part of this process is the import of the load. For every calculated load effect in Connection, the corresponding load case and the ULS combination are automatically created in Detail.

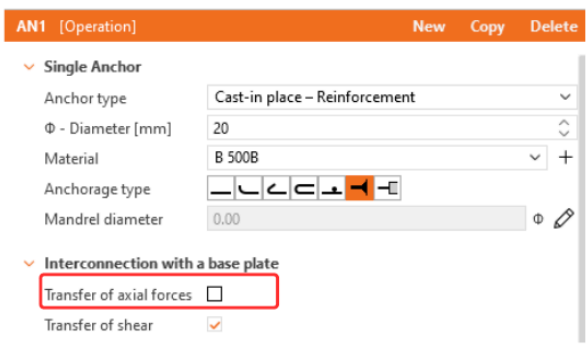
- The base plate is loaded by **forces in welds,** which are modeled as a **Group of forces.** For the loading of the base plate itself, the imported loading is represented by a group of forces following the stresses in welds between the base plate and steel members in the Connection model.



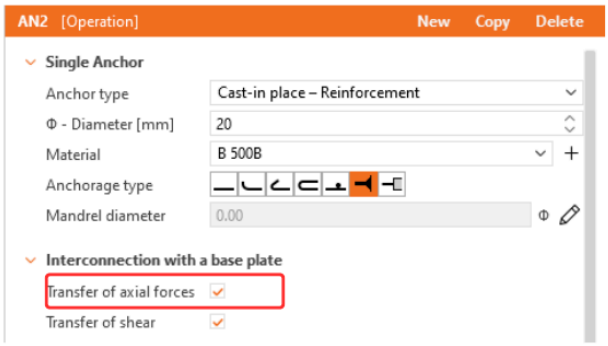
- Anchors are modeled and loaded independently from the base plate, and they are axially loaded by point loads. The loading of anchors is represented in the scene by a double of arrows in opposite directions. One arrow represents the tension force acting only on the top of the anchor. The other one represents the compression force acting on the base plate.



The Checkbox "Transfer of axial forces" is unticked by default as the anchors are loaded by forces directly.



1. Transfer of axial forces OFF - For Import from Connection (Anchors loaded directly by forces to each anchor).



2. Transfer of axial forces ON - For loading in Detail by force on the base plate (Anchors loaded through the base plate).

- Shear is transferred according to the setting in Connection by one of the options – anchors, shear lugs, or friction. If the shear force is transferred by anchors, you can turn off specific anchors by unticking the checkbox "Transfer of shear". If friction or shear lugs are set, shear in the anchors is never considered in the model.

The only step left is to add reinforcement and calculate the model.

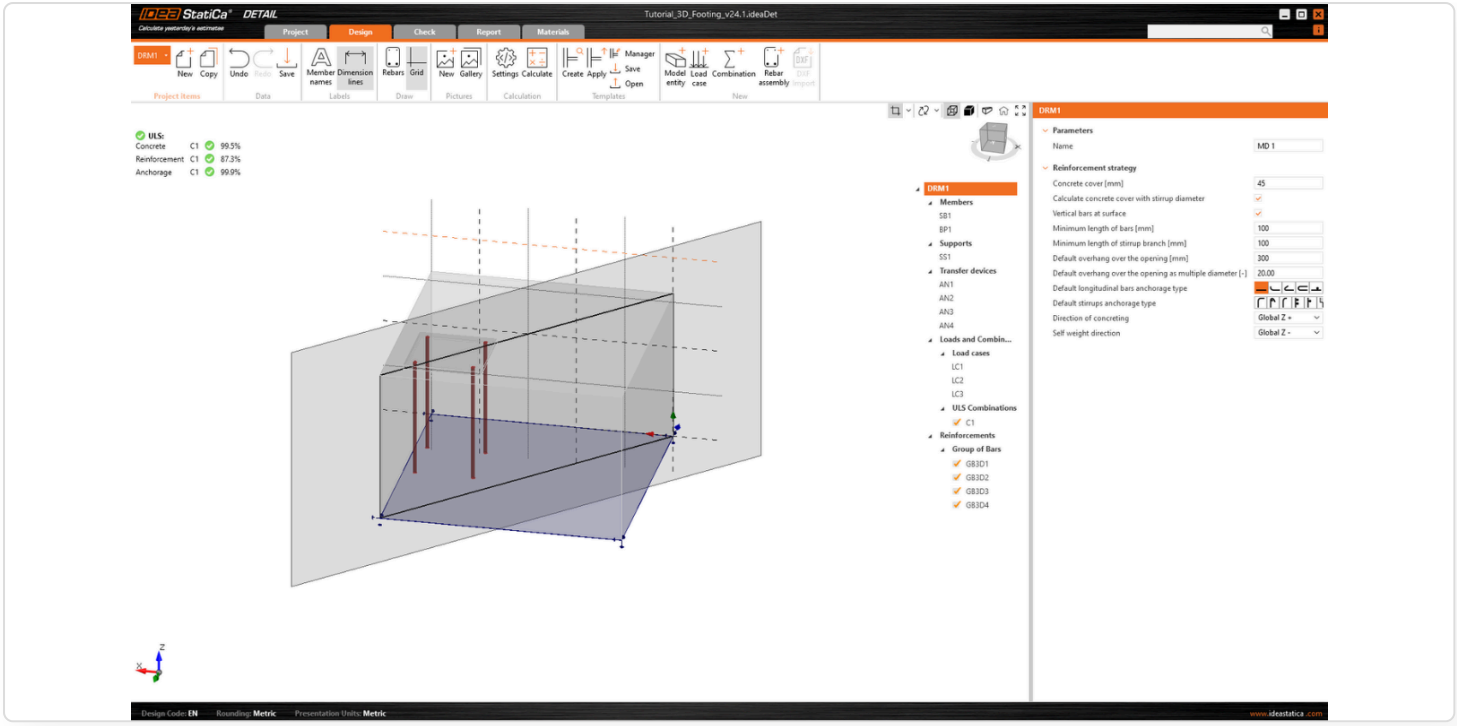
More general information about Detail as a solution for anchoring can be found in the article 3D Detail out of Beta.

Note: Code-checks in the 3D Detail are currently for the Eurocode (EN) only.

Released in IDEA StatiCa version 24.1.

Investigate reinforcement behavior with Section results and Stress check

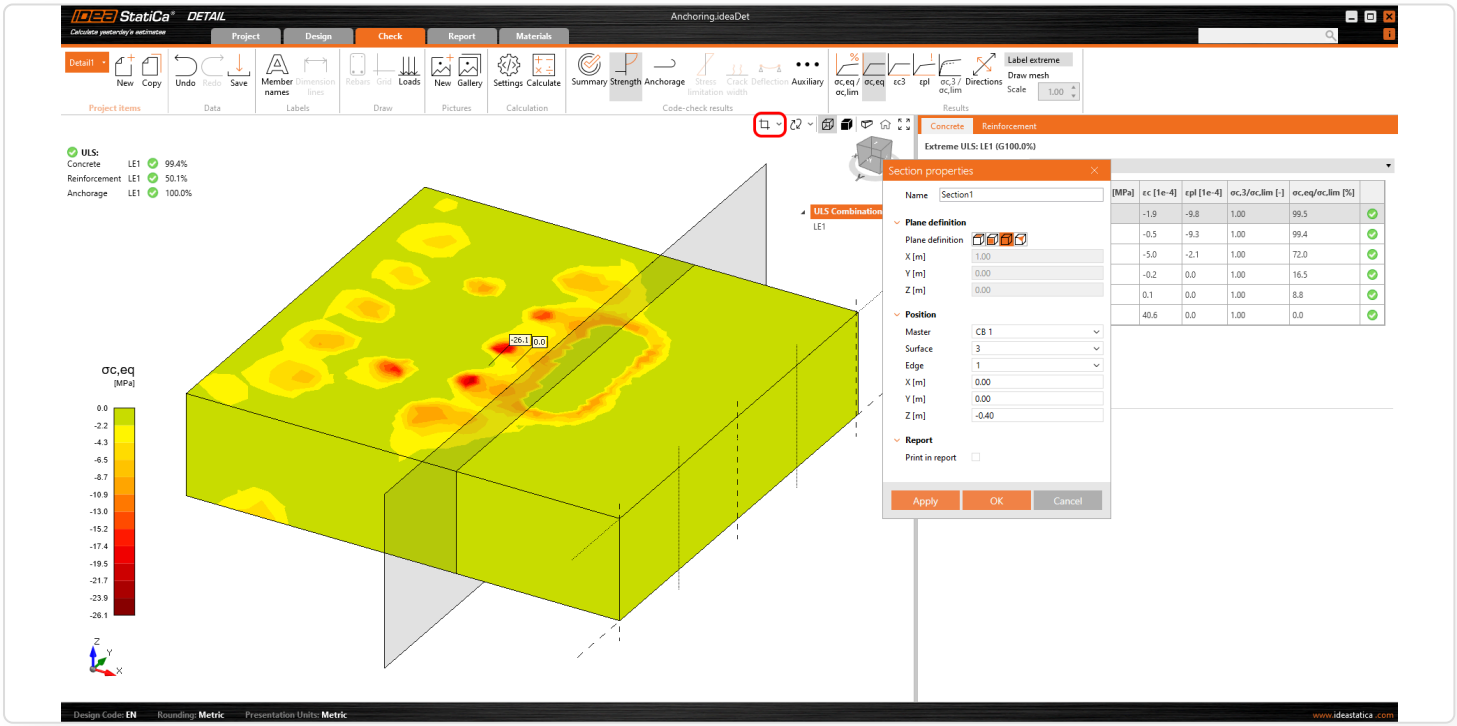
With section views, you can analyze results and their behavior within the structure, ensuring clear visualization in both graphical and tabular formats.
Section Results allow insight into the stresses within the concrete element. It is possible to create any number of sections and in any plane.



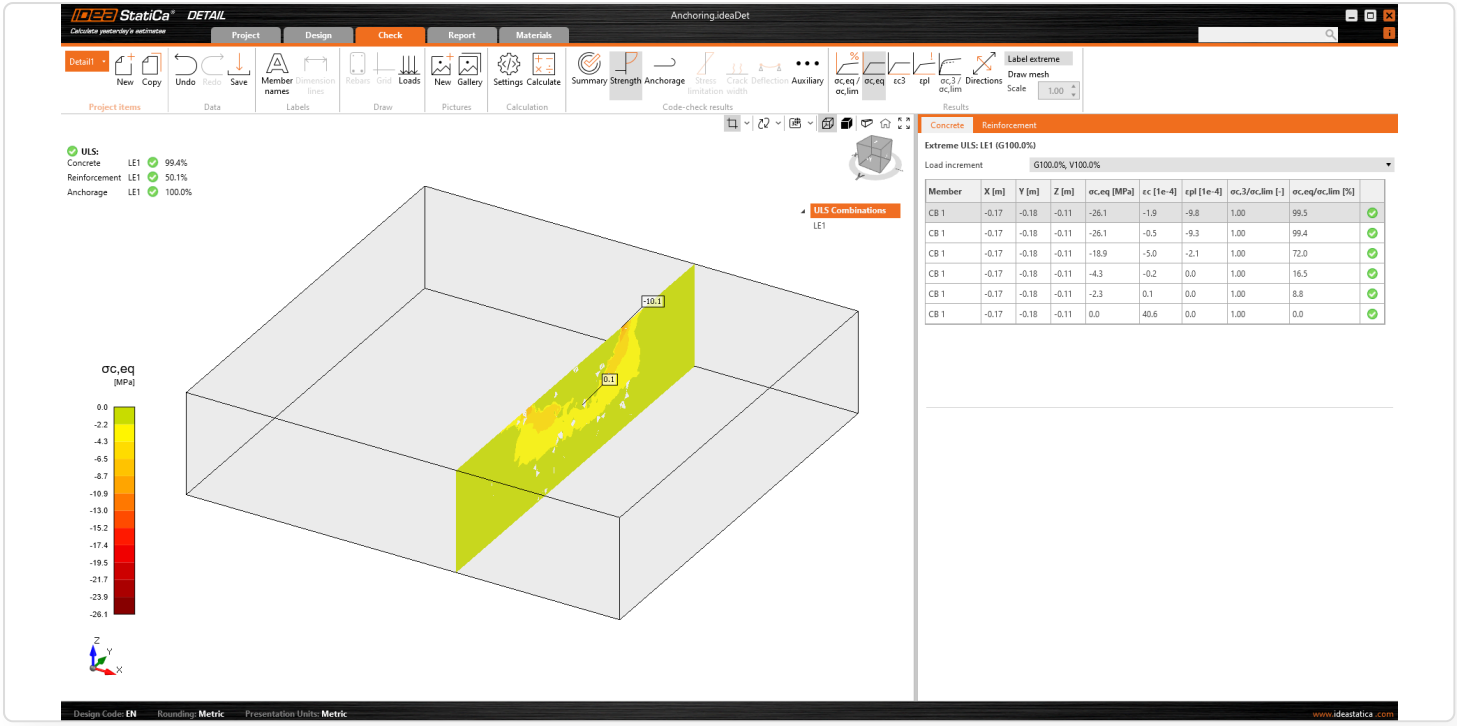
3D Detail Section results

Section results

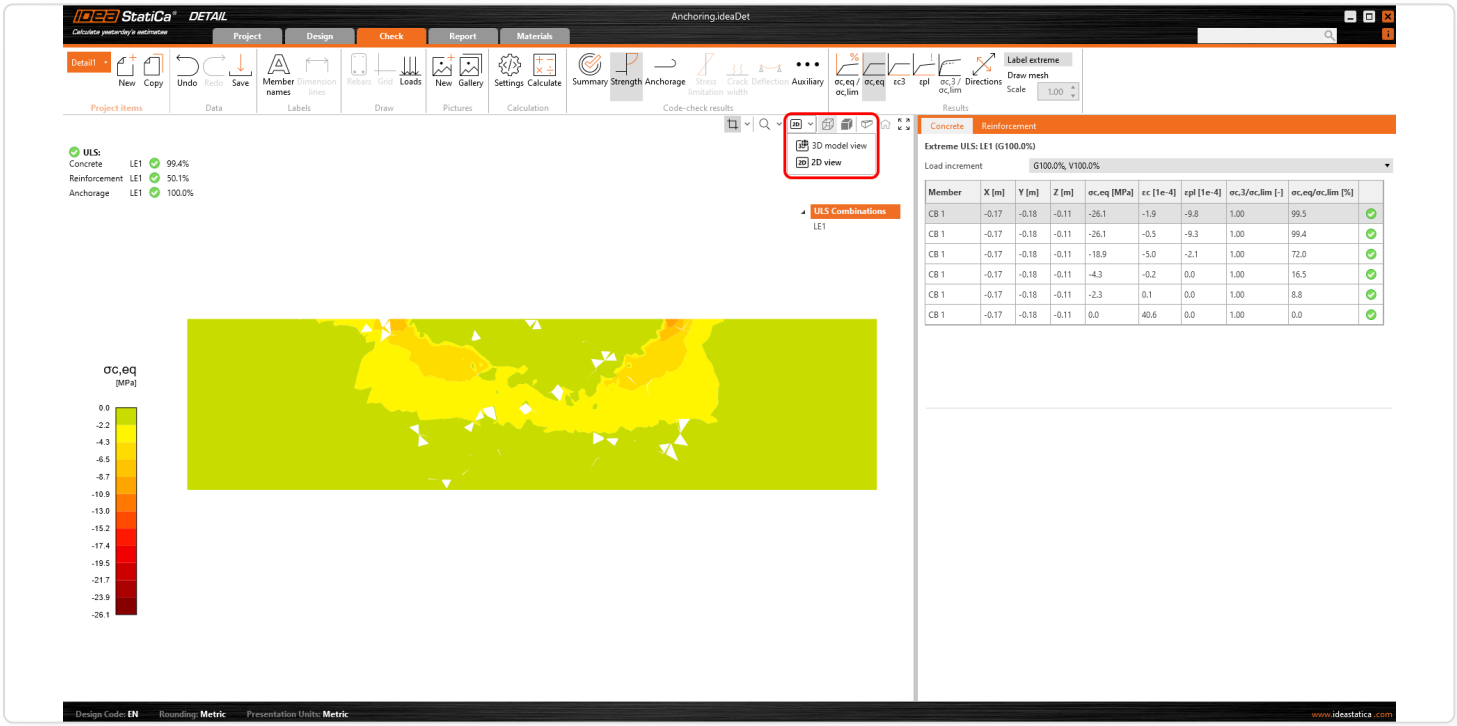
For 3D models, there is an option for displaying results for concrete - **Section results**. To define or modify the sections, you need to use the section button in the view control, which is in the top right corner of the scene.



Then you can simply turn on the section button and the results will be displayed via a specified section.



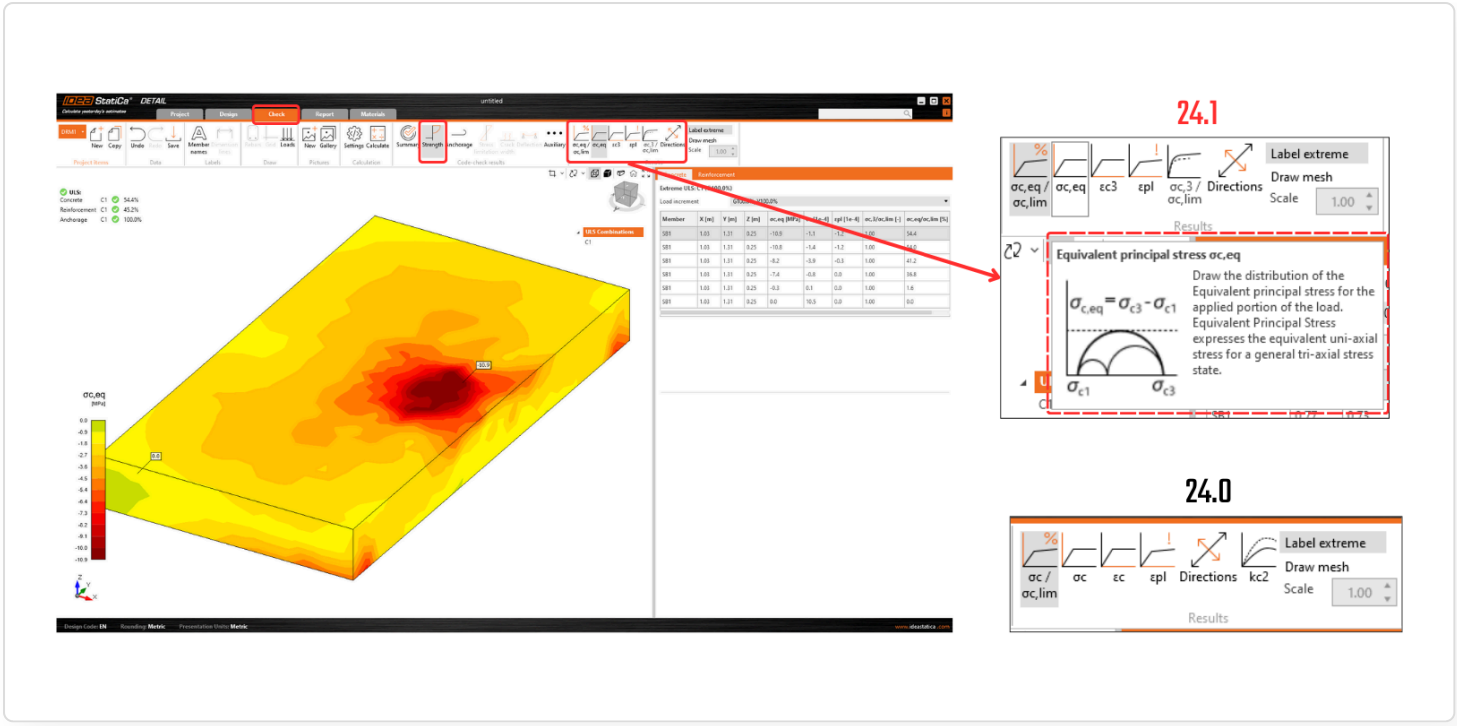
Or there is an option to switch the view from 3D to 2D and for better clarity display the selected section in 2D.



Released in IDEA StatiCa version 24.0.4

Stress check

For a better understanding of the results and the theory implemented in the 3D Detail, the iconography has been significantly improved. In the "Strength" section, under the concrete stress assessment, you will find new icons and, most importantly, tooltips explaining the basic theory. These tooltips correspond to the theoretical background.



Released in IDEA StatiCa version 24.0.2

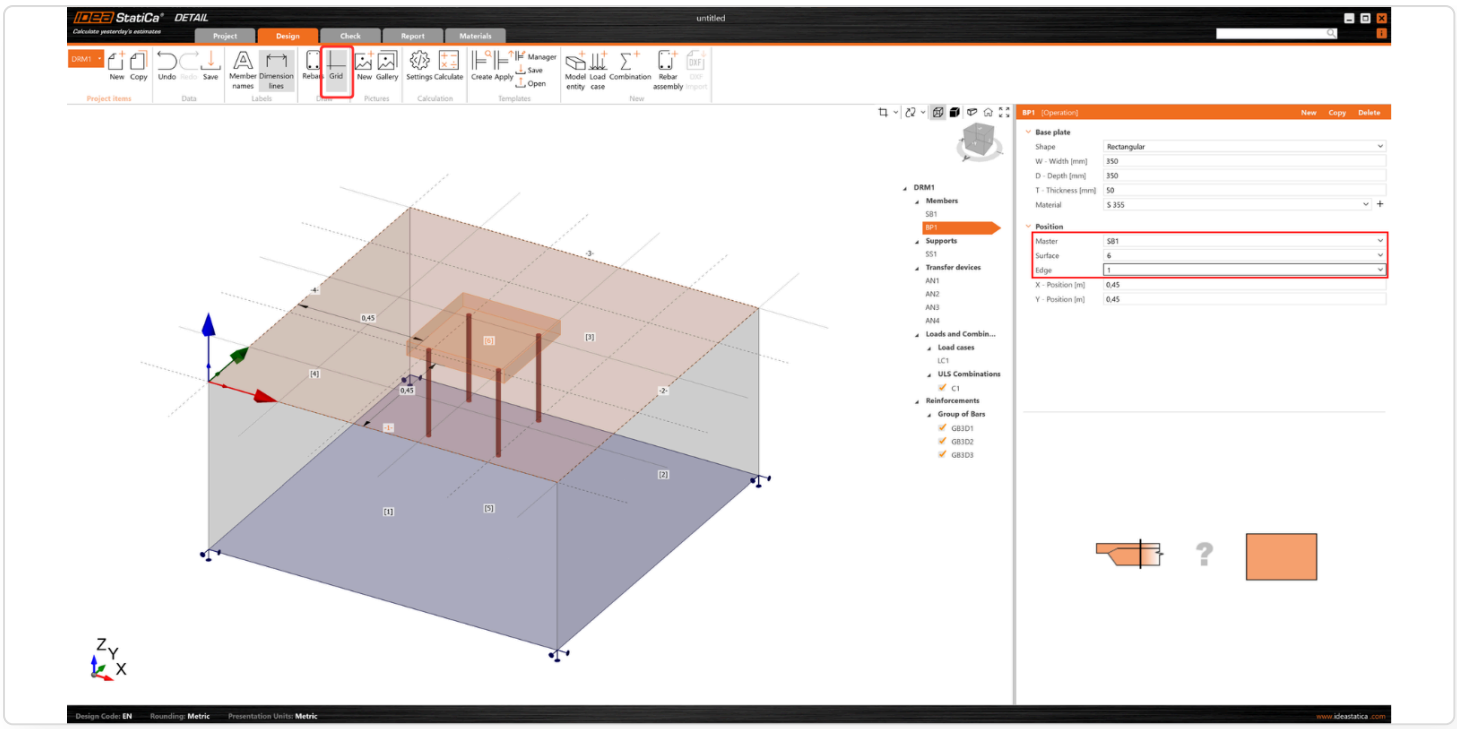
Modeling Grids and Self-weight

Development does not involve only verification and increasingly precise and stable calculations but also covers the creation of a user-friendly application. For quick input and modeling, there is the Grid for designing entities and a Self-weight type of load available.

Grid for designing new entities

When designing or editing entities in a model that includes a solid block, the grid in the local coordinate system is displayed in the scene, enhancing orientation within the 3D model. This grid is shown for base plates, supports, anchors, surface loads, and result sections.

Users can turn the grid display on or off using the button in the ribbon. The local coordinate system is established through settings in the property grid. The XY plane is defined by the selected surface, while the direction of the X axis is determined by the chosen edge. The grid size is fixed with a step of 0.25 m.

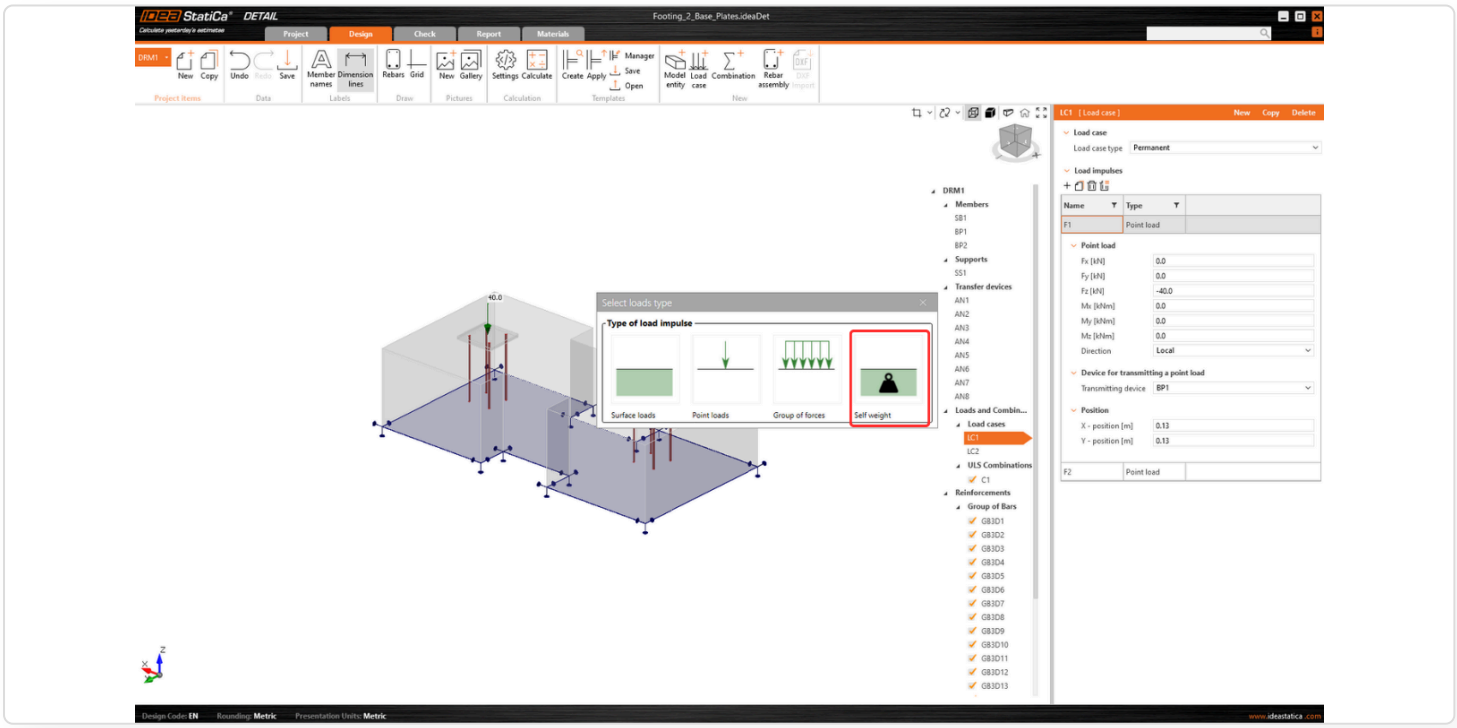


Released in IDEA StatiCa version 24.0.4

Automatic calculation of self-weight

The Detail app automatically calculates the self-weight of structural elements based on their dimensions and the selected material properties.

Four types of loads can be inserted: Surface Load, Point Load, Group of Forces, and Self-weight.



Other tips and functionality descriptions, not just related to modeling, can be found in the following article:
Comprehensive functionality description of 3D Detail

Released in IDEA StatiCa version 24.0.2

Detail 2D

Fast and intuitive modeling in Detail

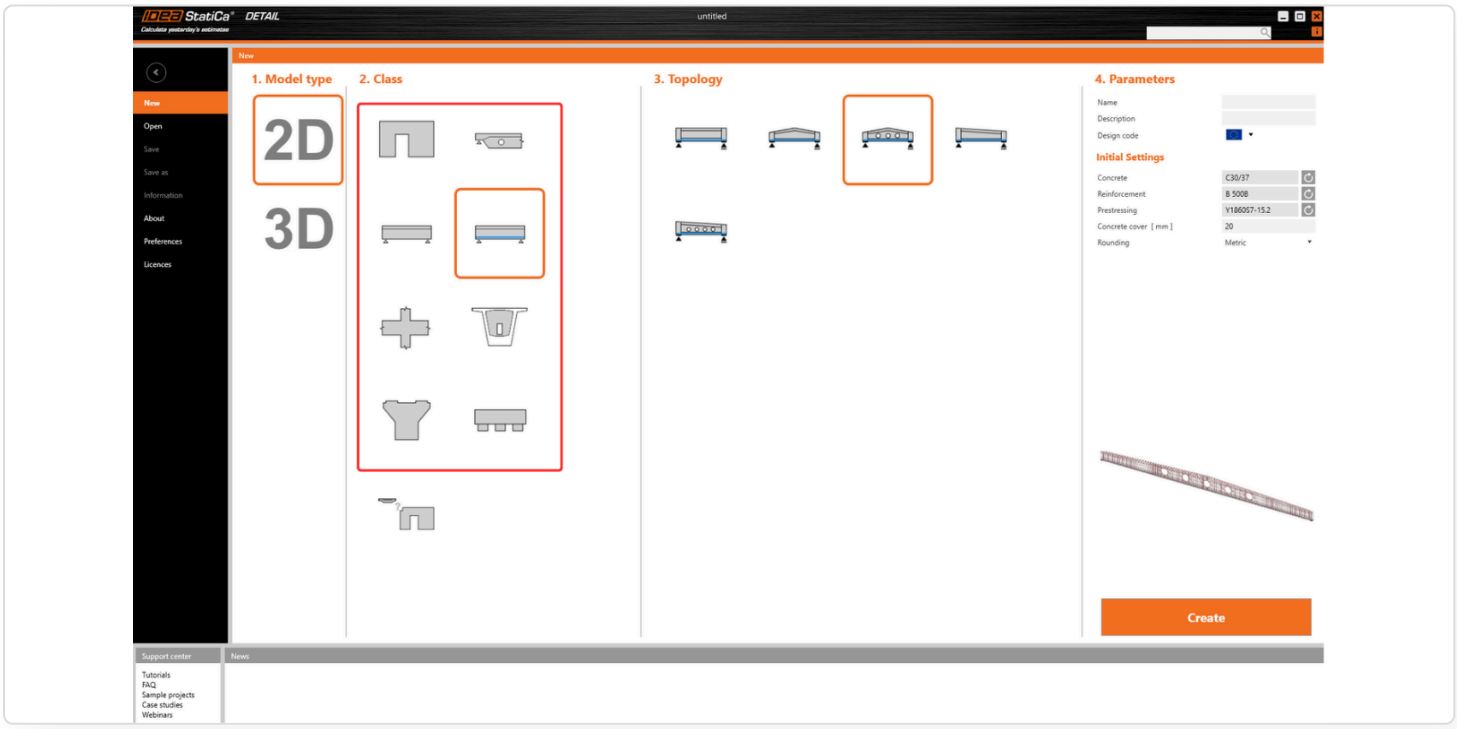
One of the greatest advantages of IDEA StatiCa Detail is its fast and intuitive modeling while still faithfully capturing the structure's behavior and displaying accurate results. Check out the features available for such efficient work, including templates for frequently used examples or others.
Whether the problem is the large number of entities in the project and lengthy modifications or the design where no one knows where to start, there is a solution.

A comprehensive set of templates

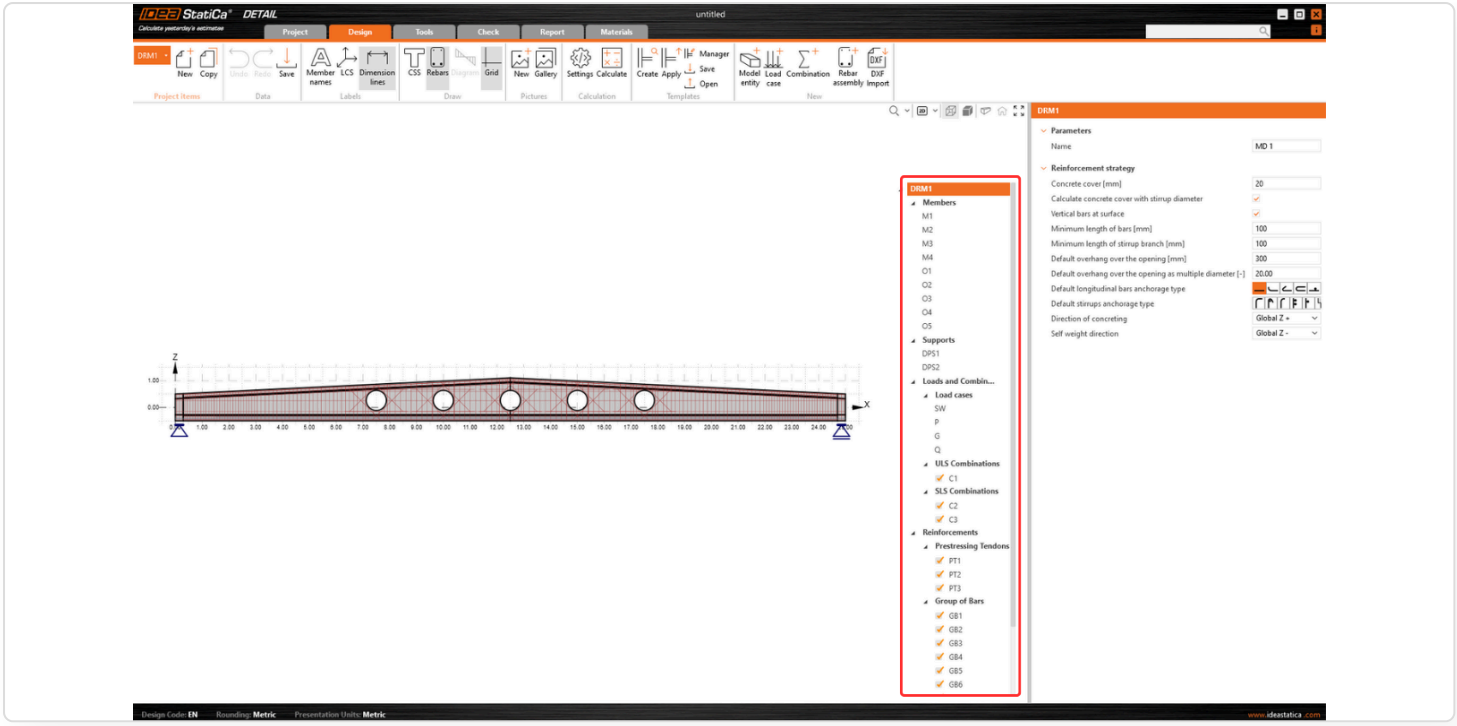
The expanded set of 2D Detail templates simplifies the design of concrete members with discontinuity regions. These templates not only make the design process easier but also inspire by showcasing various design possibilities within Detail. This template collection is available for concrete members code-checked according to **Eurocode standards**.

Supported types of members:

- Walls
- Beam cut-offs
- Reinforced concrete beams
- Prestressed concrete beams
- Frame joints
- Diaphragms
- Pier caps
- Foundations
- General

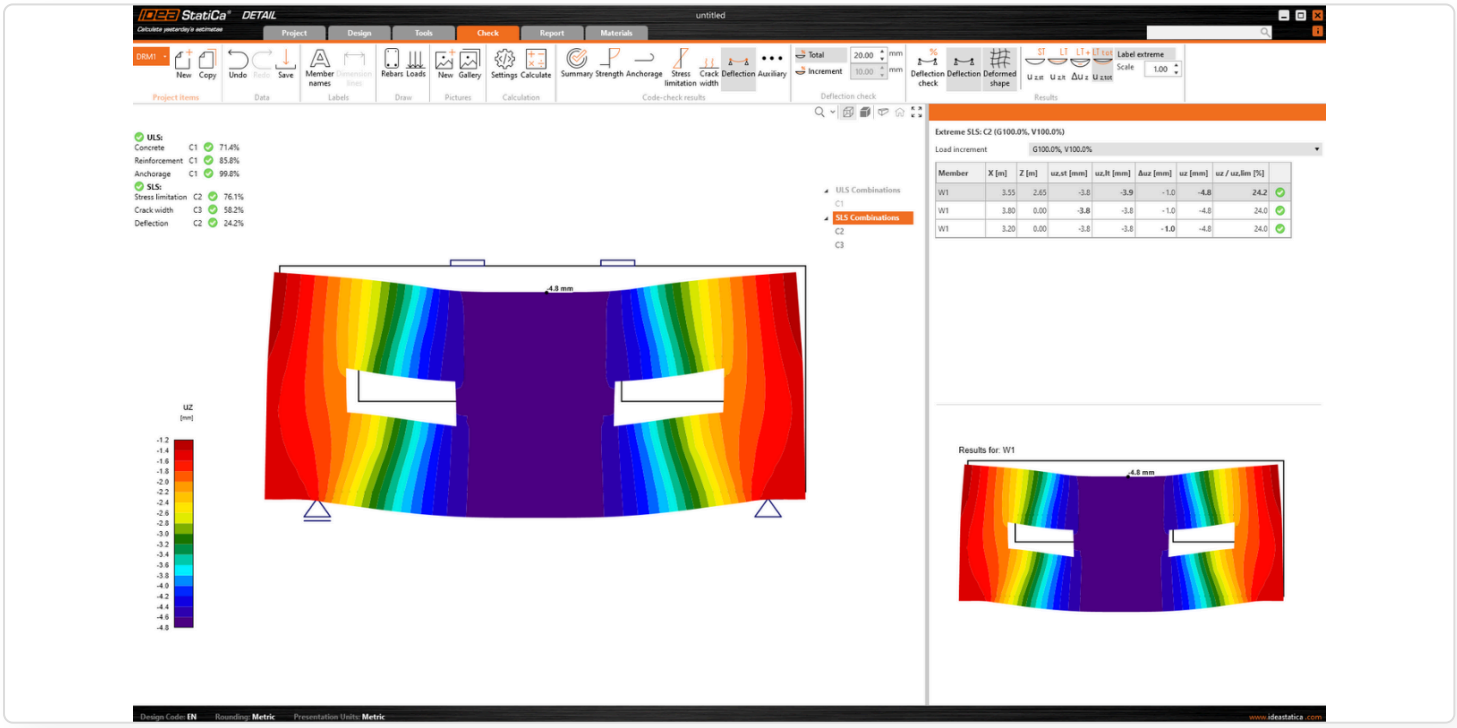


There are many options for each member type, which can be opened and modified according to the requirements. Alternatively, the settings of the template elements can be replicated for other cases. The template model is always created, including all reinforcement, several load cases, and necessary combinations.

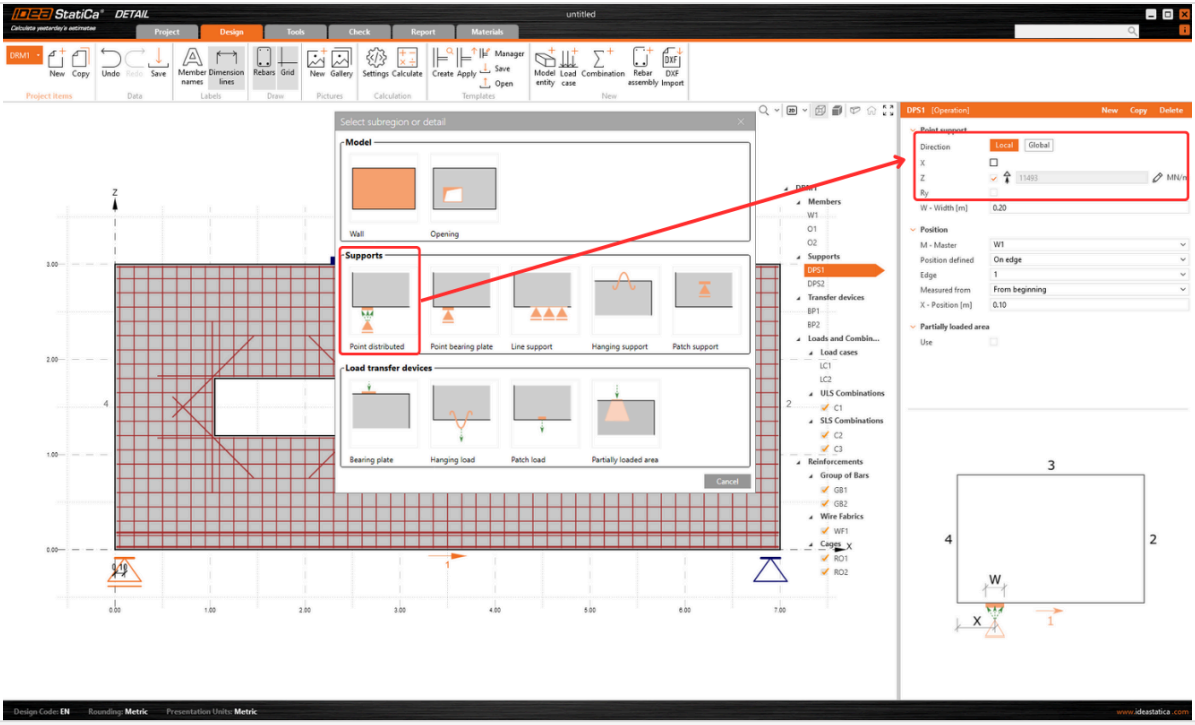


Flexible point supports with defined stiffness

Walls are frequently supported by columns with specific stiffness properties. Therefore, the stiffness modification for point supports is included to accurately model this boundary condition.



The option to set stiffness for the point support can be found when using the entity **Point Distributed Support,** and it is allowed only when **Local*** direction of the support is defined.



*If the **Local** direction is selected:

- The option to define non-linear behavior for the support appears as a button that toggles ON/OFF.
- The option to set stiffness in the Z direction is displayed. It is inactive by default with the value "Stiff". By clicking the "Edit" button, the stiffness can be defined as a positive number greater than 0.
- Stiffness can only be defined when the support is set as "On the edge". It is not available for the "Relative to the master point" option, where only stiff support is considered.

If the **Global** direction is selected:

- Stiffness cannot be defined.
- The option to define the partially loaded area is hidden.

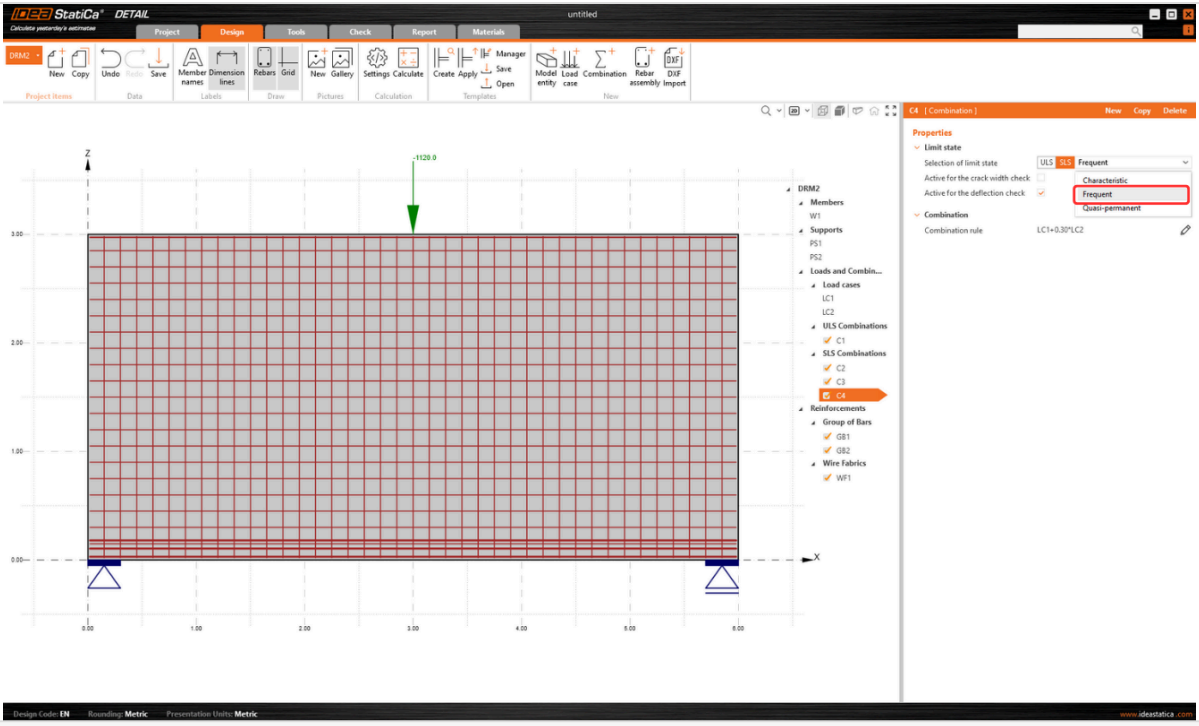
Released in IDEA StatiCa version 24.1

Complete SLS combinations in Detail

The current method for code-checking deflection and crack width aligns with the principles outlined in the General Eurocode for reinforced concrete structures. However, national annexes may impose different requirements compared to the General Eurocode.
Therefore, the "Frequent" SLS combination type exists together with the other options for customizing checks and, consequently, the display of results. This allows the calculation to be tailored precisely according to the rules in national annexes and, depending on the calculated elements, even for specific cases of prestressed structures.
Everyone now has the opportunity to fully utilize SLS code-checks in **IDEA StatiCa Detail**.

All combinations that may be needed

As already mentioned, the Frequent combination provides a set of combinations for SLS alongside the Characteristic and Quasi-permanent combinations. Each combination has its own significance and is used for a specific check.

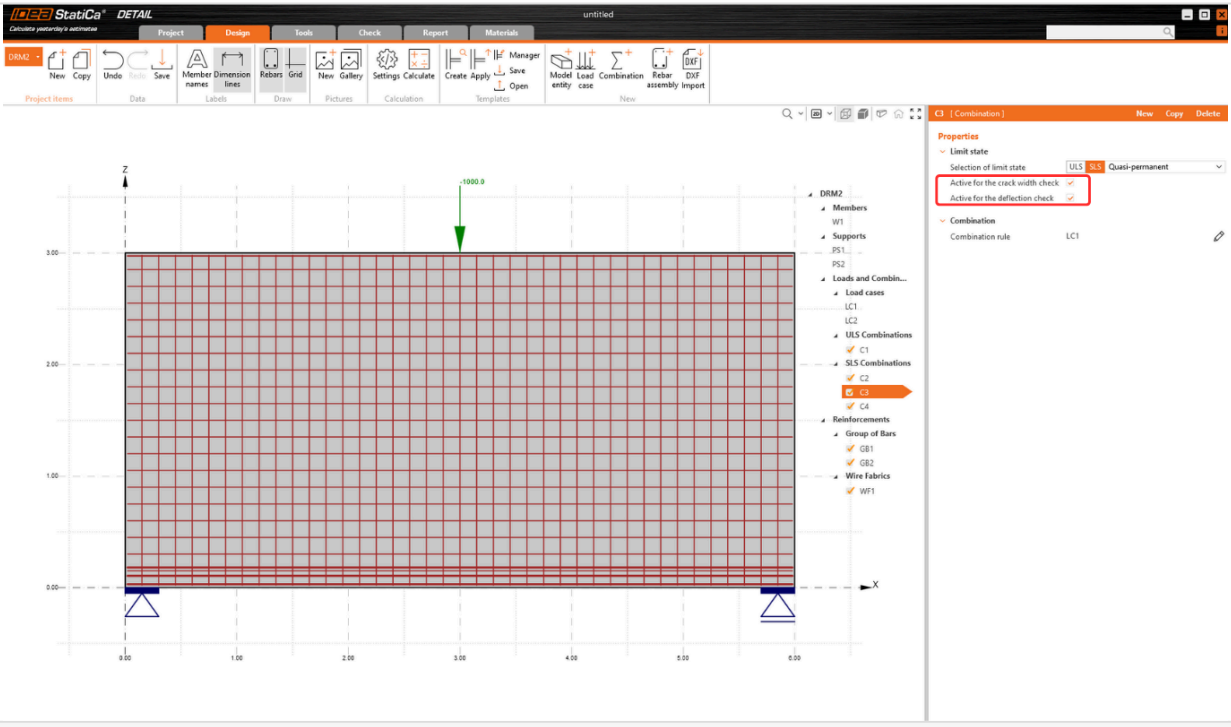


Default settings for the combination:

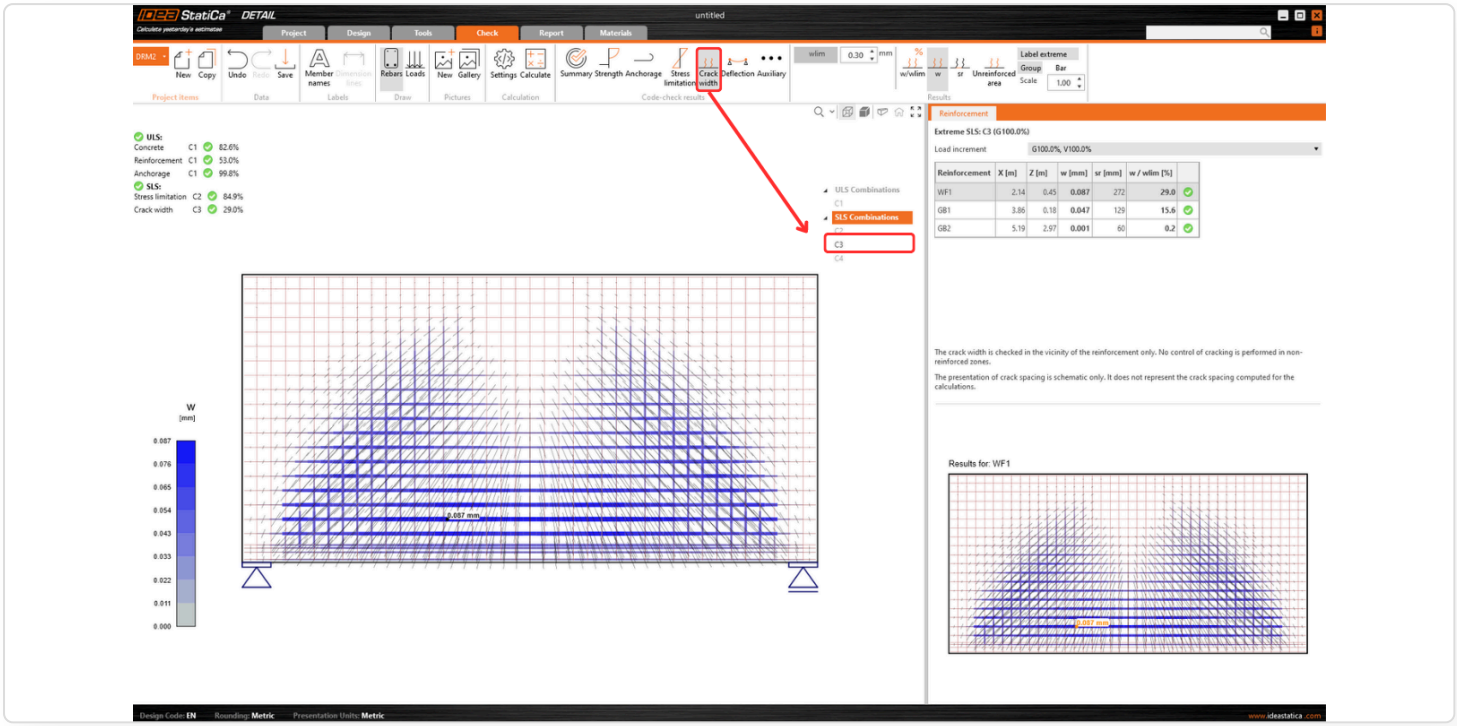
- **The Quasi-permanent combination** is used for crack width check
- **The Frequent combination** is used for crack width check
- **The Characteristic combination** is active for deflection check

Customizing checks without limitations

The checkboxes allow users to select any SLS combination for the **deflection** and **crack width** check. Users can specify which combination will be used for the deflection and/or crack width check by selecting the checkboxes in the Property grid.



This behavior is considered for both the calculation itself and the presentation of results in the Check tab. When displaying results for the crack width check, only combinations marked with the active property "Use for the Crack Width Check" will be visible in the entity tree. The same principle applies to the deflection check.



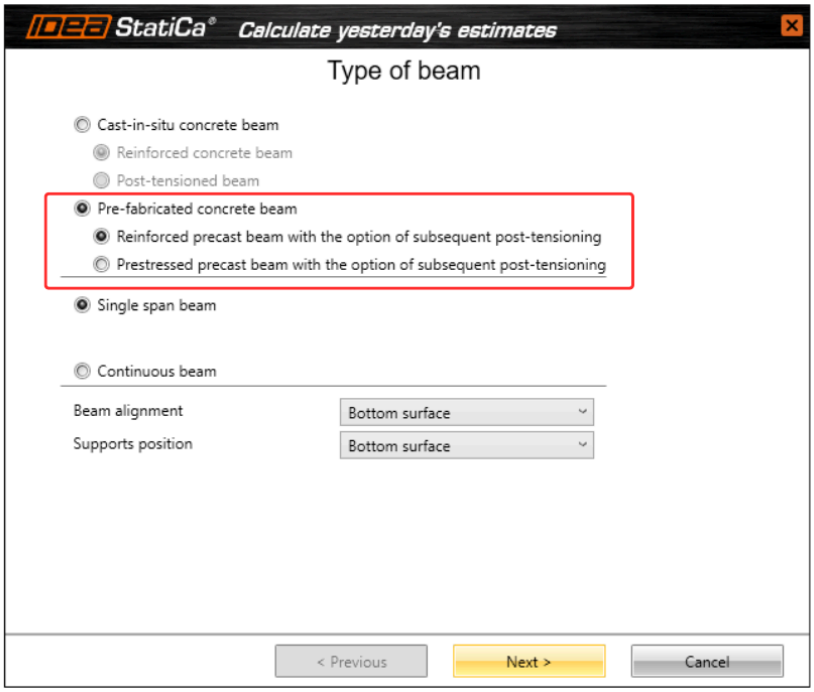
Released in IDEA StatiCa version 24.0.1

Beam

Lateral torsional buckling for prefabricated beams

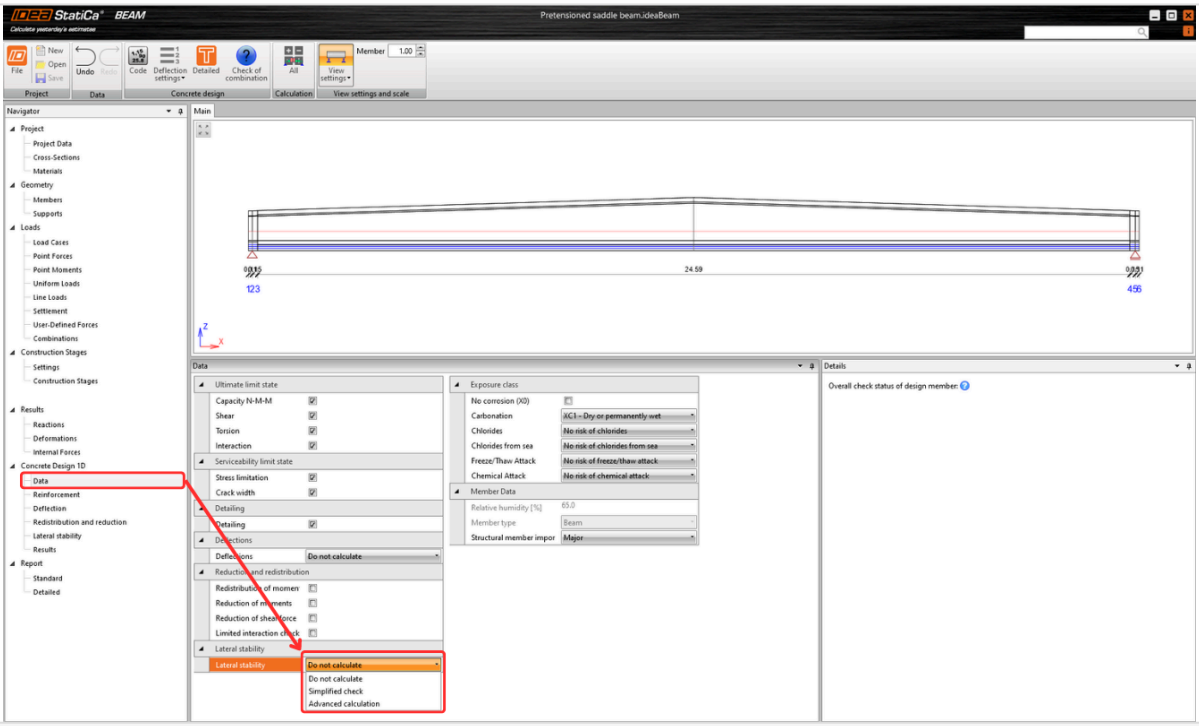
Lateral torsional buckling (LTB) is a stability issue that primarily affects slender members such as prefabricated beams. To address this issue, an advanced geometrically and materially nonlinear analysis incorporating initial imperfections is implemented in IDEA StatiCa Beam.

LTB is a stability failure that occurs in slender beams under bending, causing lateral displacement and twisting. It typically affects long and slender prefabricated beams. Checking for LTB is crucial to prevent sudden failure, optimize structural design, and ensure compliance with safety codes. It is essential to verify all construction stages, including lifting and transportation. The solution is suitable for any reinforced concrete and prestressed (pre-tensioned) **pre-fabricated concrete beams**.



Verify your design

The option can be selected in the **Design 1D - Data section** next to the Simplified Check and Do Not Calculate options. All necessary input belongs to the **Lateral Stability** section. In the case of simplified verification, only the basic dimensions need to be entered. For advanced analysis, more detailed input is required, including construction history, imperfections, and other parameters.



What impacts the analysis?

Construction stages

Each design situation requires specific inputs due to varying boundaries and times for the code-check. The times for each design situation can be set independently from the construction stages set at the beginning. Concrete properties, such as f_{ck} and E_{cm}, are automatically calculated based on the specified times but can be manually defined by the user if needed.

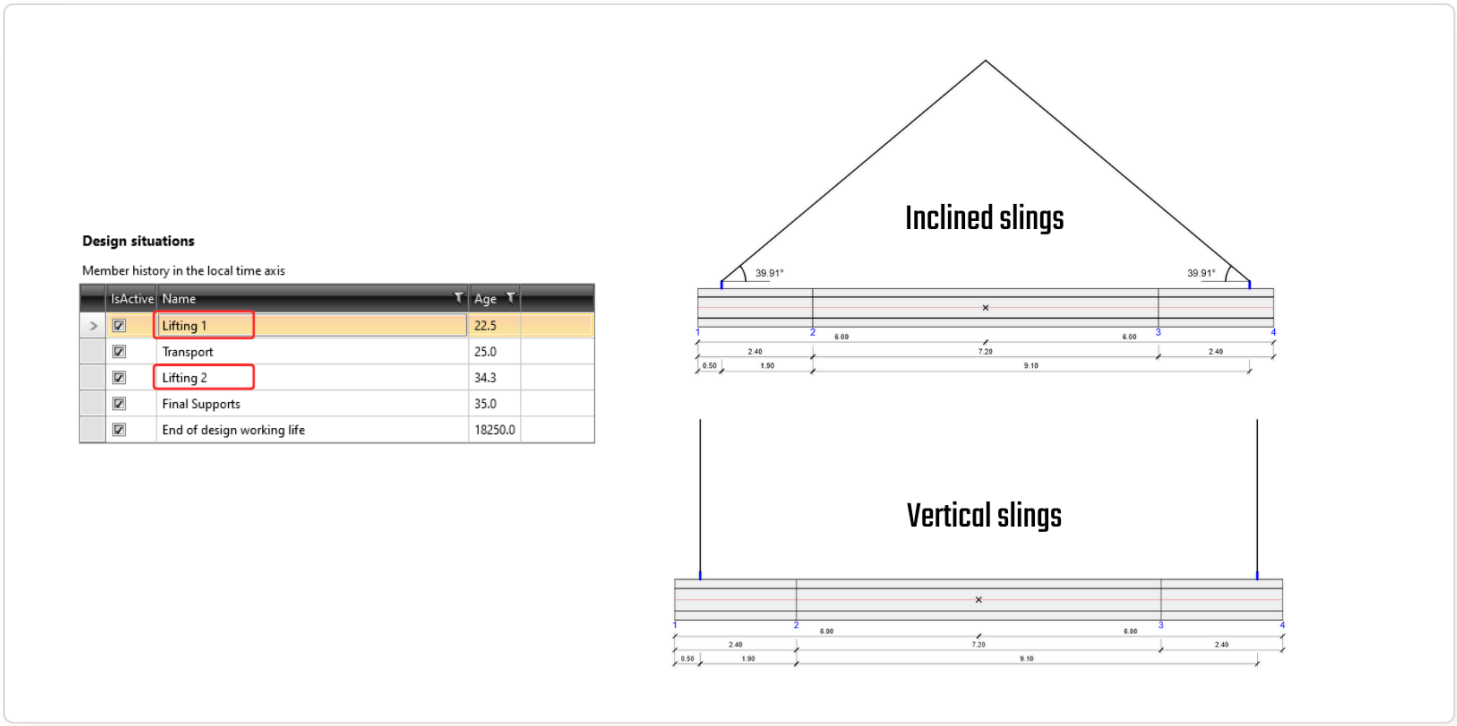
Imperfection

Also, the value of an initial lateral imperfection can be defined separately for each design situation. There are two options for the definition of lateral imperfection:

- Geometric imperfection – where IDEA StatiCa Beam calculates deformations due to creep and shrinkage. But first, the Initial imperfection needs to be set. It can be a) By code – imperfection is assumed according to EN 1992-1-1, chap. 5.9 (2) as L/300 or b) User-defined.
- Overall imperfection – resulting lateral imperfection has to be defined by user.

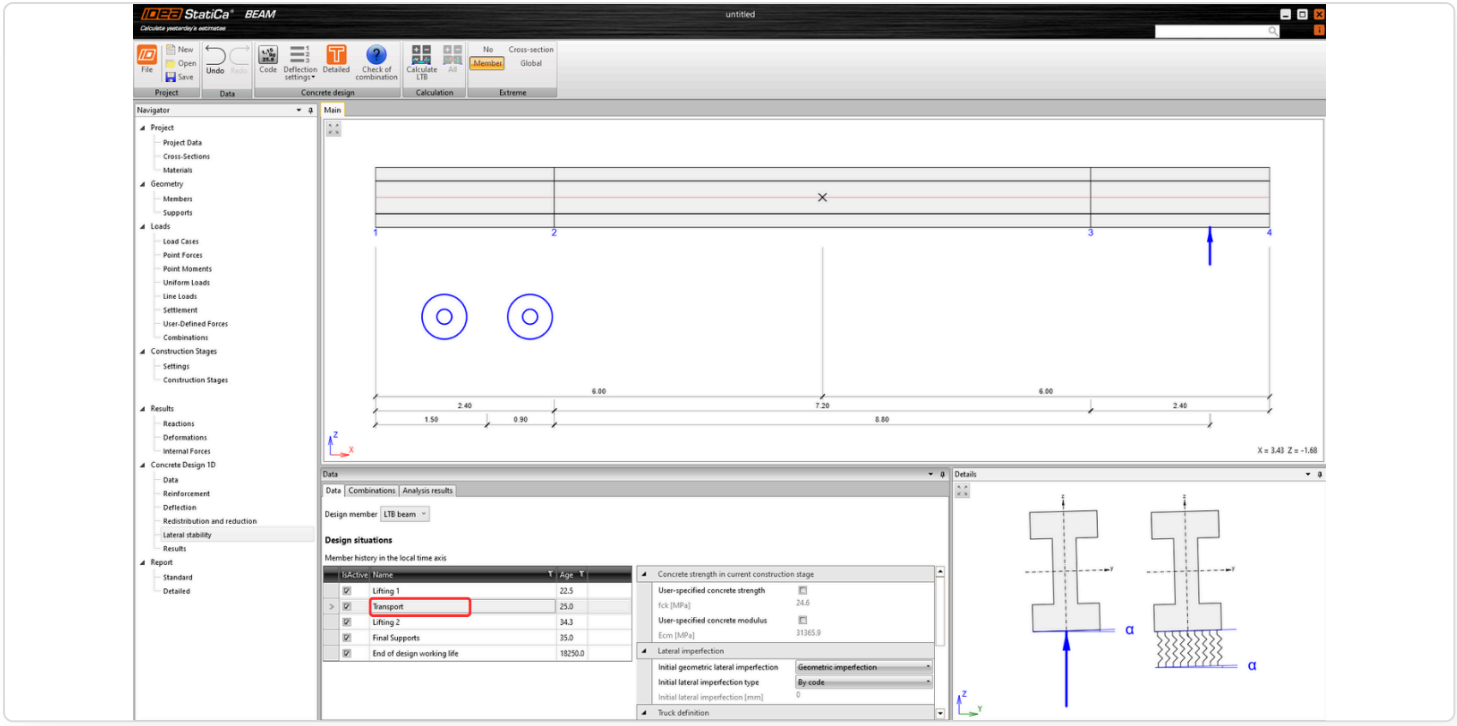
Inputs for Lifting

Two types of lifting can be defined: **vertical slings** or **inclined slings**, each with specific calculation conditions. The position of the lifting points must be specified in both the longitudinal and transverse directions of the beam.



Inputs for transport

Transport refers to the scenario where the beam is loaded onto a truck with a trailer. Deformation in the Rx direction is restrained solely by the trailer and treated as flexible support with defined stiffness. The user needs to define parameters such as the position of the truck, properties of the trailers, and others.



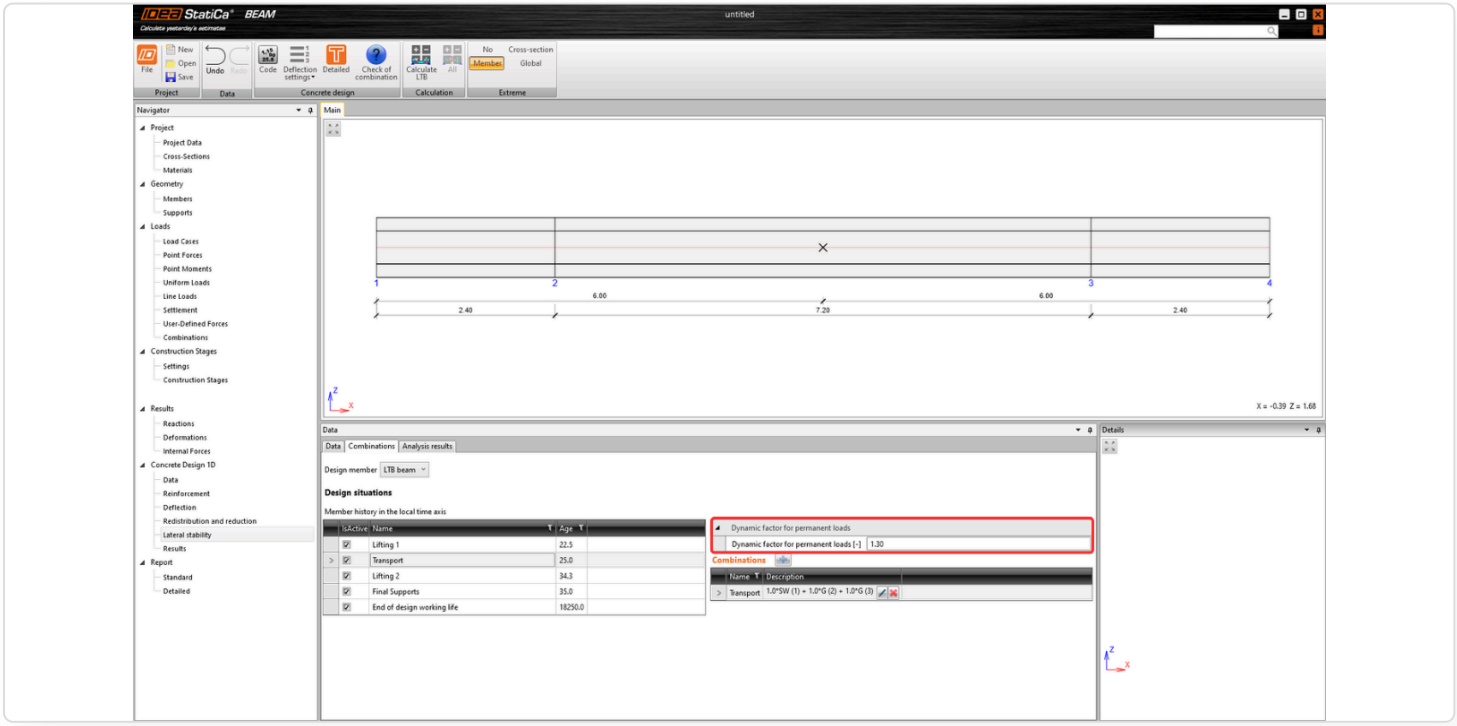
Inputs for Final supports and End-of-design working life

The static scheme for final supports and the end of the design working life is the same, with no option to define supports in the end-of-design working life scenario. The beam is always considered simply supported at its ends. Additionally, the beam can be laterally restrained at specified positions if desired.

Final supports are always positioned at the ends of the beam and can be represented by three types of support: Elastomeric bearings/Forks/Bearing pads with dowels.

Loads

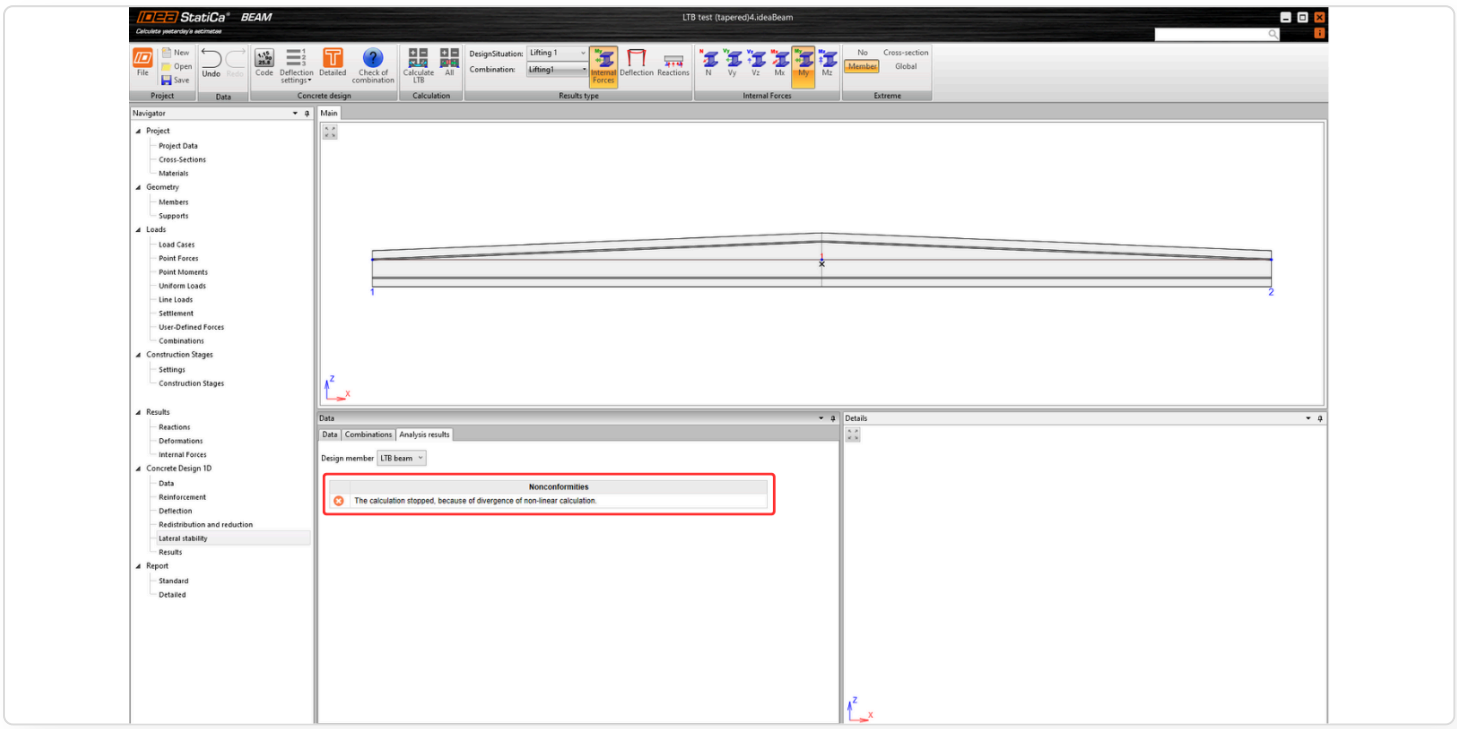
The "Loads" section in the tree of entities in previous design steps defines all the load cases, loads, and load factors. In the section Lateral Stability, dynamic factors for lifting and transport phases and correct non-linear ULS combinations for each Design situation must be defined.



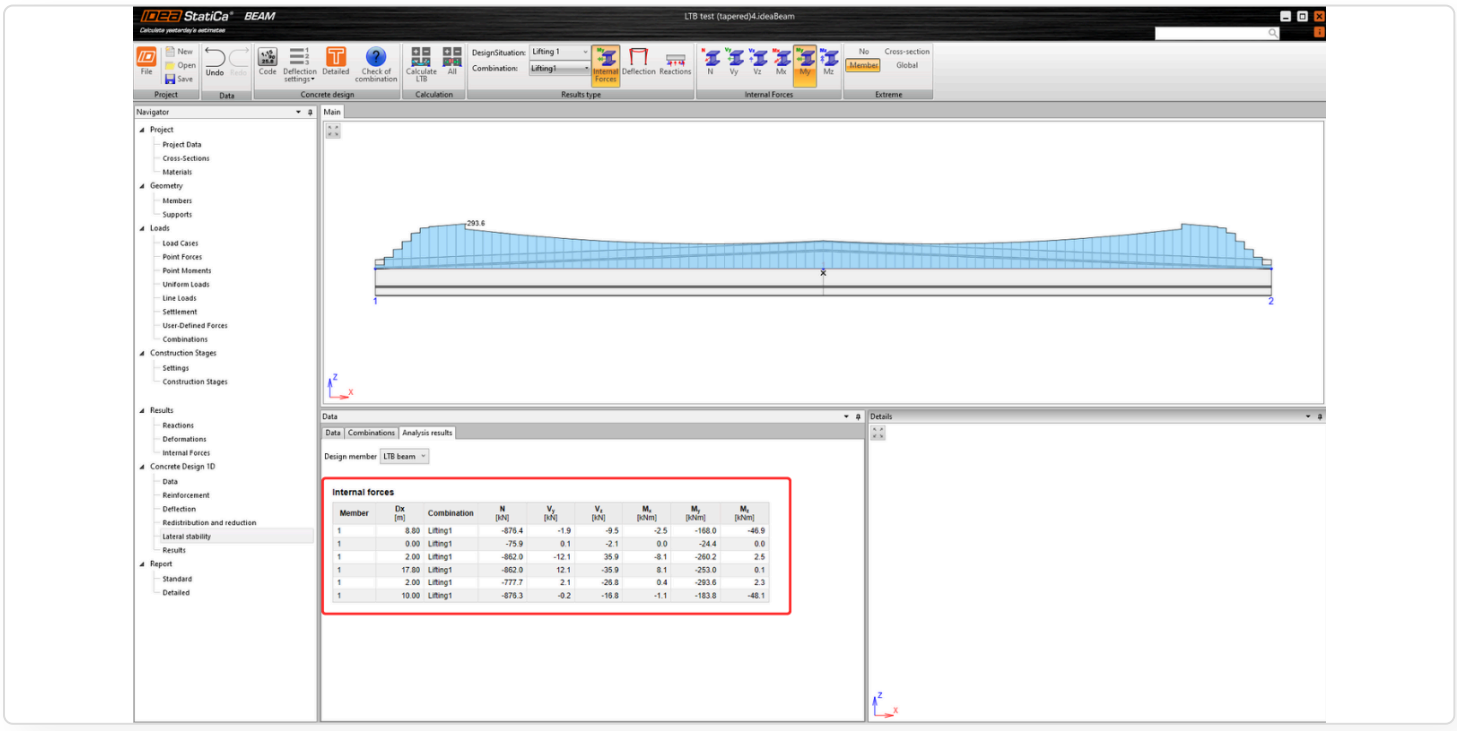
And what about that?

The advanced **Lateral torsional buckling (LTB**) analysis in IDEA StatiCa Beam provides (in addition to reactions, internal forces, and deformations) an evaluation of whether the beam is at risk of collapsing due to stability issues for each defined construction stage. **The user can then continue to work with the internal forces and use them as input for validation in the IDEA StatiCa RCS application (input must be done manually).**

In the event of a structural failure due to LTB, the calculation will not be complete, and the user will see an error message informing them of this situation.



In the opposite case, where the structure withstands, the resulting internal forces are displayed. In the graphical window above, they, as well as the reactions and deformations, are rendered visually.



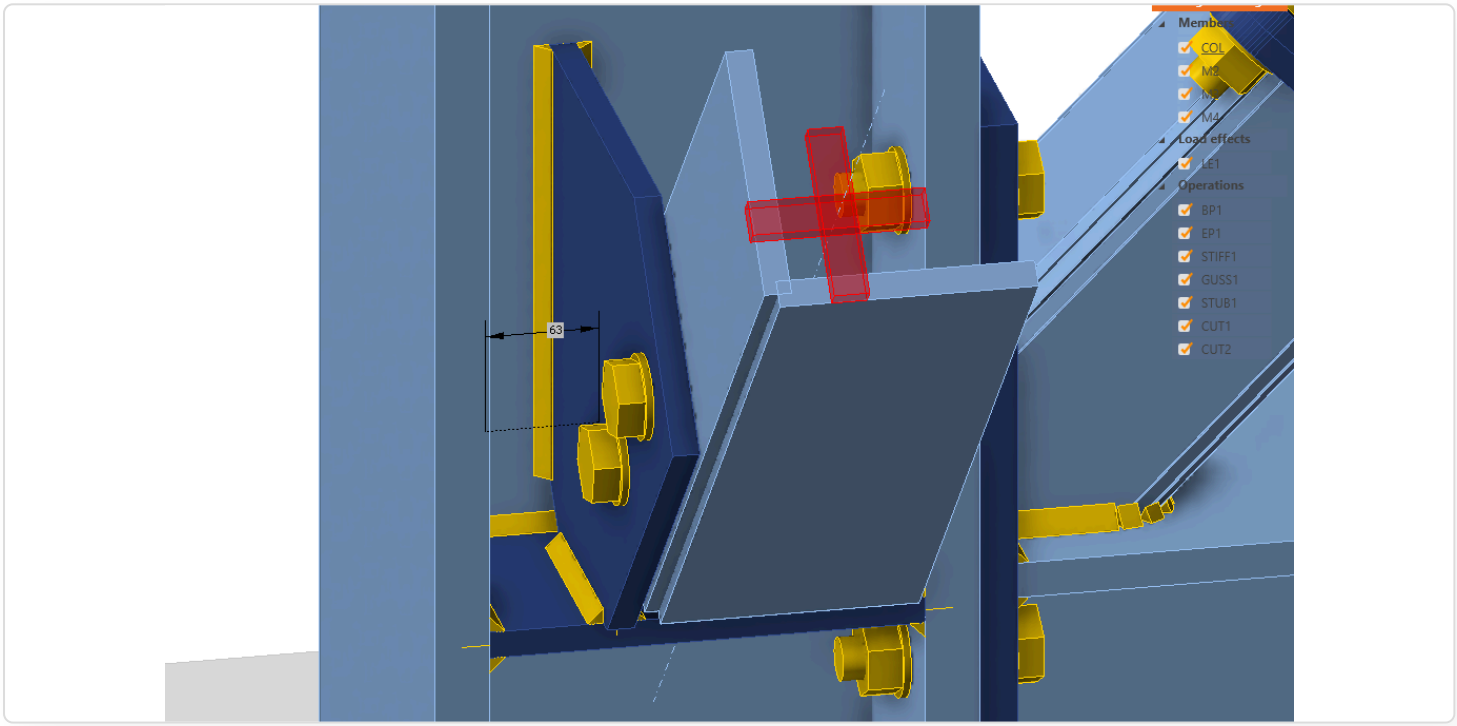
Released in IDEA StatiCa version 24.1

Steel Design

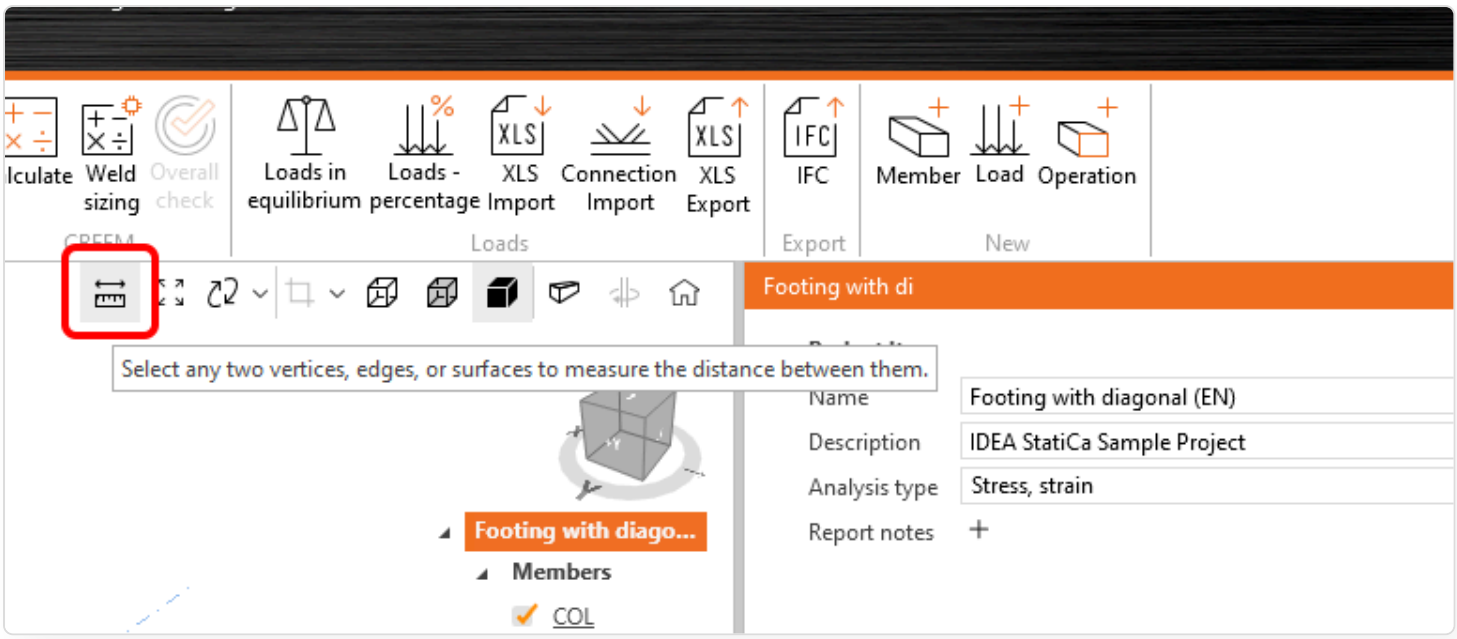
Measuring tool

The Measuring tool feature allows users to measure distances directly within the Connection app. This simplifies the process of measuring gaps, providing precise distance information without the need for external CAD software.

Users can measure critical gaps directly within the Connection app, including precise measurements for **welding gaps** to ensure proper tolerances, as well as verifying the **space required for bolt tightening** without needing to switch to an external 3D CAD software.



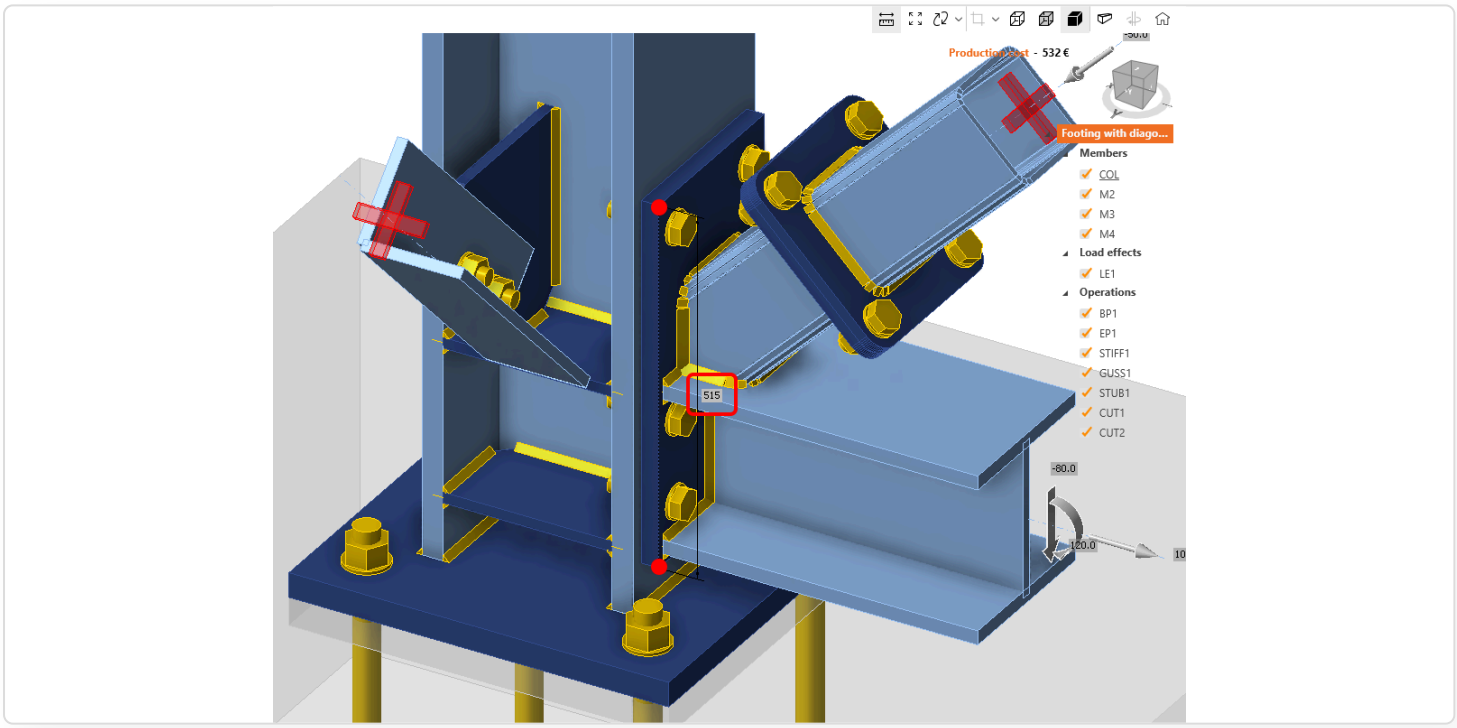
The Measuring tool can be activated in the top right corner of the 3D scene. When turned on, you can select the points for the distance measurement directly in the scene. Once finished, you can deactivate the tool by unselecting the button or using the **ESC key**.



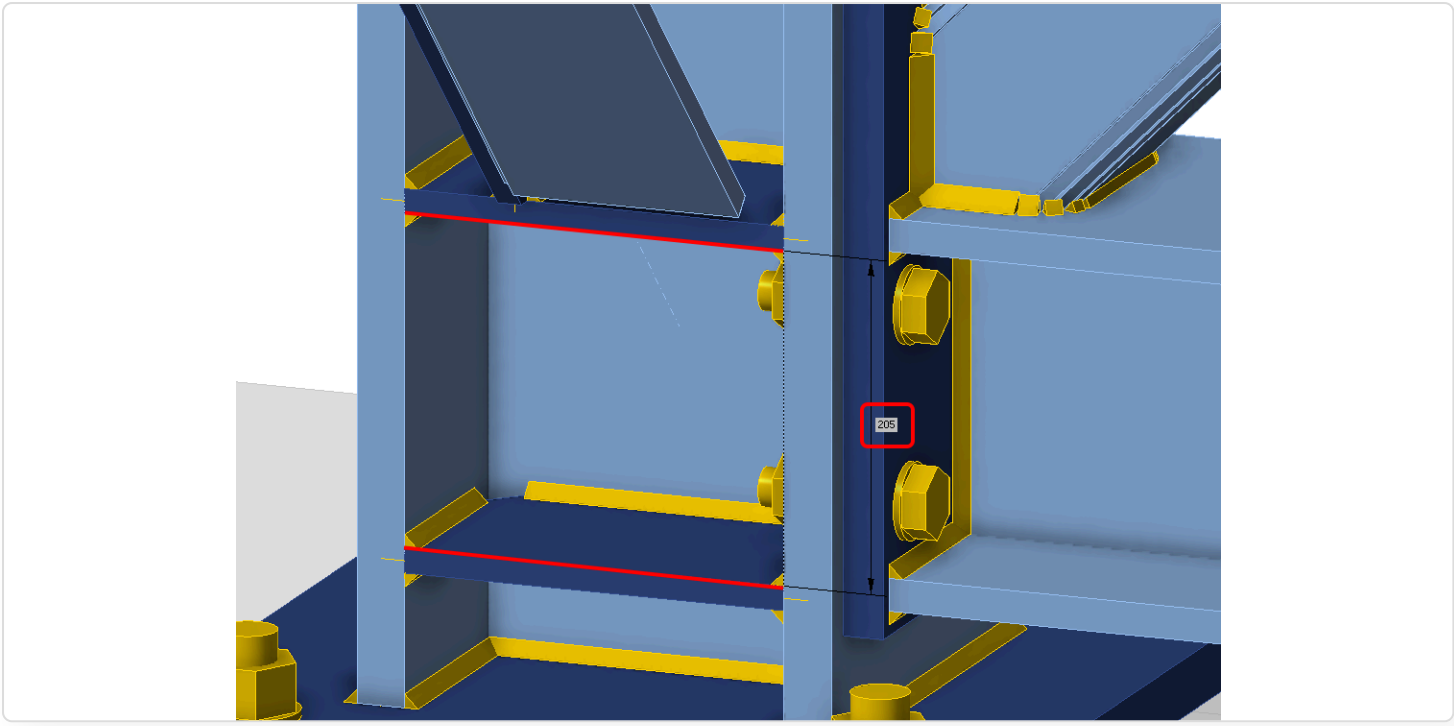
Selection options

The tool supports the measurement between the following entities:

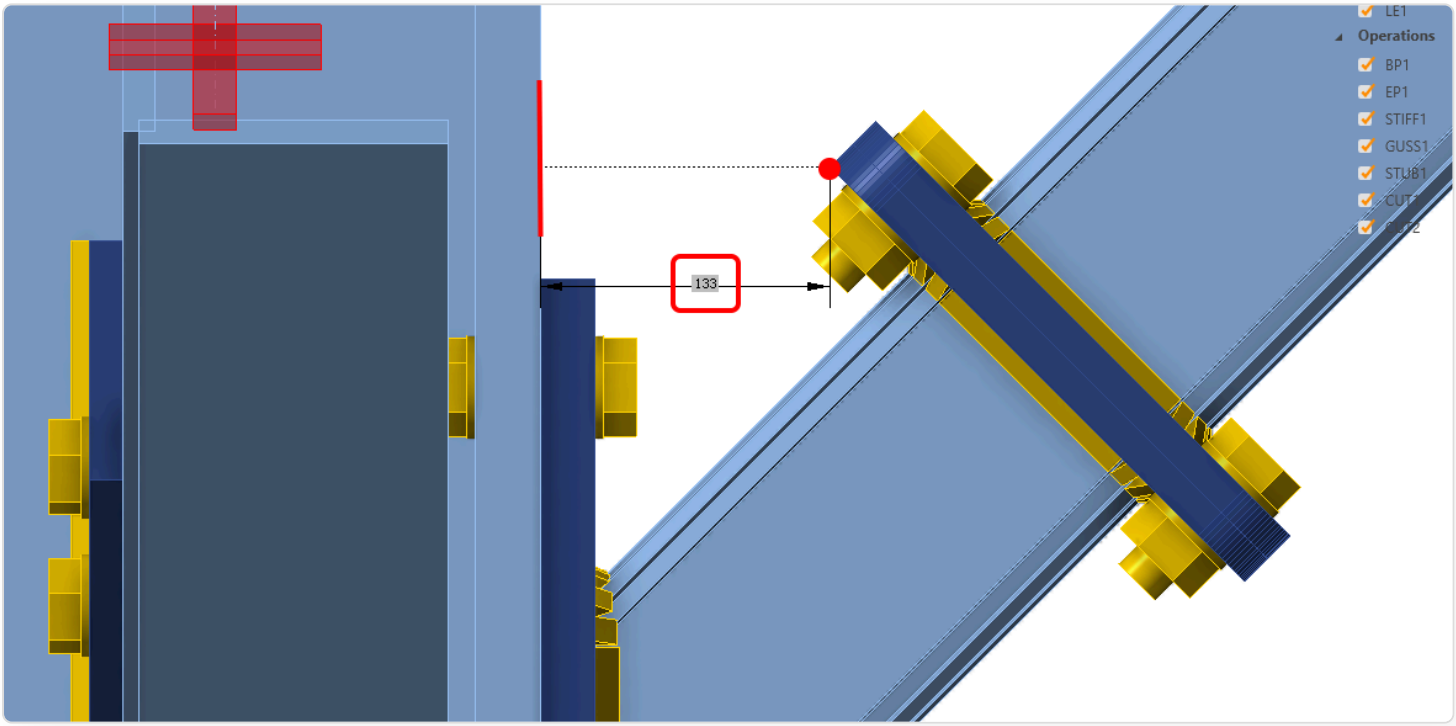
- **Corners**: Measure distances from corner points.



- **Edges**: Determine dimensions between edges.



- **Faces**: Calculate distances between plate faces.



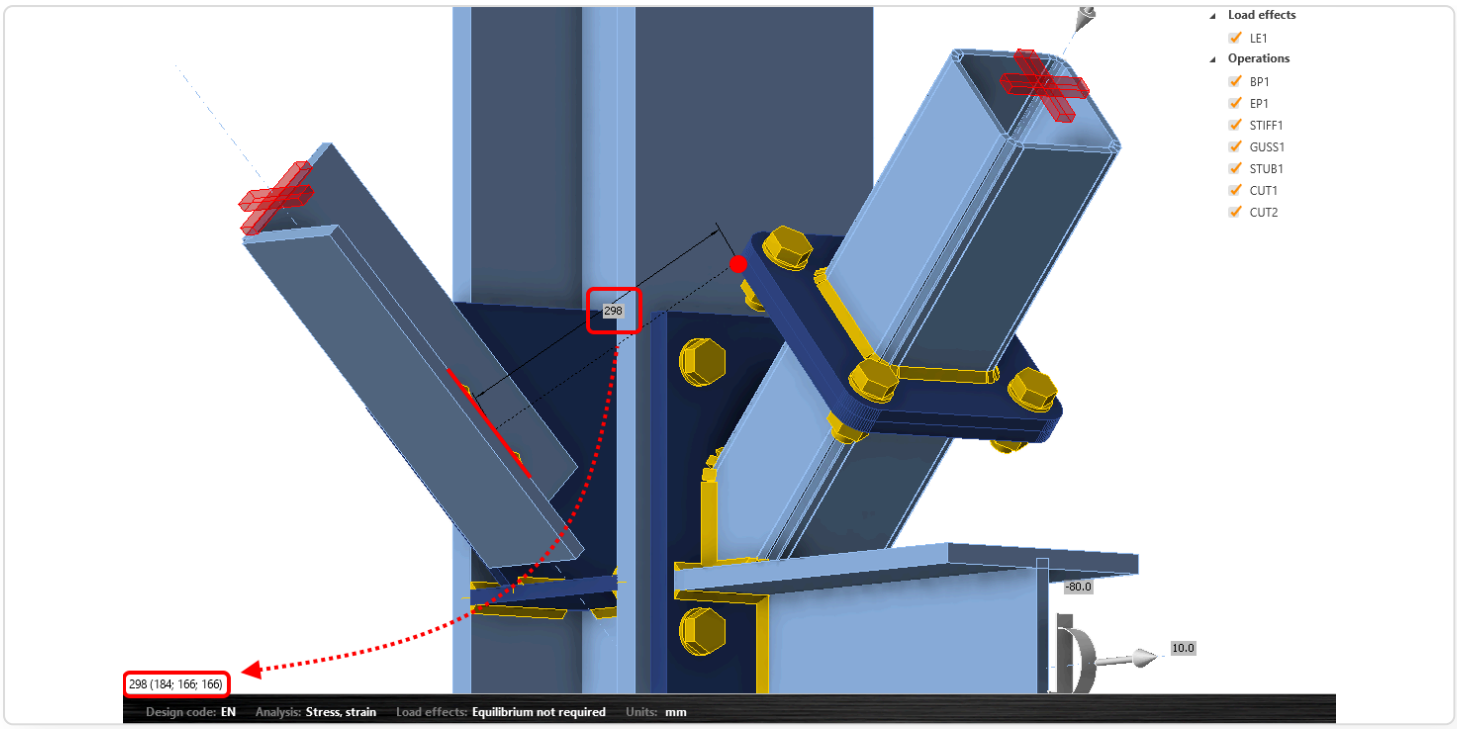
Limitations

The current version does not support multiple measuring at the same time. This feature is expected to be implemented in upcoming patches.

Distance Information:

There are two different places where you can read the distance between selected corners/edges/plates:

- **Shortest distance**: Displays the minimum distance between selected elements directly in the scene.
- **XYZ distances**: Offers detailed distance information along the X, Y, and Z axes, according to the global coordinate system, in the left lower corner in the scene.



Released in IDEA StatiCa version 24.1.

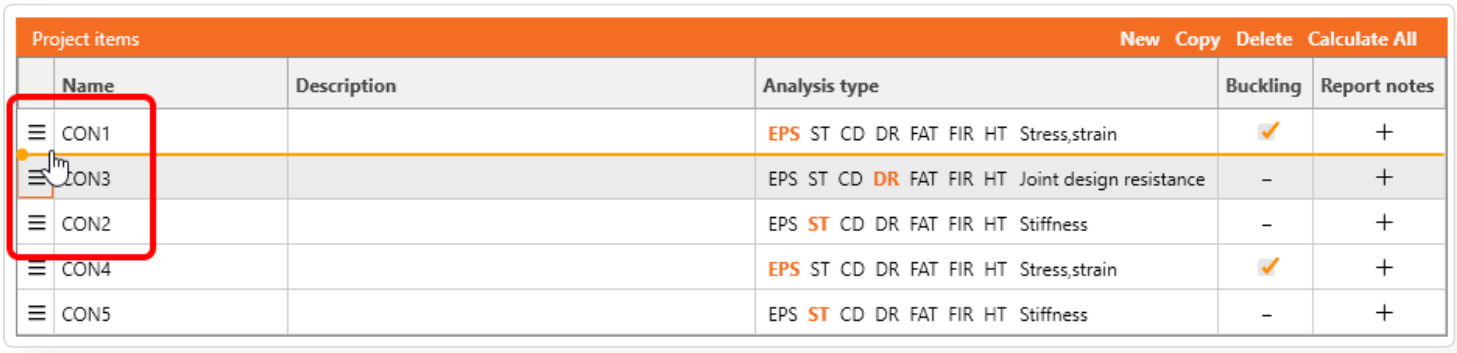
Project item and material management

The backstage menu provides control for managing project items, including reordering capabilities, while the Material defaults property in Project settings allows users to set default materials and fasteners for projects, specifically tailored to AISC or AS design codes.

Efficient project item management and reordering

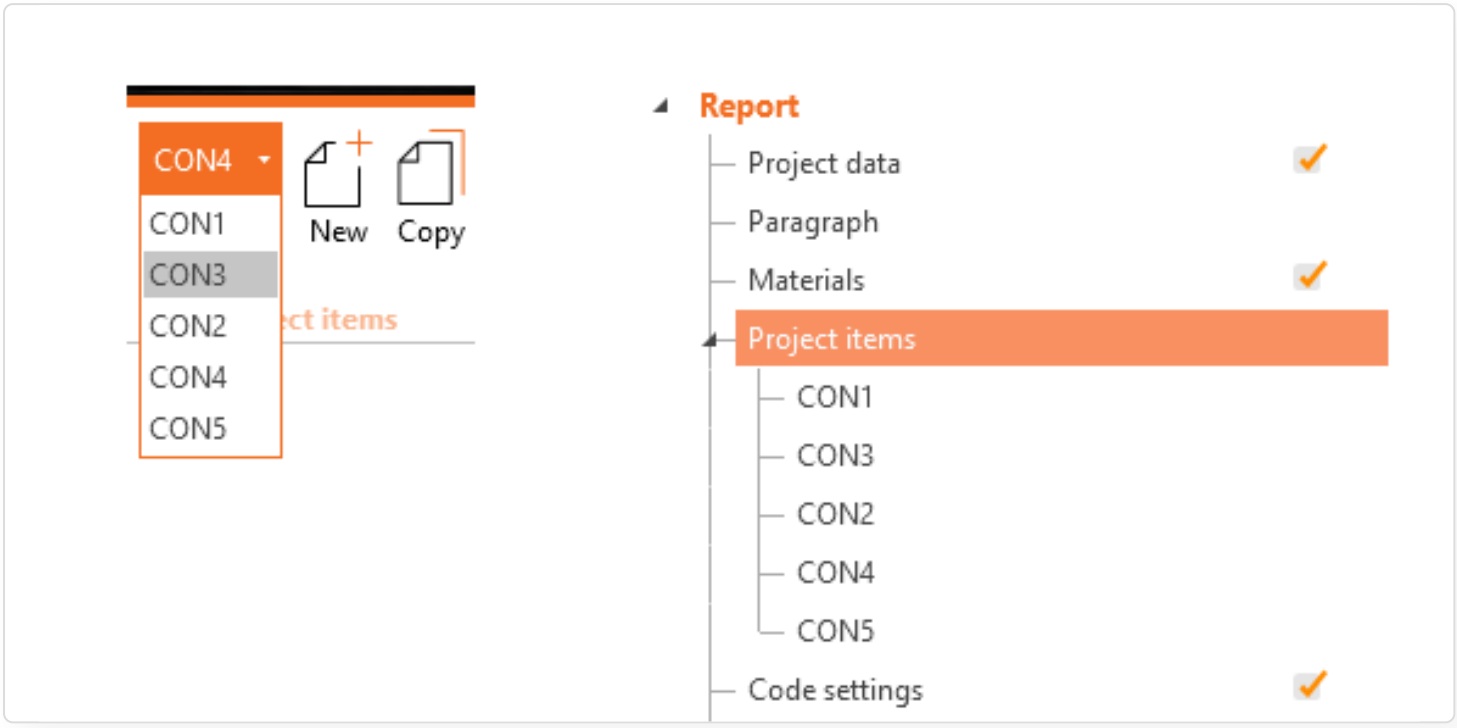
The backstage menu offers control for managing project items, allowing users to **reorder** them directly using a **drag-and-drop** function. This feature supports a more logical organization and ensures a smoother, more responsive experience for Checkbot and Connection users.

Users can adjust the order of project items without resorting to workarounds, such as copying and deleting entries. This makes it easy to organize items based on specific requirements, like analysis types or connection types, improving workflow efficiency.



The menu's **improved responsiveness** ensures that even large projects load quickly, with updates to item names, descriptions, and analysis statuses applied instantly. Read more in the Speed-up of Connection app response article.

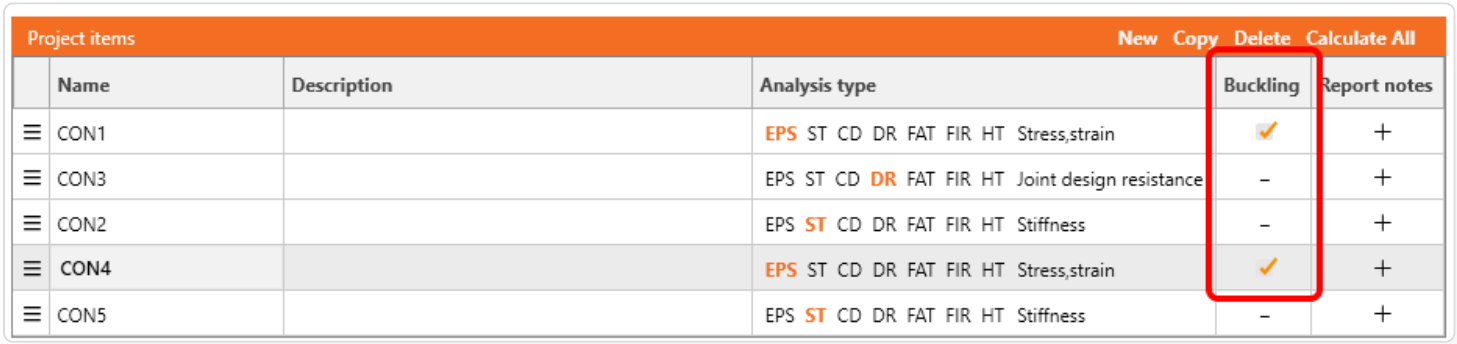
Additionally, the drag-and-drop order is **reflected in generated reports**, ensuring consistency between the modeling environment and documentation. This is especially useful for users who export complex projects from Checkbot and need a customized order for reporting clarity (when importing several models to one .ideacon file).



Released in IDEA StatiCa version 24.1.

Buckling analysis for every project item

The Project items table enables users to **activate buckling analysis** directly within the backstage menu. With a simple click, selected items undergo buckling calculations. This applies to using the Calculate all command above the table.



Users don't have to perform the buckling analysis for each project item one by one in the **Check** tab. In the Backstage menu, just check or uncheck the combo box in the column **Buckling**. This is possible for the following analysis types:

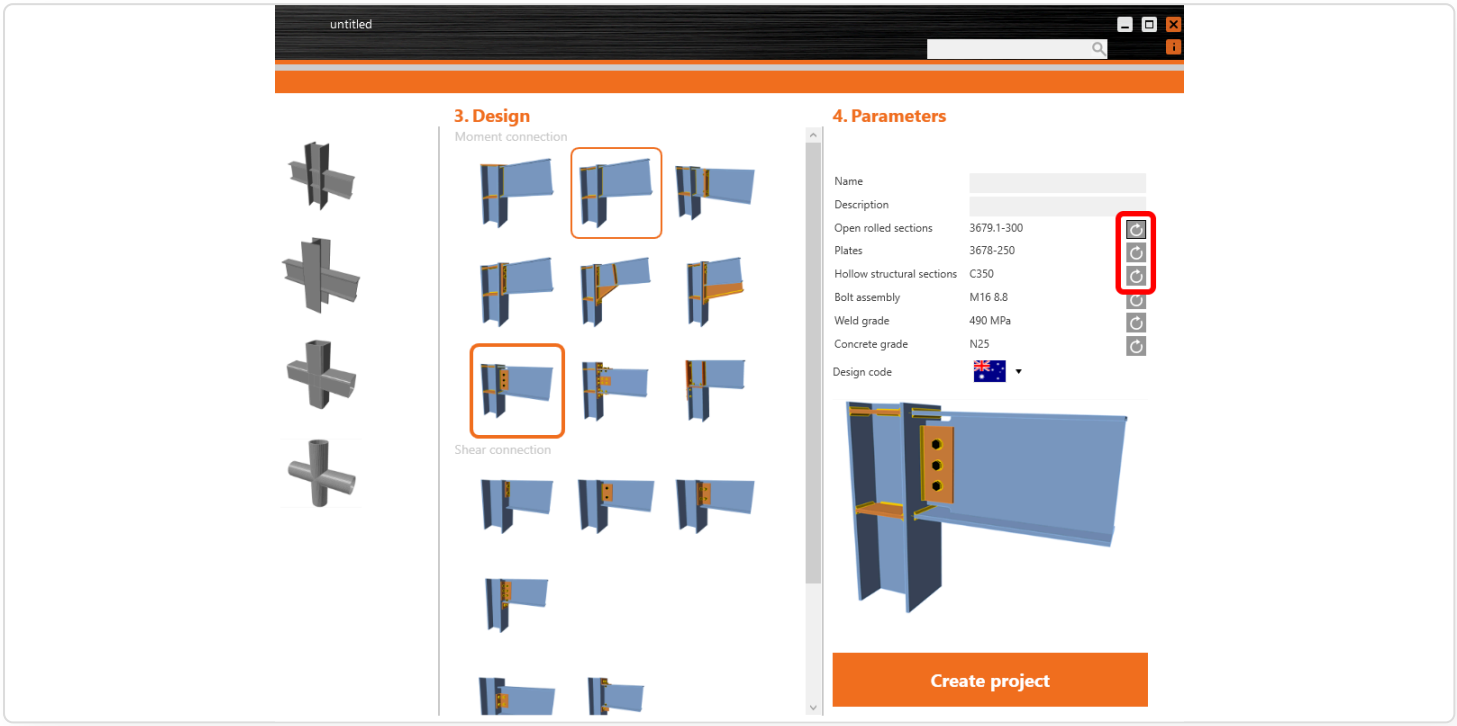
- Stress, strain (EPS)
- Capacity design (CD)
- Fire (FIR)

This setting is related only to the calculation started by the button **Calculate all** in the Backstage menu. Any other start of analysis (Design tab, Check tab, Report tab) is not affected.

Released in IDEA StatiCa version 24.0.4

Material defaults

The new Material defaults property enables users to specify default materials and fasteners for new projects or adjust them in existing ones. These default settings apply to the wizard dialog, new cross-sections, and operations, ensuring consistency and efficiency. You can define the default material of the new project in the **Parameters section**. After that, you can still review the values in the Project settings.

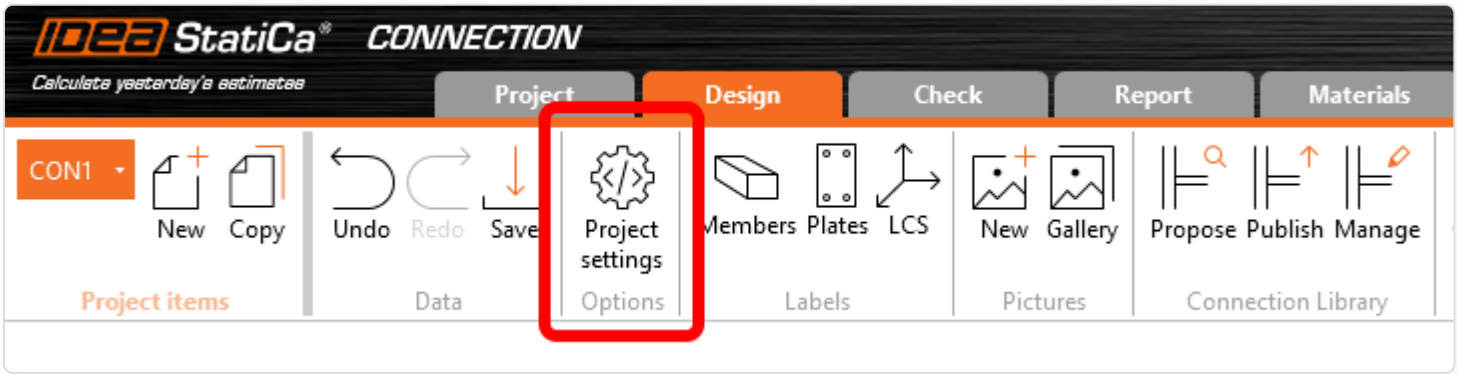


The Material defaults section in Project settings allows users to view and manage pre-set materials and fasteners for new operations. If you save the material settings as default (either in Project Settings in the Connection app or from the starting window of IDEA StatiCa), the values will be applied automatically whenever a new project is created.

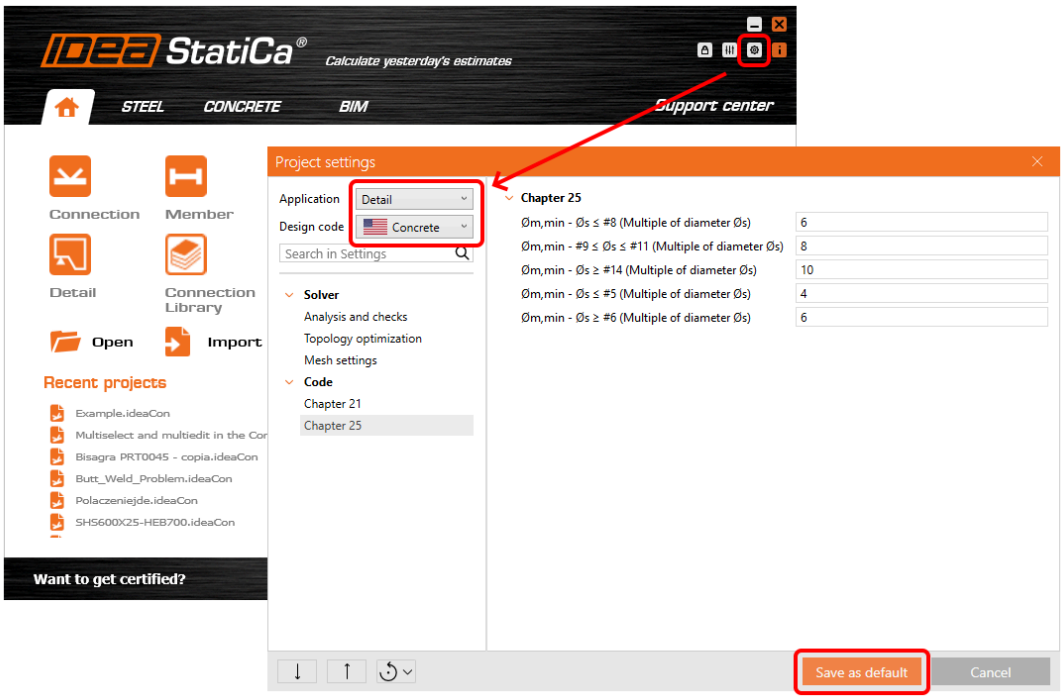
How to change the material defaults

There are three ways how to do it:

- Open the **new wizard dialog** and change the value here in Parameters. The new value will then be used only for the newly created project.
- The other option is to open the **Project settings** in the current model and change the value here. The new values will be **used for newly created cross-sections, operations, or project items.** The elements already created in the model will not be modified. You can save these values as the **new defaults** for your new projects.



- Setting the default material values for all new projects is also possible in the starting screen of IDEA StatiCa. Read more in the Project settings article.



Defaults in codes

Default materials of different cross-section types are different according to the selected code. Users who do not use AISC and AS regulations typically use a single material without dividing it into specific ones for a particular cross-section type. In AISC and AS codes, different materials for specific cross-section types are typically used.

Read more about the default materials settings on AISC and AS in the Regional improvements in 24.1 article.

Released in IDEA StatiCa version 24.1.

Multiselect and multiedit in the Connection app

The functionality of Multiselect and Multiedit in IDEA StatiCa Connection enables users to perform bulk actions for more selected items at once. Thus, creating a new connection model or modifying an imported one is fast.
Thanks to the property grid, items and objects such as Members, Load effects, Operations, Temperature, and Dissipative items can be selected in logical groups. With one click, all user actions, such as changing the offset in members, changing material in operations, copying and deleting loads, etc., can be done for the whole group.

Multiselect is implemented only for items in the tree list – it can't be used to select objects in the visual model in the 3D scene. It can be performed in two ways:

- Hold **CTRL** and select items in the tree list one by one
- Or select the first item, hold **SHIFT**, and select the last item of the desired selection in the tree list

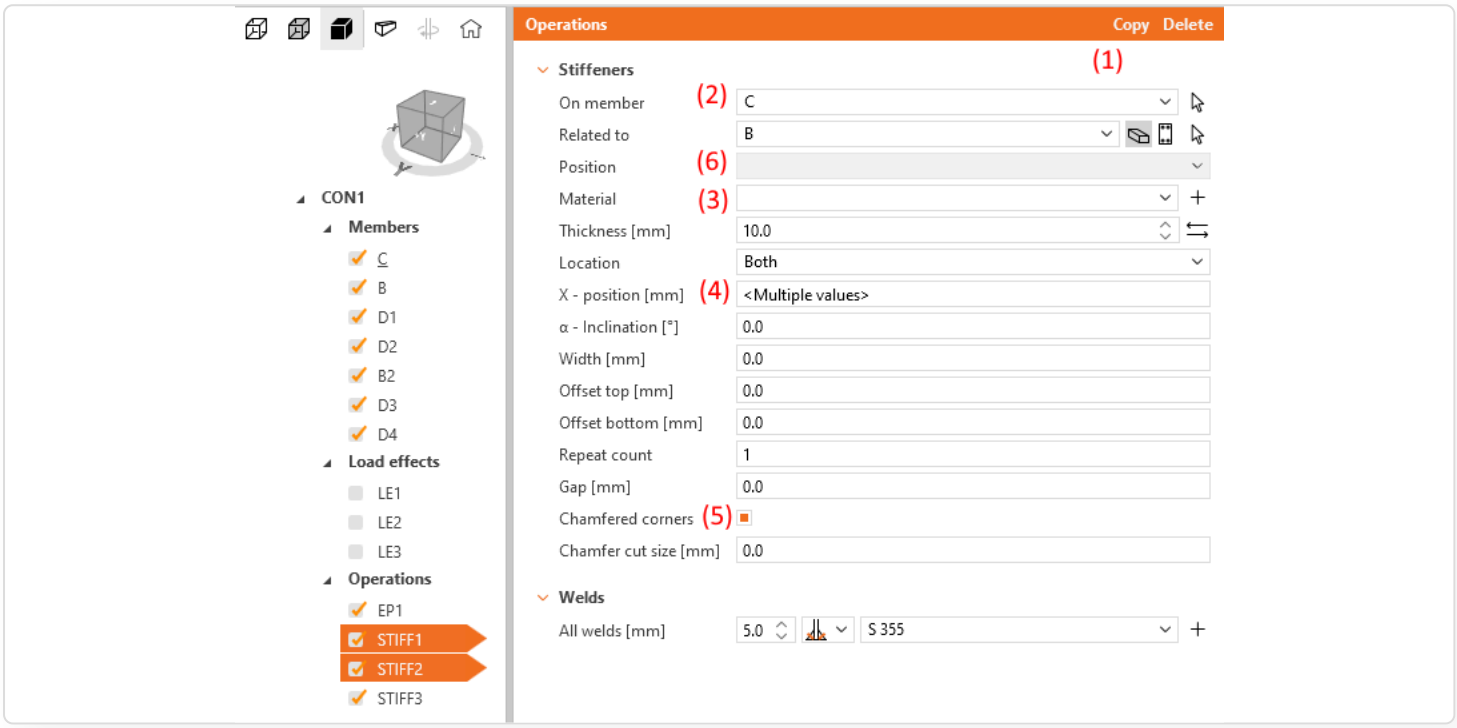
It is possible to select multiple items of the same type or different types.

Selection of the same type of items (homogenous selection)

When the same items are selected, **copy** and **delete (1)** actions in the property grid header are available. Items are copied in the same order as they were selected.

The property tab is editable respecting these rules:

- Properties with an identical value show the inputted value **(2)**
- Properties with different values are empty in a drop-down menu **(3)** or show the **<Multiple value>** in a text field **(4)**
- A checkbox with different values shows an orange dot **(5)**
- Properties illogical or impossible for bulk editing are greyed out/disabled **(6)**

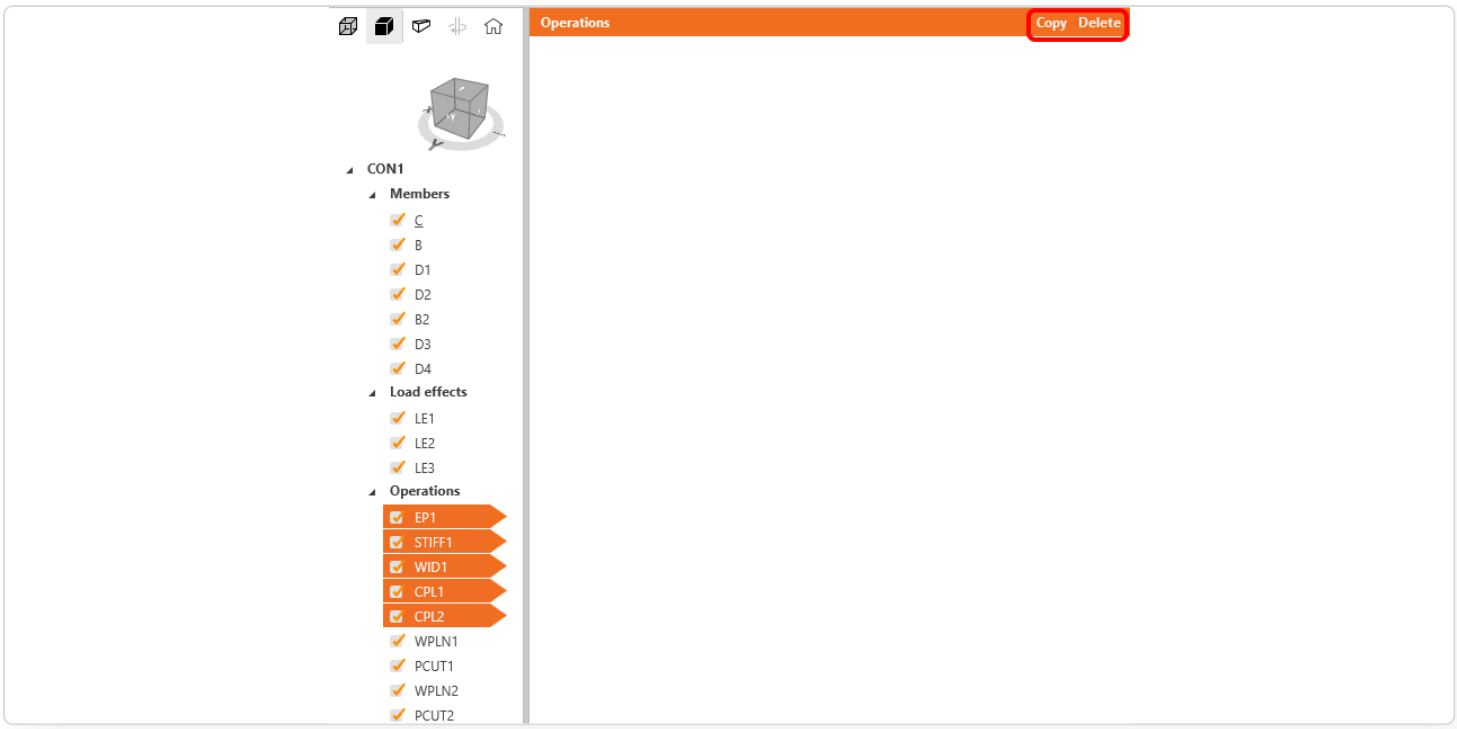


When multiple load effects are selected, only **copy** and **delete** actions are available.

Selection of different types of items (non-homogenous selection)

When different items within the same group are multi-selected, the property tab is empty, and only **copy** and **delete** actions are available.

When items in different groups (e.g. Members + Operations) are multi-selected, the property tab is empty, and **copy** and **delete** actions are not available.



Right-click context menu is not supported for the multi-selected group of items.

Released in IDEA StatiCa version 24.1.

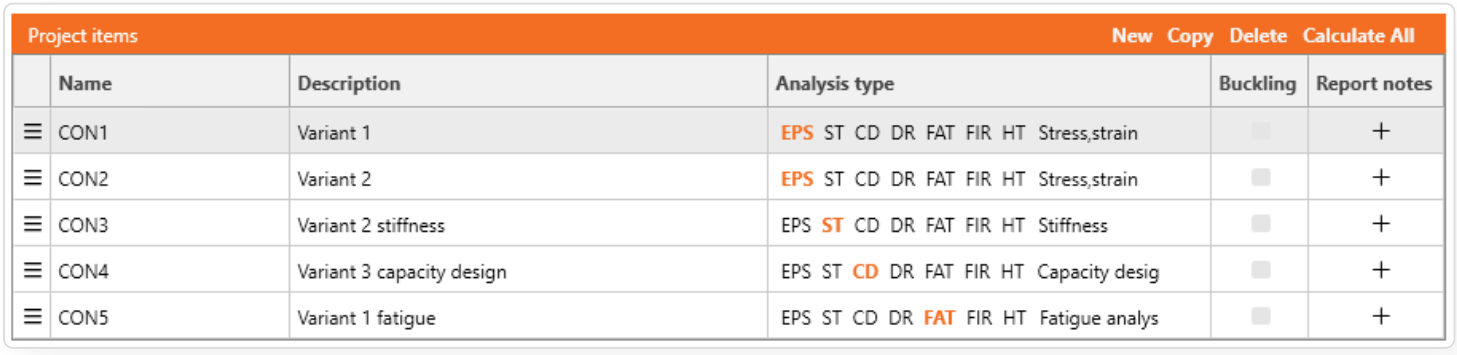
Fast Connection app response

Typical connection design work consists of an iterative process when users repeatedly switch between operations, checks, menus, etc. By making these actions fast, even during long work sessions, users can stay in flow without experiencing disruptive delays.
The improvements have touched three different areas in the Connection application - Backstage menu, results presentation, and overall computer memory usage.

Performance of the Backstage menu

The table with project items in the Backstage menu is mainly used to organize and control multiple connection models and analysis types within the project file. Users can quickly set the name, description, and analysis type for each project item, add report notes, and easily rearrange the order of project items.

The improved response is especially noticeable for projects with a large number of items, as the Backstage menu table's reaction to user actions has been dramatically accelerated.



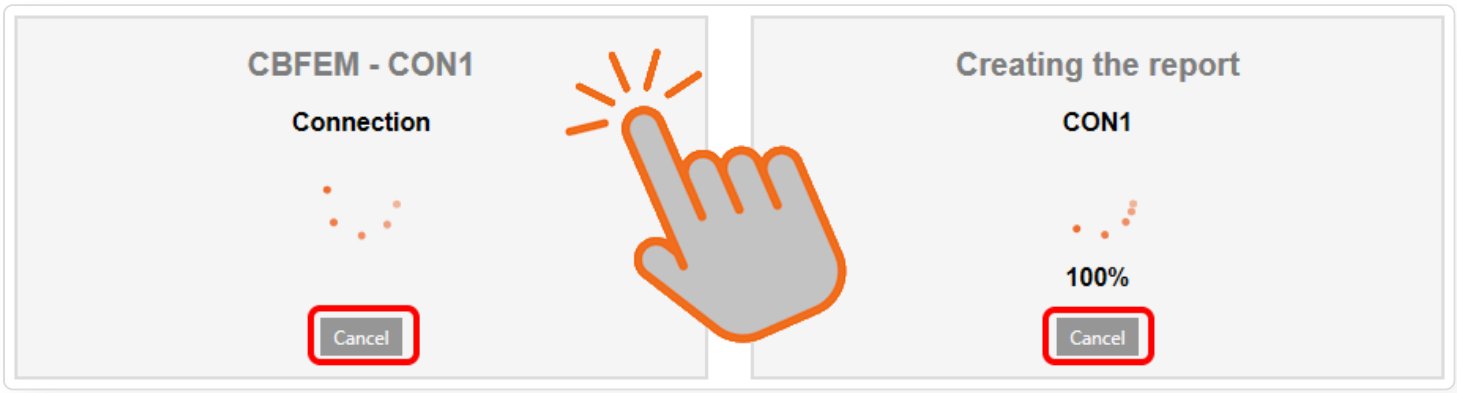
Released in IDEA StatiCa version 24.0.4

Quick cancel of calculation and report creation

The **Cancel** action is improved to stop running processes instantly. This affects the meshing part of the calculation process, report creation, and the second part of stiffness analysis calculation.

When the **Calculate** button is hit, the meshing of finite elements starts first, and then the main analysis begins. Since meshing can be time-consuming, especially for bigger models, it can now be interrupted immediately by the **Cancel** button.

A similar situation occurs while generating a **Report**, where pictures are generated for a long time. This can also be interrupted instantly.



Stiffness analysis is specific for its two rounds of calculation. The second one, which calculates the stiffness curve, can also be stopped in a second.

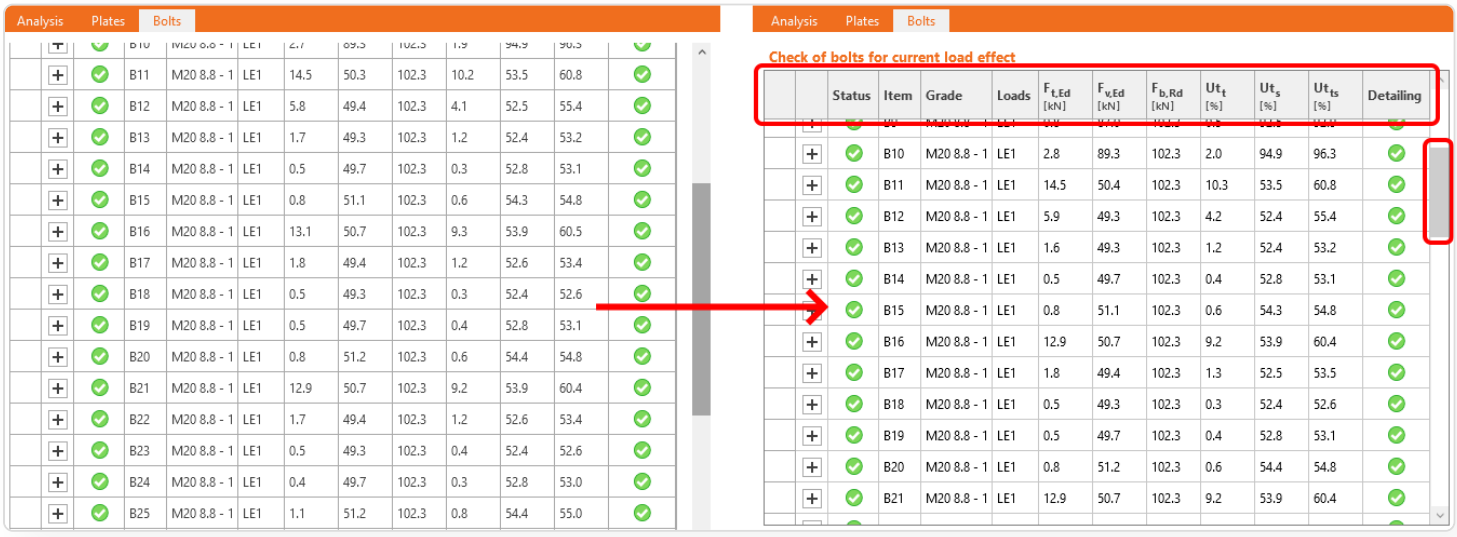
Released in IDEA StatiCa version 24.1

Speed-up of results presentation in Check

Refactoring the result tables in the **Check** tab significantly speeds up table presentation and enhances table interaction. As a result, switching between the **Design** and **Check** tabs is now much faster.

The visualization of result tables is over 150% faster when no results are displayed in the 3D model, and 25% faster when results are shown on the 3D screen, as these tables also include images derived from the 3D model visualization.

Also, the scroll bar moved inside the result tables, so the header stays visible when scrolling.



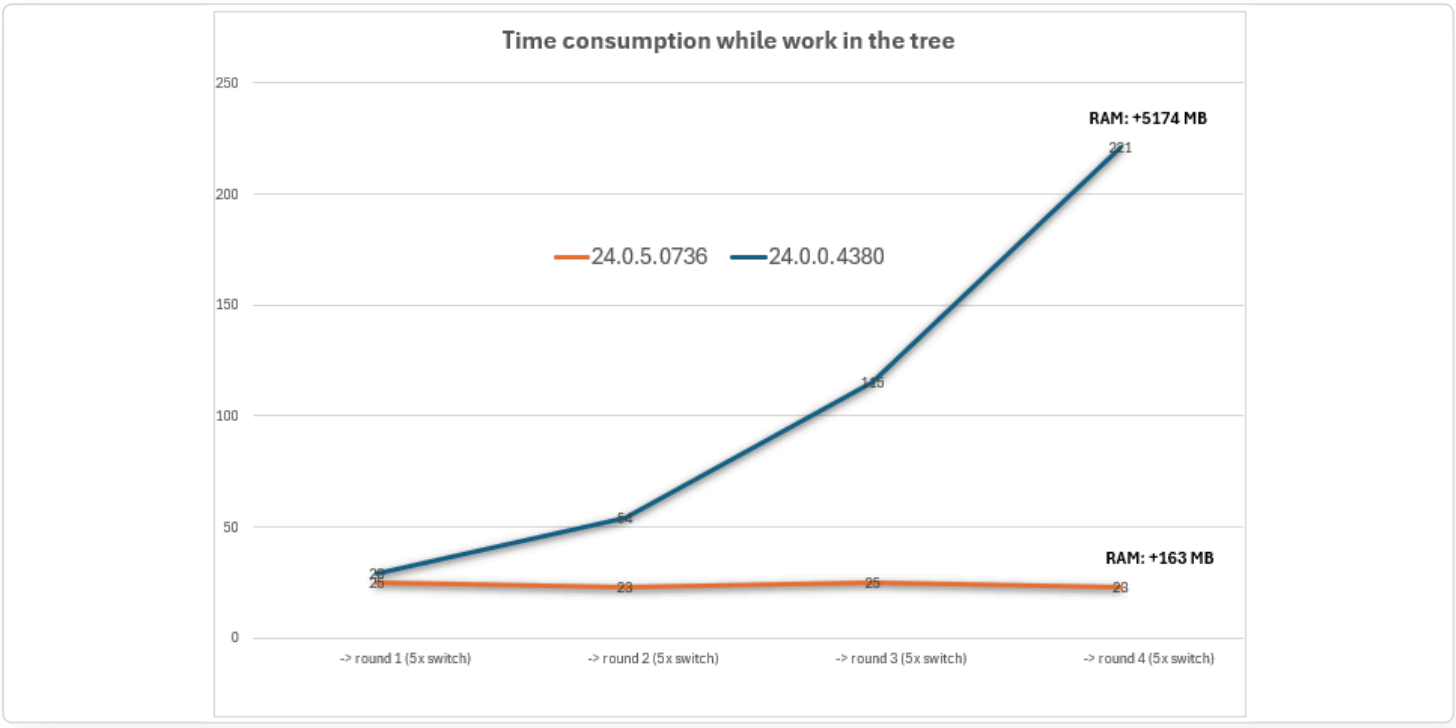
Released in IDEA StatiCa version 24.0.3

Decreased memory consumption

IDEA StatiCa Connection has eliminated memory (RAM) leaks, preventing RAM consumption from accumulating during extended work sessions. As a result, tree menus remain responsive, and modifications to operation properties continue to perform quickly.

The graph illustrates a straightforward test of switching between categories in the tree menu, with the refresh time measured for each transition:

- The blue line represents **version 24.0**, where time consumption increases with each switch between items in the tree menu. As RAM usage grows, the application becomes progressively slower.
- The orange line represents version 24.1, where time consumption remains low and constant, even during extended use of the tree menu. RAM usage increases minimally, ensuring the application's response stays fast and efficient.

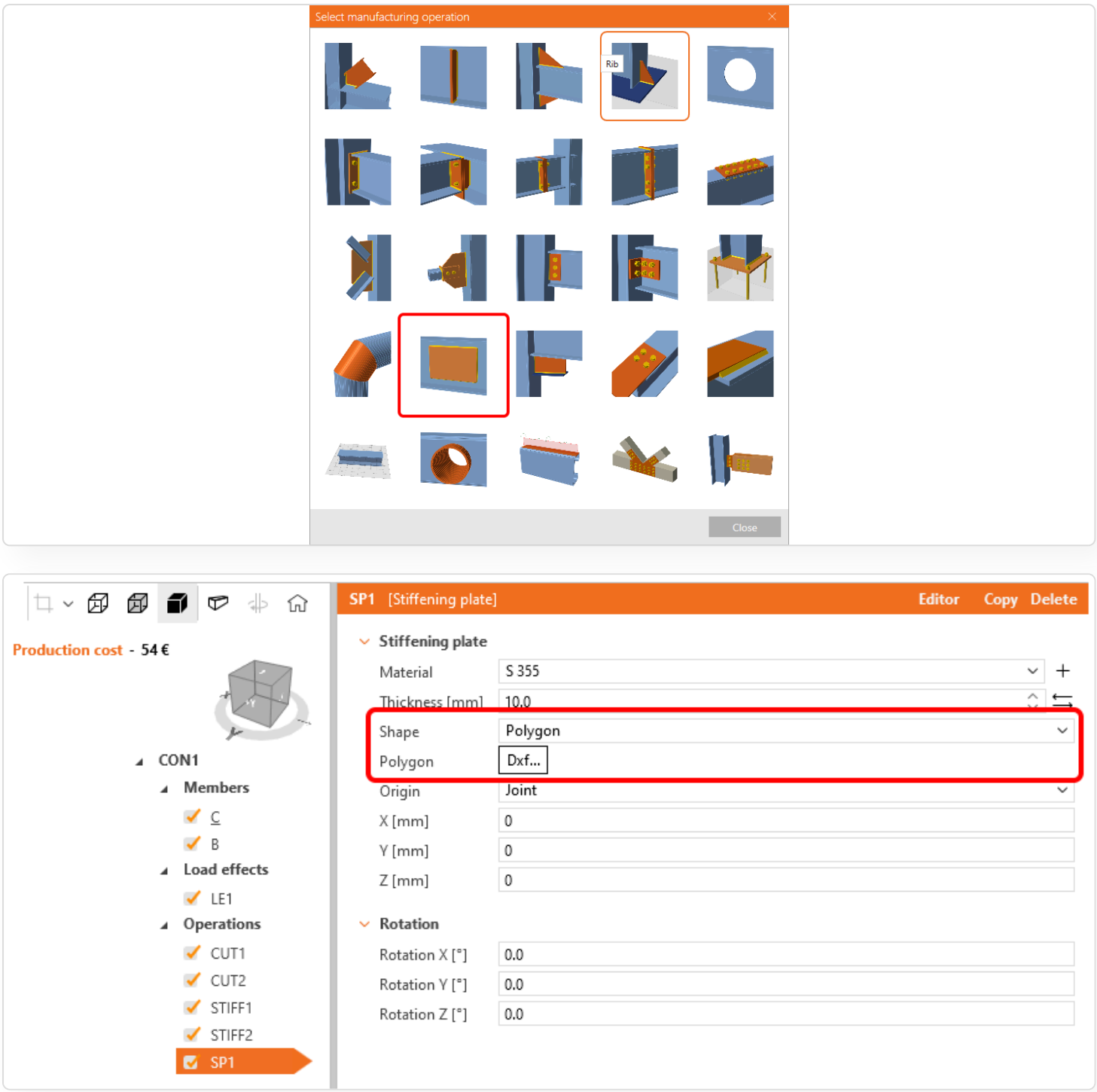


Released in IDEA StatiCa version 24.0.5

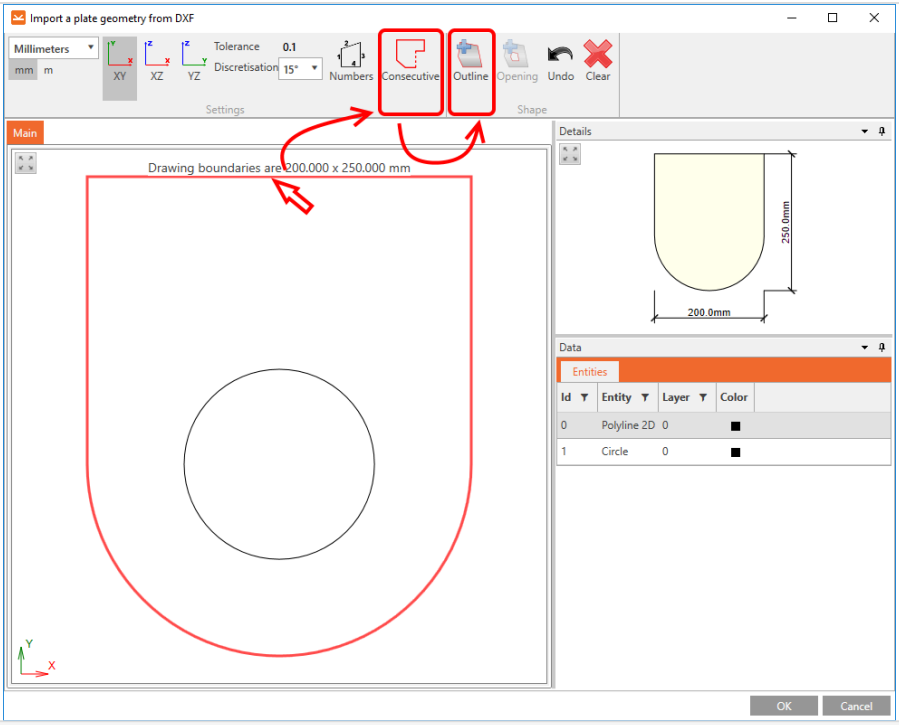
How to import a plate from DXF

In the case of plates with a general shape, instead of editing edges using Editor in the operation Stiffening plate, you can consider importing them from DXF.
Add a new operation, **Stiffening plate,** and for the parameter **Shape,** select the option **Polygon**. Click the **DXF...** button**...** and choose your DXF file containing the plate outline shape.

Please note that the plate outline shape in the DXF file must be exploded to the simple entities (line, arch), and other applications do not have the DFX file open.

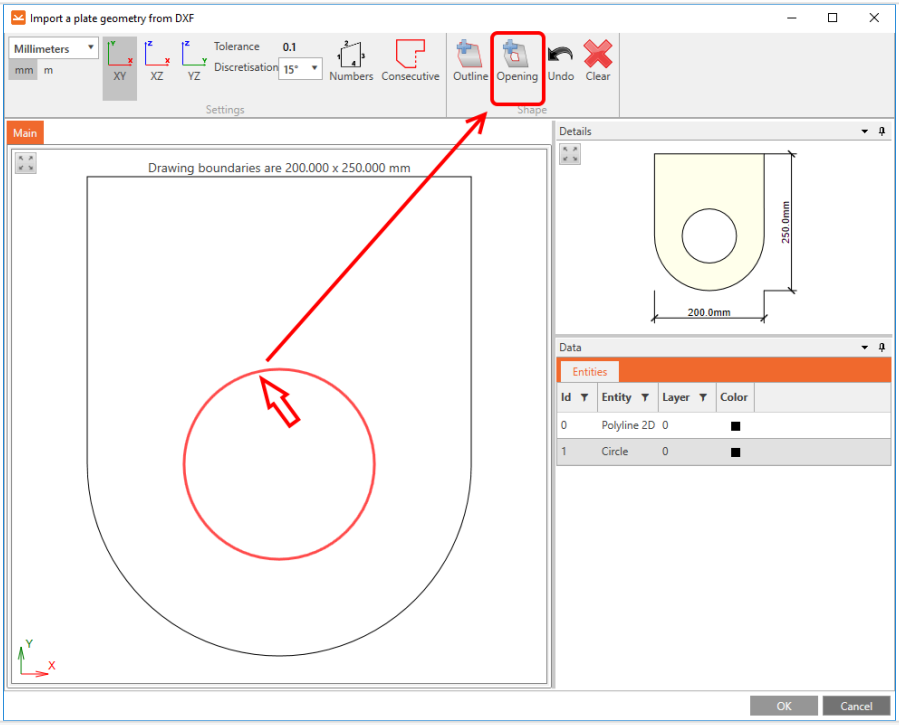


Then, click the edge of the shape to pick a line in the selection. You can manually pick the other edge lines by pressing the Ctrl button or clicking the command **Consecutive** in the top ribbon. You must select a closed shape. Then click the command **Outline**.



The shape of the imported plate is drawn in the **Details** window.

If there is an opening in the plate, pick the closed curve of the opening outline and click the command **Opening**.

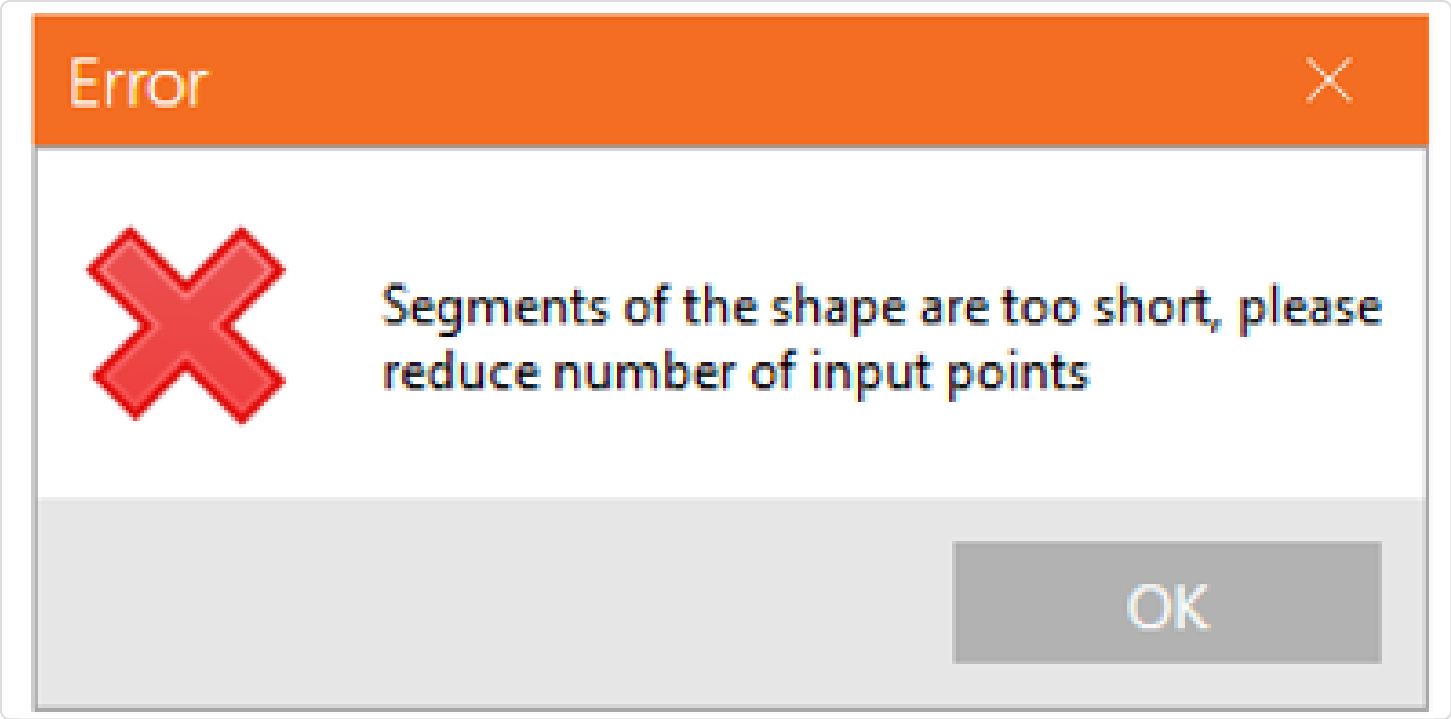


To finish the import process, click **OK**.

Limiting short lines in DXF imports (in 23.1.2 version)

Limiting short lines in DXF imports prevents issues during mesh generation and avoids any interruptions in analysis caused by excessively short lines. The minimum distance between two points is checked against a **set limit**, which is defined as **1/50** of the bounding box of the cross-section.

Read more in the release notes article: Limiting short lines in DXF imports.

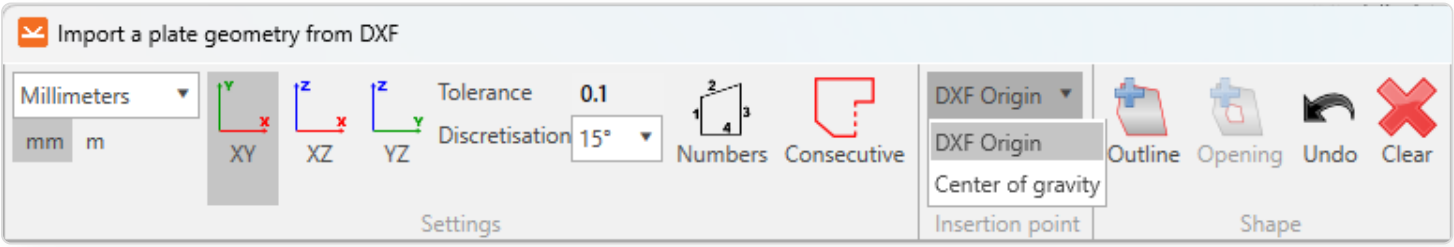


Released in IDEA StatiCa patch 23.1.2.

Defining the insertion point (in version 24.0.2)

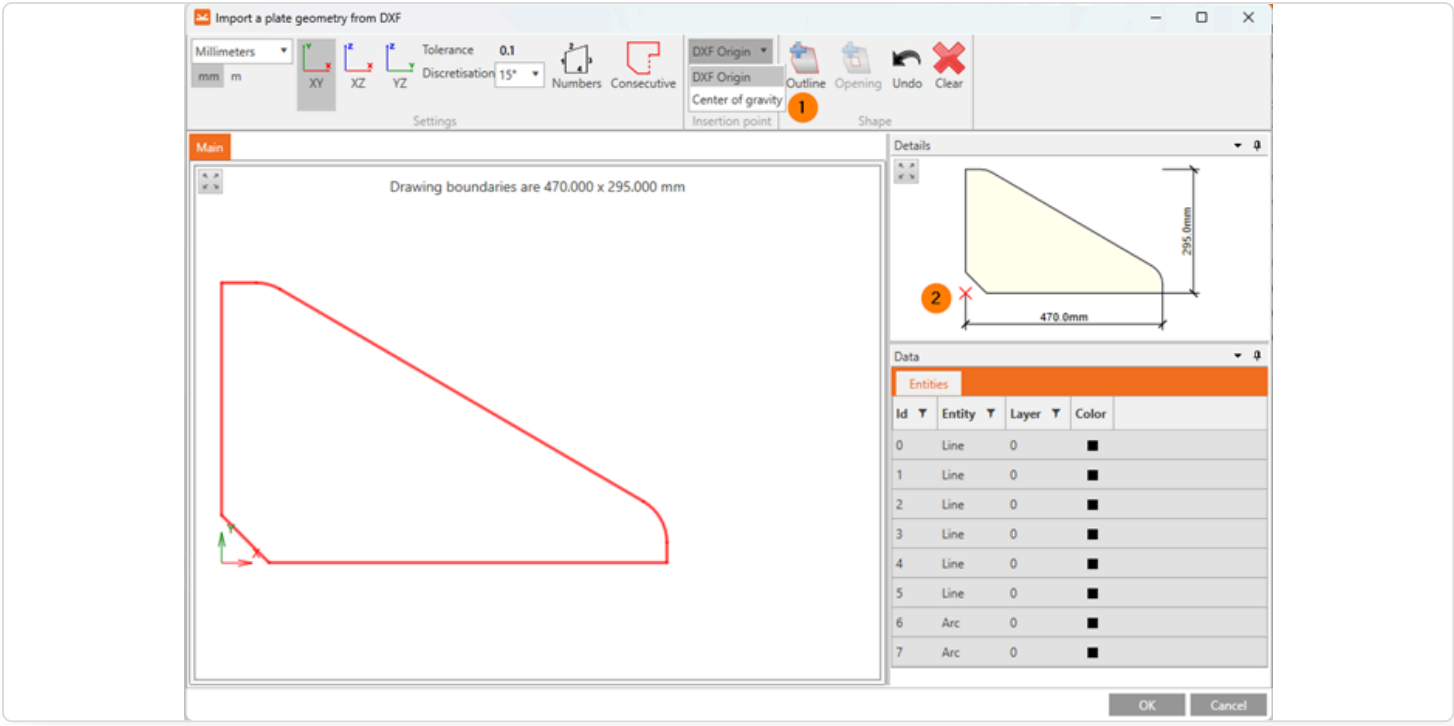
Defining the insertion point of a plate imported from a DXF file improves the accuracy of its placement within the model. Users can determine the insertion point, allowing for precise positioning of complex shapes and a more efficient workflow during DXF integration.

Selecting the insert point

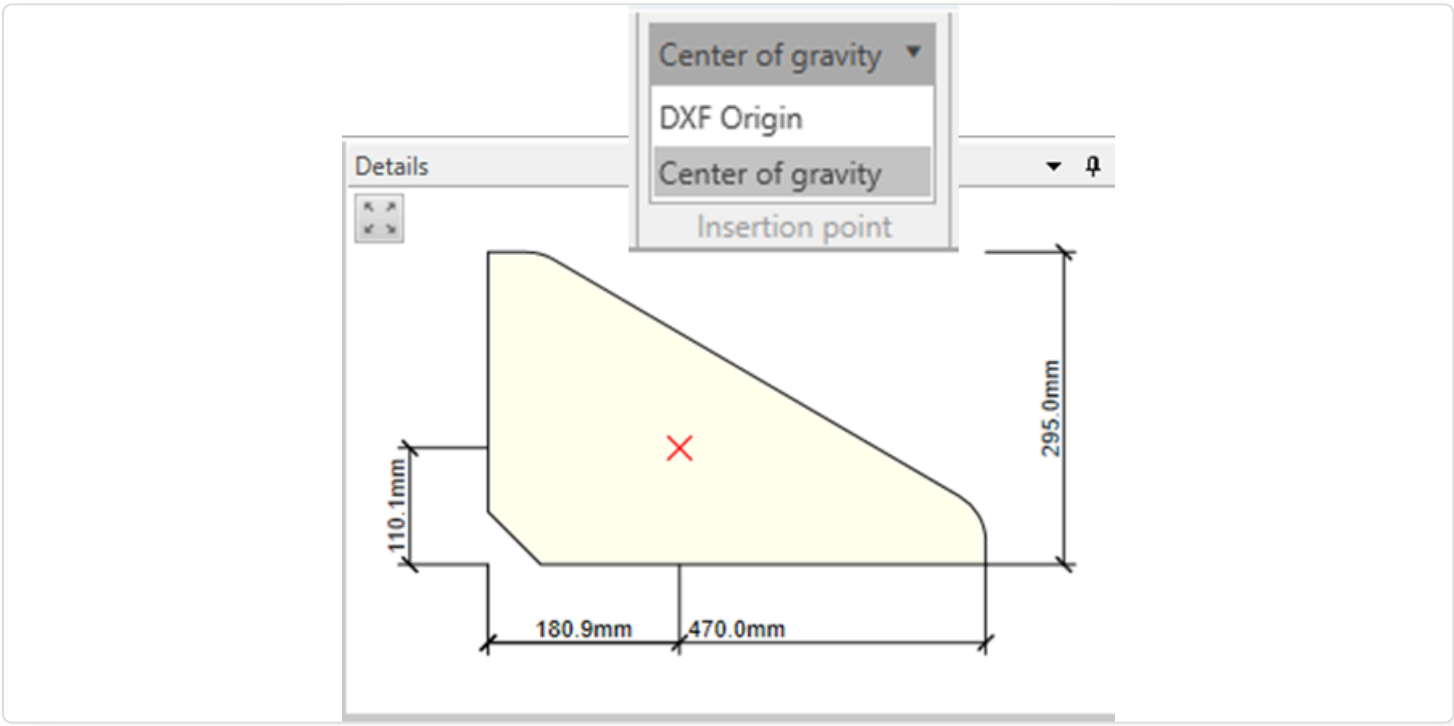


There is an option in the DXF import process that allows users to set the insertion point based on:

- **DXF Origin**: Uses the origin point as defined in the DXF file (default setting).



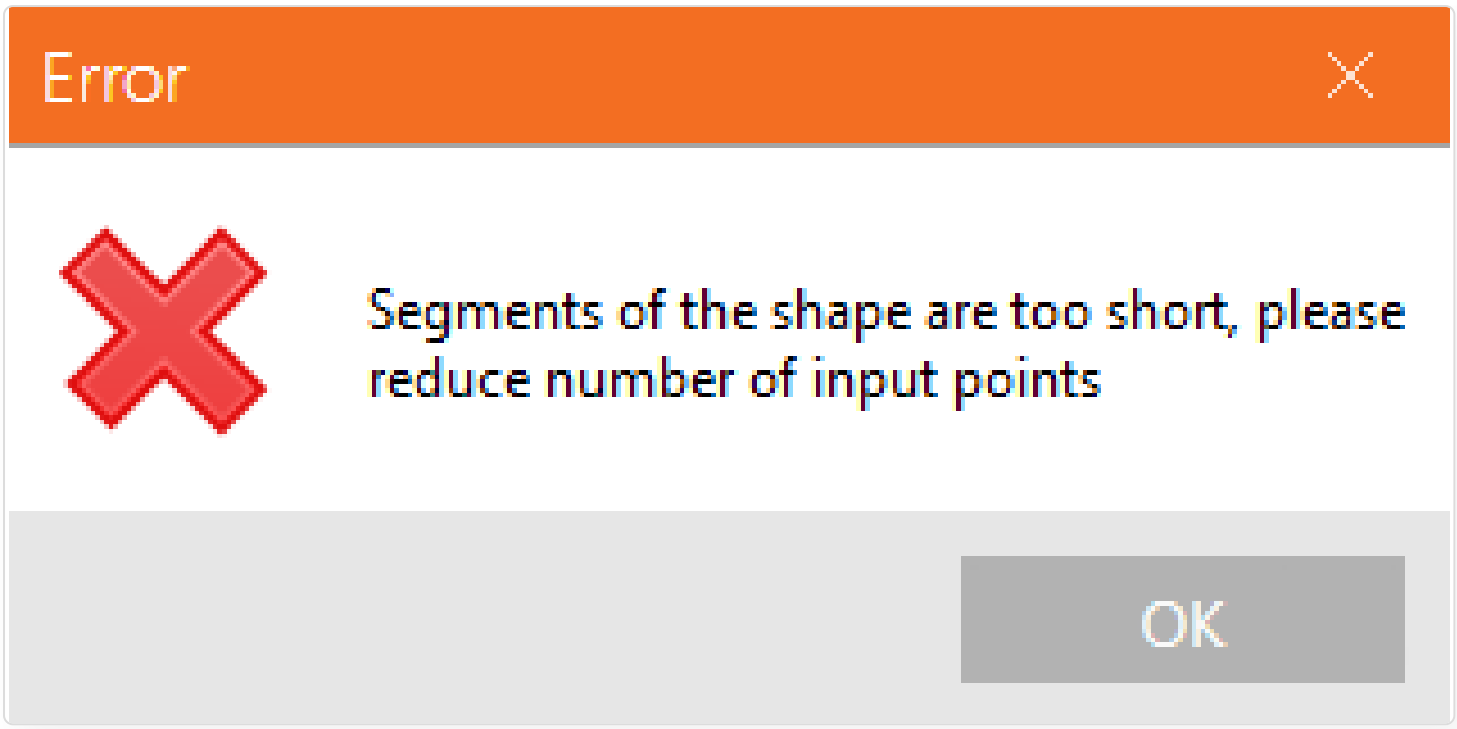
- **Center of gravity**: Positions the insertion point at the center of the shape's mass.



These options enable better control over the placement of imported shapes. The insertion point is displayed with a **red cross in the DXF editor**, alongside basic shape dimensions, providing users with a clear view of the real position before importing the shape into the model.

Enhanced Arc Import Process

The **arc import method** has been improved to address issues encountered with the previous approach. Earlier, arcs were converted into polygons composed of short line segments, often resulting in small edges that caused import errors. Users frequently had to adjust these shapes manually in software like AutoCAD to avoid such errors. The updated method allows arcs to be imported directly, maintaining their true form and eliminating the problem of small edges, which reduces the need for manual adjustments.



Released in IDEA StatiCa patch 24.0.2.

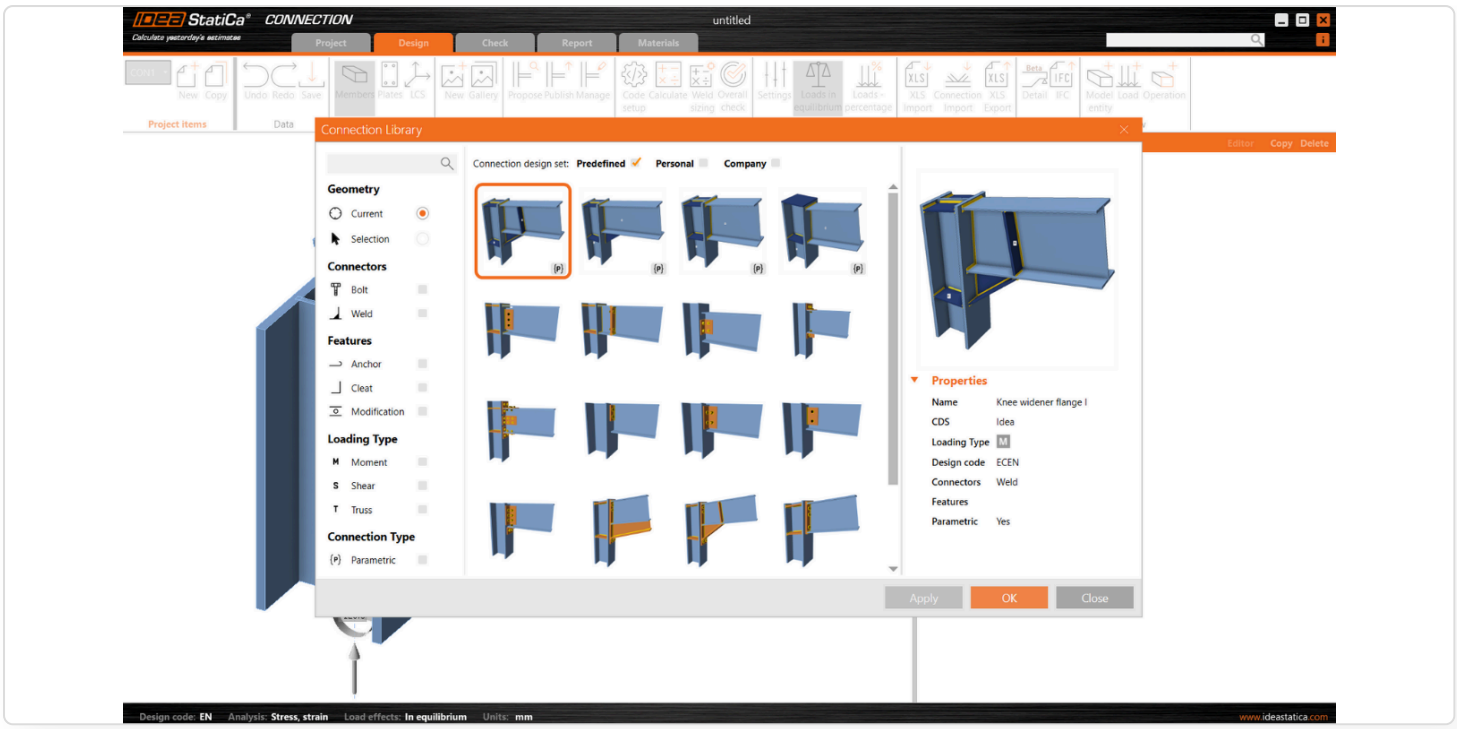
Parametric templates in Connection Library

The Connection Library supports the use of parametric templates, significantly streamlining the workflow for connection designers. This allows for the creation and utilization of a universal set of templates that can be easily customized and applied to various design scenarios.
In IDEA StatiCa Connection, we have the possibility to create connection models using parameters (relations defined between individual entities). The parametric design allows us to design standardized connections efficiently - **read about how to work with parameters** in this article.

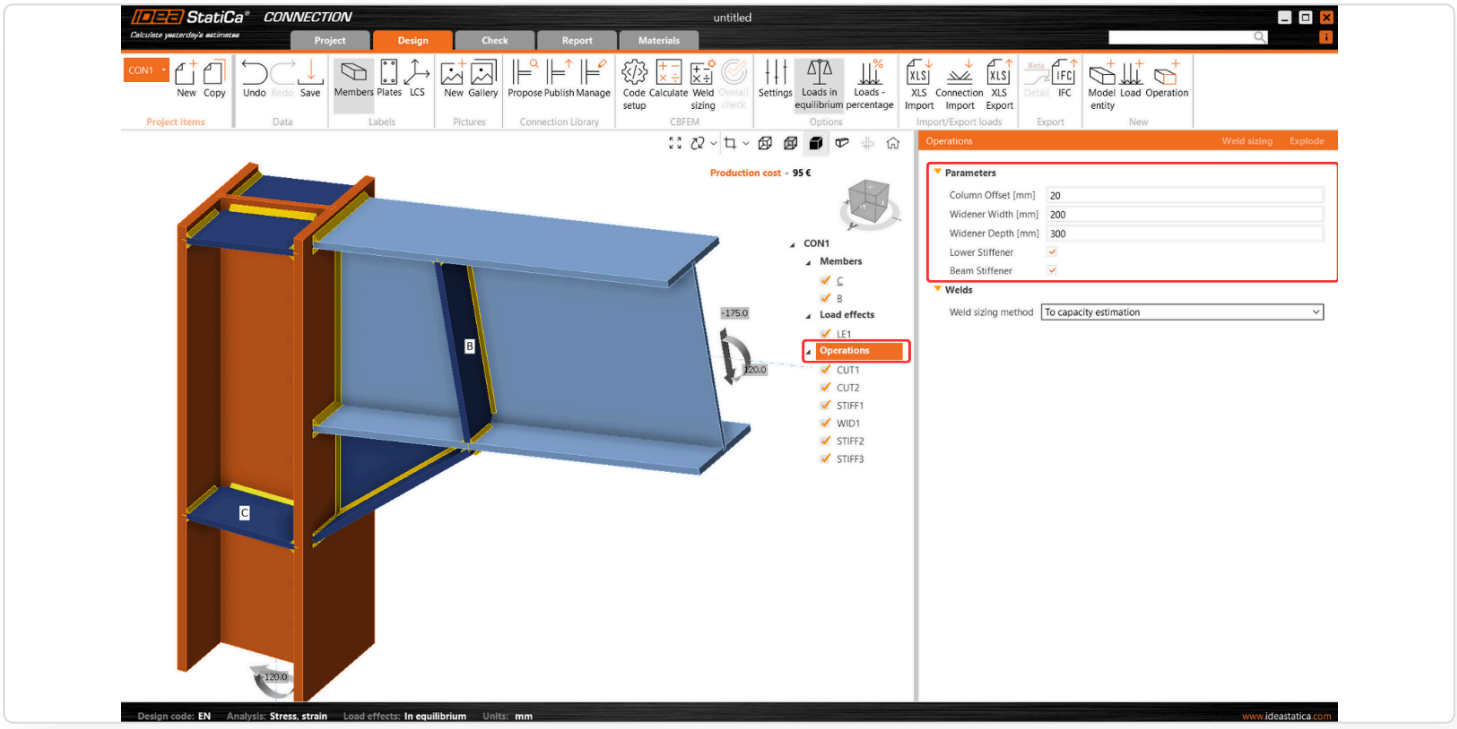
The integration of **parametric templates** into the Connection Library offers a transformational approach to connection designer workflows. **It allows users to create and use a universal collection of templates that can be effortlessly customized and deployed in different design contexts.**

How does it work?

The user can upload the created connection to their company or personal set as with any previous design, and it can even be done **with the defined parameters**. Once there is the same geometry in the project and the solution can be repeated, the user can apply this pre-prepared design (template) with all the parameters.

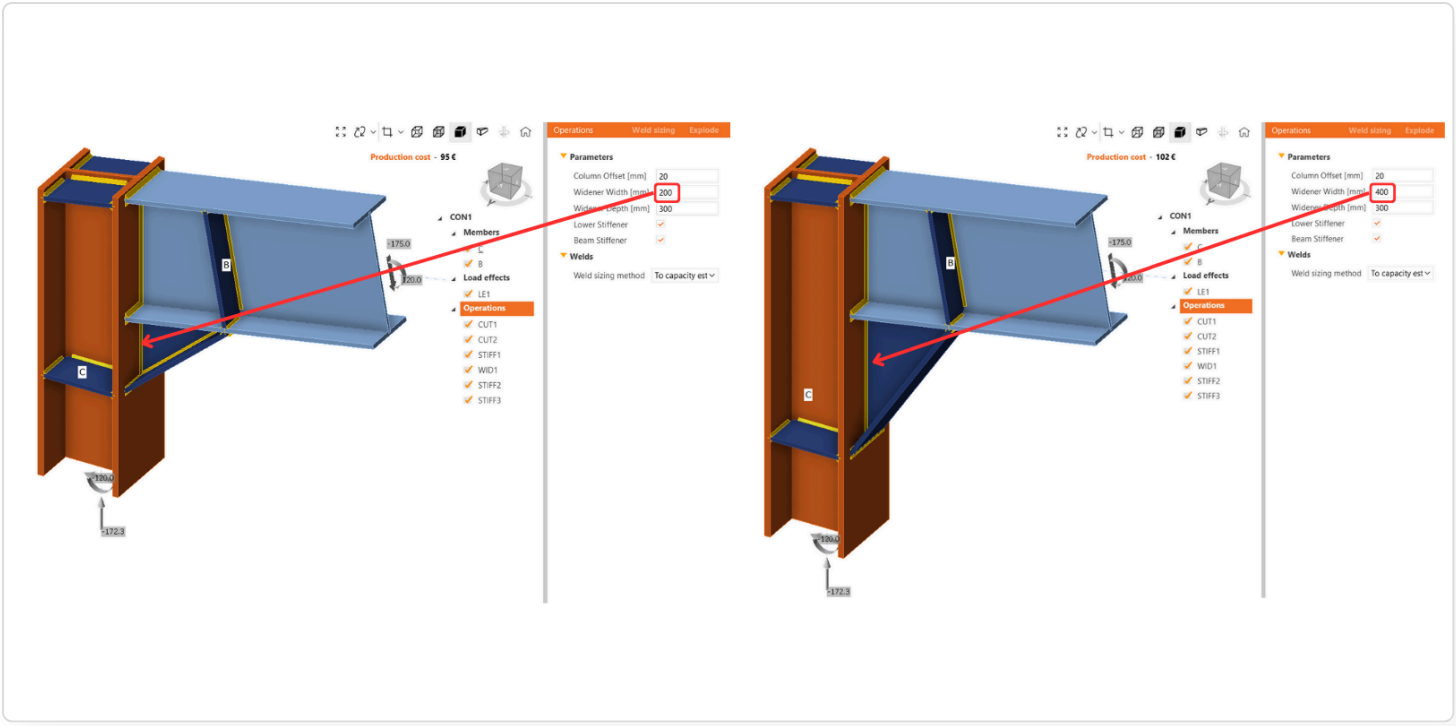


In addition, after proposing the template, it is possible to **change the parameters directly in the main design window**, and there is no need to go into developer mode. This user-friendly environment allows less experienced users to work with predefined parameters safely according to the presets of the senior designers.



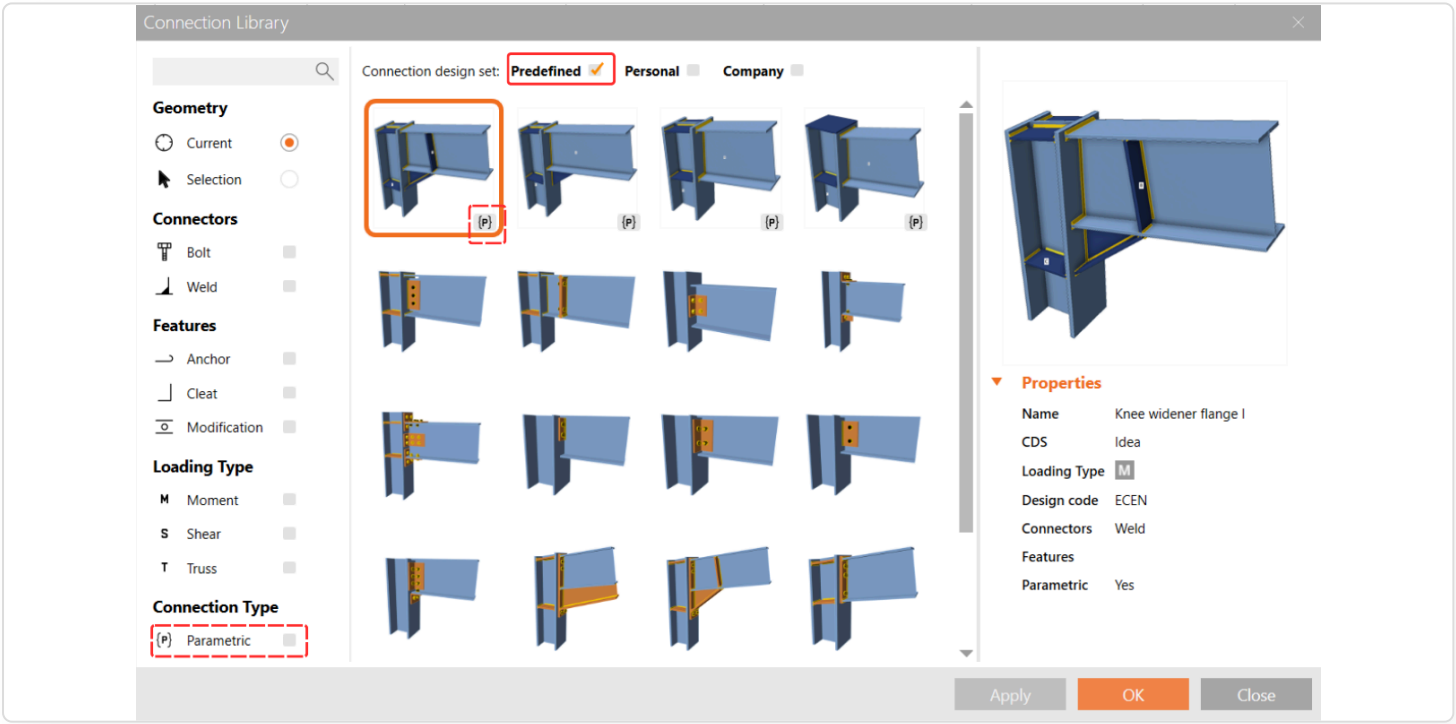
This makes further optimization very simple as there is the possibility of avoiding changing each entity one by one when using operations.

By changing one parameter, **multiple steps** can be performed at once. For example, when changing the widener width, not only the widener itself, but also all the related welds and the location of the stiffener are affected:



The locked modifications in the property list of some manufacturing operations are disabled in such cases. However, if the user wishes, they can break the parameters using the "**explode**" button. And continue with modifying operations.

The templates to which the parameters are linked are marked with a lowercase {p}. Several parametric templates have already been prepared and made available in the **predefined design set by the IDEA StatiCa team**.



What are the benefits of parametric templates?

- **Universal templates**: Designers can access a broad collection of parametric templates from the Connection Library. These templates are designed to be universally applicable, providing a solid foundation for a wide range of projects.
- **Parametric customization**: Through the Developer tab, users can define specific parameters for each template, allowing for a high degree of customization and flexibility in design.

- **Template identification**: Parametric templates are easily identifiable by the symbol {p}, ensuring that users can quickly recognize and select them for their projects.
- **Enhanced library filtering**: The filter in the Propose window of the Connection Library, enables users to efficiently find parametric templates among the library's extensive offerings.
- **Publication control**: When a connection contains parameters, designers have the option to publish these parametric templates to the Connection Library. This feature offers flexibility in sharing customized templates with the broader user community or keeping them private for individual or internal use.

Impact on workflow

The inclusion of parametric templates in the Connection Library represents a significant advancement in the design process for connection designers. This functionality simplifies the design process by providing:

- **Efficiency**: The use of templates speeds up the design phase, allowing for quicker iterations and modifications.
- **Consistency**: Parametric templates ensure design consistency across projects, which is crucial for maintaining standards and quality.
- **Collaboration**: The ability to share customized templates enhances collaboration among teams and with the wider design community.
- **Customization**: Designers can tailor templates to specific project requirements, enhancing design accuracy and effectiveness.

Released in IDEA StatiCa patch 23.1.5.

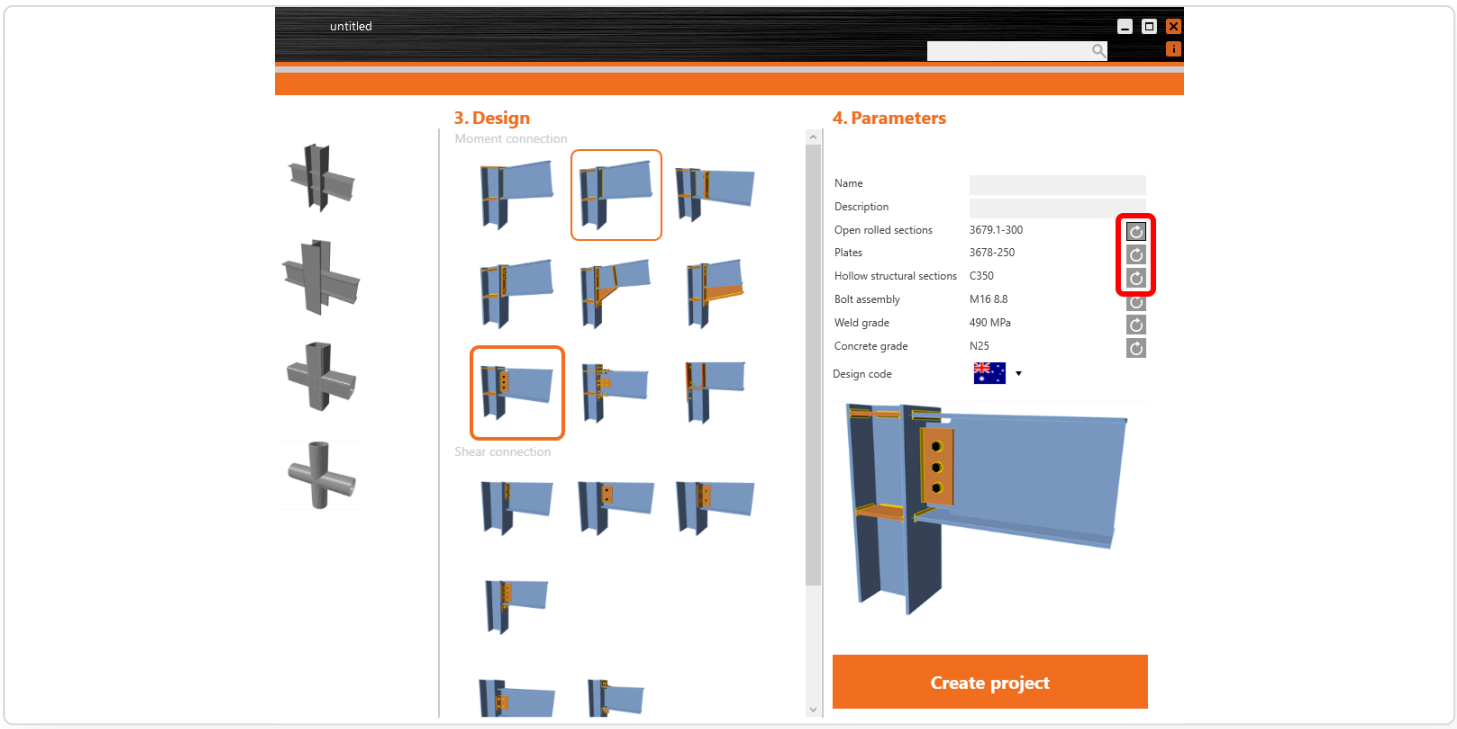
Regional improvements in 24.1

Every major version of IDEA StatiCa brings several improvements focused on the needs of specific regions. Therefore, version 24.1 also brings updates that will help make connection design easier in different countries. Five major regional improvements of version 24.1 are:

- Material defaults for various types of cross-sections (AISC, AS)
- PJP weld type for Canada and Australia
- Cross-section and material databases updated
- Through bolts code-check for AISC
- New languages for Theoretical Background

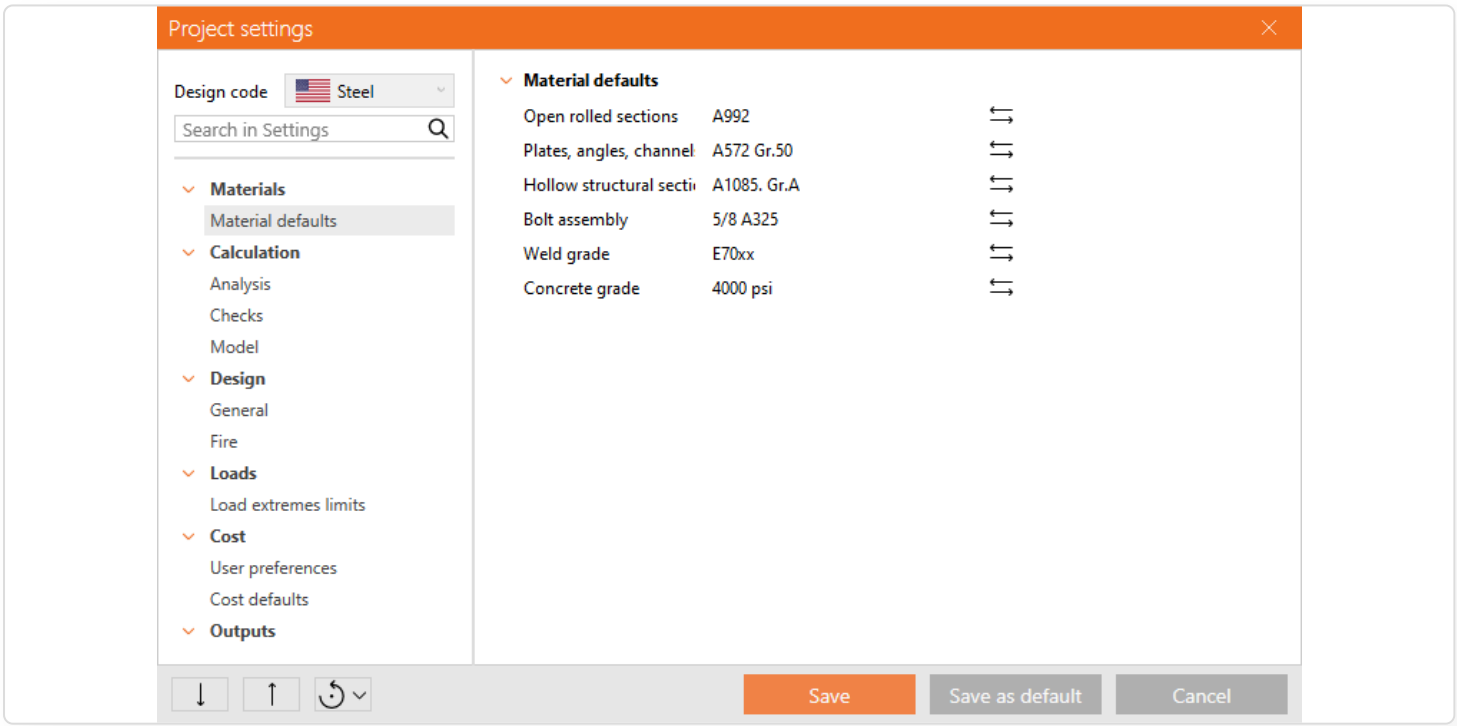
Material defaults for various types of cross-sections (AISC, AS)

Default materials of different cross-section types are different according to the selected code. In AISC and AS codes, different materials for specific cross-section types are typically used. You can assign different material to the cross-section in the **Preferences section** of the new project or in the **Project settings** of an already existing one.



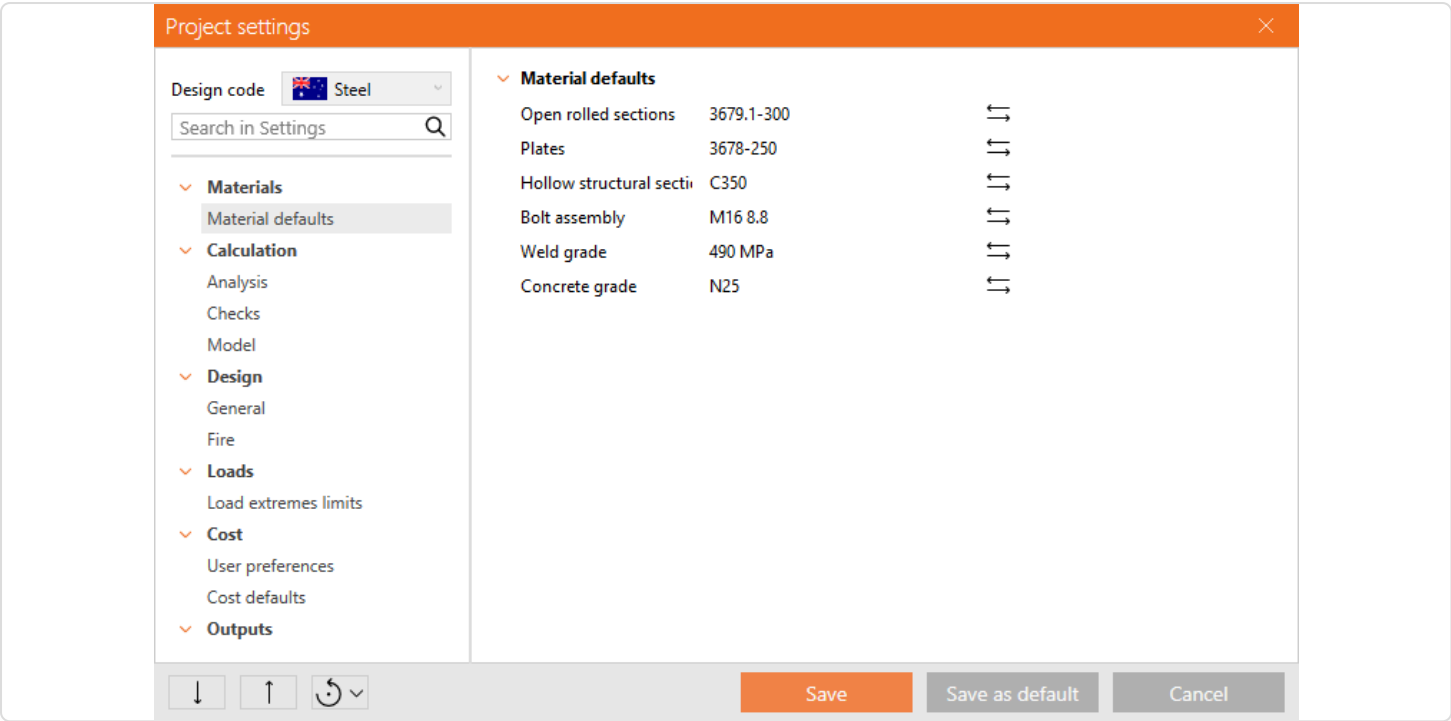
In **AISC code**, materials are divided into three groups:

- Opened rolled sections – I and T sections and all welded sections created from them
- Plates, angles, channels – plates, angles, and channels; rods; all welded sections from them; cold-formed sections except tubes
- Hollow structural sections – RHS, SHS, and tubes



In **AS code**, materials are divided into three groups:

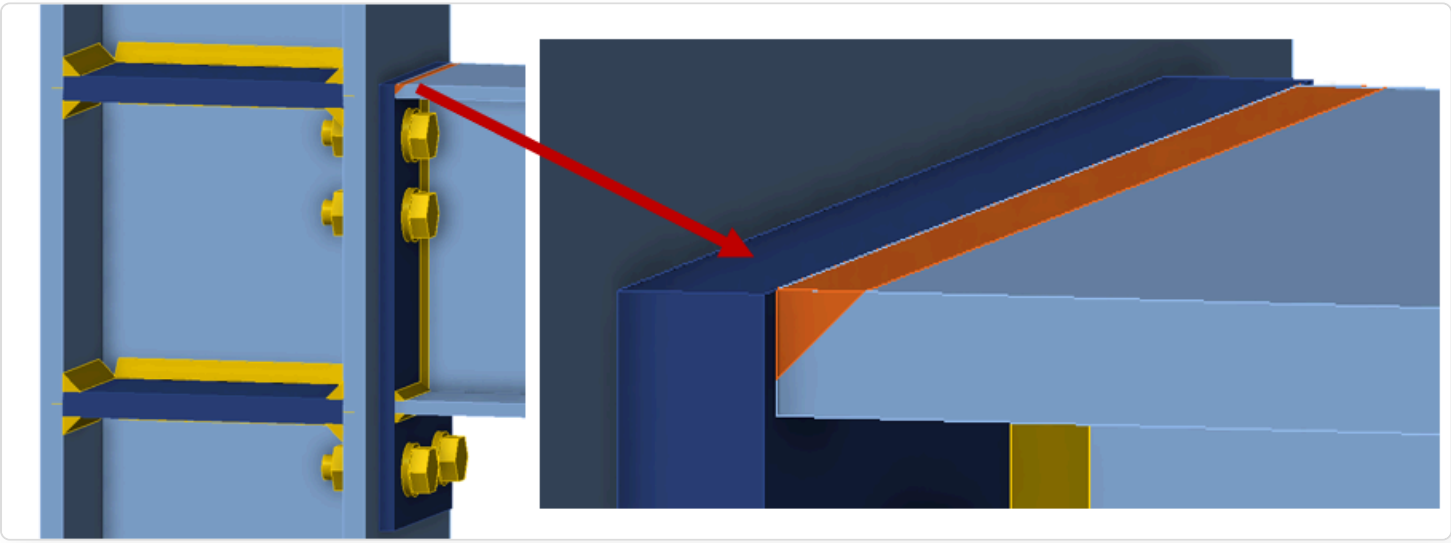
- Opened rolled sections – I and T sections, channels, angles, rods; all welded sections from them
- Plates – plates; all welded sections from them; cold-formed except tubes
- Hollow structural sections – RHS, SHS, and tubes



All the preset values can be **saved as defaults** and also shares (i.e., within a company) to be used in future projects. Read more about how the Project settings work.

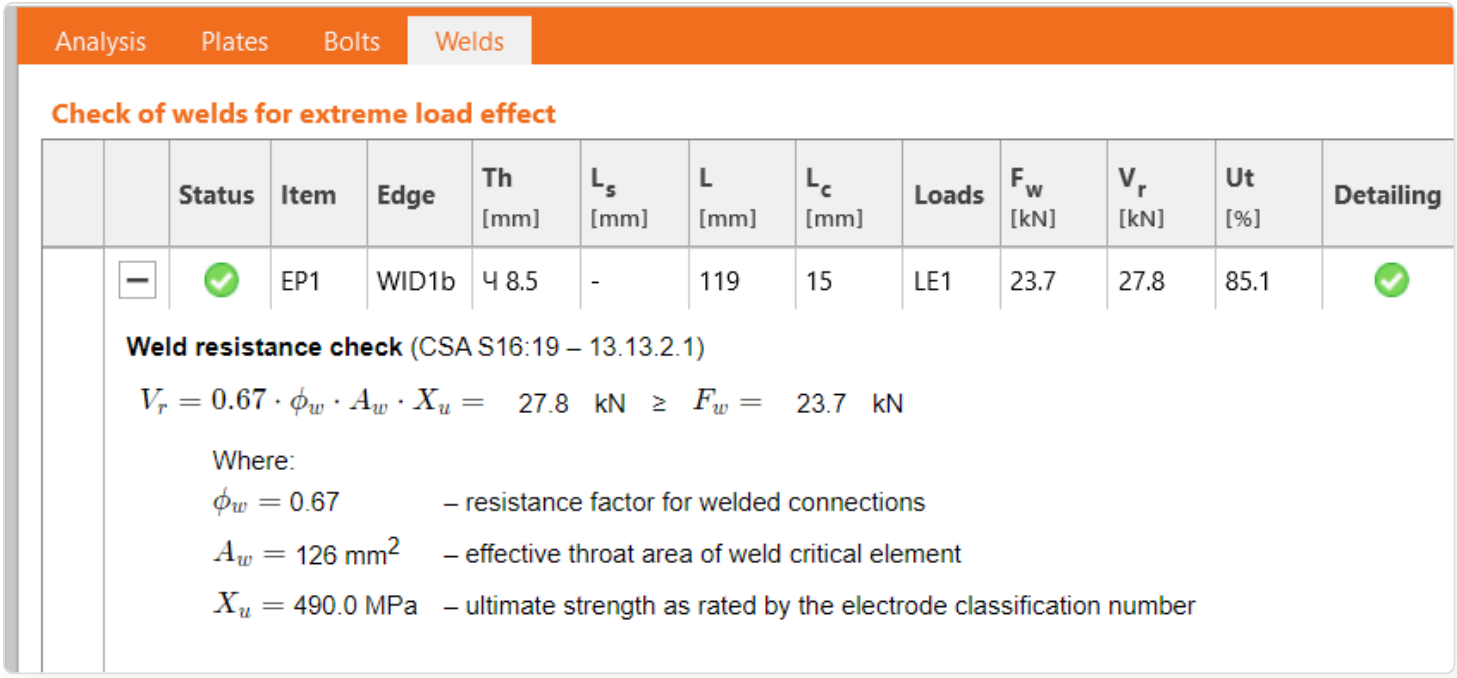
Released in IDEA StatiCa version 24.1.

PJP welds for CSA and AS

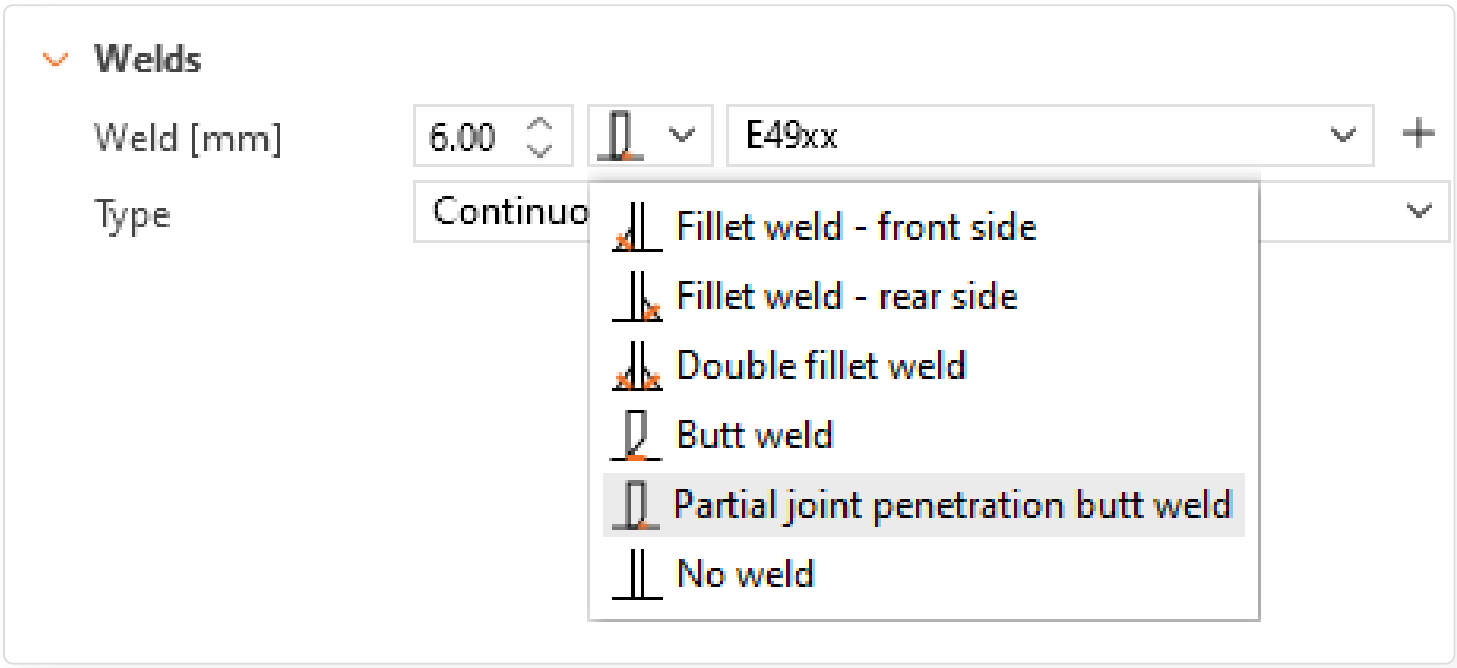


Partial joint penetration groove/butt welds are available for AISC design code and are embedded also for Australian and Canadian codes.

Partial joint penetration is a typical weld type widely used in steel structural design, but not that easy to model thanks to their specifics. However, it was successfully implemented into other design standards – CSA S16-14; CSA S16:19 and AS 4100:2020.

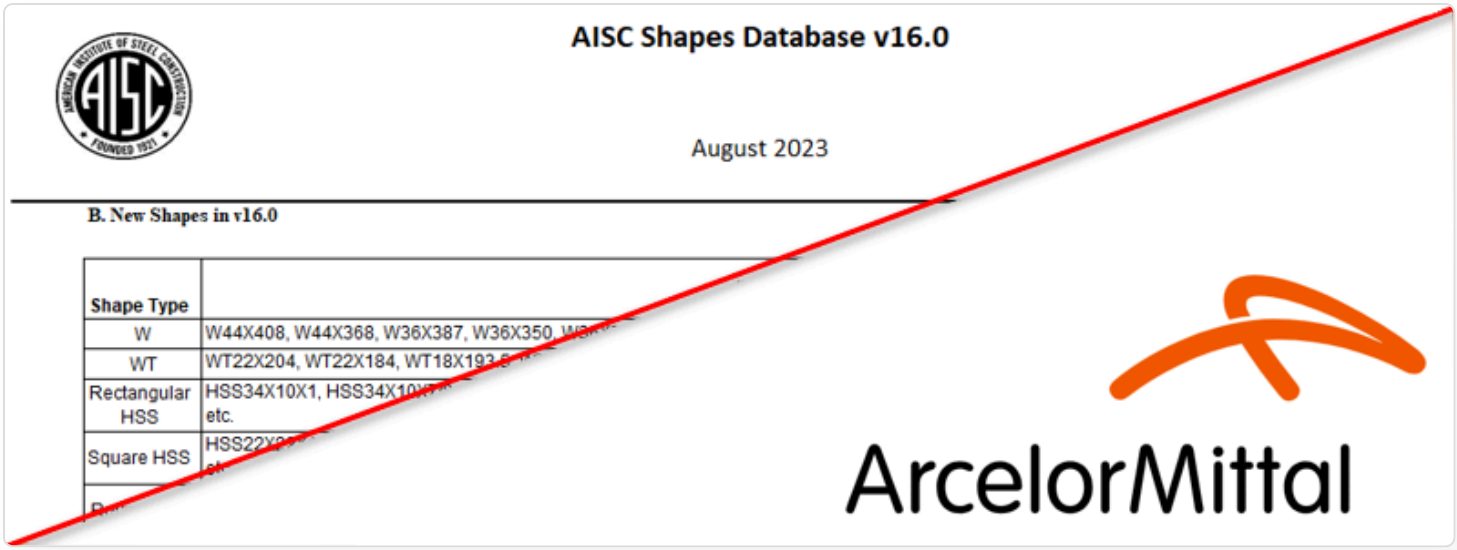


The PJP weld type can be selected in any operation with welds.



Released in IDEA StatiCa version 24.1.

Update of cross-section and material databases

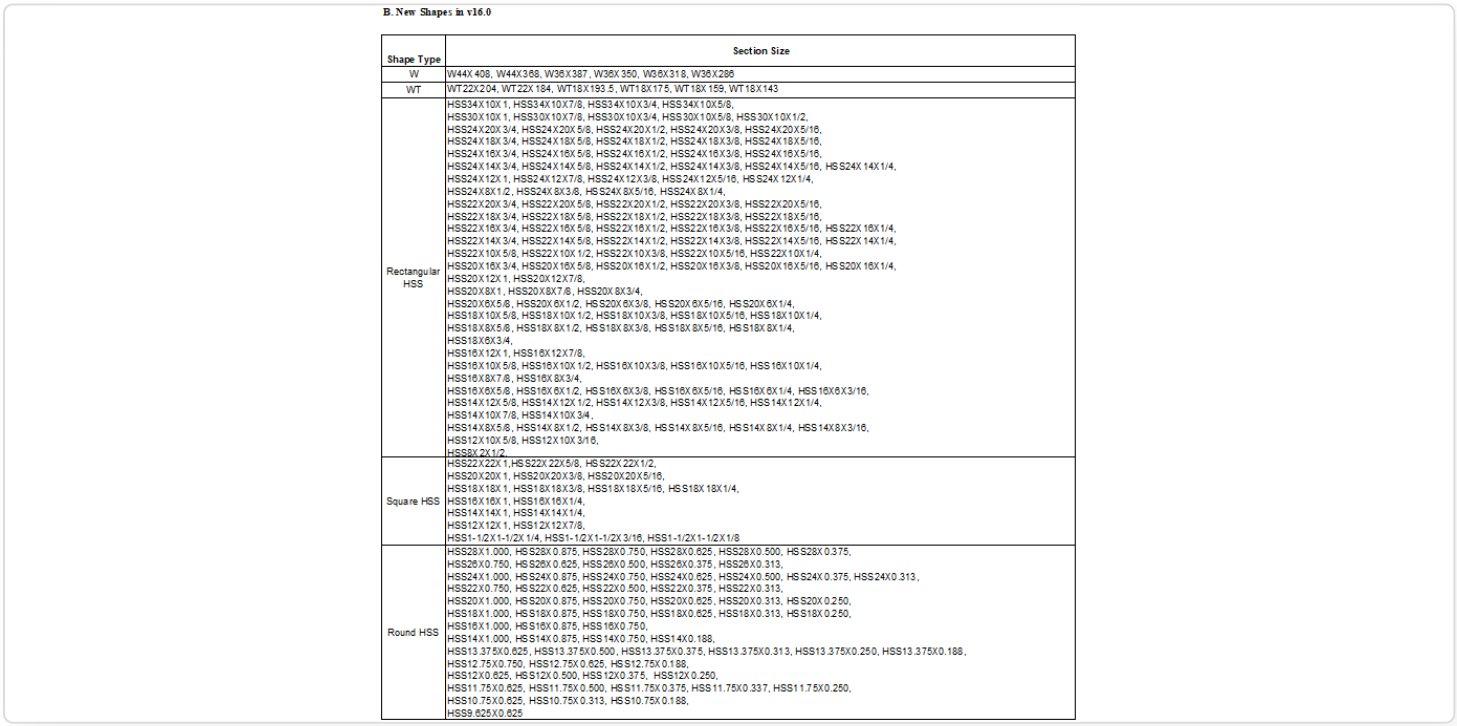


IDEA StatiCa takes the maximum effort to keep the tools up to date with any code-dependent but also best-practice-oriented changes. That's why the databases have been updated in several ways. The cross-section types were extended according to AISC standard (v16.0) and ArcelorMittal databases.

Updated AISC Shapes Database based on v16.0

Following the last release of AISC database (currently v16.0), engineers and designers with projects processed according to AISC standards can find all of these profile types in all IDEA StatiCa Steel applications.

Here is the list of updated profiles:



B. New Shapes in v16.0

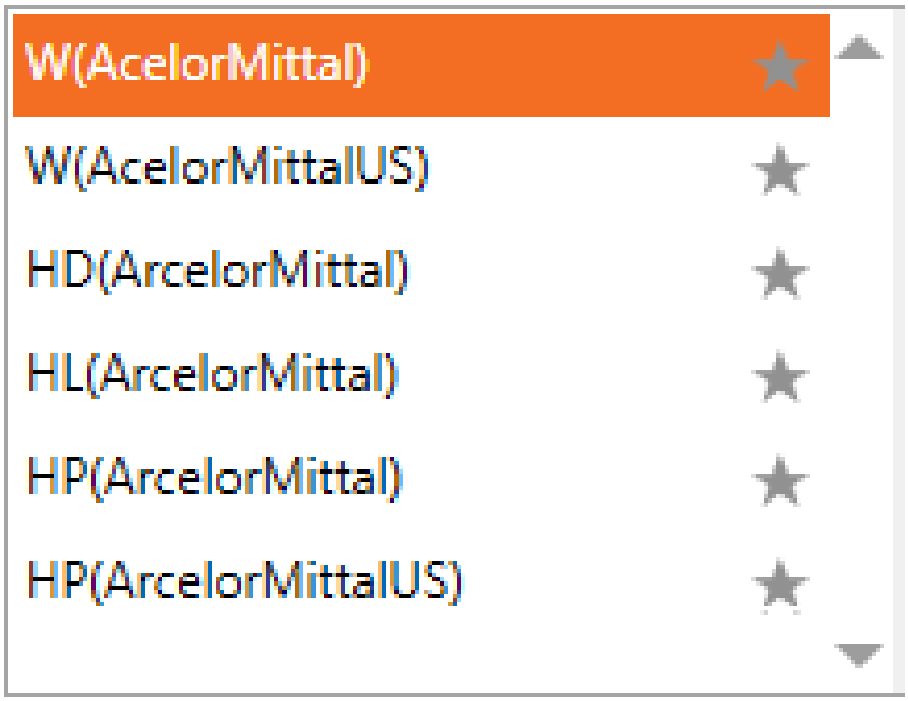
Shape Type	Section Size
W	W44X408, W44X368, W36X387, W36X350, W36X318, W36X286
WT	WT22X204, WT22X184, WT18X193.5, WT18X175, WT18X159, WT18X143
Rectangular HSS	HSS34X10X1, HSS34X10X7/8, HSS34X10X3/4, HSS34X10X5/8, HSS30X10X1, HSS30X10X7/8, HSS30X10X3/4, HSS30X10X5/8, HSS30X10X1/2, HSS24X20X3/4, HSS24X20X5/8, HSS24X20X1/2, HSS24X20X3/8, HSS24X20X5/16, HSS24X18X3/4, HSS24X18X5/8, HSS24X18X1/2, HSS24X18X3/8, HSS24X18X5/16, HSS24X16X3/4, HSS24X16X5/8, HSS24X16X1/2, HSS24X16X3/8, HSS24X16X5/16, HSS24X14X3/4, HSS24X14X5/8, HSS24X14X1/2, HSS24X14X3/8, HSS24X14X5/16, HSS24X14X1/4, HSS24X12X1, HSS24X12X7/8, HSS24X12X3/8, HSS24X12X5/16, HSS24X12X1/4, HSS24X8X1/2, HSS24X8X3/8, HSS24X8X5/16, HSS24X8X1/4, HSS22X20X3/4, HSS22X20X5/8, HSS22X20X1/2, HSS22X20X3/8, HSS22X20X5/16, HSS22X18X3/4, HSS22X18X5/8, HSS22X18X1/2, HSS22X18X3/8, HSS22X18X5/16, HSS22X16X3/4, HSS22X16X5/8, HSS22X16X1/2, HSS22X16X3/8, HSS22X16X5/16, HSS22X16X1/4, HSS22X14X3/4, HSS22X14X5/8, HSS22X14X1/2, HSS22X14X3/8, HSS22X14X5/16, HSS22X14X1/4, HSS22X10X5/8, HSS22X10X1/2, HSS22X10X3/8, HSS22X10X5/16, HSS22X10X1/4, HSS20X18X3/4, HSS20X18X5/8, HSS20X18X1/2, HSS20X18X3/8, HSS20X18X5/16, HSS20X18X1/4, HSS20X12X1, HSS20X12X7/8, HSS20X8X1, HSS20X8X7/8, HSS20X8X3/4, HSS20X6X5/8, HSS20X6X1/2, HSS20X6X3/8, HSS20X6X5/16, HSS20X6X1/4, HSS18X10X5/8, HSS18X10X1/2, HSS18X10X3/8, HSS18X10X5/16, HSS18X10X1/4, HSS18X8X5/8, HSS18X8X1/2, HSS18X8X3/8, HSS18X8X5/16, HSS18X8X1/4, HSS18X6X3/4, HSS16X12X1, HSS16X12X7/8, HSS16X10X5/8, HSS16X10X1/2, HSS16X10X3/8, HSS16X10X5/16, HSS16X10X1/4, HSS16X8X7/8, HSS16X8X3/4, HSS16X6X5/8, HSS16X6X1/2, HSS16X6X3/8, HSS16X6X5/16, HSS16X6X1/4, HSS16X6X3/16, HSS14X12X5/8, HSS14X12X1/2, HSS14X12X3/8, HSS14X12X5/16, HSS14X12X1/4, HSS14X10X7/8, HSS14X10X3/4, HSS14X8X5/8, HSS14X8X1/2, HSS14X8X3/8, HSS14X8X5/16, HSS14X8X1/4, HSS14X8X3/16, HSS12X10X5/8, HSS12X10X3/16, HSS8X2X1/2,
Square HSS	HSS22X22X1, HSS22X22X5/8, HSS22X22X1/2, HSS20X20X1, HSS20X20X3/8, HSS20X20X5/16, HSS18X18X1, HSS18X18X3/8, HSS18X18X5/16, HSS18X18X1/4, HSS16X16X1, HSS16X16X1/4, HSS14X14X1, HSS14X14X1/4, HSS12X12X1, HSS12X12X7/8, HSS1-1/2X1-1/2X1/4, HSS1-1/2X1-1/2X3/16, HSS1-1/2X1-1/2X1/8
Round HSS	HSS28X1.000, HSS28X0.875, HSS28X0.750, HSS28X0.625, HSS28X0.500, HSS28X0.375, HSS26X0.750, HSS26X0.625, HSS26X0.500, HSS26X0.375, HSS26X0.313, HSS24X1.000, HSS24X0.875, HSS24X0.750, HSS24X0.625, HSS24X0.500, HSS24X0.375, HSS24X0.313, HSS22X0.750, HSS22X0.625, HSS22X0.500, HSS22X0.375, HSS22X0.313, HSS20X1.000, HSS20X0.875, HSS20X0.750, HSS20X0.625, HSS20X0.313, HSS20X0.250, HSS18X1.000, HSS18X0.875, HSS18X0.750, HSS18X0.625, HSS18X0.313, HSS18X0.250, HSS16X1.000, HSS16X0.875, HSS16X0.750, HSS14X1.000, HSS14X0.875, HSS14X0.750, HSS14X0.188, HSS13.375X0.625, HSS13.375X0.500, HSS13.375X0.375, HSS13.375X0.313, HSS13.375X0.250, HSS13.375X0.188, HSS12.75X0.750, HSS12.75X0.625, HSS12.75X0.188, HSS12X0.625, HSS12X0.500, HSS12X0.375, HSS12X0.250, HSS11.75X0.625, HSS11.75X0.500, HSS11.75X0.375, HSS11.75X0.337, HSS11.75X0.250, HSS10.75X0.625, HSS10.75X0.313, HSS10.75X0.188, HSS9.625X0.625

Updated ArcelorMittal databases

ArcelorMittal is one of the biggest manufacturers of steel, providing a standard sort of cross-sections and materials. In addition, they are innovating to get smarter and clearer steel products.

We have updated databases of cross-section and the groups are marked by 'ArcelorMittal' and 'ArcelorMittalUS' for the US market instead of the old 'ARC' and 'ARCUS' for clear identification.

Cross-sections



Added materials

We also added a new material, A913 Gr.80, for the American market and HISTAR certified by ArcelorMittal for Eurocode.

EN code:

HISTAR (ArcelorMittal)	HISTAR 355 (ETA-10/0156)
	HISTAR 355 L (ETA-10/0156)
	HISTAR 460 (ETA-10/0156)
	HISTAR 460 L (ETA-10/0156)
	HISTAR 355 TZ
	HISTAR 355 TZK
	HISTAR 460 TZ
	HISTAR 460 TZK

Added material - AISC and CAN:

Steel ASTM	A913 Gr.80

Released in IDEA StatiCa version 24.1.

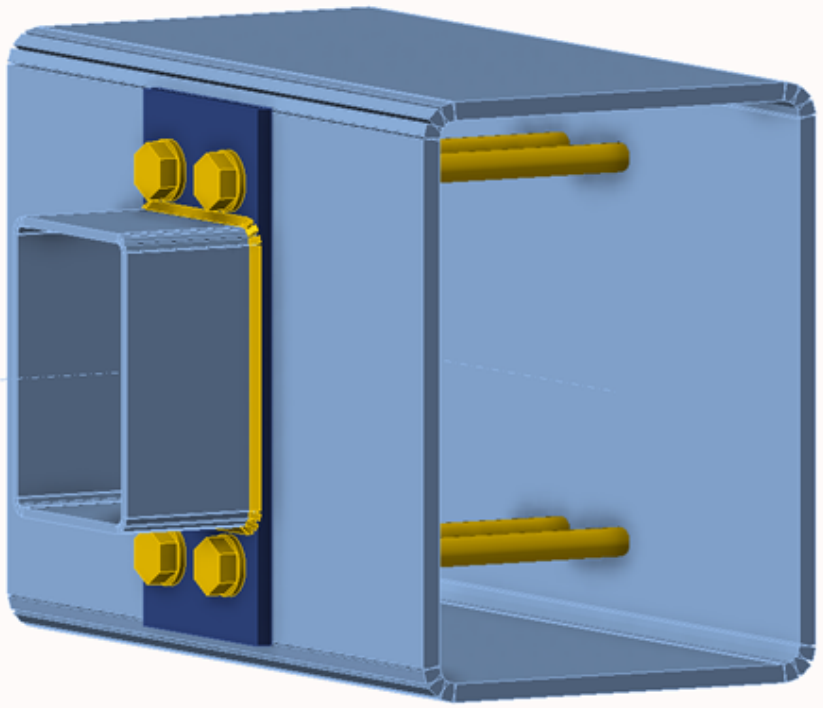
Through bolts code-check for AISC

Bearing check for pins and through bolts is embedded according to AISC 360 Specification, Chapter J7-1.

The calculation model doesn't take into account the bending of bolts while displaying the warning message. A more detailed description can be found in this article.

Bearing resistance check (AISC 360-22 – J3-6, AISC 360-22 – J7-1)

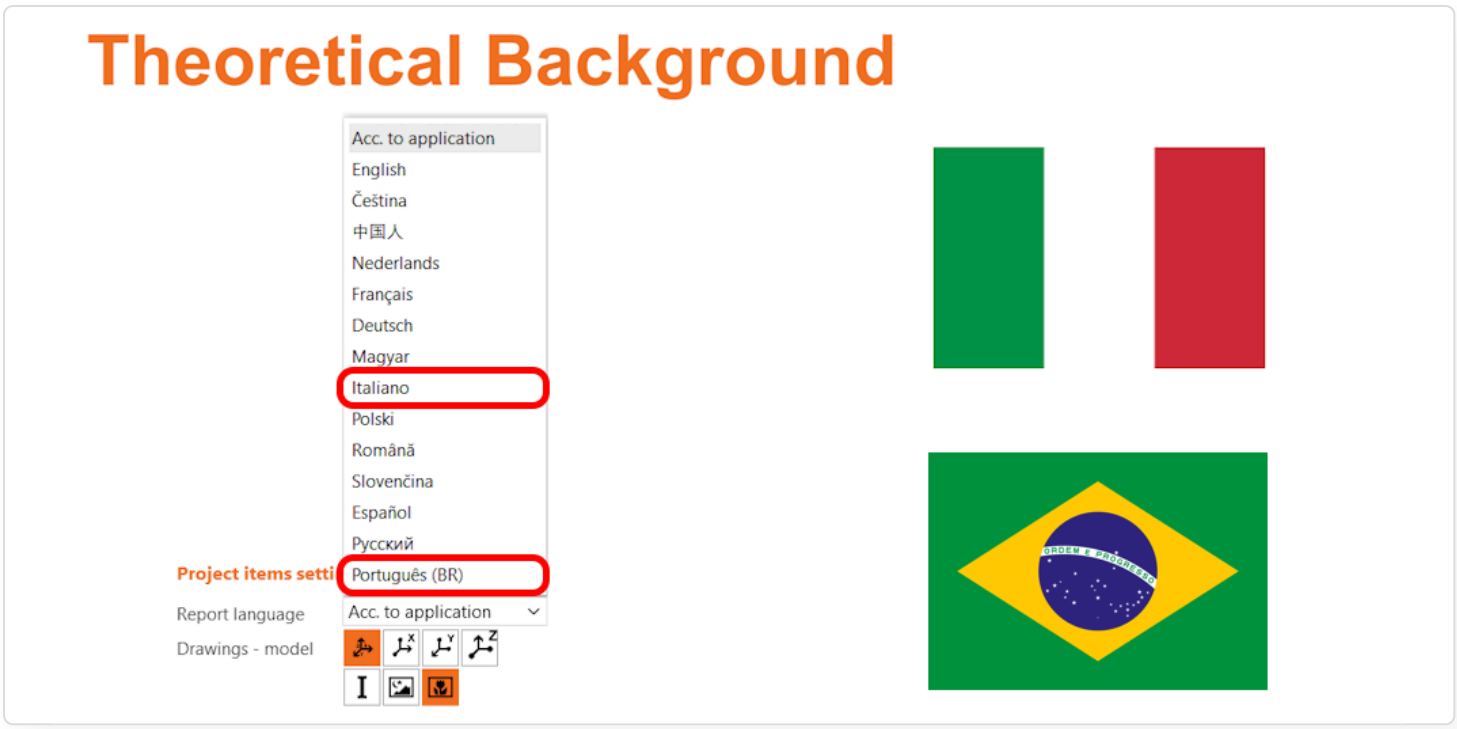
$$R_n = 1.20 \cdot l_c \cdot t \cdot F_u \quad \leq \quad 1.80 \cdot F_y \cdot A_{pb}$$



Released in IDEA StatiCa version 24.0.5.

Theoretical Background in Report updated with Italian and Portuguese

The report in IDEA StatiCa's steel apps offers you the option to change the language according to the project's requirements. The Theoretical Background has been translated into two new languages – Italian and Portuguese (Brazilian).



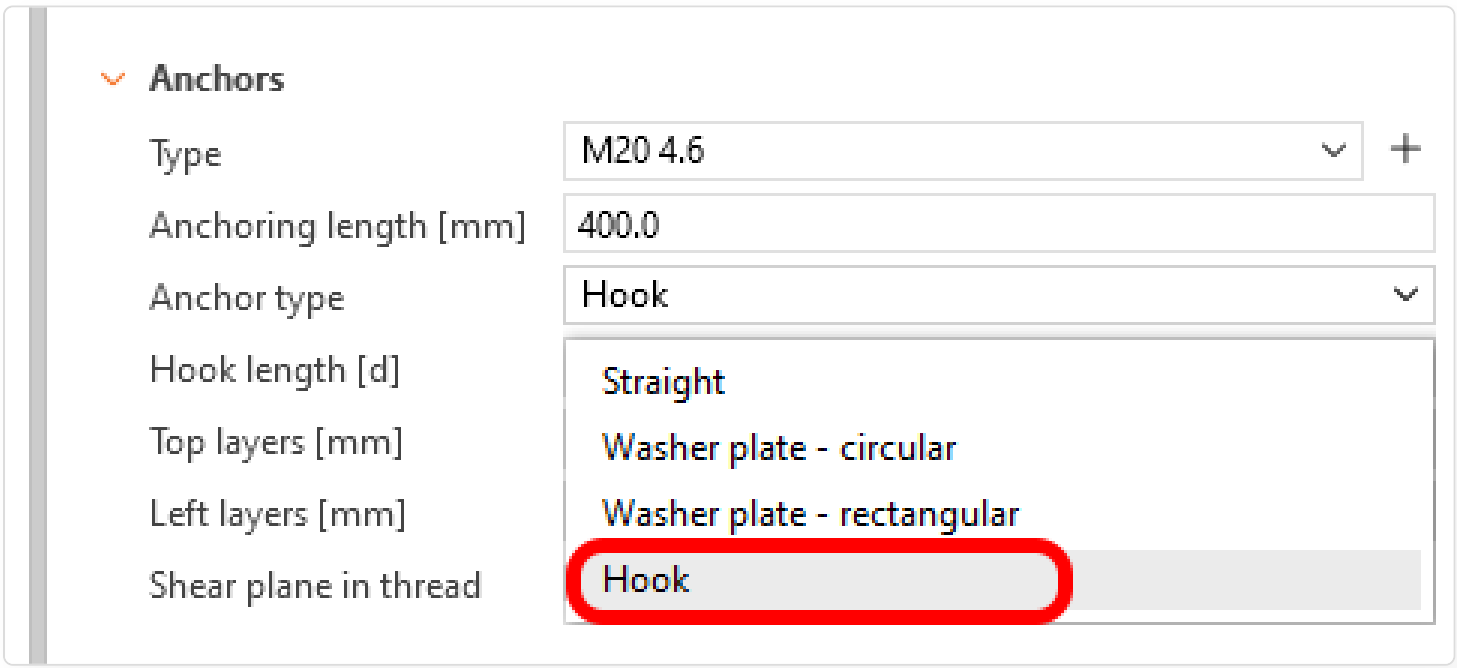
For more information about the Report's content, take a look at this comprehensive description.

Released in IDEA StatiCa version 24.0.3.

Automatic anchoring code-selection with hooked anchors

The anchor type list includes hooked anchors for EN designs. Code-check of tension and shear resistance for anchorage according to Eurocode feature automatic differentiation between 1993-1-8 and 1992-4 code requirements based on anchor type.

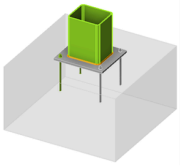
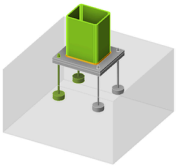
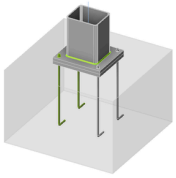
The Anchor type – 'Hooked anchors' – can be selected from the drop-down menus:



For this anchor type, pull-out resistance is checked with an assessment according to EN 1992-1-1.

The steel resistance values for anchors are taken into account according to this simple rule:

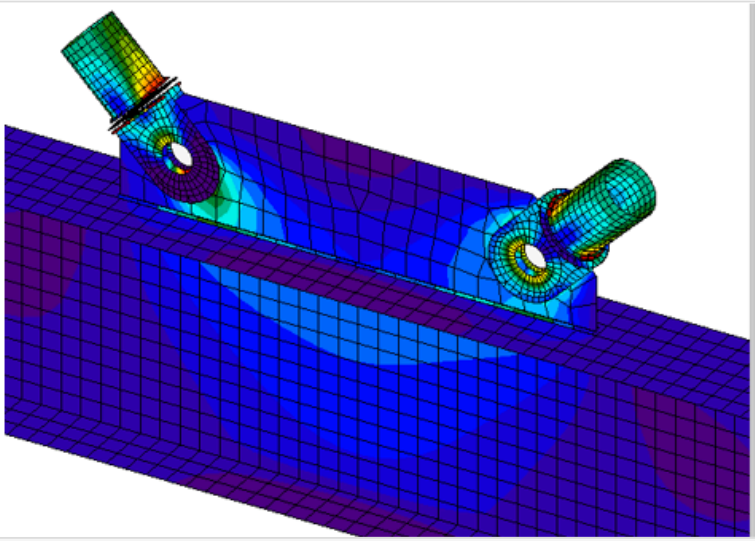
- Post-installed anchors (type = Straight) – resistance according to EN 1992-4
- Cast-in anchors (type = Washer plate or Hook) – resistance according to EN 1993-1-8

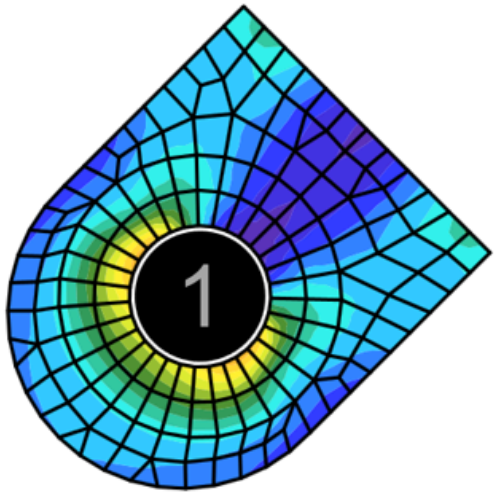
Anchor type	24.0		24.1	
	Anchor tensile resistance	Anchor shear resistance	Anchor tension resistance	Anchor shear resistance
Straight anchors 	EN 1992-4 - 7.2.1.3	EN 1992-4 - 7.2.2.3.2	EN 1992-4 - 7.2.1.3	EN 1992-4 - 7.2.2.3.2
Anchors with washers 	EN 1992-4 - 7.2.1.3	EN 1992-4 - 7.2.2.3.2	EN 1993-1-8 - Table 3.4	EN 1993-1-8 - Table 3.4
Hooked anchors 	not provided	not provided	EN 1993-1-8 - Table 3.4	EN 1993-1-8 - Table 3.4

This improvement is also based on FprEN 1993-1-8:2023 and is aligned with recommendations of the Dutch ECCS TC10 group. For more information about the approach of distinguishing between Post-installed and Cast-in anchors, see this document: Column bases in shear and normal force.

Accurate meshing around bolt and pin holes

There is significant accuracy in the field of mesh generation. The meshing around bolt and pin holes brings significant accuracy and efficiency in generating meshes for these specific areas.
The mesh in IDEA StatiCa Connection is generated automatically. The user can only adjust the mesh size, this avoids certain cases where the mesh might not be generated appropriately, leading users to attempt to set up a very fine mesh. To prevent this adjustment impacting the mesh for the entire model, causing other issues with the calculation or the default meshing around particularly large holes in relatively small plates, such as those used for pins, being inaccurately configured, the meshing algorithm ensures that the resulting mesh is accurate:





See the sizing rules for the meshing algorithm:

Number of elements in ring:

- n = (2x π x external radius)/element size
- external radius = 2x mesh ring
- n ≥ 8

Number of rings:

- number of rings = bolt hole radius/element size
- number of rings = ≥ 1

Slotted holes:

- It contains the same number of elements as a circular hole, but its shape is elongated into an elliptical form.

This provides a more precise mesh that accurately represents the material behavior, particularly in cases where larger openings are located within smaller plates.

BIM and Checkbot

Multi-management tools in Checkbot

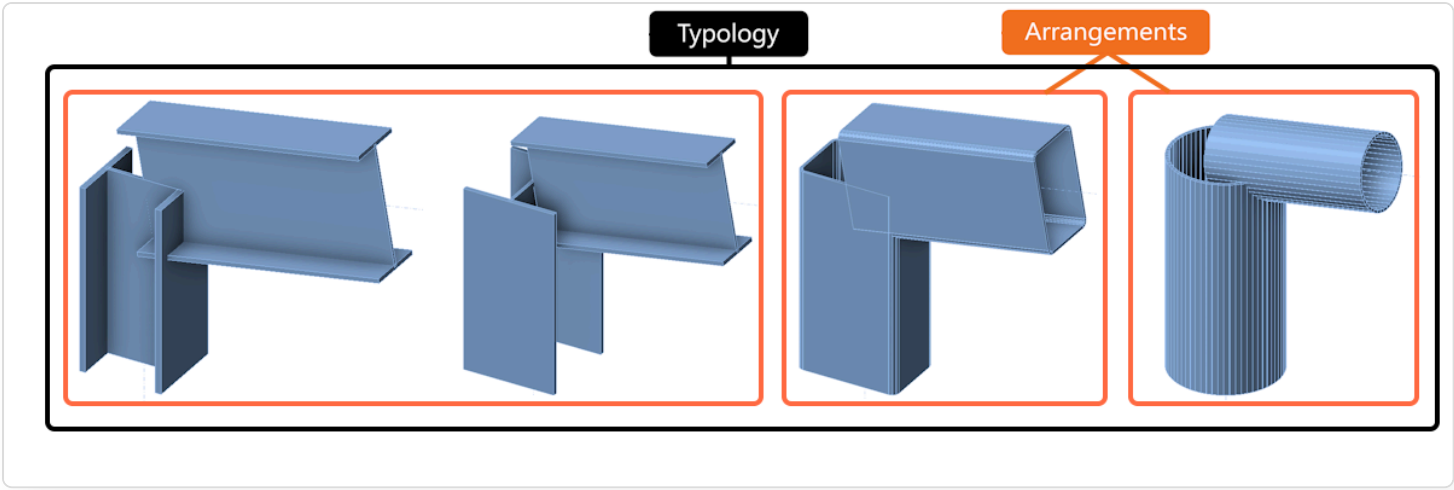
Checkbot provides engineers with a detailed view of connections and links with their BIM, FEA, and CAD software to share their global model. The tools enable quick selection, management, and bulk operations on grouped connections, improving workflow efficiency in large-scale projects.

Dynamic grouping in Checkbot

Checkbot is optimized to handle models with hundreds of connections, providing tools for bulk operations. The grouping tools allow efficient management of connections by categorizing them based on **typology** and **cross-section arrangement**. This functionality simplifies navigation, selection, and enables to perform bulk design of multiple connections.

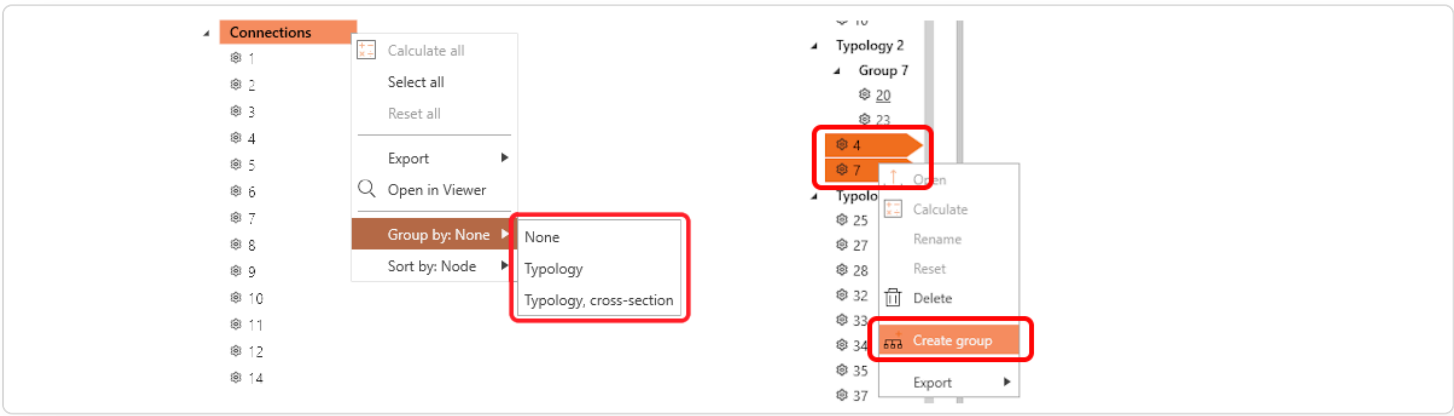
Dynamic Grouping automatically organizes connections in the project tree into two levels based on **typology** and **cross-section**:

1. **Typology** – considers the number of members and their relative positions (e.g., beam-to-beam, beam-to-column).
2. **Typology, cross-section (Arrangement)** – within the same typology, connections are further grouped by cross-section type. This is the lowest similarity that is required to enable searching for similar designs in the **Connection Library**.

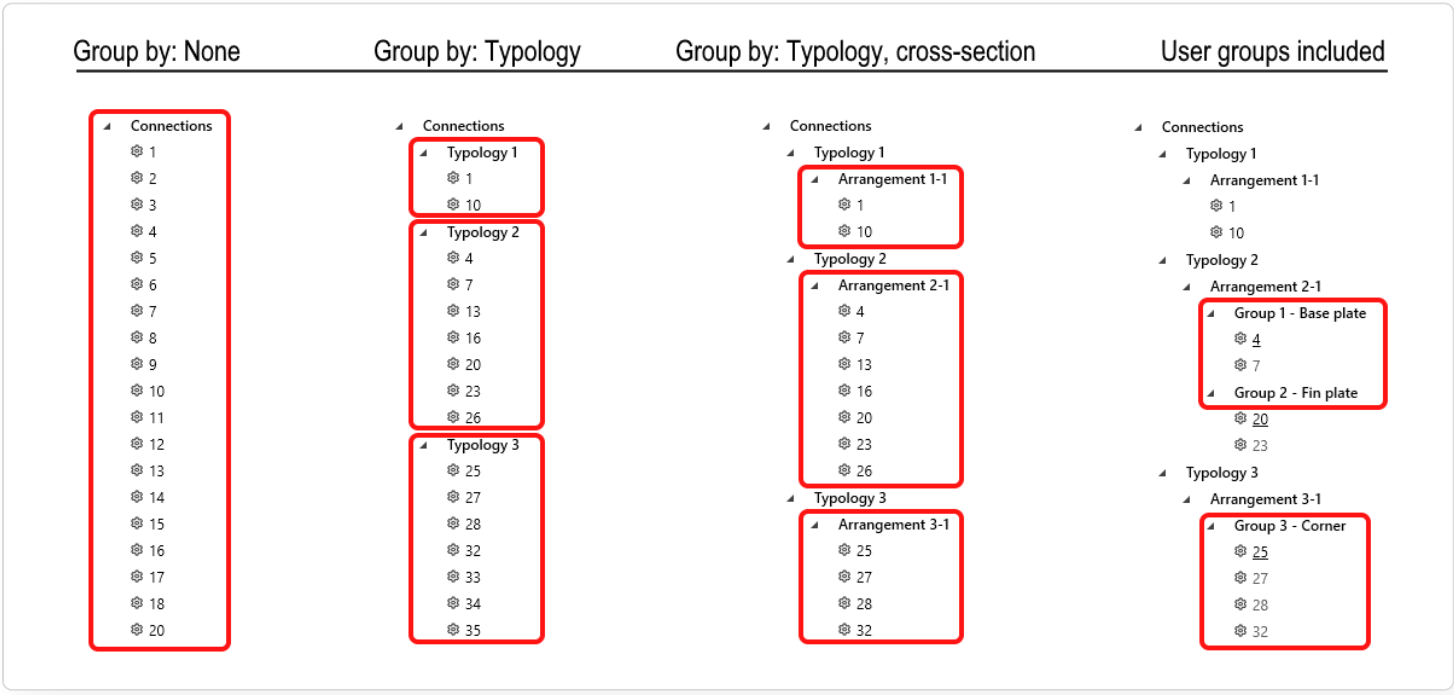


The third level of grouping is defined as **design groups**, which are created and managed by users. These groups allow for further customization, enabling users to organize specific sets of connections based on their project needs. The only condition is that connections within the same design group must share the same **typology** and **cross-section**.

The creation of all levels of groups, including dynamic grouping and design groups, is handled by the project tree.



The connection tree can have four different ordering options based on the level of grouping.

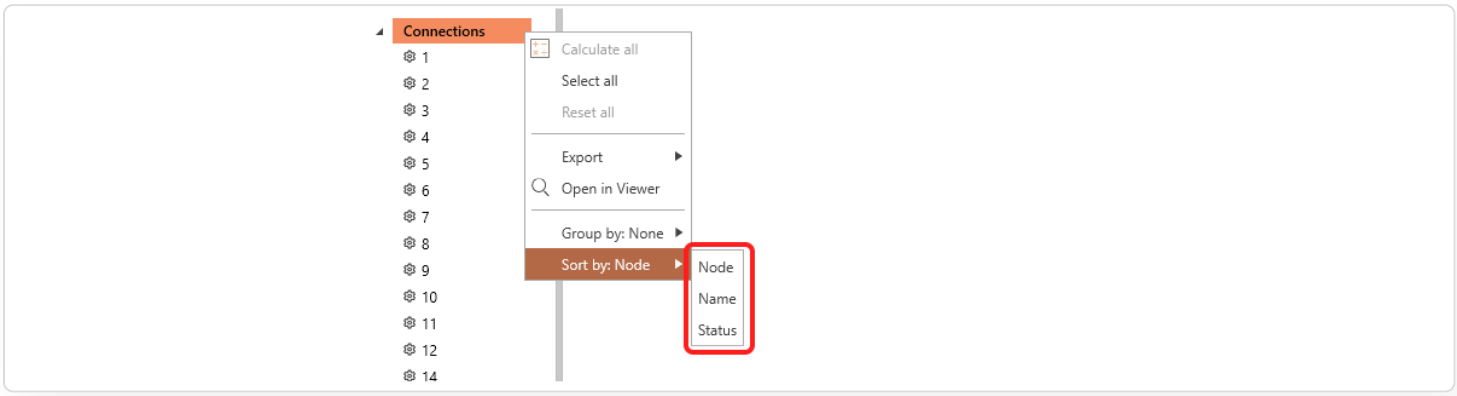


Clear Connection tree

The tree and property grid offer users a clearer project overview and faster design management. These improvements enable quicker navigation and more efficient control of design properties, simplifying the process and ensuring critical details are easily accessible.

Sorting connections in the tree is based on the following:

- Node
- Name
- Status (To be designed, To be checked, Code-check passed, Code-check failed)

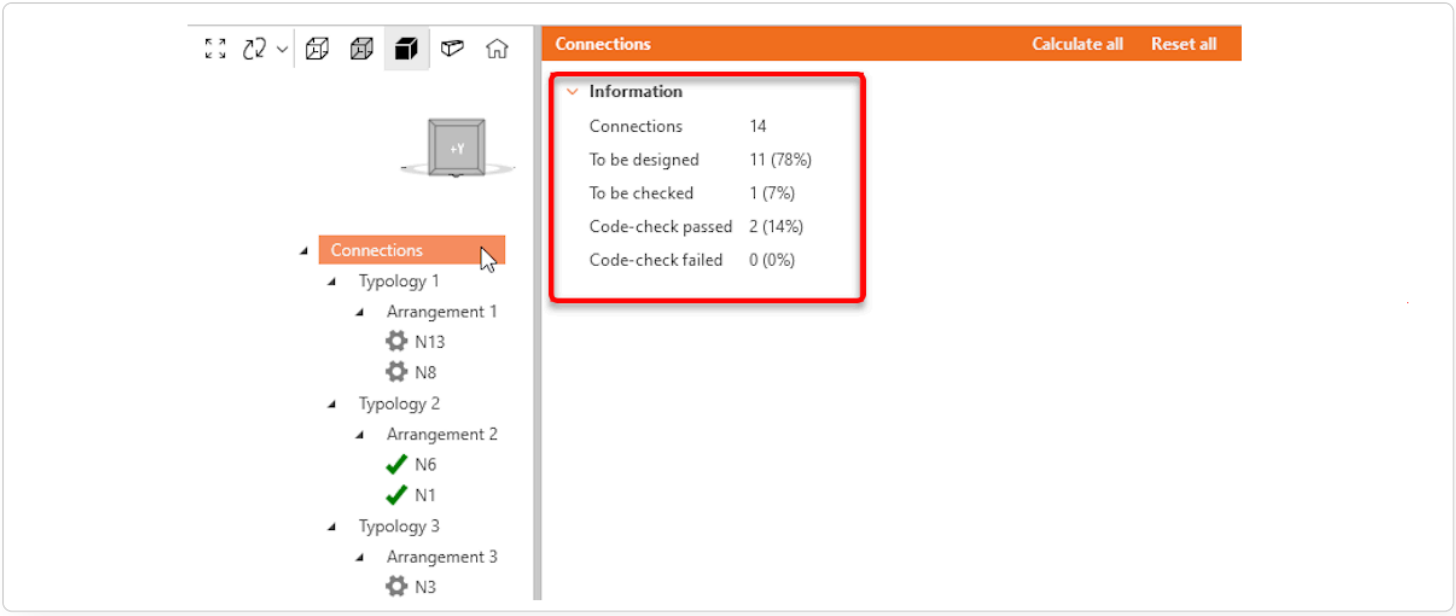


Connection groups overview information:

Users can now click the root group in the tree to instantly access an overview of information relevant to the selected group (Connections, Typology, Arrangement, Design group). This feature provides a centralized way to view key project details:

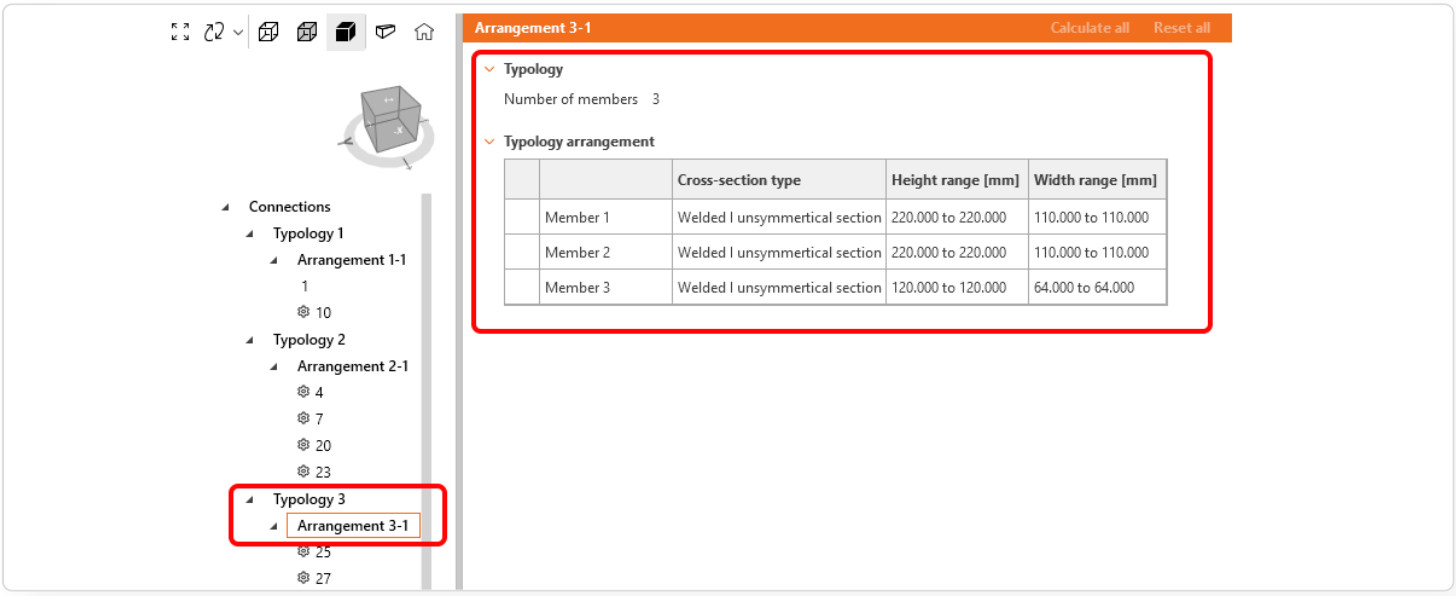
Connections

- Total number of connections
- To be designed
- To be checked (designed)
- Code-check passed
- Code-check failed



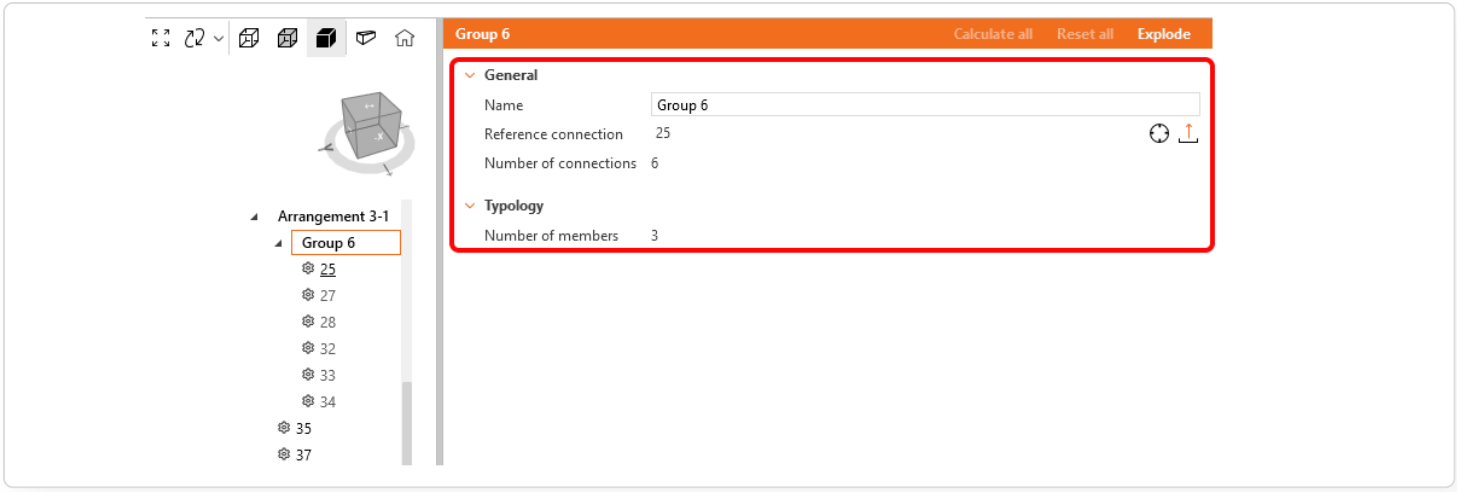
Typology and Arrangement

- Number of members
- Cross-section type of members



Design groups

- Name
- Reference connection
- Number of connections in a group
- Number of members in typology



Connection design with design groups

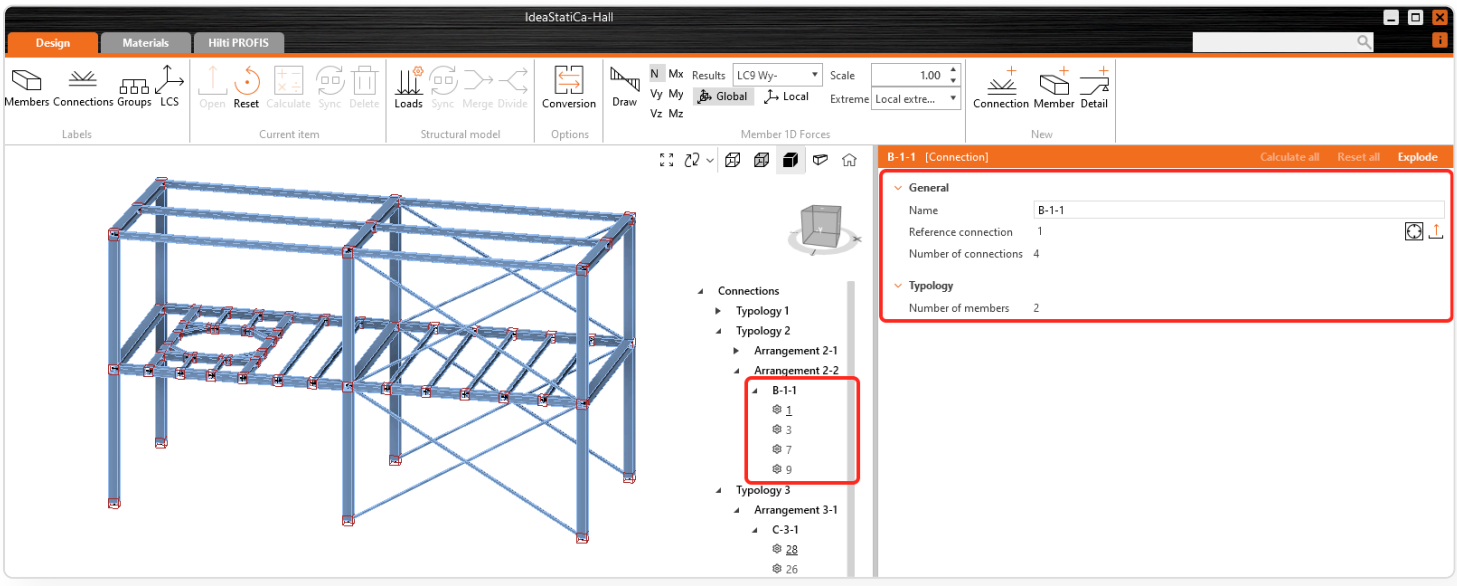
A **design group** allows a list of connections to be driven from one reference design, meaning the propagation (or templating) of a **reference connection** to all other connections in the group automatically. Groups are treated as specific objects with properties, and users can assign custom names to them. This saves users significant time when dealing with large sets of connections by automatically propagating design changes and operations to all **child connections** within the group.

User workflow

Standard group creation is as outlined below:

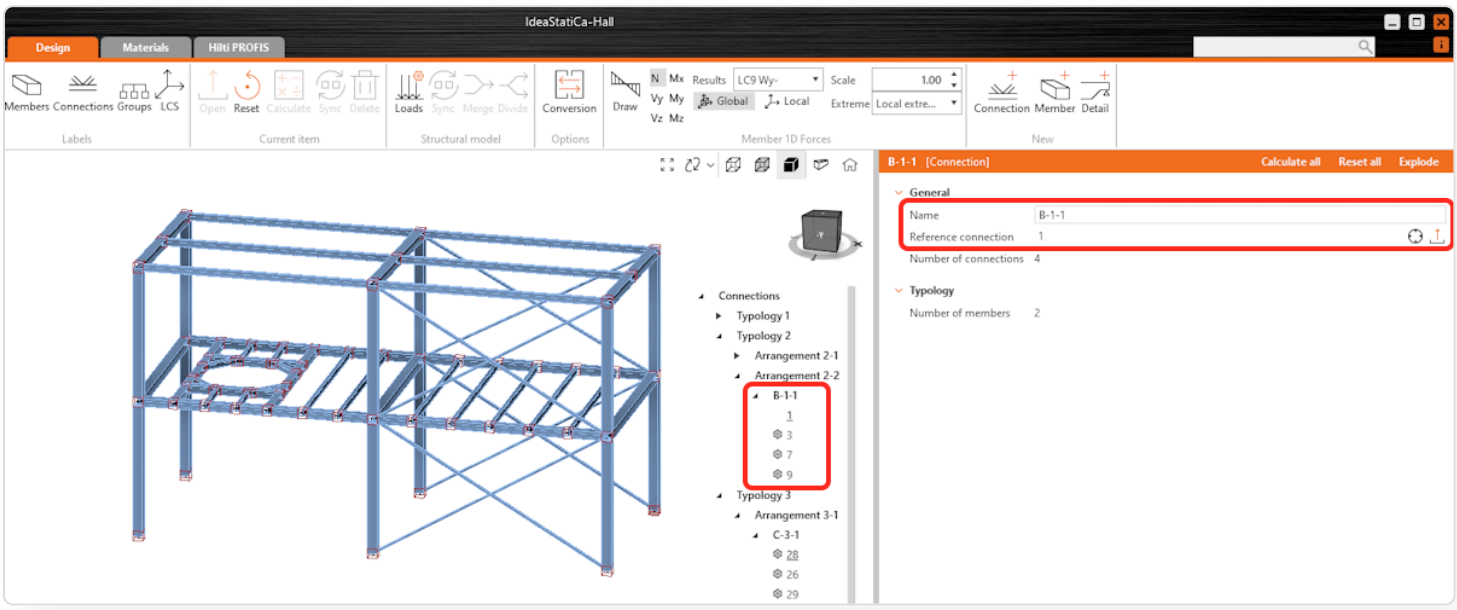
1. Group creation

- Users select one or more connections to create a group. Connections must share a minimum similarity rule: the same **typology** and **cross-section**.
- The first selected connection automatically becomes the **reference connection** (the reference connection is underlined in the tree), which drives the design for the group. All other connections are considered **child connections**.
- The design group name can be defined as desired.



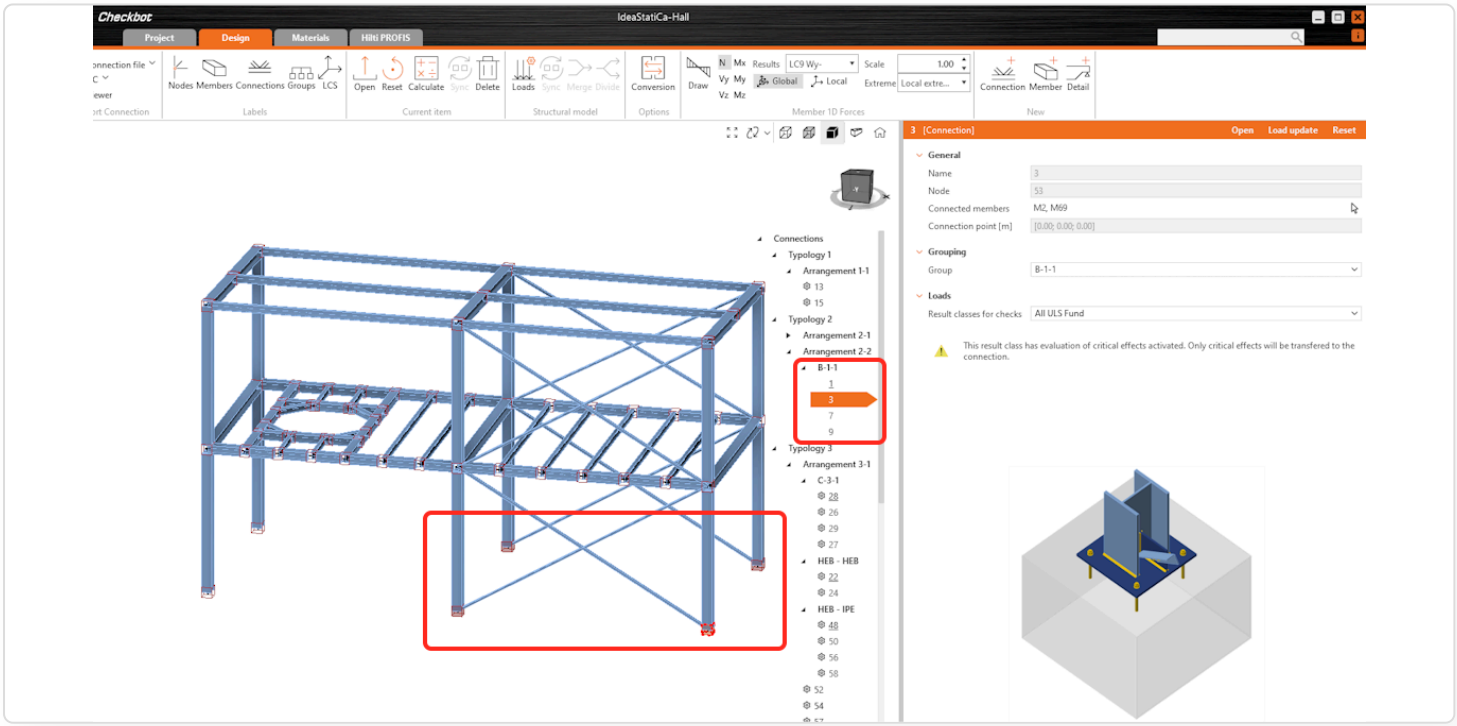
2. Design the reference connection

- The **reference connection** can either be designed manually or by applying a predefined template in the Connection app. To identify the reference connection, use the "sniper" symbol.
- All design operations and parameters from the reference connection are automatically propagated to the child connections, the section views included.



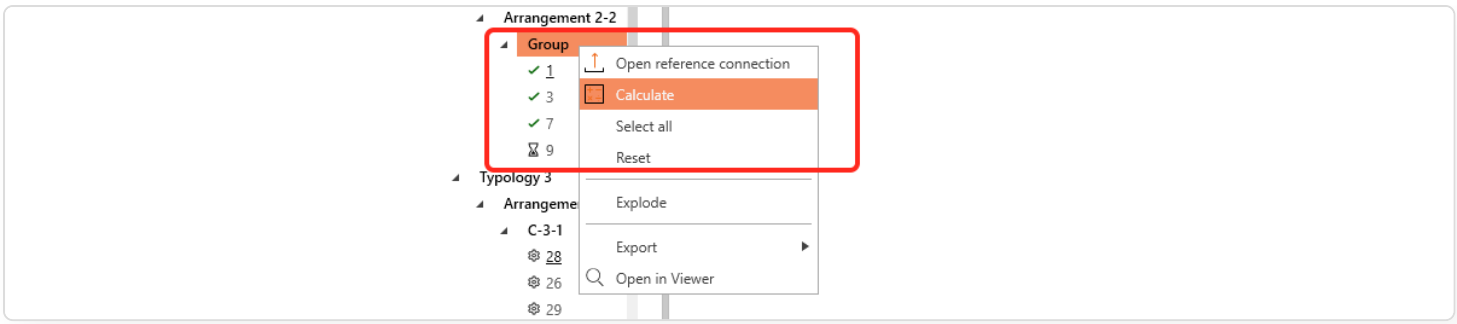
3. Automatic propagation to child connections

- Any changes made to the reference connection will automatically be applied to all **child connections** in the design group, ensuring uniformity.



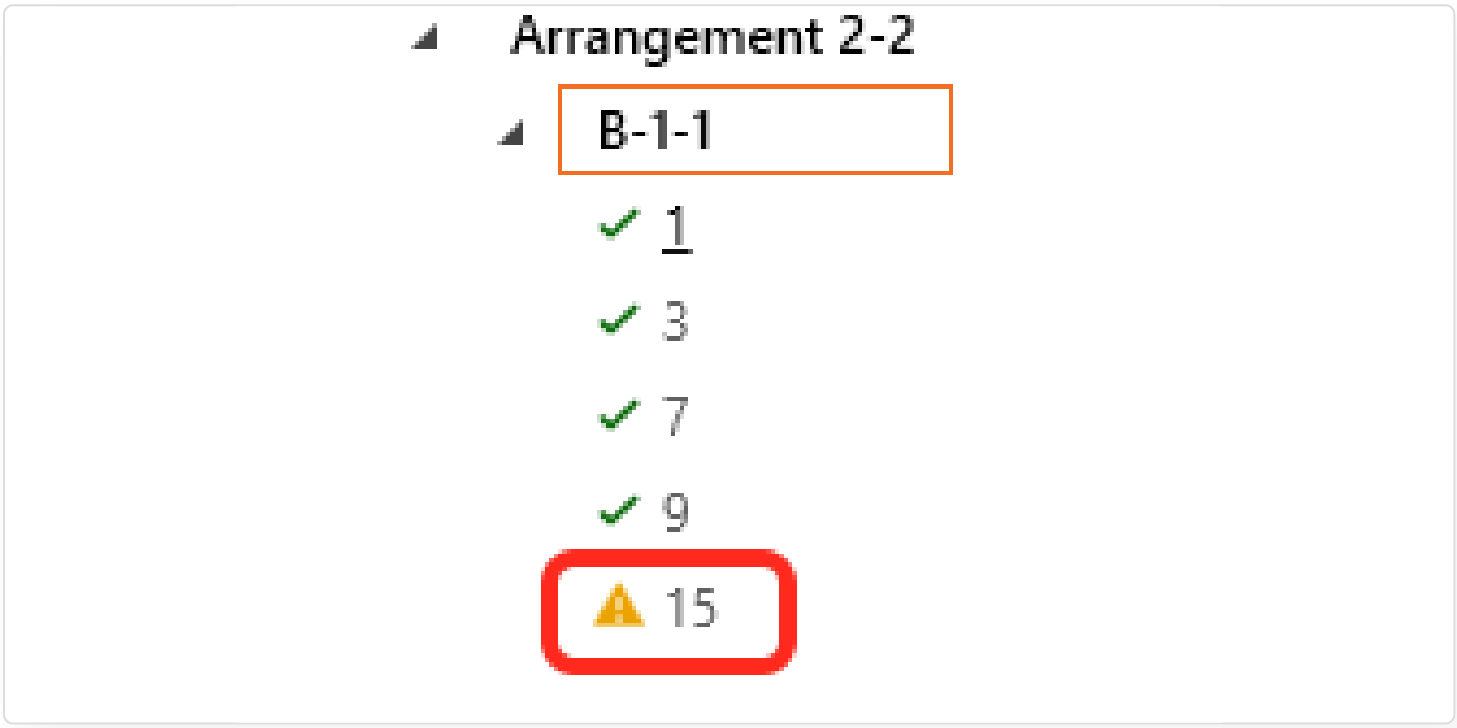
4. Group Calculation

- The connections within the **design group** are calculated collectively.



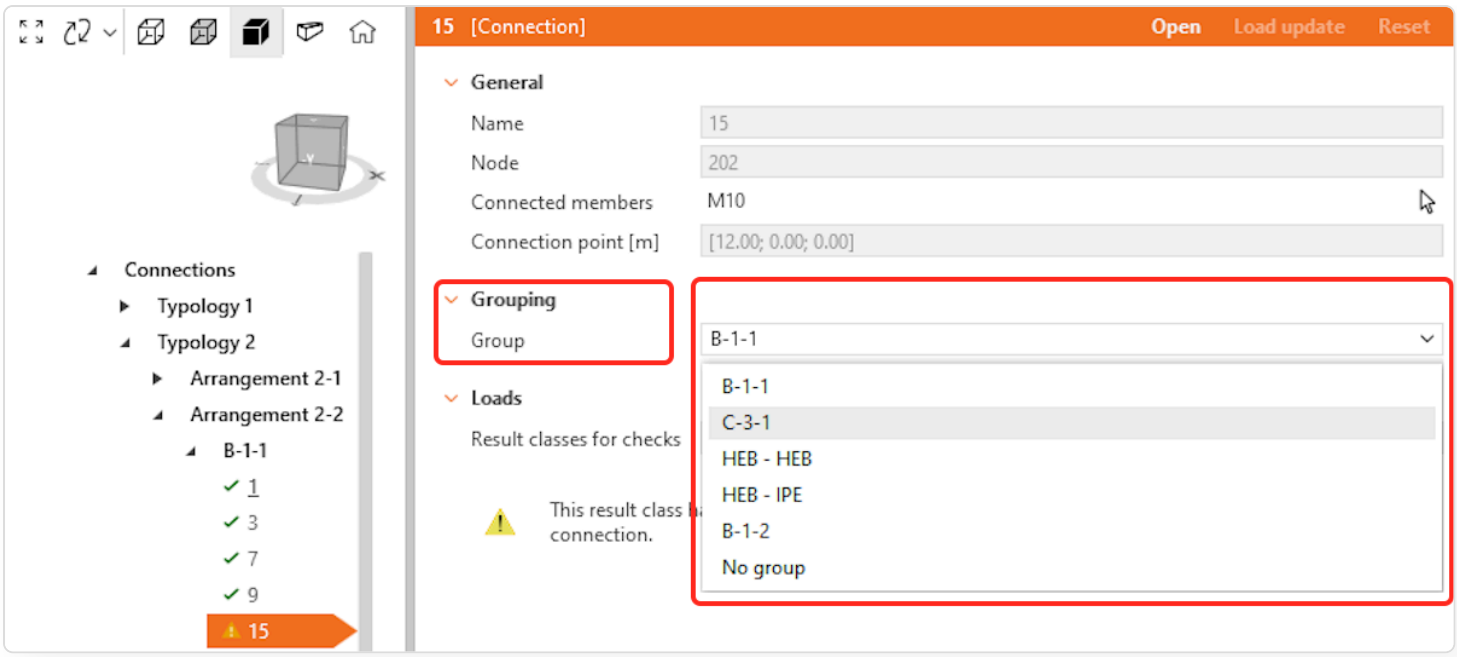
Synchronization and validation

- When syncing with or updating the structural model, all child connections are validated against the reference connection.
- If any child connection no longer meets the reference connection's parameters, it is flagged as **invalid** in the group.

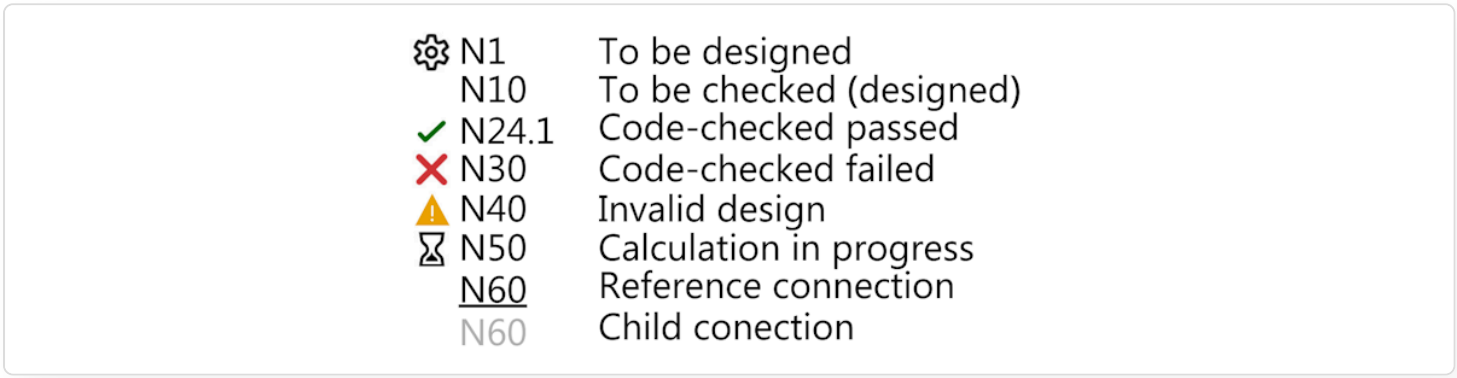


Managing design groups

- Users can add, remove, or move **child connections** between **design groups** as long as the connections still meet the group's validation rules (same typology and cross-section).
- A group can be exploded.
- **Child connections** can be removed from a group without exploding the group.
- Invalid **child connections** can be removed or corrected, and groups can be reconfigured as necessary.
- **Reference connections** cannot move between groups, the only way to remove or change the reference connection is to explode the group and re-create it. **When a connection is marked as a reference, it cannot be deleted from the structural model**.



The key of the icons in the connection tree



Known limitations

- The model type of members in the reference connection is not propagated to Child connections.
- The position of forces of members in the reference connection is not propagated to Child connections.

Released in version 24.1

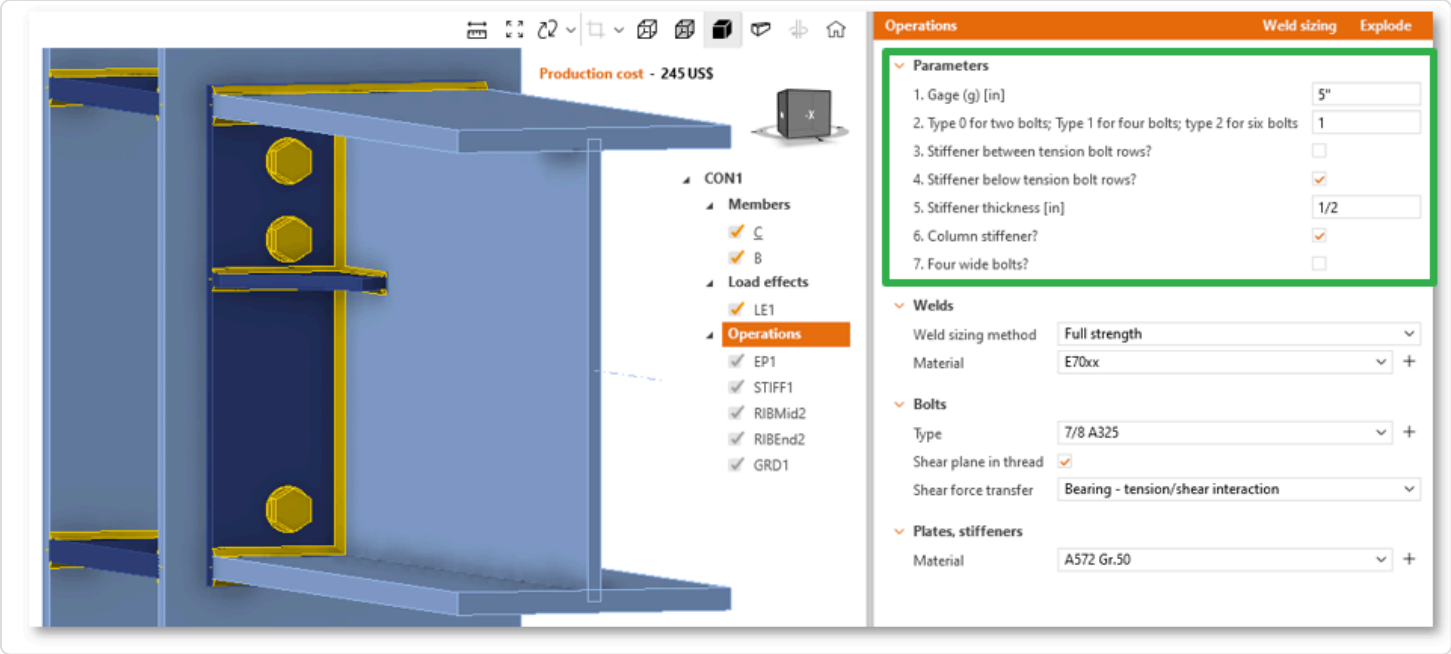
Parameters are useful for everyone

Parametric templates are designed to help simplify workflows, making template creation and use easy. Default values can be set for visible parameters in the Simple UI, guiding users with flexible options. Expanded functions simplify load effects and unit input management.

Parametric templates for any user

We advocate the creation and usage of parametric templates for everyone, whether for very experienced users or those who are just starting out. Let's divide the users of IDEA StatiCa Connection into roles and explain the differences between them and what options they have.

- **Creator**: The creator is someone who knows the key principles and rules of modeling in IDEA StatiCa Connection and is capable of correctly modeling a connection. The creator wants to parameterize a standard connection and create a template for others.
- **Basic user:** A basic user could be a junior engineer or someone who doesn't use IDEA StatiCa very often, or someone who simply hasn't developed sufficient skills yet. They mainly want to use the parametric templates and understand what's happening rather than create them themselves.



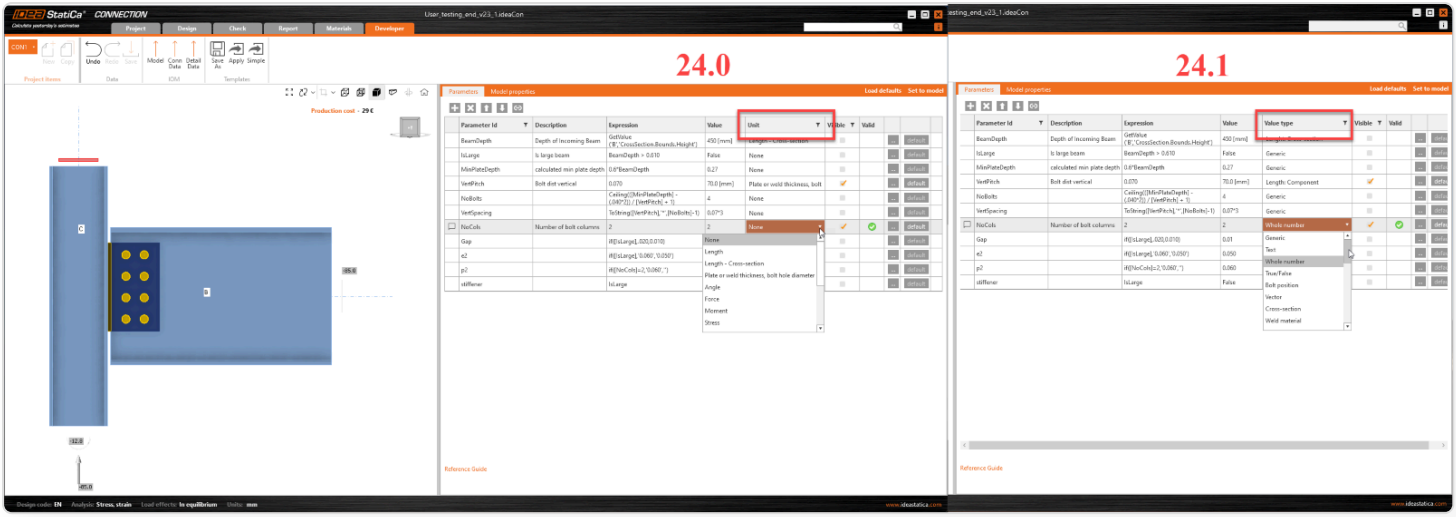
Both users will notice in the UI and in the UX that they offer intuitive functionality and simplified logic.

Key UI features

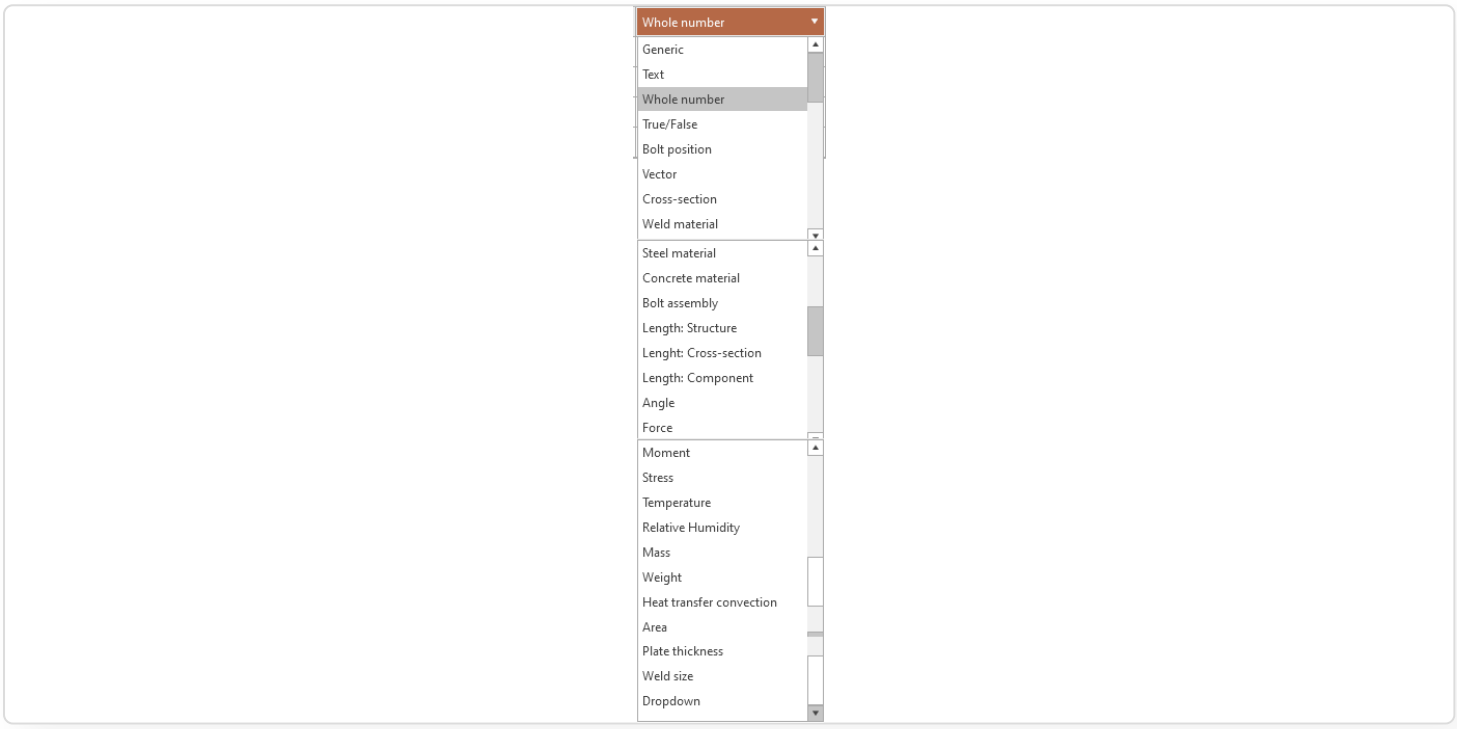
- Standard UI components for parameters
- Access to project material and cross-section through the design UI tab
- Drop-down selection
- Conversion between property types between different applications such as Tekla or Revit (the ones using specific property types)

Standard UI components for parameters

Specific unit values are included as value types, and units are linked to these value types according to the unit settings. The generic value type allows it to hold any type of data represented in string format. See the picture below for a easier understanding:



By enabling value types, we can select specific **UI components** for each value type. See all value types below:

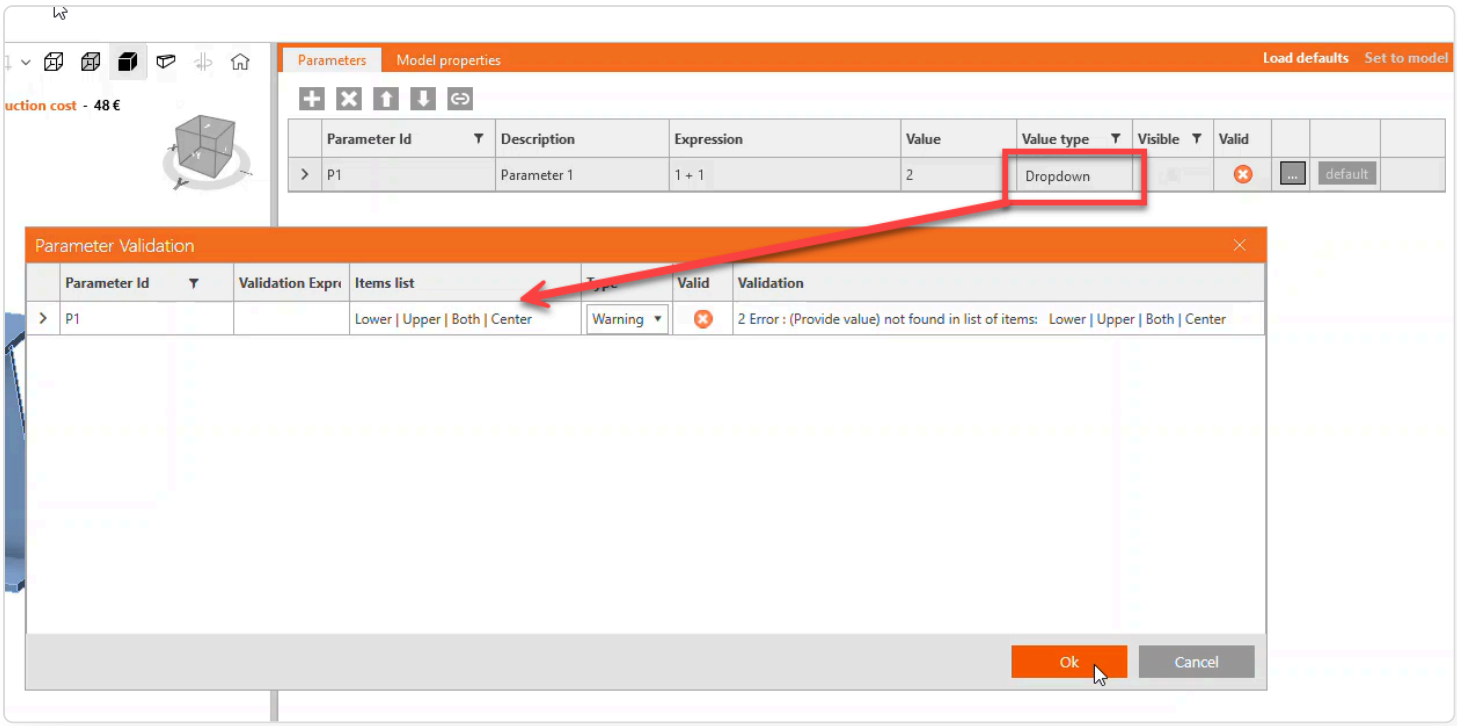


Only some value types have been added to the simple UI. Others can be added as required.

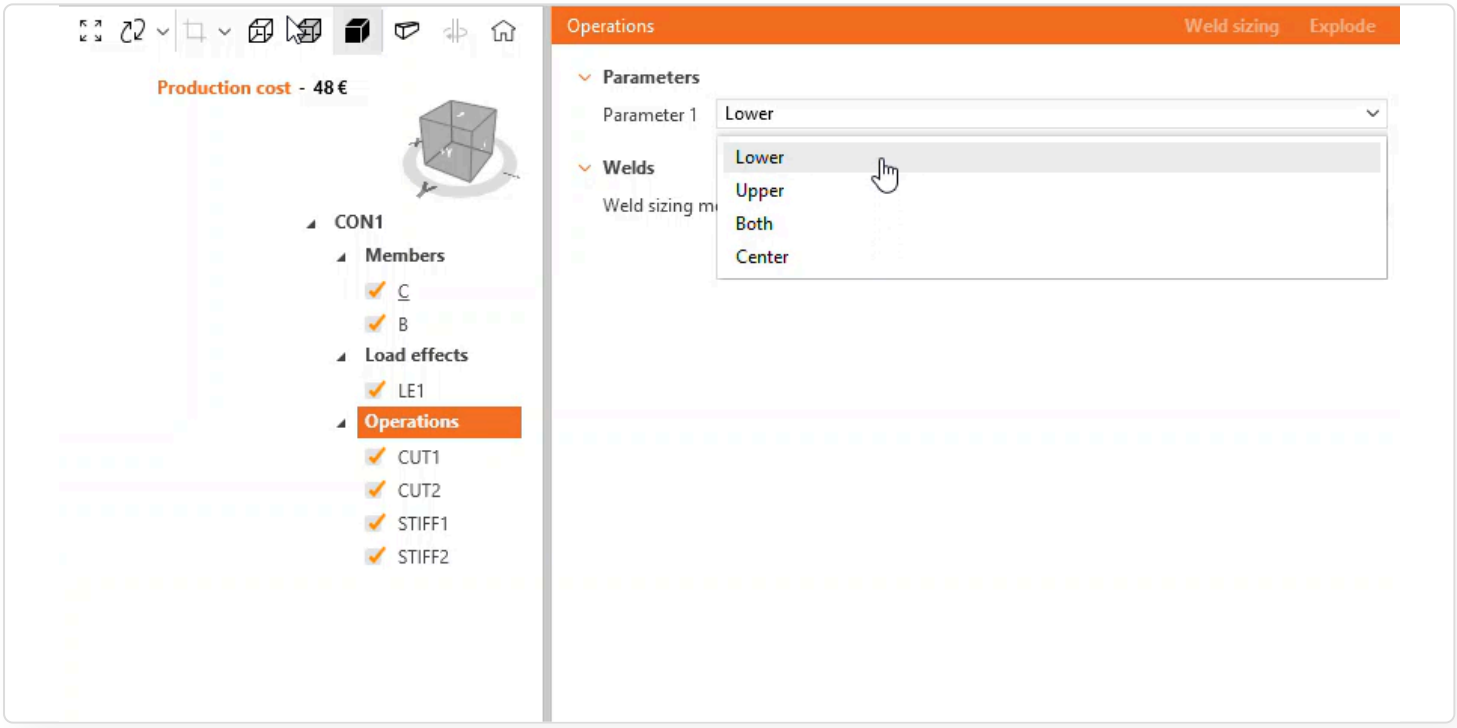
Drop-down selection

The inclusion of a dropdown selector is reflected in the work of both roles. The creator has easier input in the developer tab, while the basic user benefits from easier work in the design tab.

- **Drop-down value type (Developer tab)**



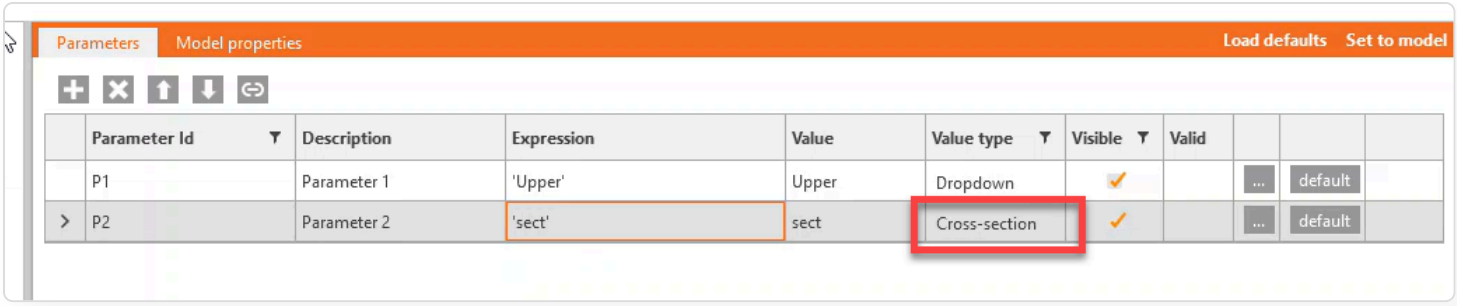
- **Drop-down value type (Design tab)**



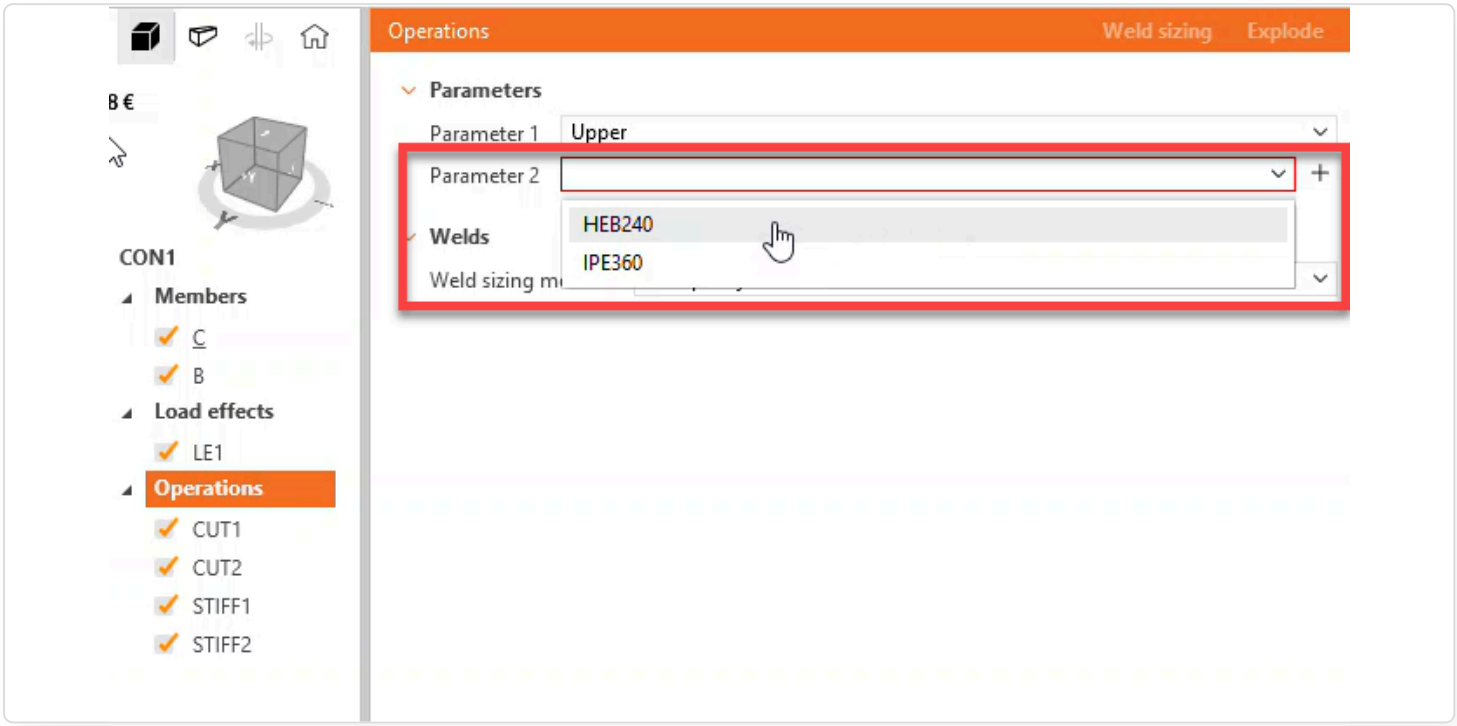
Access to project material and cross-section

Access to cross-sections and materials works very similarly, and this will again be reflected during both the creation and usage processes.

- **Cross-section value type (Developer tab)**

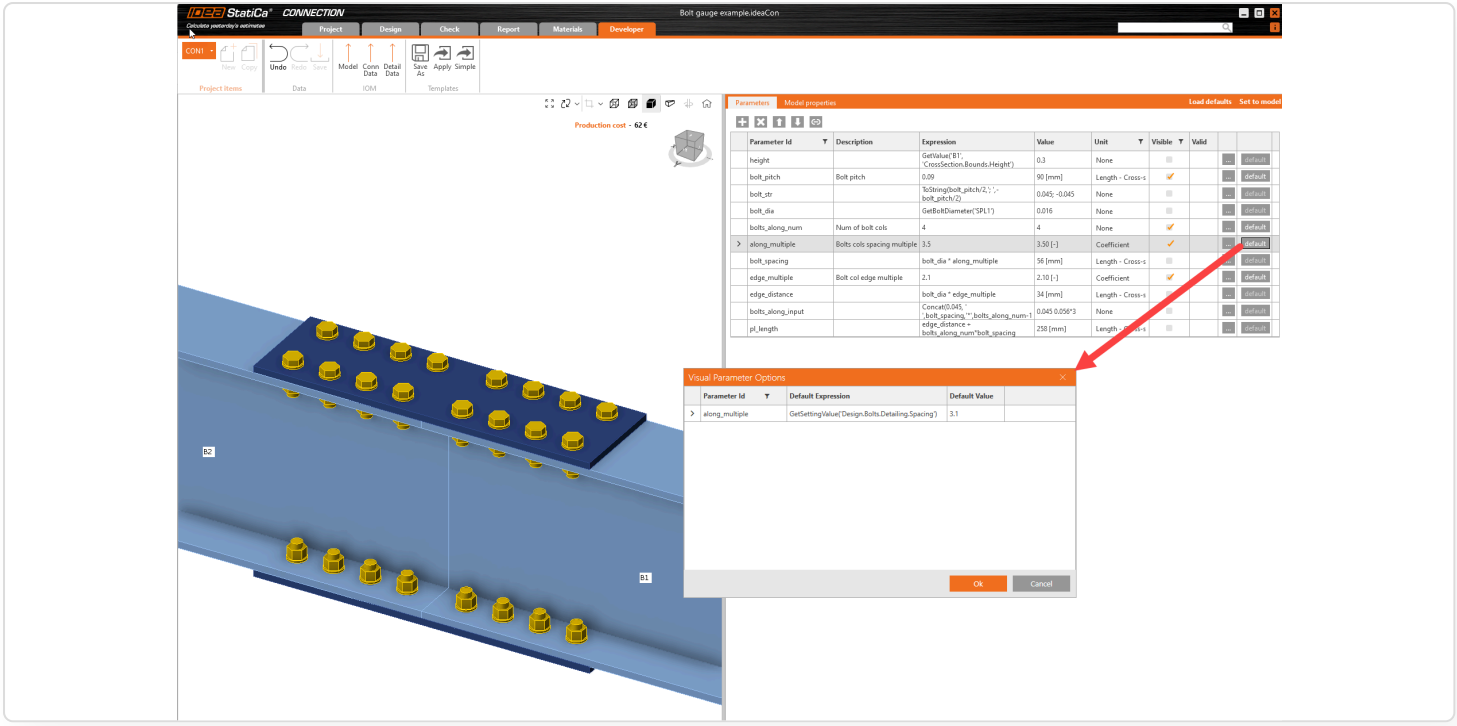


- **Cross-section value type (Design tab)**



Default values for parameters

Parametric template creators can assign default value expressions to any visible parameter within the simple operations interface. When applied, these defaults are initialized in the Simple UI, guiding junior users with pre-selected values that remain adjustable. Defaults can be set from design settings or calculated from other parameter inputs like beam heights.



Load retrieval, string input, and units

Load retrieval, string input, and unit functions are clearly defined to provide precision and usability.

Load retrieval

With the function **GetLoadEffects**, users can retrieve load effects to adjust plate thickness and other connection parameters.

- Look for GetLoadEffectEnvelope(Member, Action, Envelope, End) in the documentation.

Strings

The function of **Strings** can be used for creating strings for defining bolt spacing inputs, points, and vectors.

- Look for the String function in the documentation.

Units

The inputs for Length, Area, Force, Stress, Moment, Temperature, Angle, and Time can be inserted in various units, (other than basic SI units), e.g., the length can be defined in 'm', 'dm', 'cm', 'mm'', in', and 'ft' units.

- Look for Unit import and conversion functions in the documentation.

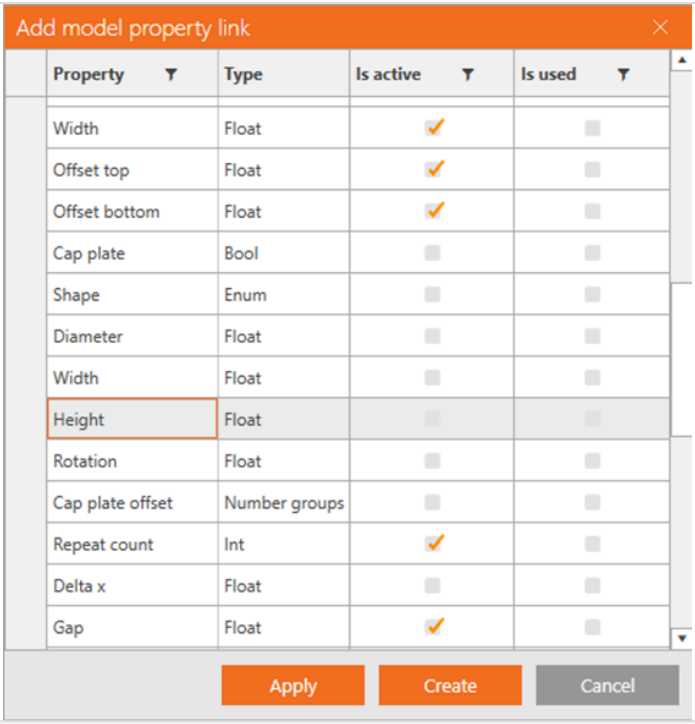
Bolt diameter retrieval

Bolt diameters for specific bolt inputs in specified operations can be defined using **GetBoltDiameter**.

- Look for GetBoltDiameter('operation_name', index) in the documentation.

Model property link

Parameters can be linked to multiple model properties by extending the “Add model property link” dialog with **Apply** and **OK**. Where Apply allows you to link more properties at once without the need to close the dialog.



HILTI PROFIS plugin in Checkbot

The collaboration between IDEA StatiCa and Hilti includes a direct BIM link from Checkbot to Hilti PROFIS Engineering Suite (PE). This enables data transfer from supported third-party FEA and CAD software to PE, improving efficiency and ensuring accurate design coordination across platforms.

What is HILTI PROFIS Engineering Suite?

Hilti PROFIS Engineering Suite is a cloud-based software for structural engineers that specializes in the design and analysis of anchor systems, base plates, and steel connections. It supports international design standards like ACI and Eurocode, and integrates with Hilti's hardware to provide optimized anchor solutions.

FEA to Checkbot to Profis workflow

Perform global analysis in your FEA

Export connection to IDEA StatiCa Checkbot

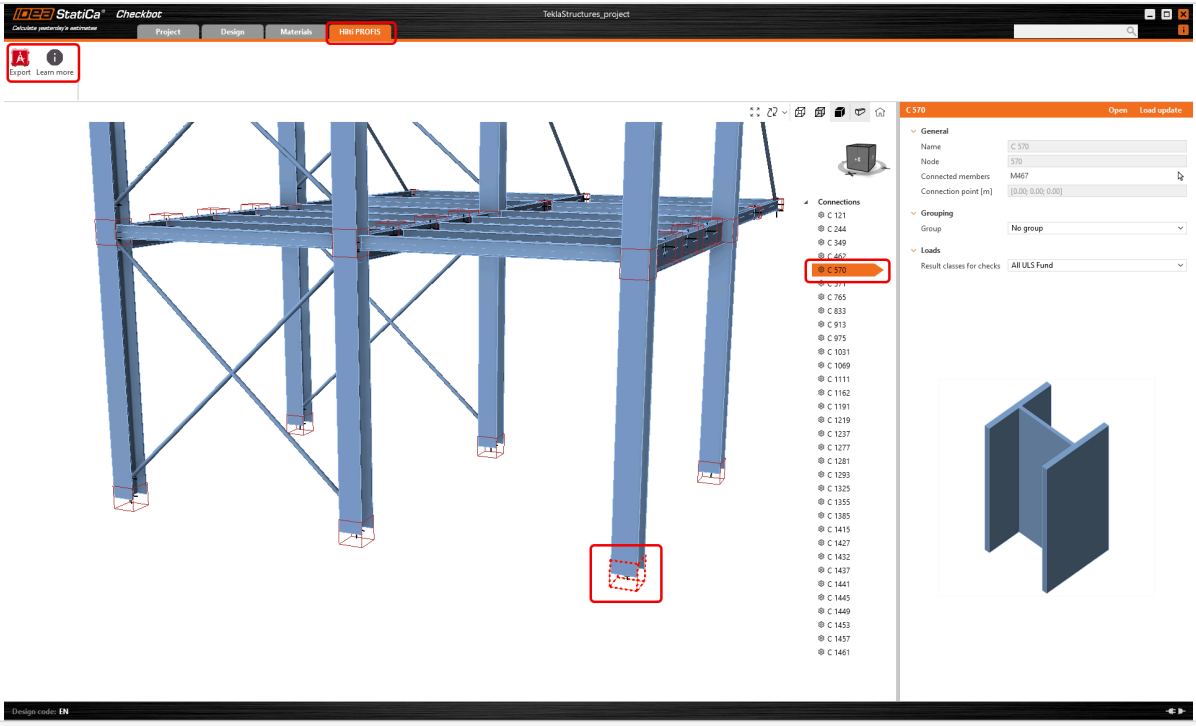
Design and Code-check your connection in PROFIS

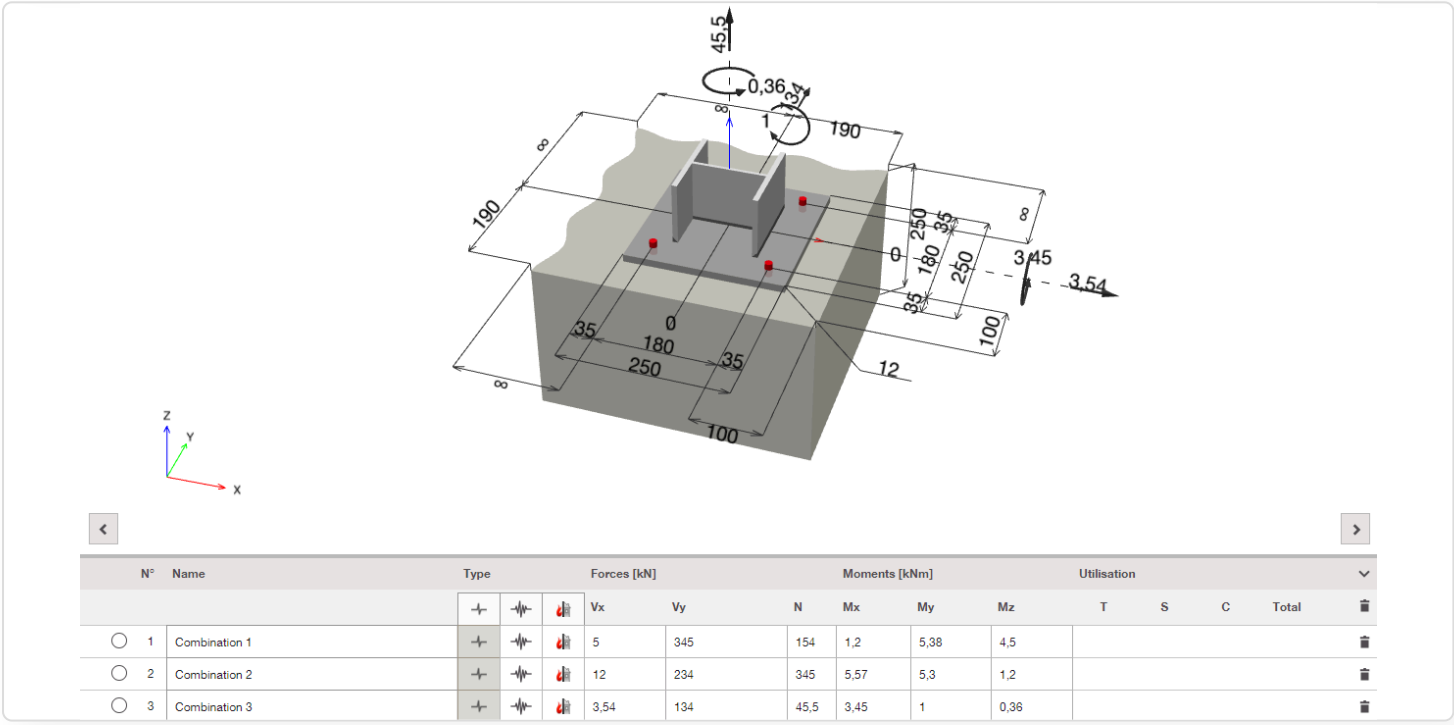
How to use the plugin

To import data from any global analysis software into Hilti PE, the information is first brought into Checkbot via any BIM link. A "HILTI PROFIS" tab in Checkbot facilitates the export to the Hilti process. By selecting a node with one anchored member, users can export the data directly to Hilti PE using the **Export** button, ensuring accurate transfer of relevant structural data for further analysis.

The **Export** button in Checkbot launches Hilti PE and, after logging in, automatically creates a new project with the same name as the Checkbot project. Within this project, a new design is generated.

This entire workflow is available even with a IDEA StatiCa Basic license, meaning that it is available for free. How to use the plugin is described step-by-step in the following article, which is also accessible from the **Learn more** button in Checkbot.





N°	Name	Type	Forces [kN]			Moments [kNm]			Utilisation			
			Vx	Vy	N	Mx	My	Mz	T	S	C	Total
1	Combination 1		5	345	154	1,2	5,38	4,5				
2	Combination 2		12	234	345	5,57	5,3	1,2				
3	Combination 3		3,54	134	45,5	3,45	1	0,36				

Data imported into PROFIS

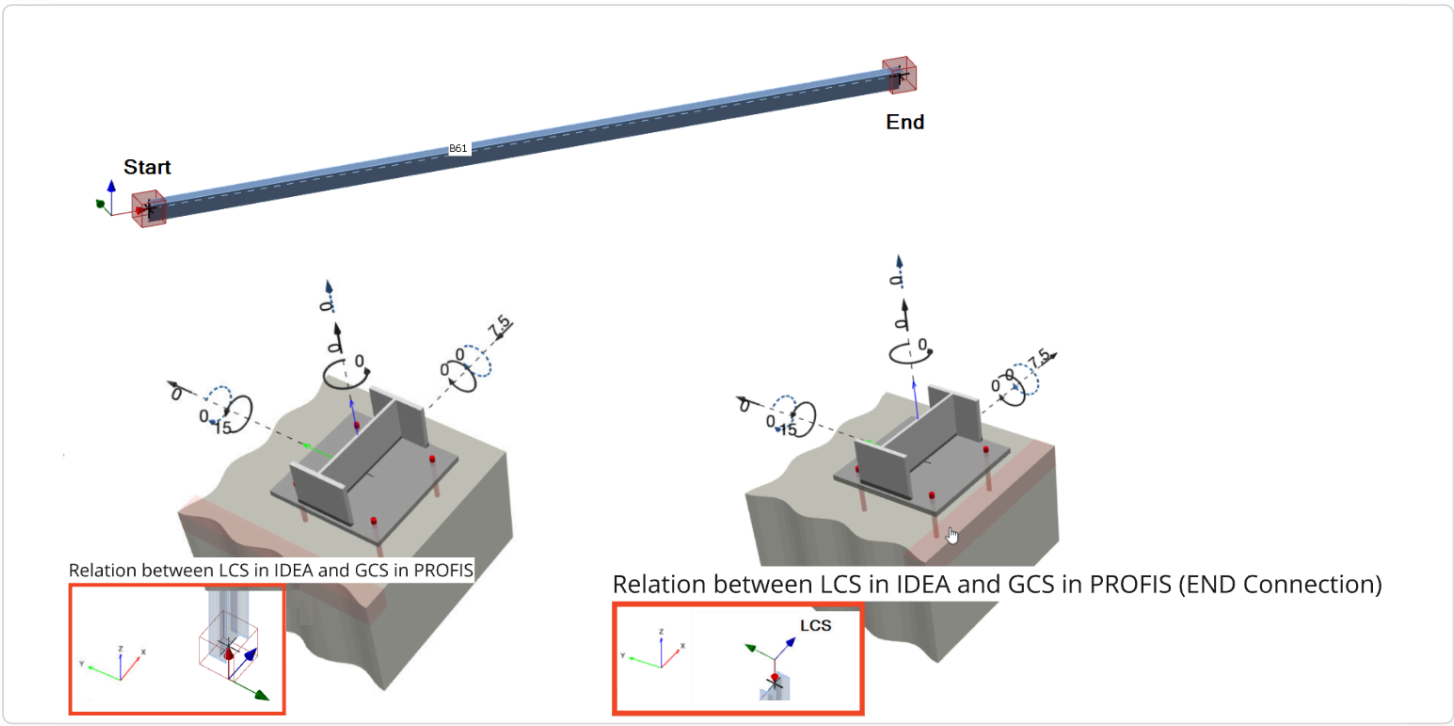
- International design standard (code)
- Profile type and material
- Internal forces from load cases and load combinations

Known limitations

- Connection design is not processed by Hilti PE (anchor layout, baseplate geometry, stiffeners, welds)
- Only connections with 1 member (simple footings)
  - Only steel-to-concrete anchoring

- No complex design (footing with braces)

- Hilti PE does not support "mirroring" of profiles at the moment. In the picture below, highlighted cases show the profile position in PROFIS does not match with the IDEA StatiCa profile position in relation to Connection at the START or END (LCS of the member in IDEA StatiCa).



Checkbot ready for big projects

The IDEA StatiCa Checkbot app is built to import design data from different sources, process it, and export it for other purposes. Its goal requires the ability to process everything in a reasonable time. The size of imported models varies from simple structures to complex ones.
The ability to link IDEA StatiCa apps to dozens of different tools brings the need to create different approaches for specific vendors. The BIM link data transfer is polished and simplified to make the processes as smooth and quick as possible.

Thanks to this, the time needed for getting the data from different sources is short. Here are some comparisons for different metrics.

Faster imports for single connection from FEA links

We improved processes during the import of a single connection and member in FEA imports by reducing unnecessary API calls we use during import. This results in significantly faster import times for specific BIM links.

FEA link	Nodes in Model	Before	After	Faster	%
Robot	2500	16:09	0:27	15:42	97%
Axis VM	1320	0:36	0:03	0:33	92%
RFEM 6	800	0:45	0:07	0:38	84%
Staad.Pro	1000	0:06	0:04	0:02	33%
SAP2000	1300	0:11	0:09	0:02	18%
ETABS	300	0:11	0:10	0:01	9%
RFEM 5	3200	0:06	0:06	0:00	0%
SCIA Engineer	350	0:40	0:40	0:00	0%

Faster opening of the existing projects

Also, the opening times for Checkbot's already existing projects has been improved by approximately 50% compared to version 23.1.5.

Checkbot	No. Connections	Before	After	Faster
Project 1	70	0:17	0:04	76%
Project 2	400	0:37	0:04	89%

Faster FEA imports by 60%

The import speed has been improved by delaying connection creation (.ideacon) till the point where they are really needed. Import speedup varies from FEA link to FEA link.

During import, we skipped the part where all imported connections were written down to the hard drive in the Connections folder in the Checkbot project since it took the majority of import time. Starting from version 24.0.2 the Ideacon files are created at the moment when the user wants to interact with them:

- Design connections in IDEA StatiCa Connection
- Export connections to Viewer or to Ideaconn files from Checkbot
- Design connections in the Member application

FEA link	Nodes in model	Before (23.1.5)	After (24.1.0)	Faster
Robot	300	0:05:20	0:03:58	25.6%
Axis VM	300	0:03:22	0:01:32	54.5%
RFEM6	300	1:40:00	0:40:00	60.0%
RFEM 5	350	0:07:25	0:03:15	56.2%
Staad.Pro	300	0:02:01	0:00:42	65.3%
SAP2000	300	0:03:38	0:01:32	57.8%
ETABS	450	0:05:32	0:02:10	60.8%
SCIA Engineer	300	0:04:04	0:01:33	61.9%

Generating a subset of Ideacons only when necessary significantly reduces the time involved compared to creating hundreds of files during the import process. This approach accelerates, especially, the import of large models, allowing users to experience quicker performance and a more efficient workflow within IDEA StatiCa Checkbot.

Exporting an IFC file from IDEA StatiCa

The Industry Foundation Classes (IFC) format is an open data, vendor-neutral format that enables the sharing of CAD data. IDEA StatiCa provides users the ability to share a connection design in a format readable by other BIM software with the touch of one button.

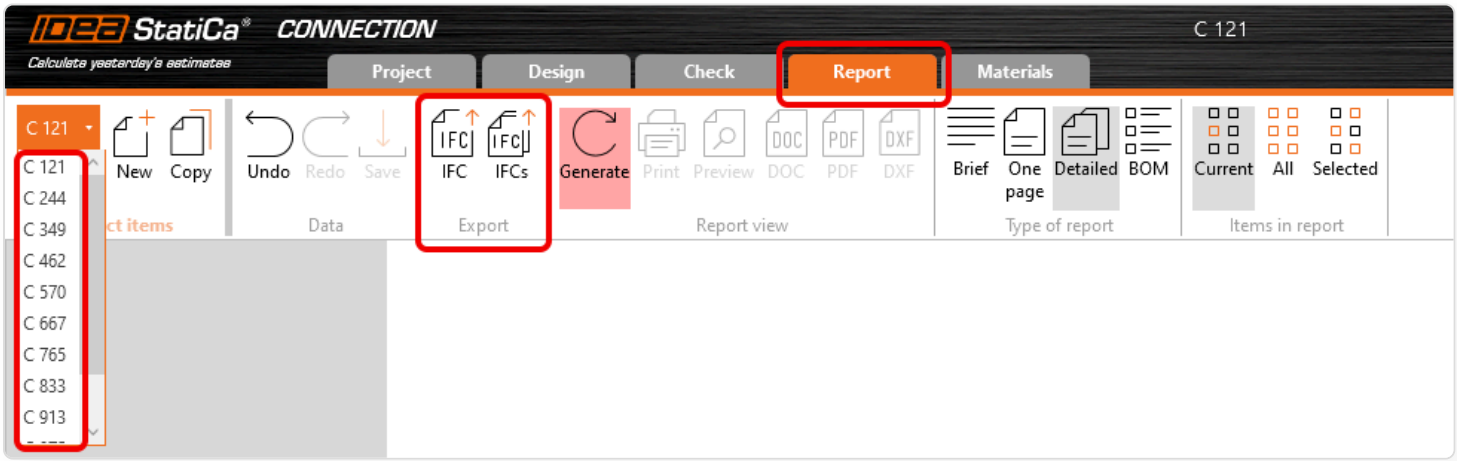
One IFC model is worth more than a dozen drawings

As a detailer, modeling a complex, or even standard, connection design according to 2D drawings provided by a structural engineer can get complicated and create room for error. Therefore, it is important to share as much information between the professions as possible. So, why not use a format that is recognized by every BIM software tool of the industry? An IFC provides you with a 3D representation of the connection, which can be linked to Tekla, Revit, etc., and all IFC viewers.

How to export your connection design into IFC format from IDEA StatiCa Connection and Viewer

All it takes is one click!

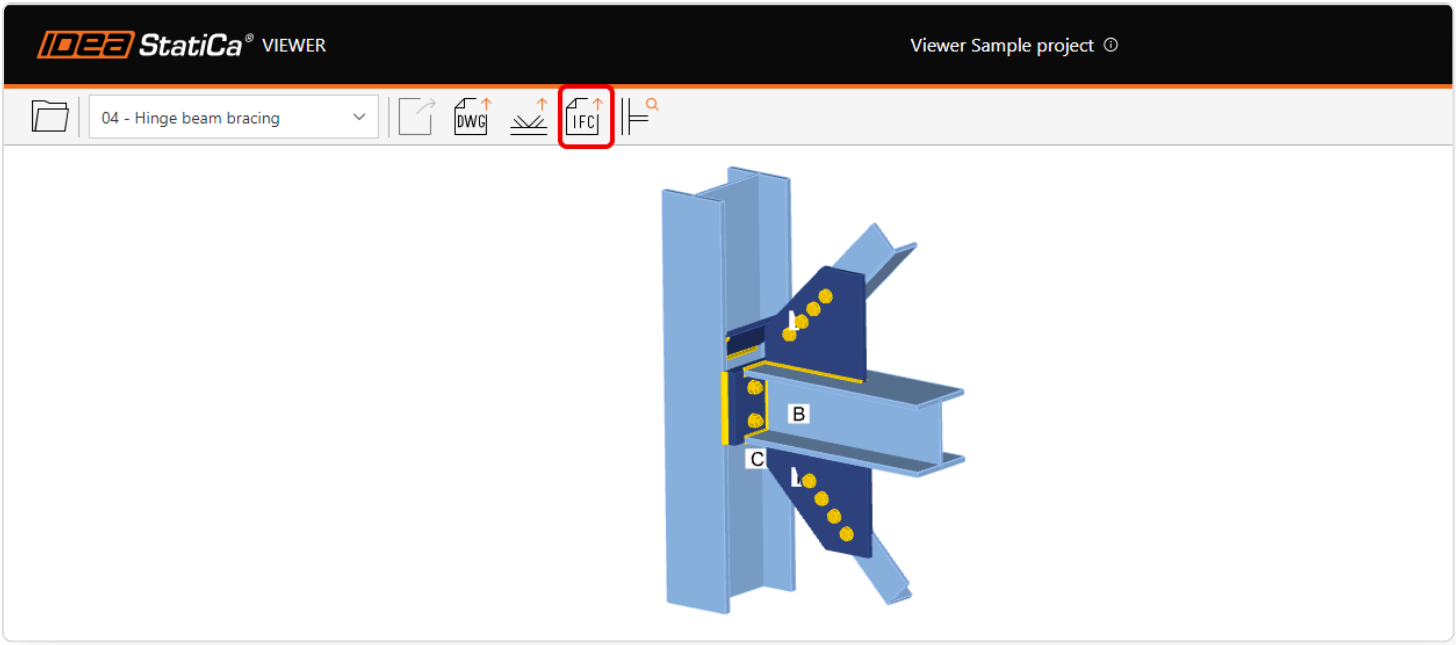
- Design your steel joint in IDEA StatiCa Connection and click the **IFC** export button in the **Design** tab or **Report** tab.

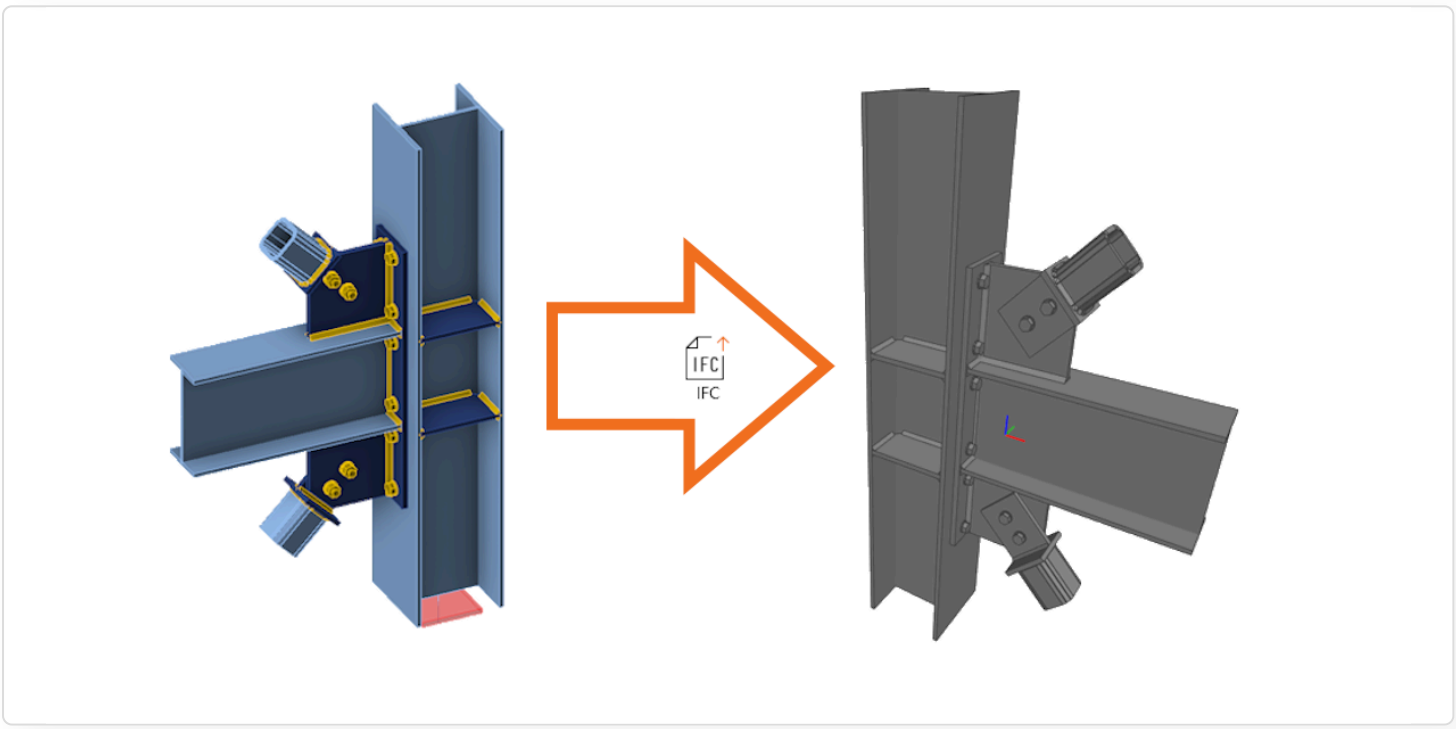


- A bulk export option is included in the "Report" tab. This feature allows users to export multiple connections included in one ideaconn file to IFC files in a user-specified folder.

You can export connections into IFC even if you don't use the desktop application!

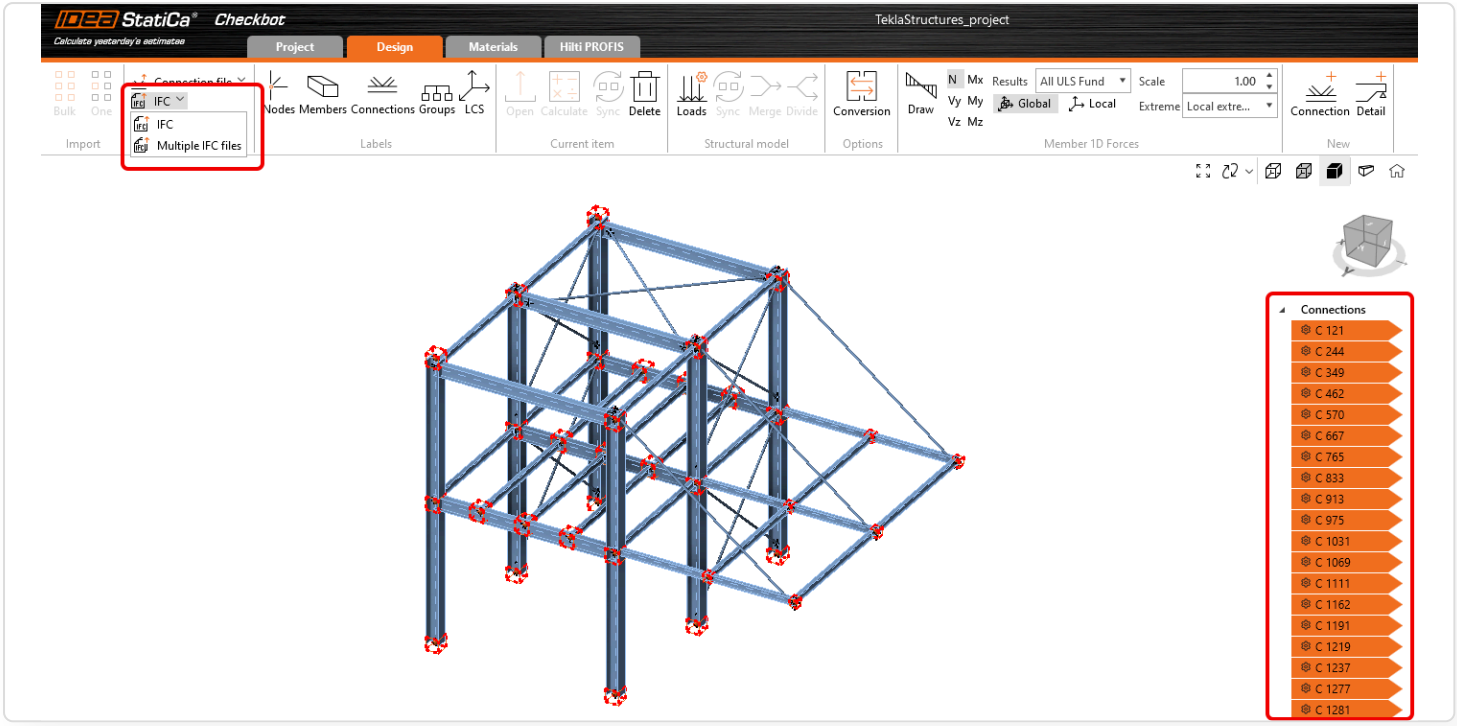
- Just upload the steel joint to our web application IDEA Statica Viewer.
- Click on the icon of IFC file and IFC of your joint will be automatically downloaded.

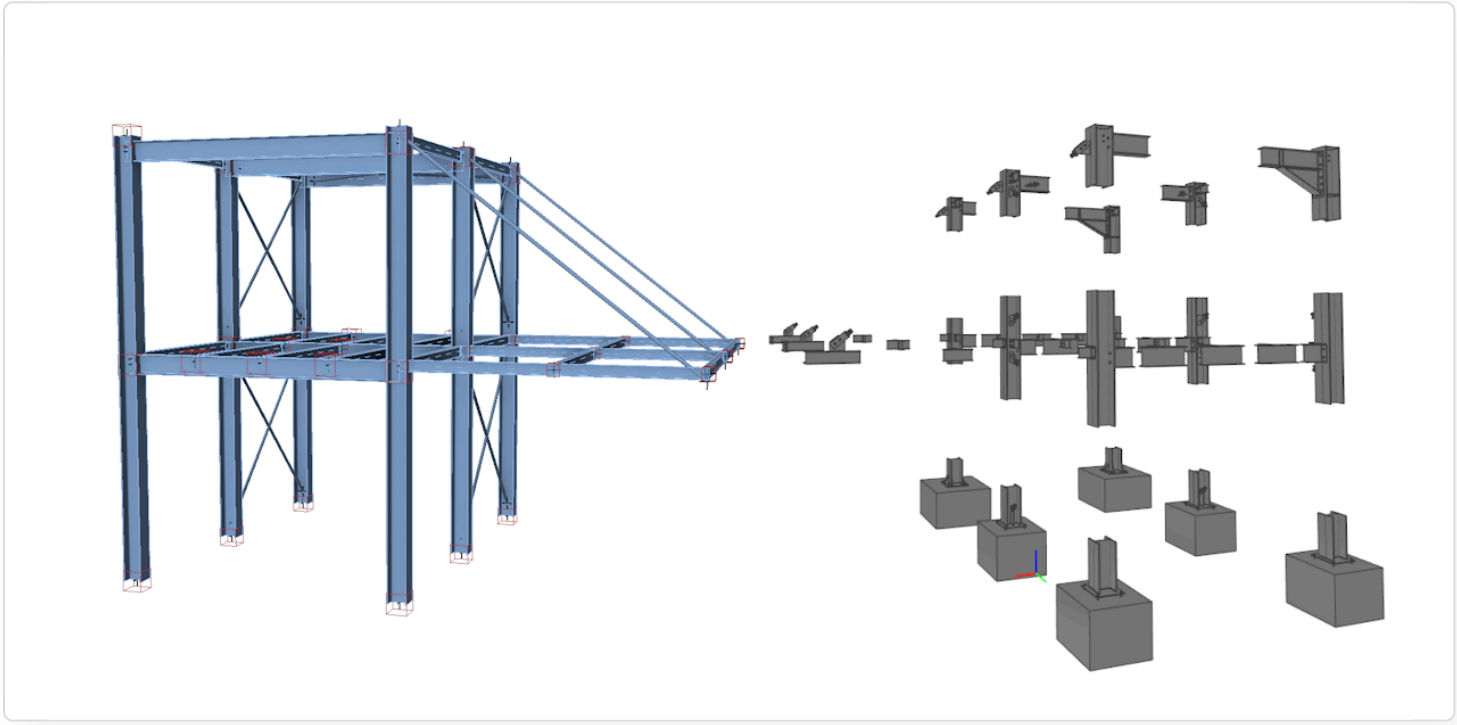




How to export your connection design into an IFC from IDEA StatiCa Checkbot

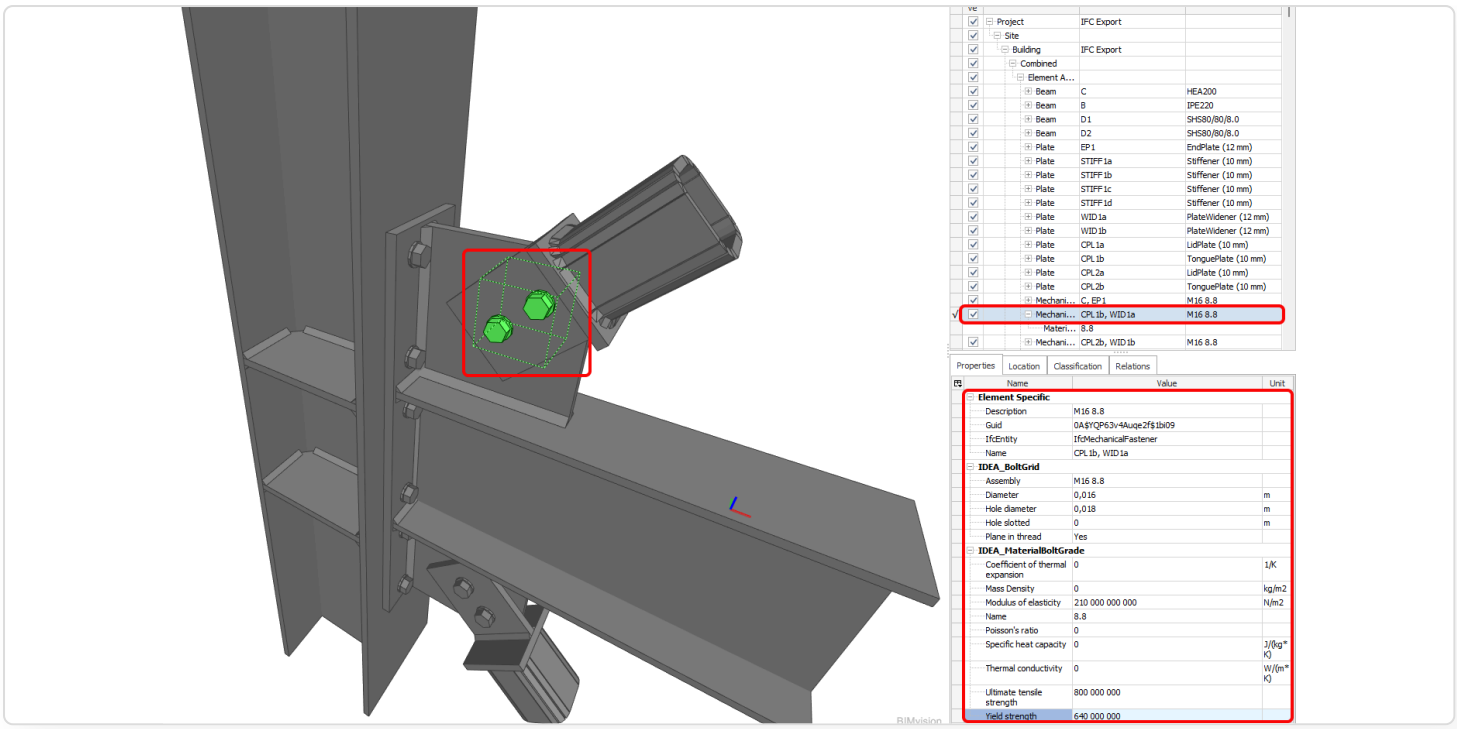
- Design your connections in IDEA StatiCa Connection and export all of them from Checkbot.
- Export the selected connections into a single file with global coordinates or each connection separately into individual IFC files.
- The Export button is accessible both from the ribbon in the **Design** tab and by right-clicking selected connections in the **tree**.



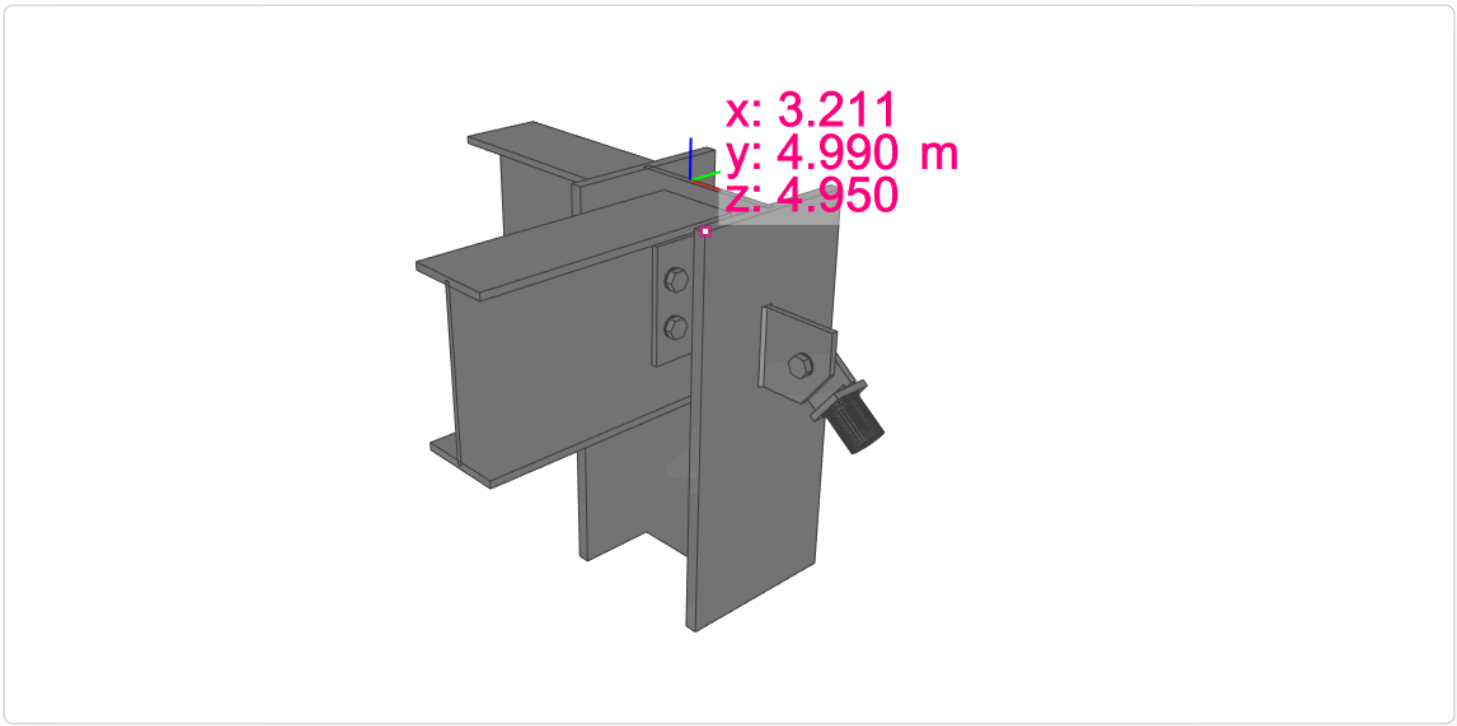


The IFC file includes:

- A geometrically accurate model of the connection, including bolts and welds.
- Basic information about cross-sections, bolts, welds, and materials.
- The model defined as an IFC2x3 coordination view

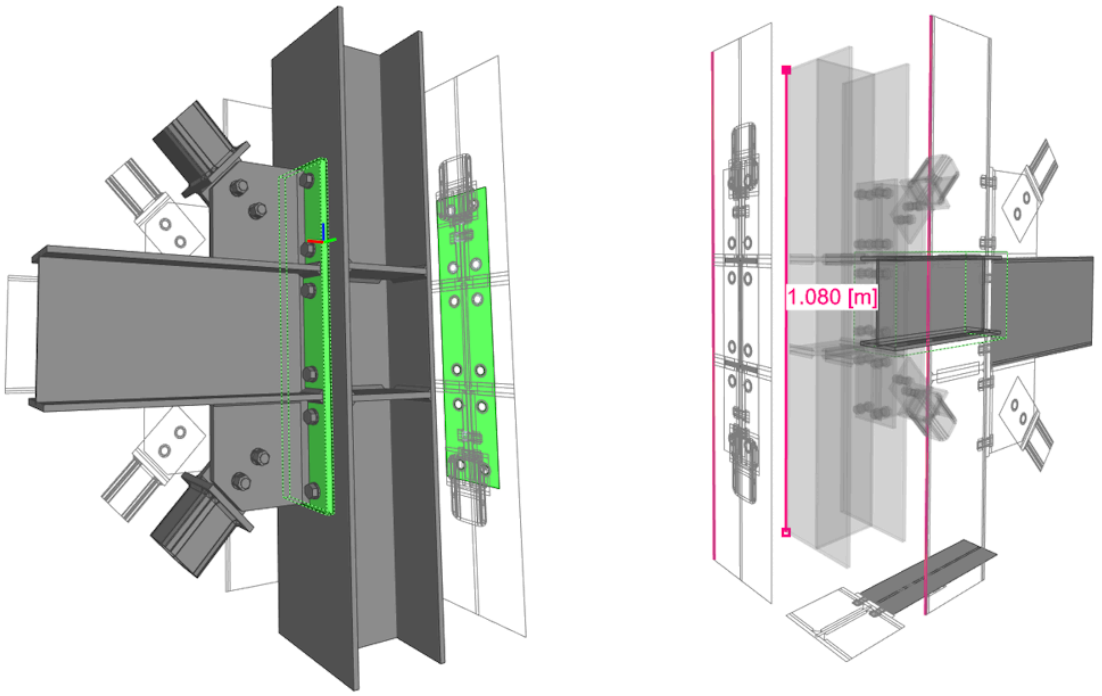


- When generating an IFC file from a connection created or exported through Checkbot, the export contains the global coordinates of the Connection Point – the real location of a connection in a project.



A visual representation of a connection

The IFC file can be imported to BIM/CAD tools (such as Tekla Structures, Autodesk Revit, etc.) as a visual representation of the connection model. This provides the detailer with a better understanding of the model. Opening the model in any IFC viewer software brings you a wholesome view and the possibility to create additional section views, cuts, and measurements.

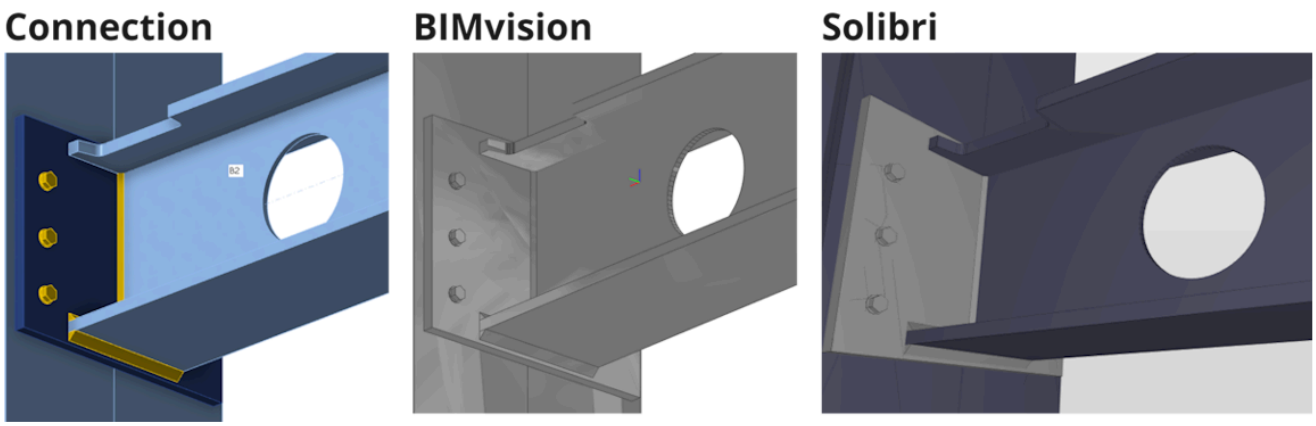


IFC export improvements (since version 23.1.1)

Here you can find listed improvements and details about the IFC export in 23.1.1 patch:

Supported and verified Viewers:

- BIMvision 2.27.3 and later
- Solibri 9.13.5.12 and later
- Autodesk Viewer (https://viewer.autodesk.com/)



Information about Cross sections and Materials

- Information about cross section and materials are contained in custom properties ("IDEA_CrossSection", "IDEA_MaterialSteel*" property sets).

Properties | Location | Classification | Relations

Name	Value	Unit
Element Specific		
Description	HEB400	
Guid	3GqU2LTnv6L9pdxvAokcS7	
IfcEntity	IfcBeam	
Name	B2	
IDEA_CrossSection		
Mprl shape name	HEB400	
Name	HEB400	
Type	RolledI	
IDEA_MaterialSteelECEN		
Coefficient of thermal expansion	0.000012	1/K
Design code	ECEN	
Mass Density	7 850	kg/m2
Modulus of elasticity	210 000 000 000	N/m2
Name	S 275	
Over strength coefficient for fu	1.25	
Over strength coefficient for fy	1.25	
Poisson's ratio	0.3	
Shear modulus	80 769 230 769.2308	N/m2
Specific heat capacity	490	J/(kg* K)
Thermal conductivity	50.2	W/(m* K)
Ultimate strength(fu,40)	430 000 000	Pa
Ultimate strength(fu40)	410 000 000	Pa
Yield strength(fy,40)	275 000 000	Pa
Yield strength(fy40)	255 000 000	Pa

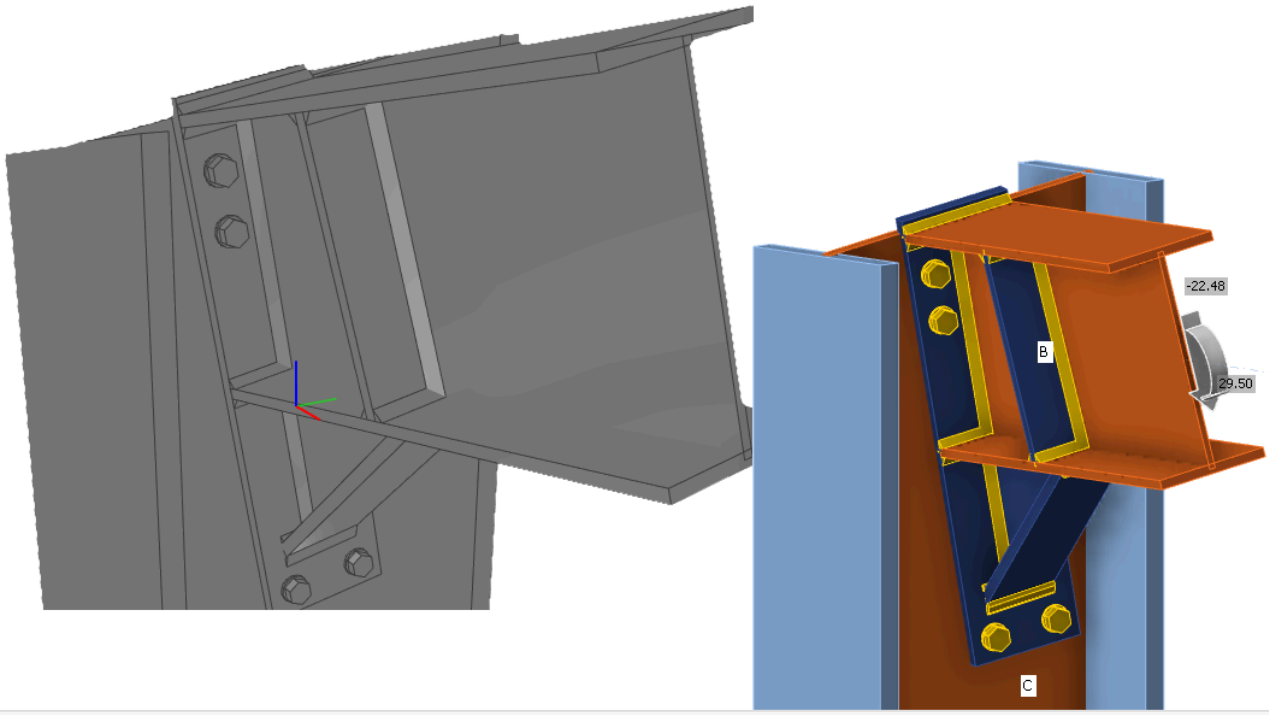
Improved Openings, Notch-cutting

- Notches and Openings operations are supported fully for IFC export and are visualized correctly in IFC viewers (Solibri, BIMvision, and Autodesk viewer checked)

Negative volumes primarily export to IFC

- Negative Volumes with limitations
  - In IDEA 23.1.1 added support for "Type" member Limitation - mirroring is not supported (planned for next iteration)
  - In IDEA 23.1.1 added support for "Type" plate Limitation - negative plate is exported to IFC and has to be disabled in IFC viewer (planned for next iteration)

Improved bolt positioning during export



Improved IFC export of circular connection plates with unequal meshing

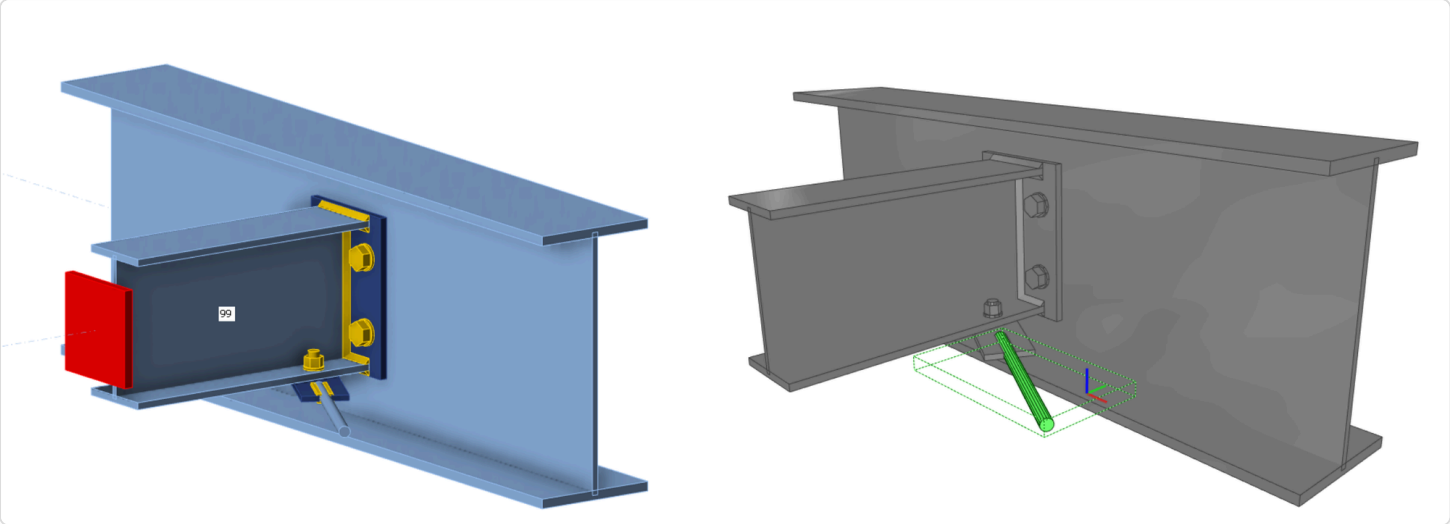
- Circular plates are exported correctly to the IFC

Validated IFC Syntax and Schema of IFC 2x3 by BuildingSmart validation tool

- https://validate.buildingsmart.org/ *(note: referencing rules are not satisfied on purpose - referencing welds to multiple edges)*

Rods can be exported to IFC from IDEA Statica

- Rods are supported for IFC export



Improved positioning of welds

- Welds are placed on the right edges according to the Connection model

Known limitations

IFC export from IDEA StatiCa Connection app and Viewer is optimized to export 3D geometry with parameters about materials and cross sections of contained objects.

Some software supports importing IFC objects and can even convert IFC objects to native design objects. We want to support this opportunity as much as possible but can't optimize it for all.

Here, you can see the status of optimization for the following software:

Tekla Structures 2022, 2023

- IDEA IFC model can be imported
- Items can be converted to native objects "as item"

Autodesk Revit 2023, 2024

- IDEA IFC model can be imported as a linked project

Advance Steel 2023, 2024

- IDEA IFC can not be imported into Advance Steel (boundary representation geometry is not supported in Advance Steel)

IFC version

IFC 2x3 as coordination view is exported. (no option for settings)

Beams and plates

Beam and plates are exported as mashed geometry, so they do not act like a dimension object. (Advance Steel import limitation)

Bolt holes

Bolt holes are exported, but not all viewers and tools support viewing them.

Properties

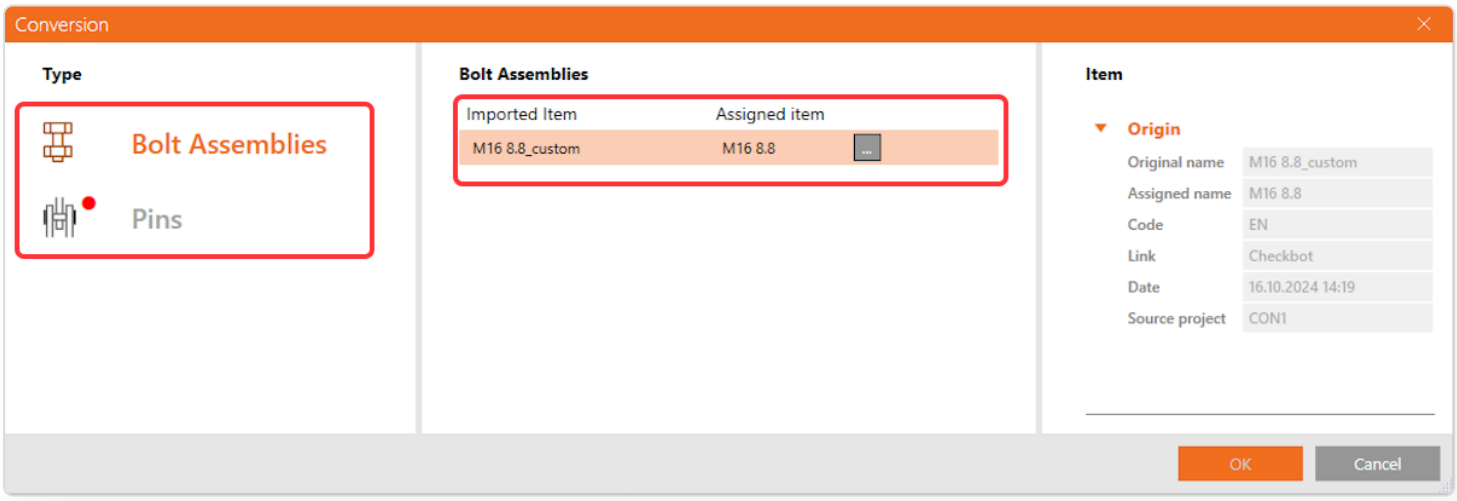
Exported properties are SI units only.

Advanced imports of connectors from CAD tools

In IDEA StatiCa, Bolt import is aligned with Bolt Assemblies with other library items like cross-sections and materials, providing consistent functionality across CAD BIM links. This simplifies the import, conversion, and clarity of Bolt Assemblies and Pins.
Bolt Assemblies are **automatically named** based on the original CAD application's bolt assembly name, eliminating the need for manual renaming.

When the bolt size in the CAD model is defined in **imperial units**, the imperial sizing is also used in IDEA StatiCa instead of being translated to the metric system.

Additionally, if the conversion of an imported bolt assembly is not processed automatically, Bolt Assemblies and Pins can be loaded directly from the IDEA StatiCa library in the **Conversion** tab. This is beneficial for CAD projects that lack data for bolt dimensions.



For export, both Bolt Assemblies and Pins are fully exported with all design information.

Usability and Licensing

Project settings

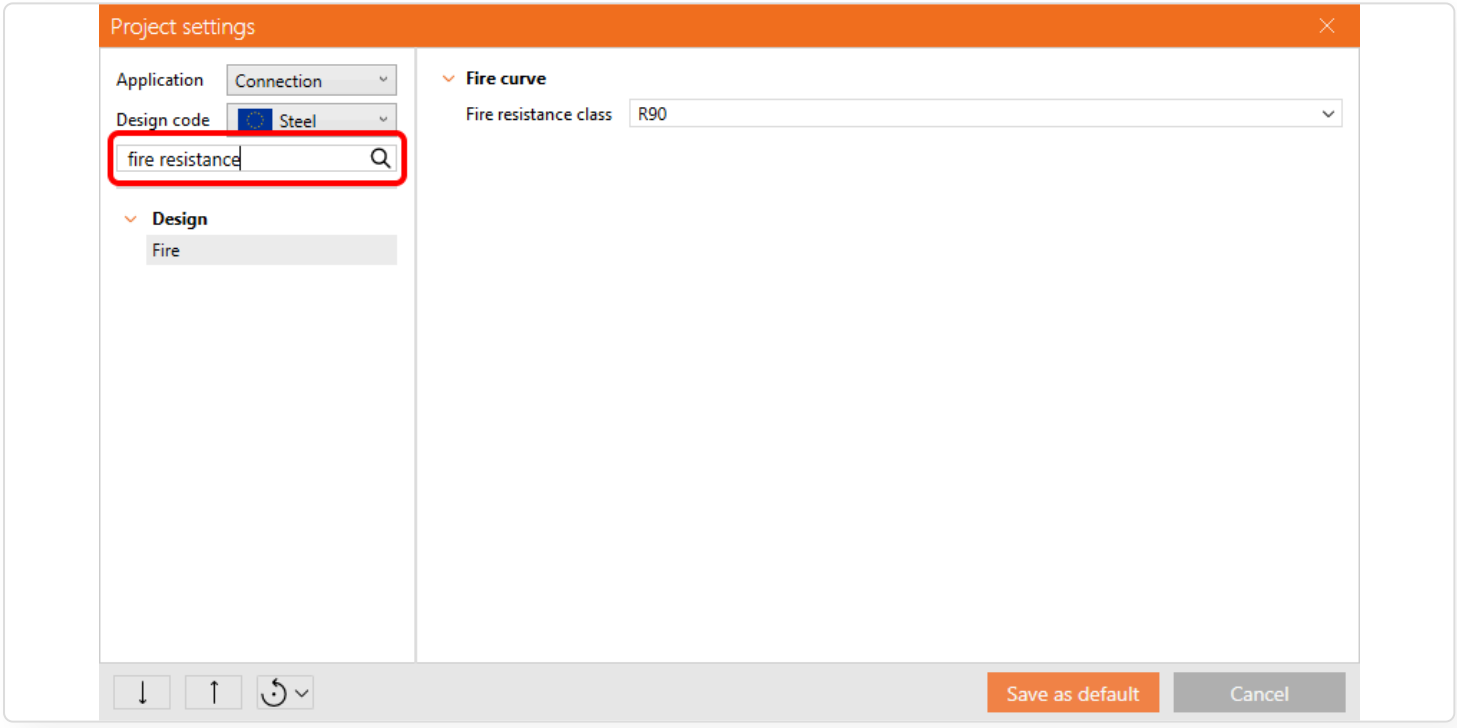
All settings related to a project or a single model are now available in one dialog called the Project settings, which can be shared among different users. The Project settings dialog was unified across different applications: Connection, Member, Detail, and the starting IDEA StatiCa app.
The Project settings replaced and united both the **Code setup** dialog and the **Settings** dialog in different applications. It is accessible from the ribbon in each app or, alternatively, from the IDEA StatiCa starting the app.

With this improvement, the ribbon design of affected applications was updated, and these apps now contain all previously missing settings.

Current project settings

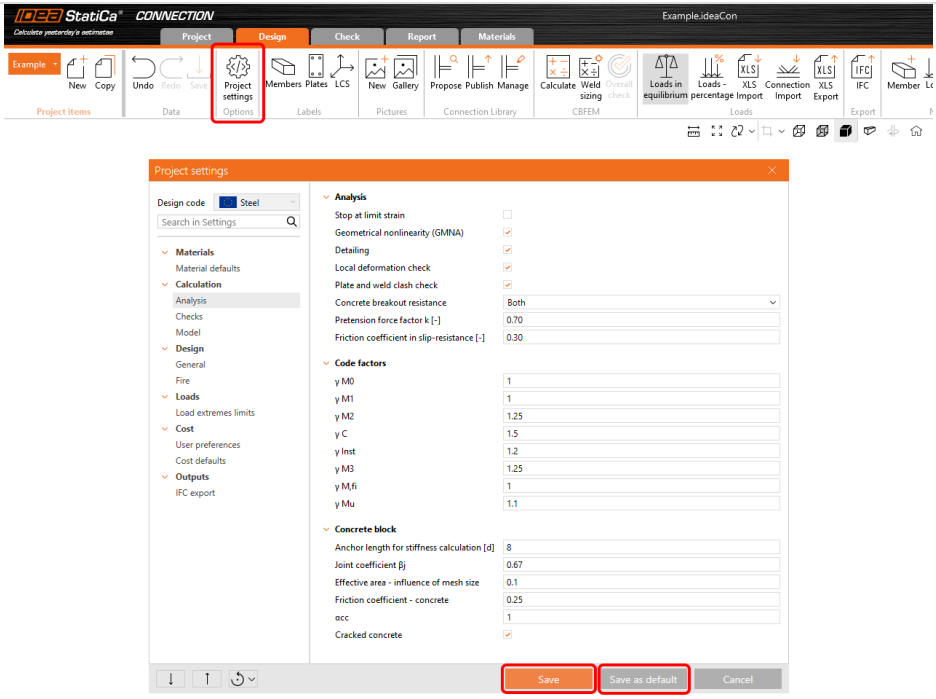
The **Project settings** dialog in the ribbon of each app (Connection, Member, Detail) allows users to change the settings for the currently opened project under the selected design code.

For example, when the dialog is opened in the Connection app, users can set the default material for new operations, the cost of steel for cost calculation, or the number of analysis iterations. To find the desired setting quickly, the **Search in Settings** can be used



By selecting **Save**, the changed settings are used only in the current project (and all project items within this project) and do not affect any other projects. If any other user opens the project, it keeps the previously saved settings.

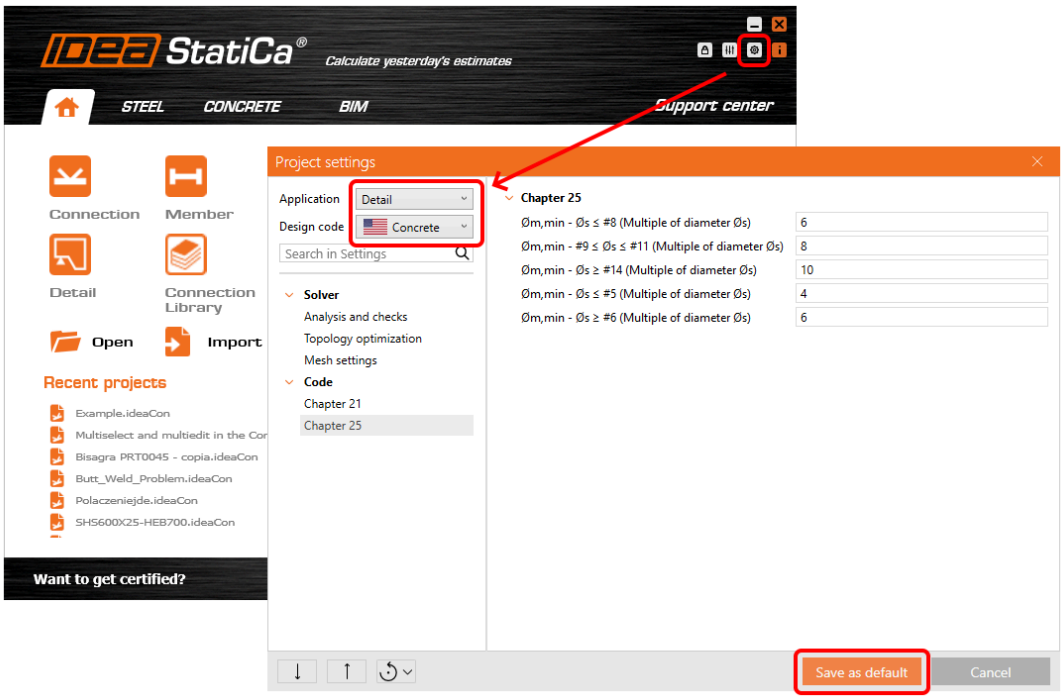
By selecting **Save as default**, the changed settings are used in the current project and in all new projects created in this app with this design code selected.



More about material defaults for the current project in the Connection app in the article Project item and material management.

Default settings

By opening the **Project settings** from the IDEA Starting app, users can control the default settings of all apps and all design codes in one place. The dialog can be opened via the cogwheel icon from the IDEA StatiCa starting app.



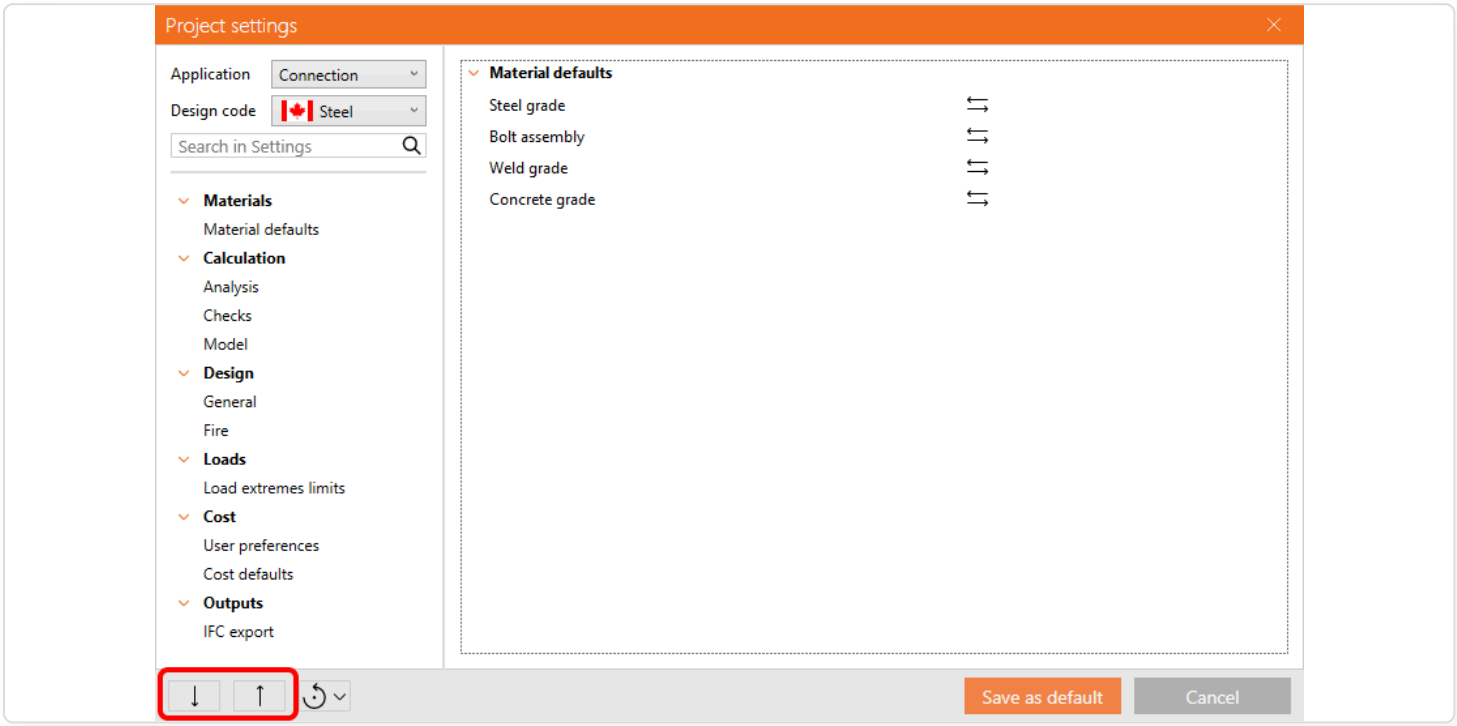
For example, users can set the default material when starting new projects in the Connection app, safety factors (gamma) in the Detail app, or limit plastic strain value in the Member app. To find the desired setting quickly, the **Search in Settings** can be used.

Users can browse the settings for each **Application** and **Design code**. By selecting **Save as default**, users can change the default settings for all new projects in different apps and under different codes.

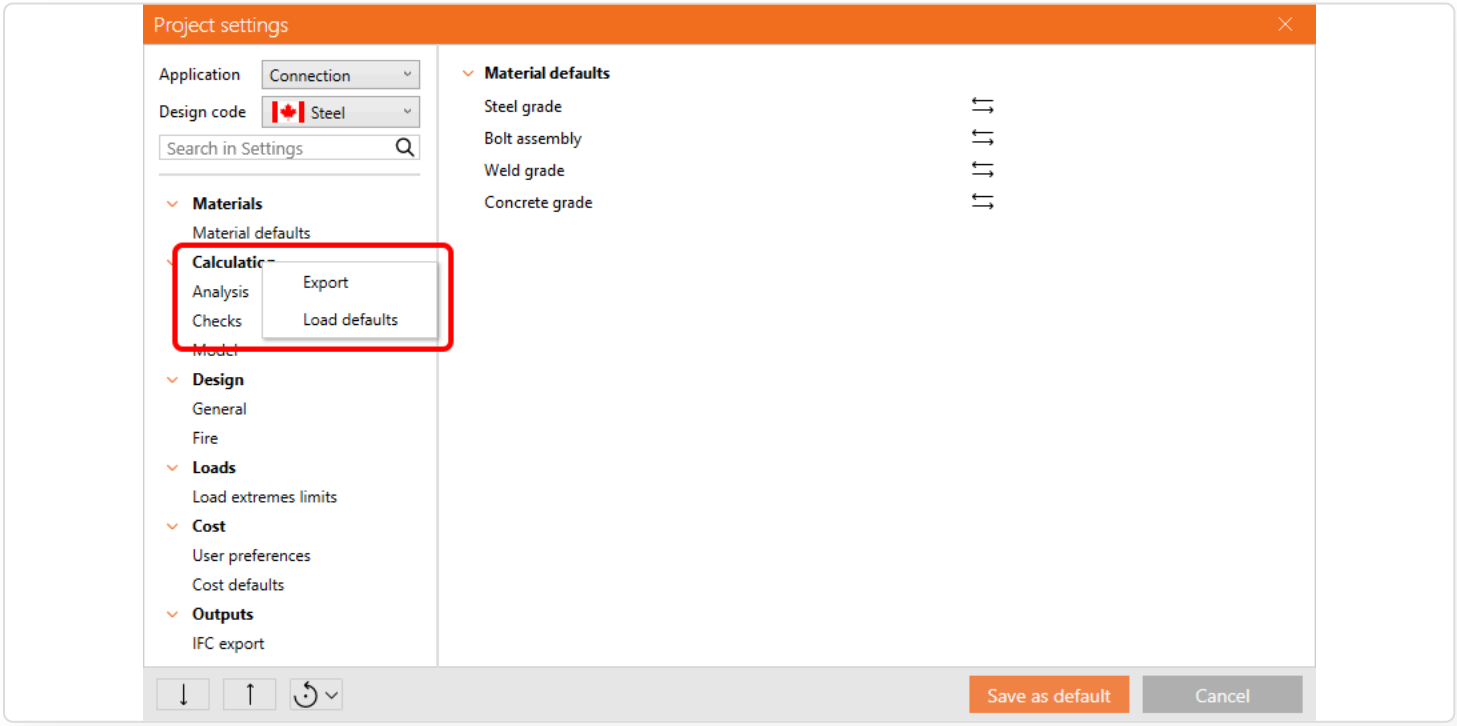
Export and import of Project settings

Users can **Export** and **Import** all project settings or just a part of them via the .json file and share them with colleagues so that the whole office keeps the same setup.

To export and import all **project settings**, select the appropriate icons in the bottom left corner of the dialog.

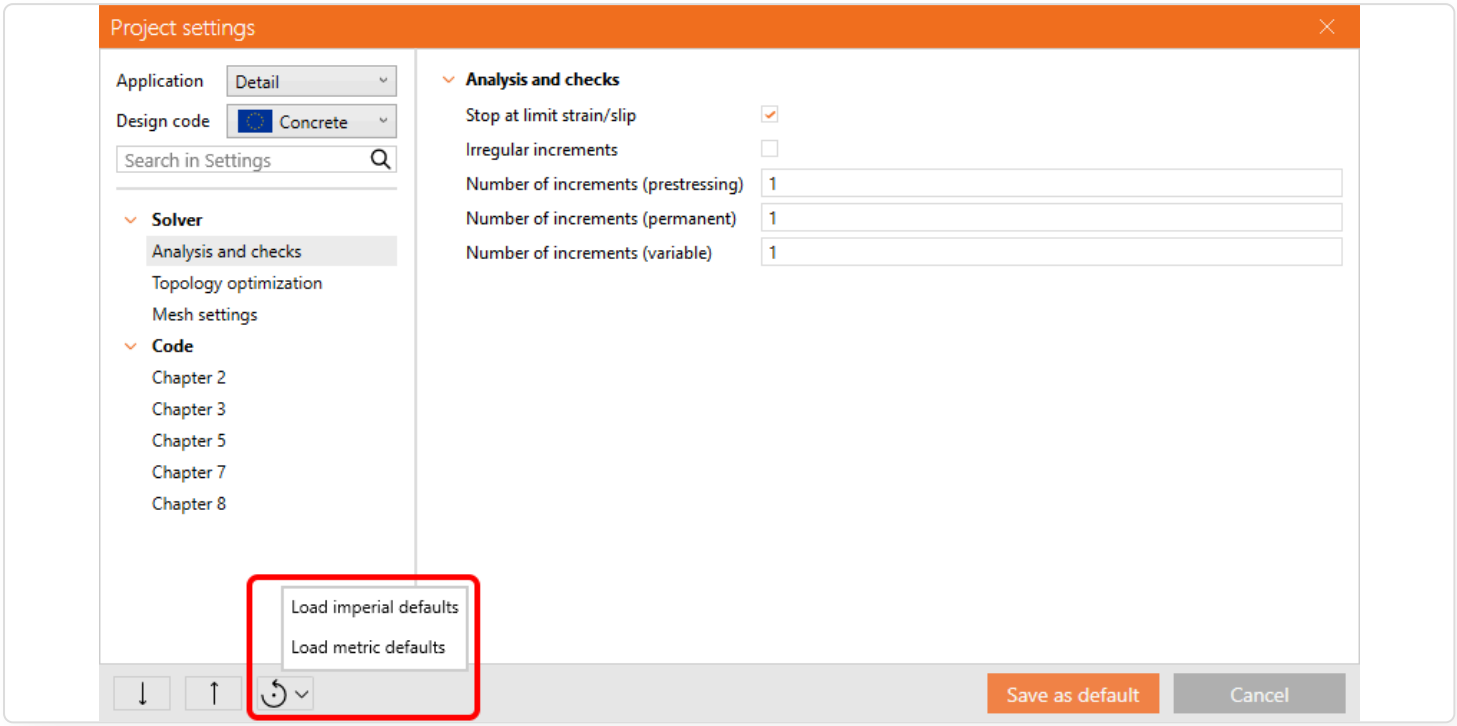


To export and import **only one chapter** of the settings, a context menu is available when you right-click on the chapter in the menu.



Reset project settings to factory defaults

The **factory settings** (the default settings after a new installation of IDEA StatiCa) can be recalled by clicking the round arrow icon in the bottom left corner of the Project settings dialog. You can load metric or imperial defaults.



To learn about IDEA StatiCa preferences, such as language, unit system, or display colors, read the article Shared preferences across the whole tool range.

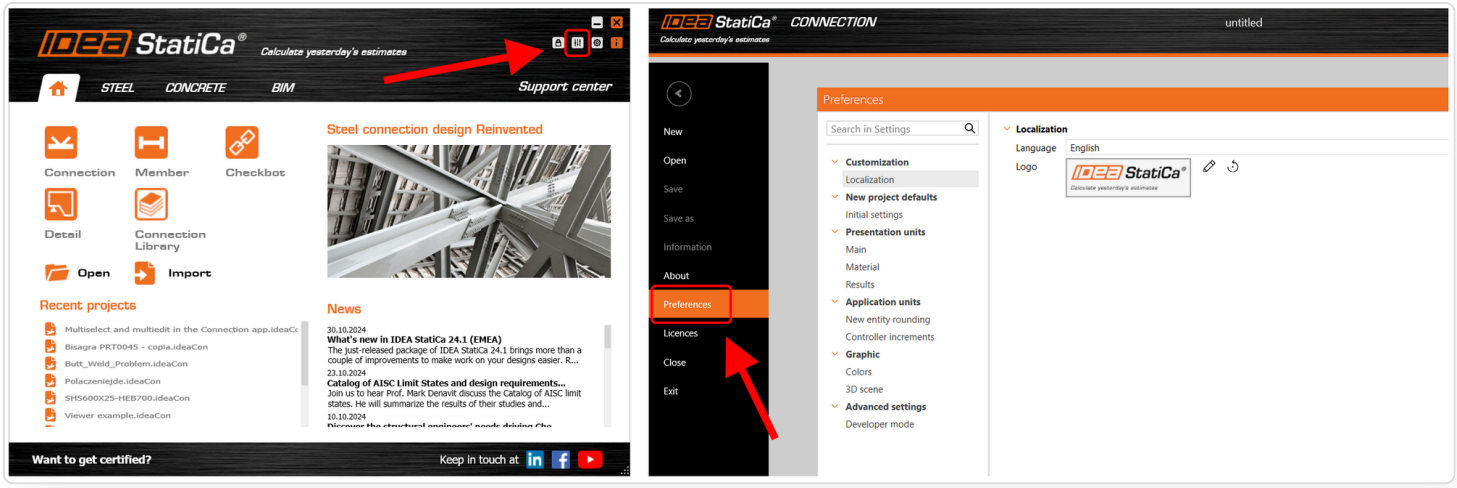
Released in IDEA StatiCa version 24.1.

Shared Preferences across the whole tool range

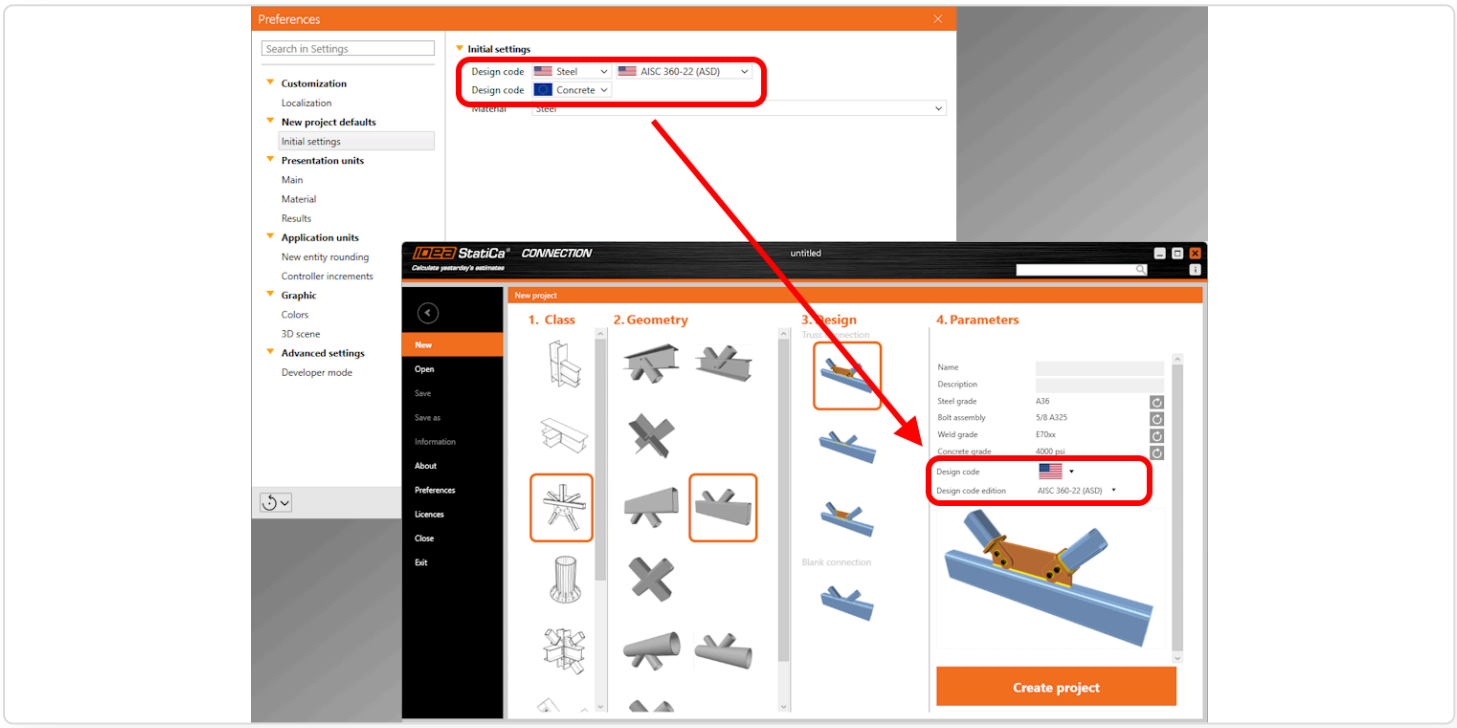
Ensuring consistently shared preferences across Connection, Checkbot, Detail, and Member applications significantly enhances the user experience and prevents the time spent on general settings.
The **Preferences** dialogue can be accessed from the starting IDEA StatiCa app or in a backstage menu of each listed app. Once the changes in preferences are saved, they will be applied to all other apps.

Embedding this feature ensures the following experience:

- Consistent settings across all the apps
- Simplification of users' workflows
- Ability to set all the shared preferences in one place for all the tools at once



An example of usage is defining the default **Design code** and subcode in the Preferences dialog, causing these parameters to be predefined for all new projects.

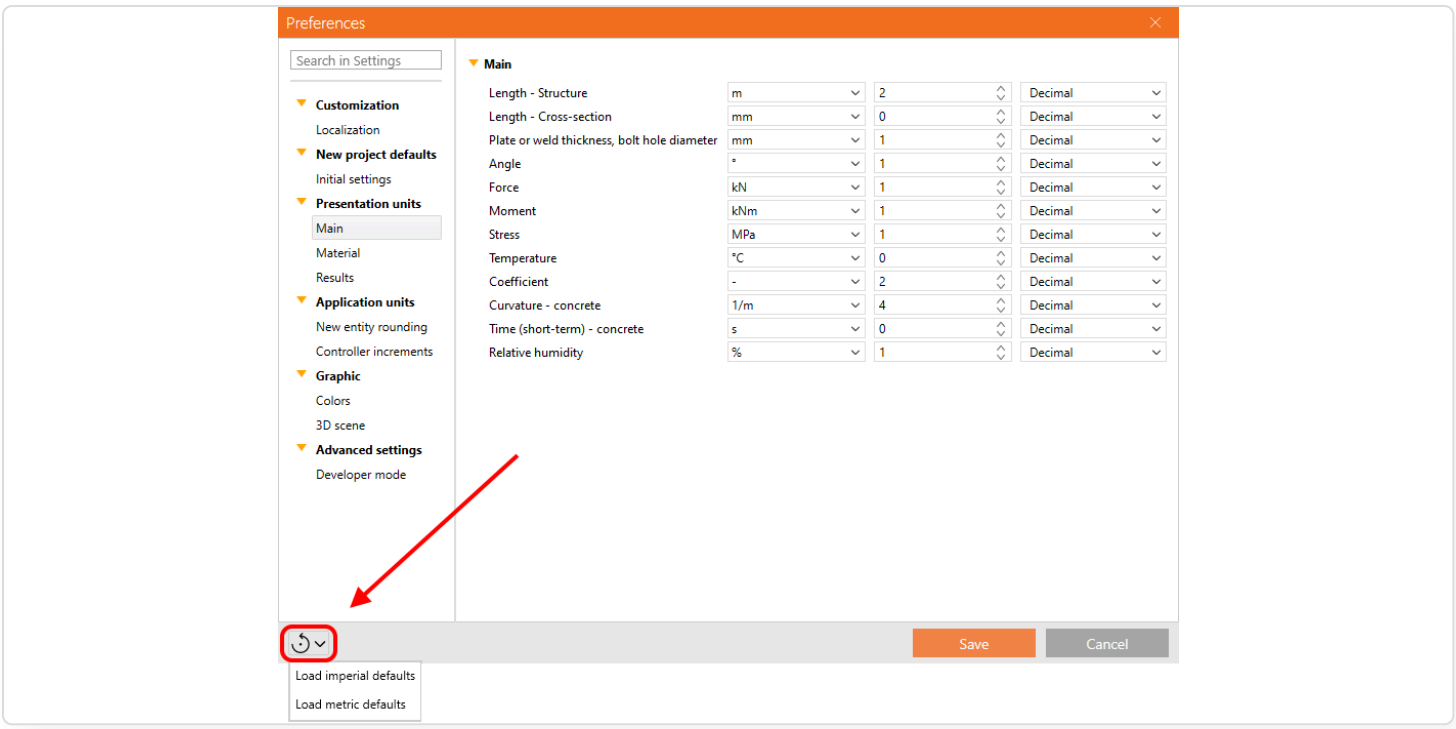


More about material defaults for the current project in the Connection app in the article Project item and material management.

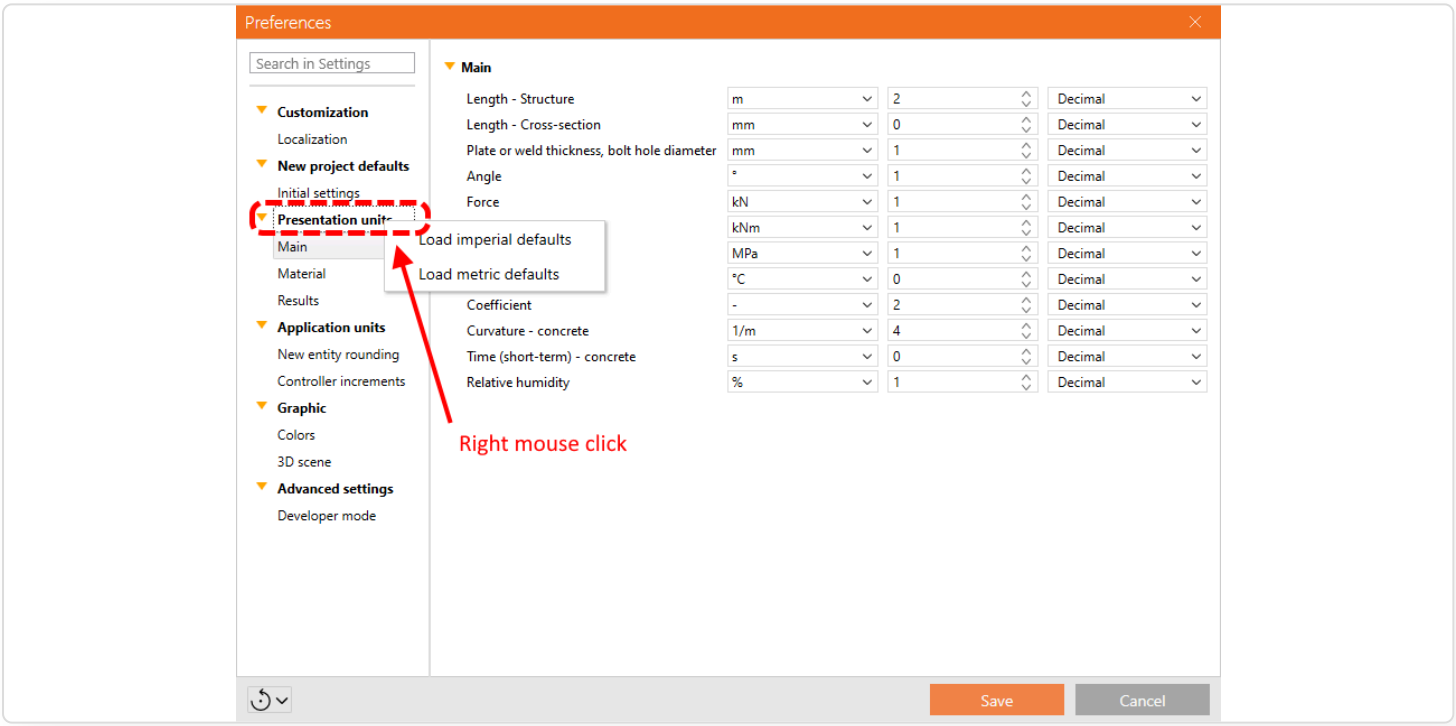
How to set the Default state

There are two ways to restore the parameters to their default settings. The default values are prepared in two versions: **Imperial** and **Metric**.

1) Reset the **whole list of preferences**



2) Reset just the **specific category**



When starting in Checkbot

This integration ensures that settings defined in Checkbot are automatically applied and recognized in the Connection, Detail, and Member items opened within the same Checkbot project, promoting efficiency and reducing the need for manual reconfiguration.

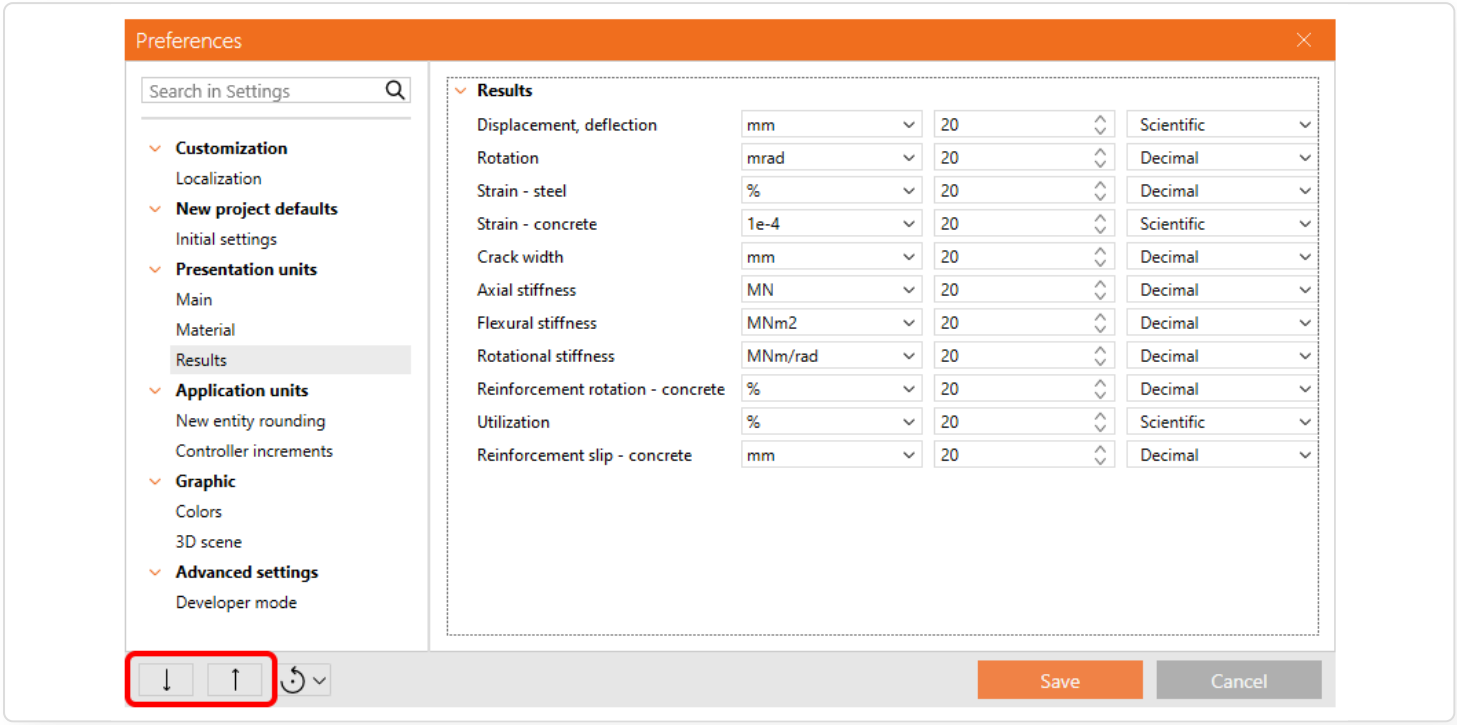
The advantages of shared preferences are currently available for Connection, Member, Detail, and Checkbot applications. The set of preferences available in the dialogue is continuously extended.

Released in IDEA StatiCa version 24.0.

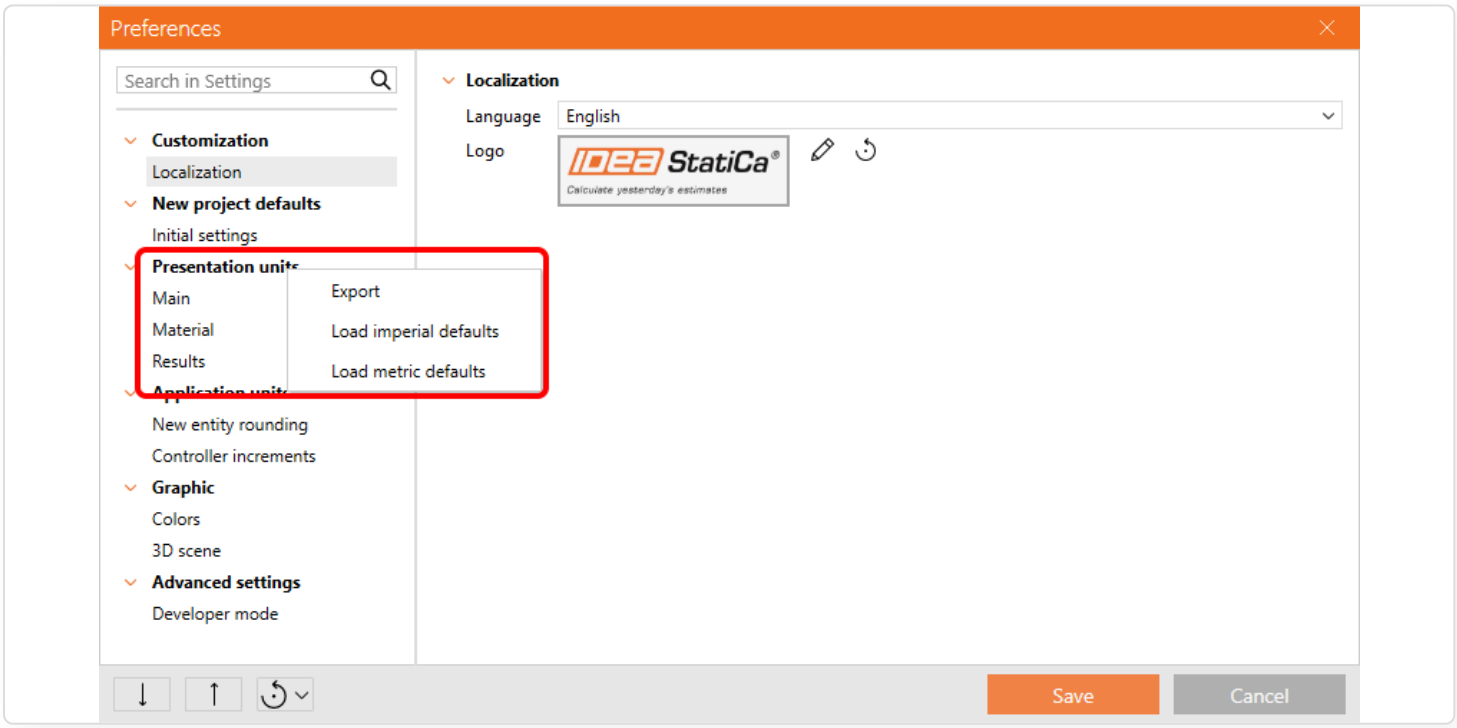
Sharing of preferences via Export/Import

Users can **Export** and **Import** the whole preferences set or just a part of it via a .json file and share it with colleagues.

To export and import **all preferences**, select the appropriate icons in the bottom left corner of the dialog.



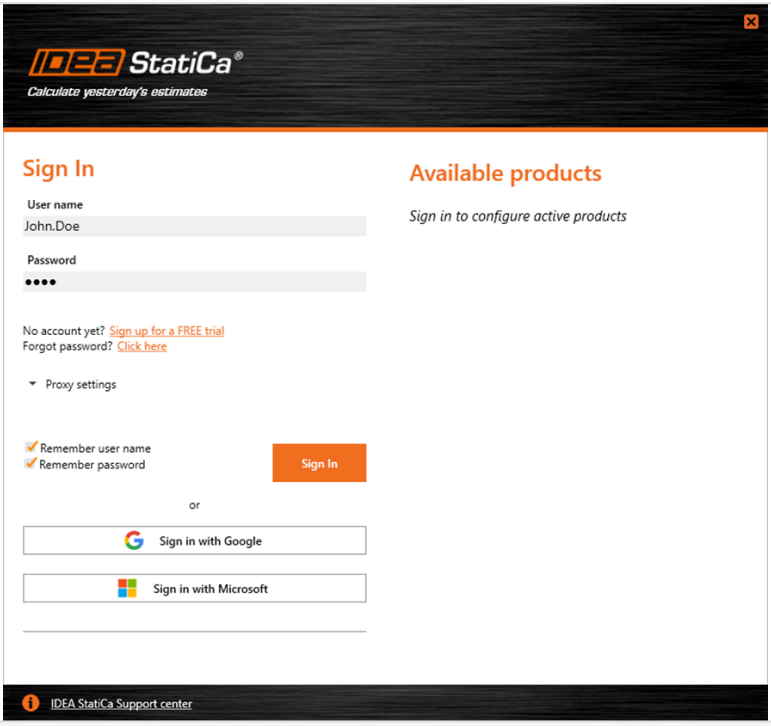
To export and import **only one chapter** of the preferences, a context menu is available when you right-click the chapter in the menu.



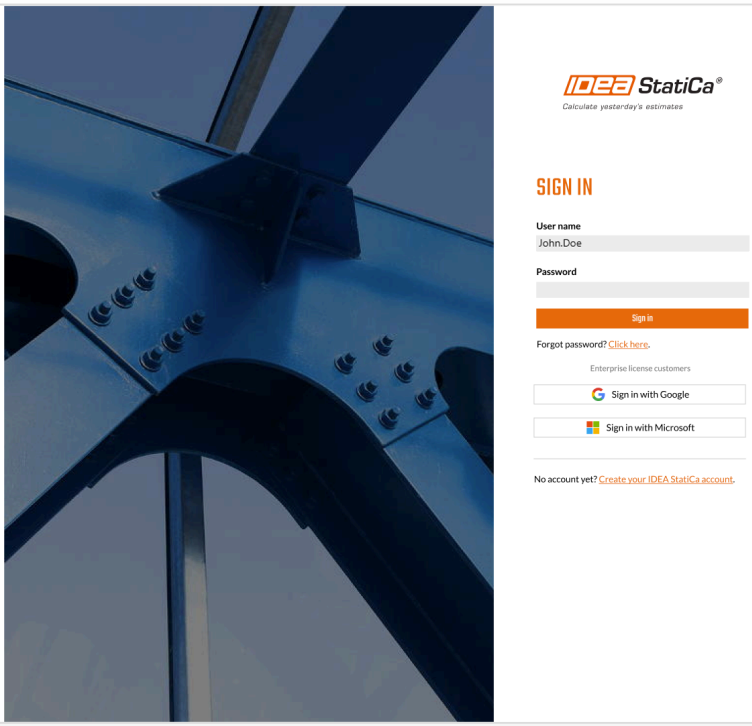
Released in IDEA StatiCa version 24.1.

Single Sign-on (SSO)

Single Sign-on (SSO) is implemented for both desktop and web user authentication. It is a feature that enables users of the Enterprise license type to use their Microsoft or Google credentials to sign in and for company admins to easily assign and manage their user base.
Single Sign-on authentication enables centralized authentication. With this feature, users can enjoy direct access to all IDEA StatiCa applications using a single set of login credentials. This simplifies the login process, providing a smooth user experience by eliminating the hassle of managing multiple passwords.



Single Sign-on facilitates access, enhances security, and boosts customer productivity, without the need to remember passwords. It empowers users to access both desktop and web applications and improves access management for company license admins.



The UI of the IDEA StatiCa Viewer tool

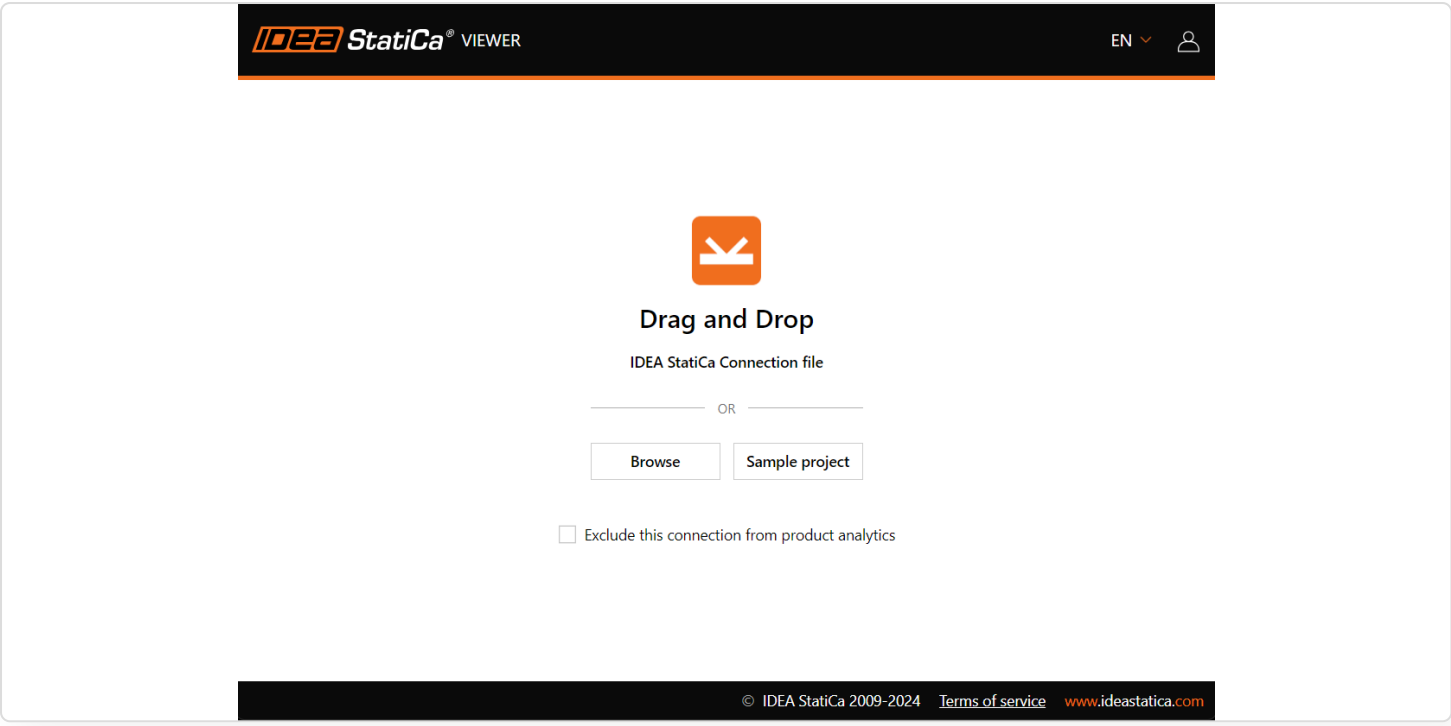
IDEA StatiCa Viewer is a free cloud tool for sharing structural data among the designers involved in connection or anchoring design. It was tailor-made for visualizing the detailed connection arrangements in the 3D scene and sharing the model data among connection designers in different stages.
You can find even more comprehensive information about the possible workflows with the Viewer tool in Project Viewer – useful and costless article.

How does it work?

The user can access the Viewer tool either through the URL viewer.ideastatica.com, directly from the Checkbot app, or from the Starting screen menu.

If the first option is used, the initial screen offers several options:

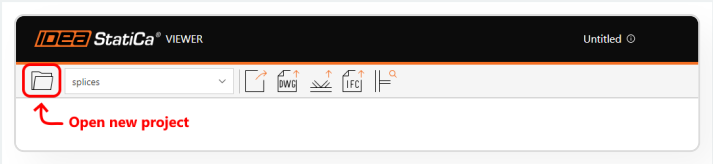
- **Drag and Drop** – for already existing model files
- **Browse** – for opening the project file from a hard drive
- **Sample project** – for opening a Demo project file with different types of connections

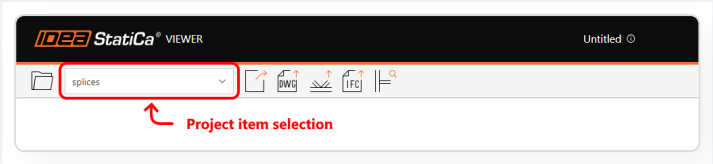


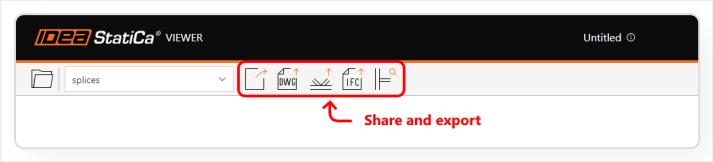
After dropping the model onto the Internet browser window, the 3D scene shows all the components of the connection model. If the users want to share the link, export the model as a DWG or IFC, or simply browse through the detailed information about the model's components, they need to be signed in.

If the current IDEA account hasn't been created yet, it is sufficient to create a free Basic account according to this simple guided process.

The top ribbon offers several essential options

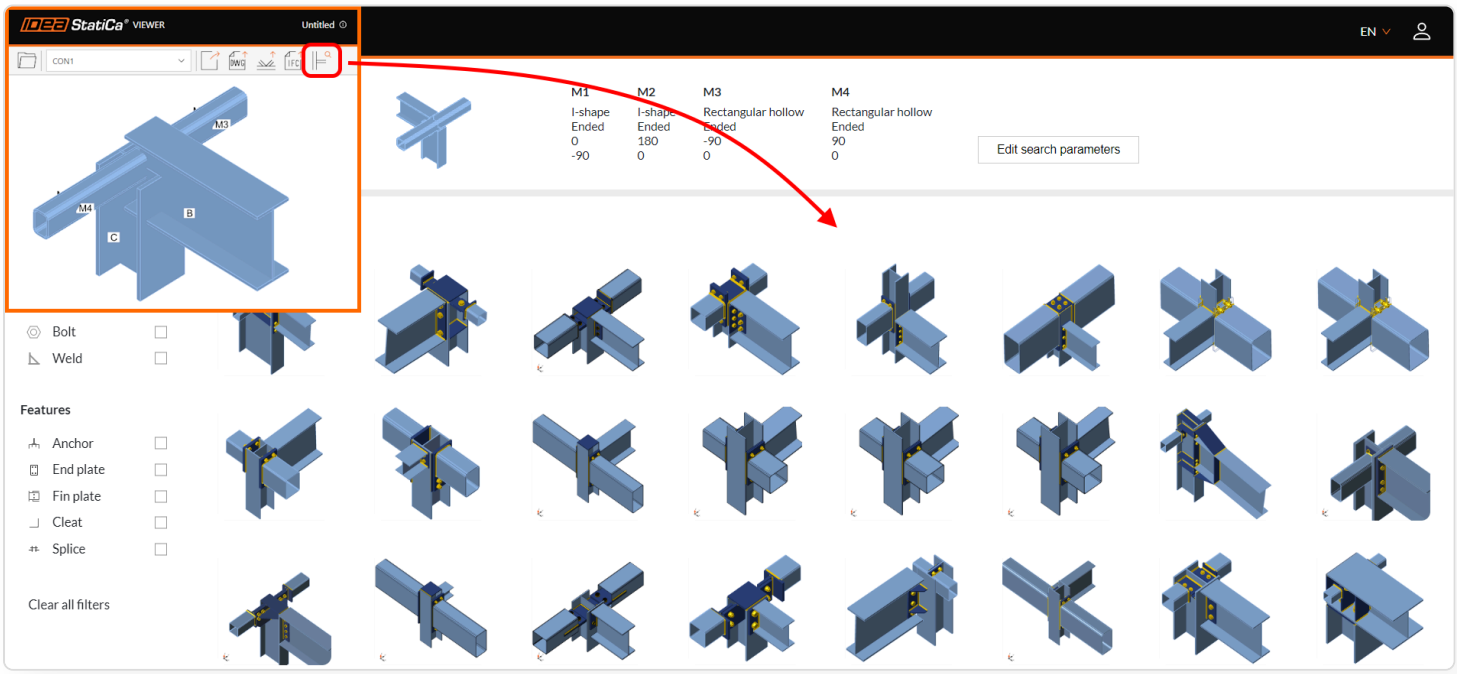




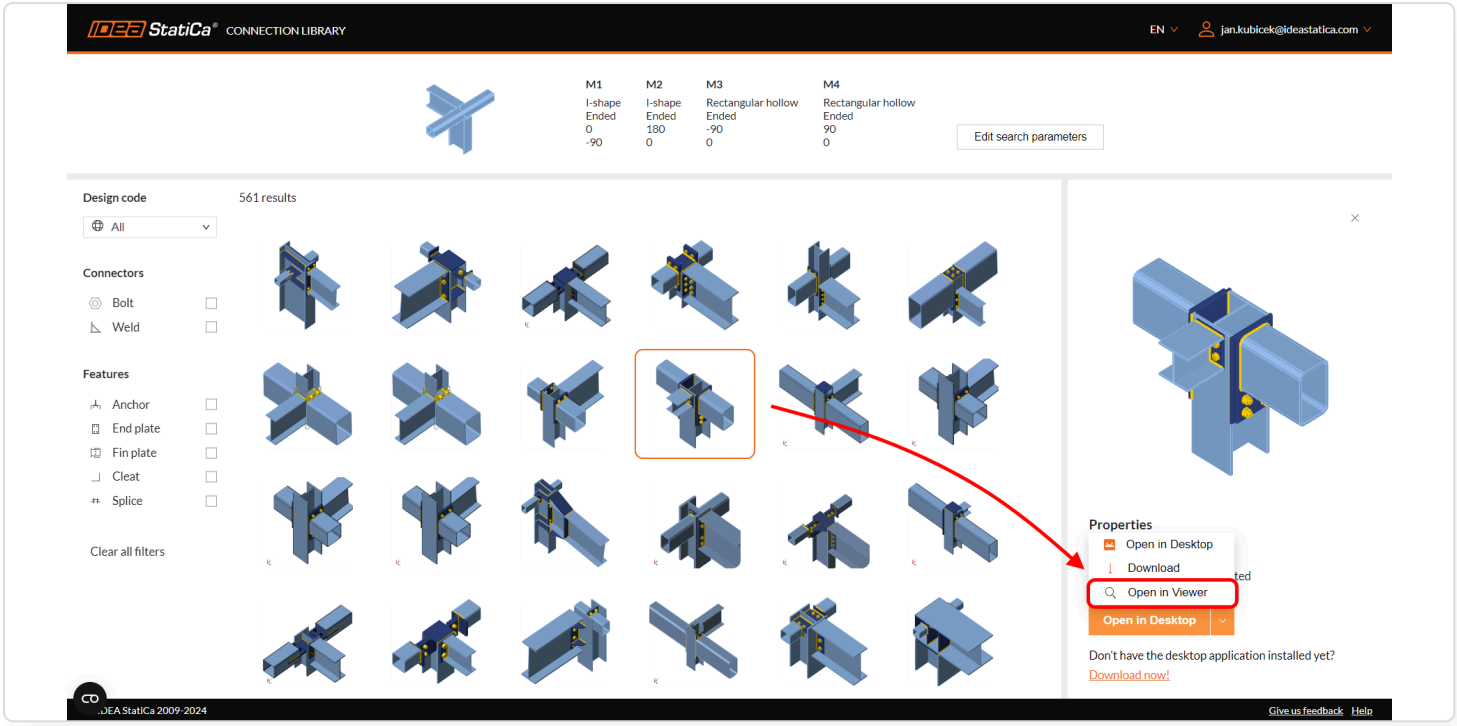


Exporting options in Viewer

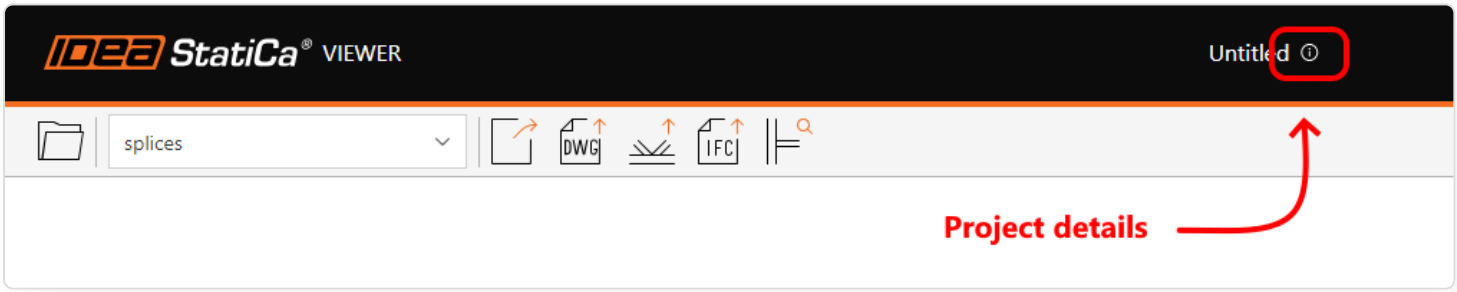
- **Share**: One of the easiest ways to share the model information is to use the URL link directly to the Viewer web page with the model already uploaded. This button copies the URL link, which will be stored on the cloud server permanently, so the link can be kept and referred to in the designers' communication for the whole length of the project.
- **Export to 3D DWG**: Exports the 3D model in DWG file format with the solid 3D elements and breaks down all plates into separate 2D line blocks.
- **Export to IDEA StatiCa**: This option creates and downloads the model in .ideaCon file format according to the current IDEA StatiCa version.
- **Export to IFC**: Exports the 3D model in IFC (International Foundation Class) file format.
- **Explore in Connection Library**: This option launches the IDEA StatiCa Connection Library web page and offers hundreds of possible alternatives to the current arrangement. The filtering in Connection Library is preset according to the arrangement of the currently opened model in Viewer. The arrangement follows the number of members and their geometry with specific cross-section types.



After finding a valid connection design solution, the user can open the specific design back in the Viewer tool.



Besides these options, the project parameters can be displayed by clicking the info button at the top of the browser window.

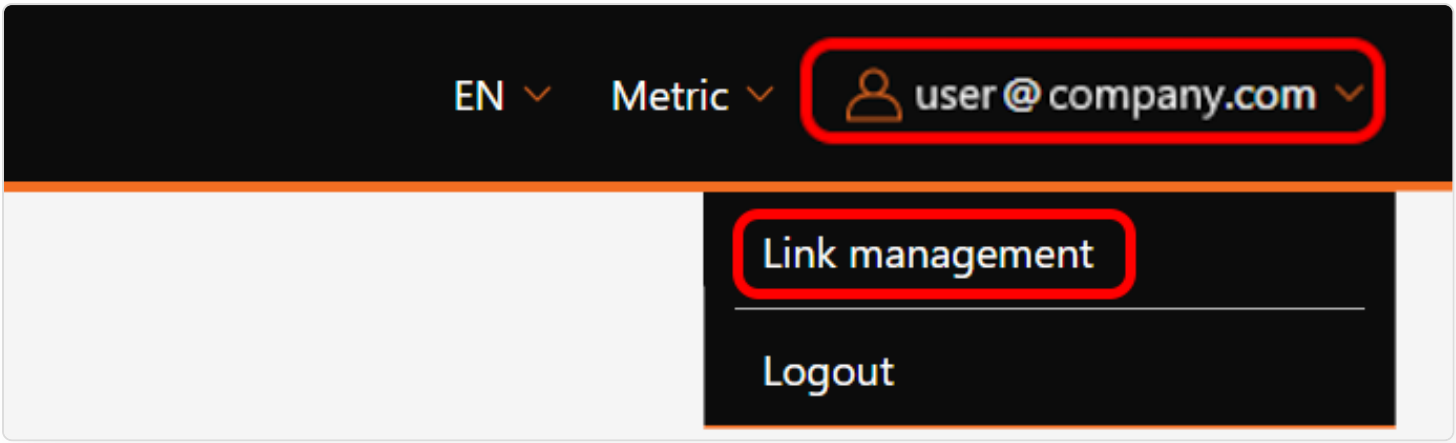


Additional model properties

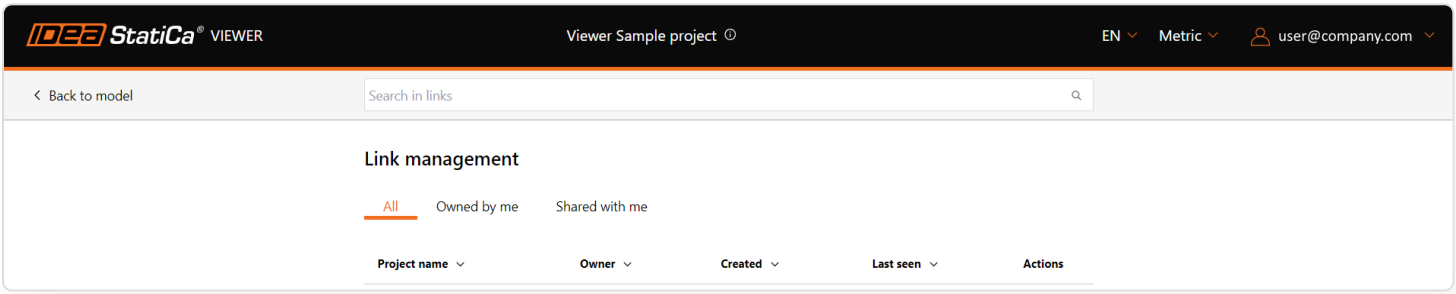
The inclusion of parametric templates in the Connection Library represents a significant advancement in the design process for connection designers. This functionality simplifies the design process by providing:

- **Language selection**: Two options are available – (EN) English and (CZ) Czech
- **Units**: A switch between Metric and Imperial units is available
- **User login**: Link management and Sign-in/Sign-out.

Link management



This tool provides the management of used URL hyperlinks. Users can browse through their own links or the ones shared with them. There is also an option to delete obsolete links (in Actions) which are not useful anymore.



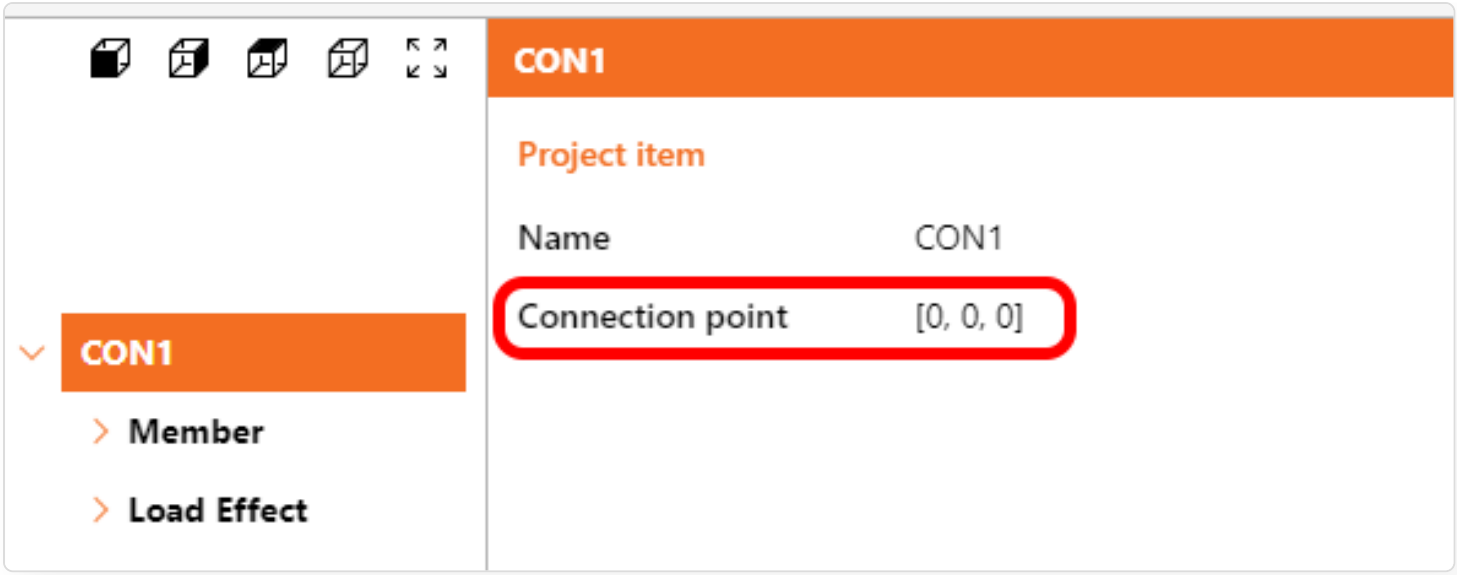
The Property panel

In the right part of the screen, a whole set of model parameters is stored.

Connection space coordinates

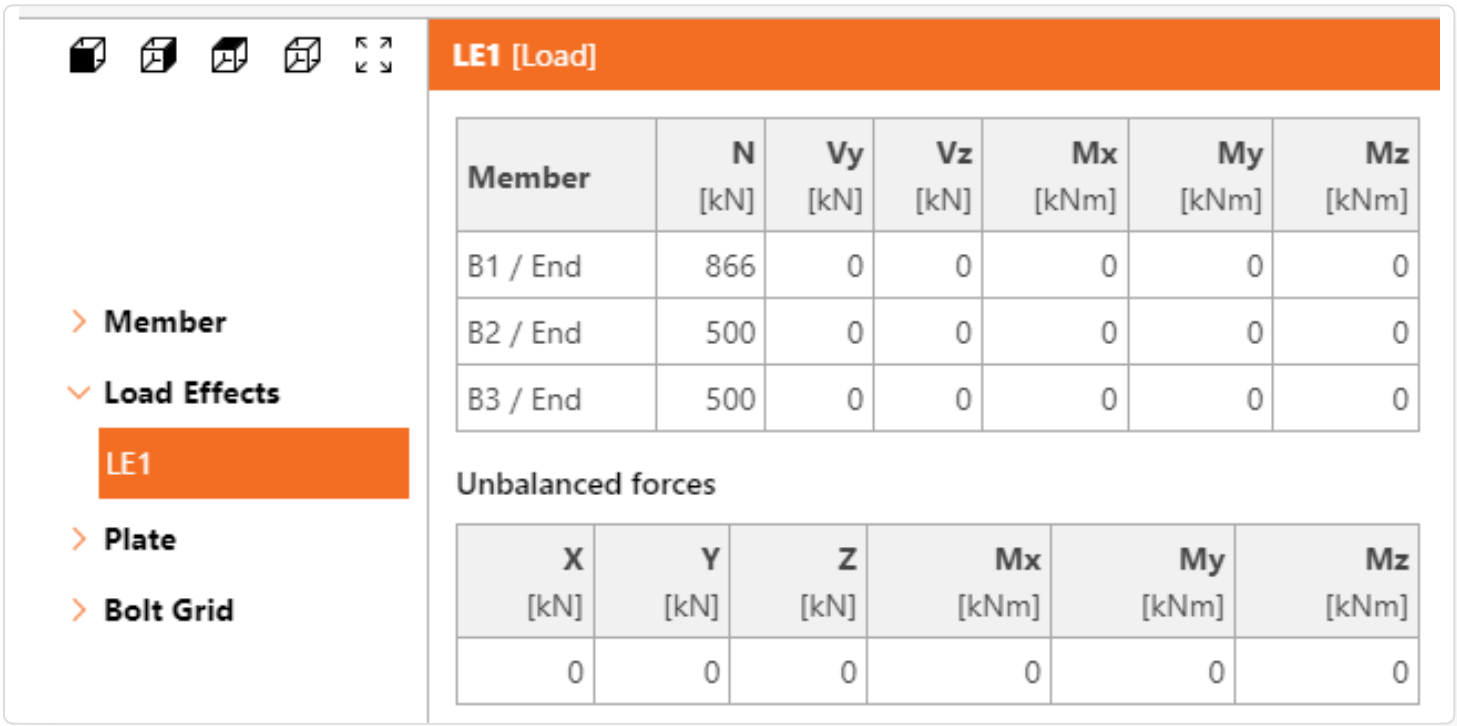
After selecting the connection name (Project item) in the tree, two parameters are displayed: the Name of the item and the Connection point.

These coordinates indicate what is the spatial position of the connection node. For models created from scratch in the Connection app, the position will be zero in all directions. However, for models coming from the Checkbot app, the values will follow the spatial position of the node within the whole structure.



Load Effects

Besides information about the component materials and sizes, unique and important information about how the connection is loaded can be found under '**Load Effects**'.



Member	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
B1 / End	866	0	0	0	0	0
B2 / End	500	0	0	0	0	0
B3 / End	500	0	0	0	0	0

X [kN]	Y [kN]	Z [kN]	Mx [kNm]	My [kNm]	Mz [kNm]
0	0	0	0	0	0

Together with load values, another crucial design info is visible - **Model type** and **Force position**.

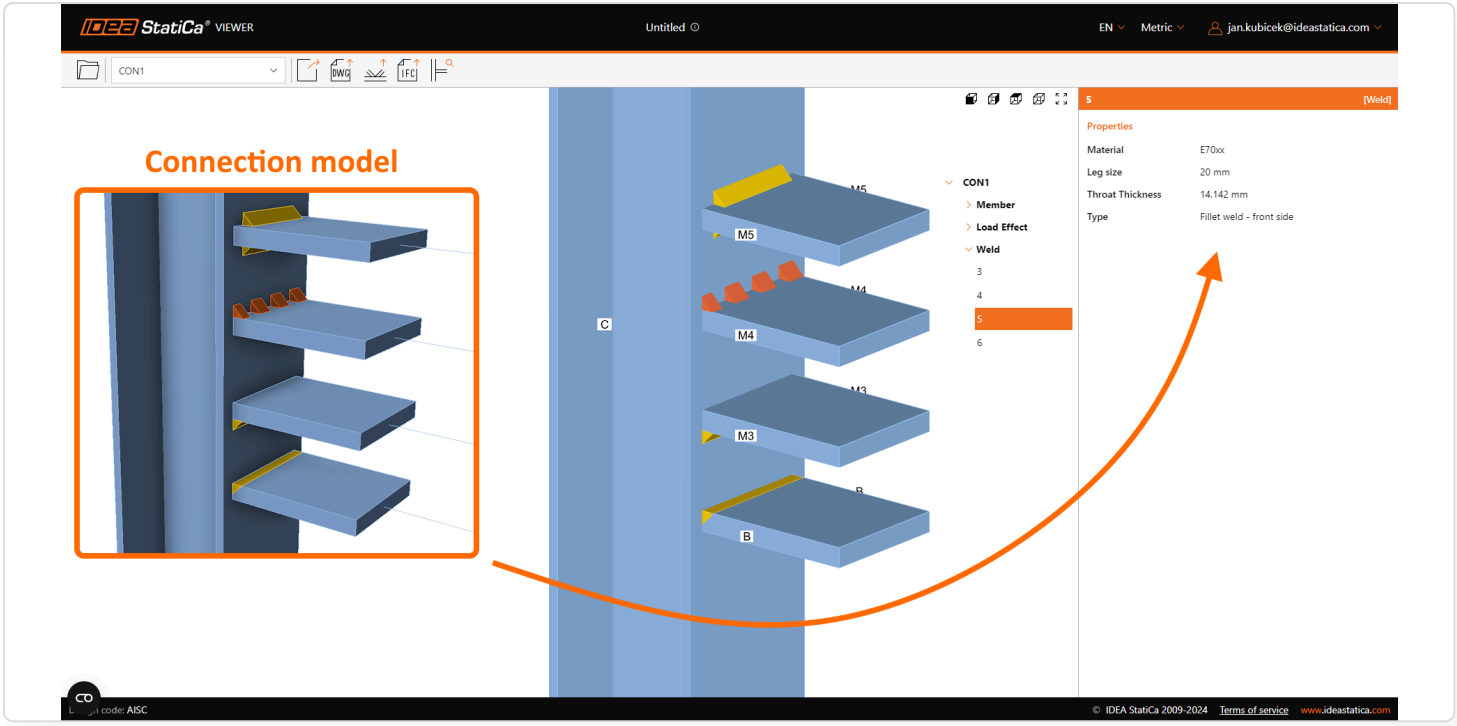
Weld properties

The right panel also contains the info about the weld type

- fillet (one-sided, double)
- butt weld
- partial joint penetration

with visual representation in the 3D scene also for

- continuous
- partial length
- intermittent welds



For clarity of outcomes, both values of weld size are displayed at the same time - Leg size and Throat thickness, independent of the units selected.

Connector properties

Additional information is displayed for different connector types:

Pins

- Material
- Diameter

Anchors

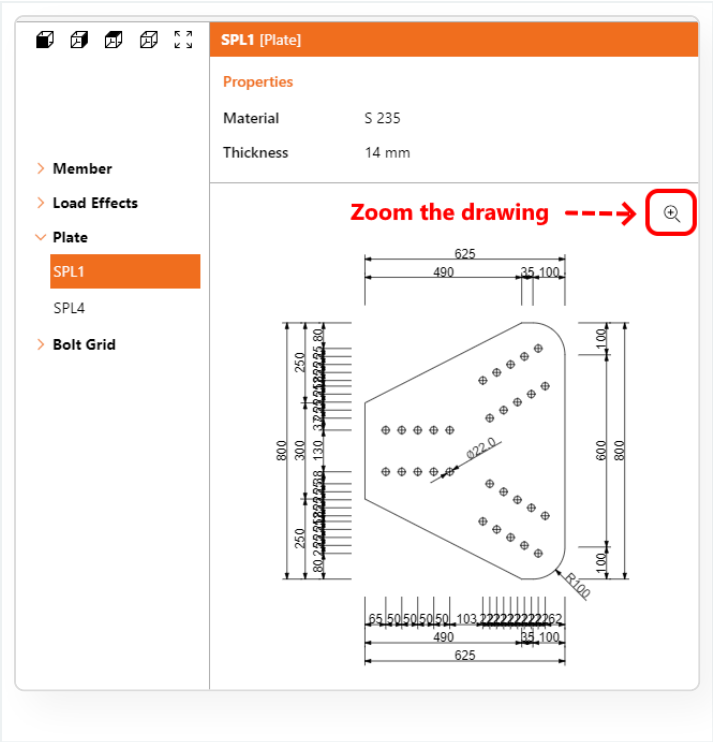
- Anchor name
- Anchor length

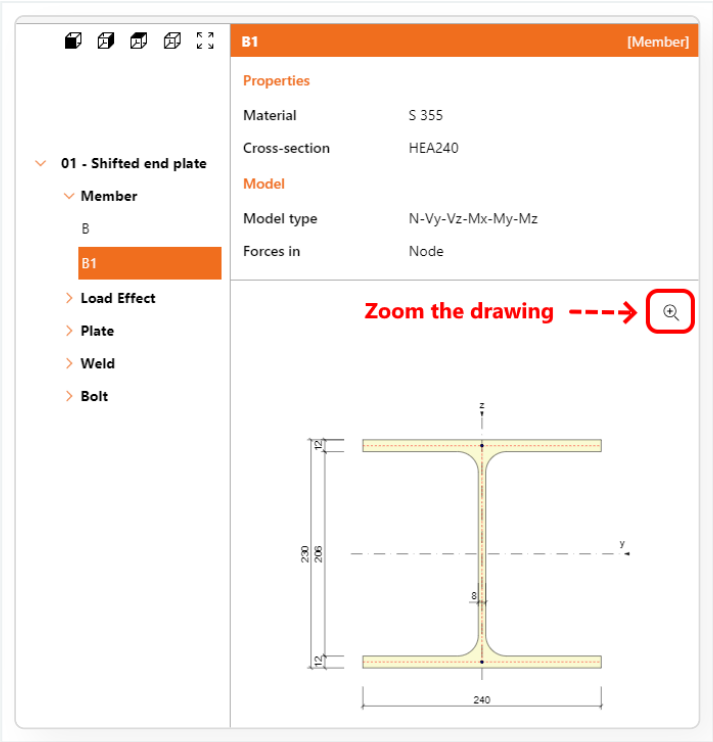
Bolts

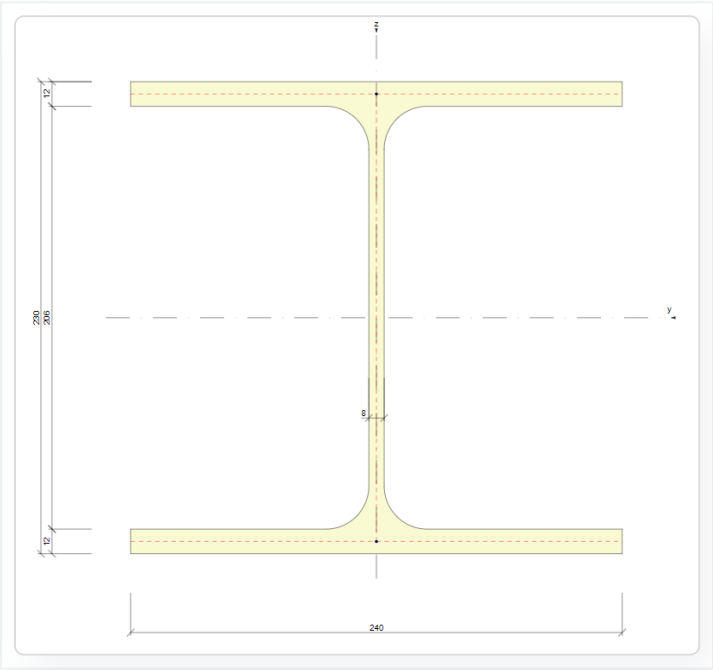
- Bolt assembly name
- Shear in thread

Element detail drawing

Detailed drawings of specific cross-sections and plates are displayed in an additional graphics window with the basic dimensions. For profiles and models with more complex plate shapes and overly dense dimension lines, there is an additional option to open the Zoomed vector drawing in a new subwindow.







Released with IDEA StatiCa 24.1.

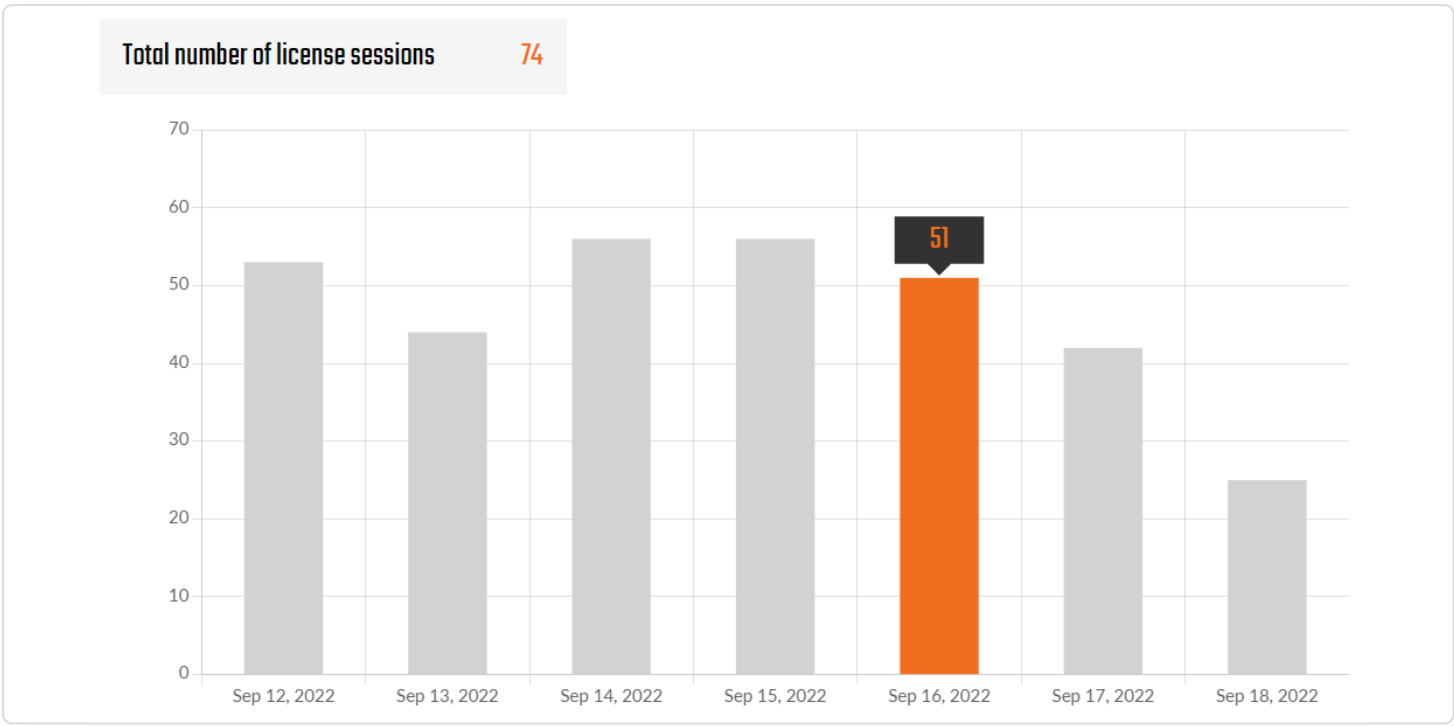
License usage analytics in the User Portal

Every customer can investigate how their IDEA StatiCa licenses are used. In the User Portal, you can find the category Analytics. Overviews displayed here help admins to recognize how often the IDEA products are used within the office.

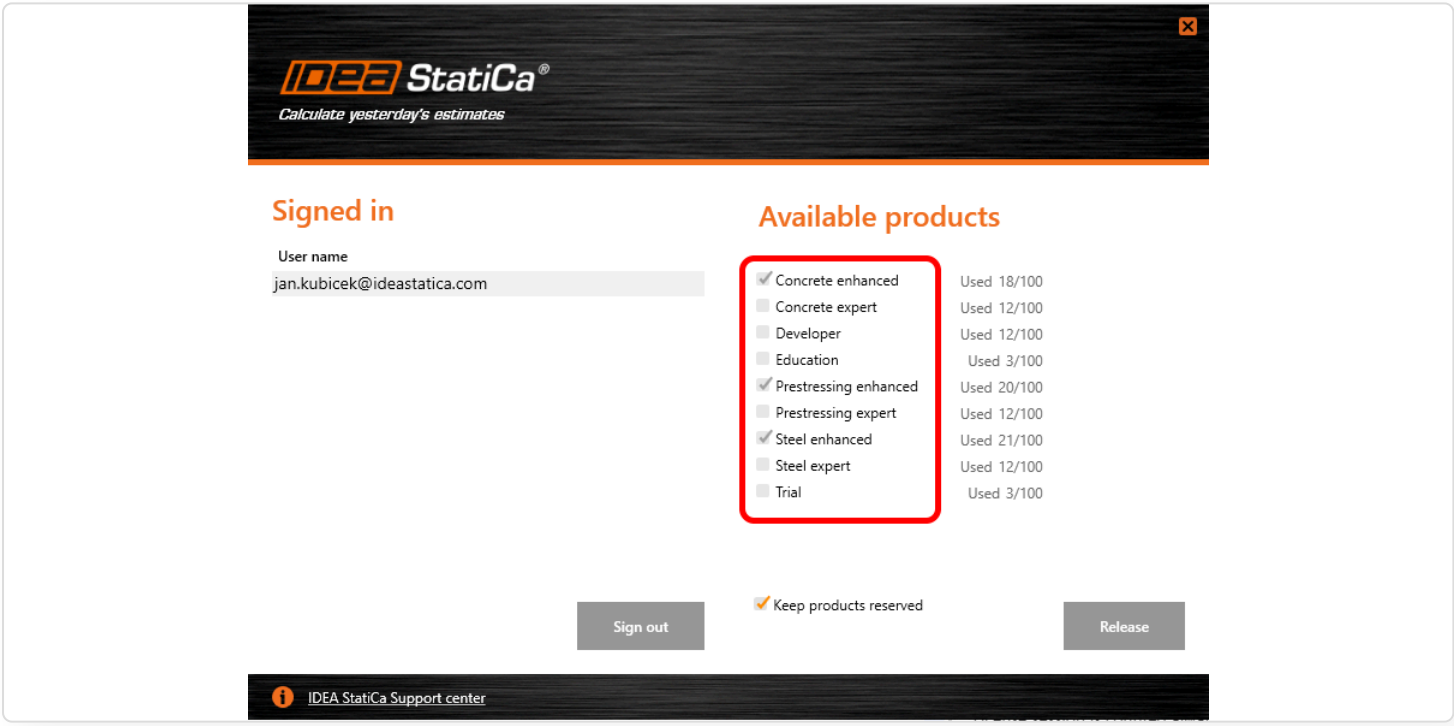
Analytics overviews:

Number of License sessions

License sessions are always counted when a license seat is occupied by running any IDEA StatiCa app.

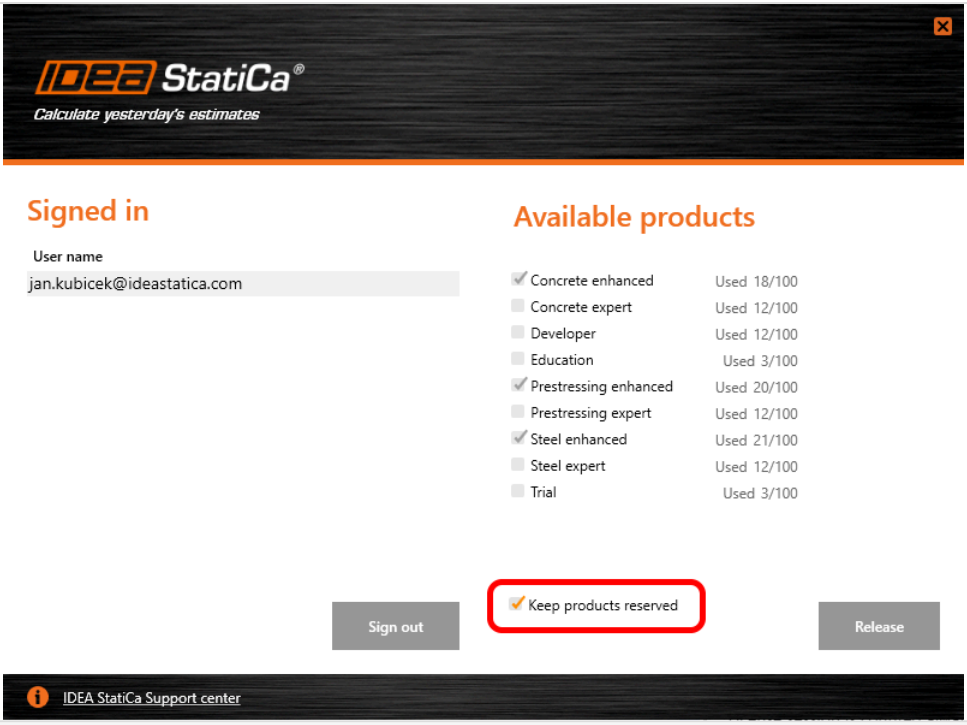


The number of License sessions depends on how many Products the user has activated in the License setting:

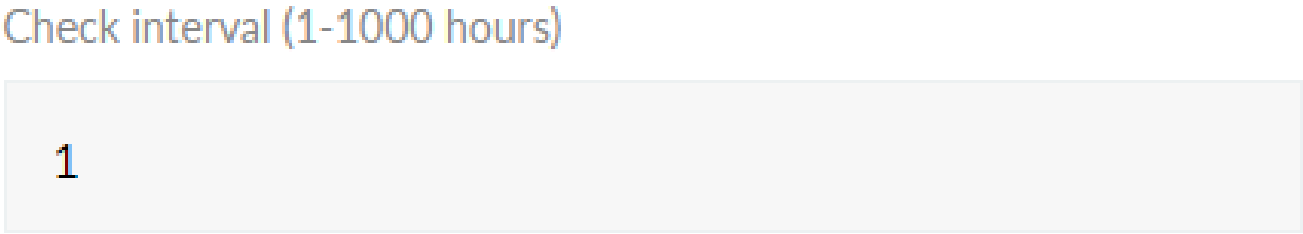


(In this example, four license sessions would be counted by the starting of any IDEA StatiCa app by this particular user.)

The user always has an option to Keep the products reserved for a longer time period, which can be set in the range of 1-500 hours after closing the app. It can be useful, e.g., for planned work without an Internet connection.



With the option 'Keep products reserved' turned on, the license session will be ended by running out of the 'Check interval' time period (set in the User Portal license setting).



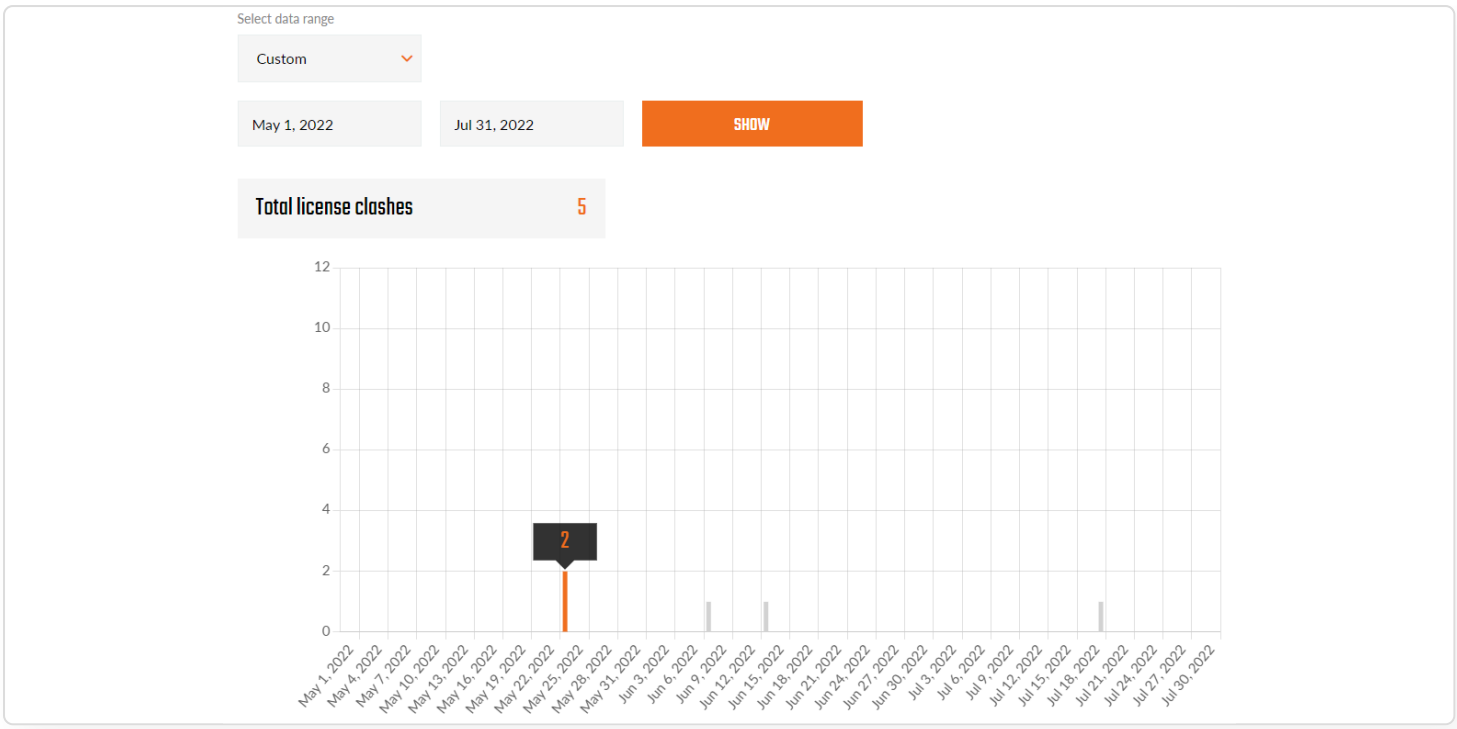
Otherwise, with the option 'Keep products reserved' turned off, the license session will be ended by closing the last active IDEA StatiCa app.

The check interval is set to one hour by default and can be edited by the license admin in the User Portal.

Number of License clashes

A clash is always counted when a user attempts to use a product while all the purchased license seats are occupied by other users.

The user can review the number of clashes in the history to be able to decide whether purchasing another license seat could help the users to be productive without waiting for others to release the seat.



License user analytics

You can have a better overview of the IDEA tools usage for specific license users with usage analytics. With this tool, the company license admin has a clear information about how their employees are using the application. This feature is aimed at Enterprise customers, for which this following data is available:

- License User name
- Country
- Reservation time of the seat

A typical use-case of this feature is evaluating the participation of branch offices in the total usage of IDEA StatiCa tools. Based on this, they can distribute the invoicing for their subsidiaries.

Description

- The feature is added to "License -> Analytics" as a tab Reports (accessible for users with Admin role)
- Available for customers with an Enterprise license
- Available for users with an Admin or Superadmin role
- The user selects the month/year for which the report should be generated
- The user receives a .csv file
- The .csv file contains the following columns: userid, username, usage in minutes

Usage in minutes is calculated based on the following logic:

- The usage is measured from when the given user reserves at least one seat. There is a check every 30 min to confirm the user's status – if they are still using the application. If not, then they are logged out and the measurement time ends.
- The user can have the "Keep seats reserved" check box in the license tool selected. In such cases, we use the value from the field "Time interval" in User Portal (License -> Details). Here, the license admin can define how long (in

hours) users stay logged-in when using the application in offline mode. If they do so, then we use this value and add it to the usage we measured before the user went offline. This is the value saved in the report.

- Usually, the measurement ends when the user releases the seat (by closing the last opened application or hitting the dedicated "Release" button) or logs out.

Available in the IDEA StatiCa **User Portal**.

Compatibility of versions

IDEA StatiCa model designed in older versions of IDEA StatiCa may show differences when you upgrade to the newest version.

Version compatibility

IDEA StatiCa software version (e.g. 22.1.3.0789) is described by a number represented by the major version (22), minor version (.1), patch number (.3), and build number (.0789).

Every major and minor version of IDEA StatiCa applications changes project data and makes it impossible to use in the previous versions. This means if you create or save a project in version 22.1, you can not open it in 22.0.

However, patches within the same version are compatible, so if you create or save a project in patch 22.1.3, you can still open it in patch 22.1.0.

The Viewer always generates project files in the latest IDEA StatiCa version and patch released.

Opening old project in a newer version

Models designed in older versions of IDEA StatiCa may show differences when you upgrade to the newest version. When upgrading to a newer version of the program, you may encounter some differences associated with opening models created in older versions.

To get updated, please read the release notes for every major and minor version released, together with patch updates and lists of resolved bugs.

Below is a list of the most important changes for each new version of the program that may lead to some changes in results:

Changes in version 24.1

STEEL

- Automatic code selection for anchoring check
- Meshing around bolt and pin holes improved

Read the highlights and the full list of improvements in Release notes IDEA StatiCa 24.1.

Changes in version 24.0

STEEL

- Out of surface load warning in Member (patch 23.1.1)
- Extend the member using the cut operation

CONCRETE

- Accurate calculation of the shear resistance of slabs (patch 23.1.2)

BIM and CLOUD SERVICES

- Checkbot Free structural design hub for all supported FEA and CAD and terminating of the free Viewer plugins

Read the highlights and the full list of improvements in Release notes IDEA StatiCa 24.0.

Changes in version 23.1

STEEL

- Welds – autodesign, input, warnings, visualization
- Shear force position input and visualization (since patch 23.0.5)
- Yield strength reduction for high-strength steel hollow sections
- Warning for welds and bolts connecting the same plates (since patch 23.0.4)
- Filler plate (packing plate) recognition (since patch 23.0.3)
- Detailing improvements for bolts and welds in Eurocode (since patch 23.0.2)
- Limitations to checks of anchors (since patch 23.0.2)
- Detailed calculation of connection design material values displayed in the plates' result table
- AISC steel and bolt grade materials for AISC360-22 (since patch 23.0.4)
- Singularity detection in Member (since patch 23.0.3)

CONCRETE

- Interaction code-check advancements in RCS (since patch 23.0.2)

Read the full list of improvements in Release notes IDEA StatiCa 23.1.

Changes in version 23.0

STEEL

- Qualification checks of seismic prequalified connections for AISC
- Update of ANSI/AISC 360-22, CSA S16:19 standards, and Taiwain sections
- Anchors with stand-off (since patch 22.1.5)
- Eurocode updates to thin-walled members and anchors
- Detailing improvements for bolts and welds in Eurocode (since patch 23.0.2)
- Limitations to checks of anchors (since patch 23.0.2)
- Edge indexing in Member and Connection models
- Load Extreme Selection (since patch 22.1.3)

CONCRETE

- Limited stress check feature in Detail
- Implementation of long-term losses in Detail
- Improvements for ACI 318-19 in Detail

- Imperial rounding improvements in Detail
- Equivalent time for deflection in Beam (since patch 22.1.3)
- Triangular mesh in concrete Member
- Interaction code-check improvements in RCS (since patch 23.0.2)

Read the full list of improvements in Release notes IDEA StatiCa 23.0.

Changes in version 22.1

STEEL

- Angle to the grain of the steel-to-timber connections

CONCRETE

- GMNIA solver extended to shear and torsion effects

Read the full list of improvements in Release notes IDEA StatiCa 22.1.

Changes in version 22.0

STEEL

- Loads in equilibrium by default
- Fatigue analysis - how the results are displayed
- Through bolts for hollow sections warning
- Connection Lite update

CONCRETE

- Update of eccentricity definition of a normal force for concrete columns according to Eurocode

Read the full list of improvements in Release notes IDEA StatiCa 22.0.

Changes in version 21.1

STEEL

- Bearing type options for bolts in version 21.1 and onwards
- Weld checks specifics as per Eurocode (EN) and Indian Standard (IS)
- Slip resistance check update according to SP 16

For users of **Template Manager**, use version 21.1 to transfer your designs into the new Connection Library.

Read the full list of improvements in Release notes IDEA StatiCa 21.1.

Changes in version 21.0

The biggest change since version 21.0 is in the analytical model of members in Connection, please read the Updated CBFEM solver article, and the Condensed superelements - invisible but essential blog post. The impact of the changes is also described in Analysis model improvements in IDEA StatiCa version 21.0 article.

Read the full list of improvements in Release notes IDEA StatiCa 21.0.

Changes in version 20.1

Here, we point out articles describing the main differences between the older and newer version.

- Improved model of contacts
- Butt welds upgraded model
- Bolt bearing distances for Eurocode
- Connecting plate eccentricity

Read the full list of improvements in Release notes IDEA StatiCa Steel 20.1 and Release notes IDEA StatiCa Concrete 20.1.

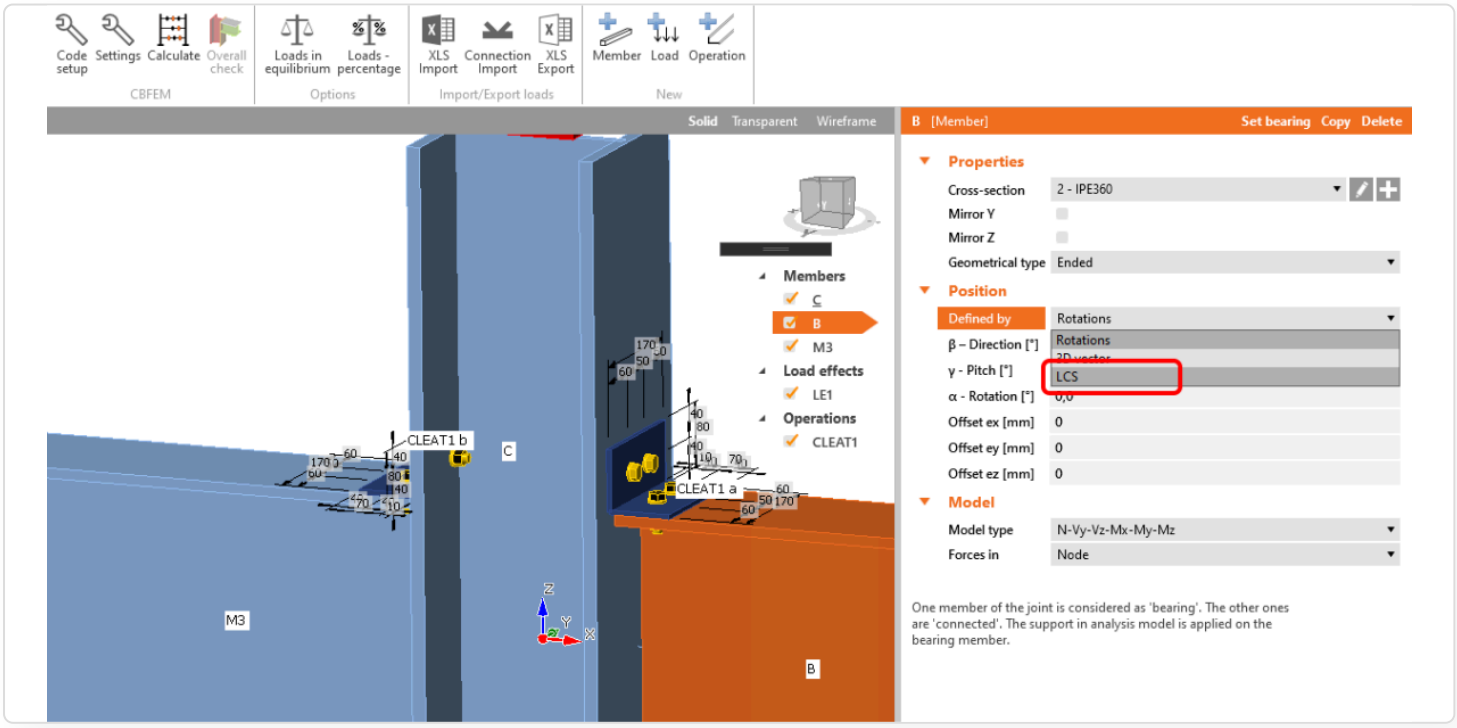
Changes in version 20

Cleat manufacturing operation refactoring

In the previous version Cleat manufacturing operation sometimes made L cross-section legs swapped, mostly while the members were rotated along the longitudinal axis. From now on, the L sections are positioned correctly, and the legs orientation is kept the same while introducing rotation to the member. The new mechanism behind is based on the new member positioning (by its LCS coordinates - rather then it's Rotations).

Local coordinate systems of the members in FEA/CAD applications

All member entities in FEA/CAD applications are created in a way that their definition axis has the start and the end. These two points are taken as a vector to define the local coordinate system of the member. In each FEA/CAD application project are these data information stored and can be used during the import. We took advantage of that, and it helps to improve the correctness of our BIM link geometry import. On the other hand, it changes the rules of the game, and the user must pay attention to the way how the model is created in FEA/CAD because it has an impact on the CBFEM Connection model also. We recommend to pay attention while importing from FEA/CAD projects into the version 20, the local coordinate system of the members may change the model to and difference compared to the model imported into the previous versions is eminent.



Member application projects compatibility

IDEA StatiCa Member application passes through agile development, especially the data storage architecture and the guided user interface. Based on this it's obvious that the projects created in the older versions may not be correctly opened in version 20 or the application may fail to open them. Please be aware of that and excuse IDEA StatiCa for inconvenience.

Read the full list of improvements in Release notes IDEA StatiCa Steel 20.0 and Release notes IDEA StatiCa Concrete 20.0.

Changes in version 10.1 and older

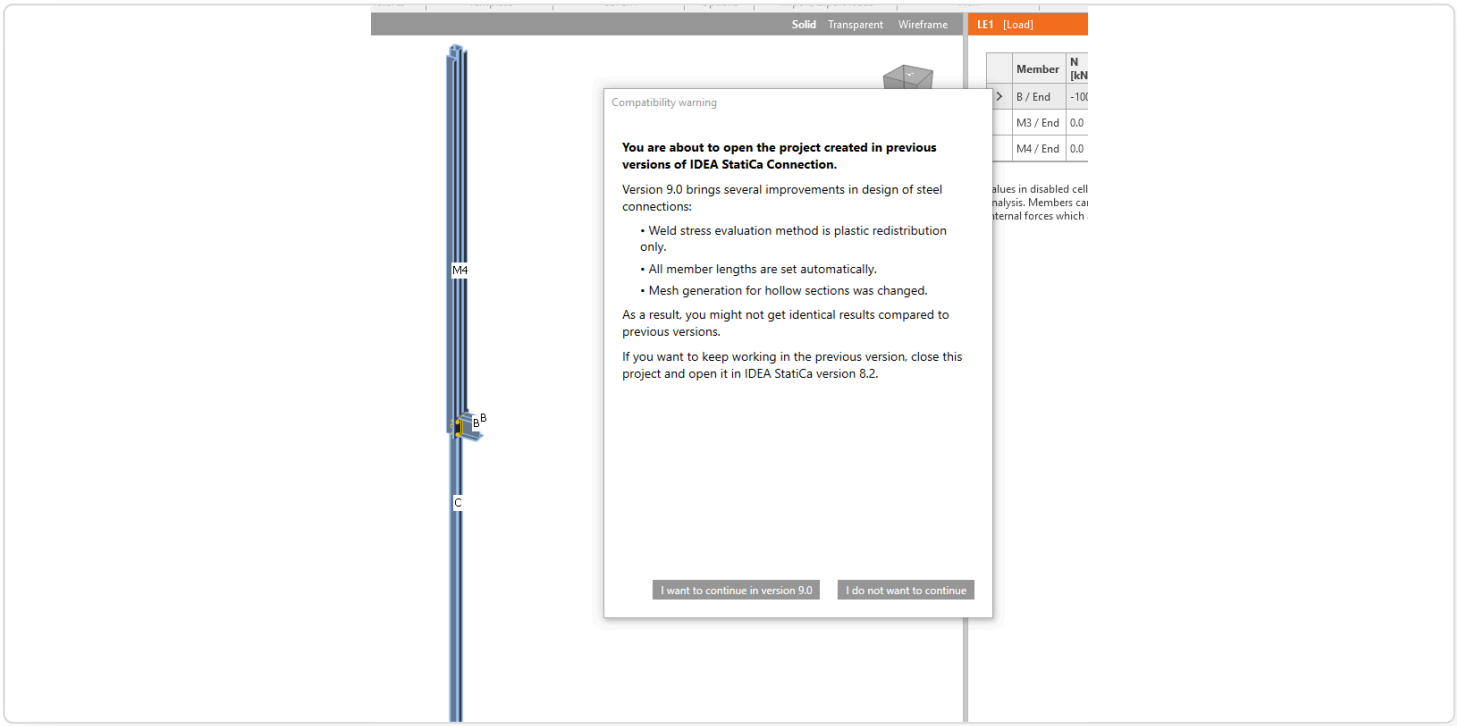
One of the reasons behind the new GUI is that a lot of engineers were saying: „IDEA StatiCa Connection needs to be more error-proof". These "errors" are usually related to:

- Setting the correct **length of members** – in cases of extremely short or long members, this can hugely influence the results. Since version 9, IDEA StatiCa Connection automatically sets an appropriate length of all members.
- **Welds** – stress plastic redistribution is by far the most accurate design method for welds and was introduced in version 7.1. During version 8 – and as a transition period - it was the default method, coexisting with the other evaluation methods. Since version 9, this method is the only option available and the other evaluation methods have been removed, to avoid confusion among the users, as we have seen through our helpdesk. This ensures that all welds in the project are safely designed and complying with the code.

We have implemented several **control mechanisms** for IDEA StatiCa Connection since version 9.0 – automatic-check when the connection is modeled in a recommend way (singularity check, member lengths, their offsets, ...). If the connection is not modeled properly, the calculation is interrupted or an error message is displayed. Please be aware that because of all of these improvements, opening projects from previous versions can lead to a different model geometry that may need some further editing.

We have also improved **meshing** of hollow sections members – IDEA StatiCa Connection since version 9.0 generates finer meshing on hollow section members which might lead to slightly different results compared to older versions.

We display the compatibility warning on each opening of a project from the older versions:



Full release notes

Below, you can download the **Release notes** for IDEA StatiCa 24.1 in PDF.

DOWNLOAD PDF →