
RELEASE NOTES

# Release notes IDEA StatiCa 25.1

Oct 21, 2025

## Table of Contents

---

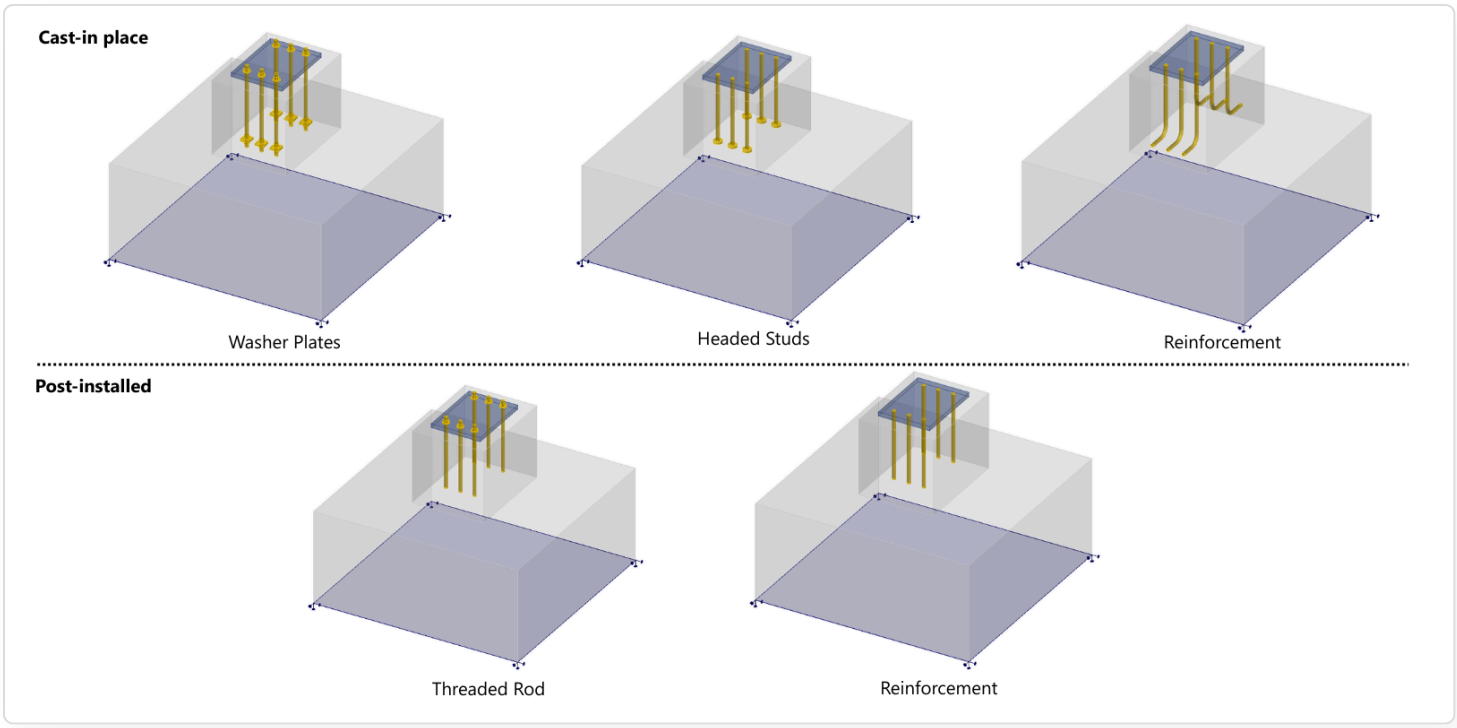
Version 25.1 is focused on anchoring, workflows for big BIM models, and the first FEA link for concrete shear walls

# Concrete Design

## Single anchor definition

Cast-in-place or post-installed anchors? IDEA StatiCa Detail provides all possibilities for reinforced concrete blocks, allowing engineers to design and analyze complex anchorage systems, including washer plates,

headed studs, and threaded rods.
The following article provides a complete overview of all anchoring cases that can be defined in IDEA StatiCa Detail.




**Cast-in-place**

1 Reinforcement (from version 25.0)

2 Washer plate (from version 25.1)

3 Headed stud (from version 25.1)

**Post-installed (adhesive) anchors**

1 Reinforcement (from version 25.0)

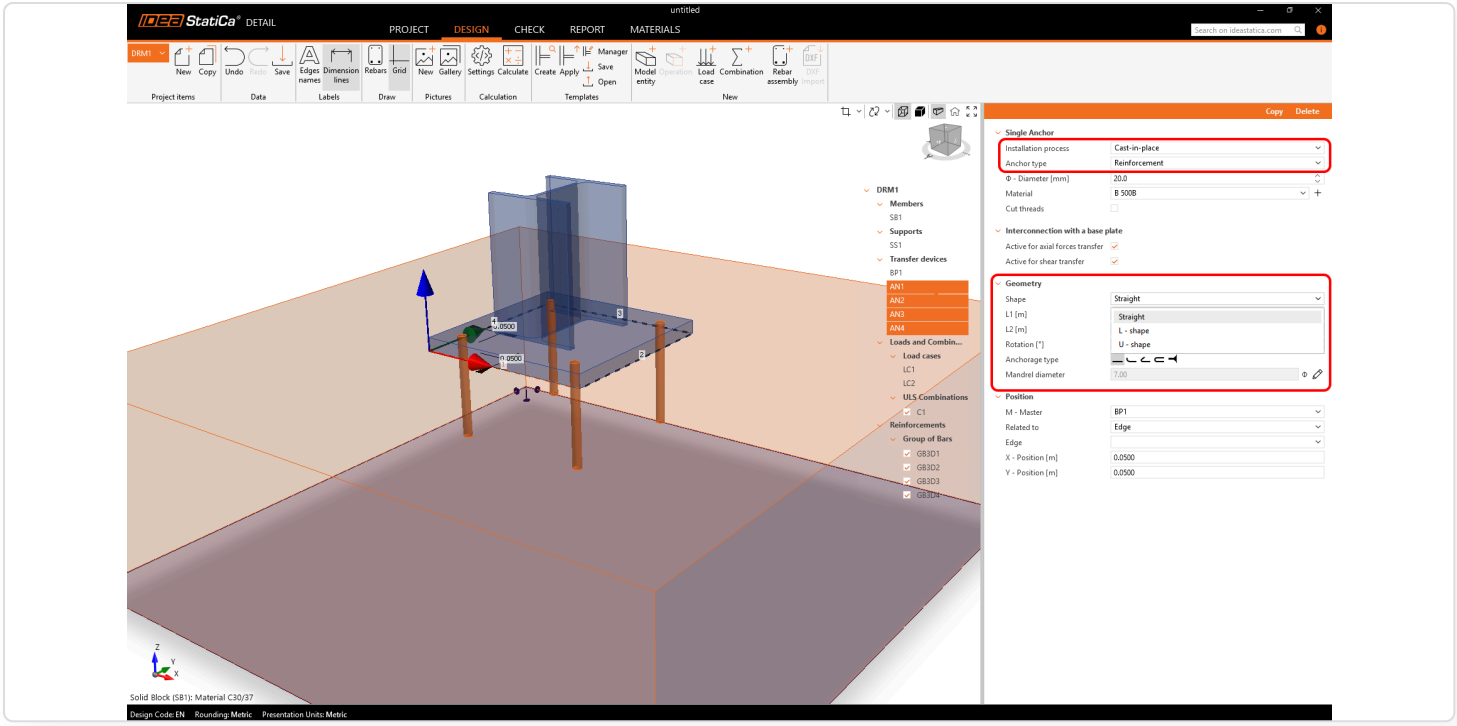
2 Threaded rod (from version 25.1)

# Cast-in-place

**1. Reinforcement**

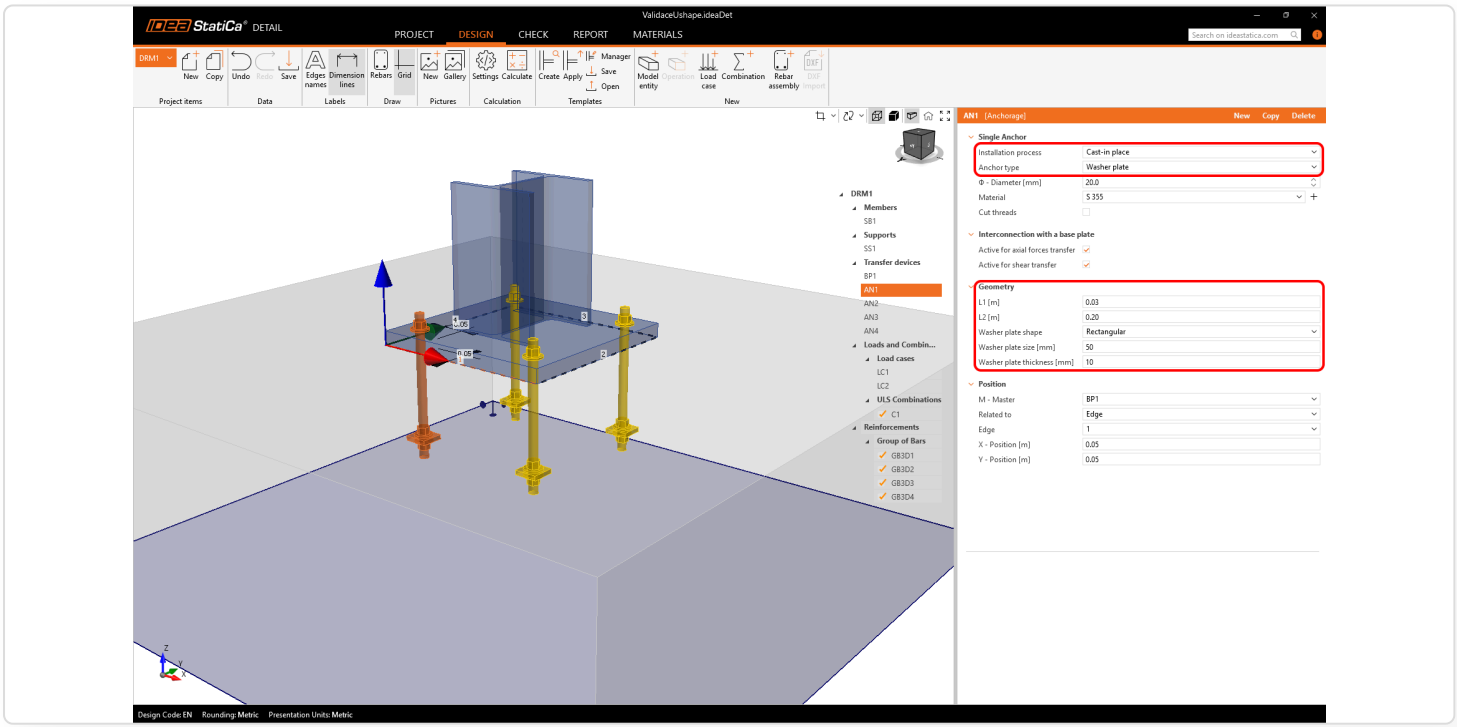
Fasteners can be defined directly for each specific plate. Users may add an unlimited number of fasteners and even combine different types within a single plate. These are modeled as ribbed reinforcement embedded in concrete, with bond strength calculated according to code provisions in the same way as for standard reinforcement. At the anchor end, an anchorage type can be specified, functioning identically to reinforcement anchorage. In this case, an anchorage spring is applied, with the β-factor determined by the selected code.

- **Straight:** Anchors can be arranged in rows and positions following a predefined key (similar to bolt placement in Connections). Material, length, and diameter, as well as specify an anchorage type can be defined.
- **L-shape:** Defined as two rows of anchors with adjustable spacing and row distance, with the option to specify anchorage type.
- **U-shape:** A continuous series of reinforcements forming a U-shape, connected on both sides.



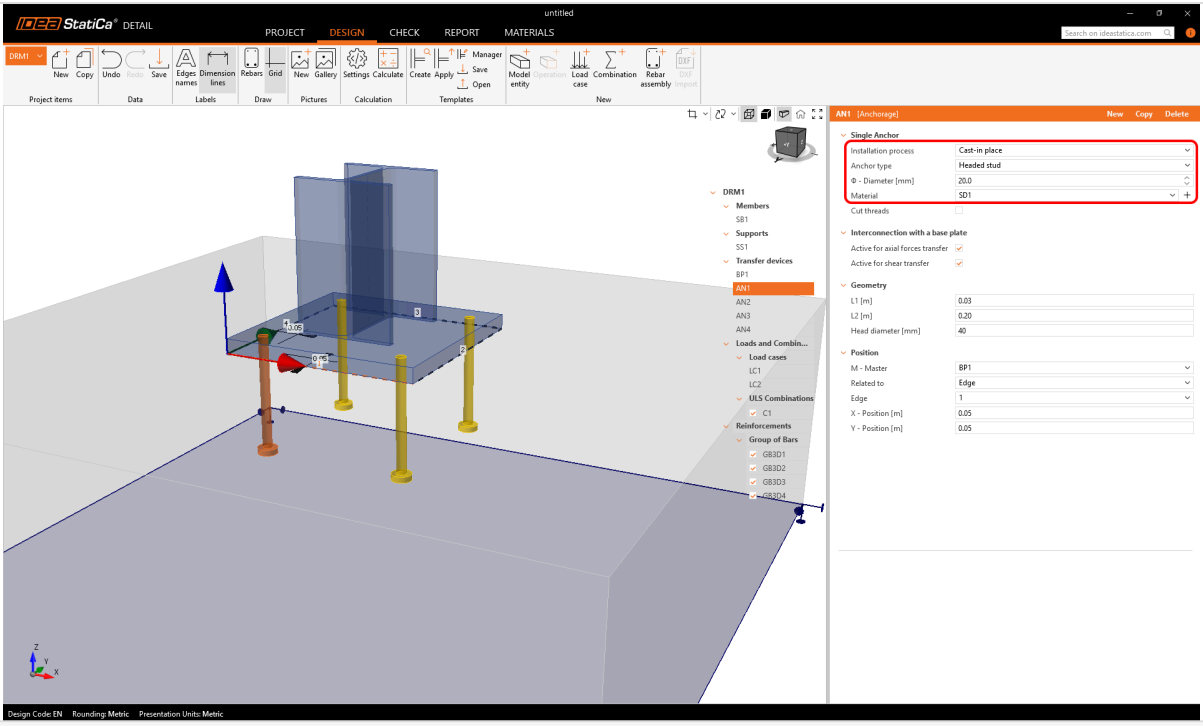

### 2. Washer plate

The washer plate is modeled as a plate-shell element directly attached to the anchor shank, transferring load to the concrete exclusively through compression contact. It can be defined in either a circular or square shape, with fully customizable dimensions. The plate itself is modeled linearly, without plasticity, and is not subjected to resistance checks. Since the shank has **zero bond strength**, the entire load is transferred to the concrete through the washer plate.

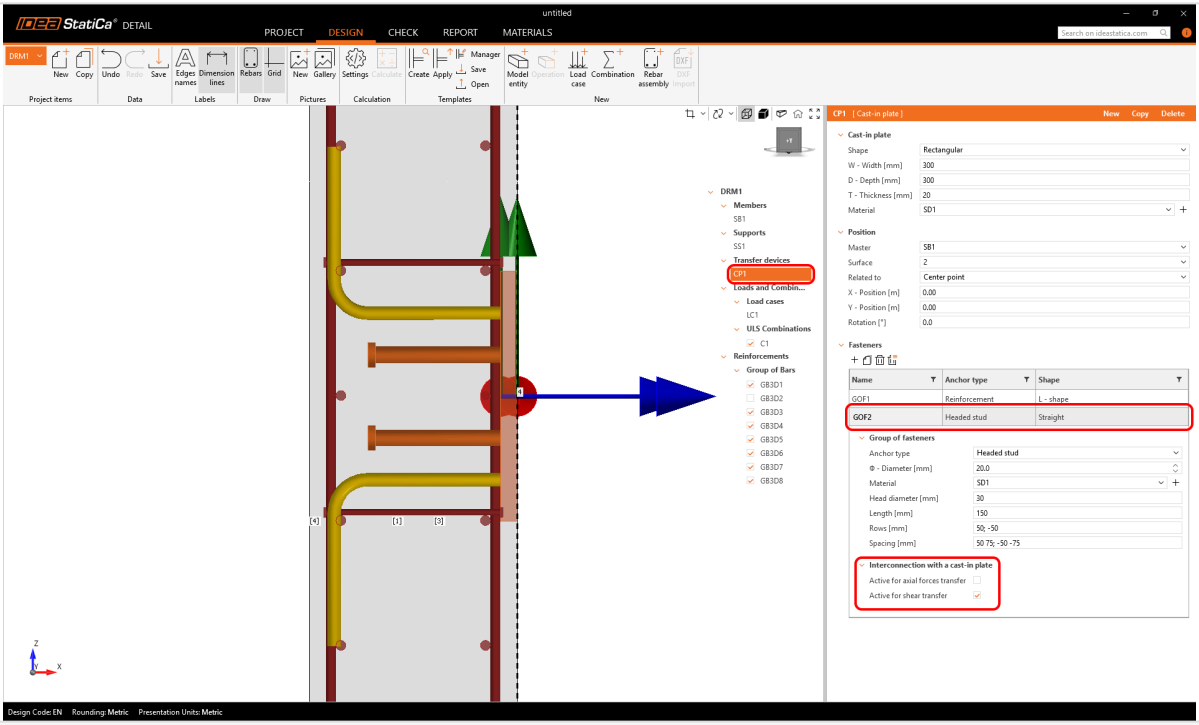



### 3. Headed stud

This element mechanically behaves like a washer plate, but is assigned corresponding material grades available in the MPRL. Headed studs can also be arranged in groups within the Cast-in place entity.




A new control option allows defining which anchor group transfers specific load types, for example, assigning shear to studs while reinforcement carries tension and compression.



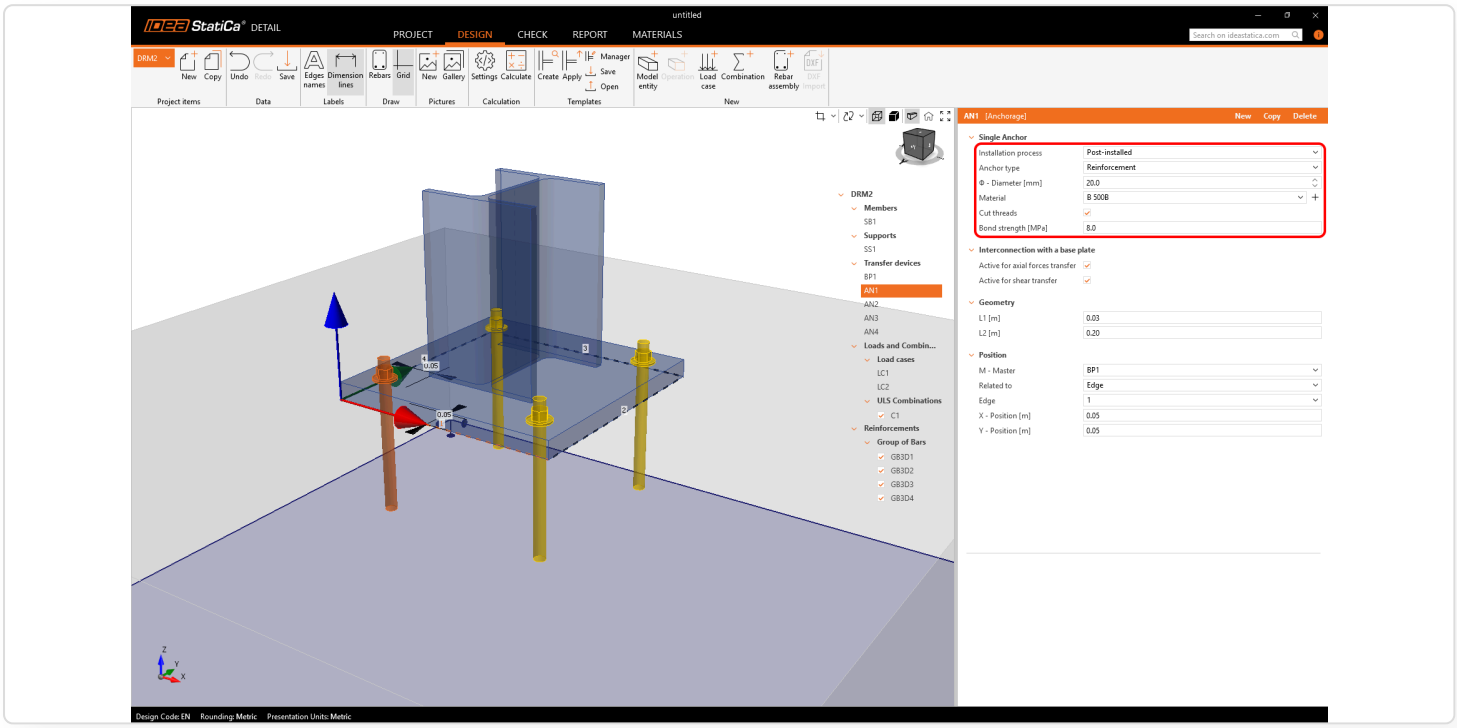

In addition, a stop criterion has been introduced to verify pull-out capacity in accordance with code requirements. Limit values for head-to-concrete or washer plate-to-concrete contact are calculated per the relevant standards and presented in a dedicated table, including the normative reference and the governing formula for the limit force.

| Code | Chapter | Equation | Calculation example | Note |
|---|---|---|---|---|
| EN 1992-4 | 7.2.1.5 | $N_{Rd,p}=k_2*A_h*f_{ck}/Y_c$ | $7.5*A_h*f_{ck}/1.5$ | According to the EN, there is the same Yc for concrete and for pull-out. |
| ACI 318-19 | 17.6.3.2.2a | $N_{pn}=\phi*\psi_{c,P}*8*A_{brg}*f'_c$ | $0.7*1*8*A_{brg}*f'_c$ | Strength reduction factor is according to Table 17.5.3c = 0.7 |
| AS 5216:2021 | 6.3.4 | $N_{Rd,p}=\phi_{Mp}*k_2*A_h*f_{cd}$ | $1/1.5*7.5*A_h*f'_c$ | $\phi_{Mp}$ is accroding to Table 3.2.4 |

# Post-installed anchors

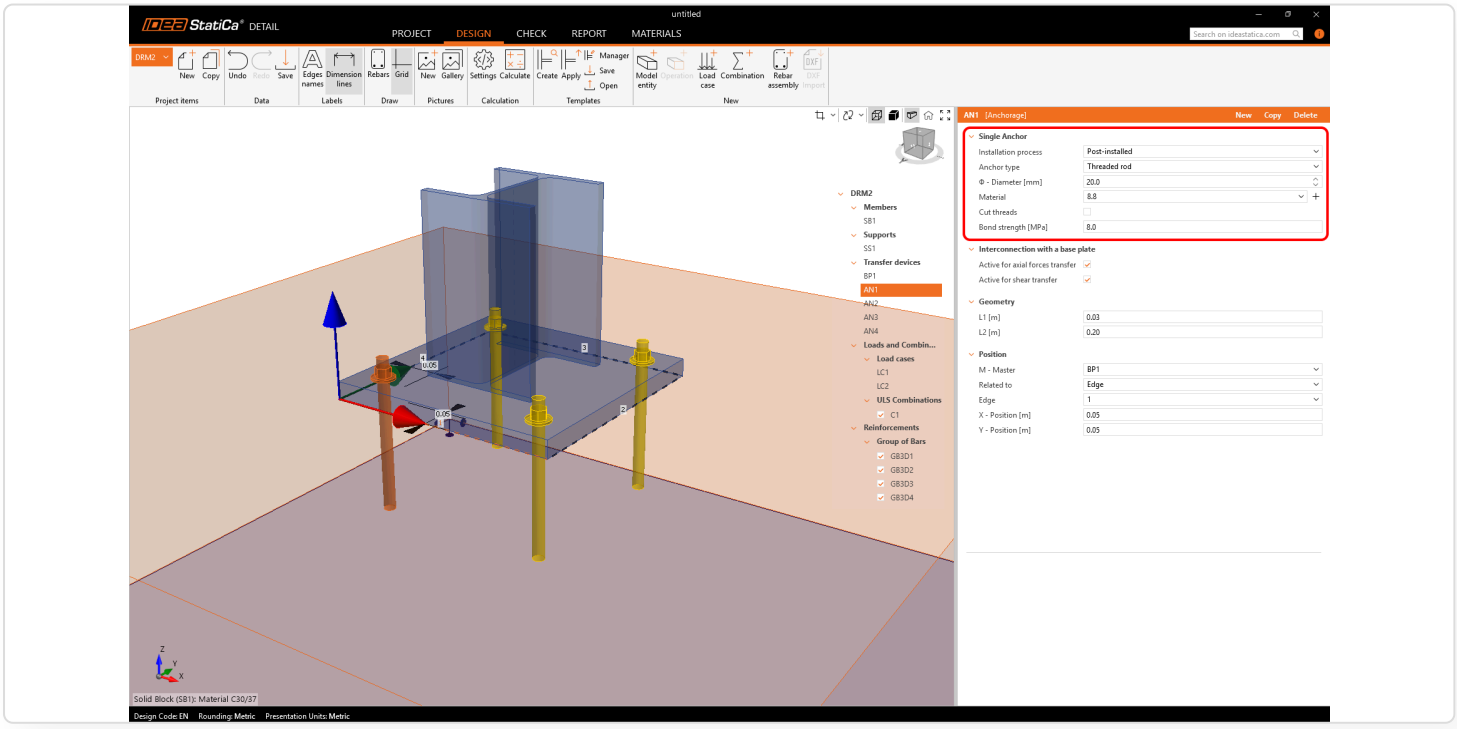
### 1. Reinforcement

These anchors are designed as bars installed into drilled holes and bonded to the surrounding concrete using adhesive. The **design bond strength** is specified directly by the engineer based on the technical documentation of the chosen adhesive product.




### 2. Threaded rod

It works in the same way as post-installed reinforcement, but with the advantage that the MPRL offers predefined standard material grades specifically for threaded rods.



# Integration with IDEA StatiCa Connection and codes

These options are also consistent with IDEA StatiCa Connection. How this is aligned to different standards can be seen in the following table.

| Installation process | Anchor type | Stand-off | Connection | | | Compatibility | | | 3D Detail | | |
|---|---|---|---|---|---|---|---|---|---|---|---|
| | | | EN | ACI | AUS | EN | ACI | AUS | EN | ACI | AUS |
| Cast-in-place | Reinforcement | Direct | ☑ | ☐ | ☐ | ☑ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Mortar | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Gap | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | Washer plates | Direct | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | | Mortar | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | | Stand-off | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | Headed-studs | Direct | ☑ | ☐ | ☐ | ☑ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Mortar | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Gap | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | Hooked threaded-rods | Direct | ☑ | ☑ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |
| | | Mortar | ☑ | ☑ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |
| | | Gap | ☑ | ☑ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |
| Post-installed | Reinforcement | Direct | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Mortar | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | | Gap | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☑ | ☑ | ☑ |
| | Threaded-rods | Direct | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | | Mortar | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | | Gap | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ | ☑ |
| | General anchor | Direct | ☑ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |
| | | Mortar | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |
| | | Gap | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ | ☐ |

| | |
|---|---|
| **3D Detail** | The functionality is implemented in the 3D Detail |
| **Compatibility** | process, anchor type and stand-off between Connection and 3D Detail |
| **Connection** | The functionality is implemented in the Connection |
| Note | Notification about not supported hooked anchors in Detail, and washer plate anchors are created in Detail instead |

More about the anchoring plates (base plate, cast-in plate) options can be found in a separate article.

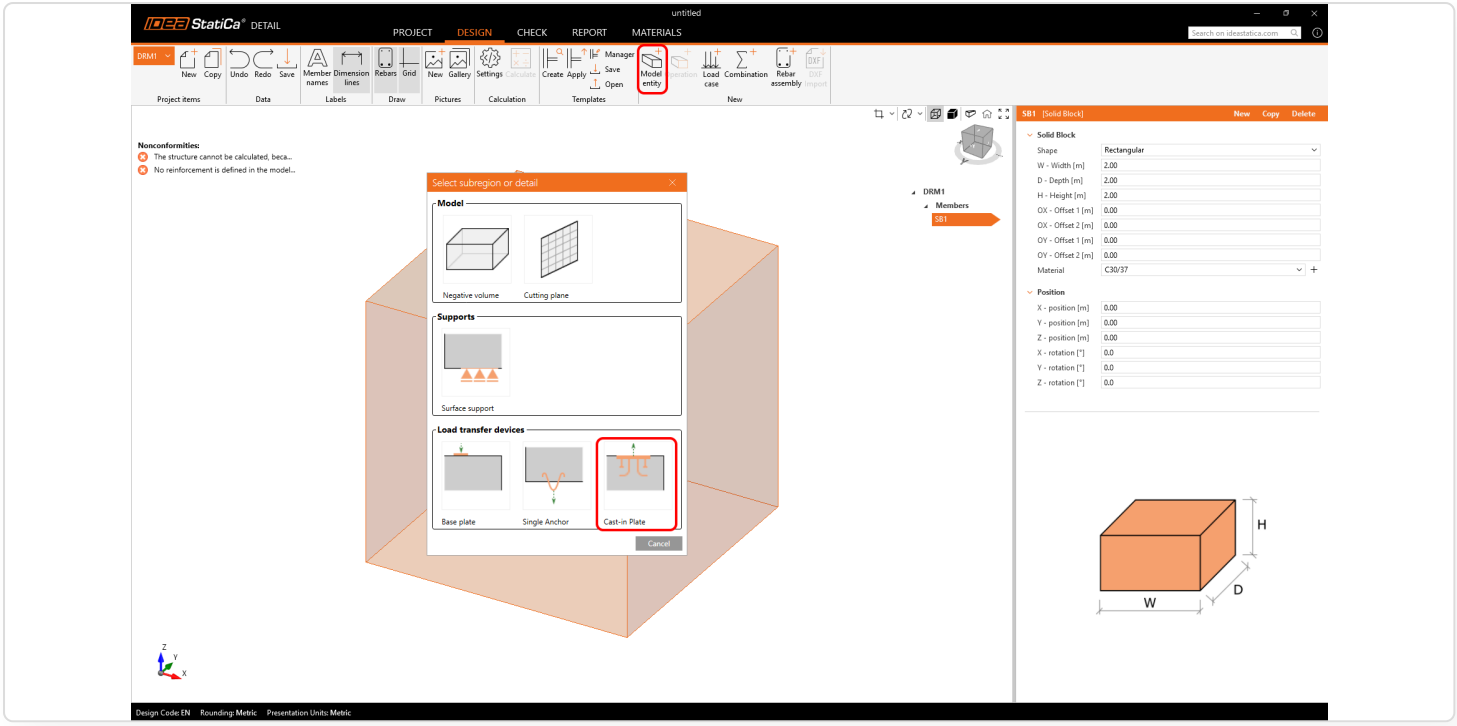
*Released in IDEA StatiCa version 25.0, updated in version 25.1.*

# Cast-in plates in Detail

When implemented in IDEA StatiCa Detail, cast-in plates unlock more anchoring possibilities for reinforced concrete blocks, allowing engineers to design and analyze complex anchorage systems. This integration enables optimized designs and a more efficient workflow in reinforced concrete detailing.
In Detail, you can find **cast-in plates** with fasteners/reinforcement of several types commonly used in construction for securing beams, columns, or façade elements. Cast-in plates ensure strong and reliable load transfer between steel and concrete structures.

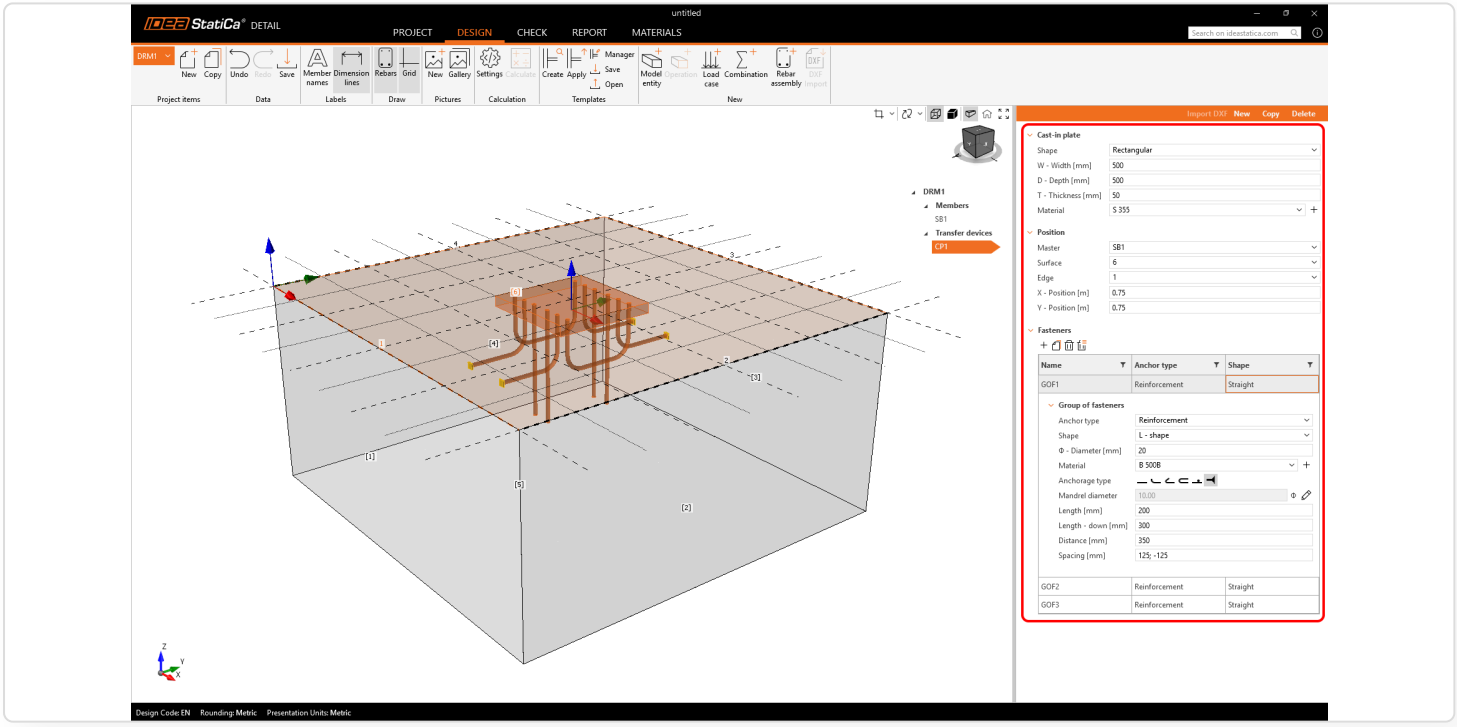
# Cast-in plates in Detail

Cast-in plates are available as a **Load Transfer Device** in the model entity selection. The **plate's geometry and position** can be defined in a single property grid, while an additional table allows users to add and combine multiple **groups of fasteners**. Fasteners can be defined directly for each specific plate. Users are able to add **an unlimited number** of fasteners and even combine multiple types within a single plate.

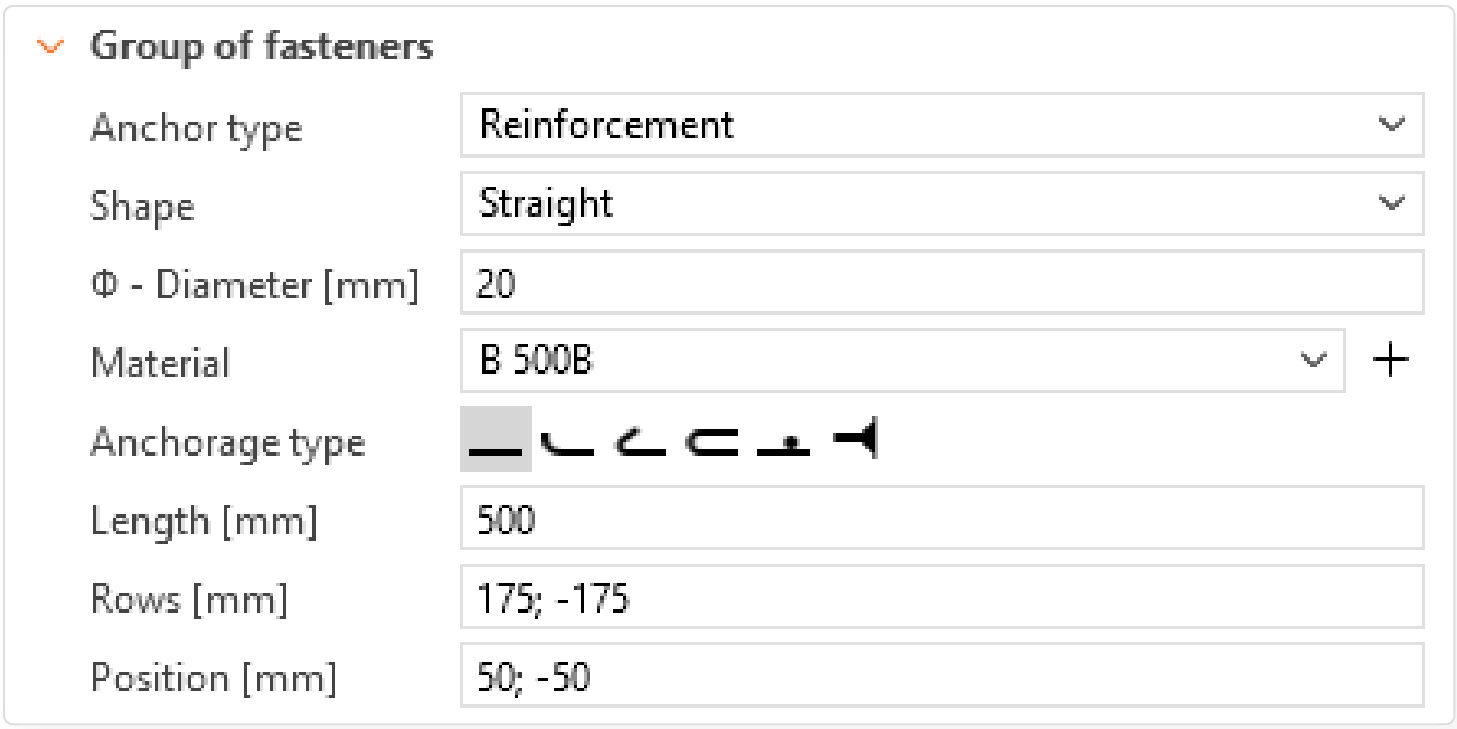



# Reinforcement types

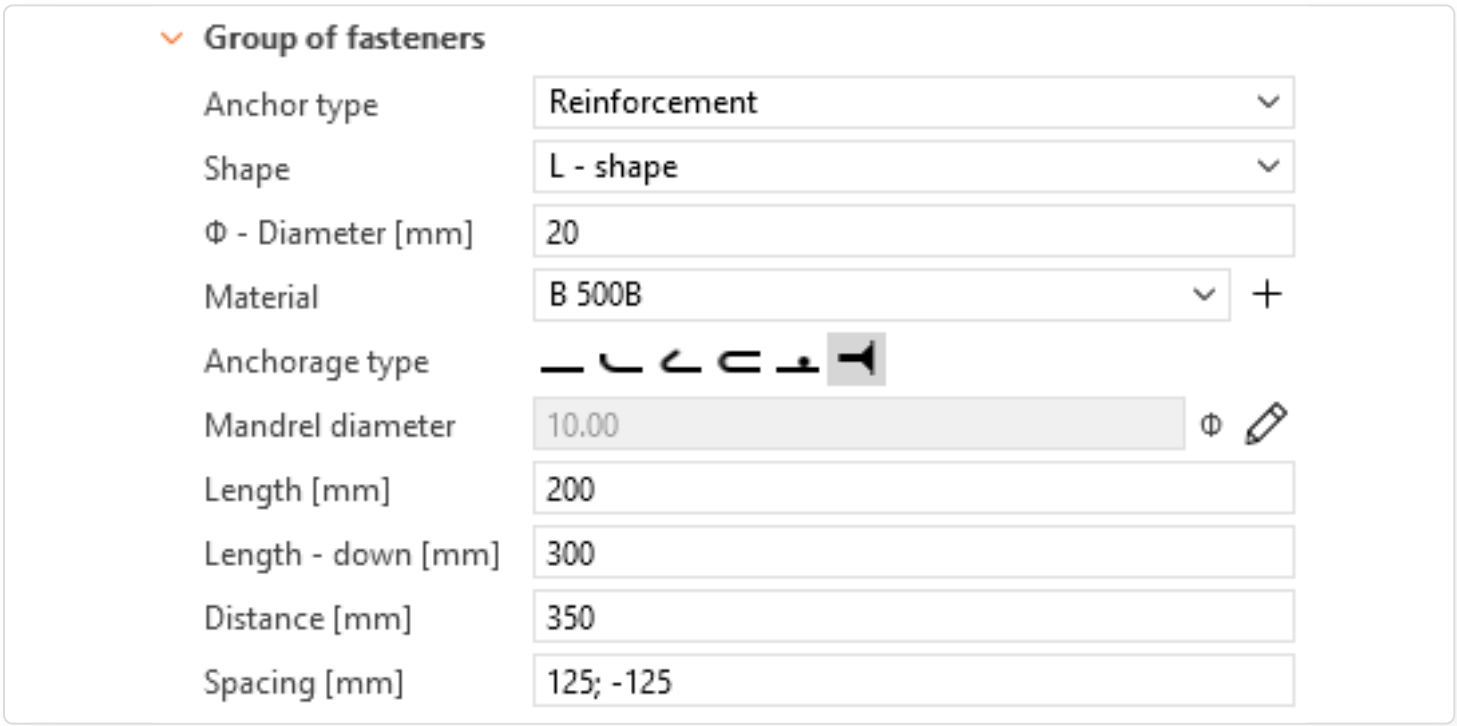
The Cast-in plate allows for **welded reinforcement** in three shapes: **Straight, L-shape, and U-shape**.



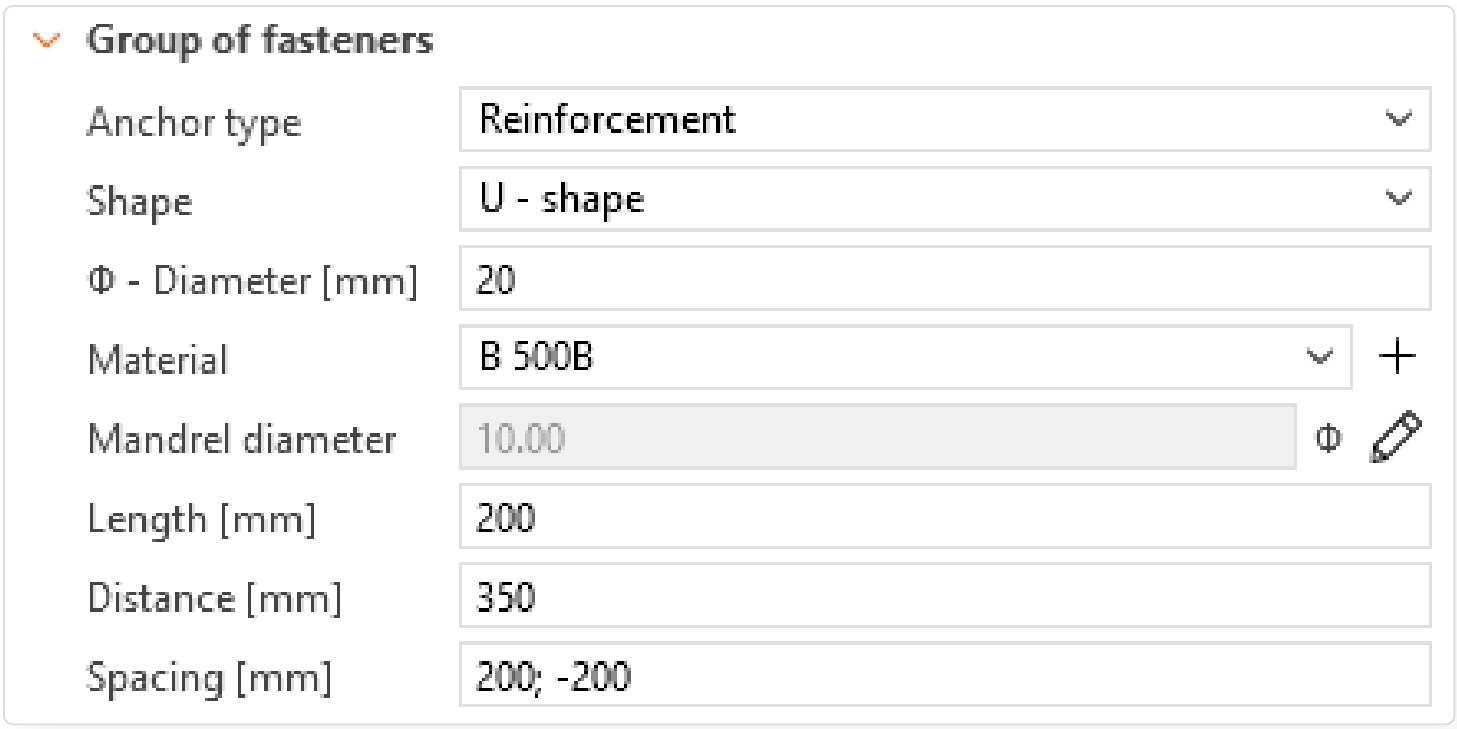

- **Straight:** Anchors can be arranged in rows and positions following a predefined key (similar to bolt placement in Connections). Users can define material, length, and diameter, as well as specify an anchorage type.




- **L-shape:** Defined as two rows of anchors with adjustable spacing and row distance, with the option to specify anchorage type.




- **U-shape:** A continuous series of reinforcements forming a U-shape, connected on both sides.

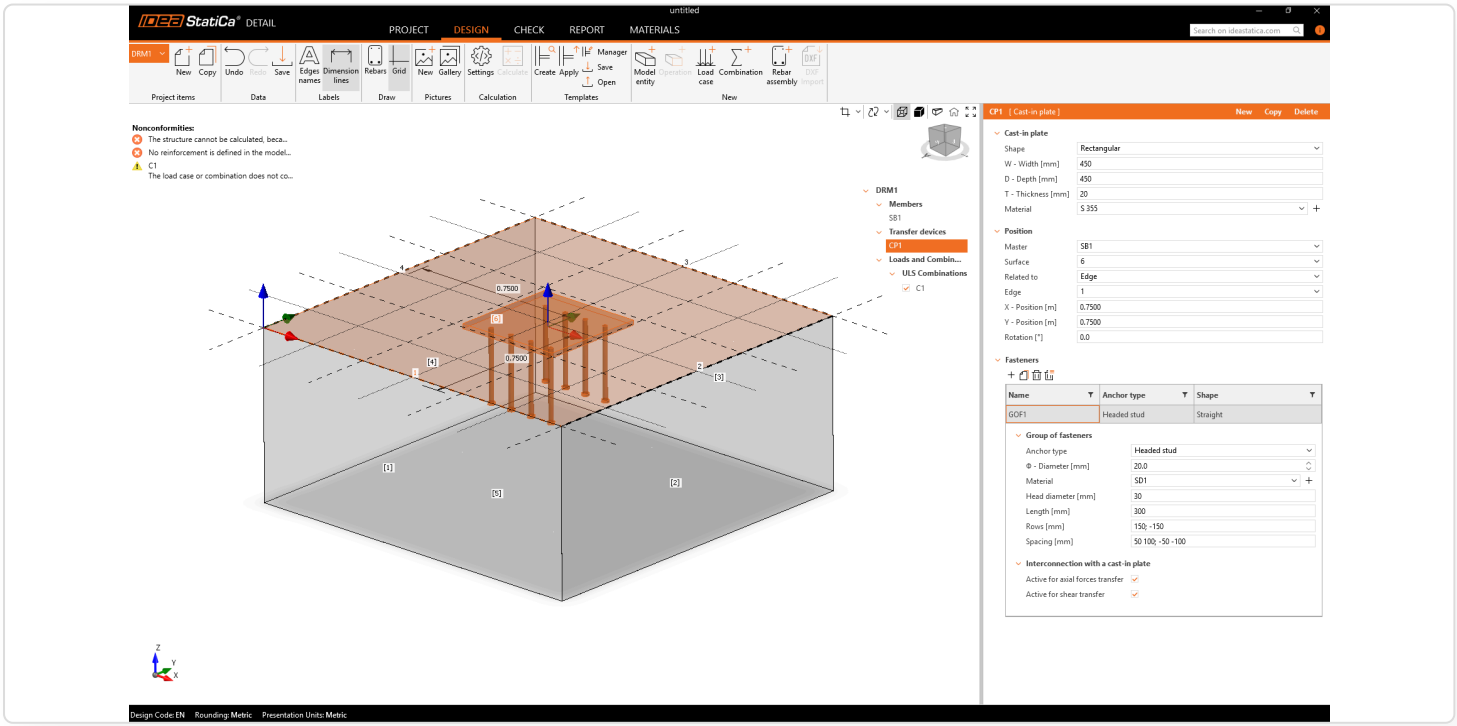



*Released in IDEA StatiCa version 25.0.*

# Headed stud

The headed stud transfers load to the concrete exclusively through compression contact. The head is modeled as a plate-shell element directly attached to the anchor shank with fully customizable dimensions. The plate itself is modeled linearly, without plasticity, and is not subjected to resistance checks. Since the shank has **zero bond strength**, the entire load is transferred to the concrete through the head.

This element is assigned corresponding material grades according to ISO 13918 available in the MPRL.



# Model Behavior

Anchors are treated as elements resisting both **shear and tension**, as they are welded to the plate. Their evaluation follows the same principles as standard anchors, with similar limitations (e.g., shear in the anchor cannot yet be assessed in the application). For more information, see the Theoretical Background.

A control option allows the defining of which group of fasteners transfers specific load types, for example, assigning shear to studs while reinforcement carries tension and compression.

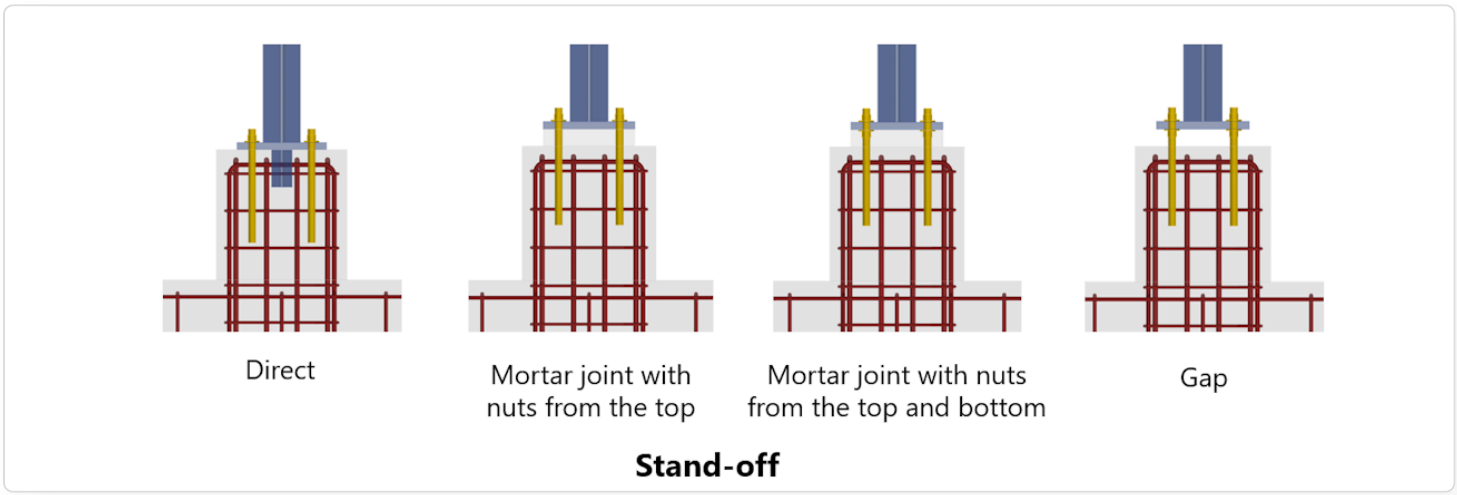
*Released in IDEA StatiCa version 25.1.*

# Anchoring plates options

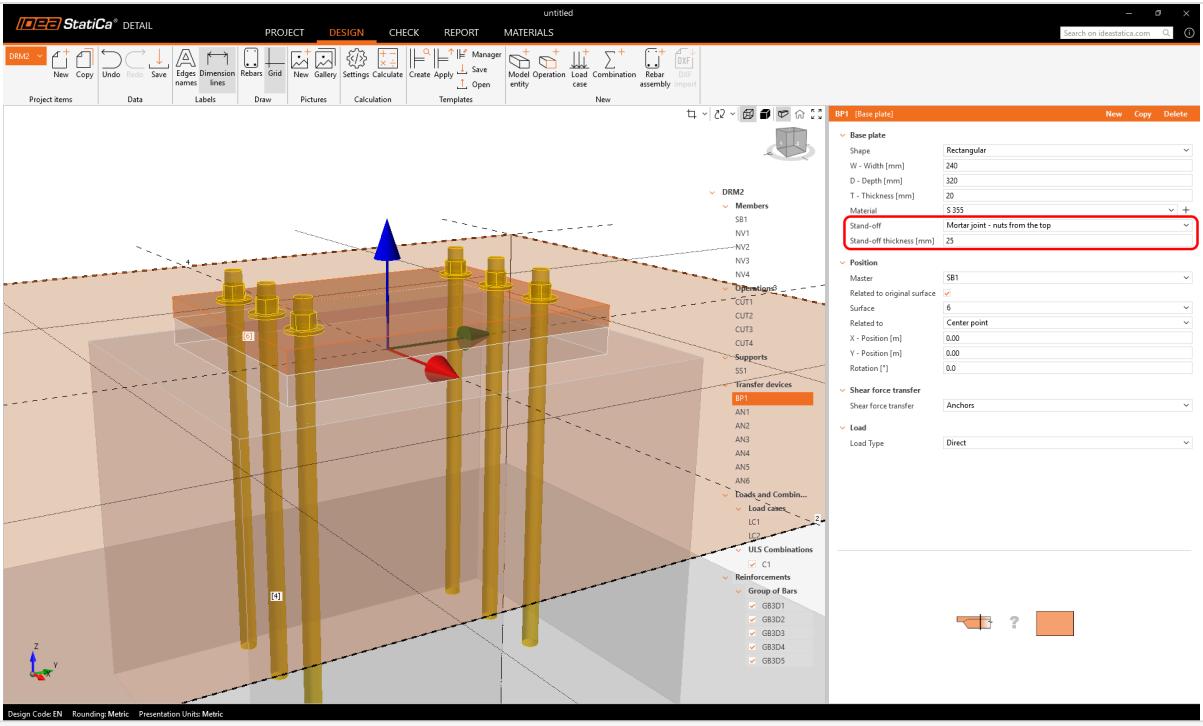
IDEA StatiCa Detail provides you with complete control over anchoring plate stand-off, including mortar joints and gaps. Together with cast-in plates, it offers users a comprehensive solution for most anchorage cases. Here is an overview of common stand-off scenarios, including **mortar joint**, **nuts**, and **gaps.** Further in the article, you can find an explanation of the **Cut thread** function, which, when used correctly, can help you achieve an accurate representation of hinged versus fixed anchor–plate behavior. The feature enhances physical accuracy without compromising calculation efficiency.

# 1. Base plate options

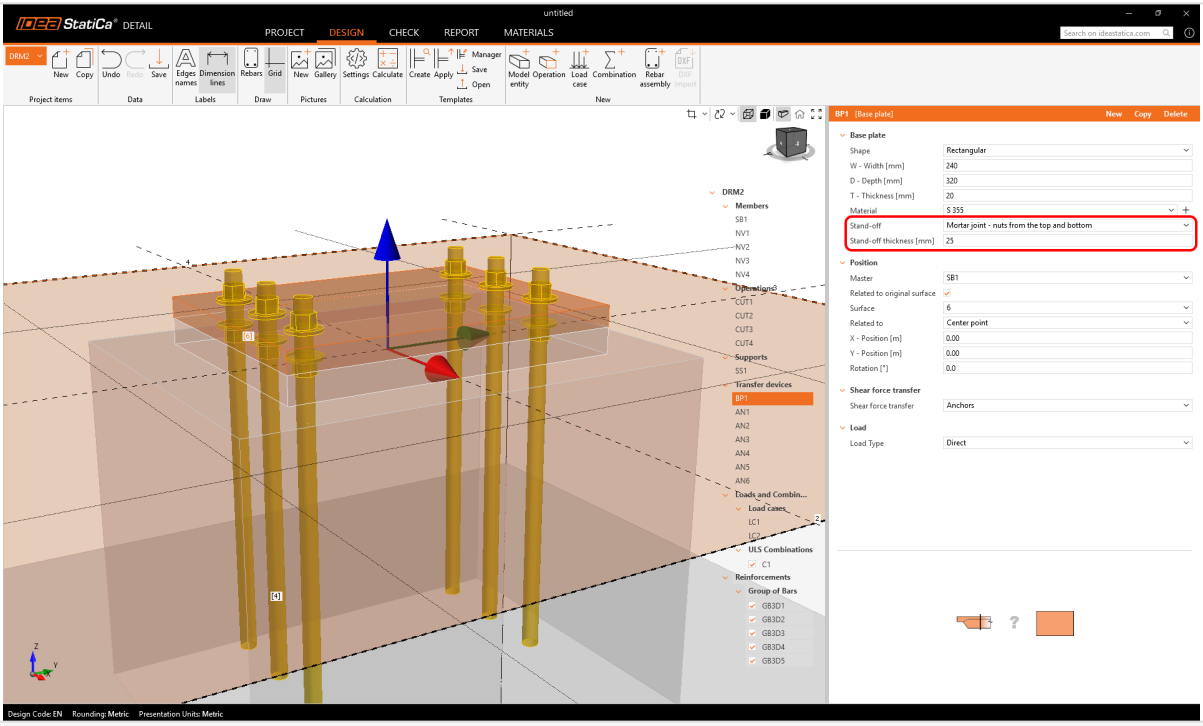
Stand-off



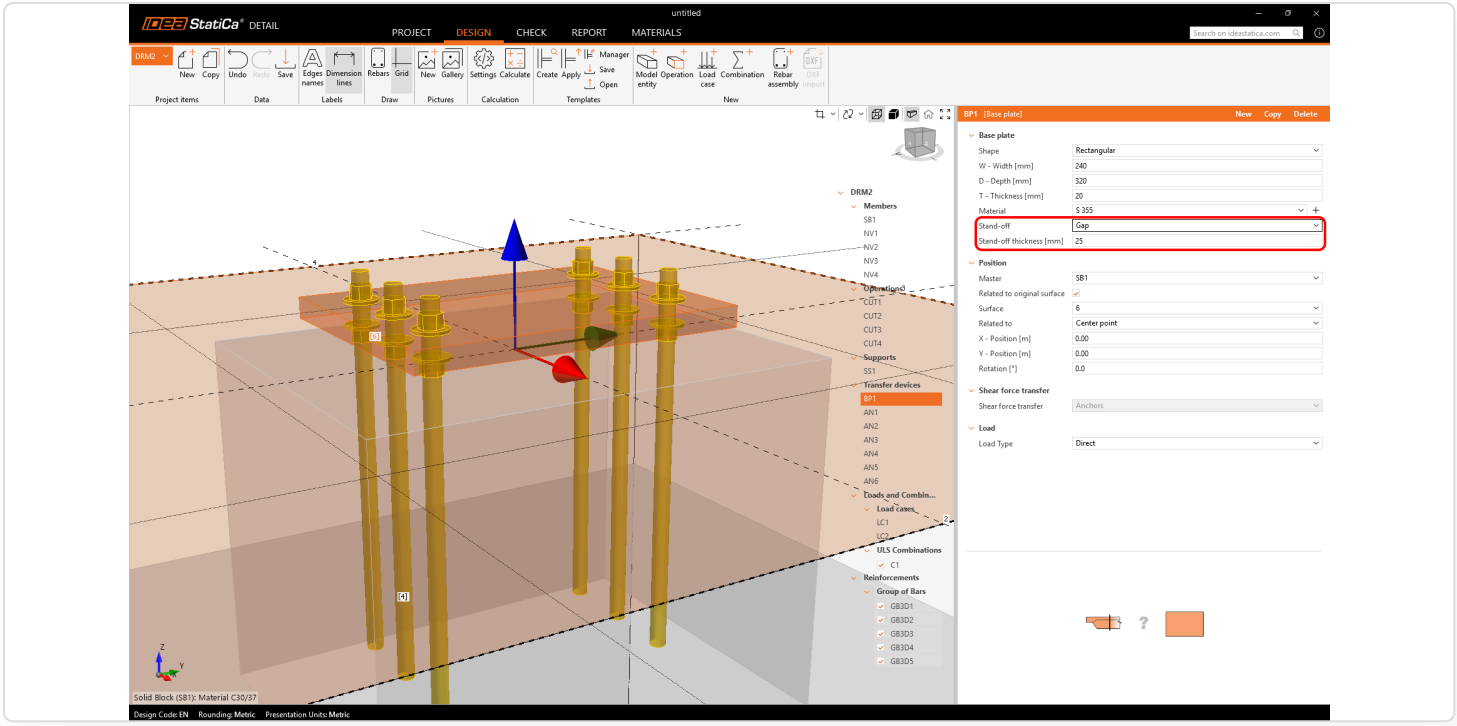

- **Mortar joint – nuts from the top**: a mortar layer can be defined with a specified thickness. The anchors are connected only from the top, modeling a hinged behavior.




- **Mortar joint – nuts from the top and bottom**: a mortar layer with nuts on both sides, allowing a fixed anchor–plate connection.




- **Gap**: a vertical clearance under the plate can be specified. Anchors are then directly loaded, with no contact between the plate and concrete.



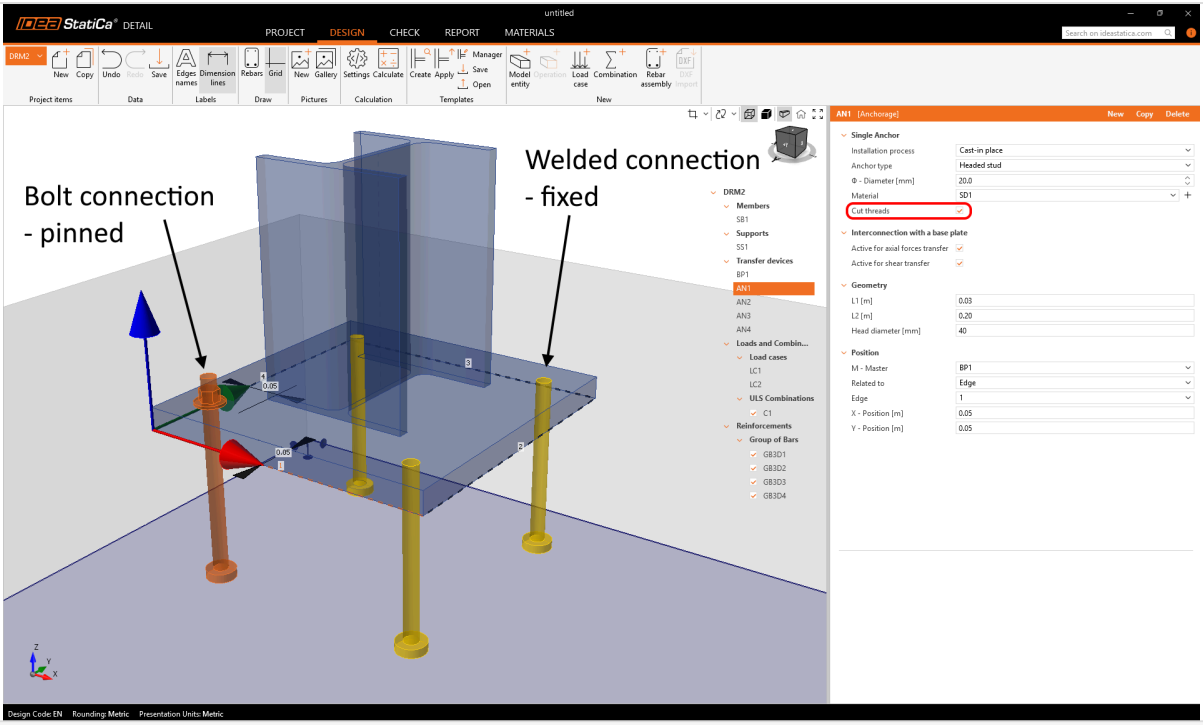

## Anchor types

Users can add **an unlimited number** of fasteners and even combine multiple types within a single plate. More information about anchor types (cast-in-place and post-installed) can be found in the Single anchor definition article.

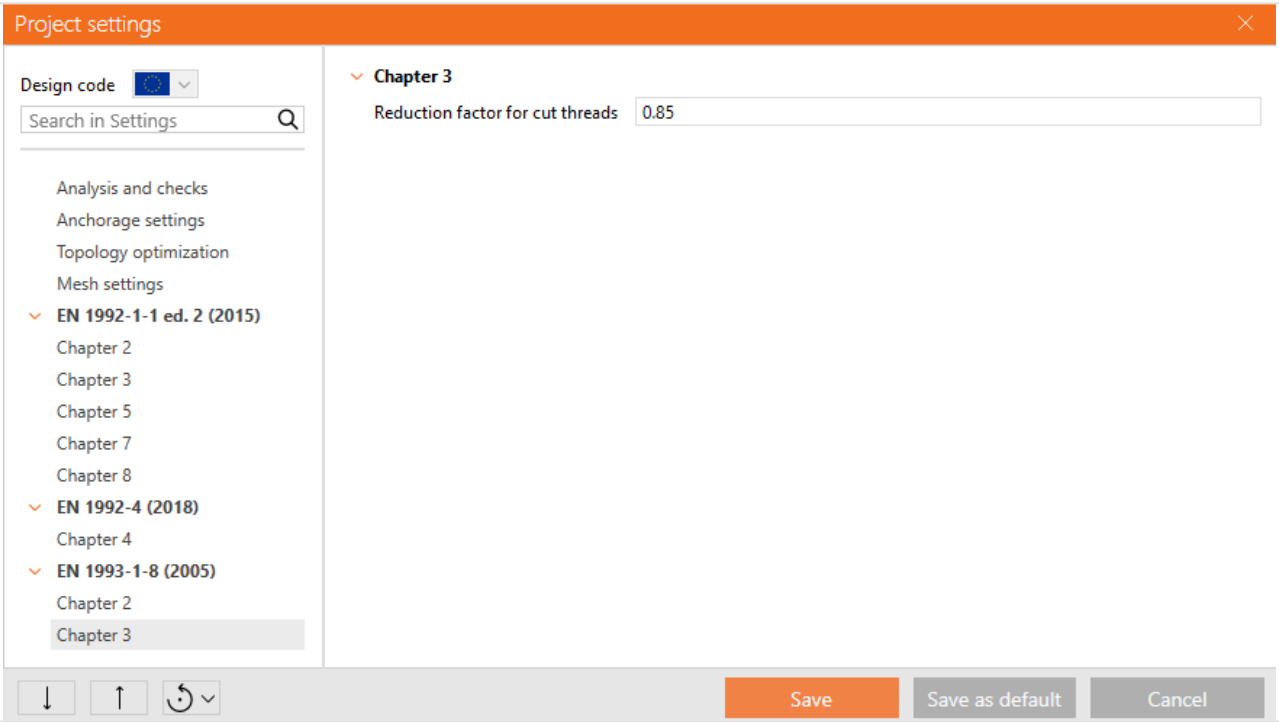
## Cut thread option

The checkbox in the anchor properties lets you control how the anchor connects to the base plate and defines what behavior to expect from the steel-concrete connection.

- For headed studs and cast-in reinforcement connected to the base plate (not for Cast-in plates), it distinguishes between a **bolt connection (pinned)** and a **welded connection (fixed)** — visible in the 3D scene.
- The way of anchor-to-plate connection has a significant influence on the shear resistance from the point of view of the bearing of the concrete




For EN, the resistance of the anchor with cut threads is reduced according to EN 1993-1-8 3.6.1 (3). It can be set in Project settings.




## Technical notes

- The mortar layer is modeled as a **shell element**, with its stiffness taken into account, while it is incompressible. This helps to redistribute local forces to the concrete and is valid for typical bedding thicknesses used in practice - 25-50 mm.
- The distinction between nuts only from the top (pinned interconnection between anchor and base plate) vs. top and bottom (fixed interconnection between anchor and base plate) strongly influences the shear capacity from the point of view of concrete bearing.

Below is a summary table explaining axial and rotational constraints between the base plate and the anchor.

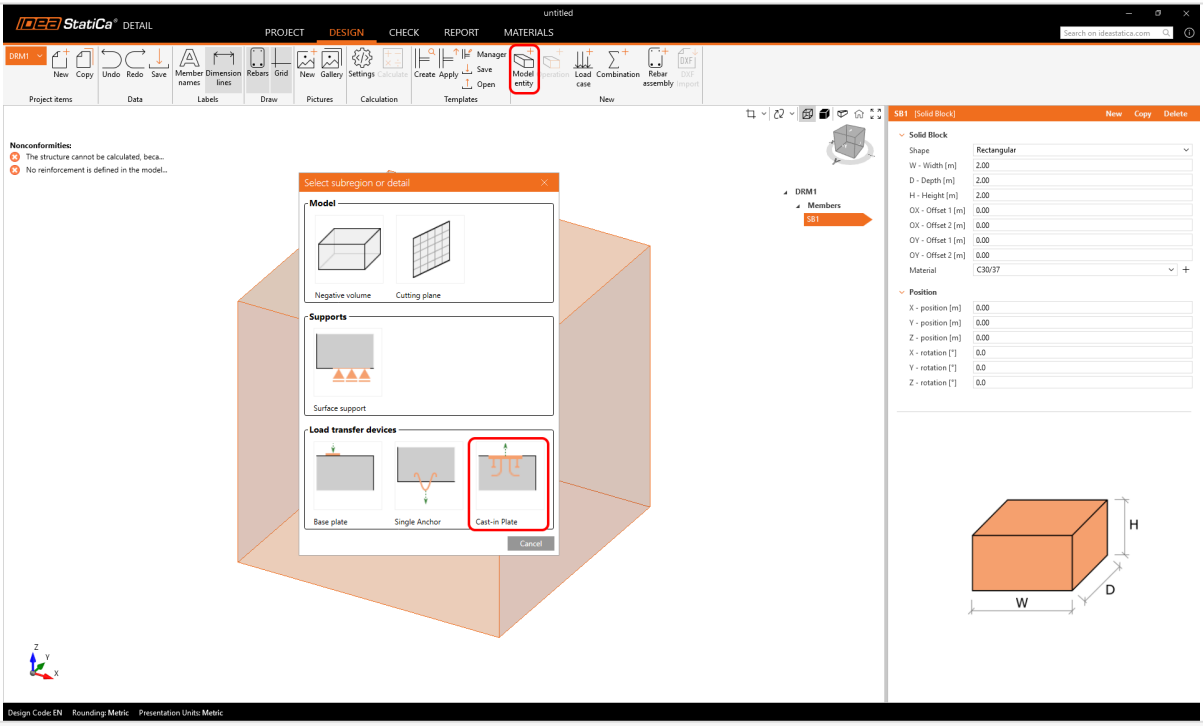
| Base plate installation (stand-off) | "Cut threads" is active | Installation process | Anchor type | Presentation in the scene | Anchor - base plate constraint | |
|---|---|---|---|---|---|---|
| | | | | | Axial constraint | Rotational constraint |
| Direct + Mortar joint (nuts from the top) | Yes | Post-installed | Reinforcement | Top nut | Tension only | Hinged |
| | | | Threaded rod | Top nut | Tension only | Hinged |
| | | Cast-in | Reinforcement | Top nut | Tension only | Hinged |
| | | | Washer plate | Top nut | Tension only | Hinged |
| | | | Headed stud | Top nut | Tension only | Hinged |
| | No | Post-installed | Reinforcement | Weld | Tension + Compression | Fixed |
| | | | Threaded rod | Top nut | Tension only | Hinged |
| | | Cast-in | Reinforcement | Weld | Tension + Compression | Fixed |
| | | | Washer plate | Weld | Tension + Compression | Fixed |
| | | | Headed stud | Weld | Tension + Compression | Fixed |
| Mortar joint (nuts from the top and bottom) + Gap | Yes | Post-installed | Reinforcement | Top + bottom nuts | Tension + Compression | Fixed |
| | | | Threaded rod | Top + bottom nuts | Tension + Compression | Fixed |
| | | Cast-in | Reinforcement | Top + bottom nuts | Tension + Compression | Fixed |
| | | | Washer plate | Top + bottom nuts | Tension + Compression | Fixed |
| | | | Headed stud | Weld | Tension + Compression | Fixed |
| | No | Post-installed | Reinforcement | Weld | Tension + Compression | Fixed |
| | | | Threaded rod | Weld | Tension + Compression | Fixed |
| | | Cast-in | Reinforcement | Top + bottom nuts | Tension + Compression | Fixed |
| | | | Washer plate | Top + bottom nuts | Tension + Compression | Fixed |
| | | | Headed stud | Weld | Tension + Compression | Fixed |

These stand-off settings are transferable when importing models from Connection.

*Mortar joints and gaps are also available.*

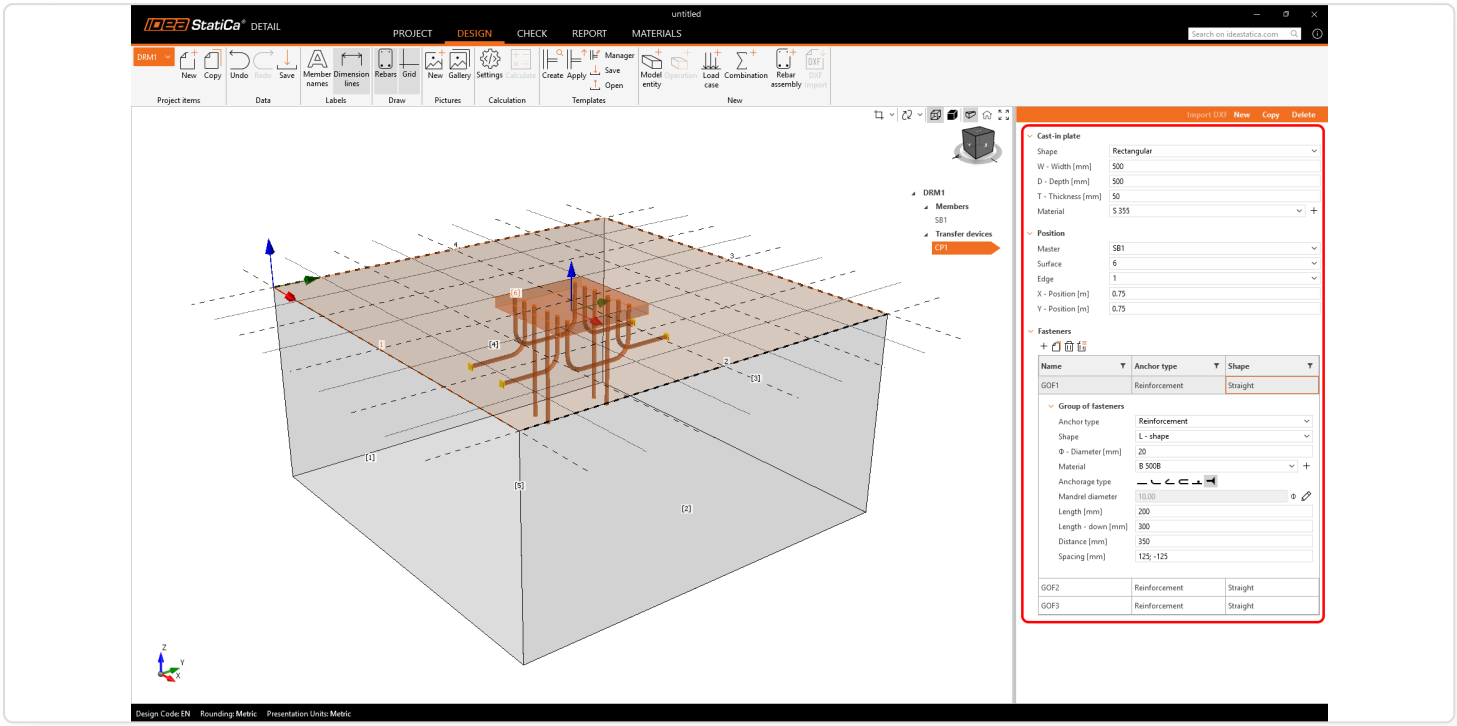
# 2. Cast-in plates options

Cast-in plates are available as a **Load Transfer Device** in the model entity selection. The **plate's geometry and position** can be defined in a single property grid, while an additional table allows users to add and combine multiple **groups of fasteners**.



## Anchor types

Fasteners can be defined directly for each specific plate. Users can add **an unlimited number** of fasteners and even combine multiple types within a single plate. More information about the anchor types related to the Cast-in plate can be found in a separate article.




## Model Behavior

Anchors are treated as elements resisting both **shear and tension**, as they are welded to the plate. Their evaluation follows the same principles as standard anchors. For more information, see the Theoretical Background.

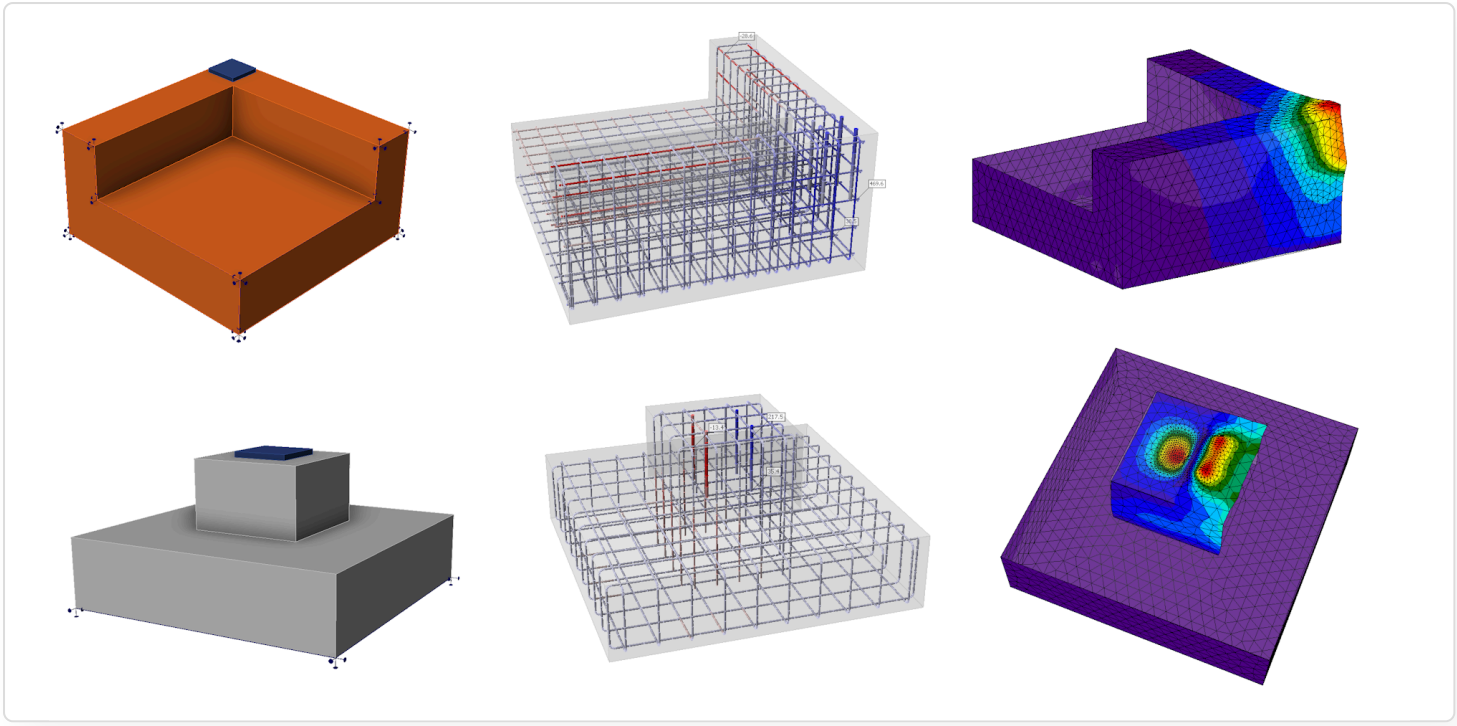
*Released in IDEA StatiCa version 25.1.*

# Modeling options – Negative volume, Cutting plane and Cut, Related to Center Point

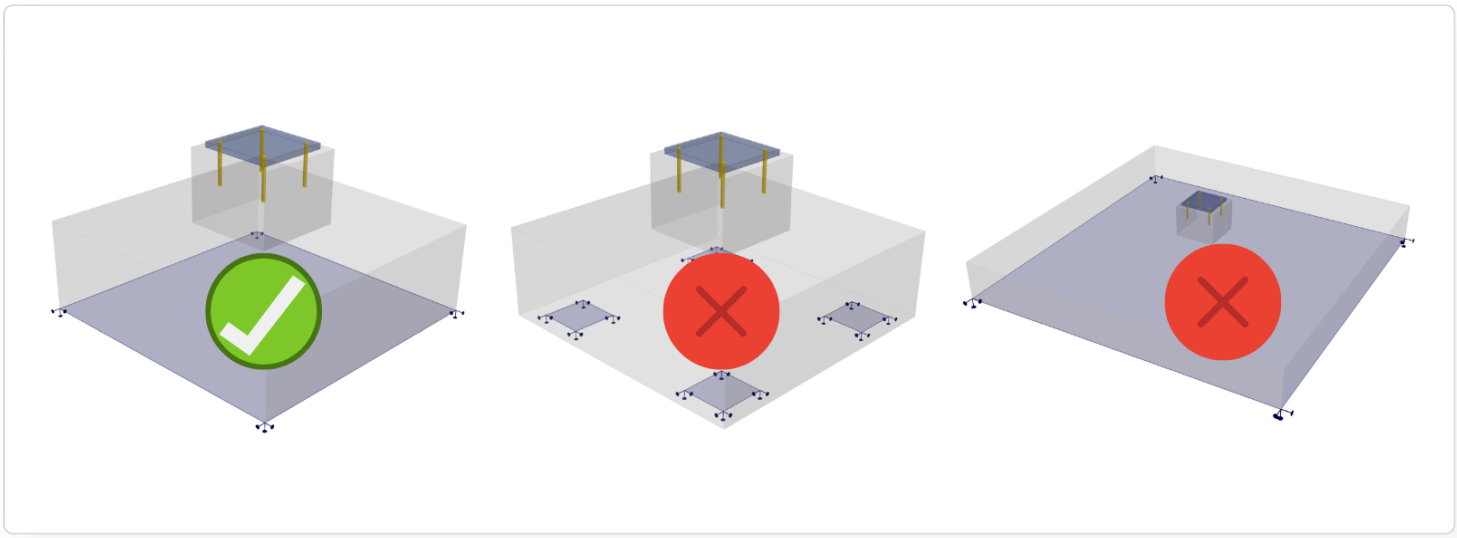
As 3D in Detail continues to evolve, the possibilities open up and we can cater for more complicated cases. IDEA StatiCa Detail features the ability to modify an anchor block using Negative Volumes, Cutting Planes, Cut Operations, and Related to the Center.

# Complex shapes of anchoring

Additional modeling operations enable the creation of more complex shapes and expand potential applications, including pedestals, non-rectangular anchor blocks, foundation strip extensions, and anchoring near openings. We are, therefore, moving towards a more general use of the application. However, it is important to note that the **application is only suitable for the anchoring cases** for which it is verified.



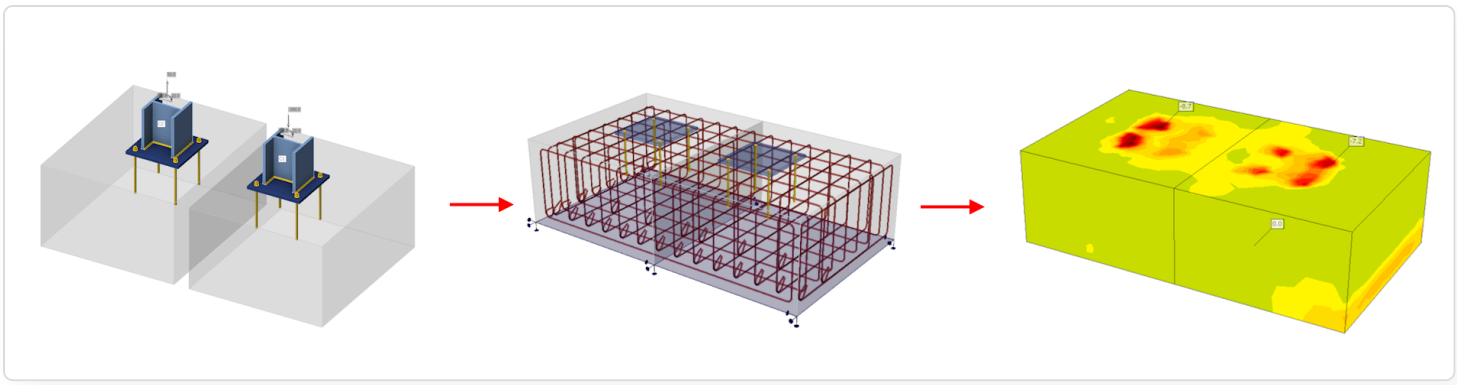
**Further development and verifications are needed to assess punching and other more complex and general cases. These cases are not supported in versions 25.0 and 25.1.**



*Note: In the case of surface support, failure always occurs at the anchorage area, and punching does not play a role. Only in cases of soils with low stiffness may punching occur, or also in the case of pile caps. For these situations, the software is not yet suitable for such use.*

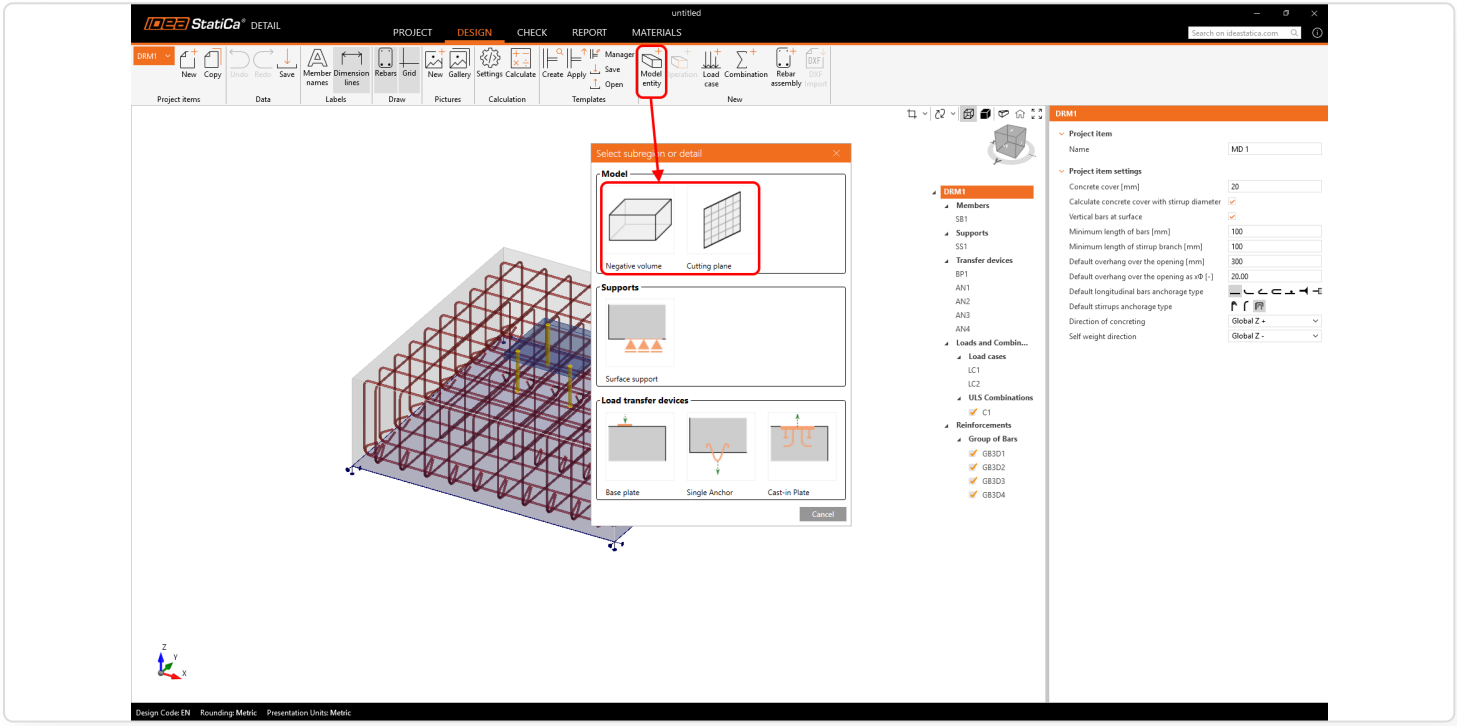
# Merging of anchor blocks

3D in Detail supports only one anchor block. However, since more blocks can be created in IDEA StatiCa Connection, it is now possible to import these multiple blocks into Detail, where they can be merged using the Cut operation and then reinforced. In the case of overlapping blocks, it is then necessary to delete one of the blocks and assign the base plates to the same block.

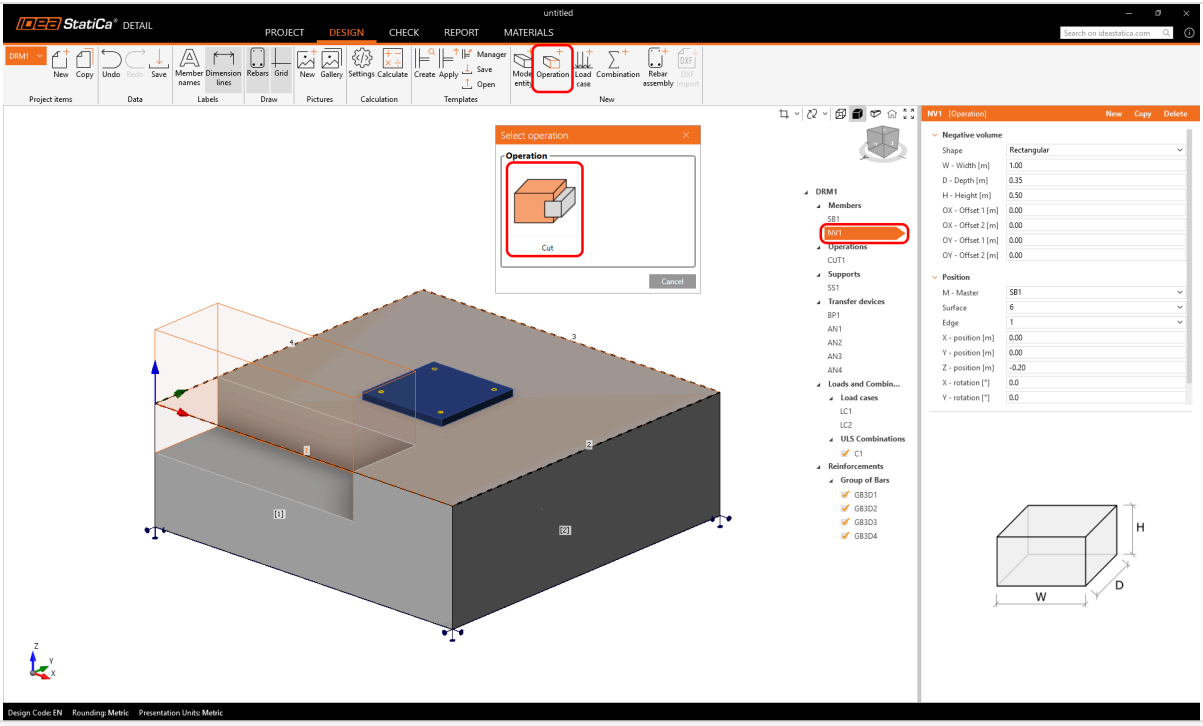


# The Cut operation

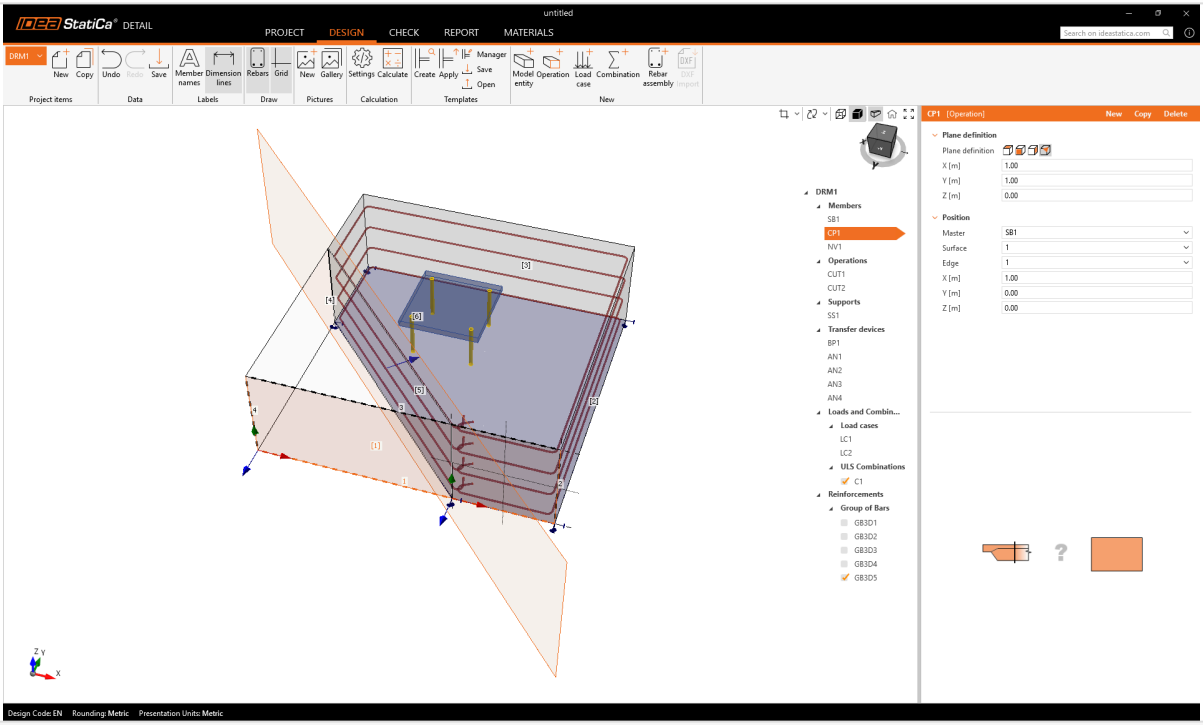
The basic operation for adjusting the block shape is the Cut operation. The Cut can be according to a negative volume or a cutting plane, these can be found under the "model entity" button.



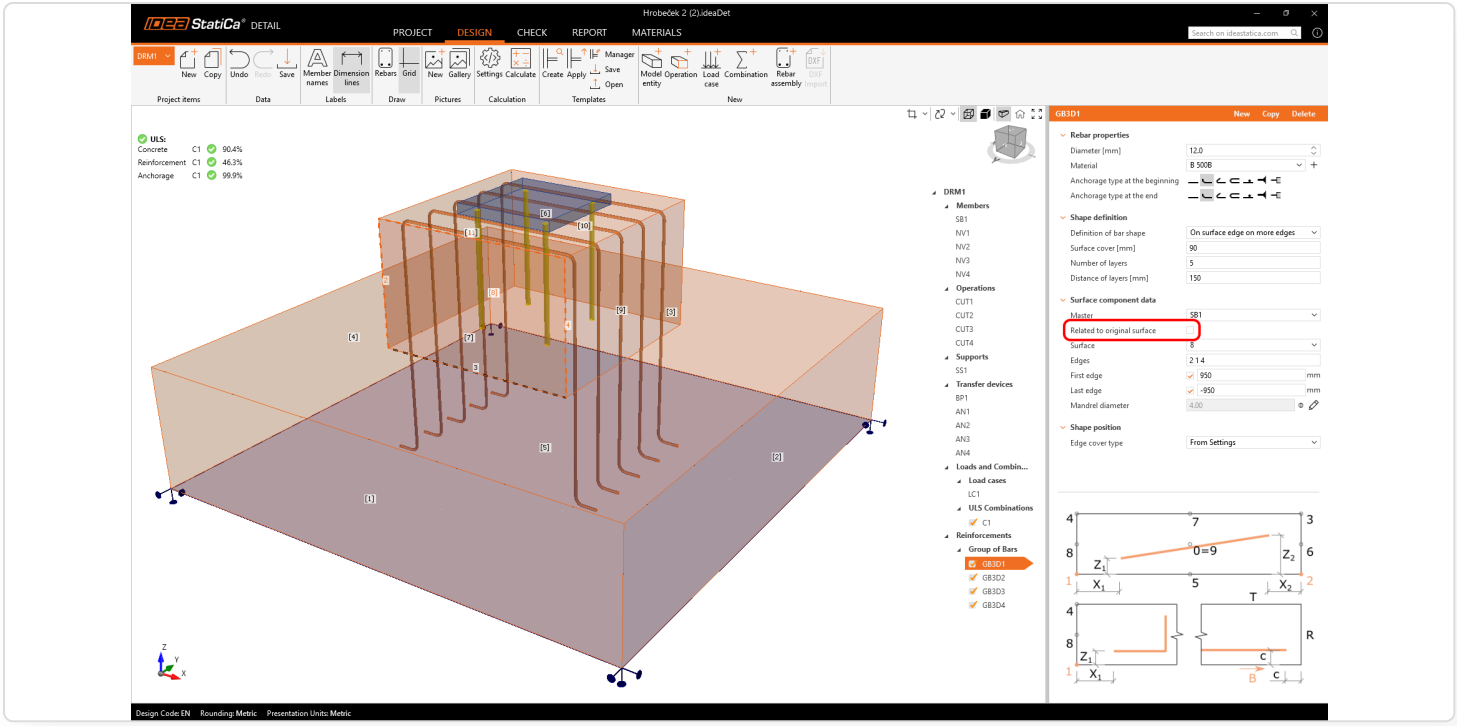

- The **Cut operation** is applied to the negative volume or cutting plane to adapt the host block using the operation.




- **A Cutting plane** works in a similar way. As shown in the following figure, any cut generates new edges and surfaces that can serve as references for placing reinforcement.



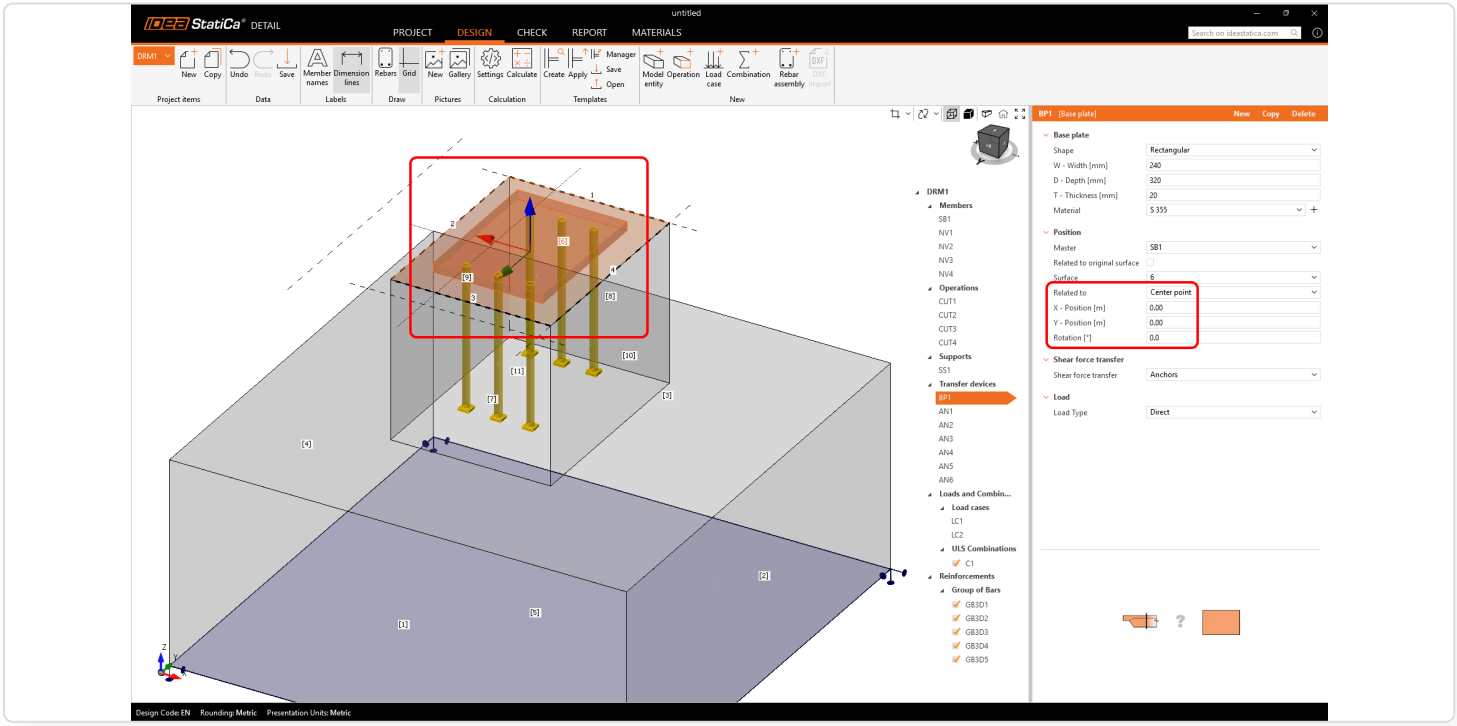

- After the first Cut operation is applied, reinforcement can be referenced to either the original or newly created faces and edges. This is controlled through a new setting in the property grid.




*Released in IDEA StatiCa version 25.0.*

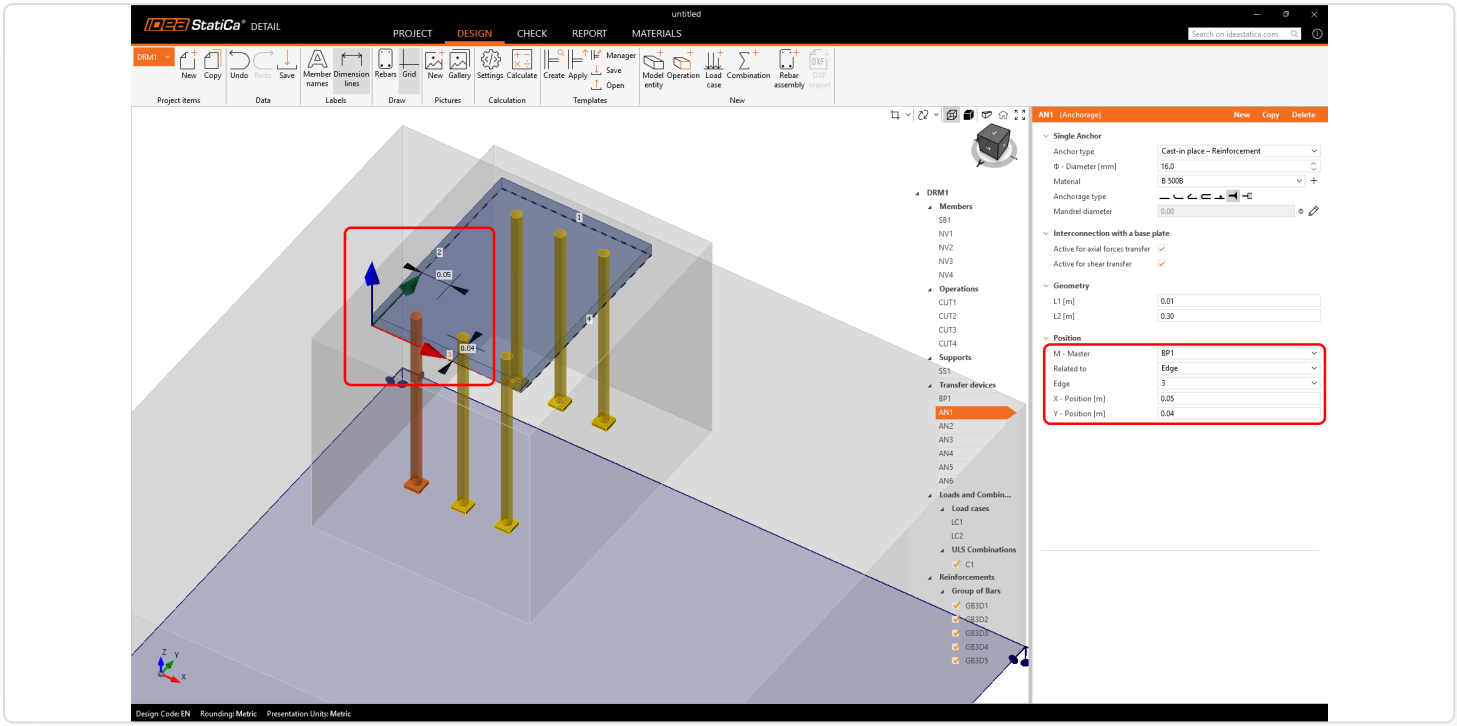
# Anchors tied to the base plate and referenced to the center

With the referencing system in Detail 3D, users can position model entities - such as base plates, anchors, supports, and loads - relative to the **center point** of their master entity instead of only from an edge. This means anchors can be placed directly in relation to the base plate, and all connected components automatically adjust when the geometry changes. The **“Related to: Edge / Center point”** option in the Property Grid makes modeling faster, more intuitive, and ensures consistent alignment without manual repositioning.




The following entities support reference to the center:

- Base plate (Related to the Center of the solid block surface)
- Support defined by polyline (Related to the Center of the solid block surface)
- Point load (only for Cast-in plate)
- Surface load defined by polyline (Related to the Center of the solid block surface)
- Single anchor (Related to the Center of the solid block surface or Base plate)




*Released in IDEA StatiCa version 25.1.*

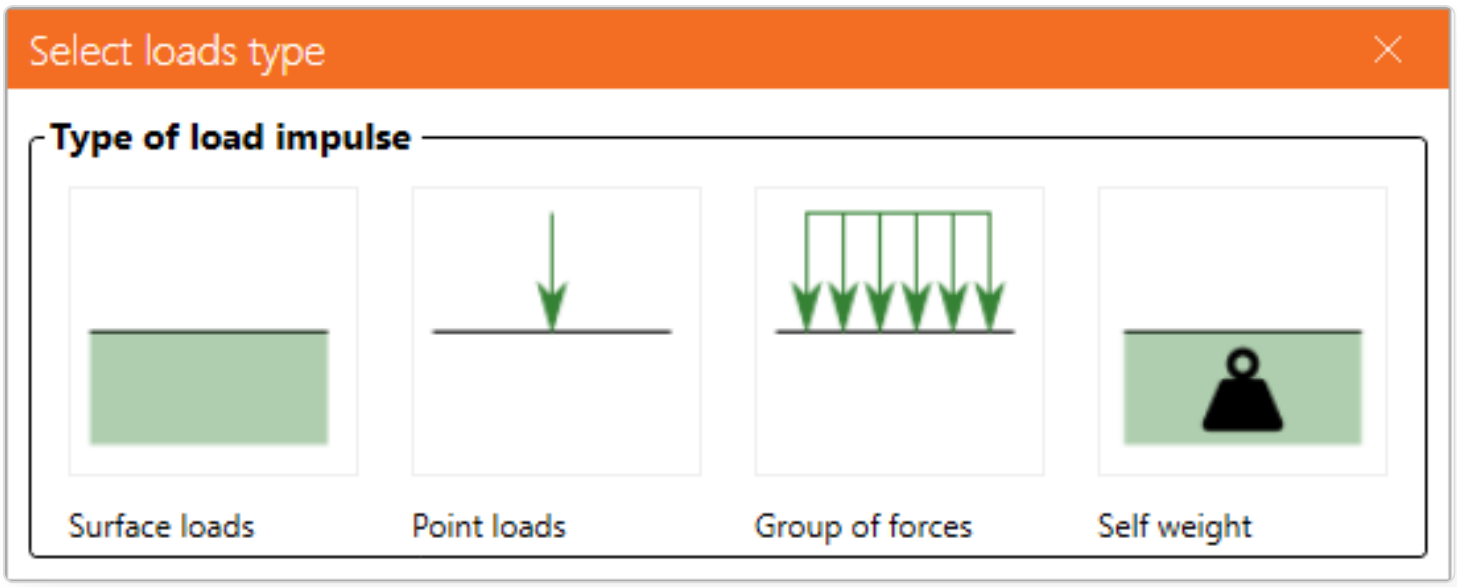
# Loading and combinations

## Loading

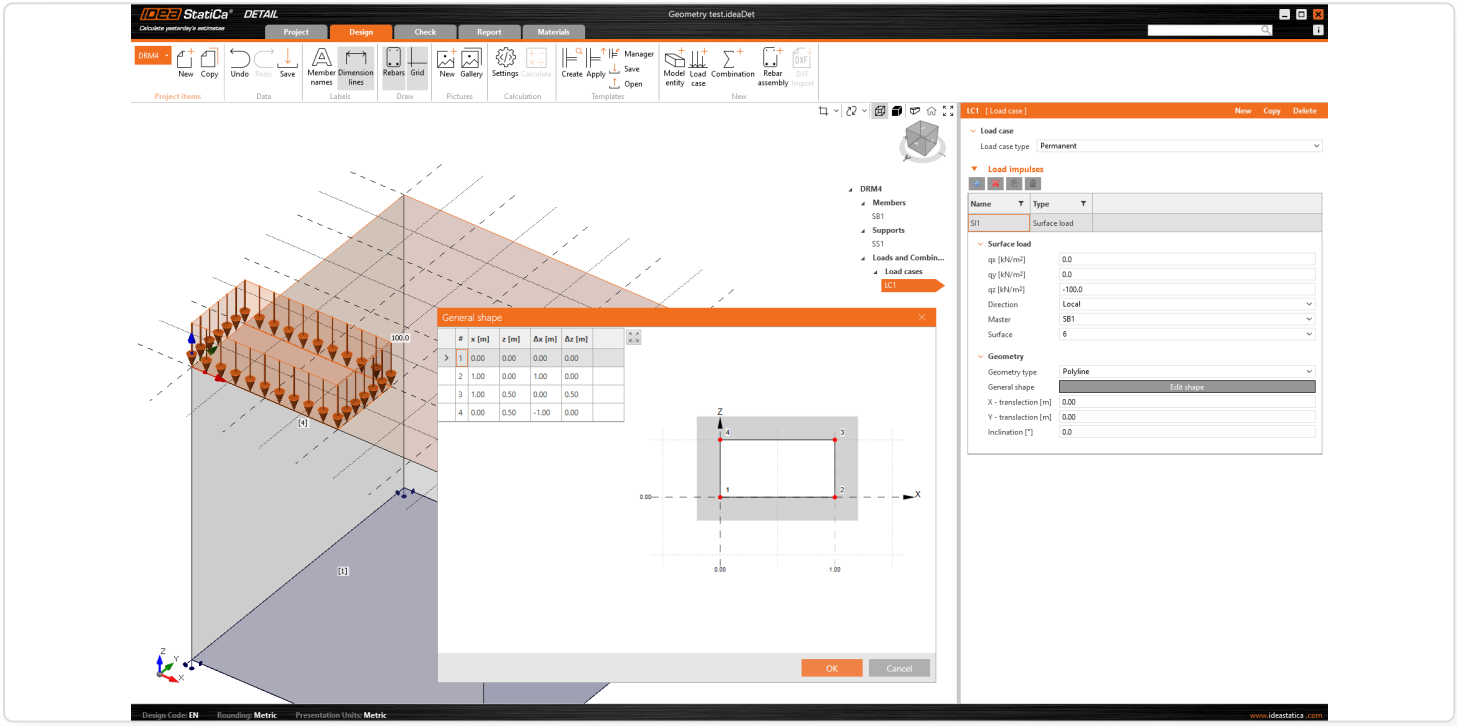
Load cases can be defined in the same way as for 2D reinforced concrete elements. This means that each load case can be assigned either a Permanent or a Variable load type. The Permanent load cases are applied to the model first, and after a successful calculation, the Variable load cases are applied.

**Type of load impulses**

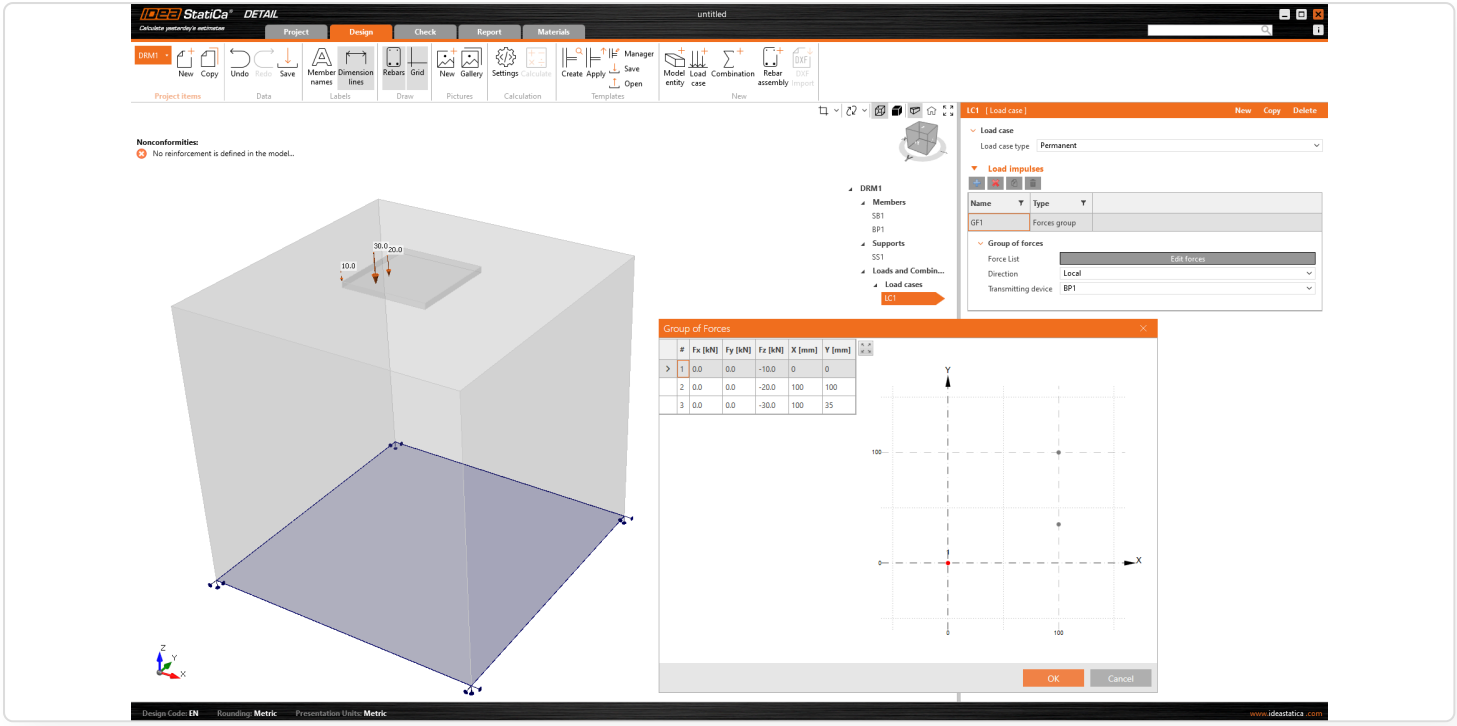
A total of 4 types of load pulses can be added to each load case.




The definition of **Surface loads** is identical to the definition of Surface support. This means it is possible to specify it in two ways: Whole surface and Polyline. In the case of Surface loads, of course, the load intensity is entered in the three general directions.

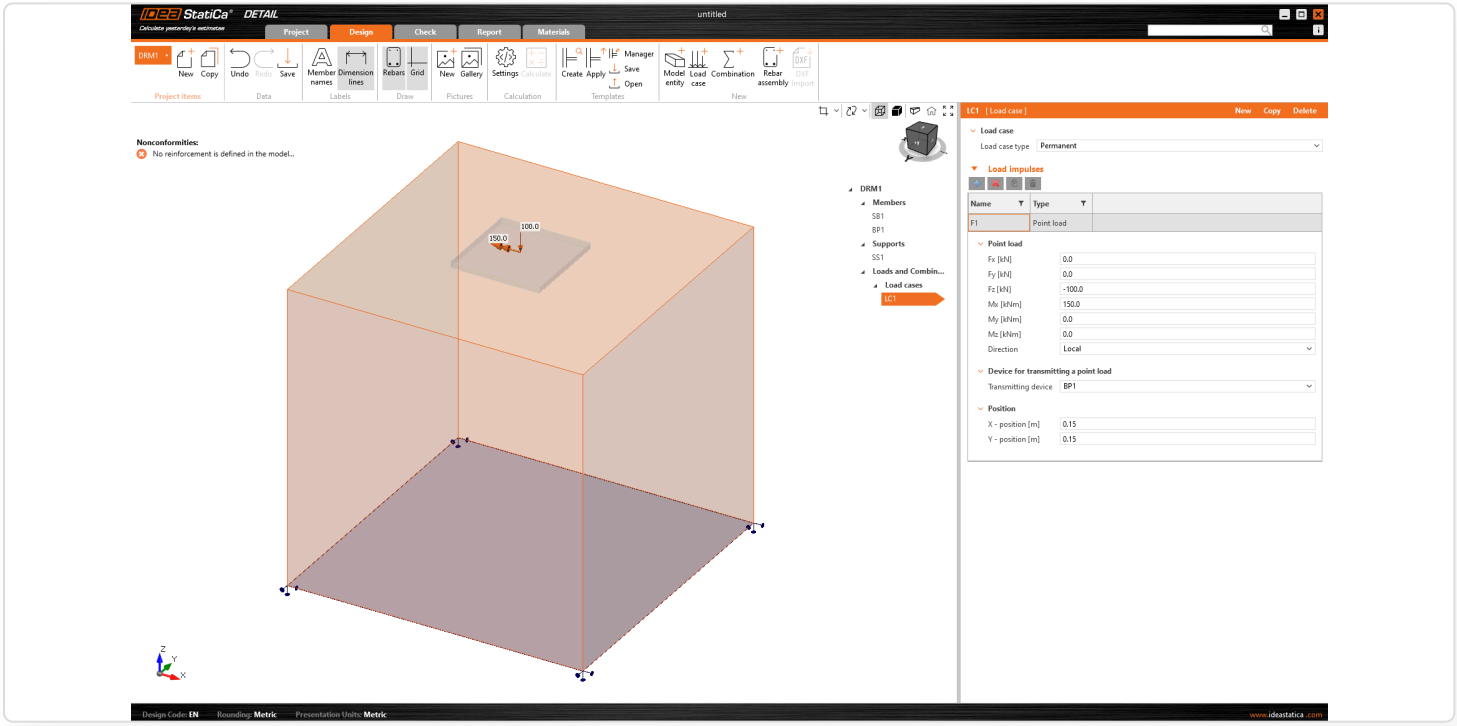



**Group of forces** is a load entity that allows you to specify forces in three directions anywhere on the model using a table. It can be referenced to the base plate or the surface of a concrete block. For tabular input, it is again possible to use the copy-paste functionality from the spreadsheet program.




**The self-weight** should be included in every model. For example, concrete foundations loaded with a bending moment will not so easily overturn.

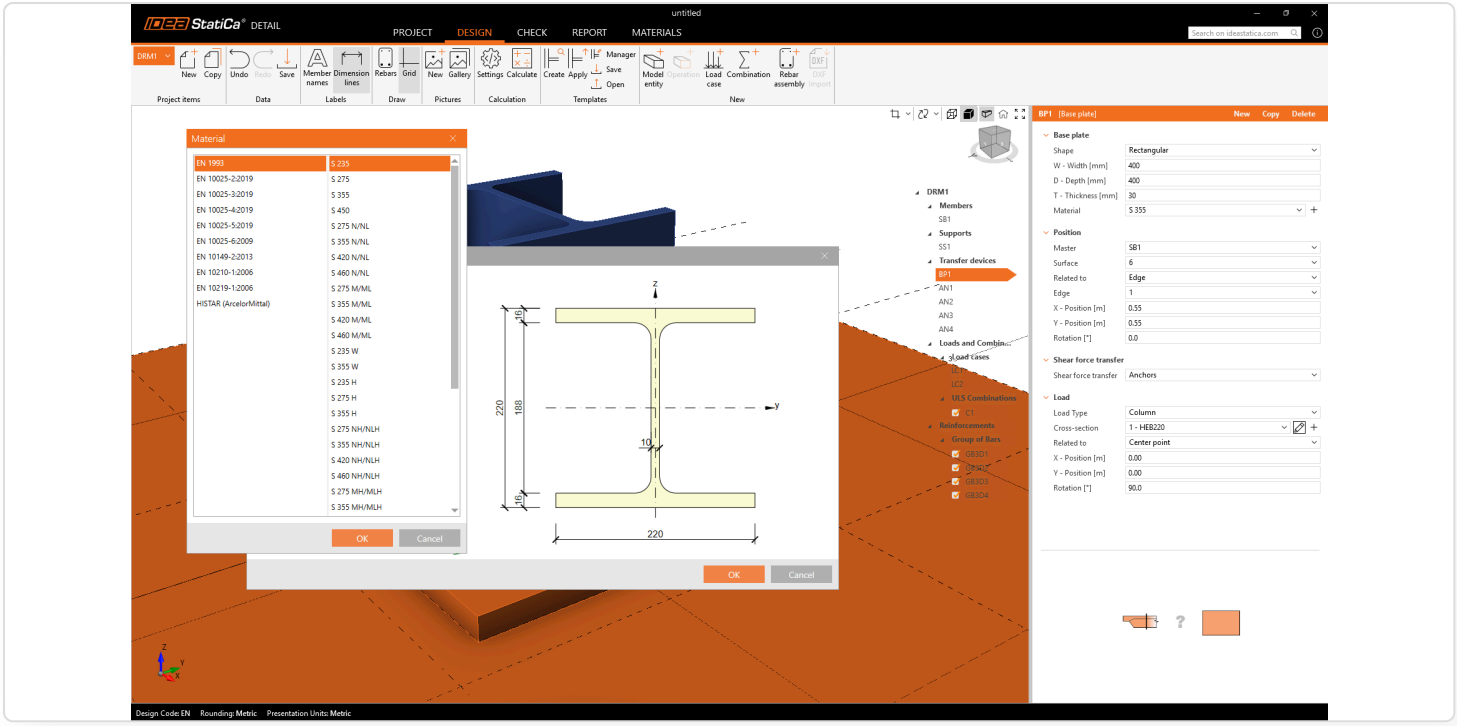
**Point loads** can be loaded directly to the base plate with six internal forces Fx, Fy, Fz, Mx, My, and Mz in the general position.



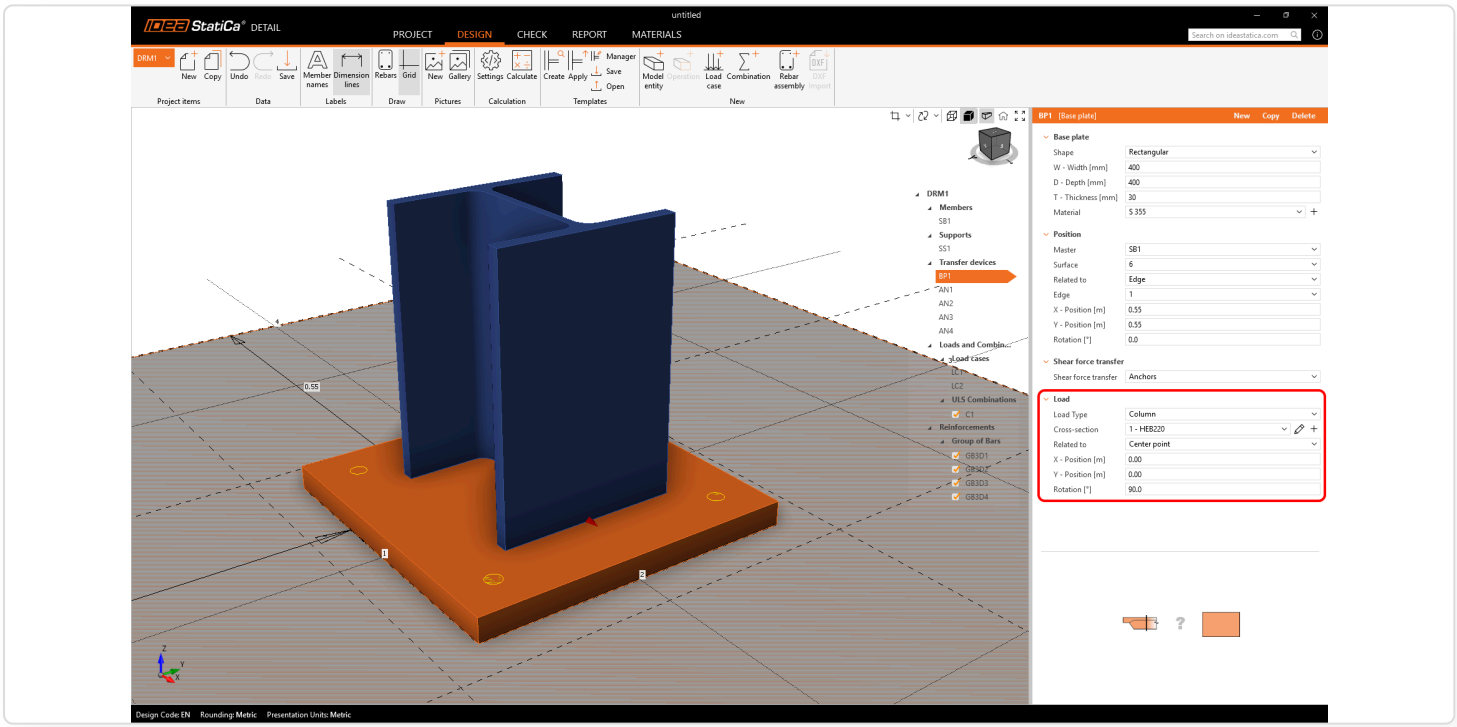

When using a base plate, applying this force directly to a realistic, deformable base plate can lead to unrealistic stress redistribution across the plate, anchors, and concrete. It is therefore more appropriate to use the second option - the stub.

### The Stub

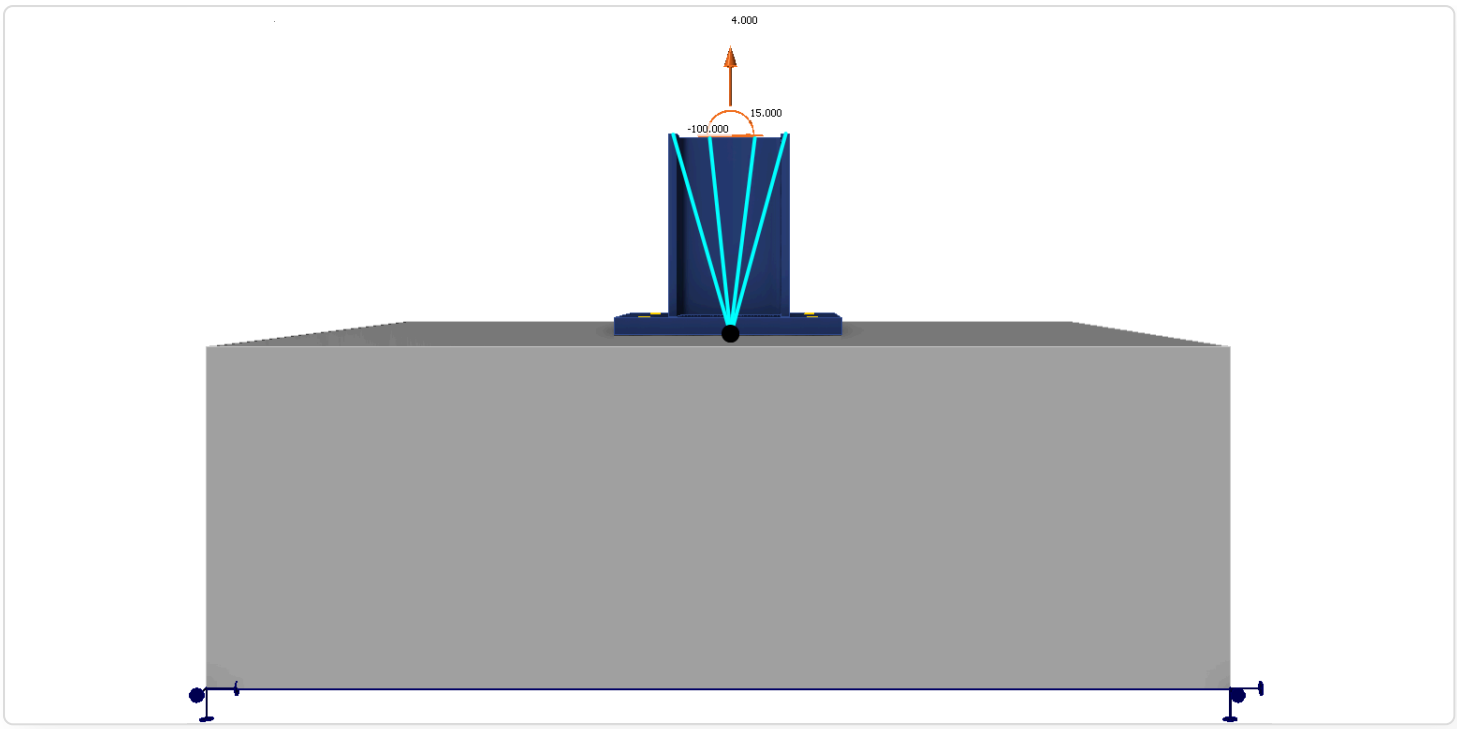
The stub is represented by a short part of the column above the base plate, which is modeled as a **shell element structure** and behaves as a physically accurate interface between the internal forces and the plate. A standard section database is used.




The **6-component internal force set** (forces and moments) is applied at a **single point** on the **bottom face of the stub** - i.e. the base of the column.




Constraints **transfer the forces** to the **top face of the stub**, from where they are naturally **redistributed through the stub** into the base plate, anchors, and concrete.



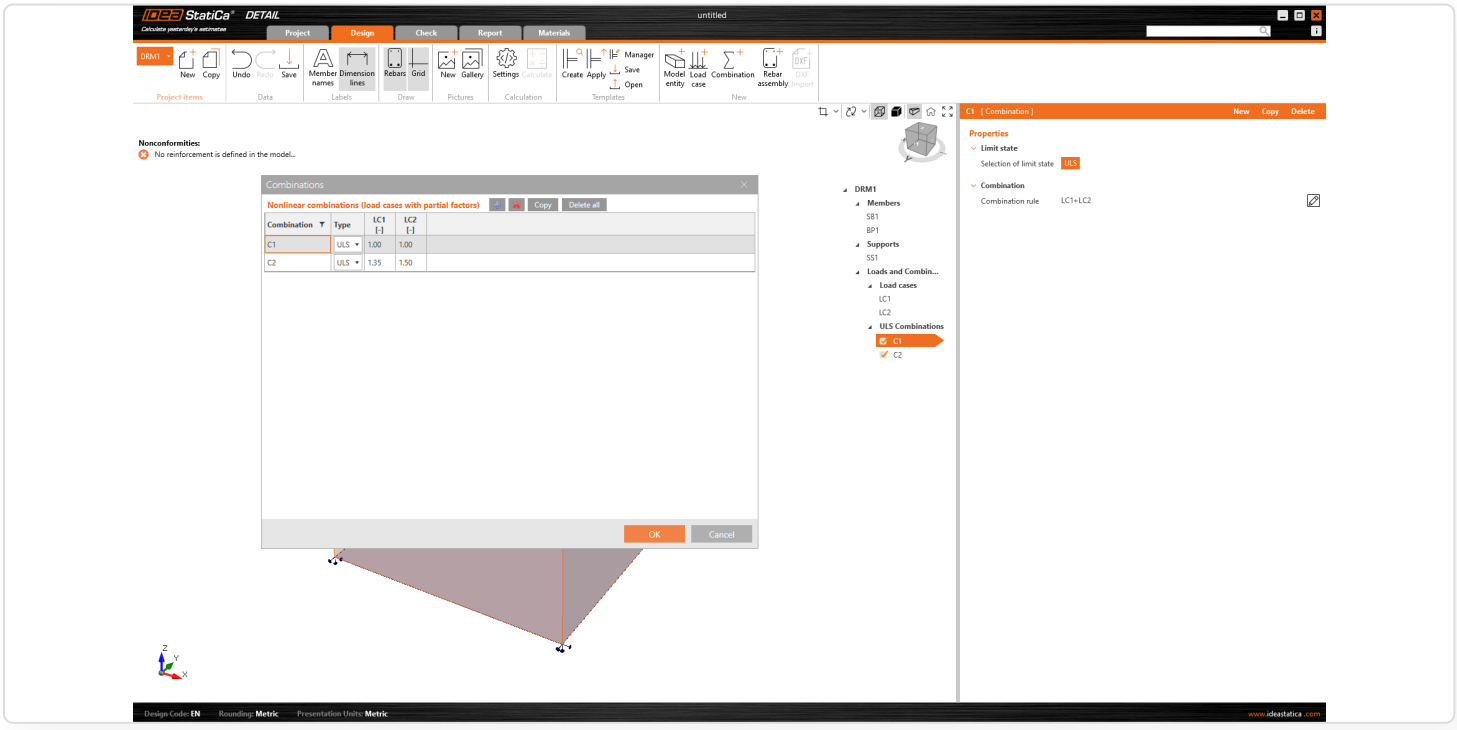

This approach preserves the realistic stiffness interaction between column and plate and eliminates the need for any manual redistribution or artificial assumptions.

*The stub was released in IDEA StatiCa version 25.1.*

## Combinations

Because the analysis in IDEA StatiCa Detail is non-linear, so-called non-linear combinations are used. This means that individual load cases are not calculated and the results are not then added together. On the contrary, load cases of the same load type are added together before the calculation, of course with the respective coefficients defined in the combinations, and the individual combinations are then calculated. This is why the existence of at least one combination is a prerequisite for starting the calculation.

Only combinations for ULS can be defined.



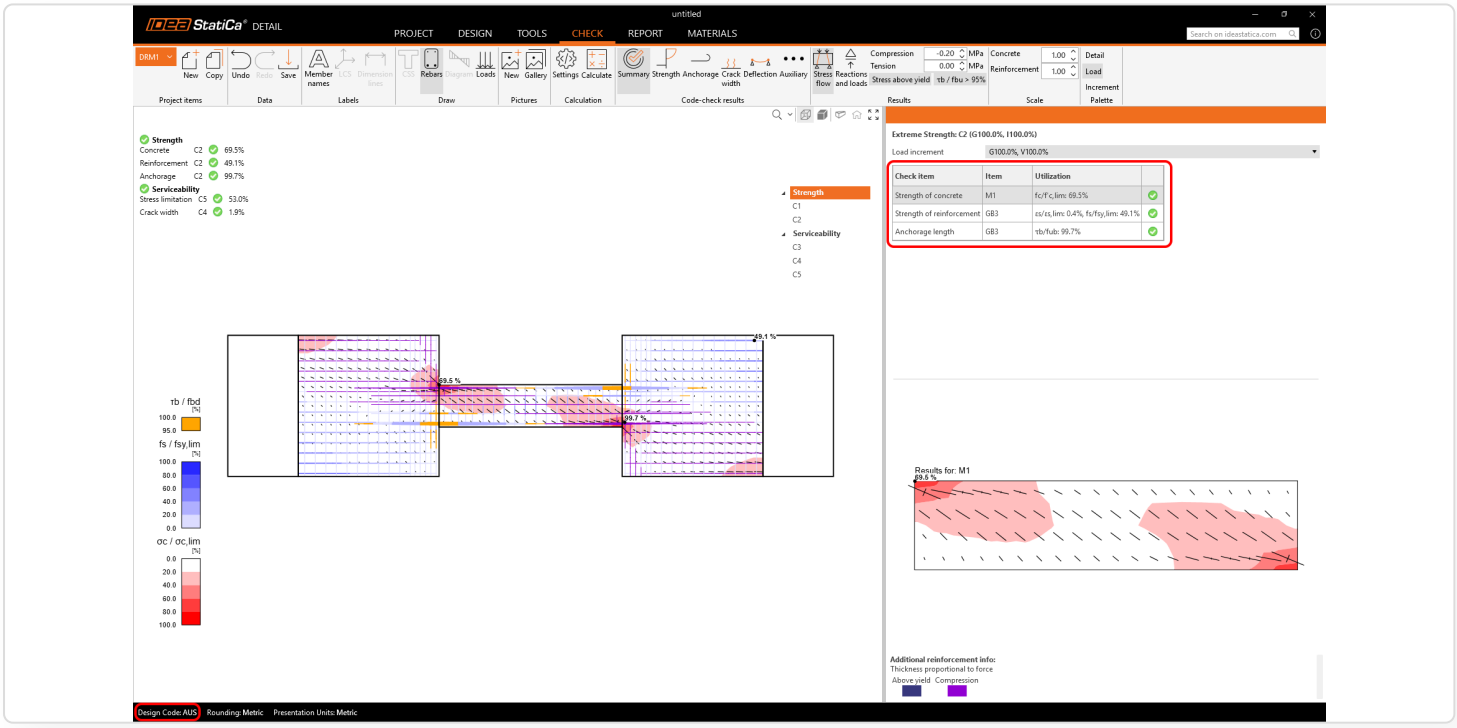
# Australian code for Detail

The Detail application includes the Australian Standard with full functionality of all concrete modules, including prestressing and anchoring.
This integration ensures that engineers working with AS 3600 have the option to design discontinuity regions using CSFM. CSFM (**Compatible Stress Field Method**) is an innovative approach that allows engineers to move away from rough estimations and solve tasks using precise calculations. This provides Australian engineers with a modern alternative to the outdated Strut and Tie method, allowing them to efficiently design and assess **walls and beams with openings, pile caps (in 2D), pier caps, diaphragms, anchoring (in 3D) and other discontinuity regions** in concrete structures.

Learn more about the method implemented in IDEA StatiCa Detail and its practical applications in the article: CSFM explained.

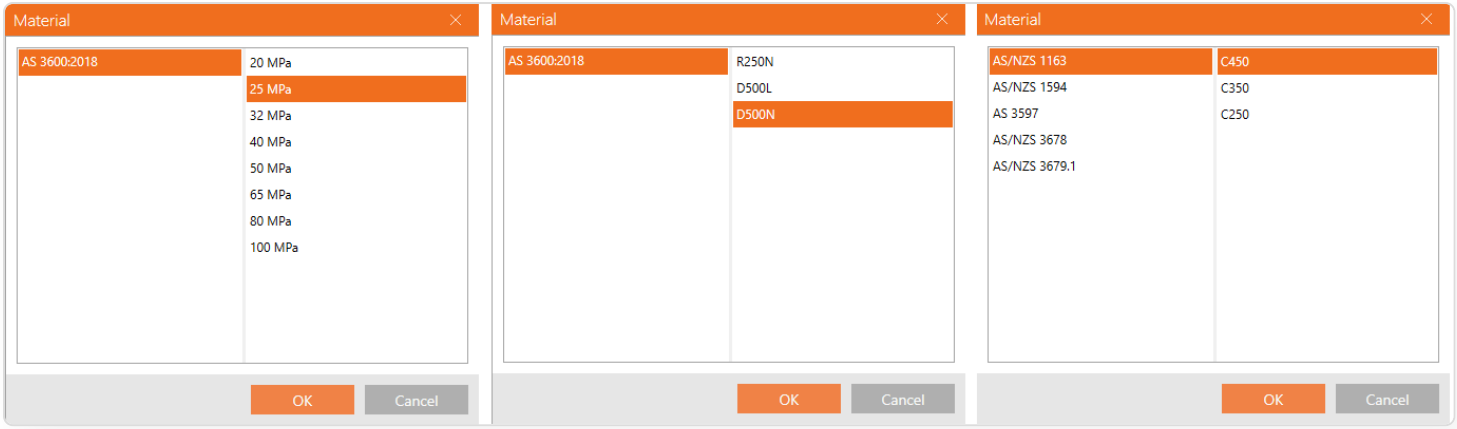
# Discontinuity regions

Users can utilize the **full functionality of Detail (in 2D)**, adapted to **AS 3600 terminology, material properties, and verification checks**. The user interface aligns with Australian standards, making the design process more intuitive and ensuring that calculations follow local industry requirements.

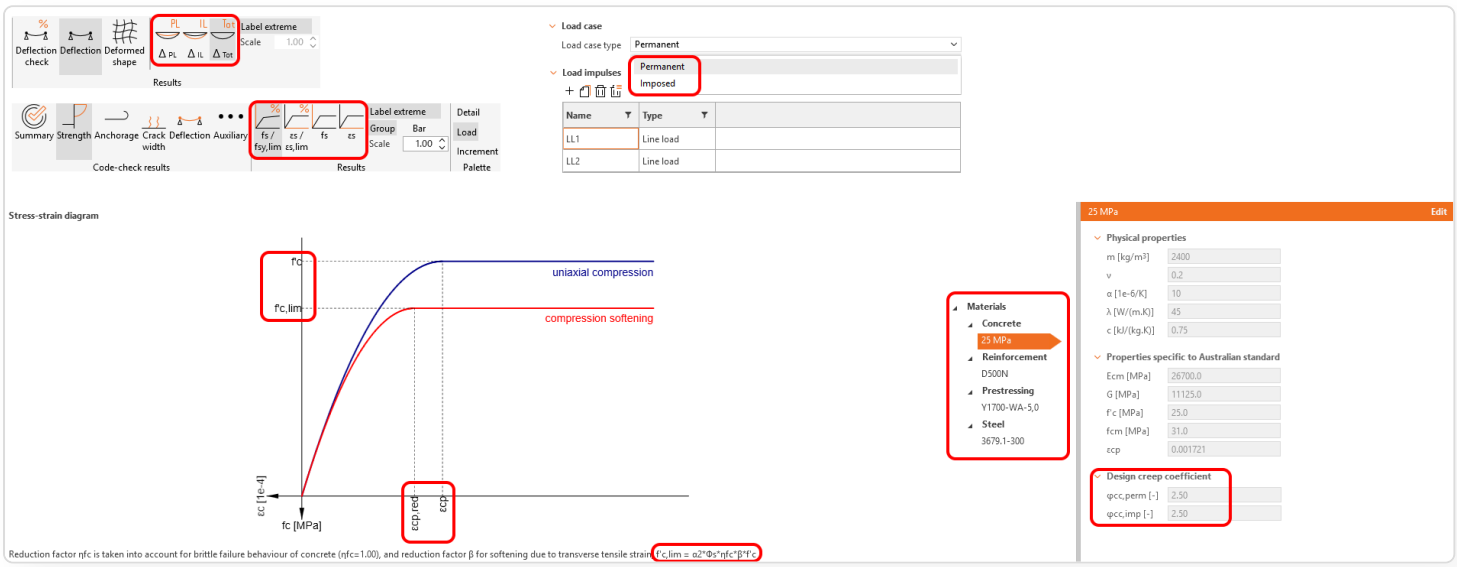



## Key features

- **Material Database** – A built-in database includes concrete, reinforcement, and steel materials defined by AS 3600.




- **Standardized Nomenclature** – Variable names and labels throughout the application match AS 3600, ensuring clarity in design and verification.
- **Refined Calculations to AS 3600**




1 **Stress-strain diagrams** for serviceability and strength, incorporating **long-term effects using the design creep factor**.

2 **Bond strength and B-factor** calculations for anchorage springs, ensuring compliance with AS 3600.

*Released in IDEA StatiCa version 25.0.*

# Prestressing

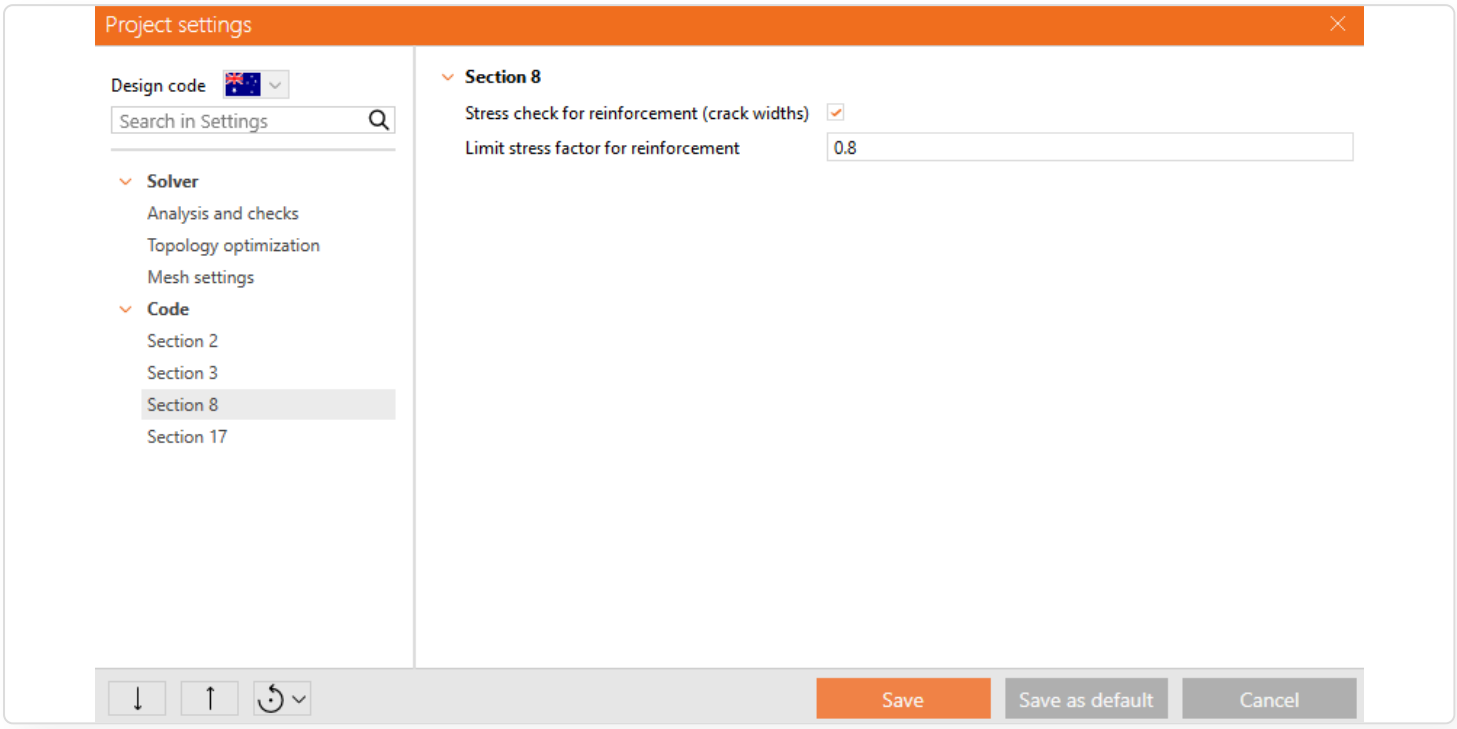
Prestressing has been fully implemented according to **AS 3600:2018** in Detail 2D. A dedicated material database for wires, tendons, and bars has been added, along with project settings specific to prestressing such as stress increments, stress reduction, and design limits. The implementation includes anchorage zones and partially loaded areas with dispersion angles as defined by AS 3600, as well as a dedicated bond model for prestressing tendons.

All relevant verification checks are available, including strength, anchorage, stress limitations, and crack width, in full compliance with AS 3600.

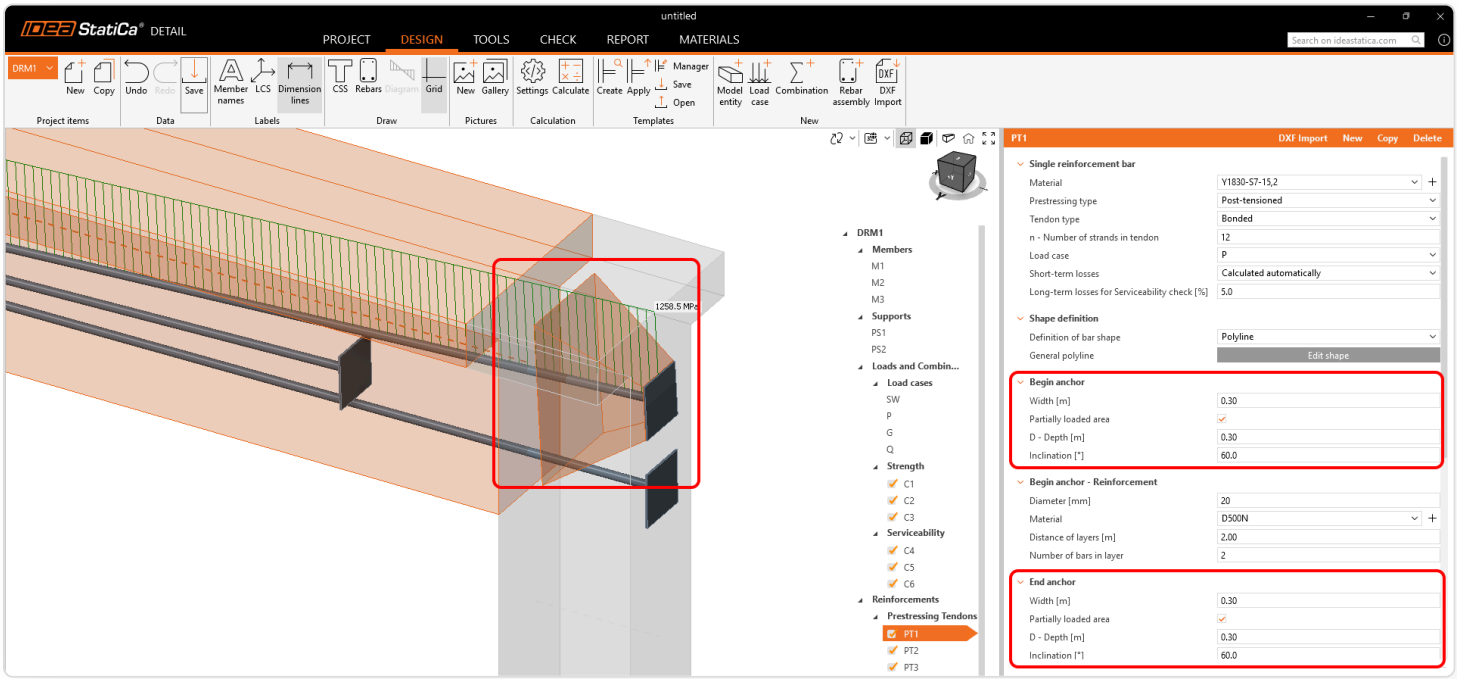
All templates are also available, just like for the other codes.

## Key features

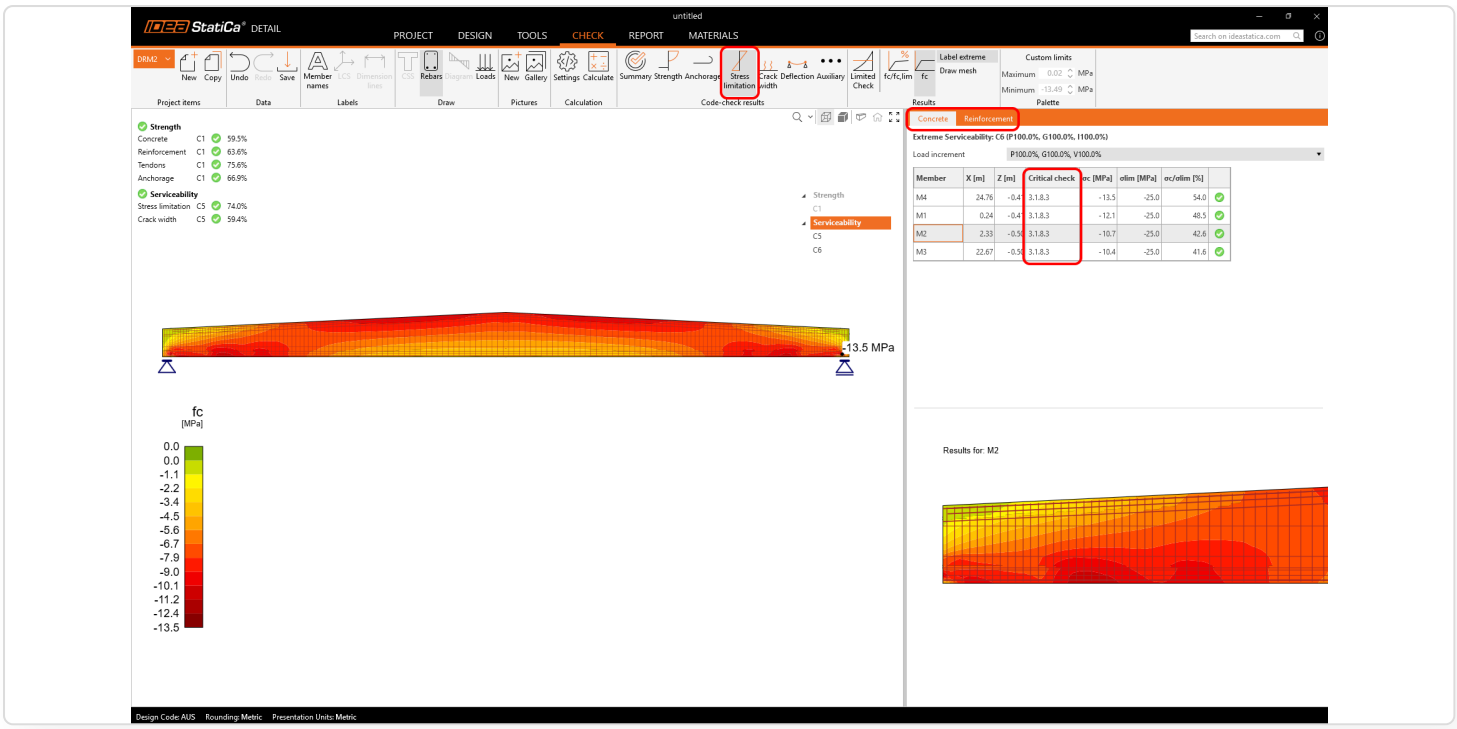
- **Material database** for wires, tendons, and bars per AS 3600 Section 3.3 Properties of tendons.
- **Project settings** with prestressing-specific factors (increments, stress reduction, concrete/reinforcement stress limits).




- **Partially loaded areas** (anchorage zones and point supports) with a 60° dispersion angle per AS 3600 chap. 8.1.4.




- **Bond model** for prestressing tendons, including neglecting bonds over 0.1 Lpt as required by the code, clause 13.3.2.
- **Expanded results** to include Strength, Anchorage, Stress limitation, and Crack width checks, with stress limitation checks aligned with AS 3600 sec. 3.4.3.3 (concrete) and sec. 8.6.2.2 (reinforcement).



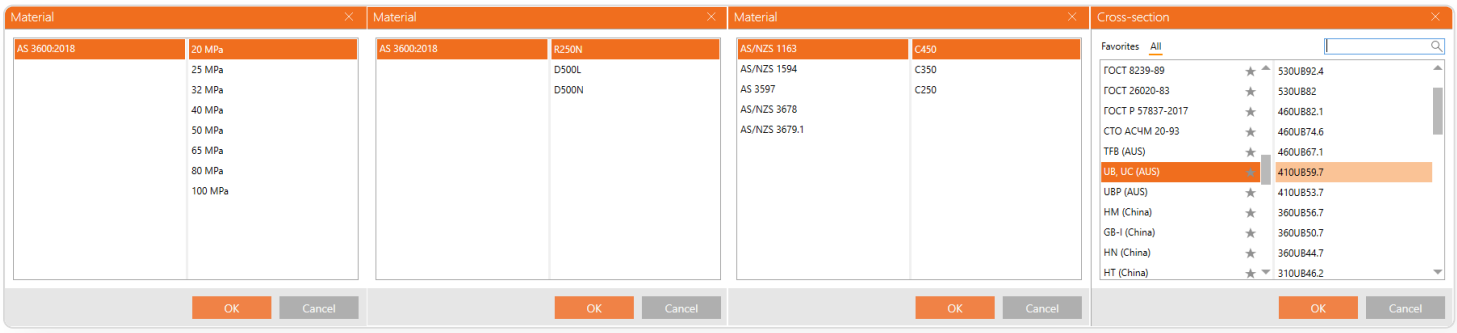

*Released in IDEA StatiCa version 25.1.*

# Anchoring

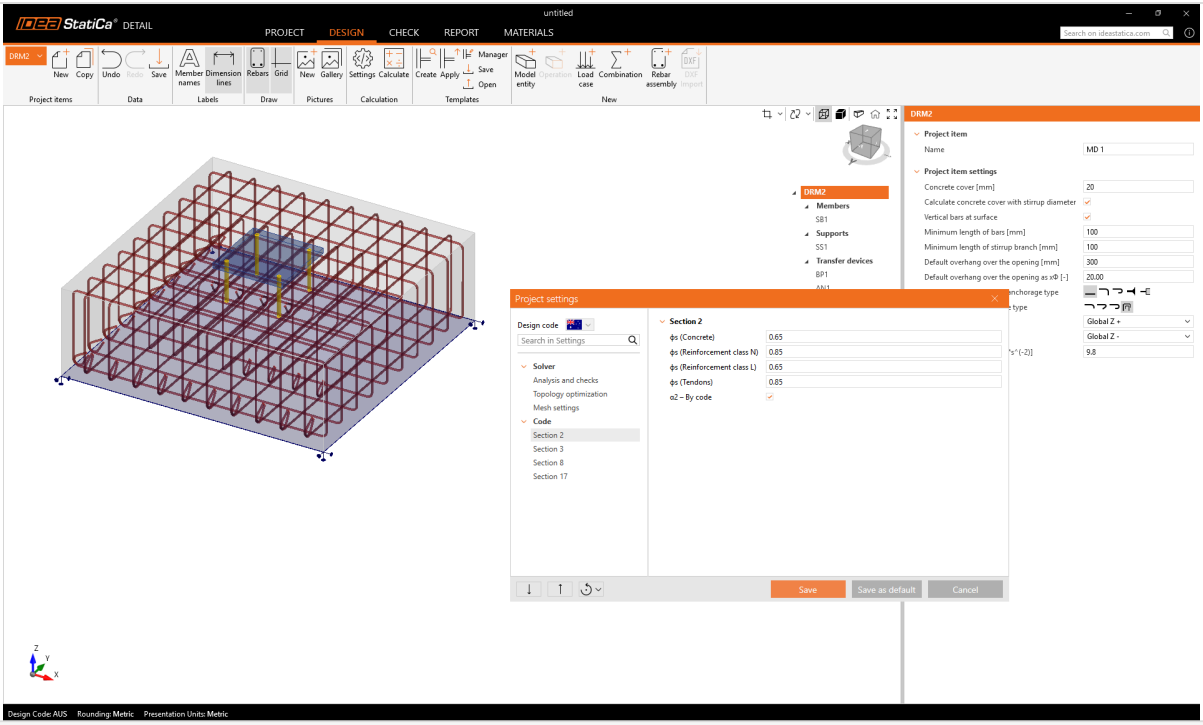
Anchoring functionality in **Detail 3D** is available fully in line with AS 3600. This eliminates the need for Australian engineers to approximate their designs using EN or ACI standards. The module includes an Australian-specific material and cross-section database, adjustments to reinforcement and bond behavior, and code settings reflecting local rules. All templates are available as for EN and ACI, with seamless exports from the Connection application.

## Key features

- **Database of Australian materials** and cross-sections.




- Code settings to reflect **local design rules**.



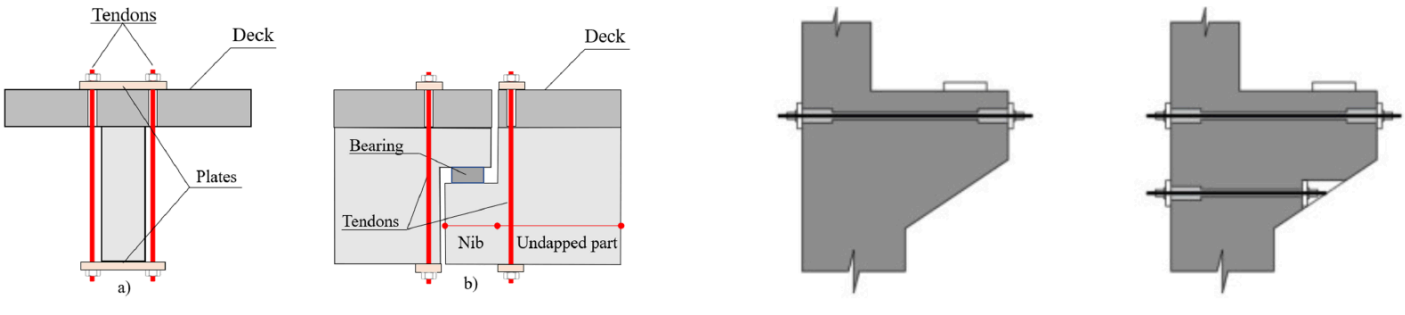

- **Export from the Connection** application into Detail 3D.

*Released in IDEA StatiCa version 25.1.*

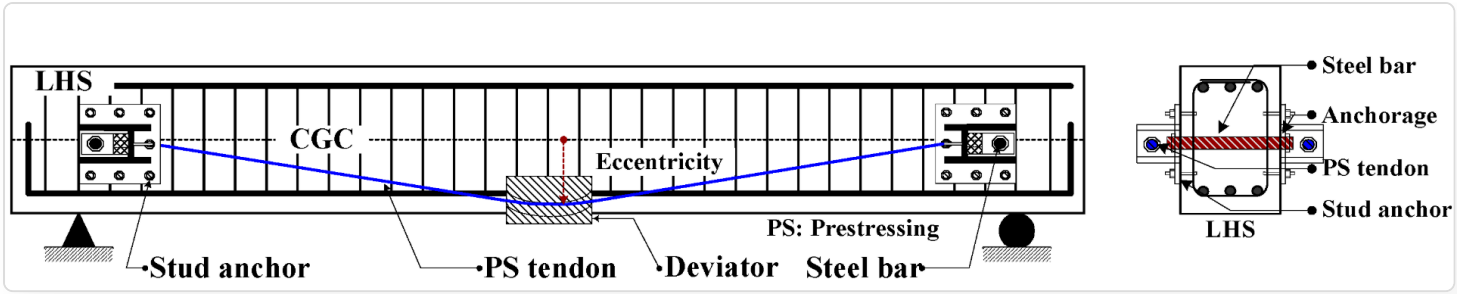
# Prestressing in Detail – unbonded tendons

Detail supports unbonded tendons. This feature is available for all three supported standards (EN, ACI, AUS) and allows users to bring new life to existing concrete structures.
Together with the smooth bar option, this addition makes complex prestressing and retrofitting tasks far more efficient and accurate. Suitable, for example, for the following types of structures:

- **Designing new prestressed members** – unbonded tendons simplify construction and allow future adjustments.
- **Retrofitting D-regions** – for example, reinforcing dapped ends with vertical tendons to close cracks.

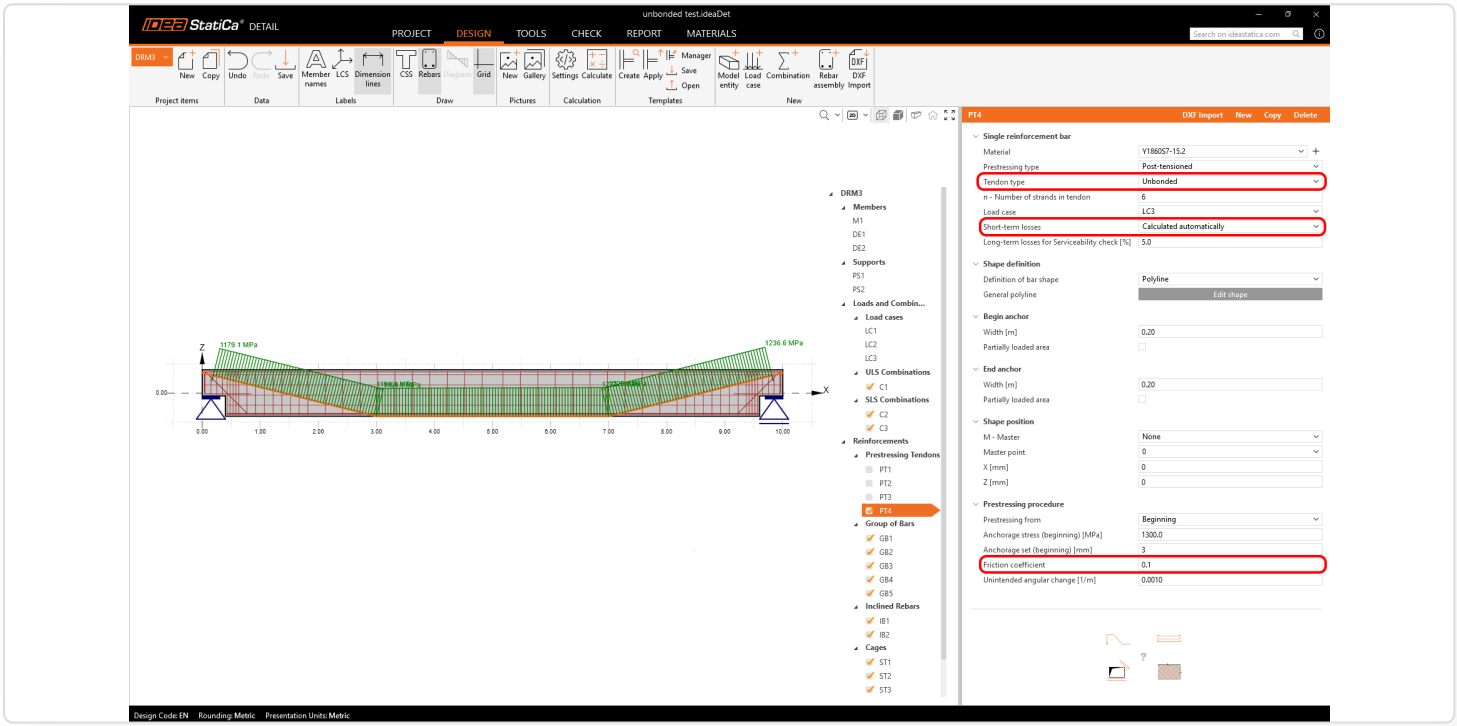



- **Strengthening existing structural members** – such as old precast reinforced beams or ribbed slab cross beams.




A **Bonded/Unbonded switch** has been added for post-tensioned tendons in Detail 2D, enabling users to easily model both tendon types. Outputs follow the same logic as bonded tendons, with the exception that **anchorage and decompression checks are skipped**, reflecting the behavior of unbonded tendons.

For unbonded tendons, when short-term losses are set to be **calculated automatically**, the **default reduced friction coefficient** is 0.1, in accordance with **EN 1992-1-1 Table 5.2** and **ACI 423.3R-05, chapter 3.5**. This ensures accurate modeling of prestressing losses and tendon behavior.

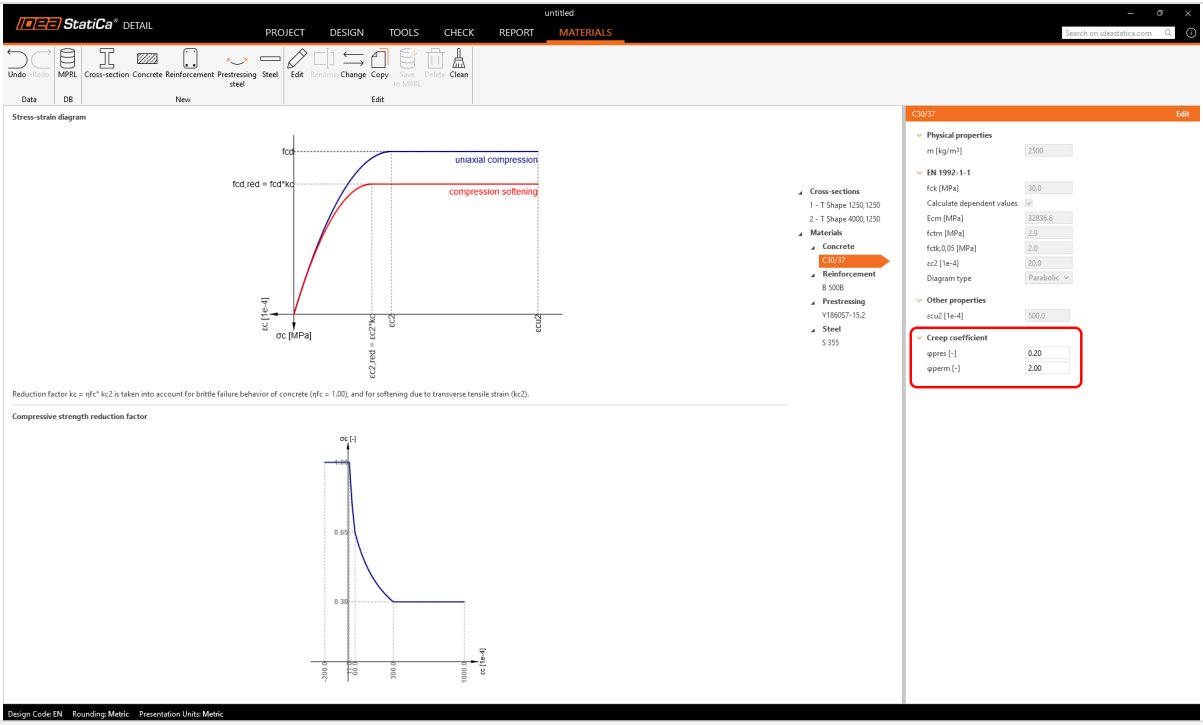



For **user-defined tendons**, the prestressing stress must be specified as a **single value along the entire polyline**.

Additionally, prestressing is **applied only at the Beginning and End** of the tendon, simplifying the definition while ensuring consistency in modeling.

# An important recommendation for creep coefficient (EN, AUS only)

When strengthening an existing reinforced concrete structure, typically done long after casting, when much of the concrete's creep capacity has already been consumed, the **creep factor for prestressing** should be set **much lower** than that for permanent (dead) loads. Otherwise, the calculated prestressing effects will be **unrealistic** because, according to the CSFM principle, prestressing is always applied in the **first increment**.




For the reinforcement of prestressed beams, there is a limitation (in the EN and AUS implementation) that the original prestressing and strengthening prestressing cannot be applied with different creep factors.

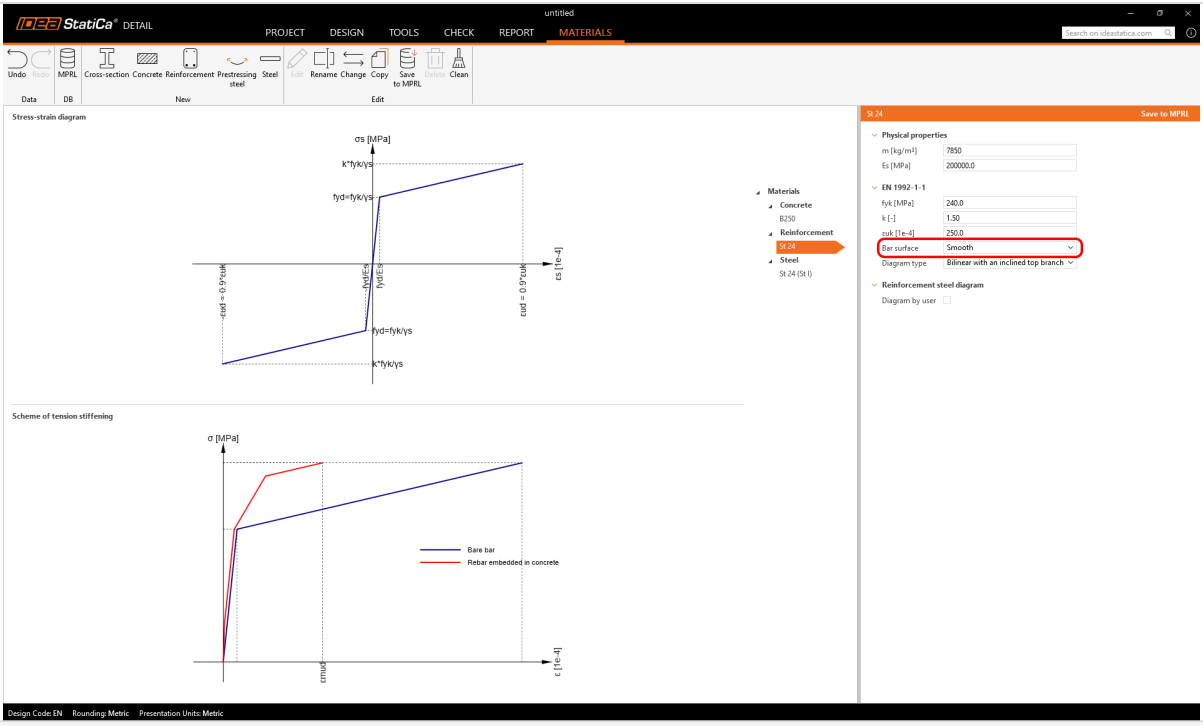
# Smooth rebars in Detail

Detail supports smooth reinforcement bars, addressing a major limitation in recalculating or retrofitting older structures.

Many mid-20th-century designs used smooth rebars, whose bond behavior is significantly weaker than ribbed bars. Since design codes (EN, ACI, AS) are primarily based on ribbed reinforcement, this difference affects anchorage length, crack spacing, and stiffness modeling. With IDEA StatiCa Detail, it is possible accurately model and verify structures with smooth rebars, improving the reliability of crack width calculations and anchorage checks. This makes Detail a reliable **tool for retrofitting and assessing older buildings, bridges, and precast elements**.

The option is available for all implemented codes.

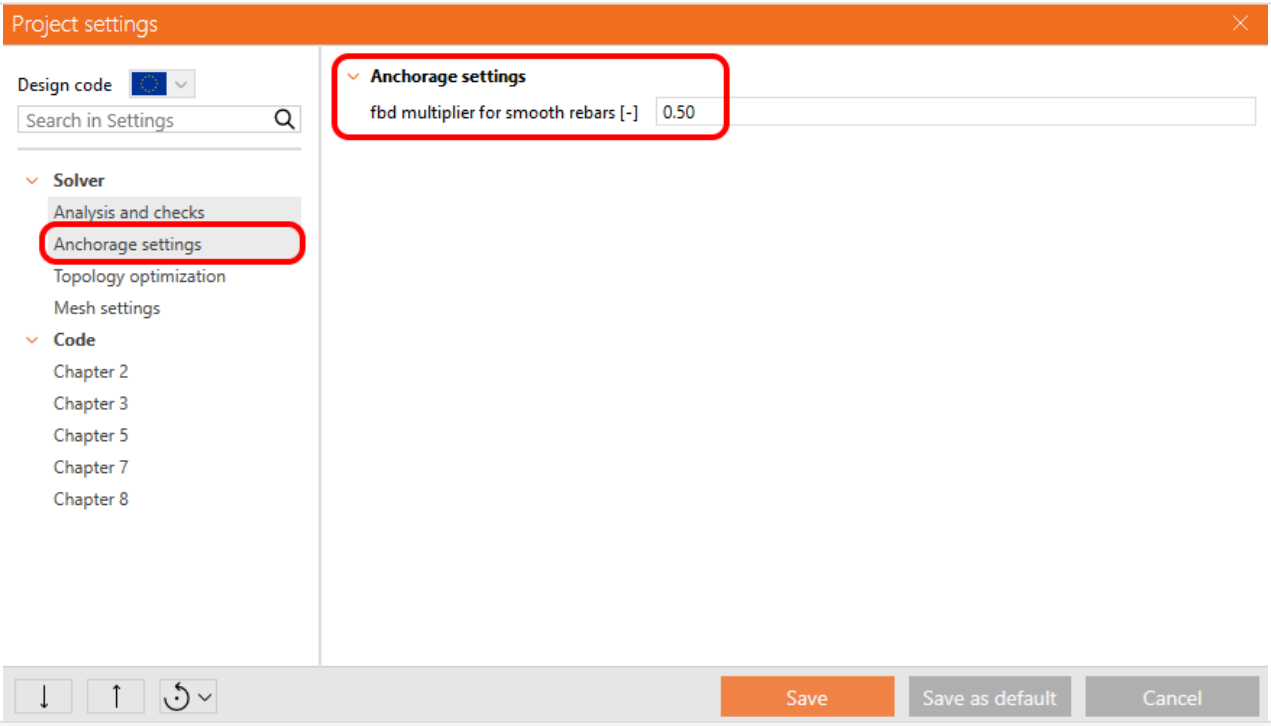
Switching the reinforcement type is possible in the **Material Tab**, where you can choose between two options: *Ribbed* to *Smooth*.




# Reduction factors

For ACI and AUS, the reduction factors for bond and anchorage length follow code definitions; for EN, values are inspired by fib Model Code 2010 but can be adjusted by the user.

For consistency, the same reduction applied to bond strength in each design code is also used to scale the **anchorage coefficient β** in the CSFM (Compatible Stress Field Method) model. This ensures that the anchorage types, straight bar, hook, loop, etc., are correctly adjusted when smooth rebars are selected.



# Code-specific adjustments for smooth rebars

- **EN 1992-1-1**: Eurocode 2 does not explicitly cover smooth reinforcement in terms of bond strength (EN 1992-1-1:2004, CL. 8.4.2). For crack spacing calculation (CL. 7.3.4), the code implicitly assumes ribbed reinforcement. Based on research and guidance from fib Model Code 2010 (CL. 6.1.5.3.3, Table 6.1-3), the bond of smooth rebars is taken as 0.5–0.67 of ribbed rebars. In Detail, this is implemented as a user-defined multiplier “fbd multiplier for smooth rebars”, defaulting to 0.5.
- **ACI 318-19**: Current ACI 318 (CL. 25.4.2.1) addresses development length only for deformed bars. Historical provisions in ACI 318-63 (§1402) specify that the required development length for plain (smooth) bars is twice that of deformed bars. Accordingly, Detail adopts a 0.5 bond reduction factor, consistent with this long-standing definition.
- **AS 3600:2018**: Clause C13.1.3 explicitly defines the development length for plain (smooth) bars as 1.5 $L_{sy,t}$, compared to $L_{sy,t}$ for deformed bars. This corresponds to a bond reduction factor of 0.67, which is directly applied in anchorage and crack spacing checks.

The **Tension chord model** and **Pull-out model** were updated to reflect the reduced bond and different crack development of smooth rebars. For more information on the models, see the **Theoretical Background.**

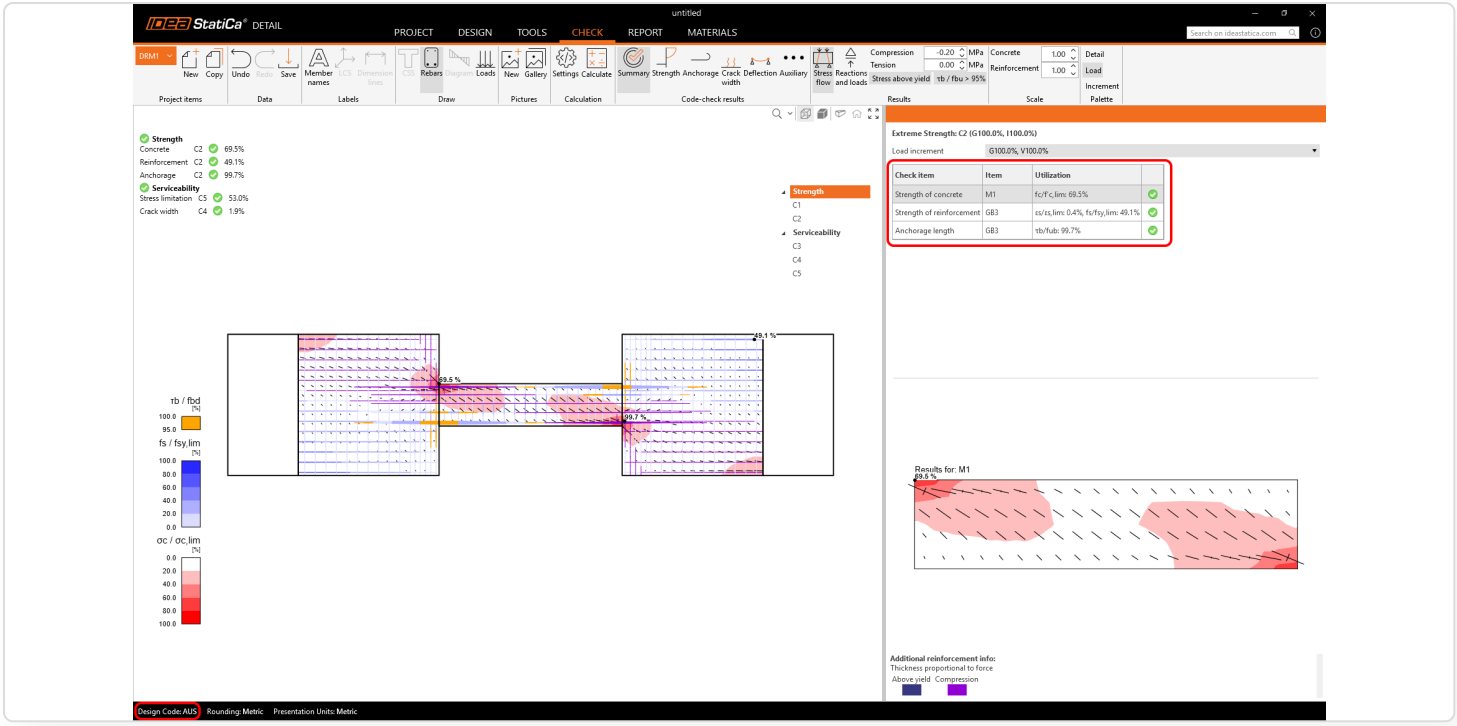
# Australian code for Detail

The Detail application includes the Australian Standard with full functionality of all concrete modules, including prestressing and anchoring.
This integration ensures that engineers working with AS 3600 have the option to design discontinuity regions using CSFM. CSFM (**Compatible Stress Field Method**) is an innovative approach that allows engineers to move away from rough estimations and solve tasks using precise calculations. This provides Australian engineers with a modern alternative to the outdated Strut and Tie method, allowing them to efficiently design and assess **walls and beams with openings, pile caps (in 2D), pier caps, diaphragms, anchoring (in 3D) and other discontinuity regions** in concrete structures.

Learn more about the method implemented in IDEA StatiCa Detail and its practical applications in the article: CSFM explained.

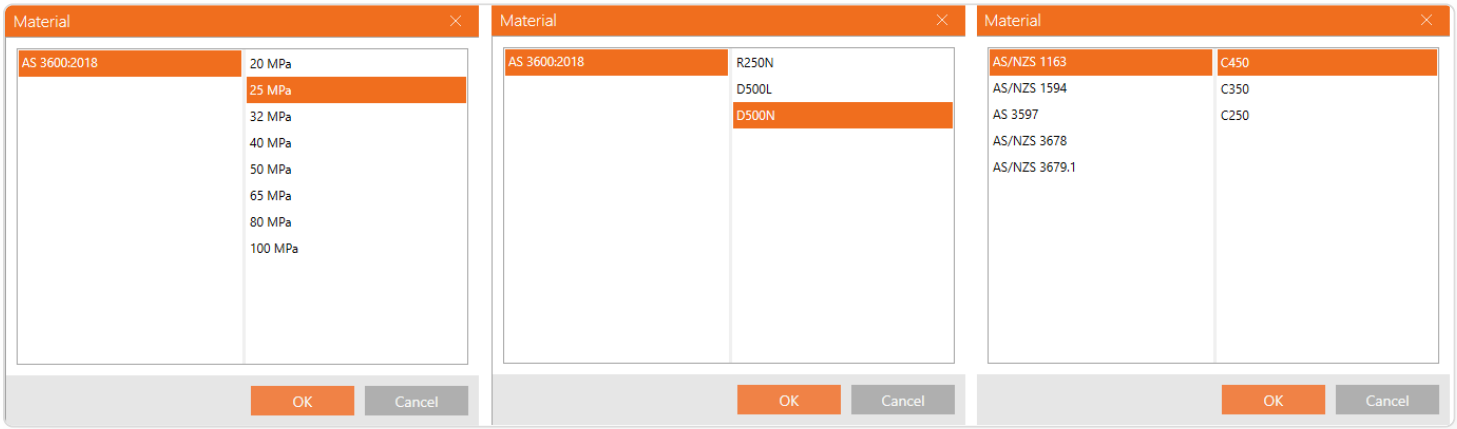
# Discontinuity regions

Users can utilize the **full functionality of Detail (in 2D)**, adapted to **AS 3600 terminology, material properties, and verification checks**. The user interface aligns with Australian standards, making the design process more intuitive and ensuring that calculations follow local industry requirements.

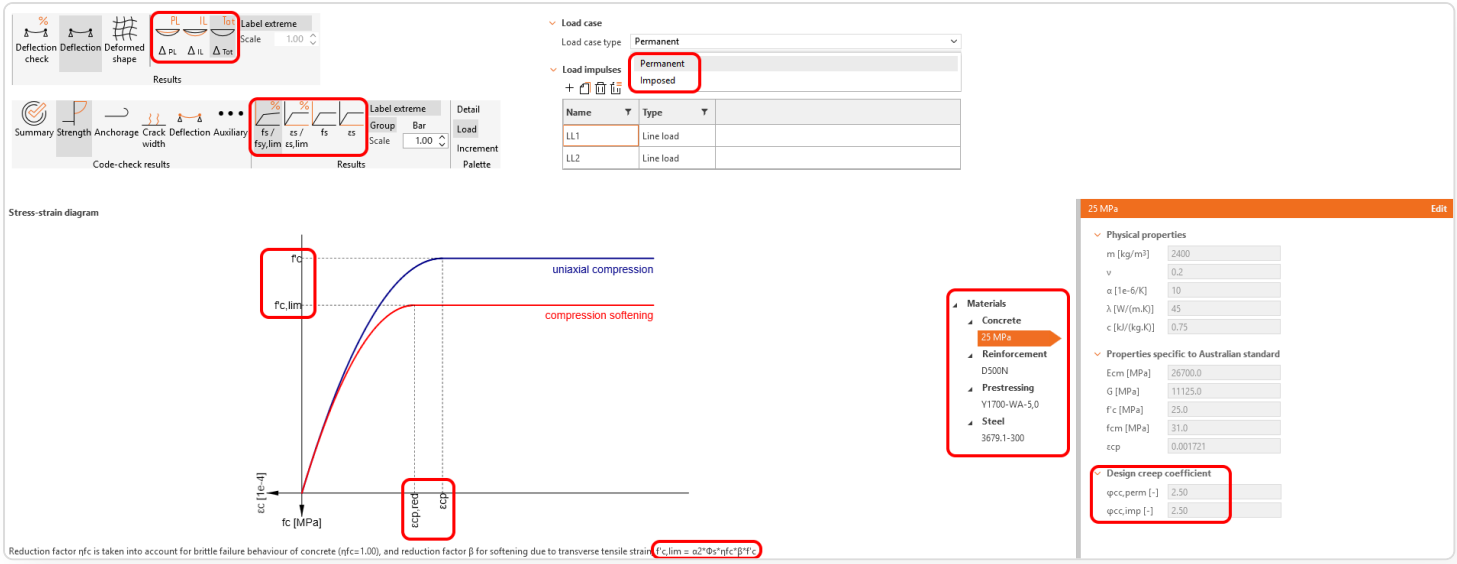



## Key features

- **Material Database** – A built-in database includes concrete, reinforcement, and steel materials defined by AS 3600.




- **Standardized Nomenclature** – Variable names and labels throughout the application match AS 3600, ensuring clarity in design and verification.
- **Refined Calculations to AS 3600**




1 **Stress-strain diagrams** for serviceability and strength, incorporating **long-term effects using the design creep factor**.

2 **Bond strength and B-factor** calculations for anchorage springs, ensuring compliance with AS 3600.

*Released in IDEA StatiCa version 25.0.*

# Prestressing

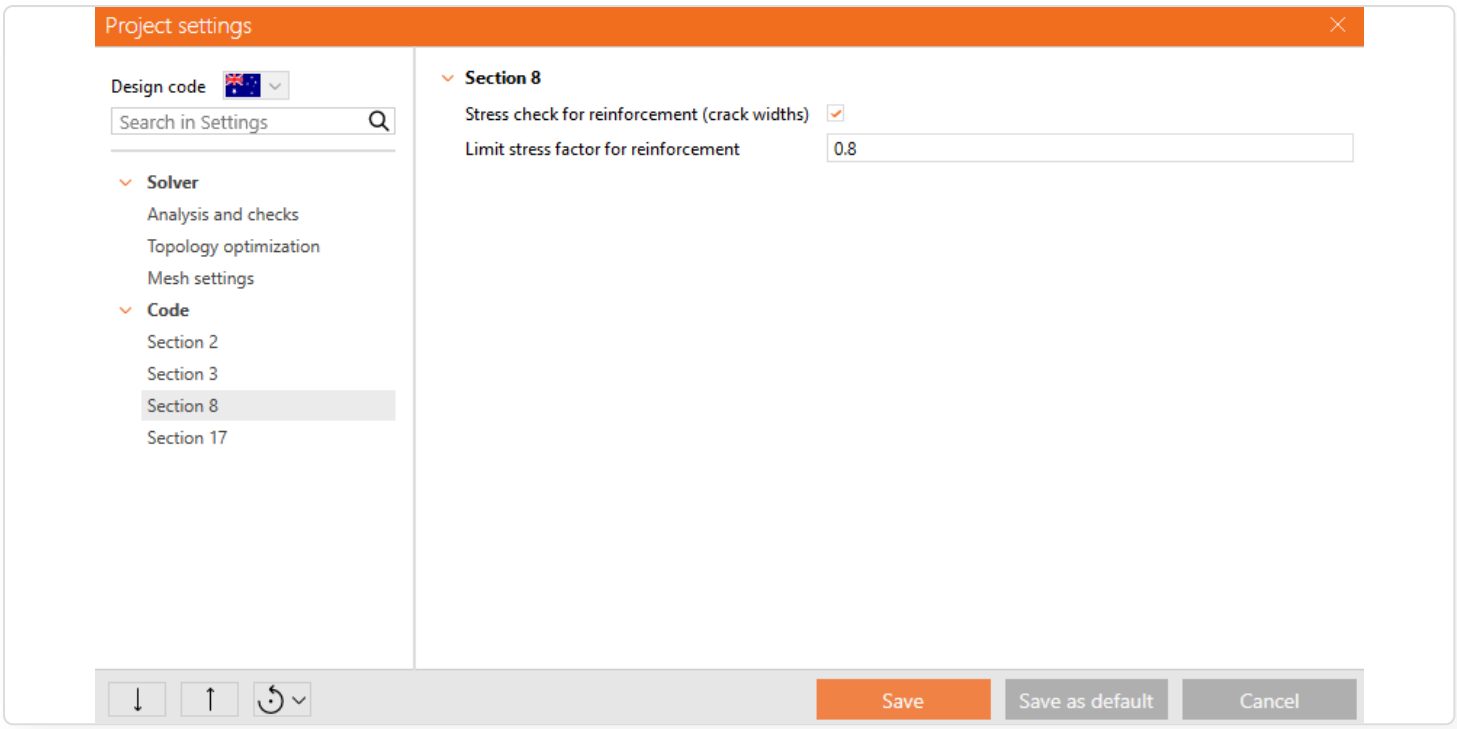
Prestressing has been fully implemented according to **AS 3600:2018** in Detail 2D. A dedicated material database for wires, tendons, and bars has been added, along with project settings specific to prestressing such as stress increments, stress reduction, and design limits. The implementation includes anchorage zones and partially loaded areas with dispersion angles as defined by AS 3600, as well as a dedicated bond model for prestressing tendons.

All relevant verification checks are available, including strength, anchorage, stress limitations, and crack width, in full compliance with AS 3600.

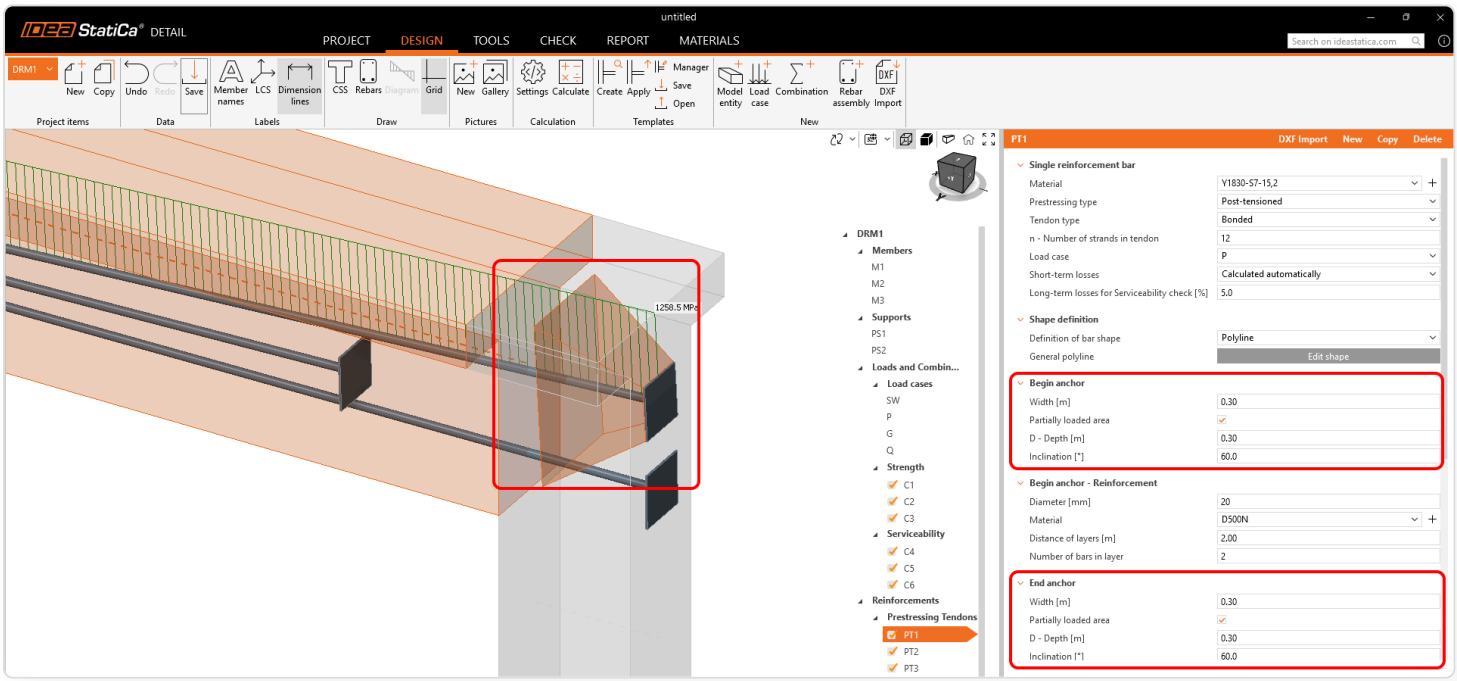
All templates are also available, just like for the other codes.

### Key features

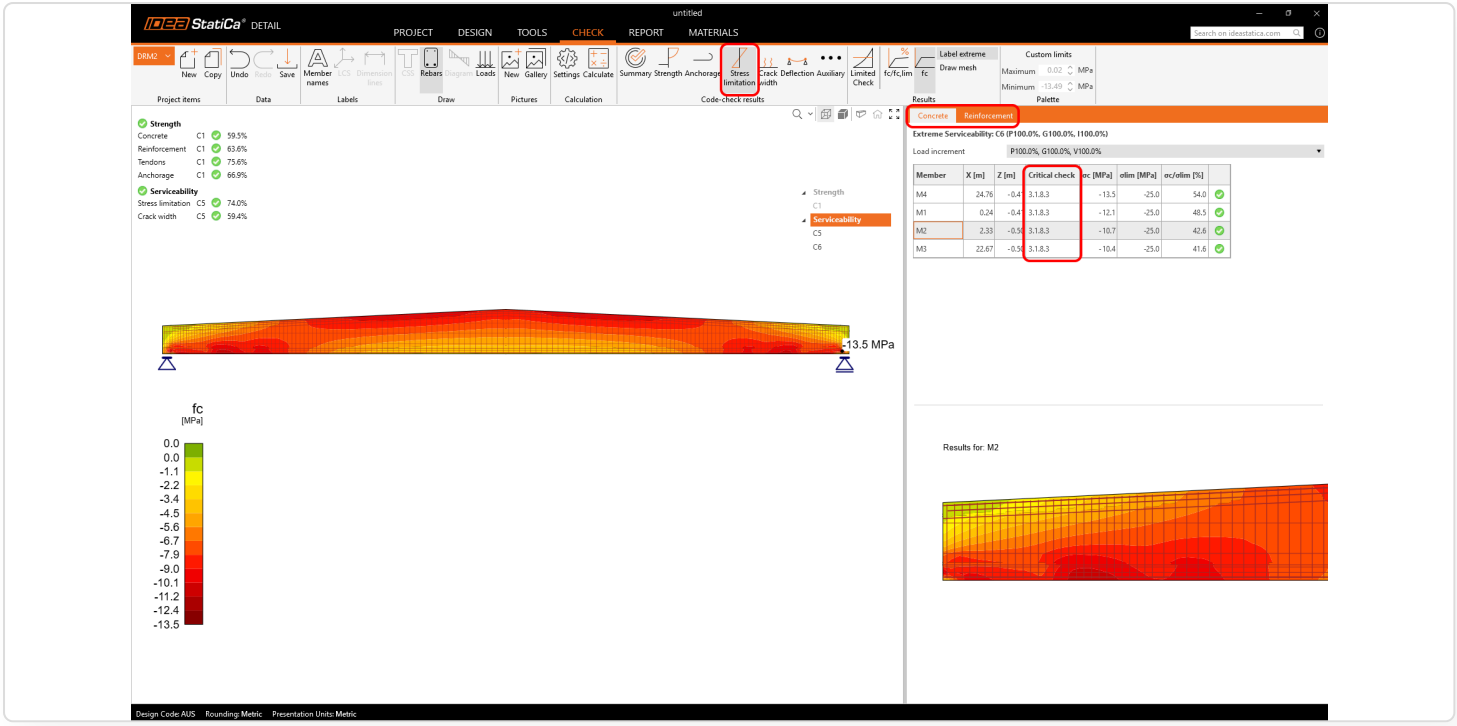
- **Material database** for wires, tendons, and bars per AS 3600 Section 3.3 Properties of tendons.
- **Project settings** with prestressing-specific factors (increments, stress reduction, concrete/reinforcement stress limits).




- **Partially loaded areas** (anchorage zones and point supports) with a 60° dispersion angle per AS 3600 chap. 8.1.4.




- **Bond model** for prestressing tendons, including neglecting bonds over 0.1 Lpt as required by the code, clause 13.3.2.
- **Expanded results** to include Strength, Anchorage, Stress limitation, and Crack width checks, with stress limitation checks aligned with AS 3600 sec. 3.4.3.3 (concrete) and sec. 8.6.2.2 (reinforcement).



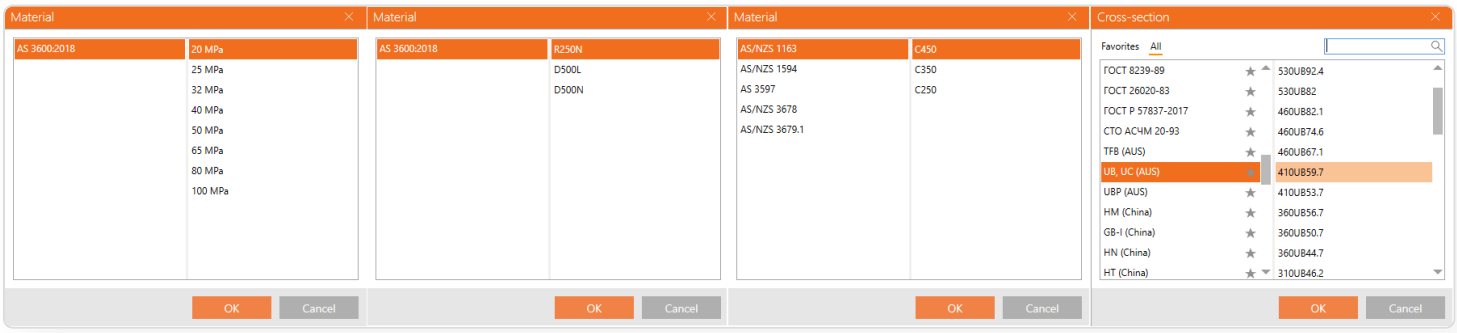

*Released in IDEA StatiCa version 25.1.*

# Anchoring

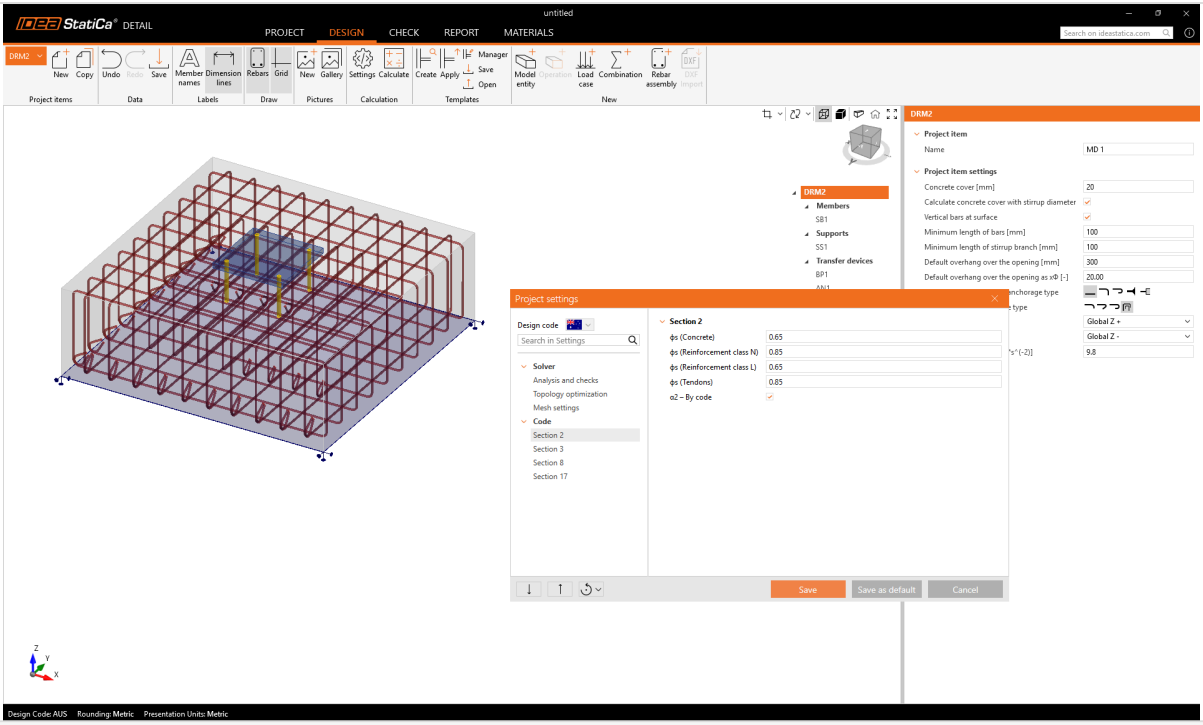
Anchoring functionality in **Detail 3D** is available fully in line with AS 3600. This eliminates the need for Australian engineers to approximate their designs using EN or ACI standards. The module includes an Australian-specific material and cross-section database, adjustments to reinforcement and bond behavior, and code settings reflecting local rules. All templates are available as for EN and ACI, with seamless exports from the Connection application.

## Key features

- **Database of Australian materials** and cross-sections.




- Code settings to reflect **local design rules**.




- **Export from the Connection** application into Detail 3D.

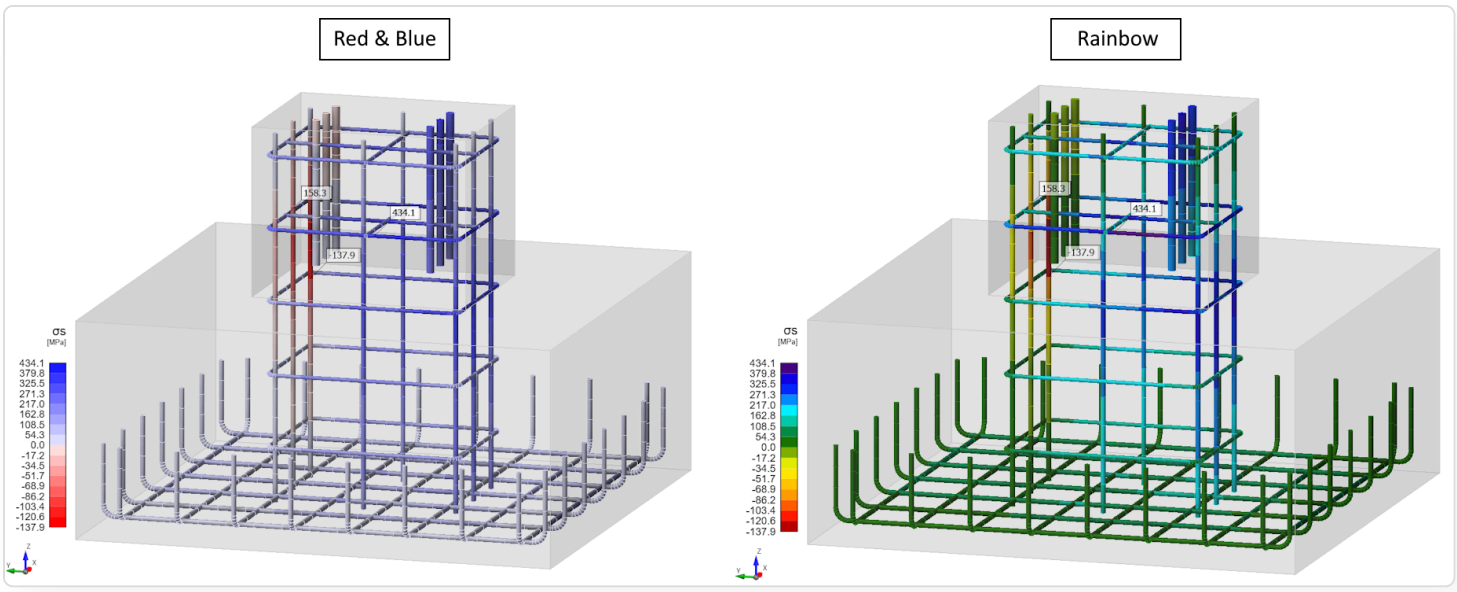
*Released in IDEA StatiCa version 25.1.*

# Advanced results presentation

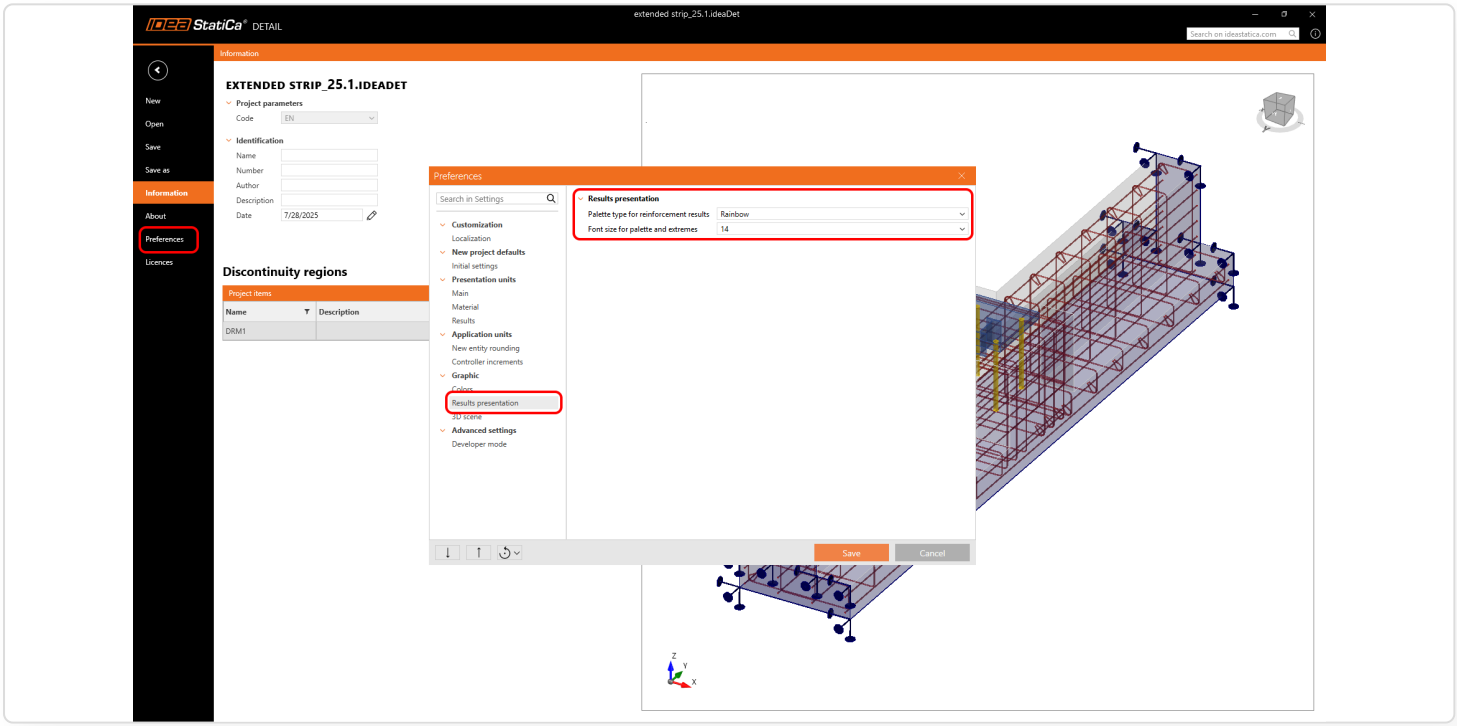
The Detail application offers a user-customizable view of results to match individual preferences. It can be used not only for browsing results but also for generating more user-friendly reports. Advanced options include adjustable palette limits, value fonts, and more.

# "Rainbow Colors" for reinforcement

For displaying reinforcement results, it is possible to choose between so-called rainbow coloring or red-and-blue coloring (which distinguishes between tension and compression).

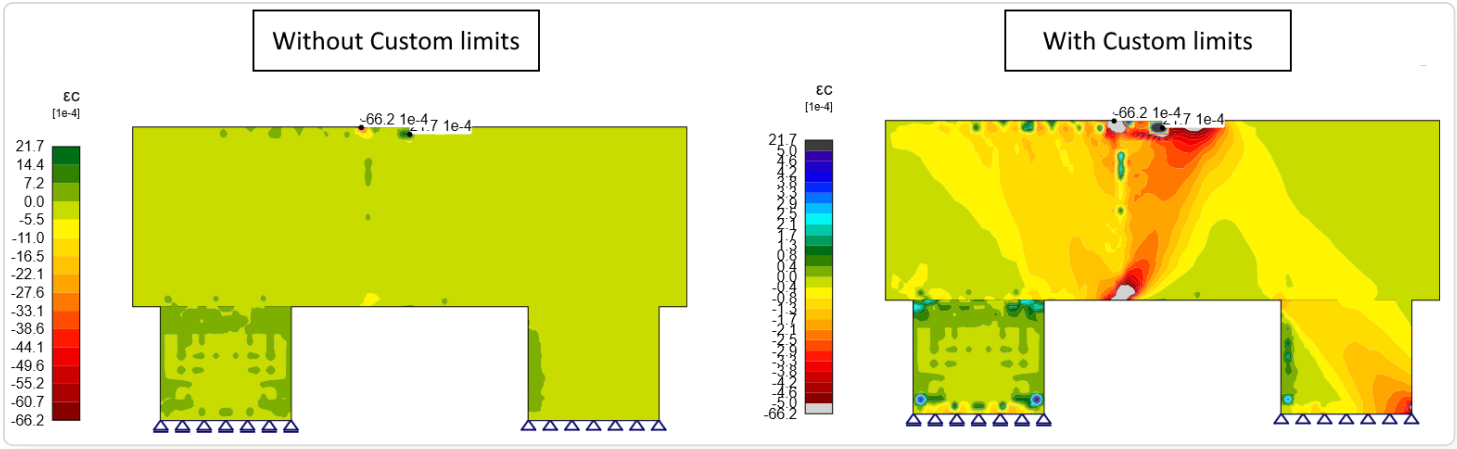



This setting can be adjusted in the preferences dialog and is saved with the application settings for future projects. Similarly, it is possible to define the font size for values displayed in the scene.

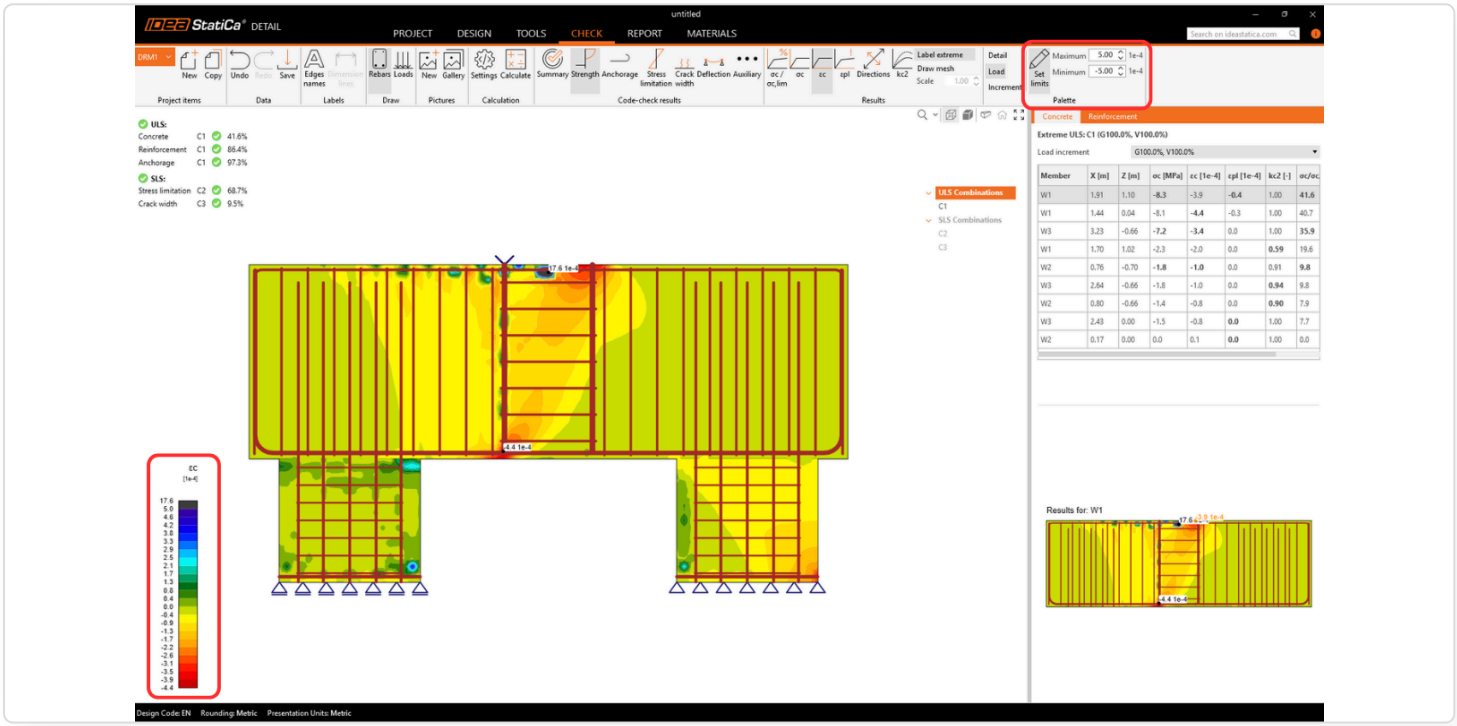



# Custom limits for the results palette

In the main ribbon, it is also possible to set custom limits for the color palette used to display results. This is particularly useful in cases where peaks with concentrated stresses occur, which would otherwise make it difficult to read the results and see the stresses in the rest of the structure.




These settings are temporary (not saved with the project), but **they are saved for each view.** So when switching from one result to another and back, your settings are persistent. Moreover, figures can be saved to the gallery and then added to the report.

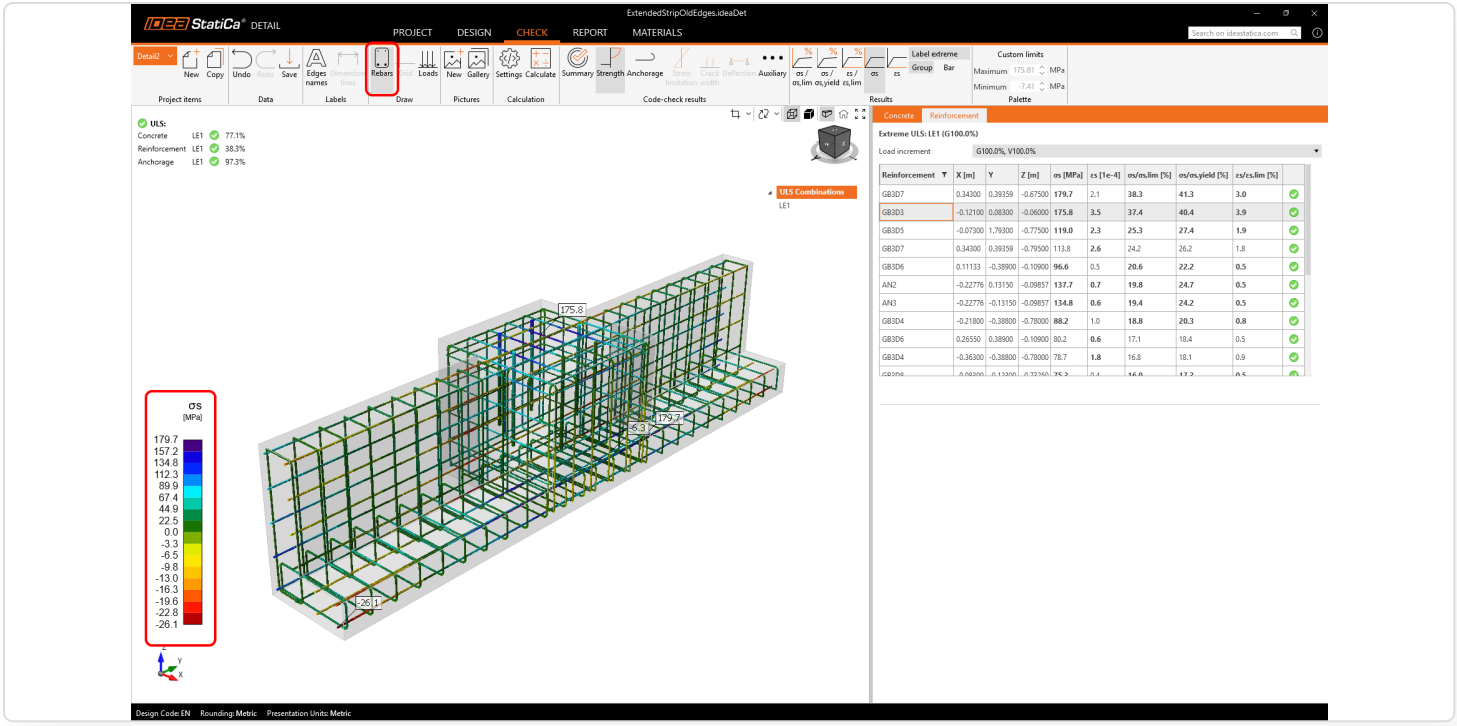



# Drawing results for selected reinforcement only

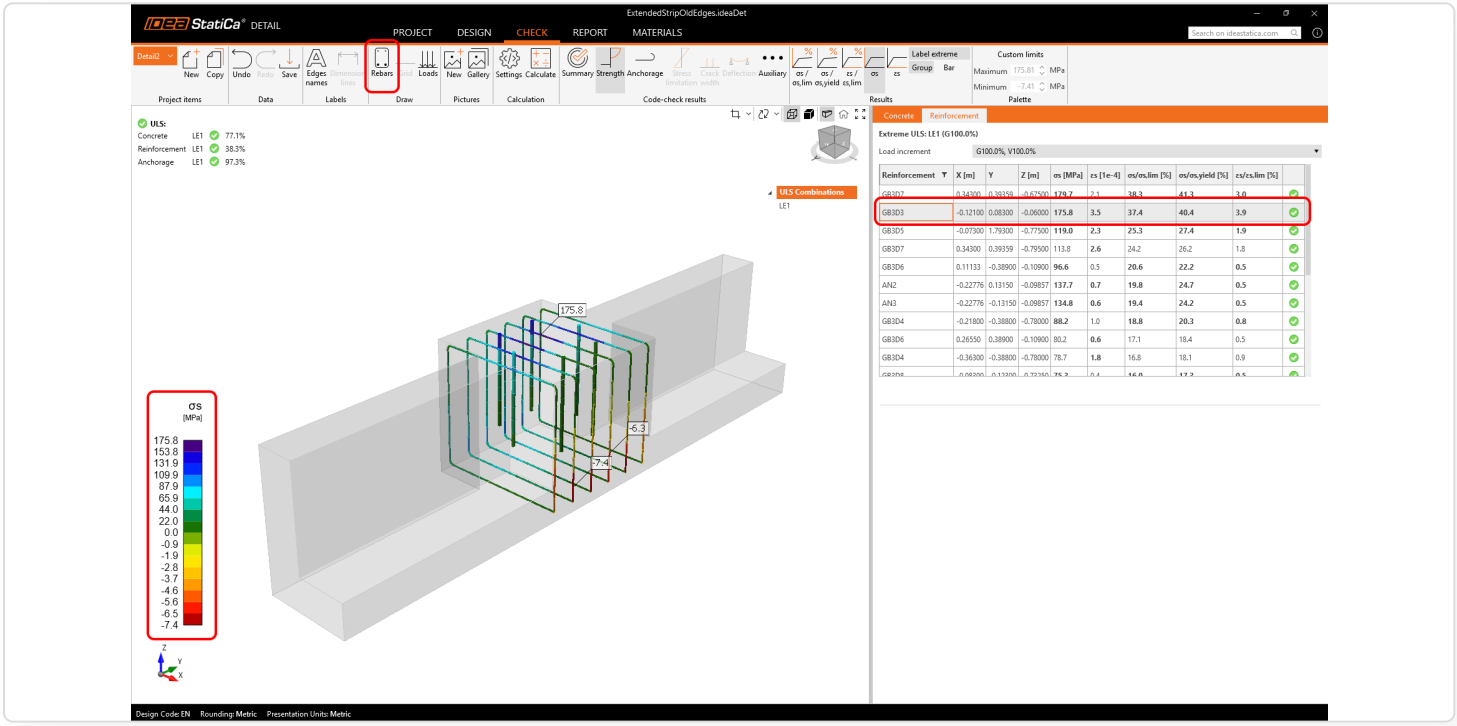
For **Strength** results on reinforcement (e.g., stress, strain) and **Anchorage** results (e.g., bond stress, Ftot, Flim) there is option to display only selected reinforcement.

The option can be found it the main tab:

- When **Rebars = ON**, result plots display on all reinforcement groups (default behavior).




- When **Rebars = OFF**, the scene shows results only on the reinforcement group currently selected in the Results table (vprvoright Property Grid).

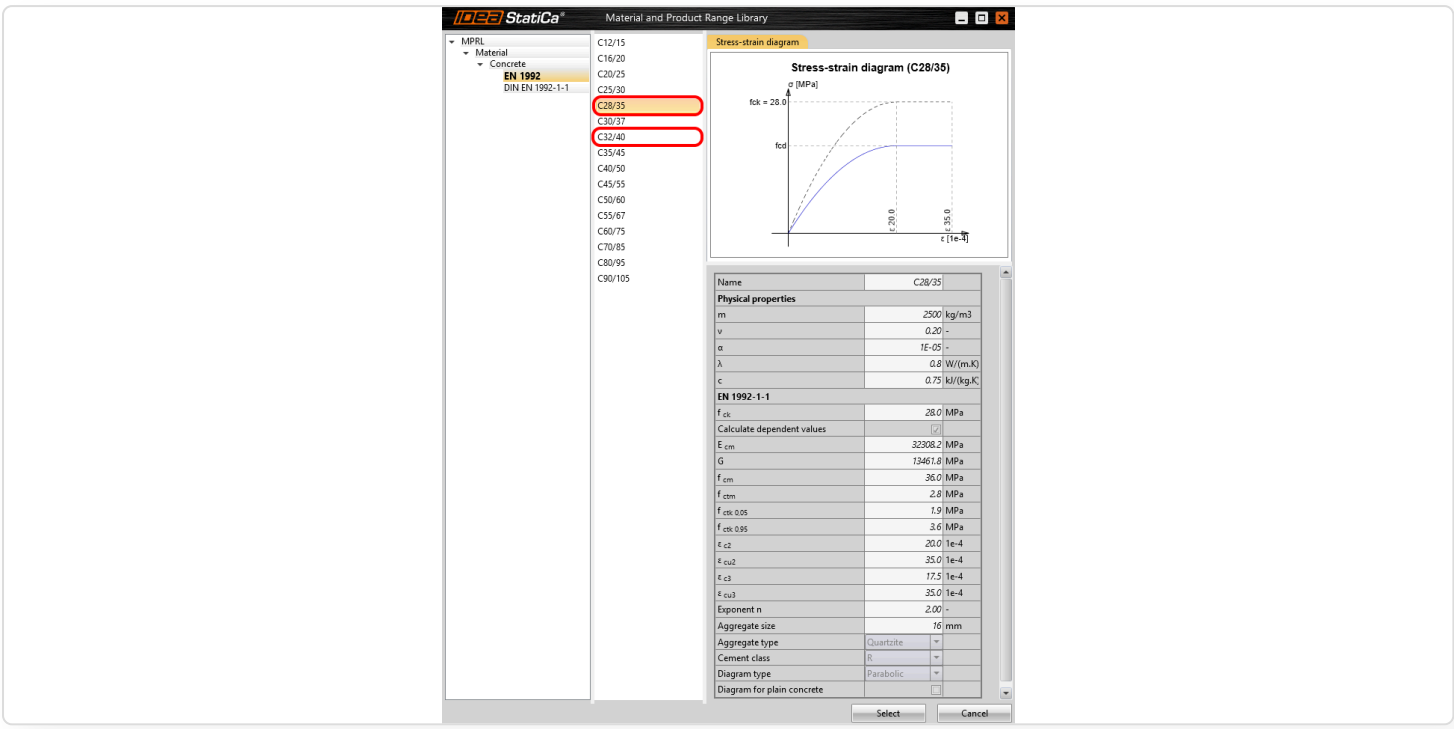



Any filtered view can be **saved to the Gallery** as a user-defined image and then inserted into the report, eliminating the need for external editing.

# Concrete materials for the UK market

Detailed support of concrete material commonly used in the UK. The specific grades are included in the default materials library, meaning no need for manual modification for each project.
The concrete grades **C28/35** and **C32/40** are implemented in default materials library, the MPRL.

These grades are immediately available in all applications where concrete materials are used, primarily Detail, RCS, and Beam, and they come with correct properties aligned with EN 206.

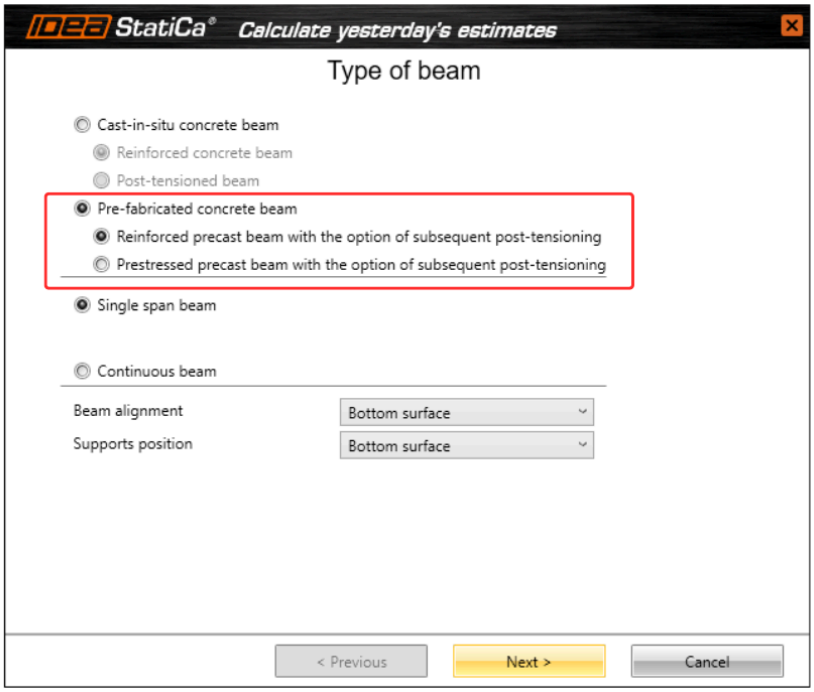



**Beam**

# Lateral torsional buckling for prefabricated beams

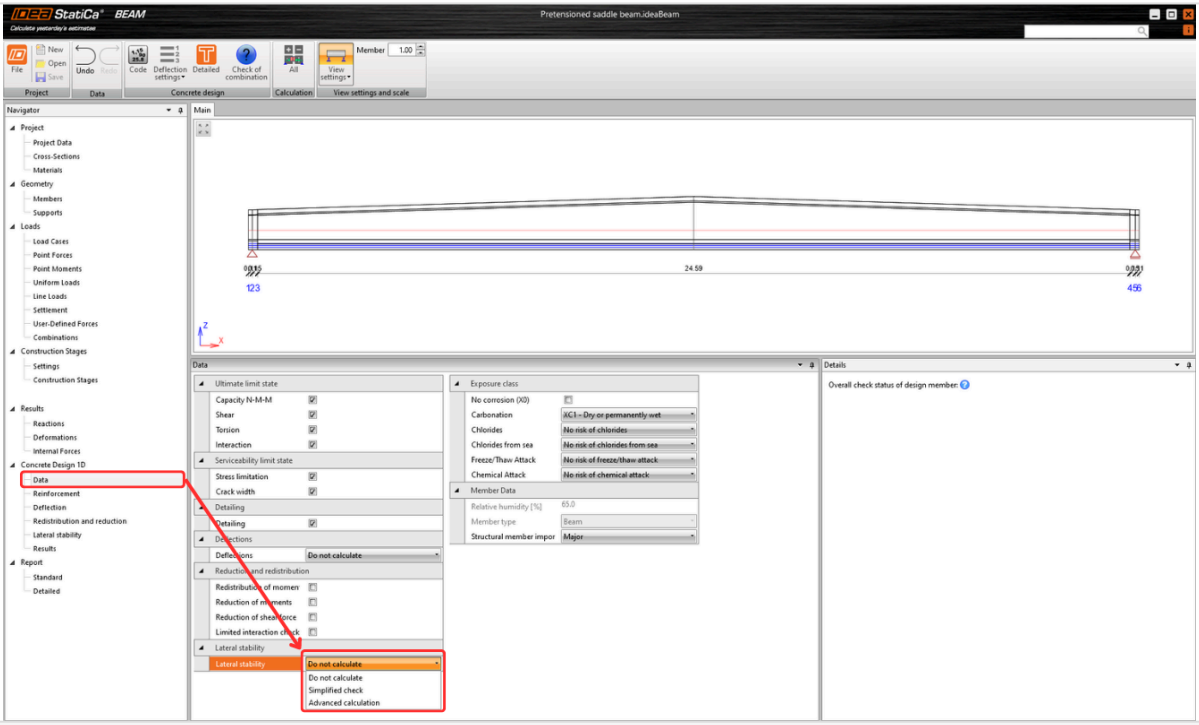
Lateral torsional buckling (LTB) is a stability issue that primarily affects slender members such as prefabricated beams. To address this issue, an advanced geometrically and materially nonlinear analysis incorporating initial imperfections is implemented in IDEA StatiCa Beam.
LTB is a stability failure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

The geometrically and materially non-linear calculation released in 24.1 can capture the mentioned effects and provide results of internal forces including second-order effects. Calculated **internal forces for selected sections are now automatically collected and sent to RCS for detailed code-checks (available from version 25.0).**




# LTB

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.



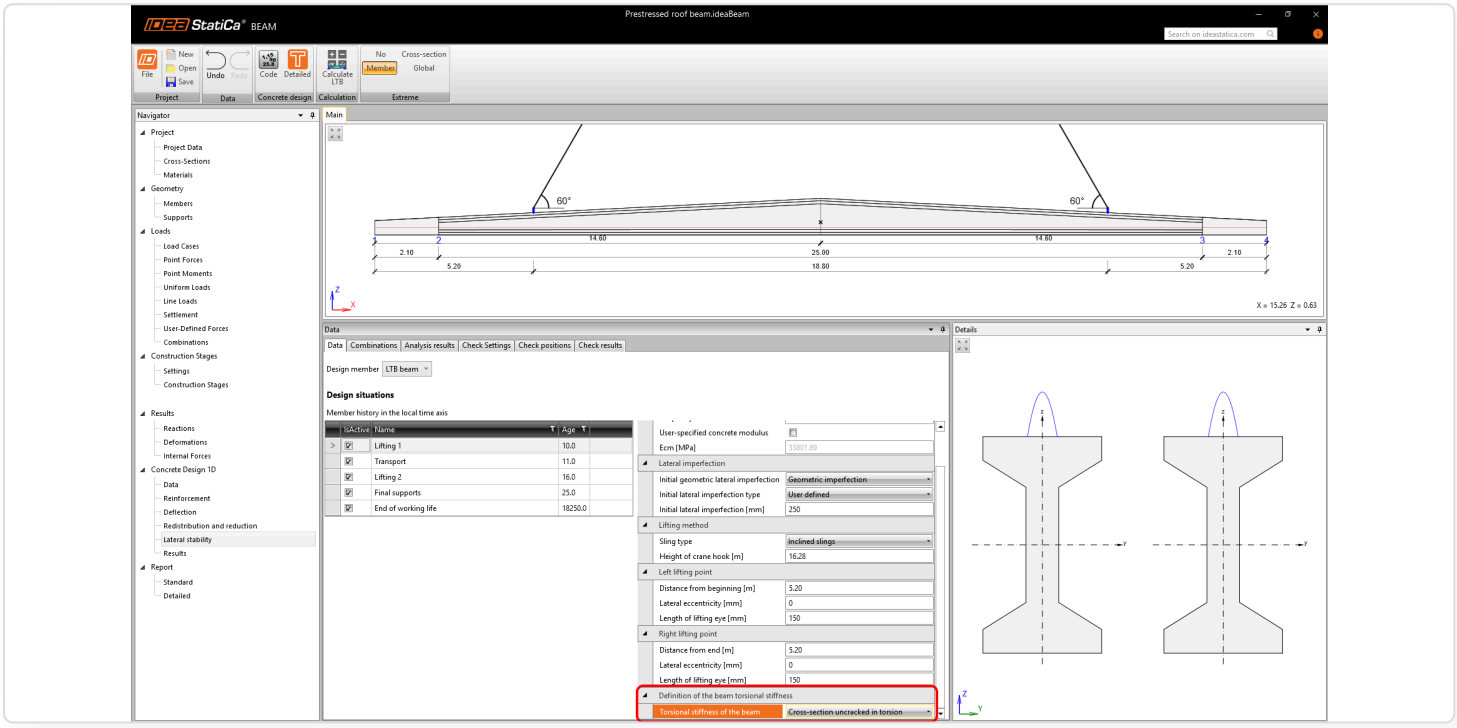

# Setting the analysis

**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.

### Torsional stiffness

In the setting, there is the option (since v25.1.) to define torsional stiffness more accurately. In the drop-down menu *"Torsional stiffness of the beam",* whether the beam cross-section should be considered **uncracked** or **cracked** in torsion can be selected.




- *Uncracked in torsion*: torsional stiffness is calculated as the linear torsional stiffness of the reinforced concrete cross-section.
- *Cracked in torsion*: torsional stiffness accounts for the torsional resistance provided by longitudinal reinforcement (including prestressing), stirrups, and compression diagonals, safely neglecting the concrete contribution.

The setting can be applied separately for each Design situation, giving engineers flexibility to adapt the calculation approach depending on the limit state or load combination.

*Recommendation for use: As a guideline, the option Uncracked in torsion is recommended for fully prestressed members where the decompression condition is satisfied. In contrast, for partially prestressed beams where, under ULS loading, the tensile stress in extreme fibers exceeds the design tensile strength of concrete ($f_{ctd}$), and for reinforced concrete members, we recommend switching to the more conservative Cracked in torsion assumption.*

### Imperfection

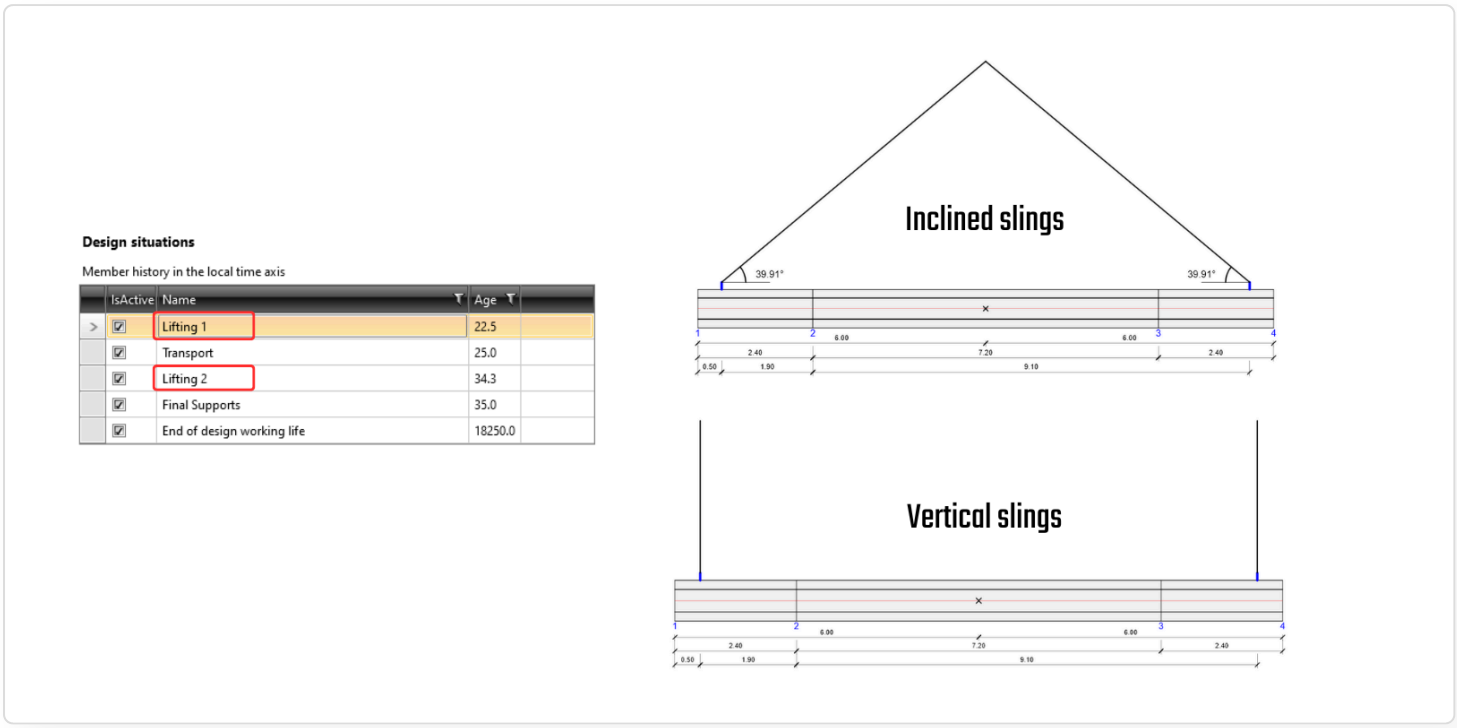
Also, the value of an initial lateral imperfection can be defined separately for each design situation. There are two options for the definition of lateral imperfection:

- 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.

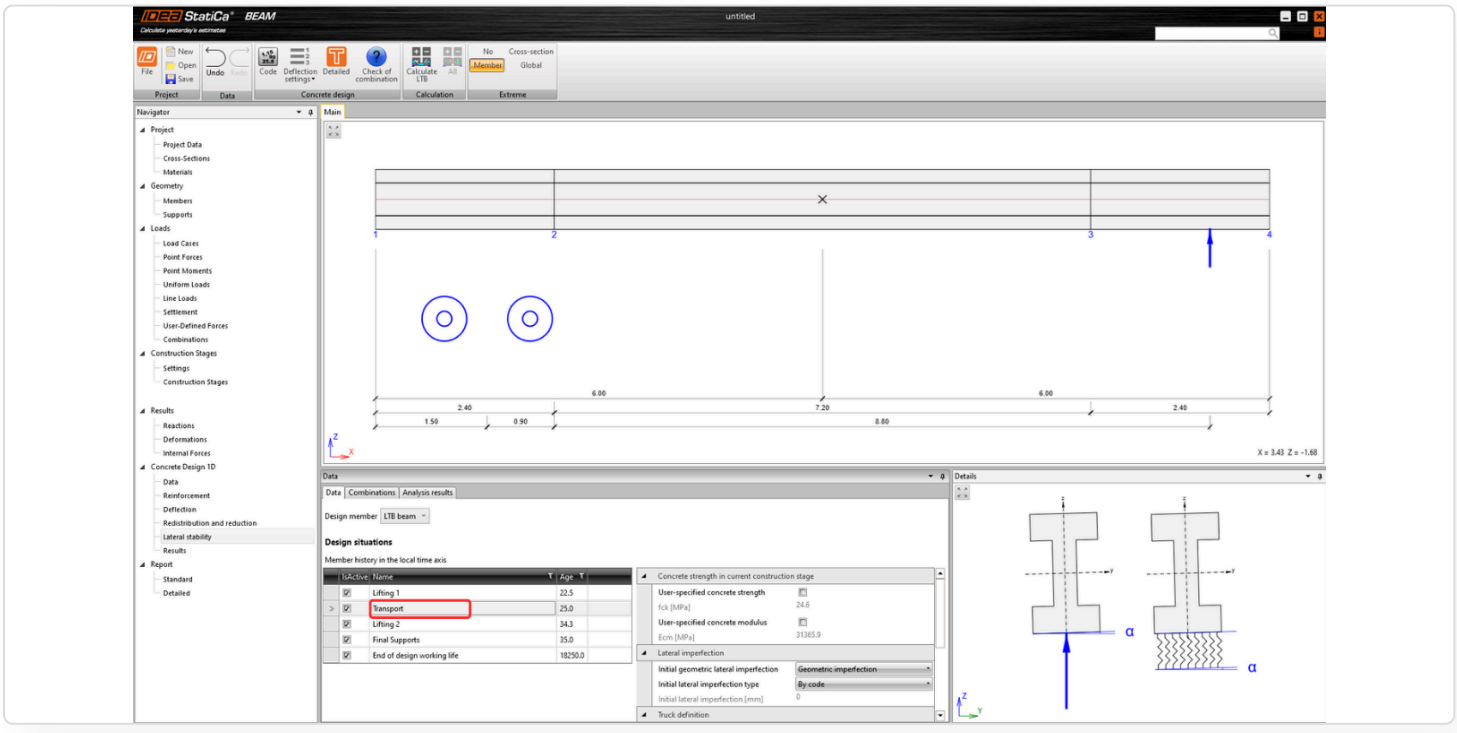
### Inputs for lifting

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.



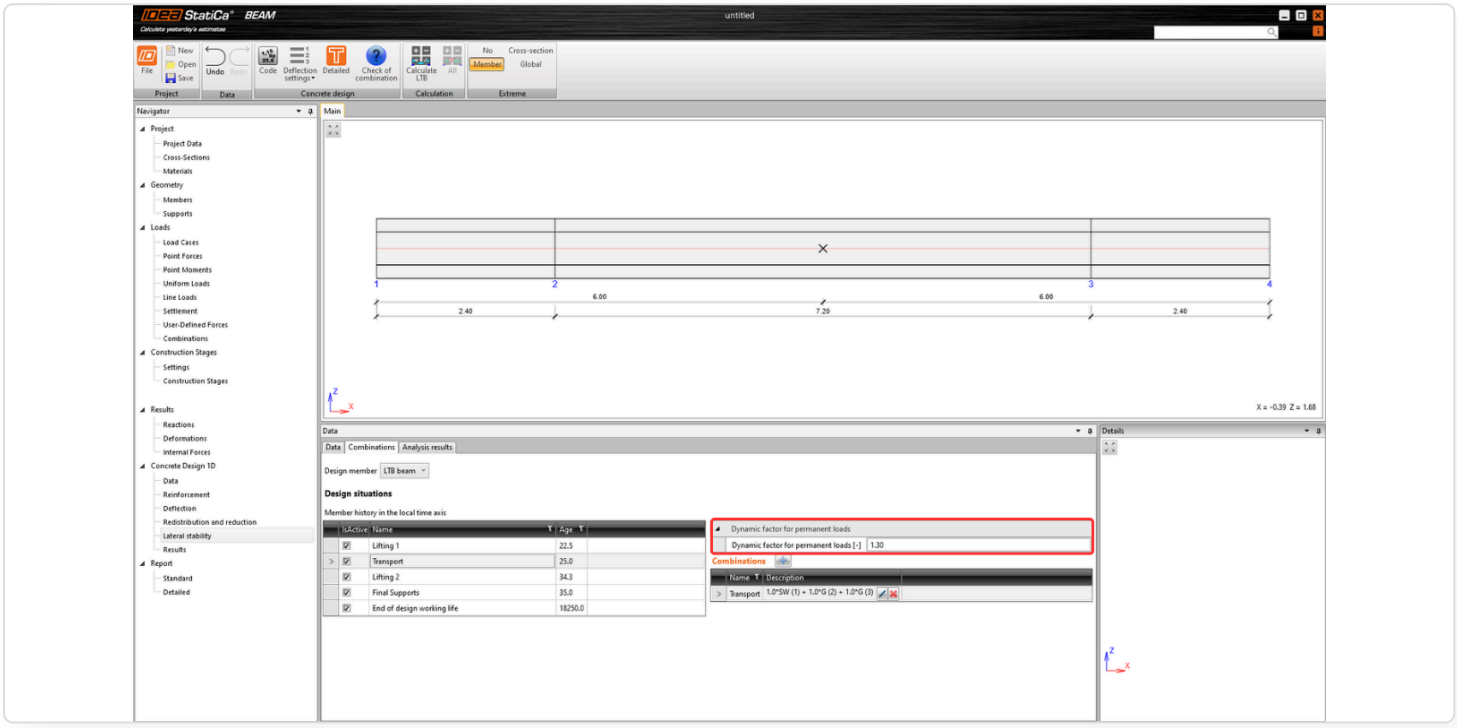

### Inputs for final supports and end-of-design working life

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.

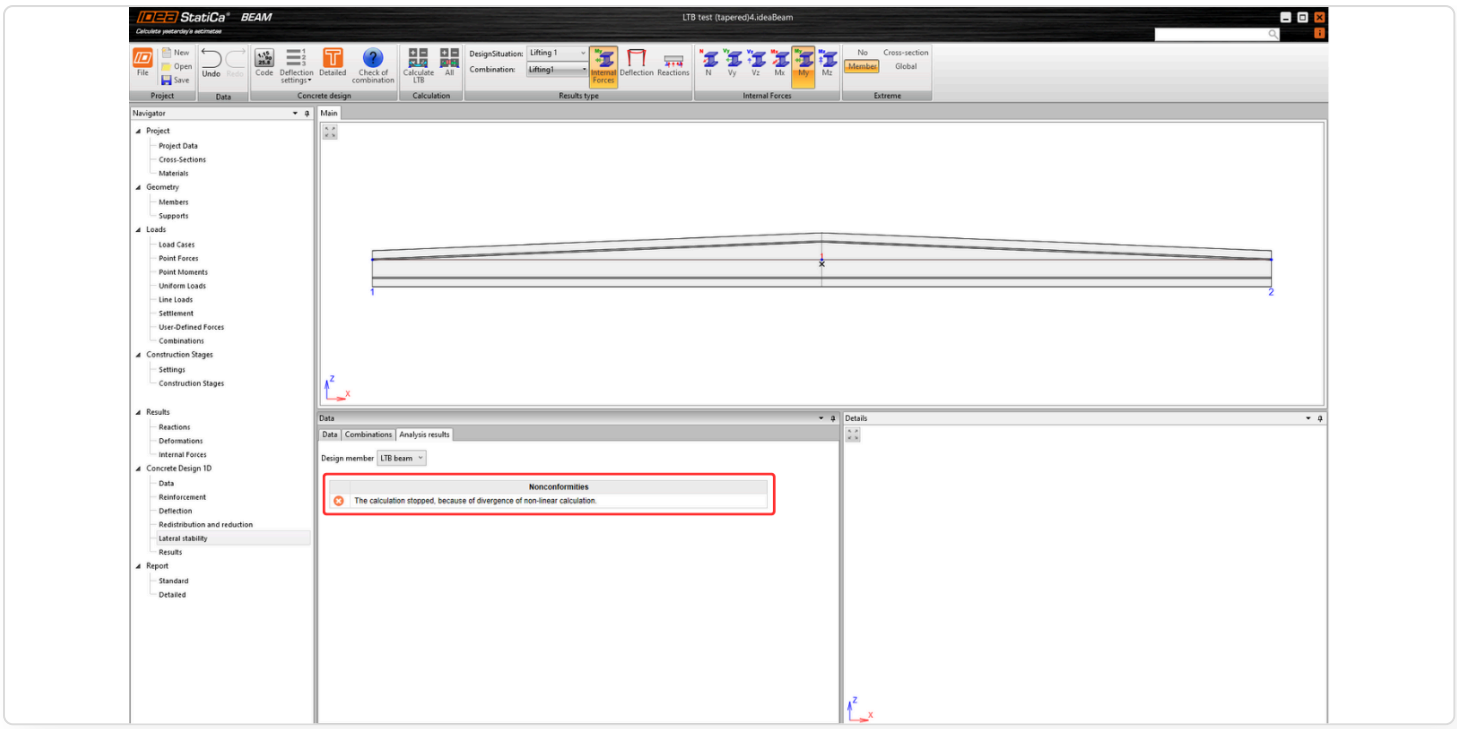
**Loads**

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.

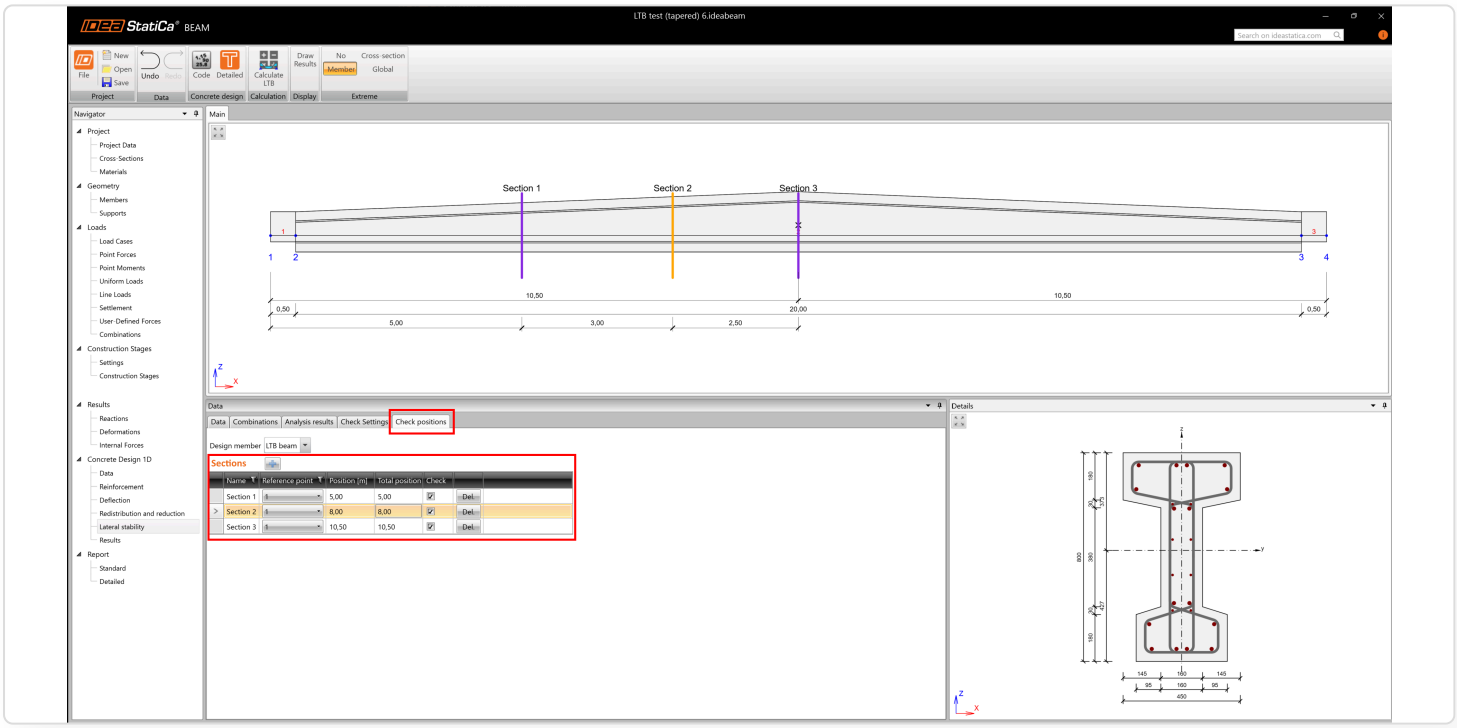



# Results

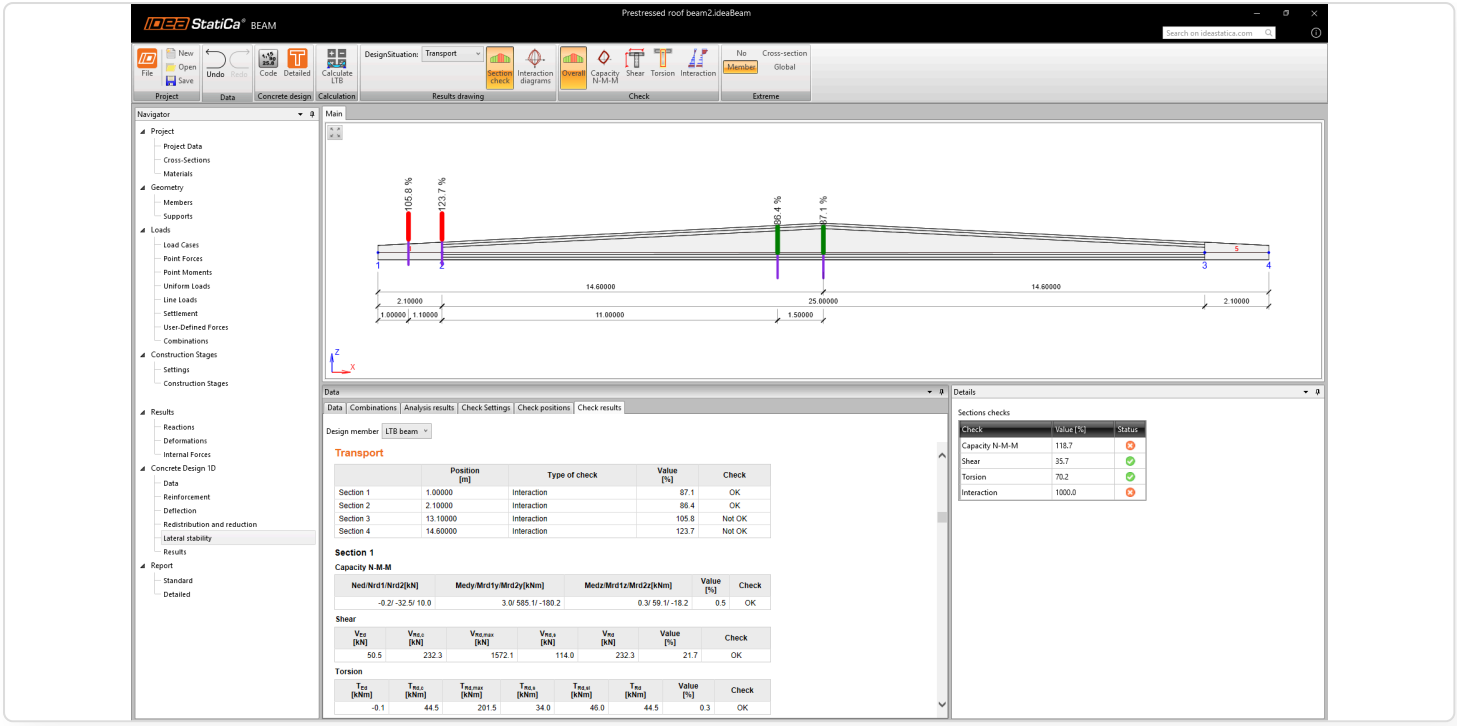
The advanced **Lateral torsional buckling (LTB**) analysis in IDEA StatiCa Beam provides (in addition to reactions, internal forces, and deformations) an evaluation of whether the beam is at risk of collapsing due to stability issues for each defined construction stage. In the event of a structural failure due to LTB, the calculation will not be complete, and the user will see an error message informing them of this situation.




The **Check Settings** tab allows you to define ULS code-check types for the selected member. In the **Check Positions** tab, you can specify multiple section locations where detailed code checks will be performed using RCS.




Detailed RCS checks for lateral stability are integrated directly in the Beam app (since v25.1).




### Note

Please be aware that advanced LTB analysis only works for models created in v24.1 and higher. The calculation will not proceed for older projects, and they must be remodeled.

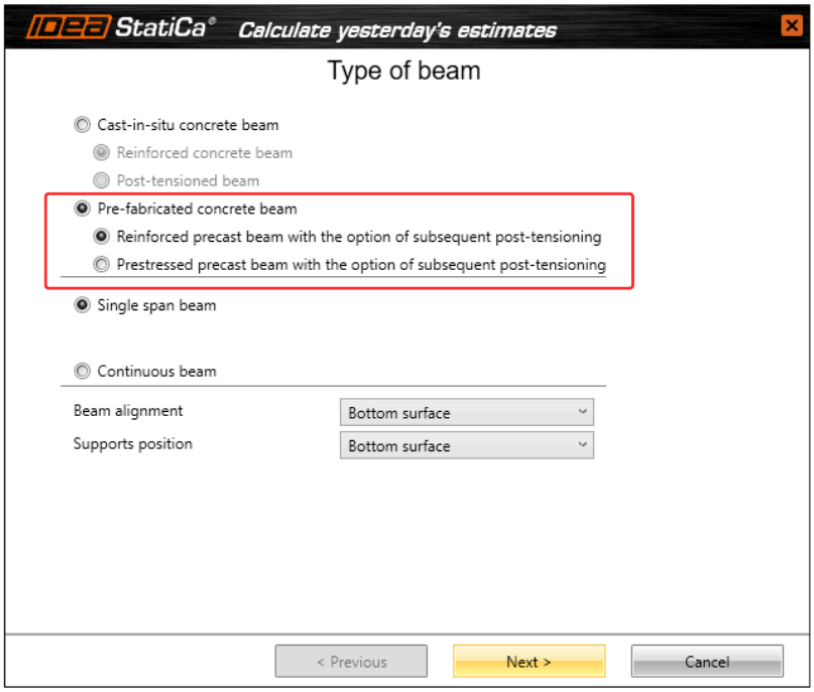
*Released in IDEA StatiCa version 24.1, improved for detailed results and torsional stiffness definition in IDEA StatiCa version 25.1.*

# 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.

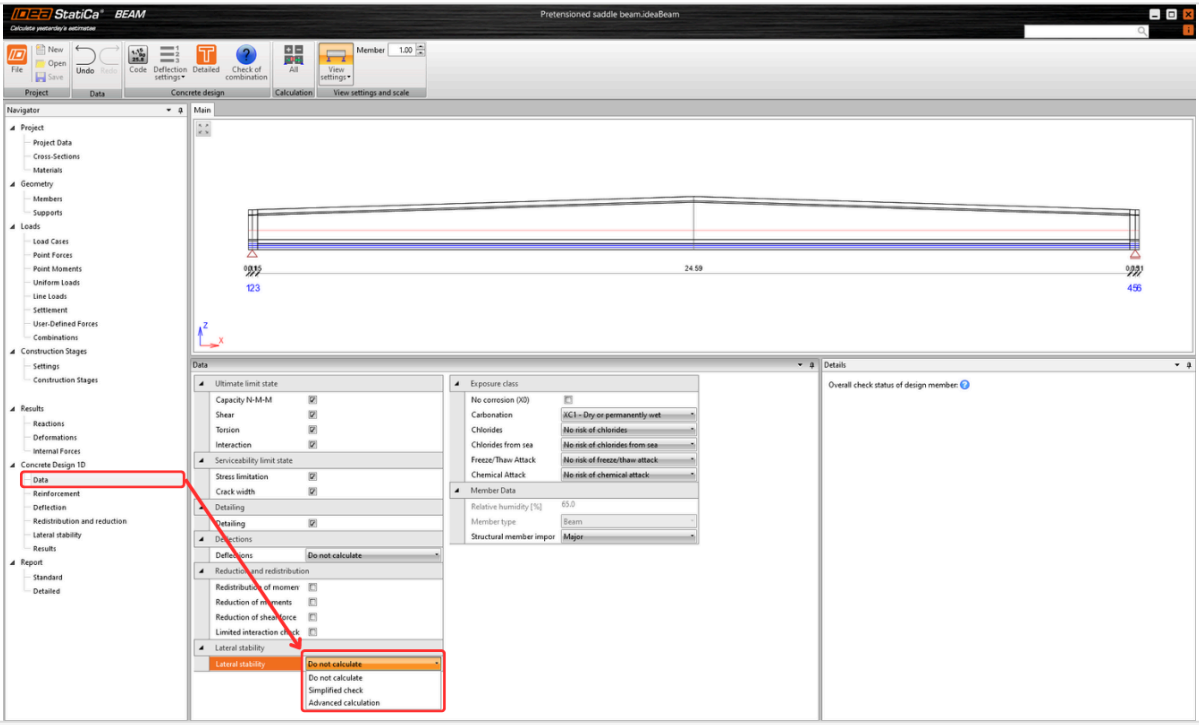
LTB is a stability failure that occurs in slender beams under bending, causing lateral displacement and twisting. It typically affects long and slender prefabricated beams. Checking for LTB is crucial to prevent sudden failure, optimize structural design, and ensure compliance with safety codes. It is 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**.

The geometrically and materially non-linear calculation released in 24.1 can capture the mentioned effects and provide results of internal forces including second-order effects. Calculated **internal forces for selected sections are now automatically collected and sent to RCS for detailed code-checks (available from version 25.0).**




# LTB

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.



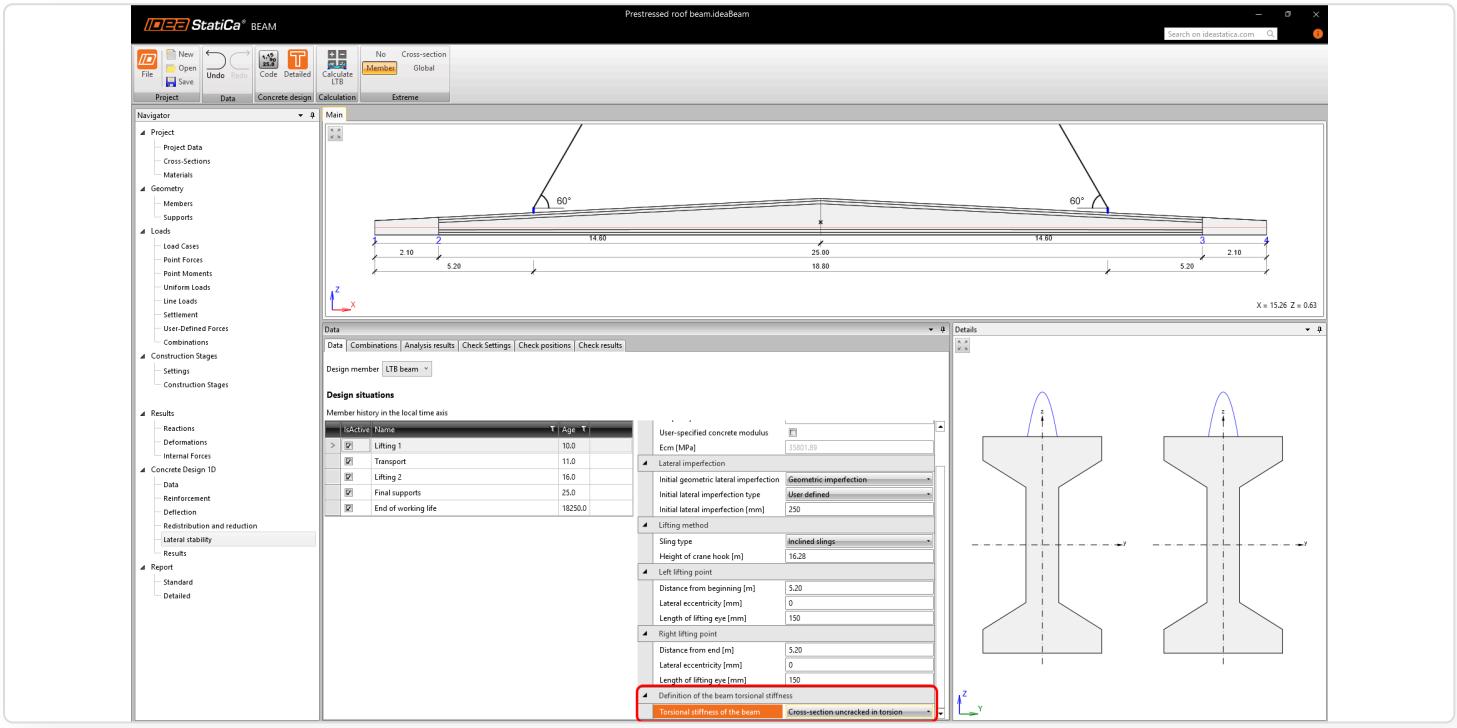

# Setting the analysis

**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.

### Torsional stiffness

In the setting, there is the option (since v25.1.) to define torsional stiffness more accurately. In the drop-down menu *"Torsional stiffness of the beam",* whether the beam cross-section should be considered **uncracked** or **cracked** in torsion can be selected.




- *Uncracked in torsion*: torsional stiffness is calculated as the linear torsional stiffness of the reinforced concrete cross-section.
- *Cracked in torsion*: torsional stiffness accounts for the torsional resistance provided by longitudinal reinforcement (including prestressing), stirrups, and compression diagonals, safely neglecting the concrete contribution.

The setting can be applied separately for each Design situation, giving engineers flexibility to adapt the calculation approach depending on the limit state or load combination.

*Recommendation for use: As a guideline, the option Uncracked in torsion is recommended for fully prestressed members where the decompression condition is satisfied. In contrast, for partially prestressed beams where, under ULS loading, the tensile stress in extreme fibers exceeds the design tensile strength of concrete ($f_{ctd}$), and for reinforced concrete members, we recommend switching to the more conservative Cracked in torsion assumption.*

### Imperfection

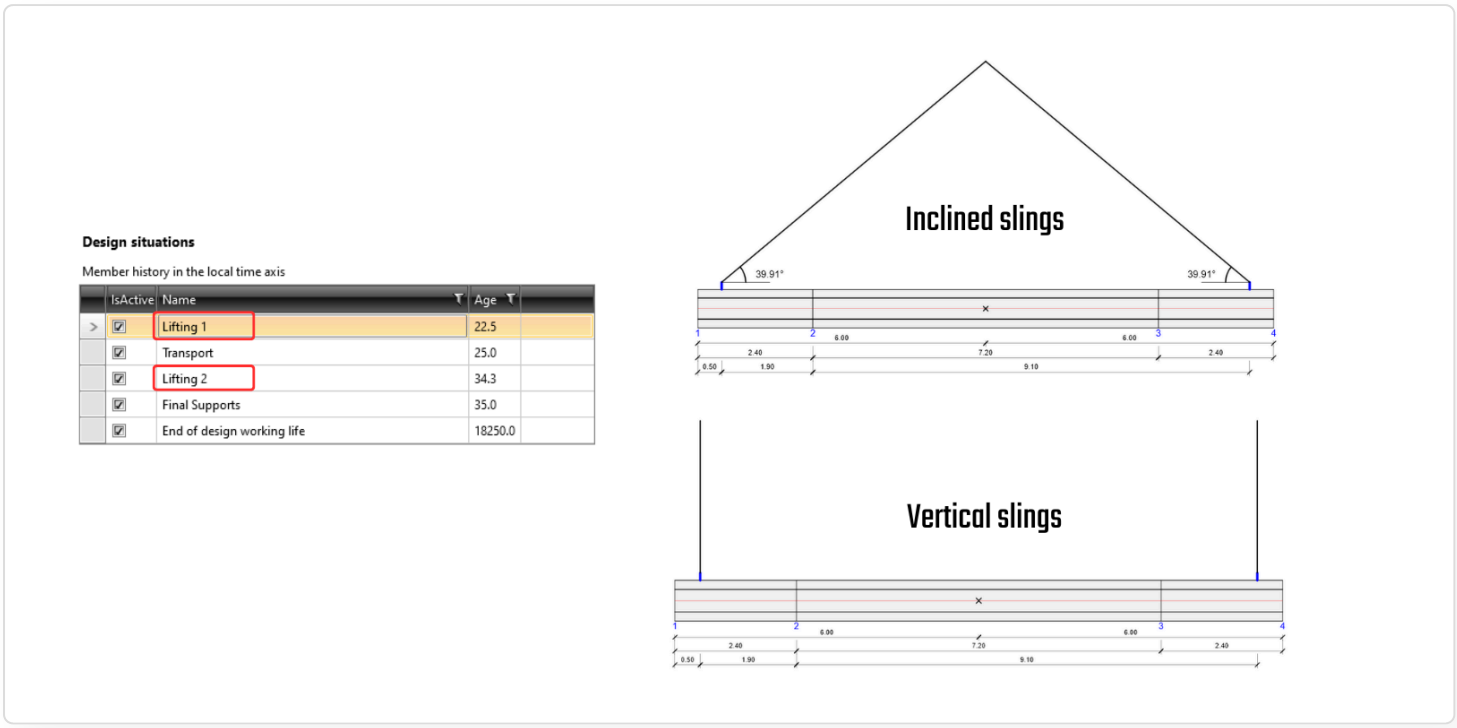
Also, the value of an initial lateral imperfection can be defined separately for each design situation. There are two options for the definition of lateral imperfection:

- 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.

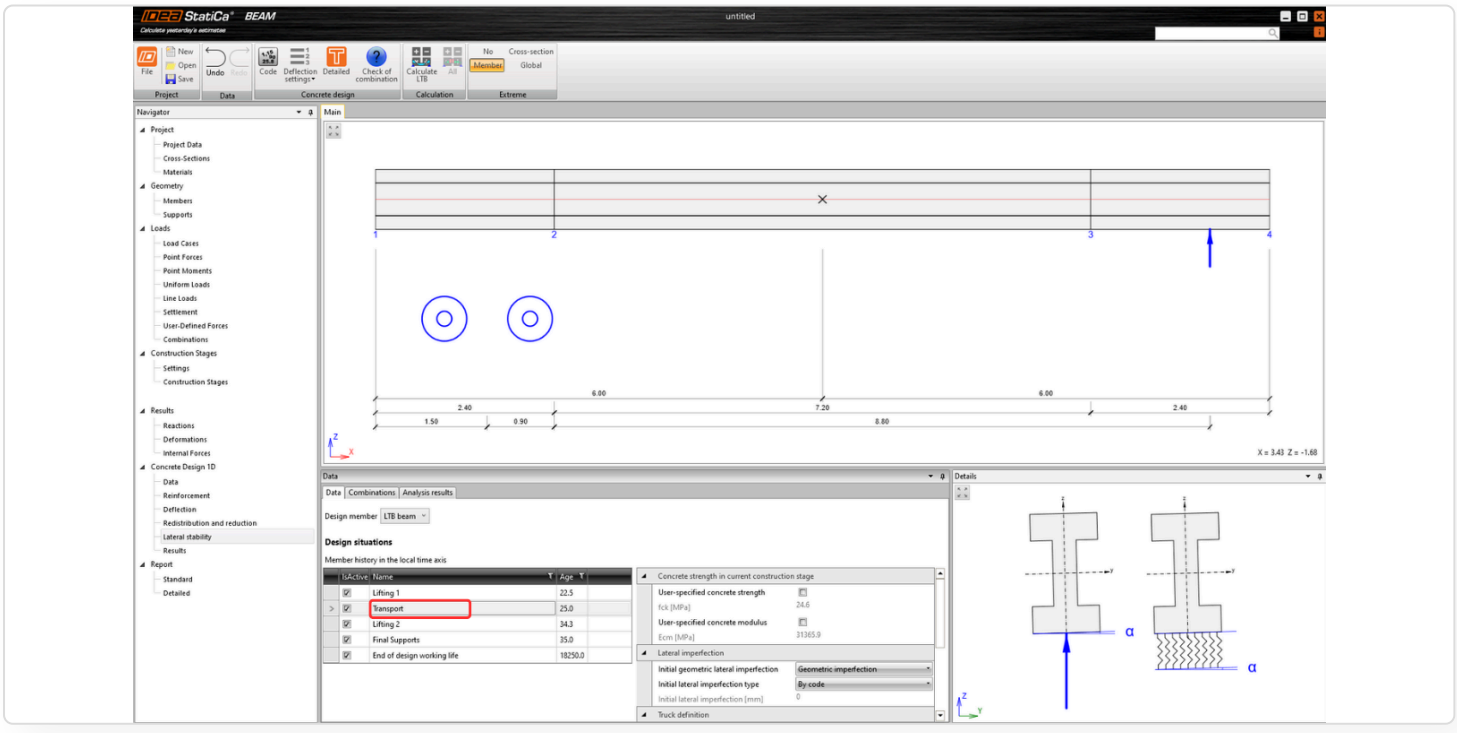
### Inputs for lifting

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.



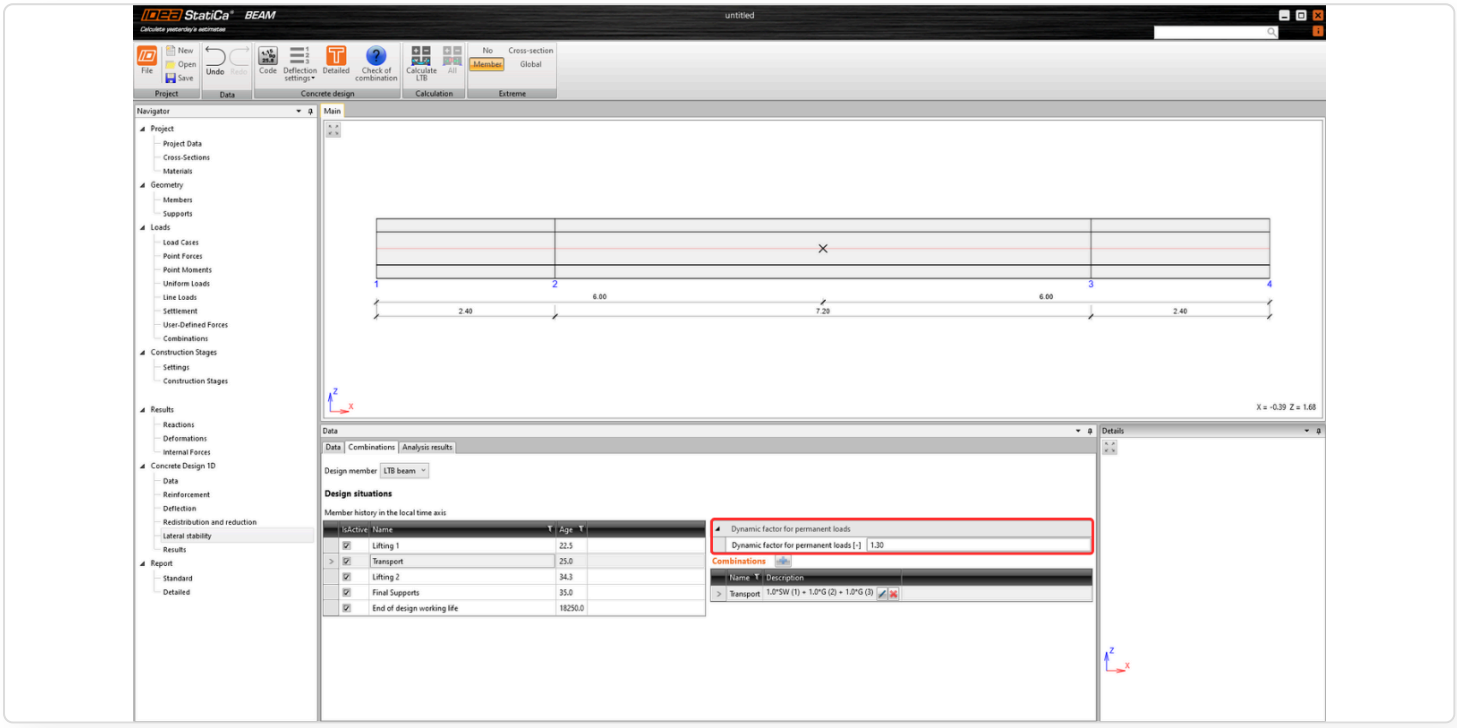

### Inputs for final supports and end-of-design working life

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.

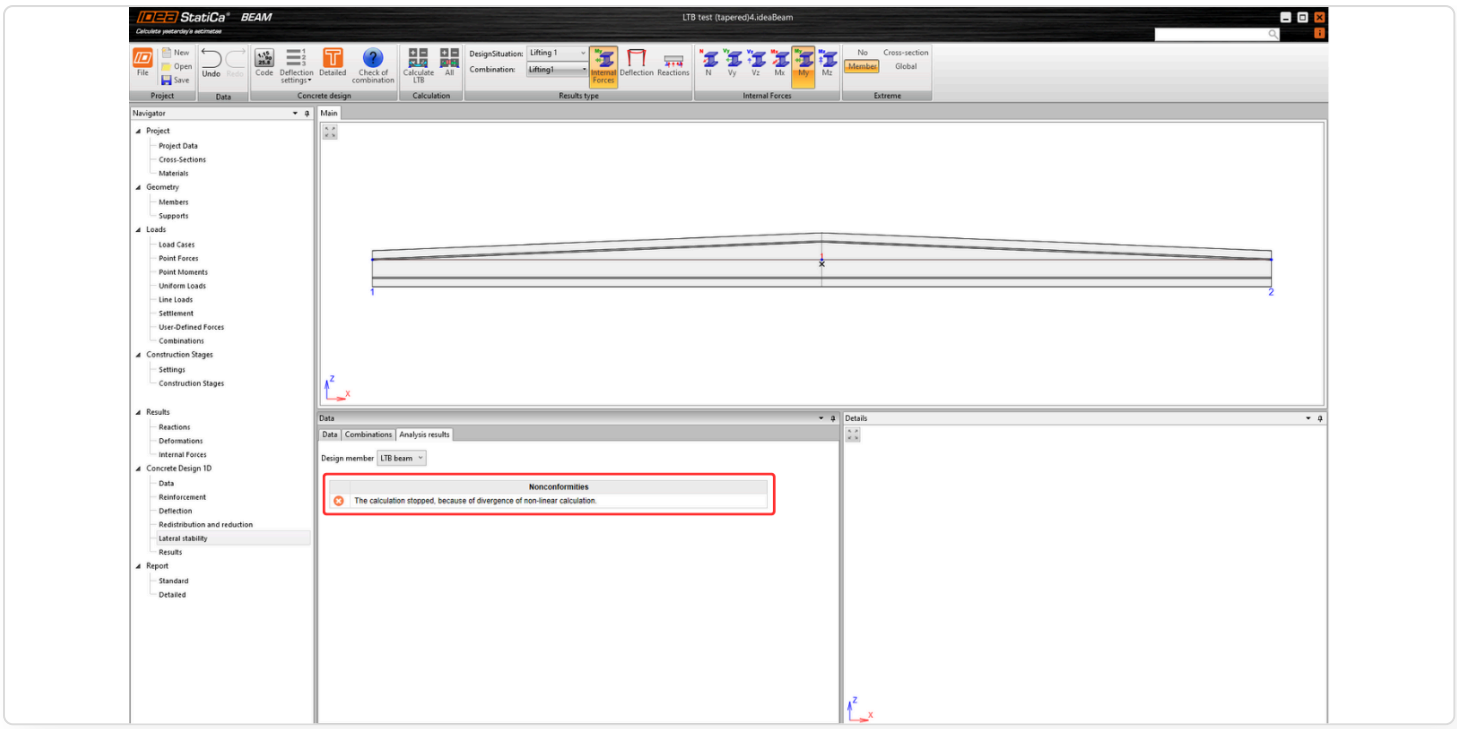
**Loads**

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.

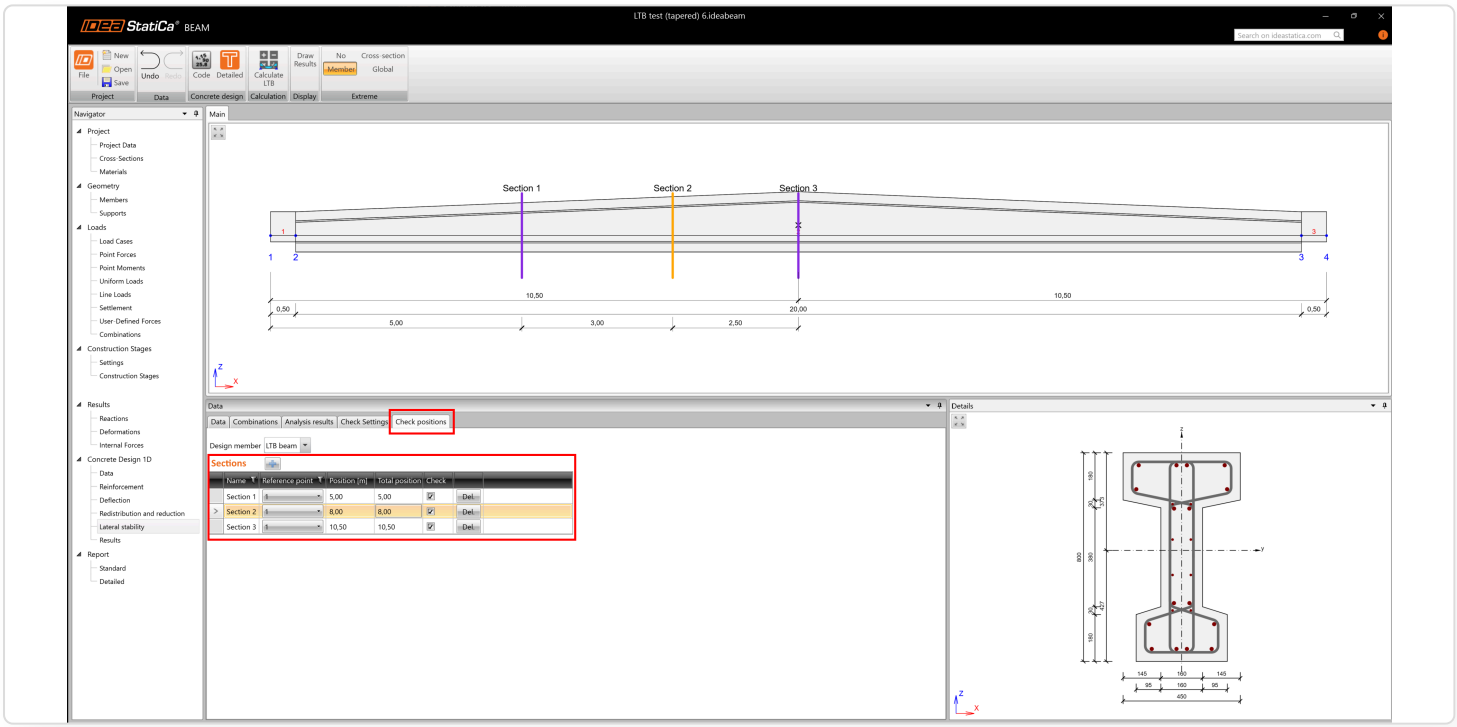



# Results

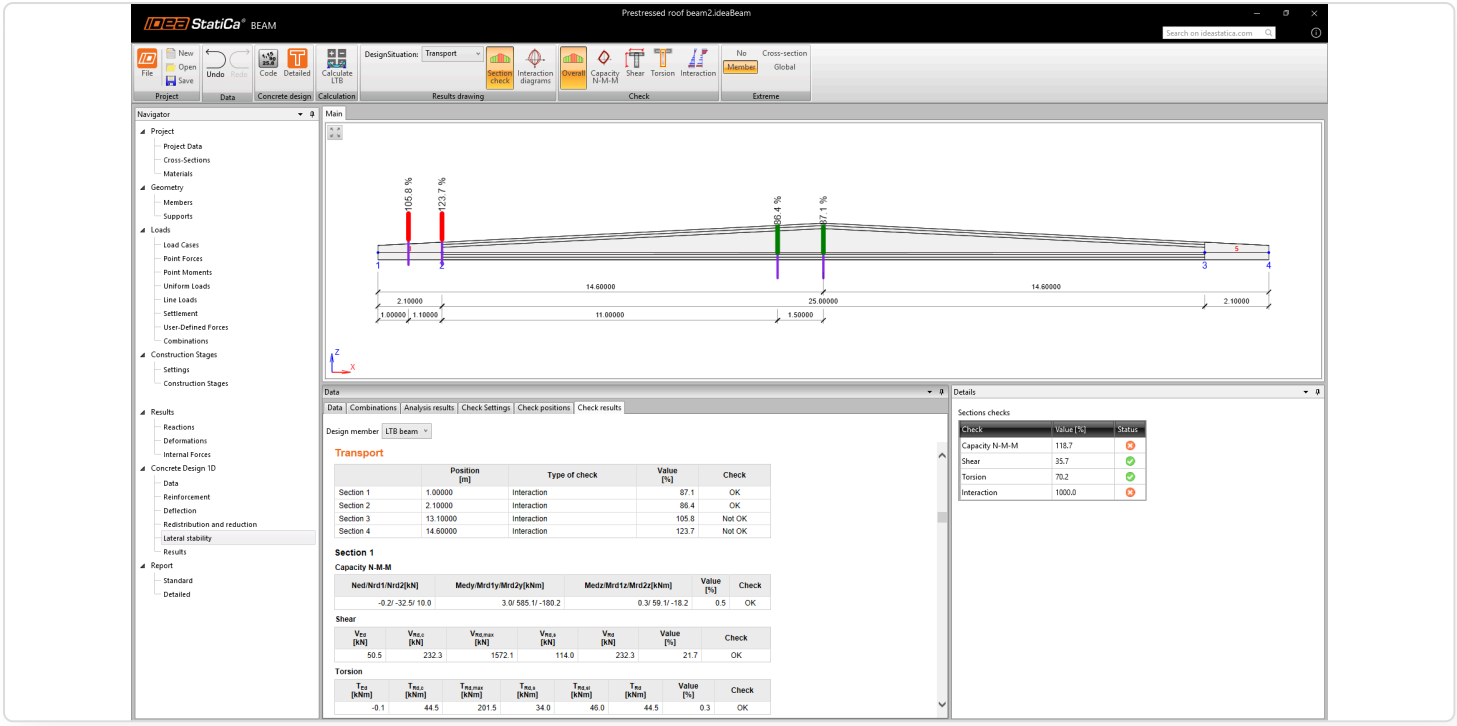
The advanced **Lateral torsional buckling (LTB**) analysis in IDEA StatiCa Beam provides (in addition to reactions, internal forces, and deformations) an evaluation of whether the beam is at risk of collapsing due to stability issues for each defined construction stage. In the event of a structural failure due to LTB, the calculation will not be complete, and the user will see an error message informing them of this situation.




The **Check Settings** tab allows you to define ULS code-check types for the selected member. In the **Check Positions** tab, you can specify multiple section locations where detailed code checks will be performed using RCS.




Detailed RCS checks for lateral stability are integrated directly in the Beam app (since v25.1).




### Note

Please be aware that advanced LTB analysis only works for models created in v24.1 and higher. The calculation will not proceed for older projects, and they must be remodeled.

*Released in IDEA StatiCa version 24.1, improved for detailed results and torsional stiffness definition in IDEA StatiCa version 25.1.*

# Steel Design

Enhancements in the anchoring design

# Advanced anchoring design and anchors in Connection

Anchoring functionality has been enhanced with process-based selection of anchors, cast-in plates, multi-surface anchoring, and a general anchor database for the integration of third-party anchor types, enabling precise modeling, code-compliant checks, and safe connection design.
Find out more about the features for anchor design in IDEA SttaiCa Connection:

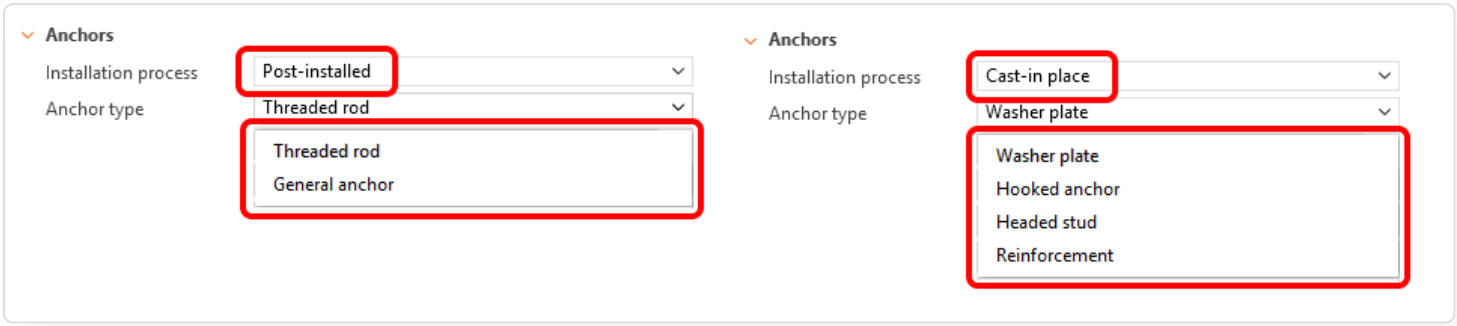
Anchor type selection for better understandability
Cast-in plates (Eurocode)
Anchoring on different planes
Two base-plates (operations) in one block
General anchor database

# Anchor type selection for better understandability

When defining the anchor configurations, there is now a **two-step anchor selection process** in IDEA StatiCa Connection, designed to improve clarity, consistency, and user control. The new workflow separates the **installation process** from the **anchor type**, making it easier for designers to select the correct anchoring method and ensuring a unified experience between the Connection and Detail applications.

Anchor definition now follows a two-step logic:

1. **Installation process selection** – first define whether the anchor is *Post-installed* or *Cast-in-place*
2. **Anchor type selection** – based on the installation process, relevant anchor types become available:
   - *Post-installed*: Threaded rod, General anchor
   - *Cast-in place*: Washer plate (Rectangular or Circular), Hooked anchor, Headed stud, Reinforcement



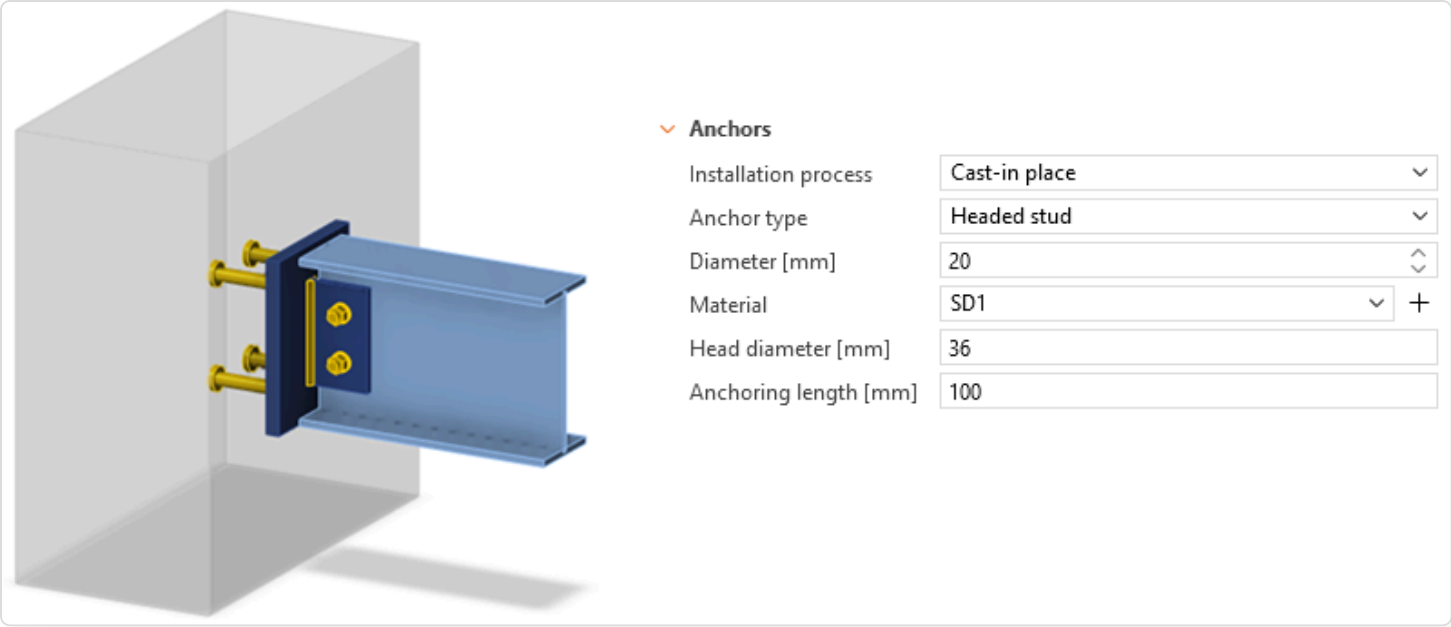

The feature is implemented in all operations supporting anchor definition, including **Base Plate** and **Fastener Grid or Contact** operations.

# Cast-in plates (Eurocode)

IDEA StatiCa Connection introduces comprehensive support for **cast-in plates anchored with headed studs and reinforcement** in IDEA StatiCa Connection. This enhancement allows designers to perform anchorage checks directly in the application without relying on external tools. The implementation follows the provisions of **EN 1992-4**, enabling verification of both tension and shear resistance for cast-in connections.

## Headed studs

A device type for **headed studs** can be found under the *Design* tab. The shear force transfer is limited to the *Anchors* option, with *Direct* stand-off only. Supported materials include **ISO 13918 grades SD1 and SD2**, ensuring compatibility with standard construction practice. Headed studs are defined by diameter, material, head diameter, and anchoring length, with visualization indicating the plate embedded in concrete.



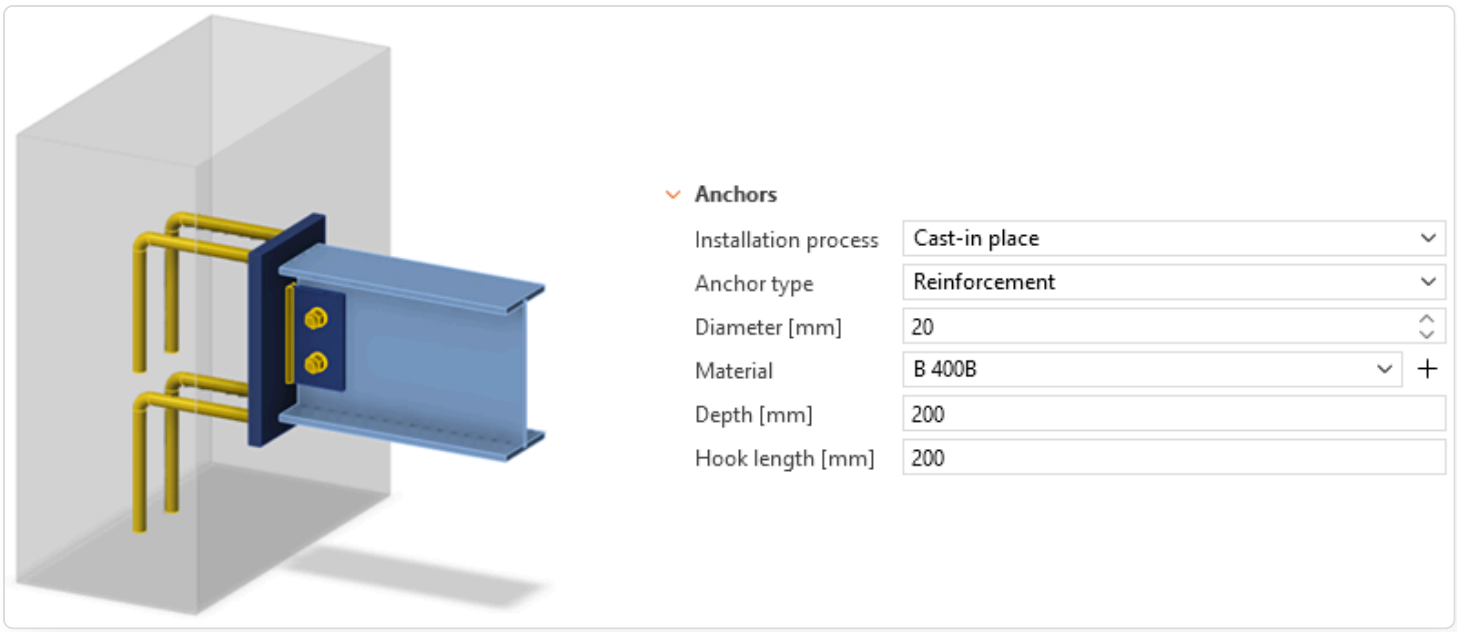

Implemented design checks follow **EN 1992-4** and cover the following:

- Tension
  - Anchor tensile resistance: EN1992-4: 7.2.1.3
  - Concrete breakout resistance of anchor in tension: EN 1992-4: 7.2.1.4
  - Concrete pullout resistance: EN 1992-4: 7.2.1.5
  - Concrete blowout resistance: EN 1992-4: 7.2.1.8
- Shear
  - Shear resistance
  - Concrete edge failure resistance: EN 1992-4: 7.2.2.5
  - Concrete pryout resistance: EN 1992-4: 7.2.2.4
  - Interaction of tensile and shear forces in concrete: EN 1992-4: Table 7.3

## Reinforcement

An option for **reinforcement anchorage** is also introduced in the *Design* tab. Again, the shear force transfer is limited to the *Anchors* option, with *Direct* stand-off only.

The only available shape is the L-bar, using materials already implemented in the concrete apps (EN 1992, ÖNORM B 1992-1-1). Reinforcement properties include diameter, material, hook length, and embedment depth.



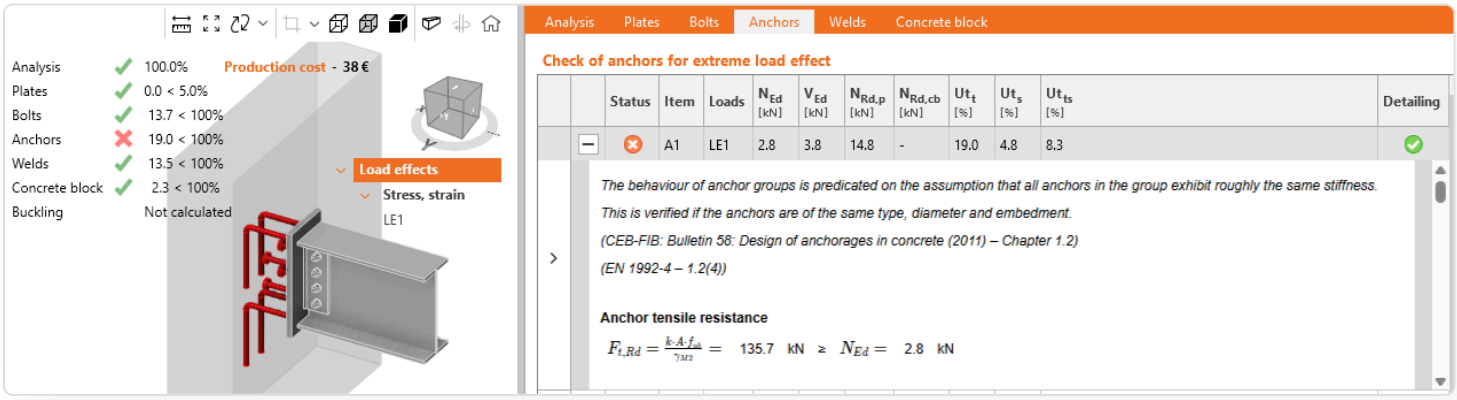

Design checks follow **EN 1992-4** and **EN 1992-1-1**, covering

- Tension
  - Anchor tensile resistance
  - Concrete breakout resistance of anchor in tension: EN 1992-4: 7.2.1.4
  - Concrete pullout resistance: EN 1992-1-1: 8.4.4
- Shear
  - Shear resistance
  - Concrete edge failure resistance: EN 1992-4: 7.2.2.5
  - Concrete pryout resistance: EN 1992-4: 7.2.2.4
  - Interaction of tensile and shear forces in concrete: EN 1992-4: Table 7.3

**Limitations**: It is not possible to define different directions of specific rebars. The group of rebars, together with the cast-in plate, can only be rotated as a whole using the rotation parameter of the base plate.

## Combining headed studs and reinforcement

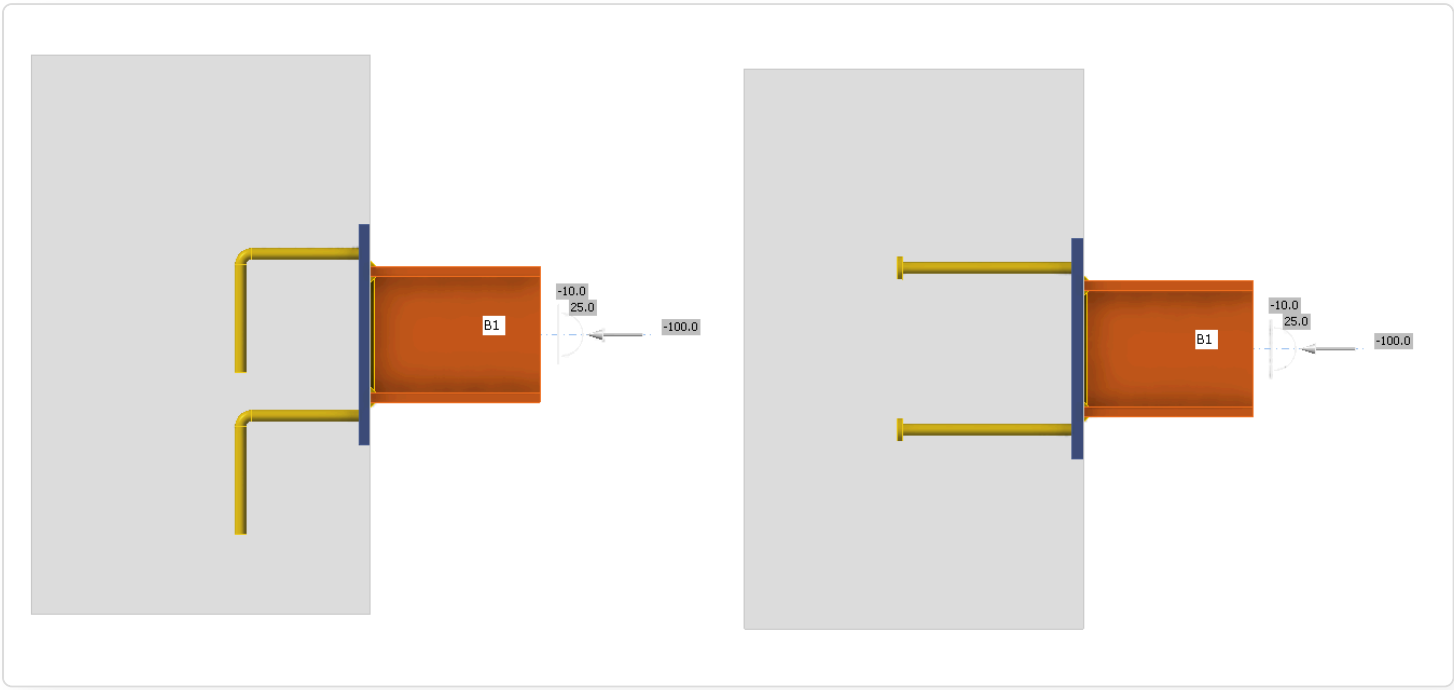
In case of higher load, the **headed studs and reinforcement can be combined**. This is practically modeled using the *Fastener Grid* or *Contact* operation.



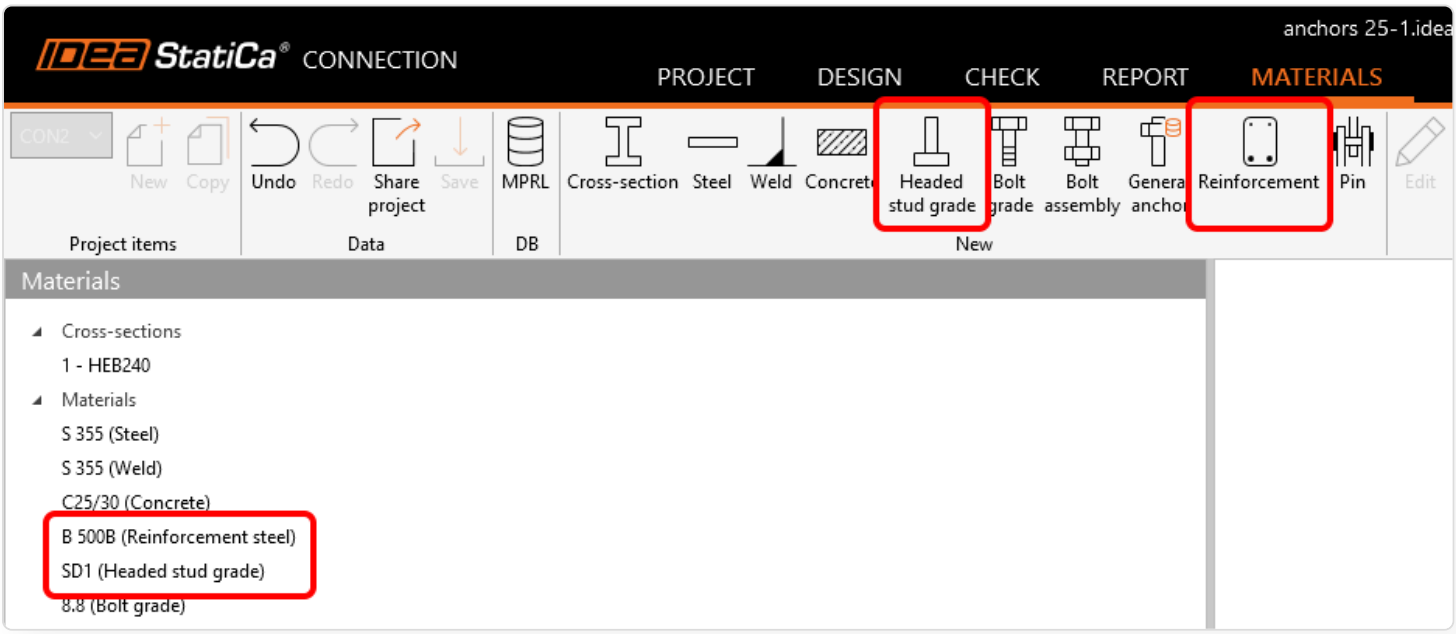

The redistribution of tension and shear is among all anchors based on their stiffness. The users can‘t change the force distribution (SCI 416).

The checks are evaluated in line with **CEB-FIP Bulletin 58**, marking anchors as failed, even below 100% utilization. The other steel component checks remain available (except anchoring). And for advanced analysis of the concrete block with reinforcement, the model can be exported to **IDEA StatiCa Detail**.

For cast-in plates with headed studs and/or reinforcement, the base plate is embedded in the concrete.

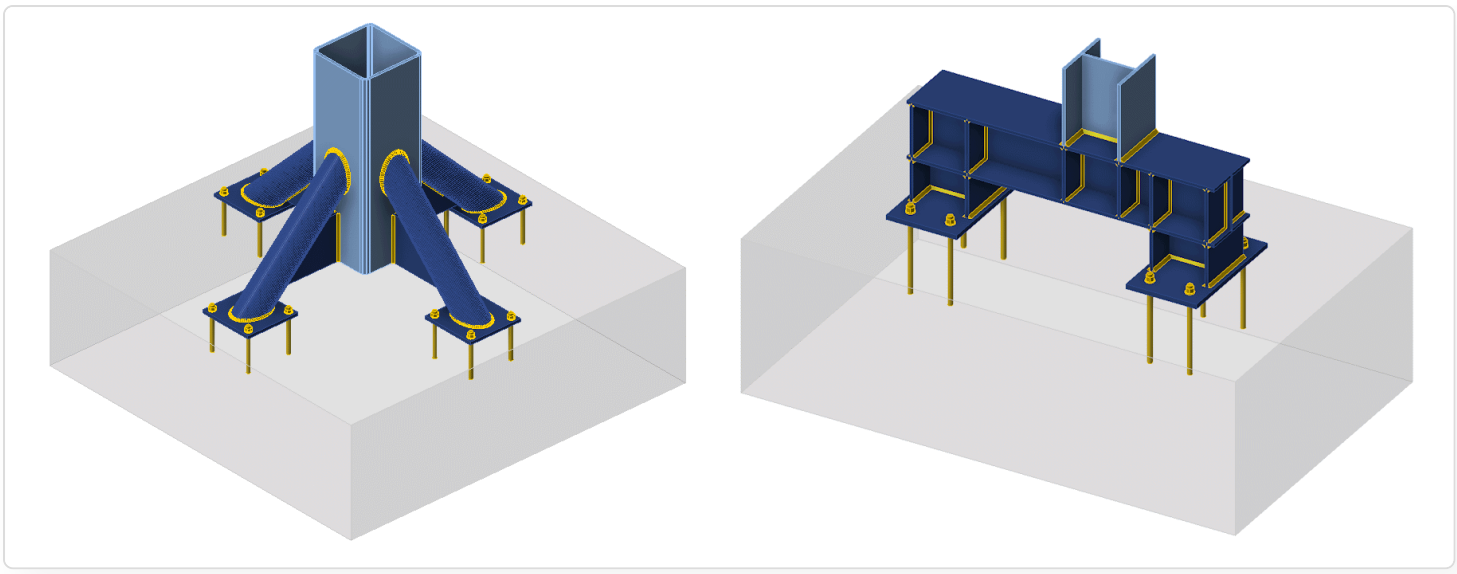



In the *Materials* tab, the Headed studs grade and Reinforcement material are placed in the top ribbon as well as in the left tree when used in the model.




# Multiple base plates on one block

To model such a geometry, the workaround with the stiffening plate is no longer necessary. You have the possibility to use the **Base plate** operation directly to add another set of anchors to an existing foundation block.

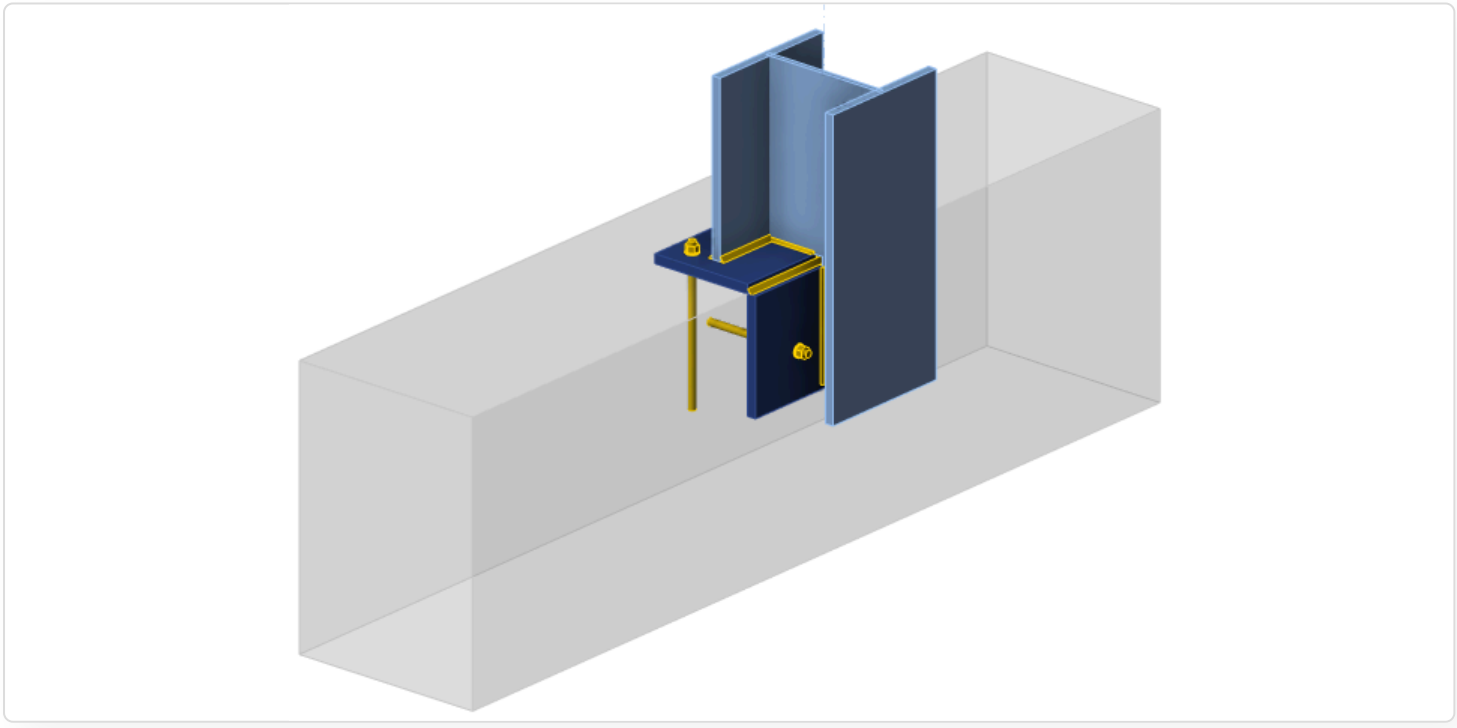


To add the baseplate operation onto a stiffening member, place the baseplate at the origin of the stiffening member. For inspiration, take a look at the sample project.

Please note that the "one node" principle still needs to be followed (i.e., when modeling, e.g., more columns anchored to a single foundation block, the internal forces may not be correct).

# Anchoring at the concrete block edge

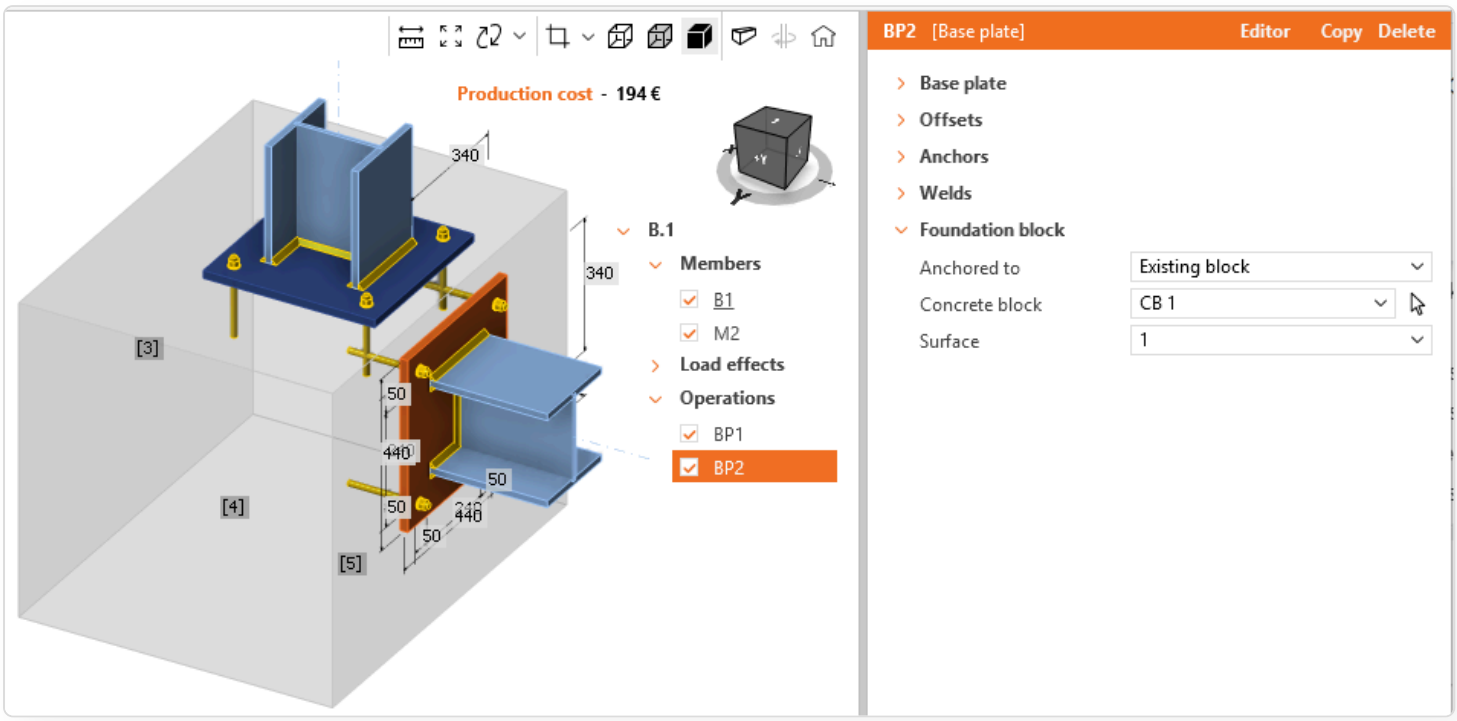
IDEA StatiCa Connection supports **anchoring to multiple concrete block surfaces**, significantly extending modeling capabilities for complex base plate configurations. Designers can now define anchors on two planes of a single concrete block, enabling accurate representation of real-world connection details such as members anchored at both horizontal and vertical faces.



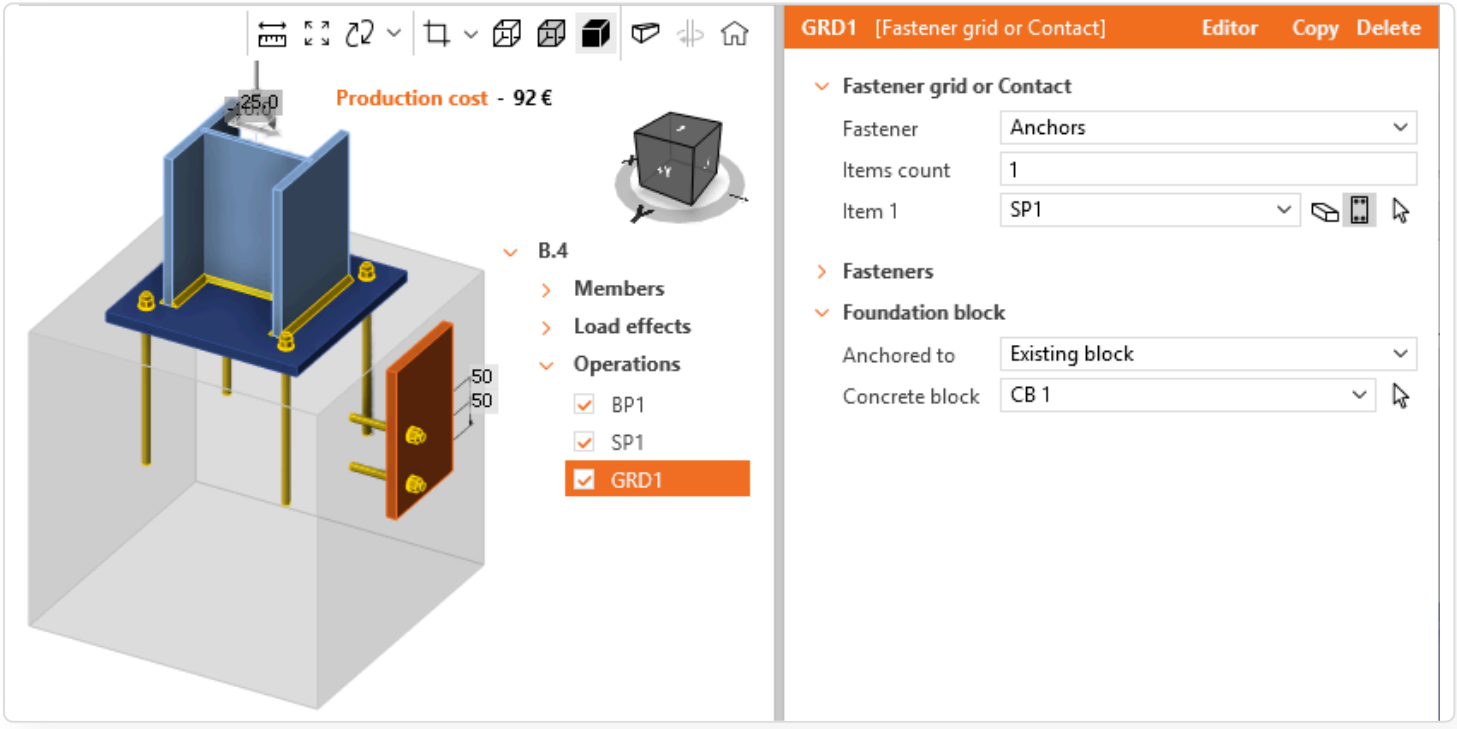
This removes the need for time-consuming workarounds involving stiffening plates, manual cuts, or multiple block simulations, and ensures consistent, traceable anchoring behavior across design codes. Integration with the **Detail app** ensures proper concrete checks (EN, ACI, AS).

## Define the base plate on a given surface

In the *Base Plate* operation, you can **choose between creating a new block or selecting an existing one**. In case of an existing block, there is a new option for specifying the surface. Surfaces are numbered in the same way as in the Detail app.



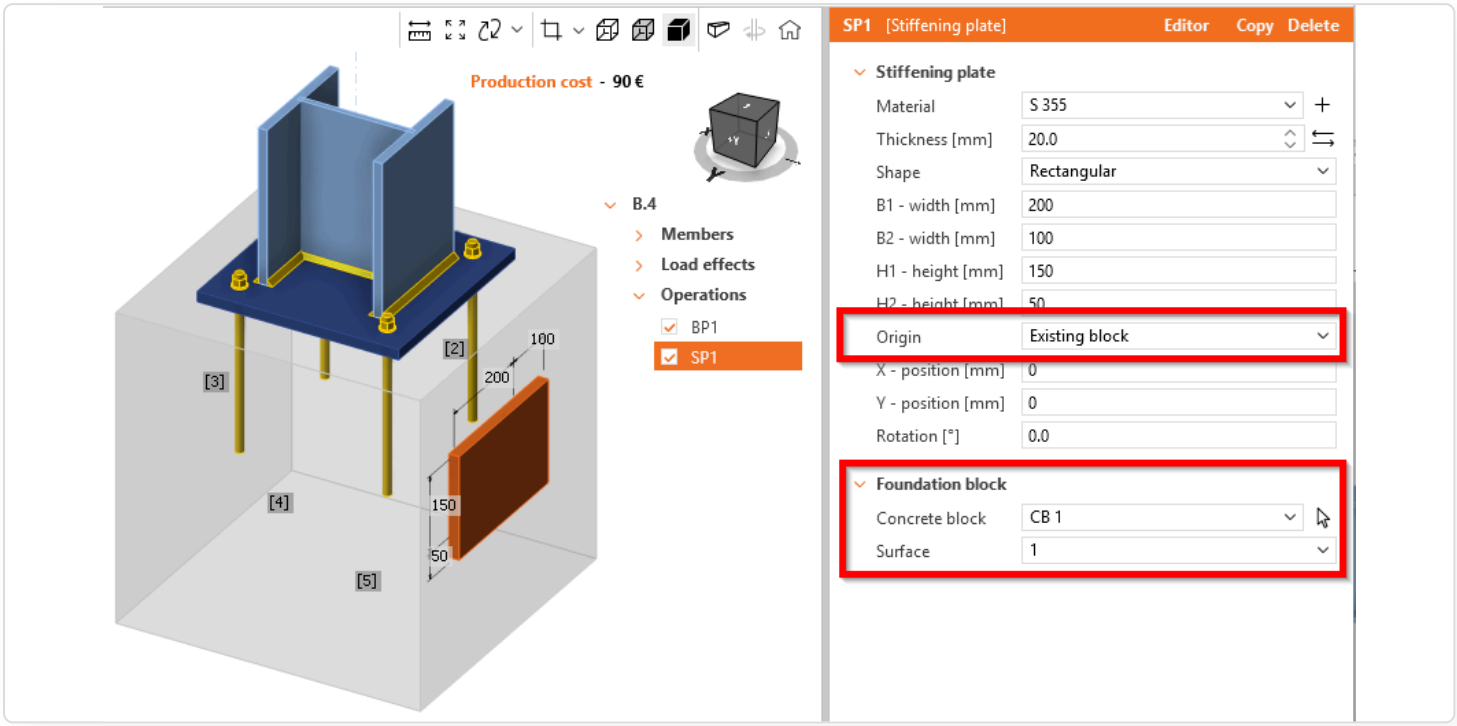

For the *Fastener grid or Contact* operation, a new logic has been implemented. When the referenced plate is located on the face of the concrete block, and the operation is set to *Anchors*, the surface is automatically recognized and used for the creation of the subsoil model.




When an existing block is selected, the anchor properties (Offset, Depth, Shear force transfer, Stand-off) are automatically reused.

## Locate the stiffening plate on the surface

There is also a new option for the Stiffening plate – its *Origin* can now be defined on the *Existing block* of concrete. When selected, the stiffening plate is automatically placed on the selected concrete block and its surface. Location is in the middle of the surface.




## Check of anchors

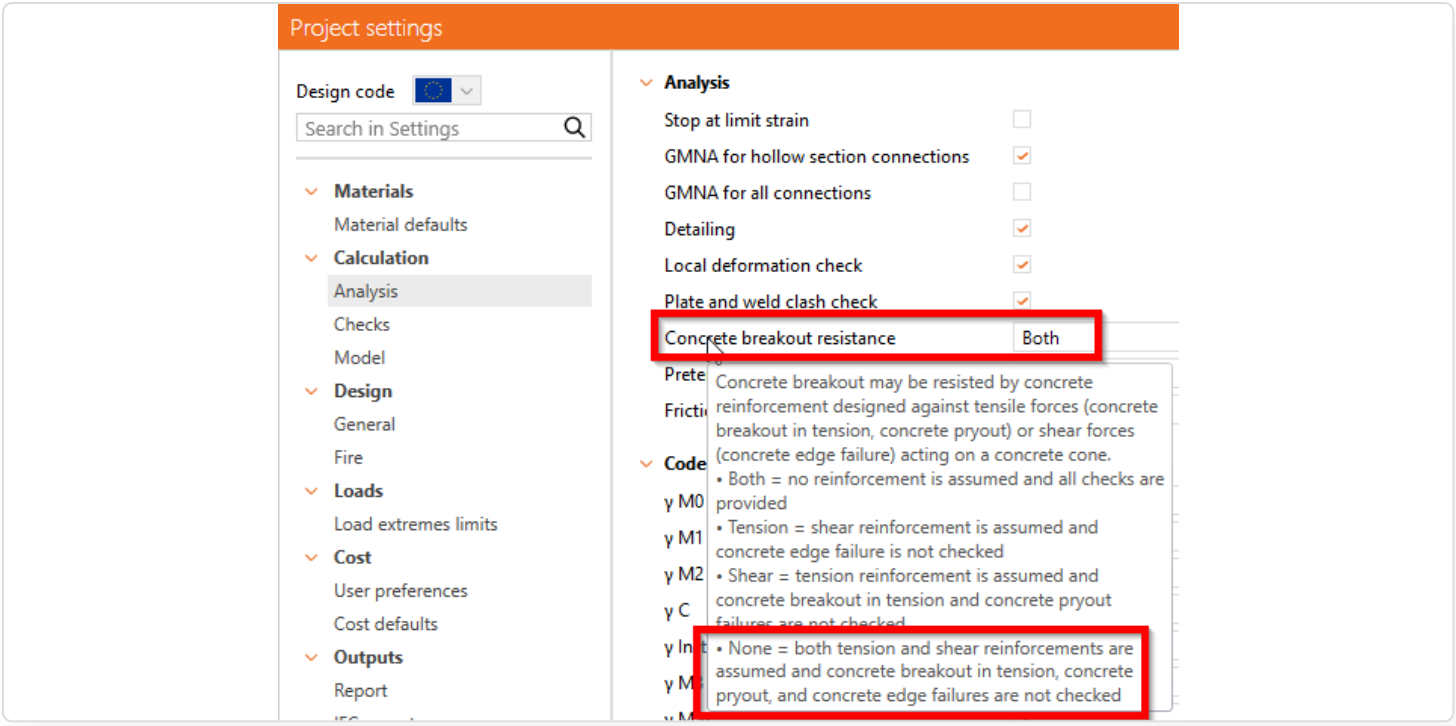
By default, the anchor checks are marked as failed due to the interaction of the base plates, which is not checked in the Connection app.

**Check of anchors for extreme load effect**

| | Status | Item | Loads | $N_{Ed}$ [kN] | $V_{Ed}$ [kN] | $Ut_t$ [%] | $Ut_s$ [%] | $Ut_{ts}$ [%] |
|---|---|---|---|---|---|---|---|---|
| − | ✖ | A1 | LE1 | 42.0 | 32.6 | 59.0 | 64.9 | 76.9 |

*Anchor groups at separate base plates interact with each other in one concrete block. This is out of the scope of standards for anchorage design. Concrete breakout in tension and concrete pryout are not checked. Concrete edge failure is not checked. (CEB-FIB: Bulletin 58 - Design of anchorages in concrete (2011) – Chapter 1.2: Figure 1.2-8 and Figure 1.2-9)*

You can change this status in the Project settings and modify the *Concrete breakout resistance* to **None**. In this case, both tension and shear reinforcement in the concrete block are assumed, and the concrete checks are not performed.



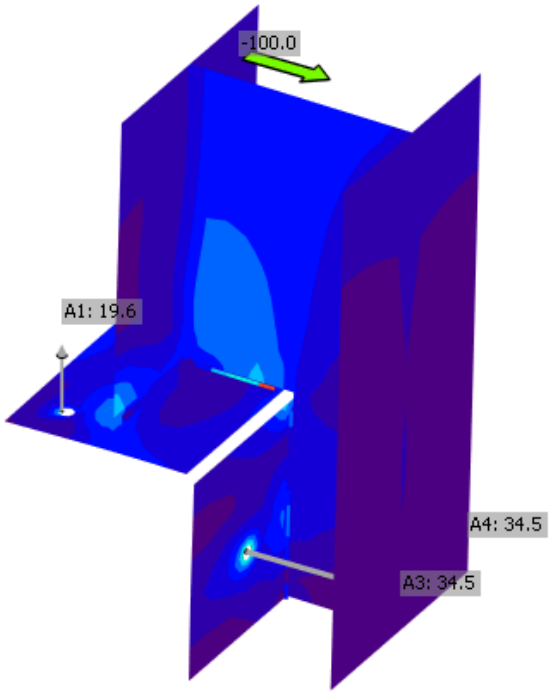

The steel code checks remain valid, and users can **export the model to Detail** for advanced concrete verification. The software automatically redistributes forces according to anchor stiffness, compression subsoil stiffness, and load paths, allowing engineers to investigate and validate critical load transfer conditions manually.

For the overview of the anchor types in IDEA StatiCa Detail, see the Single anchor definition article.

## Safe behavior of the model in Connection

The force distribution and, therefore, the safety of the results depend heavily on engineering input. By default, all forces are redistributed into all devices in the model, tension/shear stiffness of the anchors, compression stiffness of the Winkler subsoil, according to their stiffnesses.

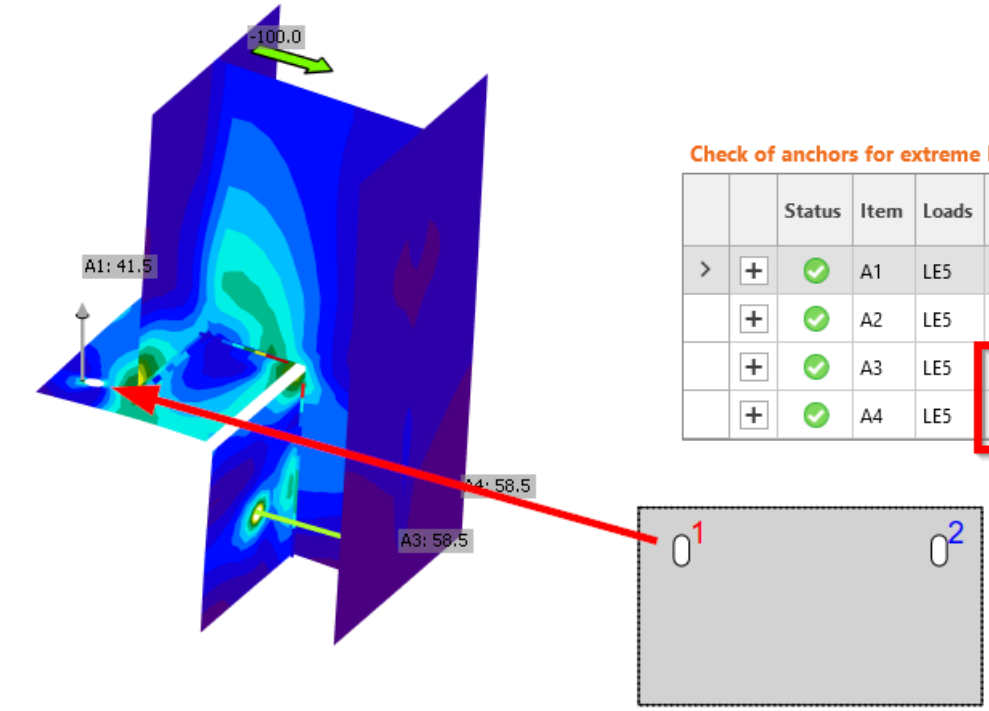
See the example, where the anchoring is modeled at two surfaces. By default, all anchors transmit tension and shear. That assumes precise holes or fit-in holes. In a two-plane anchoring case, a horizontal force of 100 kN is transferred as tension through anchors on the vertical face (A3, A4) and as shear through anchors on the horizontal face (A1, A2).




Check of anchors for extreme load effect

| | | Status | Item | Loads | $N_{Ed}$ [kN] | $V_{Ed}$ [kN] | $Ut_t$ [%] | $Ut_s$ [%] | $Ut_{ts}$ [%] | Detailing |
|---|---|---|---|---|---|---|---|---|---|---|
| > | + | ✓ | A1 | LE5 | 19.6 | 24.0 | 27.5 | 47.7 | 30.3 | ✓ |
| | + | ✓ | A2 | LE5 | 19.6 | 24.0 | 27.5 | 47.7 | 30.3 | ✓ |
| | + | ✓ | A3 | LE5 | 34.5 | 3.5 | 48.5 | 6.9 | 24.0 | ✓ |
| | + | ✓ | A4 | LE5 | 34.5 | 3.5 | 48.5 | 6.9 | 24.0 | ✓ |

If slotted holes are applied to the horizontal anchors, shear transfer is released, doubling tension in vertical anchors.




Check of anchors for extreme load effect

| | | Status | Item | Loads | $N_{Ed}$ [kN] | $V_{Ed}$ [kN] | $Ut_t$ [%] | $Ut_s$ [%] | $Ut_{ts}$ [%] | Detailing |
|---|---|---|---|---|---|---|---|---|---|---|
| > | + | ✓ | A1 | LE5 | 41.5 | 0.2 | 58.4 | 0.4 | 34.1 | ✓ |
| | + | ✓ | A2 | LE5 | 41.5 | 0.2 | 58.4 | 0.4 | 34.1 | ✓ |
| | + | ✓ | A3 | LE5 | 58.5 | 7.7 | 82.2 | 15.4 | 70.0 | ✓ |
| | + | ✓ | A4 | LE5 | 58.5 | 7.7 | 82.2 | 15.4 | 70.0 | ✓ |

Such examples highlight the importance of the user's engineering judgement. The ideas about correct behavior should always be critically evaluated in the most critical state. By switching the anchor groups on and off, you are able to find out the most critical state not only for the anchor check but also for other components, obtaining correct structural behavior.

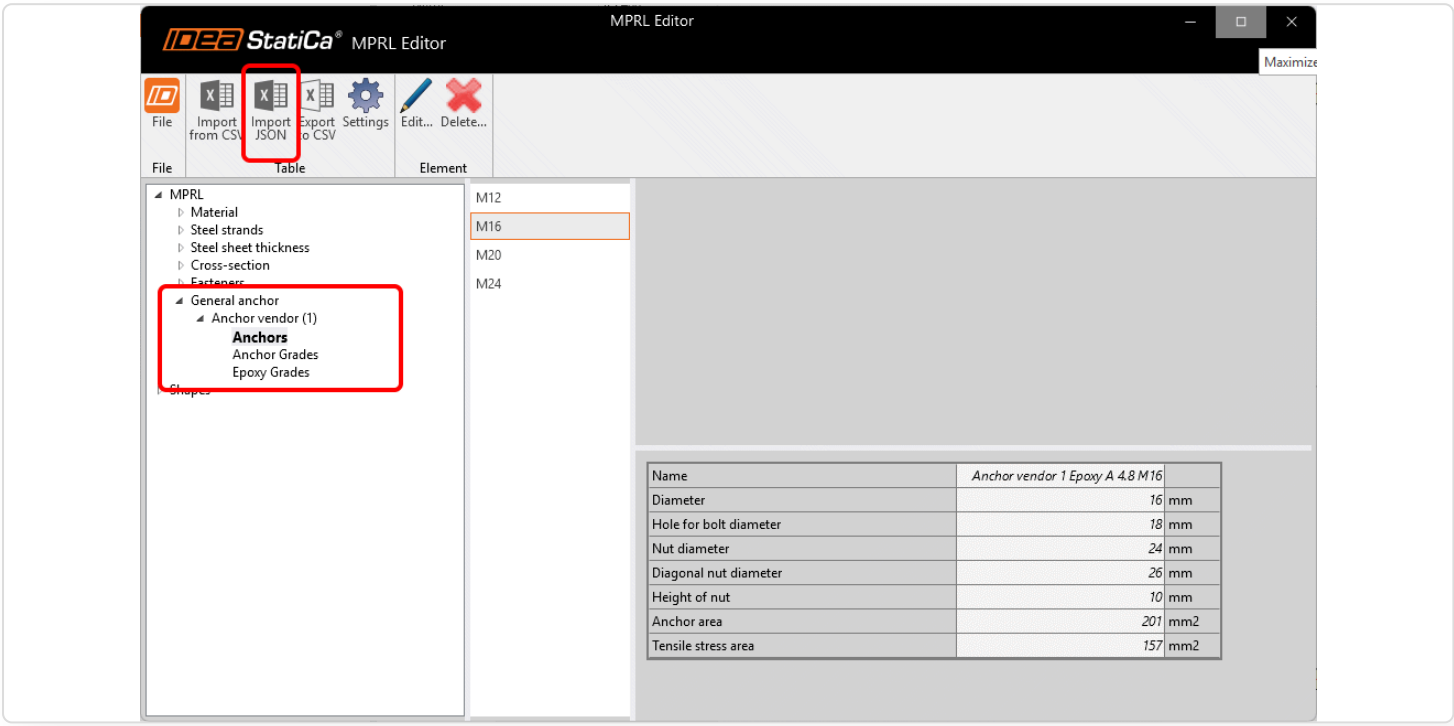
# General anchor database

The **General Anchor Database** introduces a new framework for managing and integrating third-party anchor data within IDEA StatiCa Connection. This functionality allows expanding the range of available anchors beyond the predefined library.

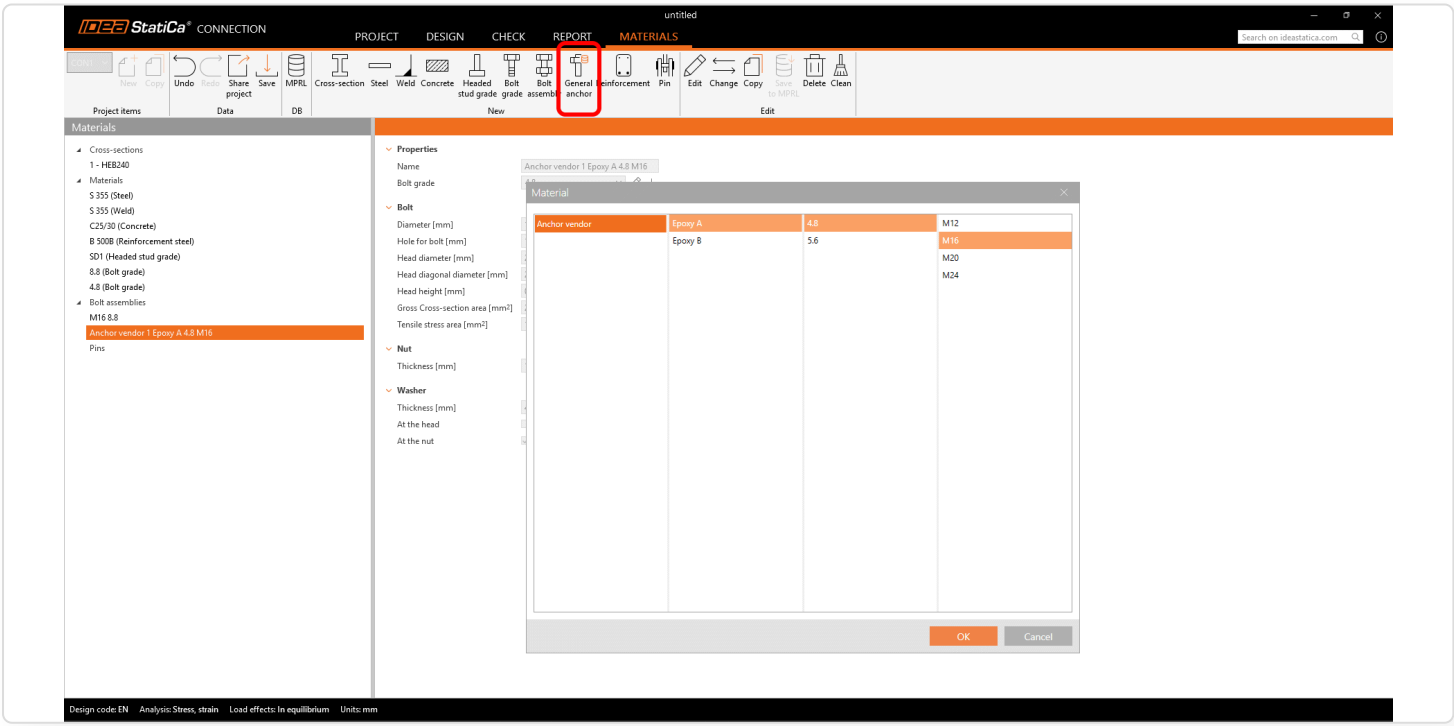
The new system allows **anchor suppliers** to prepare and maintain their own anchor databases in a structured format (.json) that can be imported into the MPRL database of IDEA StatiCa by the end-user. Each dataset retains ownership on the supplier side, ensuring data accuracy and regular updates.

***Remark:*** *Currently, the .json files are not available yet. Publicly shared databases will be accessible to all users once they have been approved and validated, supporting collaboration with leading anchor manufacturers (the list will be published upon approval).*

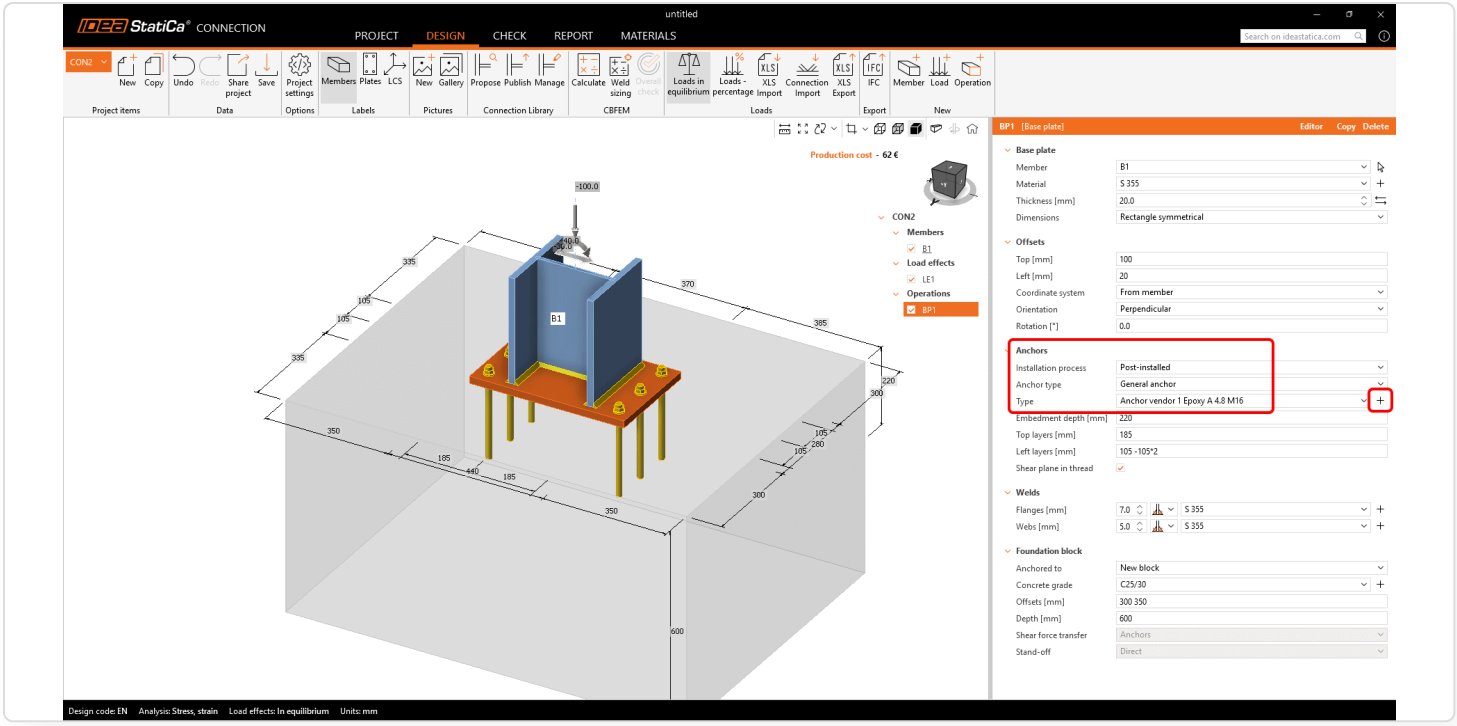
Import the .json file provided by the vendor in the MPRL database (under the *Materials* tab - *MPRL*).




Once imported, the material of the *General anchors* may be reviewed using the button in the top ribbon.




The uploaded anchor types will then be available in the drop-down menu under the *Post-installed* anchor type. The + button opens the library of available general anchors.




Improvements in the design templates

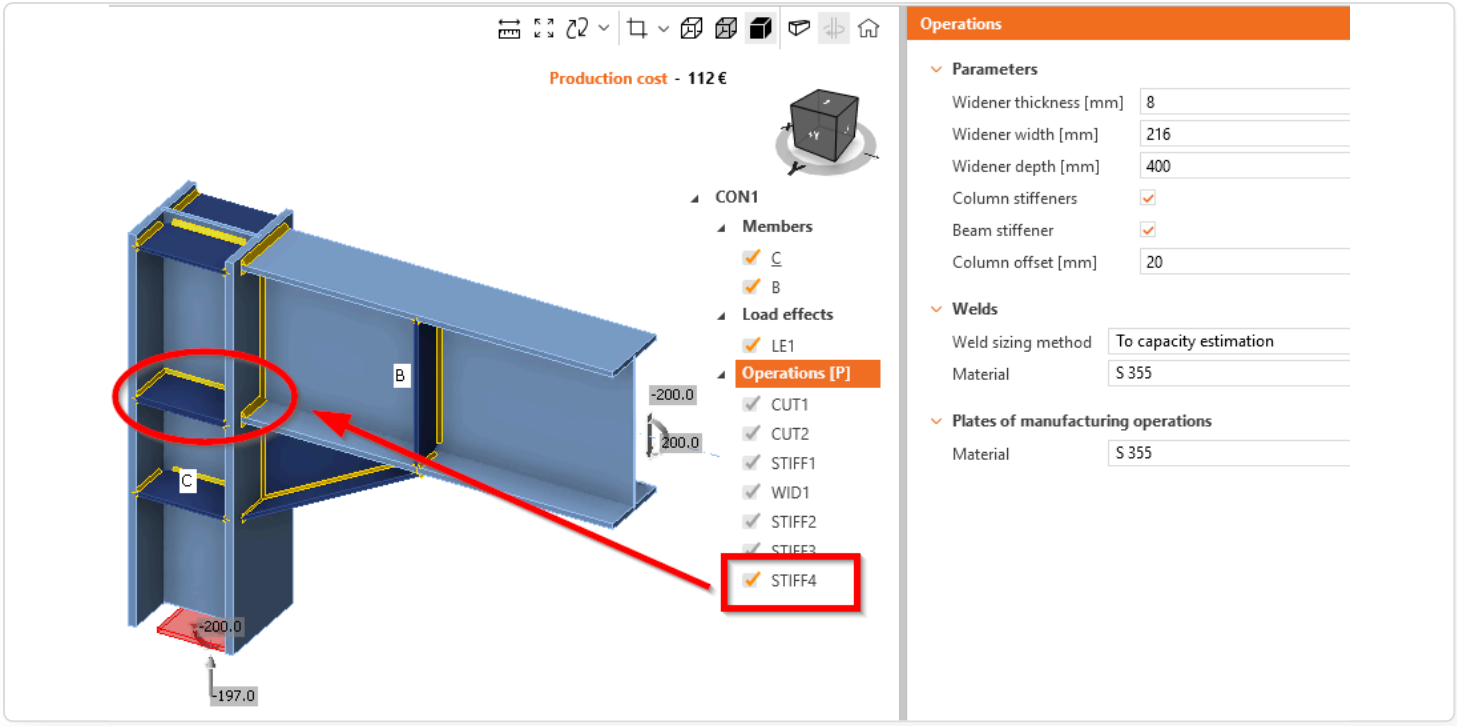
# Combining parametric templates and operations

By combining more parametric templates with a standard operation, the design stays still simple thanks to parameters used in a standard connection, but can cover any geometry with an additional operation.
Many nodes in a real structure can't be covered one-to-one by a standard connection type. Designers divide these nodes into separate connections and check them in simple tools or in Excel spreadsheets without considering their interaction, which can cause mistakes in design.

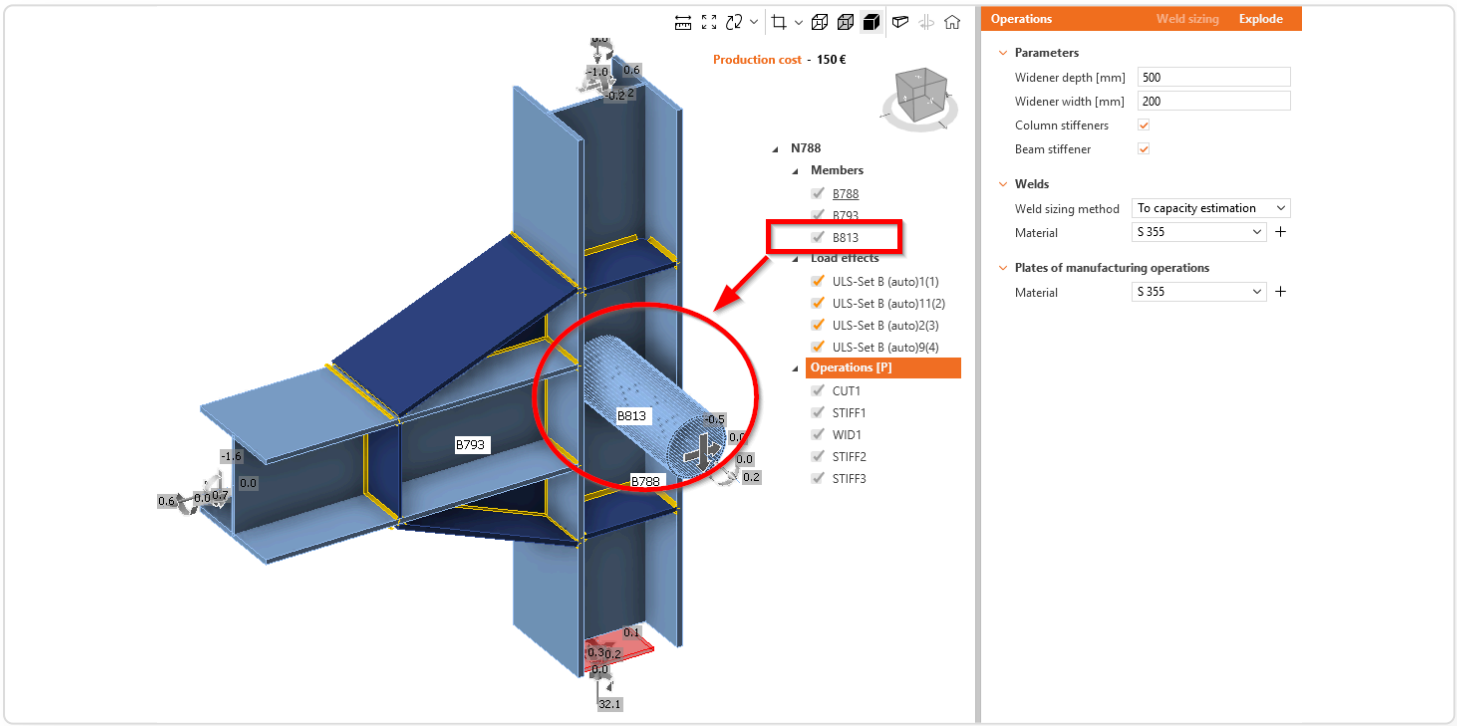
## Parametric templates can be combined with other operations

When using a parametric template, there is still an option for adding new operations, so it is possible to add a single operation to an existing parametric template.

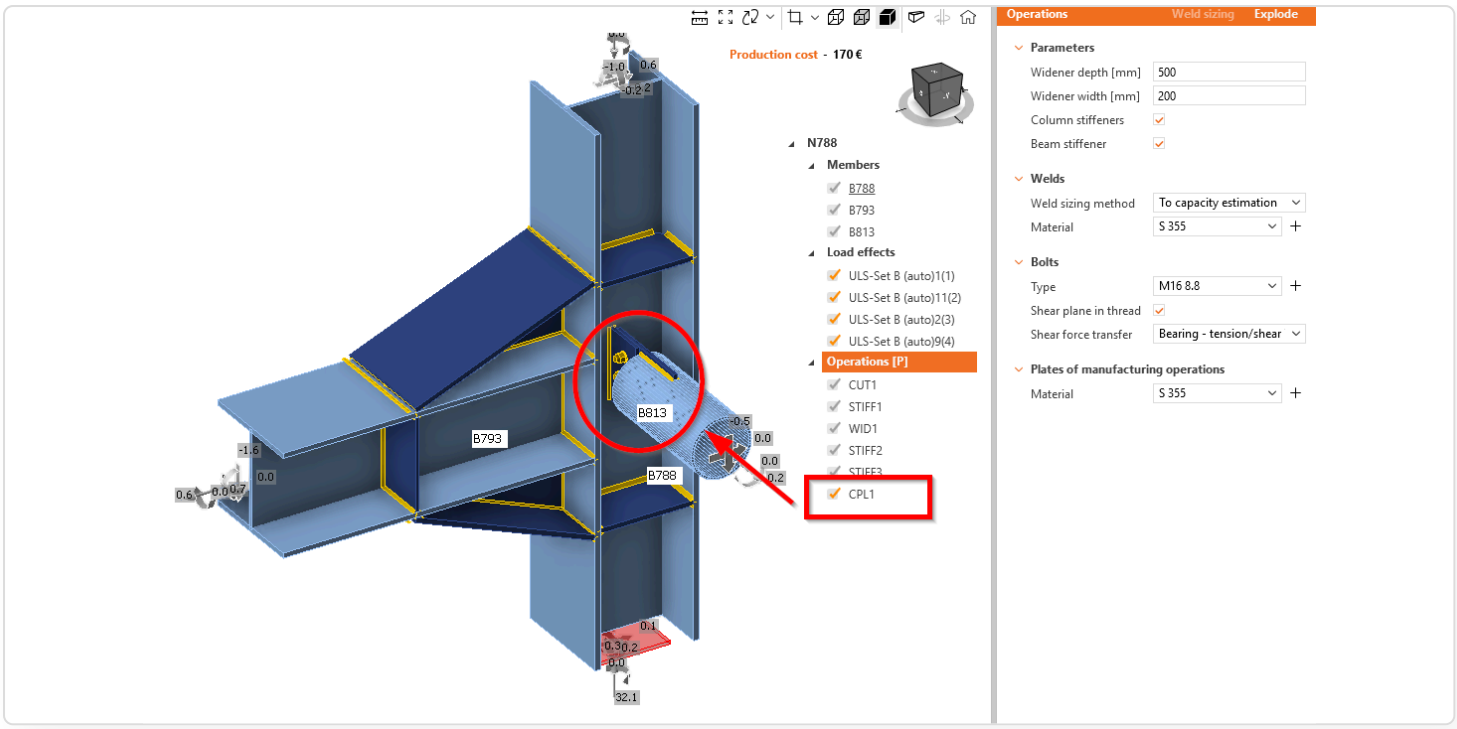
In this way, adding, e.g., more stiffeners to the chosen template can be easily reached.




Another example can be a joint that is composed of different connections – some members are not considered in the chosen template.



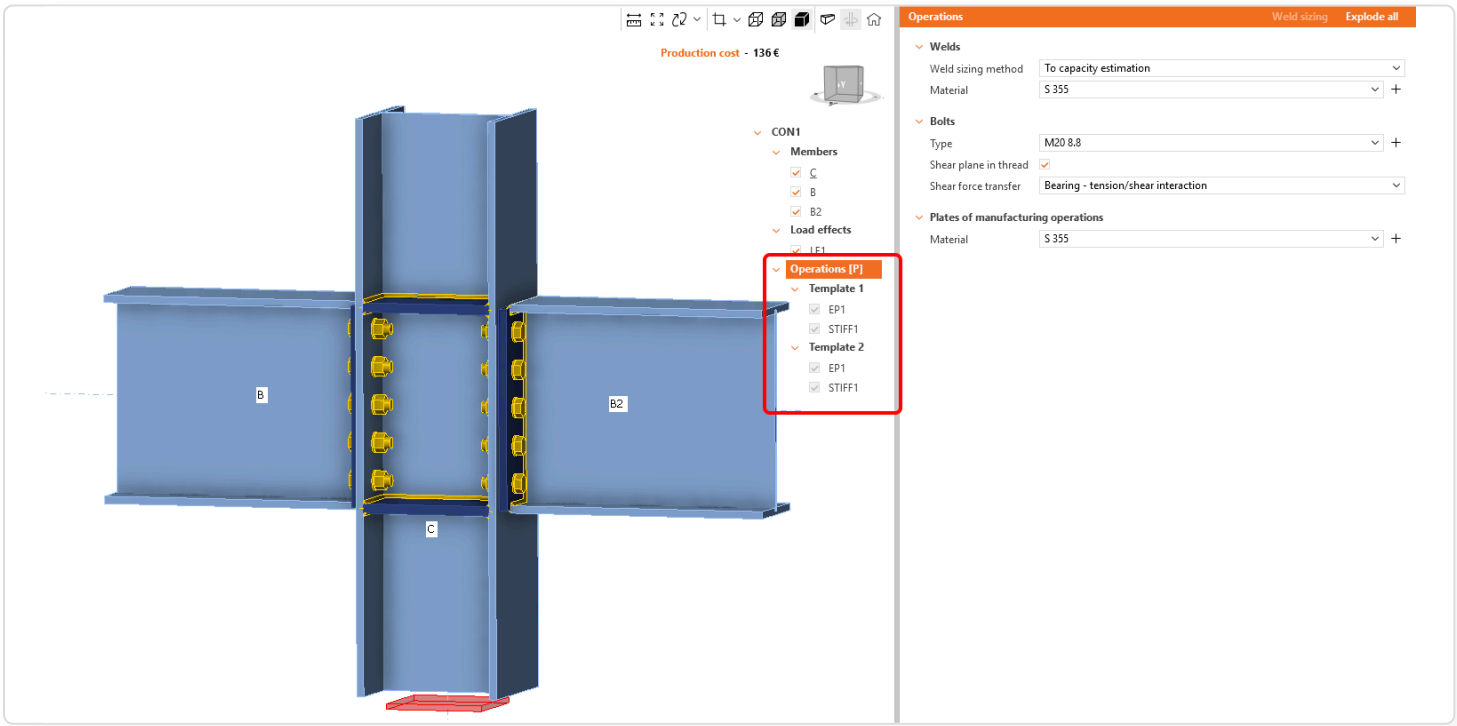

Connecting this additional member with a standard operation is easy to do.



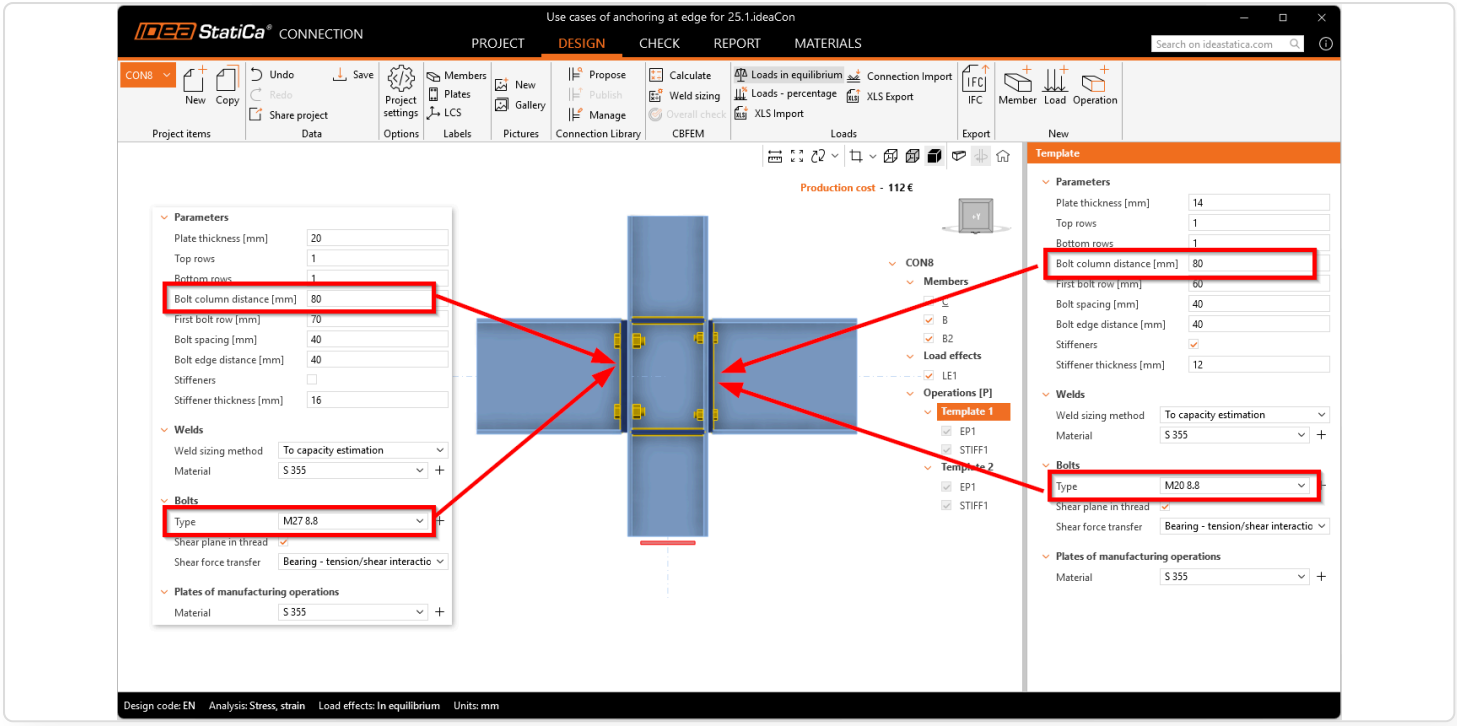

*Released in IDEA StatiCa version 25.0.*

# Combine more parametric templates in one model

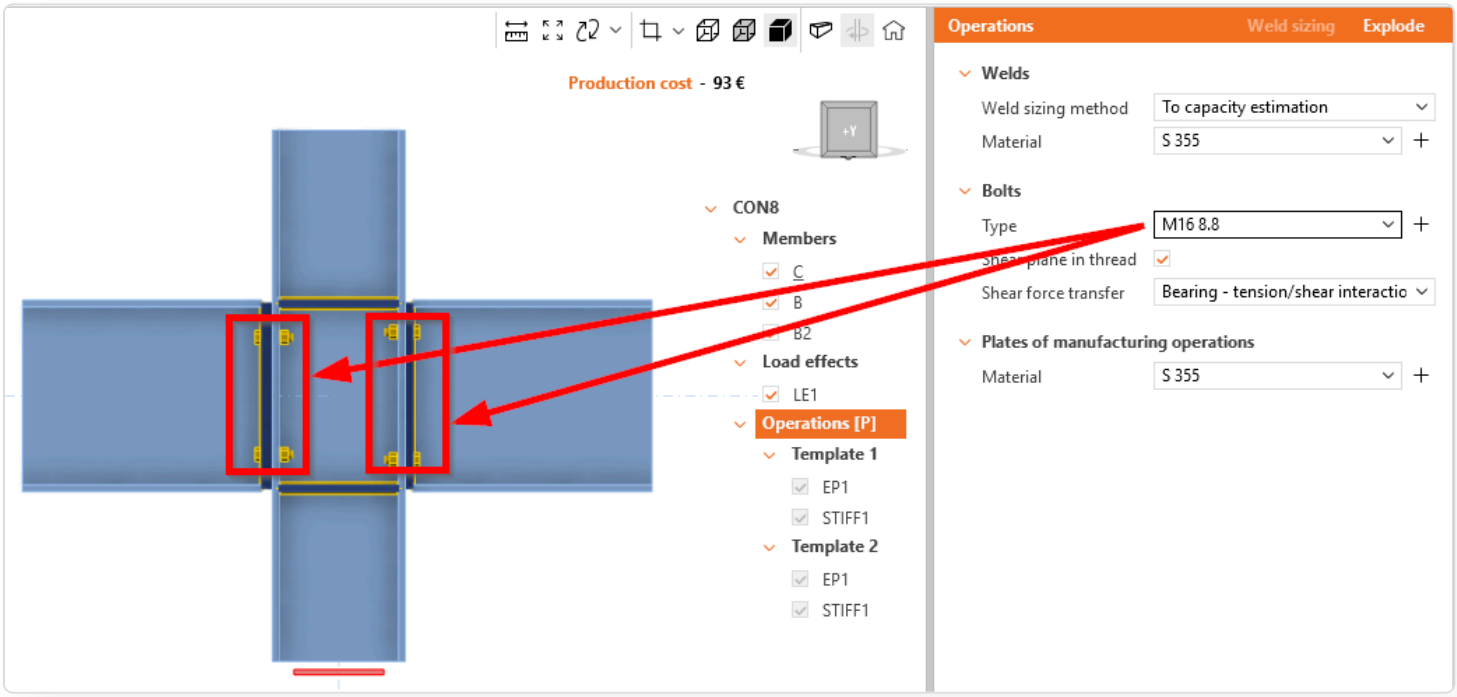
To connect multiple members in one node, you can apply several parametric templates. Create your standard connections as a parametric template, and then combine them effectively into the desired connection design of the whole structure.




Each applied template is now represented as a separate **template header** (Template 1, Template 2, etc.) in the operations tree. Selecting a template displays its specific parameters and controls, enabling users to modify local properties, such as bolt layouts, materials, or plate thicknesses, independently for each template.




Global properties remain accessible under the **Operations root**, allowing the user to apply common material or code parameters to the entire connection. The new structure supports mixed-template designs where individual templates retain their parametric flexibility.




*Updated in IDEA StatiCa version 25.1.*

# Parametric templates in Connection Library

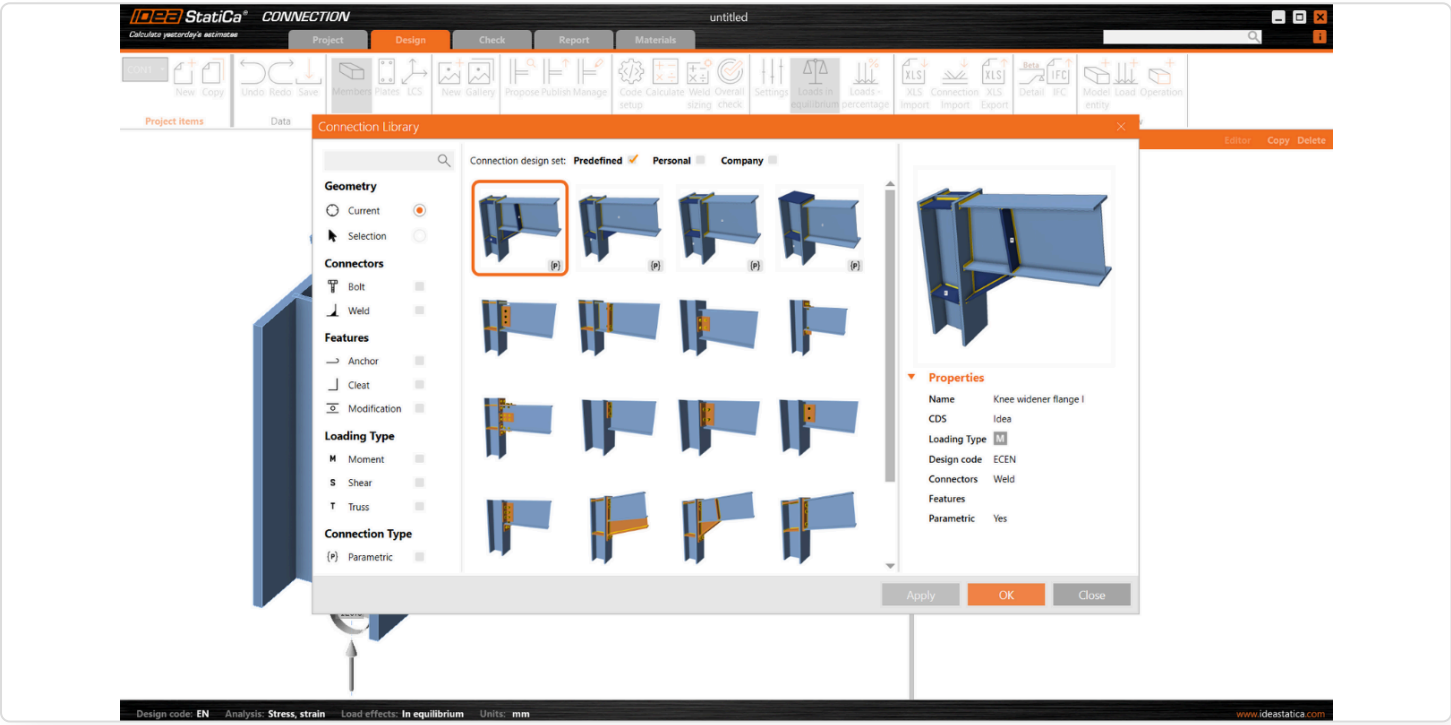
The Connection Library supports the use of parametric templates, significantly simplifying the workflow for connection designers. This allows for the creation and utilization of a universal set of templates that can be easily customized and applied to various design scenarios.

In IDEA StatiCa Connection, users have the possibility to create connection models using parameters (relations defined between individual entities). The parametric design allows users to design standardized connections efficiently – **read about how to work with parameters** in this article.

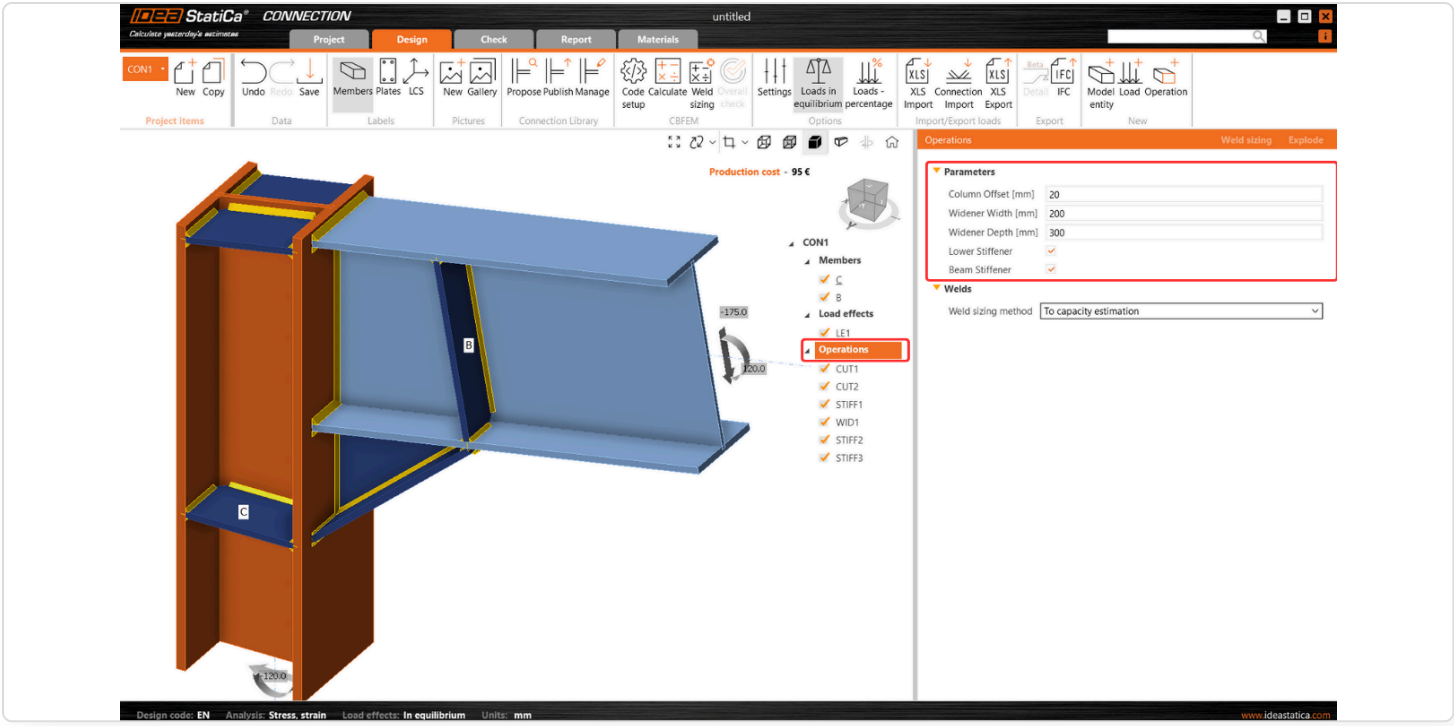
The integration of **parametric templates** into the Connection Library offers a transformational approach to connection designer workflows. **It allows users to create and use a universal collection of templates that can be effortlessly customized and deployed in different design contexts.**

# How does it work?

The user can upload the created connection to their company or personal set as with any 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.

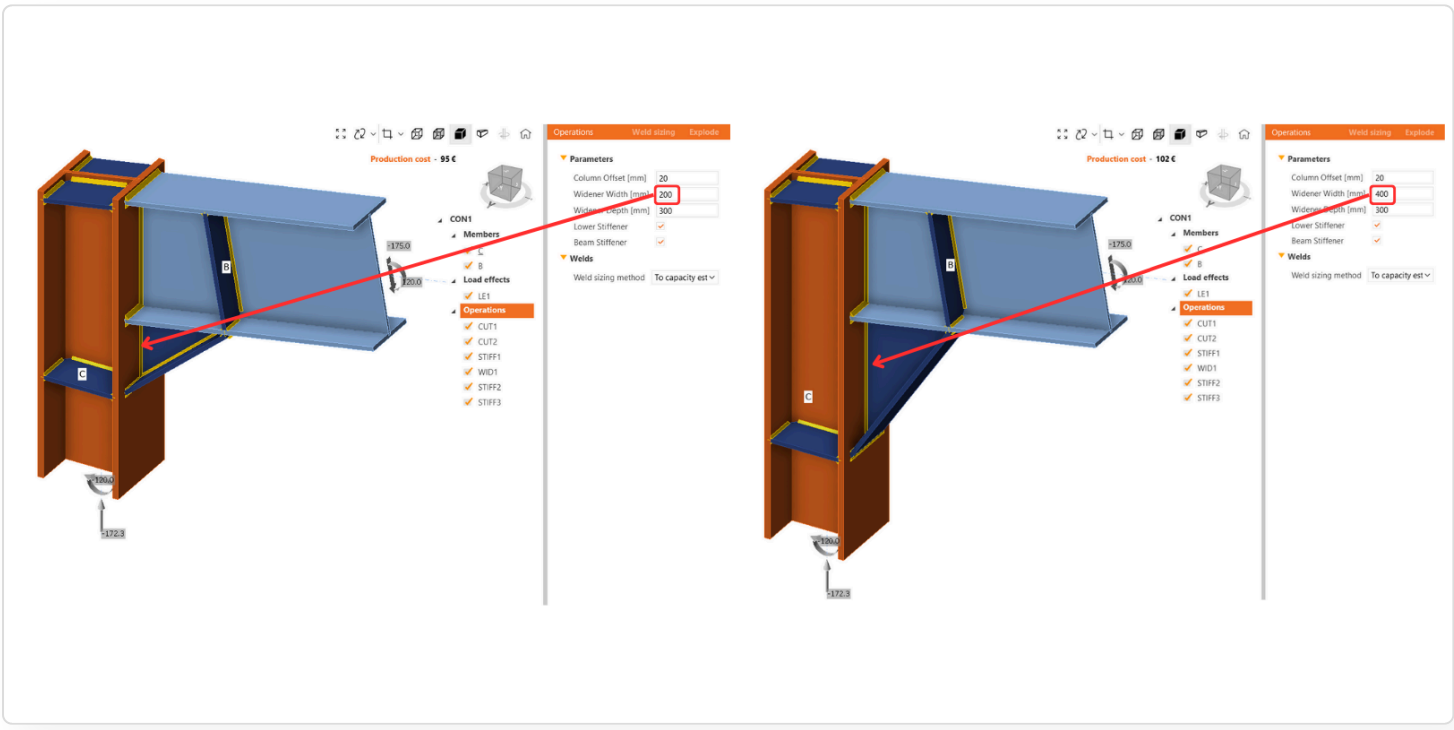



In addition, after proposing the template, it is possible to **change the parameters directly in the main design window** and there is no need to go into developer mode. This user-friendly environment allows less-experienced users to work with predefined parameters safely according to the presets of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



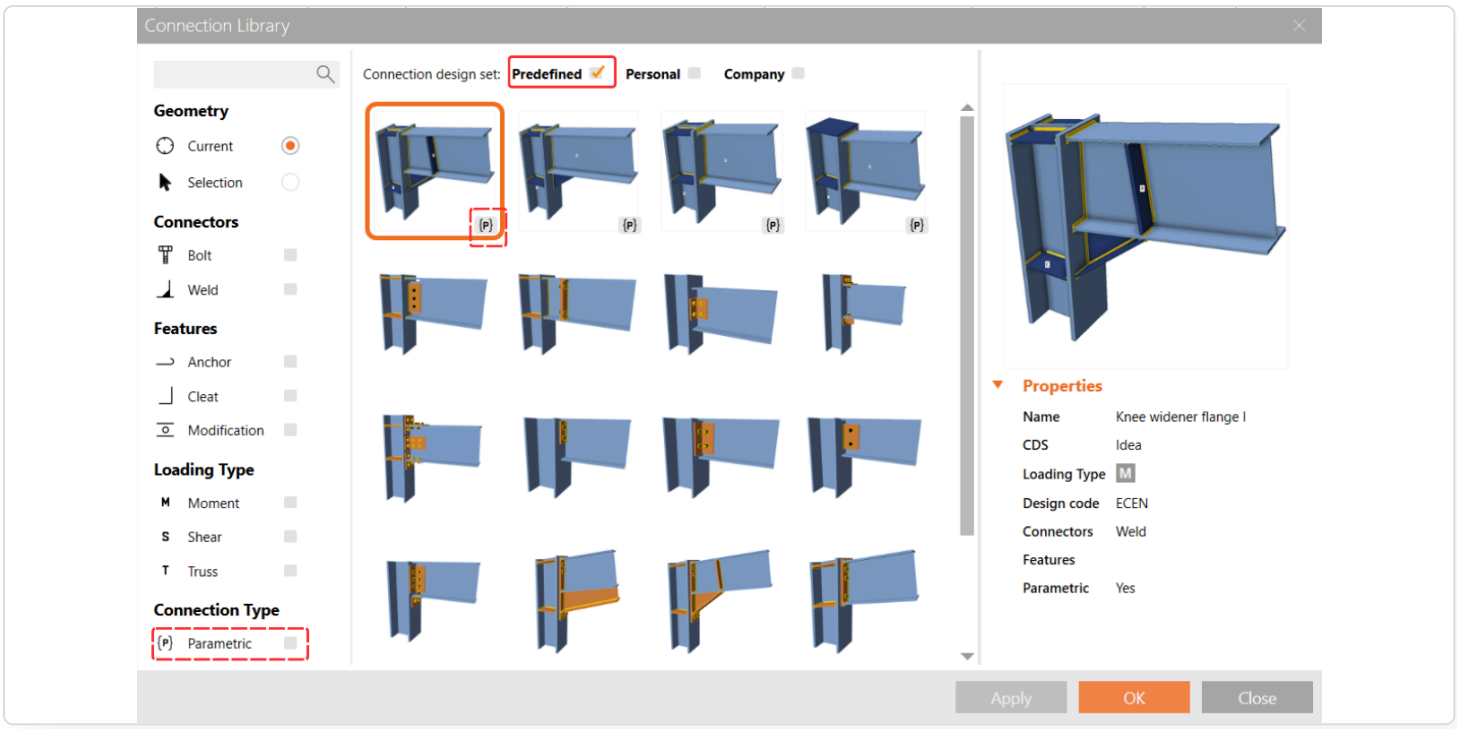

This makes further optimization very simple as there is the possibility of avoiding changing each entity one by one when using operations.

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 offering.
- **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.*

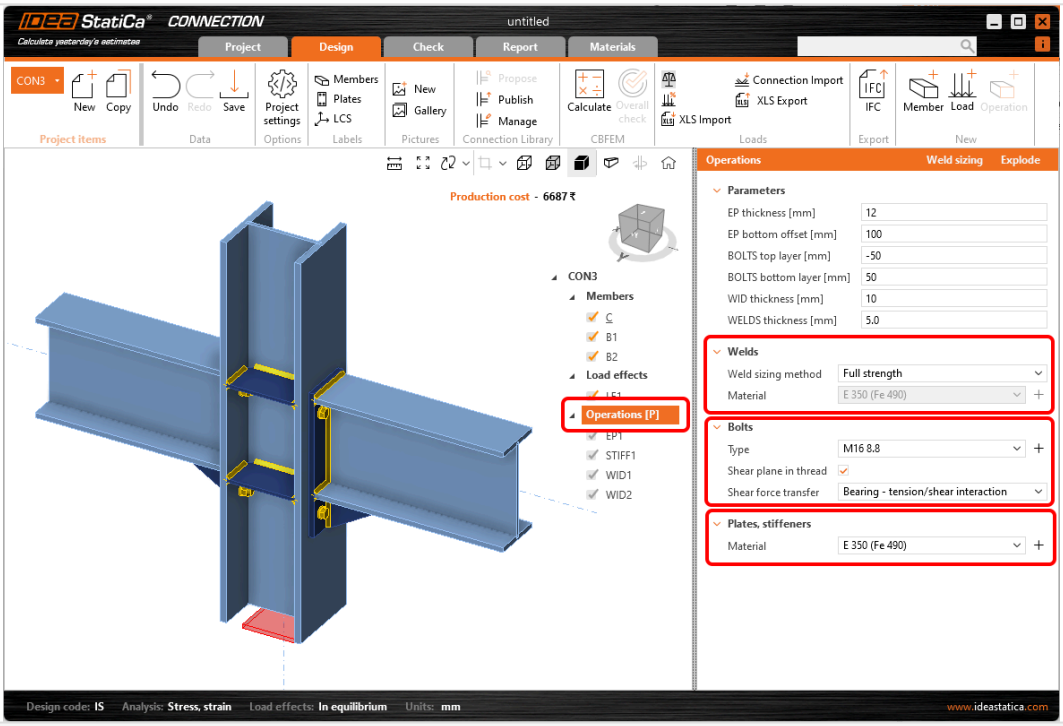
# Default control sets in the Operations

The design of a connection by parameters consists of the setting of very unique parameters specific for every single connection type on one hand. On the other hand, there are parameters that are used almost always due to common engineering practices, such as a set of welds, fasteners, and materials.

To simplify the creation of parametric templates and to reach this setting anytime, we include these controls as default sets in the operation root.

When you select Operations, there are, by default, the following settings:

- Weld setting: **Weld sizing method** and weld **Material** (24.0.5)
- Bolt setting: **Type**, **Shear plane in thread,** and **Shear force transfer** (24.1.0)
- Plates setting: **Material** (24.1.0)



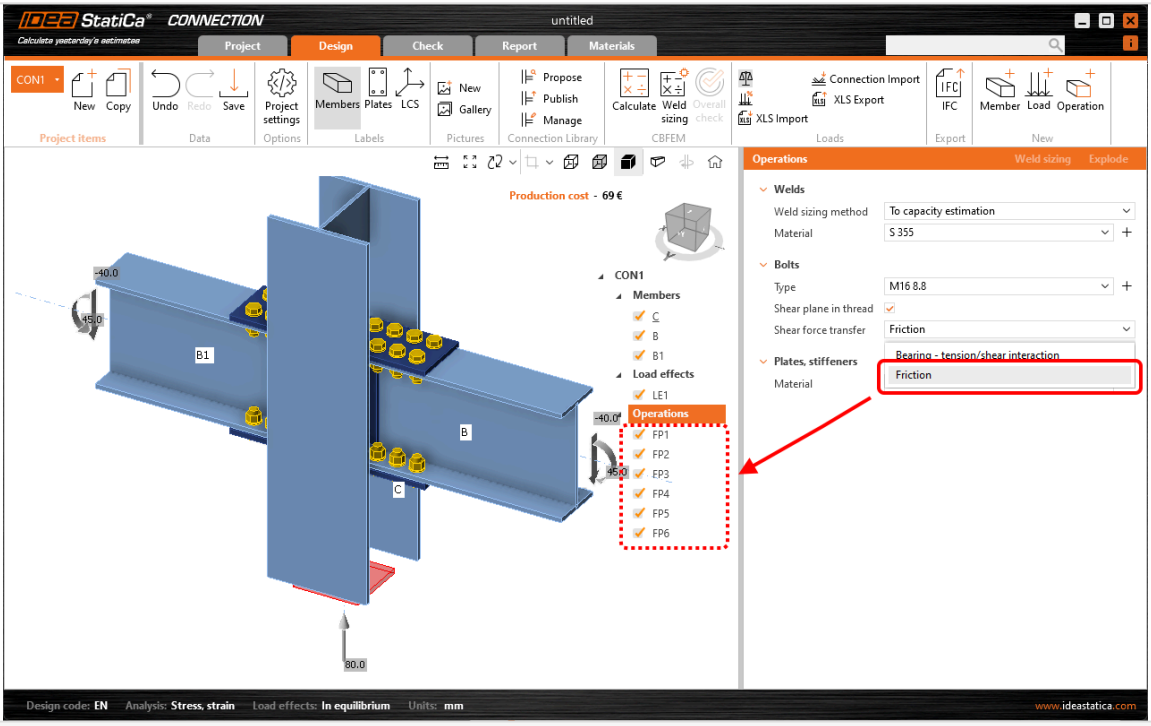

These controls set the specific property for the whole connection model to all operations and are shown in the Operations root by default. They do not change the properties of the structural members, only the manufacturing operations.

In case any of the properties related to the control **is used in any parameter** of the parametric template, the control is automatically disabled. By this behavior, we prevent the possibility of influencing the same property from two places and, thus, ruining the connection detail by loops in parameters.

As a benefit, these controls are also shown without a parametric template, enabling quick changes and unifying properties of any connection model.

# What are the benefits?

- For the **parametric template user** (e.g., junior user), all operations are disabled (read-only). However, **changing the material, bolt class, or plate material** is still a commonly expected action. Although, if there is a single parameter missing in the parametric template, it makes the template unusable. Common properties such as welds, bolts, and material can now be controlled in each parametric template using the default controls shown under the parameters.
- Adding these controls **simplifies the work of the template creator** (e.g., senior user) **in preparing the parametric templates**. Now, it is not necessary to manually create special parameters for modifying common properties such as welds, bolts, and materials.
- Sometimes, you want to **unify bolt grades or plates and weld material** in the whole connection detail even when the **parametric template is not used**. It can be easily done either by multi-selecting in the tree and editing operations of the same type (nevertheless, in this case, it has to be done multiple times for each group of operations separately), or it is now possible **to change the requested property** or just check if the same property is used **in one click**.

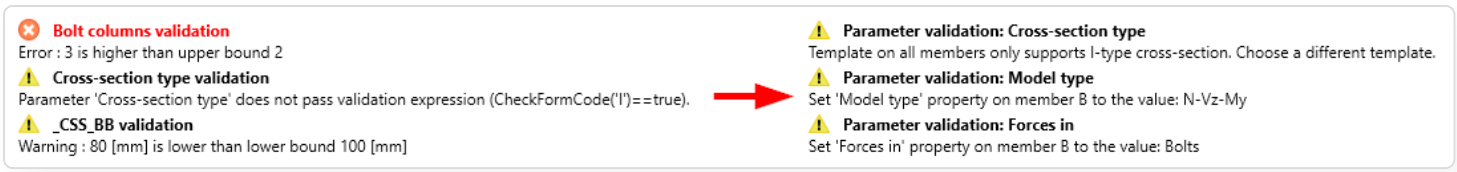



*Released in IDEA StatiCa version 24.1.*

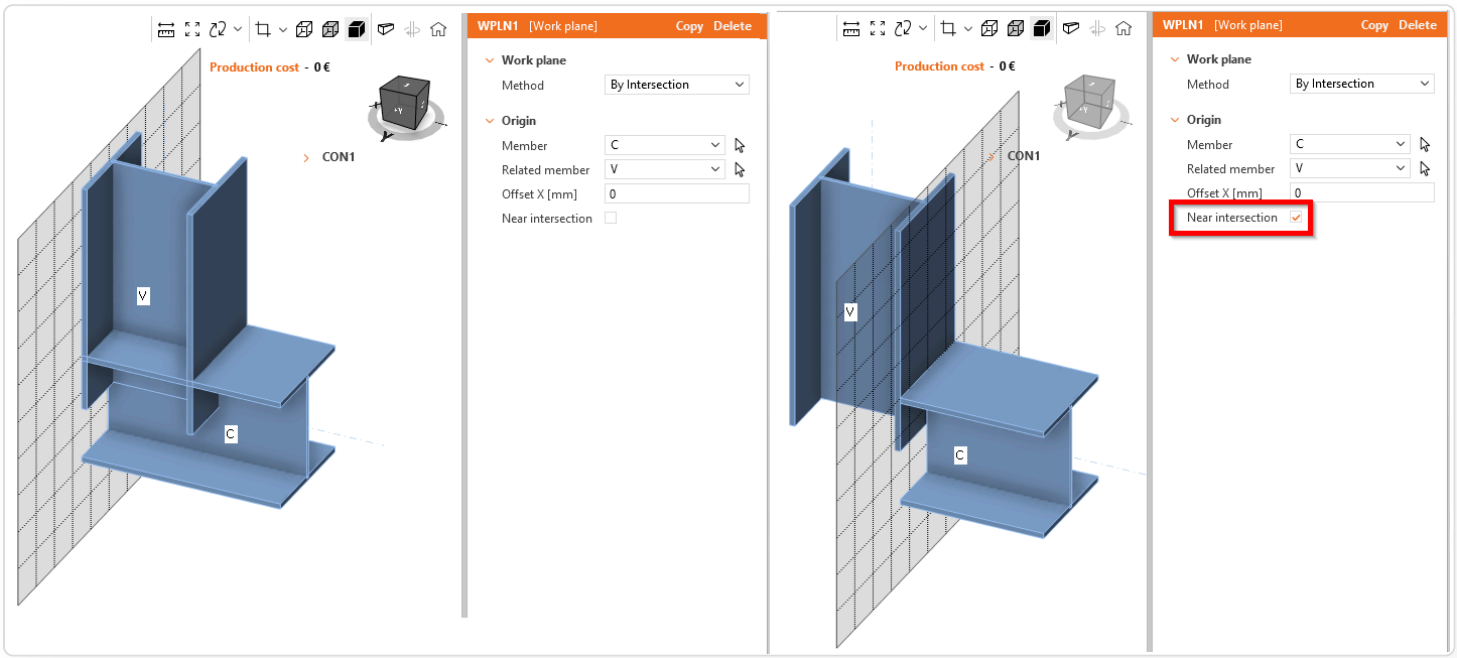
# Improvements of the predefined set

Several enhancements have been made to **parametric templates created via the Wizard** to improve usability, clarity, and model consistency.

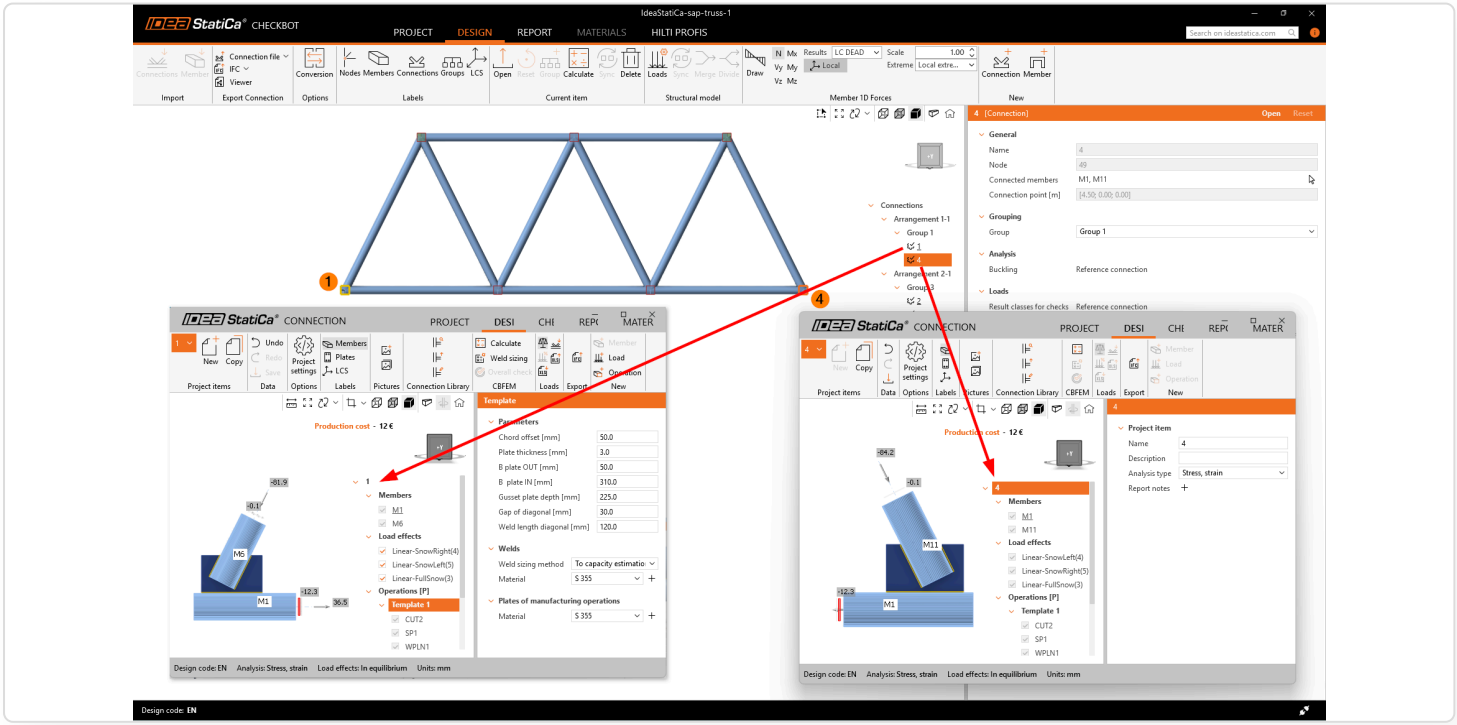
- Validation messages have been refined, only functional and relevant messages are now displayed, such as incorrect member type, cross-section compatibility, or load definition issues.
- Message texts have also been rewritten for better readability and technical accuracy.




- Templates now correctly retain **cross-section data** for cleats and stiffening members, ensuring accurate geometry generation.
- **Custom cross-sections (CSS) stored in the MPRL** are included in templates and automatically applied to new models.
- A new **Workplane operation method, Intersection**, allows templates to be used for members ending in both X+ and X– directions within Checkbot, with adjustable offset and positioning on the near or far edge of the related member.




Thanks to this new function, templates are applicable at both ends in the Checkkbot.



Parametric templates can also be combined with each other or with standard manufacturing operations.

*Released in IDEA StatiCa version 25.1.*

# Bulk import of templates to Connection Library

This feature allows users to efficiently upload a large number of connection design templates to the Connection app. Enabling bulk uploads saves huge amounts of time compared to one-by-one hand publishing.
To save a single connection design template into the **Connection Library** in the IDEA StatiCa Connection app, users can use the Publish command. Design templates can be published to a **Company set** and **Private set,** and can be standard (a compilation of design operations) or parametric (a simple list of defined parameters instead of operations).

To batch publish design templates, you can either download the ready-to-use tool IDEA StatiCa Bulk Publisher or prepare your own routine using Python or C# and the open API of IDEA StatiCa.

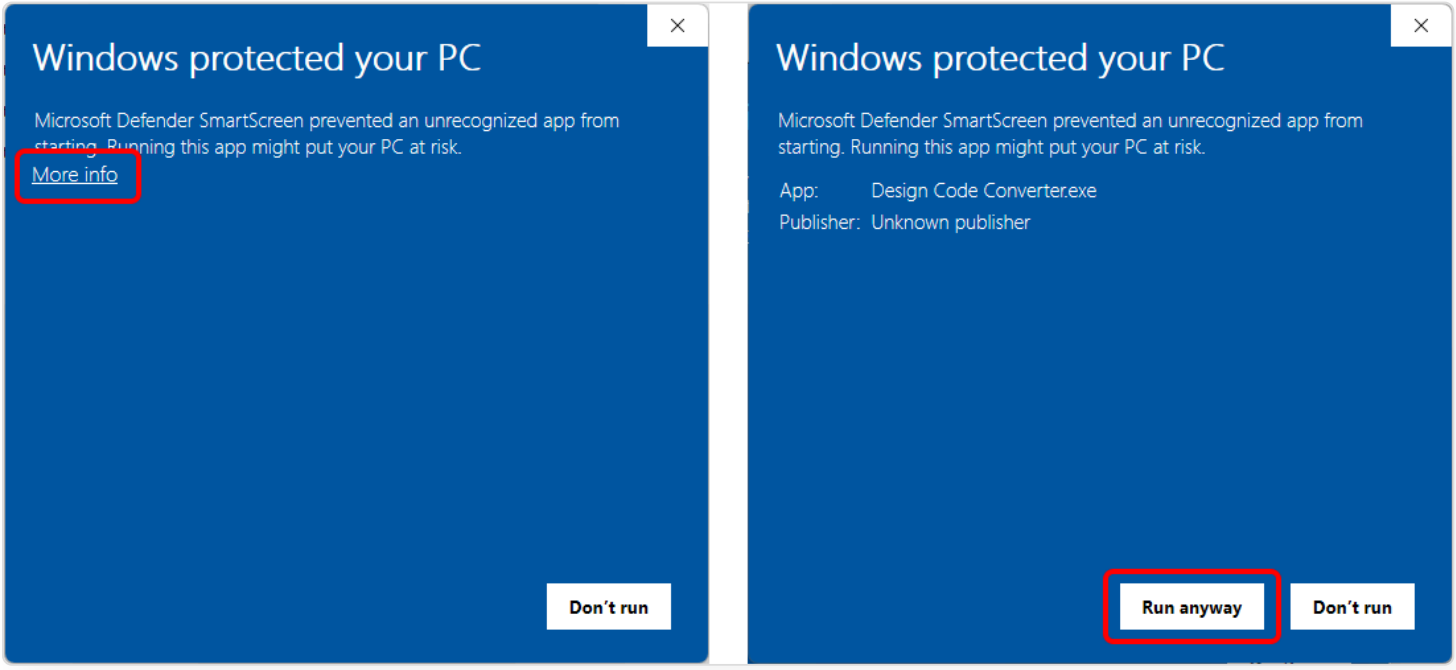
# IDEA StatiCa tool for bulk import of design templates

With the IDEA StatiCa Bulk Publisher (download here), you can import a folder containing multiple Connection files as the design templates into the Connection Library. You can also modify the provided tool to your needs using the provided code resources.

To run the bulk import tool, you need to have ASP.NET Core installed on your computer (usually it's part of the Windows system and the .NET framework).

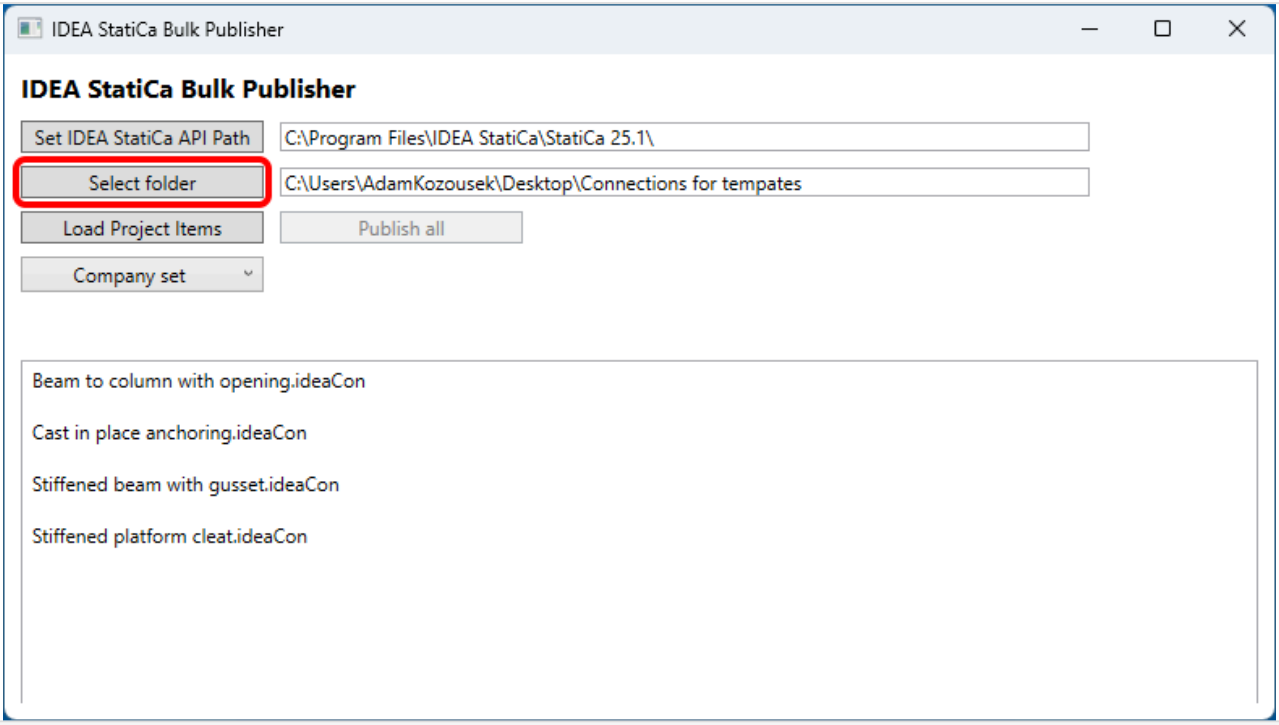
The bulk import tool uses the IDEA StatiCa API. The default path to the API is set to:
*C:\Program Files\IDEA StatiCa\StatiCa 25.1*
You can change the path to the API when you install newer versions of IDEA StatiCa.

In case your Windows user settings don't let you run the .exe file, click **More info** and then **Run anyway** to open the tool, or ask your IT admin for help.

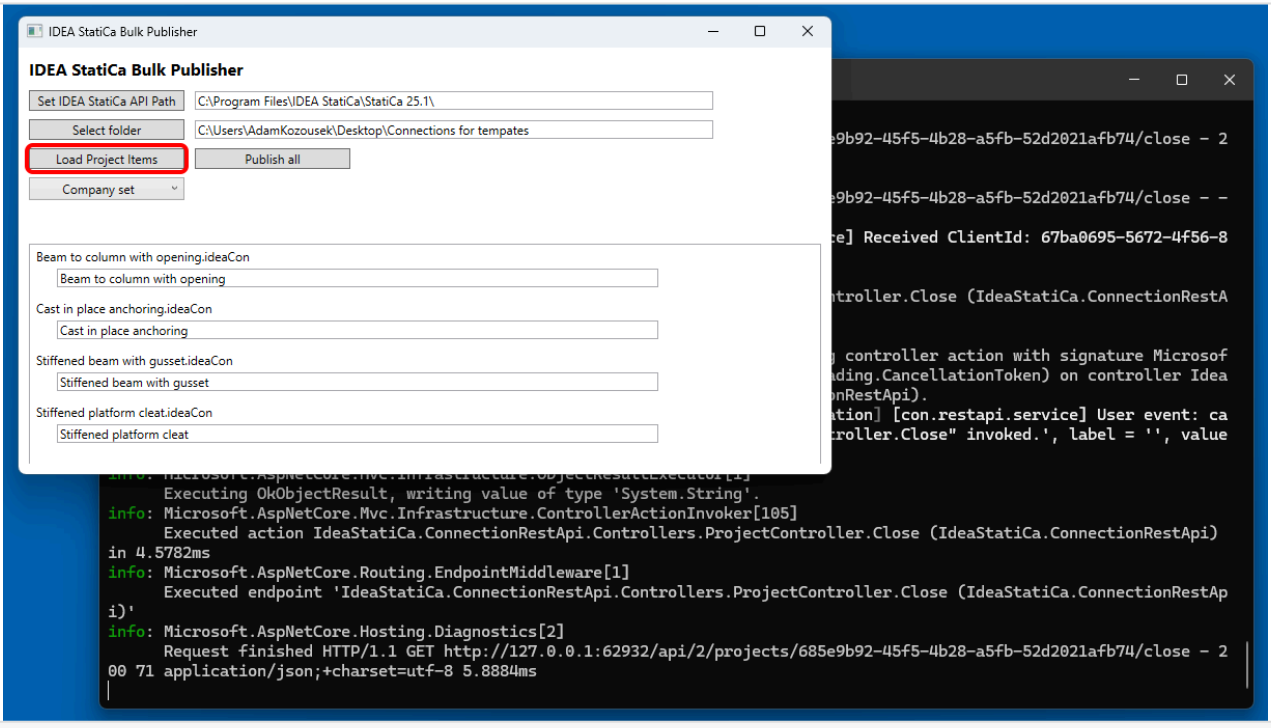


To bulk publish the folder of connection models into the Connection Library set of templates, follow the steps:

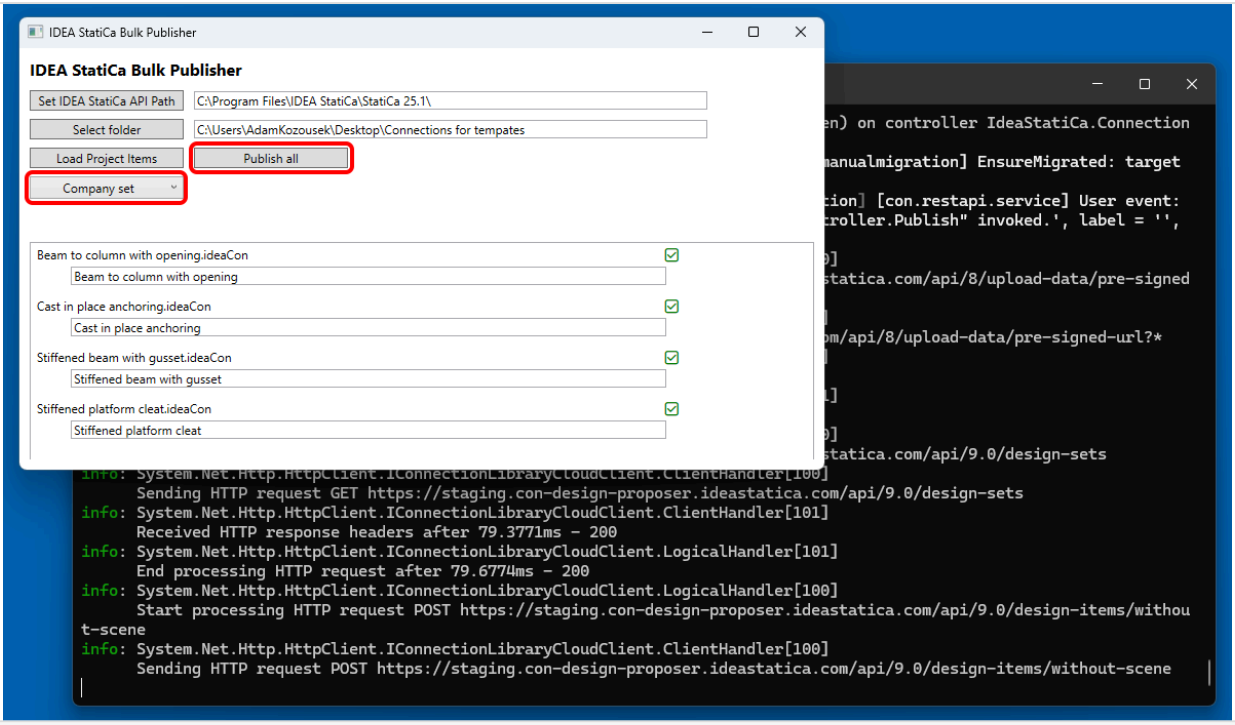
- Open the IDEAStatiCaBulkPublisher.exe.
- Click **Select folder** and navigate to the folder containing your Connection files. The list of files appears in the window.



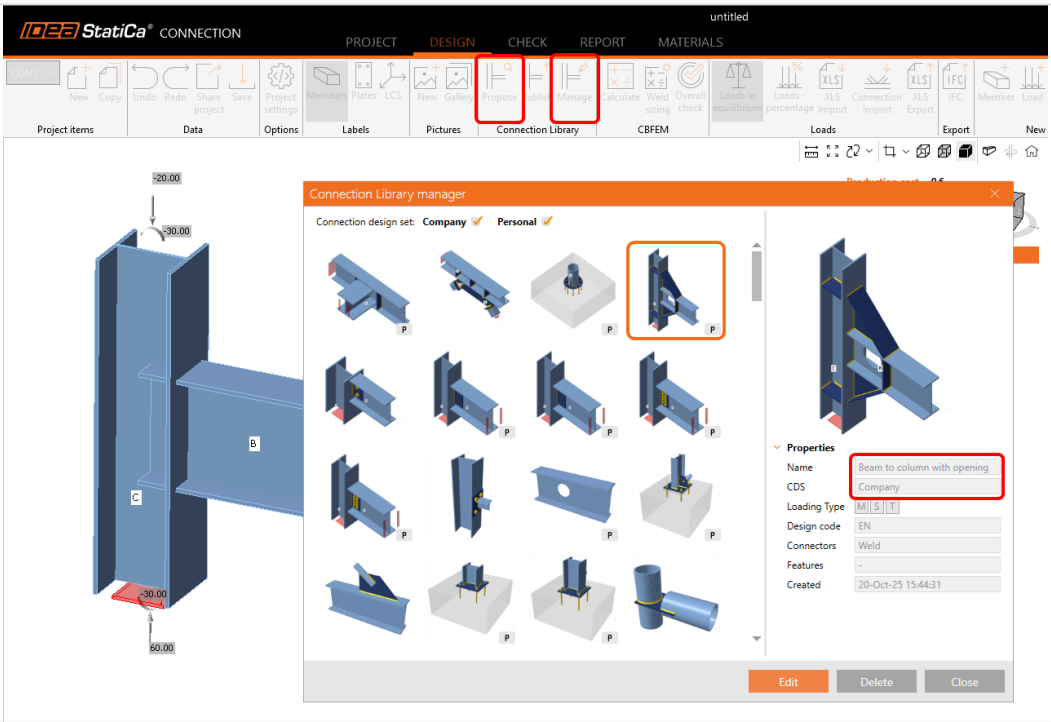

- When you click **Load Project items**, the design data are loaded from the files. The names are automatically taken from the **Project item name** of each connection model (not the file name). You can edit the names manually during the process.




- Select the **Company** or **Private** set, and click **Publish all** to upload the design templates to the Connection Library cloud database.




- Once the process is finished, the routine screen (black) closes, and green ticks are displayed next to the uploaded templates.
- In the Connection app, you can use the templates with the **Propose** function and browse them in the **Manager** of Connection Library.

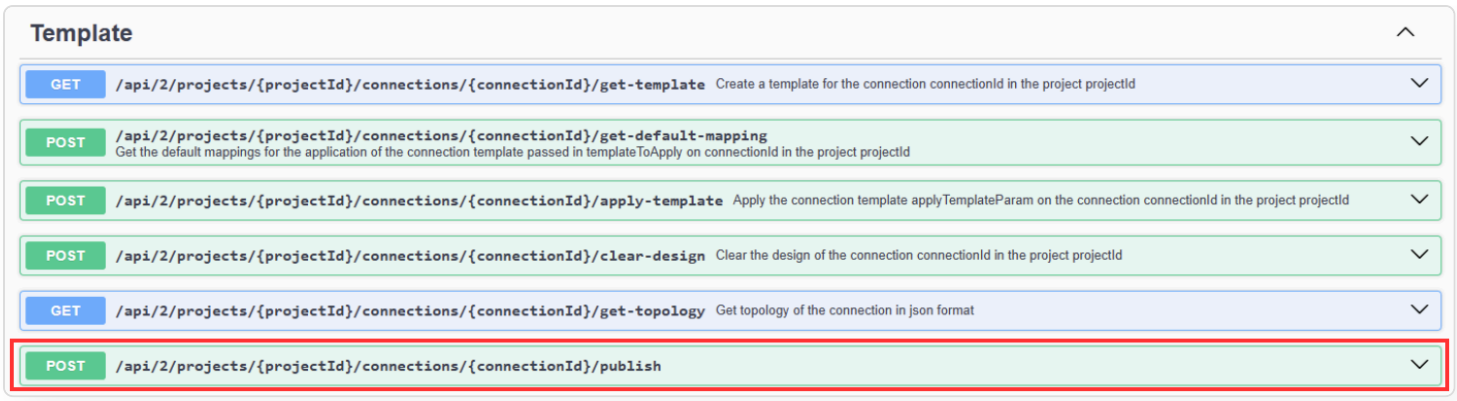



# Open API import using Python and C#

Download and edit the prepared code examples from the GitHub repository to build your own import tool.

- C# section on the developer.ideastatica repository
- Python section on the developer.ideastatica repository

When you initialize the API for the Connection app, use the publish API endpoint to utilize the import.




The API is open, and broad options are available for importing multiple Connection designs into the Connection Library.

# Known limitations

- It is not possible to bulk delete imported connection design templates from Connection Library sets. Only one-by-one deletion is available in the Manage window of the Connection Library in the Connection app.
- No automated validation of connection design templates is available.

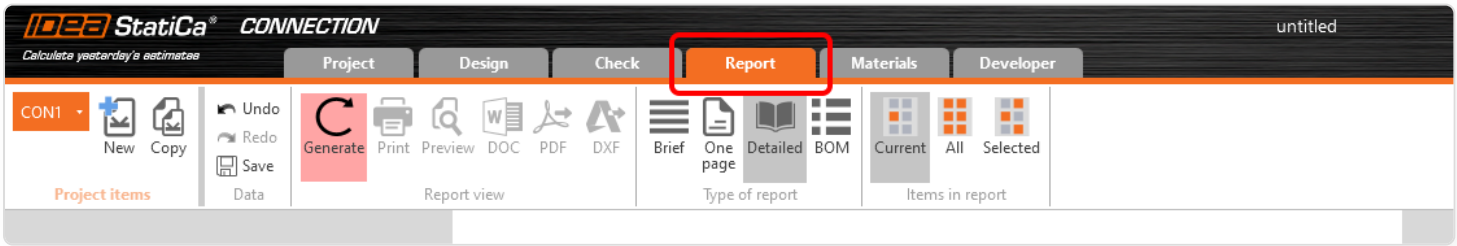
Customization of your projects

# How to create and export a report to MS Word, PDF or DXF files

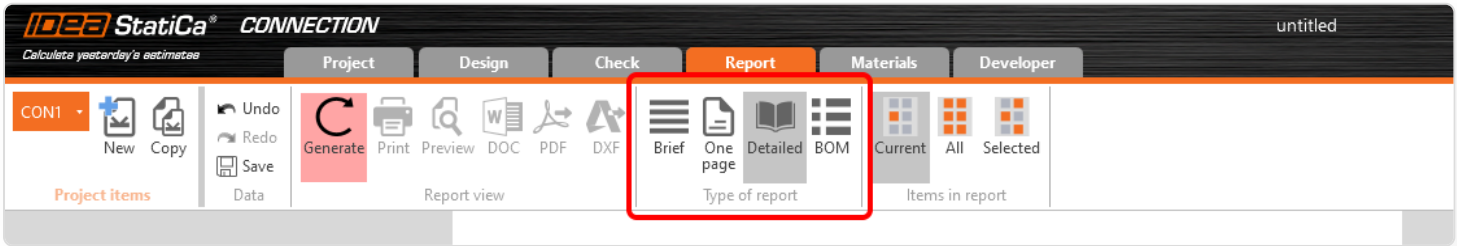
Exporting IDEA StatiCa reports to various output file types enables users to work with these reports further or share them in a generally acceptable format for additional project steps.

# Creating areport

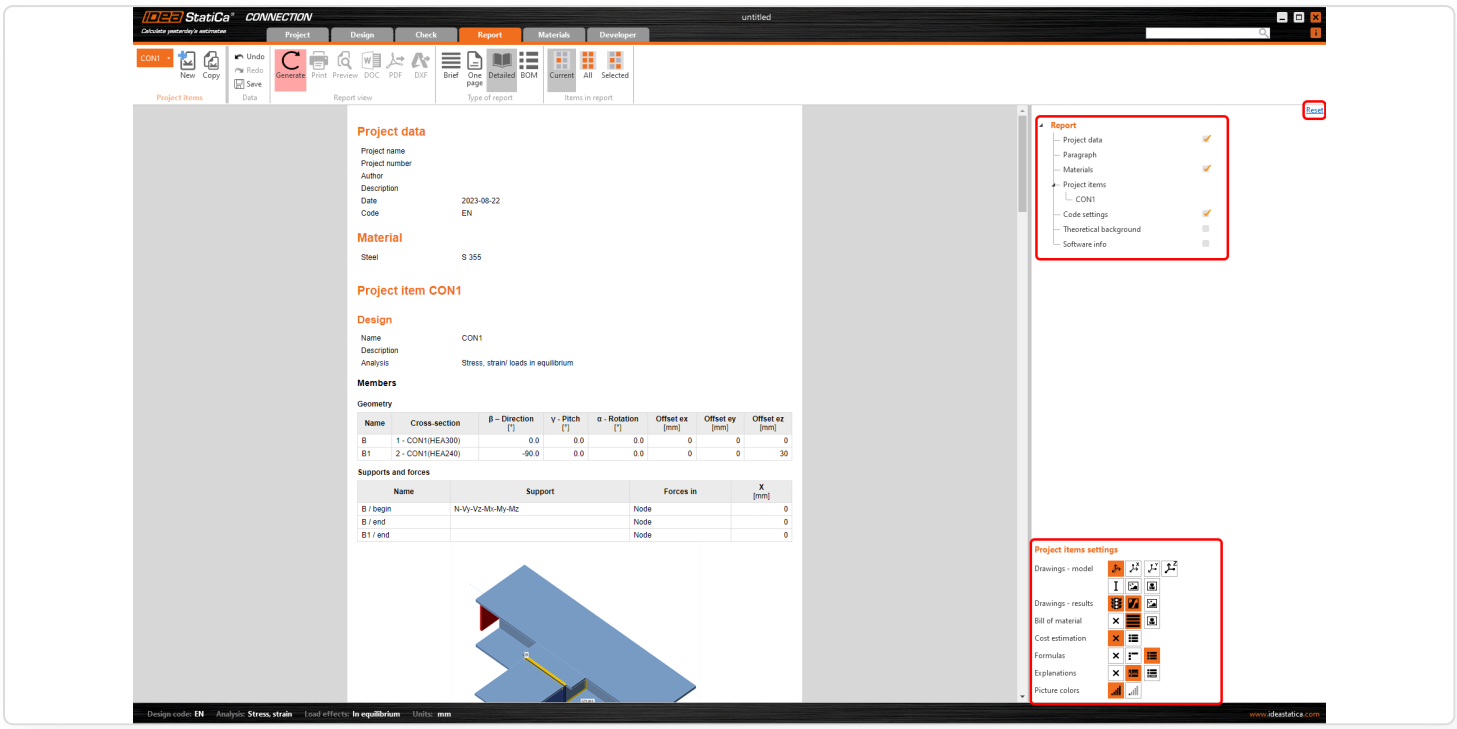
To create a report, start in the **Report** tab.



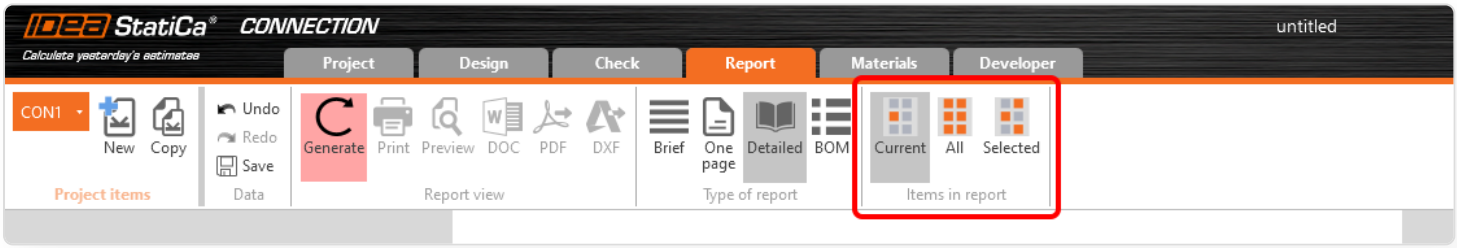

You can select the level of reporting: **Brief** gives the overall results only, **One page** puts the most necessary results on a single page, and **Detailed** is the most descriptive option. **BOM** (Bill of materials) presents the 2D drawings of the plates in the connection.



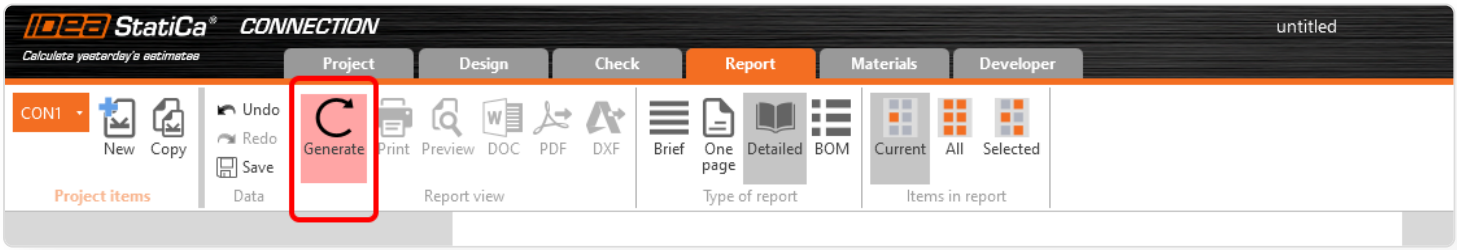
The report can be adjusted based on your preferences – use the **Report settings** (top right) and **Project items settings** (bottom right corner) menus. Preferences can be saved for future projects.




You can also select if you want the report to include the current project item only, the selected items, or all of them.

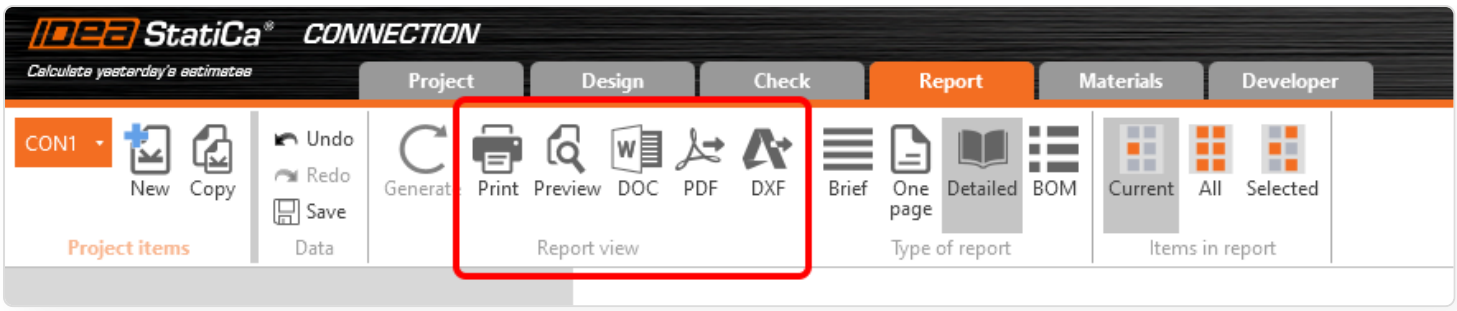



After each change, the report must be re-**generated** using the button in the top ribbon.



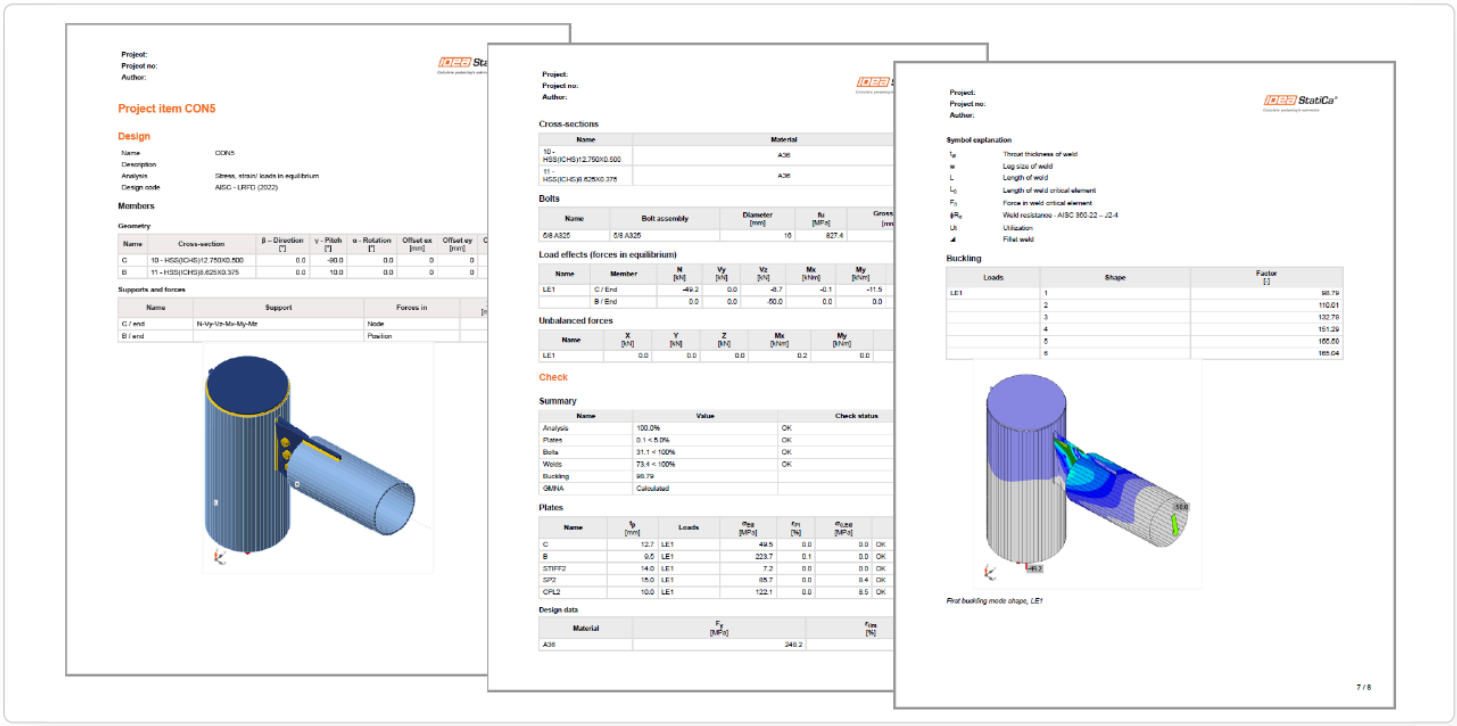

# How to export report to MS Word, DXF or PDF files

Once the report looks as you wish, you can directly **Preview** or **Print** the output or export it into **DOC** (editable Microsoft Word document) or **PDF** formats. The bill of materials can be exported to an editable **DXF**.




# Improvements according to versions

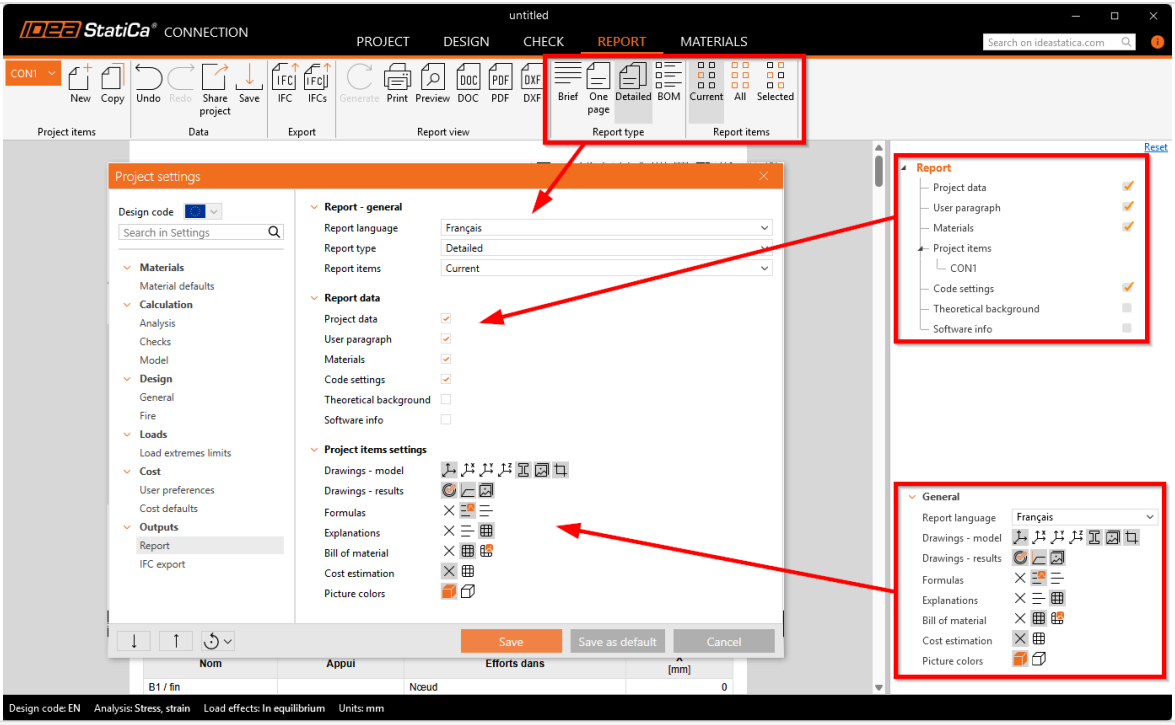
Check also the improvements in version 23.0 in the Detailed model and loads reporting release notes article.



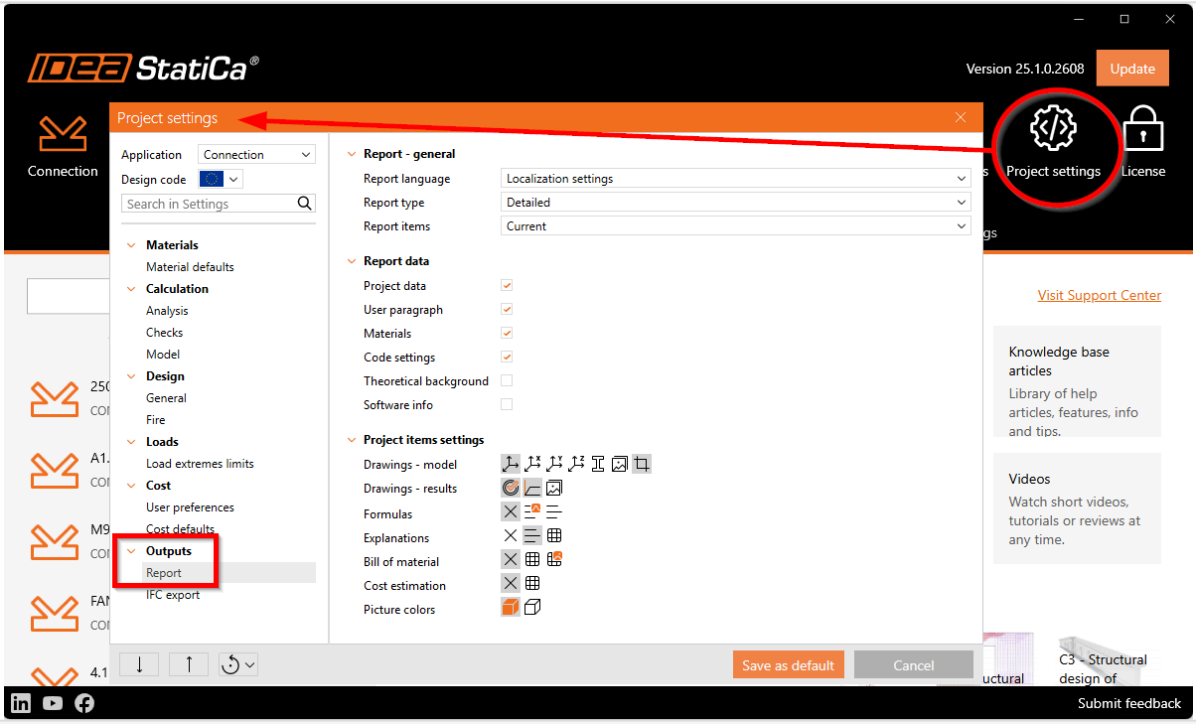
# Save the report settings for the upcoming projects

Report settings can now be saved as **default configurations** and reused across all IDEA StatiCa Connection and Checkbot projects. Users can define preferred report parameters, language, format, scope, and detail level, once and apply them automatically to new projects.

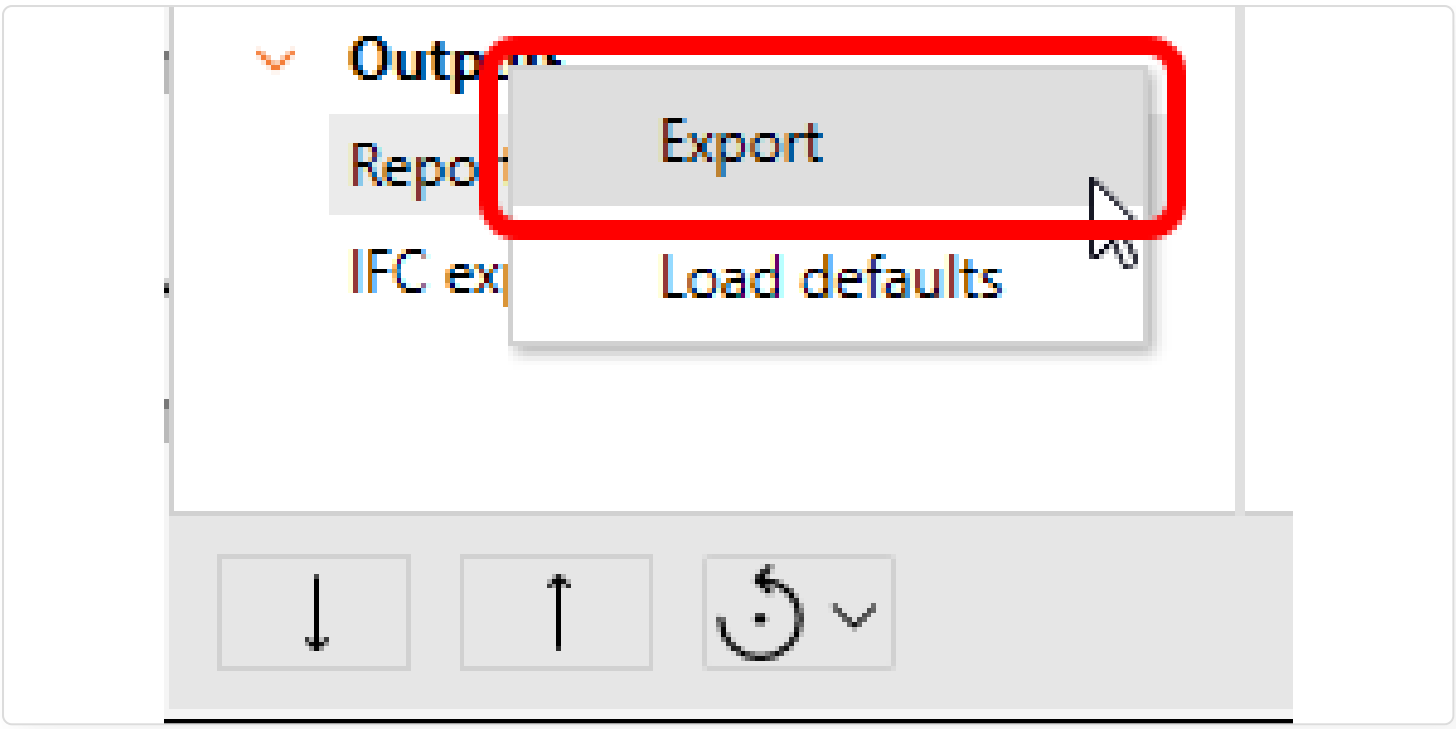
Every change made in the **Report** tab is now automatically stored in the current project's settings. Using the **Save as Default** option in Project Settings → Outputs → Report, these settings can be saved globally and applied to all newly created projects.



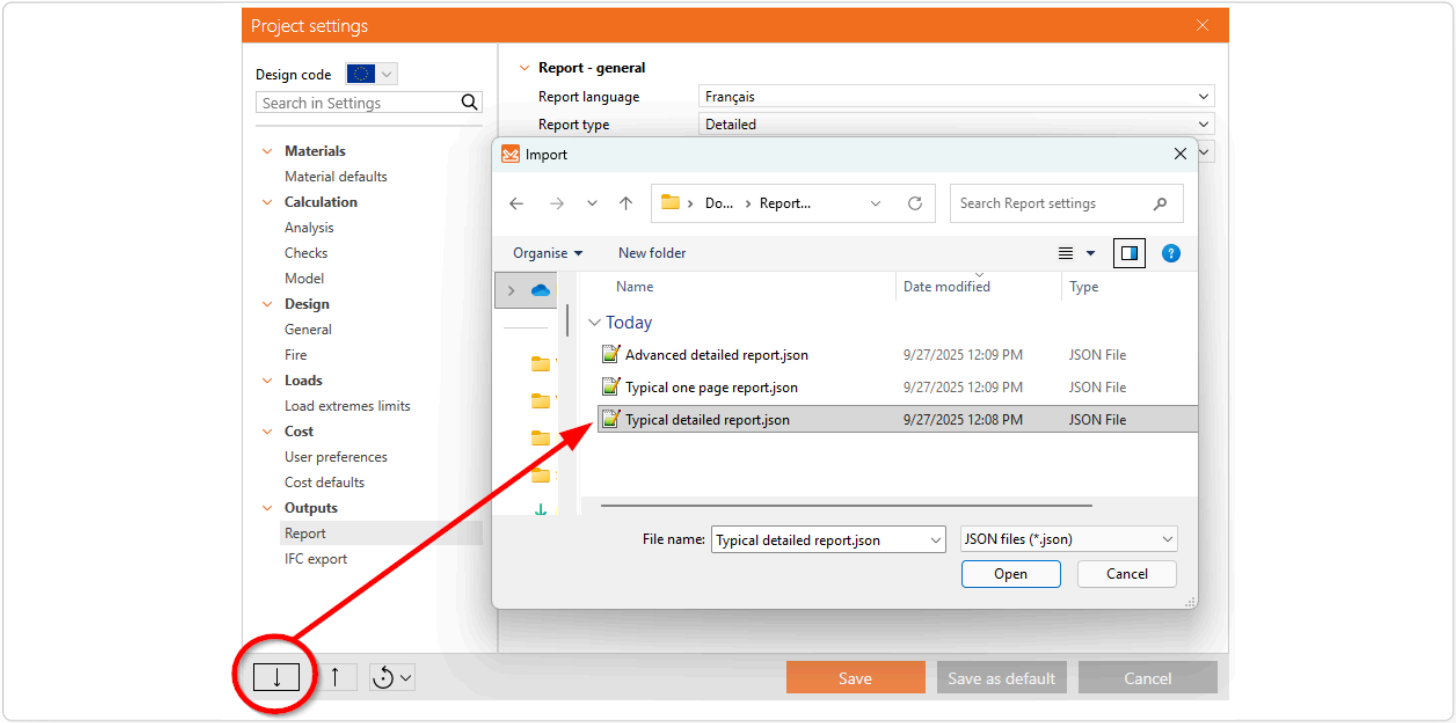

In addition, default report configurations can be managed directly from the **Launcher** without the need to open any project under the Project Settings.




Report templates can be **exported and imported**. Using the **Export** option under the **Outputs** section, you can store customized report setups, for instance, short summaries, full detailed outputs, or client-specific formats.




The **Import** function allows consistent use of these configurations across teams or projects, ensuring uniform report generation company-wide. Only the Outputs part of the settings that was exported is imported.




# Webinar recording

Watch Adam describing how to get the report in our Tackling IDEA StatiCa Connection - The basics webinar.

# Saving of user-defined cross-sections to MPRL

The user defined cross sections can be saved into the MPRL library where they are then available to use in all future projects in IDEA StatiCa Connection or IDEA StatiCa Checkbot (Steel).

The Material and Product Range Library (MPRL) has the ability to save custom cross-sections, just like users can do with materials and bolt assemblies. From MPRL, the cross-section is available for using in users' other projects.

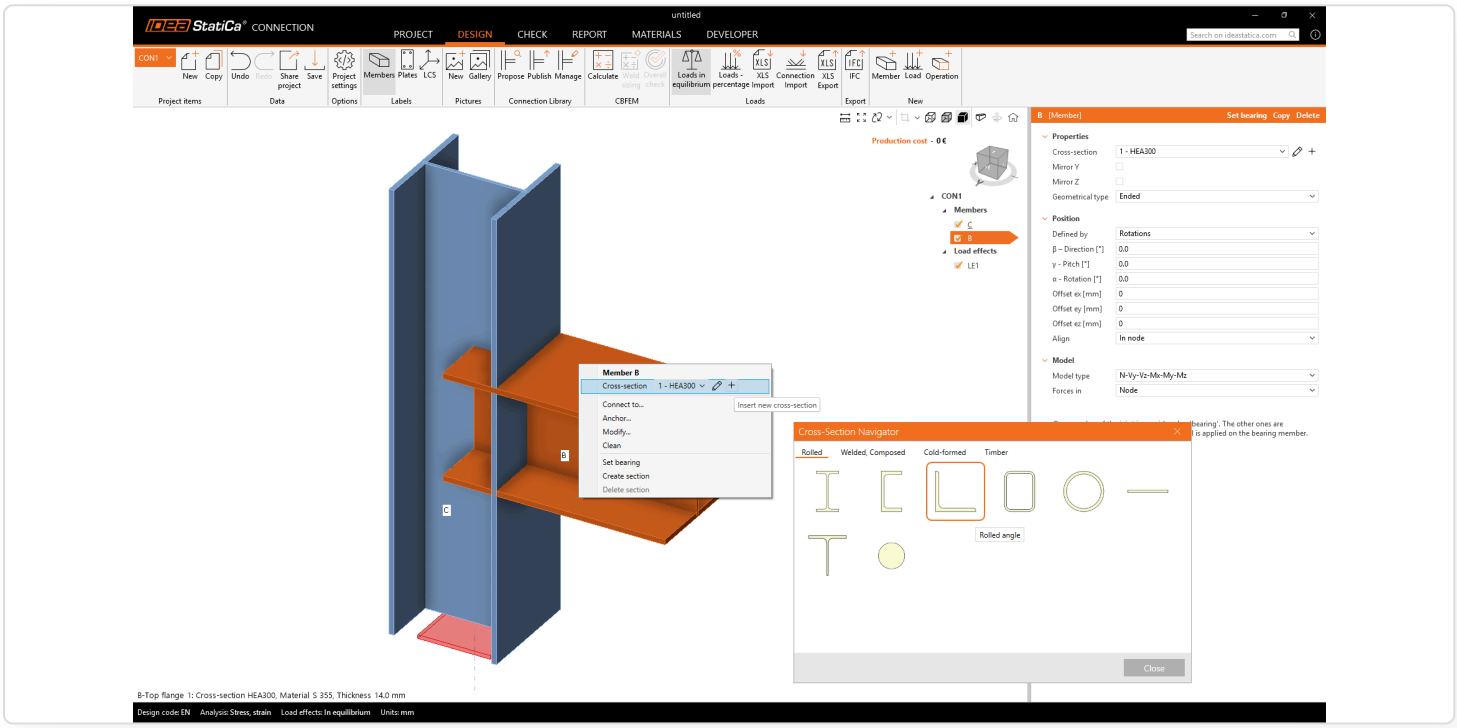
The saved custom sections are available in **IDEA StatiCa Connection** and **IDEA StatiCa Checkbot** – both applications can open the saved MPRL library.

Currently, the user can save the rolled, composed, welded, thin-walled, timber, and general cross-sections. To save the paired rolled, I-cuts (T), a workaround is necessary (see below).

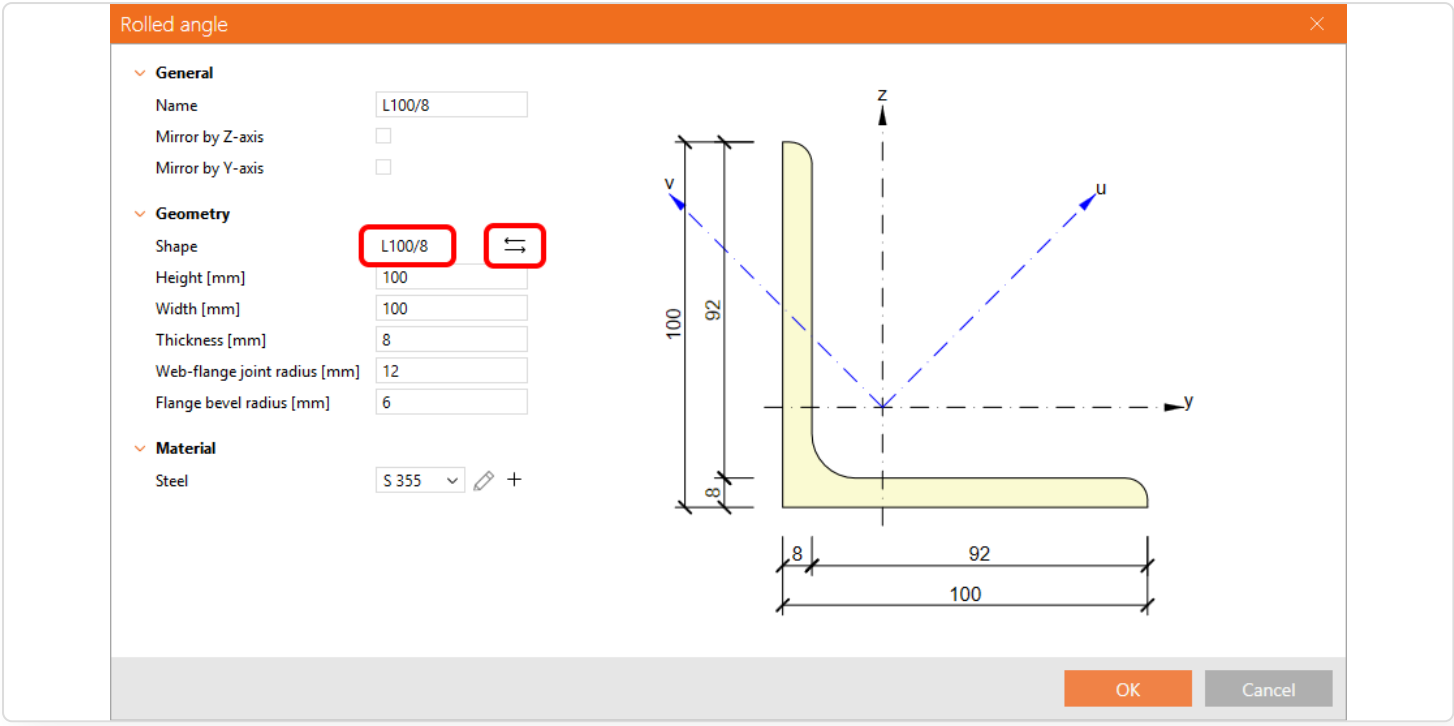
# Definition of a custom/parametric cross-section

The layout of the library of cross-sections merges rolled cross-sections and standard and parametric/custom sections together.

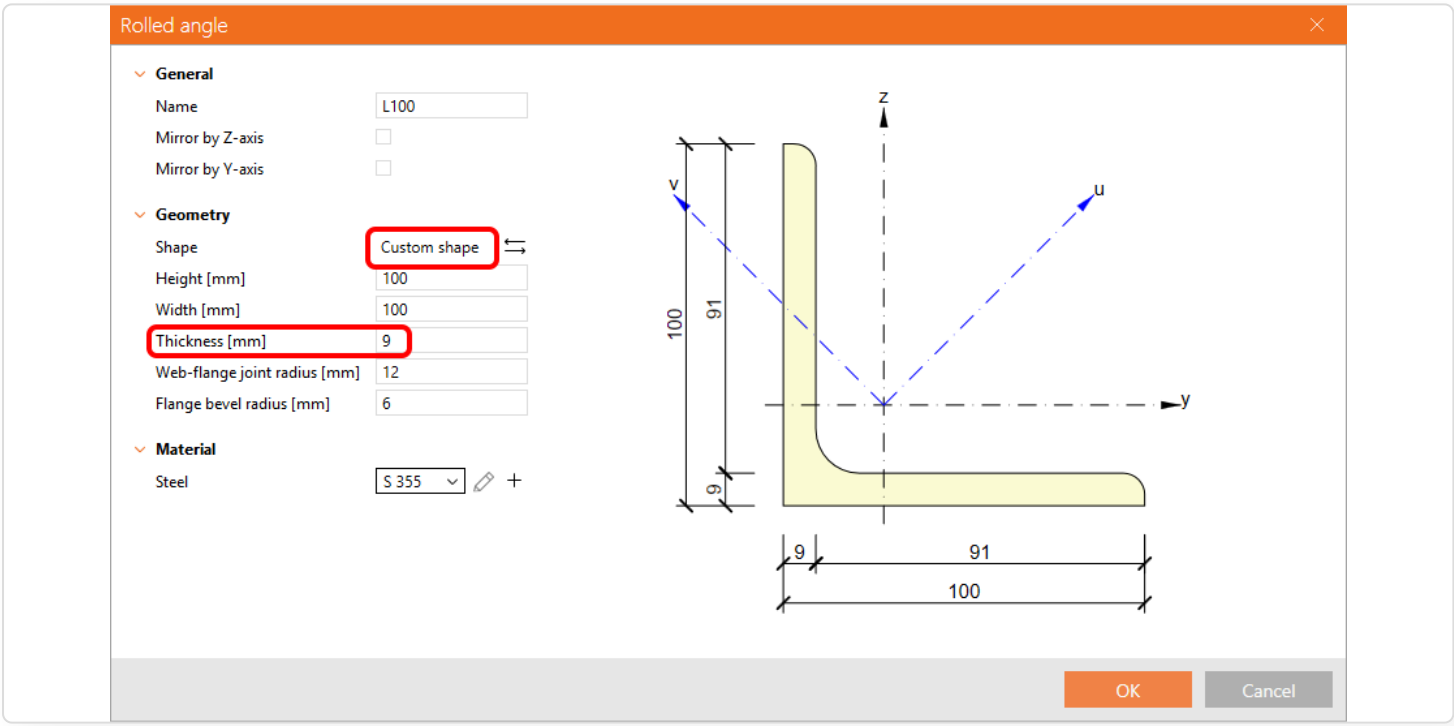
As an example, the definition of a Rolled Angle is presented.



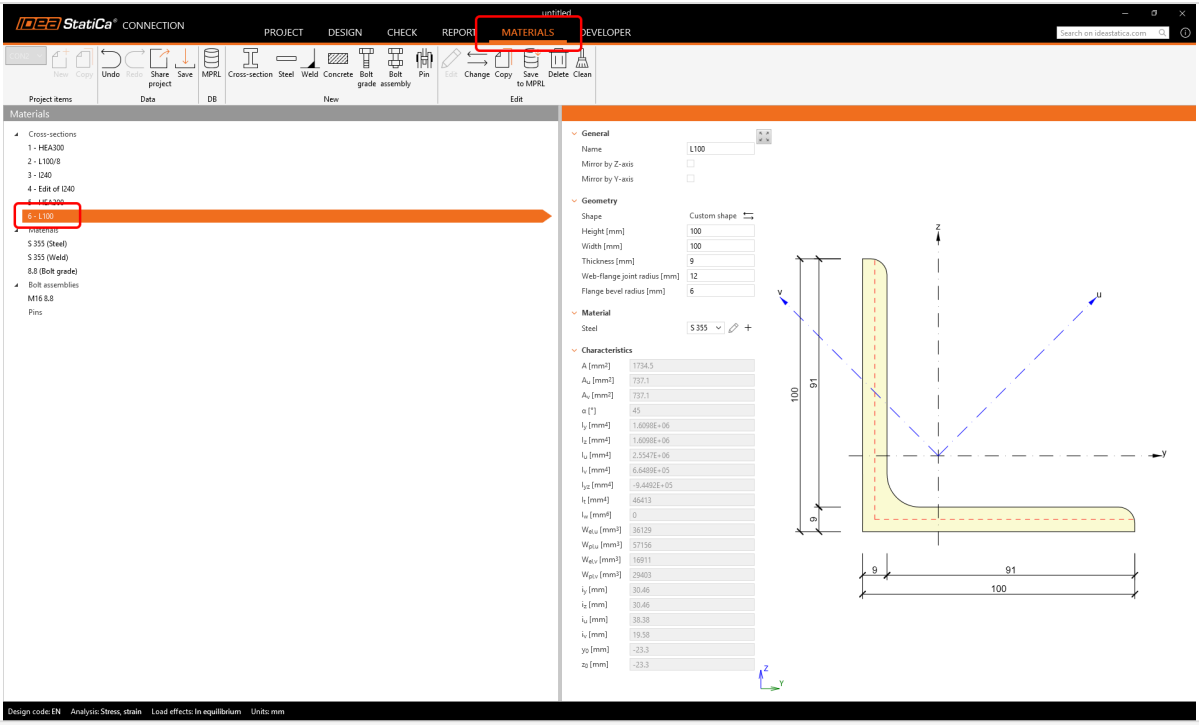

First, the standard section is selected. Using the arrow button, the section can be changed to another one from the library of angles.




When some parameters are changed (the thickness of the standard section, etc.), the section becomes custom.

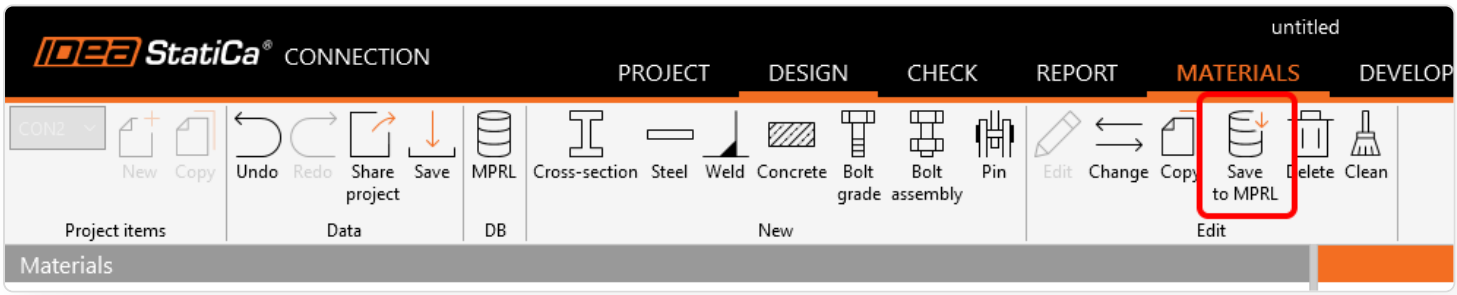



The custom section is displayed in the Materials tab.

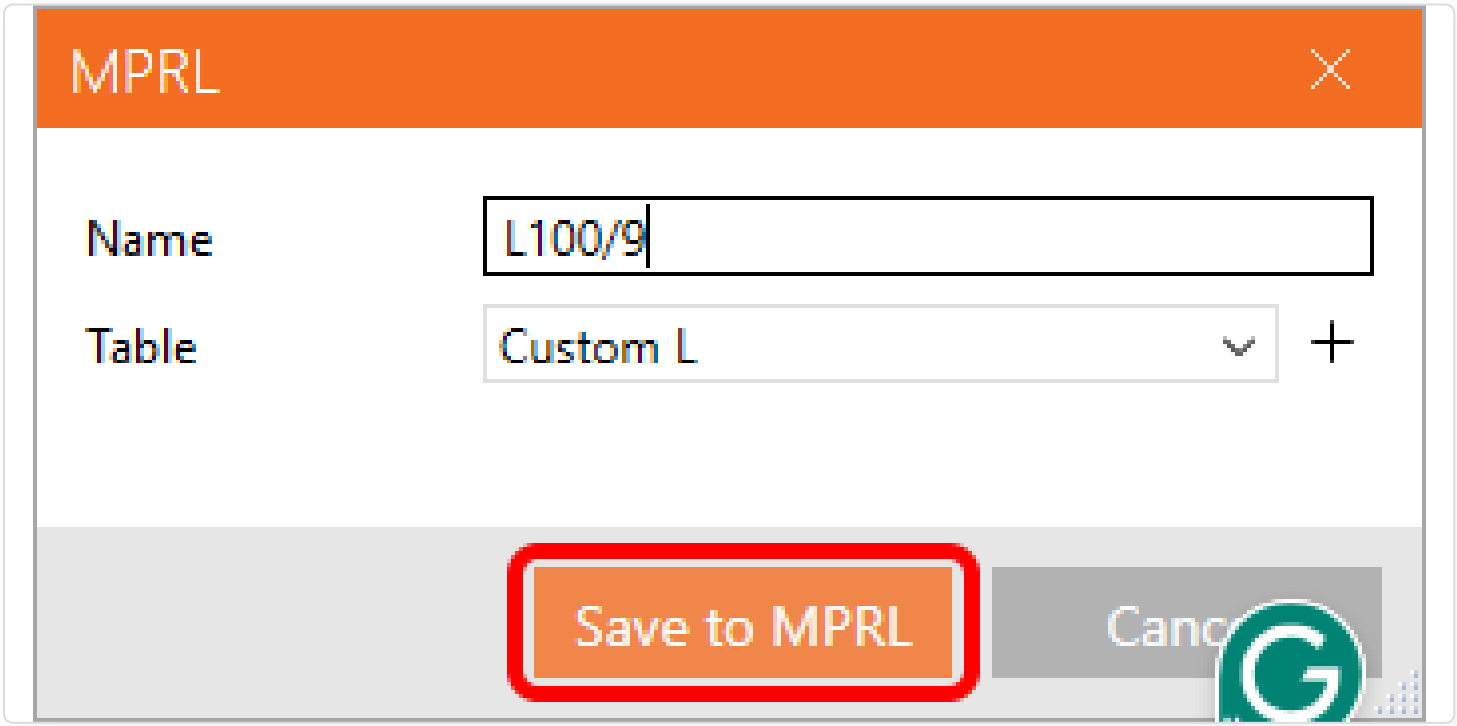



# Saving a library section into the MPRL

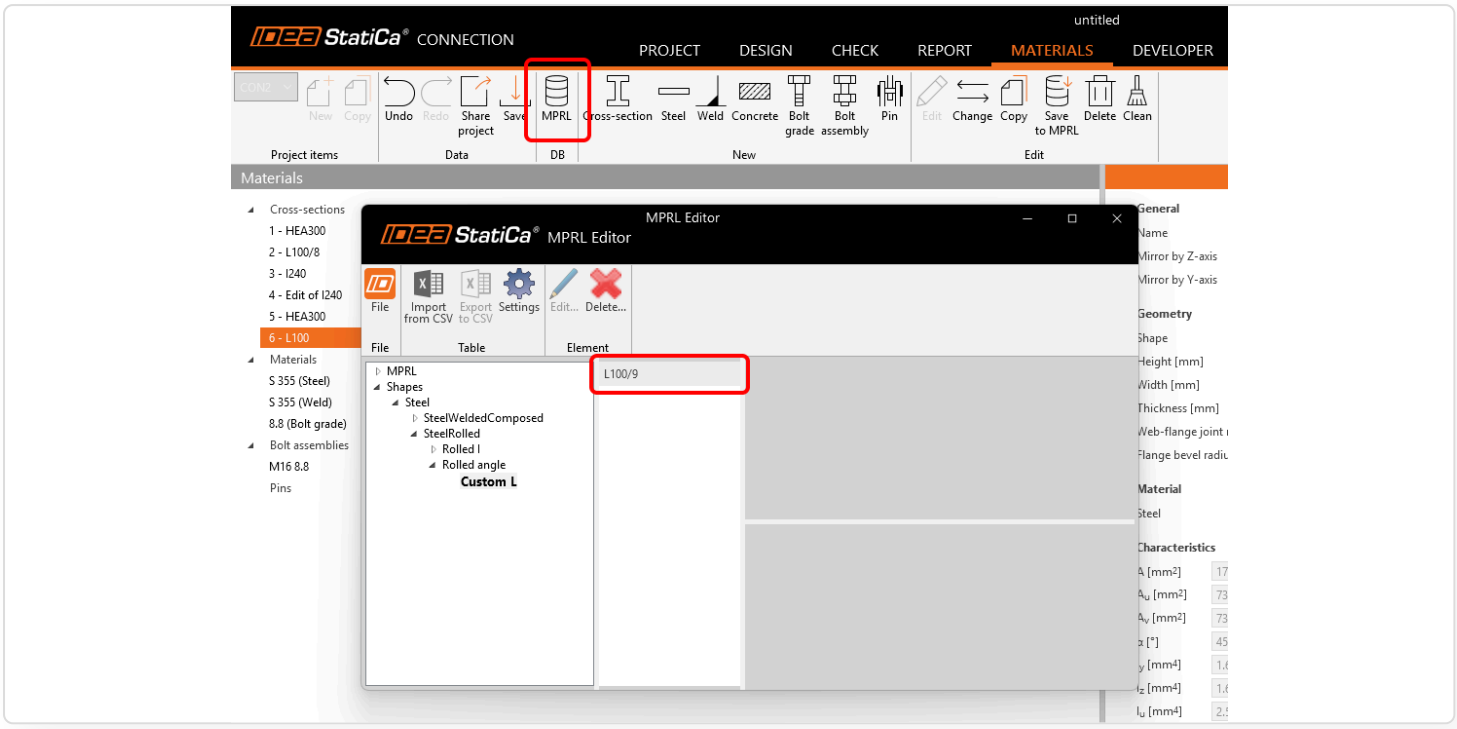
The custom section can be saved into the MPRL using the button in the top ribbon.




The name of the cross-section can be adjusted and saved to a selected/created table in the MPRL.



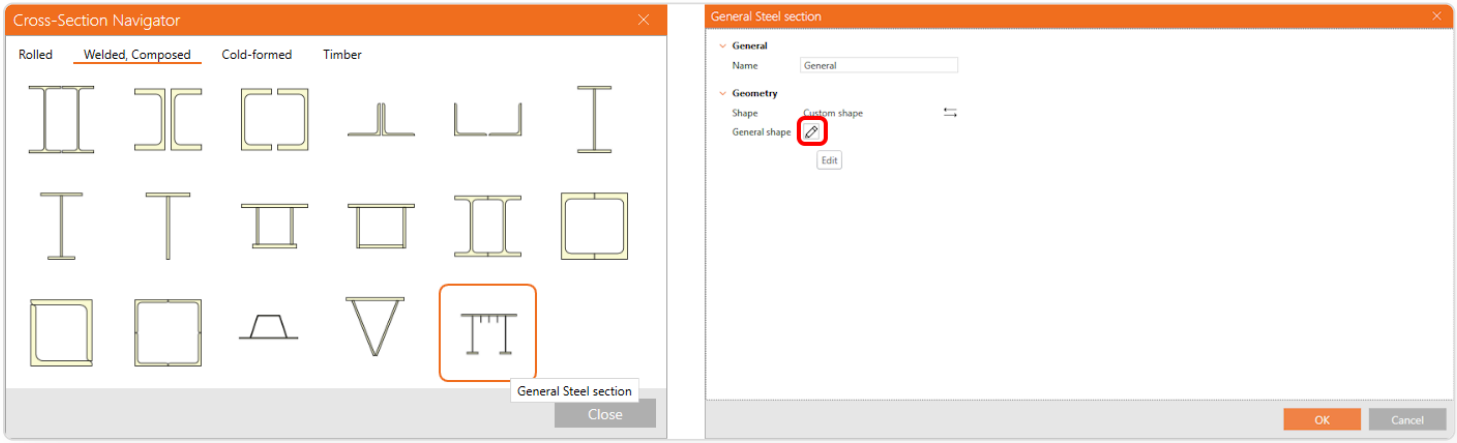
In the MPRL, the custom L (100/9) is saved to the selected location.



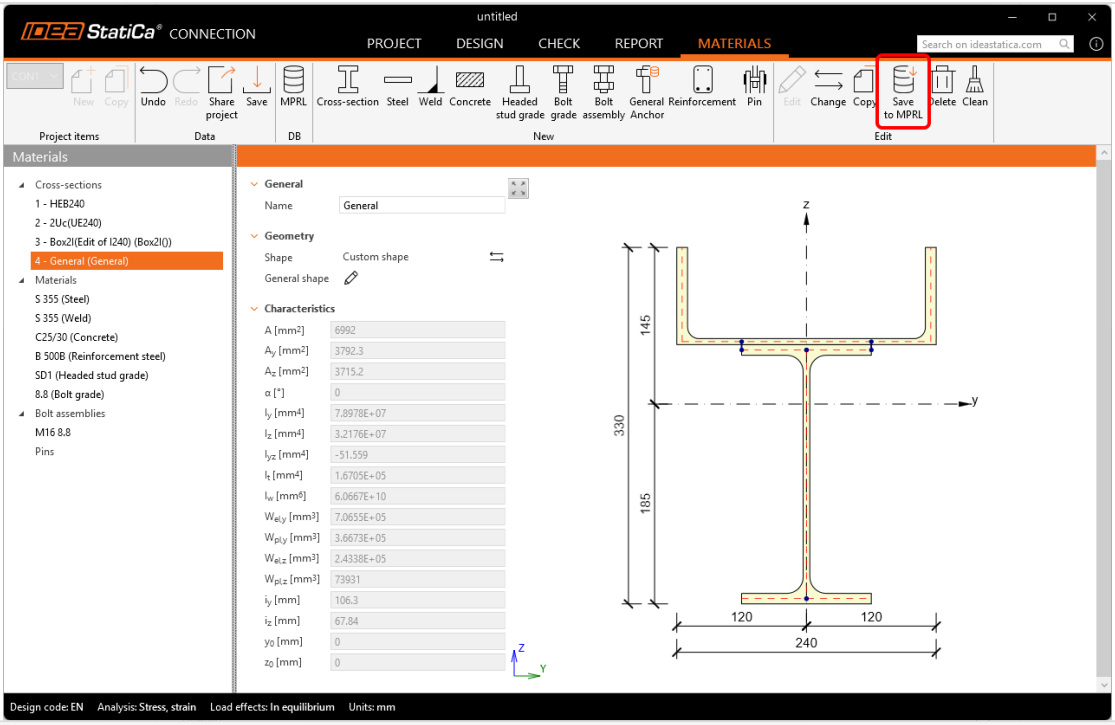

# Saving a general cross-section into the MPRL

It is also possible to save the custom cross-section created in the **Cross-section editor** into your MPRL (since version 25.0.3).

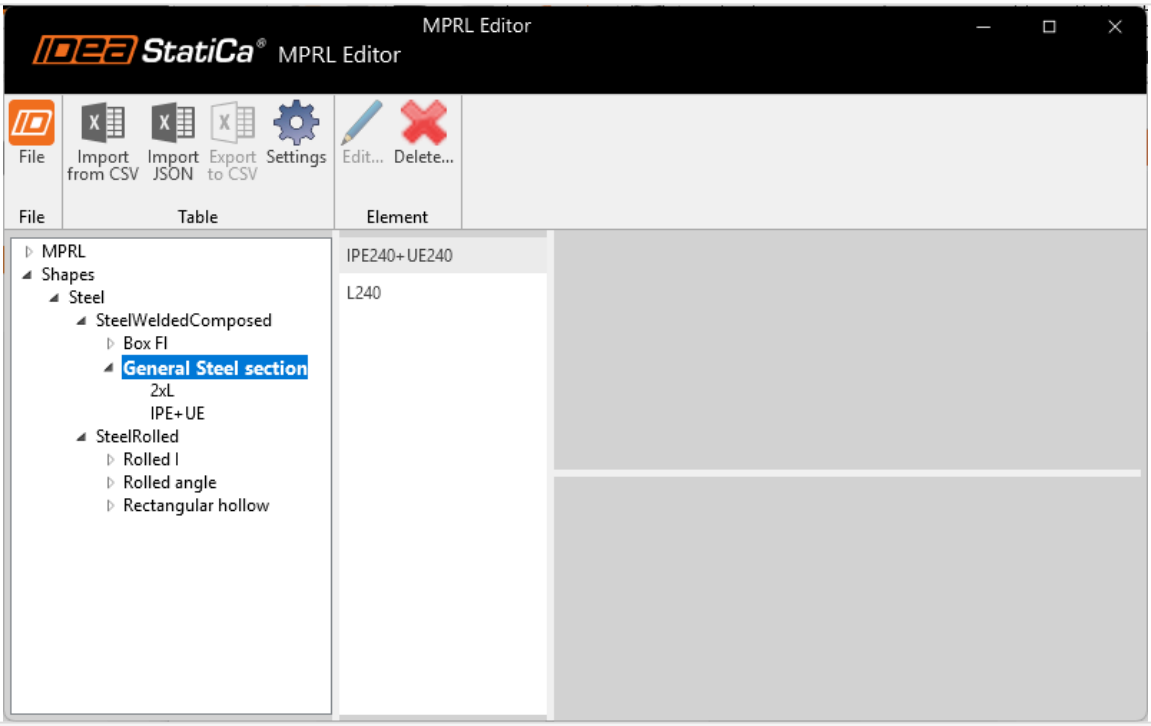
The general cross-section can be defined in the **Editor**, which can be opened from the cross-section library.




In Materials, save the general section into your MPRL.




These sections are saved together in the dedicated place in the MPRL under welded sections.



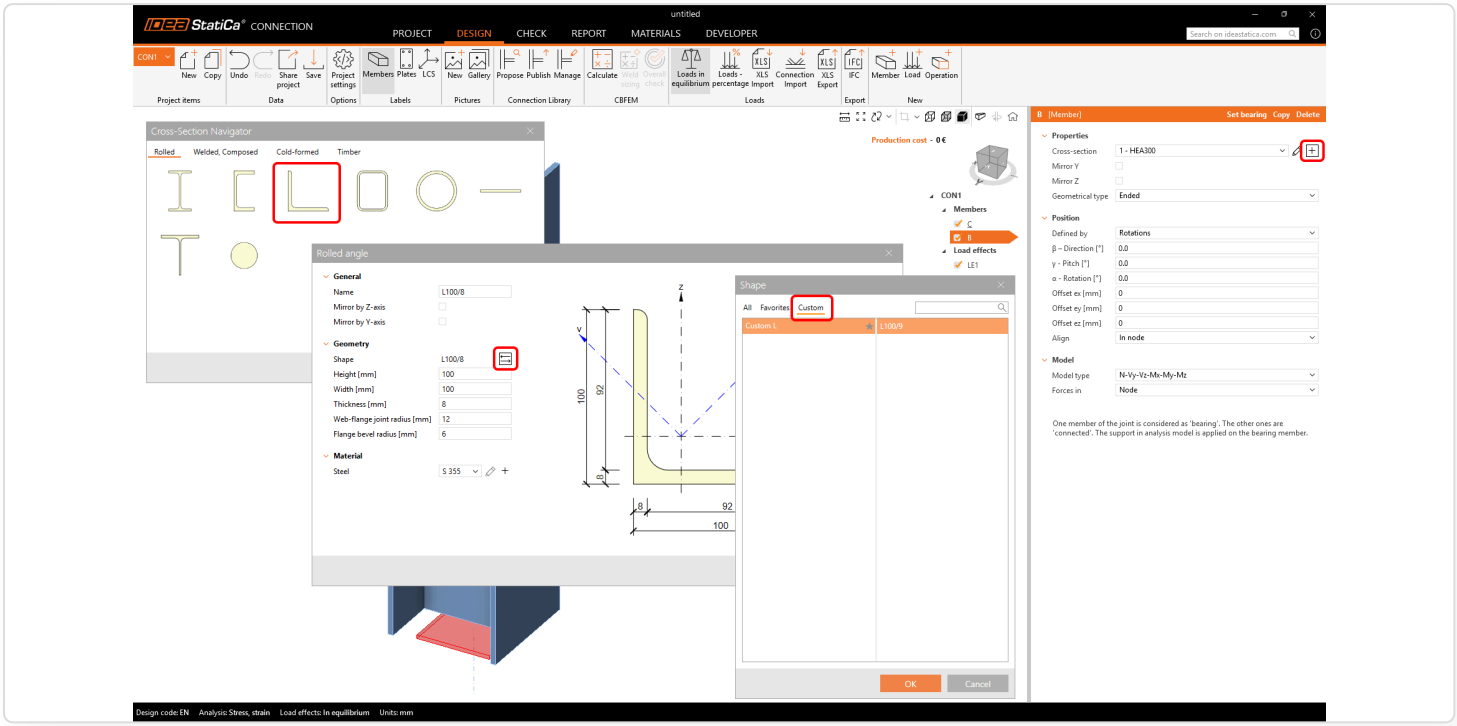

## Limitations

- Saving the compounded (paired) sections: 2I, 2Uc, 2Uo, 2Lt, 2Lu, Box 2i, Box 2U, Box 2L, Box 4L is not available at the moment.

  The suggested workaround is to build these as a general steel cross-section.
- Saving concrete sections is not available at the moment.

# Loading a section from the MPRL

In a new Connection or Checkbot project, you can load the previously saved custom cross-sections from the MPRL using the standard cross-section library.




The shape selector includes a **Custom** section, where only custom shapes stored in the MPRL under certain types are available. Note that custom tables are also available in All and can be marked as Favorites.

# Exporting and importing cross-sections from/to the MPRL

The direct export/import of cross-sections from/to the MPRL is not currently available.

To share the entire custom shapes library, you can share the following file, which can be copied to another computer (to the same location).

%AppData%\IDEA_RS\user_mprl_v2.sqlite

*Released in IDEA StatiCa version 24.1.4, updated in version 25.0.3.*

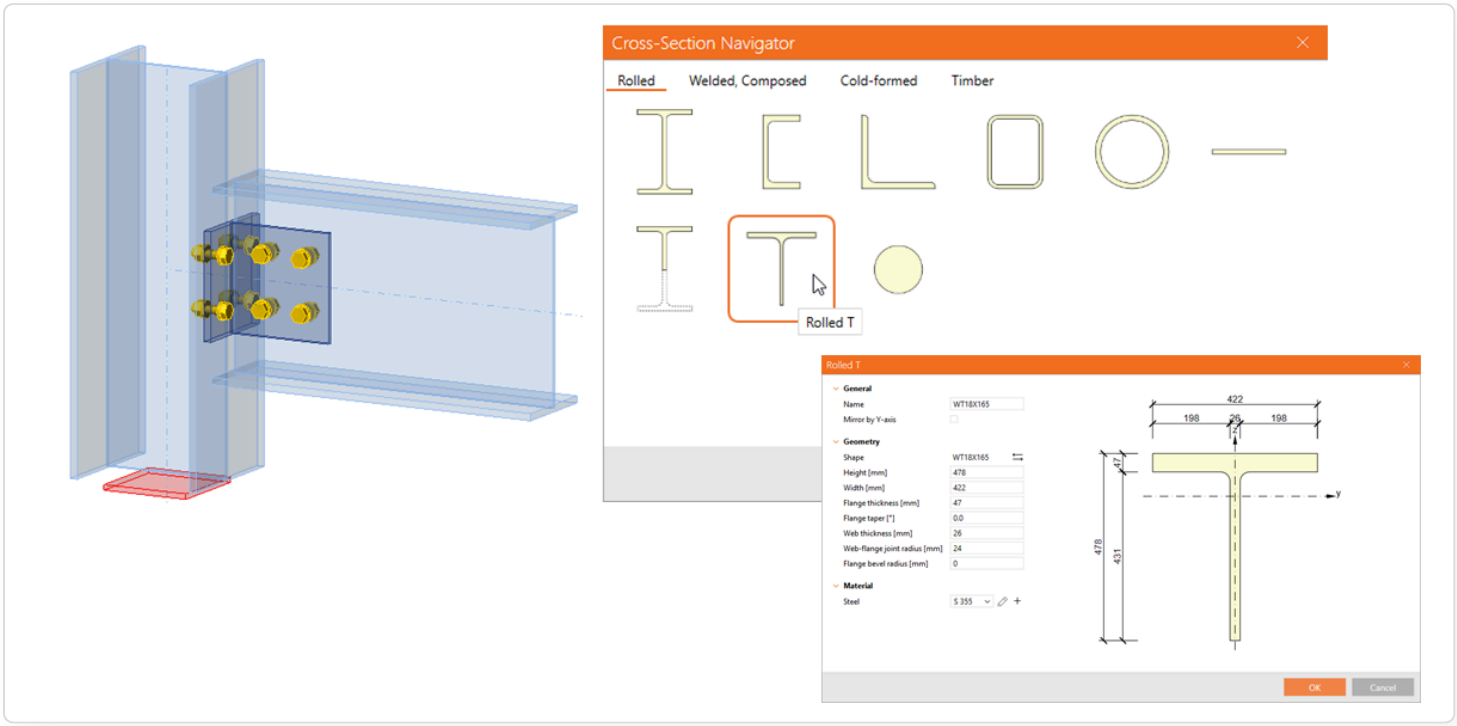
Regional improvements

# Regional improvements in 25.1

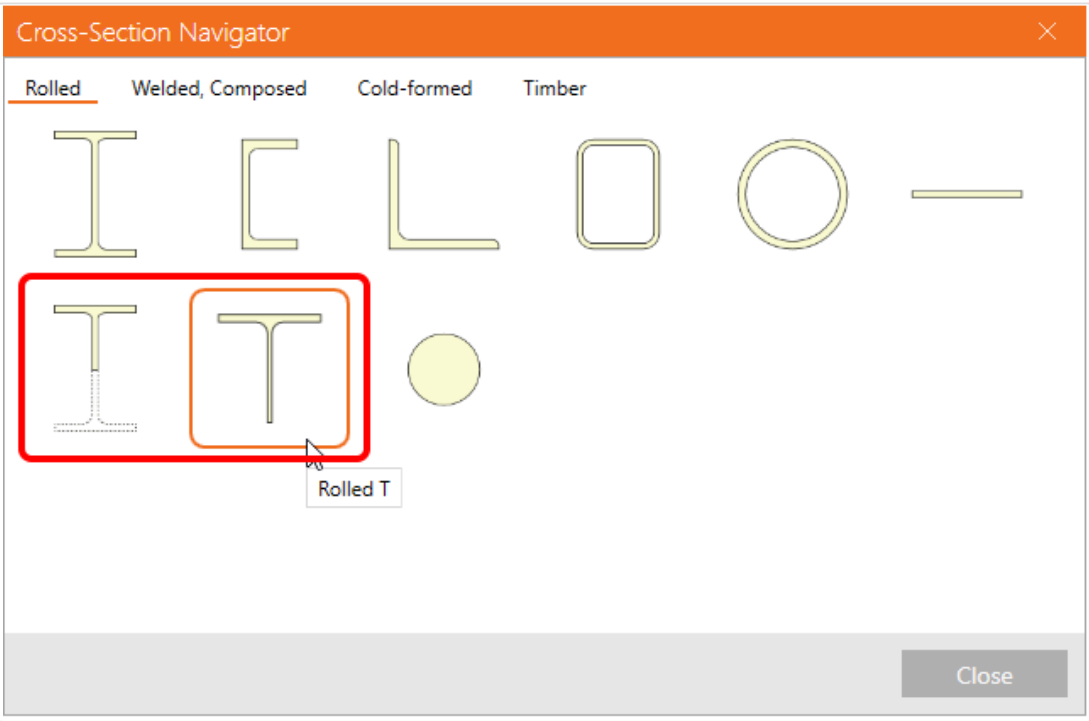
IDEA StatiCa 25.1's release extends language support (Croatian, Ukrainian, Bulgarian, traditional Chinese), adds Indian Standard anchorage checks, introduces WT sections from AISC v16.0, and implements ACI 318-25, enhancing code compliance, usability, and design efficiency across global markets.
The major regional improvements of version 25.1 are:

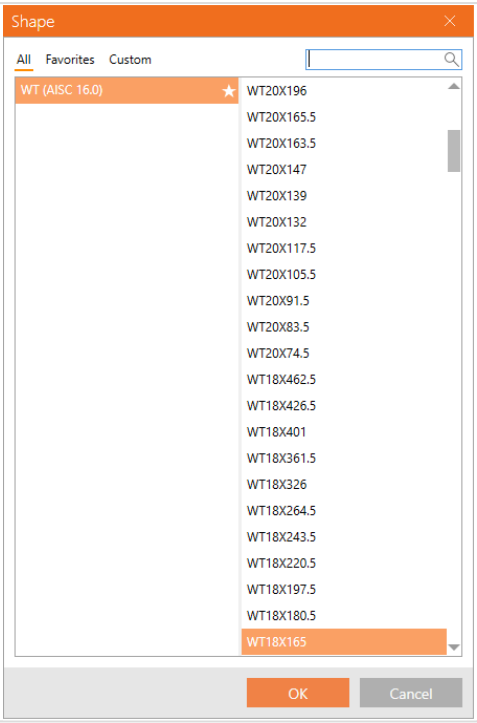
WT sections added to the library (AISC)

# WT sections added to the library (AISC)

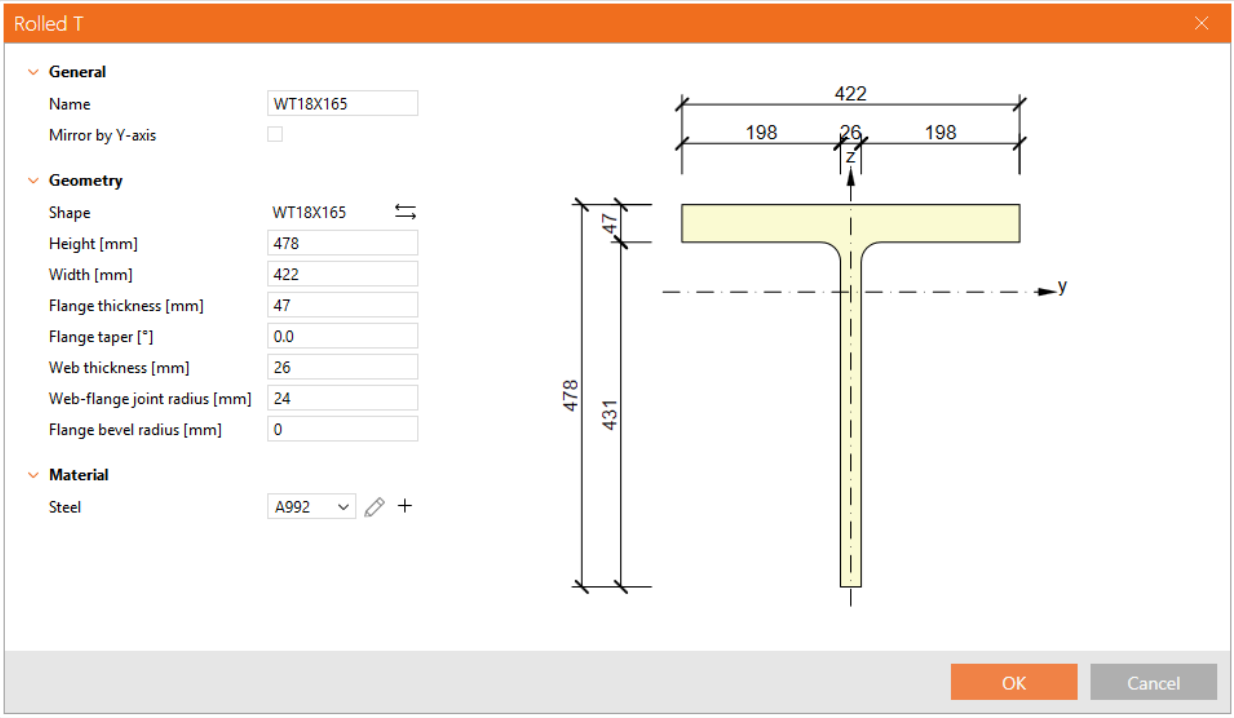



IDEA StatiCa Connection introduces **WT sections from the AISC v16.0 database**, significantly improving workflows for engineers designing with rolled T profiles. A new cross-section type for rolled T sections has been added in the Cross-Section Navigator, clearly separating two groups: **T sections cut from I-shapes** (e.g., ½ W40x277) and rolled **WT sections** directly available from the database.

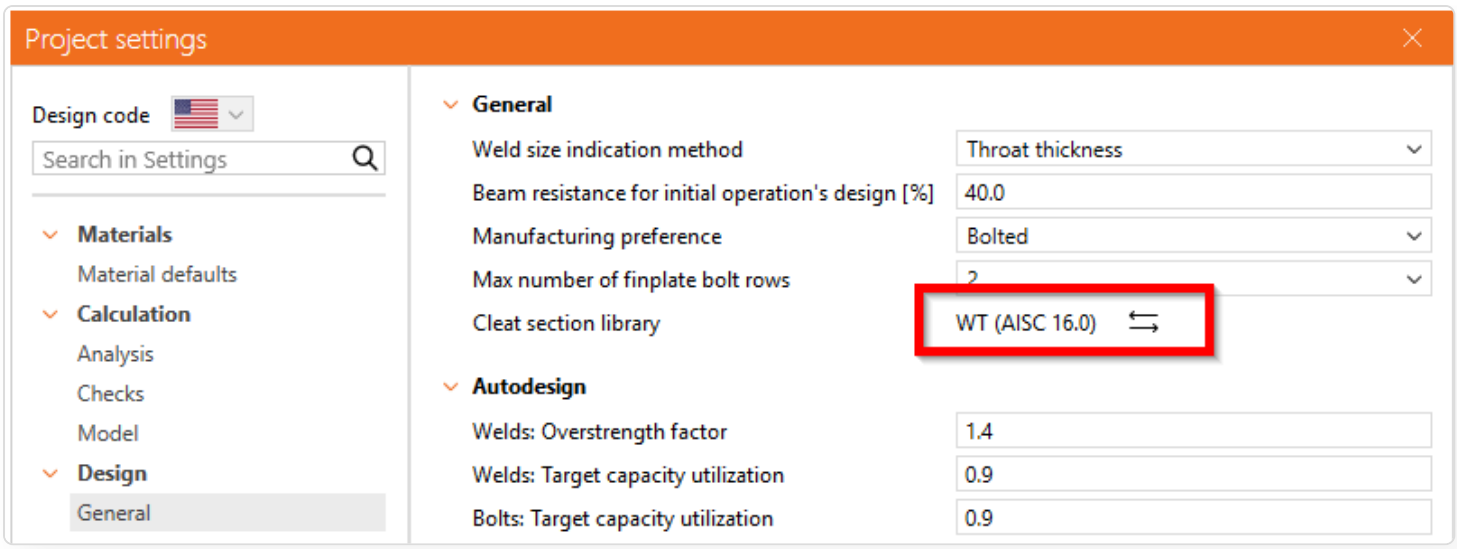





The new dialog for rolled T sections enables users to select a WT shape, adjust geometry if needed, rename the section, and assign materials, removing the need to manually derive WT profiles from W sections.



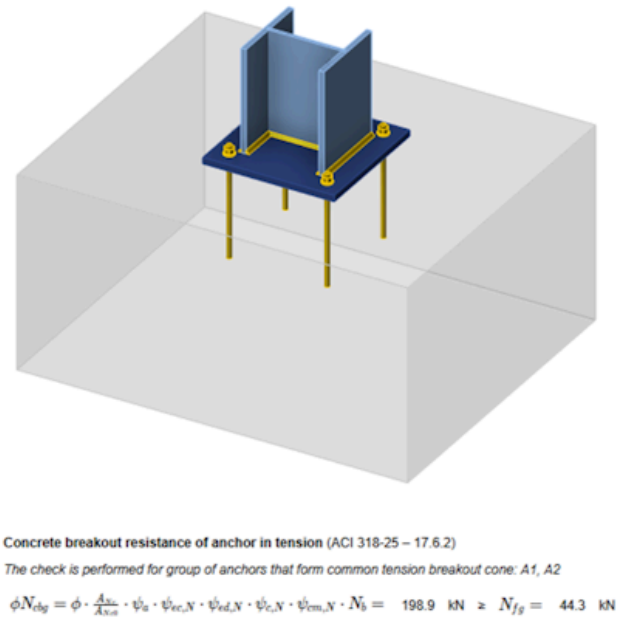

To enhance usability in connection design, the Cleat (clip angle) cross-section library is now filtered to show only applicable L-type and T-type profiles. WT sections can be chosen directly from the database and set as the default option in Project settings under **Design – General: Cleat Section Library**.




This eliminates manual workarounds, ensures consistency with American labeling practices, and improves accuracy in calculating dimensions and weights, particularly for clip angle designs.

# Implementation of ACI 318-25 (AISC)

The **ACI 318-25 standard** has been implemented in IDEA StatiCa Connection and Member applications, extending the range of available concrete design codes for users working under AISC frameworks.

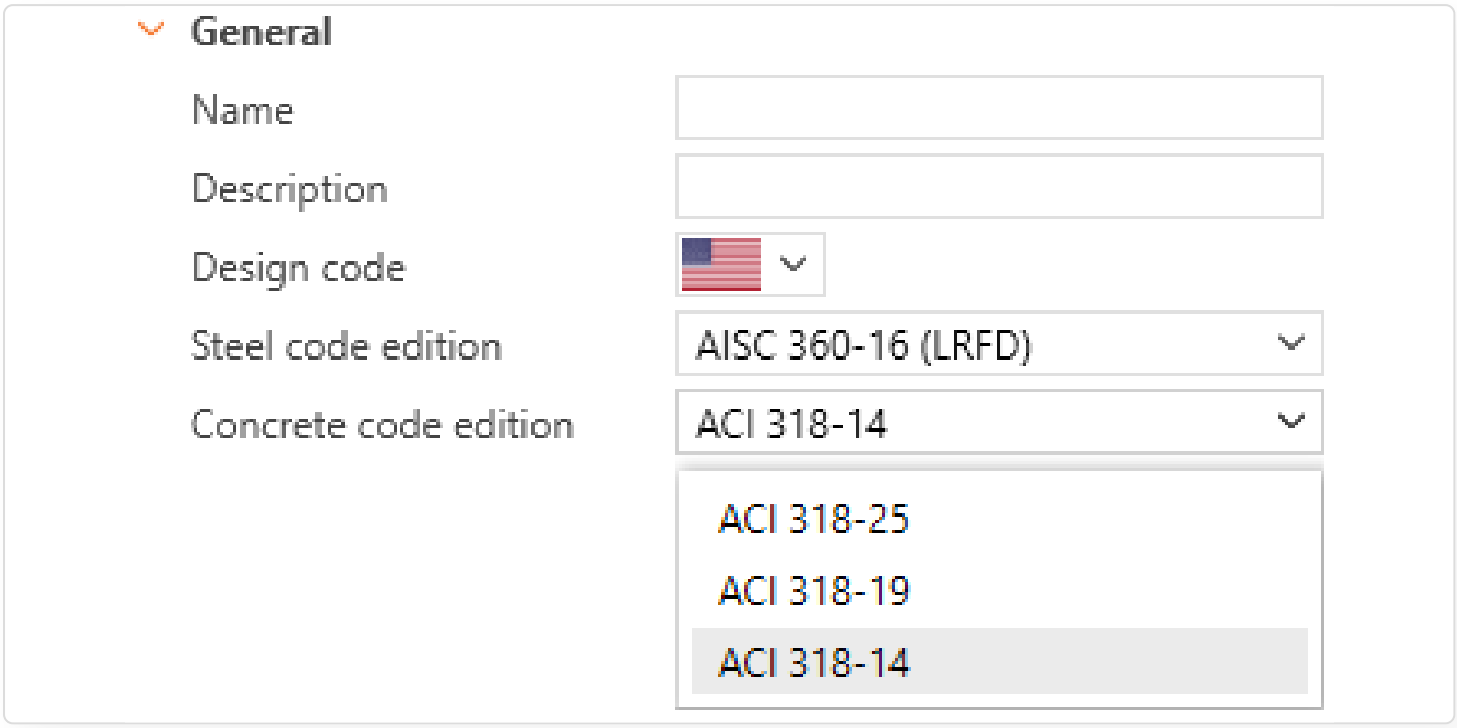




Code versions can be defined during project creation in the wizard. Currently implemented codes for AISC users:

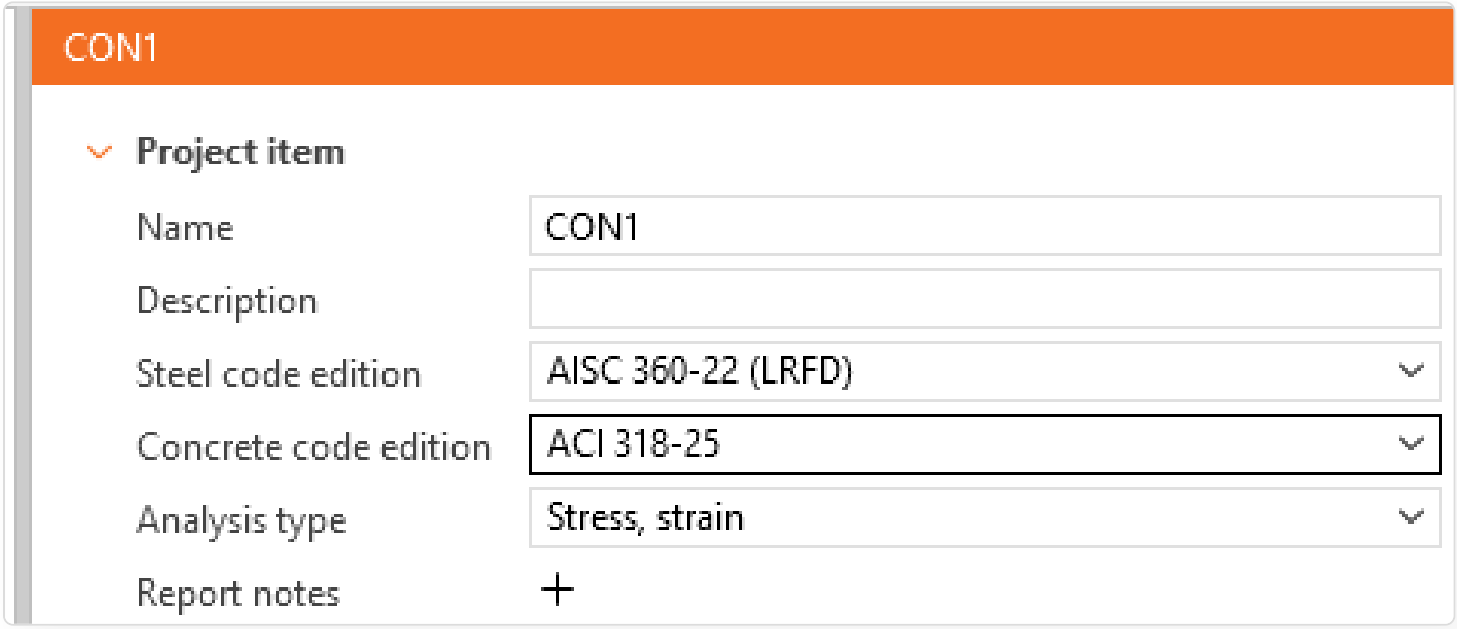
- AISC
  - AISC 360-10 / 360-16 / 360-22 (LRFD)
  - AISC 360-10 / 360-16 / 360-22 (ASD)
- ACI:

- ACI 318-14 / ACI 318-19 / ACI 318-25

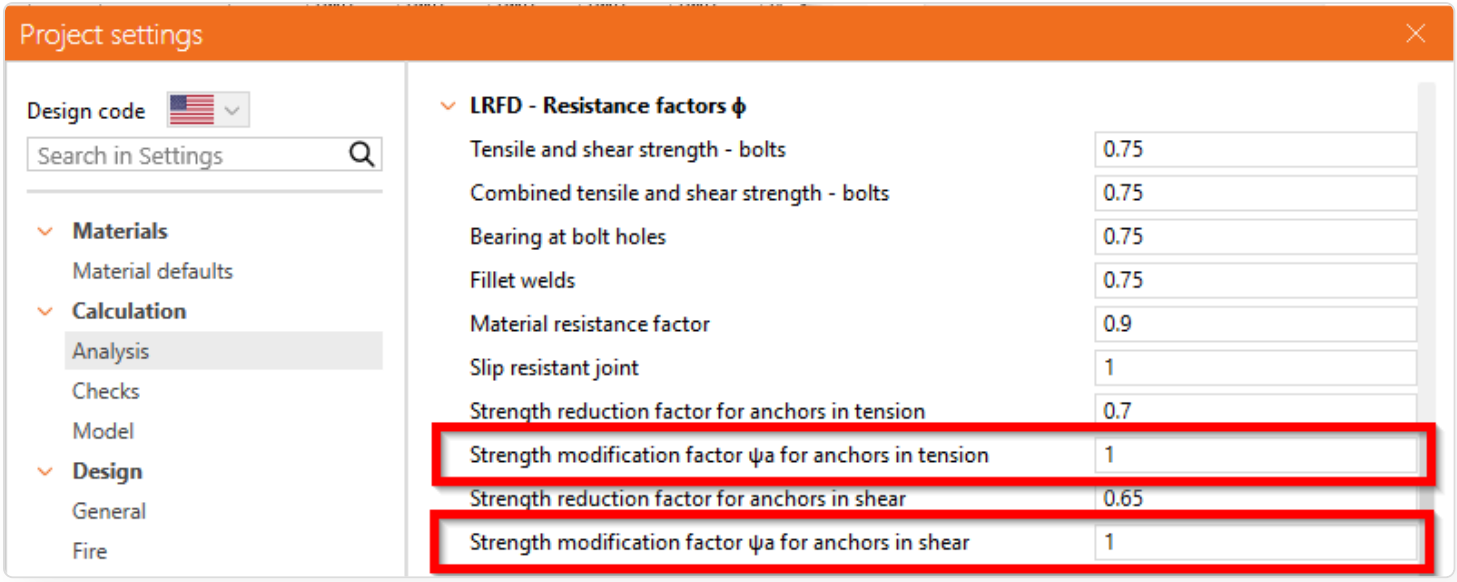
This flexibility allows users to combine different AISC and ACI versions in a single project according to specific design or contractual requirements.



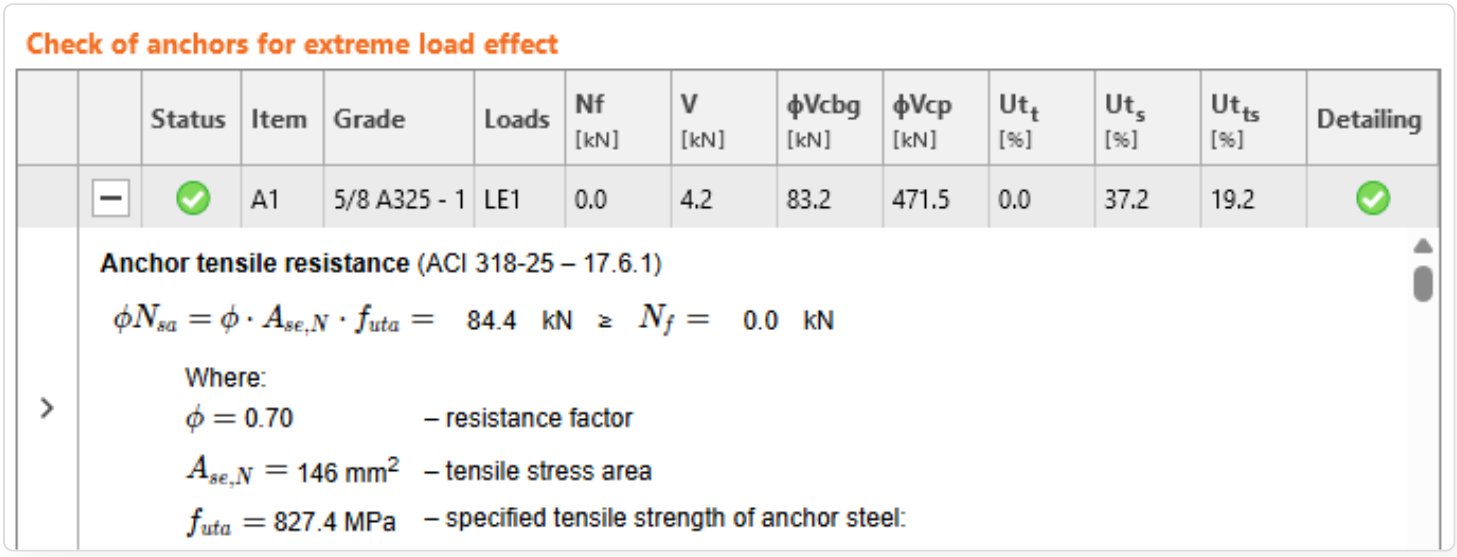

The steel/concrete code edition can also be adjusted later in the Project Info dialog. The same options are available in Checkbot.




The update incorporates new factors in Project settings and modifies several checks in accordance with ACI 318-25 provisions, as documented in the corresponding U.S. code references.

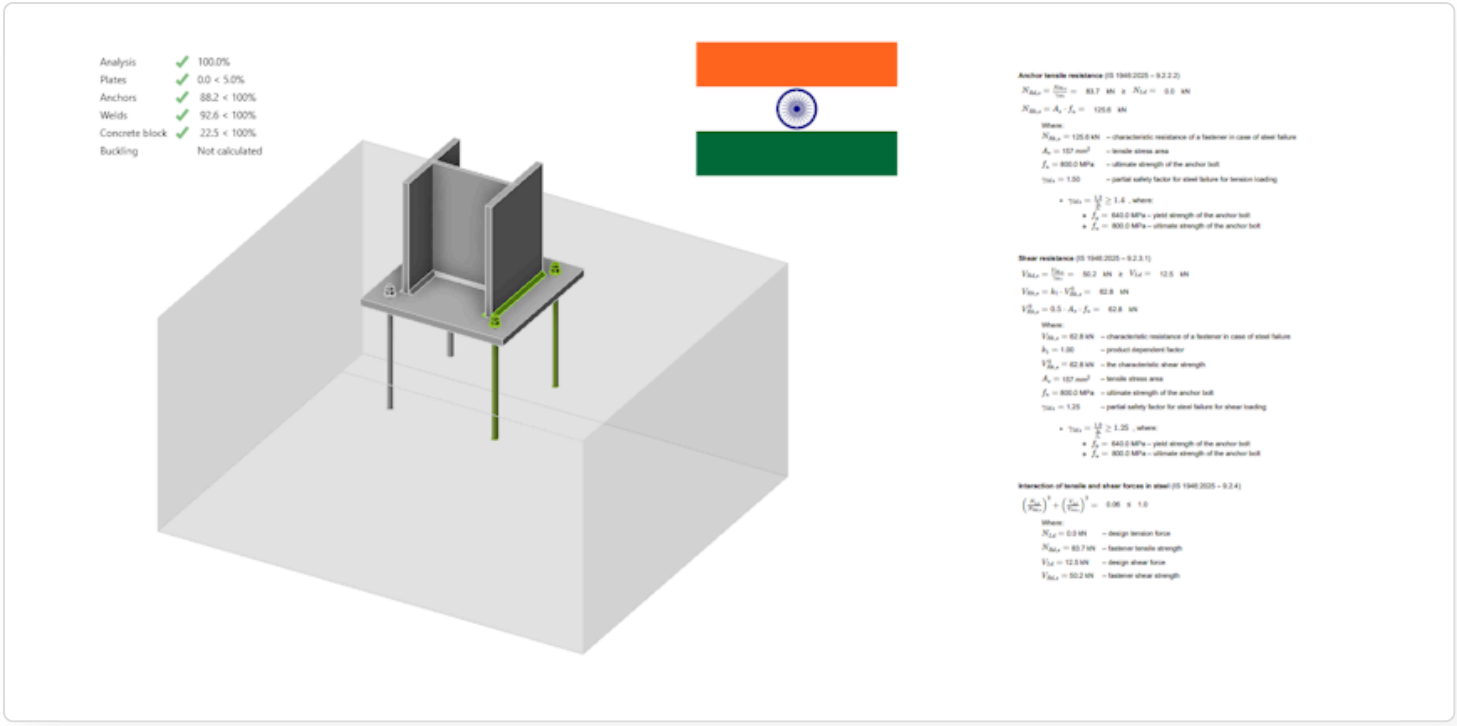



Code references selected by the user are automatically reflected in generated reports, ensuring traceability and compliance with the latest standards.

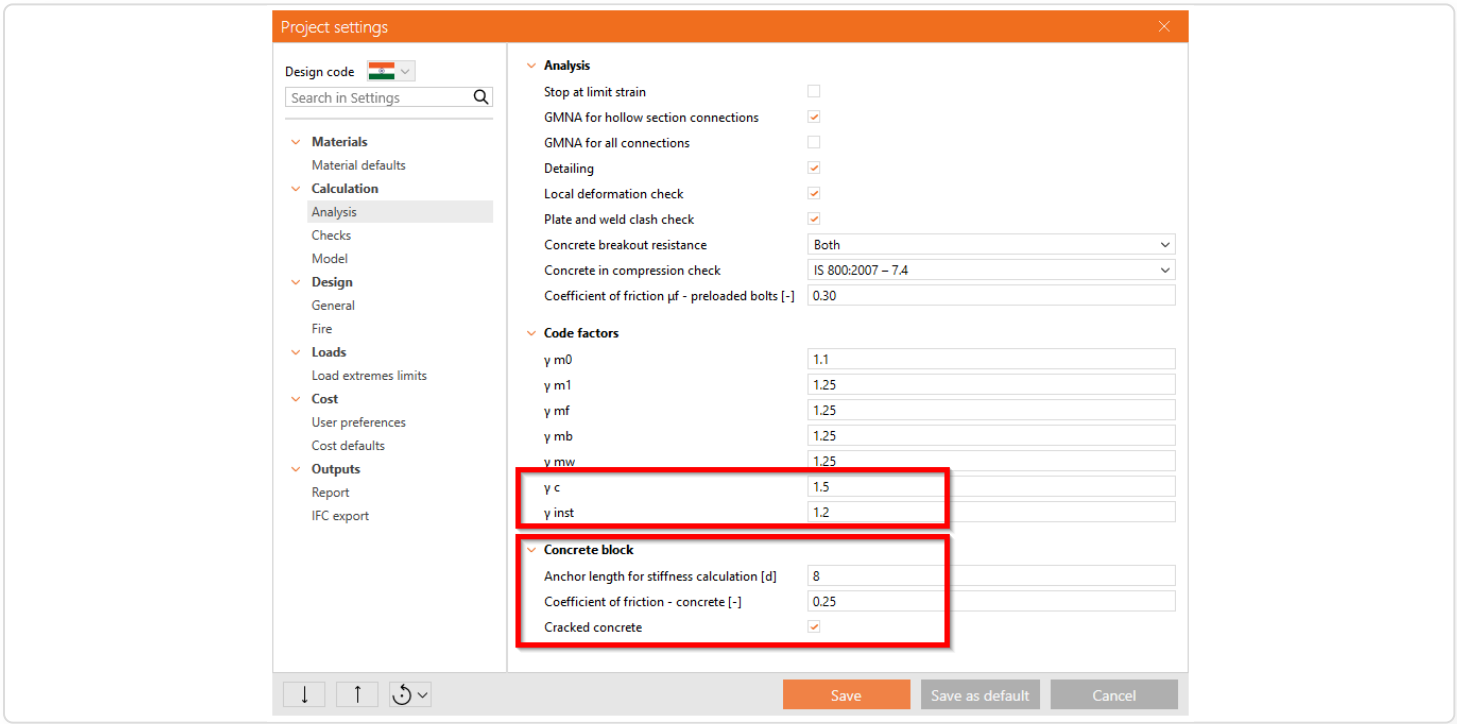



# Anchorage checks in the Indian Standard (IS)

IDEA StatiCa Connection introduces **anchorage checks according to the Indian Standard**. This enhancement implements the new IS 1946 (Part 2):2025 and IS 1946 (Part 3):2025 provisions, enabling designers to perform anchorage verification directly within the software.




Supported anchor types include **post-installed threaded rods** and **cast-in washer plate anchors**, with relevant parameters integrated into the Project settings for consistent application across designs. This closes a long-standing gap where only steel checks were available under the Indian code, requiring users to depend on external tools.

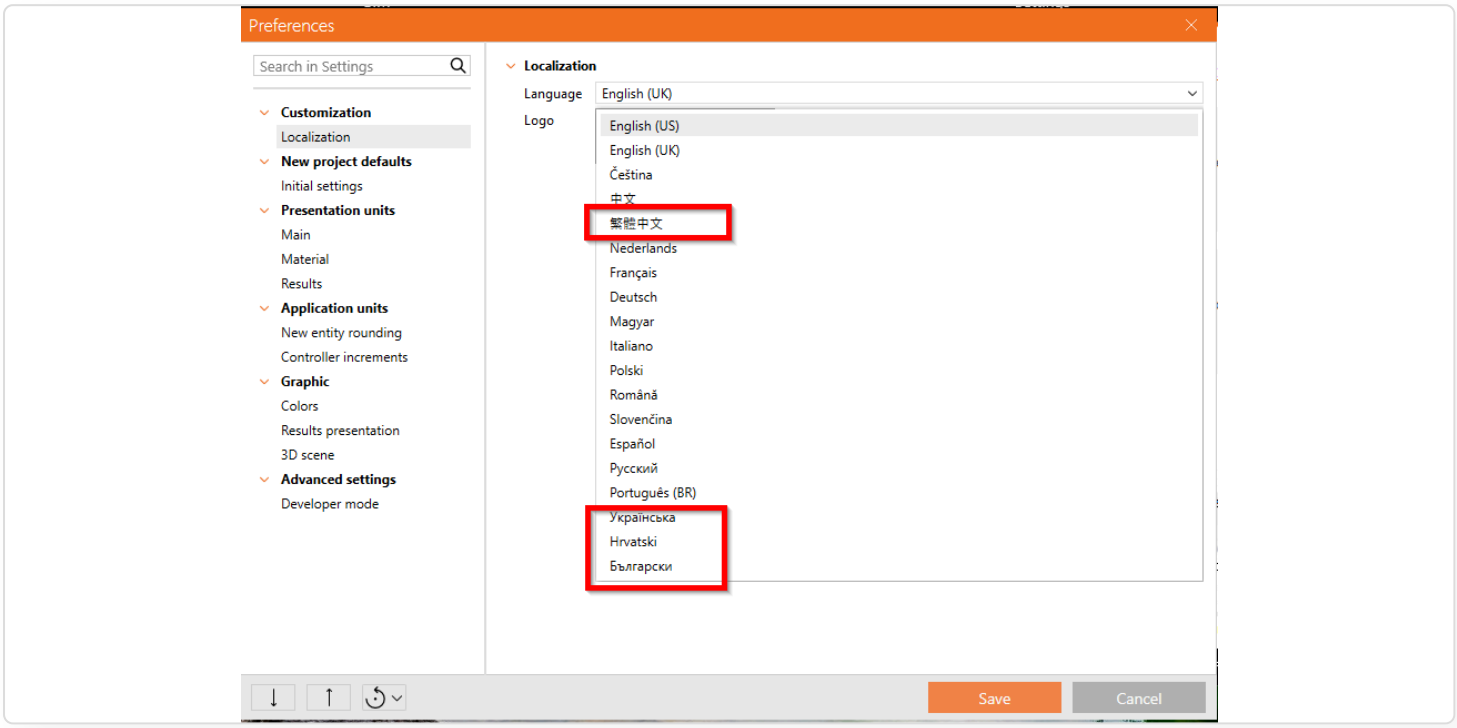



Certain checks **remain outside the current implementation scope**, including pull-out failure, combined pull-out and concrete failure, splitting, concrete blow-out, and concrete pry-out failures. By focusing on the most commonly applied anchorage scenarios, this release provides a reliable and code-compliant solution for connection designers while laying the groundwork for broader coverage in future updates.

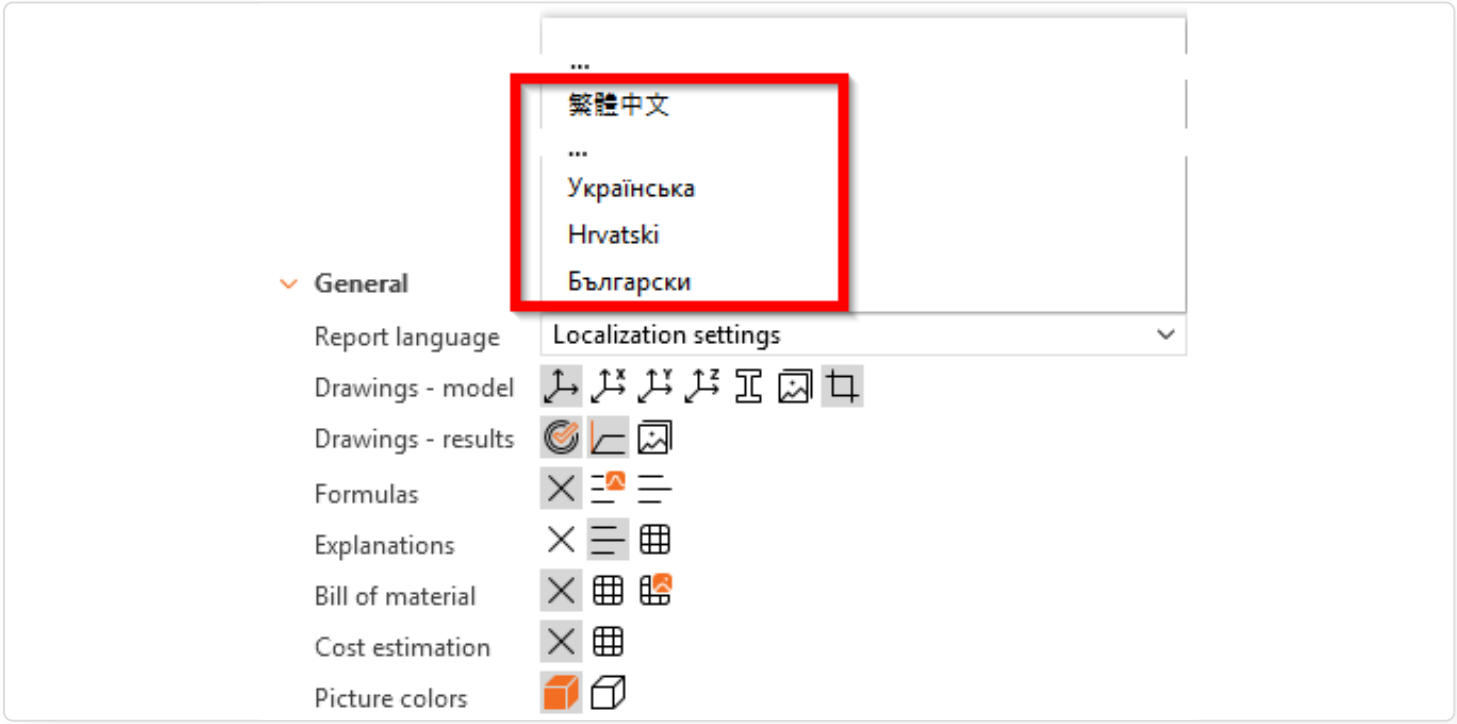
# New languages – Bulgarian, Croatian, Ukrainian, Traditional Chinese

IDEA StatiCa extends the language support to **Croatian, Ukrainian, Bulgarian, and Traditional Chinese**. Structural engineers and connection designers in these regions can now work directly in their native language, improving the user experience when designing projects and generating reports.

The language can be set in the **Preferences** under **Localization**.




The language can be changed for the report generation only, in case the user needs to create a report in a language different from the application language.

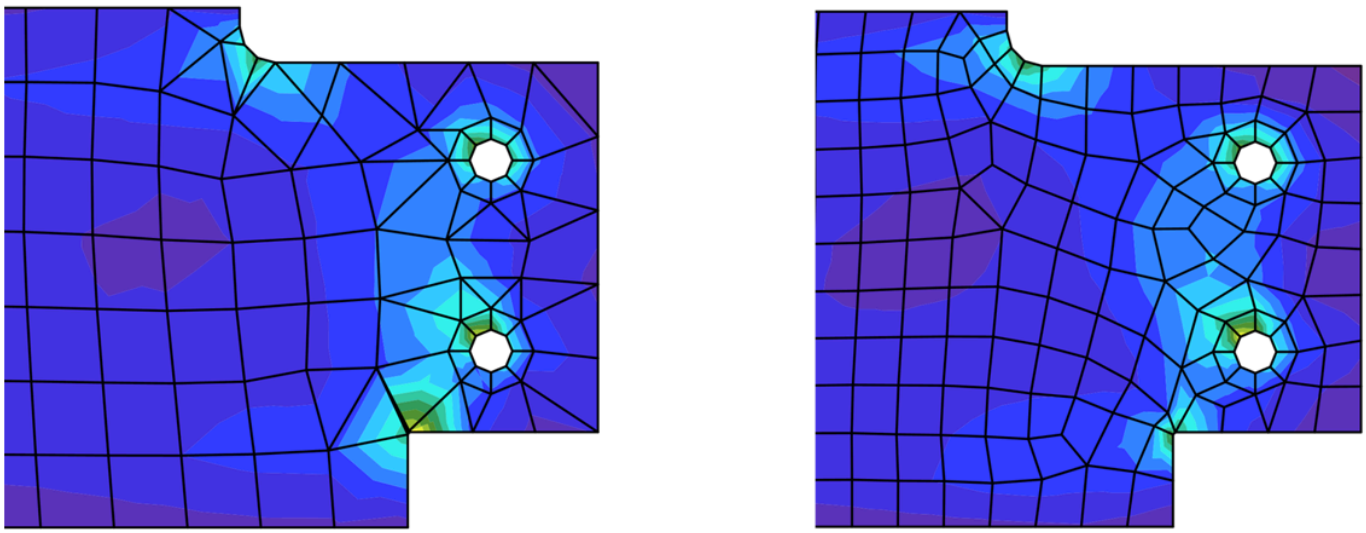



Accuracy of the results, safety of the project

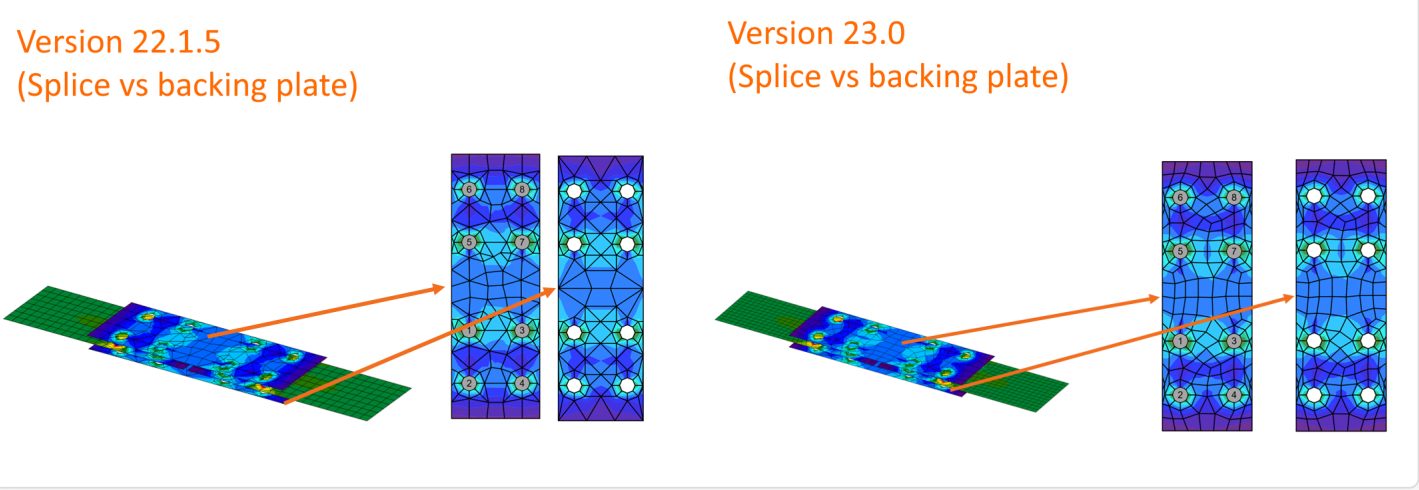
# Smooth results with precise meshing

As a result of the continuous research of our solver team, we are always bringing big improvements in mesh generation for the finite element model in the Connection and Member applications. The mesher now gives smoother and, therefore, more precise results.

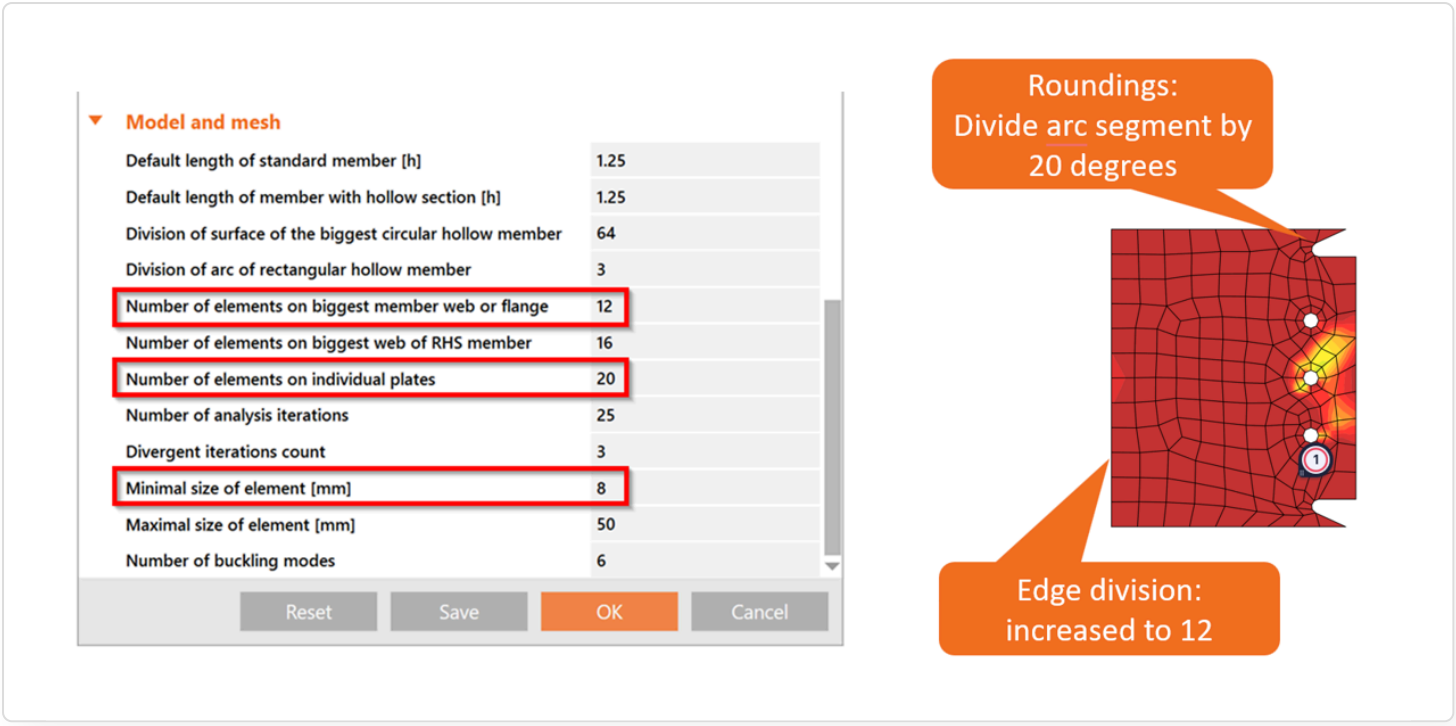
Using the mesher CM2 MeshTools SDK, which means applying the latest meshing algorithms, provides a smooth mesh structure.



We have also programmed the newest mesh generation rules to ensure better distribution in all plates. Connections composed of splice and backing plates, intermediate stiffening members, or stiffening plates used as custom end plates are tuned for even higher precision of analysis and match better with other parts of the connection.



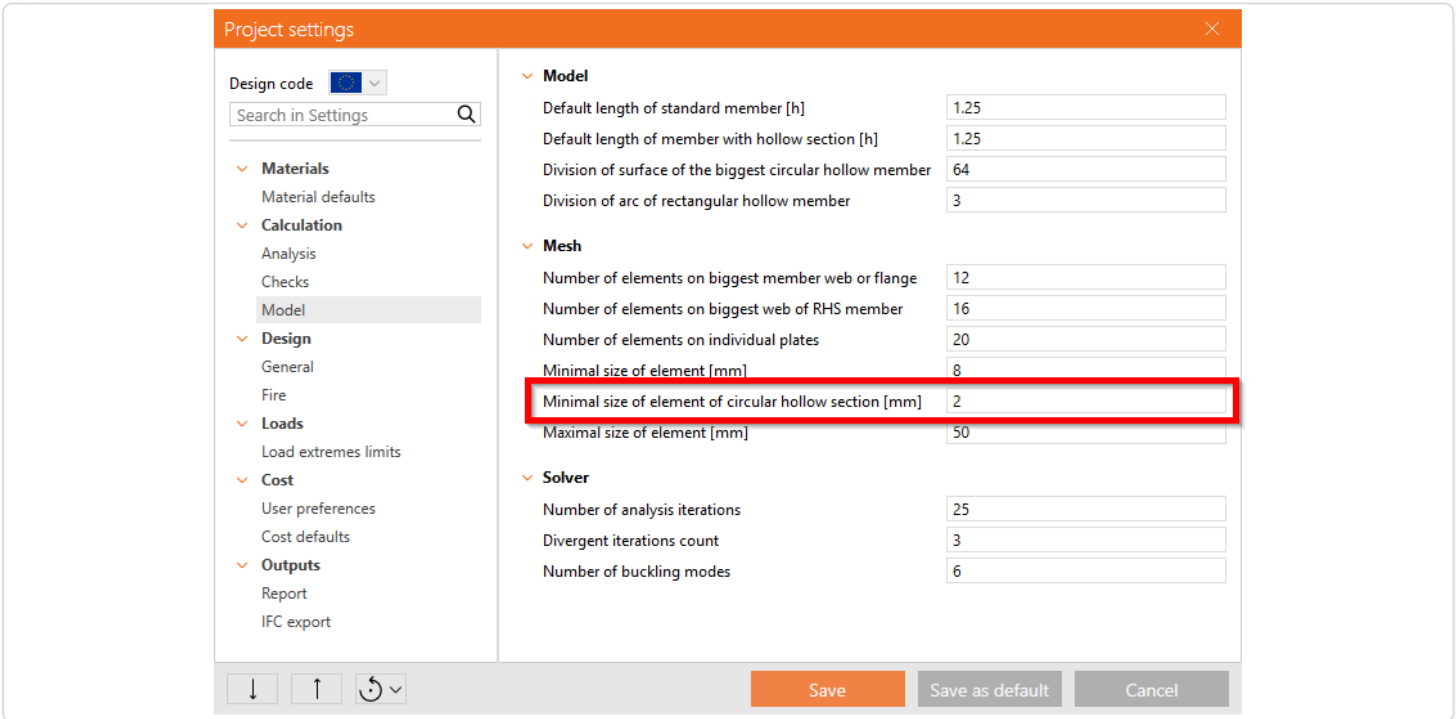

Plus, the default values for the mesh settings, such as the minimal size of elements and the number of elements on plates, are refined in the Code setup.



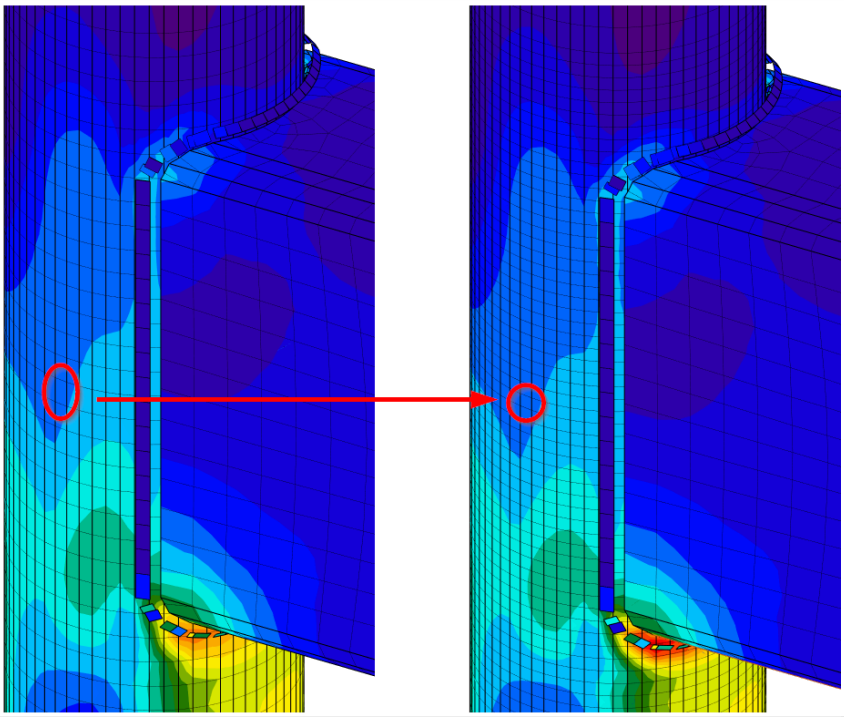

*Released in IDEA StatiCa version 23.0.*

# Changes in the mesh of CHS members

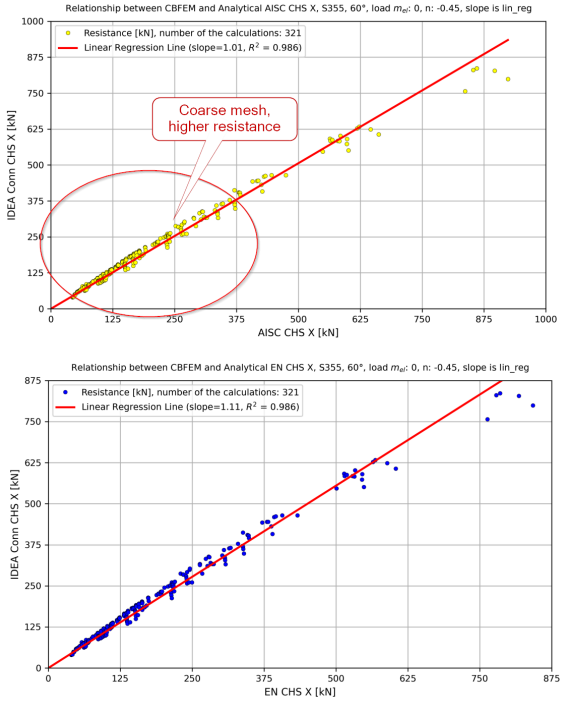
The calculation model for **circular hollow sections (CHS)** is refined to improve numerical accuracy and result consistency. The setting for the **minimal finite element size** is available in Project settings, allowing users to enforce square meshing, even for small tube diameters (D < 160 mm).




This adjustment removes any limitations where smaller diameters generated rectangular elements, which could lead to slightly unconservative results.



By standardizing the mesh pattern across all CHS geometries, this approach delivers **more stable and realistic stress and deformation outputs**. Smaller tubes now exhibit behavior consistent with reference curves, ensuring that the predicted resistances remain within safe, code-compliant limits.




*Released in IDEA StatiCa version 25.1.*

# Updated CBFEM solver

The following article details the improvements to the CBFEM solver for IDEA StatiCa Connection and Member applications. The cross-section deformations of hollow sections and numerical iterations of the calculation are covered.

The computational solver is the heart of CBFEM.

We are constantly improving and refining this numerical method. In version 21, we completed several years of development in the area of geometric nonlinearity and large deformations, including initial imperfections (GMNIA). This allowed us to move the Member application from beta to live.

Wherever the member assessments in their 3D FEA program are not enough for our users, Member is now available to them. The elements are completely modeled including the end connections now, which is beneficial for the designer and frees them from the necessity of estimating the end connection effects on the load capacity and the stability loss. Member can deal with the influence of transverse and longitudinal stiffeners, openings, changes in the cross-section height, but also with the influence of connected secondary members. The effect of torsion and warping is not a problem for this method.

At the same time, the GMNIA method significantly serves to refine the connection models in the Connection app. It is now used in all hollow cross-section connections – circular and rectangular pipes. According to standards, pipe joints are assessed according to empirical formulas, the validity of which is limited to certain geometric conditions. The compliance of these formulas with reality is quite questionable, especially at the edges of the validity intervals. The improved GMNIA method shows very good compliance with standard formulas, especially in the middle parts of the validity intervals. In the marginal ones, it was thoroughly validated against higher mathematical models (ABAQUS) and physical experiments.

Every change in the numerical model necessarily brings a change in the results. This is also reflected in version 21, and in the vast majority of cases, it differs within percentage units.

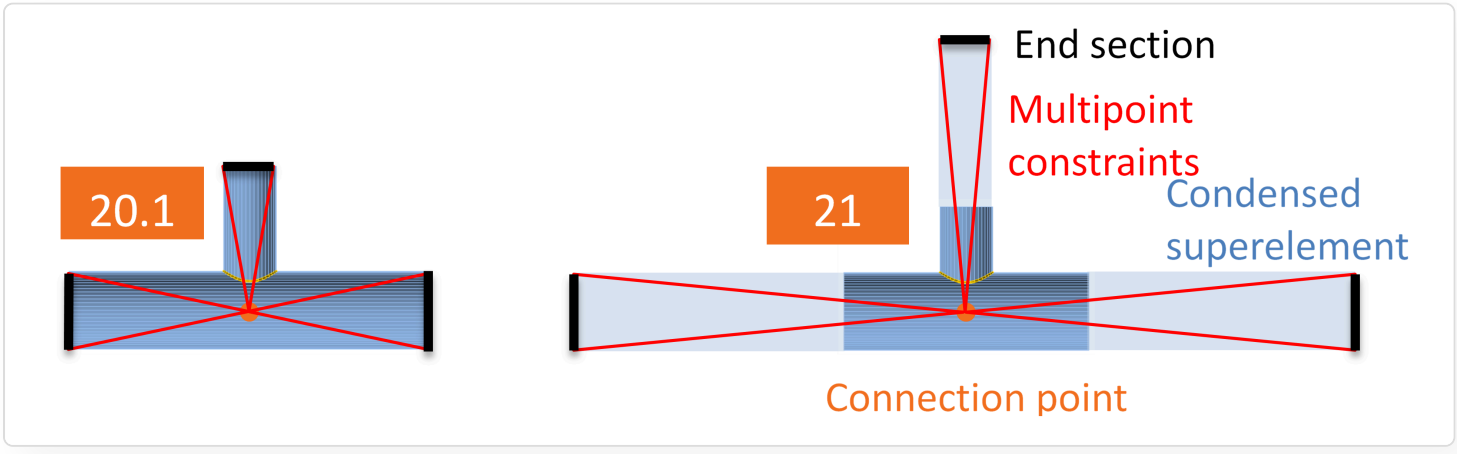
Moreover, the model improvement also manifests in significant acceleration of the calculation, in the range of up to 30%.

*Released in IDEA StatiCa version 21.0.*

# GMNIA solver

The solver used for the hollow section joints in IDEA StatiCa Connection and also for GMNIA in IDEA StatiCa Member was improved. Now, it contains a nonlinear formulation of not only shell elements (which was already present in previous versions) but also of links and constraints used in components, such as bolts or welds.

The model of the connection is greatly improved by the insertion of a condensed superelement. This element is added behind the member end and has the same properties as the elastic shell model of the member. It is just one element but it allows any elastic deformation and stress to develop in the member ends. Because of this, the part of the member consisting of shell elements may be shorter and still even improve the model behavior.

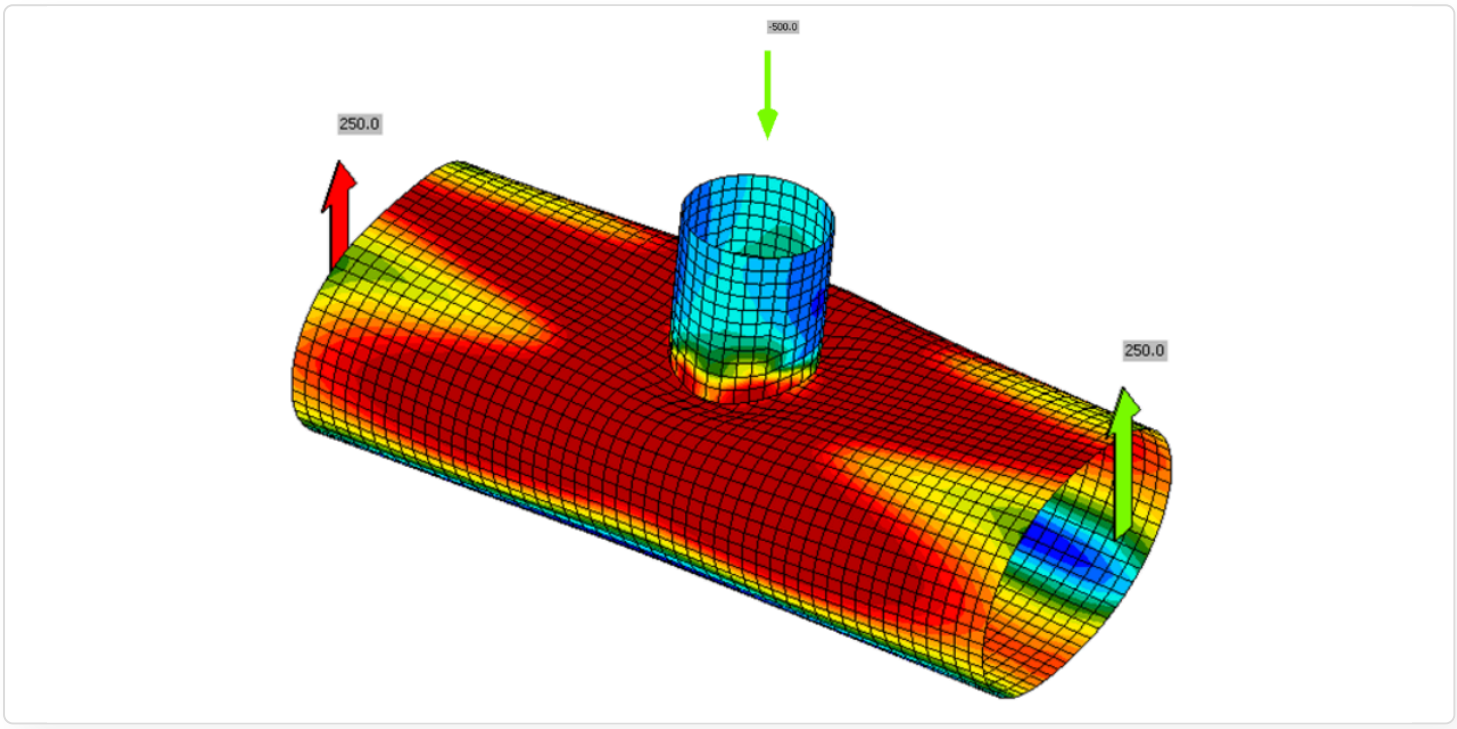


*A condensed superelement has been added at member ends*

This allows shortening the part of the model where shell elements are used and still increases the precision of the model. This results in fewer elements in the model, leading to faster calculation time and visualization of the results.

## Cross-section deformations at the end of the shell model

This is the main reason the change was made. The cross-section may deform at the ends of a model consisting of shell elements. Joints of hollow sections require relatively long members – up to 10 times cross-section diameter. By introducing the condensed superelement behind the part of the model consisting of shell elements, the calculation is much faster with the same precision.

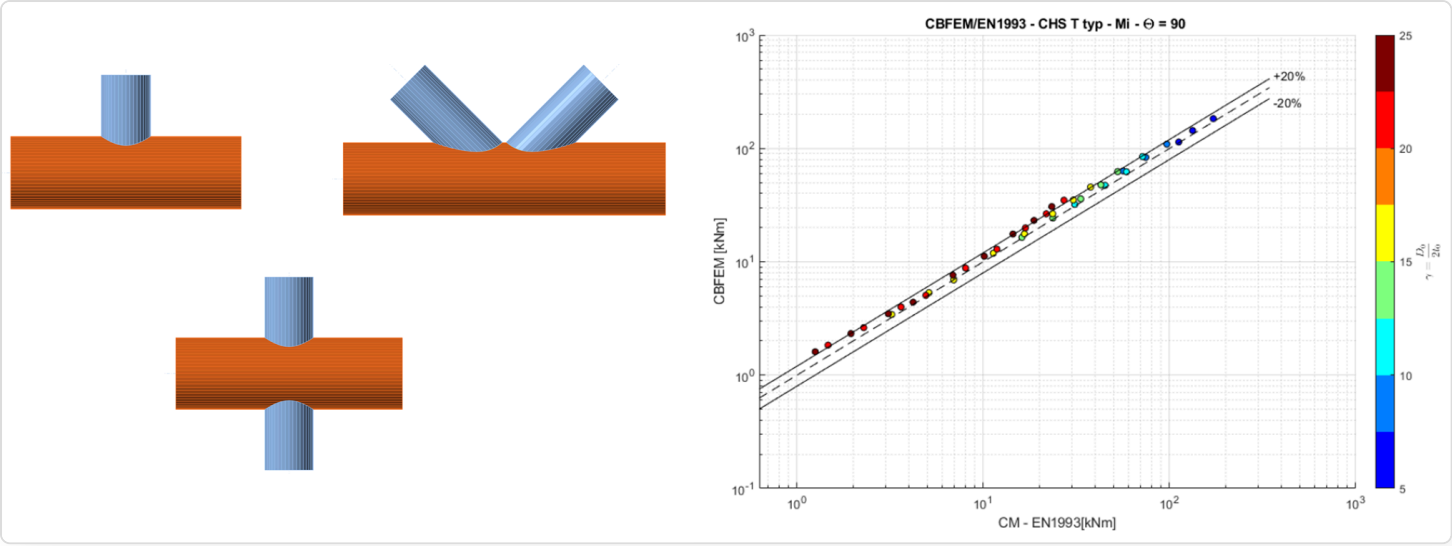



*Released in IDEA StatiCa version 21.0.*

## Shell bending resistance reduced for hollow sections (imperfections)

Load resistances of hollow section joints in the codes are determined by the Failure Mode Method, which uses curve-fitting models determined from experiments and advanced numerical models. The real structure contains initial imperfections and residual stresses, which are not captured by shell models in IDEA StatiCa Connection. To achieve closer compliance with the results of codes, the influence of residual stress and initial imperfections was introduced in IDEA StatiCa models by reducing the bending resistance of shells of hollow sections with a high $D/(2t)$ ratio.

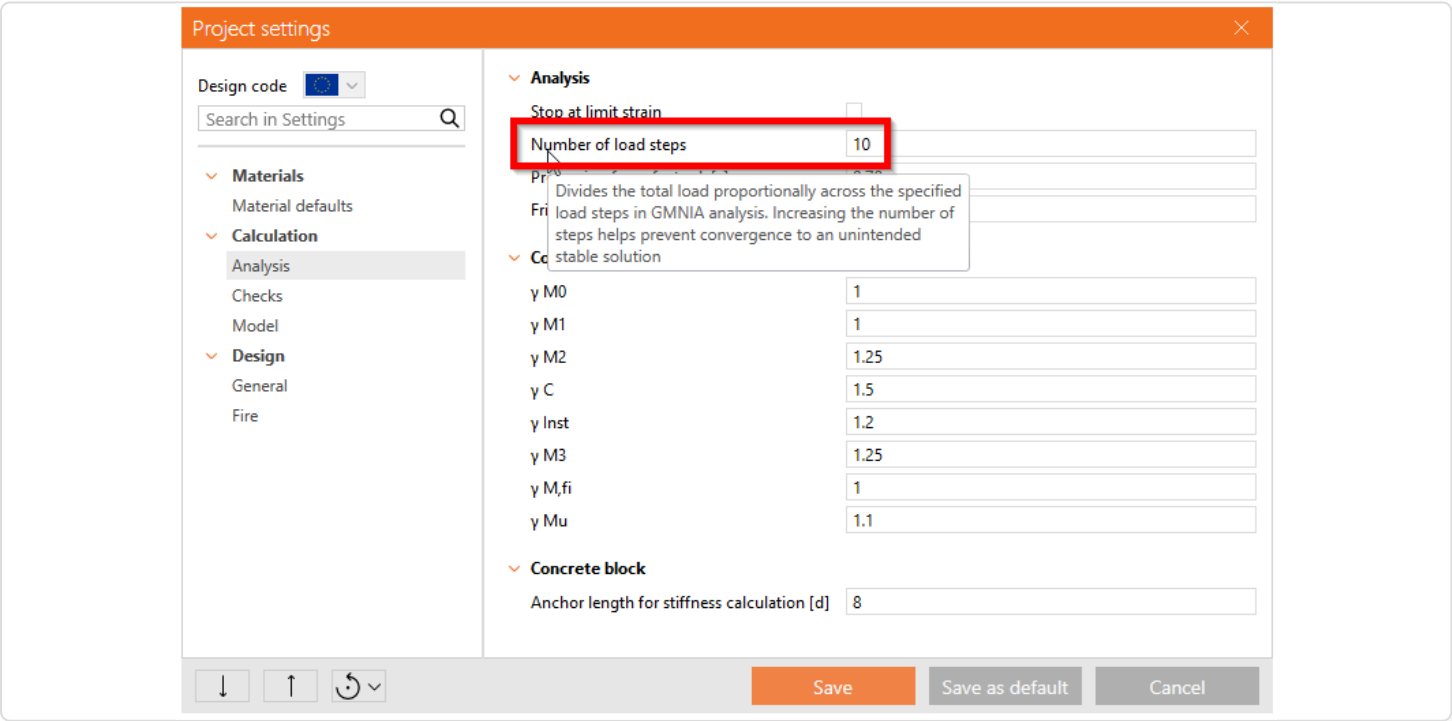
These combined changes allowed us to achieve close compliance with the results of the Failure Mode Method contained in design codes.




*Released in IDEA StatiCa version 21.0.*

# Stepped calculation for GMNIA in Member

The parameter, **Number of load steps for GMNIA analysis**, has been added to Project settings in Member to improve stability and safety in nonlinear member calculations. The total load is divided into incremental steps (default 10), reducing the risk of the solver converging to an unintended stable branch of the load–deformation curve. This stepped approach ensures smoother load progression, more accurate equilibrium tracking, and safer prediction of ultimate resistance in geometrically and materially nonlinear analyses.




*Released in IDEA StatiCa version 25.1.*

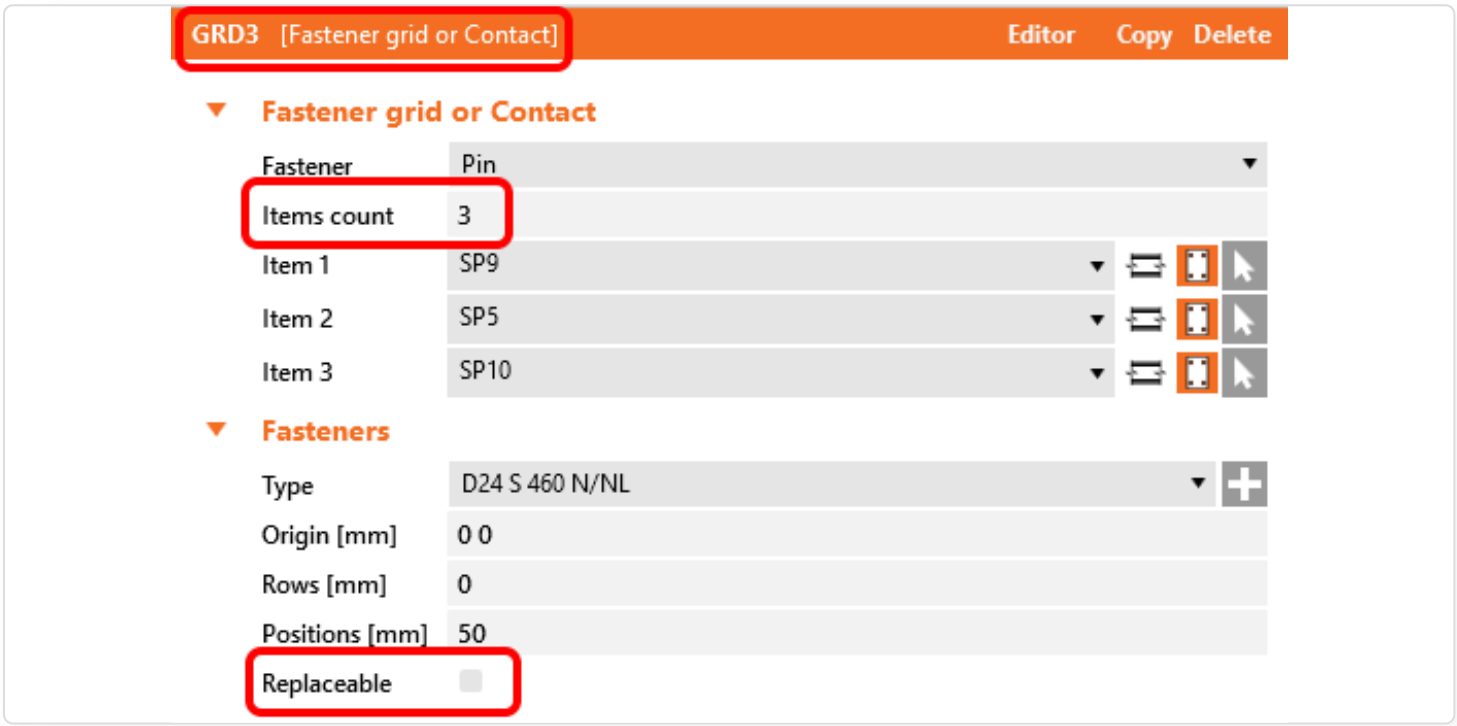
# Steel pins

Pins are widely used in all kinds of steel structures, typically connecting bracing, diagonal, rod, bar element, or other structural members that transfer tension and compression forces. Such a connector type is a ready solution available in the IDEA StatiCa Connection app.
Pins can be used for modeling pin connections as well as help in cases where a one-bolt connection doesn't meet the requirement in EN 1993-1-8, 3.13.1 (2) and has to be checked as a beam element with bending – pin.

# How to model a pin

There is a **Pin** type of fastener within the operation **Fastener grid and Contact**. When **Pin** is selected, the **count of items** is automatically changed to three, which is the minimum number of connected plates.

The pin can also be designed as **Replaceable**, which governs the used checks according to code.



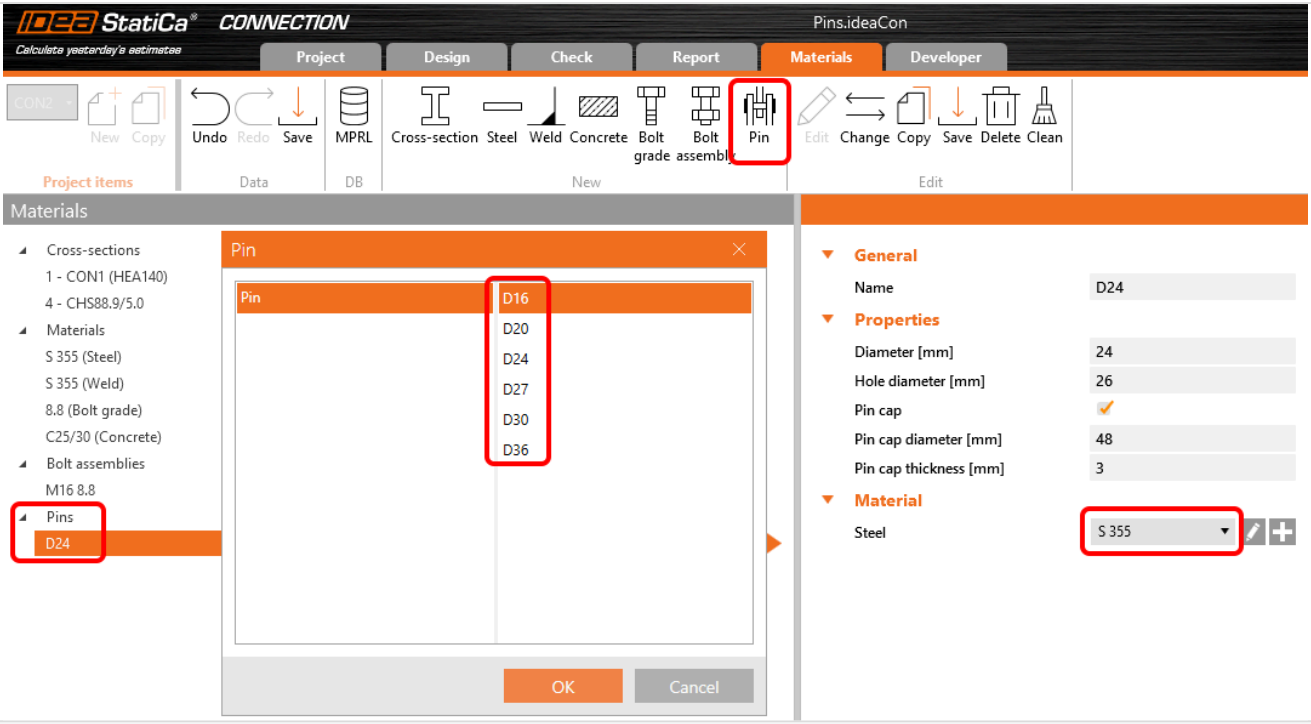

A member connected by a pin must have the model type **N-Vy-Vz** specified, otherwise, the analysis leads to a singularity.

The **contact between plates** is created automatically if the distance between connected parallel plates is less than 2 mm. If the gap is bigger than 2 mm, the contact is not created.

# Pins in the project material database

The geometric parameters of a pin (pin diameter, material, hole size, pin head...) can be defined in the **Materials** tab similarly to bolt assemblies. The **diameter** of a pin can be selected from the predefined material library.

Structural **steel material** (e.g., S355) is used as the default material for pins. In case a steel **bolt material** (e.g., 8.8) or any other type of material needs to be defined for pins, it can be done by editing the properties of added structural steel material ($f_u$, etc.).




A **Pin cap** can be turned on, but it doesn't influence the checks or analysis, only the visual presentation of pins in the 3D scene.

# Code checks of pinned connections

Checks are provided according to EN 1993-1-8, 3.13.2.

**Table 3.10: Design criteria for pin connections**

| Failure mode | Design requirements |
|---|---|
| Shear resistance of the pin | $F_{v,Rd} = 0{,}6\,A\,f_{up}/\gamma_{M2} \geq F_{v,Ed}$ |
| Bearing resistance of the plate and the pin<br><br>If the pin is intended to be replaceable this requirement should also be satisfied. | $F_{b,Rd} = 1{,}5\,t\,d\,f_y/\gamma_{M0} \geq F_{b,Ed}$<br><br>$F_{b,Rd,ser} = 0{,}6\,t\,d\,f_y/\gamma_{M6,ser} \geq F_{b,Ed,ser}$ |
| Bending resistance of the pin<br><br>If the pin is intended to be replaceable this requirement should also be satisfied. | $M_{Rd} = 1{,}5\,W_{el}\,f_{yp}/\gamma_{M0} \geq M_{Ed}$<br><br>$M_{Rd,ser} = 0{,}8\,W_{el}\,f_{yp}/\gamma_{M6,ser} \geq M_{Ed,ser}$ |
| Combined shear and bending resistance of the pin | $\left[\frac{M_{Ed}}{M_{Rd}}\right]^2 + \left[\frac{F_{v,Ed}}{F_{v,Rd}}\right]^2 \leq 1$ |

$d$ is the diameter of the pin;
$f_y$ is the lower of the design strengths of the pin and the connected part;
$f_{up}$ is the ultimate tensile strength of the pin;
$f_{yp}$ is the yield strength of the pin;
$t$ is the thickness of the connected part;
$A$ is the cross-sectional area of a pin.

If the pin is set as replaceable, different checks are used, according to Table 3.10 and Equation 3.14.

If the pin is intended to be replaceable, in addition to the provisions given in 3.13.1 to 3.13.2, the contact bearing stress should satisfy:

$$\sigma_{h,Ed} \leq f_{h,Rd} \quad \text{... (3.14)}$$

where:

$$\sigma_{h,Ed} = 0{,}591 \sqrt{\frac{E\, F_{Ed,ser}\, (d_0 - d)}{d^2\, t}} \quad \text{... (3.15)}$$

$$f_{h,Ed} = 2{,}5 f_y / \gamma_{M6,ser} \quad \text{... (3.16)}$$

where:

d is the diameter of the pin;

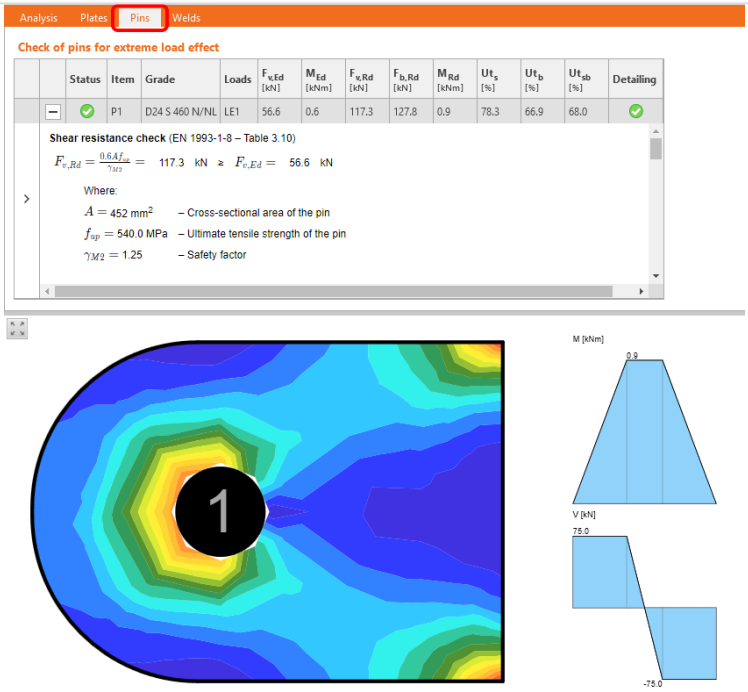
$d_0$ is the diameter of the pin hole;

$F_{Ed,ser}$ is the design value of the force to be transferred in bearing, under the characteristic load combination for serviceability limit states.

When the fire resistance analysis type is performed, the **replaceable** setting of used pins is ignored, and all pins in the model are checked as **non-replaceable**.

# Pins in the results and report

There is a **check tab for pins** with detailed results and code equations listed in the **Check** tab. Also, the **bending moment diagram** for the pin is displayed along with the stress/strain in the connected plate.




All relevant input data and result data of pins are included in the **report**.

**Bolts / Pins**

| Name | Diameter [mm] | $f_y$ [MPa] | $f_u$ [MPa] | Gross area [mm$^2$] |
|---|---|---|---|---|
| D24 S 355 | 24 | 355.0 | 490.0 | 452 |

**Pins**

| Item | Grade | θ [°C] | Loads | $F_{v,Ed}$ [kN] | $M_{Ed}$ [kNm] | $F_{v,Rd}$ [kN] | $F_{b,Rd}$ [kN] | $M_{Rd}$ [kNm] | $Ut_s$ [%] | $Ut_b$ [%] | $Ut_{sb}$ [%] | Detailing | Status |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| P1 | D24 S 355 | 0 | LE1 | 55.0 | 0.4 | 106.4 | 153.4 | 0.7 | 71.7 | 60.9 | 63.8 | OK | OK |

**Symbol explanation**

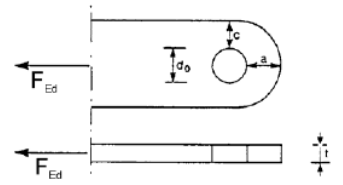
| | |
|---|---|
| θ | Temperature |
| $F_{v,Ed}$ | Maximum shear force in the pin |
| $M_{Ed}$ | Maximum bending moment in the pin |
| $F_{v,Rd}$ | Shear resistance of the pin |
| $F_{b,Rd}$ | Bearing resistance of the plate and the pin |
| $M_{Rd}$ | Bending resistance of the pin |
| $Ut_s$ | Utilization in shear |
| $Ut_b$ | Utilization in bending |
| $Ut_{sb}$ | Utilization in combined shear and bending |

# Detailing checks for pinned connections

Basic **detailing checks** are provided according to **Table 3.9 – type A**. For finding the **edge distances** "*a*" and "*c*", identical logic is used as the one used for determining the edge distances of bolts (dependent on vector direction).

**Table 3.9: Geometrical requirements for pin ended members**

Type A: Given thickness t

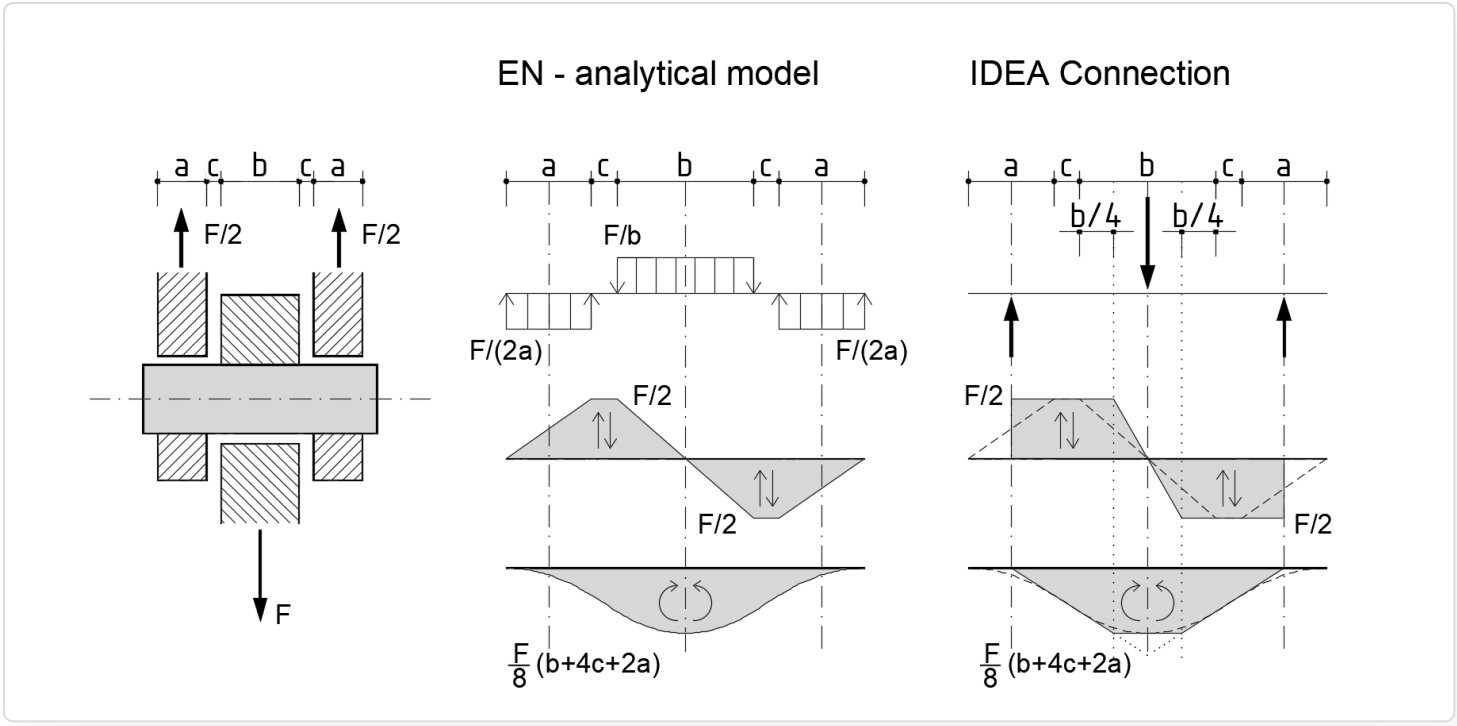



$$a \geq \frac{F_{Ed}\gamma_{M0}}{2\,t\,f_y} + \frac{2\,d_0}{3} \;:\; c \geq \frac{F_{Ed}\gamma_{M0}}{2\,t\,f_y} + \frac{d_0}{3}$$

# The CBFEM analytical model of a pin

Pins in CBFEM are modeled as linear elastic beam elements. The pin model is implemented in such a way that individual plates are not connected in the direction of the pin axis, and, thus, **no normal force is generated** in it. Only **shear force and bending moment** are active.

In the Connection app, the forces are applied to the pin as **point loads** in the axis of connected plates, and the bending moment diagram in a pin has a **simplified linear shape** compared to the analytical model in the Eurocode. Internal forces in the pin are evaluated and presented in a quarter of the inner plate thickness, which ensures that shear force and bending moment **maximum values used for the code-checks** are identical to extreme values from the analytical Eurocode model.

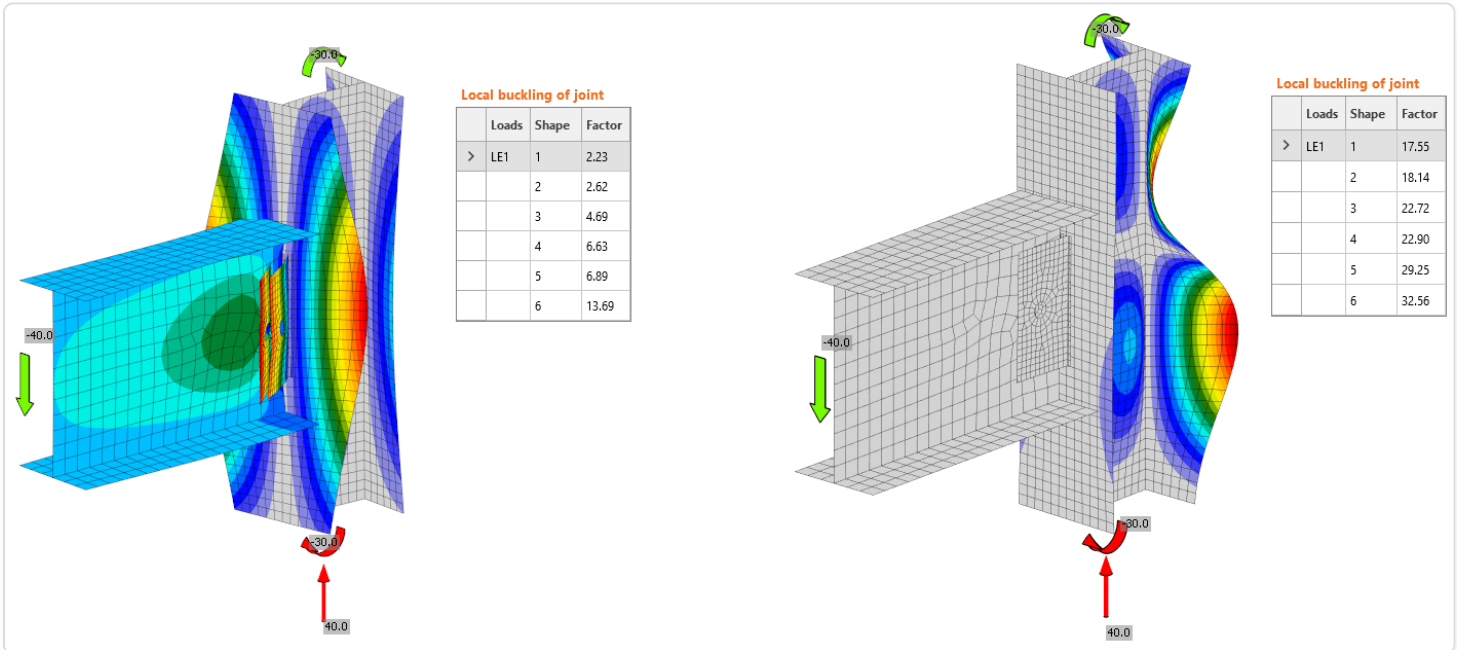



*Note: Pins are available for Eurocode (EN) only.*

*Released in IDEA StatiCa version 24.0.*

# Buckling calculation of models with pins

The **buckling analysis method for pinned connections** has been refined to deliver more realistic and less conservative results. The updated formulation in version 25.1 more accurately represents the rotational stiffness and boundary conditions of pinned joints, aligning the calculated buckling behavior with that of equivalent bolted configurations. This improvement ensures a closer correlation between analytical prediction and physical performance, enhancing the reliability of stability assessments in models containing pins.



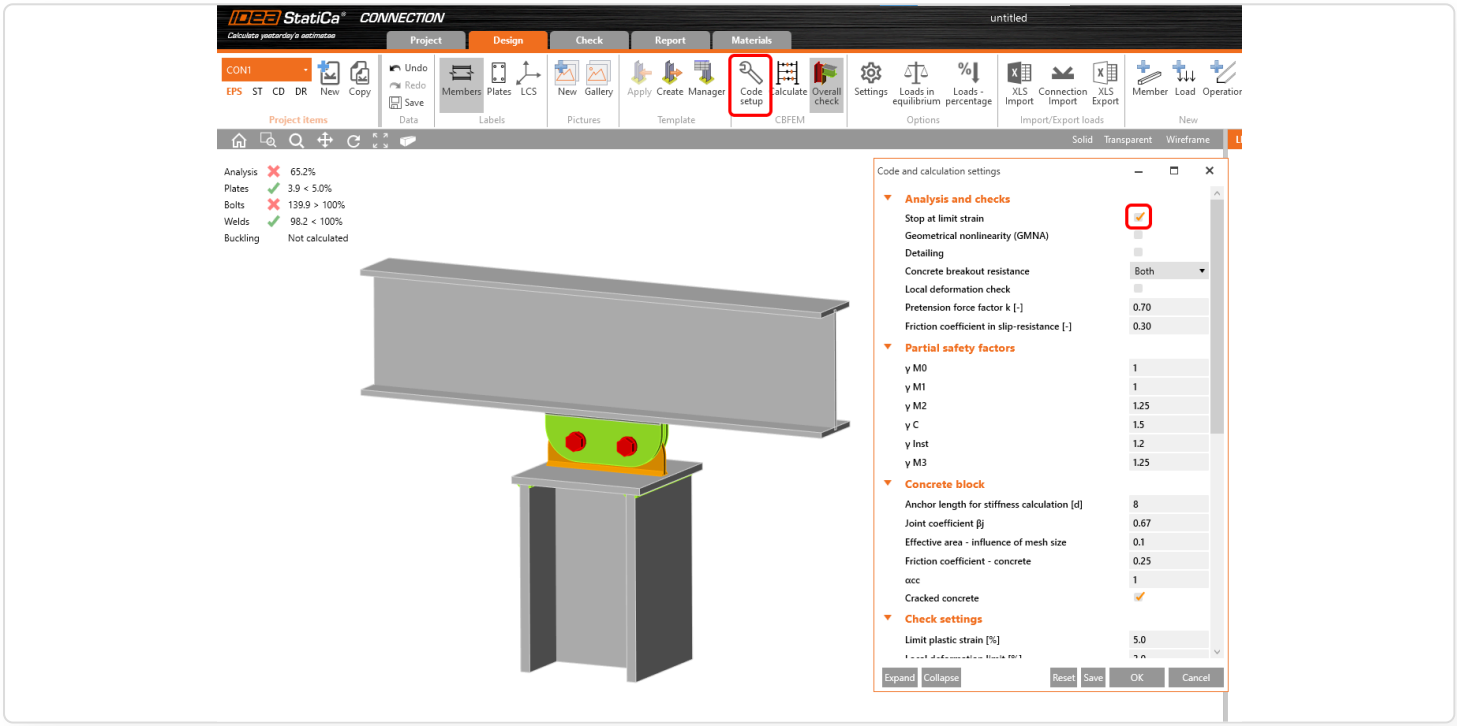

*Released in IDEA StatiCa version 25.1.*

# Connection analysis 0% or doesn't reach 100%

There could be several reasons for calculations failing, stopping, not completing and showing result of Analysis 0%, a value below 100%, or large deformations. The analysis engine detects these conditions and ensures users can see the potential sources of these incomplete or unstable results.
Analysis may also stop because of a singularity. Read more about singularities in What is the singularity warning.

# Stop at limit strain

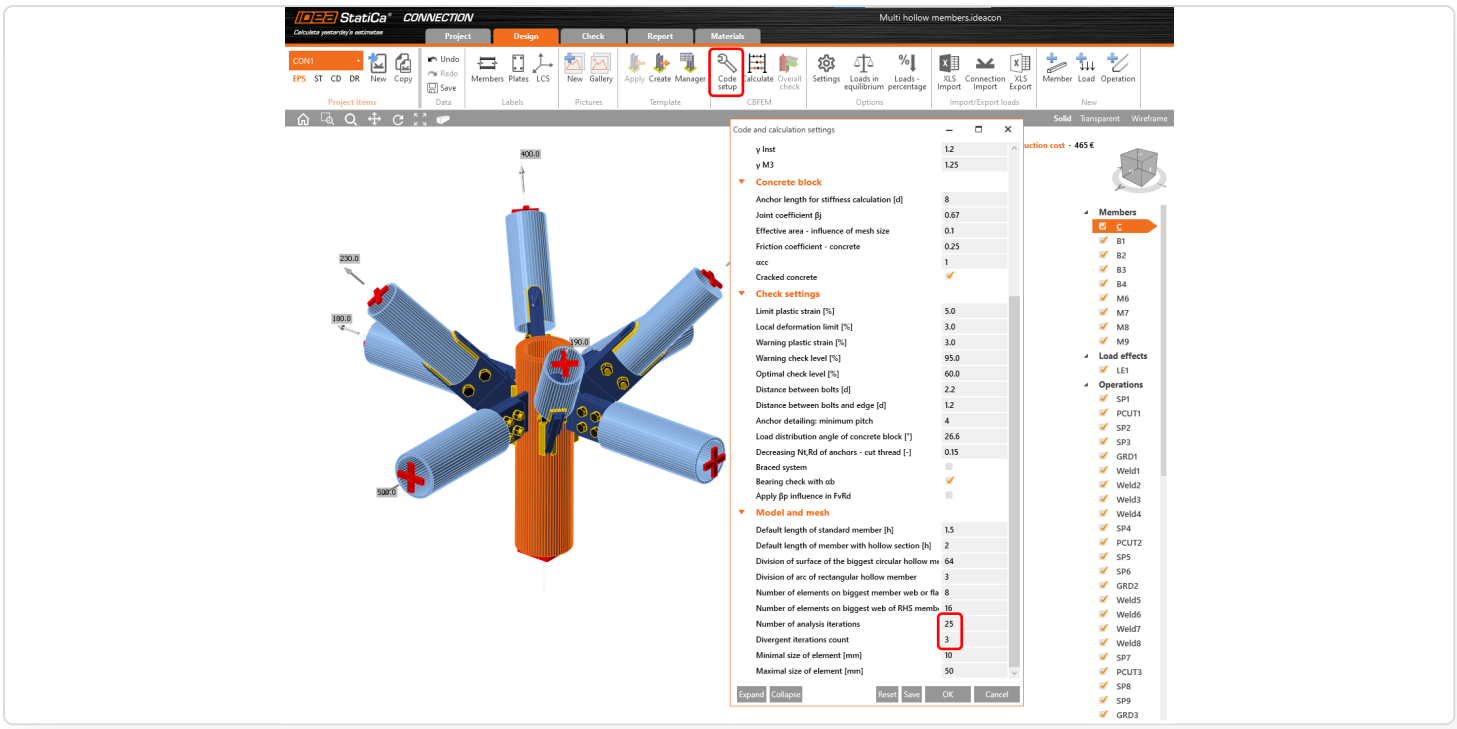
In the **Code setup**, there is an option to activate the **Stop at limit strain** feature. If this is turned on, the analysis halts when the capacity of any part of the connection (e.g., a plate, a weld, etc.) is reached. In case the connection is overloaded, the analysis stops before the inputted load effects are fully applied and the actual percentage of used loads is displayed.




# Too complex joint

When the joint model is very complex, the default capacity of the finite element analysis might not be sufficient, resulting in the analysis 0%. In such a case, the calculation capacity can be adjusted to enable the analysis to be completed.

In the **Code setup**, raise the values of the **Number of analysis iterations** from default value 25 to higher (e.g., 50) and the **Divergent iterations count** from default value 3 to higher (e.g., 5). This adds to the calculation capacity but also results in the calculation taking longer.

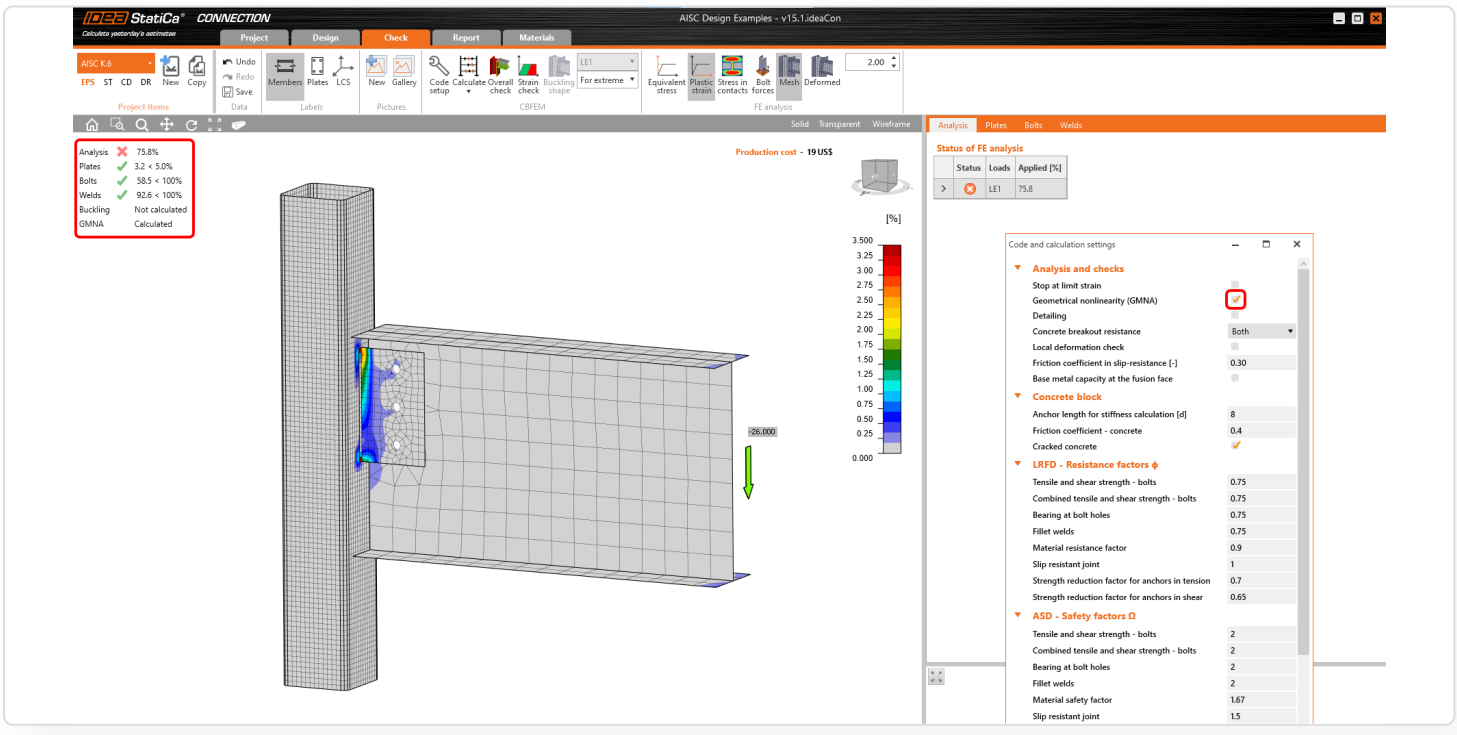



# Geometrical nonlinearity (GMNA)

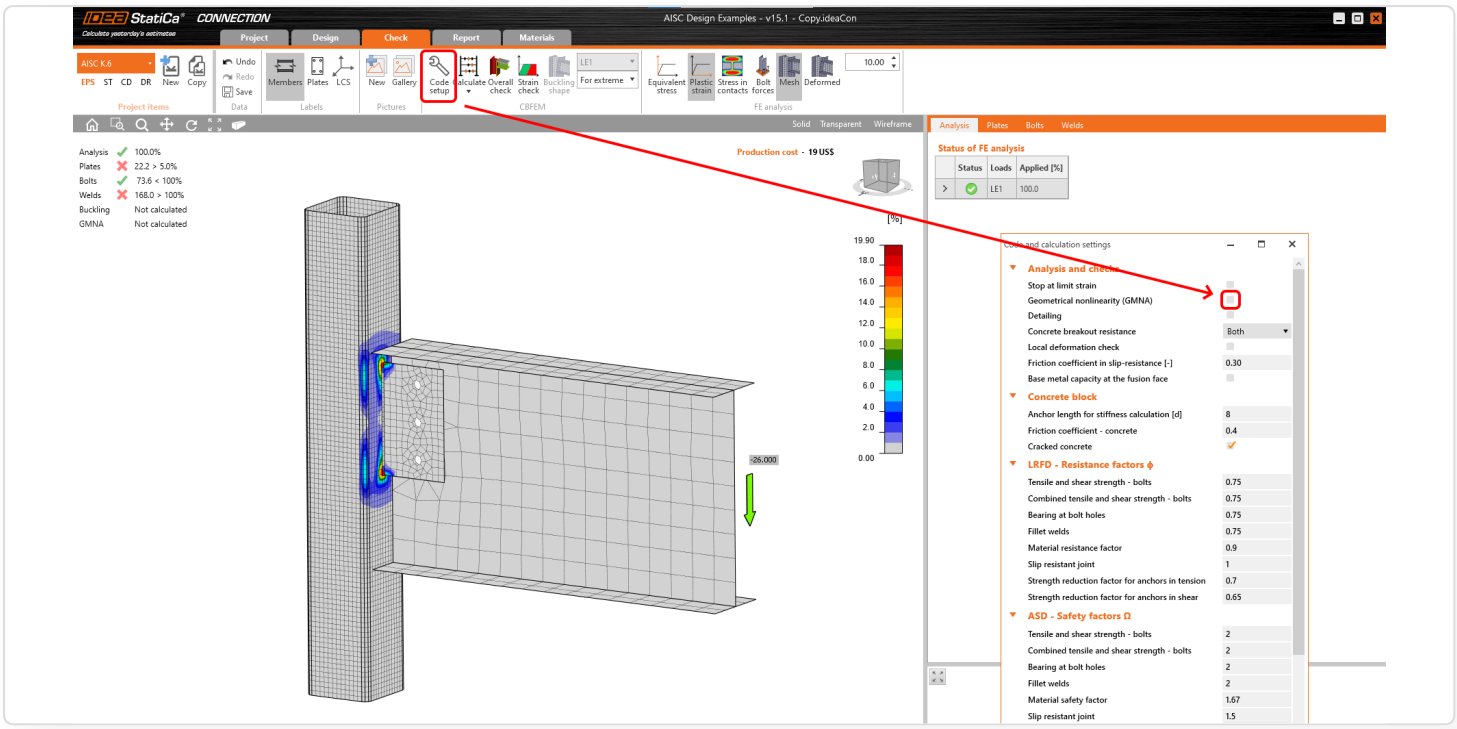
GMNA means advanced geometrically nonlinear analysis, which provides more precise results for models mainly with hollow section members. The GMNA solver is used only when the bearing member has a hollow section. GMNA analysis can be turned on/off in the **Code setup**.

*Note: If the bearing member is not a hollow section, the GMNA solver is disabled for the analysis of the whole connection model regardless of the settings in the code setup (GMNA on or off).*

When the connection is overloaded, the hollow sections might lose stability, which results in a break of the analysis at the current percentage of applied loads.



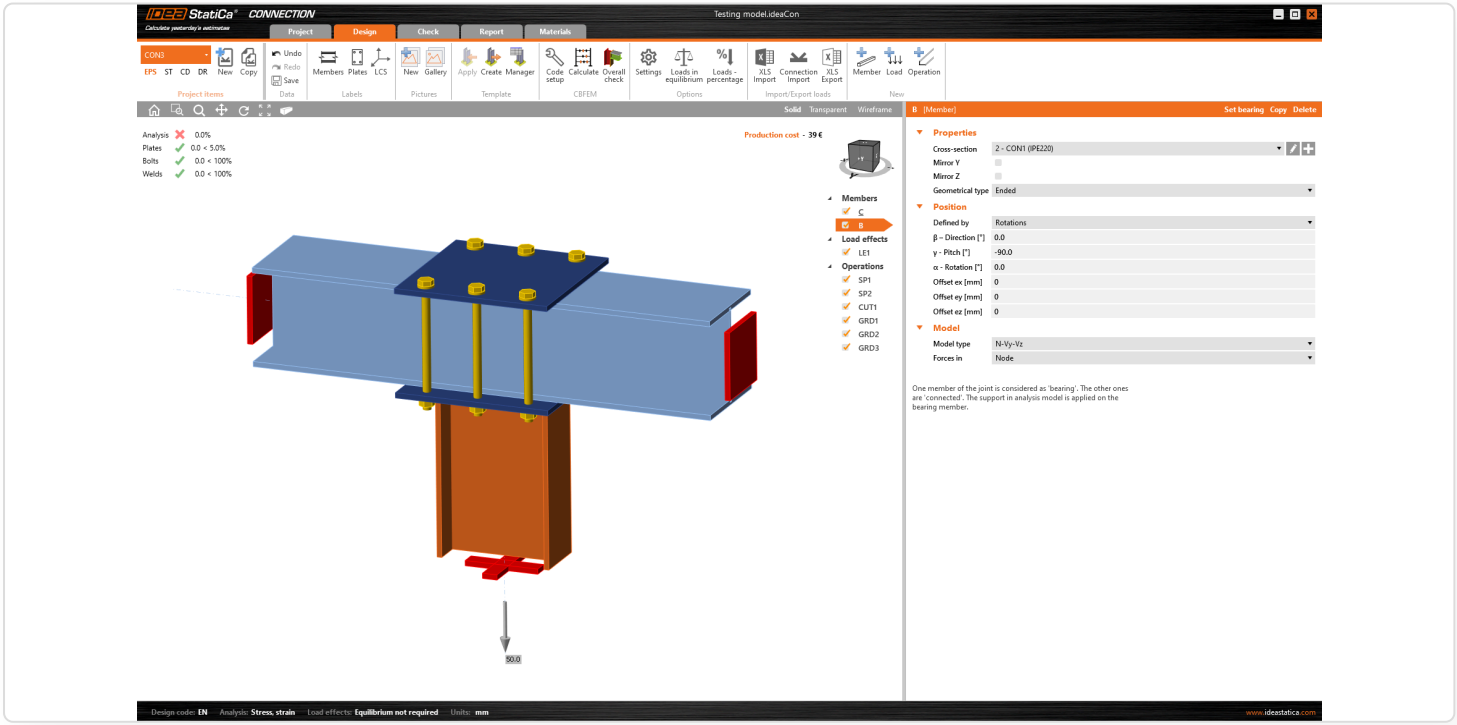
By disabling the **GMNA** in the **Code setup**, the analysis finishes with 100%, revealing the failure of the hollow section and other parts of the connections.



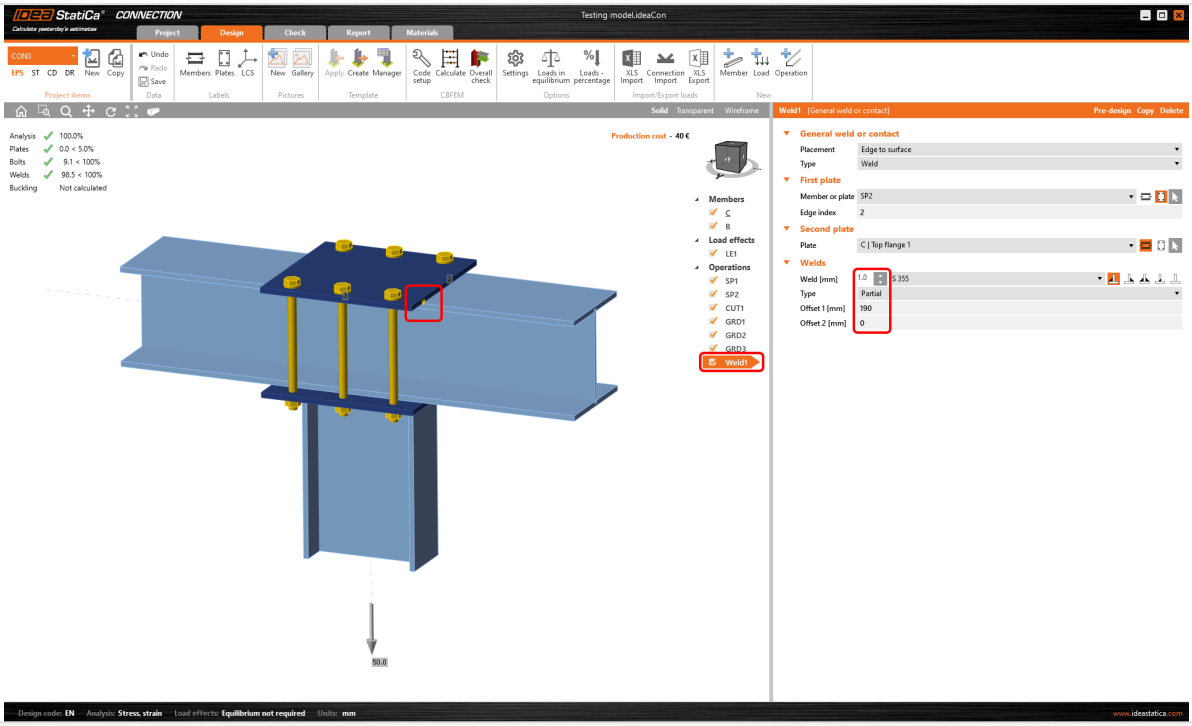

# Unstable analysis model (friction)

The CBFEM model in IDEA StatiCa can not directly calculate and code-check friction between plates when modeling a connection based on friction and preloaded bolts, such as a clamp connection.

In such a connection, where only tension/compression force is applied and transferred via contact (compression-only), there is a small prying shear force generated. Since friction is not taken into account and, thus, the shear force is not transferred, the model is unstable.

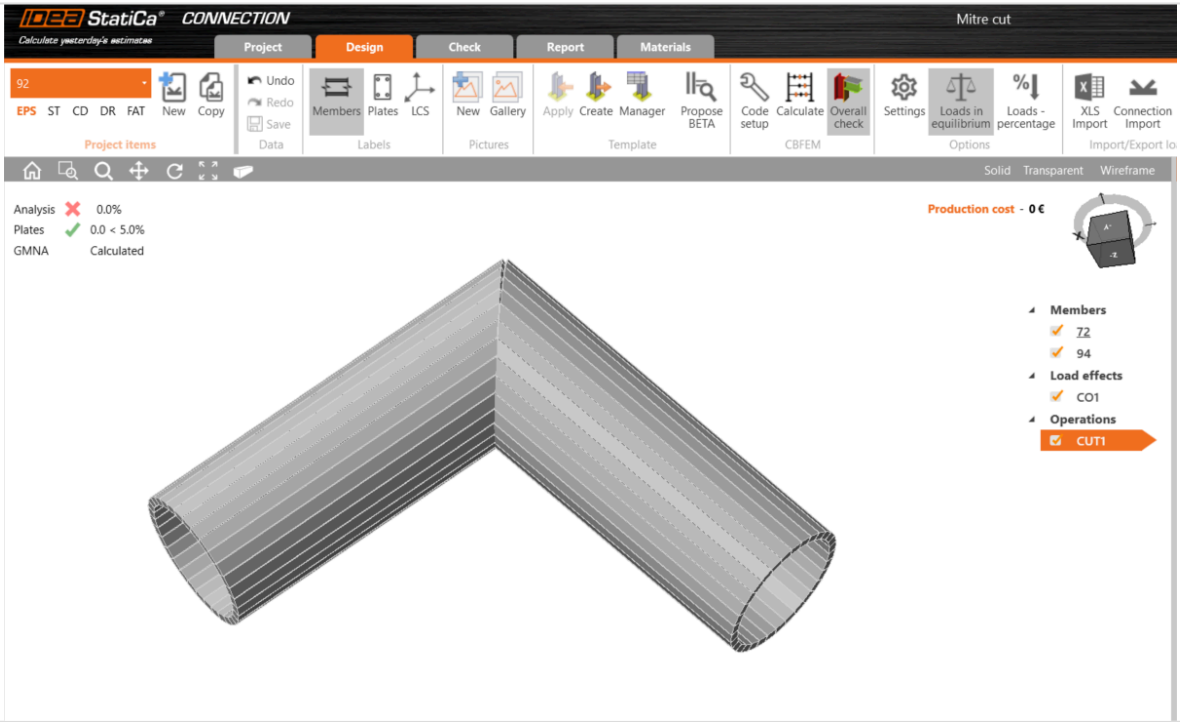


In this case, it is necessary to add a "utility" weld in order to deal with the small shear force. For this, a **Partial weld** with **Offset** can be used. The analysis can then be finished, while the impact of the utility weld on the model behavior is negligible.

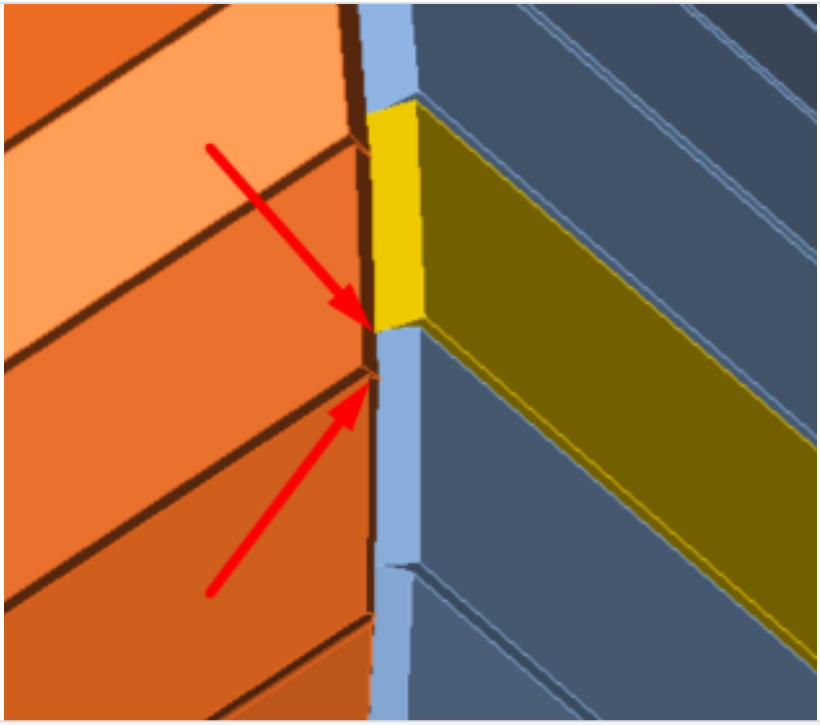



# Cut of hollow sections

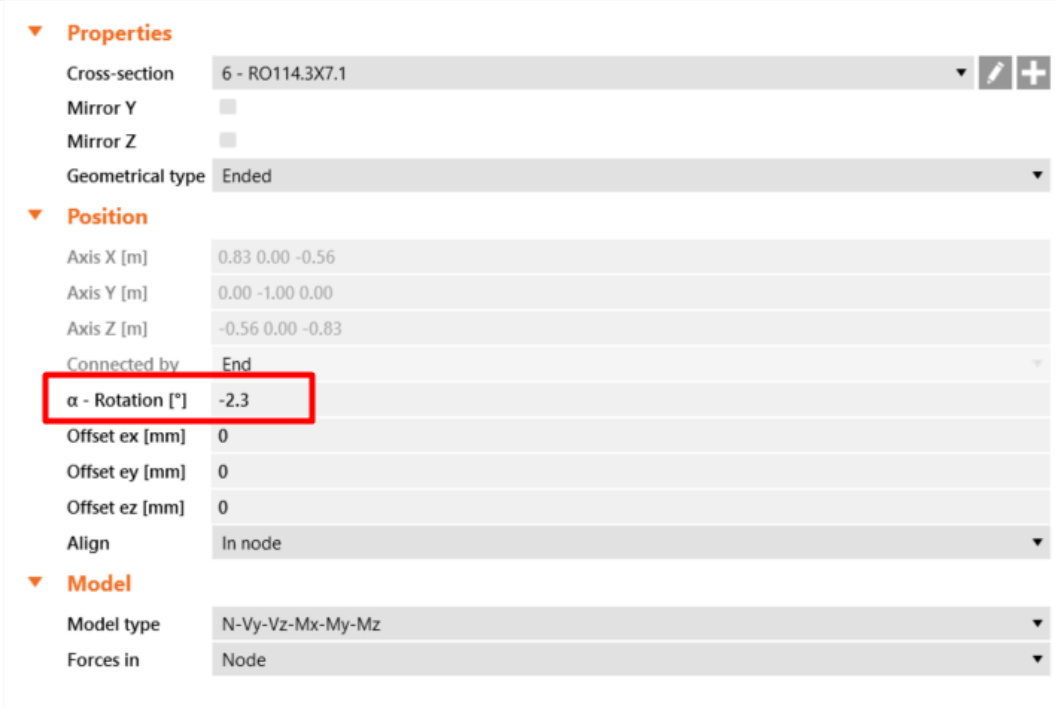
When the operation Cut and its cutting method **Mitre cut** is used to cut and weld members with circular hollow sections, sometimes you the analysis is not done, resulting in analysis 0%.




This is due to the alignment of 1D elements of the connected members. Having different angles or sizes, the operation may not be able to create the butt weld needed for the analysis.



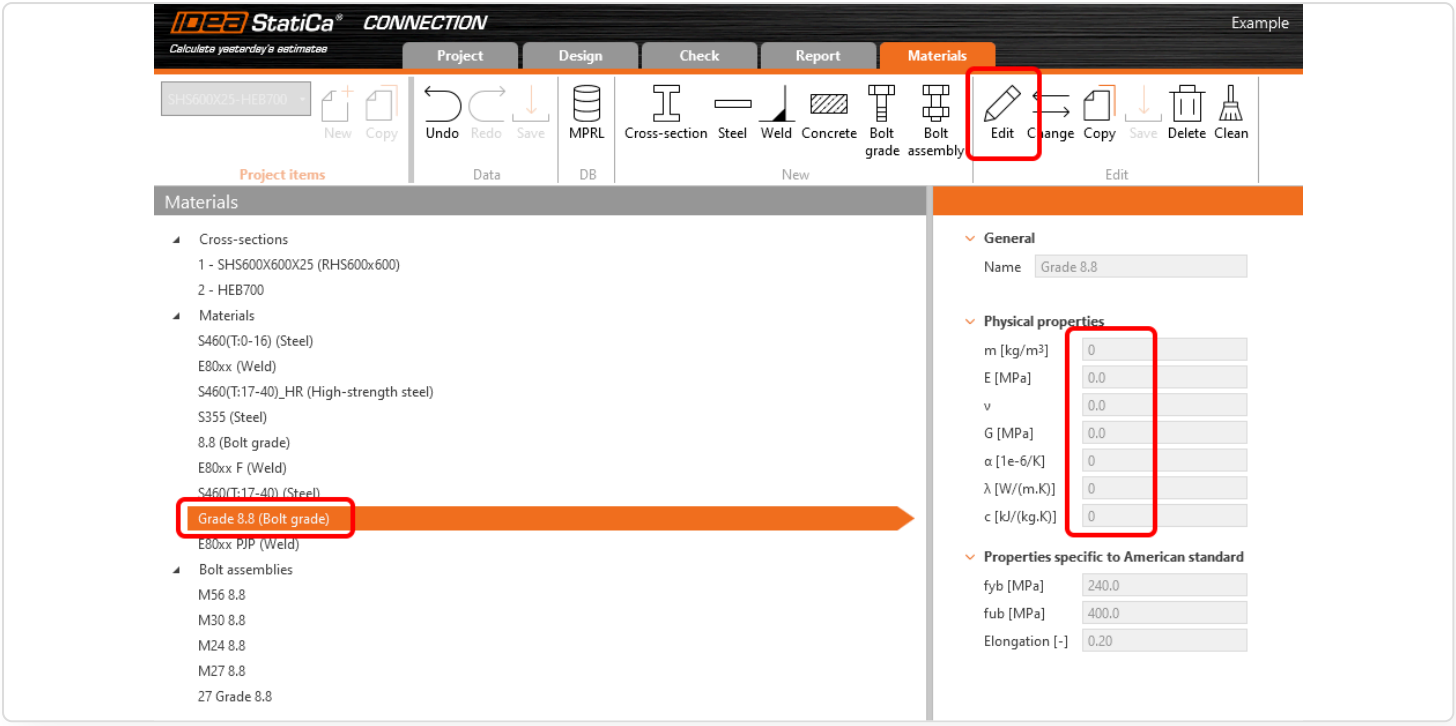
In such a case, change the value of **α - Rotation** for one of the connected members so that the elements are aligned and the butt weld is created. The butt weld is represented by a yellow line visible when the 3D screen is switched to transparent mode.




# Material properties are 0 (zero)

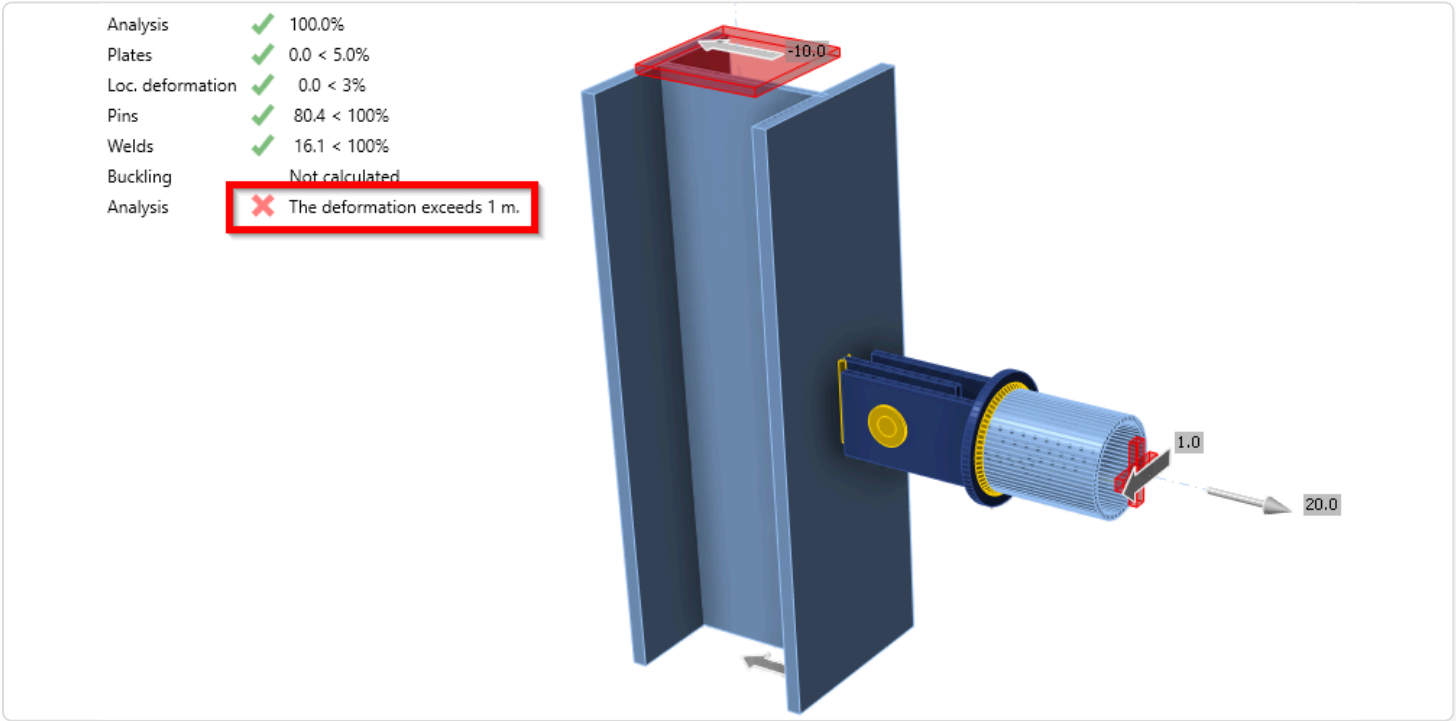
If a material property is filled with 0 (zero) or some non-acceptable value and this material is used in the project, the finite element model can not be calculated, and the analysis stops at 0%. This can be a property of a cross-section, steel material, bolt grade, etc.

Material parameters can be edited in the **Materials** tab, or possibly imported with a 0 value via a BIM link or CSV file (MPRL database).




# The deformation exceeds 1 m

A warning system for excessive deformations improves the model stability assessment. The overall results display a warning when large displacements over 1 m are detected, even if all code checks formally pass. This immediately alerts to a potential instability or unrealistic structural behavior without the need to manually review deformation results in the **Check** tab.




*Released in IDEA StatiCa version 25.1.*

# BIM and Checkbot

Checkbot workflows

# Checkbot – bulk BIM workflows

Checkbot is an IDEA StatiCa application that manages and synchronizes structural details imported to IDEA StatiCa from other programs. Checkbot as a structural data hub makes sure that you can work with all of your steel elements from other applications like Axis VM, Scia Engineer, Revit, or Tekla.
IDEA StatiCa Checkbot manages your BIM workflows and gives you:

- A clear list of all imported items, including status checked/not checked
- 3D visualization of imported members and loads
- A conversion table for materials and cross-sections
- Load combinations management

The tool's major advantage is that it understands not only the geometry of the original structure but, more importantly, the loading and internal forces resulting from the analysis. Every engineer who transforms internal forces from one analysis to another understands how tricky it can be to keep all the forces in the proper coordinate system. Checkbot can do that for hundreds of load combinations and tens of different tools originating from any supported software.

# Different ways of importing data into Checkbot

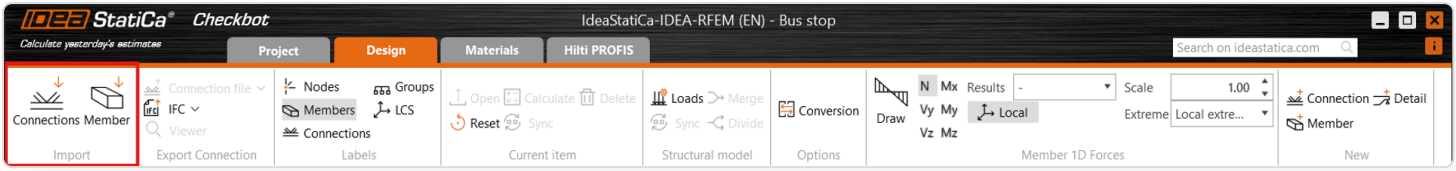
## BIM Links

Checkbot works as a stand-alone application, allowing users to open any structural design using **BIM links** directly in third-party software. The import command bar slightly differs depending on whether the source program is a CAD or FEA (finite element analysis software) type. You can always check supported integrations for steel CAD/FEA software, plus their known limitations, and see a list of supported versions of 3rd party applications.

Following step-by-step tutorials, you will learn how to design and code-check your connections and members using the BIM link between IDEA StatiCa and other software.

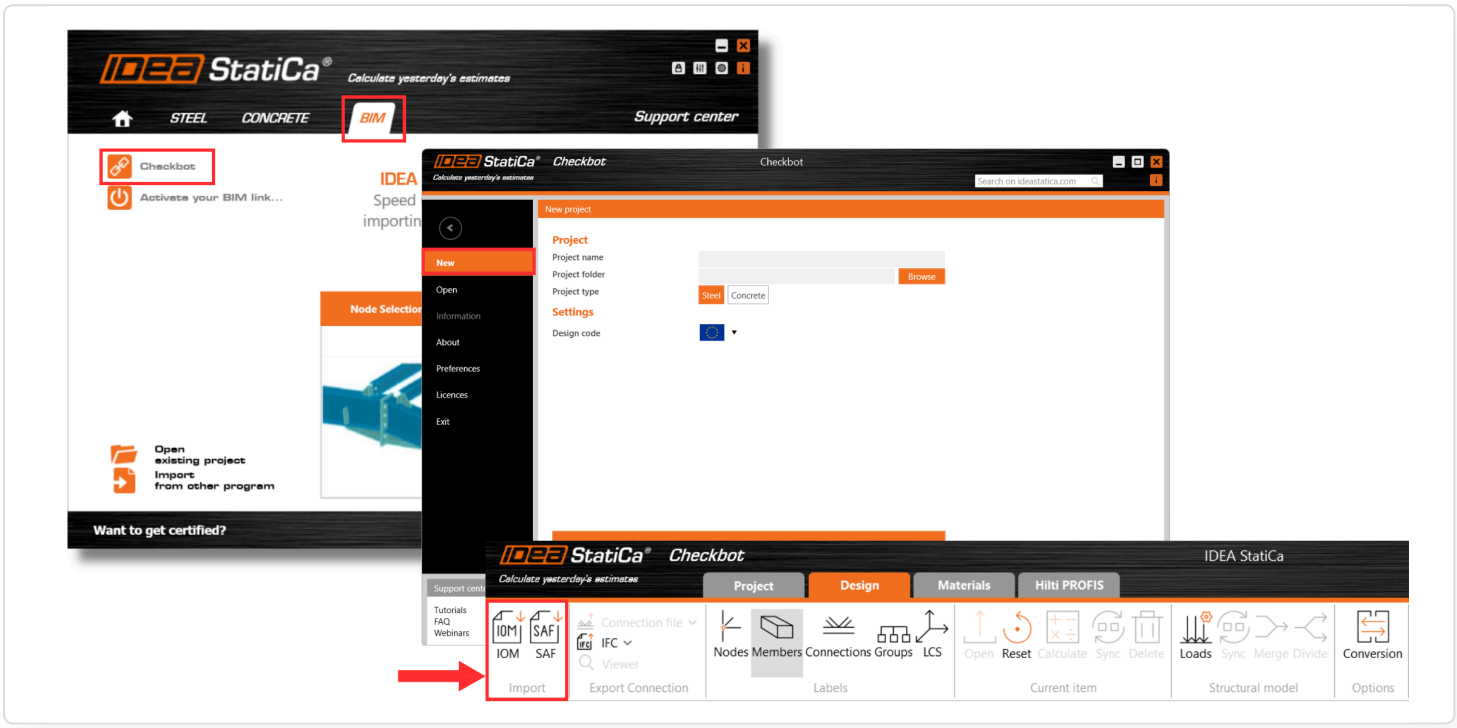
- SCIA Engineer
- SAP2000
- Tekla Structures
- Revit
- Robot Structural Analysis
- Dlubal
- ETABS
- ... and more

When a Checkbot project is created utilizing the BIM link, the nodes and members need to be chosen to import connections or members.



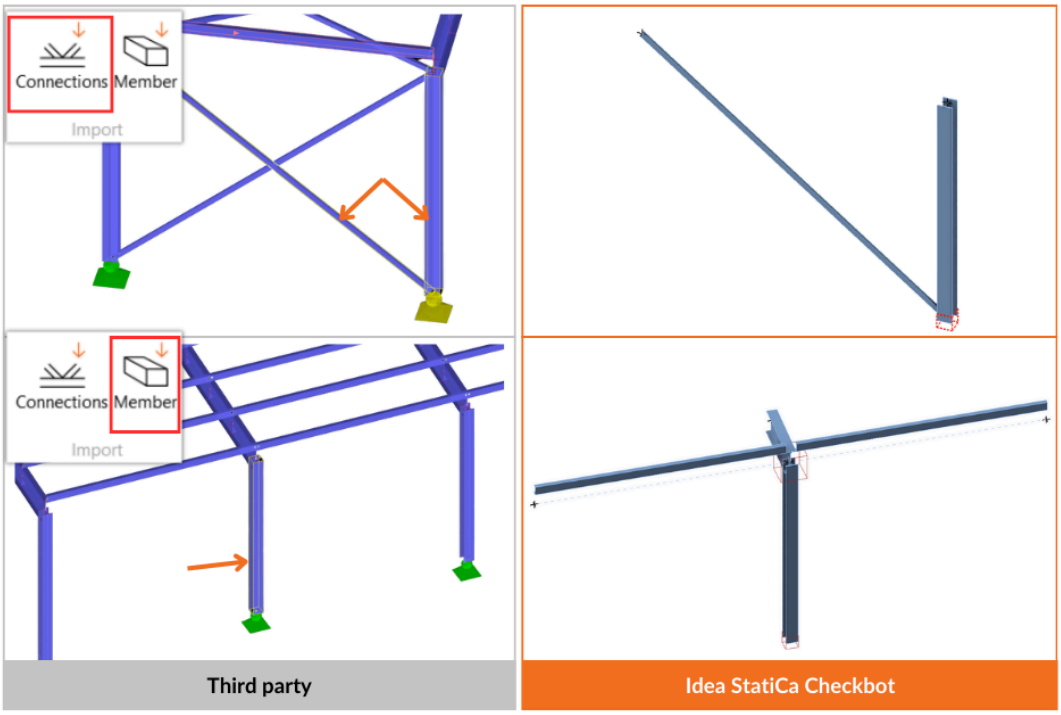

## SAF/IOM

Another way of importing data from CAD/FEA software is provided using **IDEA Open Model** (IOM). It also enables the importing and processing of widely used **Structural Analysis Format** (SAF). An SAF file can be exported from SCIA Engineer, Dlubal software, FEM-Design, SOFiSTiK, Risa 3D, FRILO, Allplan, AxisVM, ConSteel, and many others.



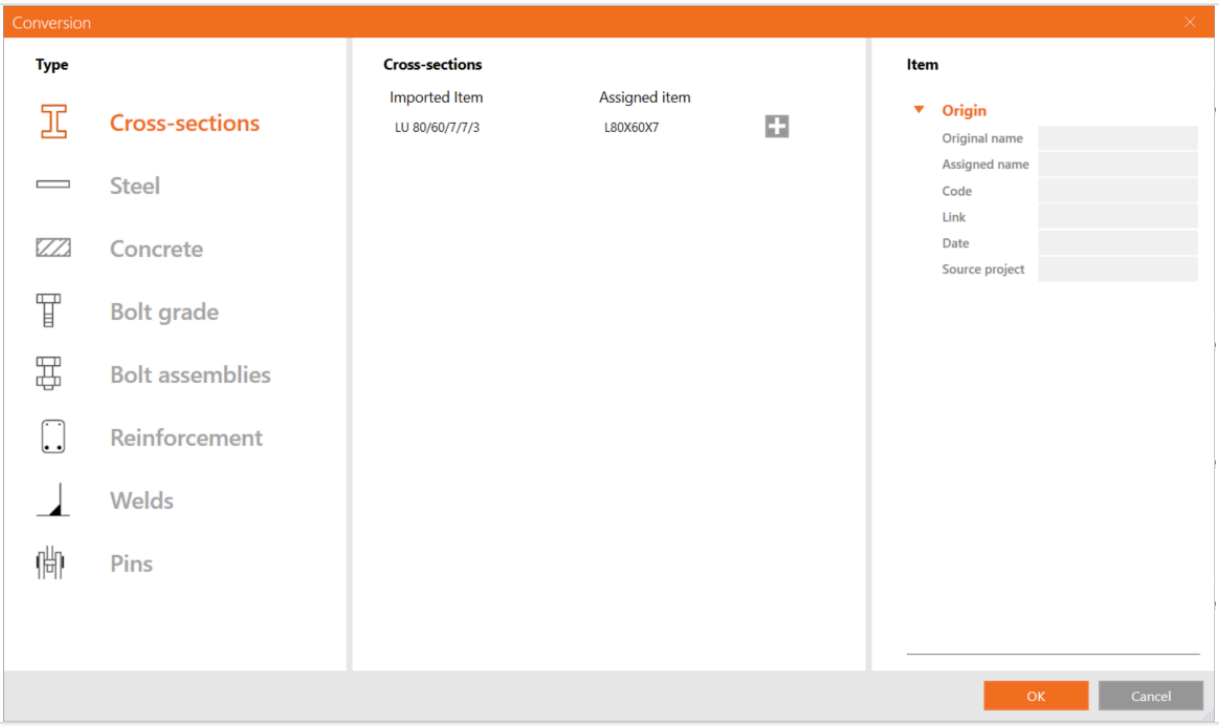

## Importing connections/members

- **Connection** – imports only a selected node and connected members into the Checkbot application.
- **Member** (analysis software only) – imports selected nodes and beams, which are fundamental for verification in the application IDEA StatiCa Member.

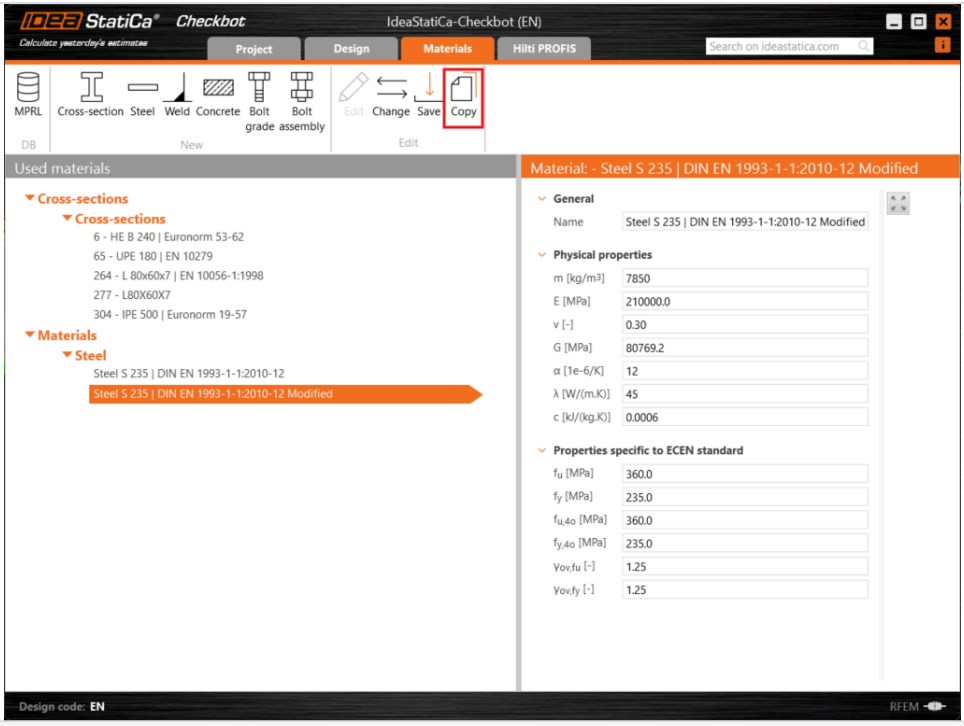



# Conversion

When a cross-section or a material is not recognized automatically during import, a new conversion tab appears to manually assign it from our section/material library. These pairs are then saved for future use within your user account and, therefore, do not need to be defined again.




All materials used in Checkbot are listed in the **Materials** tab. All imported materials and cross-sections are disabled for making any changes. To modify their parameters, a copy needs to be created.




The conversion tab is saved for each design code in a separate .XML file in the following folder:
C:\Users\YOURUSERNAME\AppData\Local\IdeaStatiCa
Here, the conversion files can be modified manually, deleted completely, backed up, or shared with other users.

### Numbering and local coordinate system

The numbering and local coordinate system of members in Checkbot and in third-party software might be different. Although Checkbot is based on a load-mapping algorithm, there is no need to be worried. Checkbot recognizes and identifies the corresponding members and reliably assigns the correct load effects to them.

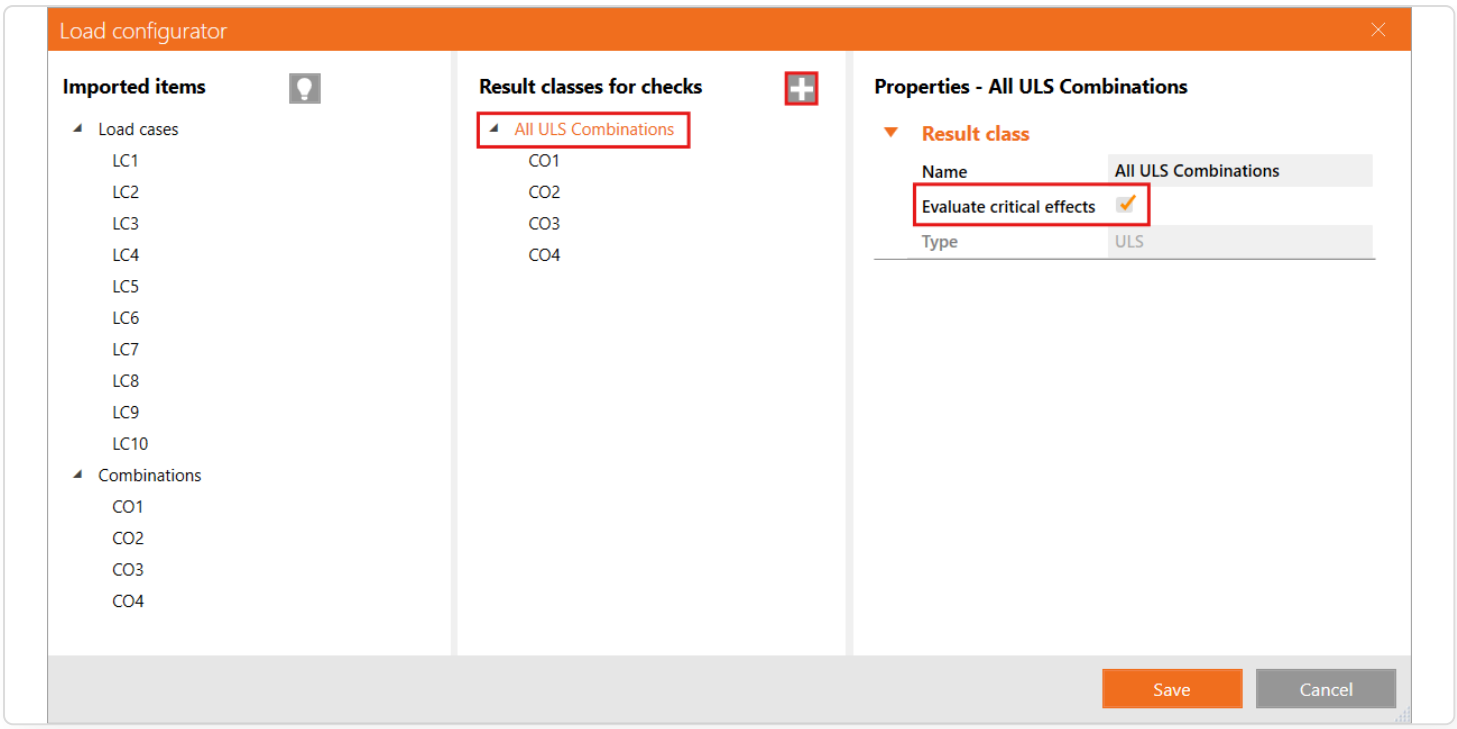
## How to work with Checkbot

### Load configuration

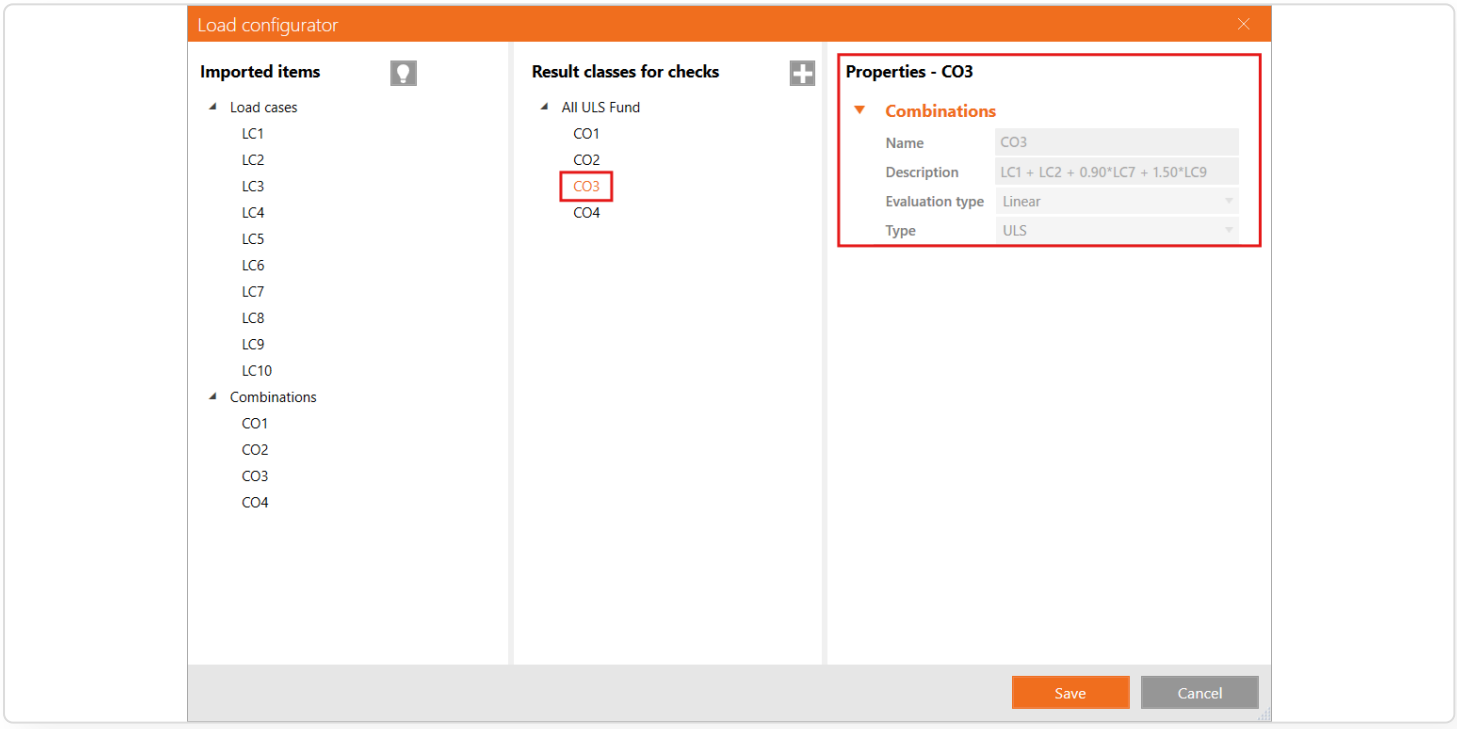
The **Load configurator** displays the imported load cases and load combinations. The first column lists all the Load cases and Combinations imported from the linked FEA model. The second column shows the result classes used for the Checkbot project.

Imported load effects can be assigned to **Result classes**. Result classes are meant to sort load effects to specific groups, which can simplify and speed up the design process. Result classes can be created using the "**+**" icon and removed with a right-click.

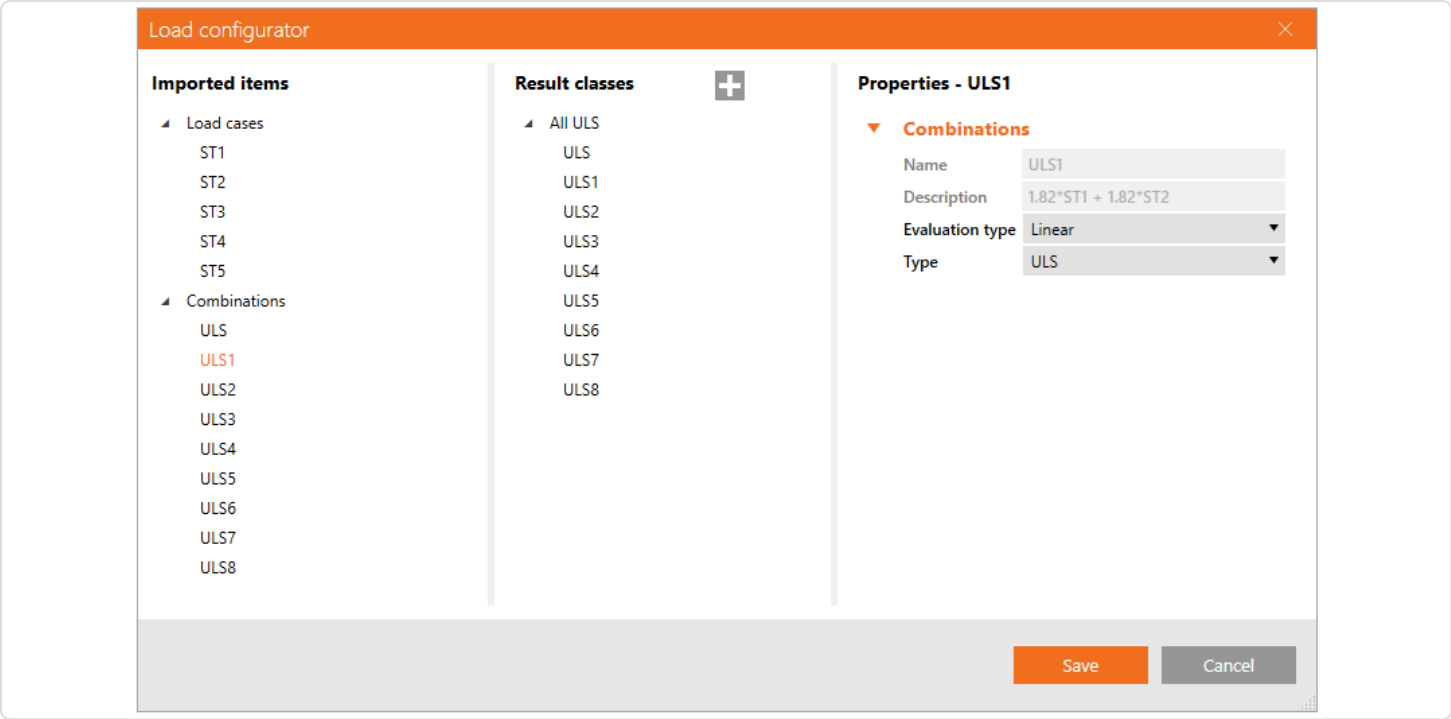
The load configurator allows you to activate the evaluation of critical effects in complex structures. This feature selects load cases and combinations with maximum and minimum load effects taking into account the stresses in the top and bottom fibers of the cross section to speed up the code-check calculations.



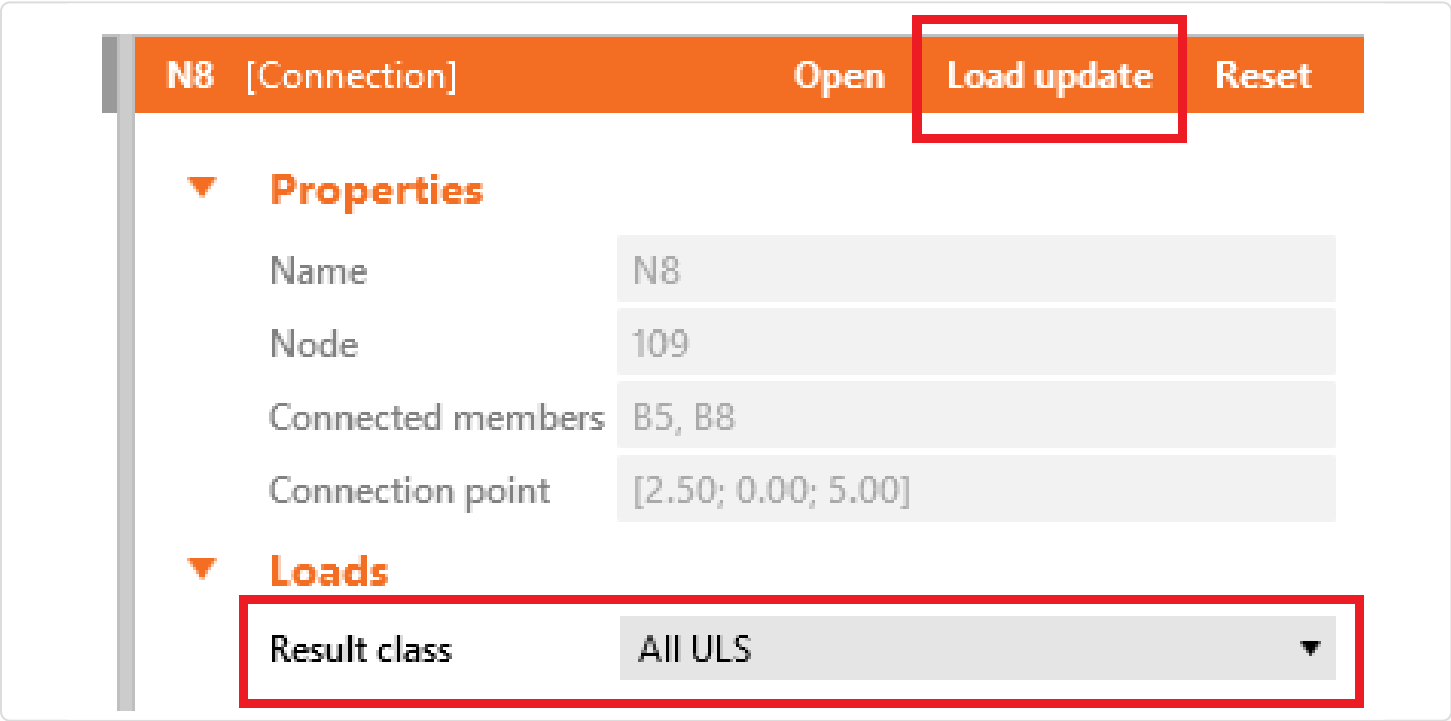

Furthermore, the third column displays detailed descriptions of the currently selected entity.



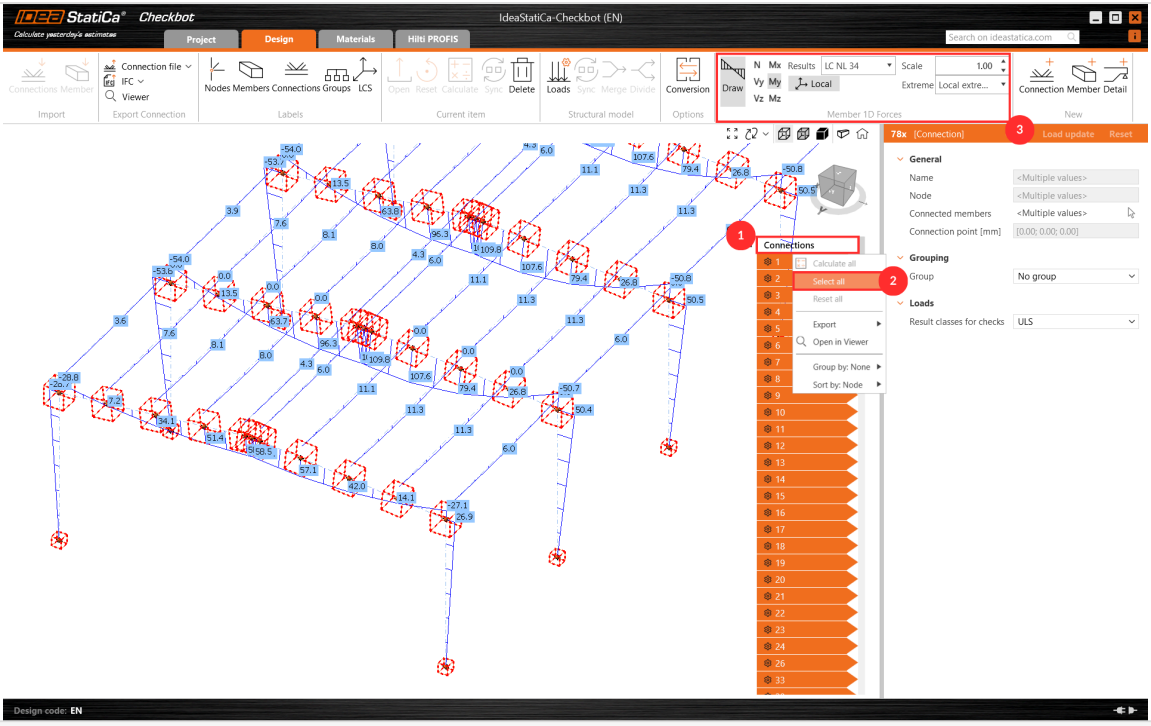

Any Load case or Combination can be assigned under a Result class by dragging it from the first column under the appropriate Result class in the second column. Result classes can be added using the "+" icon and removed with a right-click of the mouse.




A Result class can be assigned to any given Design item using the drop-down menu in the project item properties. After a different Result class is selected, the Load update button can be used to generate Load effects for the project item based on the newly selected Result class. In case the project item has already been calculated, this action will delete the old results.




Checkbot also visualizes the internal forces of imported combinations. Right-click connections/members or choose all of them and select the load combinations and internal forces to be displayed.

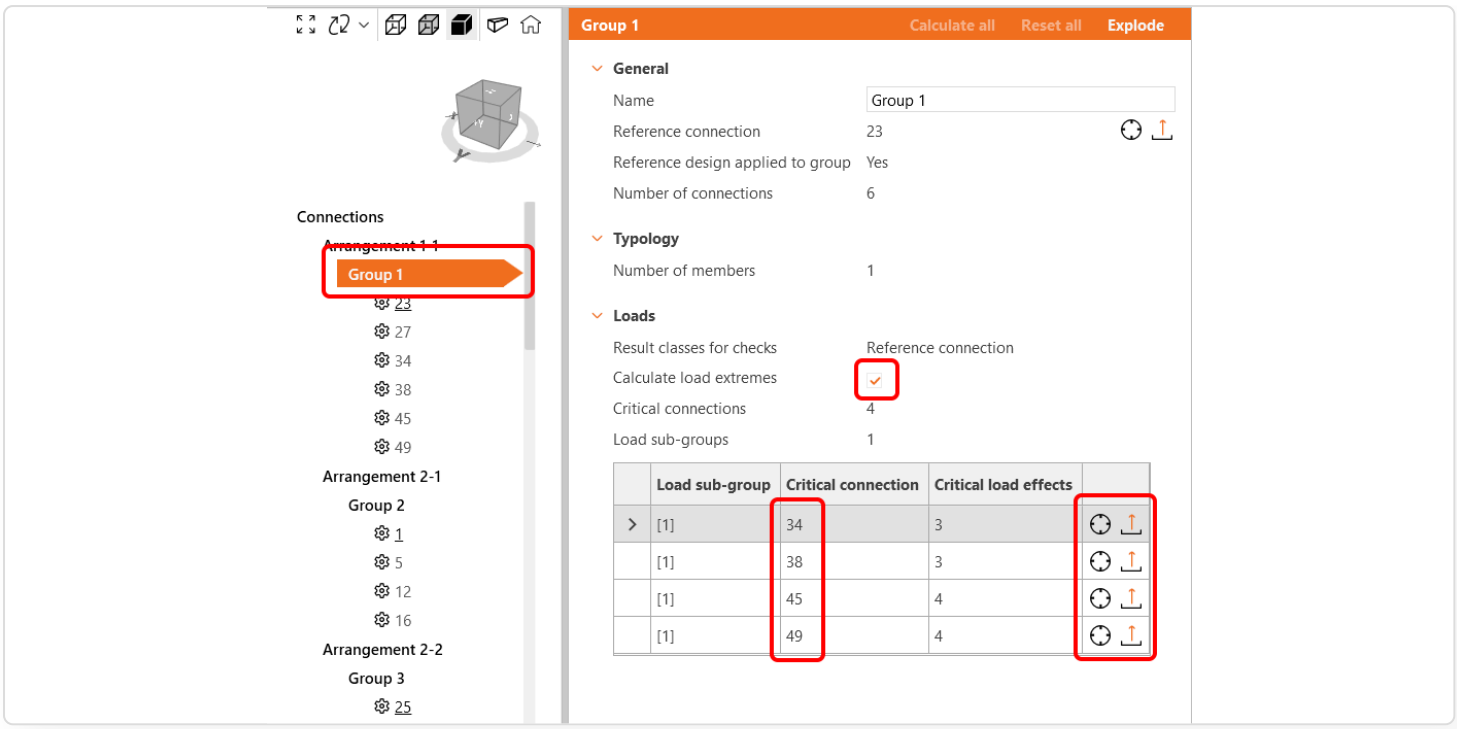



Please be aware that not all combinations are compatible with Checkbot. Some combinations might be imported as load cases, which does not affect the results. Always check the known limitations of each BIM link:

- STAAD.Pro
- Scia Engineer
- ETABS/SAP2000
- RFEM/RSTAB
- Robot Structural Analysis
- Advance Steel

## Calculation of load extremes

In projects with many nodes and load combinations, similar connections with very similar load effects are analyzed repeatedly. To speed up a calculation process, you can use the **Calculate load extremes button** to define only the load extremes containing the critical load effects. This will reduce the number of examined load effects as well as the time of calculation.




How it works is explained in this article.

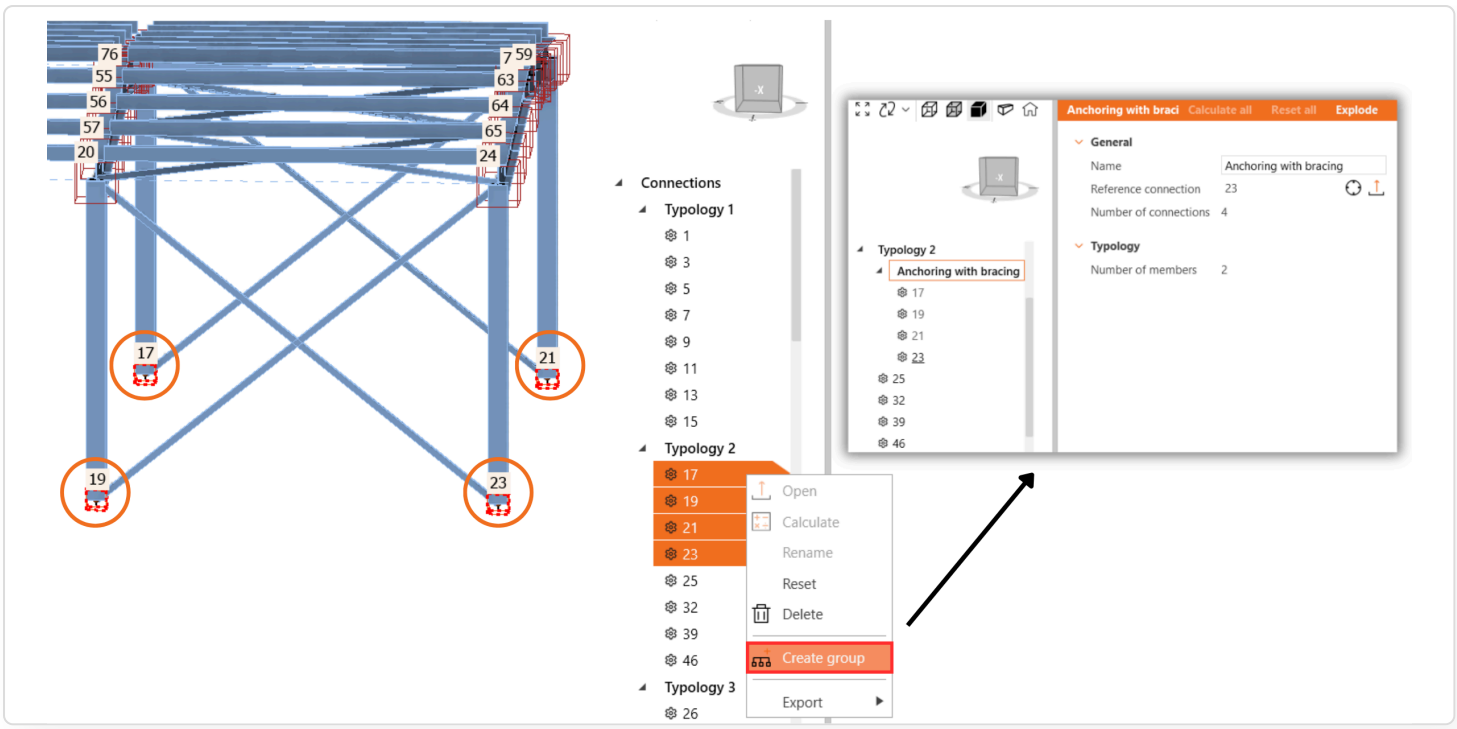
## Dynamic grouping

The grouping functionality automatically creates groups based on typology and/or cross-sections. The groups are then ordered from least complex to most complex based on the number of connected members. Every group is represented by the **reference connection** (underlined in the tree), which is automatically designated as the parent connection that drives the design for the whole group. All other connections are treated as **child connections**.

Any operations added to the reference connection will automatically be duplicated to the child connections to avoid work duplication, including project settings, position of the load, and model type. Adjustment of the child connection is disabled. If adjustments are needed, right-click the connection and select 'Remove from group.' Once a connection is ungrouped, all operations copied from the reference connection will also be deleted.

## User-defined groups

Furthermore, users can create their own groups, allowing further customization and enabling organization based on specific project needs.



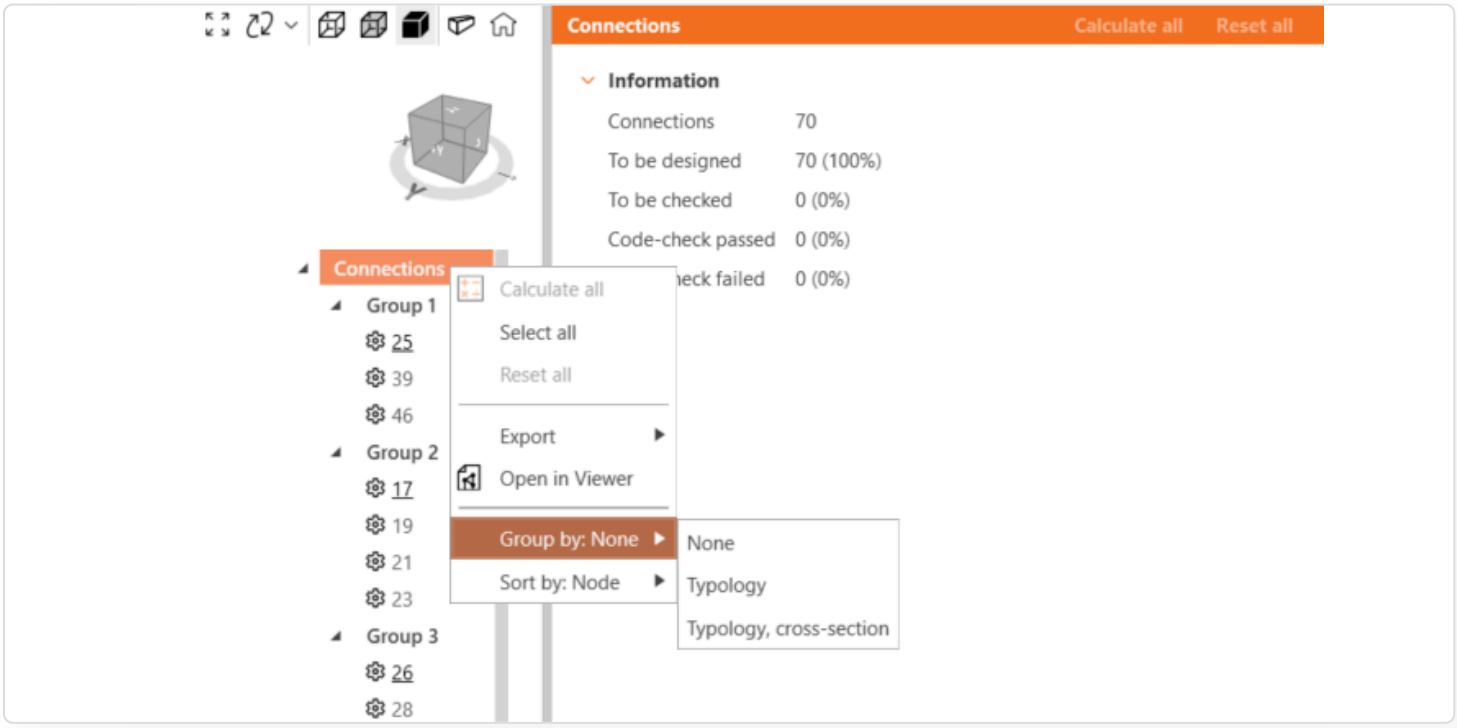

To define a specific connection as a reference, start by creating a group that includes only the desired reference connection. Once the reference connection is added, include the other connections in the group. You can multiselect connections and change their group with one click.

## Grouping in the tree

**None** arranges connections according to dynamic groups only.

**Typology** grouping considers the number of members and their relative positions (beam-to-beam, beam-to-column). It does not consider the rotation of members. The dynamic groups are considered too.

**Typology, cross-section (Arrangement)** grouping differentiates typology groups according to designed cross-section types. For example, a HEB 200 and a HEB 220 are under one cross-section type. Grouping according to typology and arrangement is set by default. The dynamic groups are considered too.



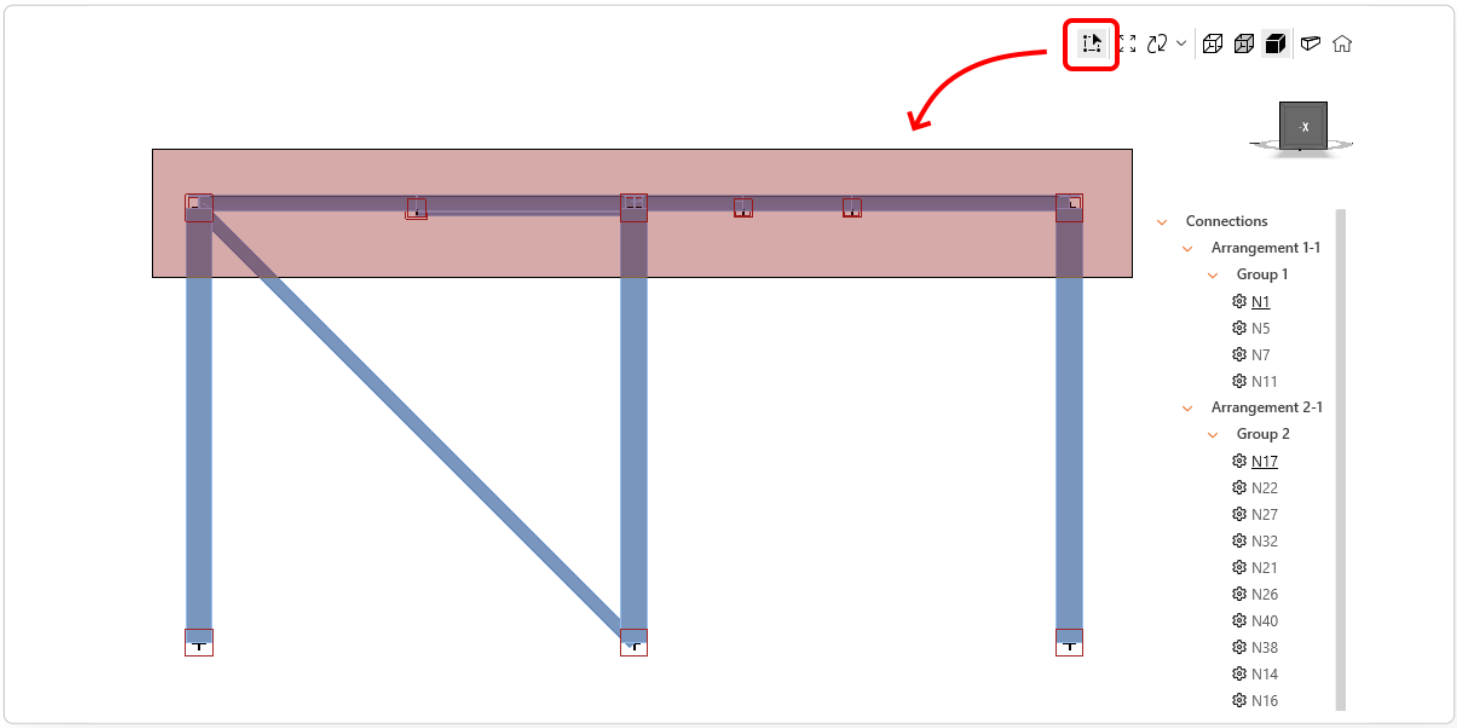
*Released in IDEA StatiCa version 25.0.*

## Alignment of members by cross-section

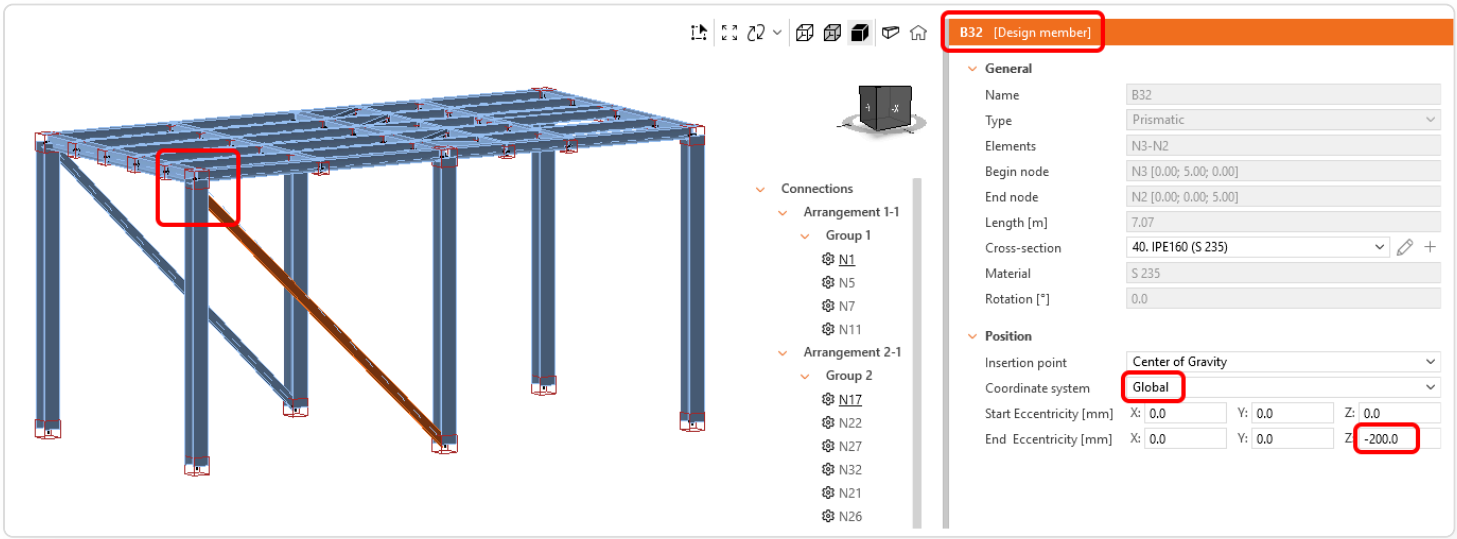
Single-select, multi-select, or area-select allow users to effectively input the local and global eccentricities and align members to match their surfaces across the entire structural model imported to Checkbot.

This is a necessary step between the global model imported from FEA software, where the geometry is simplified, and the CAD-based design model in IDEA StatiCa Connection, where the geometry matches the real design.

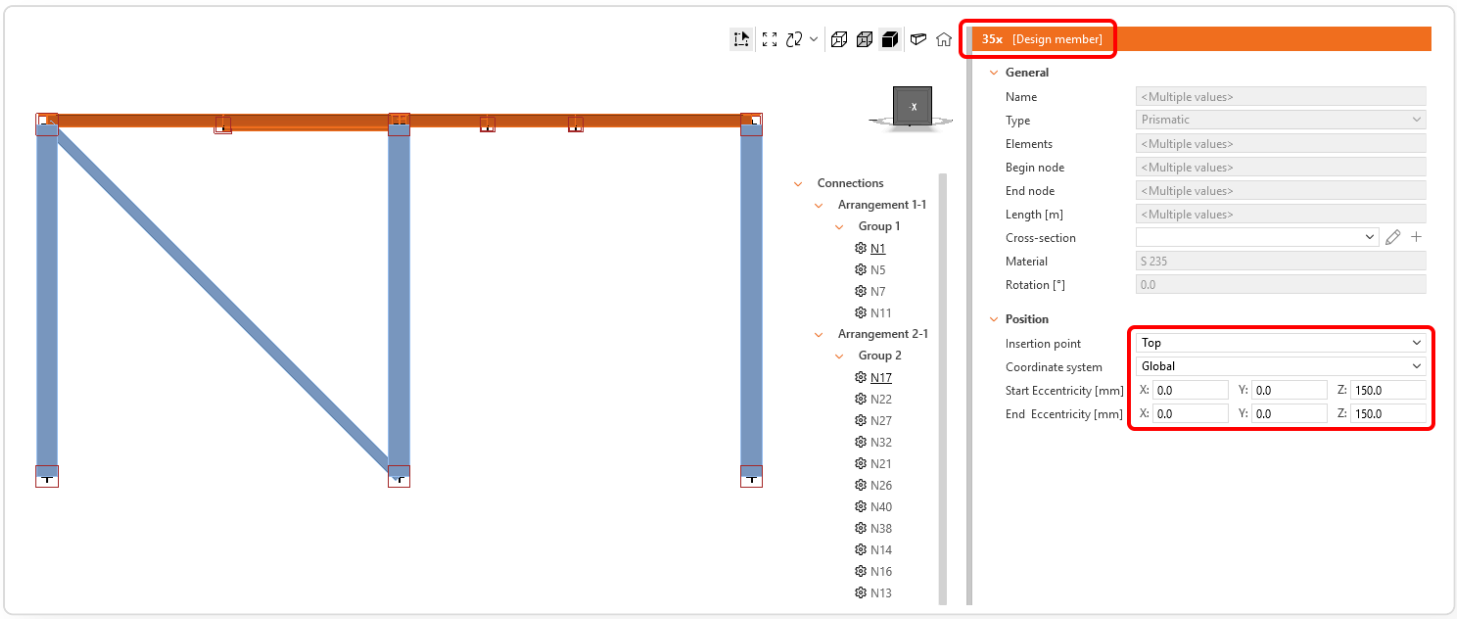
To perform the area selection, click the **Select multiple members with area selection** icon located in the 3D scene and select the members. Drag from left to right – only members inside the selection area are included, and drag from right to left – also members intersecting the selection area are included. Turn off the area selection by clicking the icon again or pressing the ESC key.




You can input eccentricities for single or multiple members at once. Select the **Local** or **Global** coordinate system and input the positive or negative eccentricity for the member's start point and end point in X, Y, or Z direction.

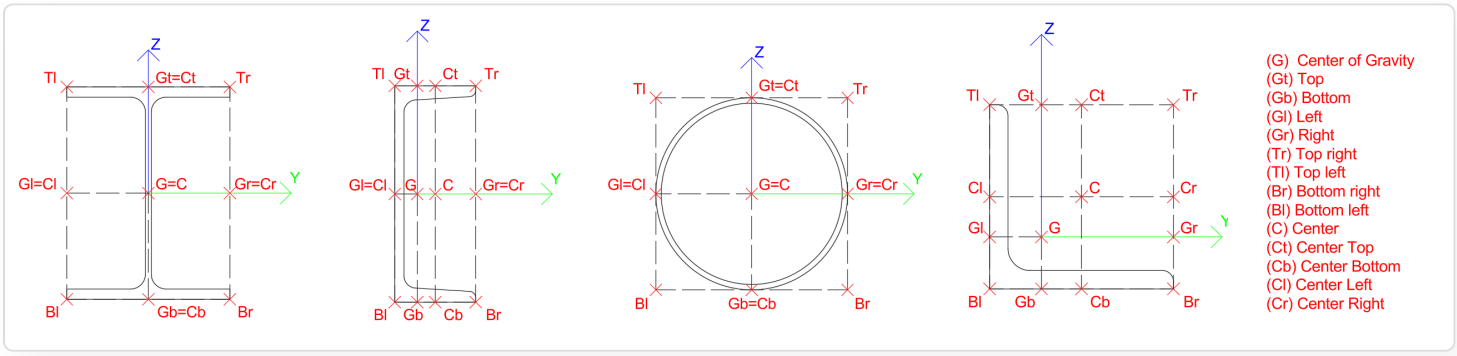


To align members, select the members that should be aligned together and choose the **Insertion point** option, to align, e.g., to the top surface, center of gravity, left surface, etc.




The **Insertion point** allows the selection of multiple points over the cross-section of the member. The FEA programs usually use shortcuts for the insertion points, which are translated in the Checkbot list as follows:

| | | | | | |
|---|---|---|---|---|---|
| G | Centre of Gravity | Tr | Top right | C | Centre |
| Gt | Top | Tl | Top left | Ct | Centre Top |
| Gb | Bottom | Br | Bottom right | Cb | Centre Bottom |
| Gl | Left | Bl | Bottom left | Cl | Centre Left |
| Gr | Right | | | Cr | Centre Right |




This feature might be limited for FEA software where the BIM link with IDEA StatiCa is developed by the FEA producer or third party – check the supported integrations list.

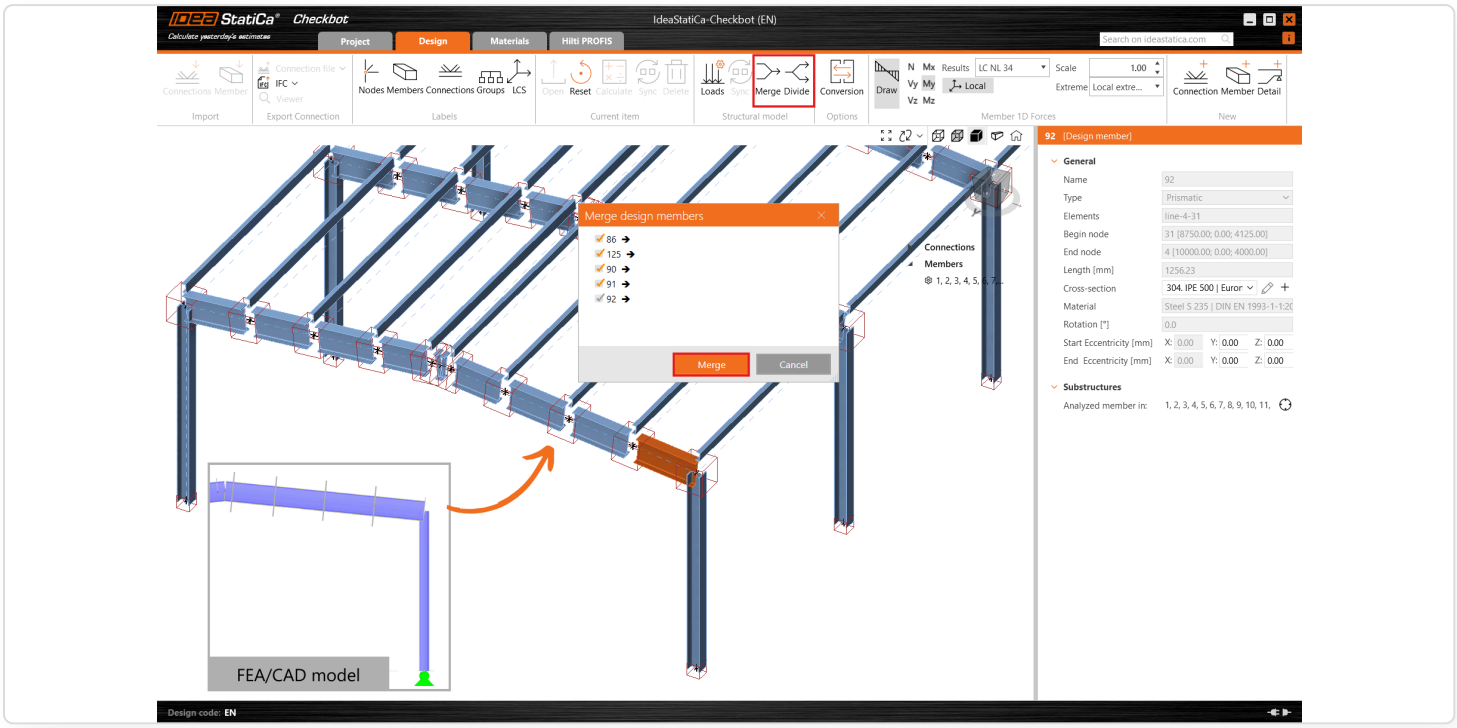
*Released in IDEA StatiCa version 25.1.*

## Merge members

In some cases in the analysis software model, a member might be split into multiple members but in reality, it is one continuous member. For these cases, the **Merge** functionality can correct it. Right-click one of the split members you want to merge and tick other members that you want to connect. Please be aware, that only members aligned along

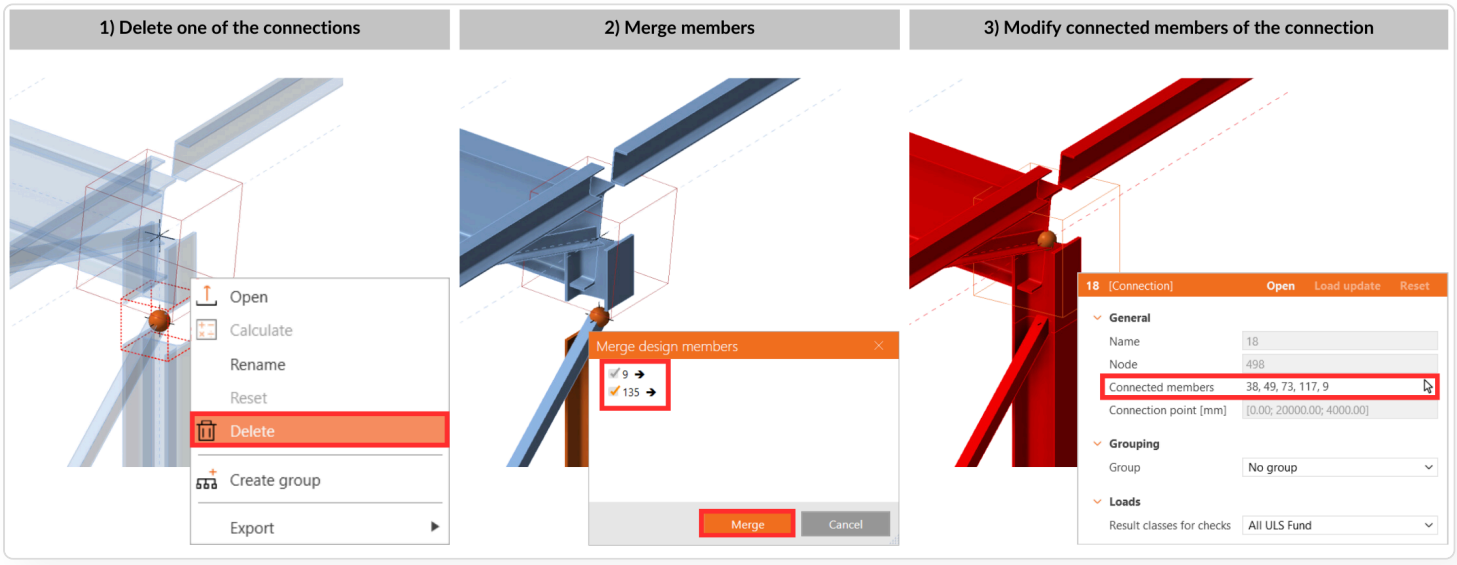
the same X vector are able to be merged.

Additionally, if you need to undo a merge, the **Divide** button allows you to separate any members that have already been merged back into their original state.




## Merge connections

Connection arrangements with close structural joints that should be analyzed in one connection model might be imported as two separated connections, typically appearing for eccentric diagonals.

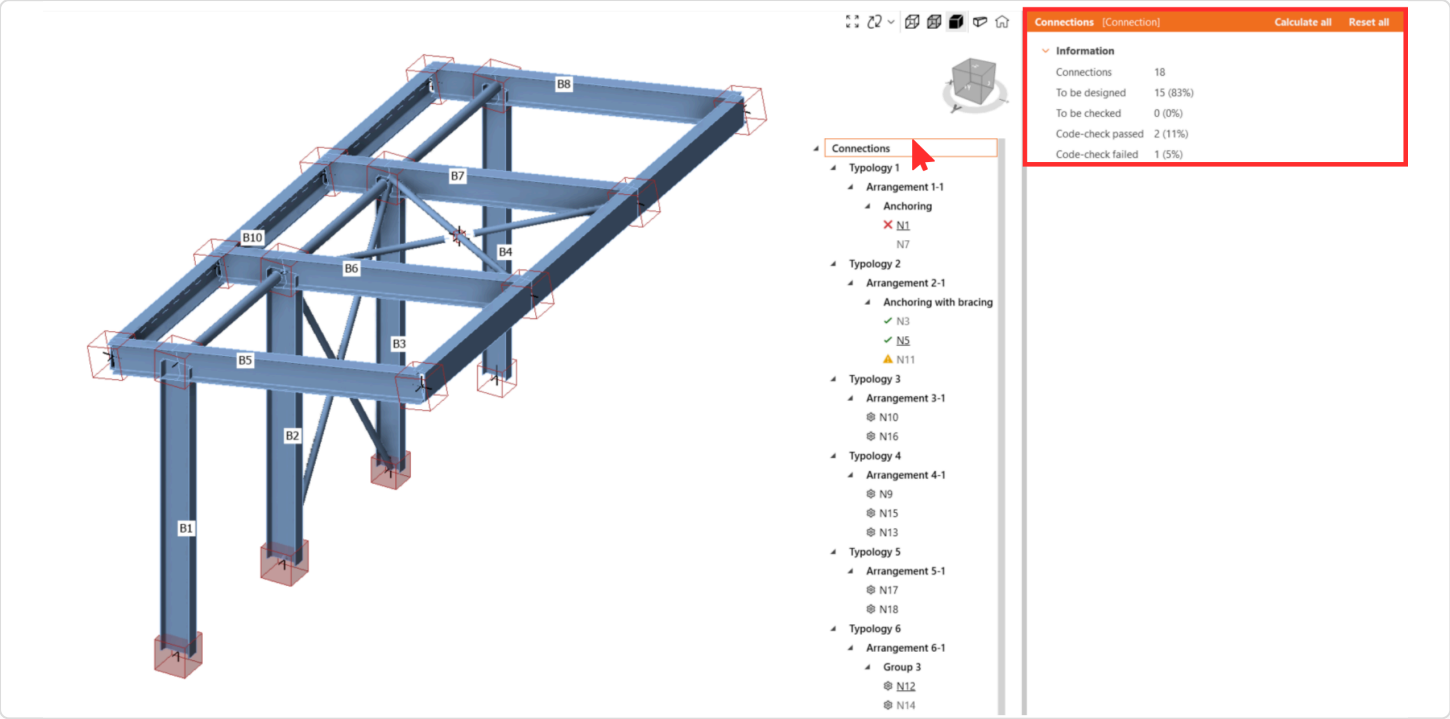



Delete one of the connections with a right-click and add members separated by the deleted connection. Now you need to modify the connected members of the analyzed connection. Click the mouse icon, select all members relevant to the connection (highlighted in red), and confirm the changes by clicking the tick box.

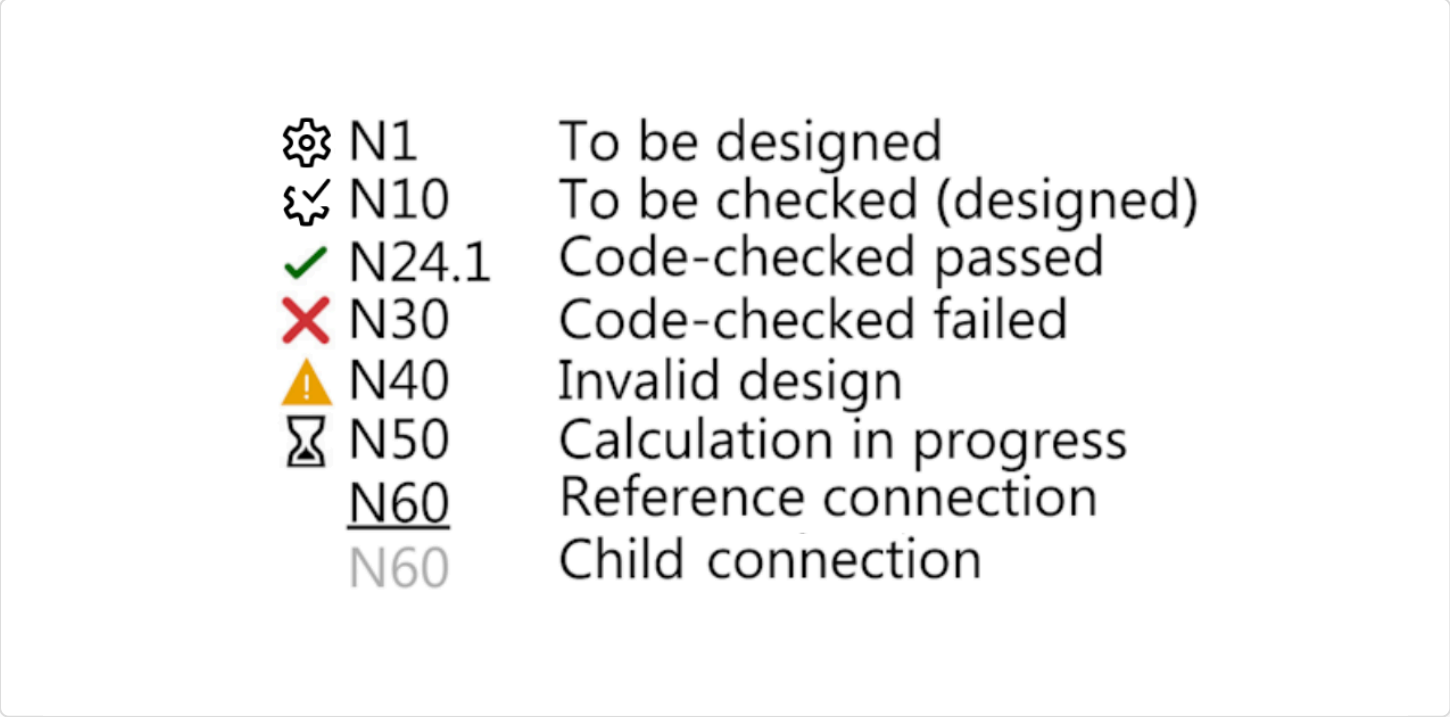
## Connection manager

For an organized overview of the project, Checkbot offers a summarized project status displaying the total number of connections, how many still need to be designed, and the number that has been checked. It also shows how many connections have passed or failed the code check, giving users a comprehensive view of the project's progress at a glance.

To access the overview, click **Connections** in the tree and all data will be displayed.




Furthermore, every connection is marked by the following icons to provide a quick overview.

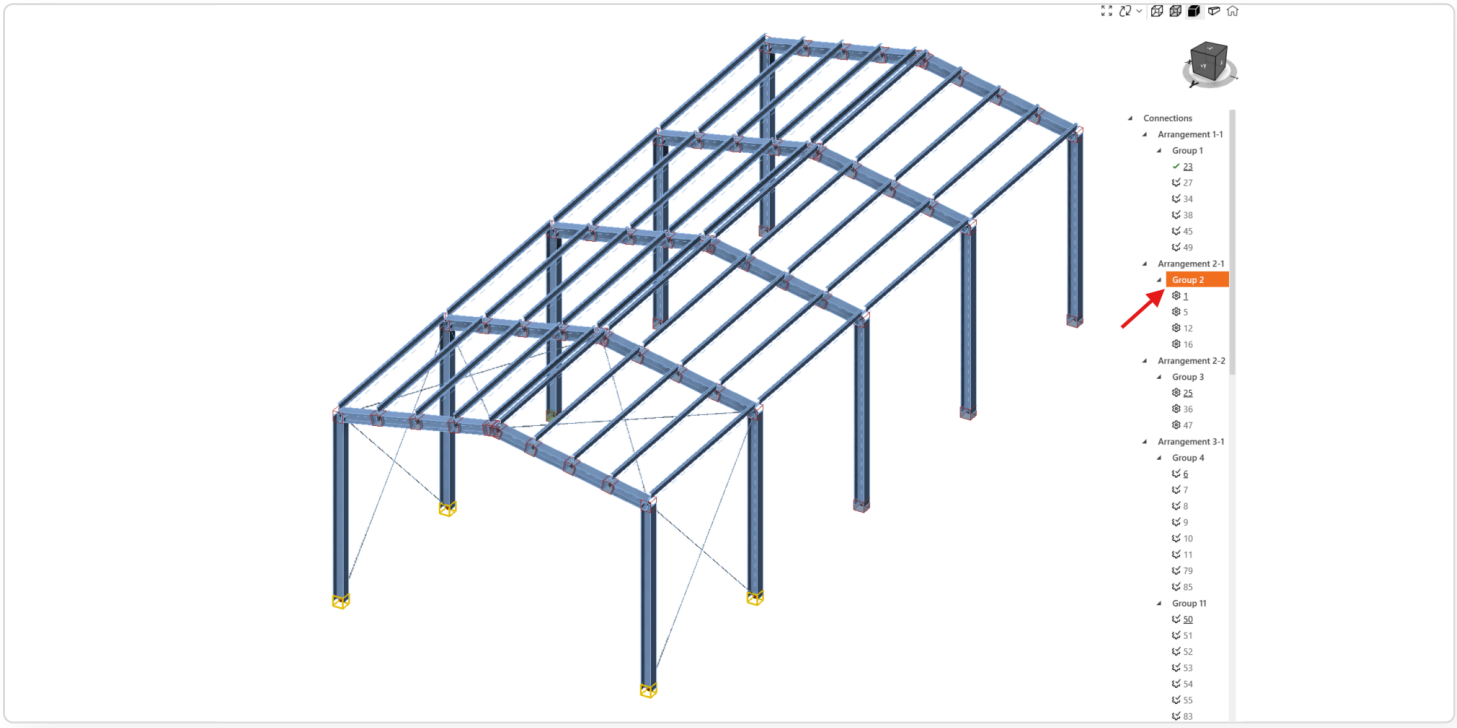



- Cogwheel – more inputs needed (geometry, load cases, …)
- Cogwheel with tick – ready to be calculated
- Green tick – calculated and code-checks passed

- Red cross – calculated but code-checks failed
- Invalid design – with a sync or update of the structural model, the child connection no longer meets the reference connection's parameter

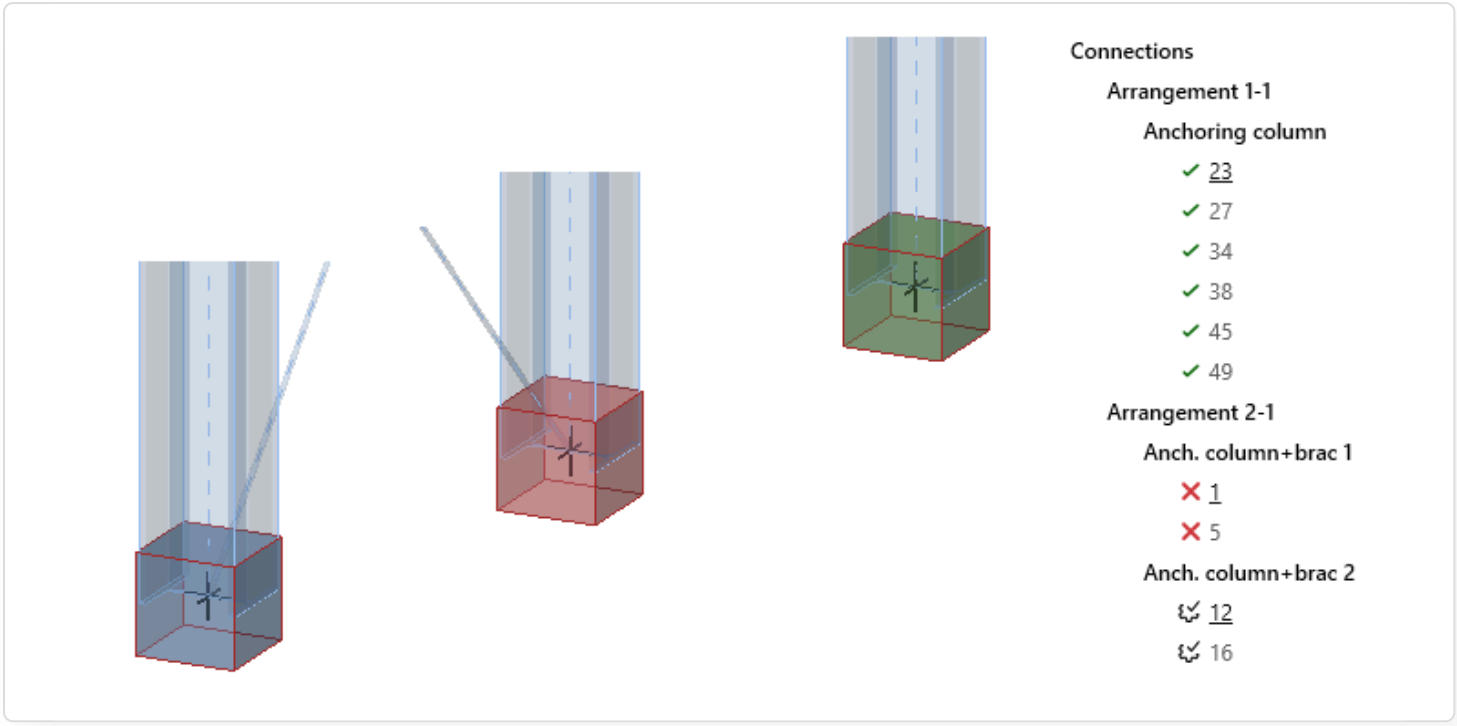
## Colors of boxes in the 3D-Scene

All connections related to the group are marked **yellow** if the group is selected in the tree.




Status of the connection is visible not only in the tree as mentioned above, but also directly in the 3D scene, and it is represented by the following colors:

- **none** - to be designed
- **green** - calculated and code-checks passed
- **red** - calculated but code-checks failed
- **blue** - ready to be calculated

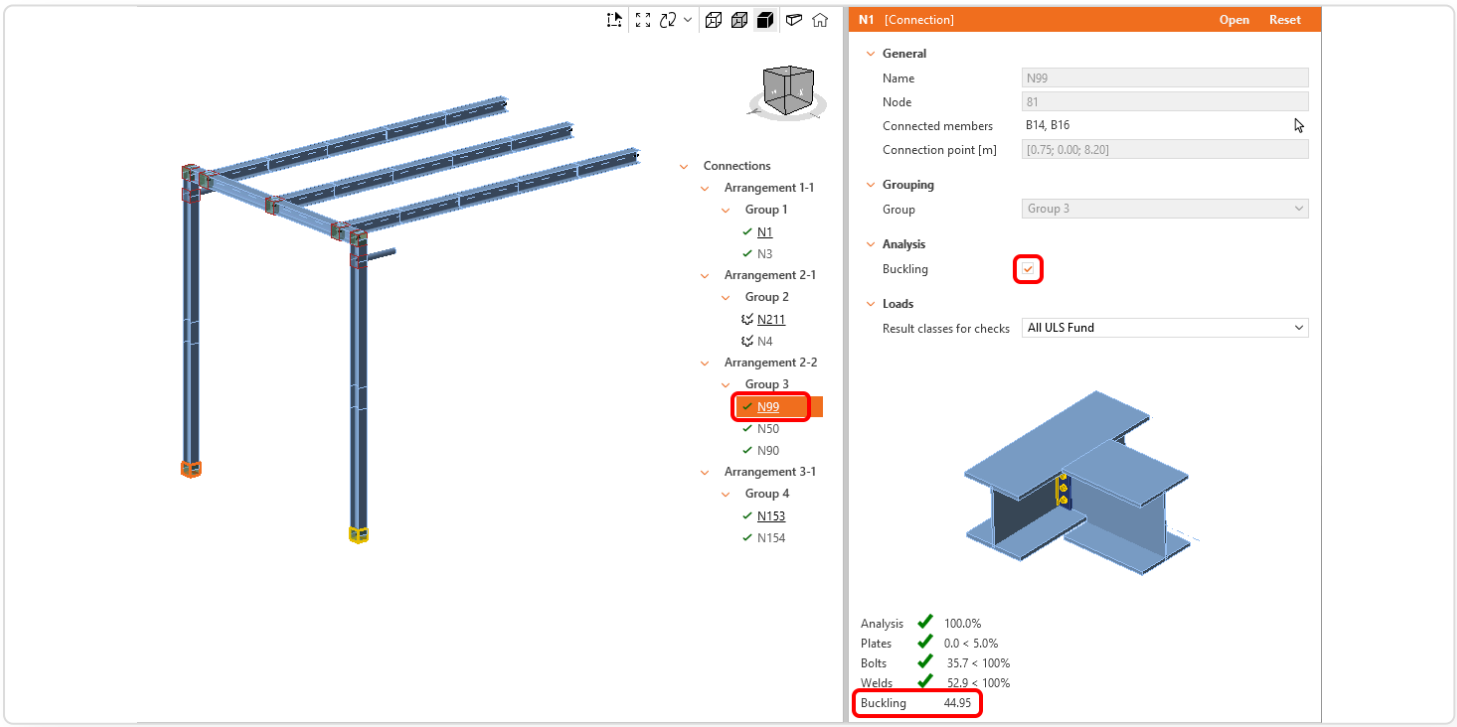



*Released in IDEA StatiCa version 25.0.*

## Calculate buckling analysis for all connections

Buckling analysis can be calculated in bulk for selected connection groups. By default, the buckling analysis is turned off. To turn the buckling analysis on, select the **Reference connection**, and check the **Buckling** combobox.

The lowest buckling factor is displayed in the overall results tab in the Checkbot window and also added to the report.




*Released in IDEA StatiCa version 25.1.*

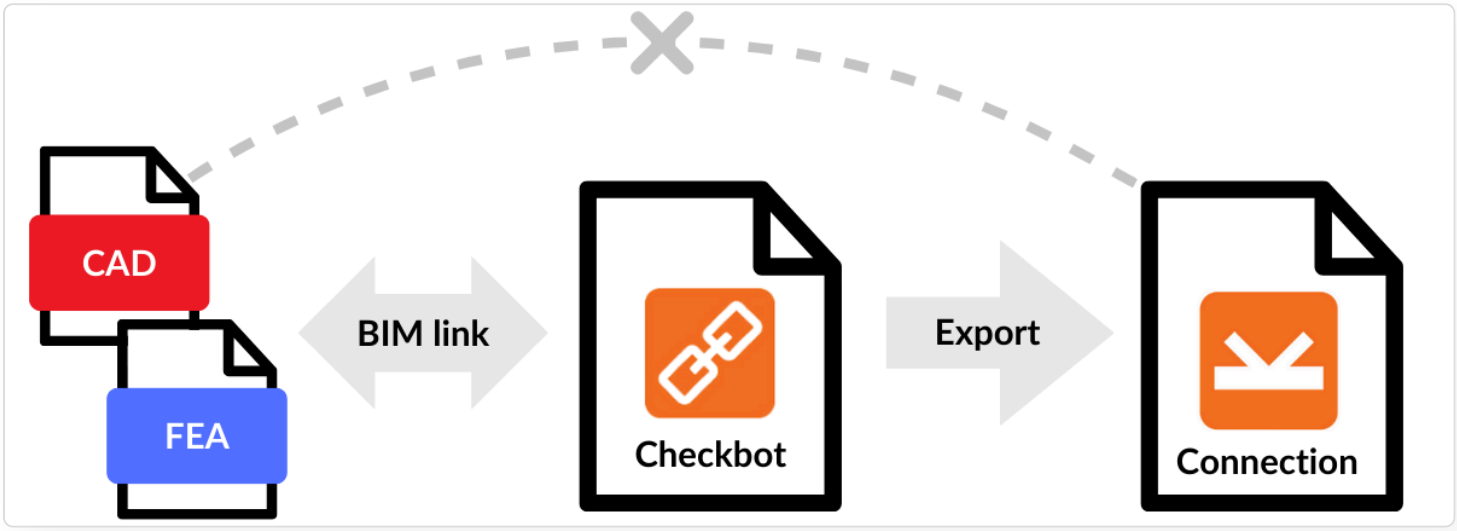
## How to make Checkbot even faster

Once the Connection application window is open, do not close it. Simply return to the Checkbot application and open a new connection, this will load much more quickly.

## Export to Connection

When Checkbot is linked to third-party software via a BIM link, it does not allow you to edit the properties of a member. If such changes are necessary, you can easily export the model to IDEA StatiCa Connection independently from the synchronized Checkbot model. You can export either only one connection or select multiple connections and save them all in one file, enabling you to print all reports of modelled connections at once.

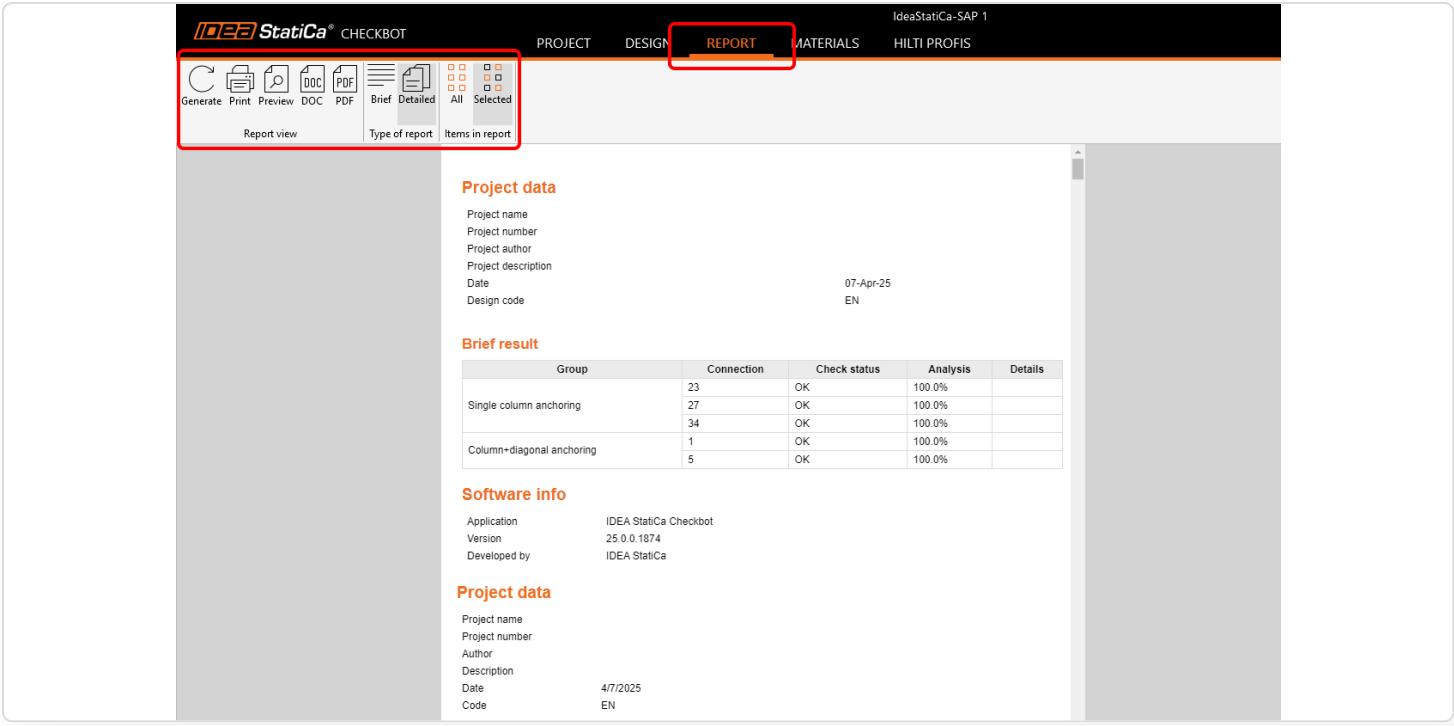
Once the separated project files are exported, they are no longer linked to the original Checkbot structural model. Therefore, no further synchronization with the third-party model or their management in Checkbot is not possible. Changes in member properties are enabled.




*Released in IDEA StatiCa version 25.0.*

## Bulk Report

The report can be generated for all connections in the project at once or for selected connections only.

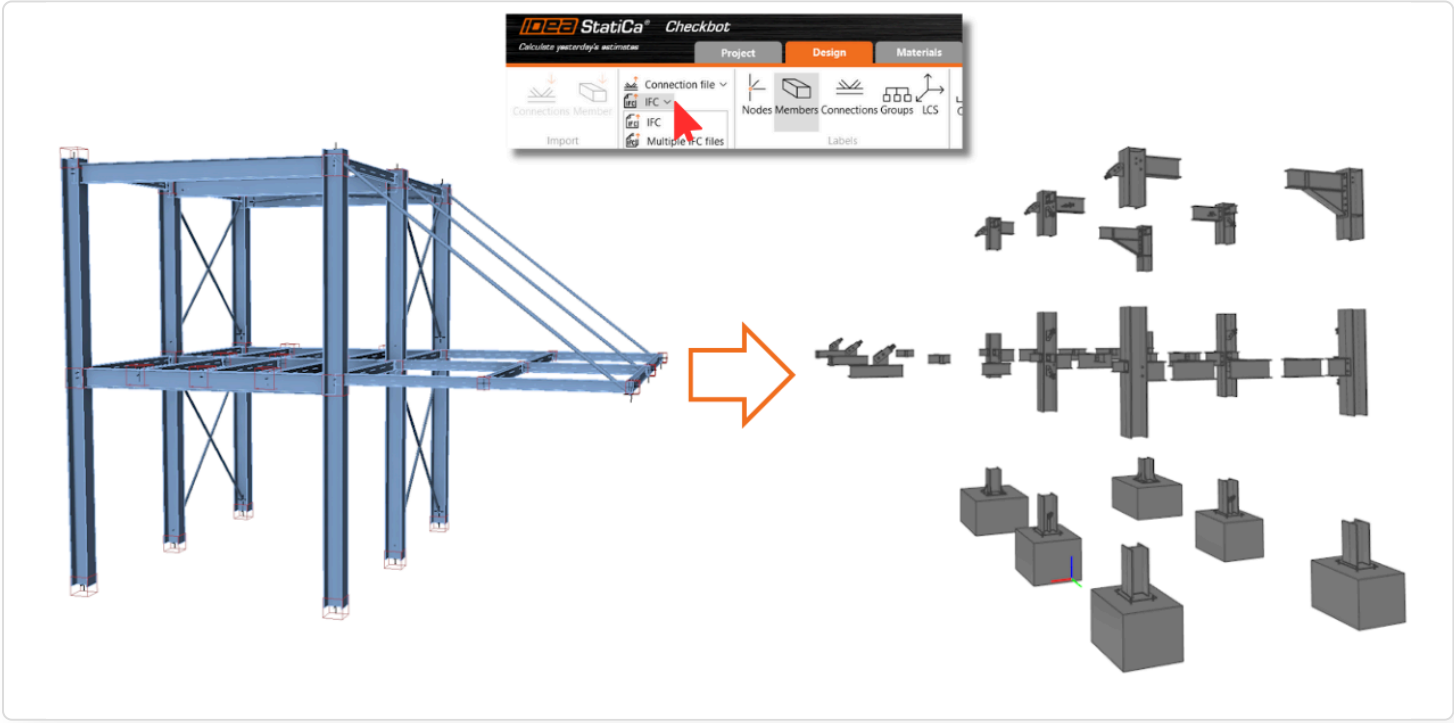



How to adjust the report accordingly is explained in this article.

## Export to IFC

The Industry Foundation Classes (IFC) format is an open data, vendor-neutral format that enables the sharing of data. Checkbot allows you to export all selected connections to one IFC model or export connections to individual IFC files to a specific folder.

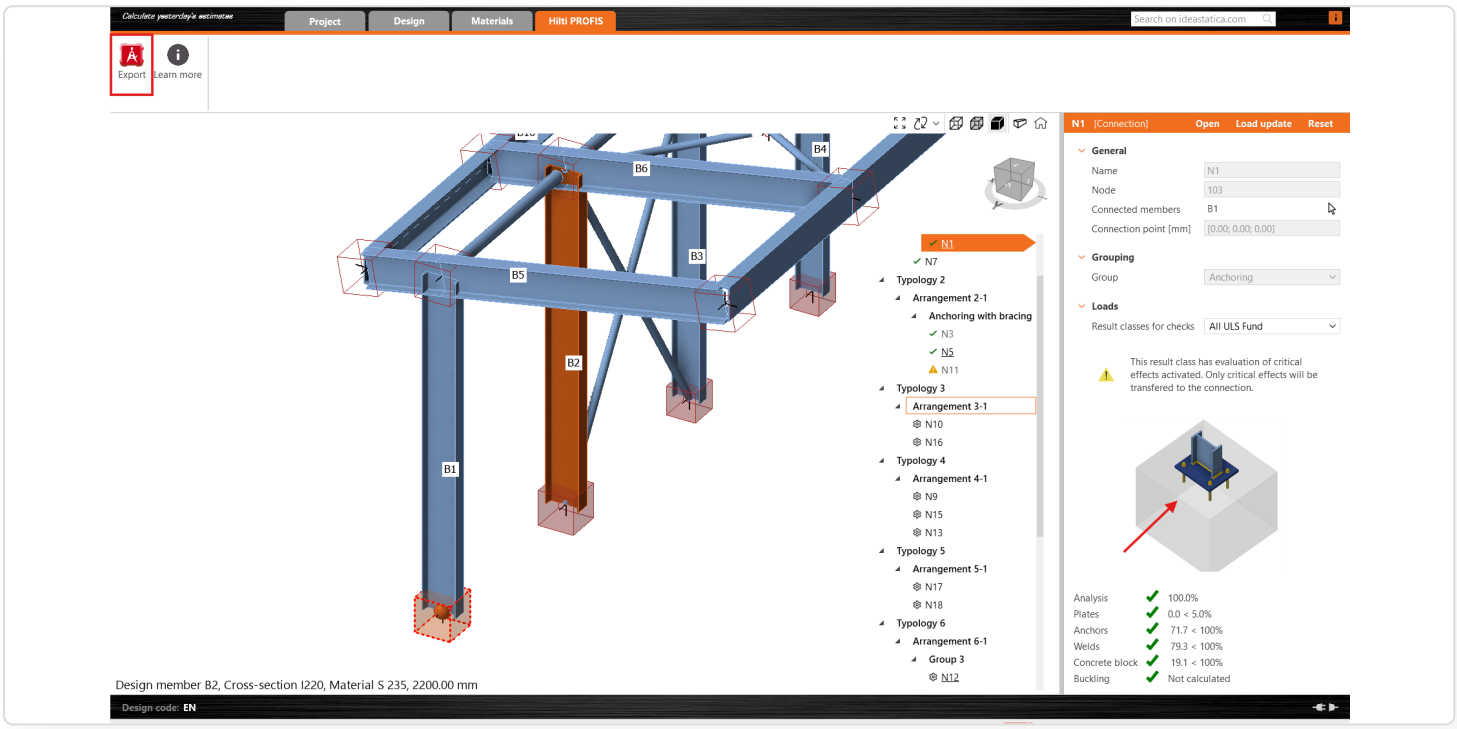
The export contains the global coordinates of the Connection Point – the real location of a connection in a project.



## Export to Hilti PROFIS Engineering

**Hilti PROFIS Engineering Suite** is a user-friendly, cloud-based software for designing and analyzing anchors. By selecting a node with one anchored member, users can export the data directly to Hilti PE using the **Export** button, ensuring the accurate transfer of relevant structural data for further analysis.

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.



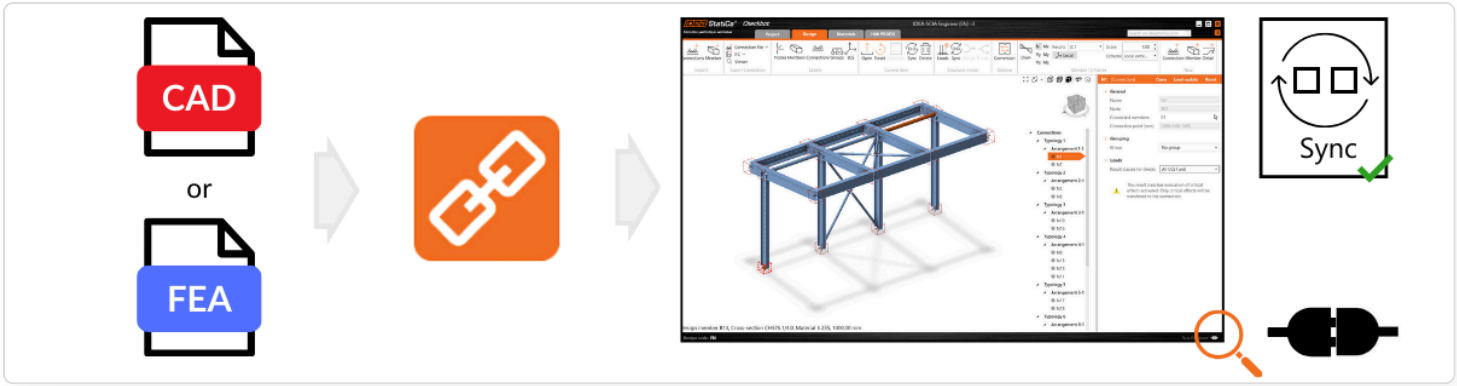

## Sync

After importing all necessary items, the **Sync** button lets you easily detect and apply changes from the source project to the IDEA StatiCa model. These changes may include updates to thickness, cross-sections, or modifications to the properties of welds and bolts. However, it's important to note that updates **cannot include new or deleted components** such as plates, members, repositioned elements, or load combinations. In such cases, you must delete the current Checkbot project (the folder) and import it again.

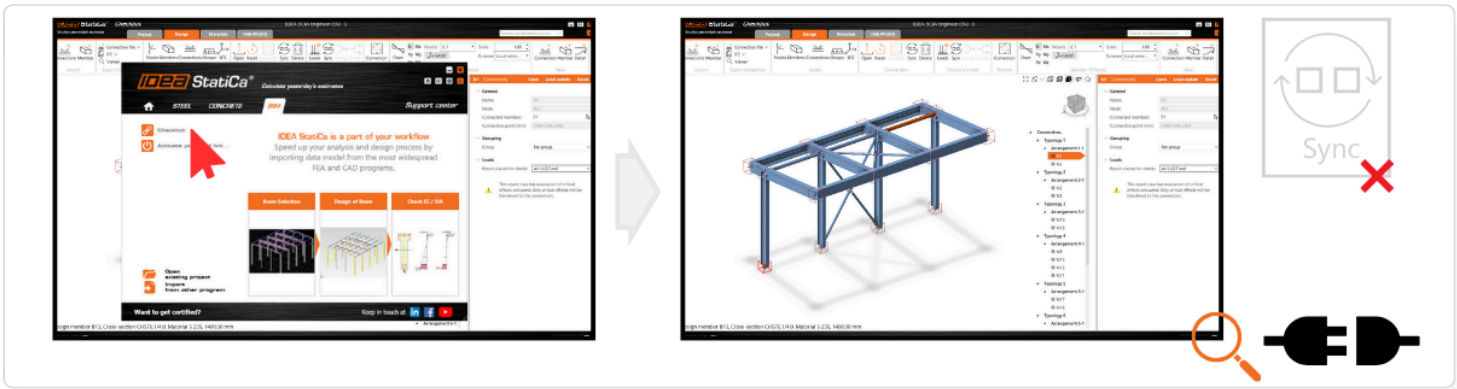
Please keep in mind that synchronization in IDEA StatiCa works one way, from the source program to IDEA StatiCa, not the reverse.

- **FEA solution** – sync does not affect design operations (cut, end plate, ...)
- **CAD solution** – sync does not affect load effects (except for Revit, which can hold the analytical model with results)

If any changes are made in the third-party software model later, you must re-open the Checkbot project and press the **Sync** button to ensure all updates from the third-party software model are reflected in Checkbot. The sync button is enabled only if the Checkbot file is open through the third-party application.




Remember, the Sync button is only active if Checkbot is opened through a BIM link from third-party software.




Be aware that any modifications made directly in IDEA StatiCa (e.g., to cross-sections, eccentricities, load combinations, or operations) will be overwritten during synchronization with the source project. The same rule applies if the eccentricities are modified in IDEA StatiCa.

For example, if you adjust a connection design in IDEA StatiCa Connection that was initially imported from Tekla Structures and then synchronize, your changes in IDEA StatiCa will be replaced by the latest design data from Tekla Structures.

*Released in IDEA StatiCa version 25.0, updated in IDEA StatiCa version 25.1.*

# Bulk workflows and critical load filter in Checkbot

The Checkbot app is ready to handle big projects in no time. Its ability to perform bulk actions, such as designing connections in a group with one report generation for the whole project or identifying critical load effects, makes it the most effective tool for designing standard connections.

# Calculate load extremes for all connections

In projects with many nodes and load combinations, similar connections with very similar load effects are analyzed repeatedly. To effectively reduce the calculation time, the **Calculate load extremes** algorithm identifies critical load combinations in a defined group of connections and speeds up the calculation time by up to 80%.

This function, together with the bulk group actions in Checkbot, enables users to design all connections in a project with minimal time. After that, users can deactivate the function and calculate all connections with all load effects as the final check.

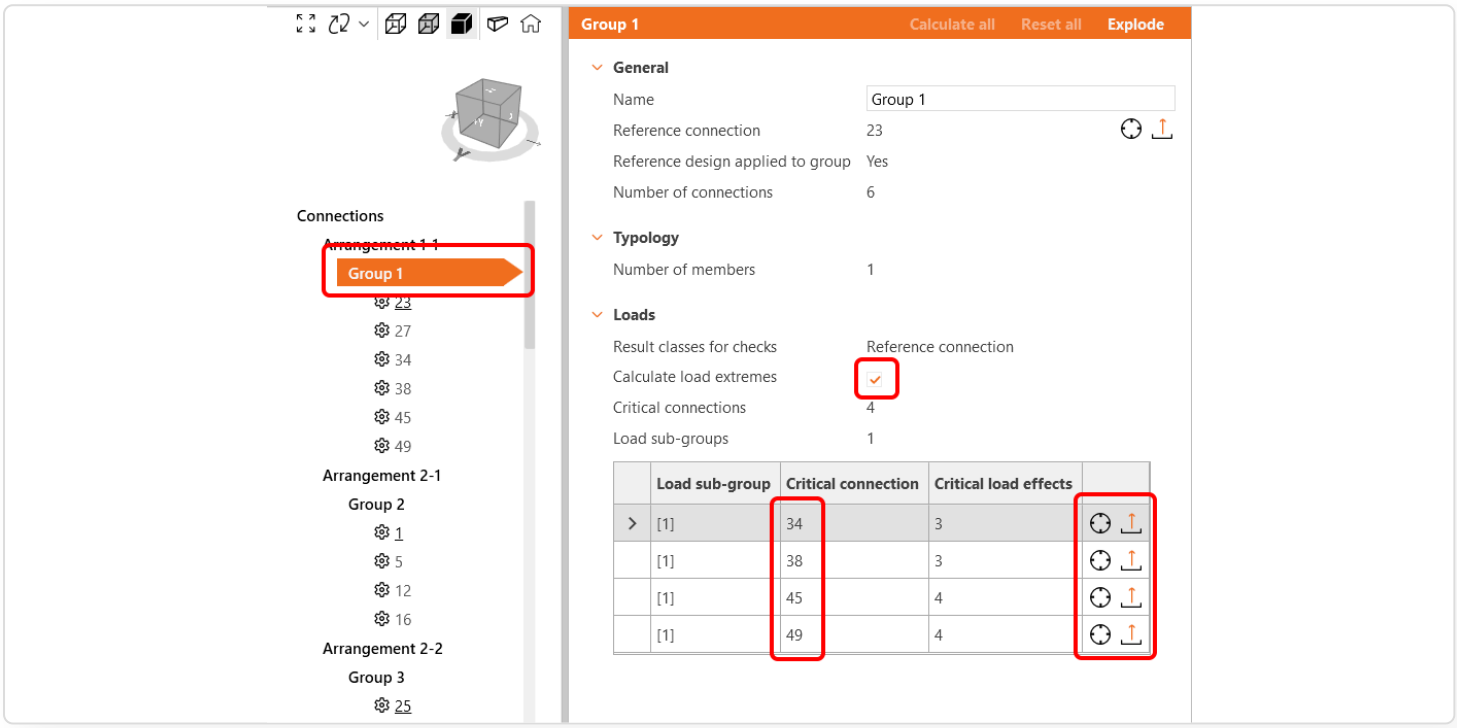
The Calculate load extremes functionality is available only for Checkbot projects imported from FEA models, since CAD models usually don't contain any load effects.

## How to use the Calculate load extremes function

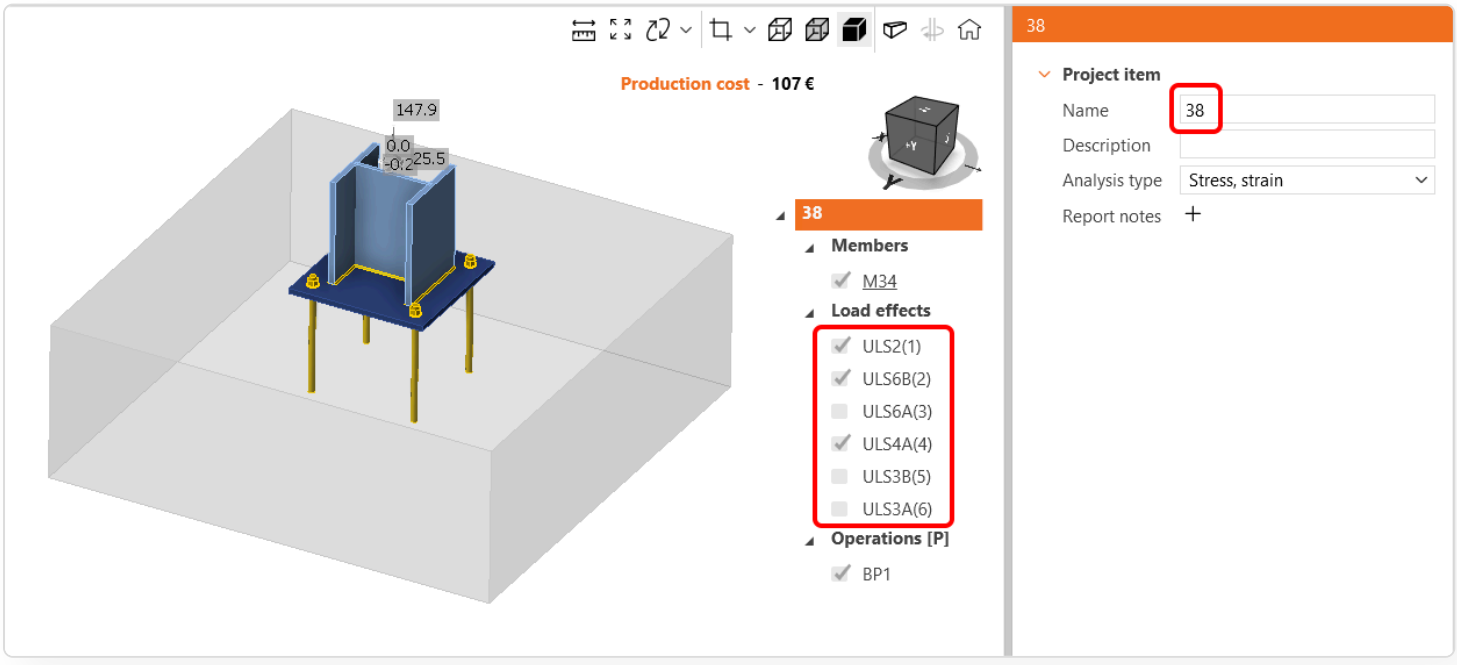
Calculate load extremes represents the Checkbot bulk variant of the function Calculate load extremes, which is available for single connections in the Connection app.

First, for an automatically or manually created group of connections, design the reference connection.

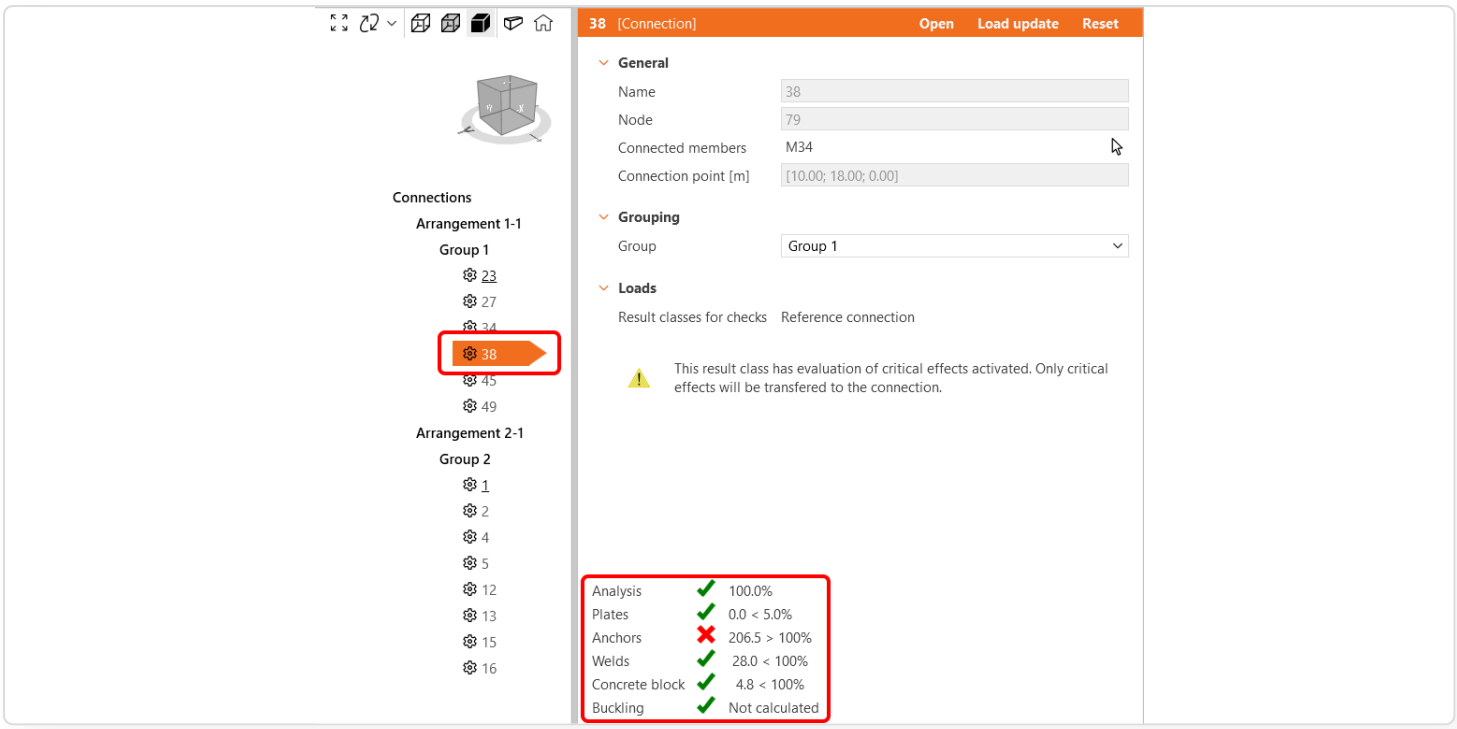
Then select the group and tick the **Calculate load extremes** combo box. This runs the algorithm, and a tab appears showing only connections containing the critical load effects.



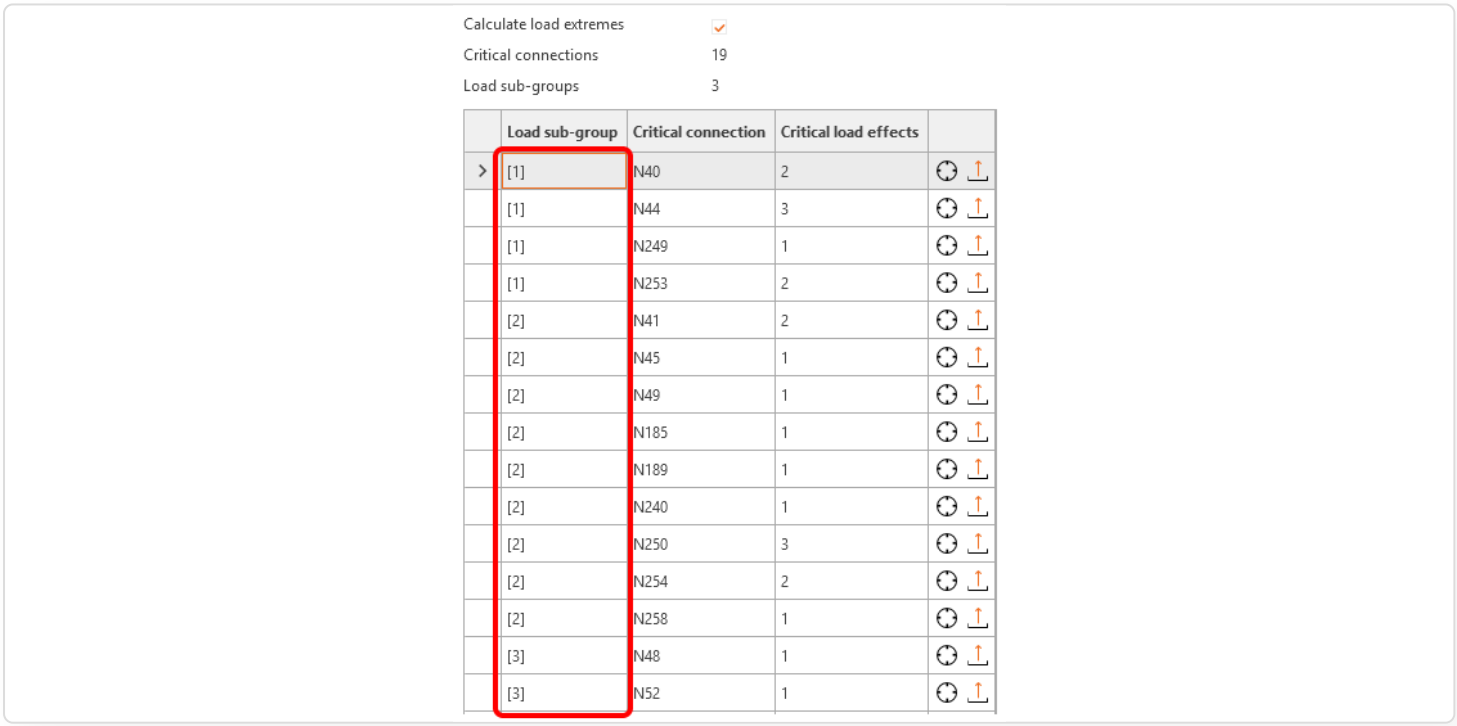

Now, only the critical load effects detected in several connections can be calculated. All other load effects in this group are disabled (for some connections, all load effects are disabled).



Calculate the group of connections and browse the results for critical load effects. Optimize the design of the reference connection or even reshuffle groups to finish with the best result.




If the Calculate load extremes function is applied to a group of nodes where a member has more cross-section variants (e.g., a group of 10 nodes where a column has cross-section HEB300 in 6 nodes and HEB340 in 4 nodes), then more Load sub-groups are created to cover all the variants and stay on the safe side.




Limits for the Calculate load extremes function are currently not editable and are set to 0.1 for all internal forces.

**Load extremes limits**

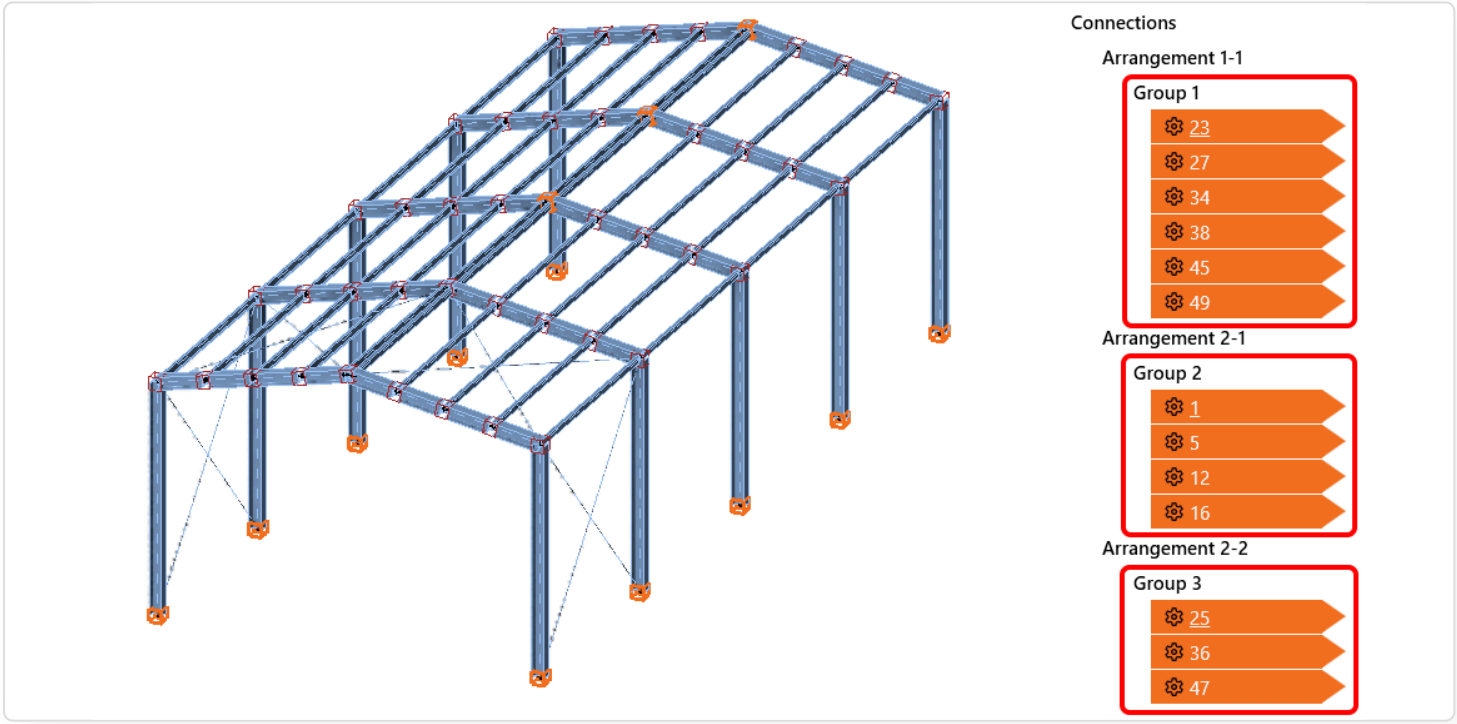
| | |
|---|---|
| Normal force N [kN] | 0.1 |
| Shear force Vy [kN] | 0.1 |
| Shear force Vz [kN] | 0.1 |
| Torsion Mx [kNm] | 0.1 |
| Bending moment My [kNm] | 0.1 |
| Bending moment Mz [kNm] | 0.1 |

# Easy work with groups

A series of improvements is ready to boost your experience when working with multiple connections and groups in Checkbot.

## Automatic group creation

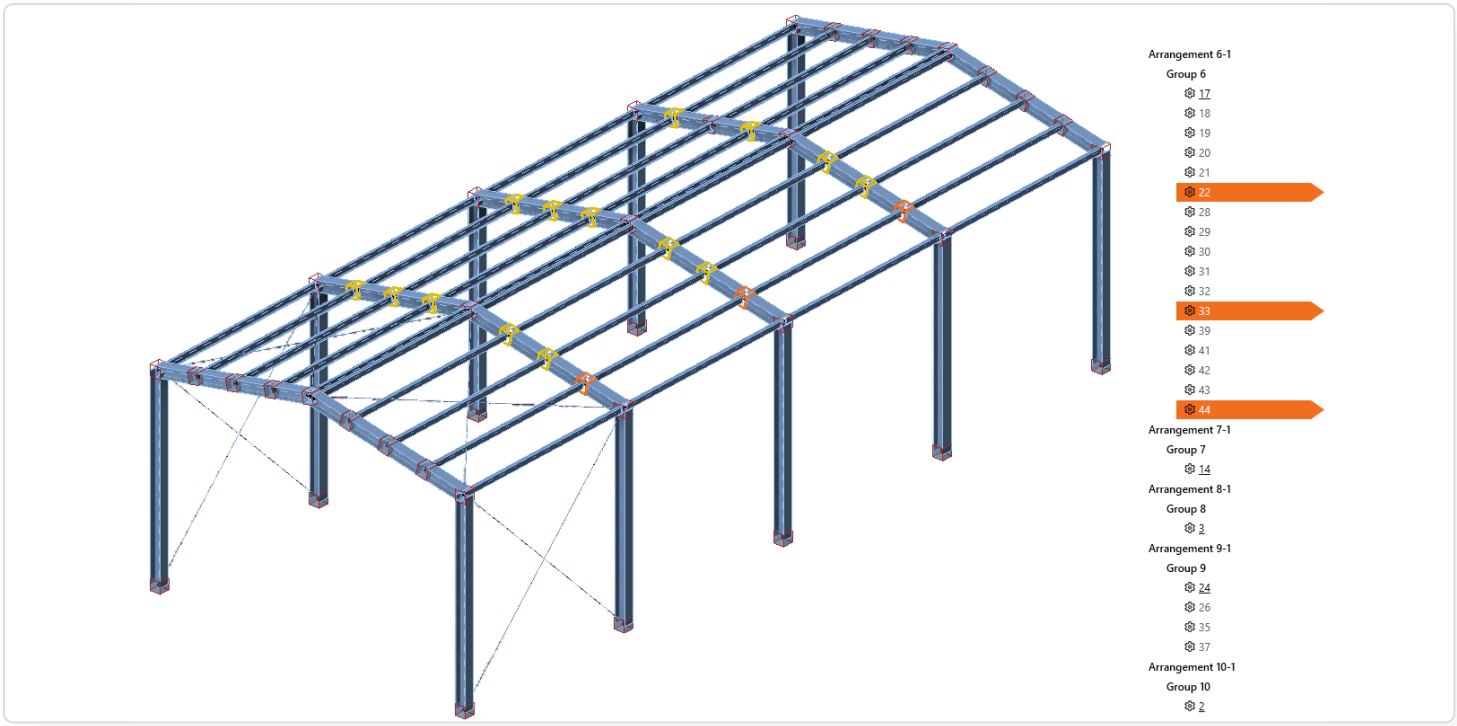
When the first bulk import of multiple nodes from the FEA global structural model into Checkbot is performed, nodes are sorted into **Arrangements** based on their geometry, and the **Groups** are created automatically. For each **Group**, a **Reference** connection is set.




If there are any nodes imported later on, those are not inserted into any Group but are listed at the bottom of the appropriate Arrangement, ready to be manually sorted into existing or new Groups.

## Connection group highlighted

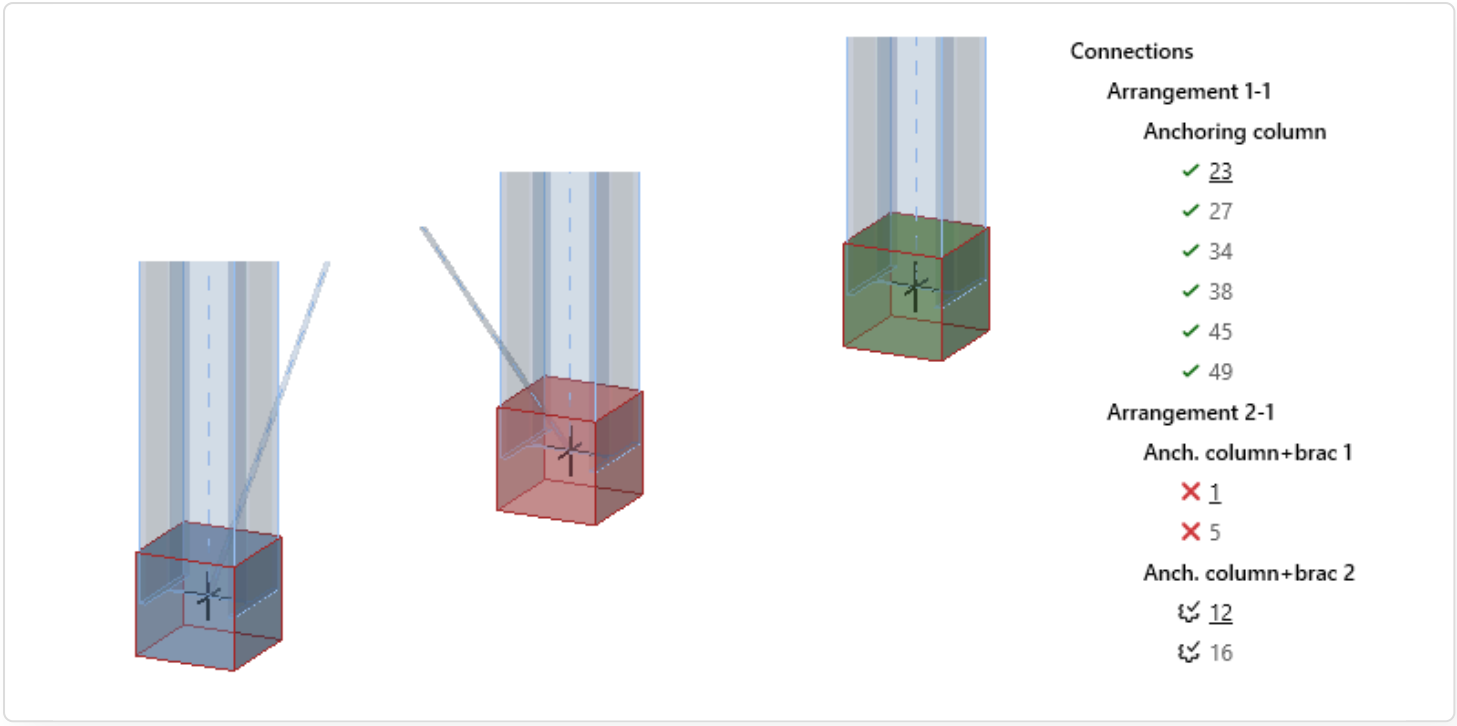
When a Group of nodes is selected in the tree list, all nodes in the group are highlighted in the scene with a yellow contour line. When a node is selected, it is highlighted in the 3D scene with an orange color while the group highlight is still active.




## New tree and scene statuses

Statutes are added for easier navigation in the project so that nodes are distinguishable at a glance in both the tree list and the 3D scene. New statuses are:

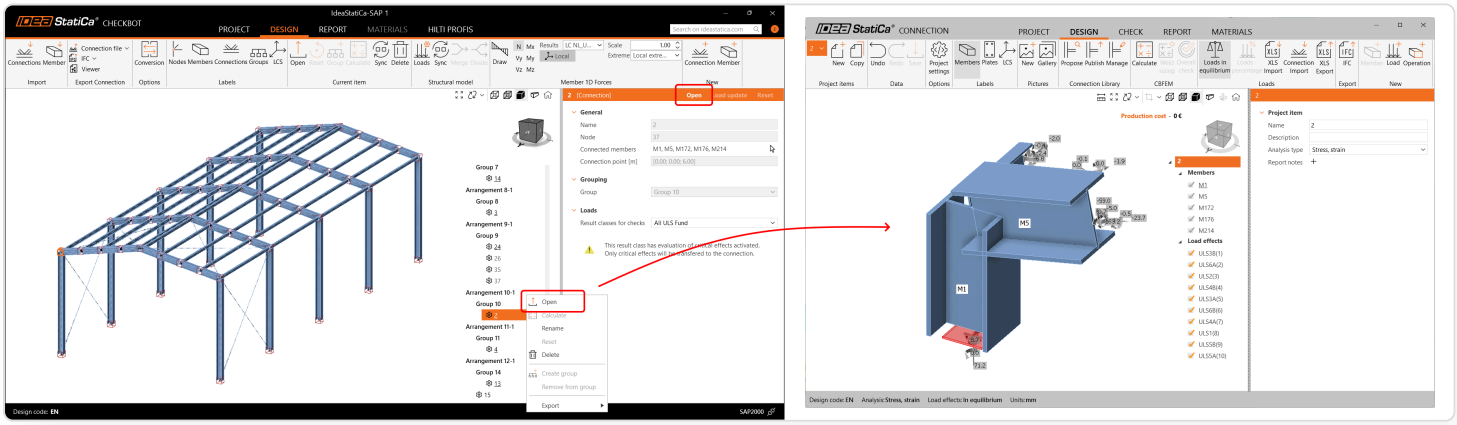
- passing the code checks – a green box in the 3D scene and a green mark in the tree list
- failing the code checks – a red box in the 3D scene and a red cross in the tree list
- ready for calculation – a blue box in the 3D scene and a marked cog-wheel in the tree list




## Smooth work with Checkbot and Connections

When a node is opened in Checkbot, a new window with the lite version of the Connection app is displayed. The Connection app can stay open, and it refreshes when you open other nodes from Checkbot, thus, you save time from repeatedly opening and closing the Connection app.

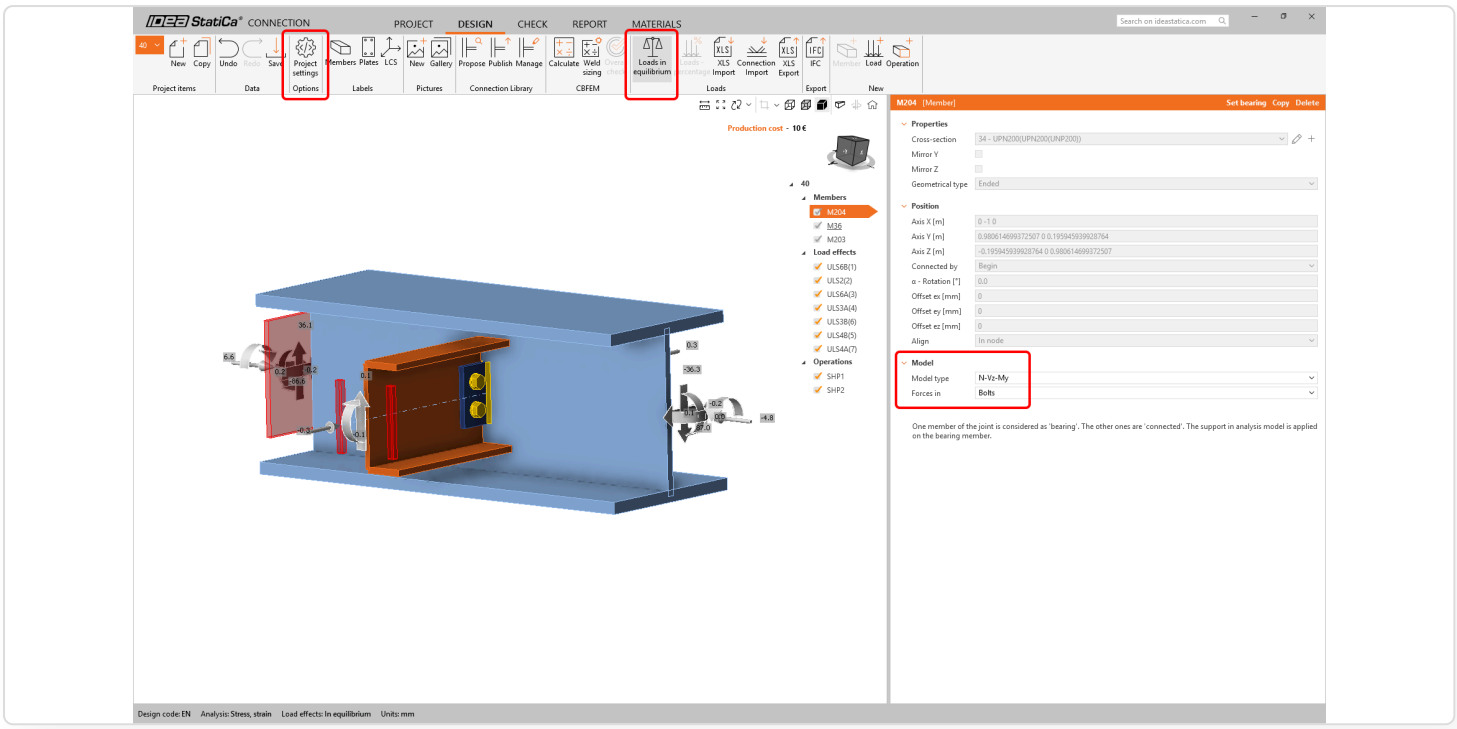
When another node is opened, the previously displayed node in the Connection app is automatically saved and dismissed.



# Consistent settings of Reference and Child connections

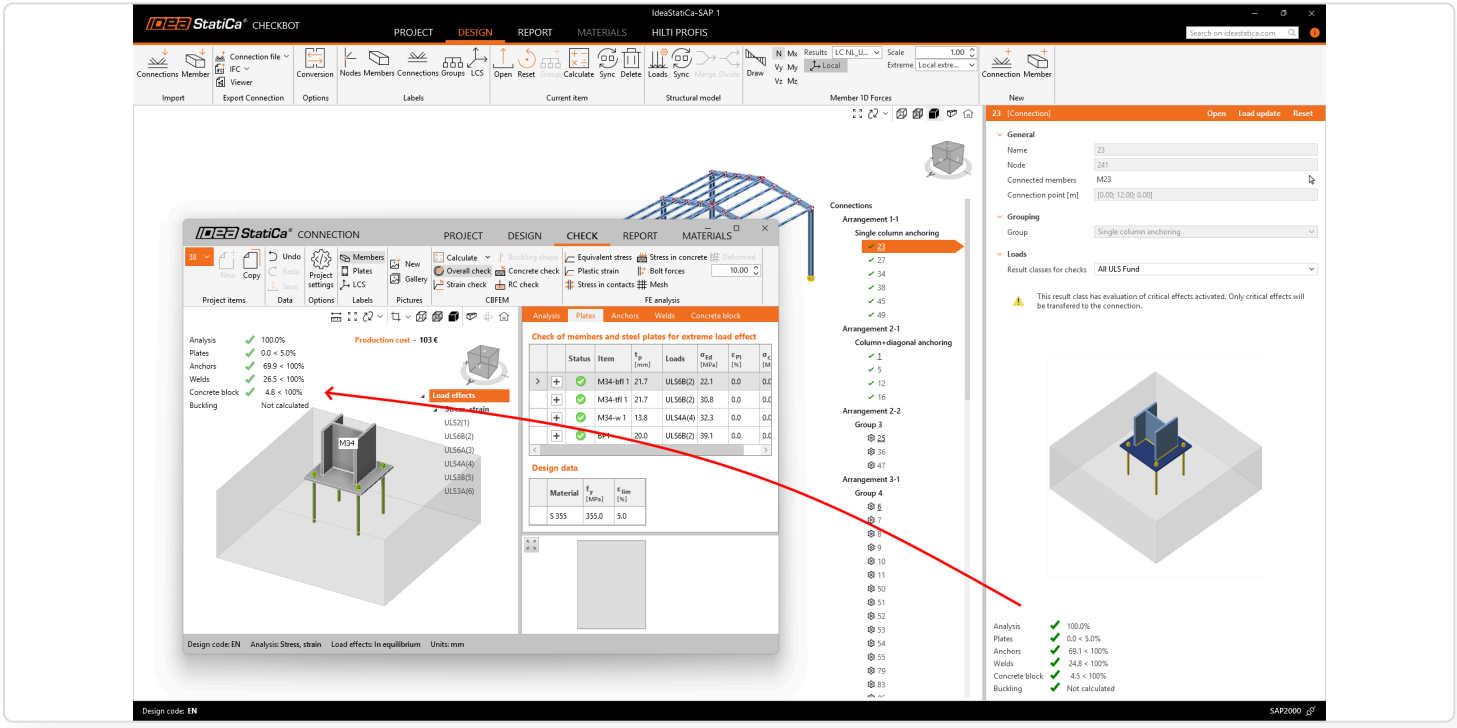
All settings applied to a Reference connection are automatically applied in bulk to all Child connections in the same group. This prevents time-consuming adjustments for nodes one by one. This includes:

- Project settings (gamma factors, detailing check, cost calculation, etc.)
- Member settings (Model Type and Forces in, including position and connected member face)
- Bearing member
- Load in equilibrium




# Calculation results stored in Checkbot

The results calculated for connections and members within a Checkbot project have been merged into one database persistently stored in the project folder. All apps interacting with the project share this one result database.




This prevents situations when the results and code-checks from already completely calculated projects in Checkbot disappeared and had to be recalculated, or situations when results in Checkbot were missing when opening nodes in the Connection app. The same mechanism applies to members opened in the Member app.

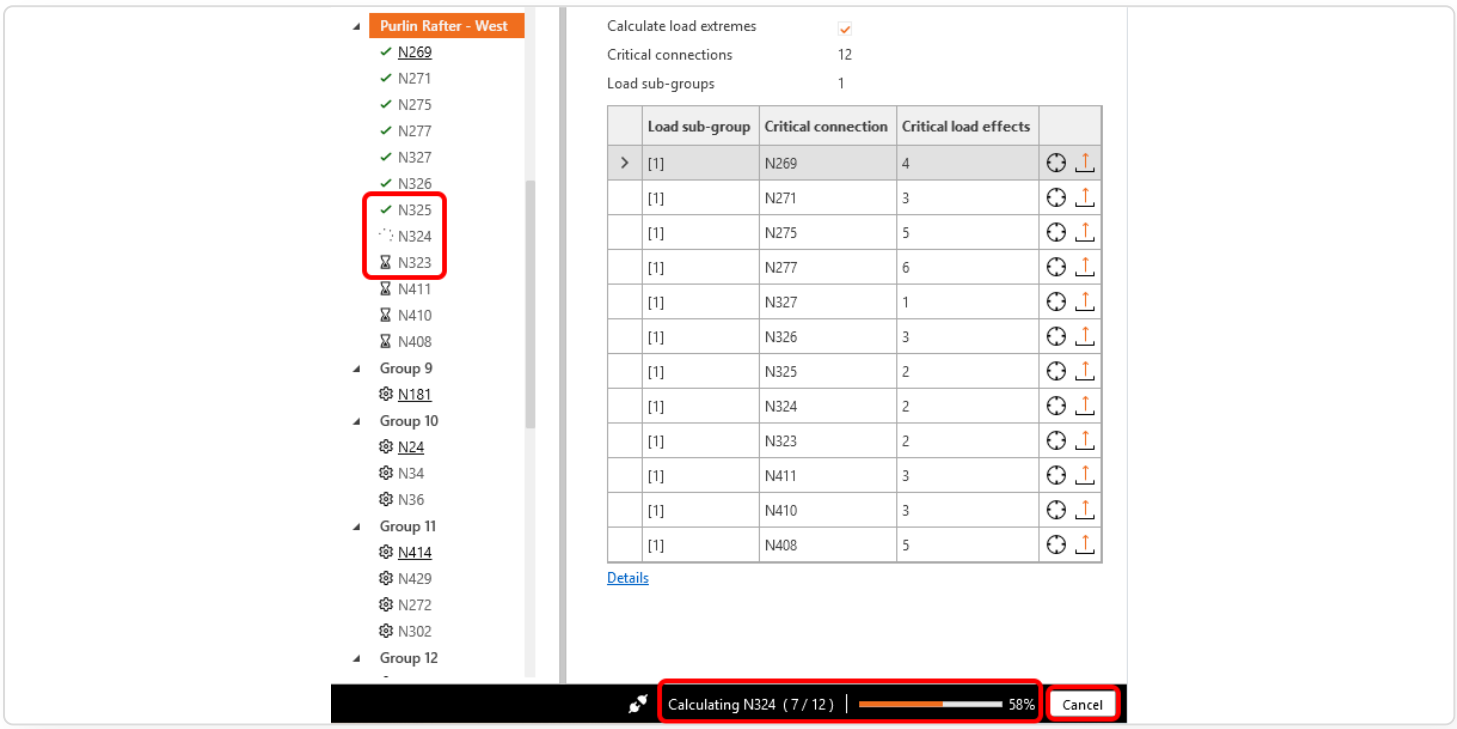
*Released in IDEA StatiCa version 25.0.*

# Bulk calculation progress bar and cancel

When calculating multiple connections in Checkbot, users can track the status on the right side of the application footer. Here, the **currently analyzed connection** is displayed together with the **number of connections** already calculated and the **percentage** of the overall progress. The percentage step is per connection, e.g., for two connections, the steps are 0%, 50%, and 100%.

In the tree list, the currently calculated connection is tagged with the **spinner icon**, and the connections waiting for the calculation are tagged with the **hourglass icon**.

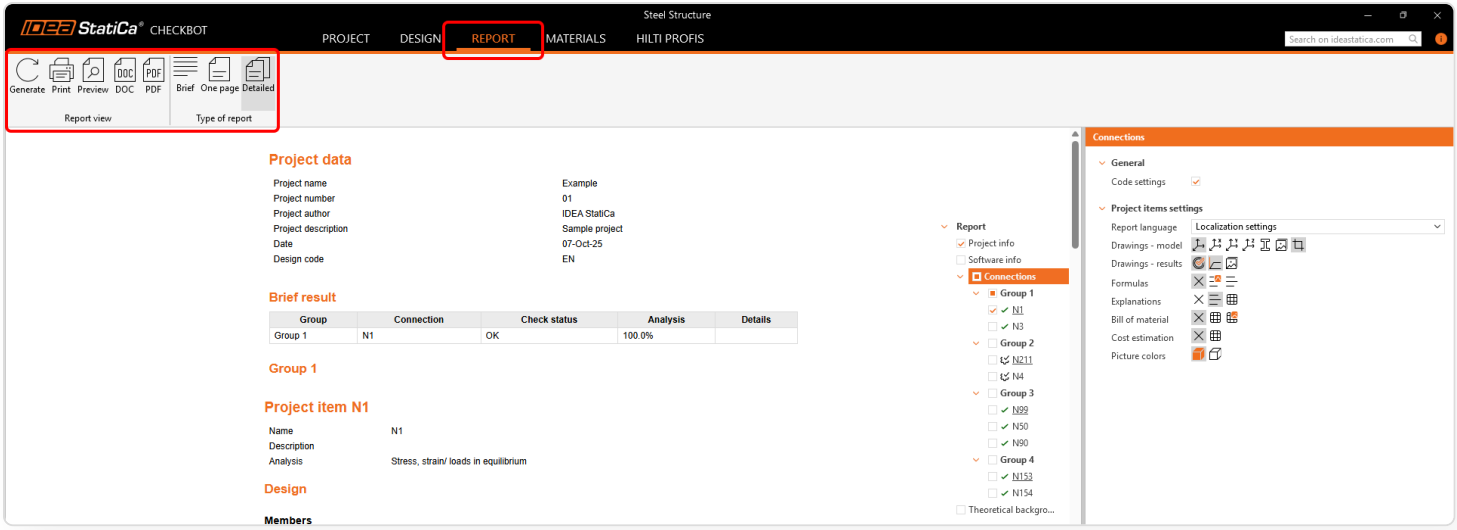
The **Cancel** button terminates the calculation of the current connection and all the connections waiting for the calculation.




*Released in IDEA StatiCa version 25.1.*

# Bulk report for all connections in Checkbot

The **Report** tab is available in Checkbot in the same format as in the Connection app. The report can be generated for all connections in the project at once or for selected connections only and can be saved in PDF or DOC format for further editing or printed out directly.

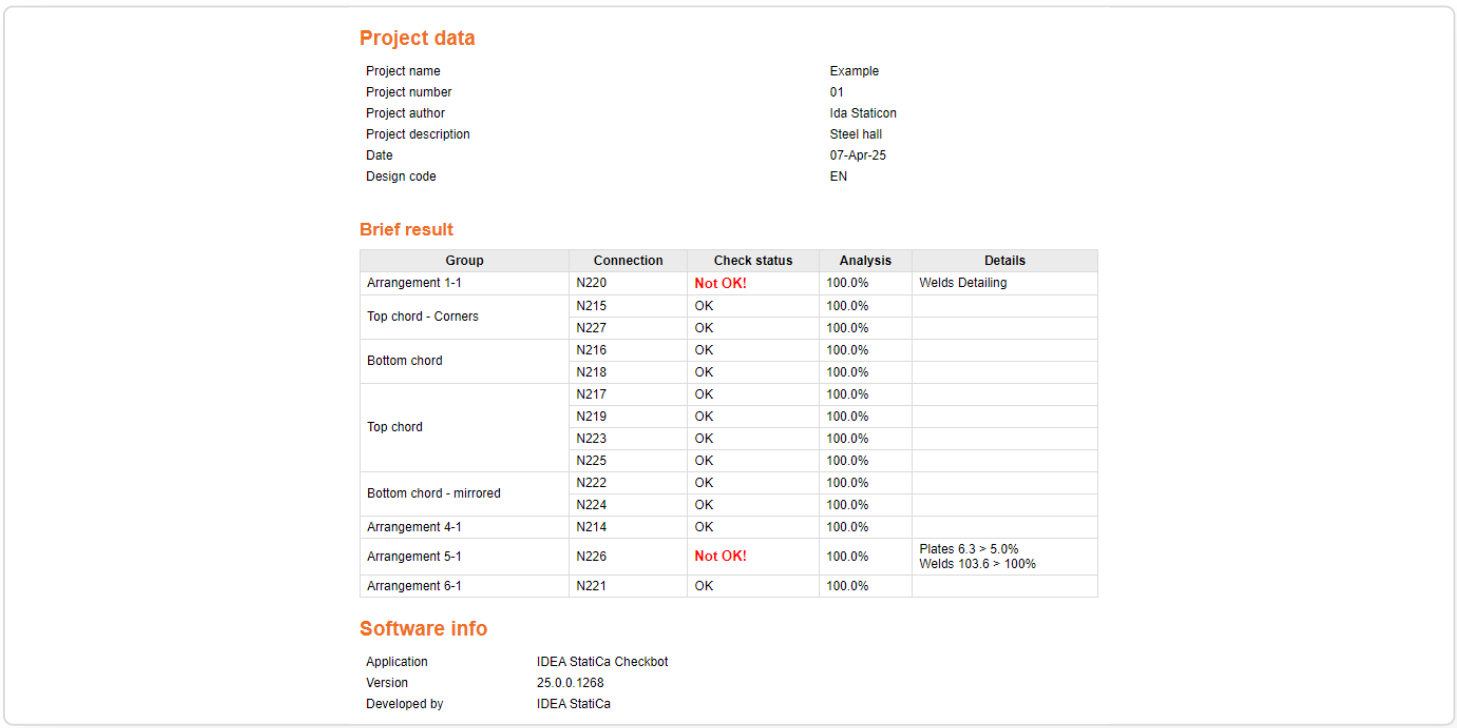



When the **Generate** command is executed, Checkbot also calculates all the selected non-calculated connections, and the report is composed of the single connection reports one by one. Depending on the size of the project, this might take several minutes for the detailed type of report.

## Types of report

There are three variants of the report per the desired level of detail:

- The **Brief** type of report is a super short version that includes the project description (defined in the backstage menu) and the simple results tab.
- The **One Page** shows the brief report part first (project description, brief results tab) and a composition of short reports for each connection, including the basic information only, results, and a representative picture.
- The **Detailed** type of report includes the brief report part first (project description, brief results tab) and a composition of full reports for each connection, including code equations, bill of materials, pictures, sketches, and cost estimations.



### Project data

| | |
|---|---|
| Project name | Example |
| Project number | 01 |
| Project author | Ida Staticon |
| Project description | Steel hall |
| Date | 07-Apr-25 |
| Design code | EN |

### Brief result

| Group | Connection | Check status | Analysis | Details |
|---|---|---|---|---|
| Arrangement 1-1 | N220 | Not OK! | 100.0% | Welds Detailing |
| Top chord - Corners | N215 | OK | 100.0% | |
| | N227 | OK | 100.0% | |
| Bottom chord | N216 | OK | 100.0% | |
| | N218 | OK | 100.0% | |
| Top chord | N217 | OK | 100.0% | |
| | N219 | OK | 100.0% | |
| | N223 | OK | 100.0% | |
| | N225 | OK | 100.0% | |
| Bottom chord - mirrored | N222 | OK | 100.0% | |
| | N224 | OK | 100.0% | |
| Arrangement 4-1 | N214 | OK | 100.0% | |
| Arrangement 5-1 | N226 | Not OK! | 100.0% | Plates 6.3 > 5.0%<br>Welds 103.6 > 100% |
| Arrangement 6-1 | N221 | OK | 100.0% | |

### Software info

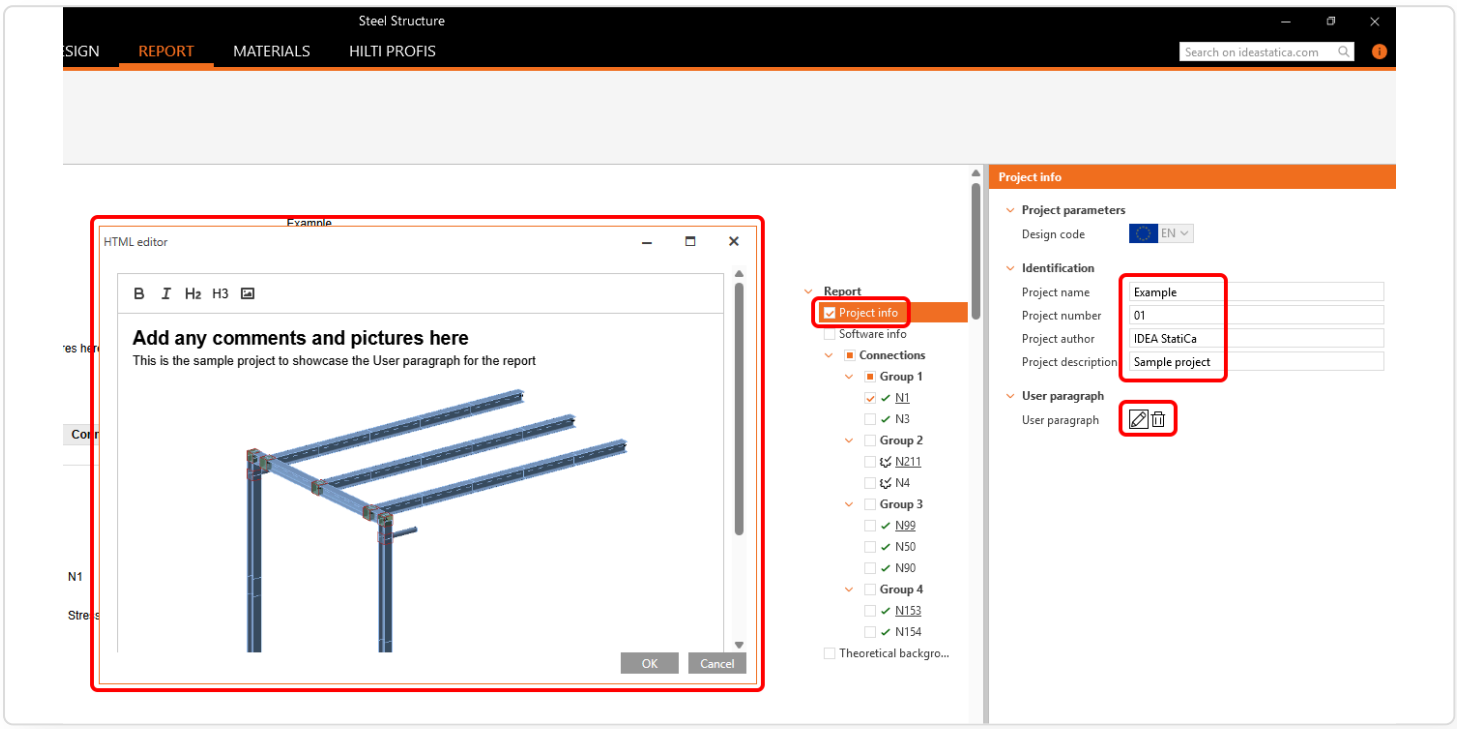
| | |
|---|---|
| Application | IDEA StatiCa Checkbot |
| Version | 25.0.0.1268 |
| Developed by | IDEA StatiCa |

# Settings for the report in Checkbot

The user can adjust the report scope in the same way as in the Connection app. Each item in the tree list can be turned on/off in order to be included or excluded from the report. Also, each item has its own settings that can be further modified.

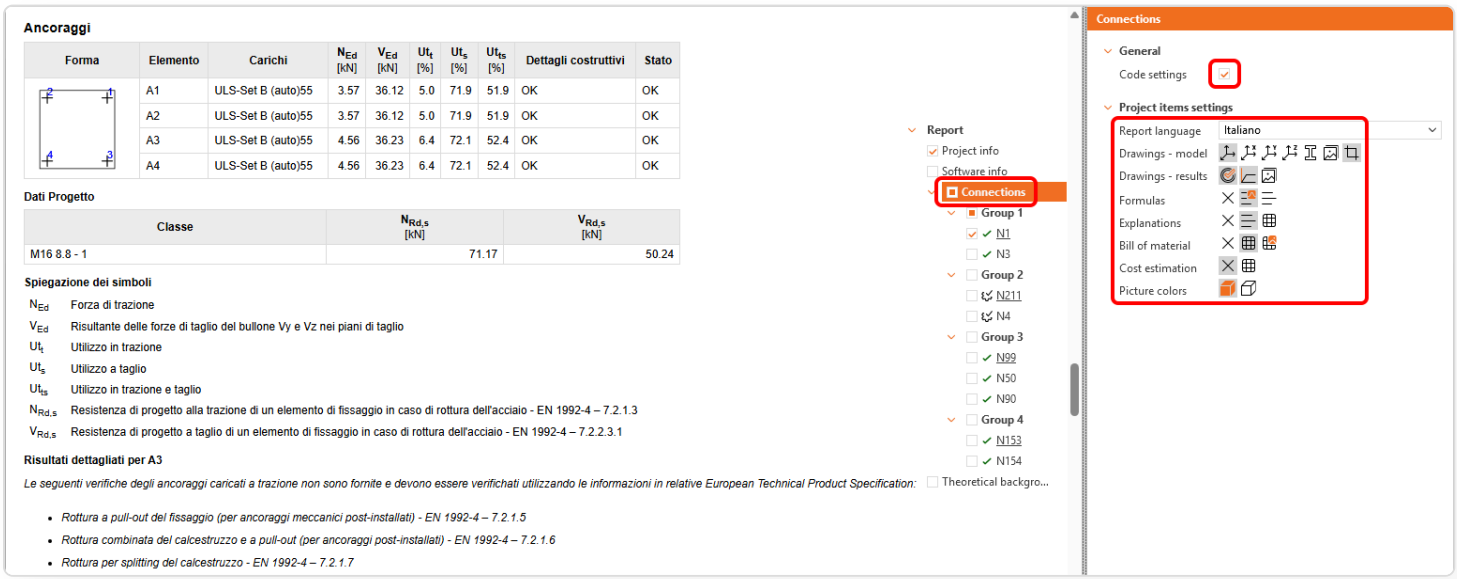
### Project info

- Edit and fill in the project name and other identification.
- Add a **User paragraph** for the whole report, a text field and pictures at the beginning of the report.



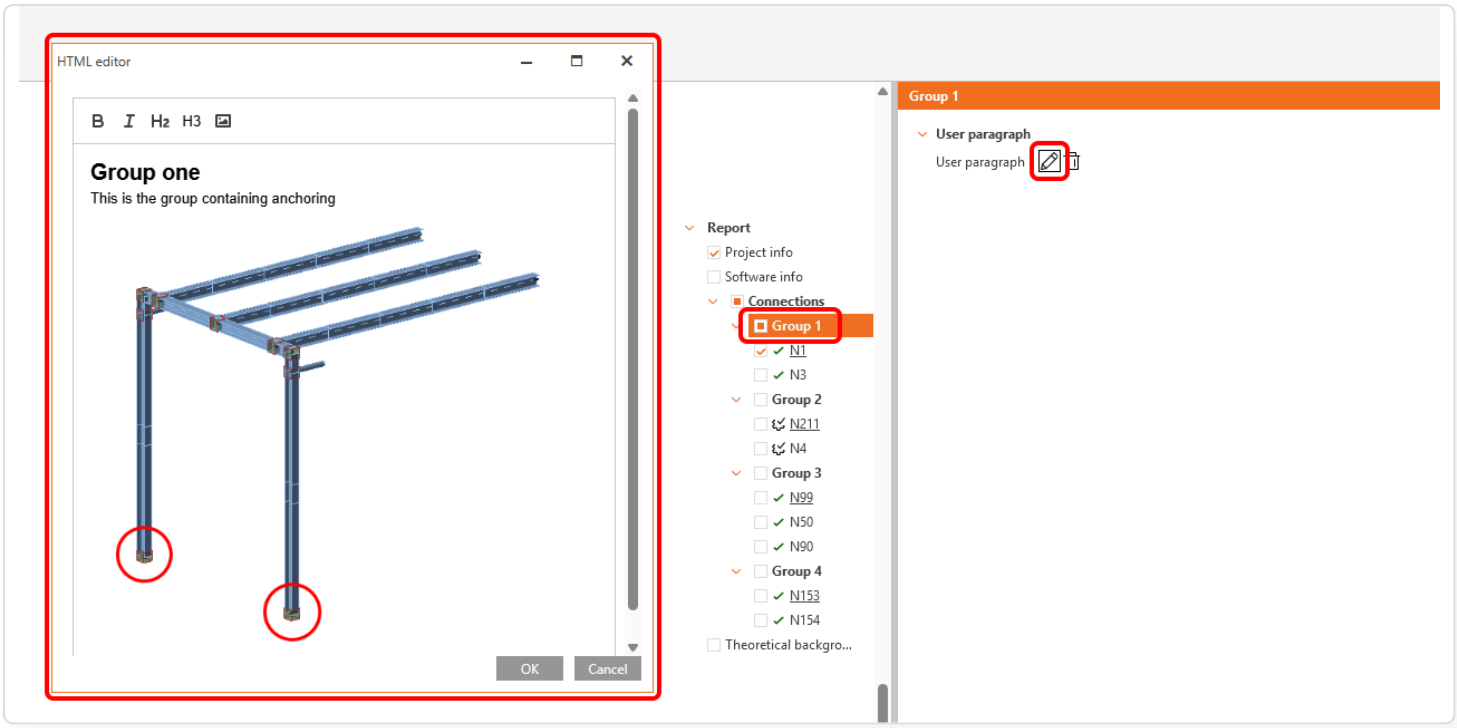

### Connections

- Turn on/off the **Code setting** (tab of references to the used codes).
- Change the **Report language.**
- Modify **Project item settings** for the Detailed type of report, drives the content of the report, whether to include formulas, explanations, bill of materials, and other data.



### Group

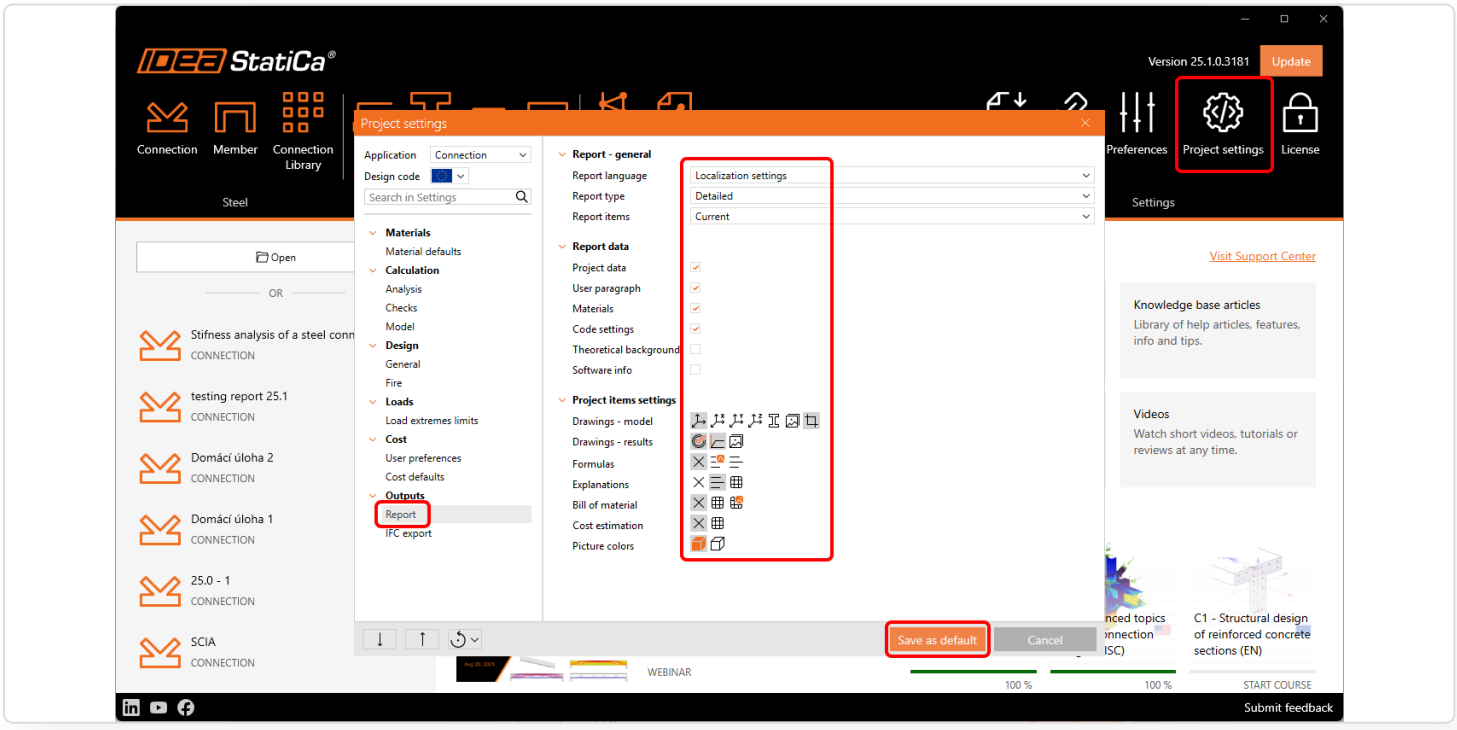
- Add a **User paragraph** for the selected group, a text field and pictures at the beginning of the part for the selected group.



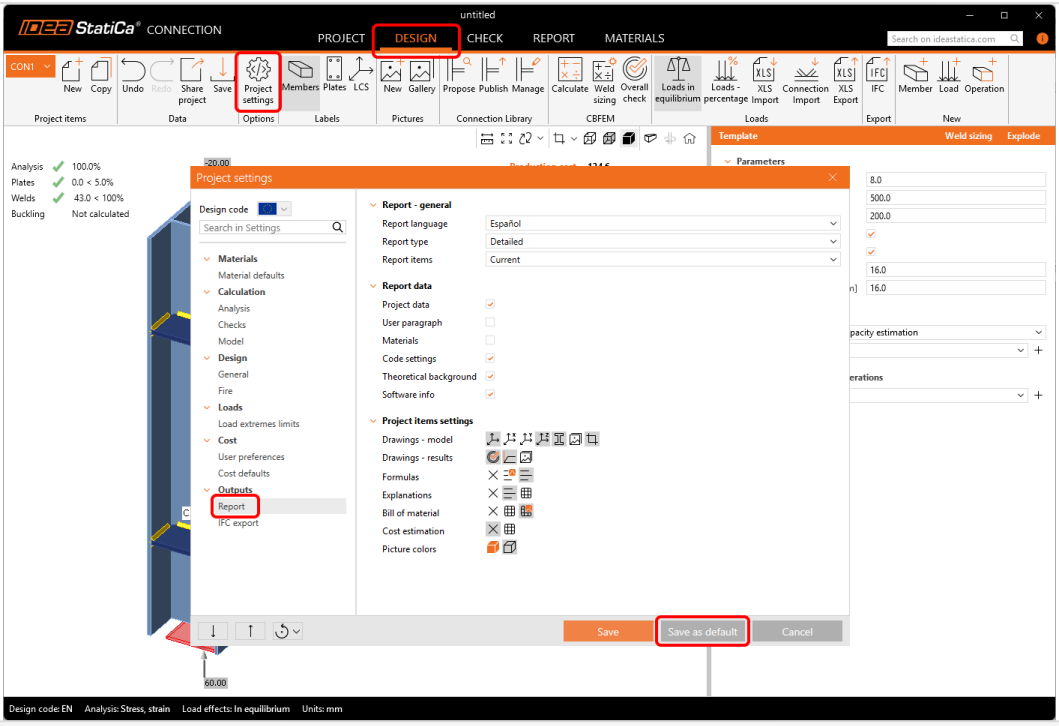

## Set the default Report settings

Users can adjust the default report settings in the Project settings and use them across all current and future projects to keep the consistent report content and format.

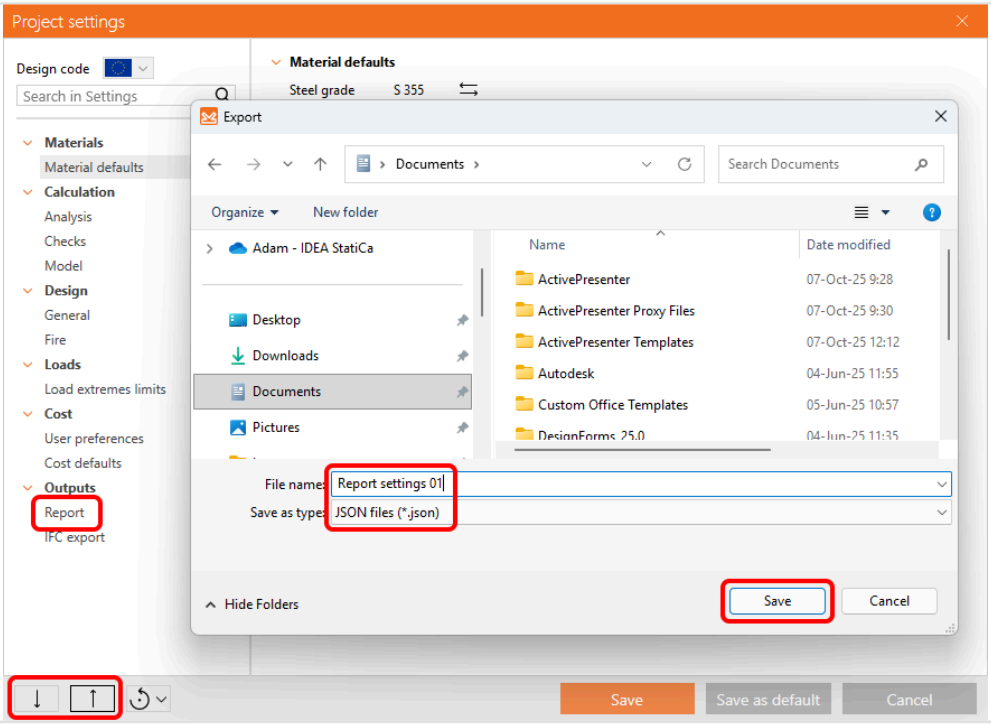
To change the report defaults, open the **Project settings** from the **Launcher** app, do the modifications and click **Save as default**.



Alternatively, users can open the **Project settings** from the **Design** tab in the **Connection** app and save the report's current settings they are using in the current project as the defaults.




The report settings can also be exported from the **Project settings** to a JSON file using the **Export** command and shared with another user.




The factory default settings can be reset with the commands **Load metric defaults** or **Load imperial defaults** in the **Project settings**.

*Released in IDEA StatiCa version 25.1.*

BIM links

# ETABS and SAP2000 BIM link to Detail

Checkbot now supports its first BIM link for concrete structures, enabling wall design transfer from ETABS and SAP2000 to IDEA StatiCa Detail. Using the CSFM method, Detail analyzes discontinuity regions and performs full ULS and SLS checks for reinforced concrete walls.

This link addresses the need for efficient continuity between global analysis and detailed reinforced concrete design. While **ETABS** and **SAP2000** are primarily intended for global structural analysis based on the **Bernoulli–Navier hypothesis** (plane sections remain plane), regions with **discontinuities** require a more advanced analytical approach. The Compatible Stress Field Method (CSFM) implemented in IDEA StatiCa Detail is specifically designed to analyze these regions, capturing nonlinear and localised behavior with higher accuracy.

Without this link, engineers must manually rebuild wall models and reapply loads in **IDEA StatiCa Detail**. Once transferred, **Detail** provides advanced nonlinear analysis, reinforcement design, and code checks, extending the capabilities of **ETABS** and **SAP2000** beyond global behaviour to local structural verification.

This link is available for all versions of **ETABS** and **SAP2000**, supporting design codes for **EC**, **ACI**, and **AUS**.

# ETABS to Detail Workflow steps

**Perform global analysis**

**Import model to Checkbot**

### Design reinforcement

### Code-check and optimize

To enable export from ETABS, add the **IDEA StatiCa plugin** used for both **steel and concrete** workflows. To add the plugin, navigate to the **“Tools”** tab in **ETABS** or **SAP2000**, then select **Add Plugin**. The plugin file can be found at:

```
C:\Program Files\IDEA StatiCa\StatiCa 25.1\net48\IdeaETABSv1PluginWrapper.dll
```

1. **Prepare the Model**: Run the analysis in ETABS or SAP2000 for all required load cases and combinations.
2. **Activate BIM link**: Launch Checkbot by activating the plugin in ETABS or SAP2000. Create a new **concrete** project.
3. **Import Walls**: Choose the wall elements to import. Other selected items might also appear in Checkbot.
4. **Create Details**: Use Detail+ to create Detail elements. Select wall elements in the 3D scene and confirm.
5. **Export to Detail**: Export the selected detail elements as a Detail model.
6. **Design in Detail:** Open the model in **Detail 2D** and add supports to match the geometry defined in **ETABS**, using either boundary conditions or wall connections as appropriate.
7. **Add reinforcement:** Define reinforcements based on stress flow.
8. **Analyse**: Run the calculation and investigate the results.
9. **Report:** Create a report.

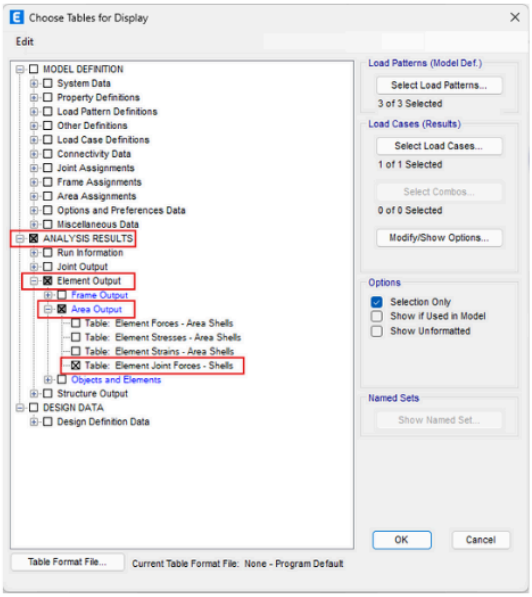
For detailed steps, please follow this tutorial.

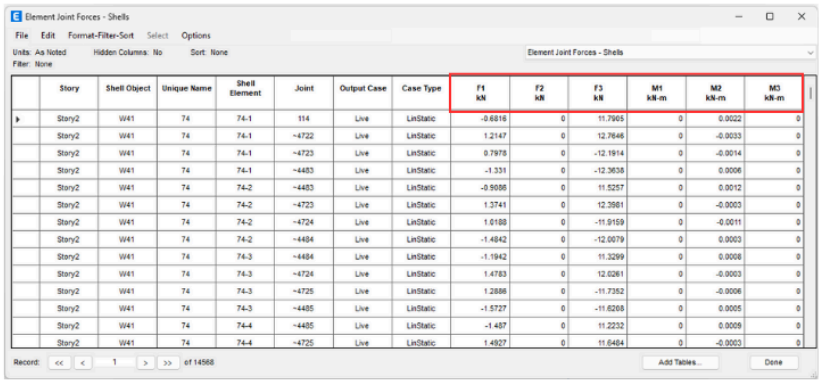
# Force transfer process

Transferring internal forces in 2D elements is a complex process that can differ between software applications. For this reason, our initial focus has been on ETABS. In this section, we explain the background and methodology for transferring forces between ETABS and IDEA StatiCa Detail 2D.

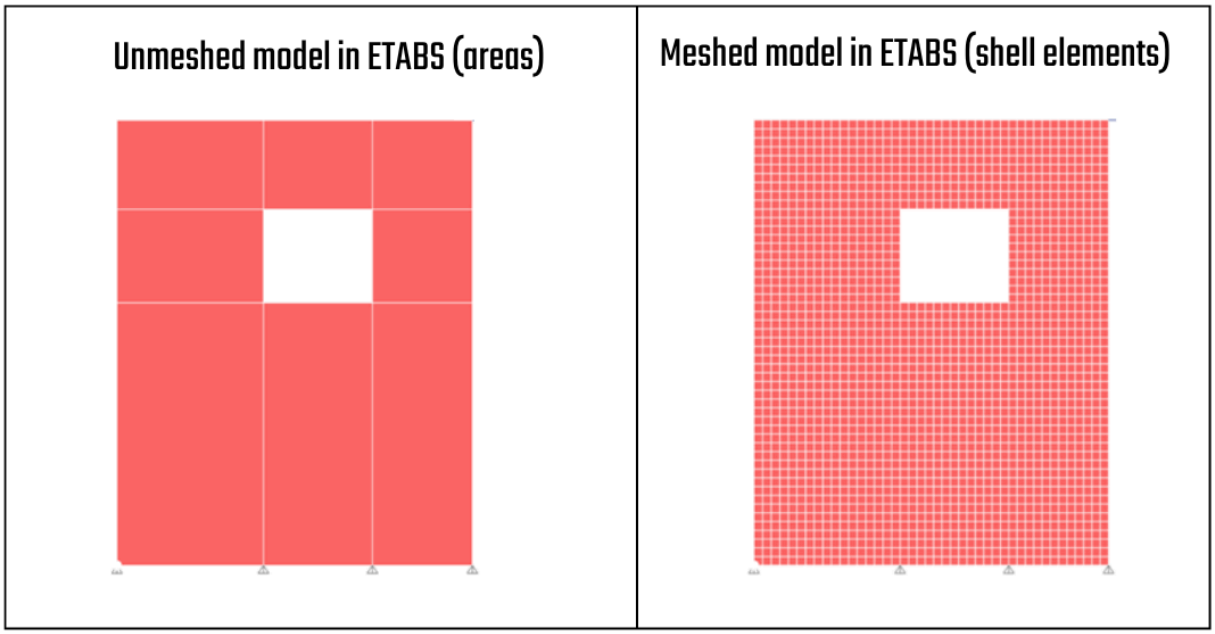
The basic process is outlined below:

- **Shell Force Extraction**
  Joint forces from shell elements in ETABS are obtained using the *Element Joint Forces – Shells* output table.
  To access these values in ETABS:
  - Select the wall elements in the 3D view.
  - Navigate to:
    **Display → Show Tables → Analysis Results → Element Output → Area Output → Table: Element Joint Forces – Shells**



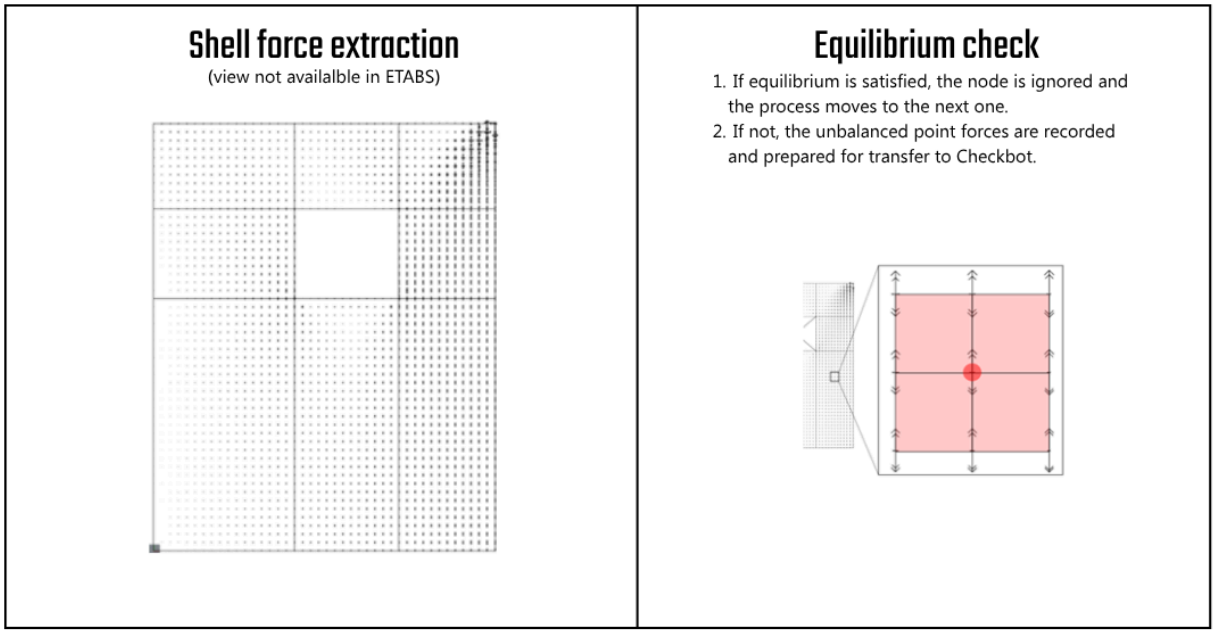




| DEFAULT CONVERSIONS | | | | | | |
|---|---|---|---|---|---|---|
| ETABS | F1 [kN] | F2 [kN] | F3 [kN] | M1 [kNm] | M1 [kNm] | M1 [kNm] |
| IDEA StatiCa | Fx [kN] | Fy [kN] | Fz [kN] | Mx [kNm] | My [kNm] | Mz [kNm] |

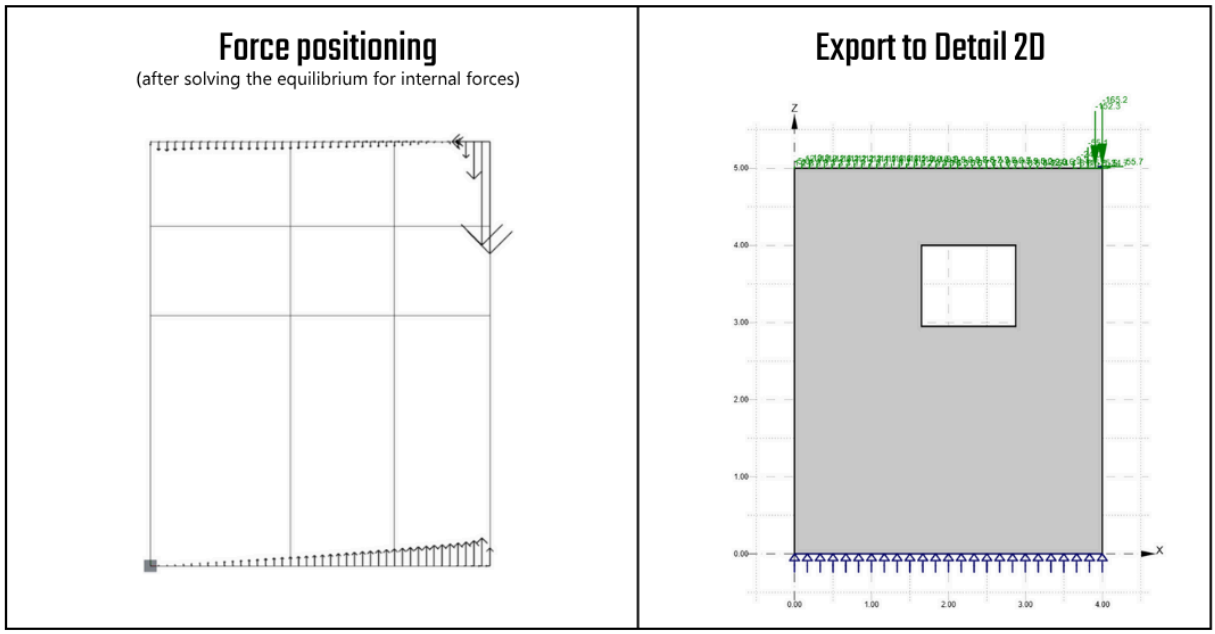



- **Equilibrium check**
  For each shell, at each joint, the corresponding shell forces are evaluated for all six degrees of freedom to check whether equilibrium is achieved.

- If equilibrium is satisfied (i.e., the sum of forces and moments equals zero), the node is ignored and the process moves to the next one.
- If not, the unbalanced point forces are recorded and prepared for transfer to **Checkbot**.
- Joint force moments (*M1, M2, M3*) are excluded from the equilibrium calculations.
- Typically, joints that are not in equilibrium are located along the edges of the area elements. However, internal joints may also show imbalance in certain cases, such as when area elements are not pre-meshed at slab intersections or when individual point loads are applied directly to a joint.




- **Force positioning**
  The resultant joint forces are then assigned at a specified 2D X-Y coordinate on the area panel in the Detail view. These forces are subsequently transferred to **Checkbot**, where they are stored and prepared for export to **Detail 2D**.




- **Additional Load Transfer**
  Certain loads acting on or within the area element are not captured by the joint force equilibrium method and must be transferred **using an alternative approach** to maintain consistency between **ETABS** and **Detail 2D**. These include:
  - **Self-weight of the element** – If self-weight is defined in **ETABS** for a given load pattern, it is calculated automatically and saved to the corresponding load case in **Checkbot**. It is then applied to the **Detail 2D** member as a uniform load.
  - **Uniform area loading** – Any uniform load applied to an area element in **ETABS** is saved to the load case and automatically transferred to the **Detail 2D** member as a uniform load.
- From the user's perspective, there is no difference in workflow, these loads are managed automatically within the transfer process, requiring no additional input or action.

# Known limitations

Refer to the article: limitations for the BIM link between **ETABS**, **SAP2000**, and **IDEA StatiCa** for concrete wall design in Detail.

# API/Developer documentation website

Why create a website about the IDEA API? In response to market demand for more information about how to connect and tailor-make data interconnections, BIM link creators, design automation tool developers, and visual programming experts can find all necessary information and documentation there.

# Where can you find it?

The page developer.ideastatica.com is the place to go.

This new developer documentation site gives IDEA StatiCa third-party developers and API scripters a single access point for information relating to IDEA StatiCa public services and code.

The site links together static documentation, code documentation, and specific examples.

# Improvements compared to the previous system

- The public GitHub web page provides a lot of useful information but may not be that easy to navigate. Therefore, this provides developers with a single access point.
- Breaking change release notes help notify developers when and how they need to update their third-party apps.
- Provides a distinction between public WIP developments (should not be used as products) and those that are working products with documentation.
- The Wiki provides the ability to create code documentation and tie it to classes and concepts.
- The Wiki stays automatically updated and maintained.
- Examples are easily accessible from the documentation.

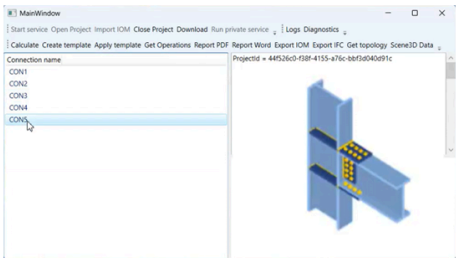
The site is split into four distinct sections:

- **IDEA Open Model (IOM)**, related to the creation and specification of the IOM Data model.
- **BIM Links (BIM API)**, related to the creation of Checkbot BIM Links using the BIM API framework.
- **Design API (API)**, related to Design App APIs.
- **Extensions**, related to Extensions created on top of the design app APIs.

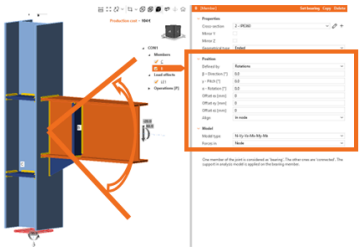
## API for IDEA StatiCa Connection updates

The updated API for IDEA StatiCa Connection includes functions to broaden its usability, such as:

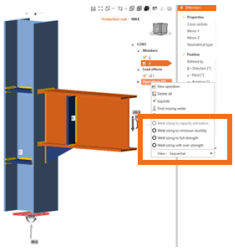
- Connection presentation
- Member positioning
- Weld auto-design
- Shared settings controller
- Improved report generation
- Project code conversion



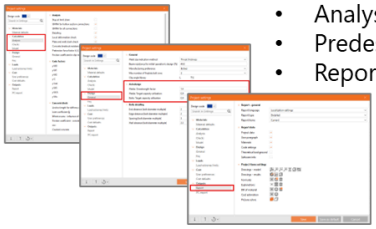
Connection presentation



Member positioning



Weld auto-design



- Analysis
- Predesign
- Report

Shared settings controller

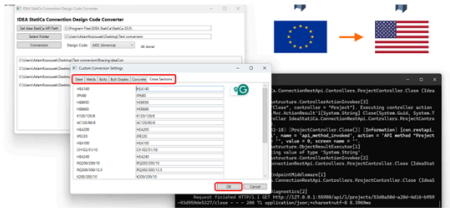
**Includes buckling results**

**Removed app loading**

**Multi report support**

**Any language**

Improved report generation



Project code conversion

*Released in IDEA StatiCa version 25.1.*

# Supported BIM links in version 25.1

We have taken several actions to ensure the consistent update process of our BIM links.

# The latest two major releases supported

In each major release of IDEA StatiCa, we support the two most recent major releases of each linked application. The older versions become obsolete, this happens in major IDEA StatiCa releases only (patches will never disconnect older versions). On the other hand, when a new major release of the BIM application comes, we develop/update the link in two months — the link appears in a patch of IDEA StatiCa.

The current state of the supported versions is always presented on our website. With the 25.1 release, we will support the versions presented in the first column of the table. The “In development” column represents the newest versions that will start to be supported in a patch of 25.1. The third column shows versions that are no longer supported.

| IDEA StatiCa - Steel | 25.1 | | |
|---|---|---|---|
| Application | Supported | In development | Obsolete |
| Advance Design | 2025, 2026 | - | 2024 |
| Advance Steel | 2025, 2026 | - | 2024 |
| AxisVM | X7.3, X8.1 | - | X7.2 |
| ETABS | 21, 22 | 23 | - |
| midas Civil / Gen | 2024, 2025 | - | - |
| midas Civil NX | 2024, 2025 | - | - |
| RAM Structural System | 23, 24 | 25 | - |
| Revit | 2025, 2026 | - | 2024 |
| RFEM / RSTAB | 5.38 / 8.38, 6.11 / 9.11, 6.12 / 9.12 | - | 5.37 / 8.37, 6.09 / 9.09, 6.10 / 9.10 |
| Rhino/Grasshopper | 7, 8 | - | - |
| Robot Structural Analysis | 2025, 2026 | - | 2024 |
| SAP2000 | 25, 26 | - | - |
| SCIA Engineer | 25, 26 | - | 24 |
| SDS2 | 2025, 2025-01 | 2026 | 2024 |
| STAAD.Pro | 2024, 2025 | - | 2023 |
| Tekla Structures | 2024, 2025 | - | - |

| IDEA StatiCa - Concrete | 25.1 | | |
|---|---|---|---|
| Application | Supported | In development | Obsolete |
| Advance Design | 2025, 2026 | - | 2024 |
| AxisVM | X7.3, X8.1 | - | X7.2 |
| ETABS | 21, 22 | 23 | - |
| midas Civil / Gen | 2024, 2025 | - | - |
| RFEM / RSTAB | 5.38 / 8.38, 6.11 / 9.11, 6.12 / 9.12 | - | 5.37 / 8.37, 6.09 / 9.09, 6.10 / 9.10 |
| Robot Structural Analysis | 2025, 2026 | - | 2024 |
| SAP2000 | 25, 26 | - | - |
| SCIA Engineer | 25, 26 | - | 24 |

The currently supported versions in the latest IDEA StatiCa patch can be found in the BIM links: Supported versions of third-party applications page.

The older versions of BIM-linked applications may still be used. Nevertheless, we will not actively support the projects or fix possible bugs.

# Cloud tools and general improvements

# Connection Library – the largest database of downloadable steel connections

Have you ever been stuck with finding a solution for a joint design? None of your previous projects have a similar geometry, all of your colleagues are too busy to help you, and you are just out of ideas?
From now on, you will know where to turn for inspiration. Simply enter connectionlibrary.ideastatica.com into your Internet browser.

With the Connection Library, you can browse examples with desired geometry and find inspiration within seconds! The use of the application is simple and intuitive! The basic set of the database consists of **1,000,000+ designs** and is expanding every day.

# How it works

Connection Library is a web app that enables users to find inspiration by providing them with a public database of thousands of available connections. Users can consequently use the provided connections in the Connection app or share the search results with others, as well as download the connections and use them as the starting point in their projects.

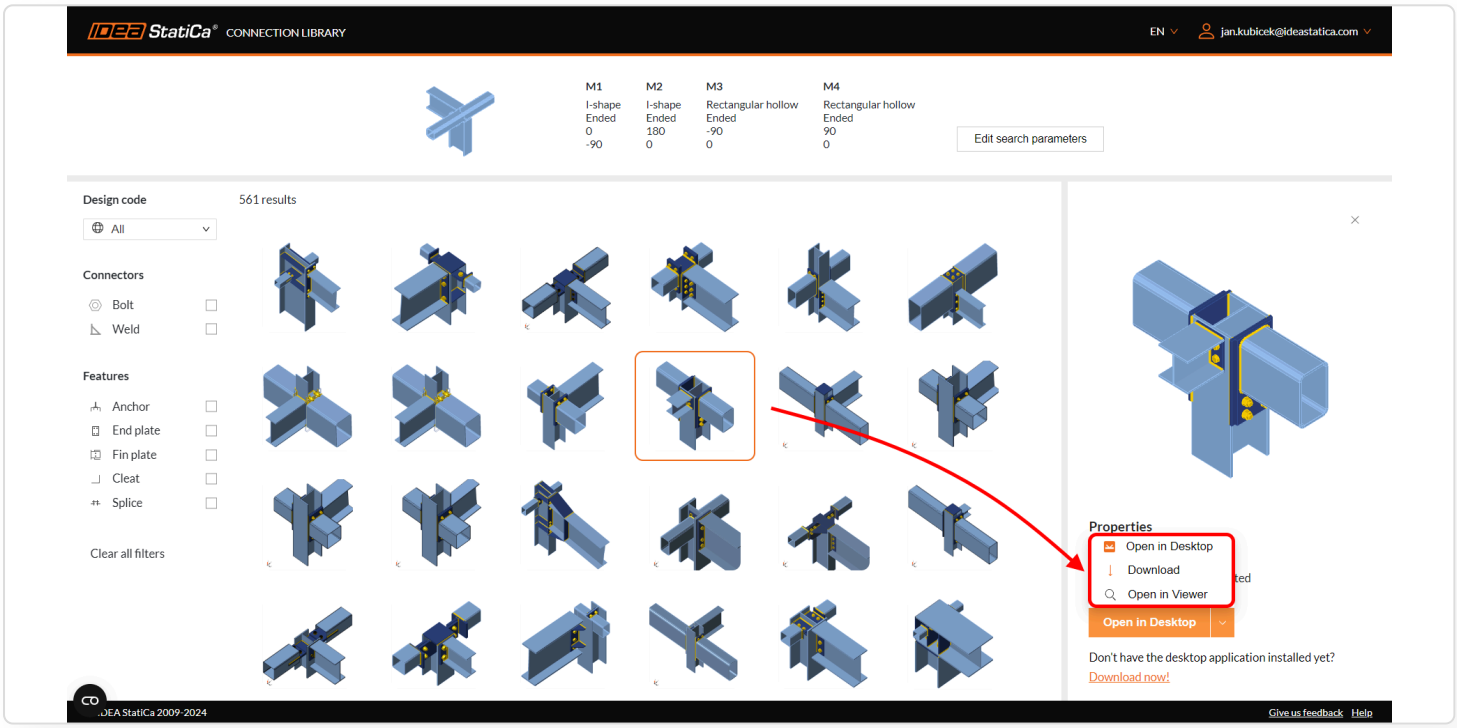
The application is comprised of two pages – the input page and the results page:

- On the **input page**, the user can define a connection they are looking for using the available input parameters. There is a 3D scene available, so it is clear what the search input looks like.
- While on the **results page**, the user has access to all the results for the input they entered previously. The user can review the search results in more detail, use filters to narrow down the search, or return to the input page and change the input.

The user can access a connection detail on the results page after they sign in. If the user does not have an account yet, then it is possible to sign up as well. In such a case, the user is redirected to a page where they can register and get a Basic license account. While on the connection detail, the user has an option to download a trial license.

The selected design can then be used further:

- Open in the Desktop app directly
- Download in .ideacon file format
- Open in Viewer



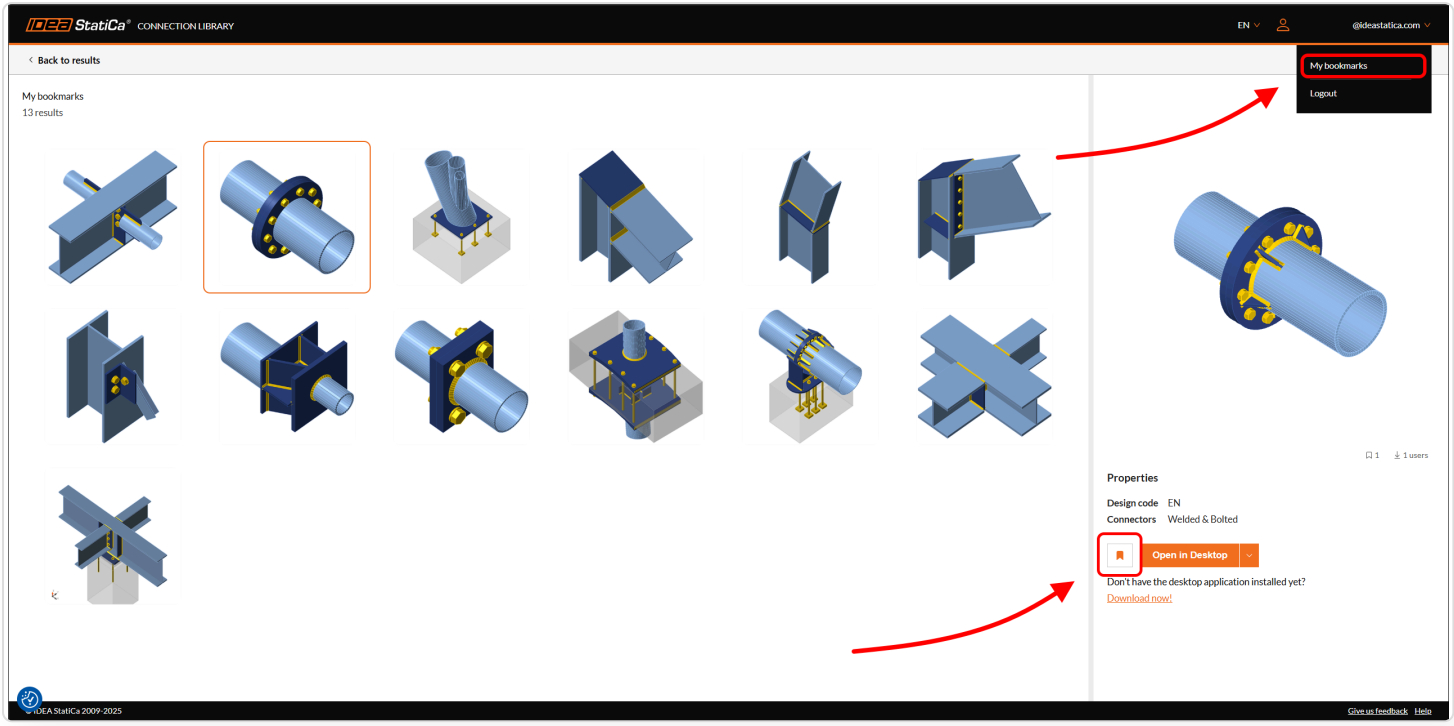

The sources of this database are the connection models created and shared by users in another web application – IDEA StatiCa Viewer.

You can find a lot more about the usage and advantages of the Connection Library cloud database in this blog post dedicated to How to get inspired during steel connection design.

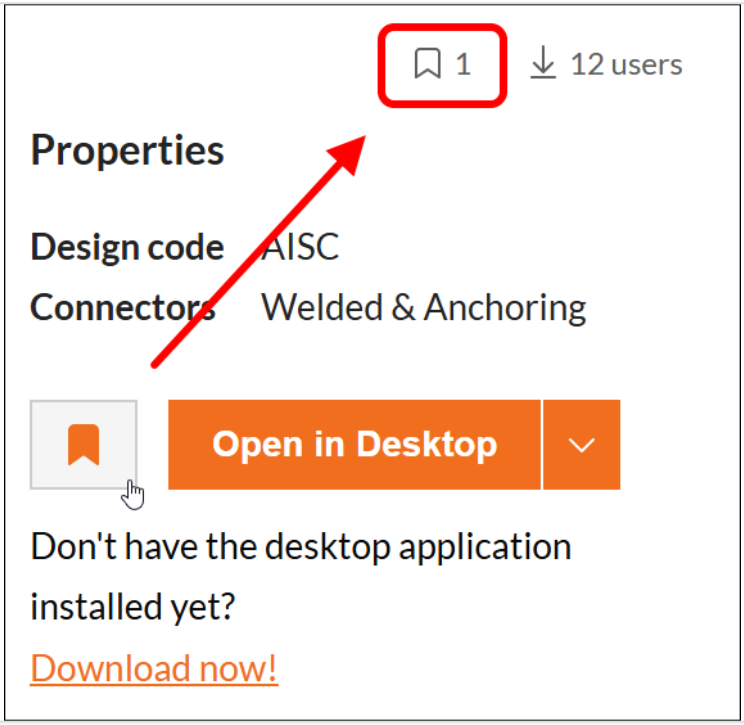
*Released in IDEA StatiCa version 23.1 and version 24.1.*

# Save your favorite template

After logging in, you can save the favorite designs through the 'bookmark' button. This saves it to a private set accessible within the drop-down menu.




The preview scene also contains the overall bookmark number, indicating how many times the design has been saved by unique users.

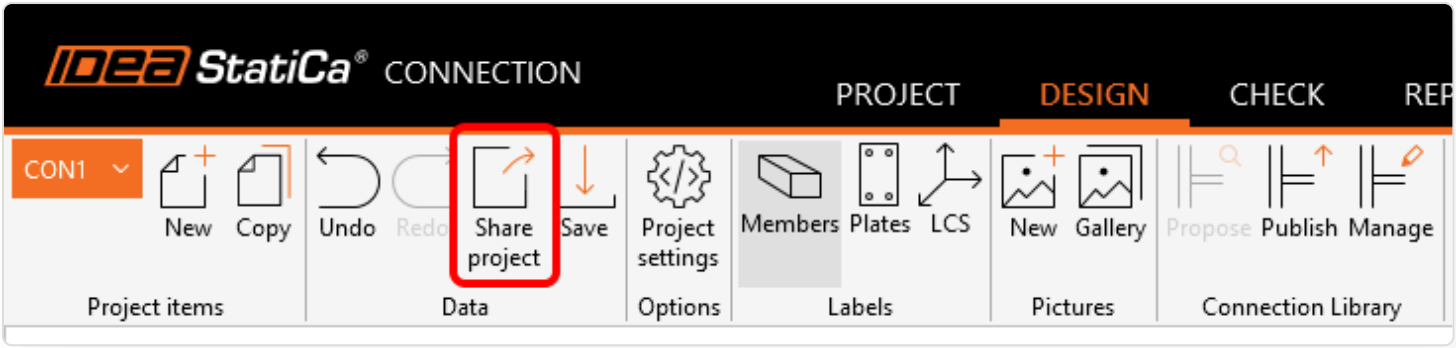



*Updated in IDEA StatiCa version 25.1.*

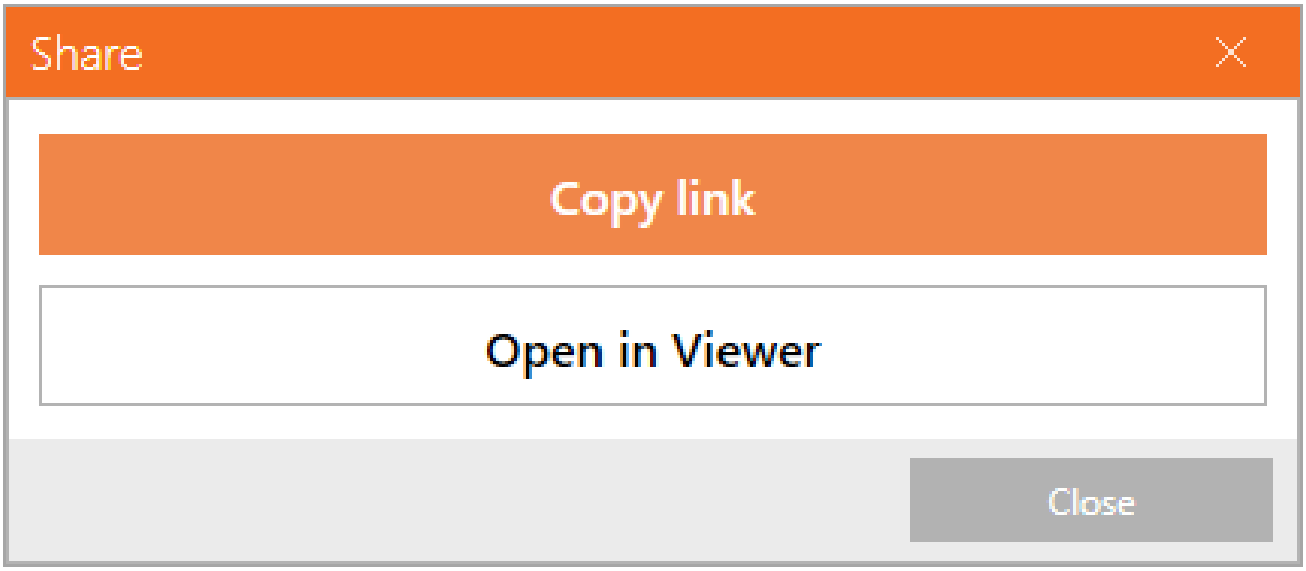
# Direct link from Connection to Viewer

Sharing projects via Viewer is easier than ever – create the link from Connection directly and send it to anyone you need. Manage the shared links in Viewer under your account.
The top ribbon in IDEA StatiCa Connection has the possibility of sharing the project via our online Viewer.

This is the easiest way to send the project to anyone, even to persons without an IDEA StatiCa license. No more sending large .ideacon files that need to be manually uploaded to Viewer.



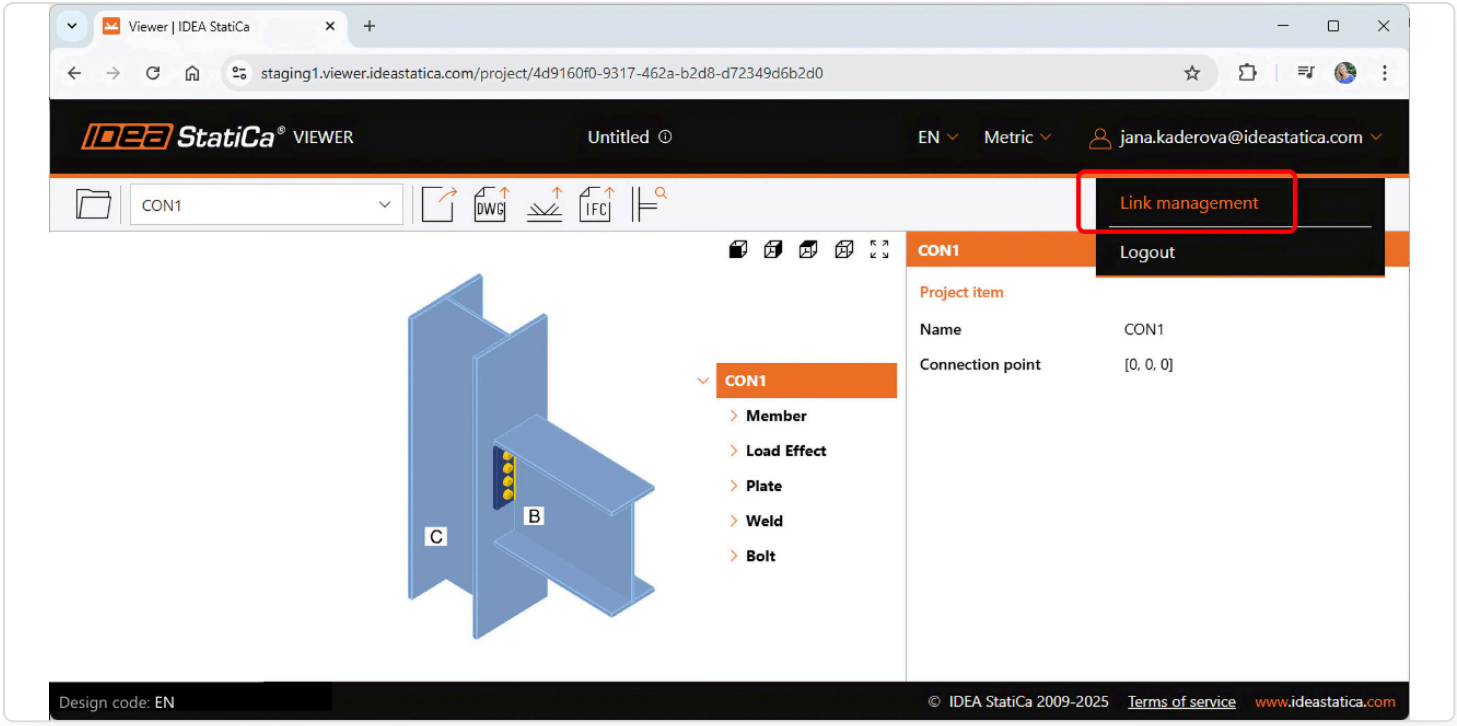

You can either copy the link onto the clipboard or open the project directly in IDEA StatiCa Viewer.



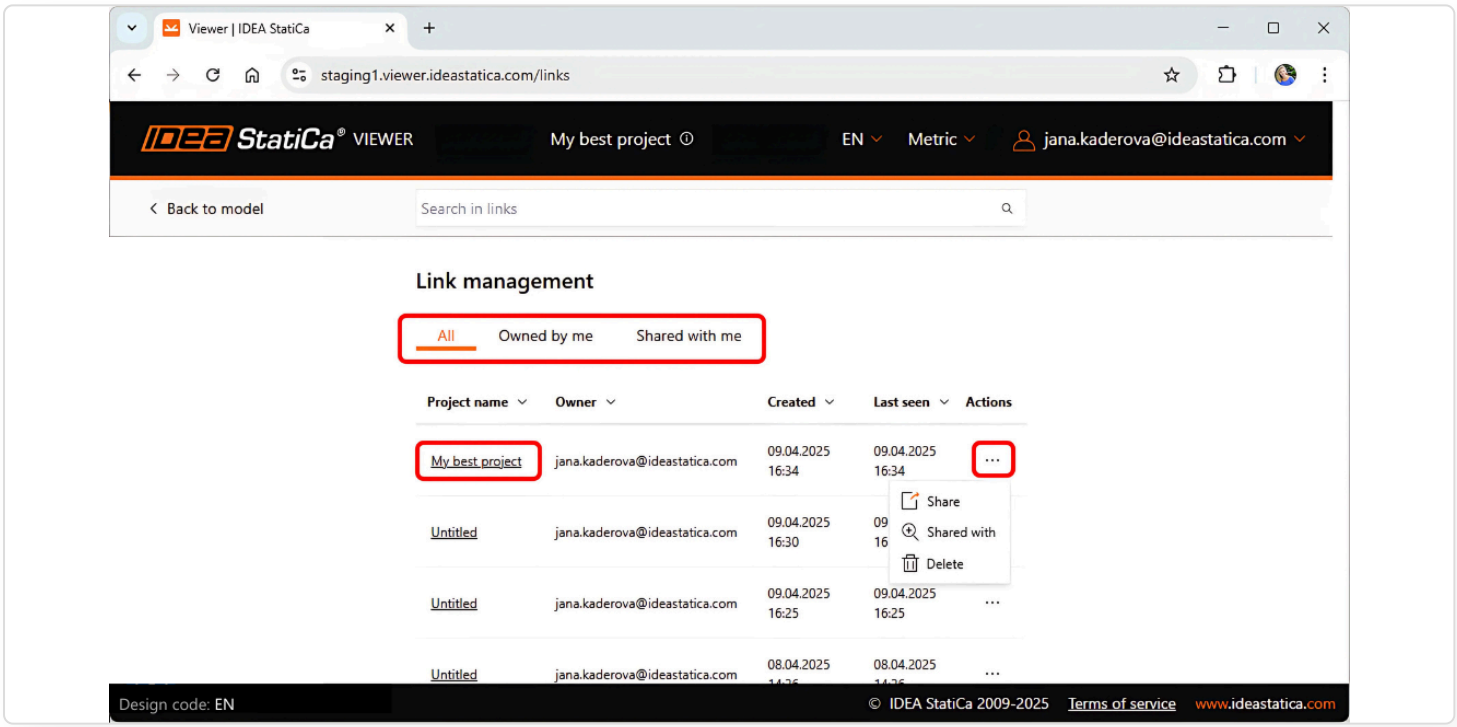

To open a shared project, you do not need to have an IDEA StatiCa license. However, to interact with the project in Viewer, you need to use your commercial, trial, or educational, license or register for the free IDEA StatiCa Basic account.

# Link management

When logged into your user account, you can find all your links to Viewer models, both the ones created and owned by you as well as the ones shared with you.



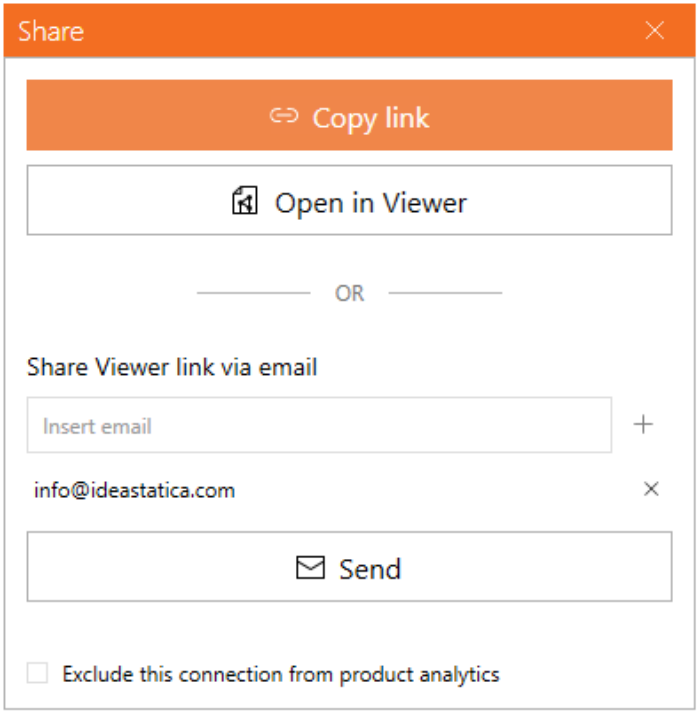

Each project with a generated link is saved on the cloud server and the URL can be copied and sent again. Projects can also be deleted from the list.




*Released in IDEA StatiCa version 25.0.*

# Share the link via email

The link to the Viewer can also be shared via email directly from the Connection app. This is possible for one or multiple email addresses at once.




**Limitation:** The current limit of project links shared via one email address is set as maximum 20 per day.

**Note:**

- All project items of the shared file are sent to the Viewer (whole projects); the users can't choose a specific project item.

*Updated in IDEA StatiCa version 25.1.*

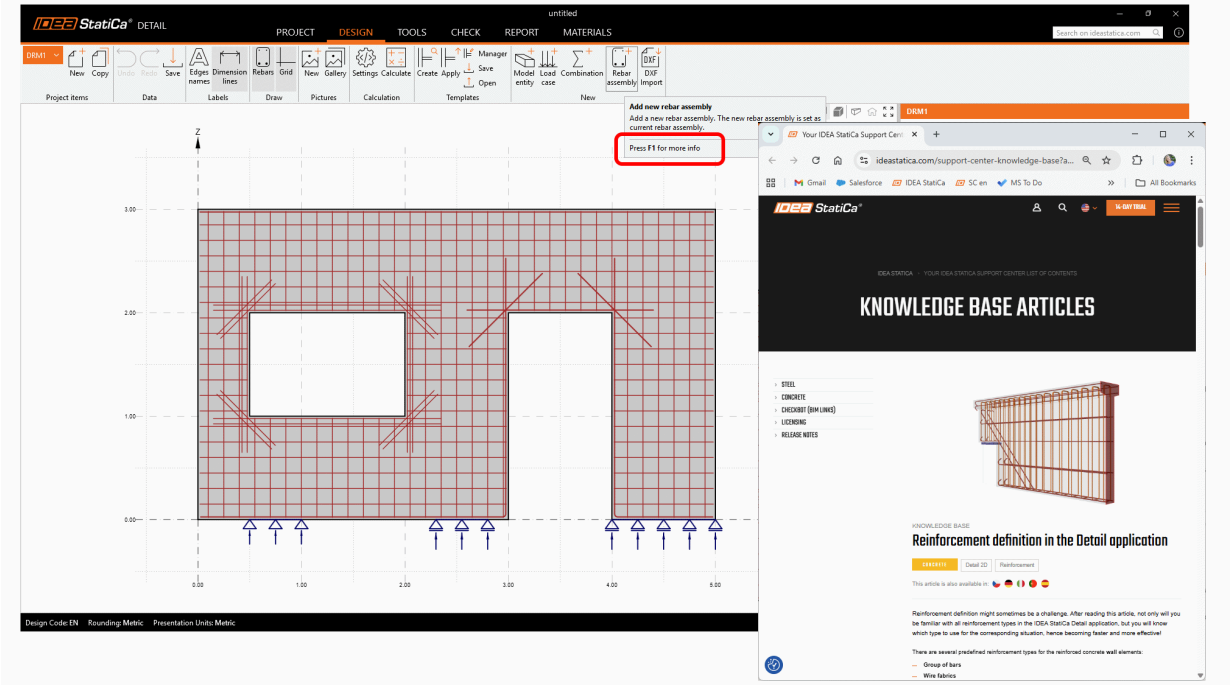
# In-app tooltips and links with Support Center

The tooltips of specific features in IDEA StatiCa products display hyperlinks to Support Center articles. This is a reaction to frequently asked questions about how the features work and where their limits are.
Feature tooltips provide instant access to Support Center knowledge base articles (KBAs) directly within the IDEA StatiCa desktop application, enabling users to find answers faster and reduce reliance on technical support.

*Released in IDEA StatiCa version 24.0.*

Upgrade of this functionality brought new types of tooltip description windows embedded in the applications. When the link to a more comprehensive explanation exists, the path is available under the keyboard function button.

When a detailed explanation is available, **pressing F1 button** while hovering over a button or UI element opens the corresponding Support Center article in a web browser.




The tooltips with the links to the KBAs are available in IDEA StatiCa Connection, Member, Detail, and Checkbot applications in the top ribbon buttons. Instant development will bring this function to even more places across the applications.

*Updated in IDEA StatiCa version 25.1.*