RELEASE NOTES

Release notes IDEA StatiCa 24.0

Apr 24, 2024

Table of Contents

CONCRETE DETAIL DESIGN

3D Detail – reinforced concrete footings (BETA)

3D Detail – Wall members subjected to general load (BETA)

Import of anchoring from Connection to Detail (BETA)

Multiselect and multi-edit in Detail

Customizable report tab in Detail

RCS API for streamlined, simplified, and efficient design processes

Simple insertion of table inputs into IDEA StatiCa Detail

Comprehensive user interface in IDEA StatiCa Detail

Presentation of relevant curve results in detailed graphs in IDEA StatiCa Detail

Unified Materials tab in Detail

SIA 262:2013 implemented in RCS and Beam

Accurate calculation of the shear resistance of slabs

BIM AND CHECKBOT

Checkbot Free – the structural design hub for FEA and CAD data

Checkbot ready for big projects

Fast project visualization in Viewer

Enhanced 3D scene interaction and property management in Checkbot

Tekla BimApi link available on GitHub

Supported BIM links in version 24.0

USABILITY AND LICENSING

Single Sign-on (SSO)

Enhanced User Portal

Shared settings across the whole tool range

Error handling with Support Center integration

In-app tooltips and links with Support Center

Keyboard shortcuts (hotkeys) in IDEA StatiCa apps

New Portuguese (Brazilian) language in apps

Version 24.0 represents two milestones. It celebrates 10 years since the release of CBFEM method and IDEA StatiCa Connection, a decade of making the right connections. It also introduces a groundbreaking 3D concrete structural design solution for footings and walls loaded out of plane.

Steel Connection Design

Parametric templates in Connection Library

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

In IDEA StatiCa Connection, we have the possibility to use developer mode and create connections using parameters (relations defined between individual entities). Parametric design allows us to design standardized connections efficiently.

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

How does it work?

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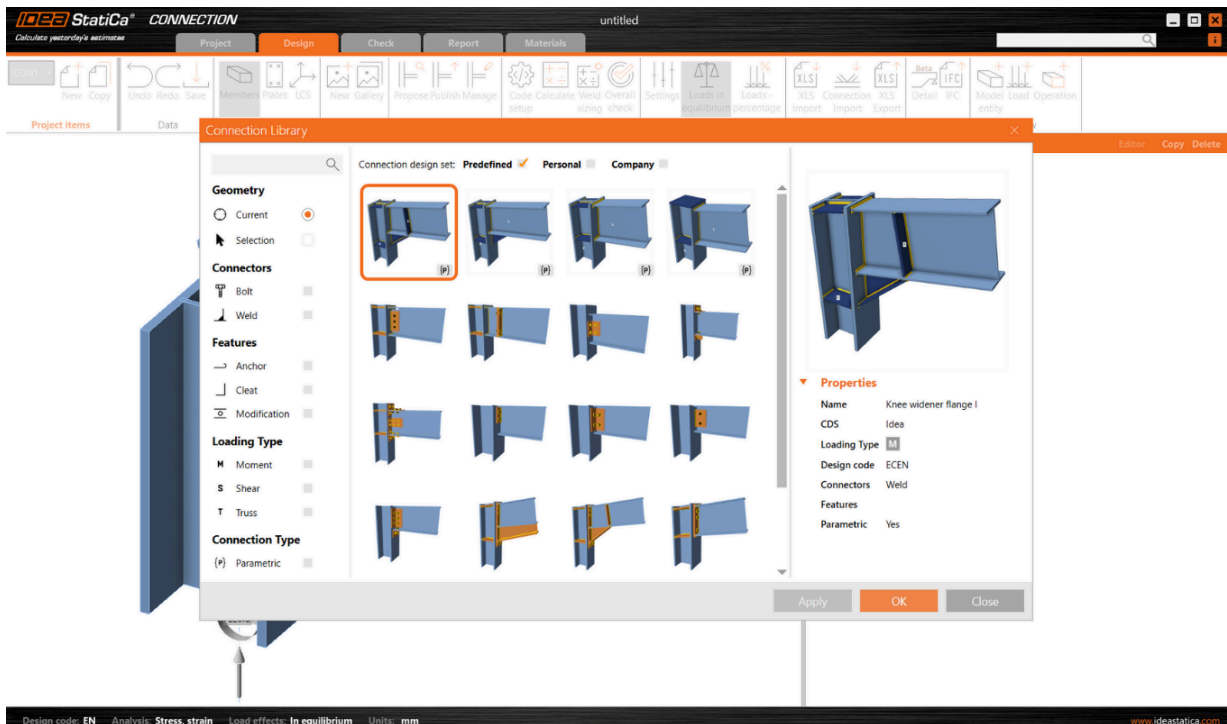

In addition, after proposing the template, it is possible to **change the parameters directly in the main design window**, and there is no need to go into developer mode. This user-friendly environment allows less experienced users to work with predefined parameters safely according to the presets of the senior designers.
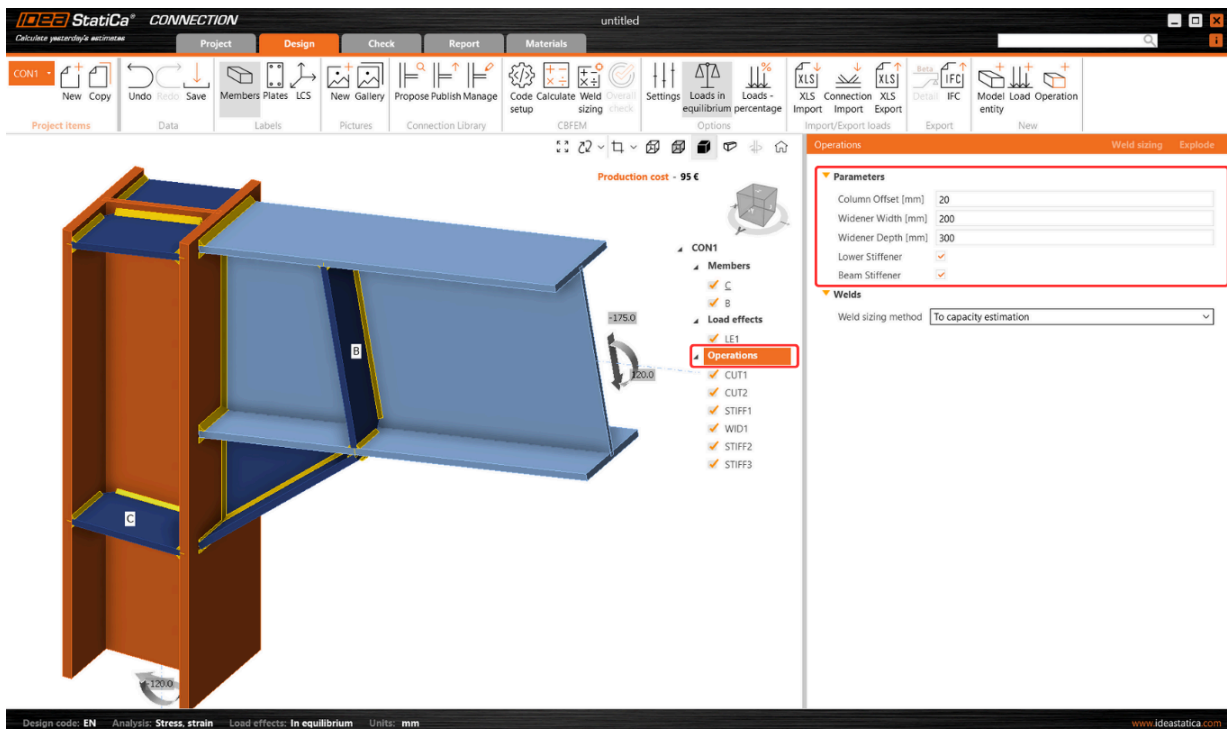
This makes further optimization very simple as there is the possibility of avoiding changing each entity one by one when using operations.

By changing one parameter, **multiple steps** can be performed at once. For example, when changing the widener width, not only the widener itself, but also all the related welds and the location of the stiffener are affected:
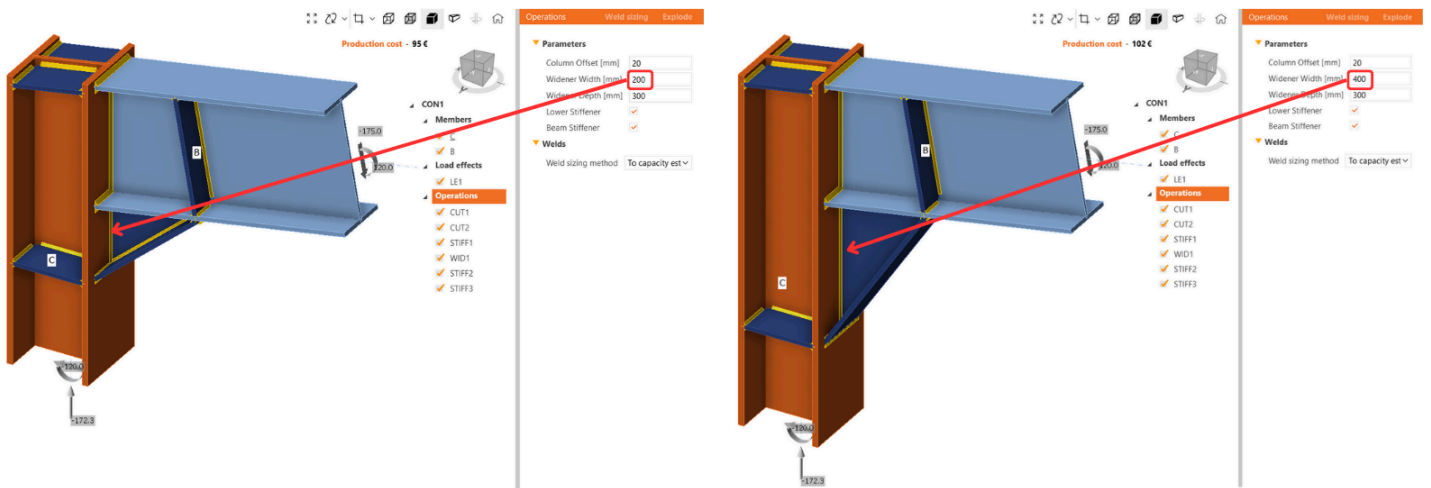
The locked modifications in the property list of some manufacturing operations are disabled in such cases. However, if the user wishes, they can break the parameters using the "**explode**" button. And continue with modifying operations.

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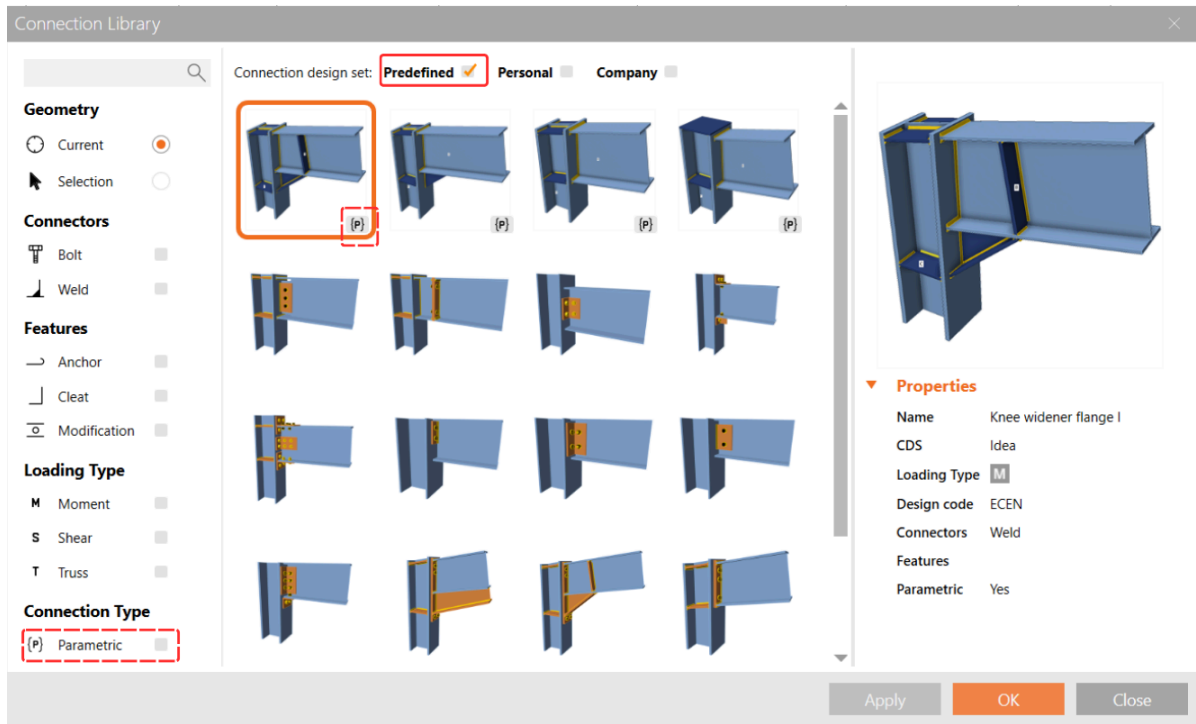
What are the benefits of parametric templates?

- **Universal templates**: Designers can access a broad collection of parametric templates from the Connection Library. These templates are designed to be universally applicable, providing a solid foundation for a wide range of projects.

- **Parametric customization**: Through the Developer tab, users can define specific parameters for each template, allowing for a high degree of customization and flexibility in design.

- **Template identification**: Parametric templates are easily identifiable by the symbol {p}, ensuring that users can quickly recognize and select them for their projects.

- **Enhanced library filtering**: The filter in the Propose window of the Connection Librar, enables users to efficiently find parametric templates among the library's extensive offerings.

- **Publication control**: When a connection contains parameters, designers have the option to publish these parametric templates to the Connection Library. This feature offers flexibility in sharing customized templates with the broader user community or keeping them private for individual or internal use.

Impact on workflow

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

- **Efficiency**: The use of templates speeds up the design phase, allowing for quicker iterations and modifications.

- **Consistency**: Parametric templates ensure design consistency across projects, which is crucial for maintaining standards and quality.

- **Collaboration**: The ability to share customized templates enhances collaboration among teams and with the wider design community.

- **Customization**: Designers can tailor templates to specific project requirements, enhancing design accuracy and effectiveness.

Released in IDEA StatiCa patch 23.1.5.

Enhanced sketches in reports

By optimizing the presentation of sketches in reports, you can create the drawings of the connection design easily.

2D drawings, named **user-defined sections** in the Connection app, are one of the most common ways to provide details of connection design.

Therefore, after refining the data contained in the sketches and avoiding overlapping of the annotations, you can achieve optimized readability. That means that every detailer is able to easily and clearly read the required data from the sketches.

The results in enhancement to the Section model view – drawings ("sketches").

Better readability of sketches

The sketches' readability is critical for avoiding mistakes in the final design in 3D CAD software due to the needs to avoid overlapping texts with the lines, etc. By providing clearer sketches where the annotation marks do not overlap with other objects anymore, the sketches are simple, clear, and fully readable.

No overlapping of texts and lines

The special allocation algorithm eliminates the aforementioned overlapping of annotation marks in sketches, preventing the occurrence of cluttered sketches.

To enhance sketch readability and annotation distribution, we use a quadrant principle: By centering the picture and dividing sketches into four quadrants around the node base point, annotations are strategically placed in the best positions within the corresponding quadrant. Using this principle, we reach a much better and more logical distribution of the annotation marks of plates, welds, and bolts.

Big display in the report

Sketches occupy approximately two-thirds of an A4 page, ensuring readability and utility in the final report. This prevents sketches from being too small and surrounded by excessive empty spaces, detracting from the overall quality and effectiveness of the documentation.

Sketches are set to ON in the report by default.

Improved annotations of operations

Annotations in key manufacturing operations such as fin plate, widener, stiffener, and gusset plate, help streamline the design process and eliminate sketch overlaps. Repositioning annotations, adding missing dimensions, and correcting inaccuracies avoid the risk of unclear sketches and incomplete details.
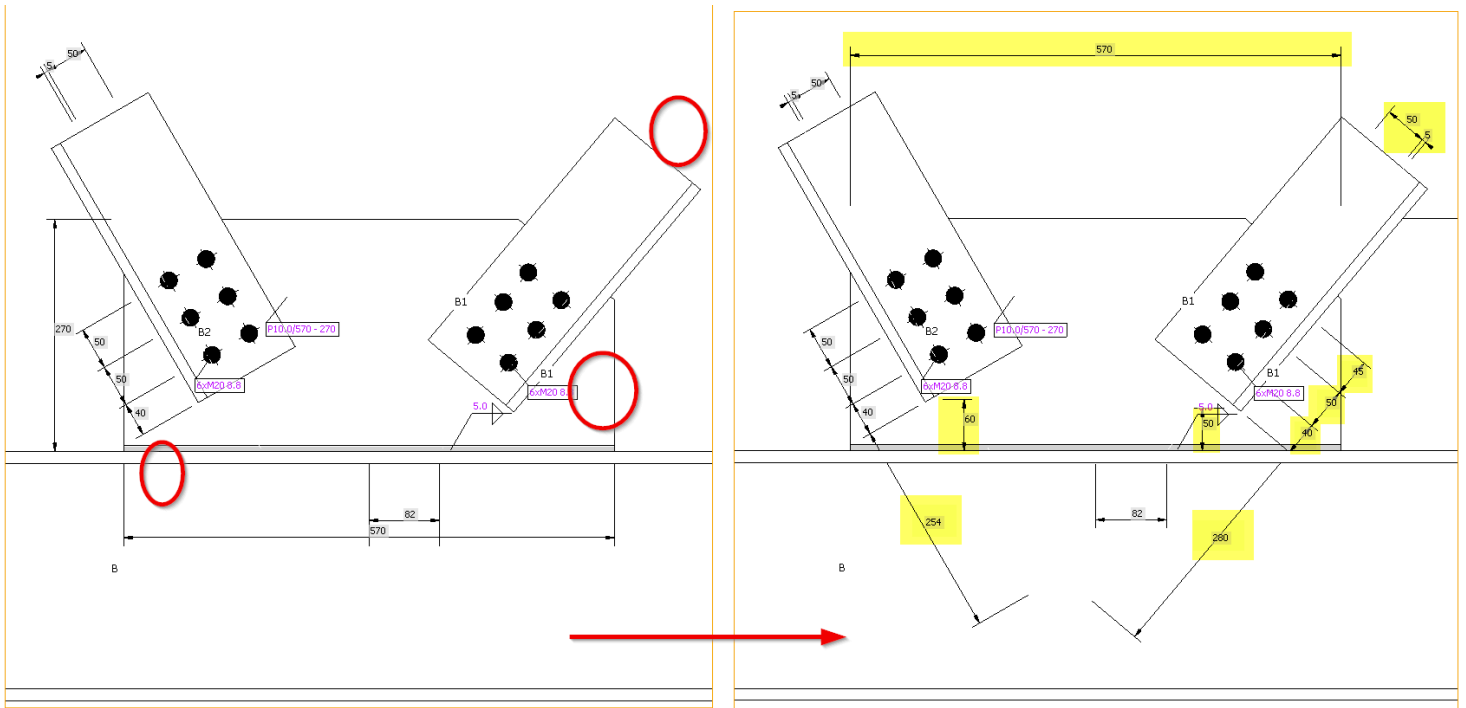
Section Editor and settings

The Section Editor accelerates the management and customization of user-defined sections within the Design tab, directly affecting ease of use. These settings are reflected in the Report and preserved with the model, ensuring consistency across documentation. You have the flexibility to easily remove sections or tailor the visibility of various annotations to suit the complexity of the section, resulting in clarity and user control over section presentations.
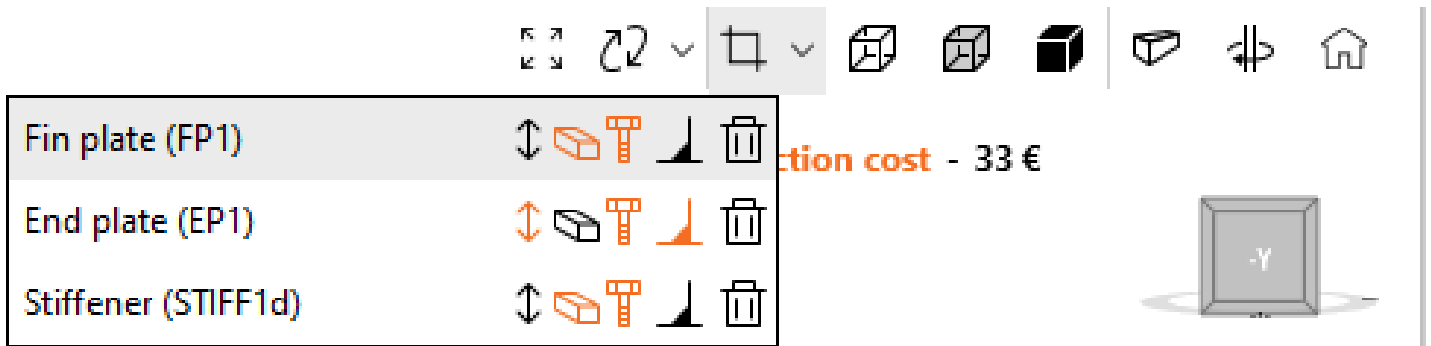
It is available in the Design tab, directly under the original button for opening the section mode view.

Functions available in the section editor are:

- Show/hide annotations
- Show/hide member and plate labels
- Show/hide bolt descriptions
- Show/hide weld symbols
- Delete sketch
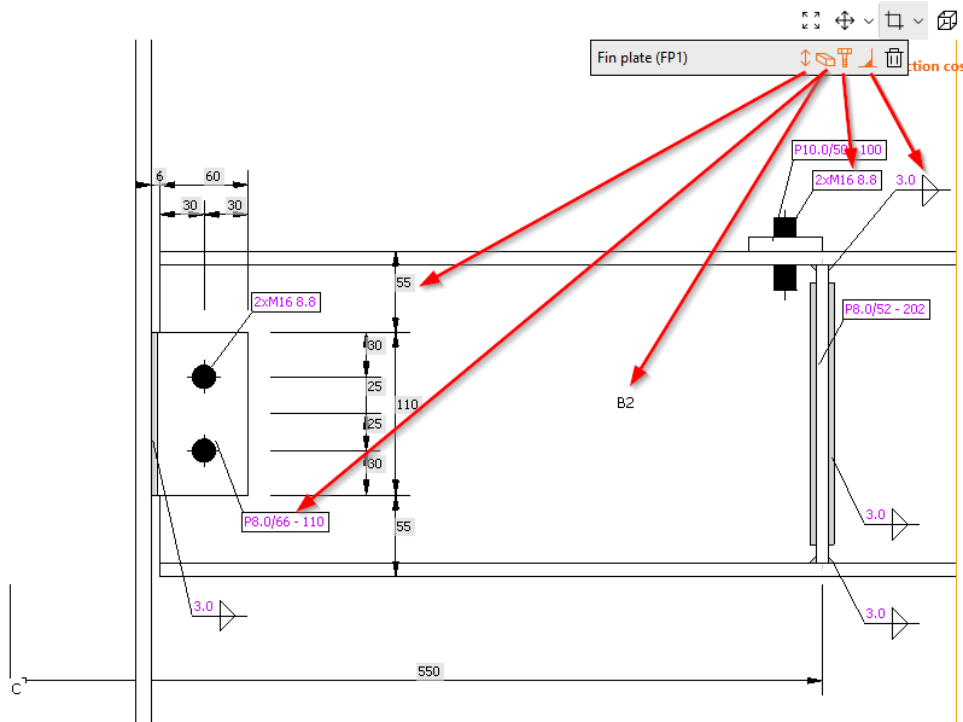
How to work with sketches

- To create a new sketch/section view, use the **right mouse button** in the 3D scene and select **Create section**.
- **Rotate** the model to view the orange box defining the section.
- **Click its boundaries** and **adjust** them using the arrows around the box.
- In the menu above the 3D scene, you can **select** which **entities** will be included in the section view.
- The sketch will be automatically **printed in the report** unless you change the settings.

You can edit the section anytime using the **Section editor** button in the menu above the 3D scene. Also, you can create as many sections as you wish.

Find out more in the How to create a section view article.

Released in IDEA StatiCa version 24.0.
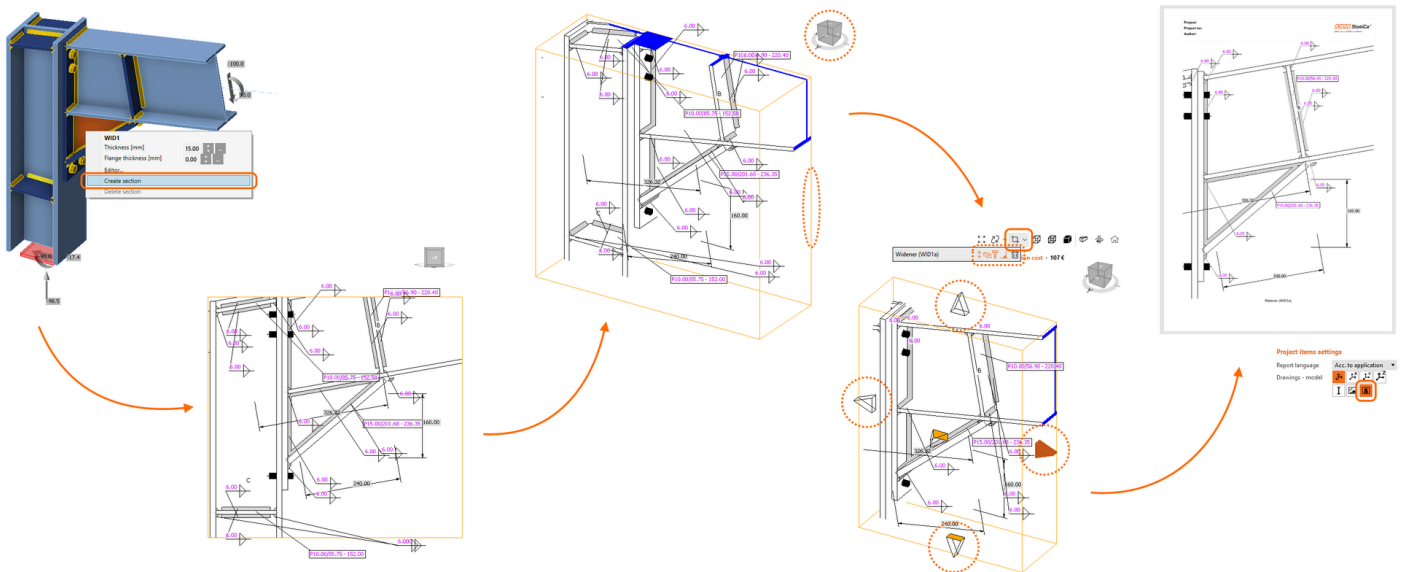
Automatic weld sizing to ductility

Automatic weld sizing removes the tedious and time-consuming manual input and check of each weld. With the automating algorithm, IDEA StatiCa provides faster modeling and absolutely safe design of welded connections.

The aim of this feature is to **control the ductility level in all welds** in the connection **with one click**! This weld sizing approach uses an algorithm that adjusts the **weld size according to the strength of connected plates** (parent metal). In the IDEA StatiCa Connection application, there are three levels of weld sizing per ductility:

- Weld sizing to minimum ductility (Eurocode only)
- Weld sizing to full-strength
- Weld sizing with overstrength

The algorithm for each level of ductility is governed by the following equations.

Minimum ductility
$$a/t = \frac{\beta_w \gamma_{M2} f_y}{\sqrt{2} f_u \gamma_{M0}} \cdot \min\left\{1.0, 1.1\frac{f_y}{f_u}\right\}$$

Full-strength
$$a/t = \frac{\beta_w \gamma_{M2} f_y}{\sqrt{2} f_u \gamma_{M0}}$$

Over-strength
$$a/t = \frac{\beta_w \gamma_{M2} f_y}{\sqrt{2} f_u \gamma_{M0}} \cdot overstrength\ factor\ in\ Settings$$

Weld sizing to minimum ductility

This method modifies weld size to minimum ductility of welded joints as per the Eurocode requirements in FprEN 1993-1-8:2023 – 6.9(4), and it applies to all welds and all different steel grades used in the model. The weld resistance is designed to be high enough to prevent brittle failures in the connection and, at the same time, allow plastification in the connected weld.

(4) Where rupture of the weld is to be avoided for ductility purposes, the welds should have sufficient strength not to rupture before yielding in the adjacent parent metal in order to comply with (3). The welds should be designed to have a resistance equal to:

— 1,1 f_y/f_u times the design resistance of the weakest of the connected parts,

— but not more than the design resistance of the weakest of the connected parts.

Minimum ductility requirements are also included in the **detailing check** of welds.

Check of welds for extreme load effect

	Status	Item	Edge	Th [mm]	L [mm]	Loads	$\sigma_{w,Ed}$ [MPa]	ε_{Pl} [%]	$\sigma_\perp$ [MPa]	$\tau_\perp$ [MPa]	$\tau_\parallel$ [MPa]	Ut [%]	Ut_c [%]	Detailing
⊟	✅	M1-bfl 1	M2-bfl 1	◢ 4.0 ◣	199	LE1	432.0	3.0	-122.4	-122.5	205.5	99.2	99.2	ⓘ

$\sigma_\perp = -122.5$ MPa   – Normal stress perpendicular to the throat

$\sigma_{\perp,Rd} = 352.8$ MPa   – Perpendicular stress resistance

Detailing check

- *Weld is too small. Throat thickness should be greater or equal to 9.2 mm where rupture of the weld is to be avoided for ductility purposes. (FprEN 1993-1-8:2023 – 6.9(4))*

	Status	Item	Edge	Th [mm]	L [mm]	Loads	$\sigma_{w,Ed}$ [MPa]	ε_{Pl} [%]	$\sigma_\perp$ [MPa]	$\tau_\perp$ [MPa]	$\tau_\parallel$ [MPa]	Ut [%]	Ut_c [%]	Detailing
➕	✅			◢ 4.0 ◣	199	LE1	432.0	3.0	-122.5	122.5	-205.5	99.2	99.2	ⓘ

*Note: Weld sizing to minimum ductility is available for Eurocode (EN) only.*

Weld sizing to full-strength

The full-strength weld sizing algorithm produces thicker welds than the minimum ductility type. The resistance of the weld is higher than the resistance of the plate connected by edge. This means, welds designed with this criteria will not brittle fail before the plate fails.

## Weld sizing with overstrength

This type of weld sizing produces even thicker welds than the full-strength, so the welds are much stronger than the connected plate. The aim is to add more safety in the plastic design approach and for cyclic loading scenarios (e.g., some seismic loads). The algorithm here is the same as for the full-strength, but the weld size is multiplied by the **Overstrength factor**.

By default, the Overstrength factor is set to 1.4 and can be changed in the project **Settings/Design/Autodesign/Overstrength factor**.
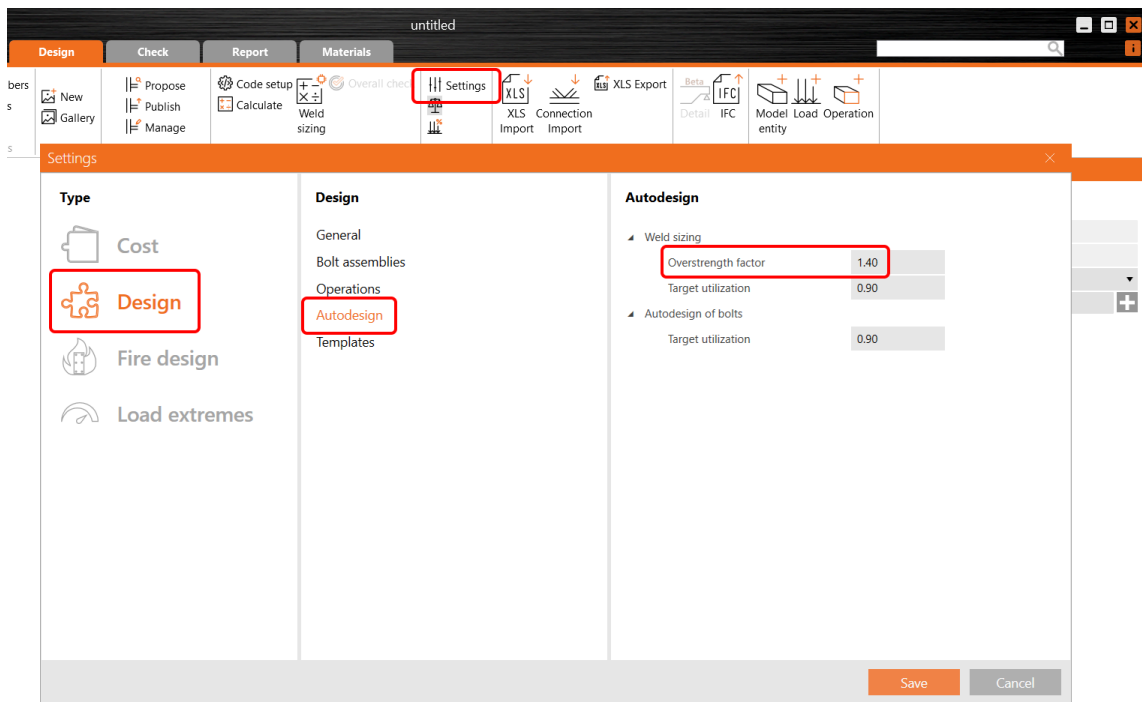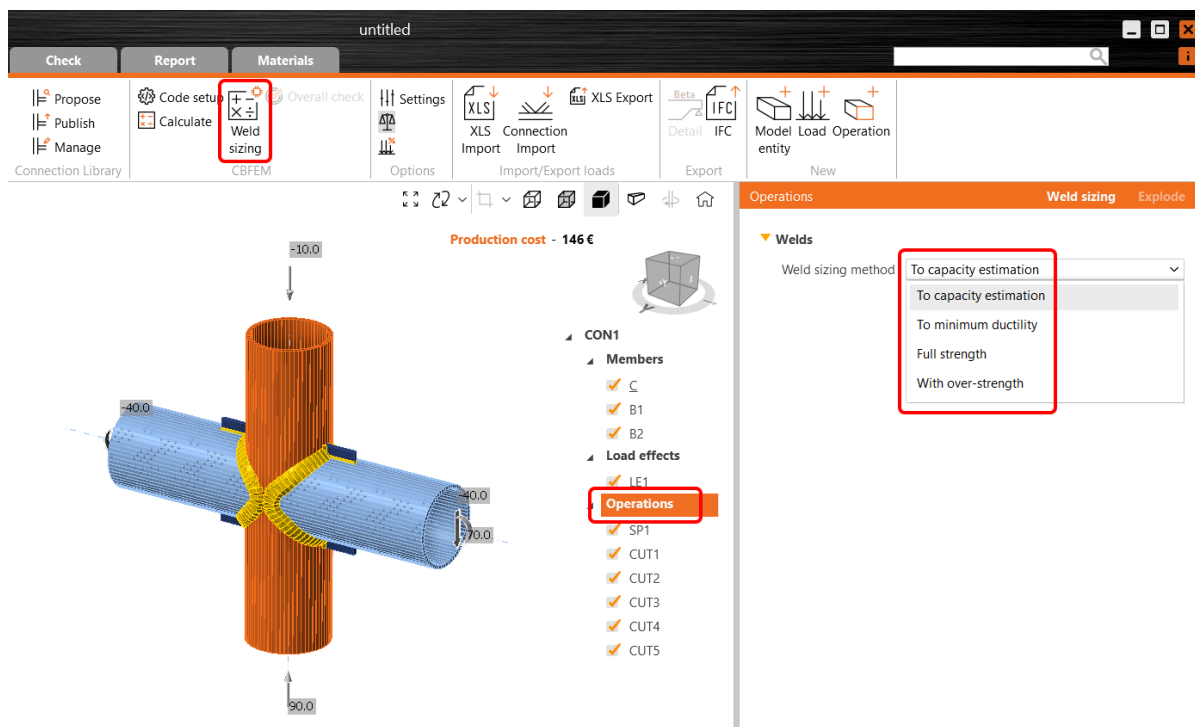
Executing the weld sizing to ductility levels

To run the automatic weld sizing, right-click the **Operations** item in the tree menu and carry out the weld sizing from here by choosing the weld sizing method.
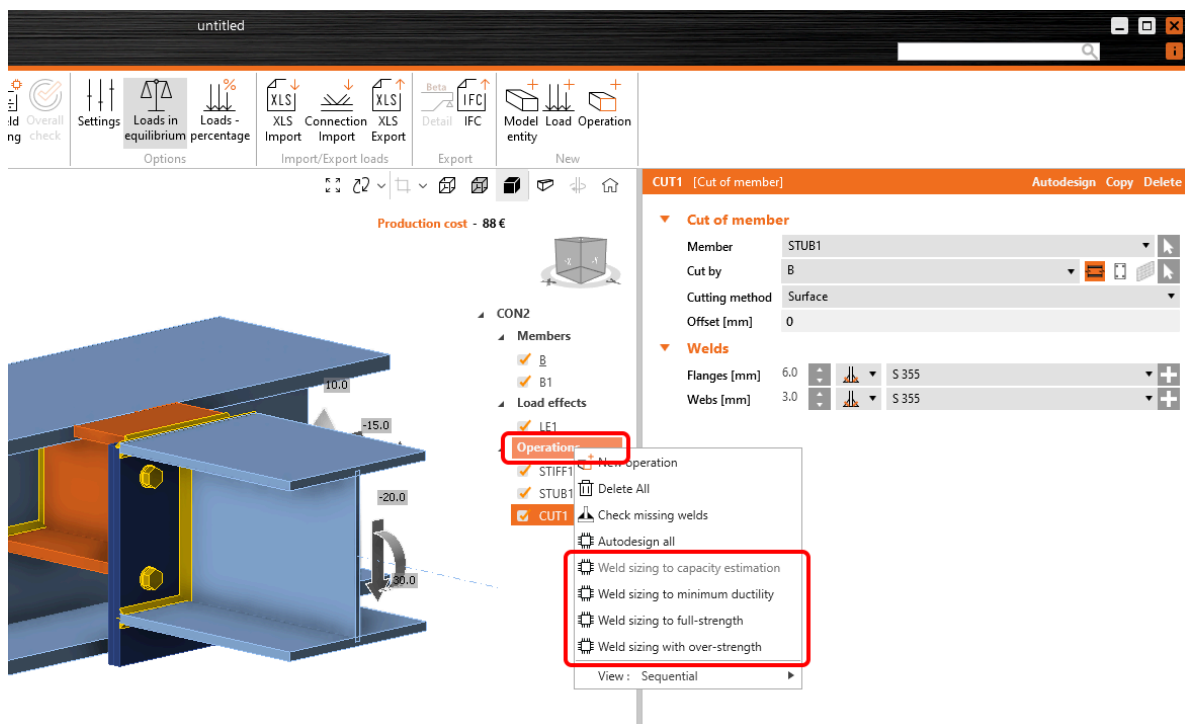
Alternatively, you can click the **Operations** in the tree menu, select the desired **Weld sizing method**, and run the **Weld sizing** command in the ribbon.

Welds size **rounding up** can be defined in application **Preferences/New entity rounding/Weld size**.
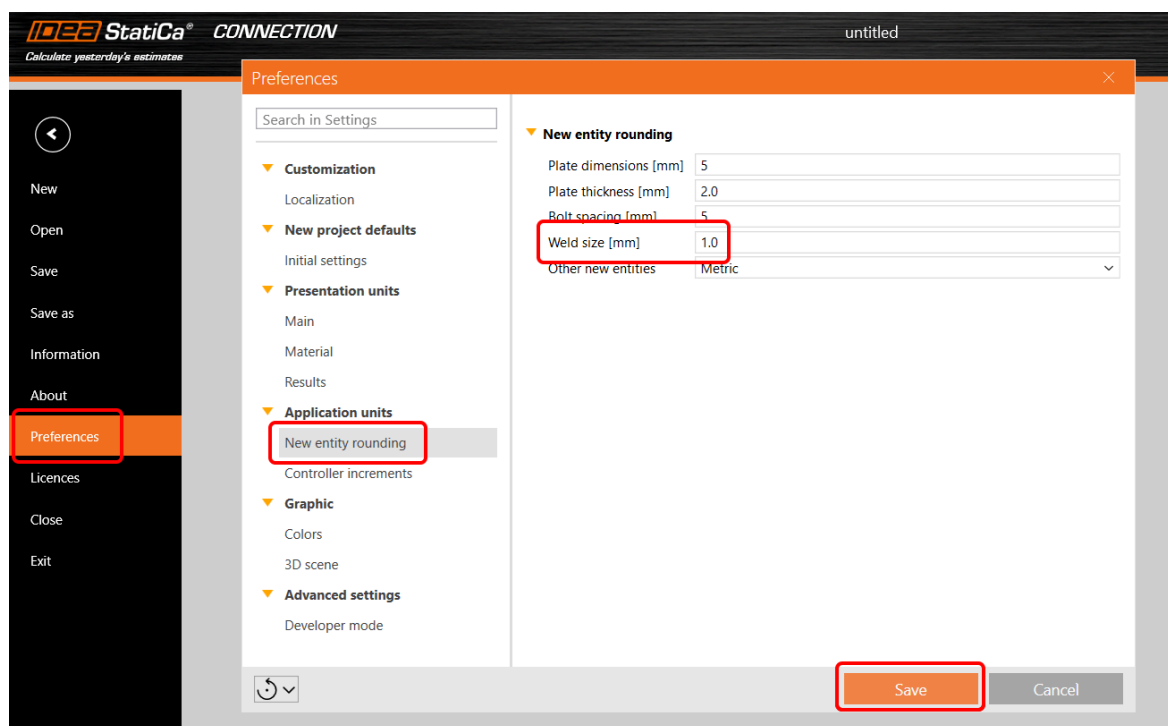
Read more about the weld sizing method in the Theoretical background chapter.

Released in IDEA StatiCa version 24.0.

Automatic weld sizing to capacity estimation

Automatic weld sizing addresses the challenge of manually adjusting each weld size, which is both tedious and time-consuming. By automating this, IDEA StatiCa significantly helps you speed up the design process and fosters more consistent weld designs across projects.

Simply put, weld sizing to capacity estimation enables you to, **with one click, optimize the size of each weld** across the whole connection model with a **preset utilization level**! Such easiness in user execution makes this feature the most advanced weld optimization tool on the market.

Machine learning technology

This feature for Eurocode (EN) projects performs the capacity estimation of all welds in the model, leveraging the machine learning data containing thousands of weld models with various loading scenarios. The capacity estimation routine considers **both connection design and the load effects**.

First, the Connection model is calculated with the original welds, and then these results are compared to the **vast library of machine-learned results database**, and weld sizes are **optimized**.
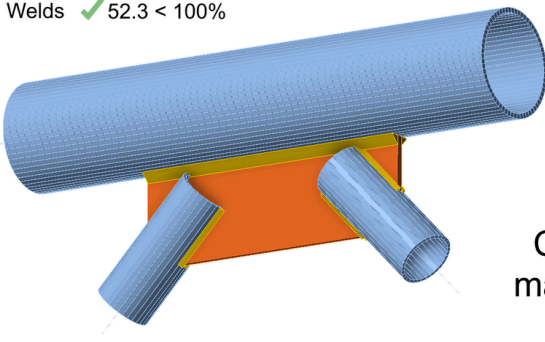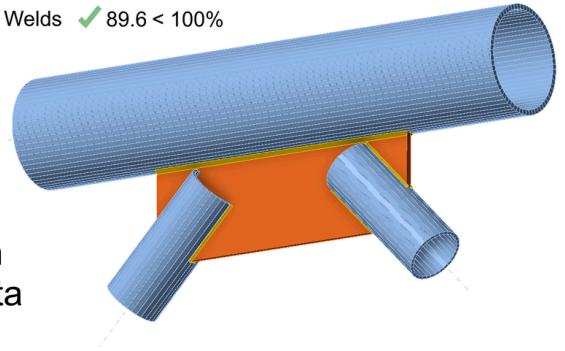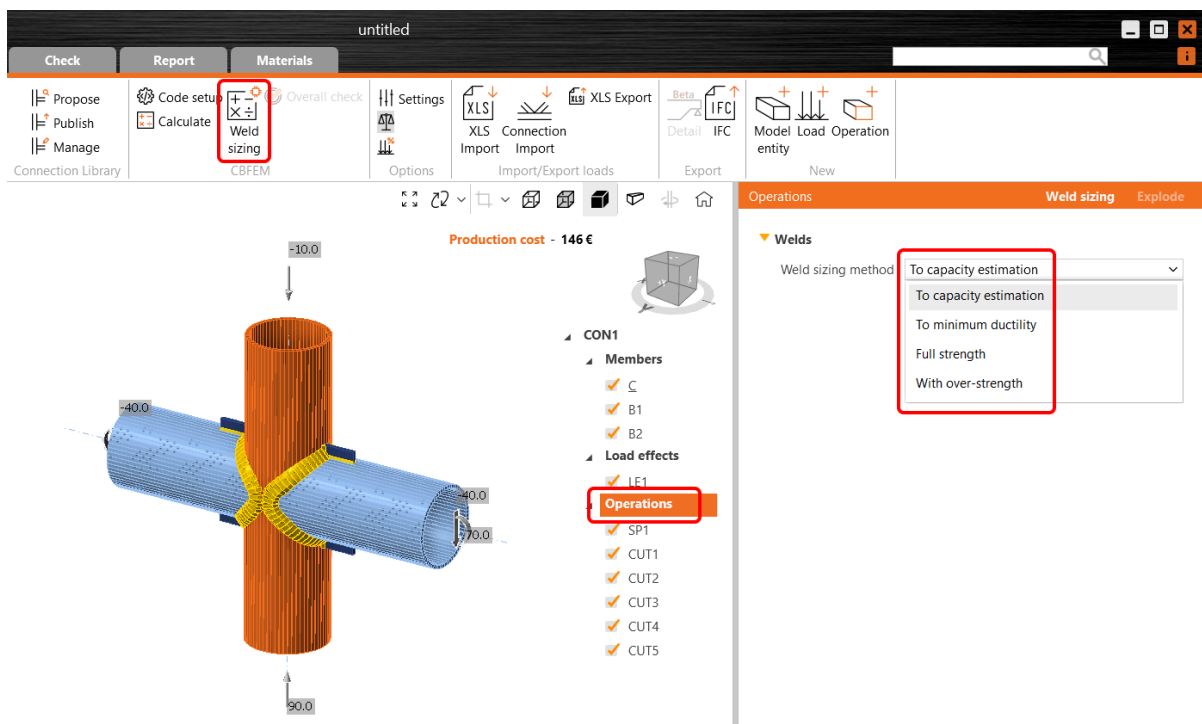

More about capacity estimation weld sizing in the article Welds – autodesign, input, warnings, visualization.

How to perform the capacity estimation weld sizing

To execute the automatic weld sizing, click the **Operations** in the tree menu, select **To capacity estimation** weld sizing method, and run the **Weld sizing** command in the ribbon.

Alternatively, you can right-click the **Operations** item in the tree menu and carry out the weld sizing from here. For **To capacity estimation**, you must first run the standard **Calculation**.
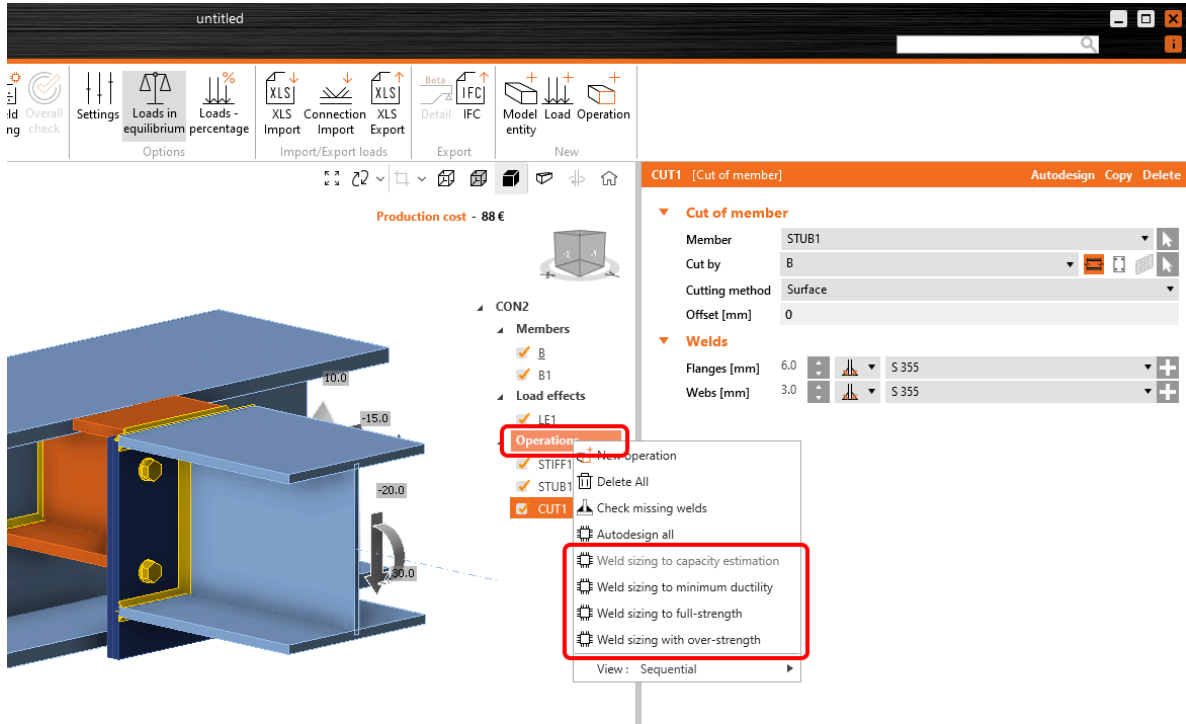

Utilization level and precision settings

Welds are automatically adjusted based on utilization using this method to calculate the optimized throat thickness:

Throat thickness$_{opt}$ = Throat thickness * Ut$_c$ / Target utilization

The utilization value is displayed in **Check** under the **Welds** tab in the U_{tc} column. More about Utc and welds in this article.

Welds **utilization level** can be controlled in the project **Settings/Design/Autodesign/Target utilization**.
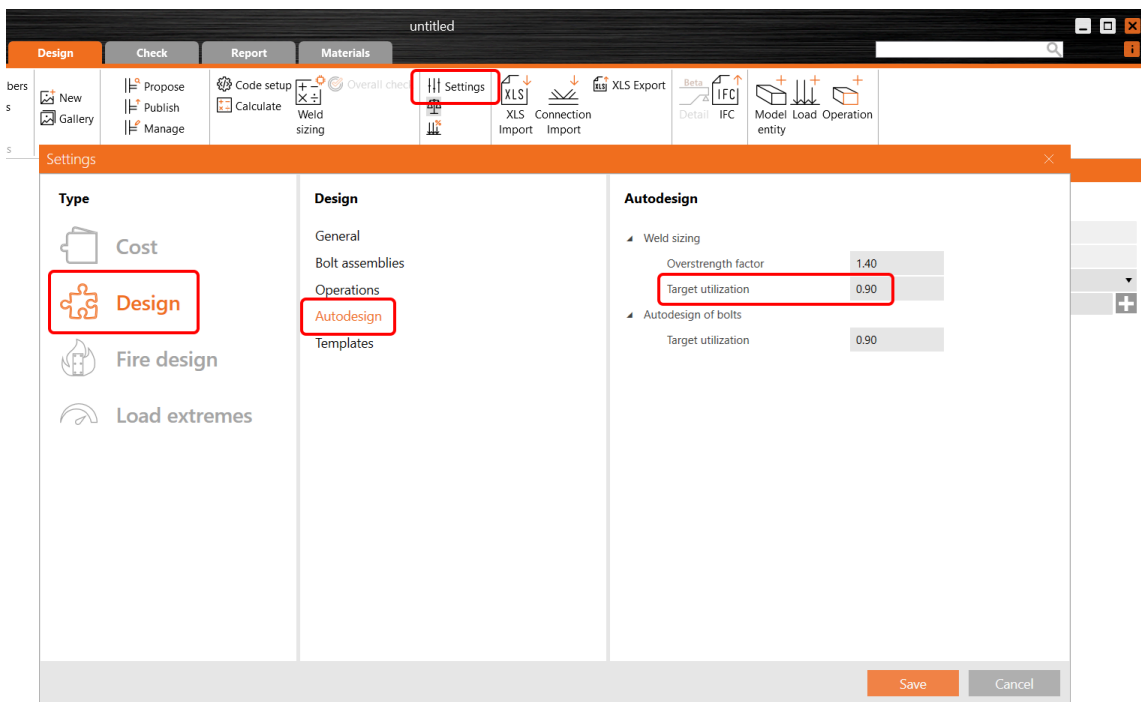
Welds size **rounding up** can be defined in the application **Preferences/New entity rounding/Weld size**.
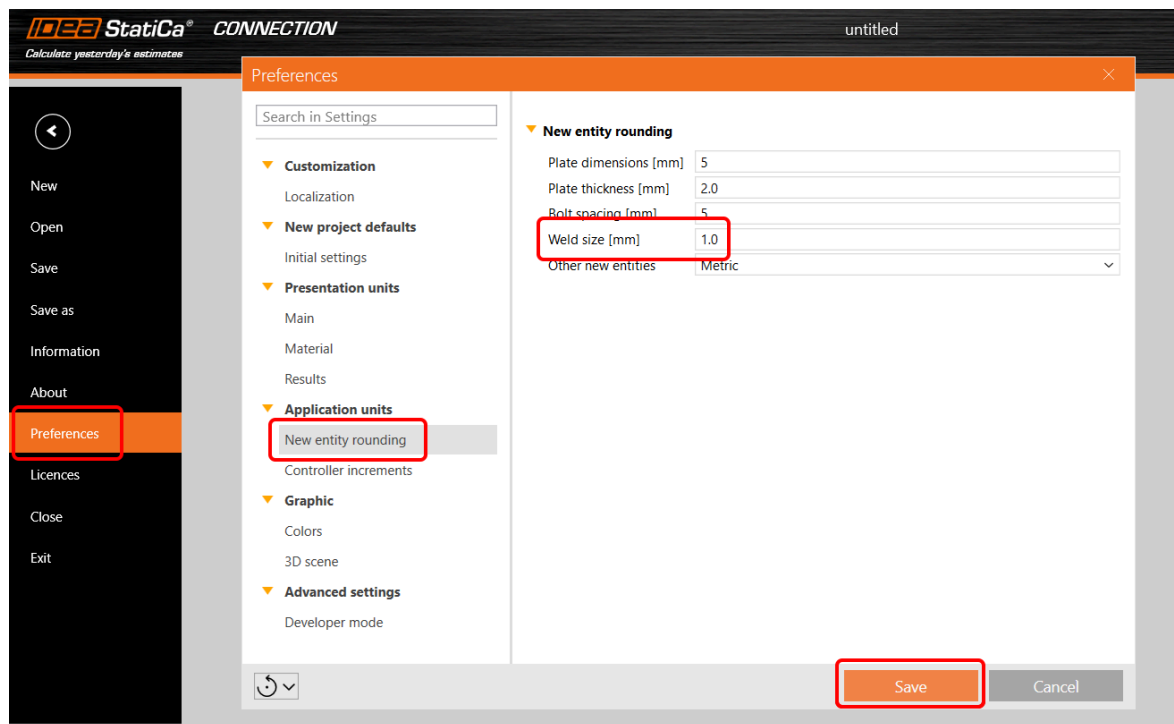
Detailing requirements are taken into account regarding the weld sizing.

Read more about weld sizing in the Theoretical background.

Note: Weld sizing to capacity estimation is available for Eurocode (EN) only.

Released in IDEA StatiCa version 24.0.

Partial Joint Penetration (PJP) groove welds

The integration of partial joint penetration (PJP) groove welds within the AISC guidelines in IDEA StatiCa Connection addresses the specific requirements set by AISC for PJP butt welds, distinct from those for fillet welds.
IDEA StatiCa Connection offers PJP groove welds as an option for welds as per **AISC 360 standards** (2010, 2016, 2022 in table J2.5). Users can specify any weld that is **edge-to-surface** as a PJP weld in every manufacturing operation, e.g., weld or contact, fin plate, etc.

The checks are affected if the Base metal capacity at the fusion face is turned on in the Code setup. The **Directional strength increase** is not accounted for when using PJP welds.

For better reporting, **special symbols** for PJP welds and **detailed formulas** are provided, along with an adjusted analysis model that reflects accurate load-deformation curves and plastic stress redistribution.

		C-tfl 1	B-tfl 1	E70xx	4 3/8	-	6.48	0.72	LE1	2.66	8.02	33.2	

Weld resistance check (AISC 360-22 – Table J2.5)

$$\phi R_n = \phi \cdot F_{nw} \cdot A_{we} = \quad 8.02 \quad \text{kip} \quad \geq \quad F_n = \quad 2.66 \quad \text{kip}$$

Where:

$F_{nw} = 42.0 \text{ ksi}$ – nominal stress of weld material:

- $F_{nw} = 0.6 \cdot F_{EXX}$, where:
  - $F_{EXX} = 70.0 \text{ ksi}$ – electrode classification number, i.e. minimum specified tensile strength

$A_{we} = 0.25 \text{ in}^2$ – effective area of weld critical element

$\phi = 0.75$ – resistance factor for welded connections

In **user-defined sections**, a butt weld symbol with the correct effective throat thickness is displayed so that the produced drawings for reports are fully reliable.
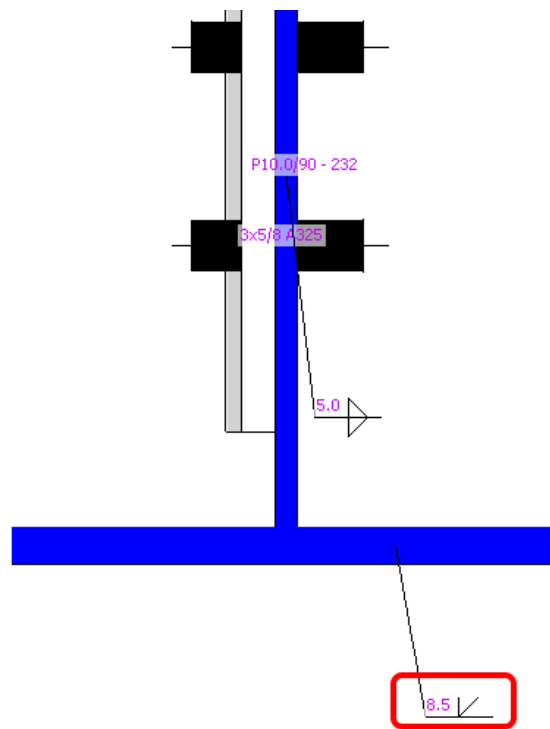
The **edge-to-rod type** of PJP welding is not available and is converted to a full-penetration butt weld, which is not checked and is considered a part of the parent metal plate. Read about how to use rod member connections.

The ability to define PJP welds broadens the scope of design possibilities, allowing for more accurate and code-compliant projects.

Note: The Partial Joint Penetration (PJP) groove welds are available for AISC code only

Released in IDEA StatiCa version 24.0.

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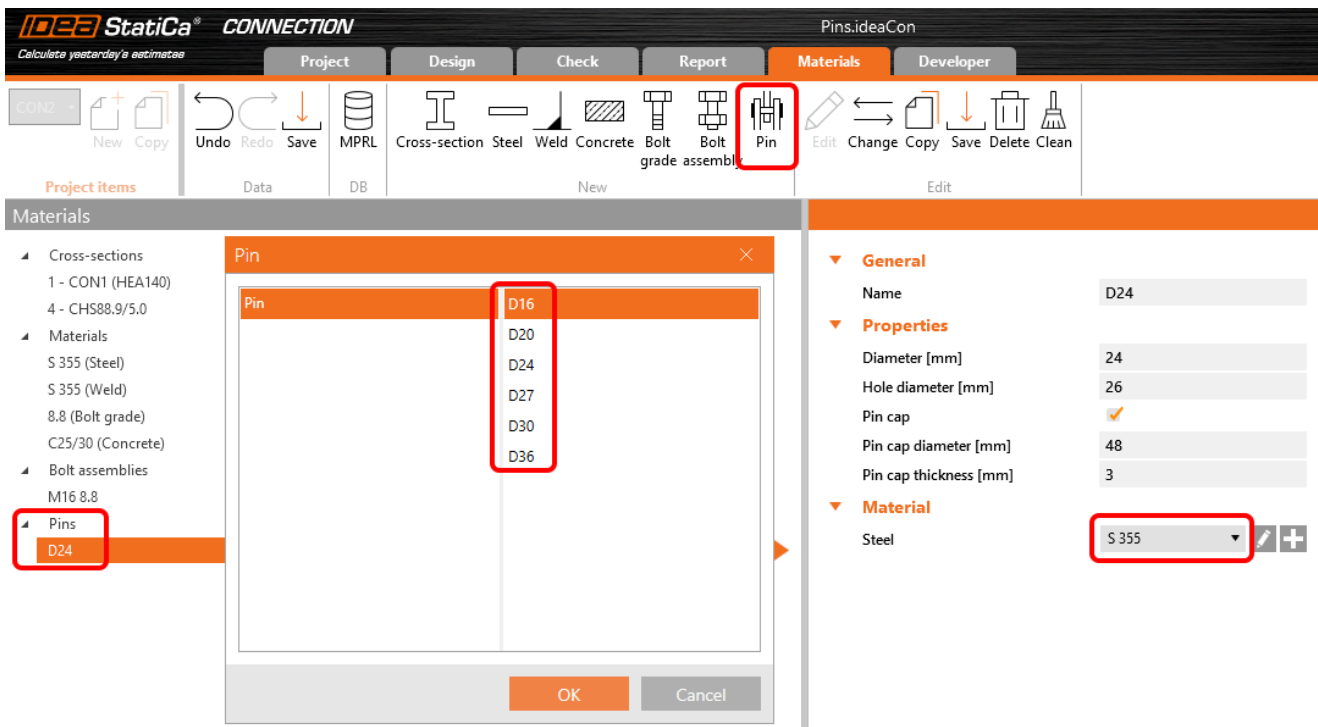
A **Pin cap** can be turned on, but it doesn't influence the checks or analysis, only the visual presentation of pins in the 3D scene.

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

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:

$$\sigma_{h,Ed} \le f_{h,Rd} \qquad \qquad ... (3.14)$$

where:

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

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

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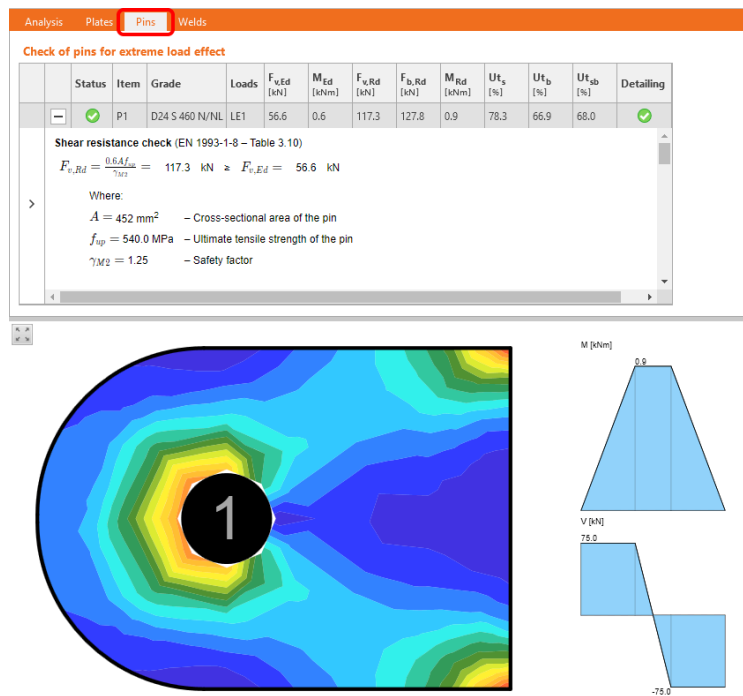
All relevant input data and result data of pins are included in the **report**.

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

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

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.

EN - analytical model   IDEA Connection
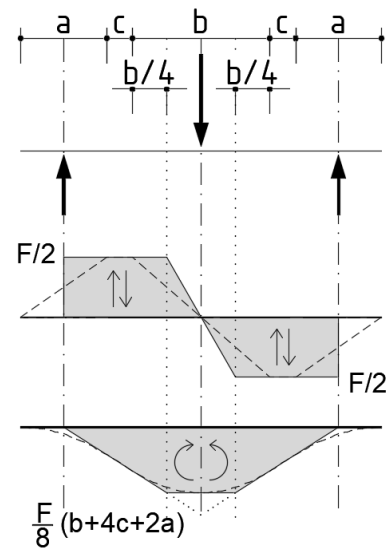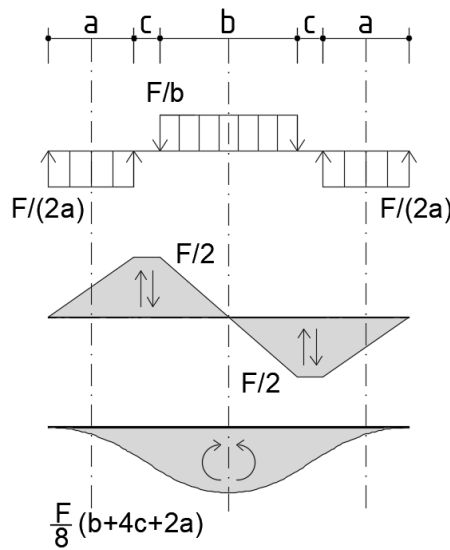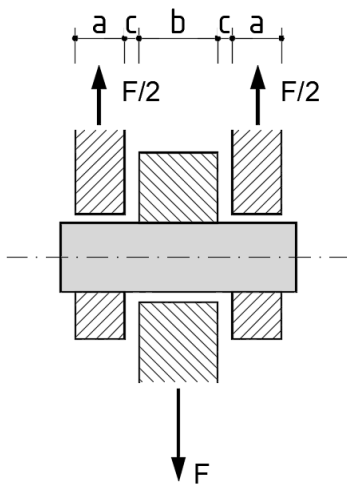
Note: Pins are available for Eurocode (EN) only.

Released in IDEA StatiCa version 24.0.

Hooked anchors/L-bolts in AISC

It is possible to define the L-shaped / J-shaped or hooked anchor bolt when using the AISC code. For this type of anchor, the pullout resistance is checked.

In the design and check of footings, an AISC project can consider a **hook type** of anchor for the foundation concrete block.

The L-shaped anchors can be defined either in the **base plate** or in the **fastener grid or contact** operations.

The hook length must be at least three times the anchor diameter, as per ACI 318-19 - 17.6.3.2.2. A warning appears when this condition is not met.

Anchors

Type	5/8 A325
Anchoring length [mm]	300.00
Anchor type	Hook
Hook length [d]	3
Top layers [mm]	40.00
Left layers [mm]	40.00
Shear plane in thread	✔

Error
×

Hook length must be at least 3 times anchor diameter

OK
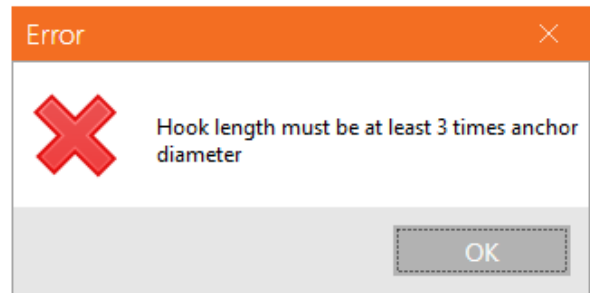
You can find the concrete pullout resistance check as per ACI in the report. For more information about the checking of anchors and a concrete block according to AISC, go to Check of steel connection components (AISC).

Concrete pullout resistance (ACI 318-14 – 17.4.3)

$$\phi N_{pn} = \phi \cdot \Psi_{c,P} \cdot N_p = \quad 1182.4 \quad \text{kN} \quad \geq \quad N_f = \quad 41.9 \quad \text{kN}$$

Where:

$\phi = 0.70$ — resistance factor

$\Psi_{c,P} = 1.00$ — modification factor for concrete condition

$N_p = 1689.1$ kN — basic concrete pullout strength for headed anchor:

- $N_{pr} = 8 \cdot A_{brg} \cdot f'_c$, where:
  - $A_{brg} = 7656$ mm^2 – bearing area of the head of stud or anchor bolt
  - $f'_c = 27.6$ MPa – concrete compressive strength

Released in IDEA StatiCa version 24.0.

Check of anchor tension as a bolt tension option (AS)

As per the Australian Standard (AS), anchor bolts are traditionally checked with the same formula as bolts. This differs from the low resistance factor in the new code for anchor bolts (especially post-installed).

In projects under the **AS standard**, for the anchor tensile resistance check, you can choose to follow the following (in the **Code Setup**):

- AS 5216:2018 – 6.2.2 (default)
- AS 4100:2020 – 9.2.2.2
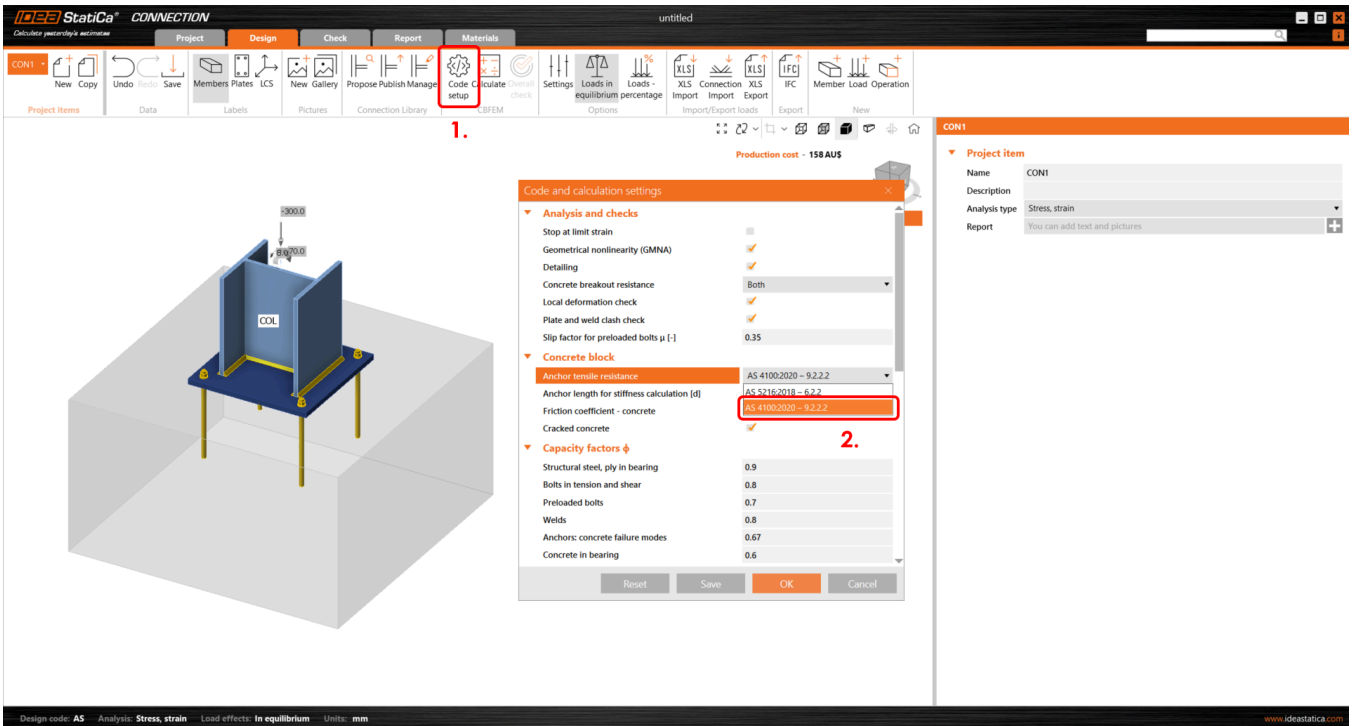
If the AS 4100:2020 – 9.2.2.2 (as it is used for bolts) is selected, the anchor tensile resistance is replaced by the bolt check as follows:
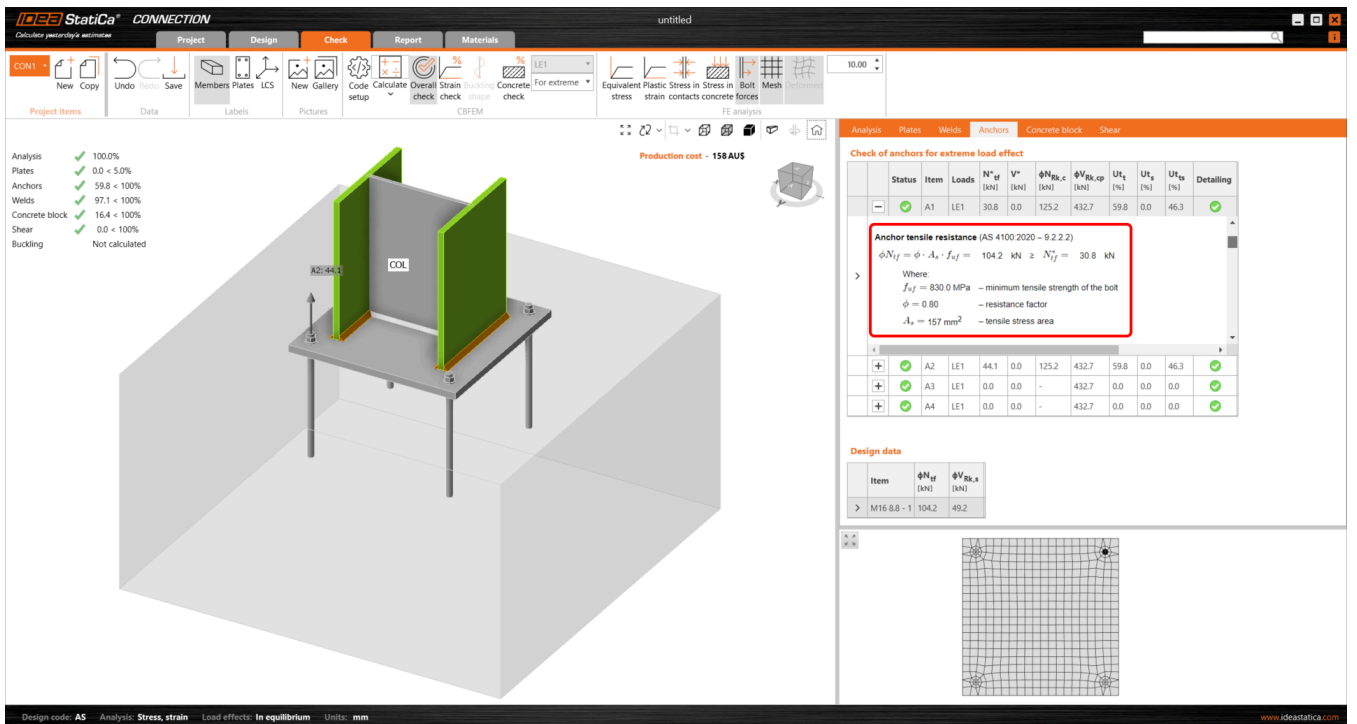
Anchor tensile resistance (AS 4100:2020 – 9.2.2.2)

Released in IDEA StatiCa version 24.0.

Change just the language of the report

Generate the report of your project in a different language while keeping the application in yours. This feature helps you easily deliver the report in the language spoken by your customer.

Sometimes, the project report should be submitted in a language that you do not speak. In some software, that means changing the language of the whole application (and restarting it) just to print the reports.

The possibility of changing the report language under the **Report** tab makes this tedious process unnecessary.
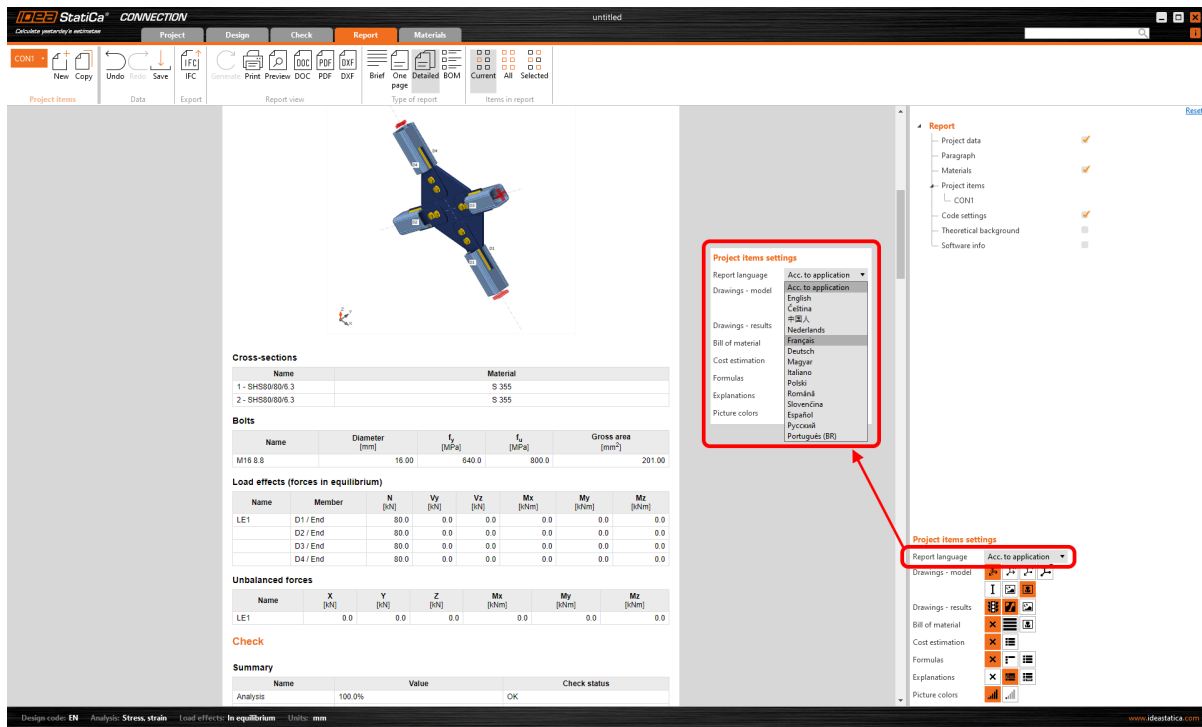
After re-generating the report, the document can be saved/printed in the changed language while the application interface stays unchanged.
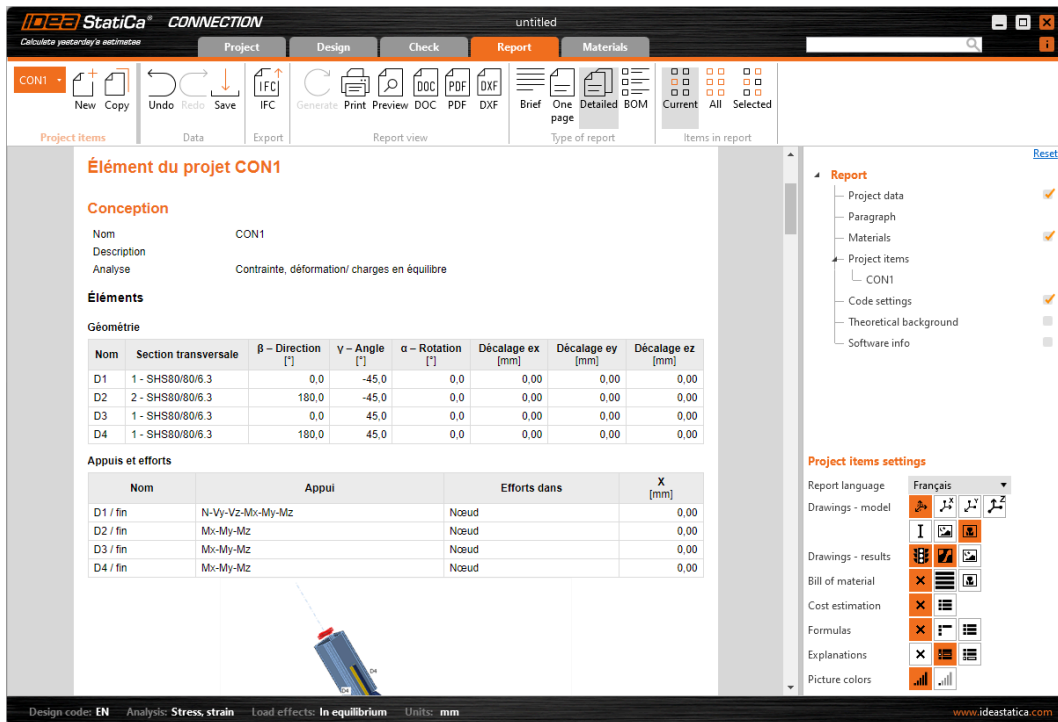
Available in IDEA StatiCa Connection and Member apps.

Released in IDEA StatiCa version 24.0.

Sliding option for a concrete foundation block

Having the option of shear force transfer for a foundation concrete block enables the simulation of a sliding hinge support model in a Member project.

When modeling a simply supported beam in IDEA StatiCa Member, you can use this option for defining the roller (sliding hinge). This option keeps the "close-to-zero" bending moment at the ends of the beam supported by a concrete block.
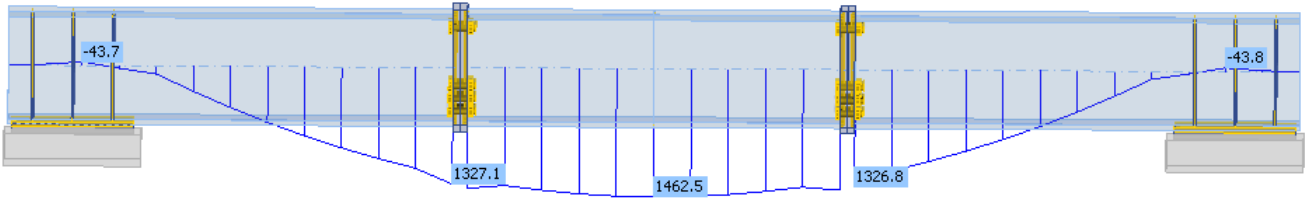
From Member, open the connection at one end of the beam. Define the anchoring/foundation concrete block and switch the **Shear force transfer** option to **Sliding**. Also, **remove all the anchors** in the **Editor**.

Please note that the other side of the beam must stay active to transfer forces in both directions so that the static system remains determined.

Although this setting also works in Connection, it is specifically designed to be used in IDEA StatiCa Member.

This feature was released in the IDEA StatiCa 23.1.1 patch.

Execute plate cuts in parallel planes

IDEA StatiCa supports the capability to execute plate cuts in parallel planes.
The **Cut of plate** operation offers the possibility to cut plates by other plates and negative volumes defined within the same plane. This is beneficial for handling general plate shapes imported from DXF files, addressing the plate cuts that could not be performed in the plate Editor.

To use this feature, select the **PCUT** (Cut of plate) operation and choose the **Surface** cutting method within the parameters of picked plates or negative volumes.
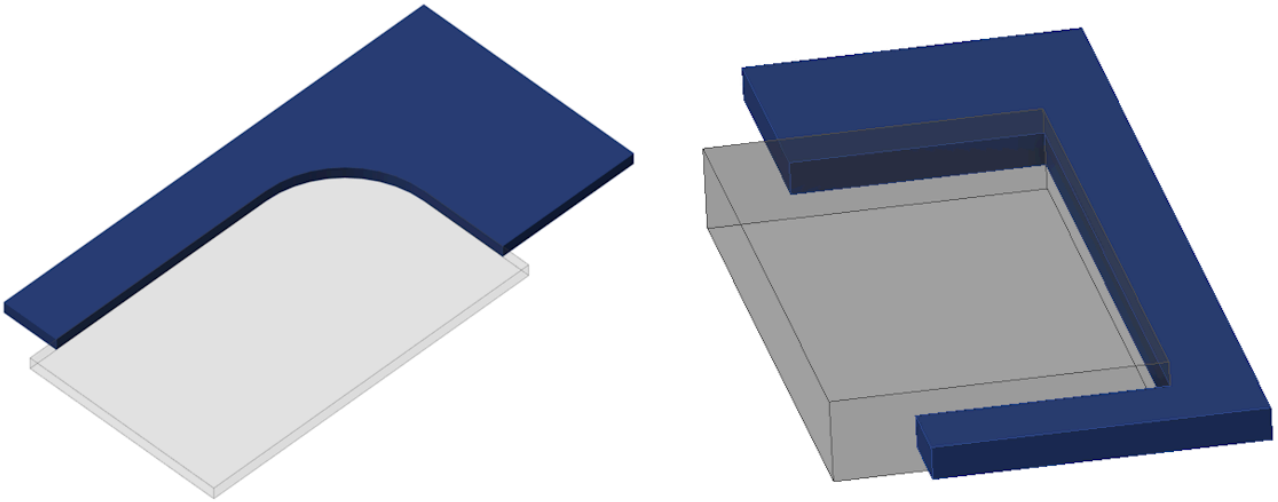
See more about modeling in IDEA StatiCa Connection in the recorded webinar Get the most out of Connection modeling.

Released in IDEA StatiCa patch 23.1.1.

Manual control over the extension of members

By enabling manual control over the extension of members, users of Connection and Member apps are afforded additional modeling options, facilitating precise and tailored project outcomes, thereby enabling more complex and accurate modeling capabilities.

This setting for "**Extend the member**" is incorporated in the Connection application inside the **Cut of member** manufacturing operation. To extend the member towards the farther point of the connected member, you also need to select the **Cutting plane** type "**Farther**".

No matter whether the Connection is opened from Member, Checkbot, or initiated from scratch, this functionality uniformly ensures a consistent experience across different workflows.

In the Member application, this feature allows users to extend members at both ends, particularly when an analyzed beam was also cut by another operation, permitting numerous design scenarios.
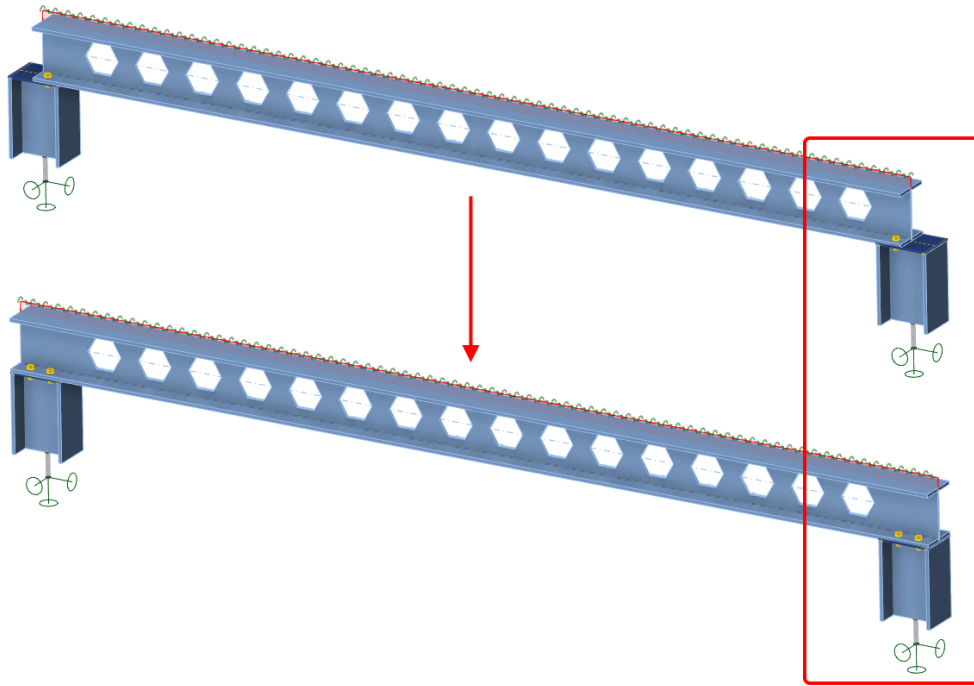
This unified approach to the extend function in the cut operation between the Connection and Member applications also prevents potential discrepancies.

Released in IDEA StatiCa version 24.0.

Template application dialogue with sketches and parameters

For applying new designs from Connection library templates, you can use the enhanced dialogue. This smart feature removes limitations with the bolt sizes, stiffening member or cleat element cross-sections and materials.

We have introduced the enhanced sketches or, in other words, simple drawings of connections that can be added and printed in the final report. Any defined sketch (user-defined section) can a part of design template published in the Connection Library.

When such a design template is applied to a new connection model, the redesigned conversion dialogue pops up enabling users to:

- See the design standard of the template and the design standard of the current project
- See an overview of members, which one is bearing (underscored), and the mapping between them
- Control mapping of stiffening members
  - Each operation has its own row
  - Each operation can have an assigned new CSS which will be added to the project
- Control mapping of materials (welds, steel, concrete …)
- Control the bolt assemblies mapping
- Control pins mapping
- Together with the template the parameters and parametric properties are copied
- Together with the template the sketches are produced
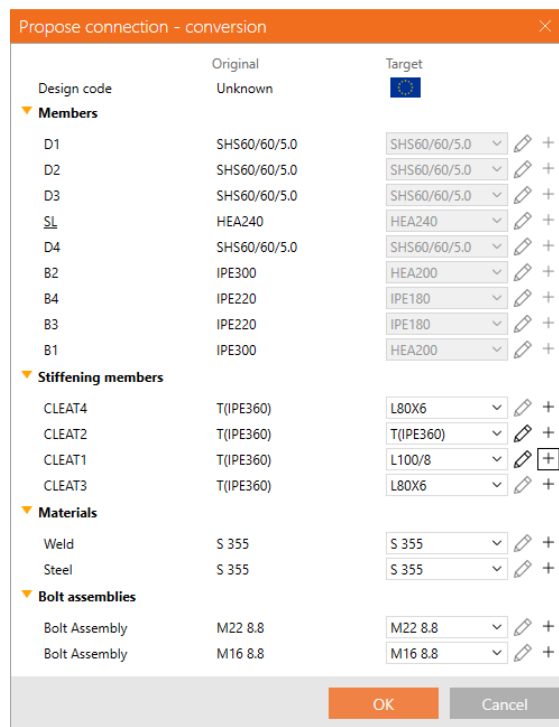
Released in IDEA StatiCa version 24.0.

Consistency in Connection and Member models

Cleat, Splice, Fin plate, and Stiffening plate operations are unified across the IDEA StatiCa Connection and Member applications in order to produce identical designs, which offers great flexibility and control over complex Member models.

The mapping of constraint points and the selection of plates in Cleat, Splice, Fin plate, and Stiffening plate manufacturing operations are consistent in the project data of all applications dealing with steel connections.

This prevents situations when in the Member app, a cut operation (utilizing cut by a plate, workplane of negative volume) is applied in one connection node of the analyzed member, causing discrepancies in the operations in other connection nodes of this member.
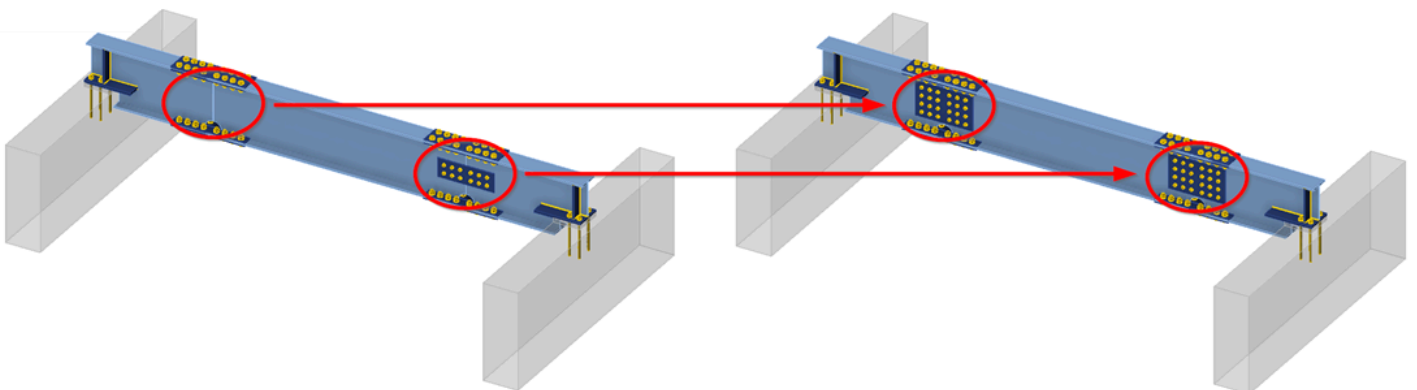
A parent-child relationship between plates and members is ensured even after one part of the Member model is touched by modifying the number of edges of plates and their ordering. This methodology ensures that even if the parent plate is removed, the subsequent operations can accurately reference the correct plate based on this relationship, maintaining the intended positioning designed in the Connection application.

Released in IDEA StatiCa version 24.0.

28

Limiting short lines in DXF imports

This functionality improves cross-section definition and, thus, the reliability of analysis by placing a strict limit on the minimum length of lines within imported DXF geometry.

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

Should the distance between any two points fall below this threshold, the user is promptly notified that the points are too close to each other, and the import of the geometry from the DXF file is halted. This measure necessitates users to adjust the geometry in the DXF file to meet the specified conditions before proceeding.
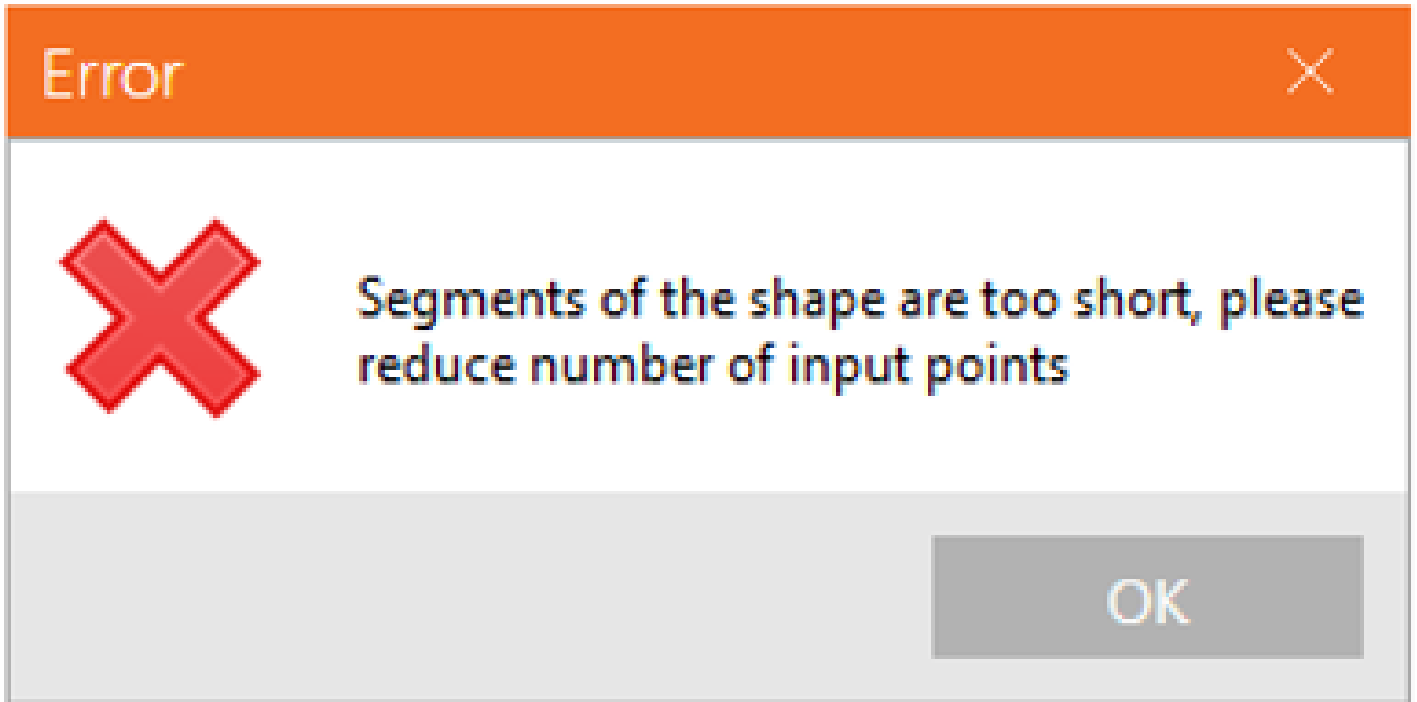
This addresses the challenge where analyses could be disrupted or the application might encounter crashes due to the presence of too short edges in the cross-section definition, leaving users uncertain of the problem's origin. Clearly informing users of non-compliant geometry at the moment of import enables the quick identification and correction of the issue, facilitating a smoother analysis process.

You can read more about creating your own build-up cross-section in How to create and use a custom cross-section and about importing a general plate from a DXF in How to import a plate from DXF.

Released in IDEA StatiCa patch 23.1.2.
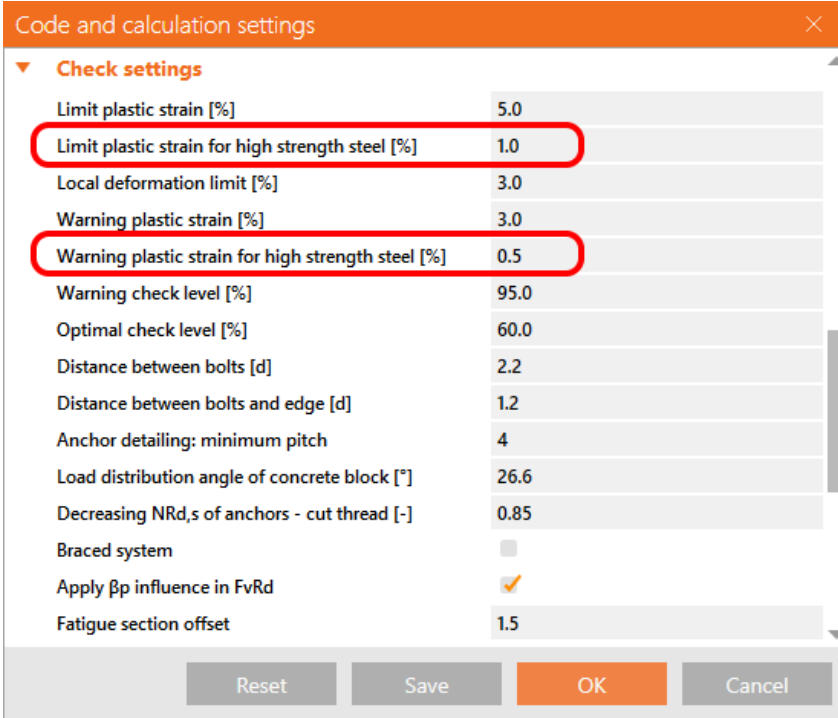
High-strength steel notifications

To enhance safety and accuracy in structural design, different values of the plastic limit strain should be used for high-strength steel grades.

While high-strength steel is not widely covered in the current edition of the Eurocode, it is included in new generations and product sheets. IDEA StatiCa's database accommodates high-strength steel, aligning with current design rules and anticipating future updates to codes and standards.

Enhanced safety and reliability

To recognize the unique characteristics of high-strength steel, a distinct plastic strain limit parameter is included in the **Code setup**, ensuring designs adhere to the most current research and standards.

The inclusion of a separate plastic strain limit for high-strength steel, with a default value set at 1%, allows for precise adherence to material-specific design principles. By specifying a more conservative plastic strain limit for high-strength steel, IDEA StatiCa contributes to the safety and durability of structures.

Design tab and material classification adjustments

For steel with yield strengths f_y exceeding 460 MPa, the software alerts the user of a tailored plastic strain limit, ensuring the integrity of designs involving high-strength steel. The automatic update from "Steel" to "High-strength steel" in the software's database underscores the importance of accurate material properties in design calculations.

The Check tab and Report sections reflect the plastic strain limit ε_{lim} as per various international codes, including EN, AISC, CSA, AS, and SP, providing a broad compatibility spectrum for global design practices.

Released in IDEA StatiCa version 24.0.

Out of surface load warning in Member

This warning in IDEA StatiCa Member ensures that the full magnitude of the load is applied to the analyzed member.

When applying a surface load on one of the member's plates, it may happen that part of the load already lies outside of the plate's surface. In such a case, the full magnitude of the load would not be applied – part of the load would not be transferred and used in the calculation. This might be unsafe and unintuitive, that's why there is a warning message.

When a line or point load distribution area is larger than the plate assigned, a warning is displayed in the top left corner of the screen, and you should adjust the position or the area of action of the load.

This warning helps you overcome these situations and, therefore, improves the safety of the model.

Released in the IDEA StatiCa 23.1.1 patch.

Autodesign of bolts in shear

Autodesign of bolts in shear can suggest the efficient design of fin plate or splice, taking into account the shear load. The layout of bolts is automatically adjusted to withstand the load and keep the given level of utilization. Autodesign of bolts in shear is another tool that helps speed up the design process and make it more straightforward and automated while keeping safety and cost efficiency. Next to Automatic weld sizing to ductility and Automatic weld sizing to capacity estimation, the feature is the next step in the automated process of connection design.

It brings the simplification of the iteration process of finding the proper design of connections. The autodesign of bolts in shear takes into account the **acting loads** on simple connections. Instead of an iteration process, the user just calls this autodesign function, and the application finds the proper design.

How to call the function

The function **Autodesign of bolts** is prepared for two operation types with bolts: **Fin plate** and **Splice.**
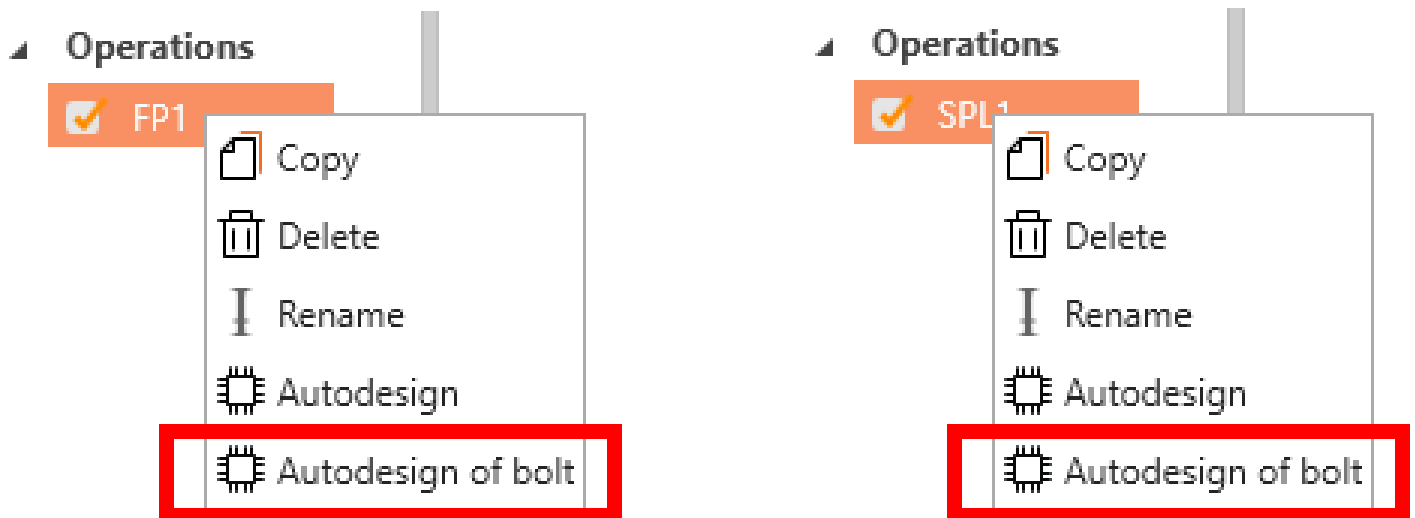
Once your bolted connection **has been calculated**, you can call the autodesign from the context menu of the operations – **right-click the operation** in the tree.
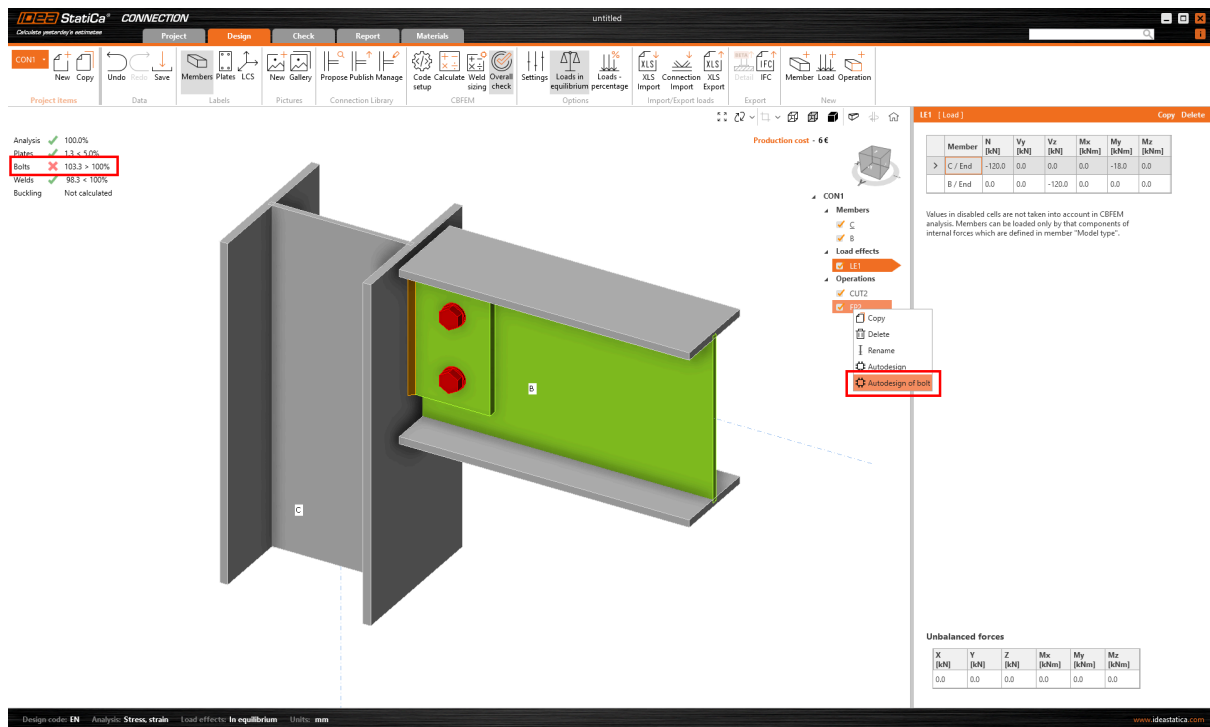
The new design is created based on the selected settings.
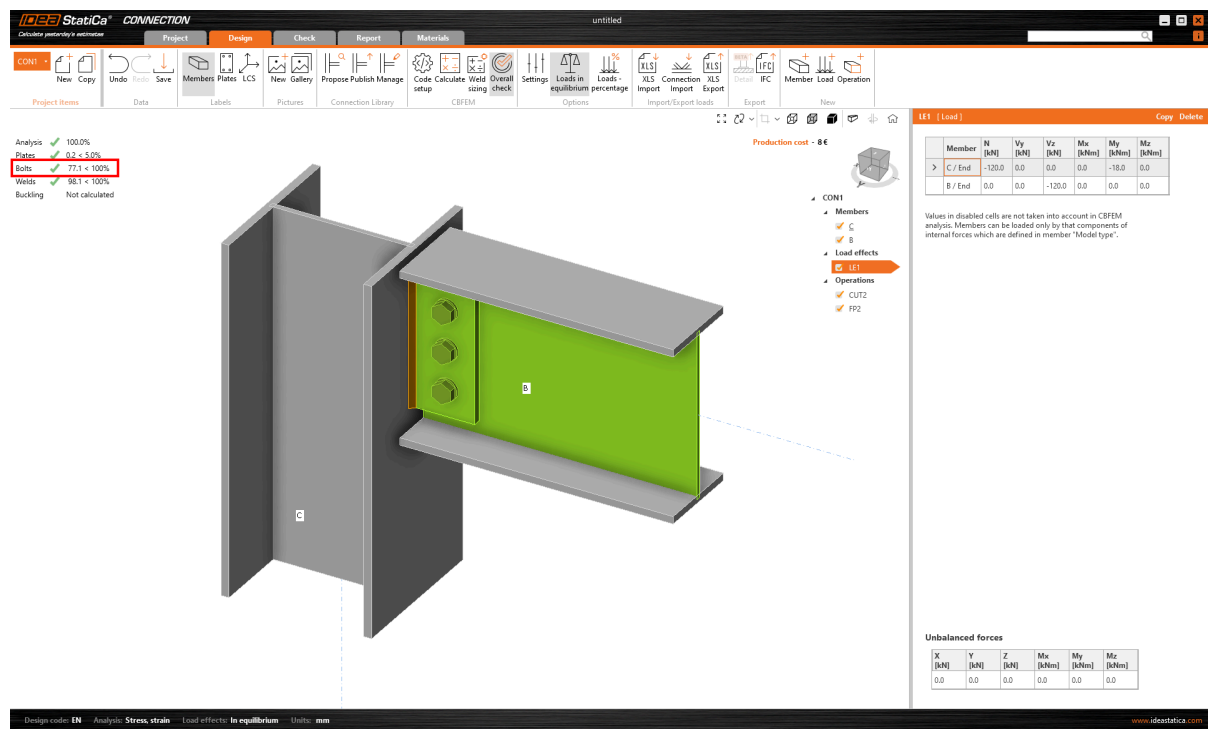
What will happen

The function will influence the following parameters:

- Number of bolts
- The geometry of the bolt grid
- Geometry of the plate (plate of fin plate or splice) according to the new bolt grid
- The thickness of the plate (only partly) – to achieve the same bearing resistance as the connected plate (e.g., the web of the beam)

While keeping the following **unchanged**:

- Diameter and grade of bolts
- The material grade of the plates

Settings

The process is influenced by the Settings. The following values will govern the autodesign process:

- The algorithm checks the **Geometry requirements acc. to the Settings:** End and edge distance, spacing, and Wall distance and finds the solution to comply with these requirements.

- The algorithm checks the **Geometry requirements according to the Preferences:** New entity roundings and finds the solution to comply with these requirements.

- The algorithm checks the **Target utilization** with the real acting loads acc. to the value set in the settings and finds the solution to comply with this requirement. The checked value is Utilization in shear (**Ut$_s$**).

Basic assumptions

- The autodesign algorithm takes into consideration **only forces and moments in the plane of the plate** (N, Vz, My - generating only shear forces for the bolts) from the acting load effects in the model. Any tension forces occurring in the bolts (generated by other loads or prying) are not taken into consideration in the autodesign process.

- If the tension utilization is high, autodesign could give inprecise results, and the user may receive a warning message. The autodesign is provided, but the results can be significantly influenced by tensile force in the bolts.

⚠️ **FP1 - Prerequisites for correct automatic design are not met.**
Significant tensile forces in a shear-type connection are not taken into account.

- Bolts in the operations have to be set with the **bearing/shear interaction** shear force type. Autodesign is not performed if bolts are set as friction and the user receives a warning message.

⚠️ **FP1 - Automatic design is not possible.**
Missing bolts with bearing transfer of shear force.

- The forces in bolts are not calculated correctly when the model is overloaded, and the analysis is finished with a **huge plastic deformation**. Therefore, the autodesign is working with incorrect forces in the bolts and can not give any correct result.

- The feature is available for the **common cross-section** types **only**, not all cross-sections are supported.

- If the selected cross-section is not suitable for placing the bolt grid according to geometrical requirements, you may also get a warning. For example, in cases with narrow flanges, it is impossible to comply with requirements for edge distances, distances from the wall, etc.

⚠️ **FP1 - Automatic design is not possible.**
Change settings for autodesign (spacing, distances, rounding).

# Concrete Design

3D Detail – reinforced concrete footings (BETA)

Anchoring in reinforced concrete blocks presents a challenge that is almost impossible to evaluate accurately using traditional formula methods. This necessity has led to the development of an innovative approach to the design and verification of these structural elements.

IDEA StatiCa introduces Detail (**3D Beta**) for solving 3D tasks such as anchoring in concrete blocks. The solution lets you perform the design without oversimplification and obtain checks based on the Ultimate Limit State (ULS). The method implemented in IDEA Statica Detail (**3D Beta**) is based on the already proven Compatible Stress Field Method, which was adjusted to be able to solve 3D problems (tri-axial stress). Also, the basic element is modified to the third dimension. The solid block is the basic element representing concrete and can be deformed in all three directions. To make it possible to create a complex model for footings, we have implemented entities such as **surface support**, **base plate**, **anchors,** and **more**. All these entities are described, as well as their functionality, in the following article.

The shape of the solid block

Concrete block can be modeled thanks to the member type called "**Solid block**", whose geometry can be defined in two ways:

- **Rectangular shape** – This option is suitable for defining the footing with a rectangular base and constant height. Defining offsets for all sides of the block also allows sloped footing to be modeled.
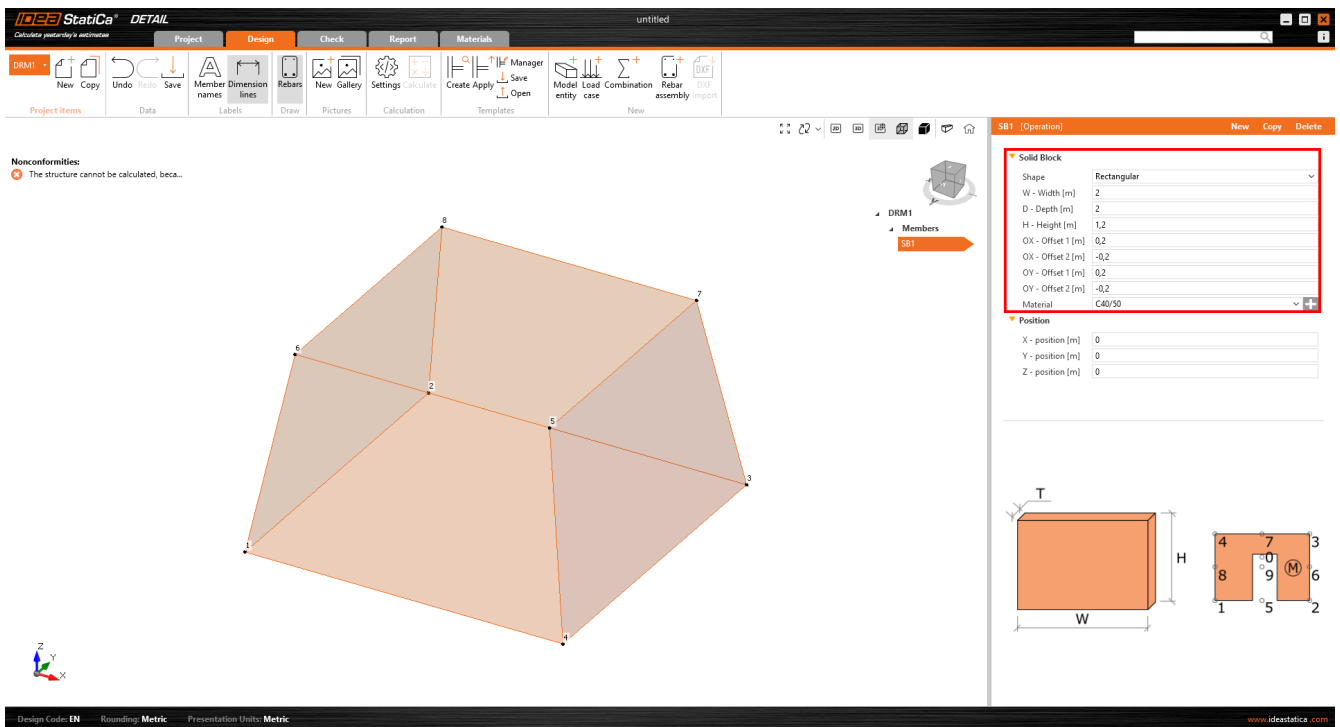
- **Polygonal base with a defined height** – The second option for defining a footing shape is the definition of the base shape as a general polygon. However, it is not possible to create sloped footing as described in the previous option.
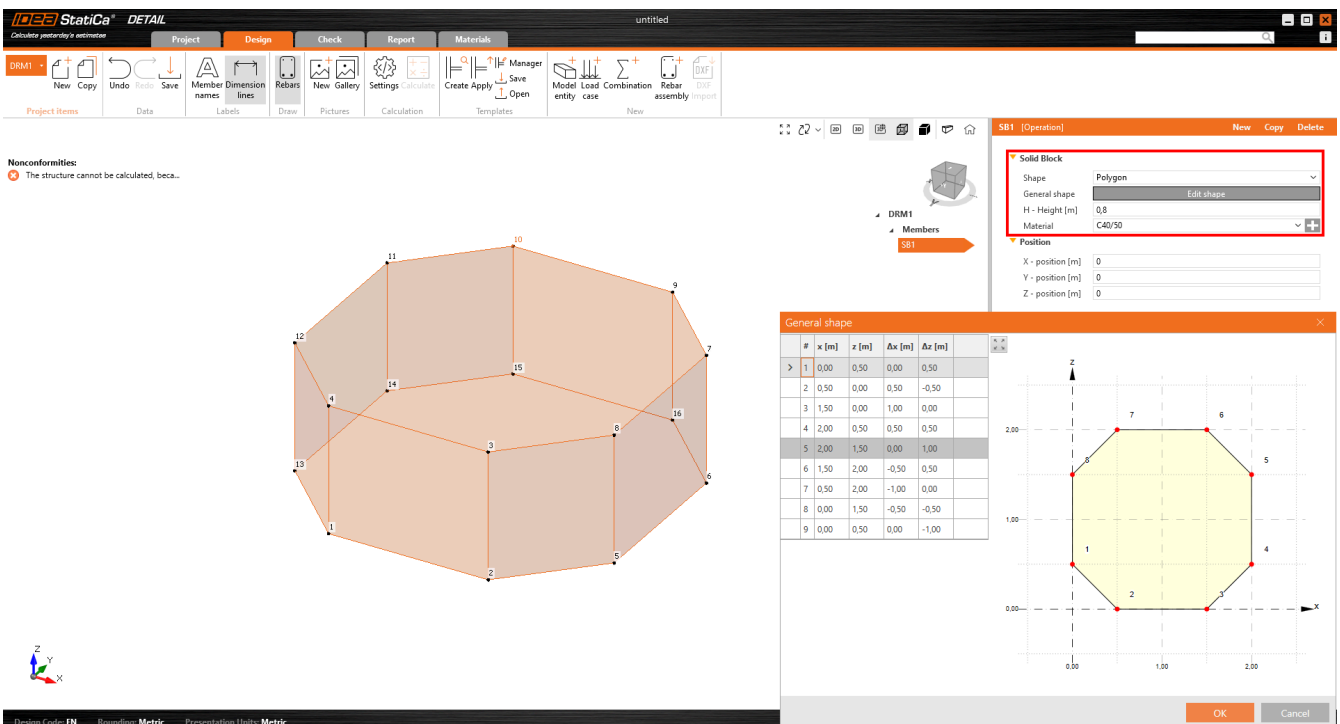
Surface supports

Surface support is the only possible support type that can be defined for the solid model. There are two options for the shape definition – it can be defined either for the **whole surface** or as a general area whose outline is determined **by polyline**.
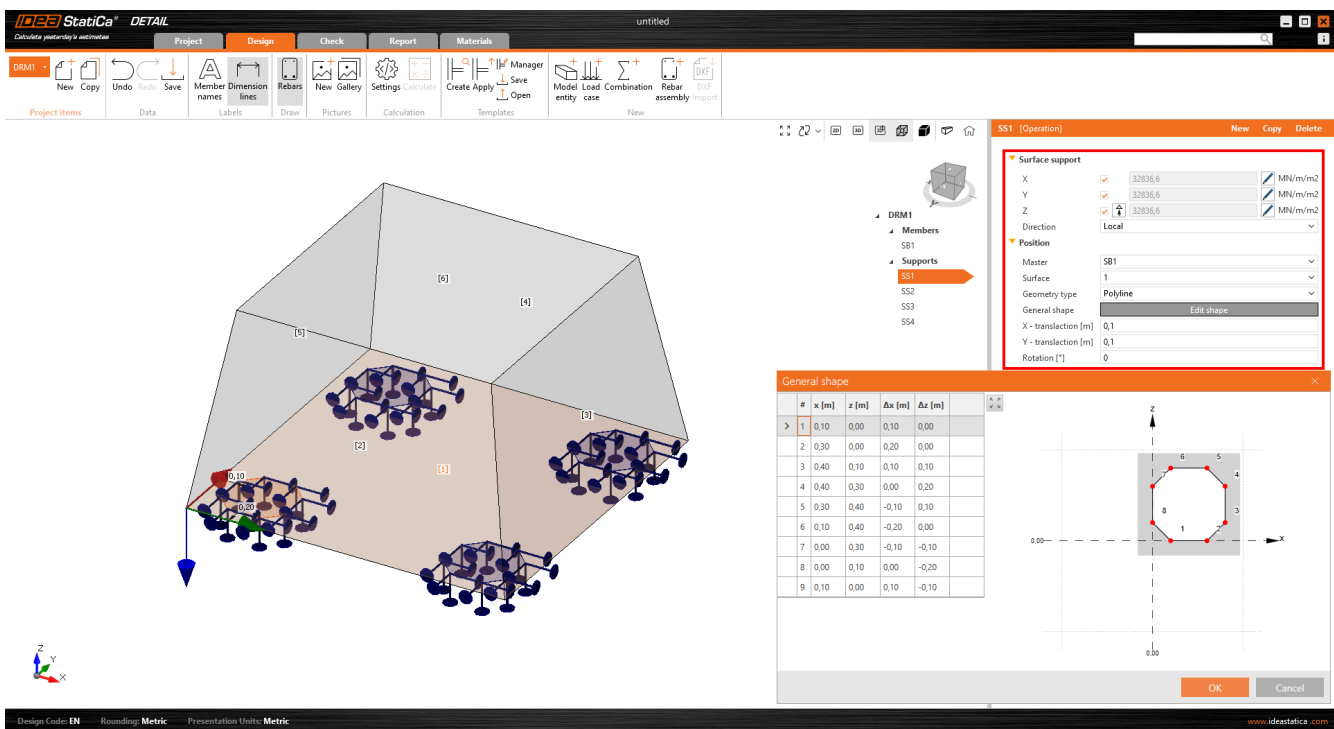
The advantage of surface support is the possibility of **defining stiffness and setting non-linear behavior** that simulates contact with the soil. Supports defined by general shape (polyline) can be used to simulate piles.

Load transfer devices

Especially for the anchoring, two entities to transfer loads were developed:

- **Base Plate** – The shape of the base plate can be defined as a rectangle or a general polyline. The base plate can be modeled with the part of the column (option "Load transfer – Column") or without the column (option "Load transfer – Forces"). When option "Load transfer – Forces" is selected, the defined load is applied directly to the base plate. When the steel column is defined (by its cross-section and length), the load is applied to the top of the column, and its stiffness helps distribute the stress correctly to the base plate.
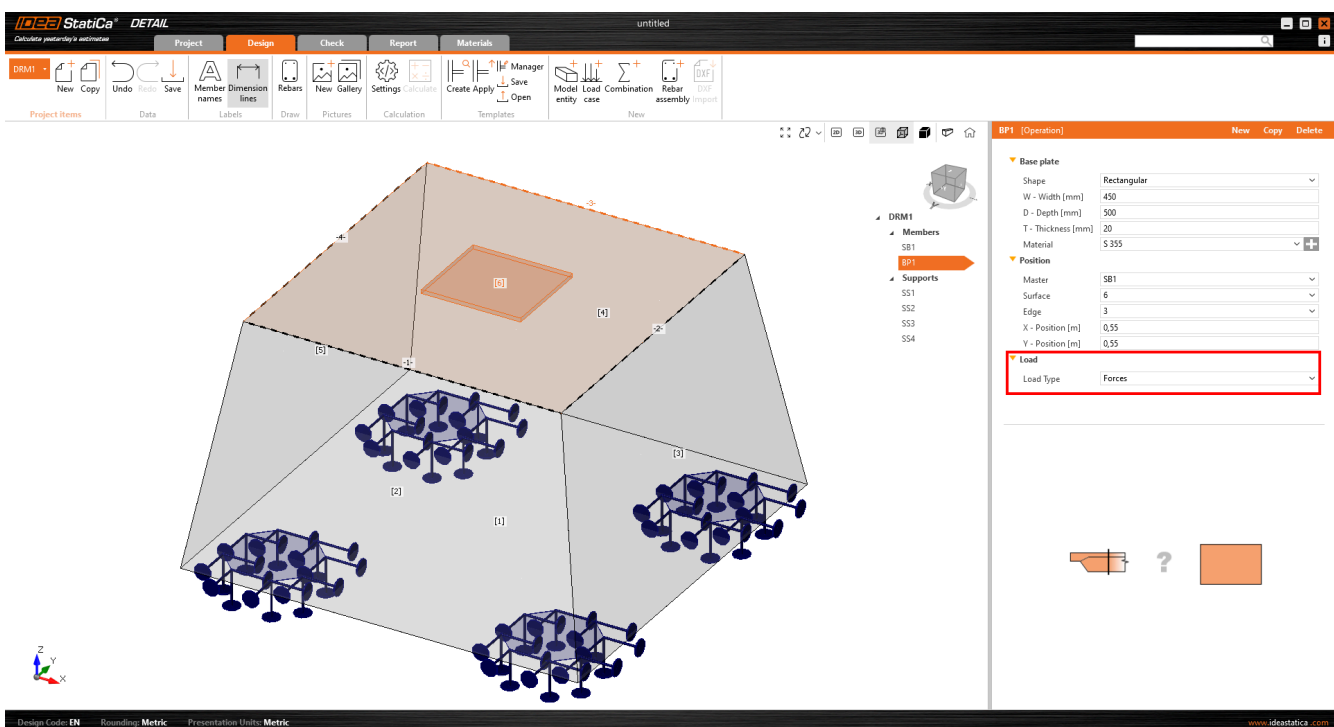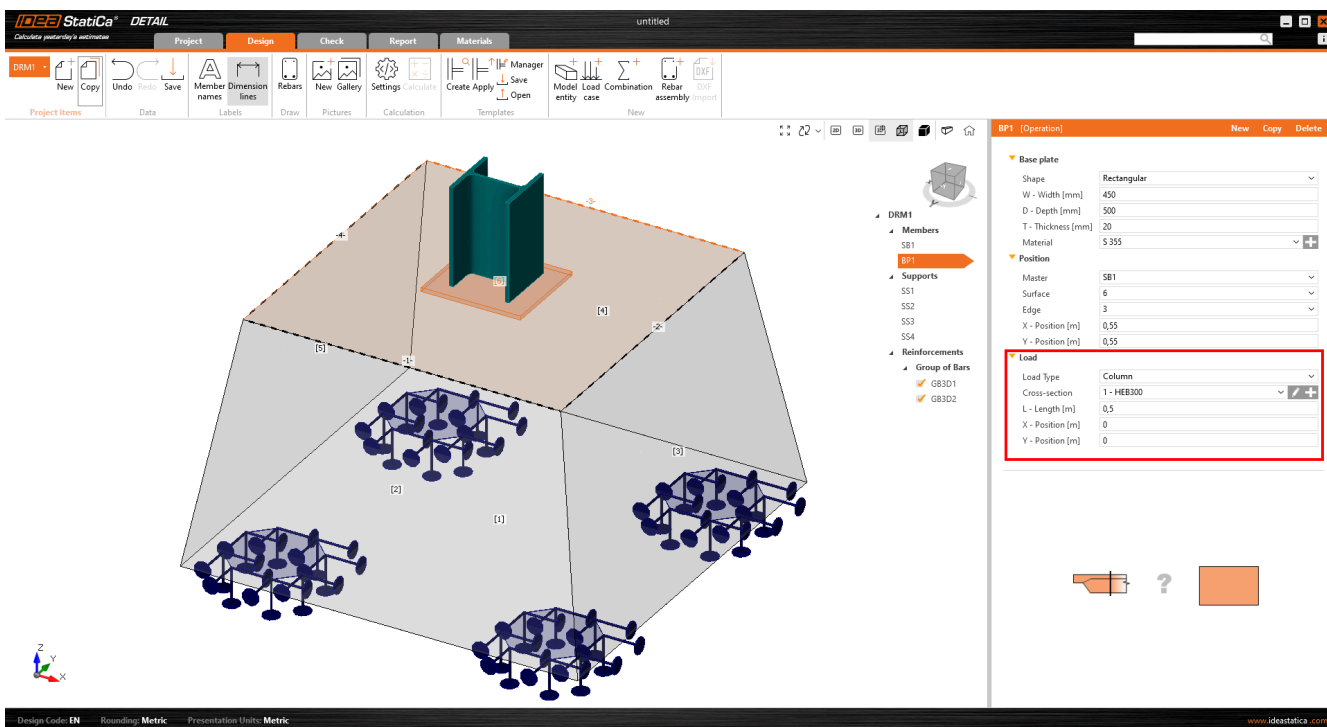
- **Single Anchor** – A single anchor is a 1D element allowing the transfer of axial load. The anchor can be automatically interconnected with the base plate or it can be loaded directly by the point load*.
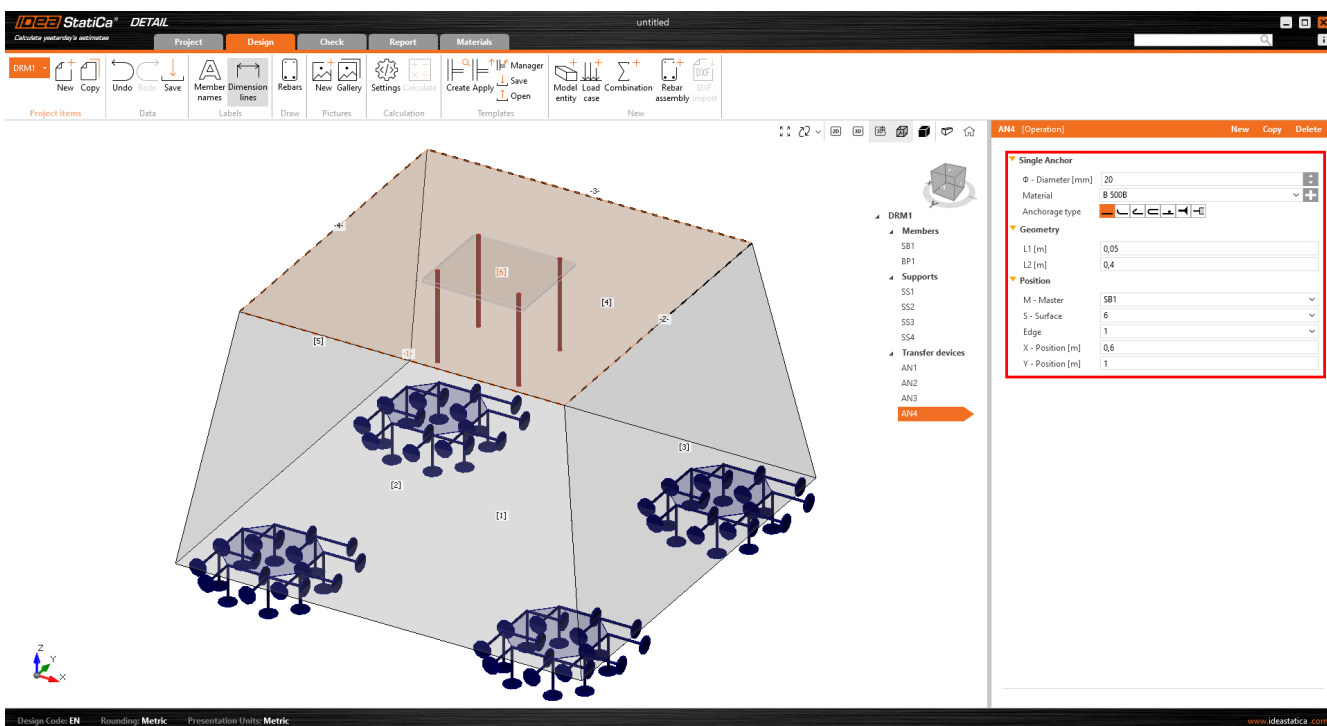
There are several limitations for anchors in BETA.

The anchor transmits only compression and tension. Shear, if imported from Connection, is transferred by friction between the concrete block and the base plate. Only axial force can be applied to the anchor.

There are two different types of anchors available in the Detail app:

- Cast-in place – Reinforcement
- Adhesive anchor

For Adhesive anchor, there is another parameter coming into the analysis – Bond strength. This value is set by default to 2 MPa and should be adjusted by the user according to the datasheet info from the anchor manufacturer.

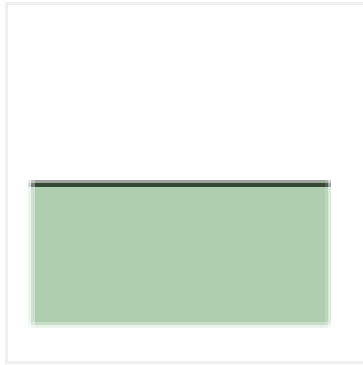# Loads

Adding a **group of forces** and **surface load** is essential for ease of use. In total, we support three load types:

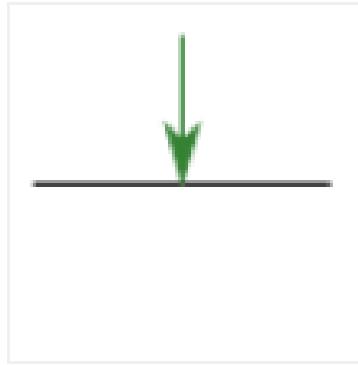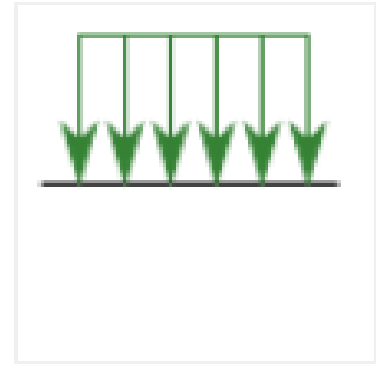- **Surface load** – Surface load can be applied to any concrete surface. It can be applied to the whole surface or just on its part over a certain area, which is determined by its outline (polyline).
- **Point load**
- **Group of forces** – A group of forces is an entity that defines several point loads acting as a group. It can be applied on the base plate or the concrete surface. This load type can be used to simulate forces in welds transferring stresses from the column to the base plate.
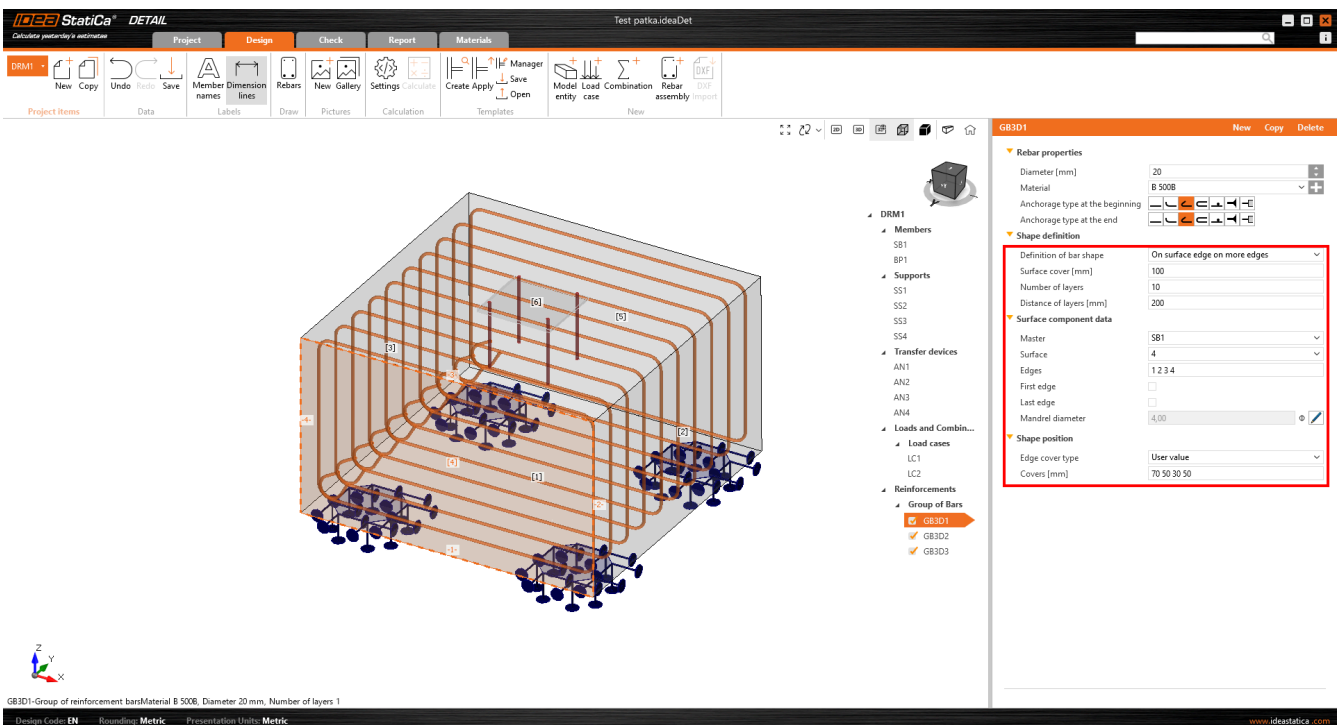

Reinforcement

A new type of reinforcement was developed, allowing the reinforcement of a footing to be sufficiently reinforced. A **Group of bars** provides several options for the definition of the rebars:

- **By two points**

41

- **On surface edge** – On surface edge – a reinforcement layer is created parallel to the selected surface. The user defines several bars in a layer and a number of layers parallel to the selected surface.
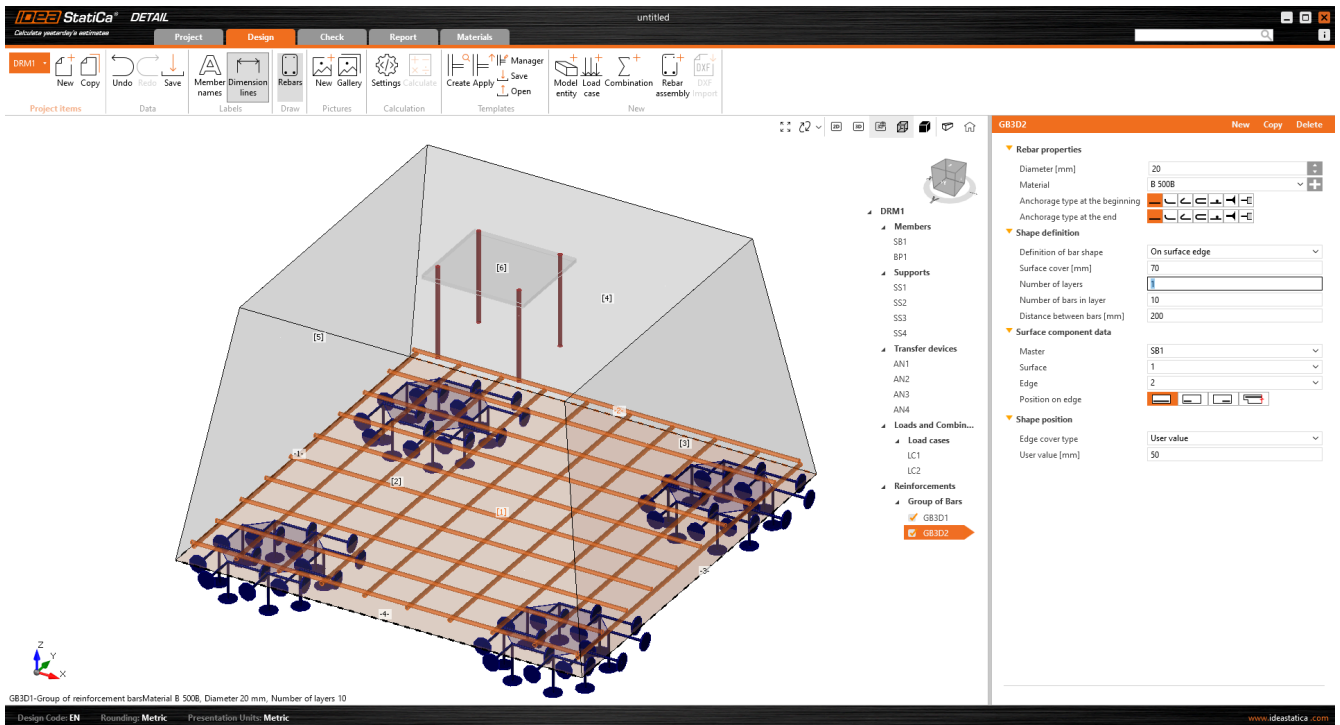

- **On surface edge or more edges** – On surface edge or more edges allow the modeling of layers of rebar with a complex shape (plane curve related to any concrete surface). The layer is, therefore, determined just by one rebar. However, several parallel layers can be defined. Concrete cover can be defined independently for each edge. This is the best option to model stirrups.


- On polyline

Results

Within the results tab, you'll discover three fundamental assessments in accordance with ULS (Ultimate Limit State).
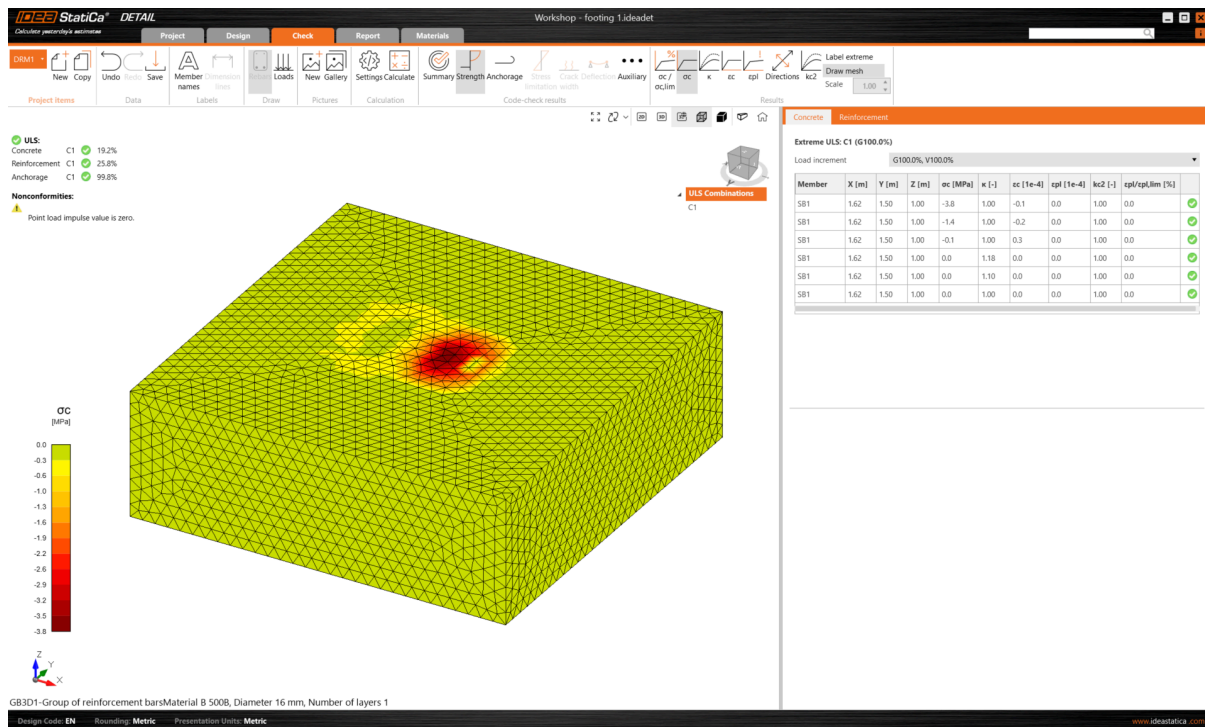
Results of stresses and strains in concrete and reinforcement are presented in the section "**Strength**". Bond stress and related values are found in the "**Anchorage**" check. A very important value of non-linear deformation is presented in the "**Auxiliary**" check.
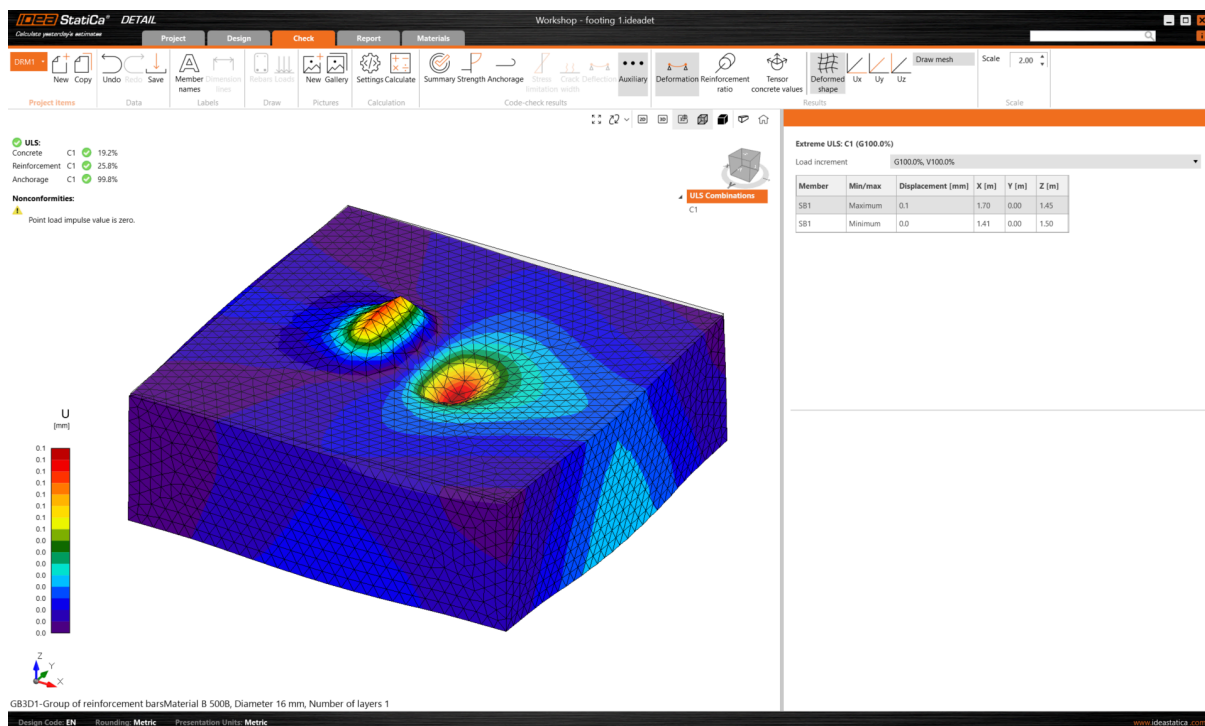
3D BETA version limitations

The goal of the released BETA version is to give our users the opportunity to get familiar with it, test the solution, and give us feedback. To clarify the expectations from the BETA version of 3D Detail, there are some limitations in the current version:

- There is **no shear force** transfer in anchors – shear is transferred only by friction between the base plate and the concrete block surface.
- Results of the analysis and code-checks are limited to **ULS** (ultimate limit state)

- The **calculation time** of a single model of a concrete block and base plate with an anchor grid may take up to 10 minutes
- **Sections** for the visualization of results will be available in BETA 2
- **Self-weight** of model elements is not taken into account
- Only **one solid** concrete **block** can be used in the model

The 3D solution for Detail is undergoing intensive development and the verification process. With version 24.0, the first BETA of 3D Detail is released, BETA 2 is expected in July 2024 with improvements in results' interpretation, reducing the current limitations' list and bringing additional features. The BETA of 3D Detail will be transferred to the ALPHA version after completing a thorough verification process and released with version 24.1.

Model description and verifications

To learn more about the 3D Detail analytical model, read the theoretical background article:

- Concrete material model for 3D Detail

Furthermore, the first verifications of the 3D Detail model and results have been published in the following articles:

- 3D Detail Verification: Simple Bend Test
- 3D Detail verification: Anchor test

Also, you can refer to the Detail theoretical background articles that are both for 2D and 3D models regarding the finite elements and ULS results interpretation:

- Finite element types
- General description of ULS results in Detail application

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

3D Detail – Wall members subjected to general load (BETA)

Out-of-plane loaded walls present a common engineering challenge, which has led to the development of an improved version of CSFM that effectively addresses such scenarios.
IDEA StatiCa Detail (**3D Beta**) is a solution that may be applied to assessing and analyzing out-of-plane stressed walls and for models consisting of different wall thicknesses. The solution represents an ideal compromise between an accurate solution using nonlinear methods but still simple modeling and calculation time.

Design

3D Detail is based on the verified and well-tested 2D Detail. It is still the same user environment, where most of the entities are kept to make working with the application as simple as possible.
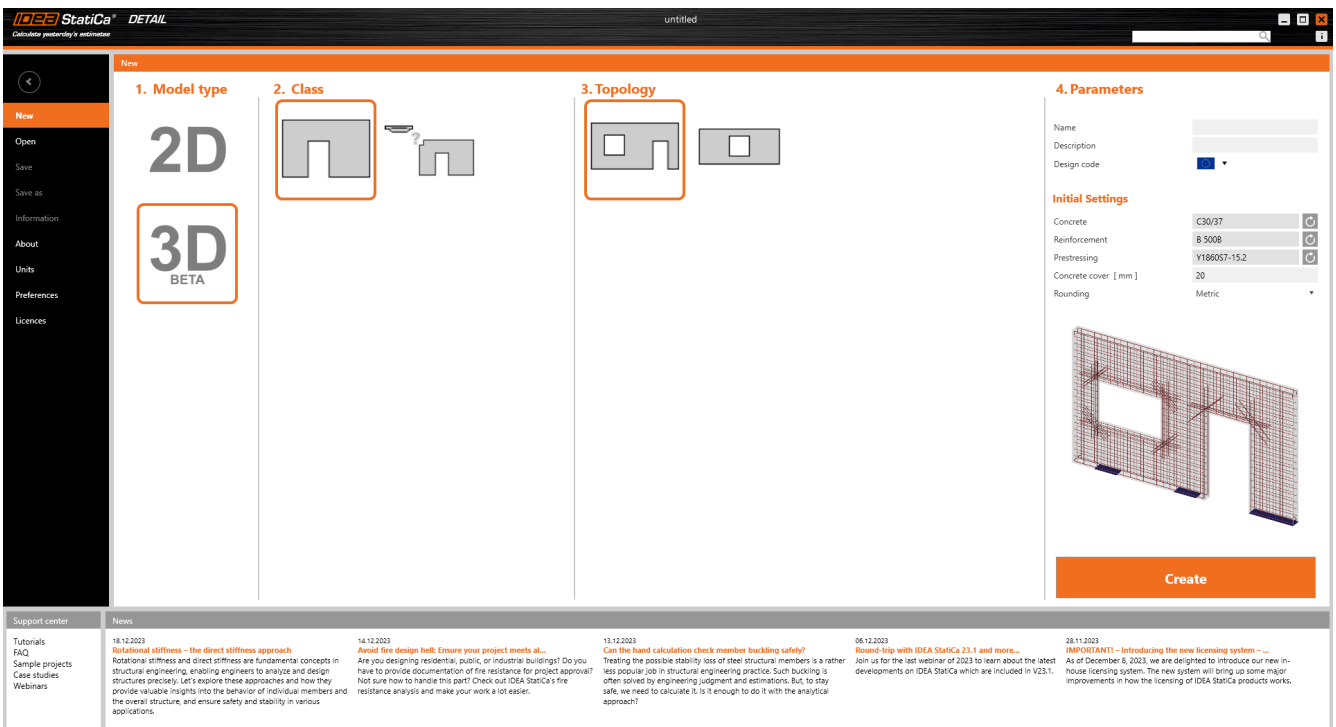
The user can still model multiple wall elements using a template or import geometry from a .dxf file. In addition, in a 3D environment, they can set the wall eccentricity "e_y" and also model walls with different thicknesses.
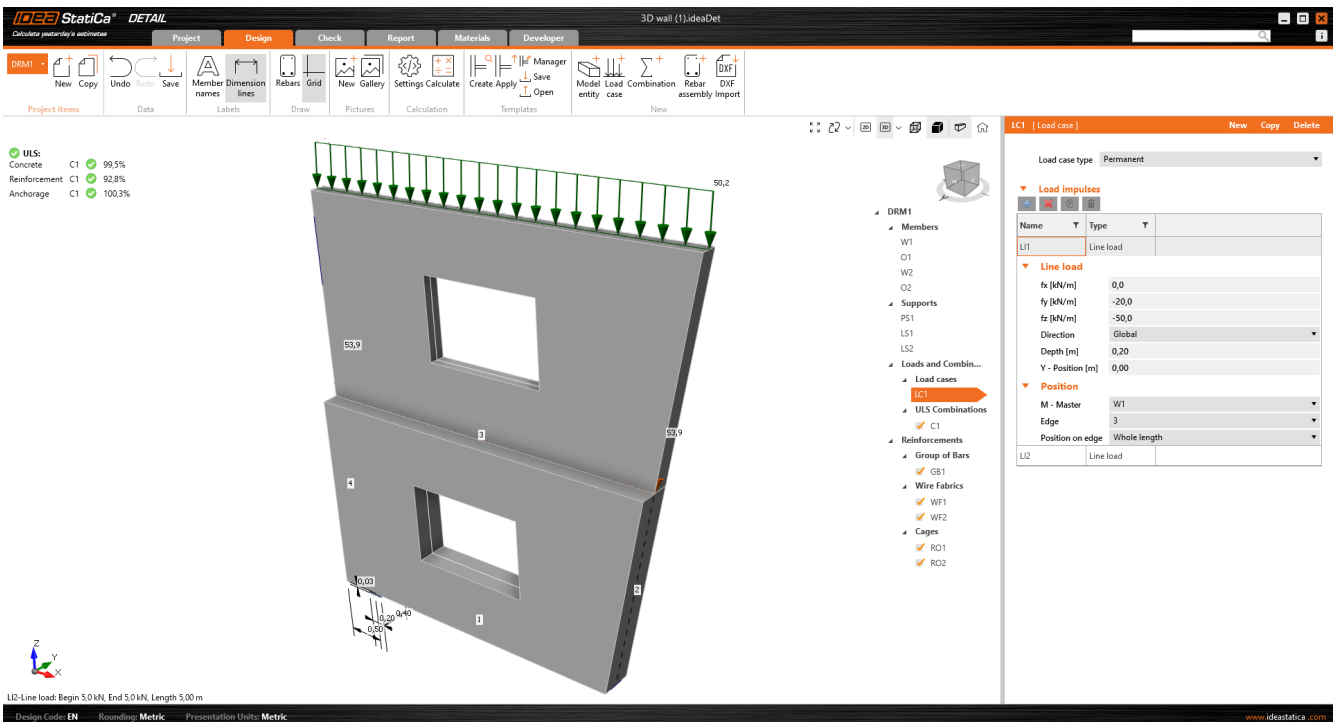

Reinforcement is defined in plane, similarly to 2D. However, there is an enhancement to define the position of the reinforcement layout over the wall thickness.

There is no possibility to define reinforcement perpendicular to the wall plane (stirrups).

Method

The approach used in IDEA Statica Detail (3D Beta) is based on the proven compatible stress field method (CSFM), which is adapted specifically for solving 3D tasks.

A special mesh formulation is applied to the wall structure to decrease the calculation time. Also, an orthotropic concrete material for walls has tensile (shear) resistance in the direction perpendicular to the wall plane. This advancement allows the calculation of the wall model without shear reinforcement defined over the thickness.
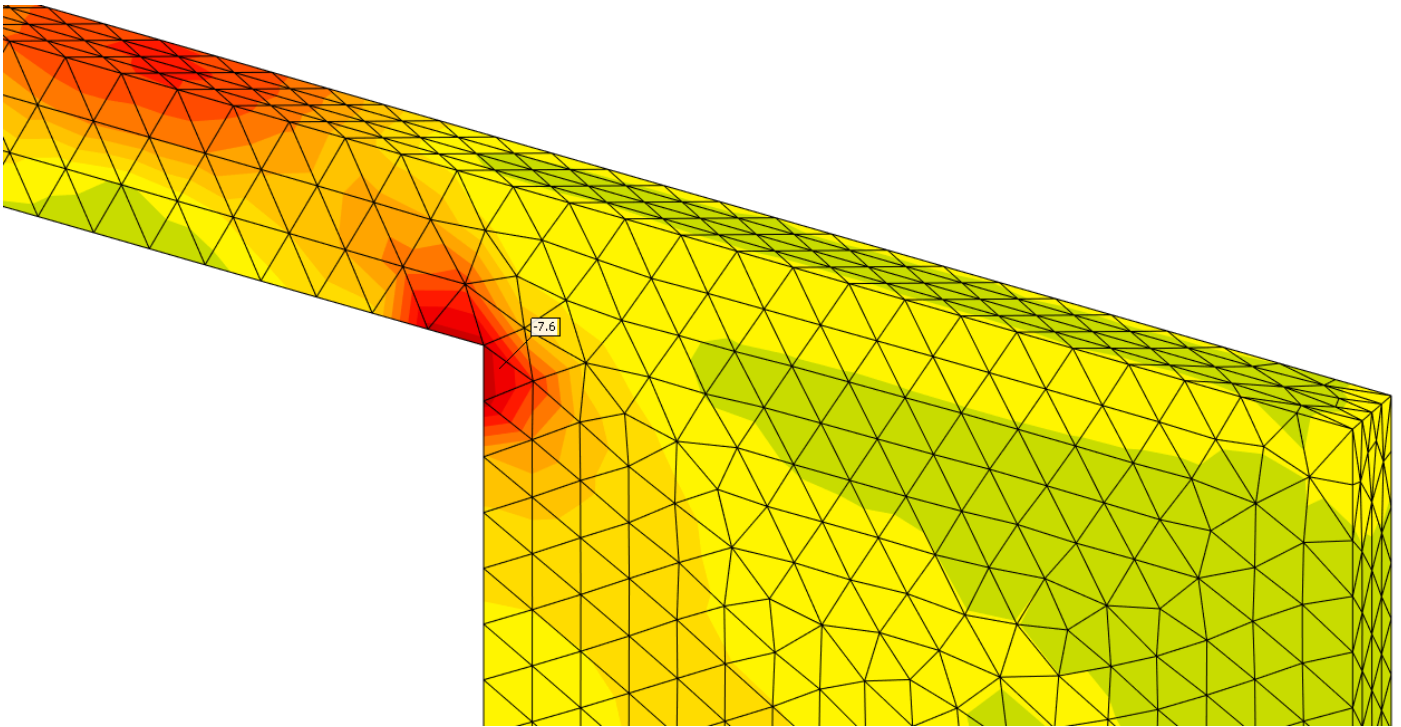

This applies specifically to the solution for walls. For the **footing solution**, it is not possible to transfer the tension in the concrete, but all the tension is captured by the reinforcement (even in the transverse direction). If you are interested in the 3D Detail for the footing solution, continue to the separate release note article).

Results

In the Results section, you will see three basic evaluations that are in line with the **ULS criteria (ultimate limit state)**.
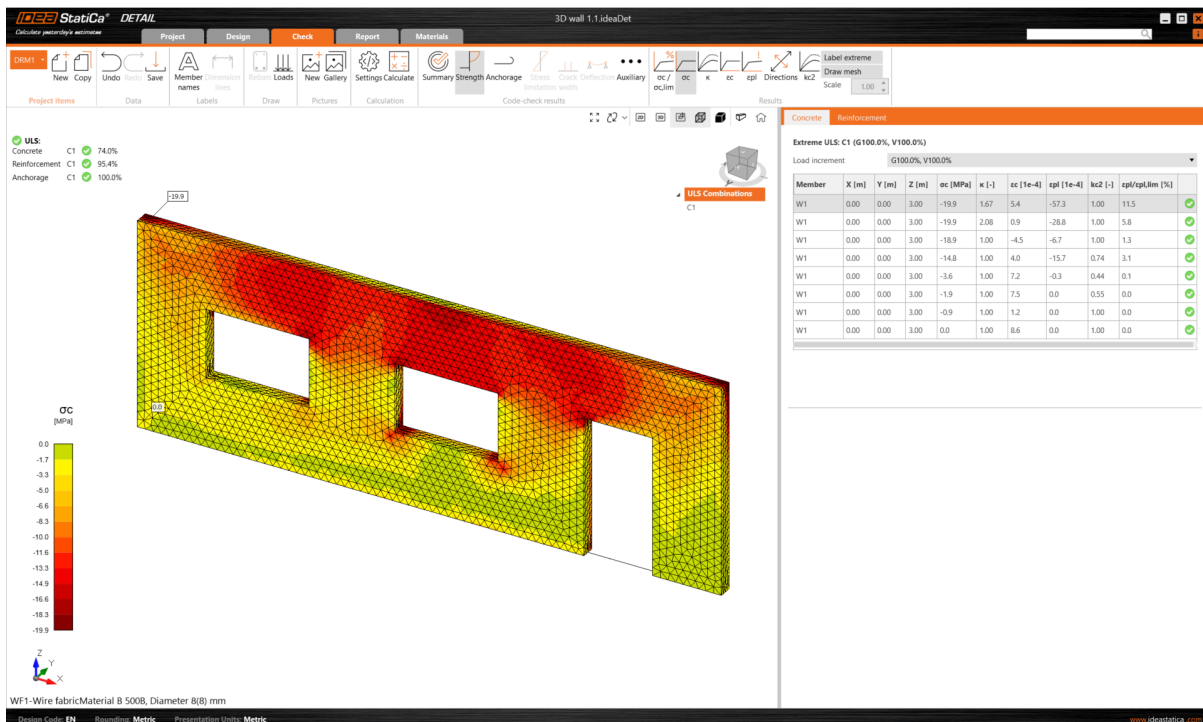
Stresses and strains related to concrete and reinforcement are defined in the section "**Strength**". The bond stresses and associated metrics are detailed in the "**Anchorage**" assessment. In addition, the non-linear deformation is provided under the "**Auxiliary**" examination.*
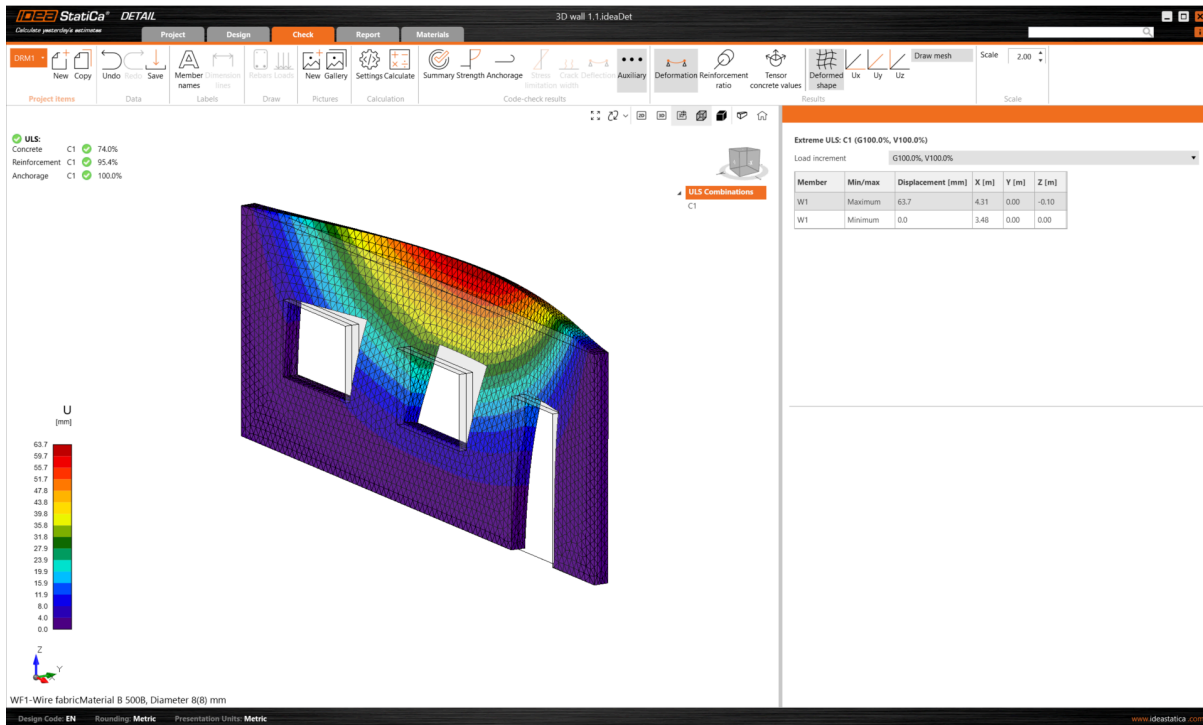
*Please be aware that the beta for the ACI code is currently unavailable.*

Released in IDEA StatiCa version 24.0.

Import of anchoring from Connection to Detail (BETA)

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

Anchoring in a plane concrete block can be modeled and code-checked in IDEA StatiCa Connection. Sometimes it might be useful or necessary to reinforce the concrete block. Unfortunately, this capability isn't available within Connection.

However, with the development of **3D Detail (BETA)**, focused on anchoring into concrete blocks and analysis of both the anchoring elements and the concrete block itself, we have a solution. Moreover, a direct link is established between our Connection and Detail applications.
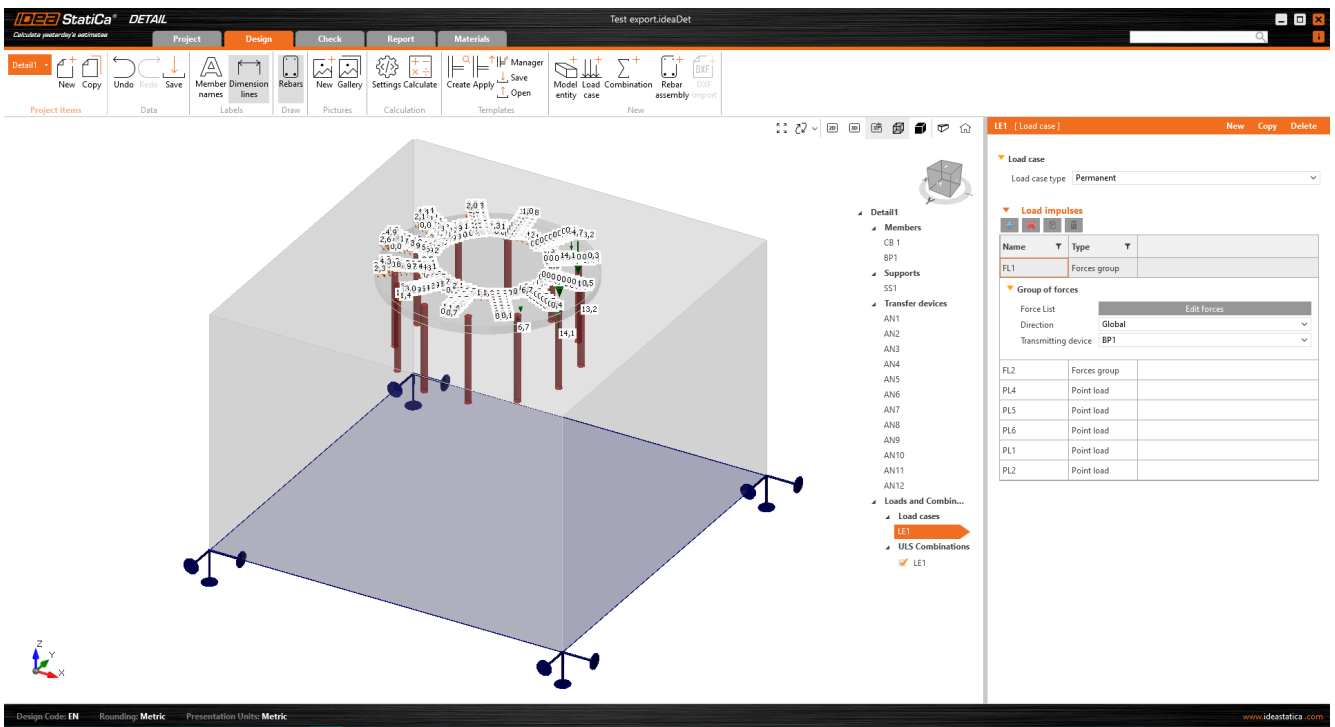
Connection users who design **anchoring according to Eurocode** can **import their model from Connection to the advanced 3D Detail (BETA) by clicking one button**.

How does it work?

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

How to export anchoring from Connection to Detail?

First, create a model of anchoring in Connection according to Eurocode and click the Calculate button.
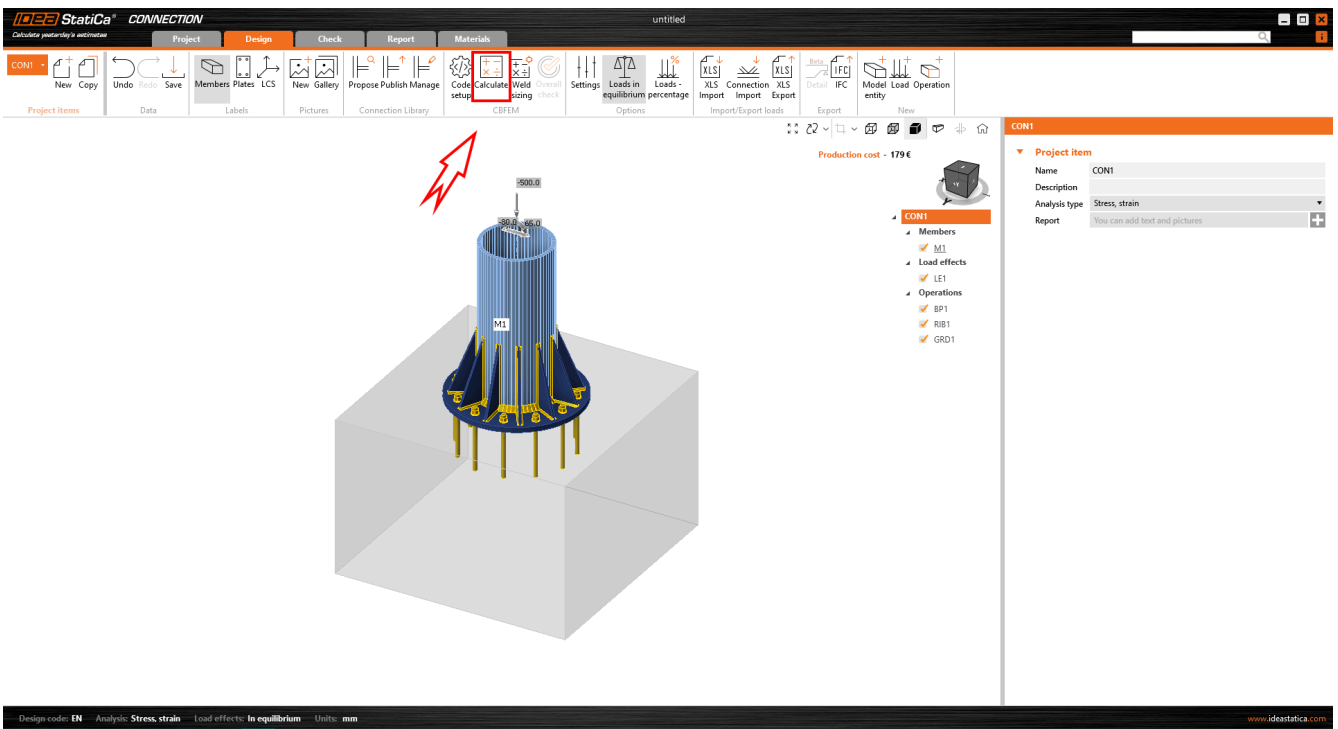
When results exist, export is enabled. By clicking the button "Detail" in the ribbon, a dialog asking for the location and the name of the newly created Detail file appears.
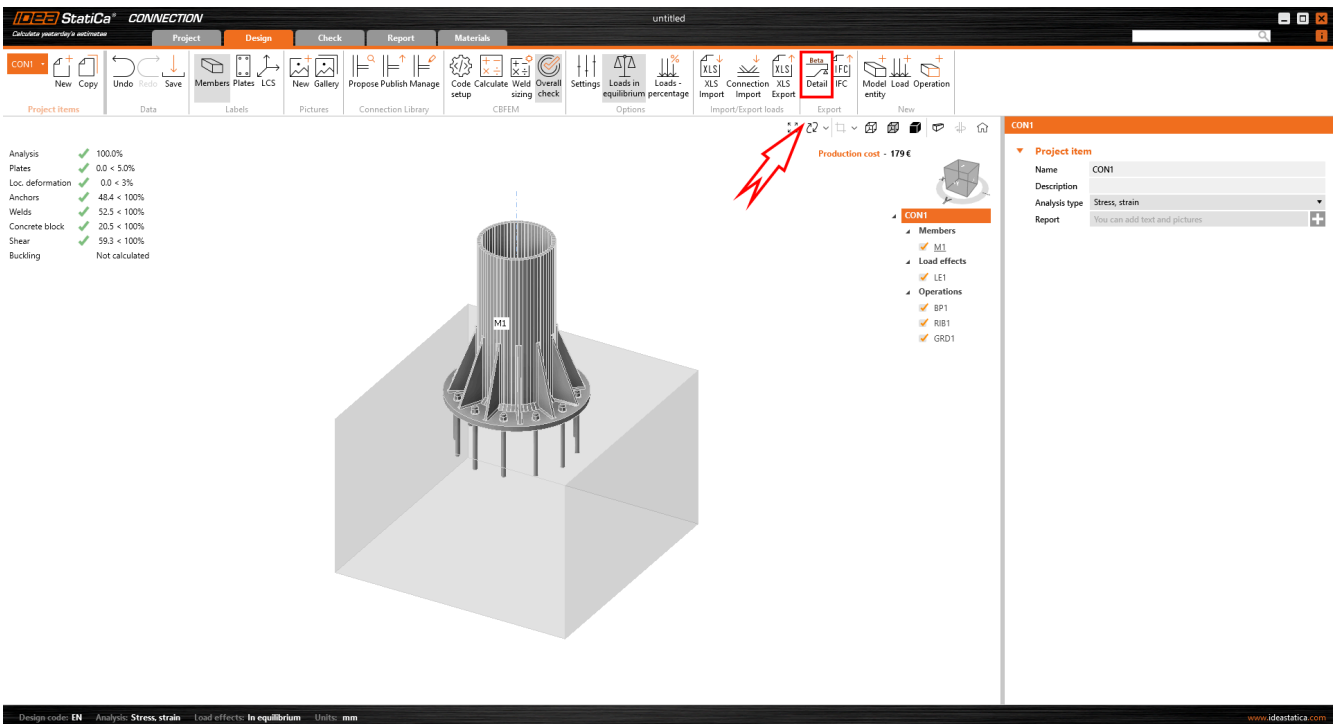
After a successful export, the project in Detail is created. **The geometry of the concrete block and the base plate, the position and properties of anchors, and the load are automatically transferred to Detail.** Surface support placed at the bottom surface of the concrete block is automatically created.
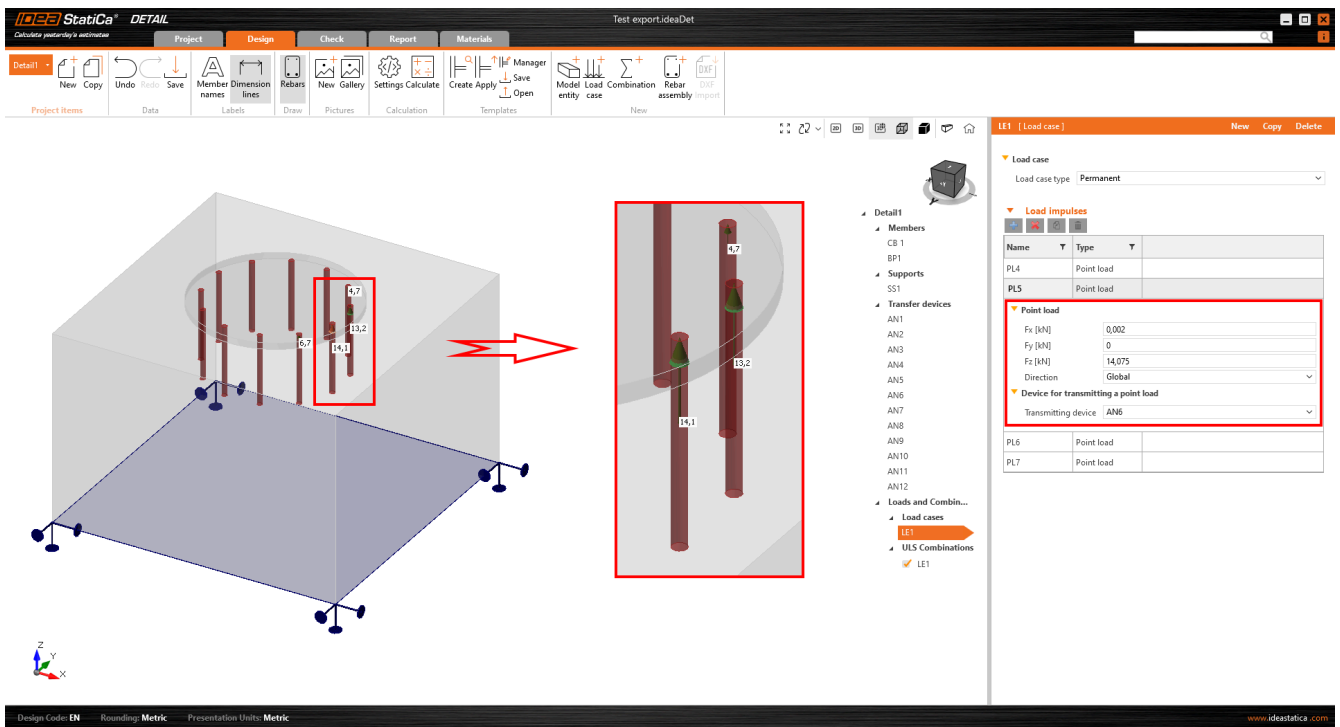
The most tricky part of this process is the import of the load. **For every calculated load effect in Connection, the corresponding load case and the ULS combination are automatically created in Detail**.
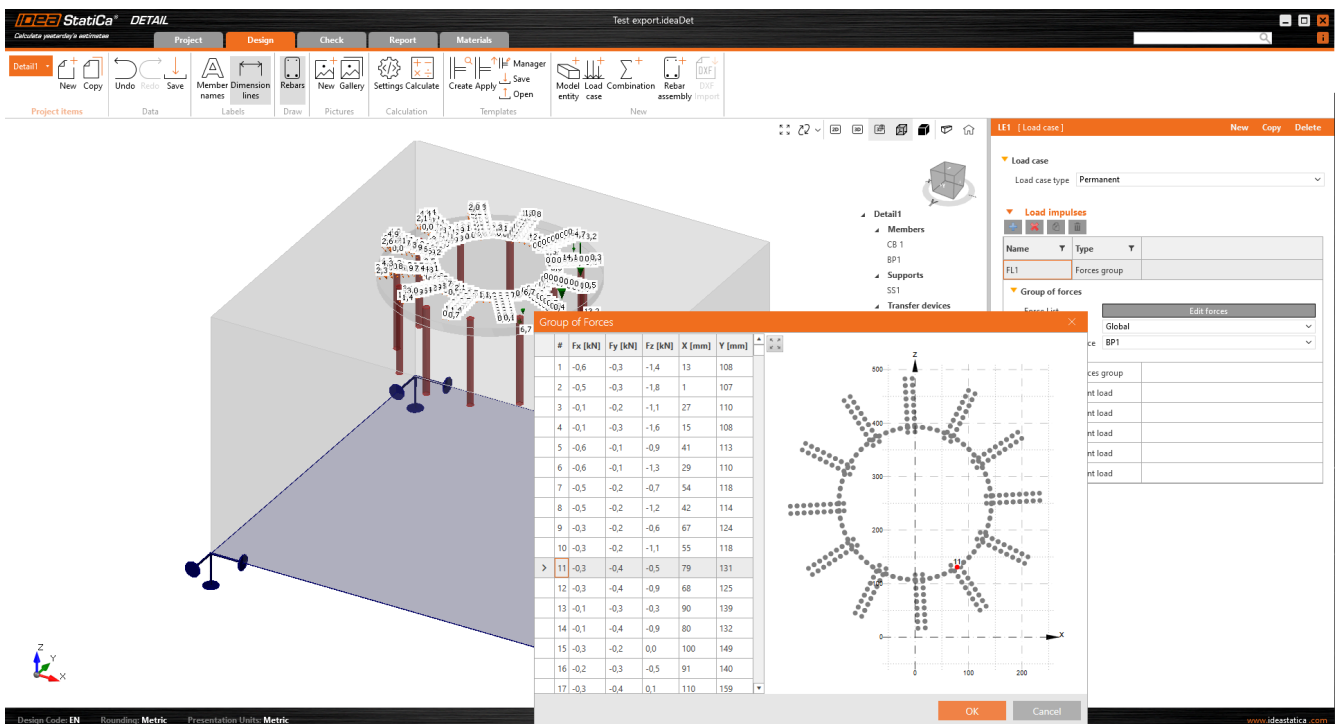
- The base plate is loaded by **forces in welds,** which are modeled as a **Group of forces.**
- Anchors are modeled and loaded independently from the base plate, and they are axially loaded by point loads. The anchor transmits only compression and tension. Shear is transferred by friction between the concrete block and the base plate.
-

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

More about 3D Detail as a solution for anchoring in concrete can be found in a separate release note.

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

Multiselect and multi-edit in Detail

Having access to and utilization of automation is the way to work faster and more efficiently. Read about how to achieve that using the multi-select and multi-edit options in IDEA StatiCa Detail.

The layout of Detail has been designed to give users easy and logical orientation within the properties. This ensures that navigating through the properties is intuitive and efficient.

When combined with the option of **bulk editing** the model entities, users can change multiple model entities simultaneously, saving time and streamlining their tasks.
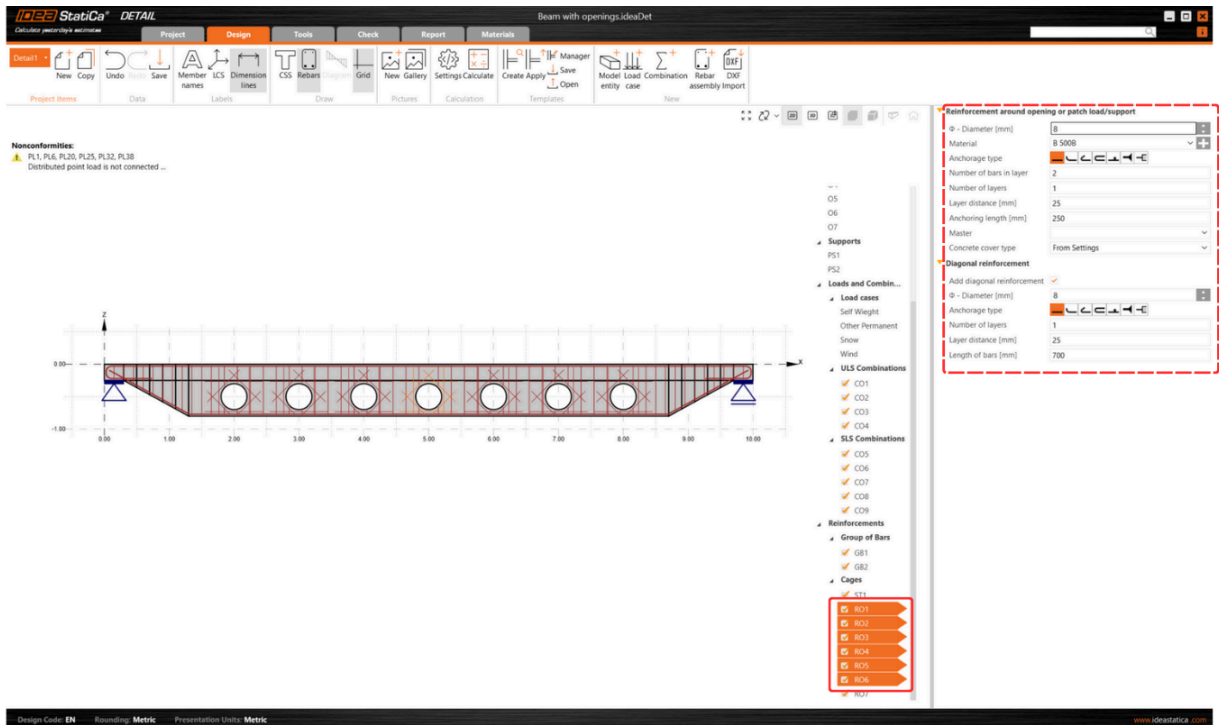
Overall, this contributes to a smooth and productive user experience.

Bulk editing of model entities

This feature is mainly for users with projects containing a high amount of model entities and reinforcement. For example, after importing reinforcements from a DXF file, **bulk editing** allows editing properties like **diameter, material, anchorage type, or layer definition** for multiple items simultaneously.
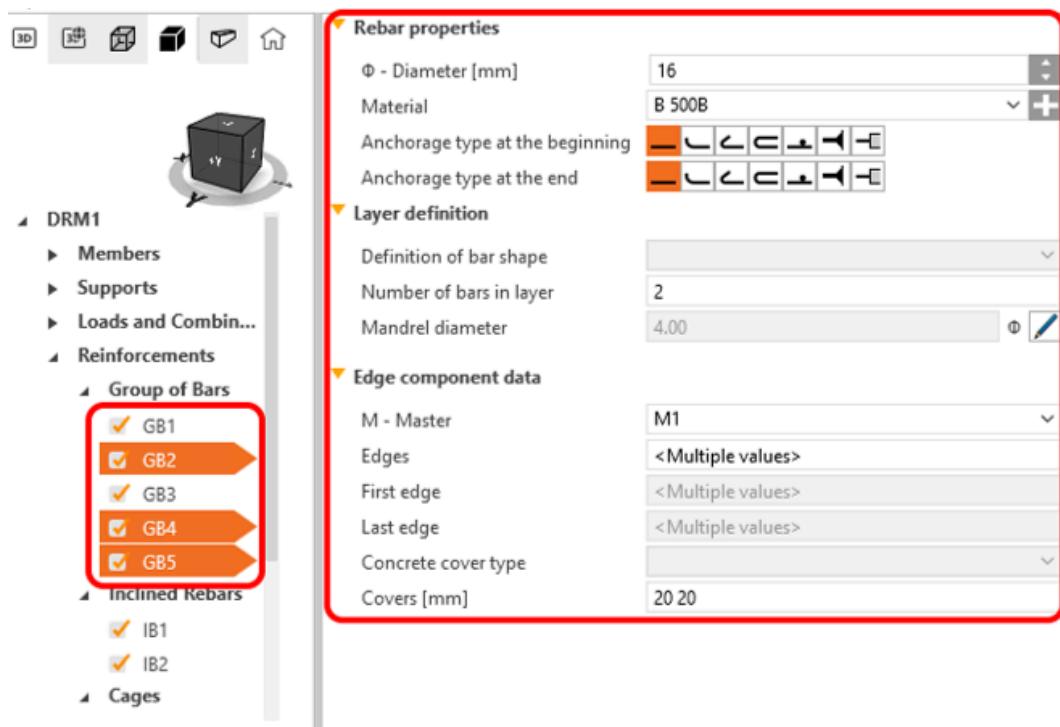
A multi-selection shortcut allows the selection of multiple entities from the tree of entities by holding Ctrl+key or a combination of the Ctrl+Shift buttons.

The multi-selection option is also available for other properties (e.g., loads, combinations, and reinforcements). This feature lets users delete or copy selected entities simultaneously, enhancing efficiency and simplifying tasks.

Released in IDEA StatiCa version 24.0.

Customizable report tab in Detail

The Report tab layout simplifies the selection of report types (Brief, Detailed, or Bill of Material) and integrates export commands directly into the ribbon, optimizing the report customization process via the tree of entities.

Enhanced interaction with entities

The **tree of entities**, a central feature of the application, streamlines report customization. Entities within the tree correspond to specific report sections, with the content being adjustable through checkbox activations or deactivations. This design supports the intuitive reordering of report sections via **drag and drop**. At the same time, the selection of an entity instantly updates the property grid and highlights the selected section at the top of the interface.
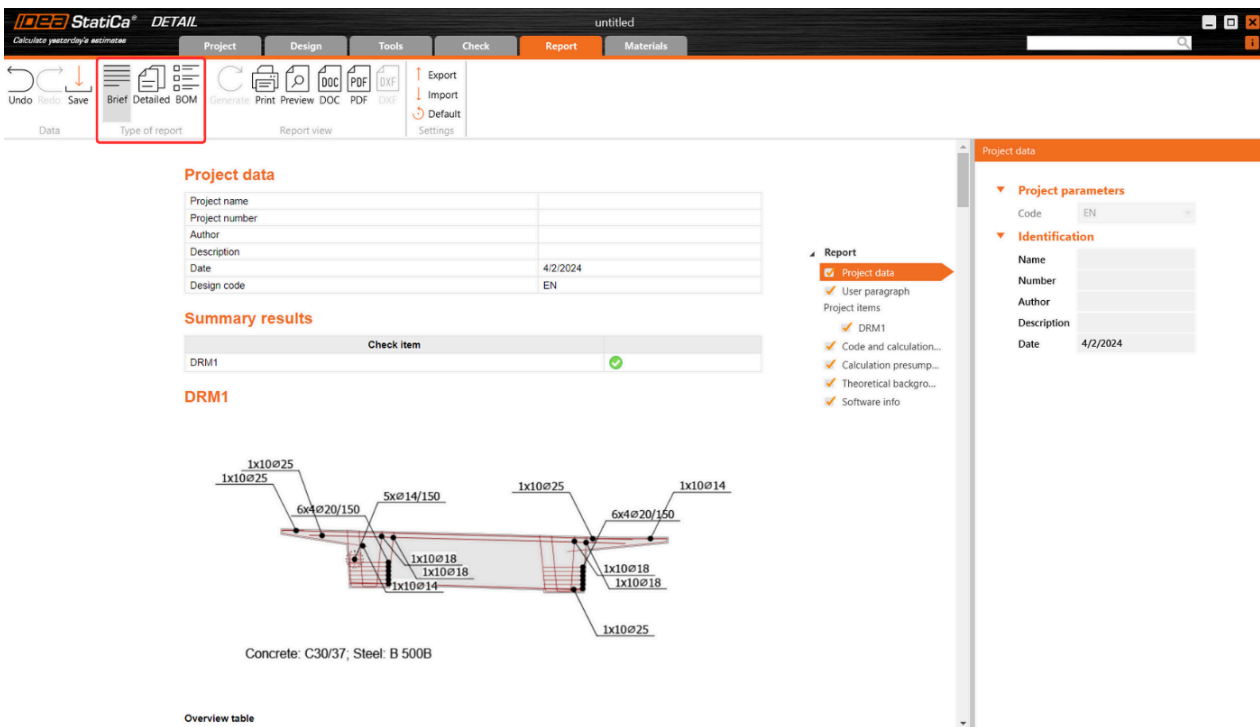
Property Grid and report content management

Selected item properties are displayed in the **Property Grid** on the right side of the application, although not all entities possess properties. This setup provides a detailed overview and control over the report's content, ensuring relevant information is included as per user preference.
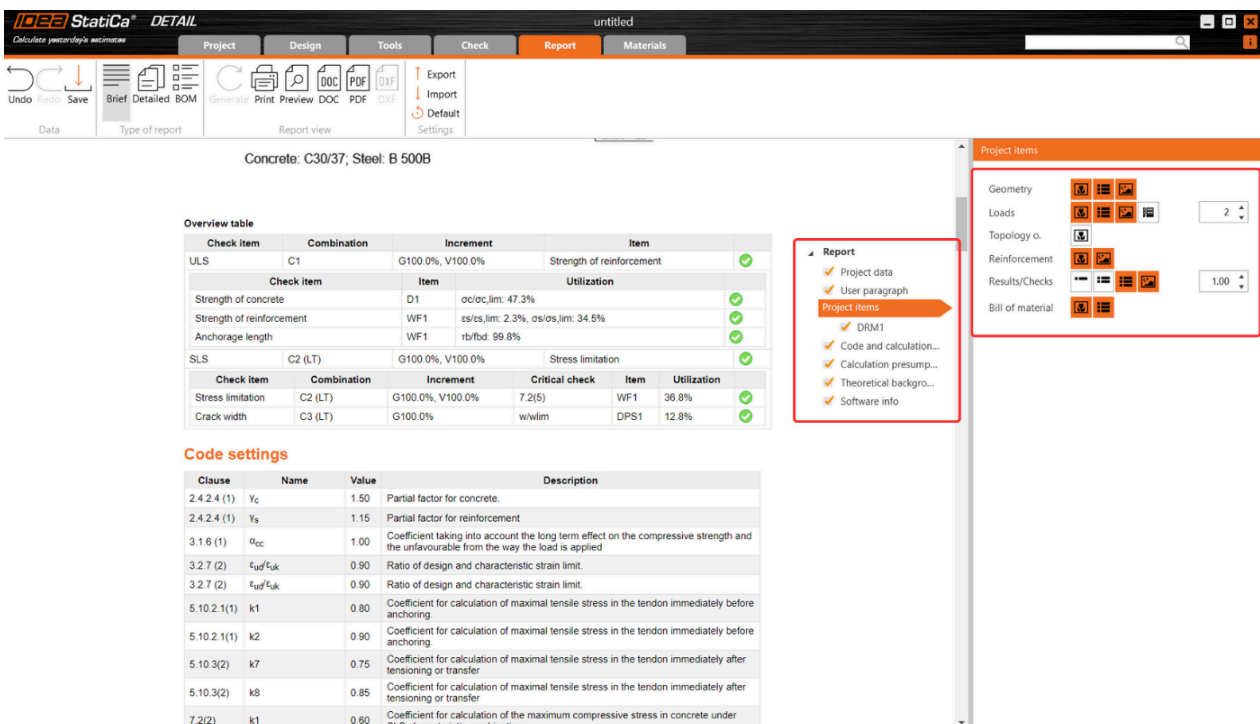

Transparent report generation

The Report tab aids in report generation, enhances transparency in calculations, and facilitates code checks. This includes:

- **User paragraph incorporation**: A "User Paragraph" can be incorporated into the report for the entire project or individual entities, utilizing an advanced editor for text and image manipulation reminiscent of MS Office tools.

Changes are saved directly within the Report tab for convenience.
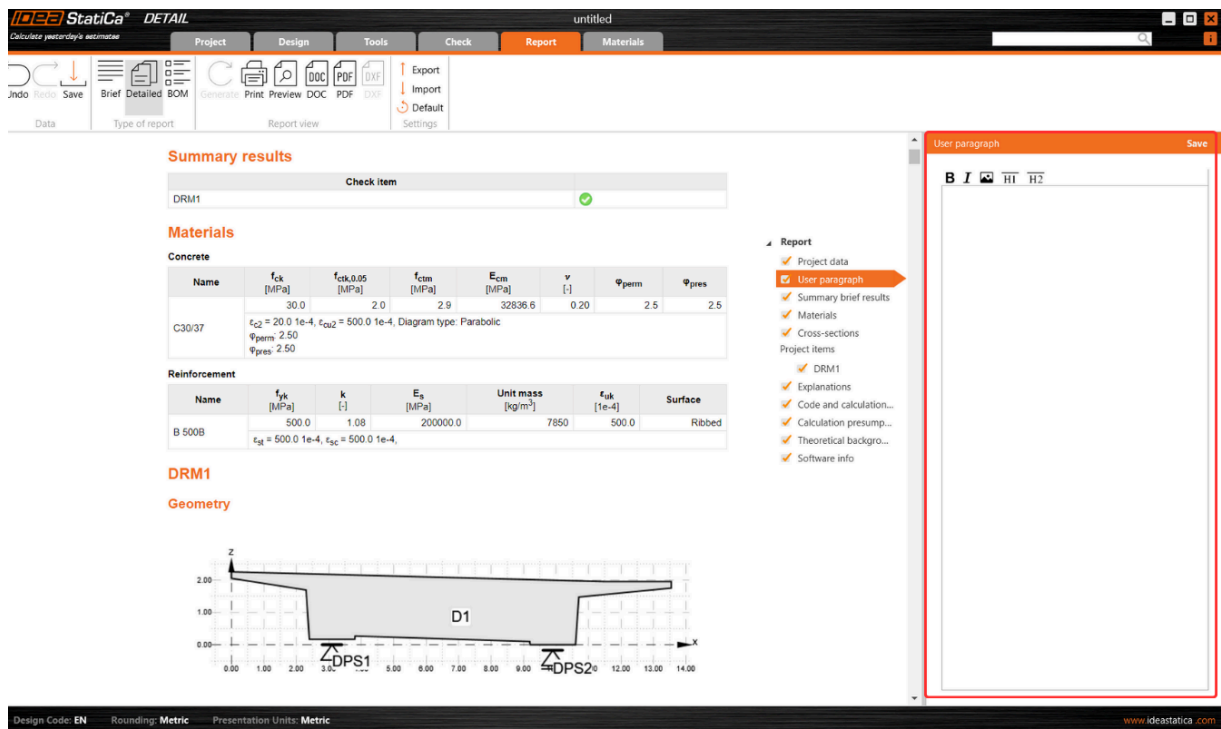
- **Inclusion of additional essential sections**: Sections like **Theoretical Background** and **Software Information** enrich the report, providing essential insights into calculation bases and software functionalities. The Theoretical Background section delves into calculation assumptions and model entity functionalities alongside code-check and design code references.
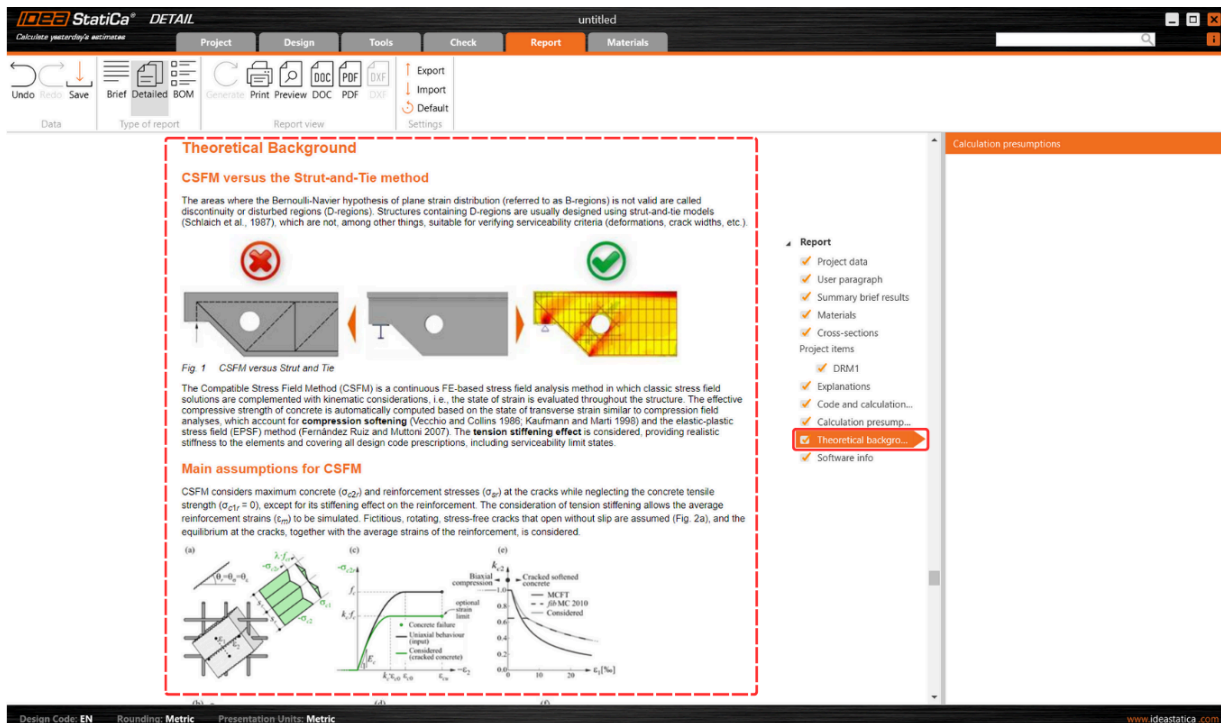

Multilingual support for theoretical background

Theoretical Background is accessible in **English, German, Dutch**, and **Czech**. For languages beyond these, an English version is provided to ensure wide accessibility.

Advanced Bill of Material features

Bill of Material (BOM) offers advanced features such as displaying rebar model entity names within the rebar table. This enhancement aids in effortlessly identifying rebars in the BOM, correlating them with their respective entities in the model. BOM also provides an option for the scheme to export reinforcement into dxf.
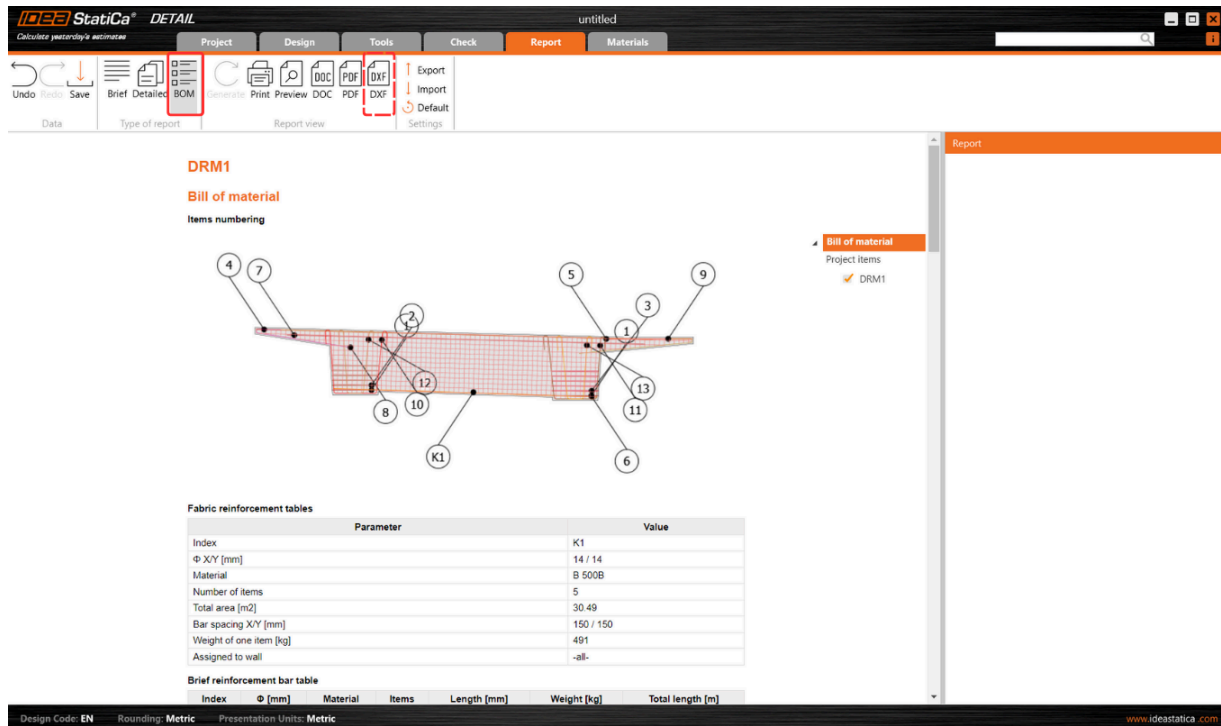
Released in IDEA StatiCa patch 23.1.5.

RCS API for streamlined, simplified, and efficient design processes

The modern IDEA StatiCa RCS API solution is comprehensive, programming-language independent, web-based, and meets the advanced needs of professionals seeking simplified and efficient design processes for reinforced concrete structures.

The RCS API facilitates integration into any project framework with minimal prerequisites. The API is based on REST technology, and it supports clients for both .Net (C#) and Python, with the flexibility to extend support to additional programming languages.

Documentation, including installation instructions and example use-cases, is readily available on the **developer website**, providing comprehensive guidance for users to get started with the RCS API quickly:

Getting Started RCS API

The RCS API allows the loading of an RCS project by:

- Import from a created IOM file, or
- Opening an existing RCS file

Once an RCS project is active, you can:

- Get information about the project, including **project data**, **available sections,** and **design members**.
- Update code settings.
- Update the Reinforced **cross-section reinforcement**, **tendons**, or the entire cross-section using a template file (.nav).
- Run the calculation for all or selected sections.
- Retrieve results for all or selected sections.

The API marks our commitment to providing a scalable, user-friendly tool for the optimization and automation of concrete structural reinforced cross-section designs. This streamlines the design workflow and opens future advancements, including the consideration of web-based calculation services and integration with platforms like VIKTOR apps or Rhino Compute.

Released in IDEA StatiCa patch 23.1.3.

Simple insertion of table inputs into IDEA StatiCa Detail

This feature allows users to efficiently fill in all tables in IDEA StatiCa Detail by copying and pasting directly from Excel or any other spreadsheet into table inputs.

Users can use this streamlined process to mitigate the risk of manual entry errors and significantly accelerate data input, improving productivity. The procedure is as follows:

1. Initiate the process by **selecting the first (top left) cell** in the table input in IDEA StatiCa Detail. **Press ESC** after the first click to prepare the cell for new input.
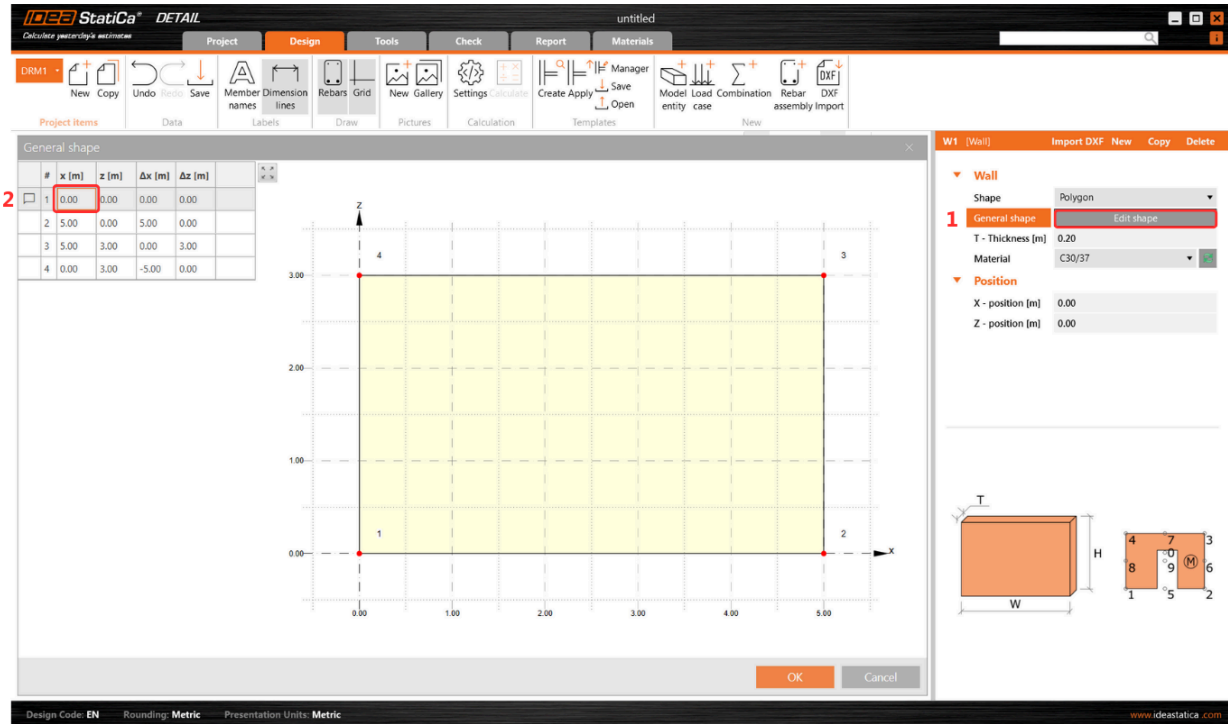
2. **Copy** the desired table from any table sheet.

2.75	2.05	2.75	2.05
5.38	9.27	2.63	7.21
4.7	9.35	-0.68	0.08
4.7	9.48	0	0.13
3.3	9.48	-1.4	0
3.3	9.35	0	-0.13
2.37	9.35	-0.93	0
1.22	6.19	-1.15	-3.16
-1.22	6.19	-2.43	0
-2.31	9.19	-1.09	3
-3.3	9.19	-0.99	0
-3.3	9.32	0	0.13
-4.7	9.32	-1.4	0
-4.7	9.19	0	-0.13
-5.28	9.19	-0.58	0
-2.75	2.05	2.53	-7.13
-2.75	0	0	-2.05
2.75	0	5.5	0
2.75	2.05	0	2.05

3. **Paste (Ctrl+V)** the copied table into the table within IDEA StatiCa Detail.
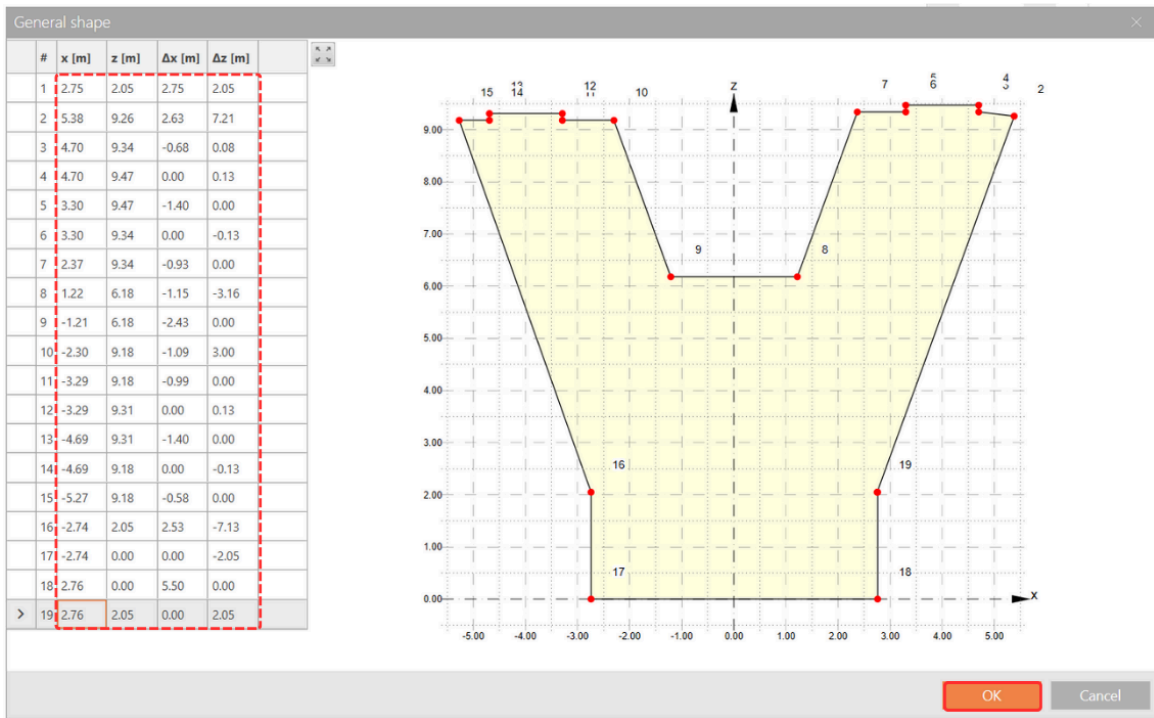
The Detail table will automatically adjust to match the dimensions of the Excel table.
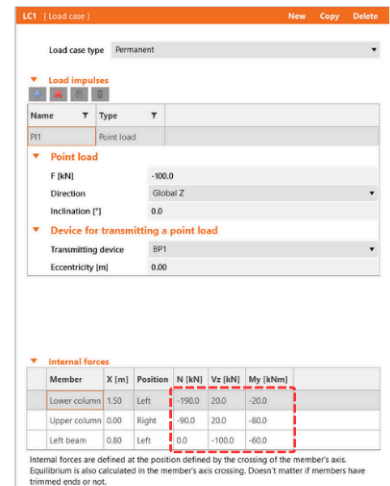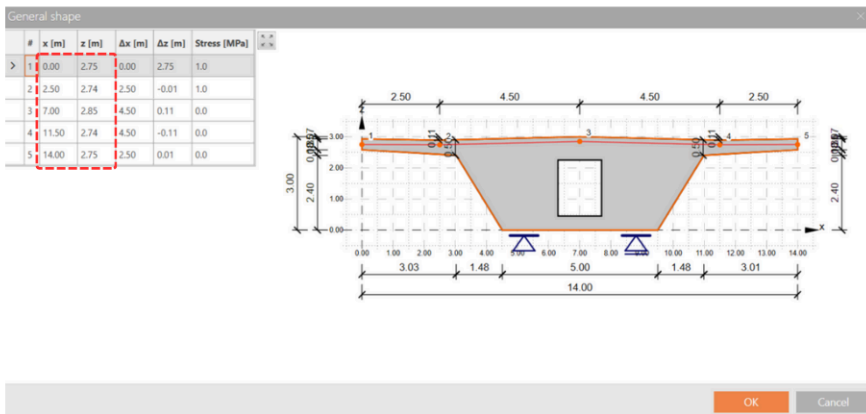
This feature applies to all tables in IDEA StatiCa Detail, including those for **Internal Forces** and **Reinforcement** or **Shape Definition**, offering a much-needed solution for efficient and error-free data entry.

Released in IDEA StatiCa patch 23.1.2.

Comprehensive user interface in IDEA StatiCa Detail

The comprehensive view control options in IDEA StatiCa Detail offer seamless manipulation of both 2D and 3D scenes.

View control options for both 2D and 3D scenes in IDEA StatiCa Detail

You can use the comprehensive control panel to seamlessly manipulate both 2D and 3D scenes within IDEA StatiCa Detail. This feature provides several display type options as well as the ability to zoom and move with the model.

The control panel offers three types of display control options:

1   Positioning tools

2   View options

3   Visual presentation
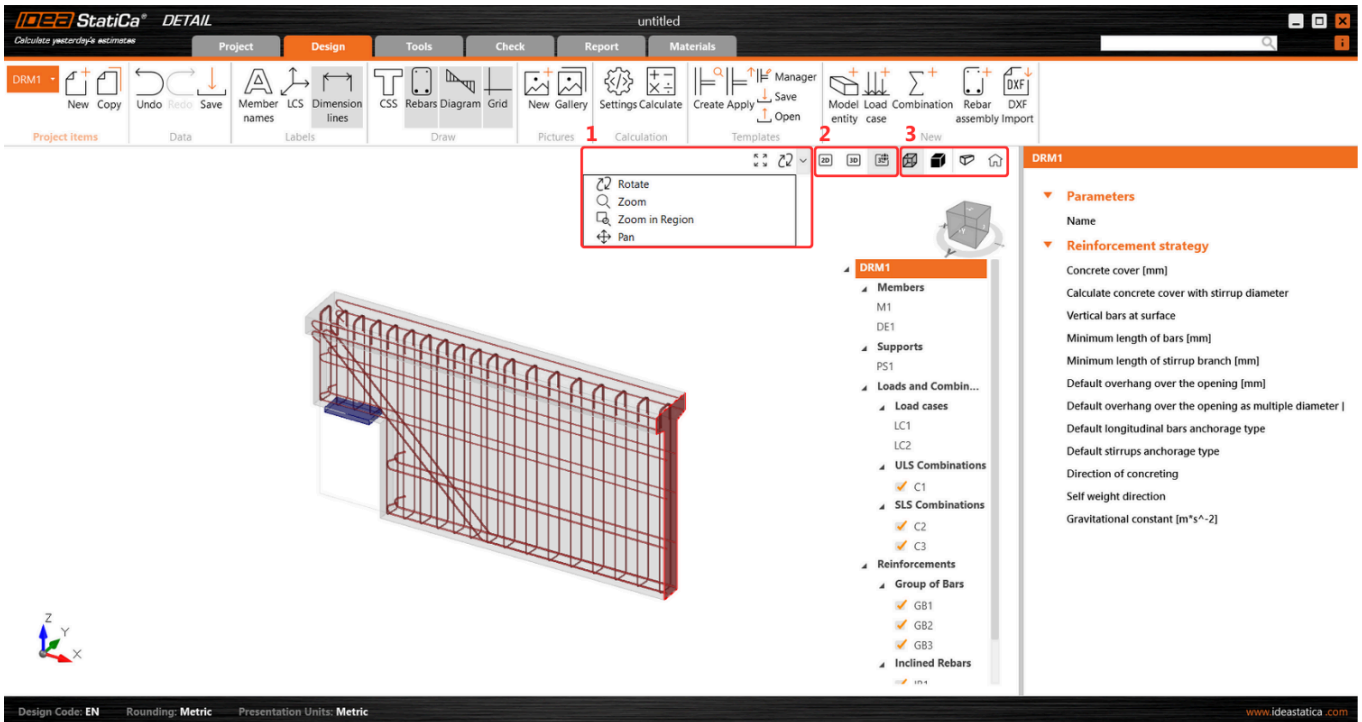
Positioning tools, essential for adjusting the model's orientation to the preferred viewpoint, are conveniently located on the left segment of the control panel. This allows for intuitive and precise adjustments to the scene's layout.

Central to the panel, users can effortlessly toggle between **2D model**, **3D model**, and **3D real** views, facilitating a fluid transition between different visualization modes to suit various analysis or presentation needs. More about the differences between the **view options** are described in General interface in Detail application.

On the right section of the panel, functionalities for altering the **visual presentation** of the model are available. Users can select between **Solid** and **Transparent** view options, which can also be integrated with a **perspective view** for enhanced depth and realism in model visualization.

A **home button** on the right side of the panel ensures easy navigation to the default view, enhancing user experience by providing a quick reset option.

This addresses key user requirements and pain points by incorporating the full suite of standard view control operations. It enables users to engage with the **perspective view** and interact with the scene without limitations, offering a comprehensive and unrestricted exploration of models.

The Information tab displays the selected Project item in a 3D transparent view
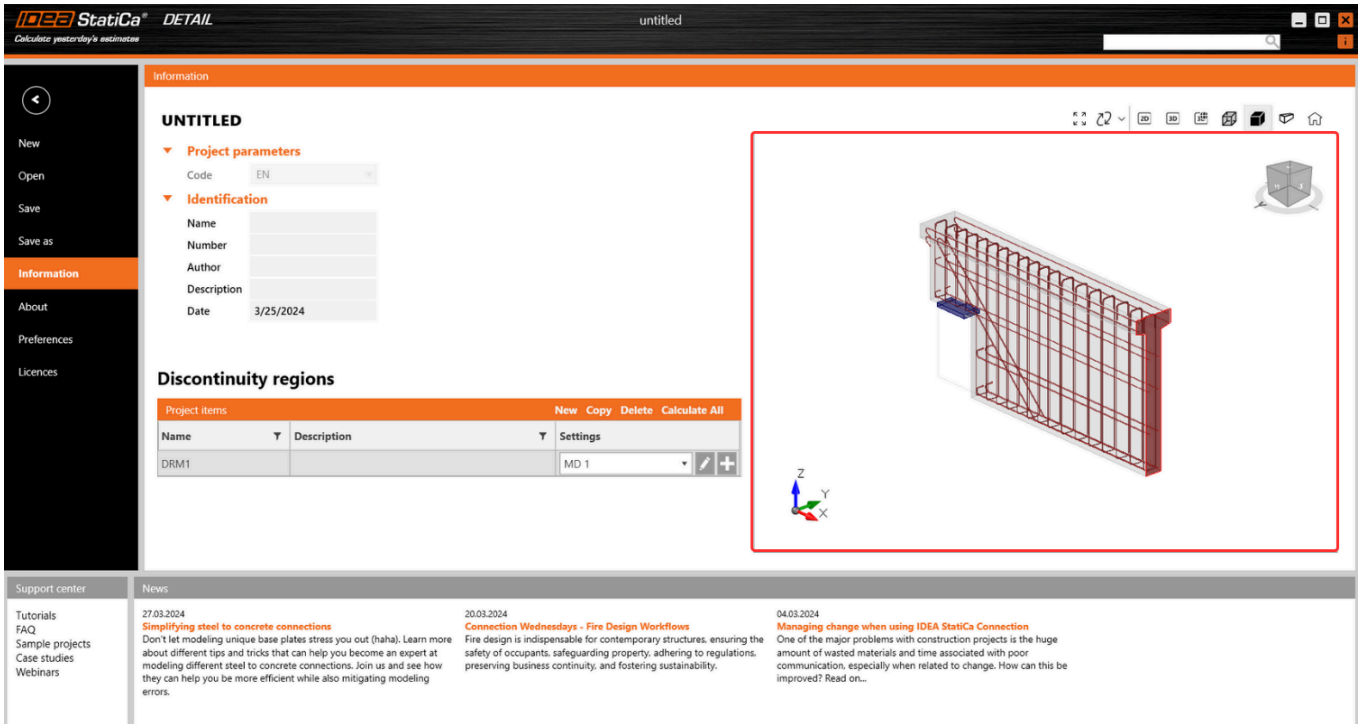
This is particularly useful when a user transitions from the Design/Tools/... tab to the Information tab as the currently active item will be automatically selected and displayed in this mode.

This is designed to facilitate easy navigation and orientation within **Project items**. By providing a 3D transparent view of the selected item, users can quickly understand the item's details, enhancing their ability to efficiently manage and interact with various project components.

Unification of screen dimensions across all graphical outputs
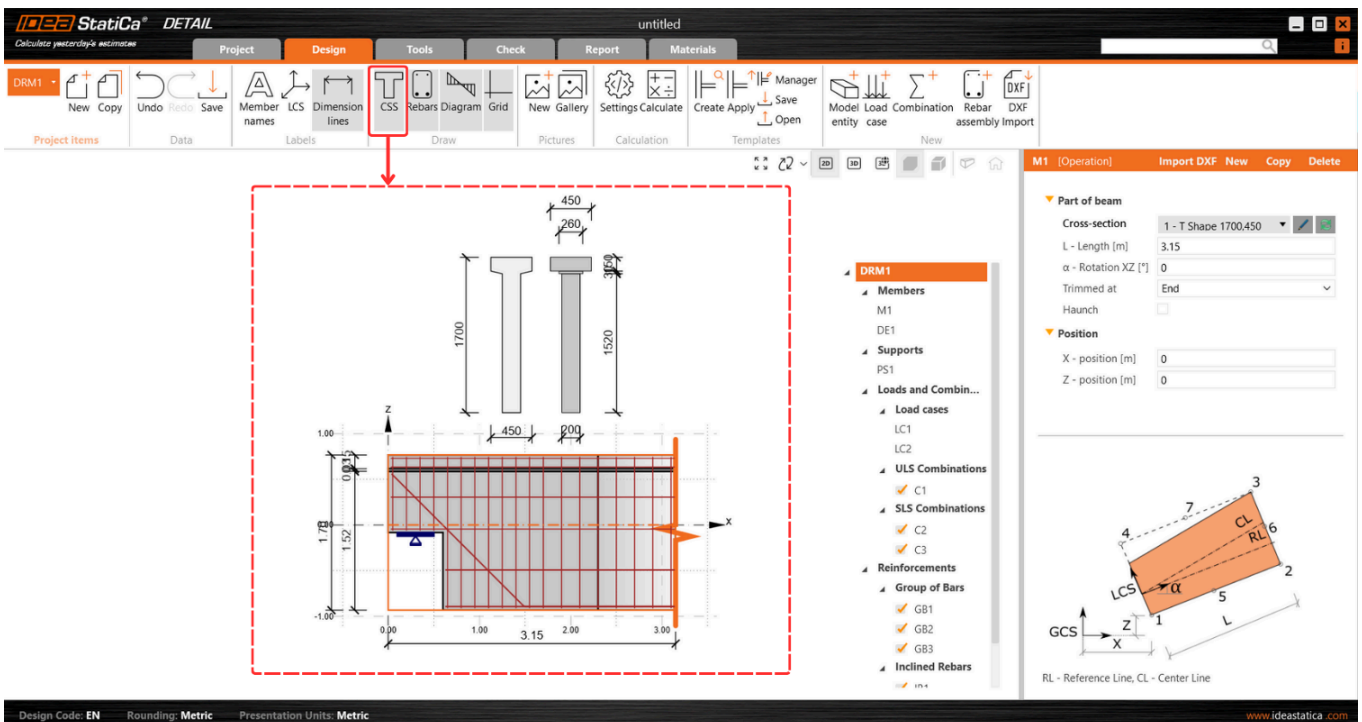
The unified dimension format significantly optimizes orientation within the 2D view, enhancing the overall user experience. To access this feature, units can be adjusted via the **CSS button** on the **Design tab's** top ribbon.

Presentation of relevant curve results in detailed graphs in IDEA StatiCa Detail

The IDEA StatiCa Detail application allows you to visualize load-deformation, stress, strain, and other relevant curves directly within the detailed scene for a selected structure component.
This is achievable through a combination of **"Shift" + left mouse click** or the **right mouse click context menu**. The displayed curve encompasses all calculated load increments for the chosen load case or combination. A specific load increment, as selected from the ribbon, is prominently **marked** on the chart.
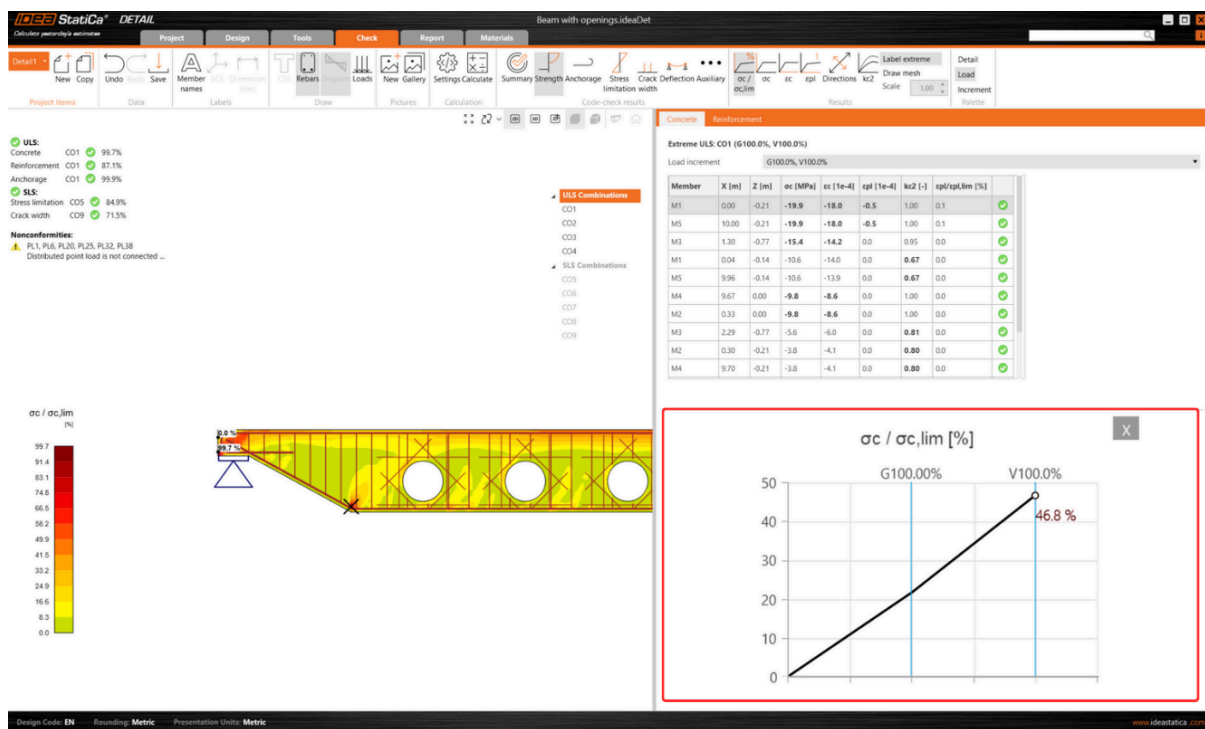
The horizontal axis of the chart denotes the applied load, with symbols representing 100% of applied prestressing (P), permanent load (G), and variable load (V).

The system provides detailed insights into stresses, strains, deformation, crack width, and anchorage forces, among other parameters.

Additionally, you can export the data displayed in the chart into **bitmap, CSV, or DXF** formats. This option is accessible through the chart's context menu, allowing for further data analysis and documentation outside the software's environment.
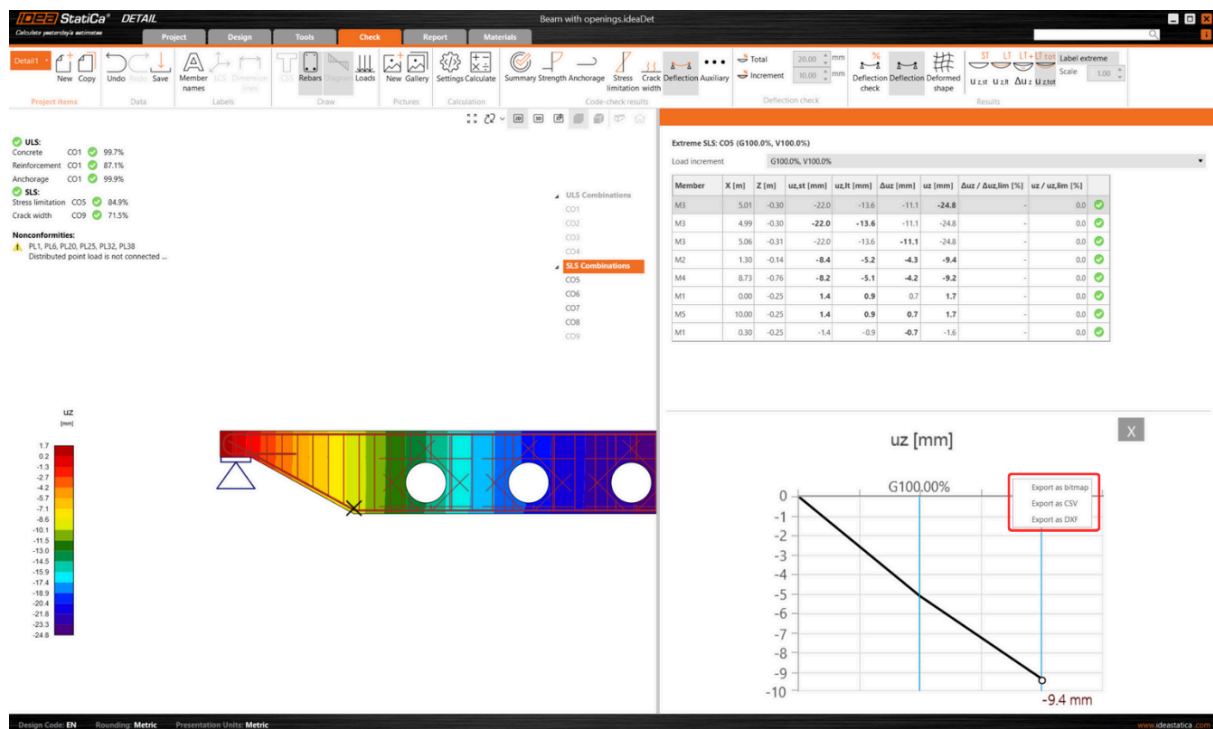
Released in IDEA StatiCa patch 23.1.3.

Unified Materials tab in Detail

The Materials tab enhances the management of cross-sections and materials such as concrete and reinforcement within projects.

The tab's user interface, which is unified with the rest of the application, presents a cohesive and streamlined design. The layout features the scene on the left side and the property grid on the right, which is aligned with the organization found in other application sections.

Key features include an intuitive exploration experience through:

- **The tree of entities**, where users can utilize the context.
- **The menu by right-clicking** an item.
- **The option to wrap and unwrap individual sections** for detailed examination.
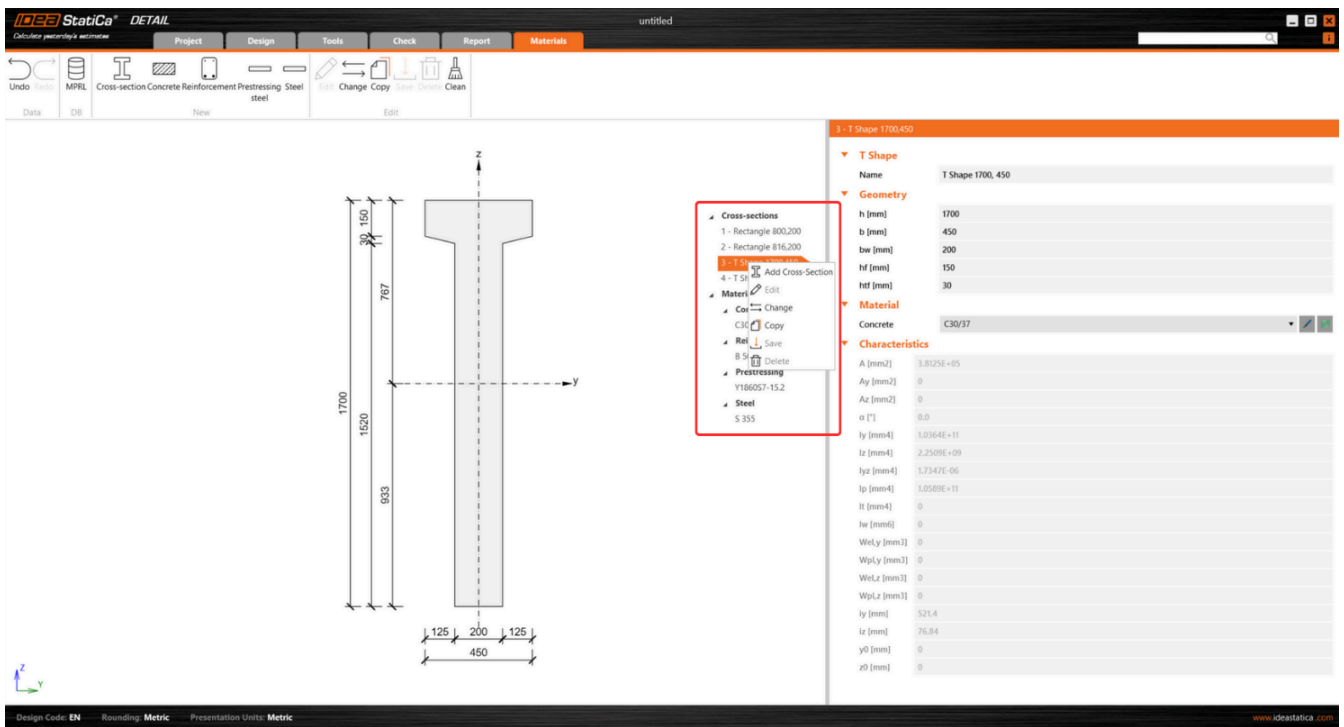
Additionally, the tab offers the ability to easily view stress-strain diagrams for the steel used in bearing plates, providing an understanding of material properties directly within the interface.

Stress-strain diagram
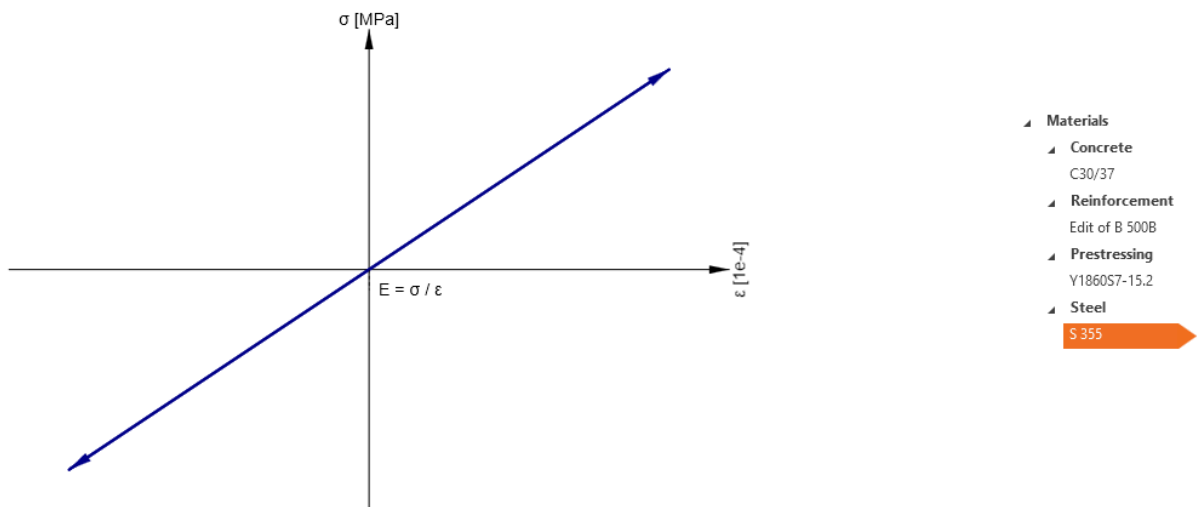
Export functionality allows users to right-click any image in the scene to **export diagrams** in a variety of formats, facilitating documentation and reporting processes.

Scene control includes two diagrams for effective scene adjustments. A notable feature is also the ability to **move the border between these graphs**, providing users with great flexibility in customizing their view.
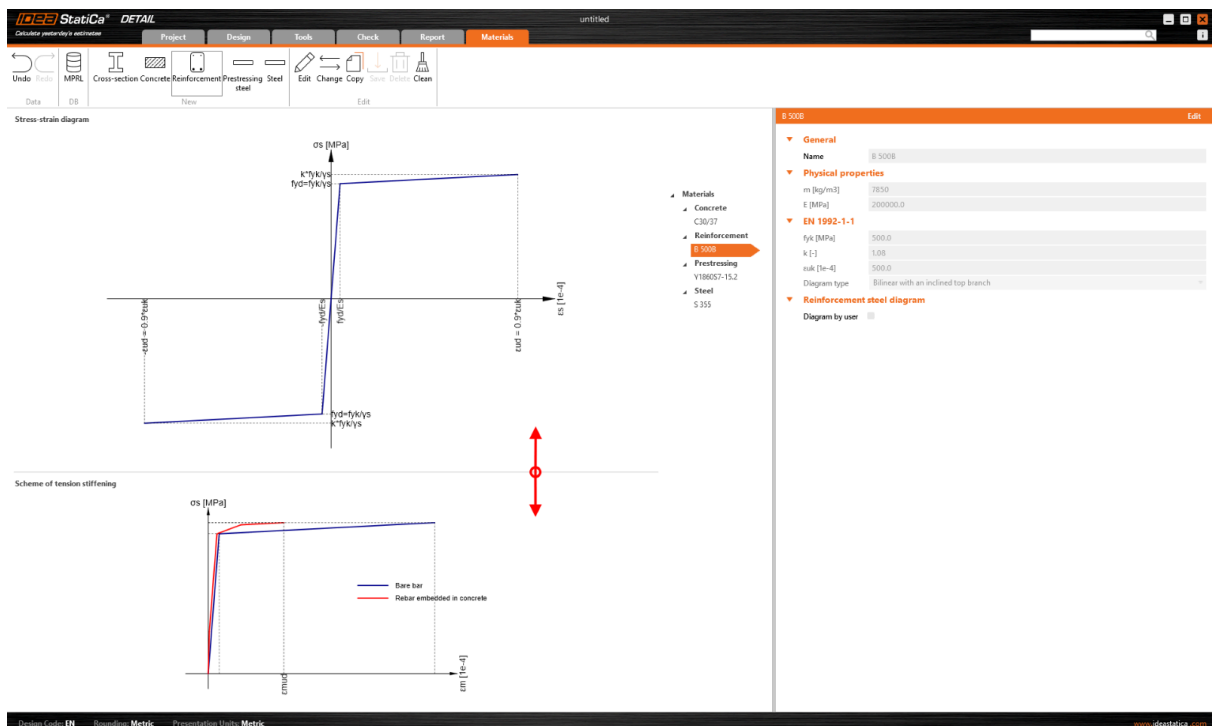
The user interface for the **Materials tab** prevents several potential critical challenges:

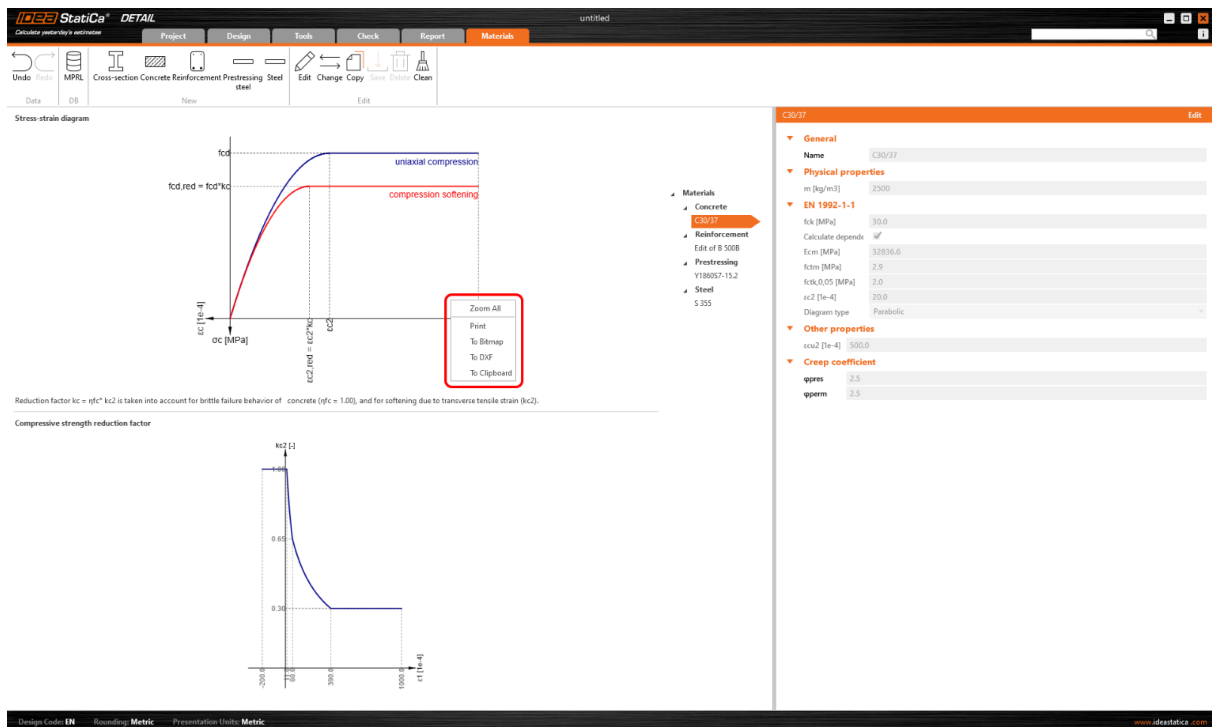- **Managing Diverse Materials**: The tab simplifies the process of managing materials and cross-sections, which is particularly beneficial for rennovation projects, large-scale constructions, or research, ensuring a streamlined workflow.

- **Complex Project Structures**: Given the complexity of projects with numerous items and varied materials, the UI promotes efficiency and ease of navigation, making it easy to manage and review material properties and cross-sections.

Released in IDEA StatiCa patch 23.1.4.

SIA 262:2013 implemented in RCS and Beam

The integration of the SIA 262:2013 material database into the IDEA StatiCa RCS and Beam applications is for users working within the Swiss standards. This provides access to concrete, reinforcement, and prestressing materials as defined in current SIA 262:2013 standards.

SIA 262:2013 can be selected when a new project is created and displayed in the **project data** and the standard is also listed in the final report. The implementation of **SIA 262:2013** includes assessments, partial factors, and **materials libraries**.

Materials for concrete, reinforcement, and prestressing reinforcement can be found in the MPRL database, which is implemented in the IDEA StatiCa RCS and Beam applications.

Access is provided to materials for concrete, reinforcement, and prestressing reinforcement as defined in the most recent **SIA 262:2013** standards, plus flexibility is maintained for users who prefer to work with the material specifications from the earlier **SIA 262:2003** standards.

Due to our users' diverse needs and preferences, materials from **SIA 262:2003** standards continue to be available, ensuring compatibility and flexibility for ongoing and legacy projects.
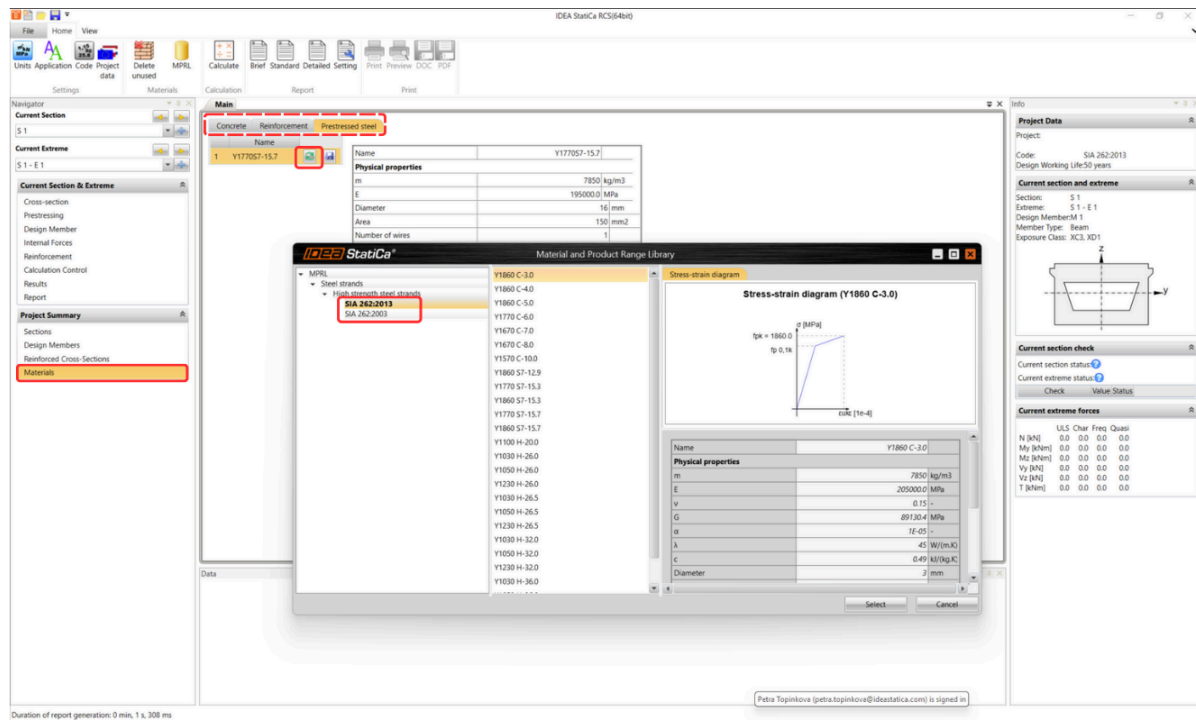
Released in IDEA StatiCa version 24.0.

Accurate calculation of the shear resistance of slabs

Find out about the enhancement of shear resistance calculation for 2D sections in IDEA StatiCa RCS leads to accuracy of the calculation of 2D slabs.

The calculation of the resulting shear force was calculated in the past according to the following formula:

$$v_{d,max} = \sqrt{v_x{}^2 + v_y{}^2}$$

Then, the force was applied to all analyzed angles between the x-axis, and directions were checked. In most cases, the shear and interaction checks were overly conservative.
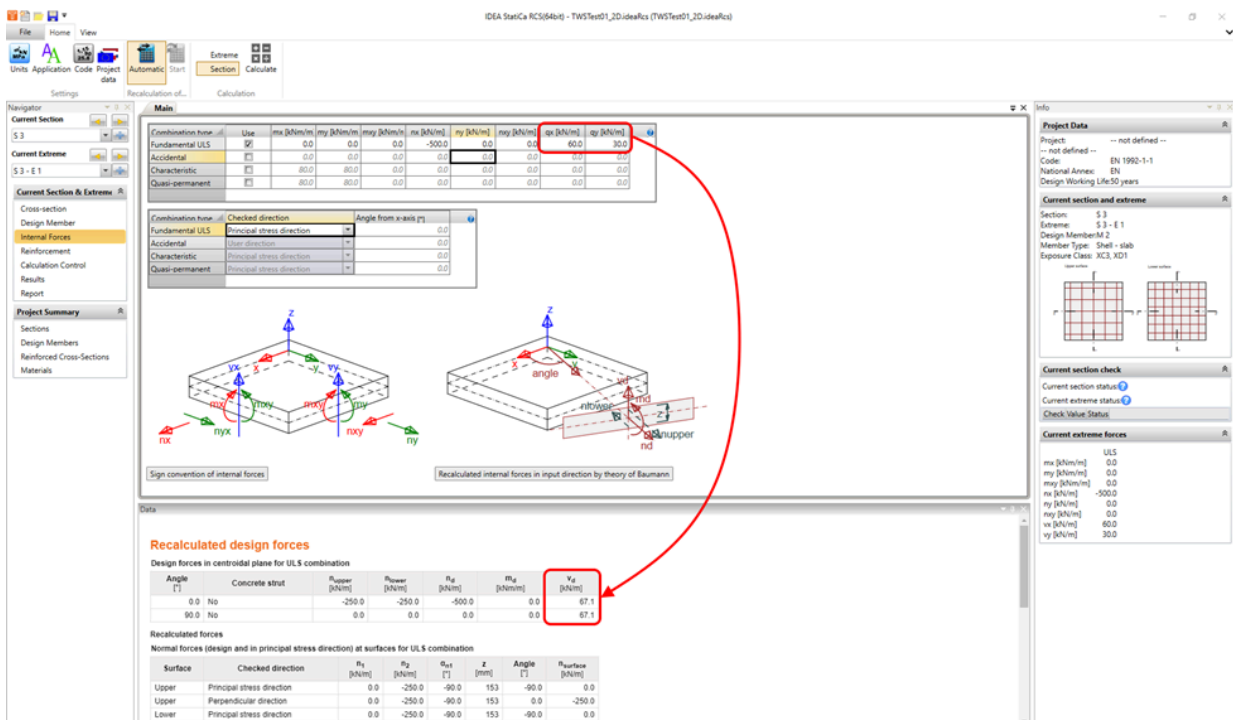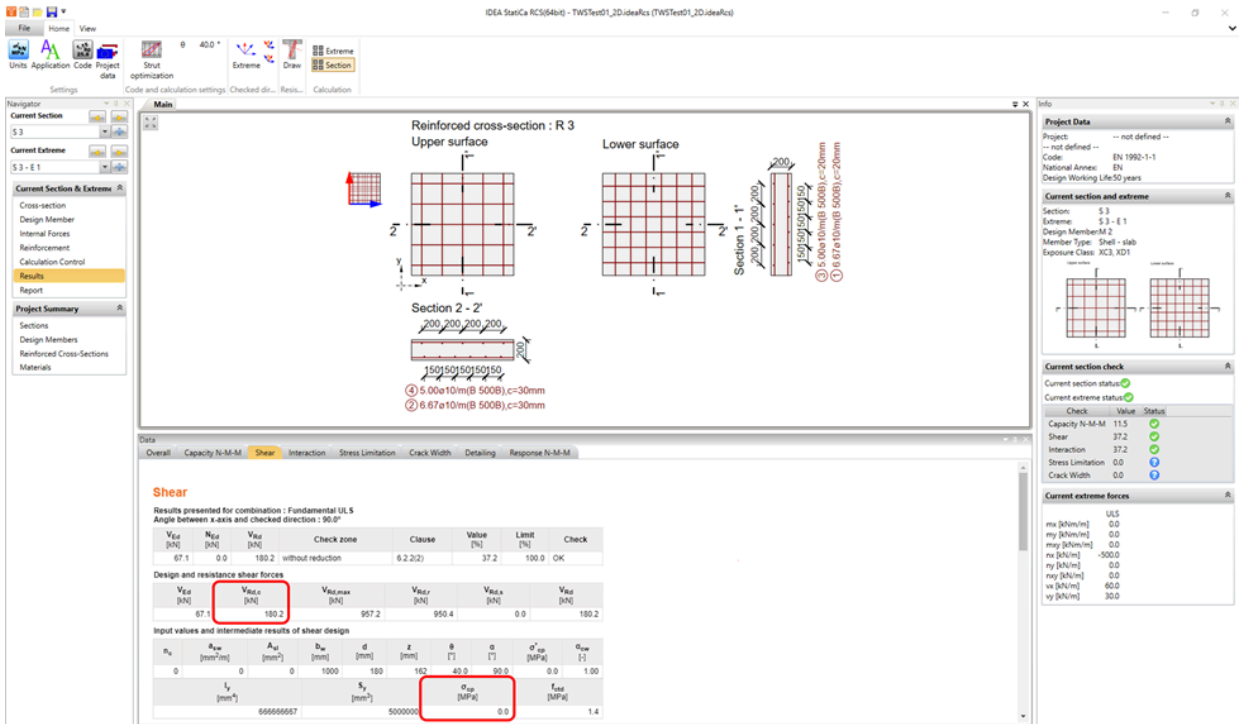
As shown in the figure above, the corresponding normal force is zero for the angle of 90°. The compressive normal force influences the design shear resistance $V_{Rd,c}$, respectively, the value of σ_{cp} according to **EN 1992-1-1 6.2.2 (1)**.

In IDEA StatiCa RCS, **additional angles correspond to the direction of the resultant shear force** (26.6°), **the perpendicular angle** (116.6)°, and **the angle of the concrete strut**. These angles are assigned a corresponding shear force as well as other internal forces.
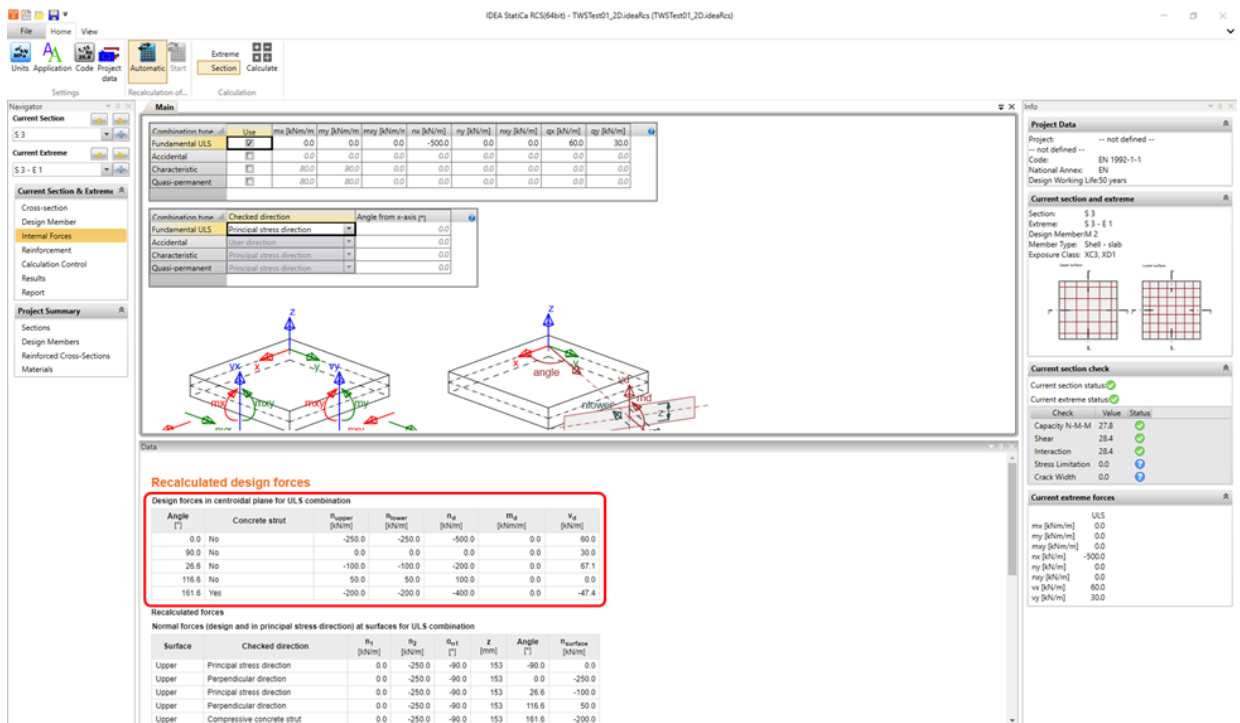
A shear check's governing (extreme) angle is now different (26.6°). The results are more accurate with proper $V_{Rd,c}$ respectively σ_{cp} values.
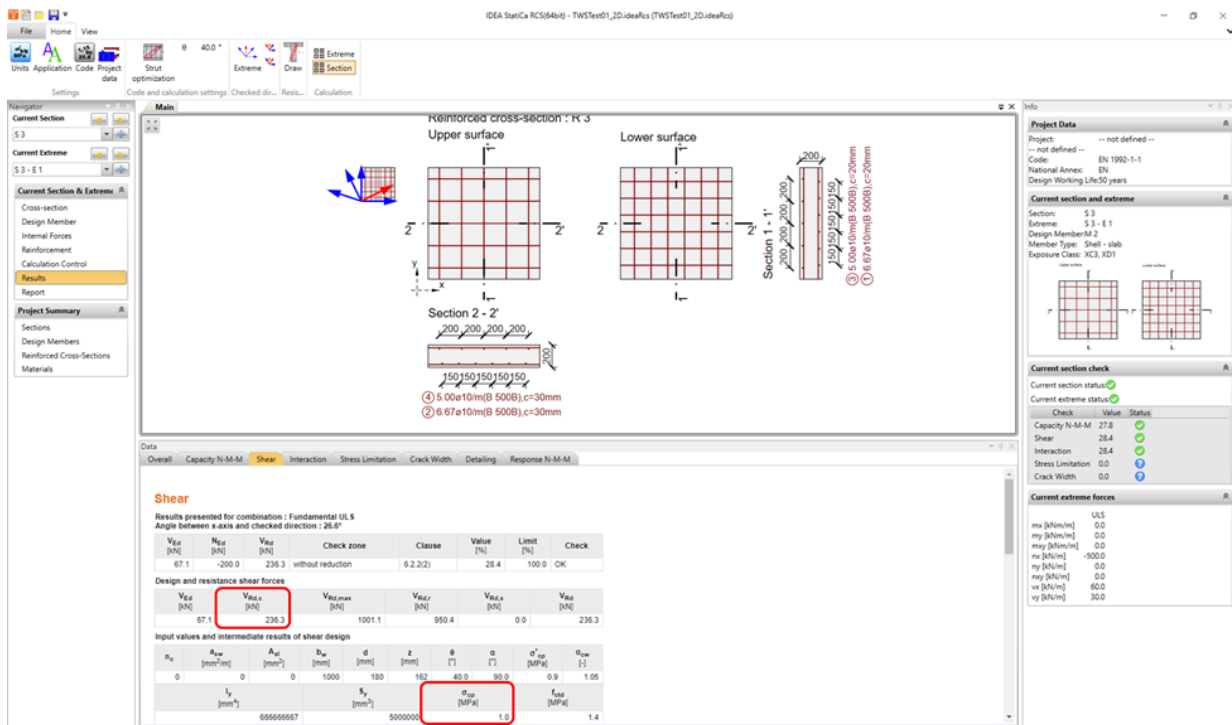
This method **accurately determines** the direction of shear force and conducts code checks accordingly, aligning with the corresponding internal forces. You can find more information on how the internal forces are recalculated in the Theoretical Background for RCS - 2D sections.

If you are interested in designing concrete slabs and would like to know more about Baumman theory implemented in IDEA StatiCa RCS, try our webinar: Baumann's method for design of concrete shells in practice.

Released in IDEA StatiCa patch 23.1.2.

BIM and Checkbot

Checkbot Free – the structural design hub for FEA and CAD data

Checkbot Free offers unification and simplification of the workflow for users across various regions. It enhances accessibility and streamlines the process of structural engineering projects through design data management and sharing between different tools.
The IDEA StatiCa Checkbot Free application works both as a plugin linked to a specific FEA or CAD tool and as a standalone application capable of importing, managing, and exporting different types of design data. For the majority of its capabilities, the paid license is not necessary. The only necessity for running it is a valid IDEA StatiCa user account. This means that also without the paid, educational, or trial license, the user can still leverage the advantages of the Checkbot Free features.

If the user has not yet created an IDEA account, they can simply follow this step-by-step guide to set up a **Basic account**.

Checkbot Free offers a range of functionalities aimed at improving project collaboration and efficiency:

- **Import compatibility**: Users can import projects from all supported CAD and FEA BIM links directly into IDEA StatiCa Checkbot.

- **Sharing of design objects**: Enables seamless sharing of design objects among project stakeholders.

- **File export**: Supports the export of `.ideaconn` files, facilitating easy exchange and collaboration.

- **Viewer access**: Allows opening selected connections in our Viewer web browser tool, ensuring project details are easily accessible for review and discussion.
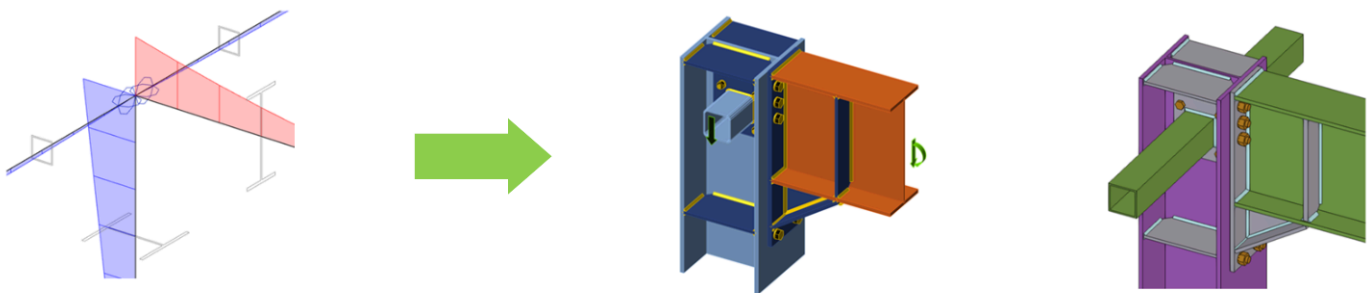
User's benefits

- **Licensing flexibility**: Free Checkbot is included in one setup package, where IDEA StatiCa applications are unlocked with a license. Even when all paid seats are occupied, users can still perform imports and exports in the Checkbot app.

- **Replacement of free Viewer plugins**: This aims to supersede the free CAD viewer plugins for Advance Steel, Revit, and Tekla, centralizing functionality within Checkbot.

- **Unified solution**: Prevents the fragmentation of tools (CAD Viewer plugin, IDEA StatiCa toolset installation, Checkbot not linked to Viewer) by centralizing Checkbot as a hub for structural engineering data management.

69

Intuitive user experience

- **Informative tooltips**: Disabled commands within the application feature tooltips explaining why they are unavailable, enhancing user understanding and navigation.
- **Clear identification**: The free version is clearly labeled as "Checkbot Free" in the title, distinguishing it from the licensed version.

Released in IDEA StatiCa version 24.0.

Checkbot ready for big projects

The IDEA StatiCa Checkbot app is built to import design data from different sources, process them, and export them for other purposes. Its goal requires the ability to process everything in a reasonable time. The size of imported models varies from simple structures to complex ones.

The ability to link IDEA StatiCa apps to dozens of different tools brings the need to create different approaches for specific vendors. The BIM link data transfer is polished and streamlined to make the processes as smooth and quick as possible.

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

Faster imports for single connection from FEA links

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

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

Faster opening of the existing projects

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

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

Released in IDEA StatiCa version 24.0.

Fast project visualization in Viewer

Viewer, as a cloud-based app, is a powerful tool enabling anybody to visualize IDEA StatiCa Connection projects in an online 3D scene swiftly and for free.

There are several major reasons why the Viewer is popular in communication between structural engineers, detailers, draftsmen, and contractors, including:

- Enhanced responsive 3D scene
- 3D model scales dynamically to fill the entire screen
- Pop-up messages – informing the user about ongoing processes
- Fast switching between Project items
- 3D dwg download processed in the background (the workspace is not blocked)
- Load effects are displayed in the navigation tree

Released in IDEA StatiCa version 24.0.

Enhanced 3D scene interaction and property management in Checkbot

The Checkbot 3D scene implements property grid technology, applicable to all selectable items within the scene, including Design Member, Substructure, Node, Connection, Detail, and Member 2D.

Benefits for IDEA StatiCa Checkbot Users

- **Multi-edit option for eccentricities**: Users can edit the eccentricities of multiple elements simultaneously, significantly streamlining the modification process and enhancing their productivity.
- **Enhanced readability for substructure elements**: The visibility and readability of analyzed or related members within the Substructure part facilitate a better understanding and inspection of these components.
- **Clear contrast for "red only" properties**: The property grid features a clear contrast for properties highlighted in red, providing good visibility of critical information and warnings.

Benefits for the entire platform

- **Unified technology across applications**: By having the same property grid technology across all IDEA StatiCa applications, maintenance and future extensions are straightforward, ensuring a consistent user experience and avoiding development overheads.
- **Powerful capabilities**: The integration of this technology enables features such as multi-select and multi-edit capabilities, enhancing the flexibility and efficiency of the applications.

Released in IDEA StatiCa version 24.0.

Tekla BimApi link available on GitHub

The Tekla link exists as a BimApi link, which streamlines the integration and functionality of IDEA StatiCa with BIM software.

This Tekla BimApi brings about several key benefits for IDEA Statica Checkbot users, including:

- **Unique naming for imported members**: Names of imported members follow a consistent format of "Member + ID", ensuring clarity and preventing confusion in project management.

- **Unified solution across BIM links**: By standardizing the approach to BIM links, including the Tekla link, the solution's overall efficiency is maximized.
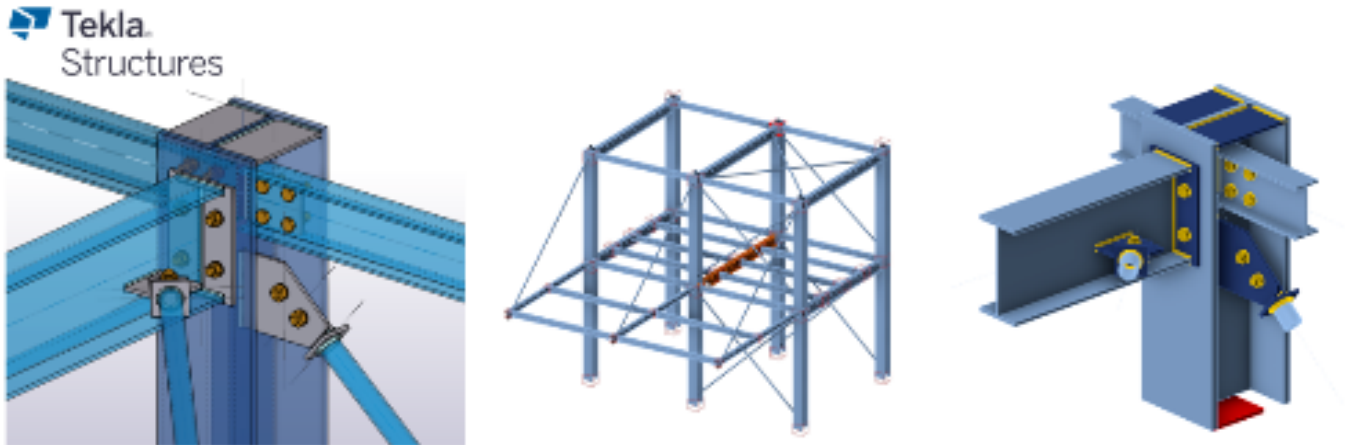
Accessibility

- **Public GitHub repository**: The Tekla link is accessible on GitHub at IDEA StatiCa Public GitHub Repository. This public availability fosters a community-driven approach to development and support.

- **Enhanced support for third parties**: With the BimApi link available on GitHub, third-party developers and partners implementing CAD to IDEA Checkbot integration can benefit from clear guidance, reducing the need for extensive assistance from the BIM team. Current integrations include Bocad, with potential future expansions to include Qnect.

Addressed Challenges

- **Low development assistance costs**: By providing a comprehensive and easily accessible resource on GitHub, the need for direct assistance from the BIM team is minimized, leading to significant cost savings.

- **Clarification of design member naming**: The unique naming convention for imported members addresses potential user confusion regarding the identification of design members within Checkbot.

User Experience (UX):

The development process for new CAD links to IDEA StatiCa Checkbot is seamless, making it straightforward and non-time-consuming for developers.
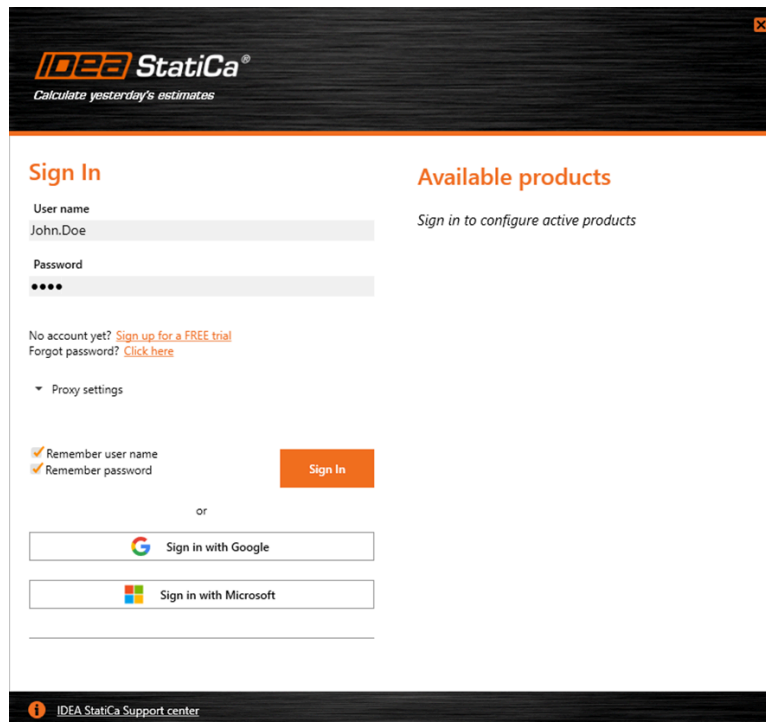
Released in IDEA StatiCa version 24.0.

# Usability and Licensing

Single Sign-on (SSO)

Single Sign-on (SSO) is being implemented for both desktop and web user authentication. It is an Enterprise feature that enables users to use their Microsoft or Google credentials to sign in and for company admins to easily assign and manage their user base.

Single sign-on authentication enables centralized authentication. With this feature, users can enjoy seamless access to all IDEA StatiCa applications using a single set of login credentials. This simplifies the login process, providing a smooth user experience by eliminating the hassle of managing multiple passwords. Additionally, this unified access approach enhances security across our platform, providing users with peace of mind and a streamlined workflow.

Single Sign-on simplifies access, enhances security, and boosts the productivity of the customers. It empowers users to access both desktop and web applications. SSO also streamlines access management for company admin users.
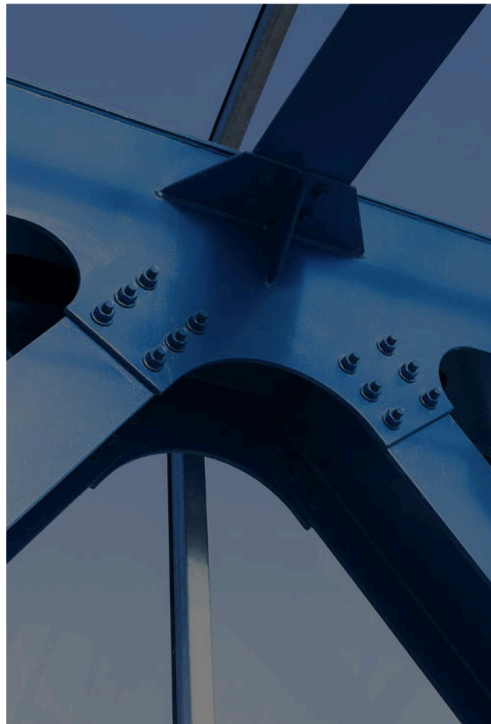
This feature brings several benefits such as, fast login time, higher security, no need to remember passwords, and easier user management for company admins.

Released in IDEA StatiCa version 24.0.

Enhanced User Portal

IDEA StatiCa User Portal is the space for maintaining the user's relationship with IDEA StatiCa and its products. It enables the administration of licenses, analytics about the tool usage or progress in Campus courses, all in one place.

Key features of the User portal environment

- User-oriented content is shown on the main platform.
- Clear communication of expired licenses.
- A **clear dashboard** with important data in one place
  - Account details
  - Reseller info
  - Easy-to-use **helpdesk cases** management
  - Access to **courses** of IDEA StatiCa **e-learning platform** available here.
- The side menu also makes it easy to
  - Browse through **License analytics** with the product usage overview
  - Access the cloud tools **Connection Library** and **Viewer**

Released in IDEA StatiCa version 24.0.

New portal layout

NAVIGATION TREE

DATA PRESENTATION
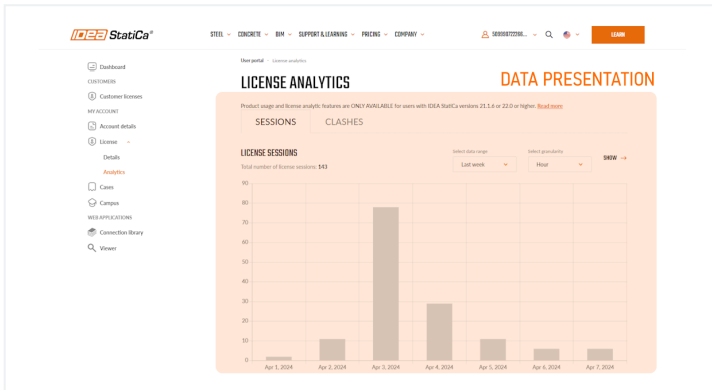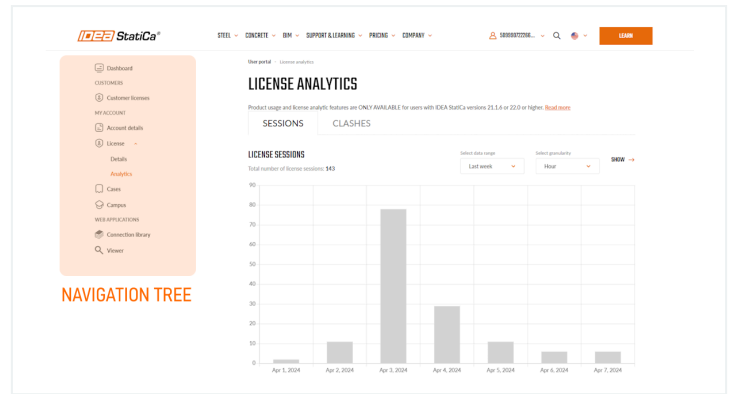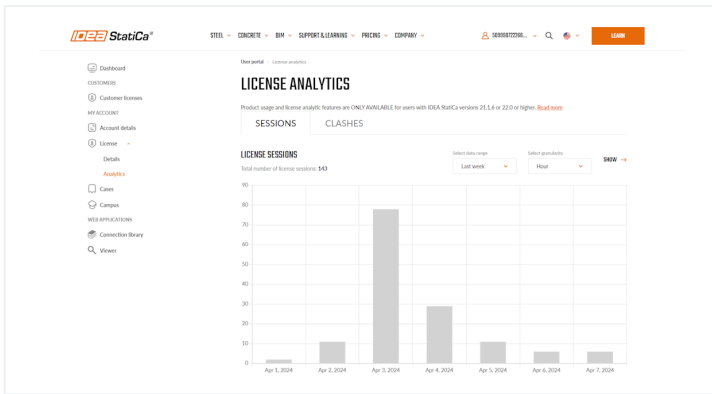
Besides administration purposes, there are links to IDEA cloud tools present there as well.

Shared settings across the whole tool range

Ensuring consistently shared preferences across Connection, Checkbot, Detail and Member applications significantly enhances the user experience and reduces the time spent on settings.
Embedding this feature addresses these major reasons:

- Consistent settings across all the apps

- Simplification of users' workflows

- Ability to set all the shared preferences in one place for all the tools at once

The Preferences dialogue is designed as a simple tool accessible either from the initial IDEA StatiCa dialogue as well as from the particular applications. The shared preferences, wherever set, will be applied also for the other apps.
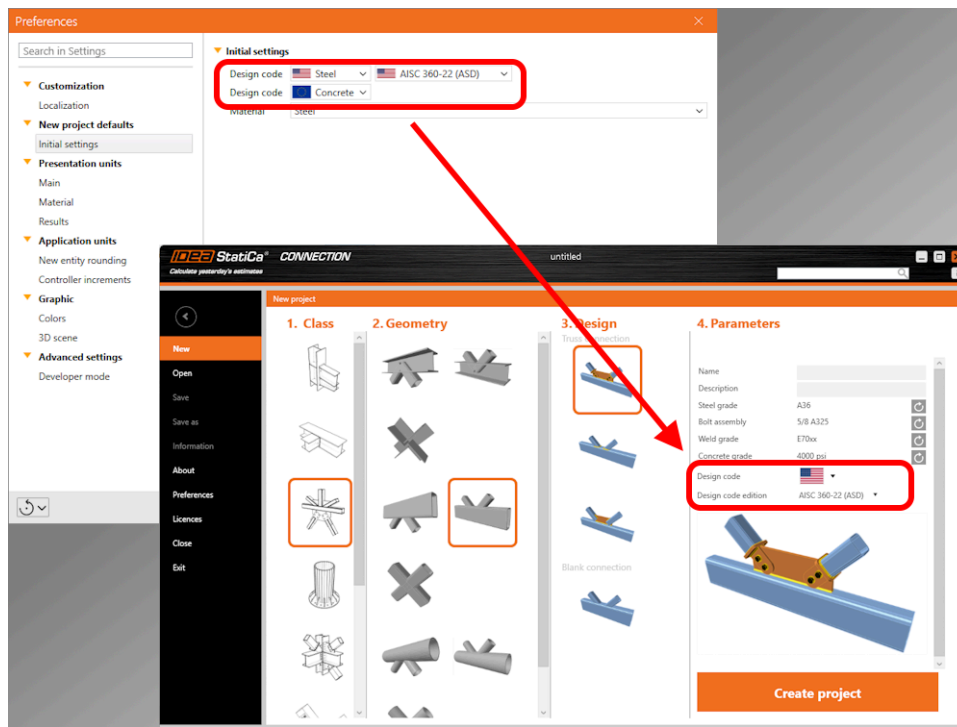
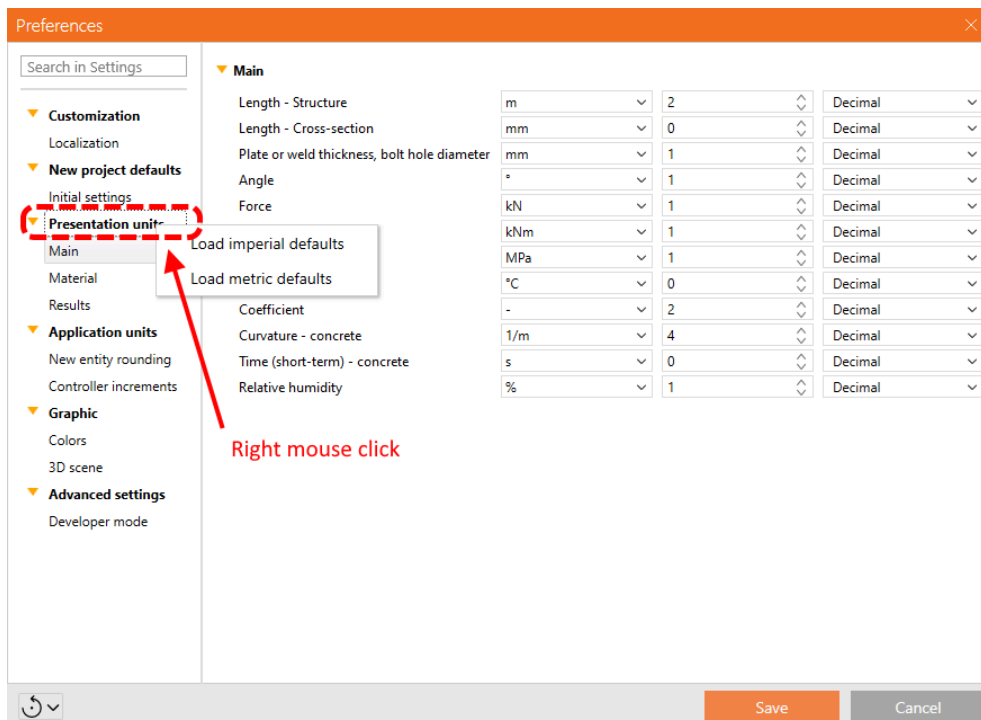An example of usage is defining the default Design code and subcode in the **Preferences** dialogue, causing these parameters to be predefined for all new projects.
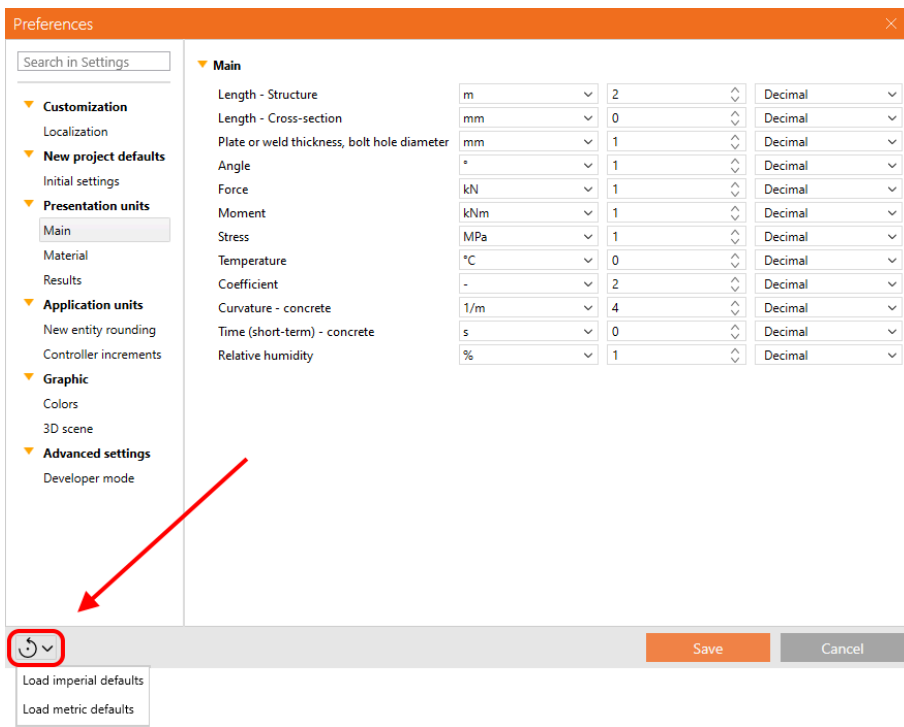
How to set the Default state

There are two ways to get the parameters back to the default settings. The default values are prepared in two versions – Imperial and Metric.

 1) Reset just the specific category

 2) Reset the whole list of preferences

When starting in Checkbot

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

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

Released in IDEA StatiCa version 24.0.

Error handling with Support Center integration

The IDEA StatiCa desktop application includes an error dialog feature informing users when a crash has occurred due to a bug already addressed in a newer version of the software.
This troubleshooting service is designed to provide immediate assistance for license-related issues encountered by users of the IDEA StatiCa desktop application with links to solutions or additional information.
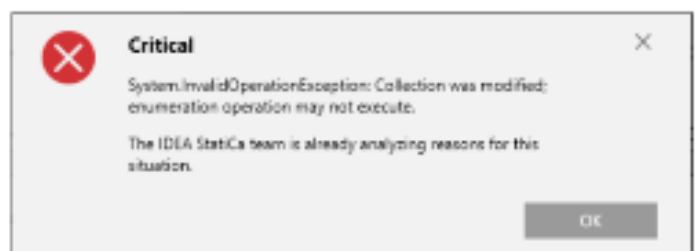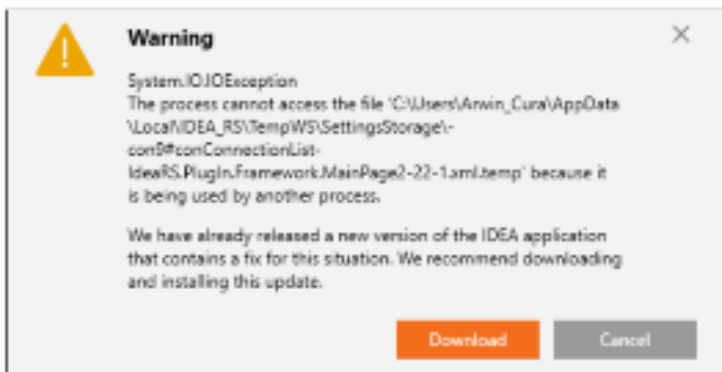
The desktop application can overcome any occurrence of crash issues by providing clear communication to the user about the cause of the error and the recommended action to resolve the issue – specifically, updating the IDEA StatiCa application. This approach ensures users are well-informed about the steps they can take to avoid future problems, thereby enhancing the user experience by offering direct guidance on how to proceed in the event of an application crash.

The error-handling tool offers explanations about what is going on during technical or licensing issues and provides direct links to articles where the solution is explained in a more comprehensive way.

Integration with the Support centre
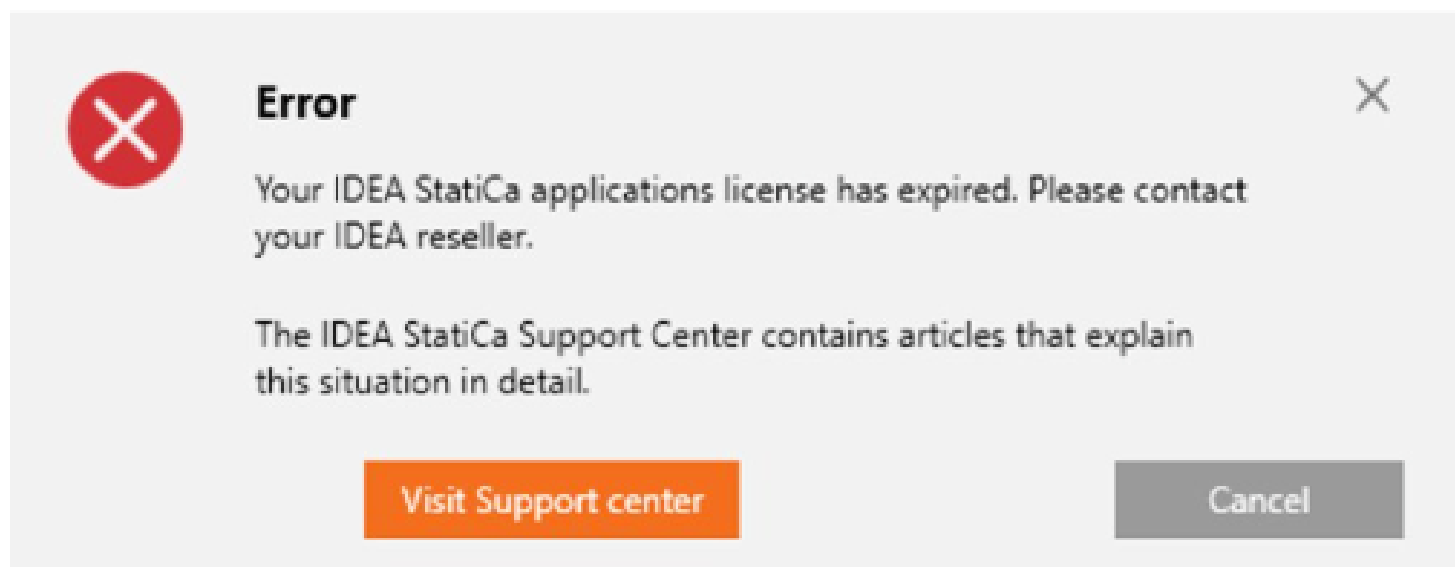
Users are equipped with the knowledge to understand the causes behind errors within the desktop application. Upon encountering an error, they are directed to a specific section of the Support Center through a direct link, where users can access solutions and detailed information pertaining to their issue.

Faster solution identification

This integration addresses several different types of situations:

- Known licensing issues – linking the user to specific knowledge base articles.
- Known technical issues that are already solved – linking the user to the current installation setups.
- Known technical issues that have already been reported – informing about no need for further actions.

This not only aids in error resolution but also enhances the user experience by providing targeted support, directly addressing the causes of common issues encountered within the application.

Released in IDEA StatiCa patch 23.1.3.

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 direct access to Support Center articles right from the desktop application, resulting in finding quicker answers and reducing the need for contacting technical support.

Users can support and educate themselves immediately regarding frequently asked topics. The tooltip stays displayed on mouseover on the tooltip itself, not just the icon.

This reduces the time necessary for users to self-help when they are uncertain about how the features work. The hyperlinks are maintained to give the user the best possible information from the knowledge base article library.

Released in IDEA StatiCa version 24.0.

Keyboard shortcuts (hotkeys) in IDEA StatiCa apps

The Connection and Detail applications feature an enhanced set of commonly used keyboard shortcuts, designed to streamline user operations and align with familiar practices observed in other engineering software.

Available keyboard shortcuts:

- **Save (Ctrl + S)**: Initiates the "Save" function for projects that have been previously saved. For new projects, it opens the "Save as" dialog.
- **Undo (Ctrl + Z)**: Performs a single undo step, allowing users to revert the last actions in steps.
- **Redo (Ctrl + Y)**: Enables a single redo step, facilitating the reapplication of the most recently undone action.
- **Open Project (Ctrl + O)**: Launches the "Open browser" in the backstage menu, displaying also the list of recent projects for quick access.
- **New Project (Ctrl + N)**: Launches the "New project" in the backstage menu, to start a new project file. It does not create a new project item within the current project.
- **Rename (F2)**: Allows to rename the currently selected item in the project tree, providing a quick way to organize and identify project components.

Released in IDEA StatiCa version 24.0.

New Portuguese (Brazilian) language in apps

Portuguese (Brazilian) has been added to the language set across all applications.