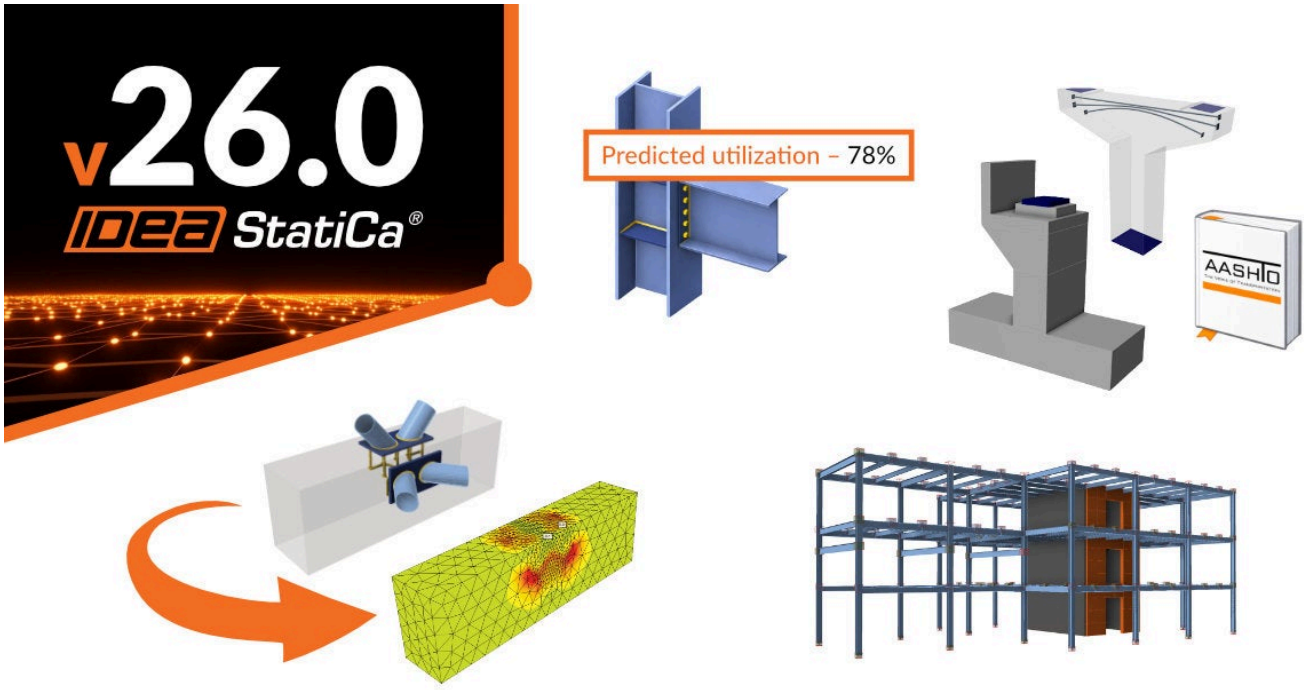


RELEASE NOTES

Release notes IDEA StatiCa 26.0

Apr 22, 2026

Table of Contents

Along with the highlights in version 26.0, read the full list of new functionalities here.

Concrete Design

Cast-in plates – Headed studs and smooth workflow

Cast-in plates are fully integrated in Detail 3D, providing a complete workflow from Connection import, faster modeling, and simpler loading. Engineers can combine welded reinforcement and headed studs on a single plate, apply realistic column forces via a stub, and follow a synchronized Connection → Detail process aligned with SCI P416.

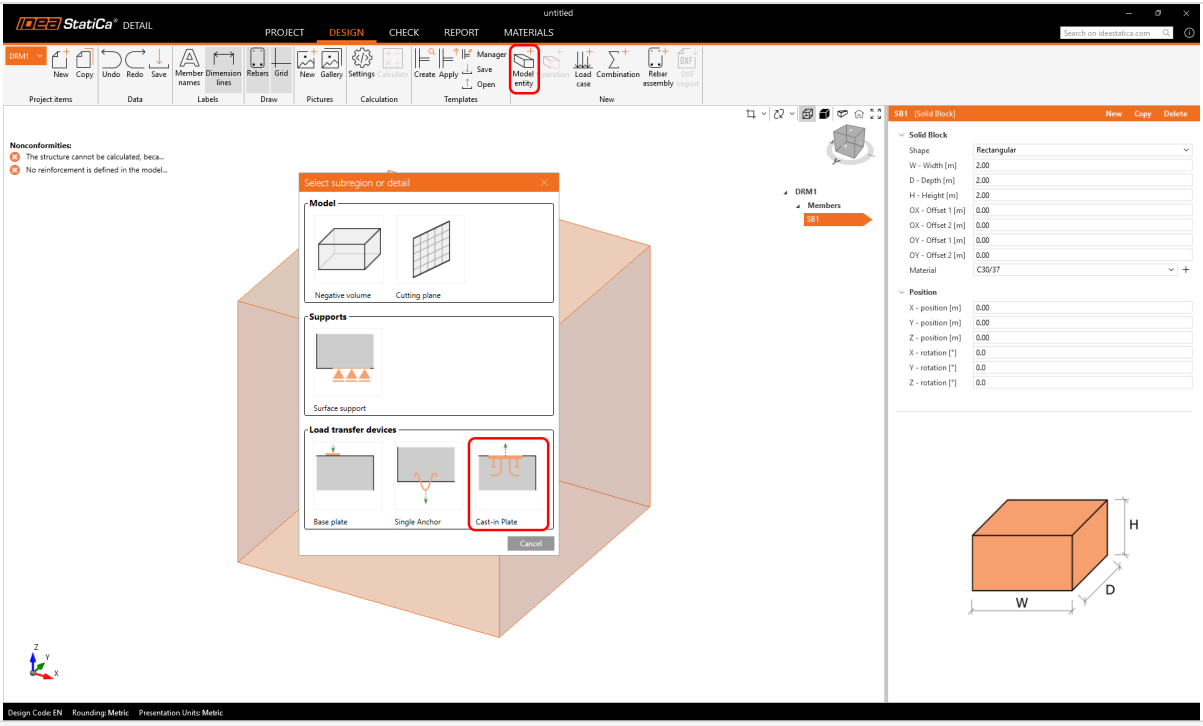
More about cast-in plates is described in the following article.

Cast-in plates in Detail

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

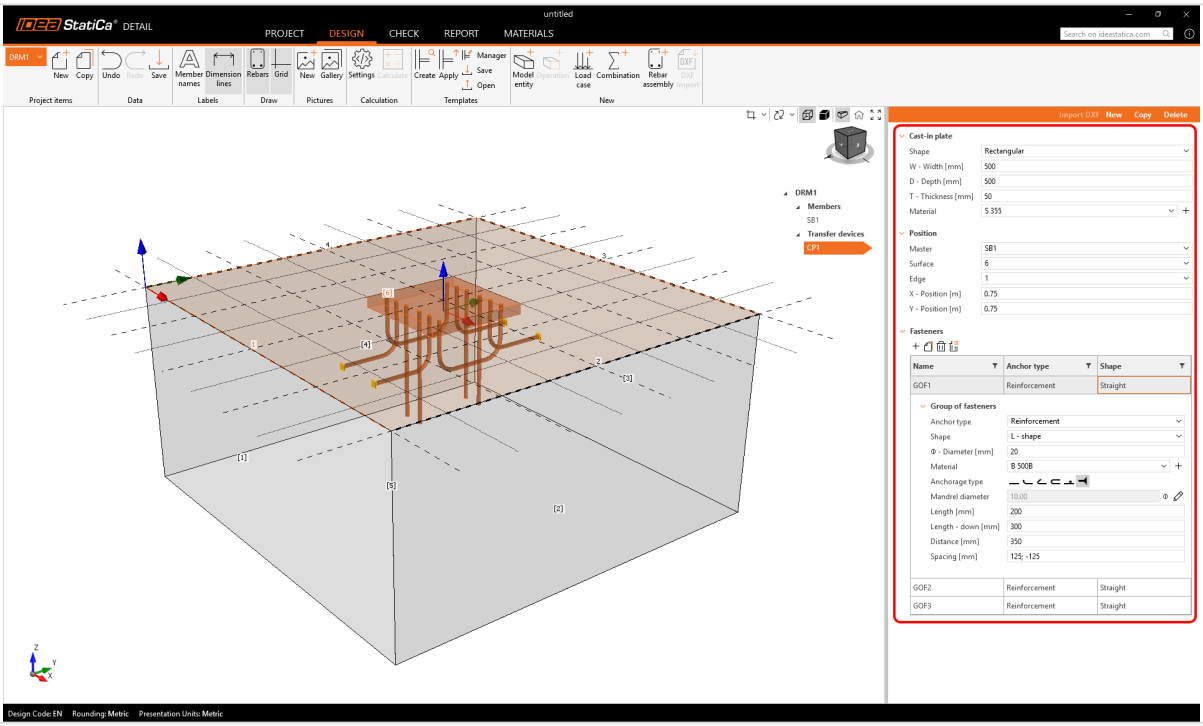
Cast-in plate device

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

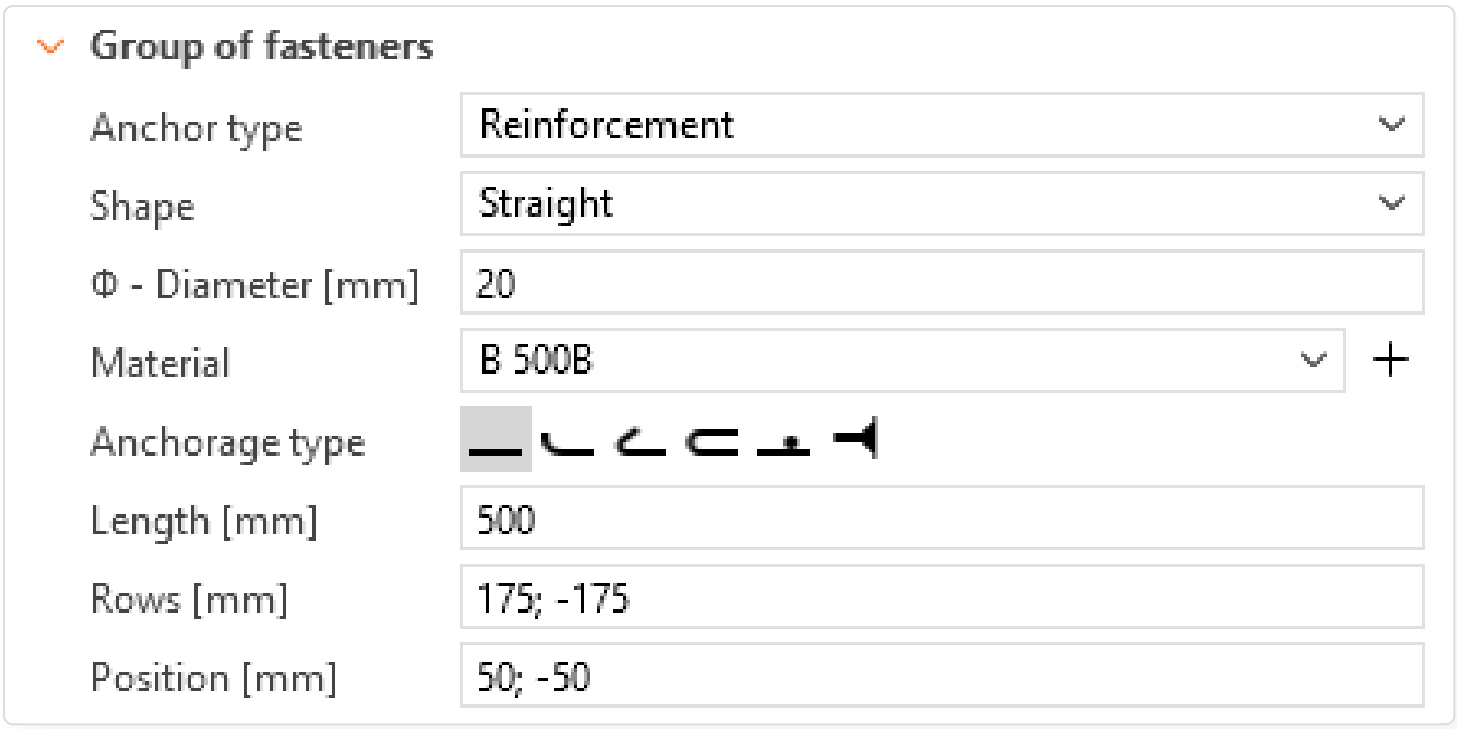


Reinforcement types

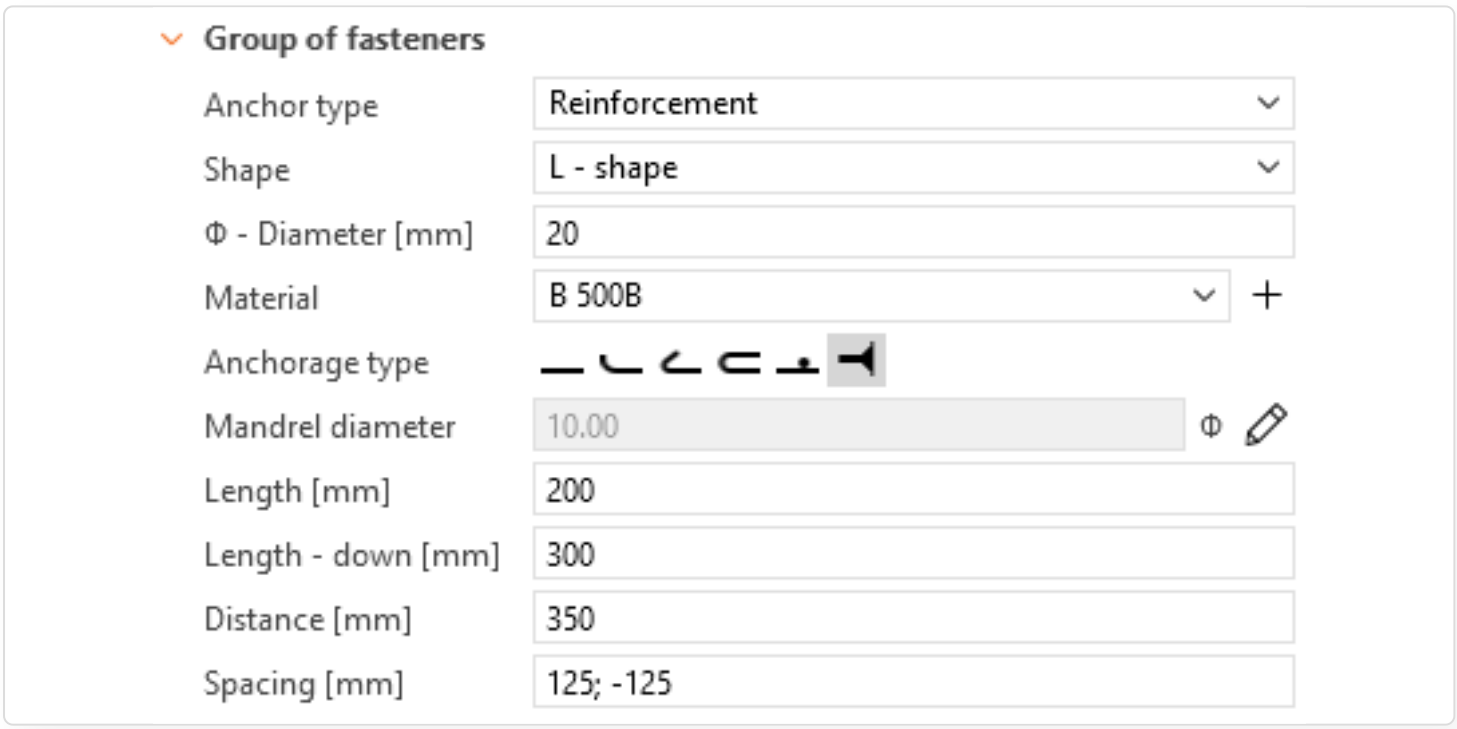
The Cast-in plate allows for **welded reinforcement** in three shapes: **Straight, L-shape, and U-shape.**



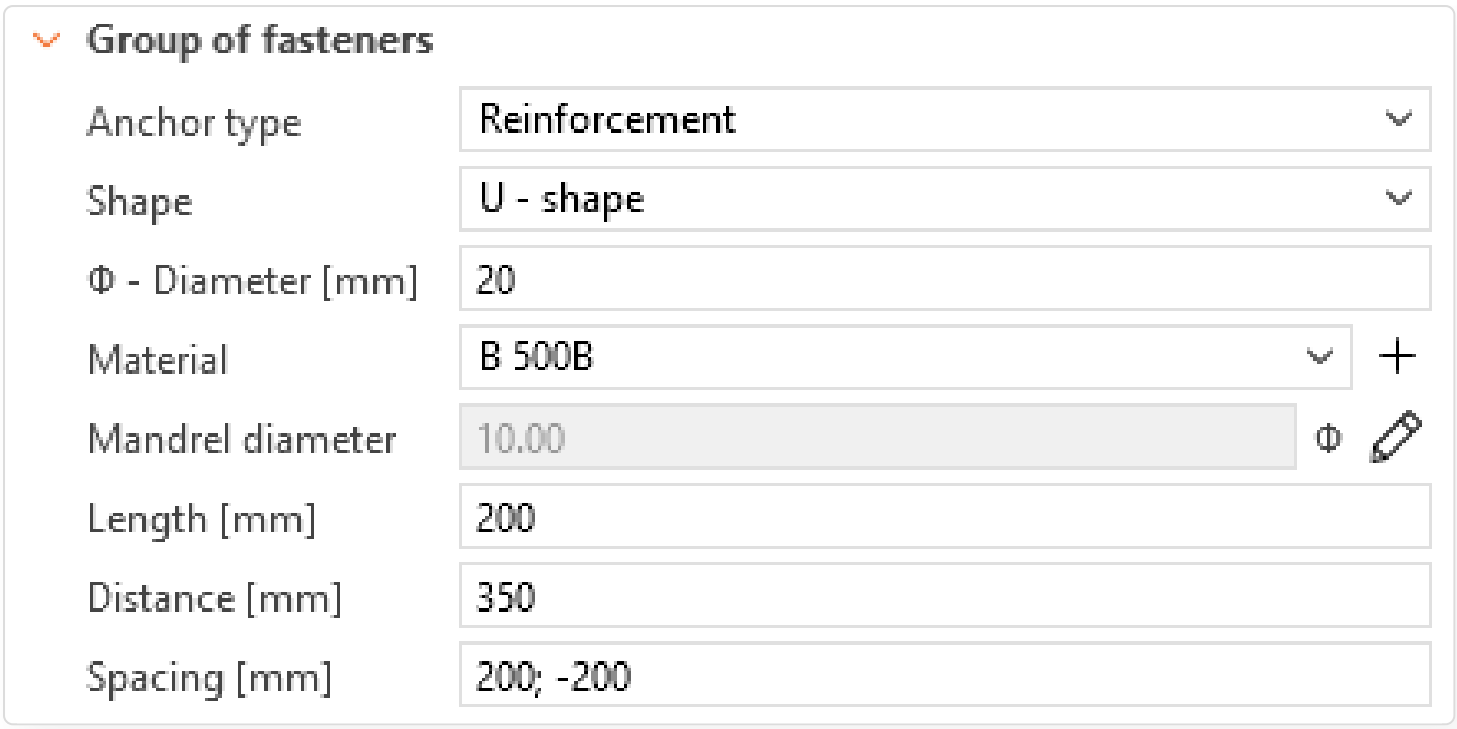
- **Straight:** Anchors can be arranged in rows and positions following a predefined key (similar to bolt placement in Connections). Users can define material, length, and diameter, as well as specify an anchorage type.



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



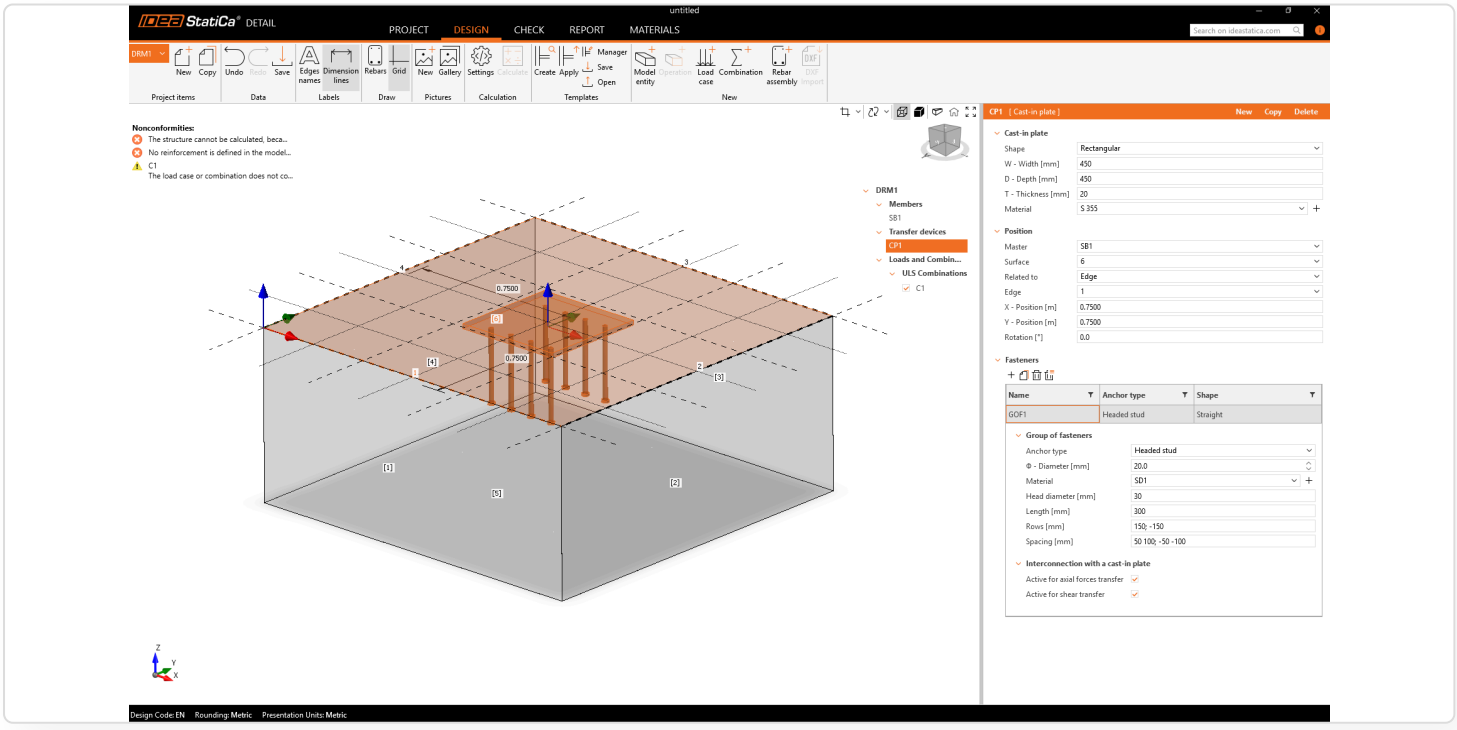
- **U-shape:** A continuous series of reinforcements forming a U-shape, connected on both sides.



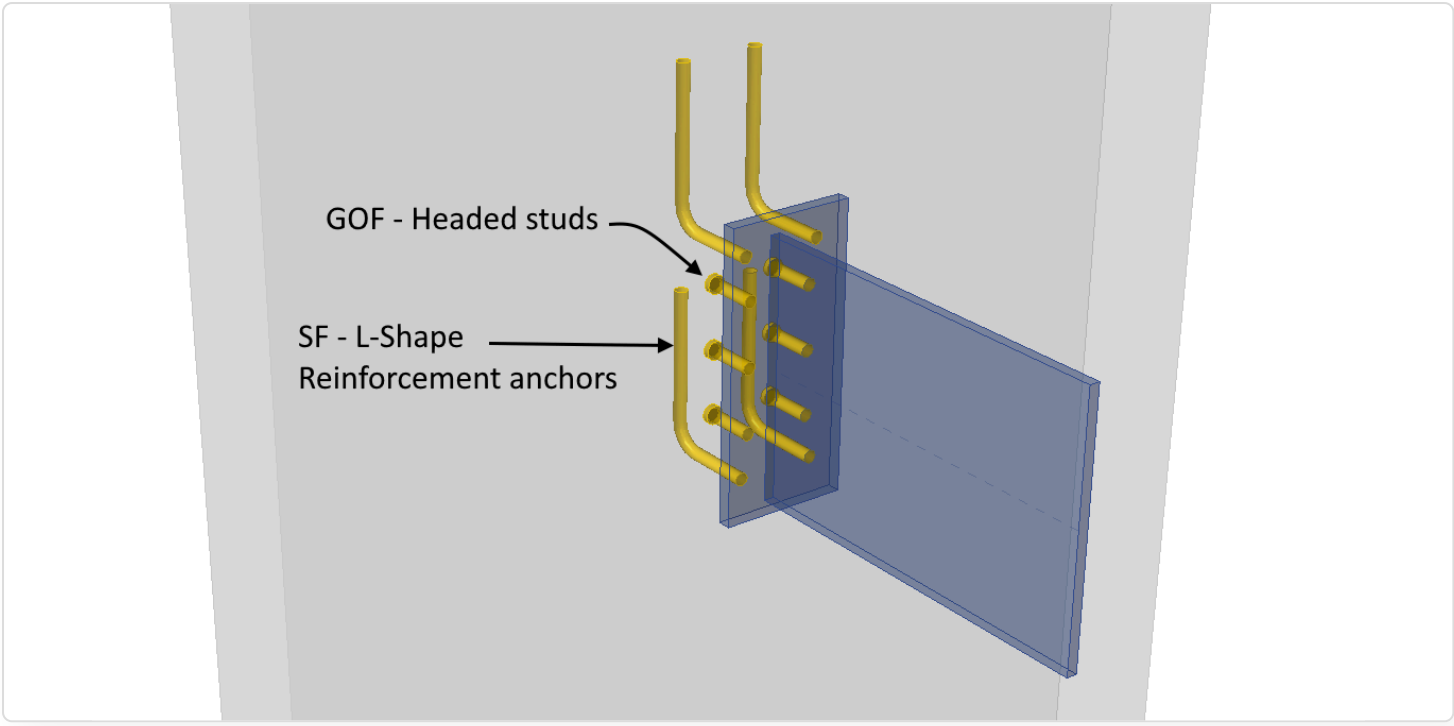
Headed stud

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

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



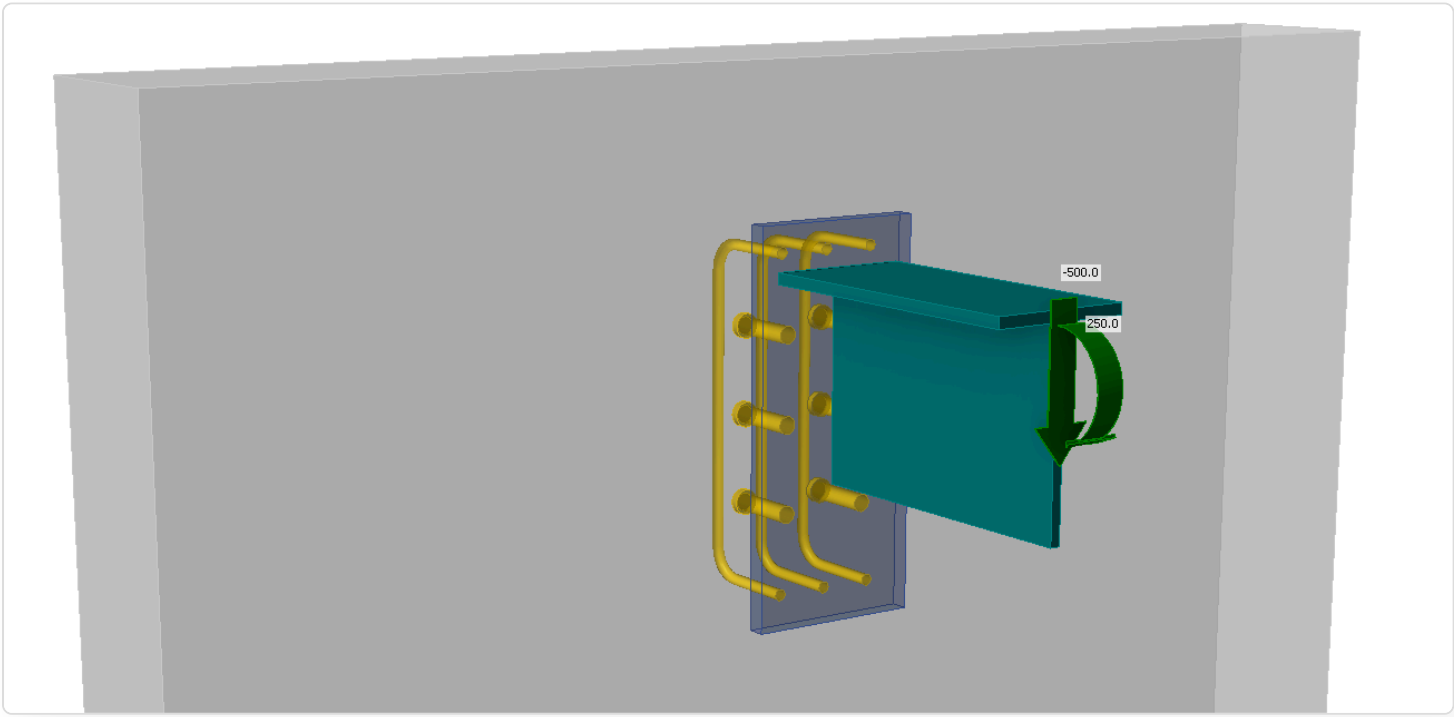
The Cast-in Plate entity supports two ways of input: Single Fastener (SF) and Group Fasteners (GOF). All the types of fasteners and input can be combined.



Approaches for specifying loads

Loads can be applied to a Cast-in Plate using two methods:

- **Direct** – Forces are applied directly to the plate.
- **Column (Stub)** – A short steel column is modeled above the plate:
  - A steel cross-section is selected from the database, specific to each design code.
  - The cross-section position relative to the plate is defined in the property grid.
  - Internal forces (Fx, Fy, Fz, Mx, My, Mz) are applied at the base of the stub and redistributed through the stub into the plate, anchors, and concrete.

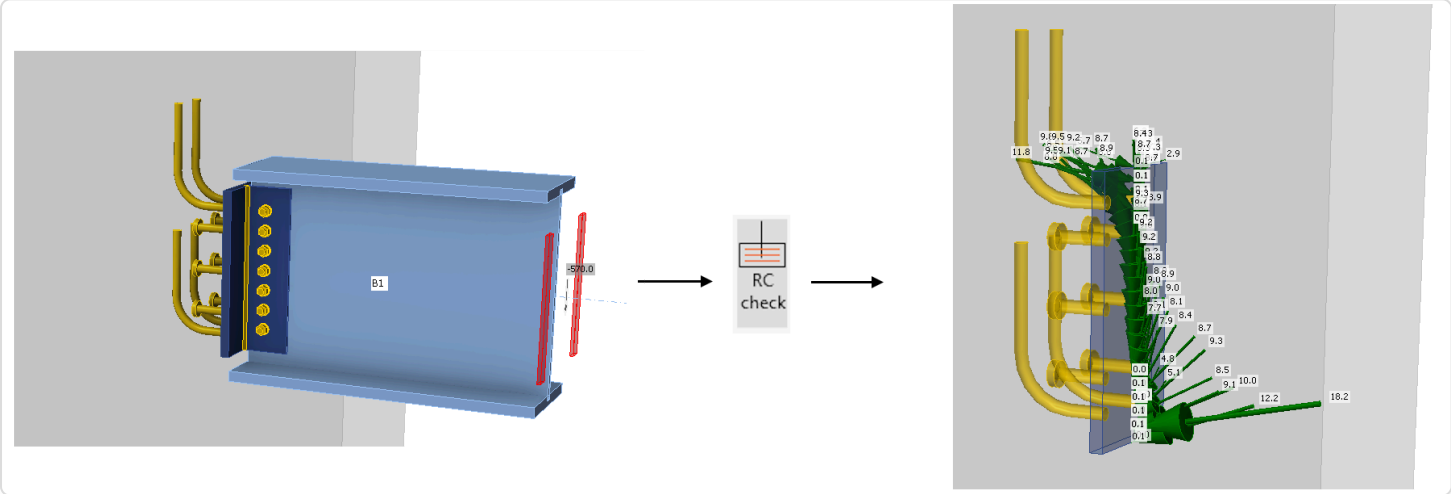


This approach provides a physically realistic transfer of column forces into the cast-in plate, consistent with the workflow already established for base plates.

Import from Connection to Detail for Cast-in plates

The export from Connection and import into Detail supports the Cast-in plate workflow:

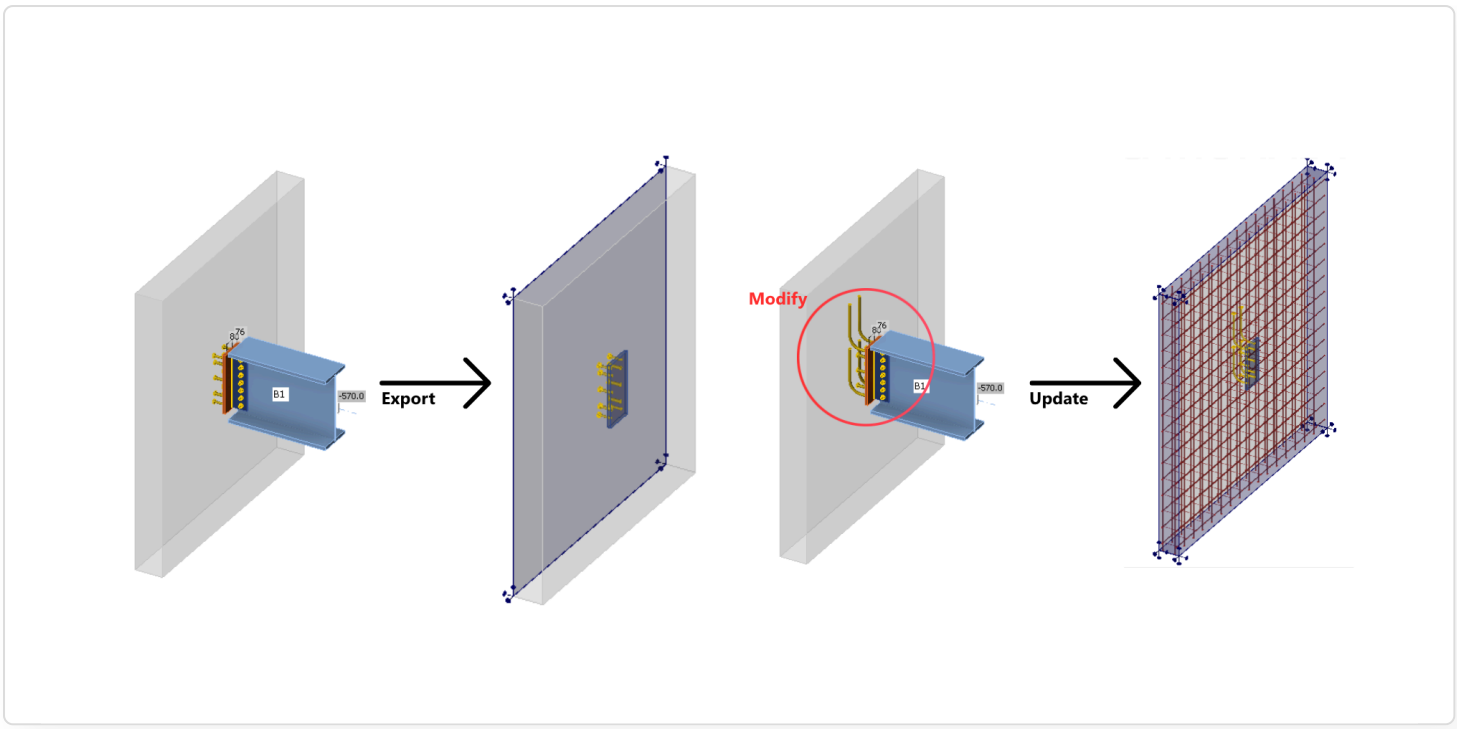
- **Plates always imported as Cast-in plates** – Any plate exported from Connection with attached headed studs or reinforcement is imported into Detail as a Cast-in Plate entity.
- **Anchors imported as Single Fasteners (SF)** – All anchors, including studs and reinforcement anchors, are imported as SF entities on the Cast-in plate to ensure a one-to-one mapping from Connection to Detail. Grouped fasteners (GOF) are not used for this workflow.
- **Load import – weld forces only** – Only weld forces are imported from Connection. Anchors in the imported model are always connected in both axial and shear. This allows the user in Detail to independently define anchor behavior, for example:
  - Reinforcement anchors active only in axial direction
  - Headed studs active only in shear



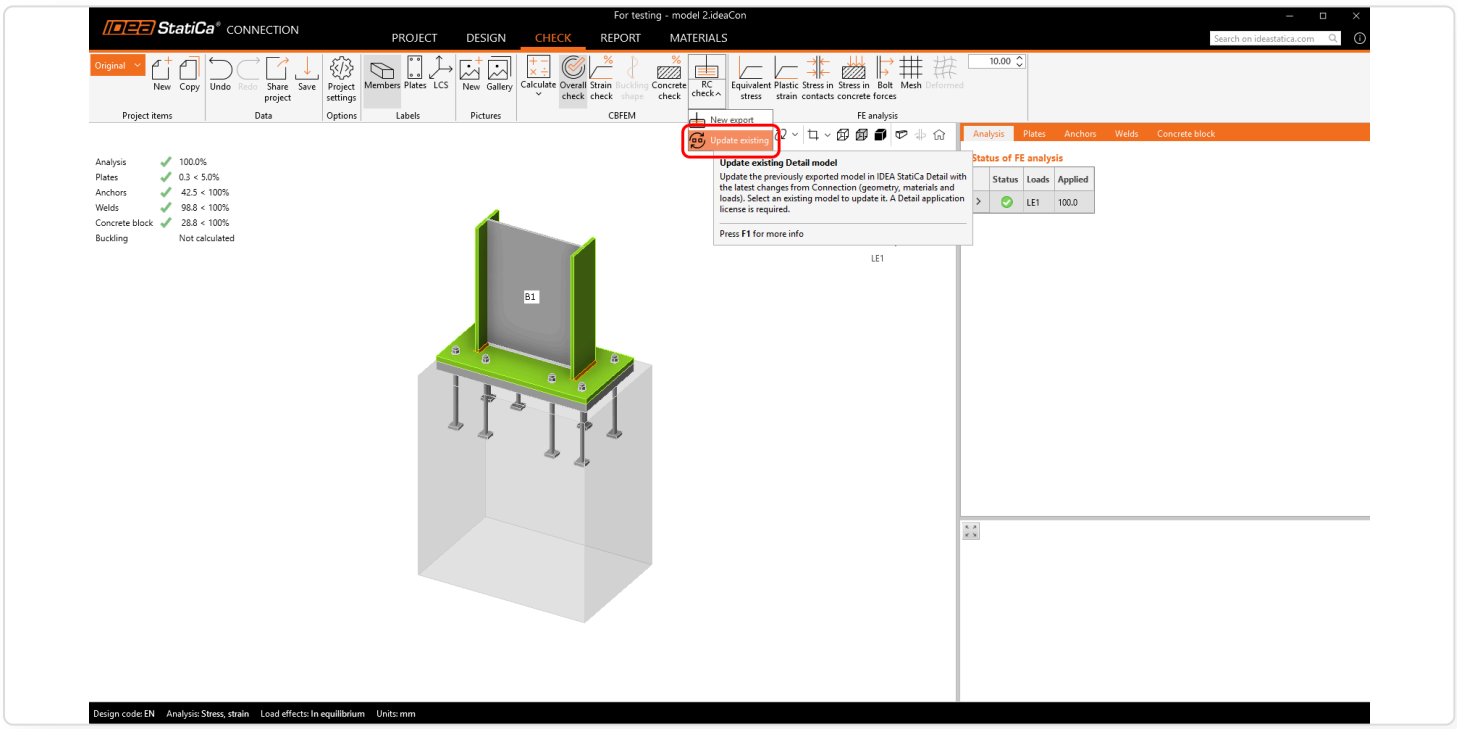
In Connection, anchors are always connected in both axial and shear, so separation of loads cannot be exported as with base plates. The “weld-forces-only” import preserves flexibility in Detail while maintaining Connection behavior. This import method is specifically designed for the Cast-in plate workflow and supports SCI P416-type approaches, enabling controlled load sharing and redistribution among different anchoring components.

One-way sync from Connection to Detail

Skip the hassle of rebuilding your model from scratch. With the “Update Existing” button in the Connection application, you can instantly sync your Detail project with the latest Connection data, saving time and keeping everything up to date without having to start over.



The Connection application provides an **"Update Existing"** function to synchronize a Detail project with the latest Connection data, eliminating the need to recreate the model from scratch.



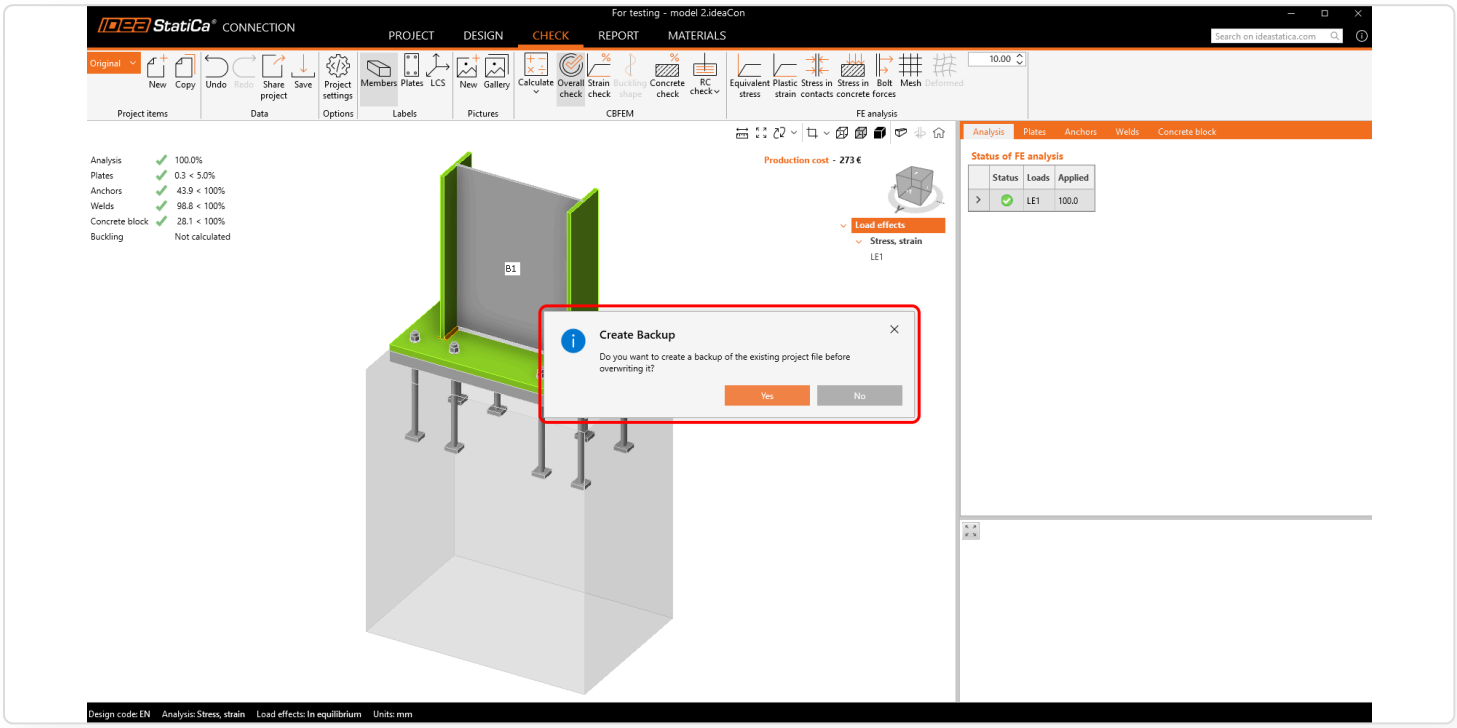
The update process synchronizes the following data:

- Concrete block: geometry and material
- Base plate / cast-in plate: geometry and material
- Anchors/fasteners: geometry and material
- Load data: load cases, impulses, and combinations

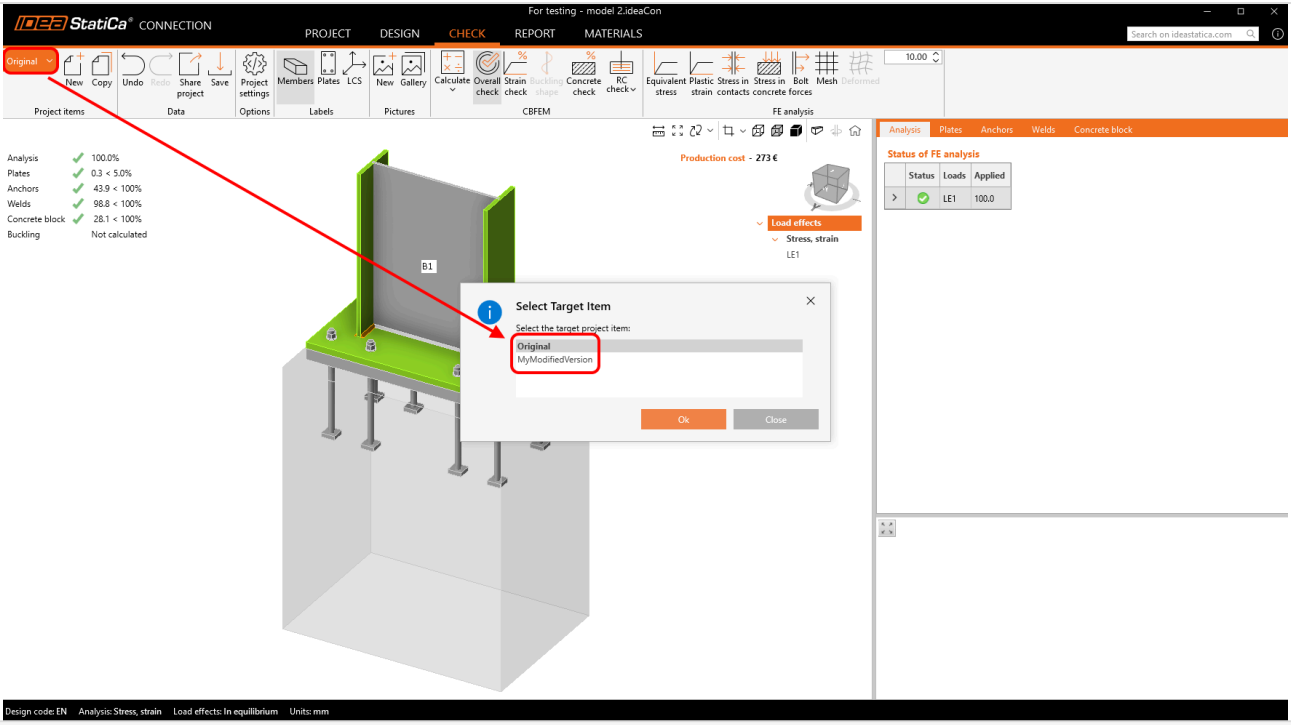
The settings are not imported/synchronized, so the code must always be set correctly.

During the update, entities originally created from Connection are handled as follows: existing entities are updated with the new data, entities no longer present in Connection are deleted, and new entities in Connection are added to the Detail project. Entities created directly in Detail remain unchanged, including negative volumes, cuts, boolean operations, reinforcement, plates, anchors, and load cases.

Before updating, the system asks to create a backup, and backups are automatically stored in the same folder to ensure recovery of the previous state.



The workflow supports multiple project items in both Connection and Detail. It is possible to copy a Connection project item to create a variant and then synchronize it with the corresponding Detail project. Updates are also supported for Detail projects containing multiple project items, allowing all relevant data to remain consistent.

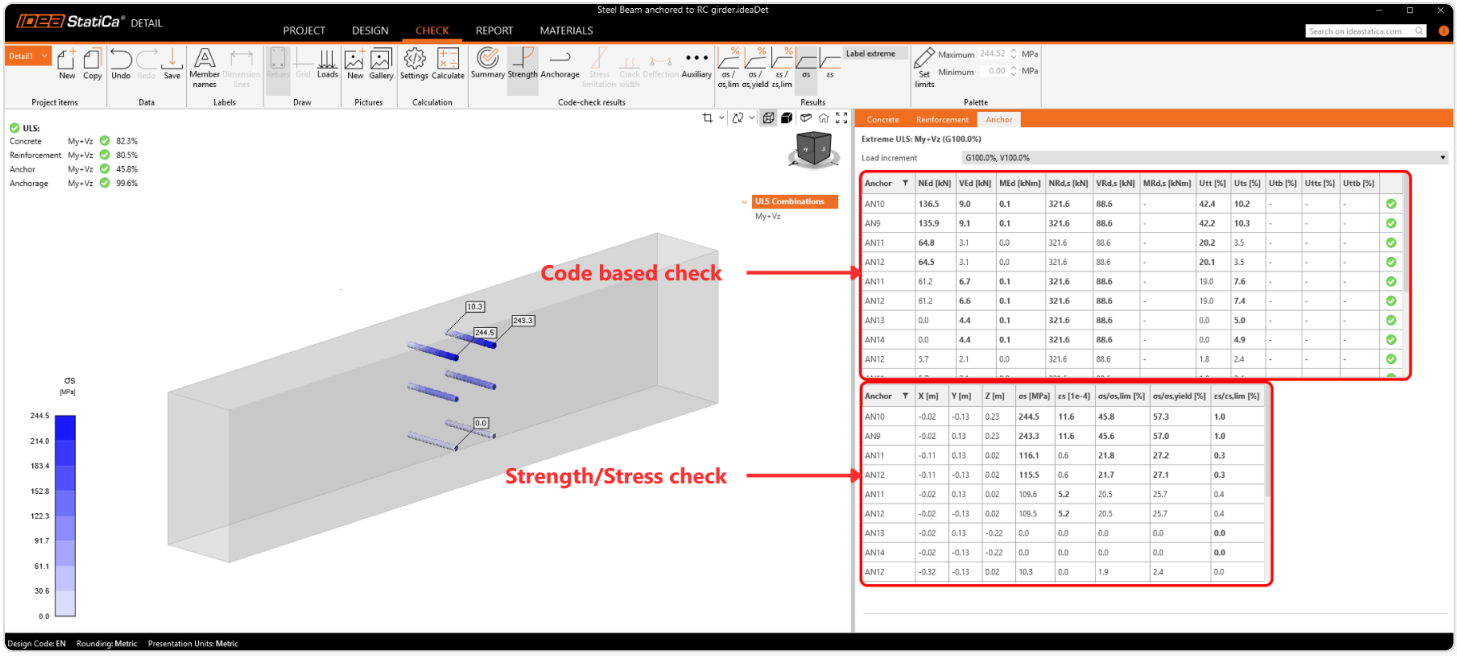


Anchor code-checks in Detail

Previously, Detail could verify the concrete block in steel-concrete connections, but anchor steel (tension, shear, and bending) was not checked, leaving a workflow gap. From version 26.0, anchor steel checks based on the codes are fully integrated into 3D Detail as part of postprocessing.

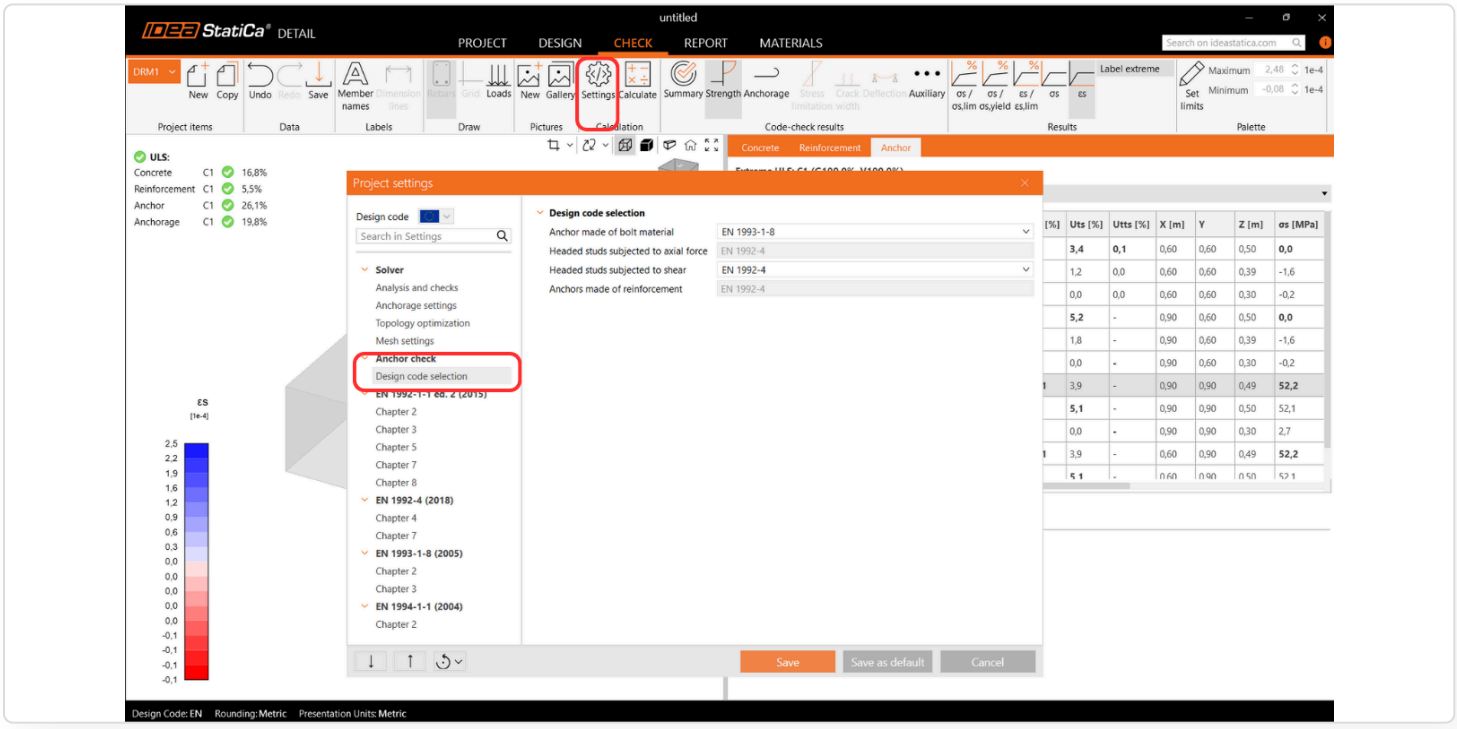
Code-checking of anchors according to the design code

In addition, we have design-code-based checks (EN, ACI/AISC, AUS), which are performed empirically in accordance with the standard. The specific standard considered can be seen in the settings, where it is also possible to select a different one depending on the type of anchorage used (base plate in direct contact with concrete, grouted base plate, and base plate with a gap), as well as the required standard based on regional practices.



Implemented codes: **EN 1992-4, EN 1993-1-8, EN 1994-1-1, ACI318-19, AISC 360-16, AS3600, AS 5216, AS 4100**

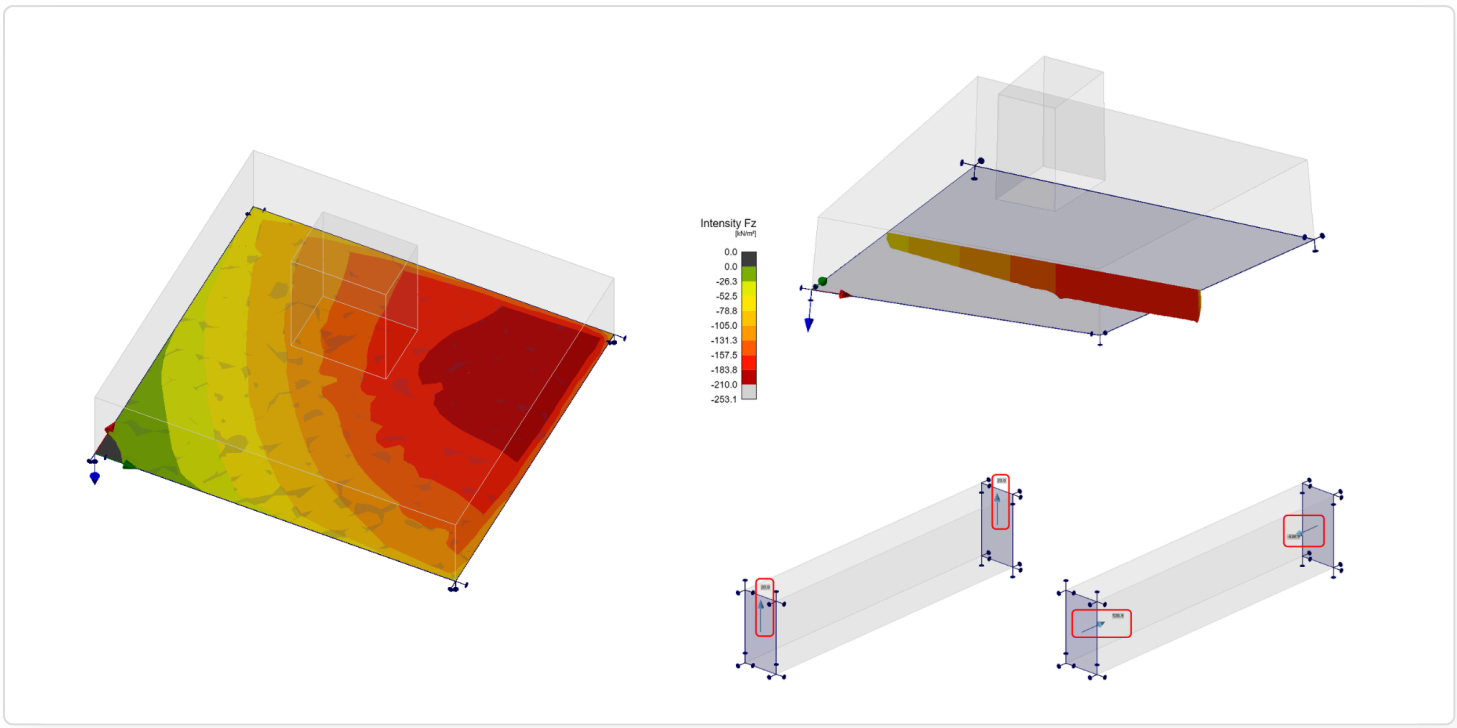
The standard settings can be changed in **Project Settings**, where the chapters will appear according to the standard selected when the project was created. When importing from Connection, It is recommended to check that the same standard is set.



In the Theoretical Background chapter - Ultimate Limit State Checks, each check is explained in detail, including all formulas used.

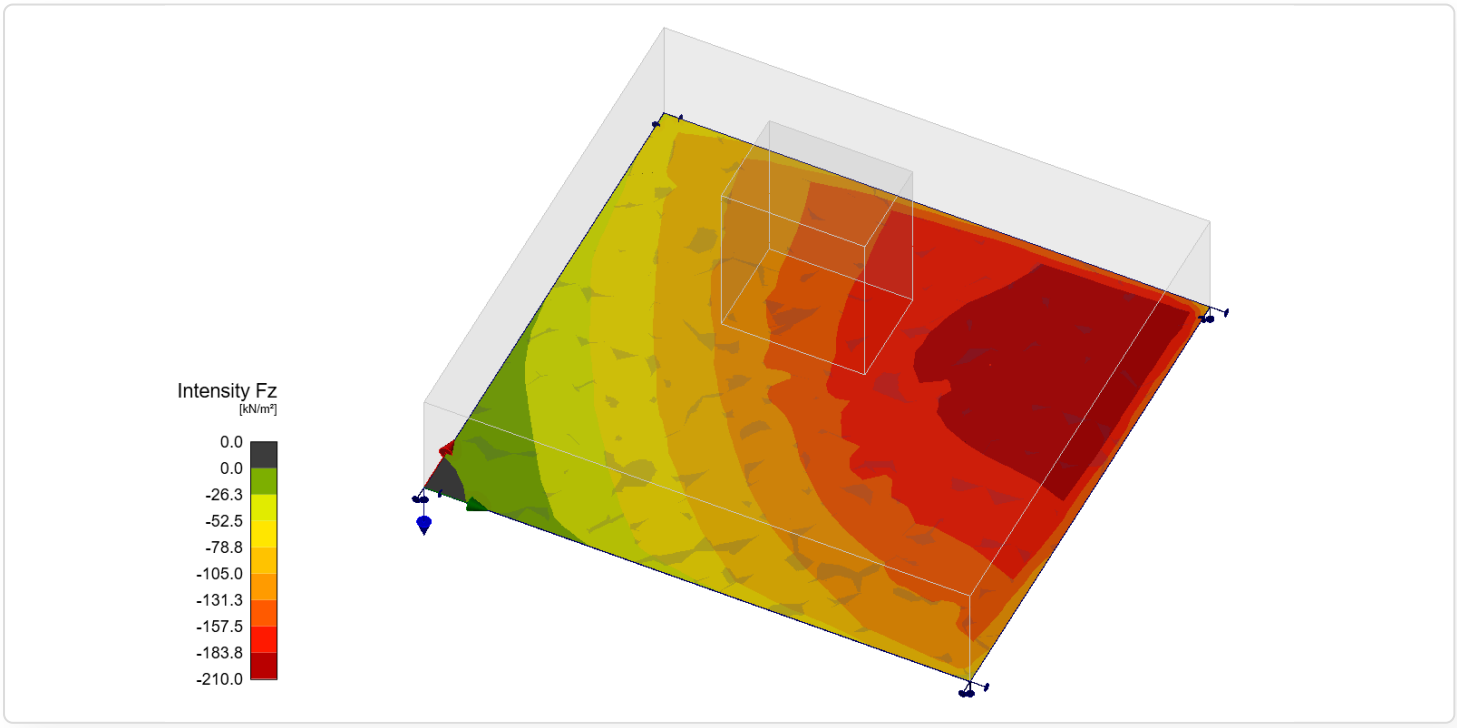
Surface support reaction presented

With the latest update, a new results visualization function has been added. The Reactions and Loads section includes a new option to display surface support reactions, allowing engineers to review them directly in two modes – Intensity and Resultant

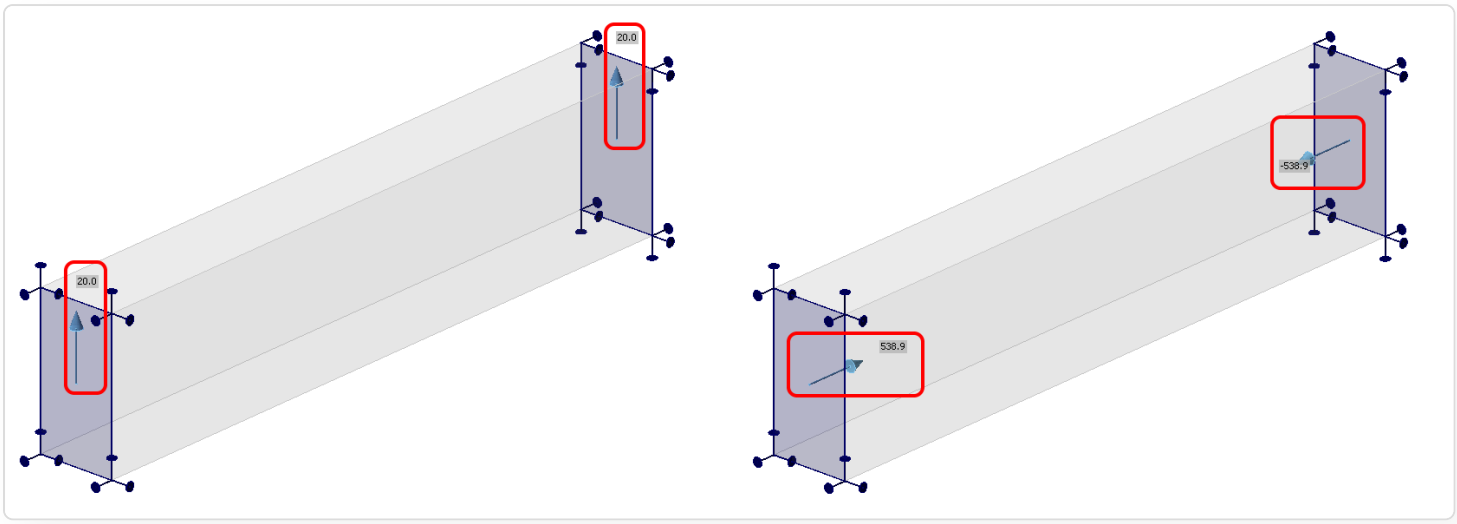


Reactions and Loads section includes an option to display **surface support reactions**. Reactions can be viewed in two modes:

- **Intensity** – Surface reactions are shown on the supported face of the concrete block using isobands to illustrate distribution over the support area.

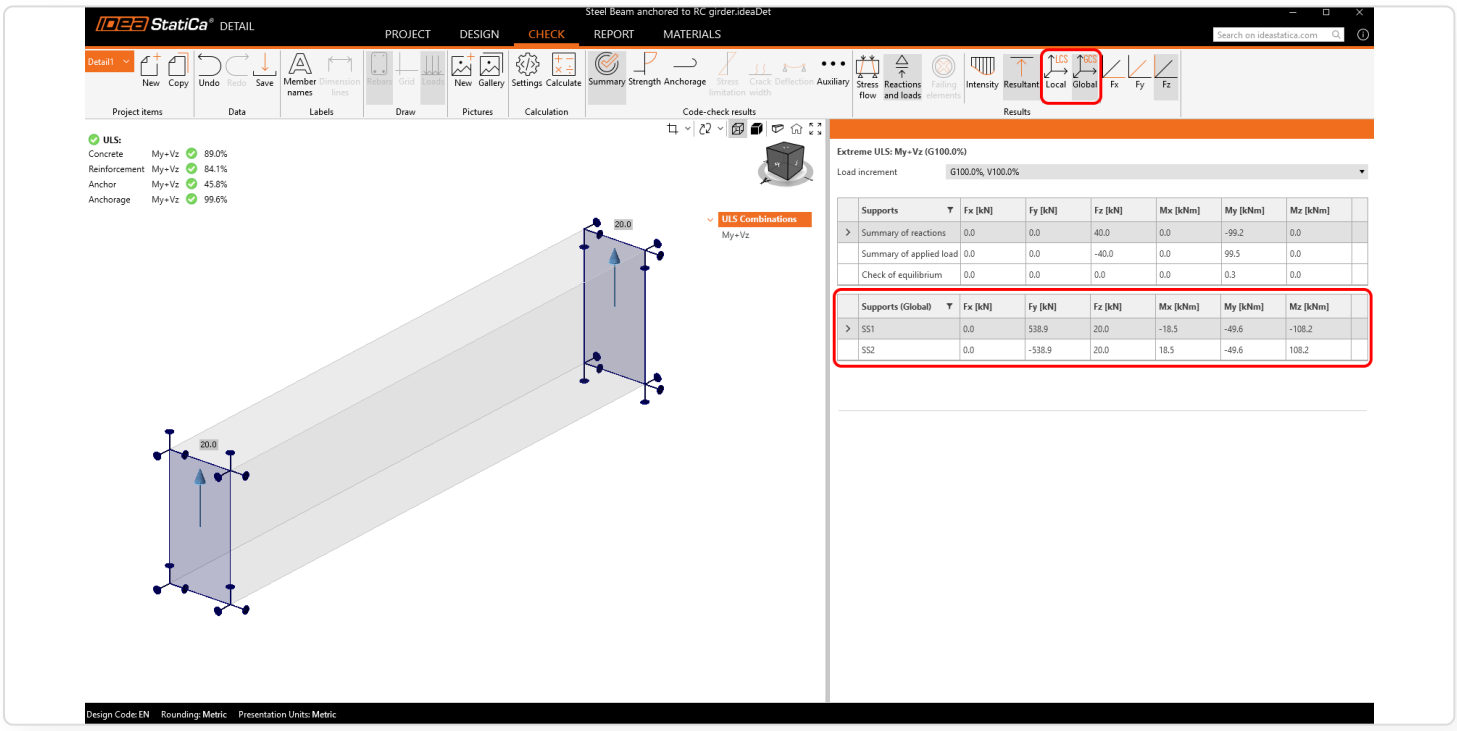


- **Resultant** – The resultant reaction for each support is displayed as an arrow at the support's center of gravity, indicating magnitude and direction.

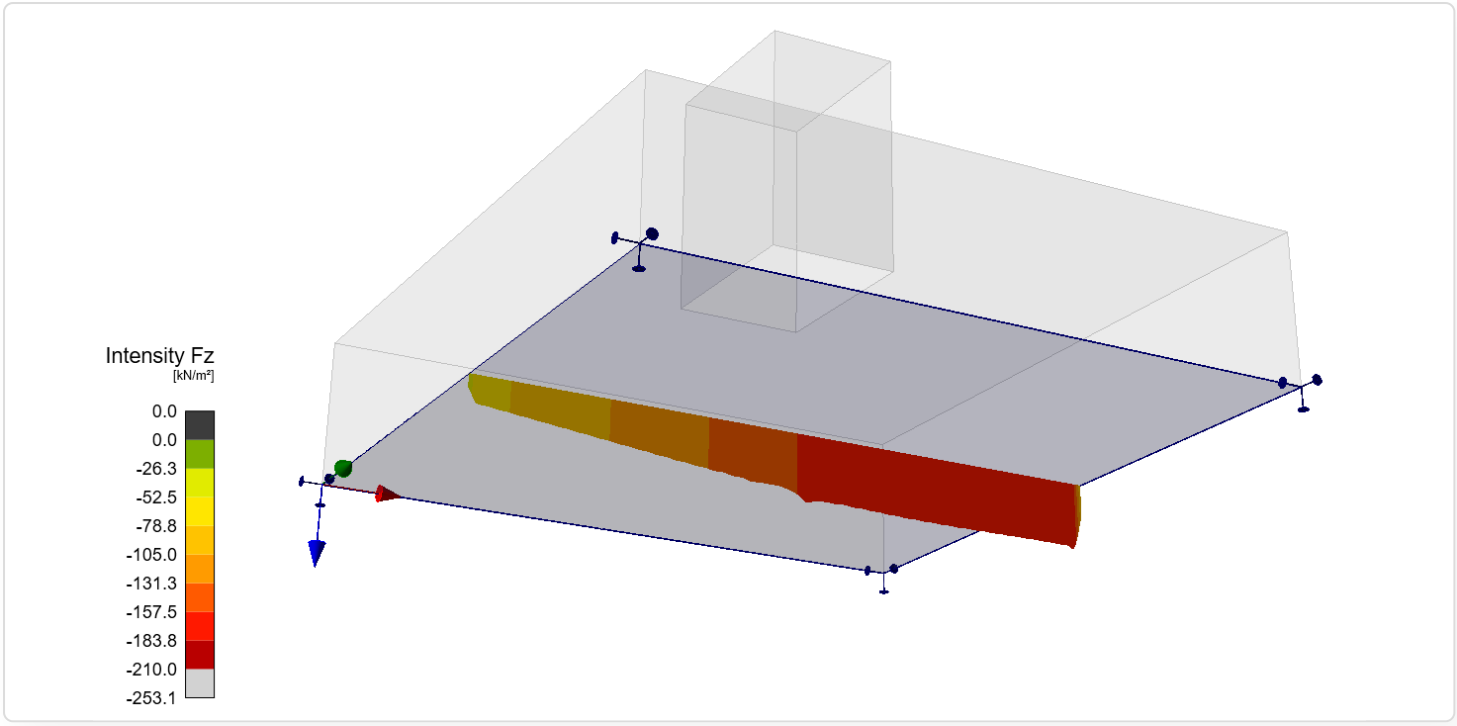


For both modes, reactions can be displayed in either the **Global Coordinate System (GCS)** or the **Local Coordinate System (LCS)** of the support.

A new table in the **Property Grid** lists summarized reactions for individual supports, also available in global or local coordinates.

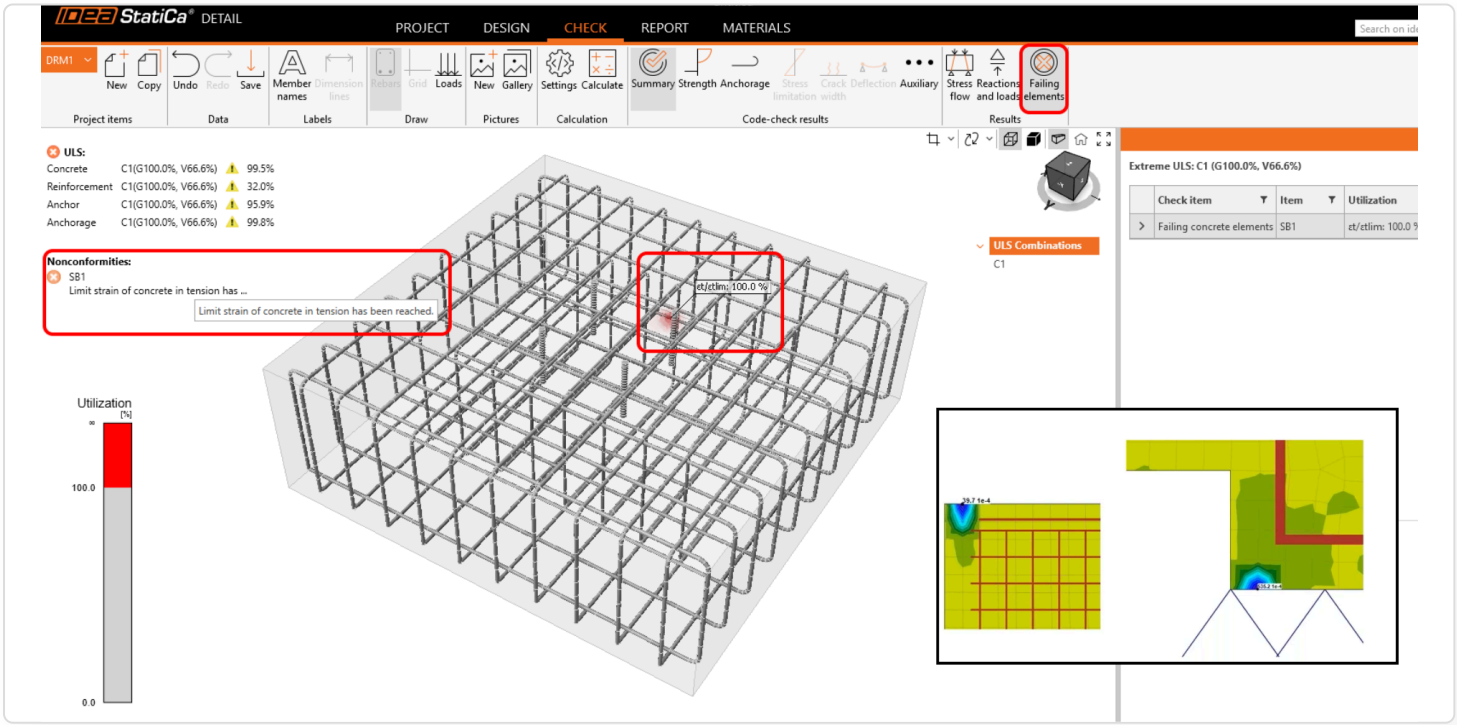


Additionally, reaction distribution can be visualized in section views created by the user.



Failing elements identification and warnings in Detail

In case of an incomplete nonlinear analysis, it could be time-consuming to determine what was wrong with the model and sometimes, purely a matter of guesswork. In 26.0, when an analysis stops due to a criterion being reached (before 100% load is applied), Detail provides clear information about what happened and where. The exact criterion is named in a warning message, and the critical location is highlighted directly in the scene.



Unsatisfactory Detail results: calculation fails, analysis 0% or doesn't reach 100%

This article addresses a previous key usability issue in nonlinear analysis workflows concerning difficulty in identifying reasons why a calculation stopped prematurely. It is now possible see the causes for failure in the graphics window.

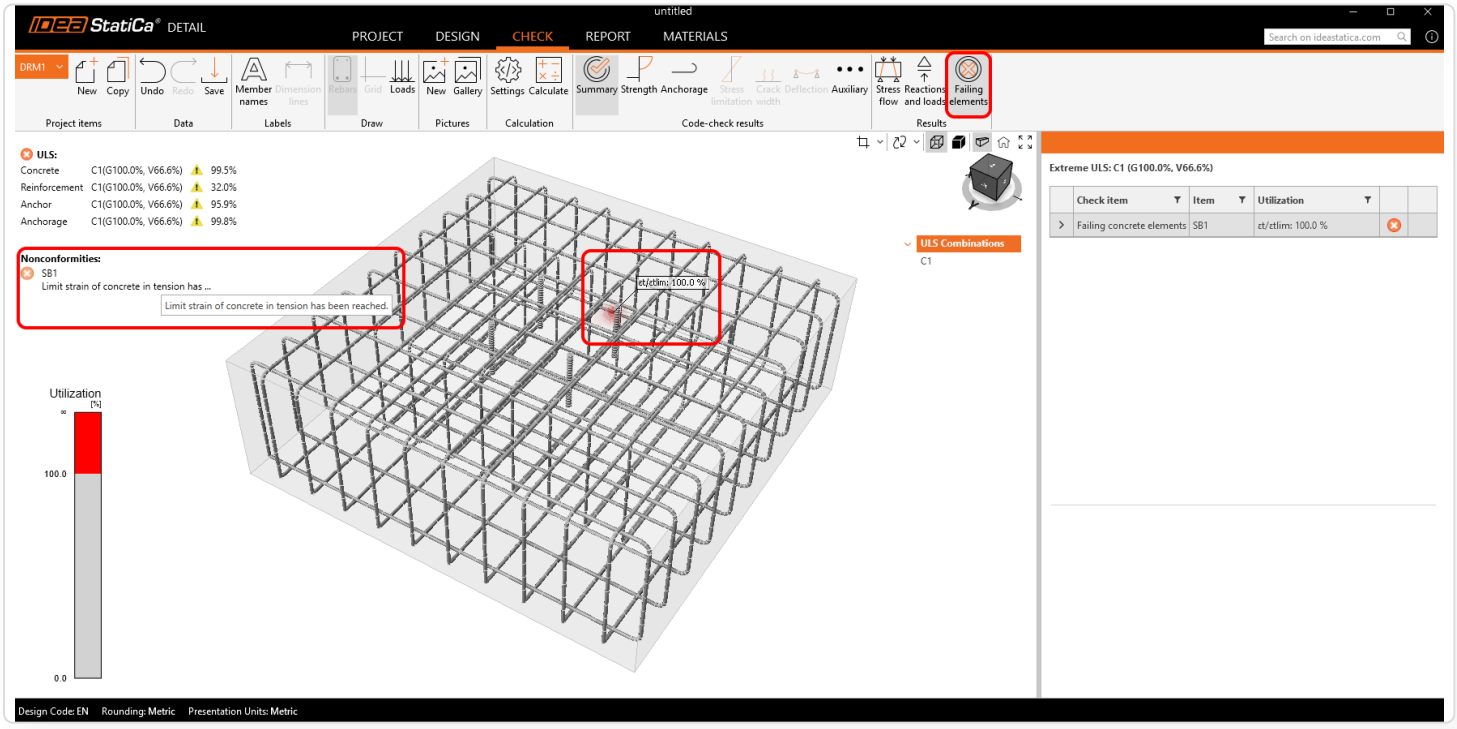
Nonlinear analysis in Detail can finish in three different ways:

- **Converged to the end** (100% of the defined load applied)
- **Stopped due to a stop criterion** (a limit was reached and the solver terminated early)
- **Stopped due to divergence** (solver could not converge)

The general principles of reading results are described in the Results articles, see the dedicated part of the Support Center.

Stop criteria

The solver terminated early because a material limit was exceeded, either in steel or concrete. To identify the exact cause, check the **warnings** displayed by the program after the calculation. Clicking "Failing Elements" will display the specific problem visually in the graphics window.



These warnings indicate which parameter exceeded the allowable limit and where to look in the model. The reason for exceeding the model limits may simply be that the load-bearing capacity has been exceeded, in which case, the design needs to be adjusted. Alternatively, it may be due to a modeling error or a violation of detailing rules, requiring closer examination. The most common warnings are listed below.

Detail 3D (3D CSFM solid model)

- The limit strain of concrete in tension has been reached.

- The limit strain of concrete in compression has been reached.
- The limit strain of reinforcement in tension has been reached.
- The limit strain of reinforcement in compression has been reached.
- The limit strain of the anchor in tension has been reached.
- The slip between reinforcement and concrete has reached the limit value.
- The slip between anchor and concrete has reached the limit value.
- Concrete crushing at the anchor–concrete interface due to shear has reached the limit.
- The pull-out resistance of the anchor has been reached.
- The base plate deformation exceeds the allowable limit.

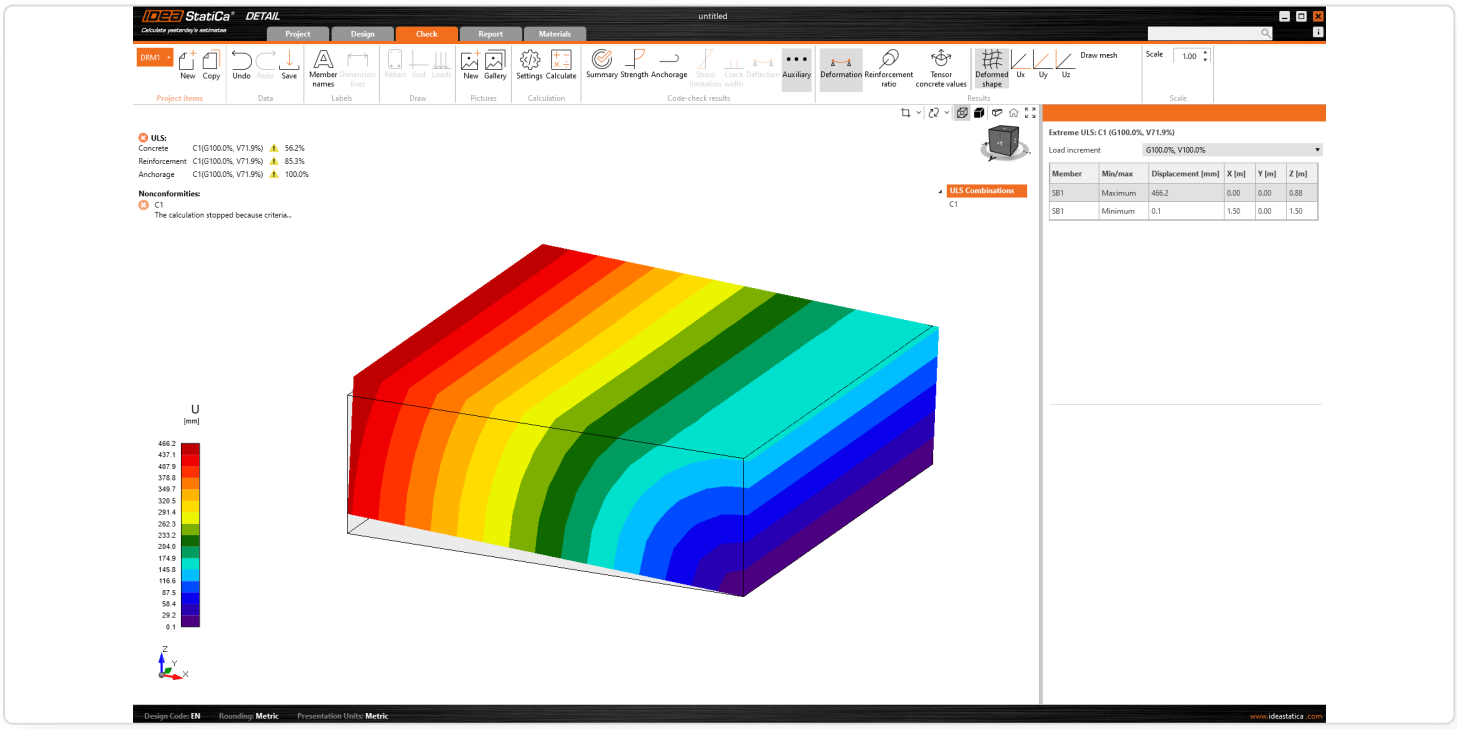
Detail 2D (CSFM shell model)

- The limit strain of concrete in tension has been reached.
- The limit strain of concrete in compression has been reached.
- The limit strain of reinforcement in tension has been reached.
- The limit strain of reinforcement in compression has been reached.
- The limit strain of prestressing reinforcement has been reached.
- The slip between reinforcement and concrete has reached the limit value.

The exact limits and the detailed operation of the analysis are described in the Theoretical Background.

Divergence

The solver was unable to converge, which means it could not find a stable solution. This can occur due to large deformations, such as overturning, incorrect mesh settings, or other modeling issues. It is recommended to check the **auxiliary** for a clearer understanding of the model's behavior.



Examples

Specific examples and a more detailed explanation are provided in these webinars:

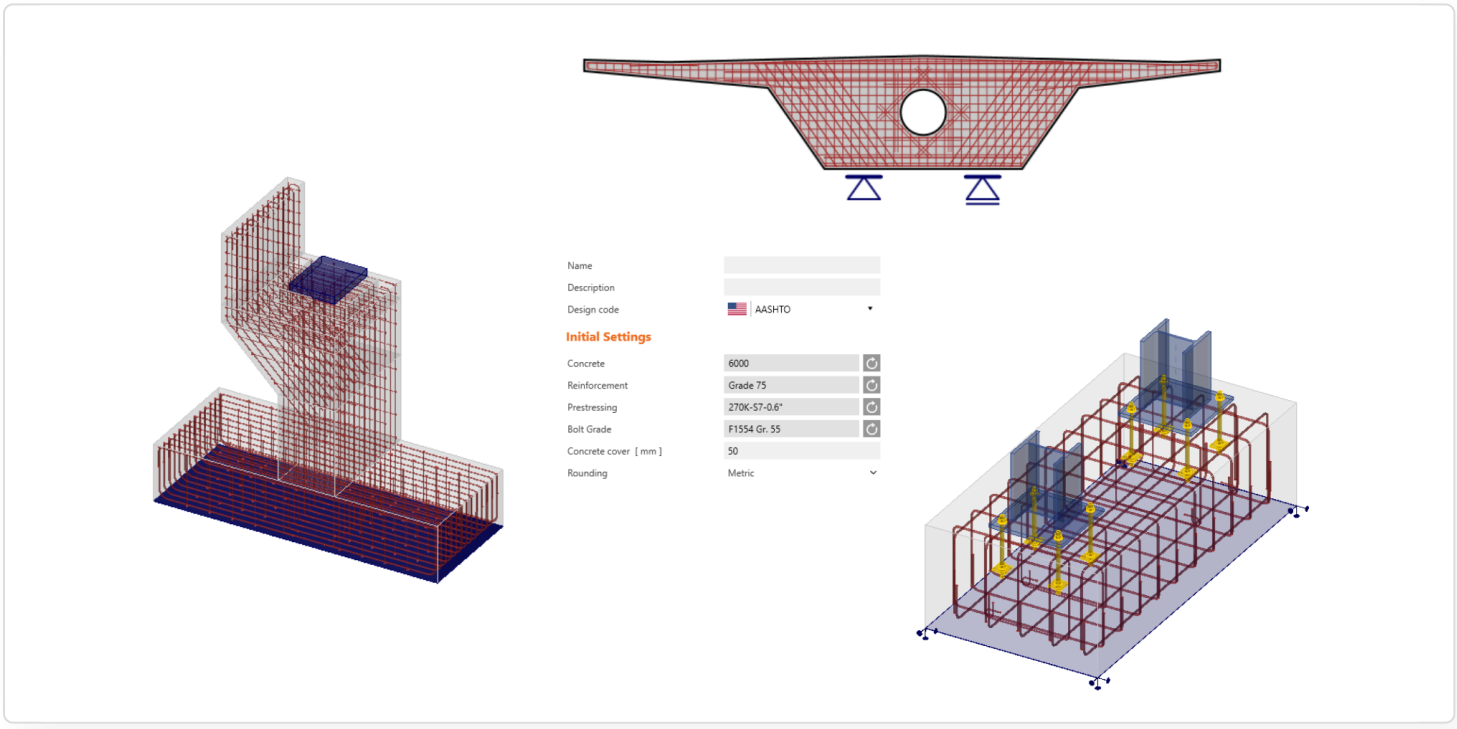
Detail 3D

Detail 2D

AASHTO in Detail

Version 26.0 introduces a native AASHTO design code in Detail to remove a limitation for infrastructural engineering projects. The implementation provides a complete, code-aligned workflow and includes dedicated 2D templates focused on typical bridge structural parts. For 3D, the existing template structure is maintained to ensure consistency across standards.

To fully support AASHTO design, we have added a dedicated material database covering normal- and high-strength concrete with the correct modulus of elasticity approach. Reinforcing and prestressing steel materials are implemented according to relevant ASTM standards. Materials are available in both imperial and metric units. More details are addressed in Theoretical Background under the AASHTO chapter.



Section BETA – An application for the new Eurocodes generation

IDEA StatiCa Section BETA introduces early support for **EN 1992-1-1:2023**, providing reinforced concrete ULS section design with clearly defined workflows and transparent calculations. It delivers a familiar UI with scalable application architecture, signaling the transition from legacy RCS technology to a next-generation Eurocode platform.

Find out more in the following chapter.

Section BETA – A New-Generation Application for RC Section Checks according to EN 1992-1-1:2023

IDEA StatiCa Section BETA represents a major step forward in how reinforced concrete sections are designed and verified. Instead of relying on legacy tools, it introduces a modern, scalable application built for the upcoming generation of Eurocodes.
This means earlier access to **EN 1992-1-1:2023**, clearer and more transparent calculations, and a workflow that better reflects real engineering practice.

IDEA StatiCa Section BETA is an application **for designing reinforced concrete sections** according to EN 1992-1-1:2023 (ULS). It allows defining arbitrary section shapes and reinforcement, performing finite-element-based calculations, and displaying results graphically, in tables, and as calculation equations. **The Beta version provides an early preview of future development and is primarily intended for user feedback and validation.**

The Beta version currently covers:

- RC sections
- ULS only
- EN 1992-1-1:2023
- Cross-section geometry
- Reinforcement geometry
- A material database compliant with the new code
- Core ULS checks
- Graphical, tabular, and equation-based result presentation

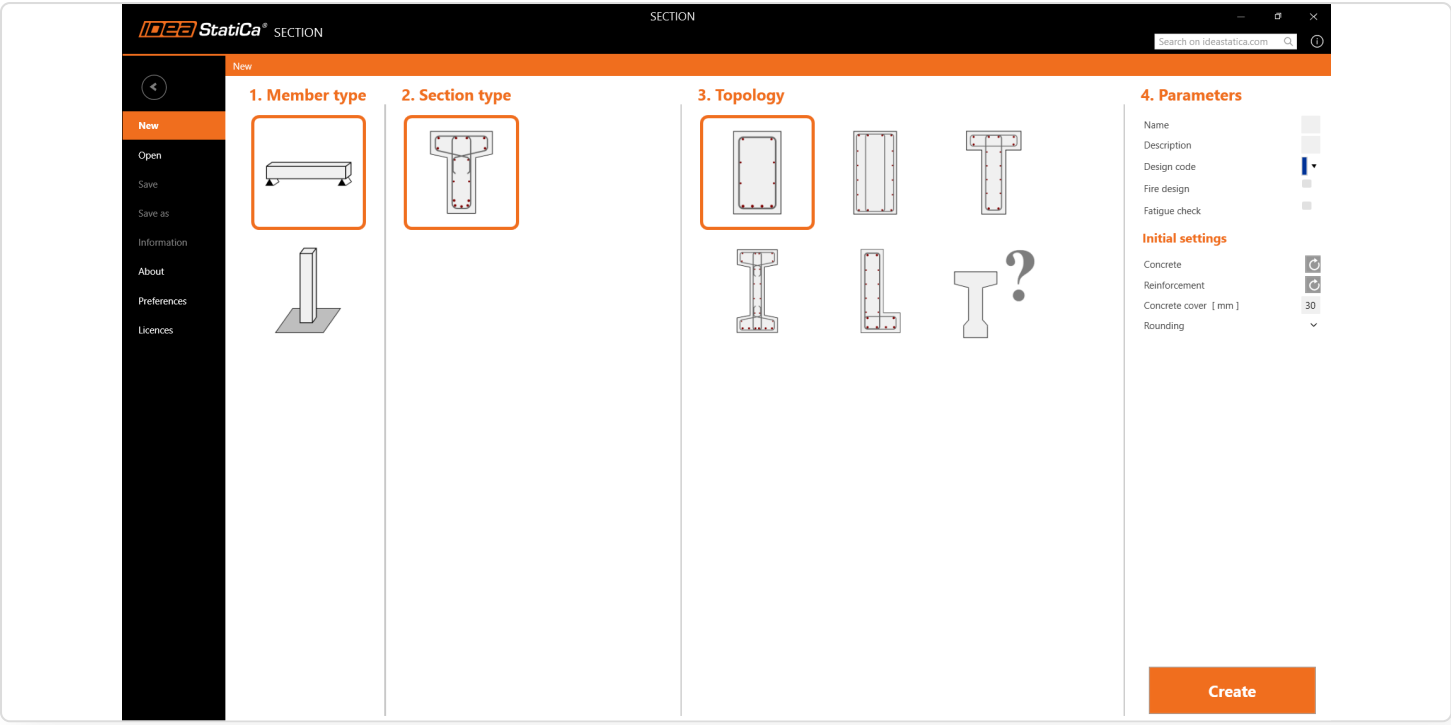
The Section application follows the style of other IDEA StatiCa applications (Connection, Detail). Its main workflow is organized into five tabs: **Project, Design, Check, Report, and Materials**.

The logic for creating cross-sections, defining reinforcement, and applying loads is the same as in the existing RCS application. For these principles, we recommend referring to RCS materials, including tutorials, etc., even though the UI differs significantly.

Compared to RCS, a major advantage is the greater insight into results, as the Section app provides all formulas, including their numerical values, making it easier to verify them. See more in the **Check** section.

Project

The Backstage view provides access to preferences, opening and saving projects, and configuring initial project data through the startup wizard. Projects can be created from templates or built from scratch.



Design

Member data and geometry

Before defining a cross-section, member data and geometry can be configured, including parameters such as exposure class, relative humidity, and member length. These values serve as a basis for future calculations, including crack width analysis and flexural slenderness checks.

Cross-section definition

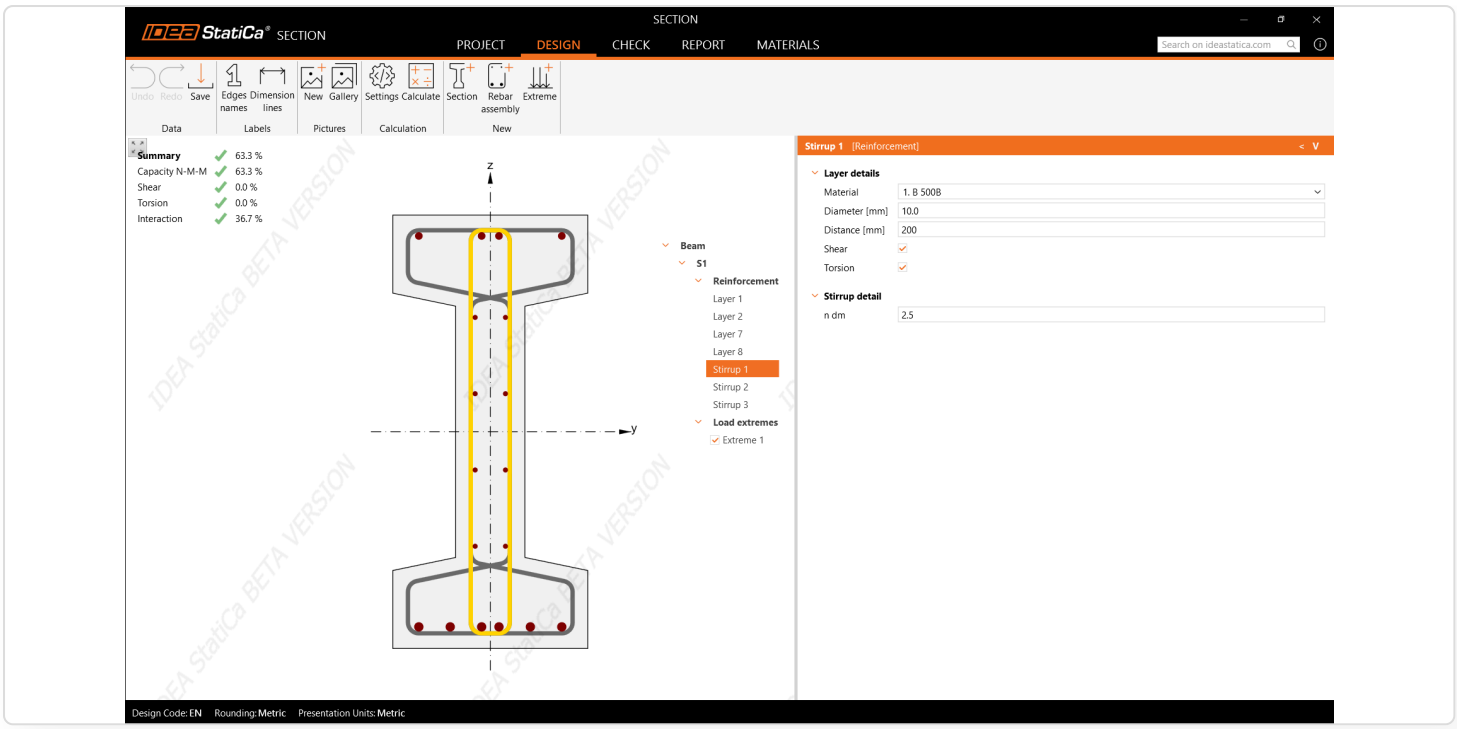
The application focuses on beam sections, supporting standard shapes like rectangles as well as arbitrary cross-section geometry, making it a general-purpose tool rather than being limited to a predefined catalog. Section properties are displayed in a read-only view, enabling immediate verification of area and other geometric characteristics within the workflow, reducing the need for external checks.

Calculation control

The "Calculation control" tab is available in the properties panel for both the entire member and individual cross-sections. Analyses can be enabled or disabled globally for the member or locally for specific sections, providing flexible control over the calculations performed.

Reinforcement definition and templates

Reinforcement is managed through reusable templates. Sections with identical geometry can share a template, while sections requiring a different layout can use a new template for independent definition. This approach minimizes repetitive input, supports families of similar sections, and allows both reuse and divergence as needed.

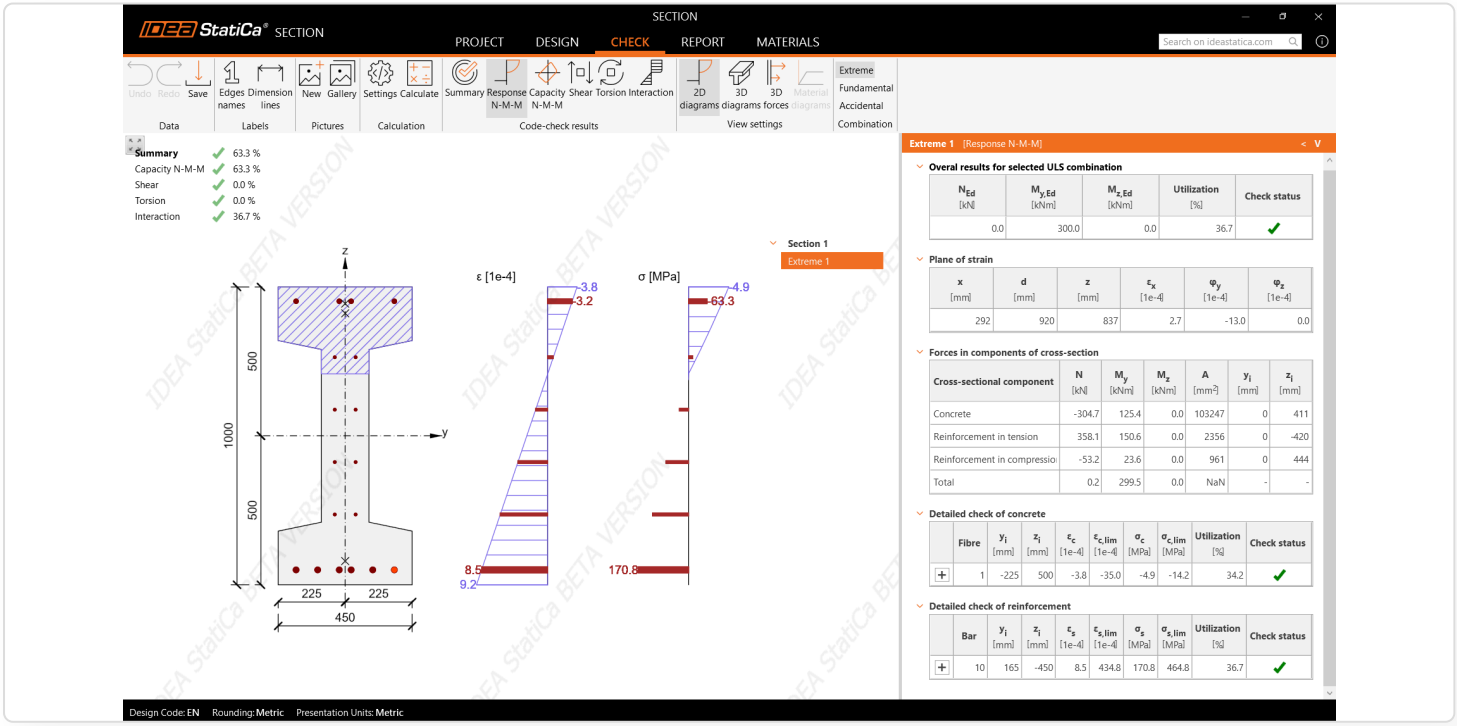


Load effects input

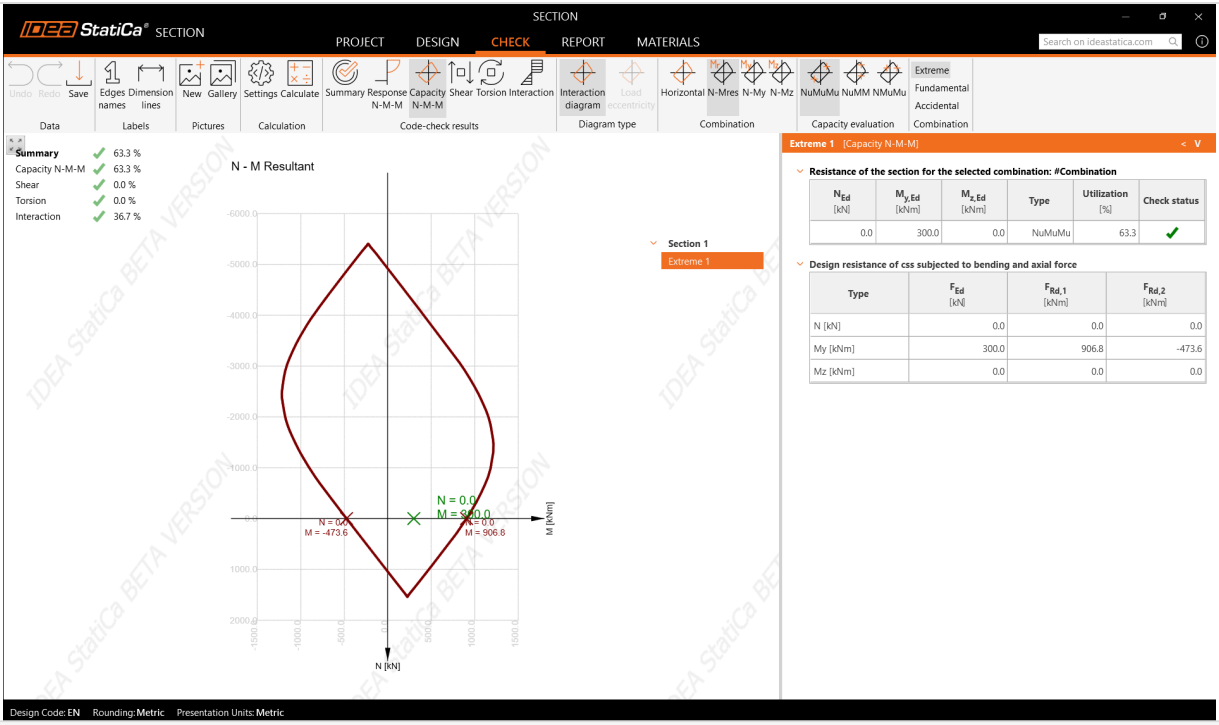
Load effects are managed through tables that support copying internal forces from Excel, pasting multiple values, adjusting and comparing extremes, and exporting data. Bulk input of load pulses is possible when "Load extremes" is selected, with extremes automatically generated from the number of rows added.

Check

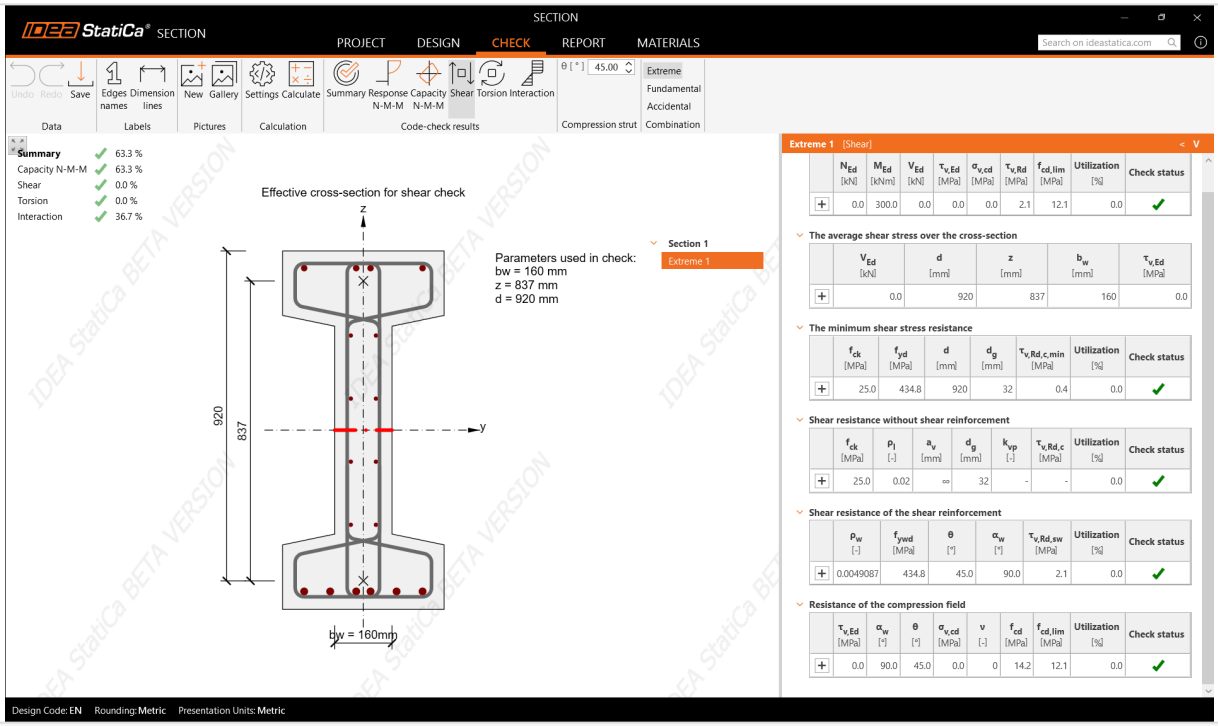
- **N-My-Mz Response**



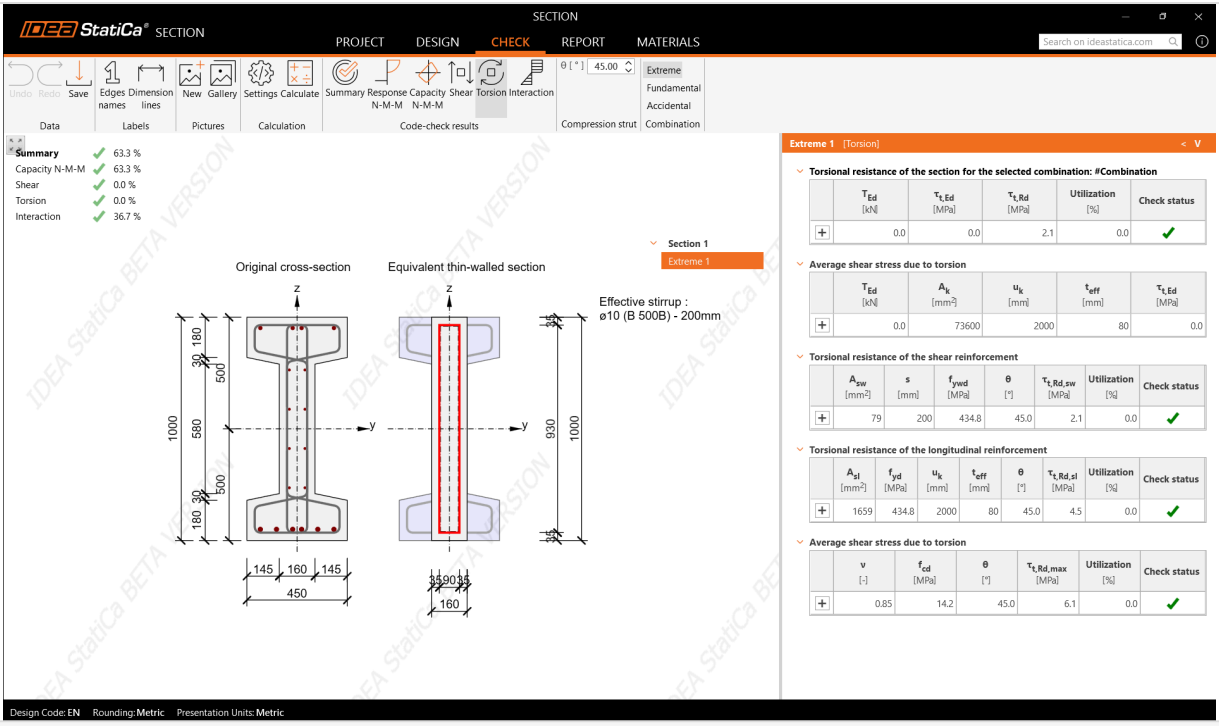
- **N-My-Mz Capacity**



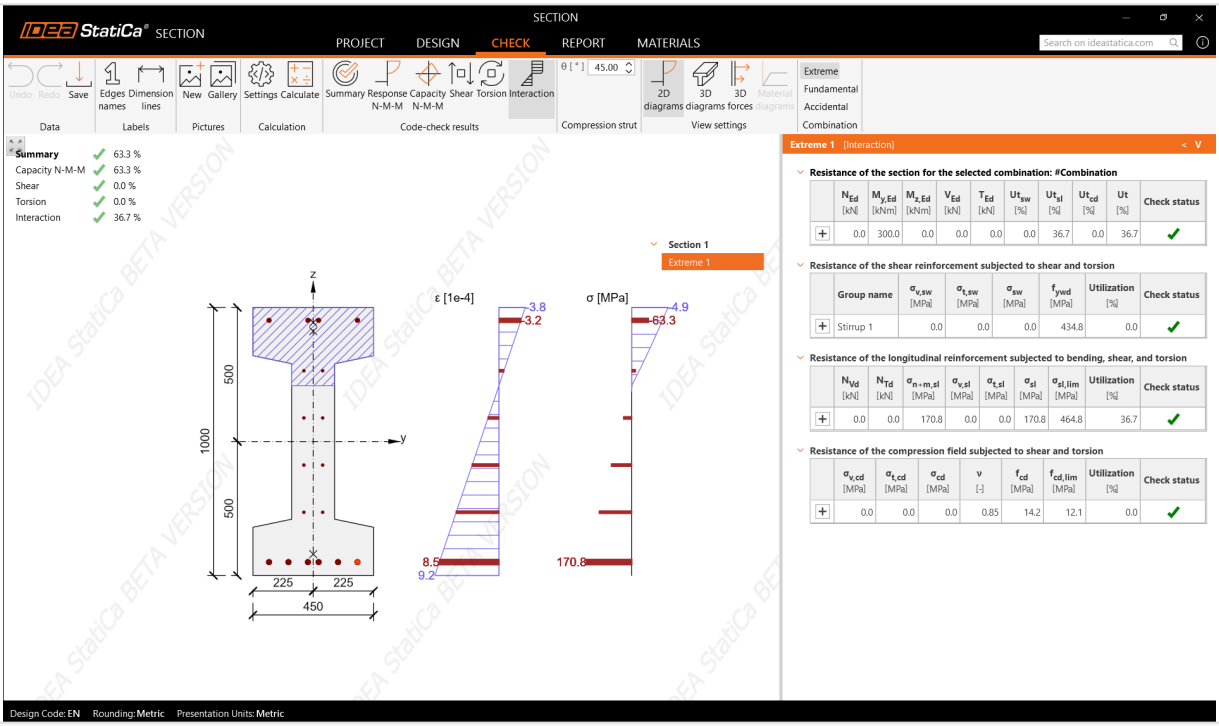
- **Shear**



- **Torsion**



- **Interaction**

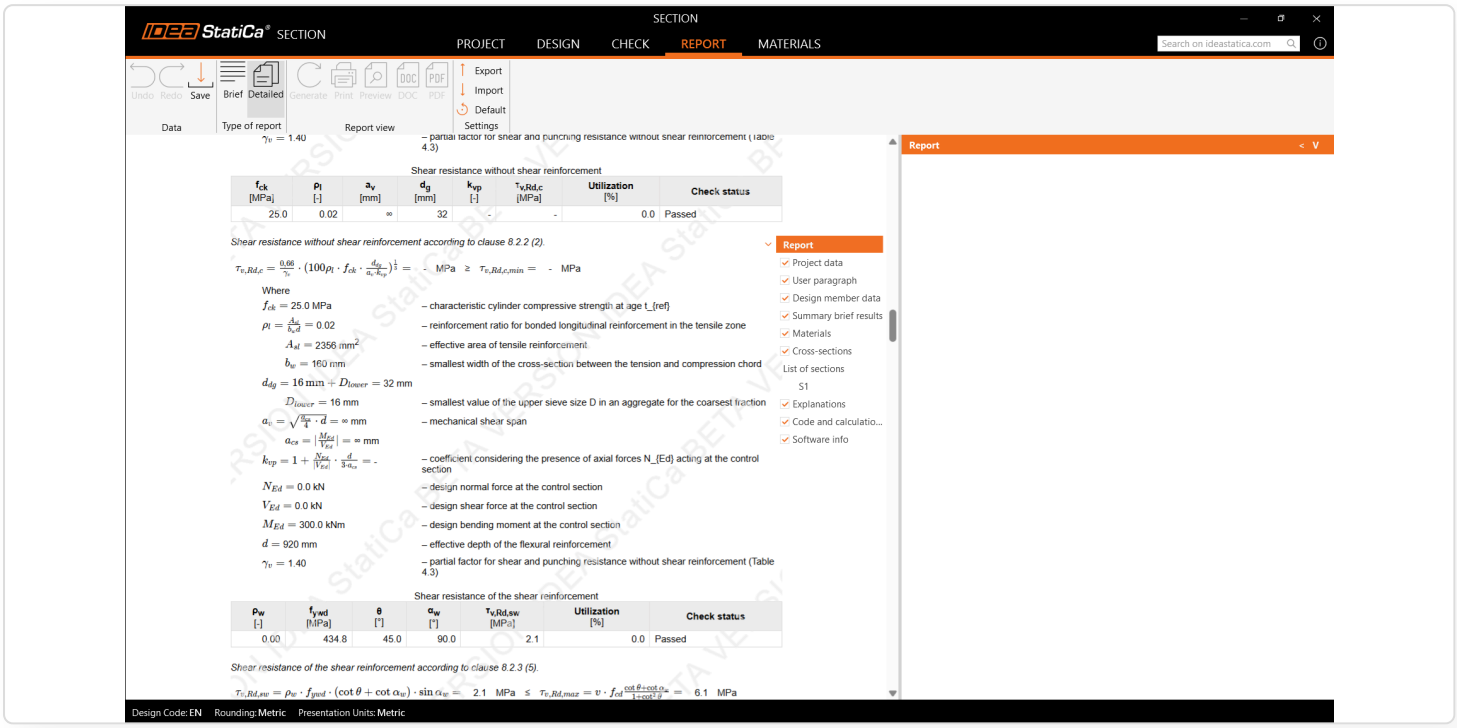


Result organization and presentation

Results are structured in a tree containing both sections and individual load extremes. Selecting an extreme displays the corresponding scene and data, while selecting a section shows the governing (worst-case) result from all associated extremes.

The application provides multiple result presentation modes, including scene-based graphical output, tables, and explicit equations. Calculations are not hidden behind summary numbers; every code check is presented as a full equation, showing the formula, input values, intermediate quantities, and relevant code references. This ensures full traceability and allows verification of the calculations directly from the displayed data.

A print preview function is available, displaying the section along with all formulas in a pop-up window, maintaining the same transparency as the interactive interface. The values shown in tables correspond exactly to those used in the governing code equations, enabling manual verification and improving clarity for advanced users, support teams, and internal review processes.



Calculations are based on finite element methods, with configurable element size for each cross-section. This provides a modern computational core rather than a simple spreadsheet-style implementation, supporting accurate, flexible, and high-resolution numerical analysis.

Report

Reports are integrated into the project and can include graphical results, tables, detailed equations, and code references.

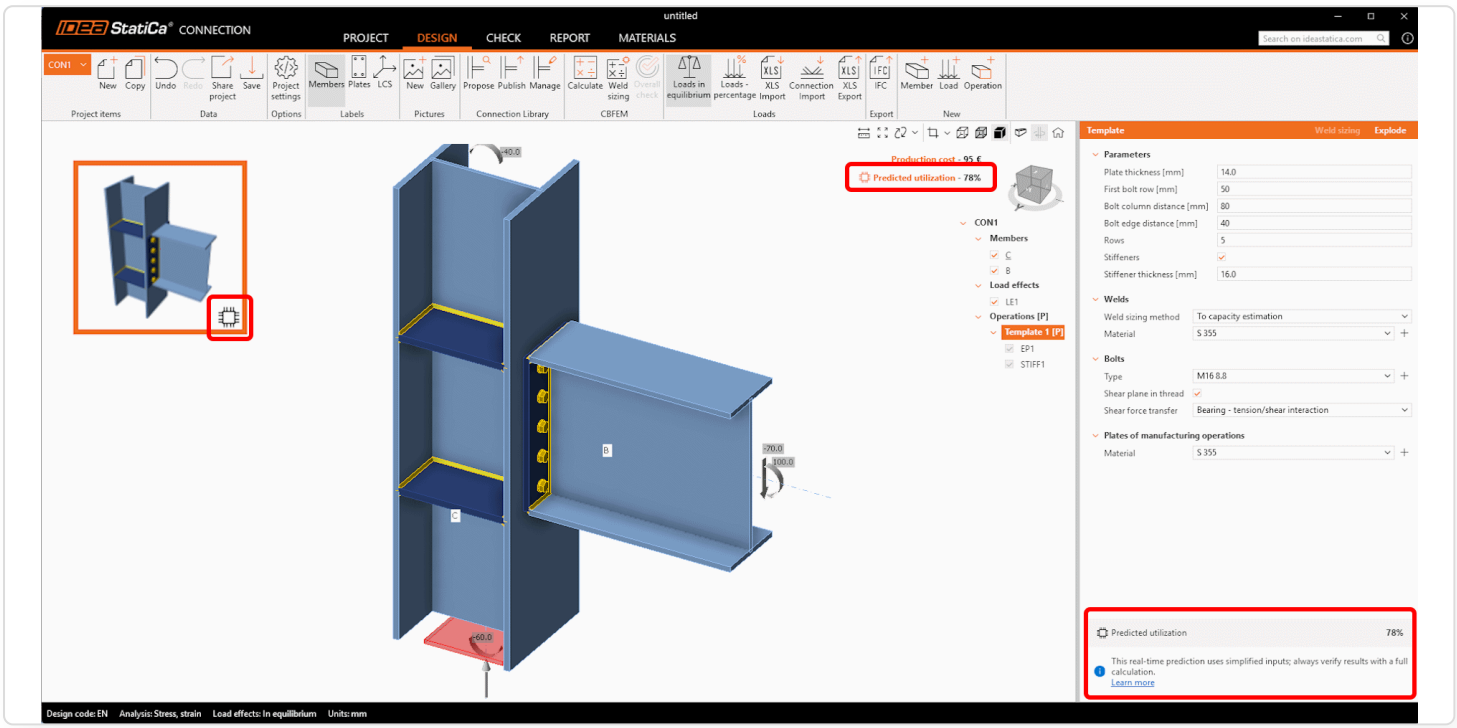
Materials

The Materials tab manages project materials, cross-sections, and reinforcement templates, consistent with IDEA StatiCa portfolio standards.

Steel Design

AI-powered prediction of connection utilization

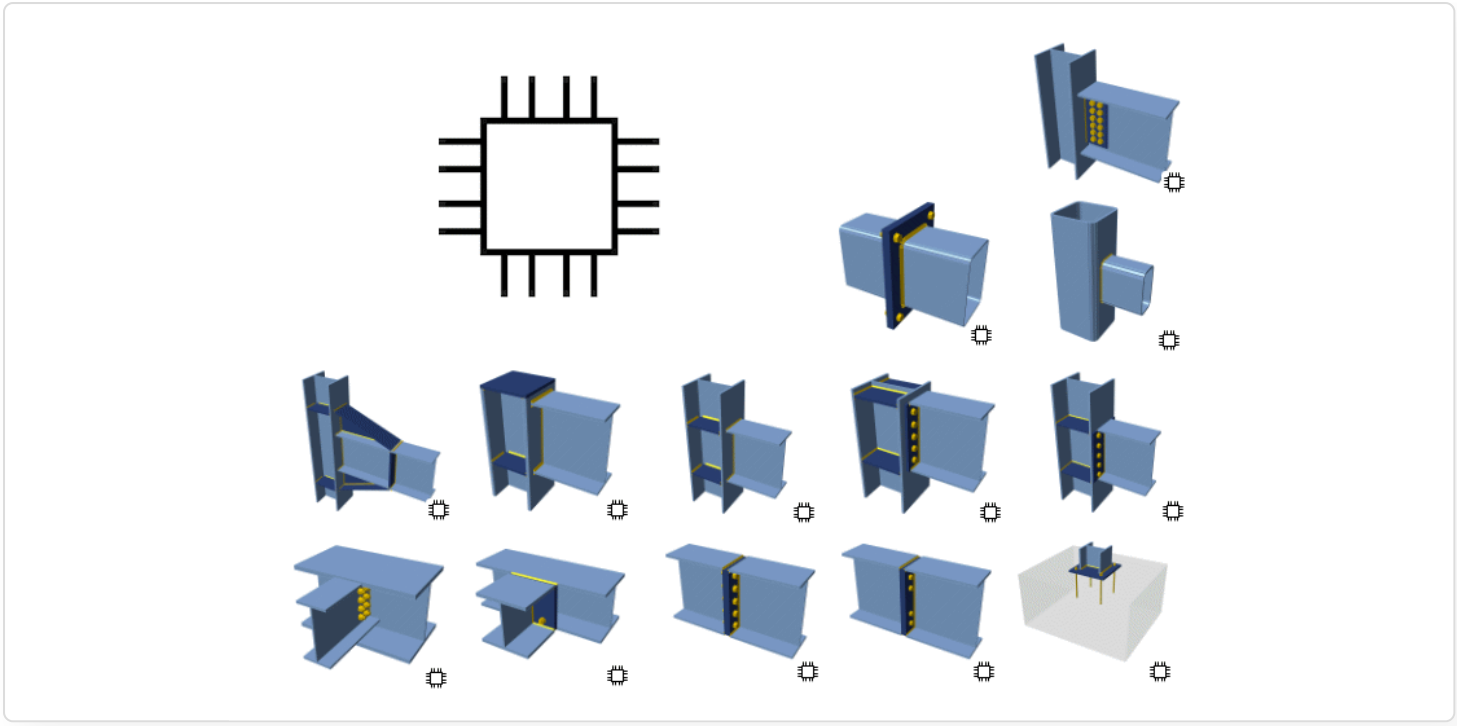
Machine-learned utilization predictions allow engineers to instantly evaluate a connection's utilization for the governing load case. This significantly accelerates the design process, as there is no need to run a full CBFEM analysis after every minor change. Powered by a dataset of thousands of trained models for each template, the tool delivers immediate utilization results across dozens of load combinations.



Utilization prediction using a machine-learned model

Instant machine-learned predictions of connection utilization are available in IDEA StatiCa Connection for the instant evaluation of a selected design.

This functionality is available for selected parametric connection templates included in the IDEA StatiCa predefined set. In version 26.0, only a limited number of templates support the prediction, but the coverage will be gradually expanded in future versions.



For supported templates, the **predicted utilization** is displayed directly in the scene, providing immediate feedback on the expected capacity of the current configuration. The displayed value represents an **estimate generated by a machine learning model.**

The prediction method

The predicted utilization is based on a machine learning model trained on a large dataset of pre-calculated connection variants. For each supported parametric template, tens of thousands of models with different parameter combinations were automatically generated.

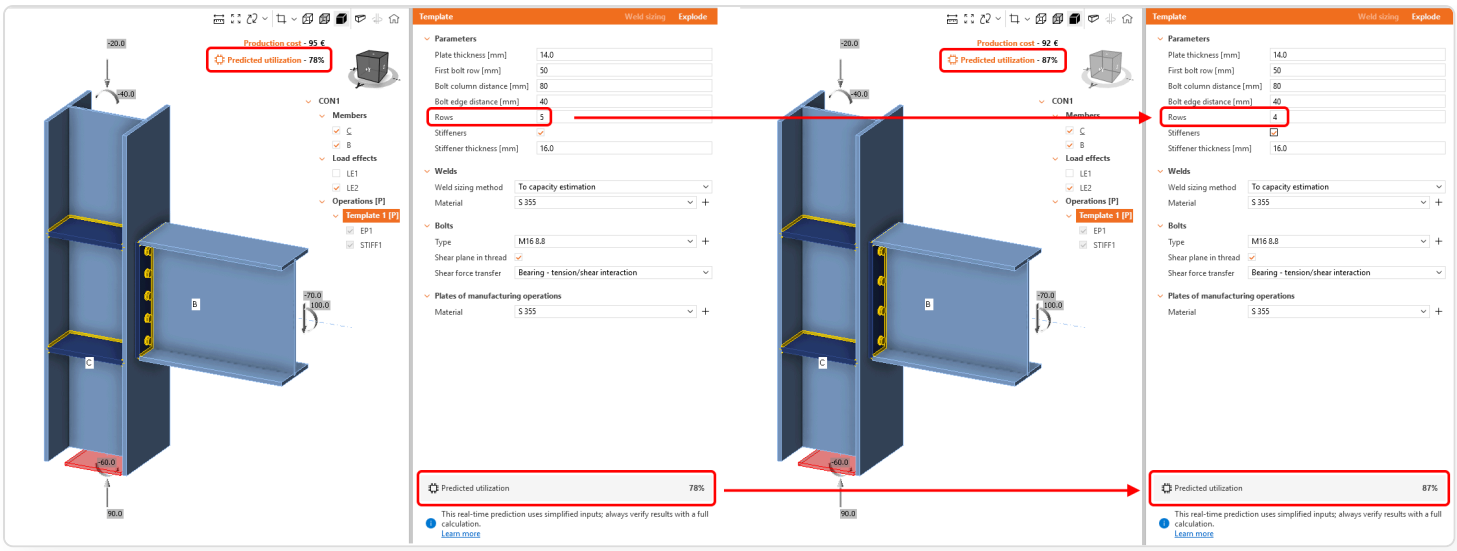
All these models were fully calculated using standard IDEA StatiCa Connection analysis. Based on the results, a machine learning model was trained to predict the template's utilization for new parameter combinations that had not been explicitly calculated beforehand.

The approach provides a fast estimate of the connection utilization without the need to run a full calculation after every parameter change.

How the prediction is evaluated

For each supported parametric template, the machine learning model considers the template's geometry and parameters, along with the load components applied during training.

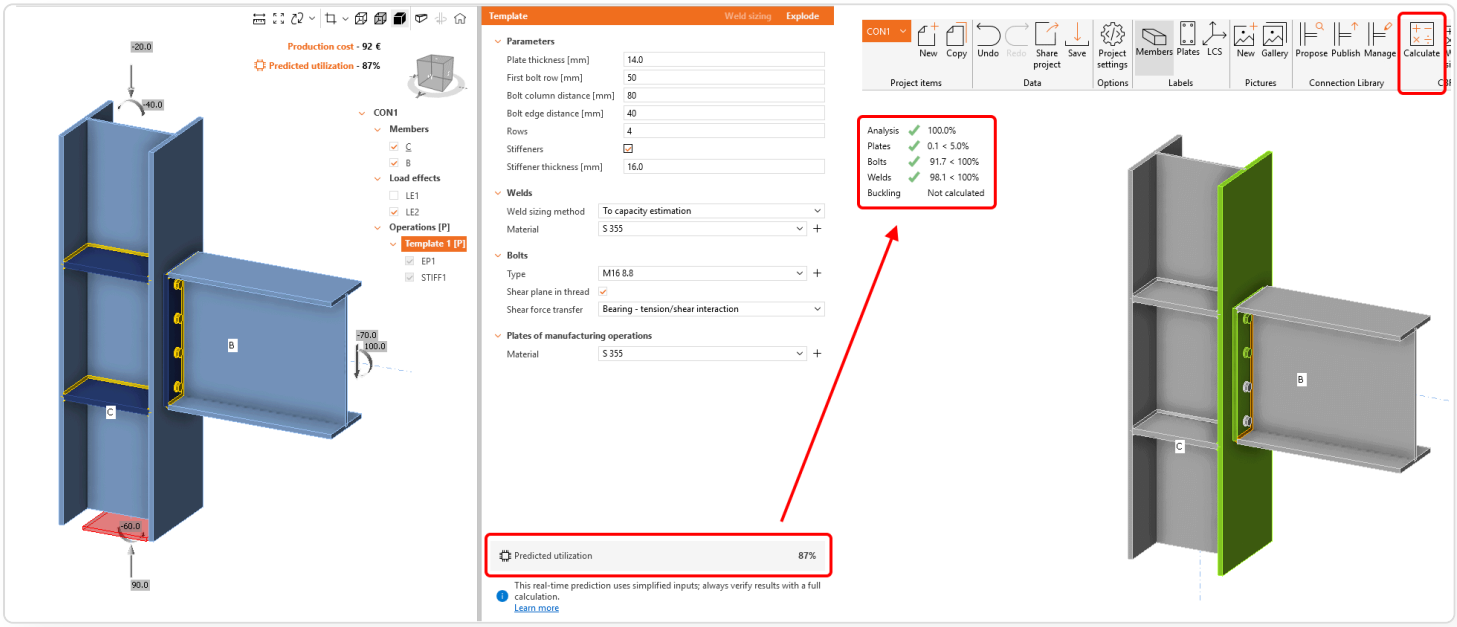
When the user modifies a template parameter (e.g., plate thickness, bolt size, or edge distances), the predicted utilization is updated immediately. This allows the user to quickly assess whether the current configuration is closer to or further from the expected capacity.



The user benefits of predicted utilization

Thanks to the predicted utilization, the user can optimize the connection configuration without repeatedly running the full calculation. The prediction provides instant feedback during parameter changes.

Once the user is satisfied with the predicted utilization, the **final verification must always be performed by running the standard stress/strain calculation**. Only the calculated result represents the actual design check according to the selected code.



This approach significantly accelerates the design process, as it is no longer necessary to verify every intermediate design variant by calculation.

Displayed prediction in the scene

If multiple load effects are defined in the project, the predicted utilization displayed in the scene always corresponds to the **load effect in which the connection reaches the highest utilization**. Predictions for other load effects are not shown simultaneously.

Similarly, when two or more parametric templates are used within a single connection model, the predicted utilization displayed in the scene represents **the most utilized template**, not the utilization of the joint as a whole, because the combined templates' interaction is not taken into account.

Predicted utilizations for individual templates can be viewed by **selecting each template separately**. This allows the user to assess the estimated utilization of each template independently, while keeping in mind that the displayed value does not represent the combined behavior of the entire connection model.

Limitations

The predicted utilization is intended to provide **a fast estimate** during connection design. When interpreting the displayed value, it is important to understand the assumptions used during the training of the machine-learning model and the inherent limitations resulting from them.

The following limitations apply to the predicted utilization:

1 Load considerations

- **Limited load components** – The training of each template included only two load components (e.g., Vz and My), with anchoring cases trained for three components. Other load components are not considered for prediction. If these components are decisive for the connection behavior, the predicted utilization may differ significantly from the calculated result.
- **Connected member only** – The prediction considers loads applied to the connected member only. Loads acting on the supporting member are not included. Where the supporting member is significantly loaded, the calculated result may be substantially less favorable than the prediction.

2 Modeling assumptions

- **Butt (CJP) welds assumed** – The models were trained assuming butt (CJP) welds. Weld size is therefore not reflected in the predicted utilization. If weld capacity is decisive, the prediction may differ significantly from the calculated result. Weld autodesign with overstrength, for full-strength, minimum ductility, or capacity estimation is available in the project item context menu.
- **Default project settings required** – The predicted utilization offers reliable results only if the project settings match those used during training. This means that **default Project settings** must be used, in particular, safety factors, mesh settings, default lengths, and segment division.

3 Availability

- Only for **EN** and **AISC** design codes.
- Only when **Stress–Strain analysis** is selected.
- An **Internet connection** is required.

4 Template modifications

- **Developer tab and structural scheme changes** – If the user modifies the template in the Developer tab (e.g., removing parameters, changing parameter limits) or selects a different supporting member, the prediction becomes unavailable. In such cases, the algorithm cannot control how much the modified connection differs from the original trained template.
- **Geometric changes** – Some geometric changes are not yet reflected in the predicted utilization. For example, changing the angle or axis eccentricity of the connected member is not taken into account, and in such cases, the predicted utilization may differ significantly from the calculated result.

- **Additional operations** – If an additional operation is added to the template (e.g., a stiffener), the predicted utilization will still be displayed, but the effect of that operation is **not** included in the prediction.

5 Multiple templates

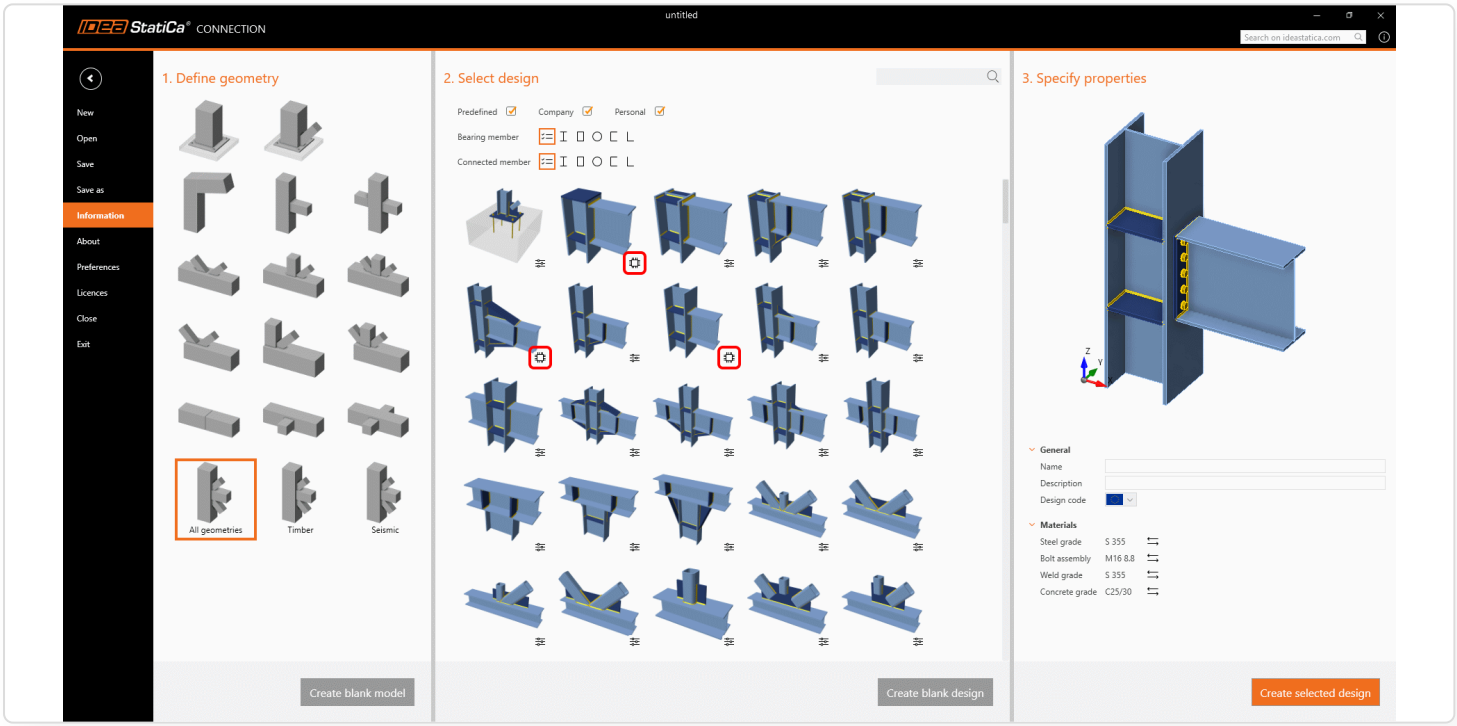
- When multiple templates are used within one connection model, the utilization of each template can be predicted separately. However, their **mutual interaction is not considered.**

Final remark

The predicted utilization should always be understood as an estimate. **Machine learning tools cannot replace a full structural calculation performed by an engineer.**

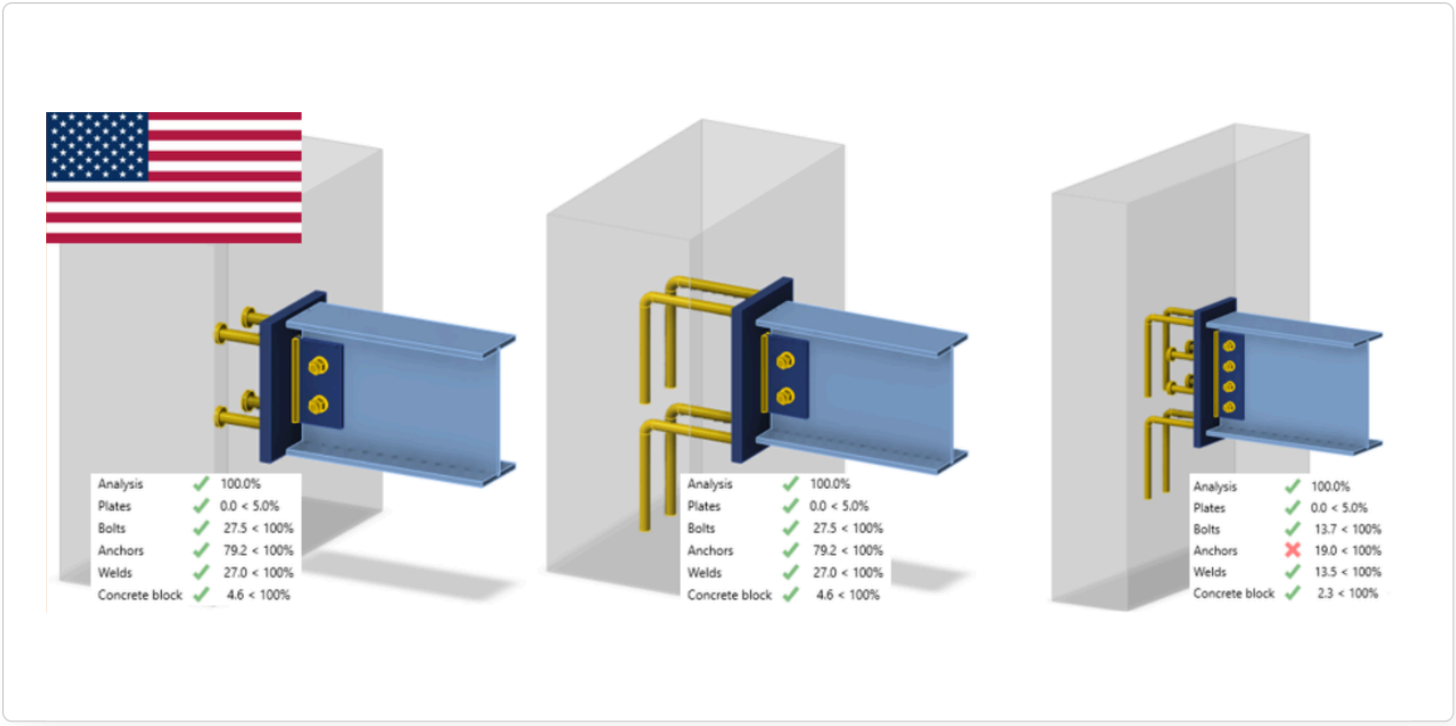
The prediction can provide **significant time savings** during connection optimization by allowing fast comparison of different parameter configurations.

The listed limitations will be gradually reduced as the machine learning process and training datasets are further expanded. At the same time, the number of supported parametric templates with prediction functionality will continue to increase.



Cast-in plate design available for AISC/ACI codes

Connection designers working under AISC/ACI codes can now model cast-in plates with realistic anchoring options, including reinforcement, headed studs, or both. This extends the workflow to support steel-to-concrete connections that were previously difficult or impossible to design directly in the Connection app, while also maintaining compatibility with export to Detail. The result is the most advanced anchoring design tool for engineers designing in accordance with these codes.



Follow the guided workflow below to learn more about cast-in plate design.

Anchor and cast-in plate design: Guided workflow for safe, efficient connections

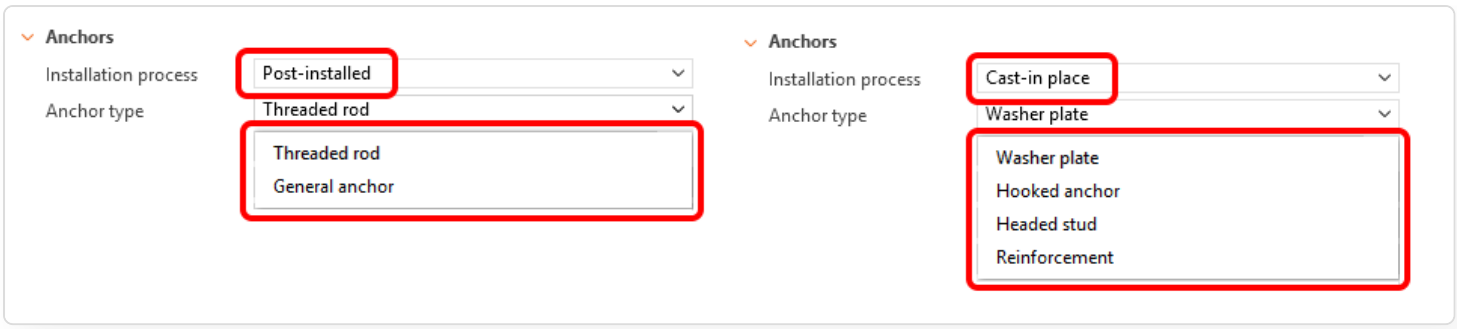
Anchoring just got easier. With a guided, process-based setup for anchors and cast-in plates, you can define details faster, run code checks with confidence, and focus on designing safe, reliable connections without the extra steps.

Anchor type selection for better understandability

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

Anchor definition now follows a two-step logic:

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



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

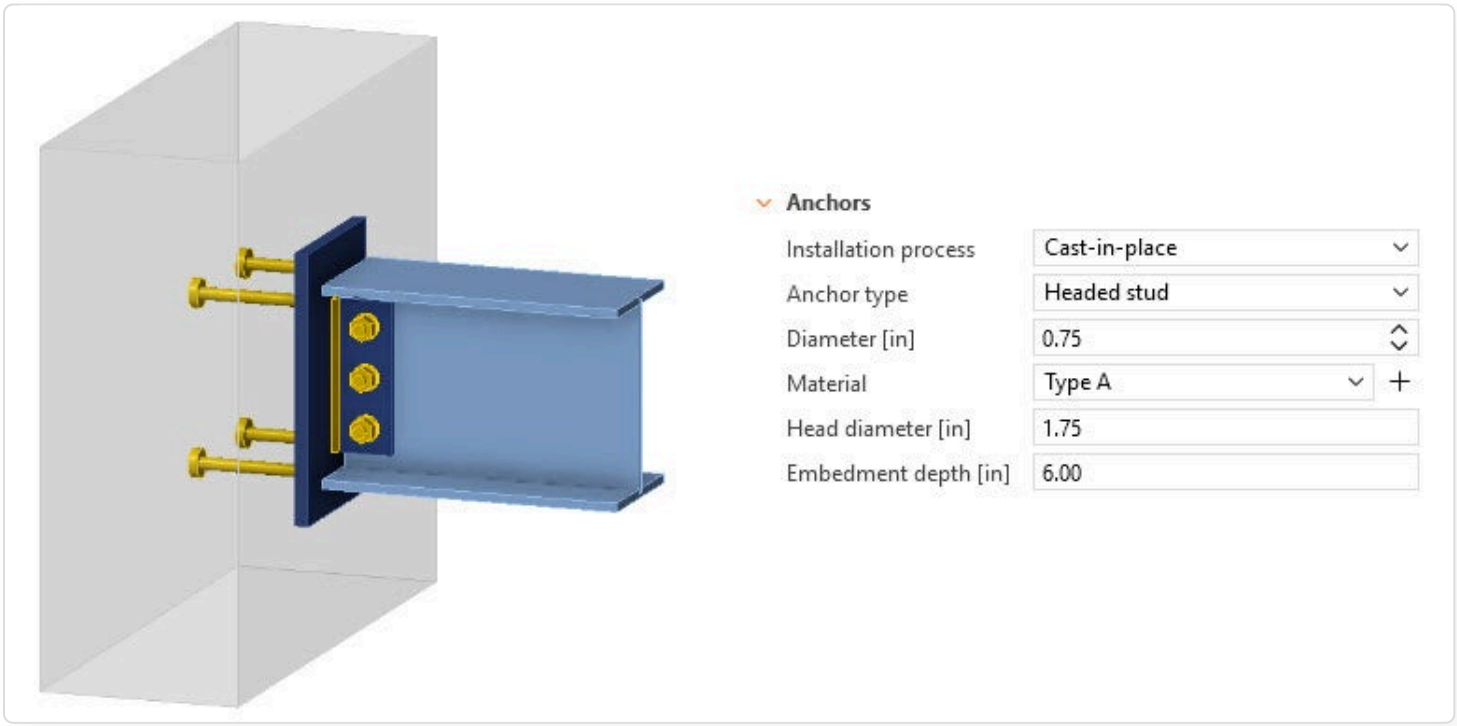
Released in IDEA StatiCa version 25.1.

Embed plates

IDEA StatiCa Connection introduces comprehensive support for **embed plates anchored with headed studs and reinforcement** in IDEA StatiCa Connection. This enhancement allows designers to perform anchorage checks directly in the application without relying on external tools. The implementation follows the provisions of **ACI 318-25**, enabling verification of both tension and shear resistance for cast-in connections.

Headed studs

A device type for **headed studs** can be found under the *Design* tab. The shear force transfer is limited to the *Anchors* option, with *Direct* stand-off only. Supported materials include **ASTM A108 Types A & B**, ensuring compatibility with standard construction practice. Headed studs are defined by diameter, material, head diameter, and embedment depth, with visualization indicating the plate embedded in concrete.



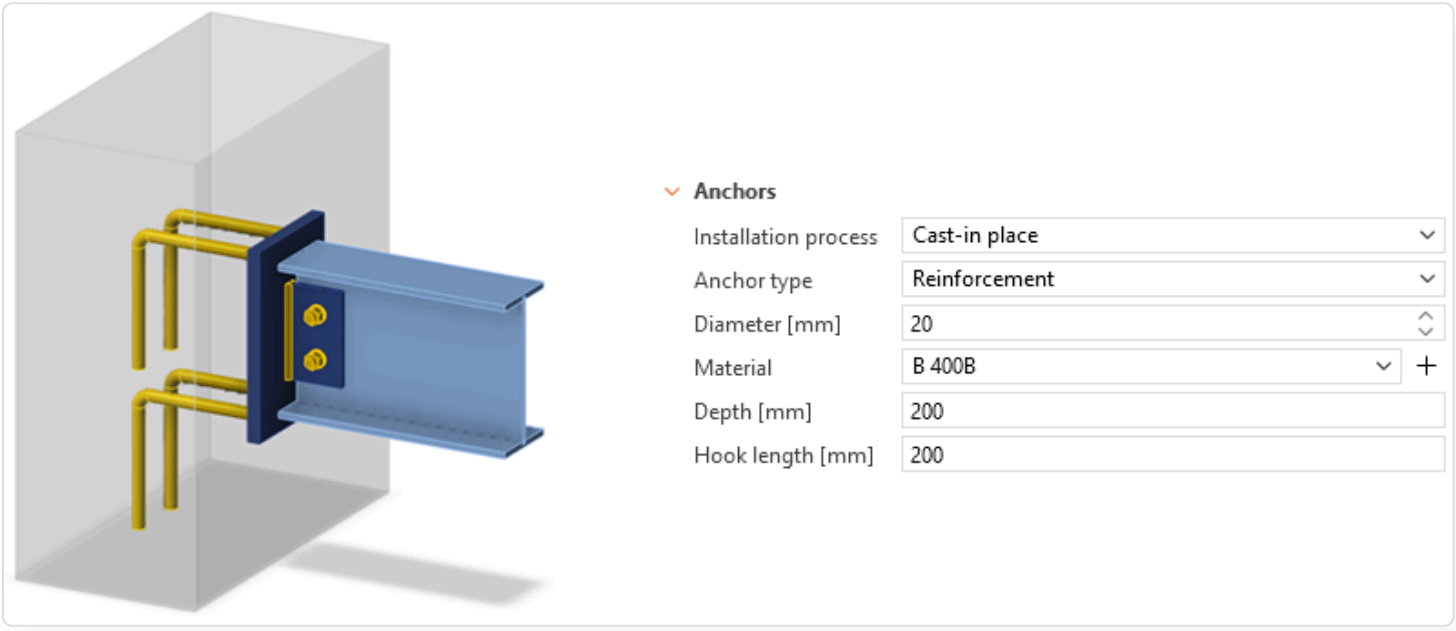
Implemented design checks follow **ACI 318-25** and cover the following:

- Tension
  - Anchor tensile resistance: ACI 318-25 - 17.6.1
  - Concrete breakout resistance of anchor in tension: ACI 318-25 - 17.6.2
  - Concrete pullout resistance: ACI 318-25 - 17.6.3
  - Concrete sideface blowout resistance: ACI 318-25 - 17.6.4
- Shear
  - Shear resistance ACI 318-25 – 17.7.1
  - Concrete pryout resistance: ACI 318-25 – 17.7.3
  - Interaction of tensile and shear forces in concrete: ACI 318-25 – 17.8.3
  - Interaction of tensile and shear forces in steel: ACI 318-25 – 17.8.4

Reinforcement

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

The only available shape is the L-bar, using materials already implemented in the concrete apps (ACI 318). Reinforcement properties include diameter, material, embedment depth, and hook length.



Design checks follow **ACI 318-25**, covering

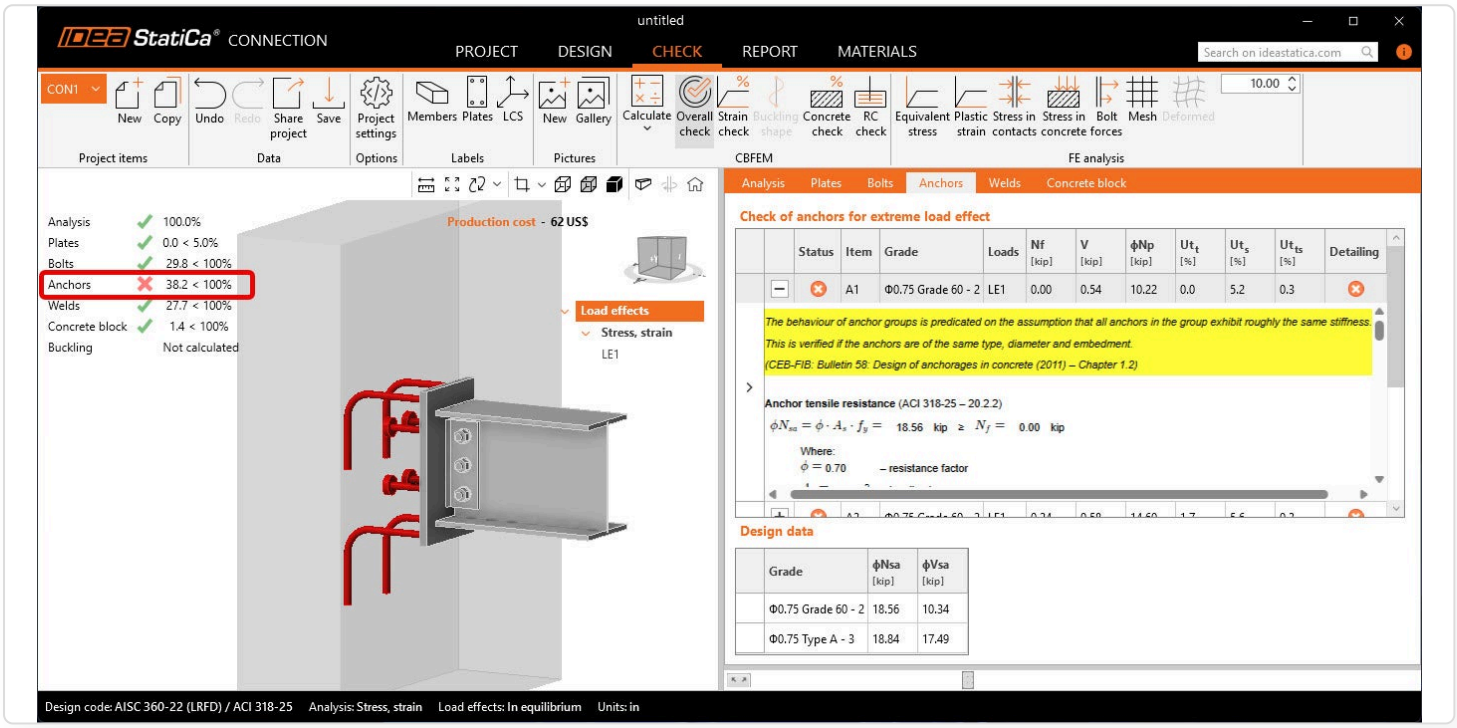
- Tension
  - Anchor tensile resistance: ACI 318-25 – 20.2.2
  - Concrete breakout resistance of anchor in tension: ACI 318-25 – 17.6.2
  - Concrete pullout resistance: ACI 318-25 – 25.4.3
- Shear
  - Shear resistance: ACI 318-25 – 20.2.2
  - Concrete pryout resistance: ACI 318-25 – 17.7.3

- Interaction of tensile and shear forces in concrete: ACI 318-25 – 17.8.3
- Interaction of tensile and shear forces in steel: ACI 318-25 – 17.8.4

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

Combining headed studs and reinforcement

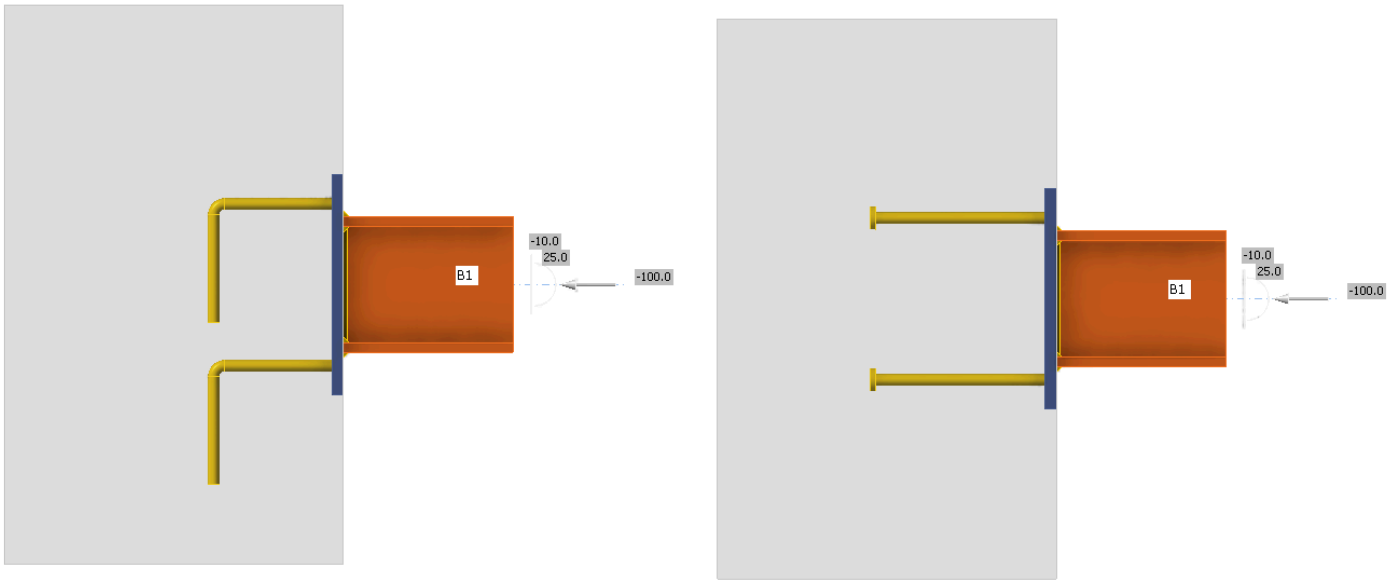
In case of higher load, the **headed studs and reinforcement can be combined**. This is practically modeled using the *Fastener Grid* or *Contact* operation.



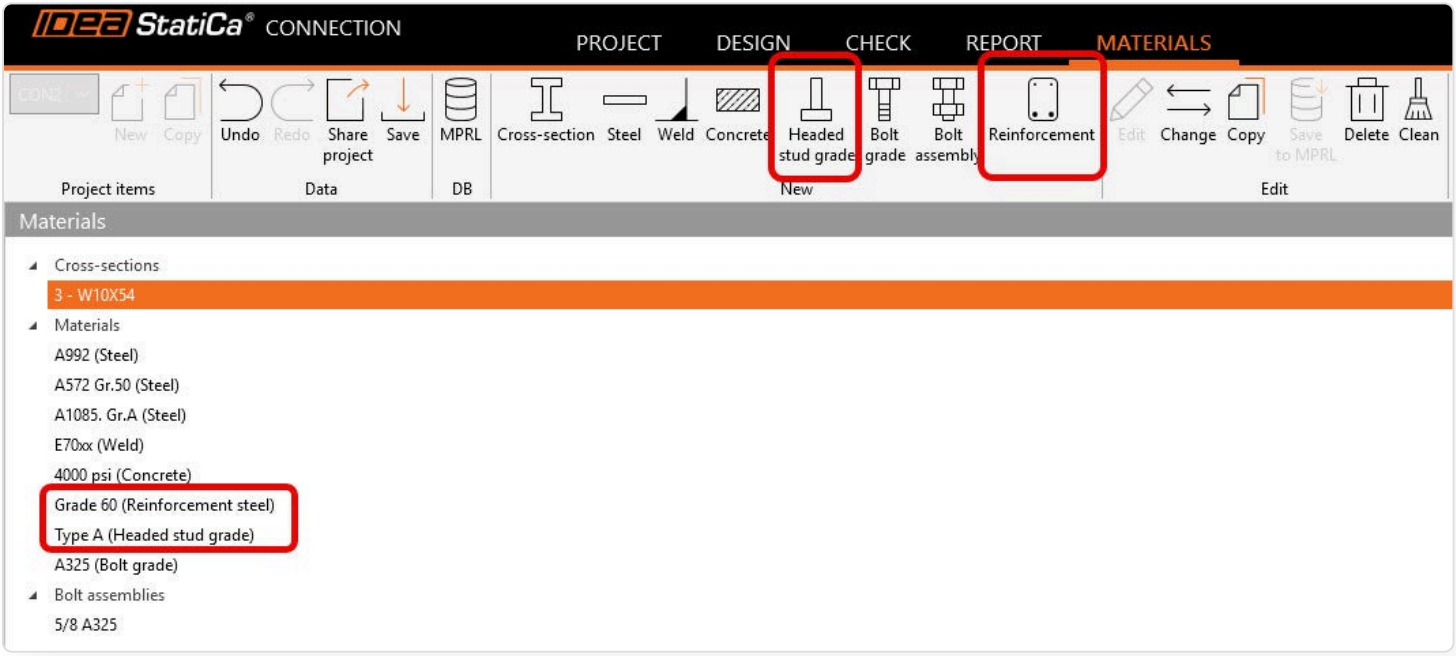
The redistribution of tension and shear is among all anchors based on their stiffness. The users can't change the force distribution (SCI 416).

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

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



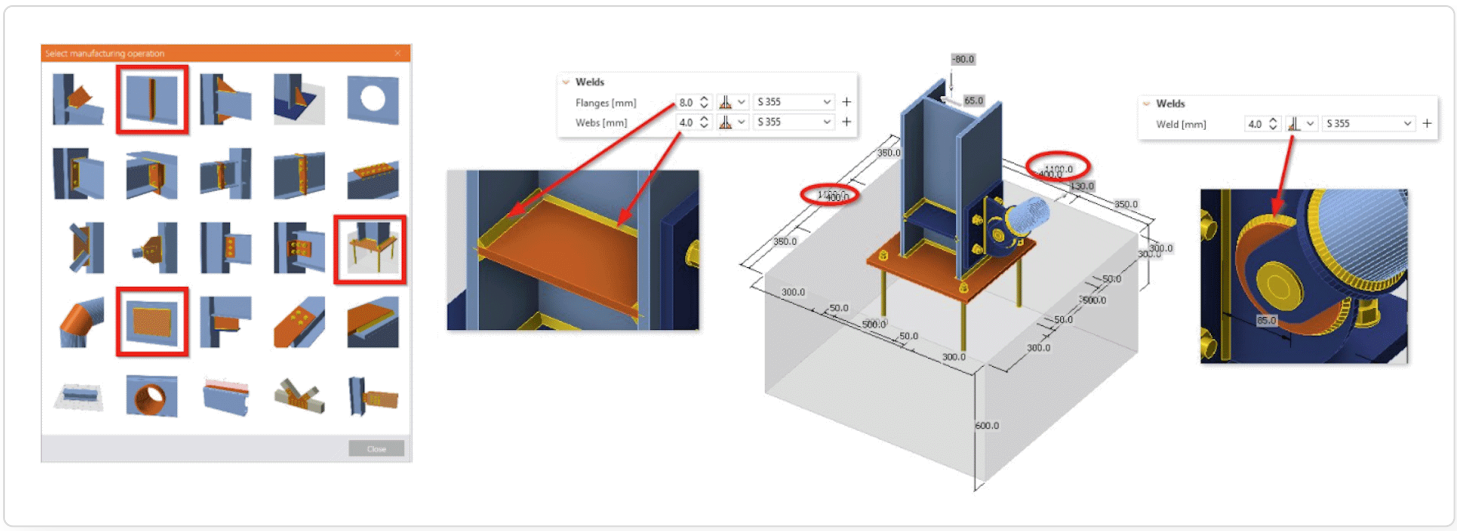
In the *Materials* tab, the Headed studs grade and Reinforcement material are placed in the top ribbon as well as in the left tree when used in the model.



Released in IDEA StatiCa version 25.1.1.

Operations usability improvements

Weld definition for stiffeners is now more flexible, allowing separate settings for web and flanges. Circular stiffening plates set as doublers can now be welded directly without workarounds. Additionally, the total dimensions of concrete blocks are displayed in the 3D scene, giving users instant clarity during modeling.

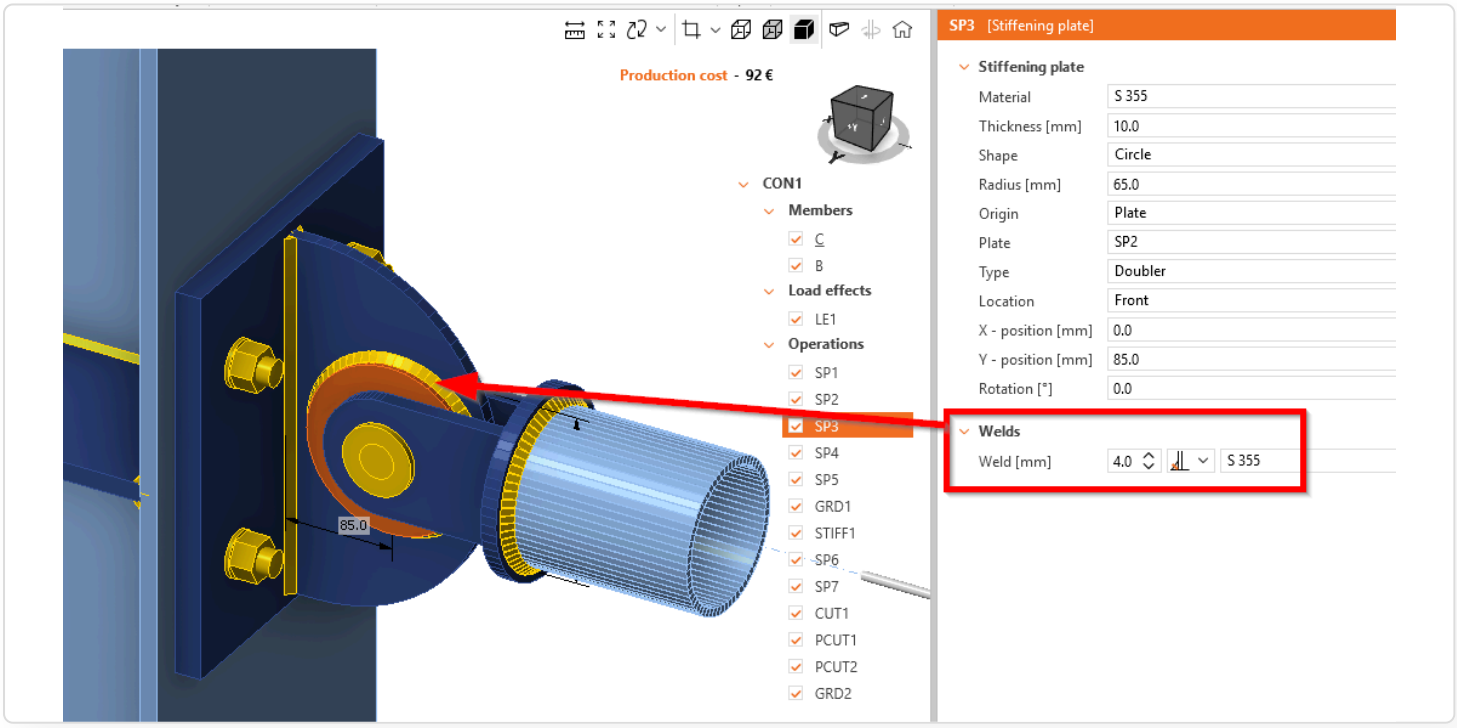


Circular plate weld

Circular stiffening plates are commonly used to reinforce thin webs, but before version 26.0, it was not possible to define welds for these plates directly.

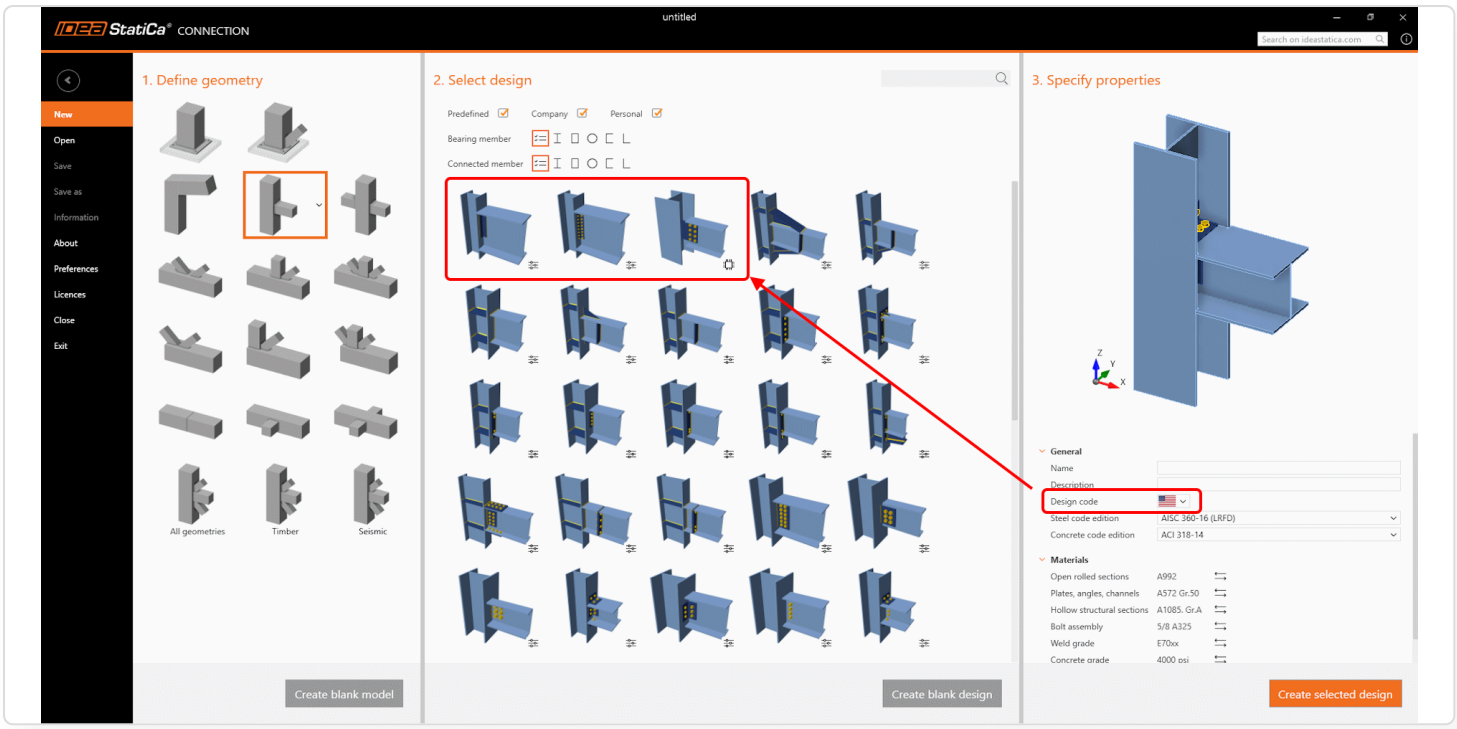
As a result, users had to rely on workarounds, typically importing the plate as a polygon from DXF, which added unnecessary steps and reduced modeling efficiency.

With this update, welds can now be defined **directly for stiffening plates with a circular shape**. The plate is internally discretized into segments, similar to circular hollow sections, ensuring accurate analysis behavior. The number of segments respects the global Project settings, providing consistency across the model.



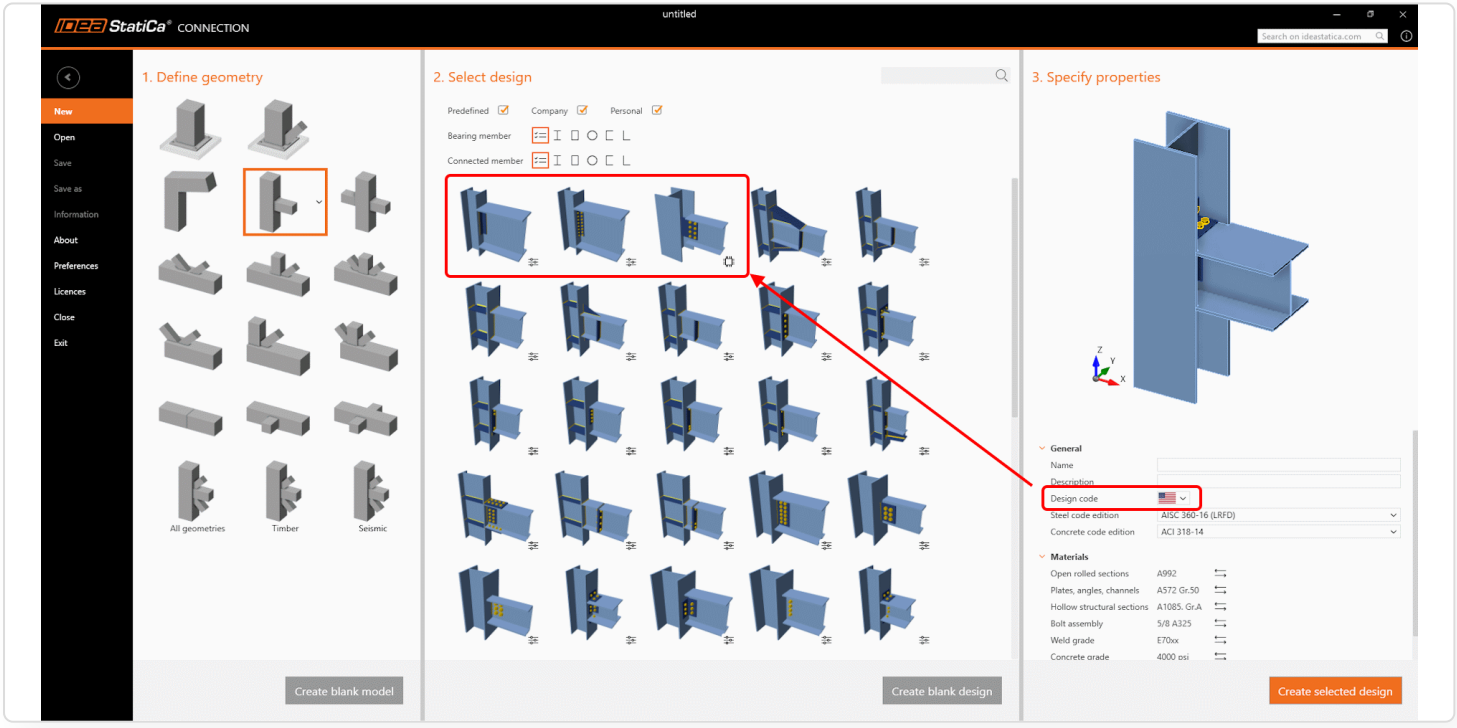
Localization of the Connection Wizard

The Connection Wizard is now better adapted to regional standards, introducing AISC-based templates, which are automatically prioritized for US users. In addition, improved rounding behavior ensures that values in imperial units are displayed in clean, expected increments based on Project Settings.



Regional templates (version 26.0)

With this update, new AISC-based templates, based on the Steel Construction Manual, are introduced and automatically prioritized when the US region is selected. While currently limited to three templates, this establishes a foundation for expanding localized content based on regional needs.



At the same time, rounding behavior for imperial units has been improved. Previously, values converted from metric templates often resulted in unintuitive numbers. Now, the wizard reads rounding settings from Project Settings and applies them to all visible parameters, ensuring clean and expected values without manual adjustments.

Connection API updates

Theoretical length for stiffness analysis and the Connection Library templates have been made available to the Connection API. The theoretical length values for M_y and M_z can be edited and set for fully automated stiffness calculations. Users can now directly access the full desktop Connection Library (IDEA, company, and private sets) without relying on external template files, while also applying multiple templates and targeting specific members within a connection model.

Learn more about IDEA StatiCa API on the developer site.

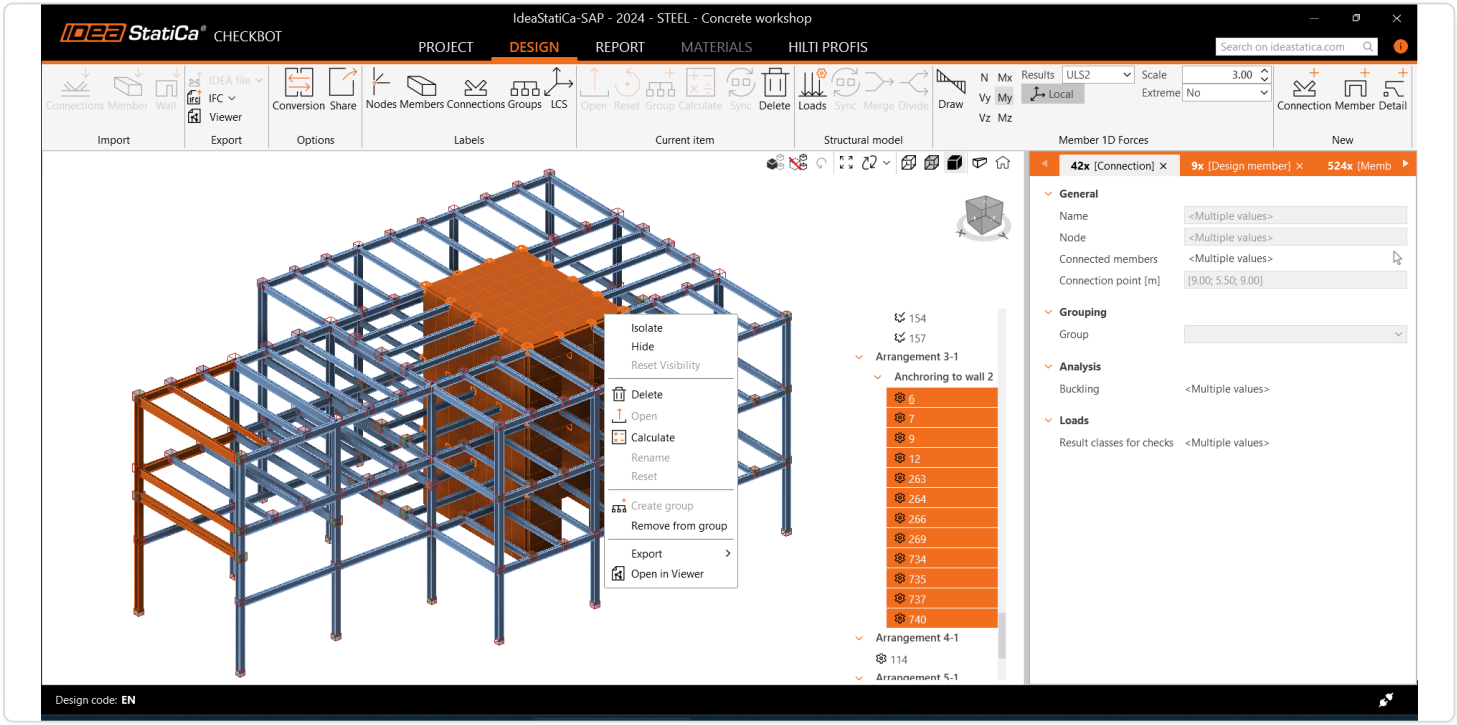
BIM and Checkbot

New 3D Scene in Checkbot

The new 3D scene in IDEA StatiCa Checkbot introduces a rebuilt engine that improves performance and reduces memory usage in large 1000+ nodes projects, enabling smooth interaction without lag or crashes.

You can now multi-select items directly by mouse drag, refine selections using Ctrl/Shift, and use new multi-selection tabs in the property grid for filtering mixed items. New visibility controls (Hide, Isolate, Reset) allow focusing on specific parts of the model, and forces can be viewed directly on the selected part of the structural model.

We have also updated the right-click context menus and added the option to remove members (beams, columns) completely from the Checkbot project.



To perform area selection, simply **drag from left to right** – only members inside the selection area are included, or **drag from right to left** – also members intersecting the selection area are included. Add item(s) by holding the **Ctrl** key and remove with the **Shift** key. Unselect with the **Esc** key. **Ctrl+A** selects all items in the 3D scene.

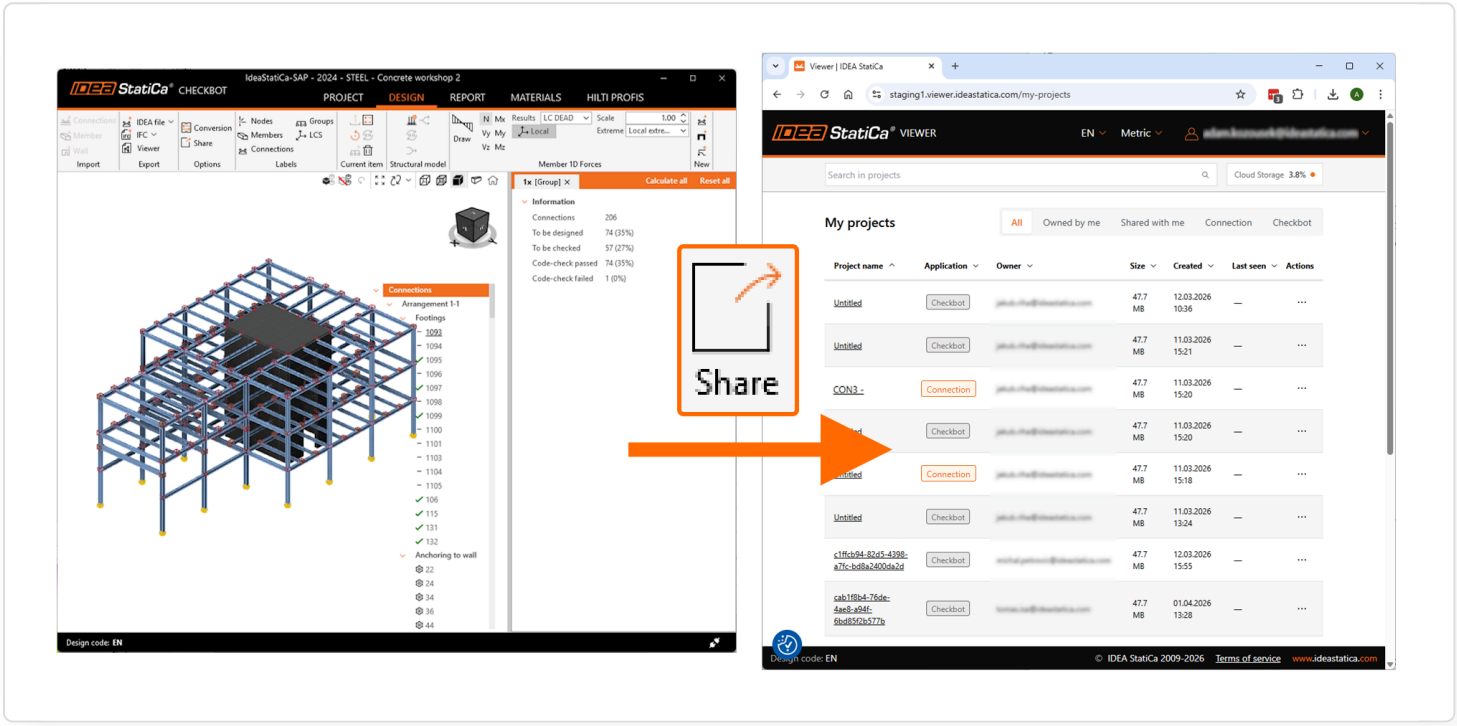
The context menu is accessible via right-click and provides actions for both selected and hovered entities. It provides execution of common operations directly within the 3D scene, such as deleting items.

Visibility controls – **Hide**, **Isolate**, and **Reset** allow users to manage the display of entities in the 3D scene. These controls are accessible via right-click menu, the 3D scene ribbon icons, or the hotkeys **Ctrl+H**, **Ctrl+I**, and **Ctrl+R**.

When multiple entities are selected, the property grid displays tabs with those entities grouped by type (Connection, Design member, etc.). Each tab provides an overview of the corresponding entities and allows for quick management of the selection by right-clicking on the tab - **Select only ...** and **Remove from selection**.

Sharing of Checkbot projects

You can now share IDEA StatiCa Checkbot projects directly from the app via email using the built-in Share function, eliminating the need for manual zipping and third-party file transfers, and enabling easy distribution through cloud storage. The shared projects are stored permanently in the cloud and managed in IDEA StatiCa Viewer, where you can also copy the direct URL link, re-share it, or delete it. The storage capacity for standard licenses is now 10 GB.



Project sharing allows users to distribute Checkbot projects directly via email message or URL link.

A project can be shared using the **Share** command in the ribbon. During the process, users can:

- Choose to include results (affects file size)
- Add one or more recipients
- Review the available cloud storage quota

Once initiated, the upload runs in the background even if Checkbot is closed. The process can be monitored in Windows Task Manager (the bottom right corner of the taskbar). Once the upload finishes or is interrupted, a Windows notification pops up. Incomplete uploads can be removed manually or are cleared automatically after a defined period. All recipients receive an automated email with access to the shared project.

Shared projects are uploaded to cloud storage and can be managed in the IDEA StatiCa Viewer under **My projects** menu (under login in the right-top corner), which provides an overview of all uploaded and shared files. Users can search and filter projects, **copy a URL link** to projects, review sharing details, and delete projects.

The interface also displays current storage usage and available quota. The quota depends on the license type - a standard license account has 10 GB of space for all its license users, an Enterprise license account has 100 GB, and single Basic / Trial users have 500 MB.

Steel and Concrete design items in one Checkbot project

Checkbot now supports multi-material projects, allowing you to manage both steel and concrete entities in a single project file under a single design code, with materials and cross-sections automatically sorted and mapped.
Note: Only steel-to-steel and steel-to-nothing (anchoring) nodes can be opened in the Connection and Member apps. Only 2D concrete walls can be exported to the Detail app.

MIDAS GEN NX - new BIM link to Checkbot

You can now import data directly from MIDAS GEN NX to IDEA StatiCa Checkbot, enabling a standard BIM workflow for steel connection design and steel member design with automated mapping of members, cross-sections, and internal forces from load combinations. Learn more in the tutorial.

Grasshopper plugin updates

We have added two new components to handle Grasshopper workflows – eccentricities and templates. With eccentricities, you can define the insertion point of members and add offsets per local and global coordinate systems to achieve member alignment. With templates, you can now fully manage the Connection Library design templates using components to propose, edit, explode, and delete them. Learn more on the developer site.

Updated BIM links with FEA and CAD apps

We have updated the BIM links to match the latest versions of the connected 3rd-party FEA and CAD software. BIM links are also continuously updated with each subsequent IDEA StatiCa patch. See the up-to-date list.

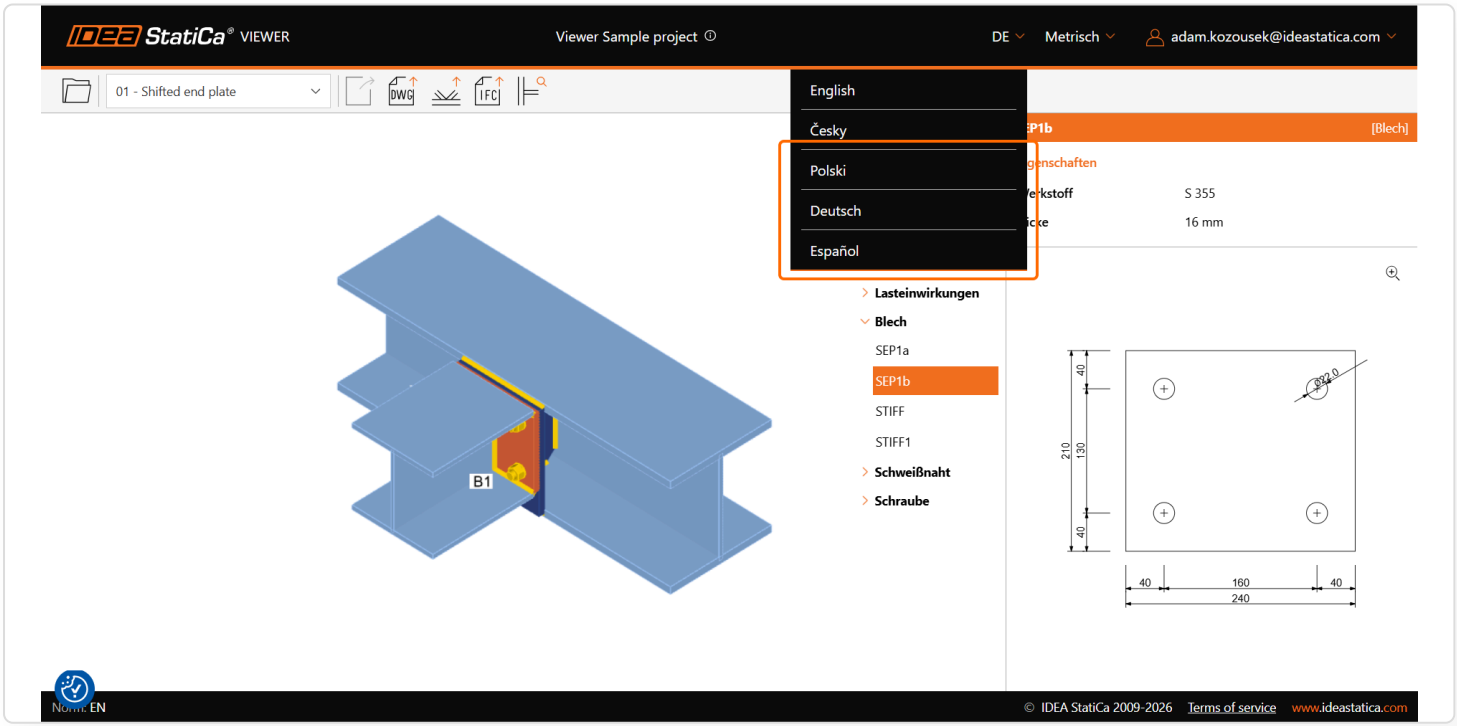
IDEA StatiCa - Steel	26.0.0		
Application	Supported	In development	Obsolete
Advance Design	2025, 2026	-	-
Advance Steel	2025, 2026	2027	-
AxisVM	X7.3, X8.1	-	-
ETABS	22, 23	-	21
MIDAS Civil	2024, 2025	-	-
MIDAS Civil NX	2024, 2025	-	-
MIDAS GEN	2024, 2026	-	-
MIDAS GEN NX	2026	-	-
RAM Structural System	24, 25	-	23
Revit	2025, 2026	2027	-
RFEM / RSTAB	5.40 / 8.40, 6.12 / 9.12, 6.13 / 9.13	6.14 / 9.14	5.38 / 8.38, 5.39 / 8.39, 6.11 / 9.11
Rhino/Grasshopper	7, 8	-	-
Robot Structural Analysis	2025, 2026	2027	-
SAP2000	26, 27	-	25
SCIA Engineer	25, 26	-	-
SDS2	2025-1, 2026	2026-1	2025
STAAD.Pro	2024, 2025	-	-
Tekla Structures	2025, 2026	-	2024

IDEA StatiCa - Concrete	26.0.0		
Application	Supported	In development	Obsolete
Advance Design	2025, 2026	-	-
AxisVM	X7.3, X8.1	-	-
ETABS	22, 23	-	21
MIDAS Civil	2024, 2025	-	-
MIDAS GEN	2024, 2026	-	-
RFEM / RSTAB	5.40 / 8.40, 6.12 / 9.12, 6.13 / 9.13	6.14 / 9.14	5.38 / 8.38, 5.39 / 8.39, 6.11 / 9.11
Robot Structural Analysis	2025, 2026	2027	-
SAP2000	26, 27	-	25
SCIA Engineer	25, 26	-	-

Cloud tools and general improvements

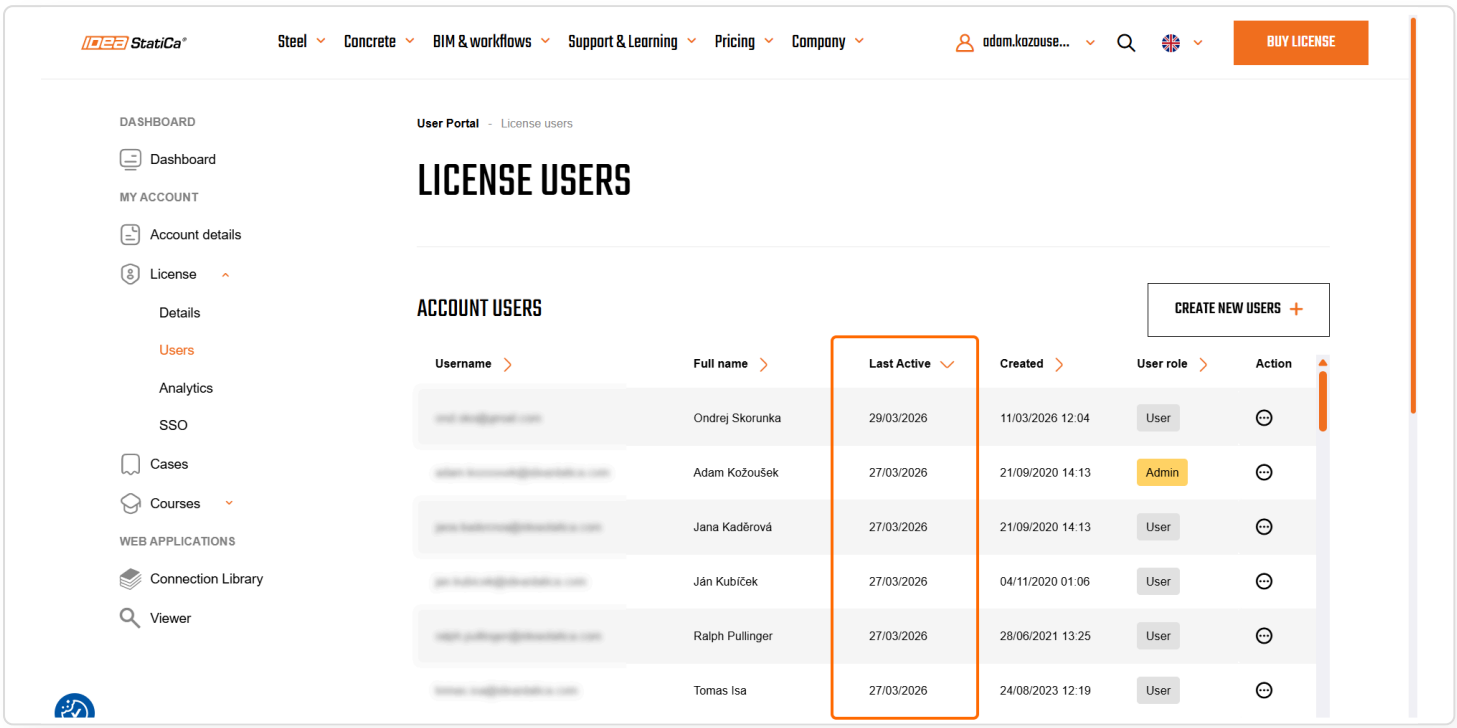
New languages in Viewer – German, Spanish, Polish

We have added three language variants to the free online Viewer app, German, Spanish, and Polish, next to the default languages of English and Czech.



User Portal – user's last active information

As a License Administrator, you can now see the "Last Active" timestamp for each user in your license. In the User Portal, section License/Users, you can identify the most active and inactive users and optimize your license utilization.



Shared login for web applications

When you work with the Viewer and Connection Library web browser apps, where you need to log in with your credentials, you no longer need to log in separately to each of them. Once you have logged into one of them, your browser remembers, and the apps open with you already logged in.

Self-service setup of Single Sign-On (SSO)

In previous versions, Enterprise customers needed to contact the IDEA StatiCa ICT department to set up SSO. This can now be set by the admin in the User Portal, section Licenses/SSO, without waiting.

Set-up guide for Single Sign-On (SSO)

Our SSO solution uses OpenID Connect (OIDC). The SSO connector lives in your tenant as an Enterprise application. Before configuring single sign-on, please ensure your organization fulfills and accepts the requirements below.

Prerequisites

- An IDEA StatiCa Enterprise license
- Your organization uses a single Microsoft tenant. Federated tenants are not supported.

- Guest accounts in Microsoft Entra are not supported.
- All your IDEA StatiCa license users must be under one license for SSO to work correctly.
- If you want to restrict access to IDEA StatiCa within your organization, you must do this on your side in the Enterprise Application settings in Microsoft Entra by assigning users or user groups, as described in the manual.
- You do not need to create users manually. Users who have access to the integration app in Microsoft Entra will be automatically created and appear in the User Portal after their first successful login.
- Your users' existing usernames and passwords will remain valid until their first SSO login. After that, they will be able to sign in only through SSO.
- Users in your organization with Trial licenses will be converted automatically to commercial users after logging in with SSO for the first time.

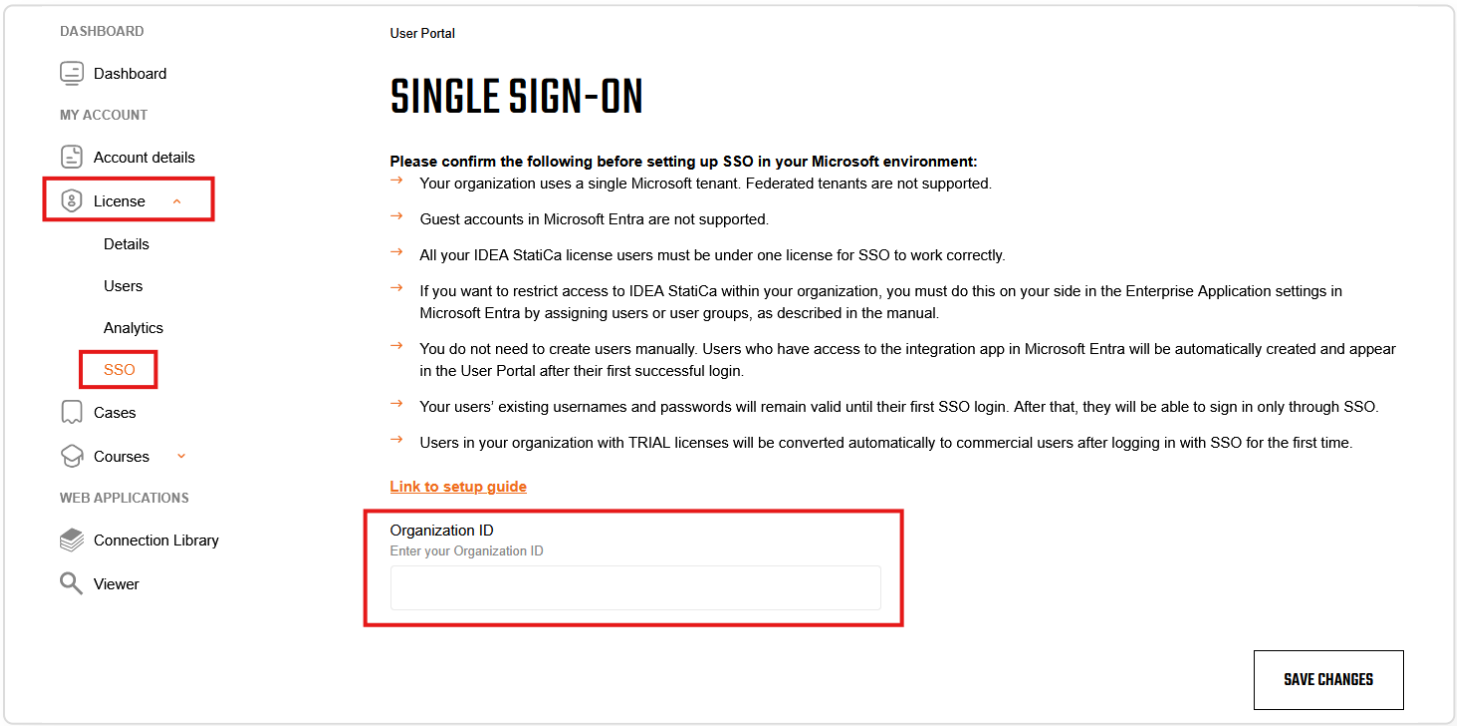
SSO Configuration

Step 1: Access the SSO Settings

1 Log in to the User Portal on the website as the license administrator.

2 In the left navigation panel, select License.

3 Select SSO.



Step 2: Pair Your Organization

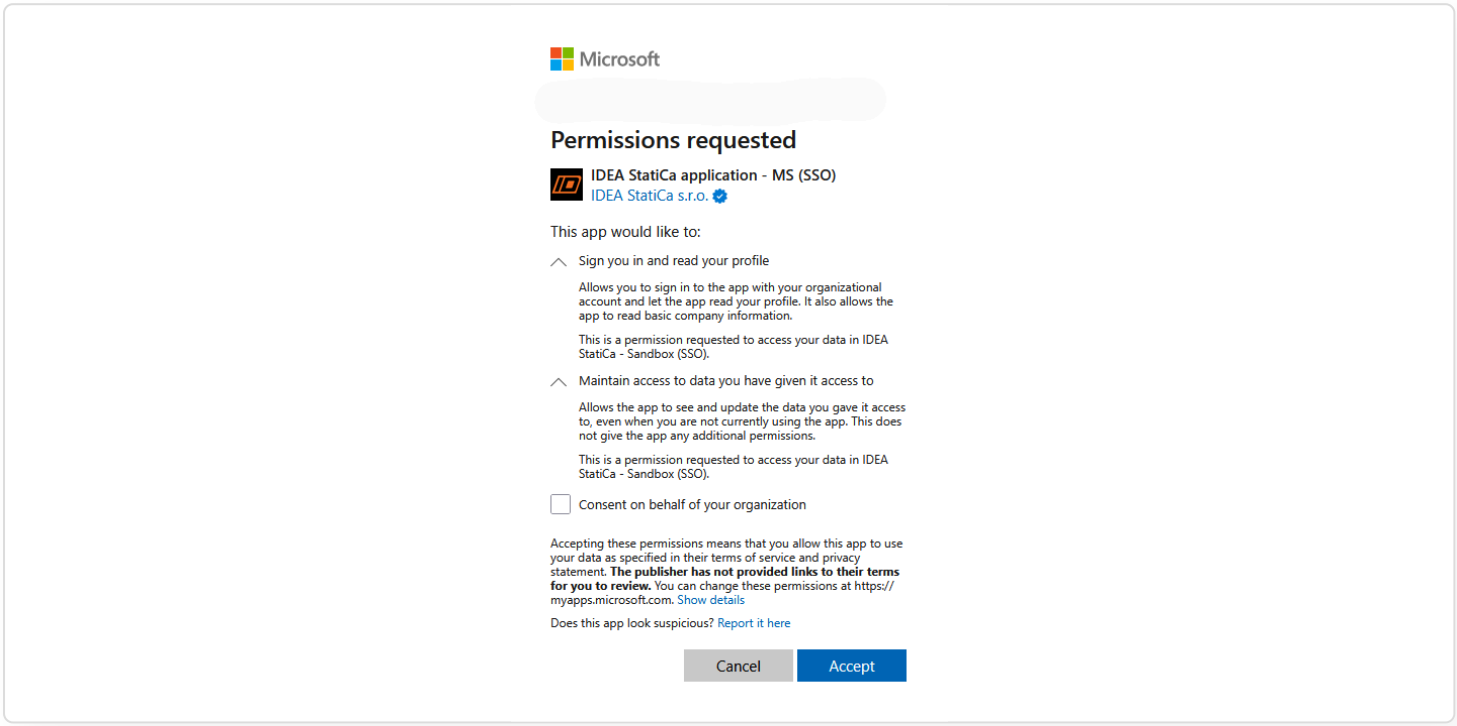
In the SSO settings page, enter your Microsoft Tenant ID. This will pair your license with your organization.

Where to find your Microsoft Tenant ID?

You can find your Microsoft Tenant ID in the Microsoft Entra admin center under Overview > Tenant ID.

Step 3: Complete SSO Activation

1. Sign in at least once using SSO, preferably as the Microsoft Entra application administrator. This initial sign-in allows you to add the enterprise SSO application to your environment.
2. When prompted, confirm the permission dialog to grant the necessary access.
3. Once confirmed, SSO is configured and ready to use.
4. Users in your organization are now able to sign in using their existing Microsoft account via SSO.



Personalized Launcher content

We have updated the content display in the initial Launcher app. You will now see news, such as webinar announcements or an important technical article, tailored to your language and region.