

# IDEA StatiCa 21.1

## Release notes

October 2021



# Contents

<b>BIM LINKS .....</b>	<b>5</b>
CHECKBOT – A 3D VISUAL STANDALONE APP.....	5
IMPORT CONFIGURATORS FOR CHECKBOT.....	7
SCIA ENGINEER BIM LINK WITH USING OF SAF INTERFACE .....	8
NEW RFEM LINK .....	9
VERSIONS OF CAD/FEA SUPPORTED IN 21.1 .....	9
<b>IDEA STATICA MEMBER.....</b>	<b>10</b>
DESIGN AND CHECK OF SLENDER COLUMNS (MEMBER FOR CONCRETE) .....	10
DISPLACEMENT VECTOR SUMMATION IN RESULTS.....	15
USE GLOBAL/LOCAL COORDINATES IN SUPPORTS SETTINGS.....	15
CUSTOMIZABLE NUMBER OF BUCKLING MODES (MEMBER FOR STEEL).....	16
<b>IDEA STATICA DETAIL AND RCS .....</b>	<b>18</b>
DESIGN OF PRESTRESSED MEMBERS (DETAIL) .....	18
SPEEDING UP THE CSFM CALCULATION (DETAIL).....	21
CRACK WIDTH CHECK OF CROSS-SECTIONS WITH A LARGE CONCRETE COVER (RCS) .....	22
<b>IDEA STATICA CONNECTION .....</b>	<b>24</b>
CONNECTION BROWSER WILL PROPOSE THE SUITABLE DESIGN .....	24
NUMBER OF BUCKLING MODES.....	29
MATERIAL DATABASE IMPROVEMENTS (EUROPEAN STEELS ACCORDING TO PRODUCT SHEETS).....	29
WELD CHECK .....	31
BOLTS: BEARING TYPE OPTION REMOVAL .....	32
<b>LICENSING .....</b>	<b>32</b>
OBSOLETE APPLICATIONS .....	32

## Introduction

IDEA StatiCa 21.1 is the second release of the year but brings more features than the spring one. What did we try to achieve with this version? Scale up the steel connection design workflows and expand what we do in the design of reinforced and concrete members and details.

Enjoy and Calculate yesterday's estimates!

## News for BIM

Reliable BIM workflows are key for effective project work. That is why we invested in the massive upgrade of the "Code-check manager" application. And we also gave it a better name – the Checkbot. This new slick-looking application is the new hub of IDEA StatiCa workflows with 3<sup>rd</sup> party software. And a big booster of the productivity for engineers working with our BIM links.

### IDEA StatiCa Checkbot gives you:

- ✓ Complete control of the transferred structure
- ✓ 3D visualization of imported members and load
- ✓ Conversion table for materials & CSS
- ✓ Load combinations management

The Checkbot aims to provide one solution for all BIM links. It is a new generation of BIM management and it will replace the Code-check manager that you already know. It will also provide the open solution for anyone who wants to link their app to IDEA StatiCa (via IOM – IDEA Open Model). The Checkbot can be started from your BIM application as well as a standalone app.

Checkbot will enable you to link your Nemetschek apps models via the new SAF format (available in SCIA Engineer 21.1).

## News for concrete and prestressing

Slender reinforced concrete columns are very sensitive to imperfections which puts extra pressure on engineers during the whole design-build cycle. IDEA StatiCa Member, equipped with a new GMNIA solver in version 21.1, provides a reliable tool for engineers who needs to deliver clear and comprehensive design reports of slender columns.

IDEA StatiCa Detail has proven to be a unique code-checking tool for critical members and details in reinforced concrete structures. Version 21.1 expands IDEA StatiCa Detail to the prestressed concrete as well. Engineers can now understand and design discontinuities in PC members and details, enabling them to slash the time needed for design of prestressed beams, diaphragms, etc.

**New concrete workflows with IDEA StatiCa Member**

- Design and check of slender columns (Member for concrete)
- Displacement vector summation in results
- Use global/local coordinates in supports settings

**Design enhancements in RCS and Detail**

- Design of prestressed details
- Speeding up the CSFM calculation (Detail)
- Crack width check of cross-sections with a large concrete cover

## News for steel

IDEA StatiCa connection is quickly becoming a standard of connection design worldwide. Version 21.1 brings various code-check and modeling improvements as well as a new approach to handle repetitive connection designs.

We are releasing the Connection Browser out of beta! This unique tool will help you find a suitable design solution from a library of predefined designs and directly apply it to your members.

**Connection design improvements**

- An innovative design advisor – Connection Browser
- Custom number of buckling modes
- Material database improvements.
- Improved code-check of welds
- Simplification of the bolt model type

**New steel features with IDEA StatiCa Member**

- Customizable number of buckling modes (Member for steel)
- Displacement vector summation in results
- Use global/local coordinates in supports settings

## Licensing

Some outdated applications will be removed from the 21.1 install file. See the list of obsolete applications.

## BIM links

### Checkbot – a 3D visual standalone app

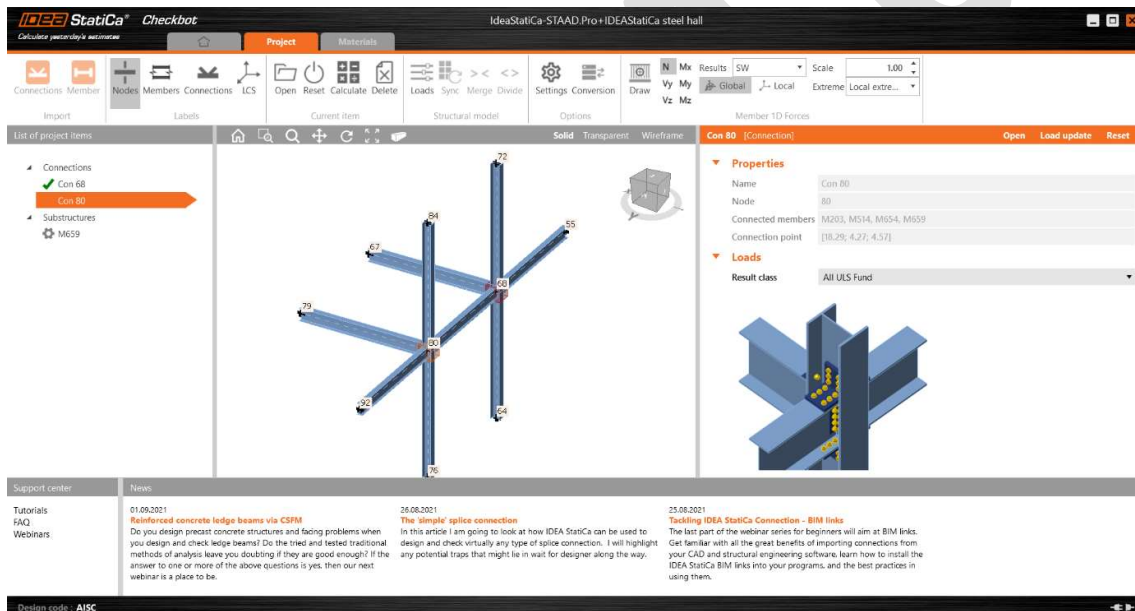
Our users around the world had one big wish – to get more clarity while working with tens or hundreds of connections or beams imported from a 3<sup>rd</sup> party applications. Basically, they wanted to see a list of connections or details that were checked, not-checked, changed, etc. That is why we responded - Goodbye to the Code-check manager, hello to Checkbot!

We are introducing a brand new application for the management of BIM workflow. Checkbot will boost the link between CAD/FEA and IDEA StatiCa with new possibilities.



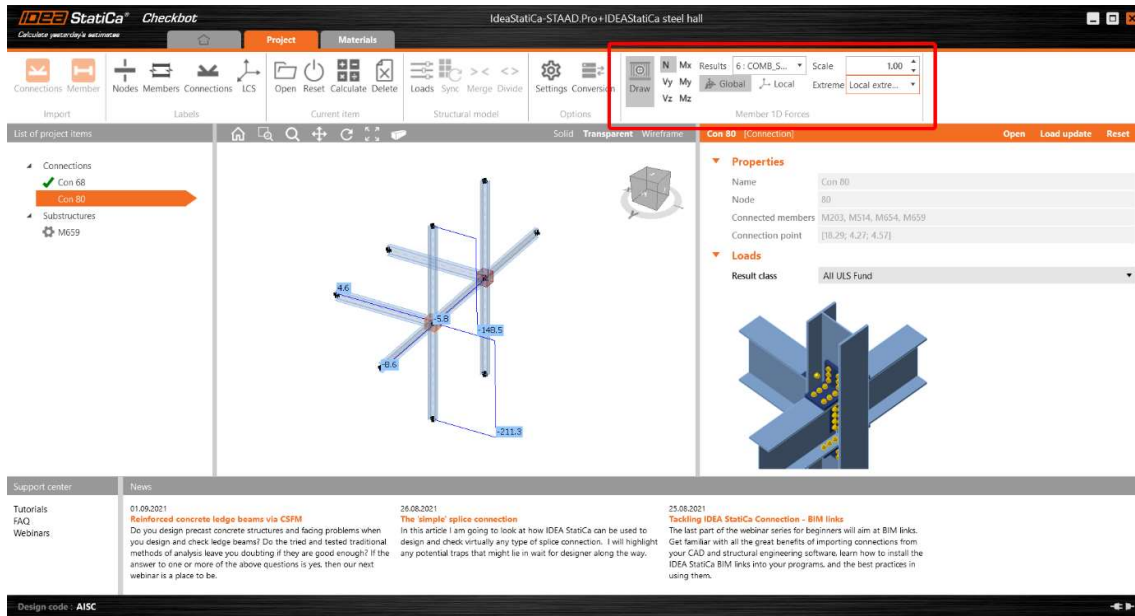
### The interface

There are three main components of the new Checkbot interface: The list of design items on the left, the 3D scene in the middle, and the properties on the right.

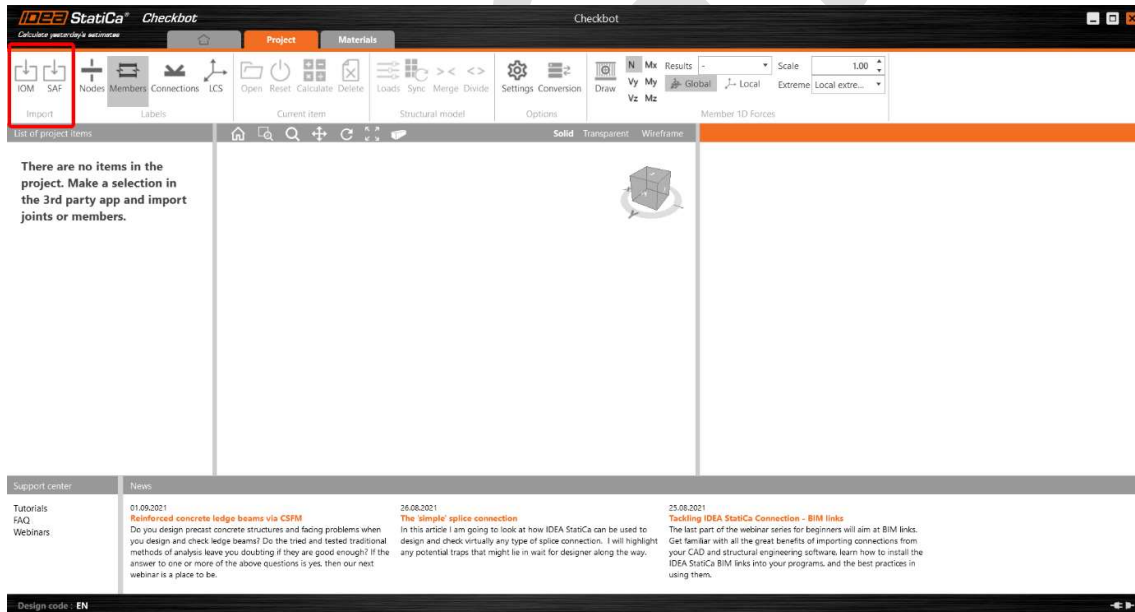


The list of project items contains all connections and members to be designed. Each project item can be in several different states. Right after import, there is a tooth wheel icon next to it, signifying the need to further define the item before it can be calculated. Once the item has been sufficiently defined (there is at least one load case and one manufacturing operation), the tooth wheel icon disappears, and the Project item can be calculated. After calculation, the item is in the calculated state and results are available (provided the calculation ran without exceptions).

The 3D scene in the middle shows the 3D Structural Model (SM), which displays the base entities in the model (nodes, beams), design items (connections, members), as well as internal forces (if available).



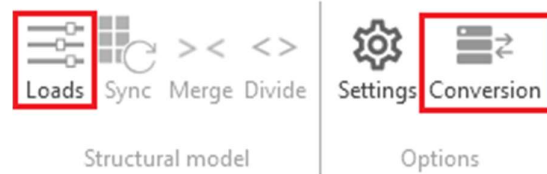
You can start the Checkbot directly from your CAD/FEA (as you are used to do) or it can also be launched as a standalone app, without a live BIM link connection. In this mode, the IOM or SAF files describing a structural model can be imported.



IDEA StatiCa Checkbot will work for FEA applications (including IOM and SAF imports) for **Expert** and **Enhanced** editions. CAD BIM links (Tekla, Advance Steel and Revit) are available in **Enhanced** edition.

## Import configurators for Checkbot

We present improvements to the management of the import of loads, cross-sections, and other materials. There are two new icons in the Checkbot ribbon.

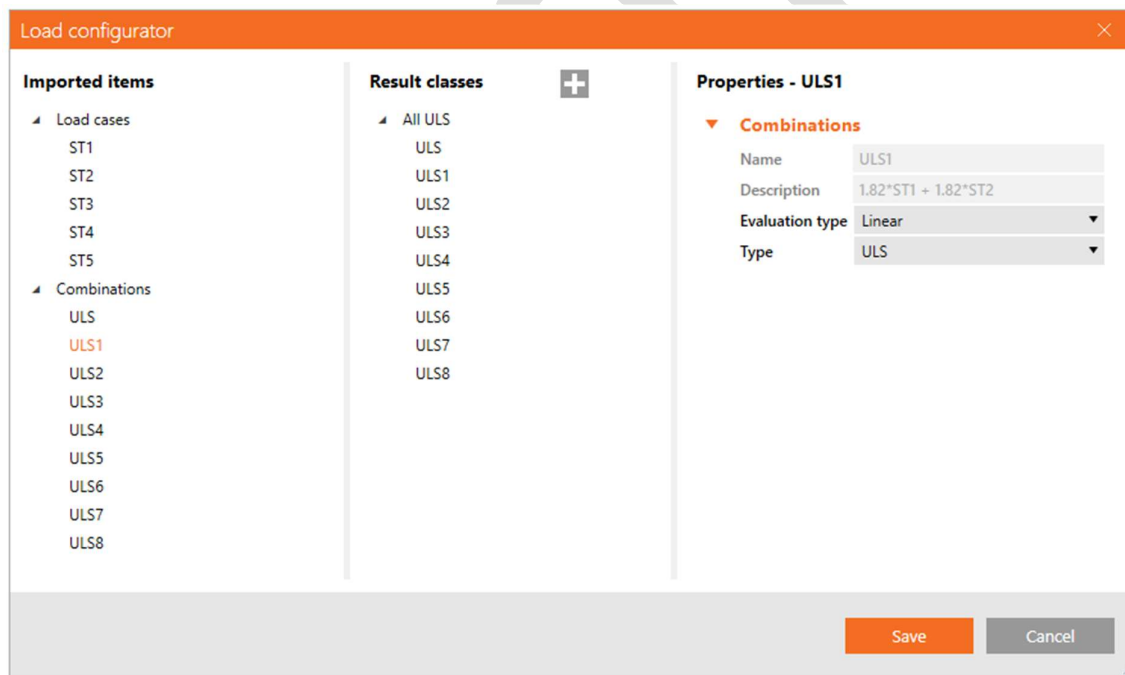


### Load configurator

The Load configurator displays the imported Load cases, Load Groups, Load Combinations and allows for their assignment under Result classes. Result classes are then used to generate Load effects for the project items in the Checkbot project.

The first column lists all the Load cases and Combinations imported from the linked 3rd party application model. The second column shows the Result classes in the Checkbot project. The third column displays the properties of the currently selected entity.

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



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

**N8 [Connection]**    **Open**    **Load update**    **Reset**

▼ **Properties**

Name	N8
Node	109
Connected members	B5, B8
Connection point	[2.50; 0.00; 5.00]

▼ **Loads**

Result class	All ULS ▼
--------------	-----------

### Editable conversion tables

Every time a cross-section or a material is not recognized during import, a new conversion table allows assigning an item from our material library to be used in place of the unrecognized material. These pairs are then saved for future use and therefore don't need to be defined again. Of course, the assigned materials can be changed, and this change will then be valid for all future imports.

Moreover, each "imported – assigned" pair has its own properties, showing further details about its origin.

**Conversion**

Type	Steel		Item
	Imported Item	Assigned item	
Cross-sections	S355JR	S 355	▼ <b>Origin</b>
<b>Steel</b>	STEEL/S235JR	S 235	Original name
Concrete	Metal - Steel - 345 MPa	S 355	Assigned name
Boltgrade			Code
Reinforcement			Link
			Date
			Source project

OK Cancel

### SCIA Engineer BIM link with using of SAF interface

An FEA link with SCIA built on the SAF format. This link is a result of close cooperation between SCIA and IDEA StatiCa. With the new version of SCIA (21.1), the Checkbot will be able to be launched directly from SCIA, data in the SAF format will be transferred from the SCIA model into Checkbot.





## New RFEM link

Significant improvement of RFEM BIM link to use IOM will improve the stability of the link and remove obstacles in functionality.

## Versions of CAD/FEA supported in 21.1

As we promised, we keep the BIM links between IDEA StatiCa and your CAD/FEA updated. In each major release of IDEA StatiCa (this year, it is 21.0 and 21.1), we will support the two most recent major releases of each linked application. The older versions will become obsolete, this will happen in the major IDEA releases only (patches will never disconnect older versions). On the other hand, when a new major release of BIM application comes, we will develop/update the link in 2 months – the link will appear in a patch of IDEA StatiCa.

The current state of the supported versions will always be presented on our web. With the 21.1 release, we will support the versions presented in the first column of the table. The third column shows versions that are no longer supported in 21.1.

IDEA StatiCa - Steel	21.1		
Application	Supported	In development	Obsolete
Advance Steel	2021, 2022	-	2020
Revit	2021, 2022	-	2020
Tekla Structures	2020, 2021	-	2019i
Advance Design	2021, 2022	-	2020
AxisVM	X5.4, X6.1	-	-
ETABS	18, 19	-	-
midas Civil	2020, 2021	-	-
midas Gen	2020, 2021	-	-
RFEM / RSTAB	5.25 / 8.25, 5.26 / 8.26	-	5.24 / 8.24
Robot Structural Analysis	2021, 2022	-	2020
SAP2000	22, 23	-	-
SCIA Engineer	20, 21	-	19.1
STAAD.Pro	22	-	-

IDEA StatiCa - Concrete	21.1		
Application	Supported	In development	Obsolete
Advance Design	2021, 2022	-	2020
AxisVM	X5.4, X6.1	-	-
midas Civil	2020, 2021	-	-
midas Gen	2020, 2021	-	-
RFEM / RSTAB	5.25 / 8.25, 5.26 / 8.26	-	5.24 / 8.24
Robot Structural Analysis	2021, 2022	-	2020
SAP2000	22, 23	-	-
SCIA Engineer	20, 21	-	19.1

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

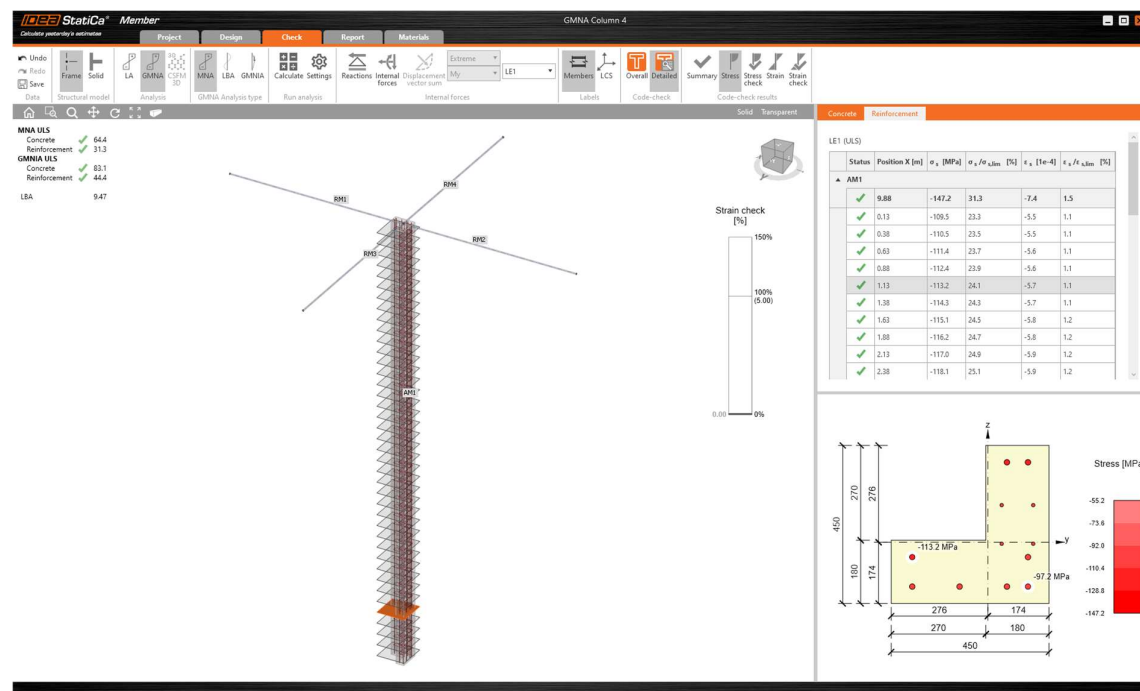
# IDEA StatiCa Member

## Design and check of slender columns (Member for concrete)

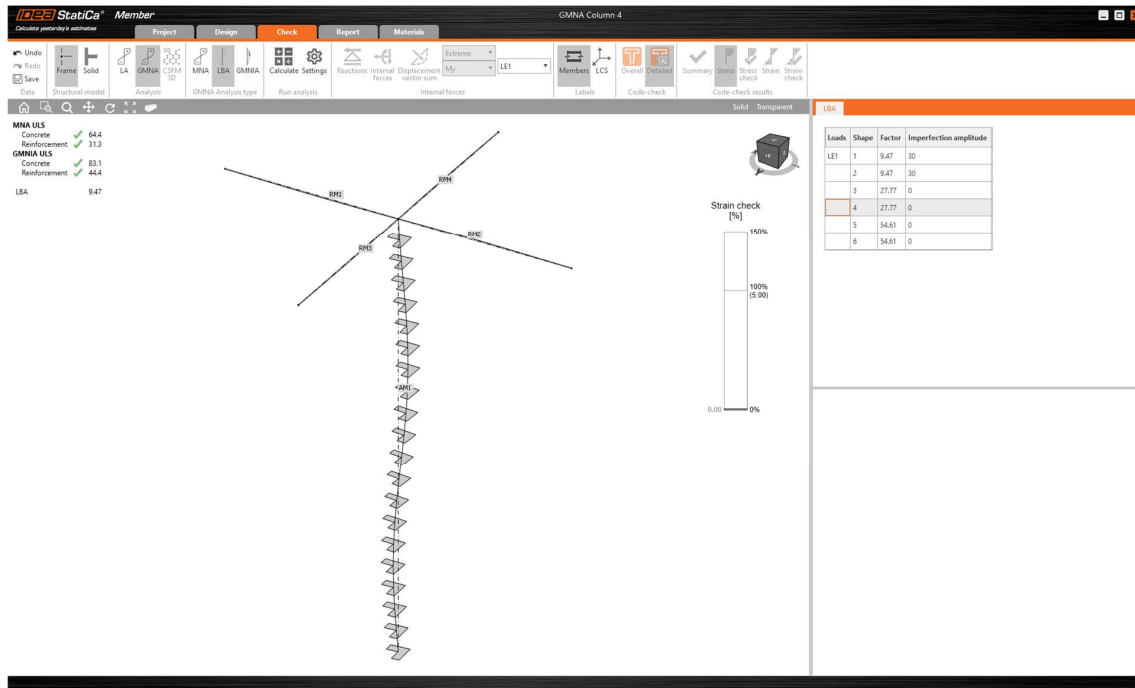
The geometrically and materially nonlinear analysis (GMNA) is a new type of analysis that allows code-check of members subjected to normal force and bending moments (such as columns) for ultimate and serviceability limit states. Limit values of stresses and strains used for code check respect the values defined in EN 1992-1-1.

The GMNA consist of three analysis types:

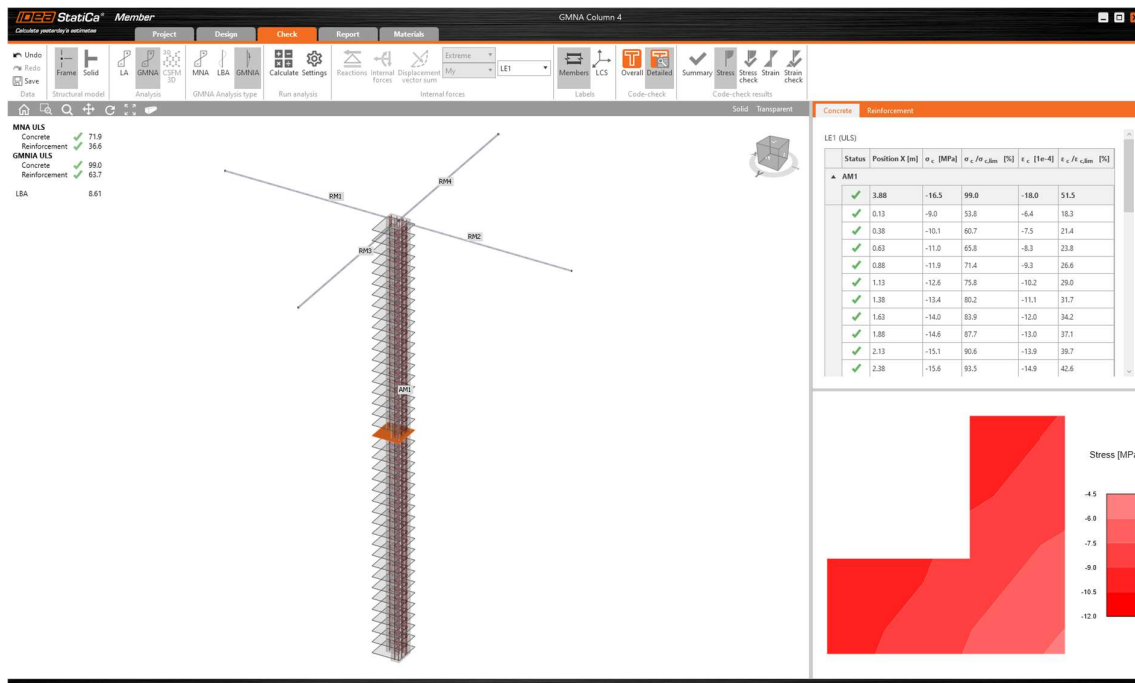
### 1. Materially nonlinear analysis (MNA)



## 2. Linear buckling analysis (LBA)

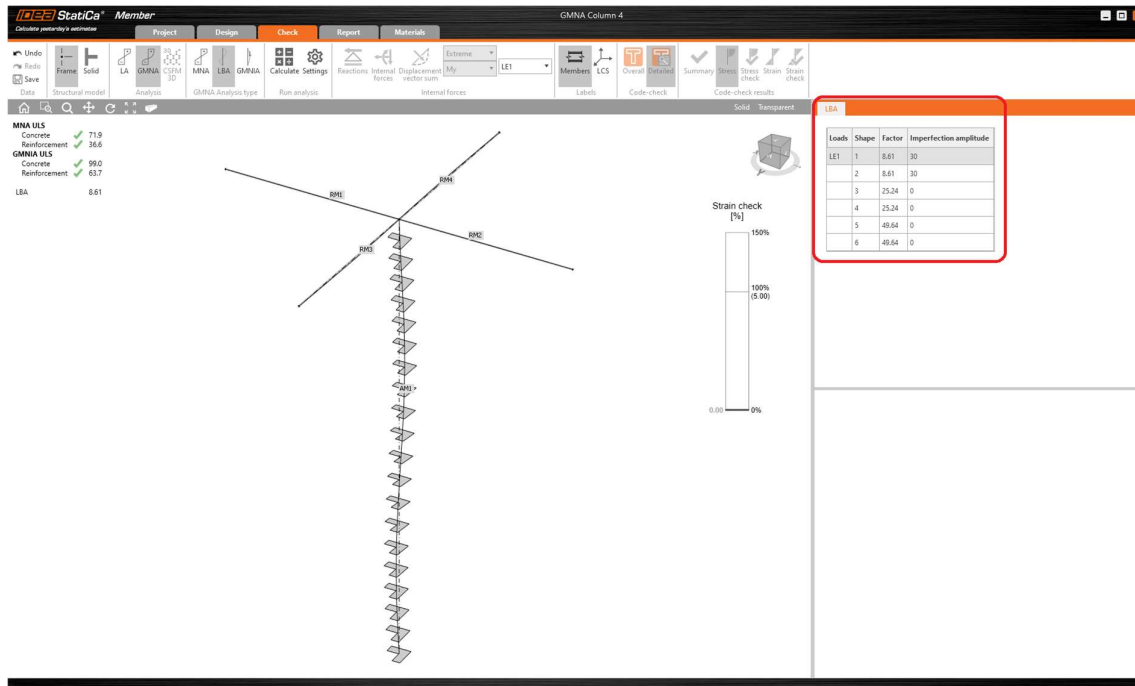


## 3. Materially and geometrically nonlinear analysis with initial imperfections (GMNIA)



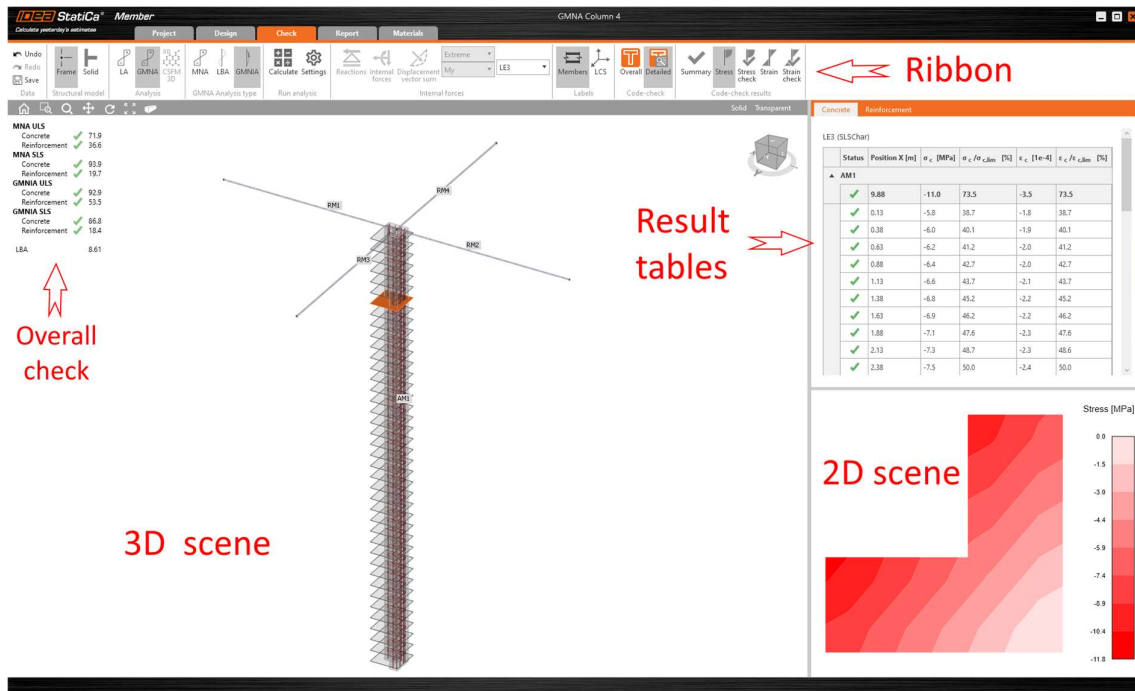
The inputs for the analysis include:

- The geometry of the structure, including boundaries
- Reinforcement definition
- Materials of concrete and reinforcement
- Load - only line load on the analyzed member is supported
- Initial imperfections - improved result table from Linear buckling analysis (LBA) allows defining the imperfection amplitude for each eigenvalue.

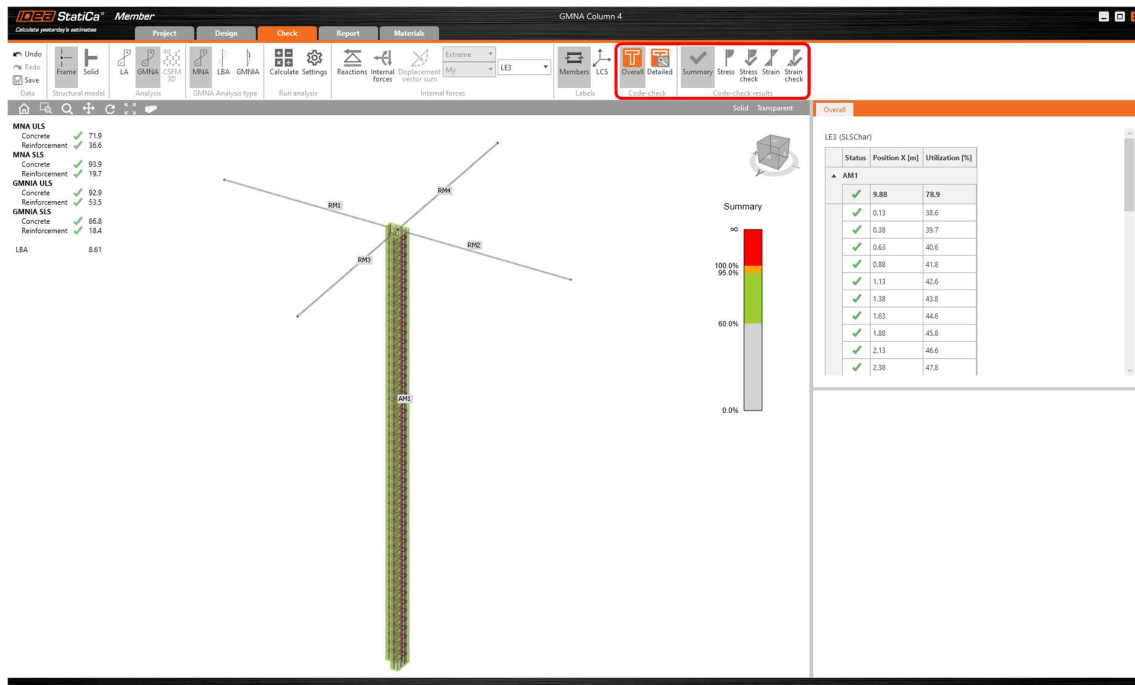


The results are presented in the Check tab, where the ribbon was reorganized and expanded by new buttons. Results in the 3D scene, result tables, and detailed results in the 2D scene are controlled by the buttons in the ribbon and by the tabs above the result table.





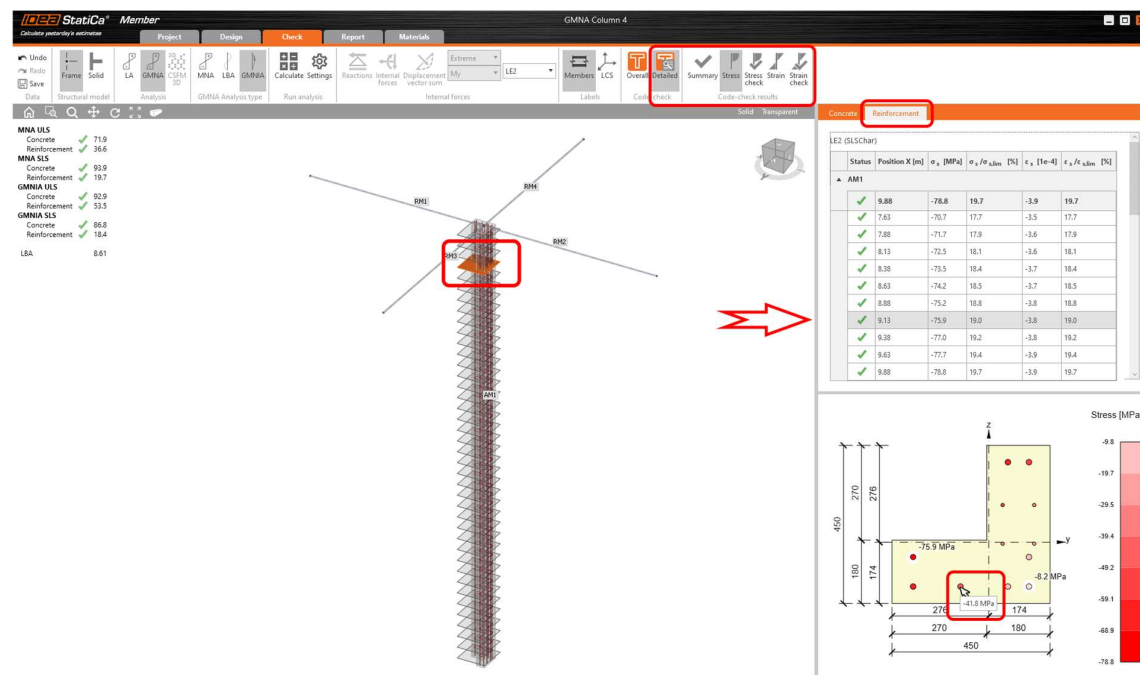
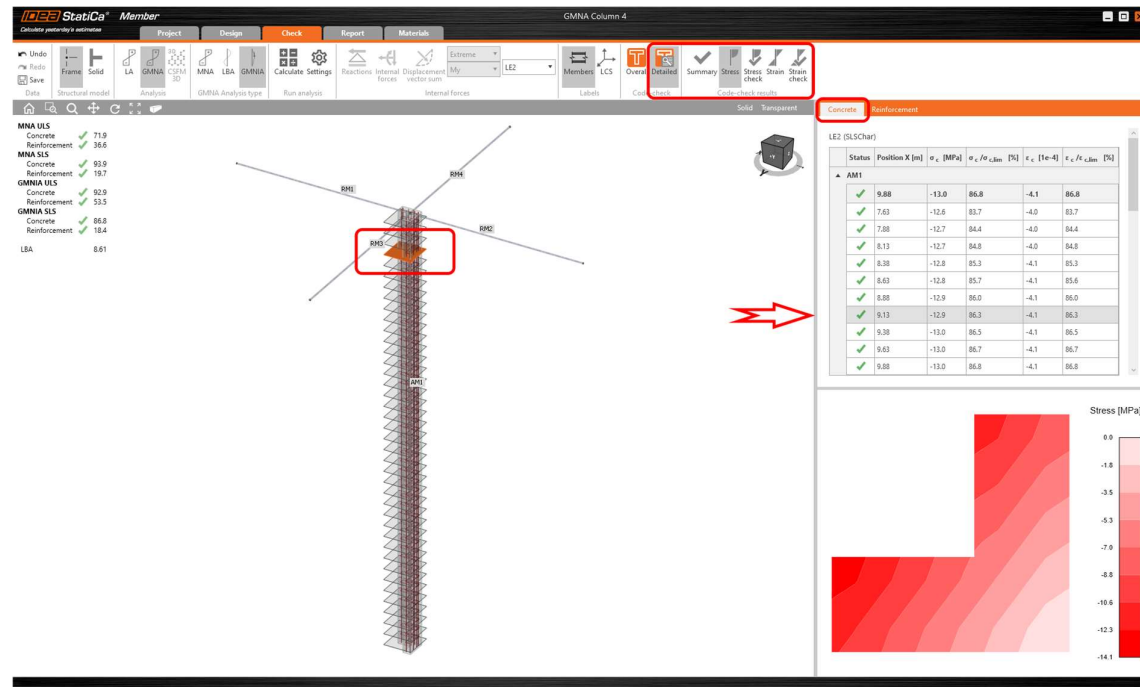
Users can display Overall or Detailed results. The Overall results show a summary of all the results in the structure, the user can immediately see whether the structure satisfies all code-checks or not.



In the Detailed or sectional results, the user selects the section in the 3D scene. The corresponding results are highlighted in the result table, and the detailed view of the selected section is displayed in the 2D window. Calculated stresses, stress checks, strains, strain checks

as well as Summary checks are presented for each section. Results for concrete and reinforcement are displayed separately, in a particular tab.

Design and check of slender columns in Member for concrete is available in **Concrete Expert**, **Concrete Enhanced**, **Prestressing Expert** and **Prestressing Enhanced** edition.

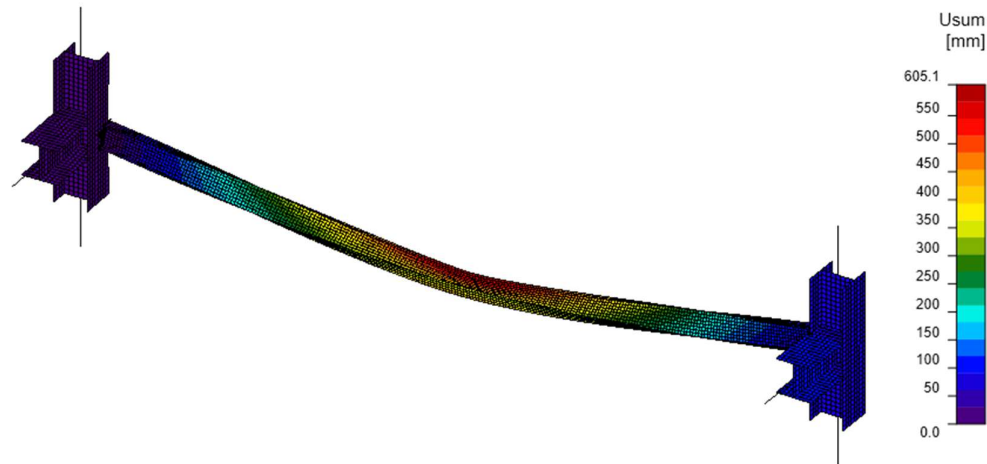


## Displacement vector summation in results

Add  $U_{\text{sum}}$  (displacement vector summation) to  $U_x$ ,  $U_y$ ,  $U_z$  in results. It is more useful than any of these 3. The summation  $U_{\text{sum}}$  is set as a default view, for steel as well as for concrete.

$$U_{\text{sum}} = \sqrt{U_x^2 + U_y^2 + U_z^2}$$

MNA	✓	100.0 %
MNA Plates	✓	0.0 < 5.0 %
LBA		0.33
GMNIA	✗	51.6 %
GMNIA Plates	✓	4.0 < 5.0 %



IDEA StatiCa Member for steel members is available in **Enhanced** edition. IDEA StatiCa Member for concrete members is available in **Expert** and **Enhanced** edition.

## Use global/local coordinates in supports settings

While defining supports of the model, you will have now an option to choose between the local system of coordinates (LCS) and the global system of coordinates (GCS). The local system is set as default.

This aims to remedy some difficulties with supporting the model. The feature is available for steel and concrete members.

IDEA StatiCa Member for steel members is available in **Enhanced** edition. IDEA StatiCa Member for concrete members is available in **Expert** and **Enhanced** edition.

**RM4** [Related member]
**Delete**

▼ **Properties**

Name	RM4		
Cross-section	2. HEB140 (Beam) ▼		
Offset ey [mm]	0.000		
Offset ez [mm]	0.000		
$\alpha$ - Rotation [°]	0.0		
Direction [m]	X 0.00	Y 0.00	Z -0.50
Length [m]	0.50		
Length 2 [m]	0.50		

▼ **Supports**

Coordinate system Local Global

Begin	X	<input checked="" type="checkbox"/>	Y	<input checked="" type="checkbox"/>	Z	<input type="checkbox"/>	Rx	<input checked="" type="checkbox"/>	Ry	<input checked="" type="checkbox"/>	Rz	<input checked="" type="checkbox"/>
End	X	<input checked="" type="checkbox"/>	Y	<input checked="" type="checkbox"/>	Z	<input type="checkbox"/>	Rx	<input checked="" type="checkbox"/>	Ry	<input checked="" type="checkbox"/>	Rz	<input checked="" type="checkbox"/>

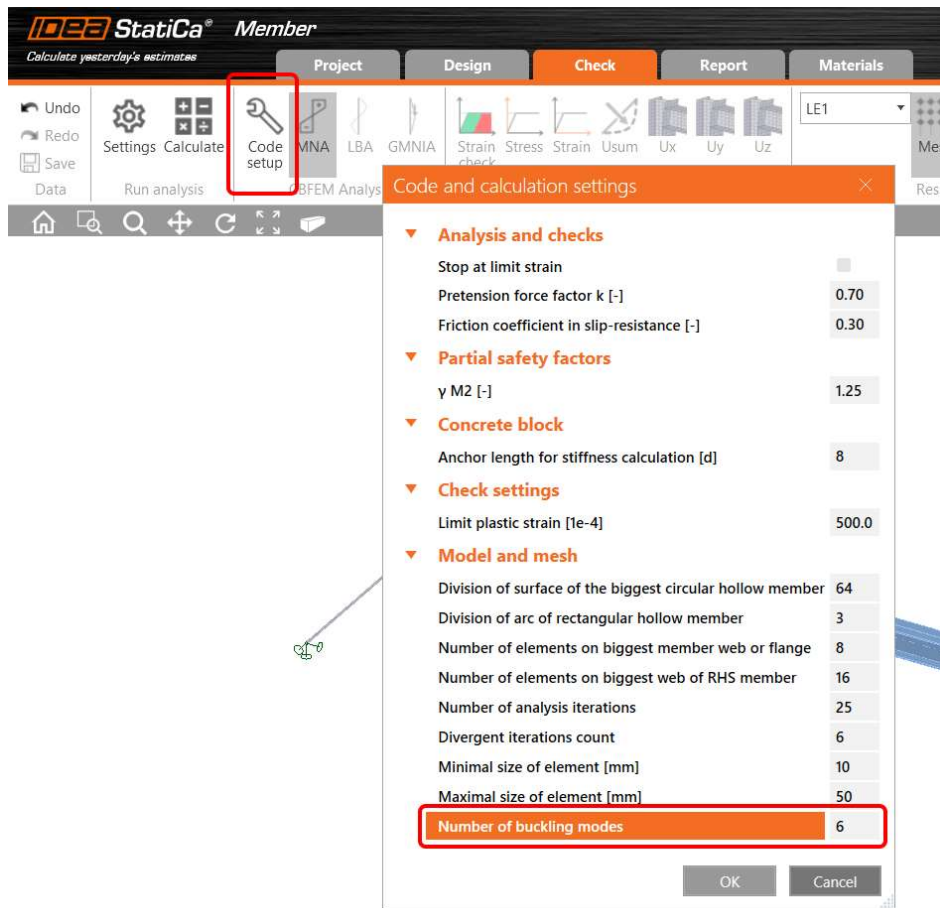
### Customizable number of buckling modes (Member for steel)

So far, the number of calculated and displayed buckling modes was set to six, which, in some cases, was not enough. While analyzing the complex members or class 4 cross-sections, this fixed value had complicated the engineers' lives.

The new version brings the possibility to change the number of modes (shapes and critical factors) evaluated in the linear buckling analysis (LBA): the value can be changed in Code setup in the range of 3–30, the default settings remain on 6.

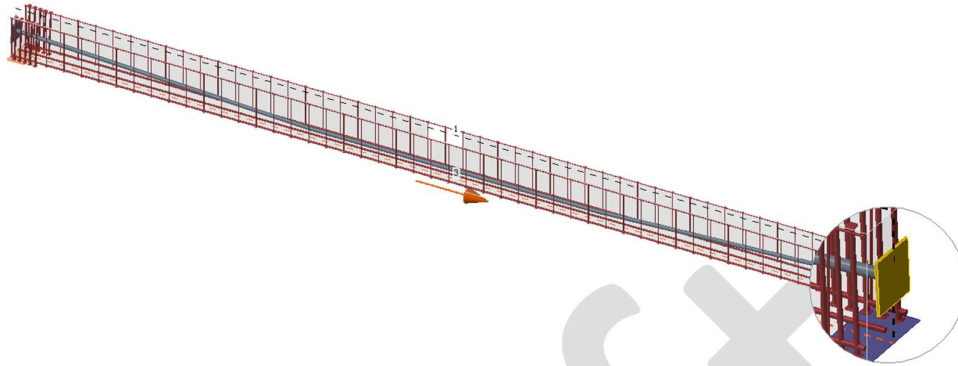
IDEA StatiCa Member for steel members is available in **Enhanced** edition.



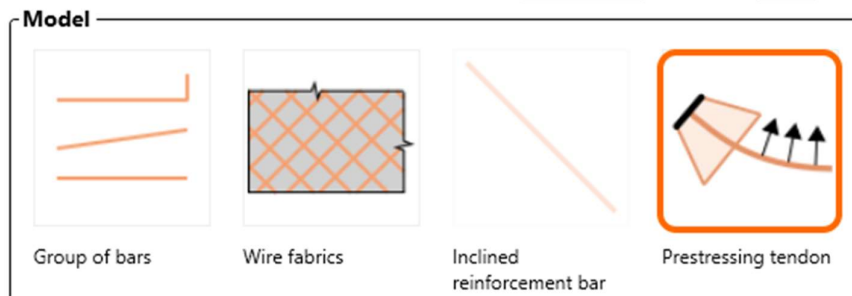


## IDEA StatiCa Detail and RCS

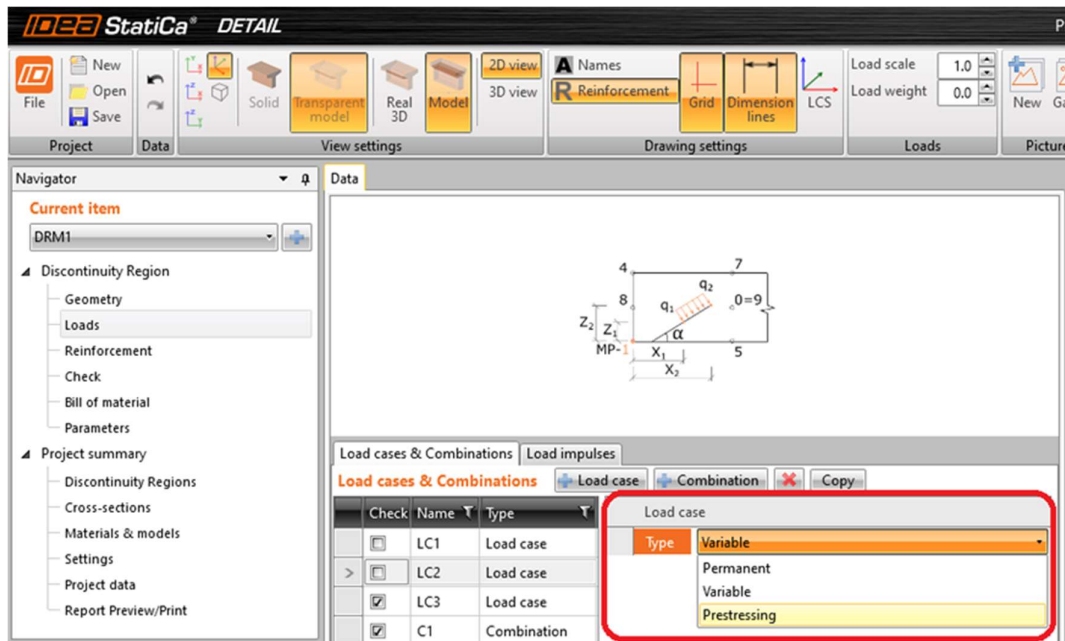
### Design of prestressed members (Detail)



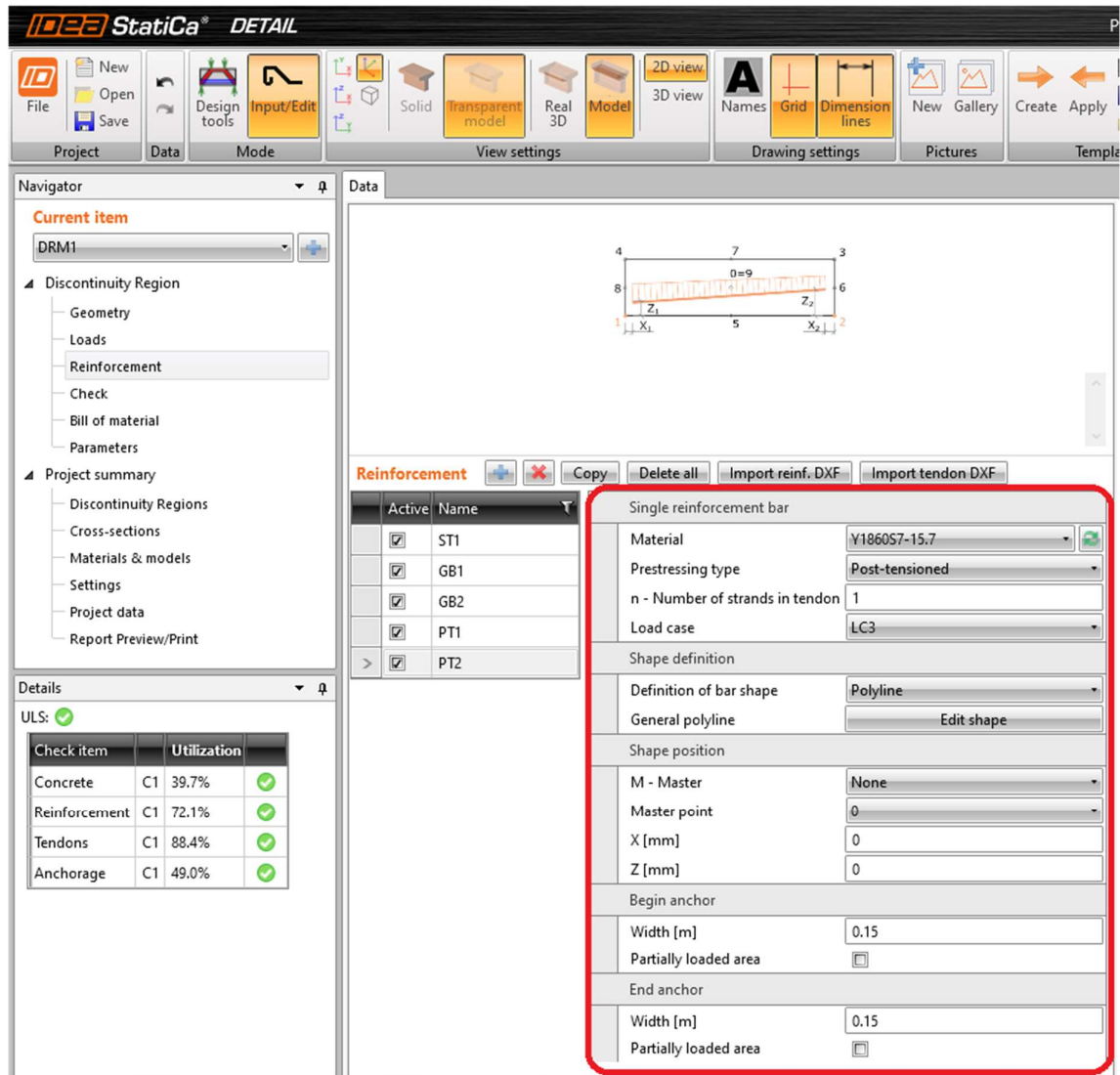
Users can now input prestressing tendons into their projects in Detail. This option makes the design of a prestressed structural member much more efficient since they do not have to calculate the equivalent load effects as it was until now. Simply input a prestressing load case, and the tendons become available.



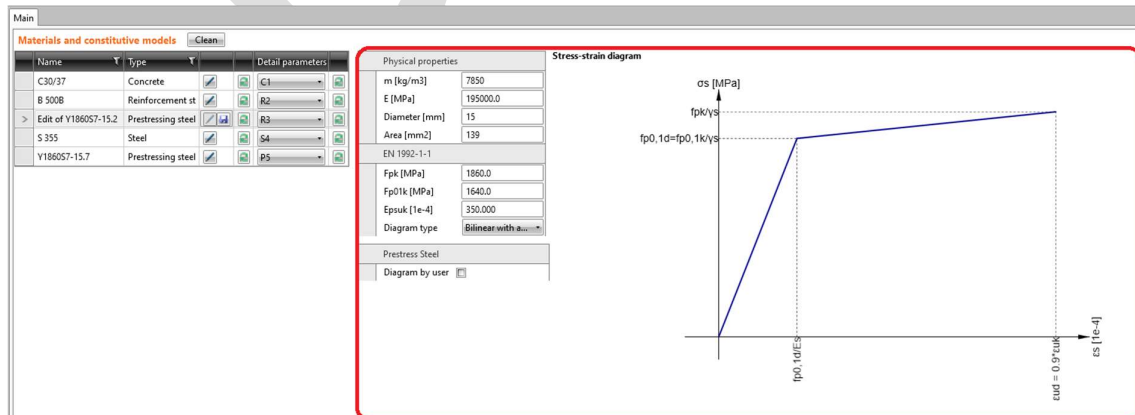
To enable this option, a Prestressing type load case must be defined.



Both pre-tensioned and post-tensioned tendons are available. You can import the geometry of the tendon from a DXF file.



We support bonded tendons and only ULS checks at this point. We also added a material library for tendons.



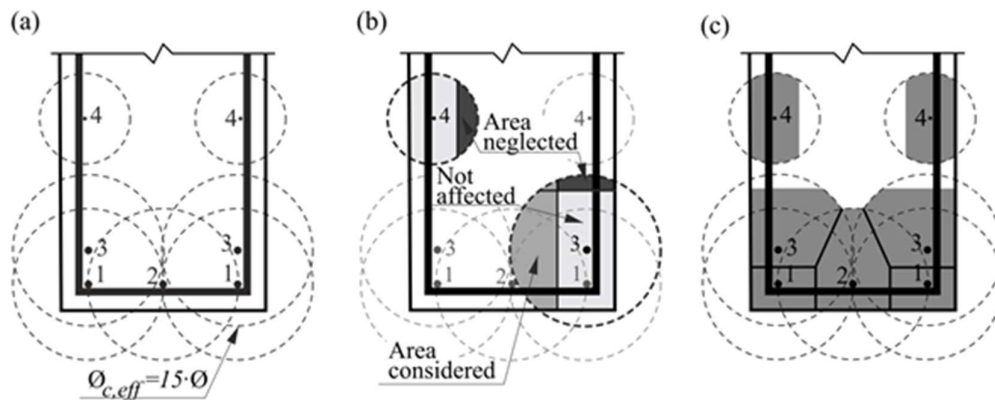
The prestressing losses need to be defined manually. The prestressing tendons can be modeled for the whole structure only. Prestressed parts of the beams (trimmed ends) are not supported.

Design of prestressed details is available in **Concrete Enhanced** and **Prestressing Enhanced** edition.

## Speeding up the CSFM calculation (Detail)

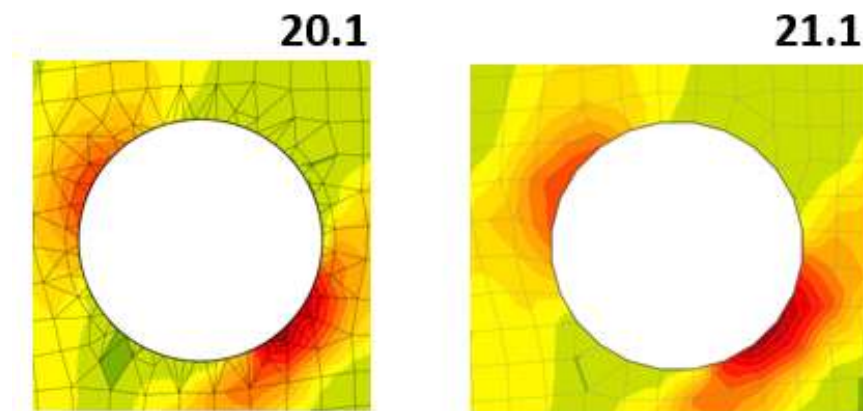
### New reinforcement ratio calculation

A new method behind the calculation of reinforcement ratio in Detail brings a significant improvement in the speed of calculations, reducing the time for analysis by approximately 30 %.



### Improved mesh around circular openings

The meshing around the circular openings in the structural details was improved in the new version. This results in not only faster calculations but also better convergence while keeping the results consistent with previous versions.



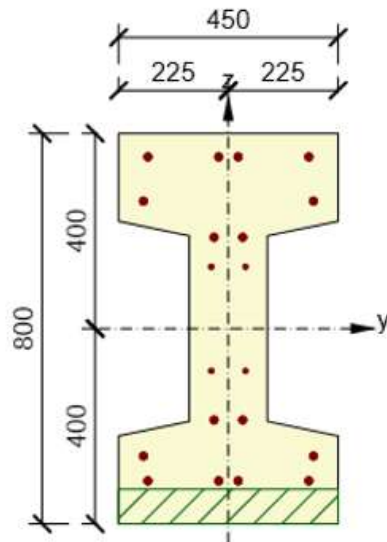
Speeding up the CSFM calculation is available in **Concrete Enhanced** and **Prestressing Enhanced** edition.

## Crack width check of cross-sections with a large concrete cover (RCS)

In previous versions, there were some situations when Eurocode methodology could not be applied, so we were not able to provide the users with the results for crack width in RCS based on the Eurocode conditions.

To calculate crack width, an effective concrete area must first be found ( $A_{c,eff}$ ). In some cases, this  $A_{c,eff}$  area did not contain any tensile reinforcement; therefore, cracks could not be calculated.

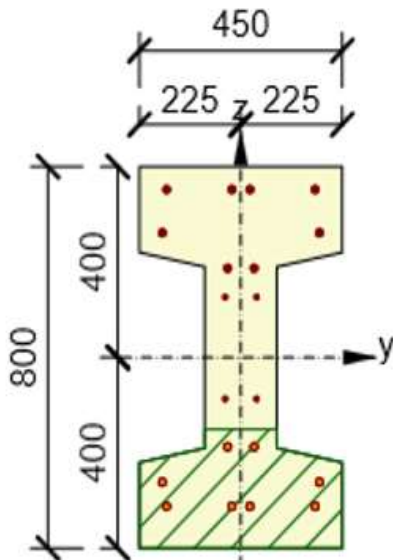
- $A_{c,eff} = \min \{2.5(h - d) ; (h - x)/3 ; h/2\}$



Check	Value	Status
Capacity N-M-M	37.8	✓
Shear	0.0	✓
Torsion	0.0	✓
Interaction	0.0	✓
Stress Limitation	42.2	✓
Crack Width	0.0	✗
Flexural slenderness	1000.0	✗
Detailing	110.7	✗

From version 21.1, if we run into this problem, RCS will run the calculation of  $A_{c,eff}$  again, but this time neglecting the condition  $(h-x)/3$ :

- $A_{c,eff} = \min \{2.5(h - d) ; h/2\}$



Check	Value	Status
Capacity N-M-M	37.8	✓
Shear	0.0	✓
Torsion	0.0	✓
Interaction	0.0	✓
Stress Limitation	42.2	✓
Crack Width	1.7	✓
Flexural slenderness	1000.0	✗
Detailing	110.7	✗

This improvement will lead to more consistent results for cracks in RCS, despite the limitations set in the code.

Crack width check of cross-sections with a large concrete cover is available in **Concrete Expert**, **Concrete Enhanced**, **Prestressing Expert** and **Prestressing Enhanced** edition.

Draft

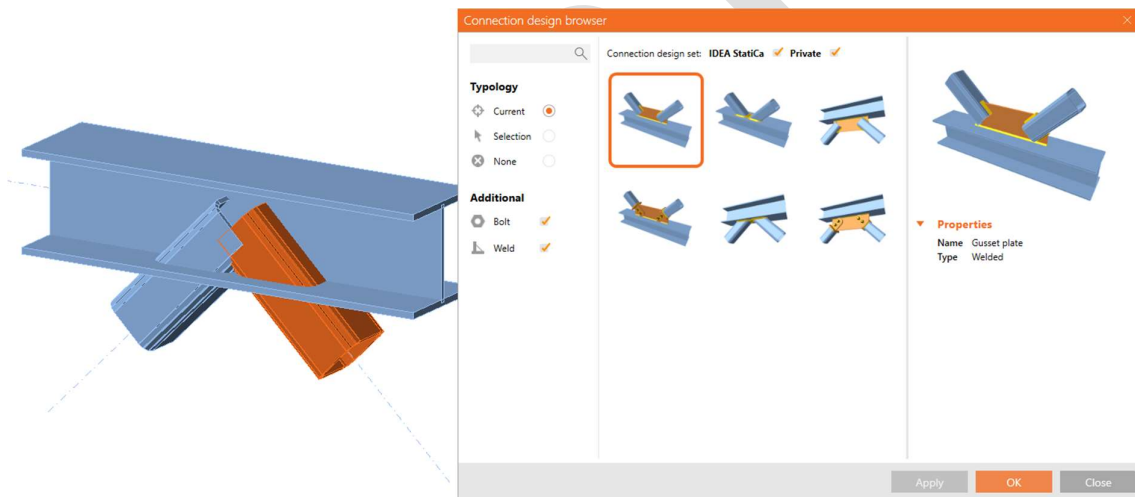
## IDEA StatiCa Connection

### Connection Browser will propose the suitable design

We are releasing the Connection Browser out of beta! This unique tool will help you find a suitable design solution from a library of predefined designs and directly apply it to your members. Connection Browser is going to replace the old Template manager in the future. Nevertheless, for the time being, you can find both in the top ribbon.

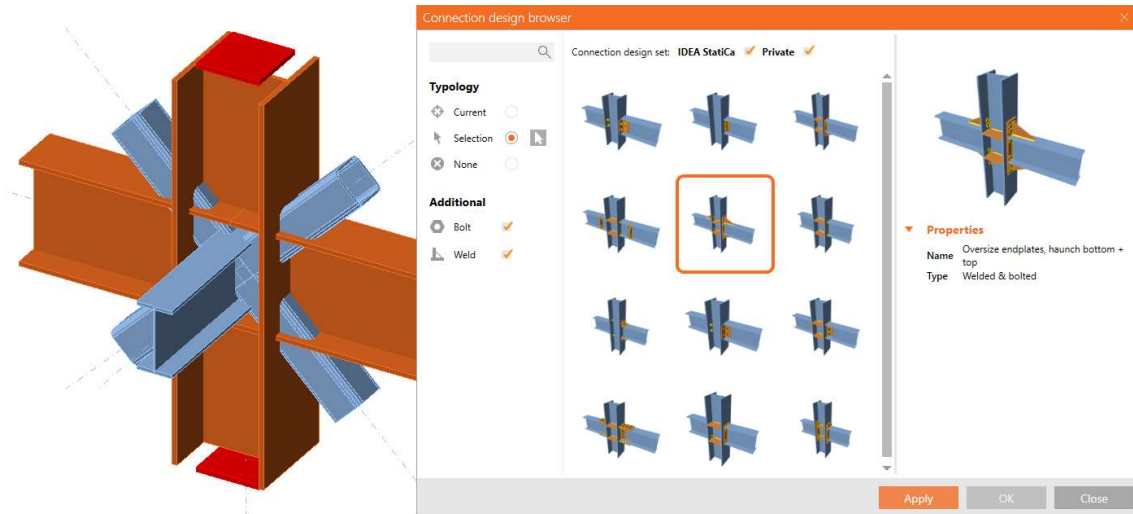


There are three new buttons dedicated to the Connection Browser. The **Propose** button opens the Connection design browser, a general library of saved designs. You can directly go through available designs for all current members in the topology of your connection.

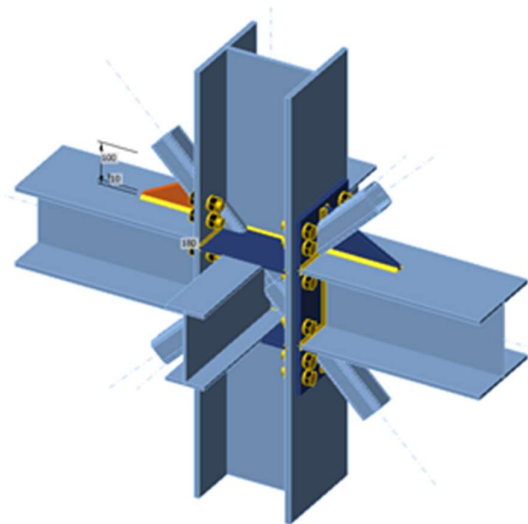


Nor the nonstandard topologies, try to select only a subset (i.e., two or three) members at a time. Switch the Typology to Selection and use the pick tool and hold CTRL to select multiple members in the scene. Confirm the selection with Enter/Spacebar or right mouse button. The proposer will filter the topologies based on your selection.

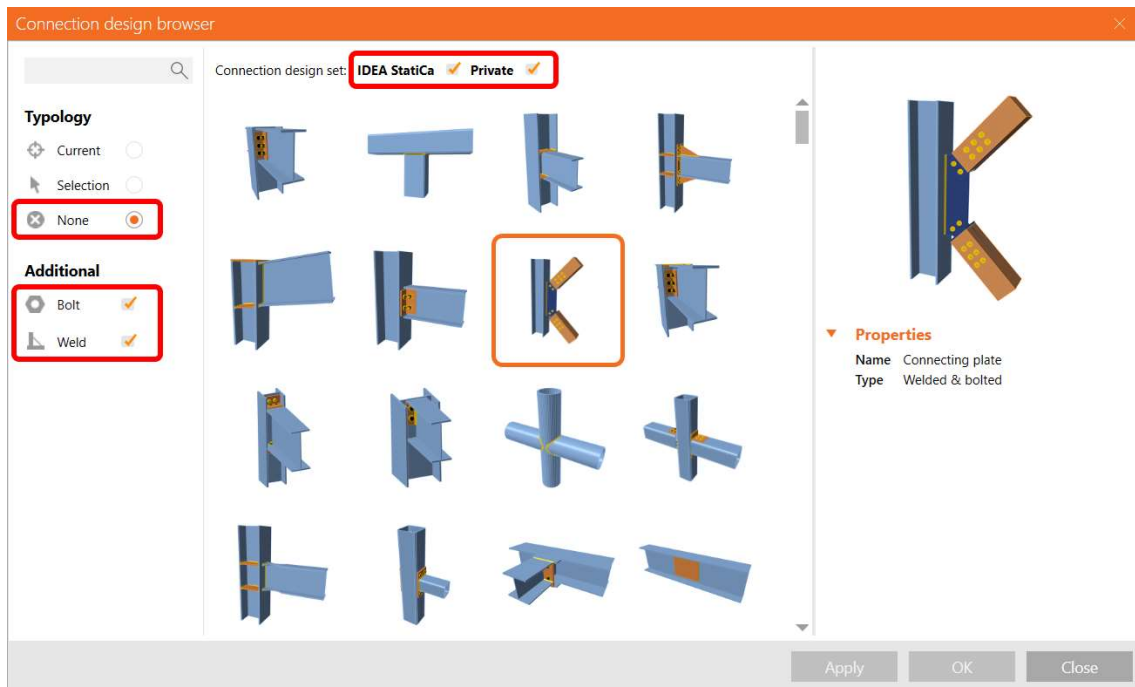




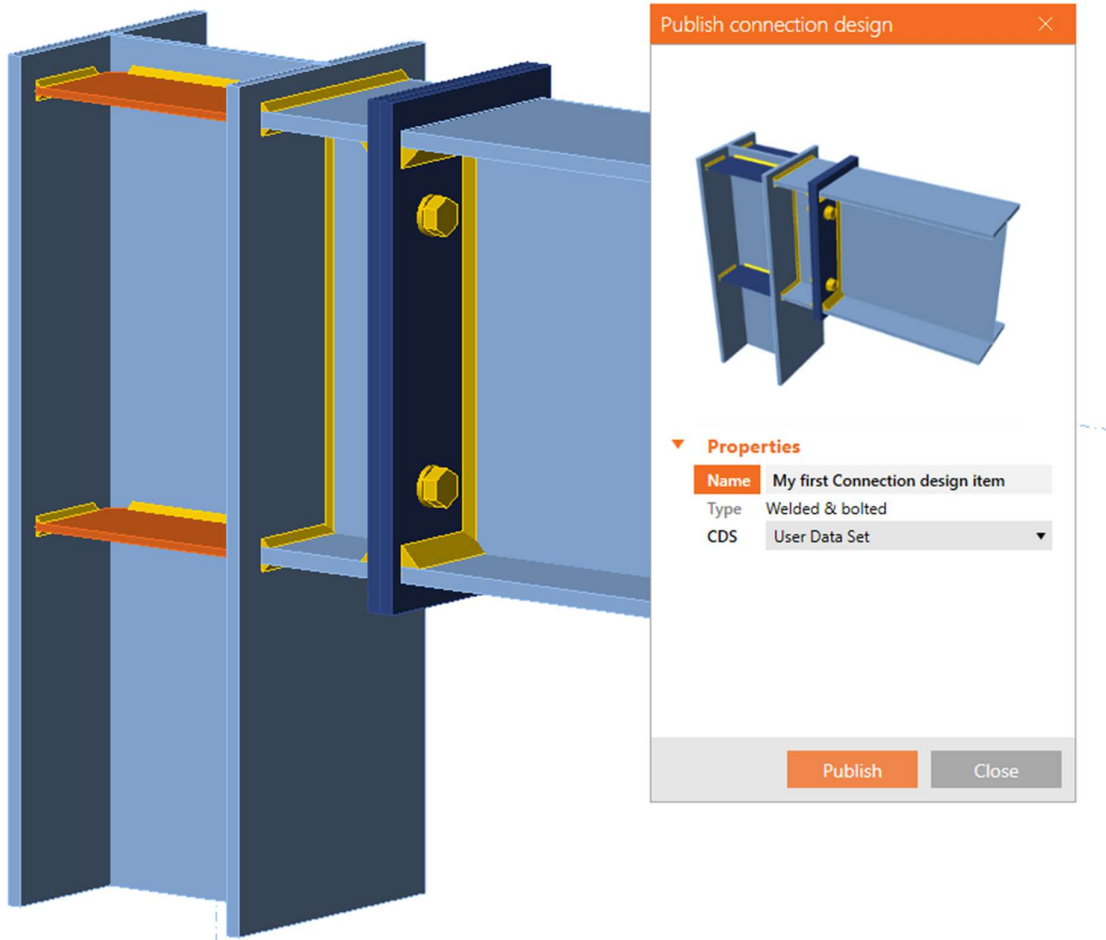
Once you find the suitable design, it is directly applied to the selected part of the joint.



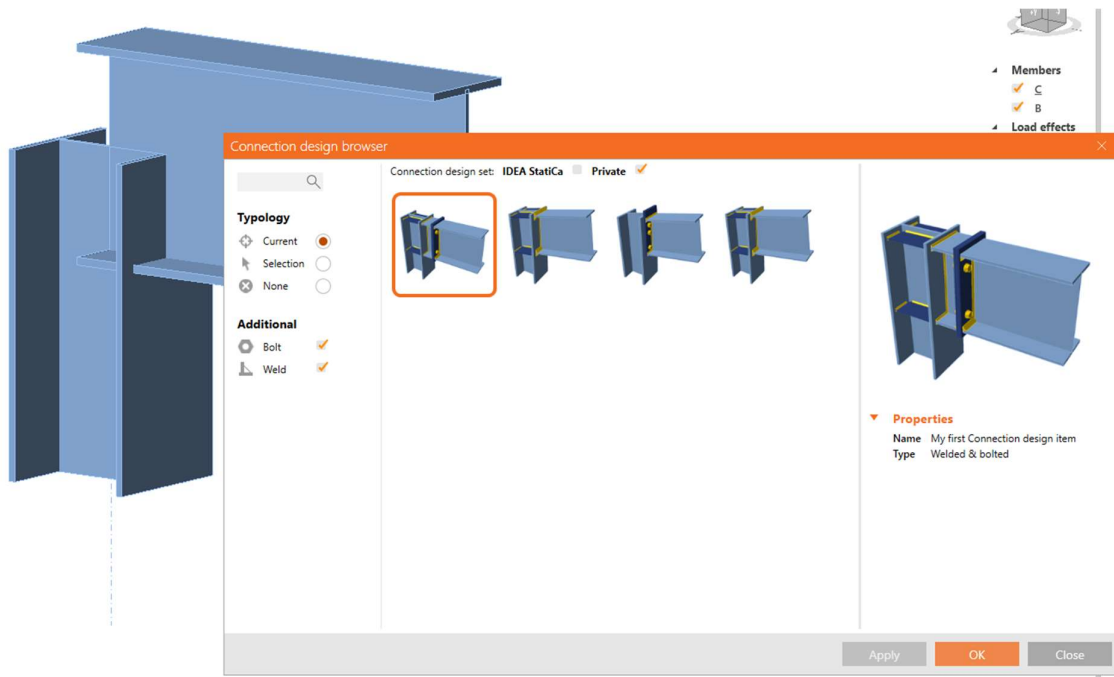
To display the full content of the library, switch the Topology to None. You can also filter the designs based on different criteria: Designs including bolts or welds or switch on/off the IDEA StatiCa default library of designs or your private (User-made) design set.



You can now add any created custom design into the library by selecting the **Publish** command from the ribbon. In the following dialogue, you can specify the Connection Design Set (CDS) where the design will be saved. At the moment, the design will be included into the user's private library – User data set. More will come in the future versions of IDEA StatiCa (the possibility to share the designs, etc.).



The newly added custom design will be available in the Connection Browser (under the Propose dialogue) for further use in the design workflow.



All the user connection design items can be managed by the **Manage** command in the ribbon: you can easily delete and edit the designs (in the current version, only the name and version can be modified, more will come soon).



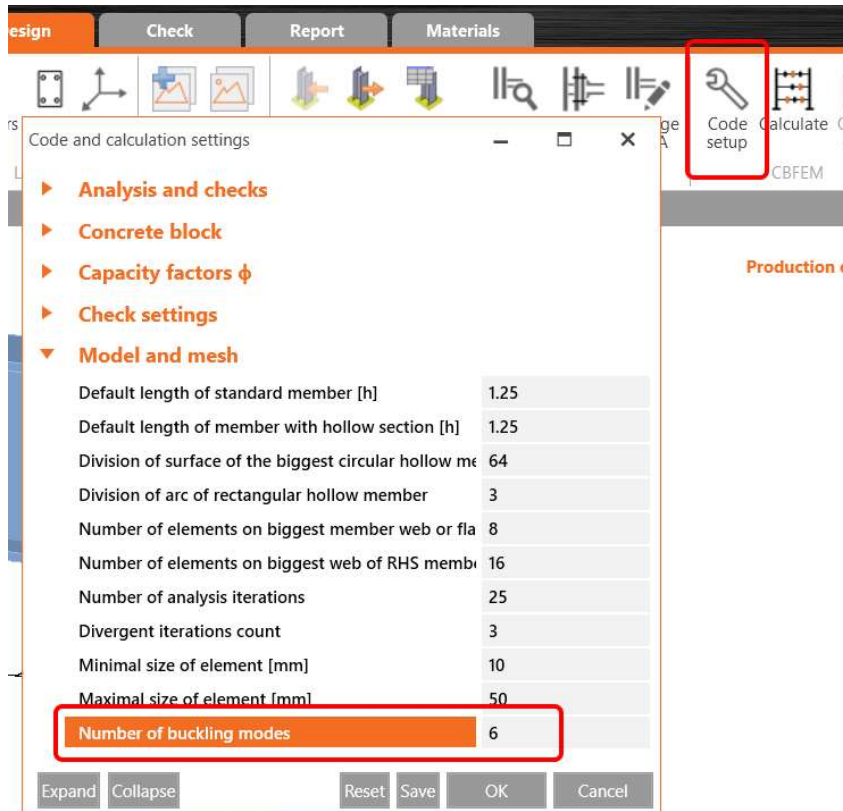
Available in **Expert** and **Enhanced** edition.

## Number of buckling modes

So far, the number of calculated and displayed buckling modes was set to six, which, in some cases, was not enough. When investigating the complex members or class 4 cross-sections, this fixed value had complicated the engineers' lives.

The new version brings the possibility to change the number of modes (shapes and critical factors) evaluated in the linear buckling analysis (LBA): the value can be changed in Code setup in the range of 3–30, the default settings remain on 6.

Available in **Enhanced** edition.



## Material database improvements (European steels according to product sheets)

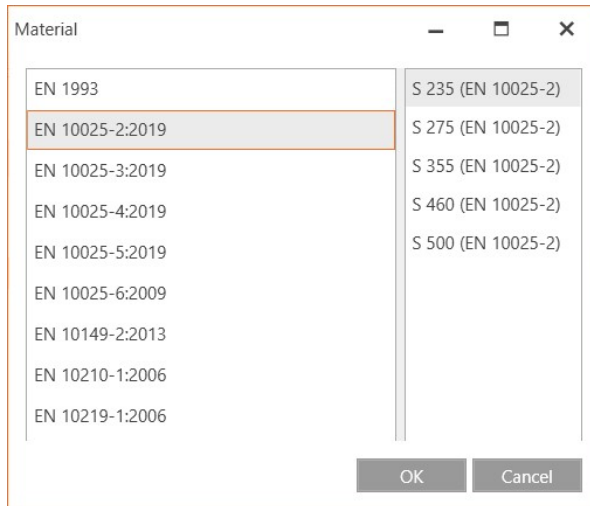
IDEA StatiCa material database (MPRL) was appended by European steel according to product sheets. This is useful especially for UK engineers where the National Annex of EN 1993-1-1 refers straight to steel product sheets, such as EN 10025-2.

List of implemented steel grades:

- EN 10025-2
- EN 10025-3
- EN 10025-4
- EN 10025-5
- EN 10025-6
- EN 10149-2

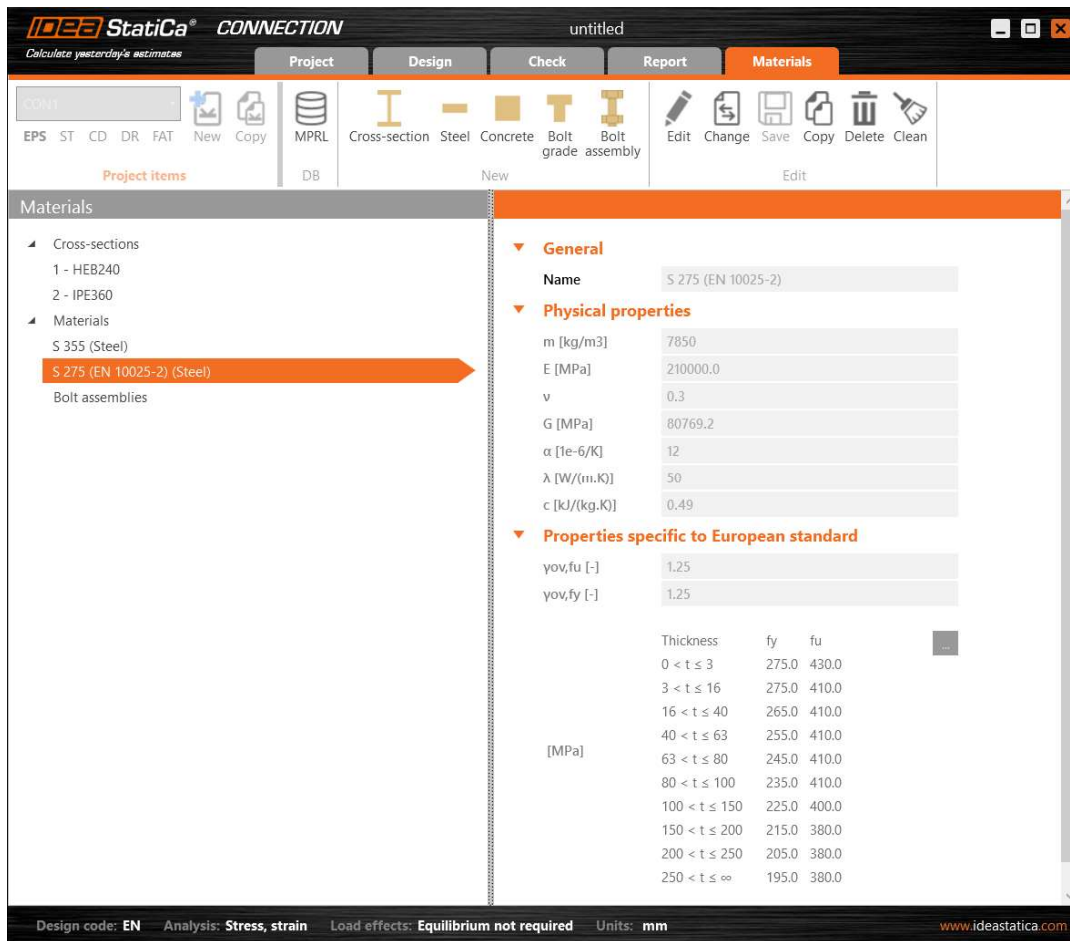
- EN 10210-1
- EN 10219-1

Steel grades according to EN 1993-1-1, which differ from the product sheets but are adopted by most countries using Eurocode, are without any change. Steel grades according to product sheets have their code name appended in brackets.



Thickness-dependent material strength is selected for each plate separately even though several plates belong to one member.

Strength of fillet welds is selected according to EN 1993-1-8, Cl. 4.5.3.2(6): tensile strength,  $f_u$ , is selected as a minimum of the tensile strength of two connected plates and the selected steel grade in weld manufacturing operation.



Updated in **Expert** and **Enhanced** edition.

## Weld check

To comply with the standards and to provide safety of the design, the strength value considered in the code check of welds is newly calculated from the strength value of the parent steel for EN and IS standards and the weld material itself.

Weld strength  $f_u$  (EN, IS) is taken as a minimum from:

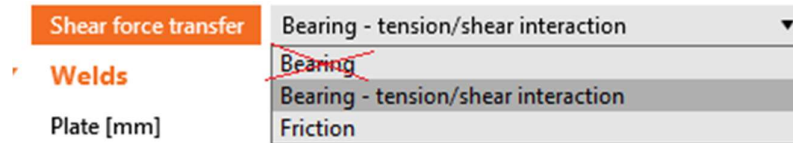
- $f_u$  of connected plate 1
- $f_u$  of connected plate 2
- $f_u$  of welding material selected in manufacturing operation

For other standards (AISC, CSA, AS, SP, GB, HKG), the strength is taken from weld electrode.

Updated in **Expert** and **Enhanced** edition.

## Bolts: Bearing type option removal

Since the 21.1 version, we will not consider the Bearing option of the shear force transfer in bolts anymore. For old projects where this type of bolt was set, the type will change from Bearing to Bearing – tension/shear interaction, which may change the results.



Updated in **Expert** and **Enhanced** edition.

## Licensing

### Obsolete applications

We decided to focus our development power on selected applications and discontinue the support of several outdated applications. These will not be included in 21.1 setups. Full functionality of all outdated applications remains included in the last 21.0 setup (21.0.4 patch).

Obsolete applications in 21.1:

- CCS
- Slab
- Corbel
- Frame
- Column
- Steel Beam
- Steel Designer (Idea.exe will remain for the Midas BIM link only)

Updated in **Expert** and **Enhanced** edition.