

# IDEA StatiCa 21.1

## Release notes

October 2021



# Contents

<b>NEWS FOR BIM.....</b>	<b>5</b>
CHECKBOT – BULK BIM WORKFLOWS.....	5
IMPORT CONFIGURATORS FOR CHECKBOT.....	7
SCIA ENGINEER BIM LINK WITH USING OF SAF INTERFACE .....	10
NEW RFEM LINK .....	10
VERSIONS OF CAD/FEA SUPPORTED IN 21.1 .....	10
<b>NEWS FOR CONCRETE AND PRESTRESSING .....</b>	<b>12</b>
DESIGN AND CHECK OF SLENDER COLUMNS.....	12
STRUCTURAL DESIGN OF PRESTRESSED DISCONTINUITY REGIONS.....	17
CRACK WIDTH CHECK OF CROSS-SECTIONS WITH A LARGE CONCRETE COVER .....	20
SPEEDING UP THE SOLVER OF IDEA STATICA FOR CONCRETE DESIGN .....	24
<b>NEWS FOR STEEL .....</b>	<b>25</b>
CONNECTION BROWSER – NEW LIBRARY OF CONNECTIONS .....	25
NUMBER OF BUCKLING MODES.....	29
EUROPEAN STEELS ACCORDING TO PRODUCT SHEETS.....	29
3D VISUALIZATION OF DEFORMATIONS .....	31
USE GLOBAL/LOCAL COORDINATES IN SUPPORTS SETTINGS.....	32
WELD CHECKS SPECIFICS AS PER EUROCODE (EN) AND INDIAN STANDARD (IS) .....	33
BEARING TYPE OPTIONS FOR BOLTS IN VERSION 21.1 AND ONWARDS .....	33
<b>LICENSING .....</b>	<b>34</b>
OBSOLETE APPLICATIONS FROM VERSION 21.1.....	34
<b>SOLVED INCIDENTS.....</b>	<b>34</b>

## 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? Push our BIM workflows to the next level, allow engineers to design more in reinforced and prestressed concrete and implement plenty of features for connection design we were asked to from around the world.

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. We also gave it a better name – the Checkbot. This slick-looking application is the new hub of IDEA StatiCa workflows with 3<sup>rd</sup> party software boosting the productivity of engineers working with our BIM links.

### IDEA StatiCa Checkbot gives you:

- ✓ Complete control over your imported connections and members
- ✓ Clear list of all imported items including status checked/not-checked
- ✓ 3D visualization of imported members and loads
- ✓ Conversion table for materials & cross-sections
- ✓ Load combinations management

The Checkbot can be started from your 3<sup>rd</sup> party application or as a standalone application and enables you to combine inputs from multiple sources. [Read more about the IDEA StatiCa Checkbot.](#)

## 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. [Read more about slender column design.](#)

IDEA StatiCa Detail has proven to be a unique structural design tool for critical members and details in reinforced concrete structures. Version 21.1 expands IDEA StatiCa Detail to pre-stressed concrete as well. Engineers can now understand, design and code-check discontinuities in pre-cast members and details. This dramatically decreases the time needed to design pre-stressed beams, diaphragms, etc. [Read more about designing prestressed discontinuity regions.](#)

Other improvements in concrete and prestressing include:

- [Speeding up the solver of IDEA StatiCa for the design of concrete discontinuities by up to 30 %](#)
- [Enhanced 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.

Connection Browser is here! This unique tool will help you find a suitable design solution from a library of predefined designs and apply them right away. The Connection Browser will work with three databases of steel connections. The first is the set defined by IDEA StatiCa in every installation. The second is the set of connection designs created and saved by each user. The third will be a company set of connections each of our customers can create and maintain (this will be released in one of the patches of the version 21.1. coming out in a couple of weeks). [Read more about the Connection Browser](#).

### Connection design and analysis improvements:

- [Custom number of buckling modes](#) – you can now set up to 30 buckling modes
- [European steels according to product sheets](#) – specifically for the UK market
- [3D visualization of deformations](#) of steel members
- [Use global/local coordinates in supports settings](#)
- [Improved code-check of welds](#) – as per Eurocode and Indian standard
- [Bearing type options for bolts](#) in version 21.1 and onwards

## Licensing

Some outdated applications will be removed from the 21.1 install file. [See the list of obsolete applications](#).

## Solved incidents

See the [current list of solved incidents](#) reported by our customers.

## News for BIM

### Checkbot – bulk BIM workflows

Our users around the world had one big wish – to get more clarity while working with all of your 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 the Checkbot!

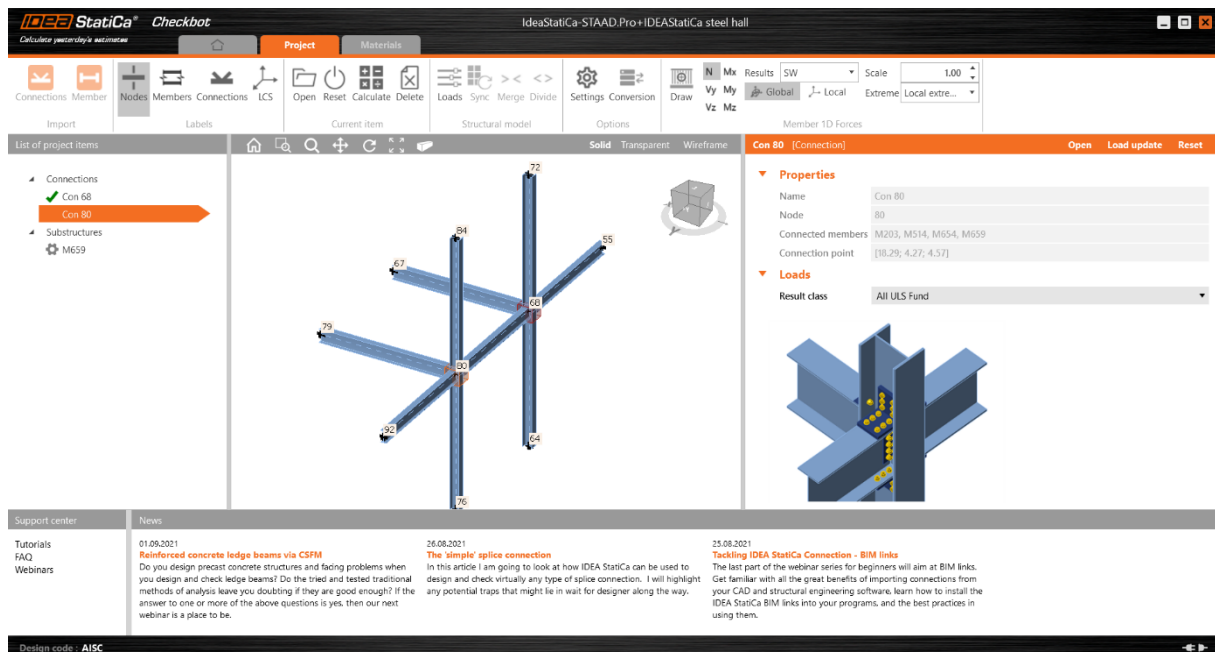
We are introducing a brand new application for the management of BIM workflows (importing and synchronizing connections and members). The Checkbot makes sure that you can work with all of your connections or members, steel or concrete, from your CAD/FEA applications. IDEA StatiCa Checkbot gives you:

- Complete control over all of your imported connections and members
- Clear list of all imported items including status checked/not-checked
- 3D visualization of imported members and loads
- Conversion table for materials & cross-sections
- Load combinations management

### The interface

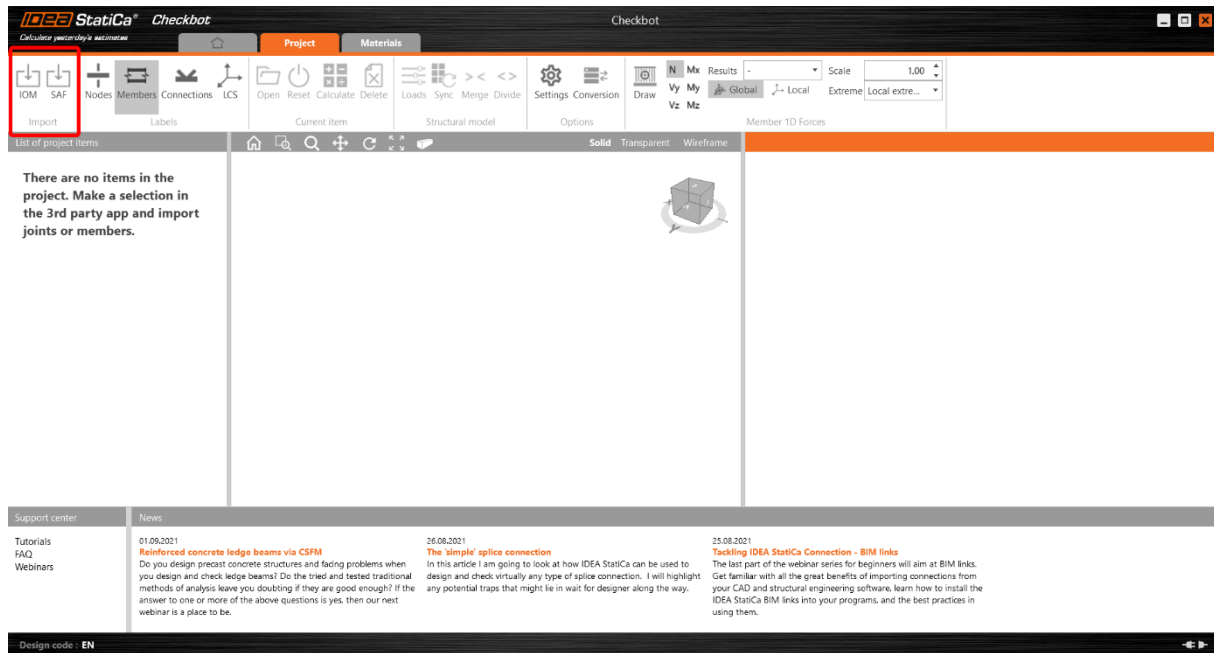


There are three main components of the new Checkbot interface - the list of design items on the left, the 3D scene in the middle, and the properties on the right.



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 cogwheel icon next to it,





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 an open solution for anyone who wants to link their application to IDEA StatiCa (via IOM – IDEA Open Model).

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

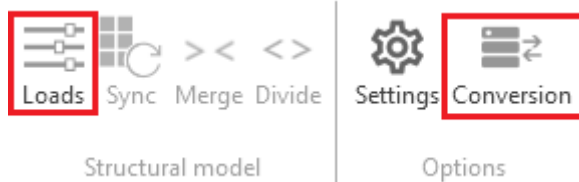
### Tips for best use of Checkbot

The Checkbot can handle an unlimited amount of connections and members. However, it is good to manage the volume of each individual import from an external application. Based on the amount of connections/members and load combinations, we recommend importing in batches of up to 100 connections/members in one go. Then review them and export another batch. When importing or synchronizing hundreds of connections/members, please respect that all the data has to be processed in the background and it might take some time.

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

You can easily manage the import of loads, cross-sections, and other materials. There are two new icons in the Checkbot ribbon.

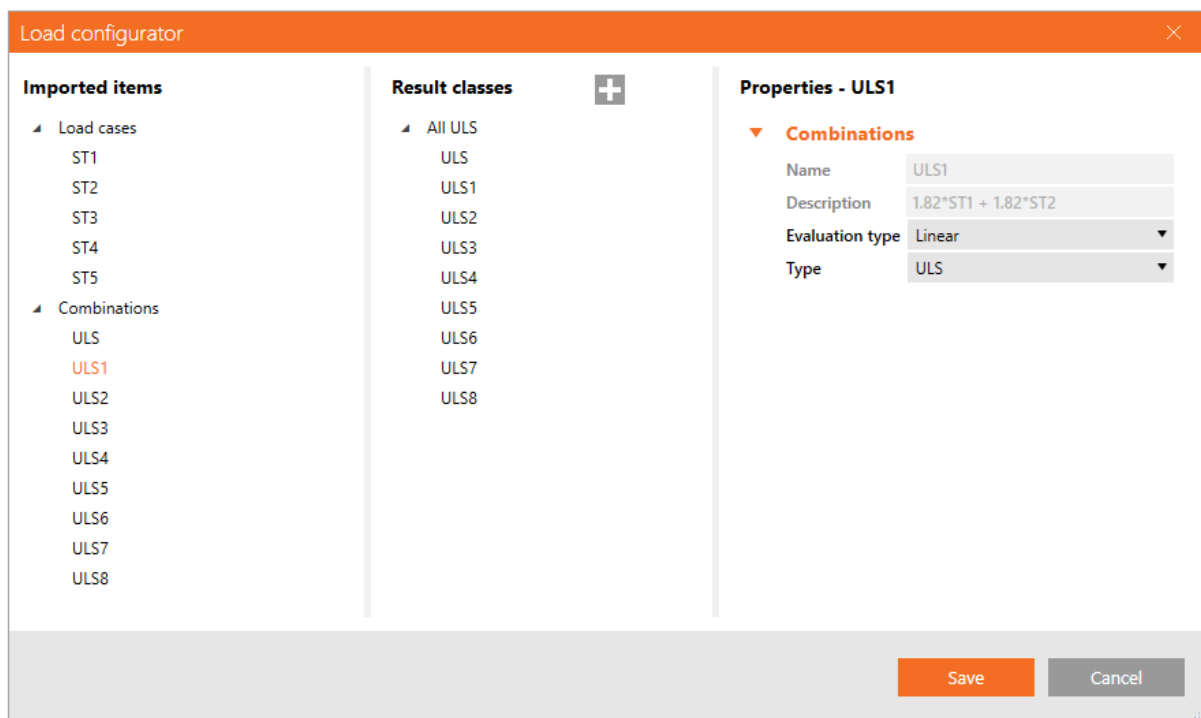


## Load configurator

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.

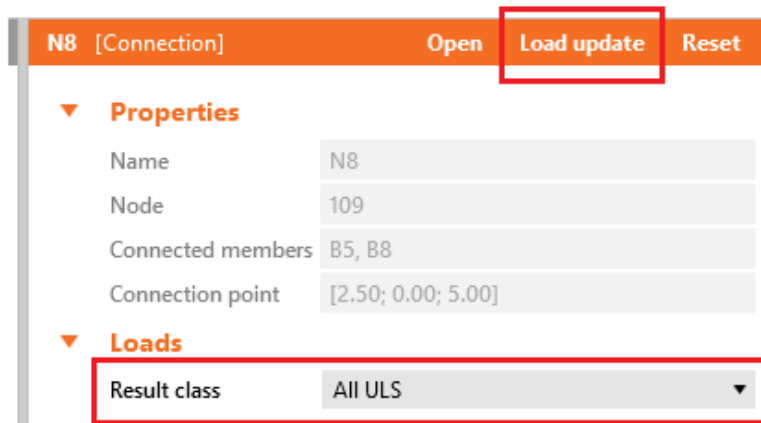
The first column lists all the Load cases and Combinations imported from the linked 3<sup>rd</sup> party application model. The second column shows the Result classes in the Checkbot project. The third column displays the properties of the currently selected entity.

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.

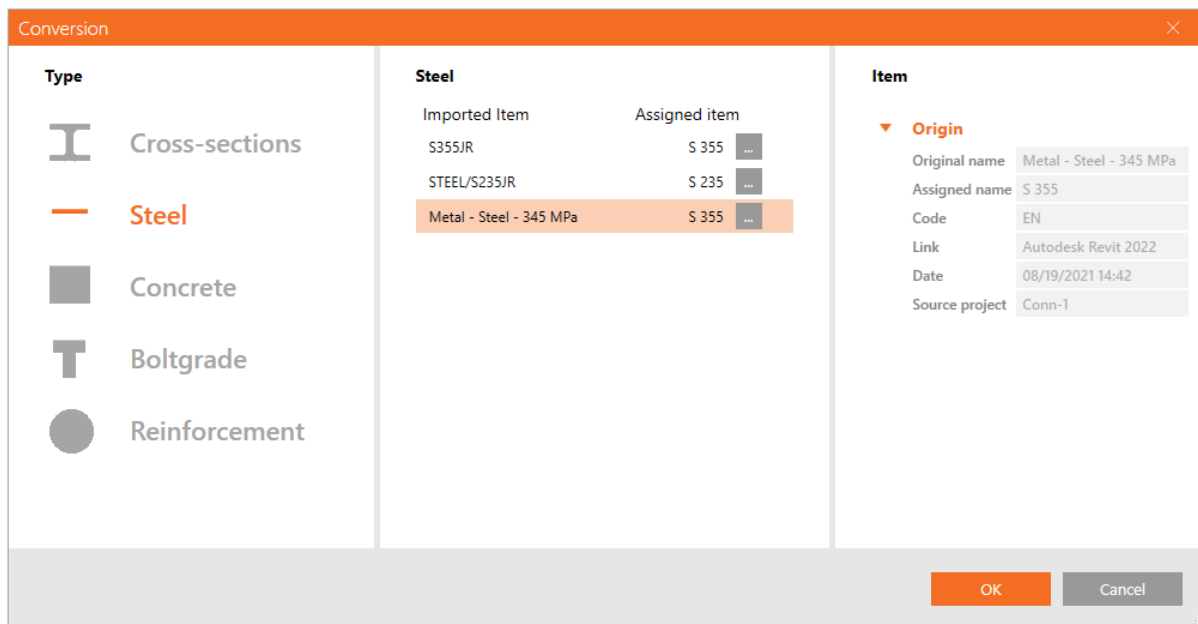




### Editable conversion tables

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.



### The installation

The Checkbot is automatically installed with the 21.1 setup of IDEA StatiCa. It also automatically detects and activates the relevant BIM links to applications installed on your PC.

IDEA StatiCa Checkbot is a part of both the **Expert** and **Enhanced** editions of IDEA StatiCa Steel. However, the functionality of the Checkbot in the **Expert edition is limited**:

- No BIM links to CAD applications (Tekla Structures, Advance Steel and Revit)

## SCIA Engineer BIM link with using of SAF interface

We have implemented an FEA link with SCIA built on the SAF format. This link is a result of close cooperation between Nemetschek and IDEA StatiCa companies. With the new version of SCIA Engineer 21.1, the Checkbot can be launched directly from SCIA, data in the SAF format will be transferred from the SCIA model into the Checkbot.



## New RFEM link

The integration between IDEA StatiCa and RFEM was reworked based on the up-to-date IOM (IDEA Open Model) framework. This will improve the stability of the link and remove obstacles in functionality.

## Versions of CAD/FEA supported in 21.1

We regularly update the BIM links between IDEA StatiCa and your CAD/FEA application. 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 best way to check the current state of the supported versions is the IDEA StatiCa website: [BIM links: Supported versions of 3rd party applications](#)

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 / Gen	2020, 2021	-	-
RFEM / RSTAB	5.25 / 8.25, 5.26 / 8.26	6.01 / 9.01	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 / Gen	2020, 2021	-	-
RFEM / RSTAB	5.25 / 8.25, 5.26 / 8.26	6.01 / 9.01	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.

## News for Concrete and Prestressing

### Design and check of slender columns

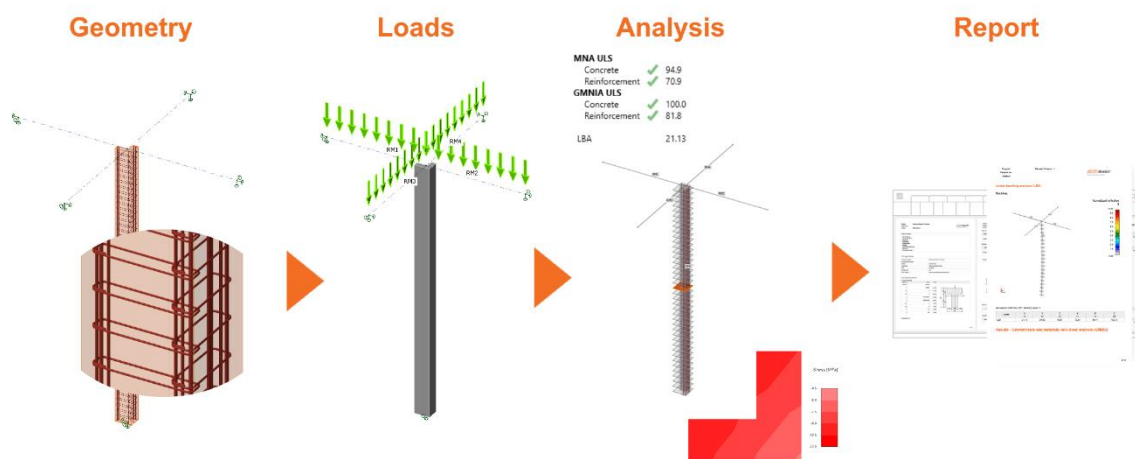
Structural engineers have to balance their load-bearing structure designs to be economical and safe. Nowadays, they need to deal with more and more slender and subtle structures either due to saving costs or satisfying architectural requirements.

What does the code say about a slender column subjected to compression and bending in both directions? How should engineers safely design and check such a column? You can use simplified methods, which may lead to a conservative design due to providing greater second-order effects. Or you can go for a more accurate approach, a general method based on nonlinear analysis. But the workflows for the general method are not described in the code. The code expects the engineers to cope.

You can now analyze slender columns via a more advanced method using both geometric and material nonlinear analysis in IDEA StatiCa. A new type of analysis allows the design and code-check of members subjected to normal force and bending moments for ultimate and serviceability limit states. The checks are performed at the comparison of stresses and strains with the limit values given by the standard, EN 1992-1-1.

Follow an easy four-step workflow to make the design and check of slender columns more understandable and engaging:

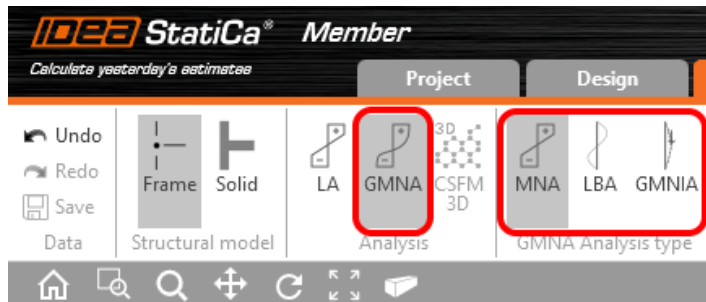
1. Create geometry of the structure, including boundaries and reinforcement
2. Subject analyzed and related members to the load effects
3. Nonlinear analysis and the assessment
4. Report with all important figures, results and checks



### Analyses types

The structure's geometry is composed of the members, boundary conditions and results. They can be created in the Member application or imported from 3<sup>rd</sup> party software (like Robot, SAP200, RFEM, etc.). After that, you add reinforcement to the member and start analyzing

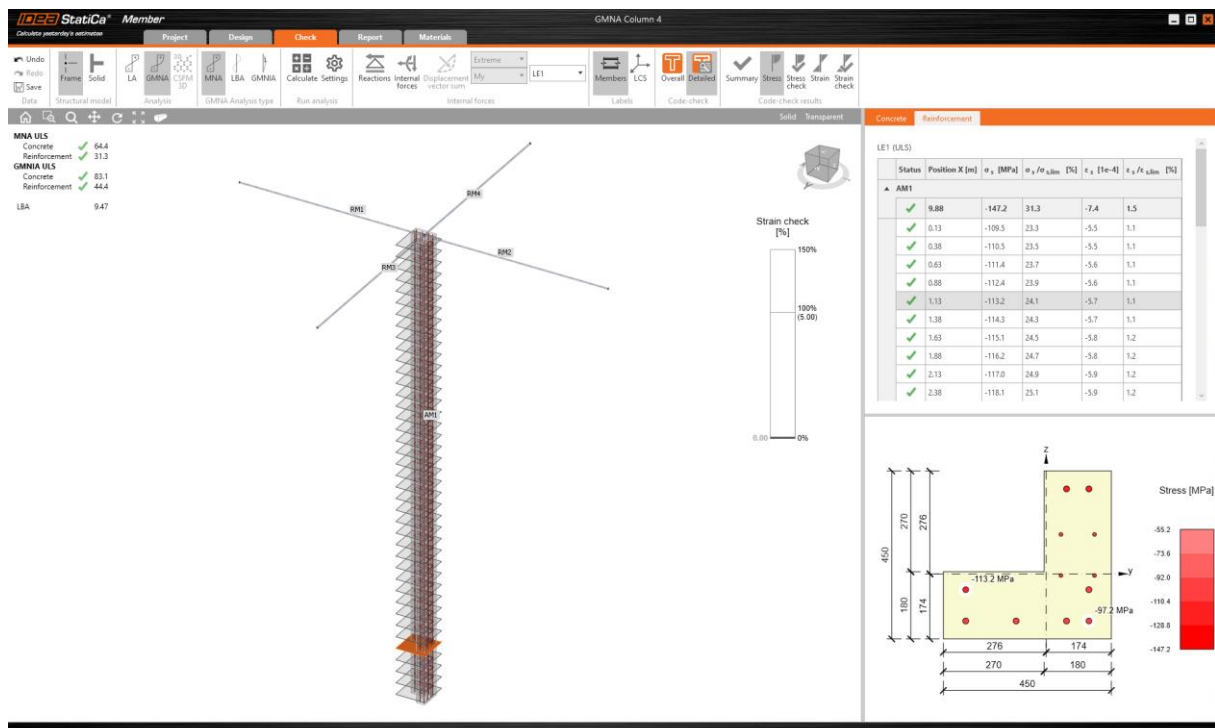
using the results obtained from the structural analysis. For slender columns, where linear analysis is insufficient, use nonlinear analysis which is available under the GMNA button.



The GMNA itself consists of three analysis types:

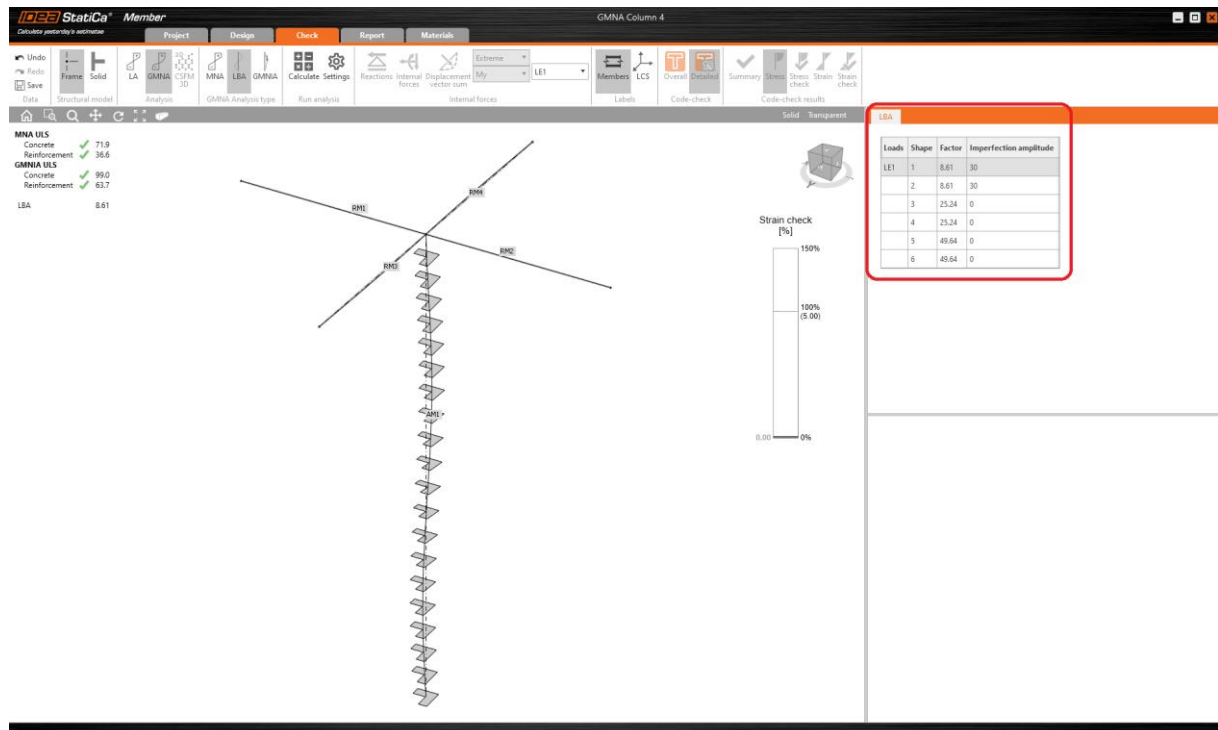
- Materially nonlinear analysis (MNA)
- Linear buckling analysis (LBA)
- Geometrically and Materially Nonlinear Analysis with Imperfections (GMNIA)

What are all these analyses for? What is an engineer supposed to do with it? You may proceed as the buttons in the ribbon suggest. First, material nonlinear analysis (MNA) is performed. An analyzed member is automatically divided into several sections where the stresses and strains are evaluated for each concrete fiber and reinforcement bar. Then, the provided values are checked with the limit values defined by the code.

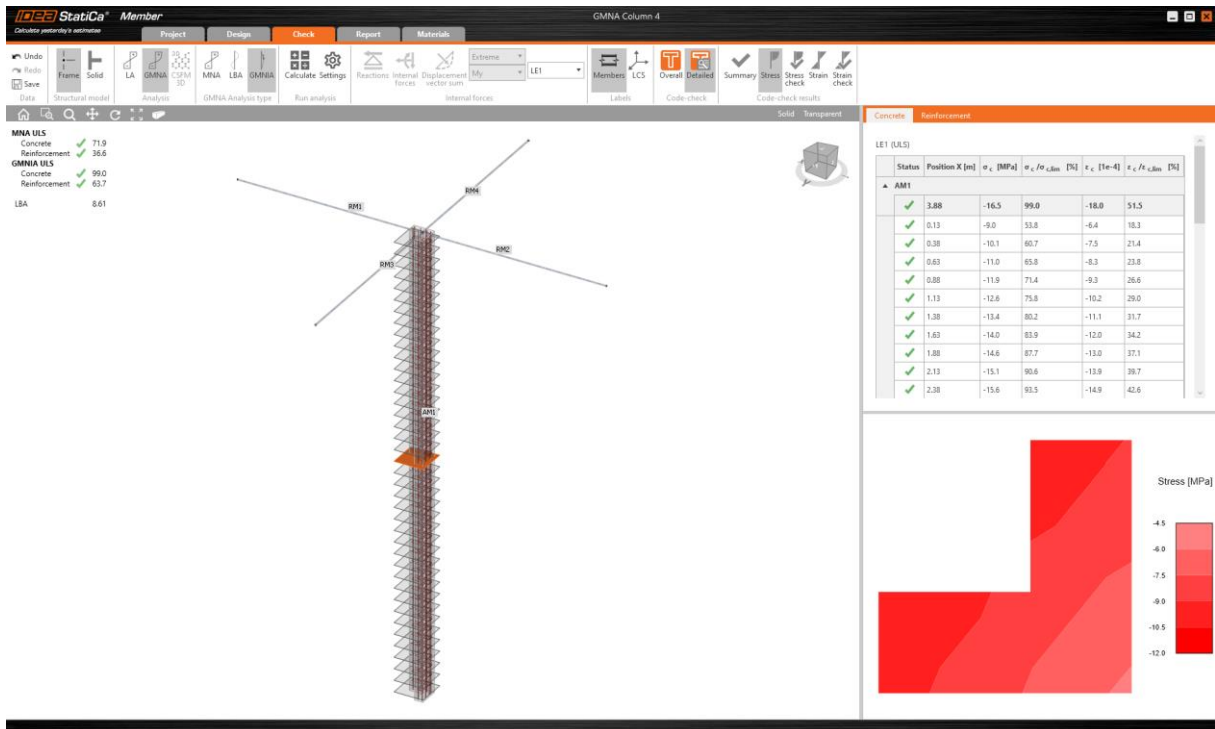


If it is not enough to consider only material nonlinearity, and it is also necessary to consider geometric one then in that case, it is time to use the linear buckling analysis (LBA), whose outputs are eigenmodes and critical loads of the analyzed member. This analysis helps an engineer determine the theoretical loss of stability of a structure subjected to acting loads. It

would not be safe to consider only the theoretical buckling shape of the structure since there are initial imperfections. That is why the result table enables the definition of imperfection amplitude for each eigenmode. The imperfection shall be defined by the engineer based on the experience or code recommendation.

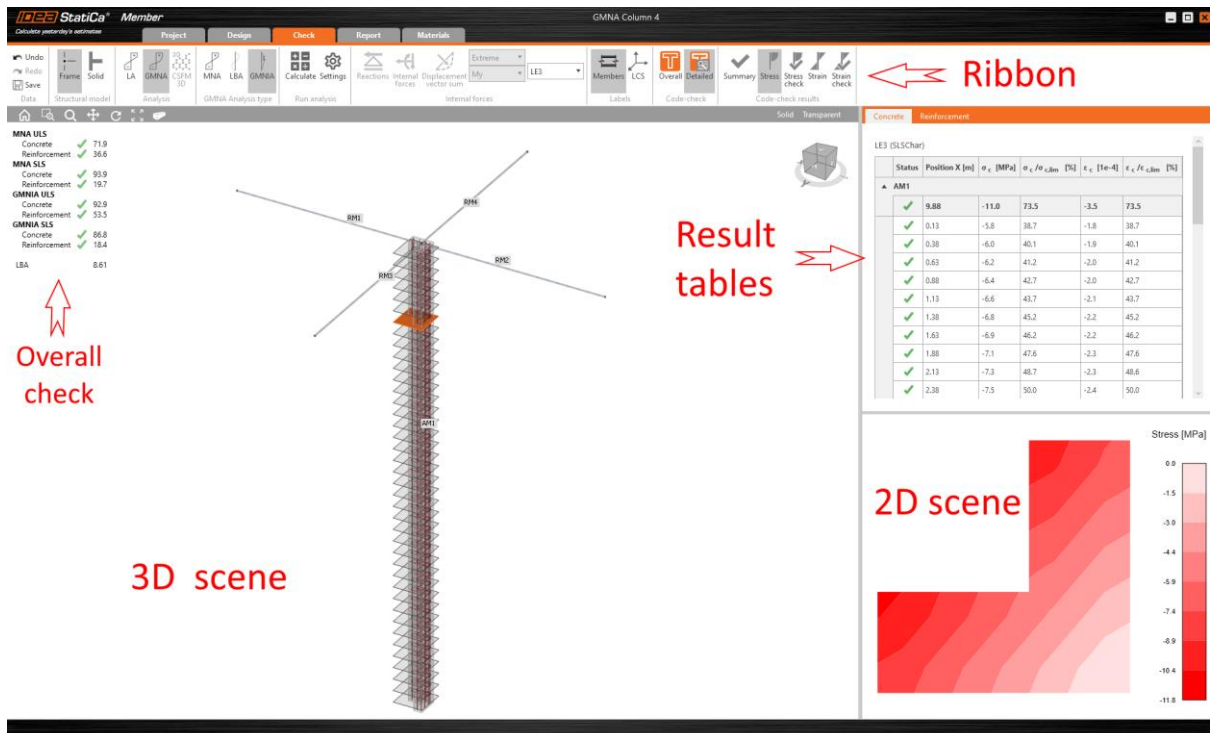


Once the imperfection is defined, it is automatically proportionally applied to the member, and you can perform the last analysis type – geometrical and material nonlinear analysis with imperfections (GMNIA). Similarly, as the first analysis (MNA), the member is divided into sections where the stresses and strains for each concrete fiber and reinforcement bar are evaluated, taking into account material and geometrical nonlinearity resulting from buckling shape with imperfections.

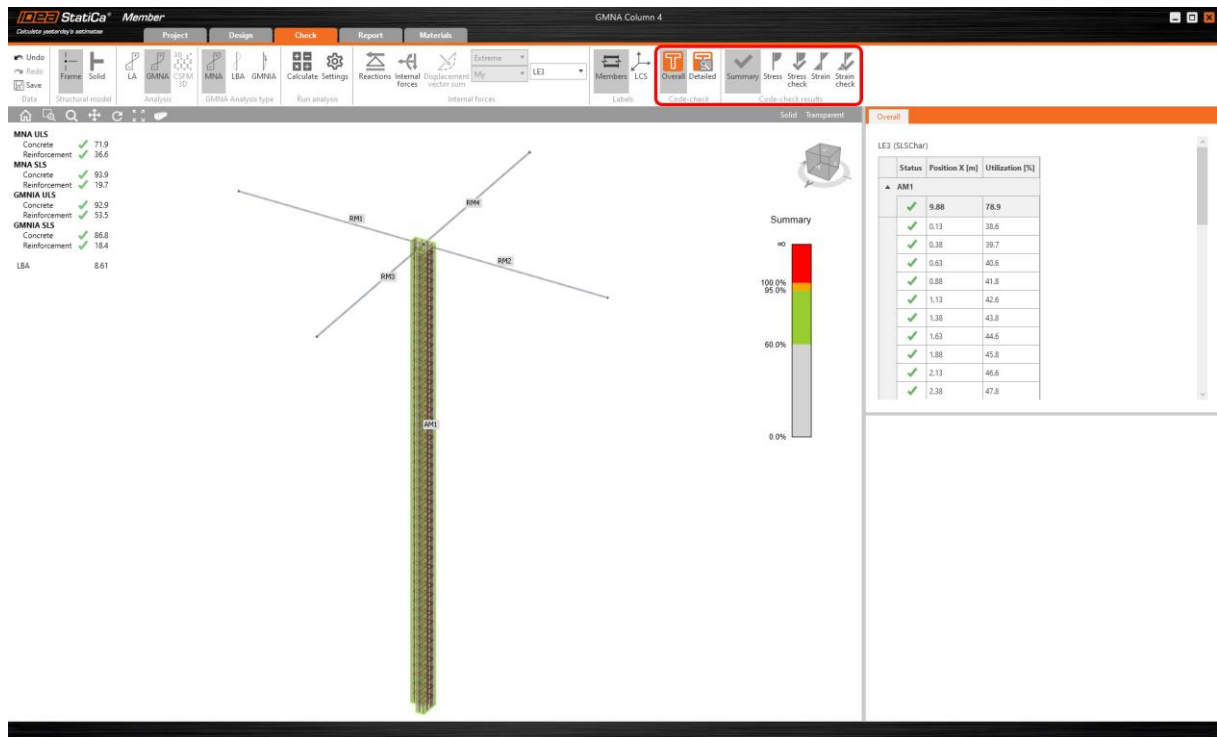


### User interface tips for Member app

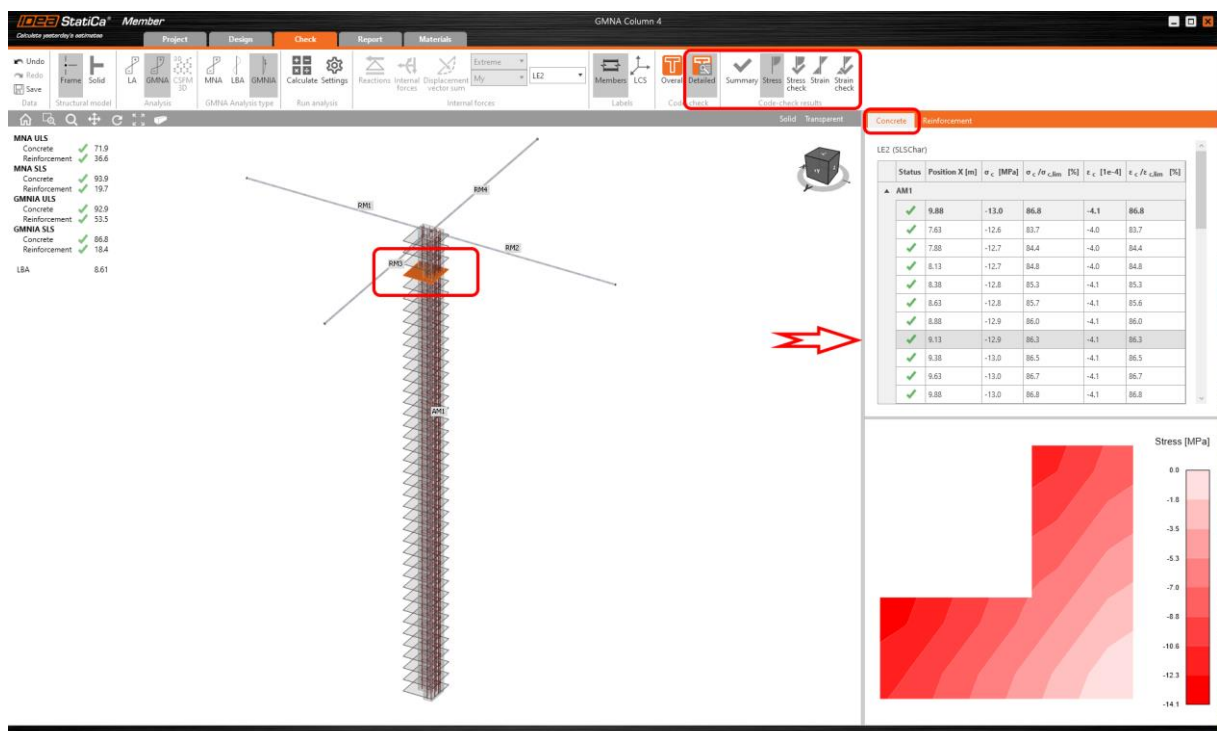
All analysis outputs are available in the Check tab, where the ribbon was reorganized and expanded by new buttons. The graphic result interpretation is displayed in the main 3D scene. At the same time, the calculated values and checks are listed in the result tables together with corresponding detailed or sectional results in the 2D scene.



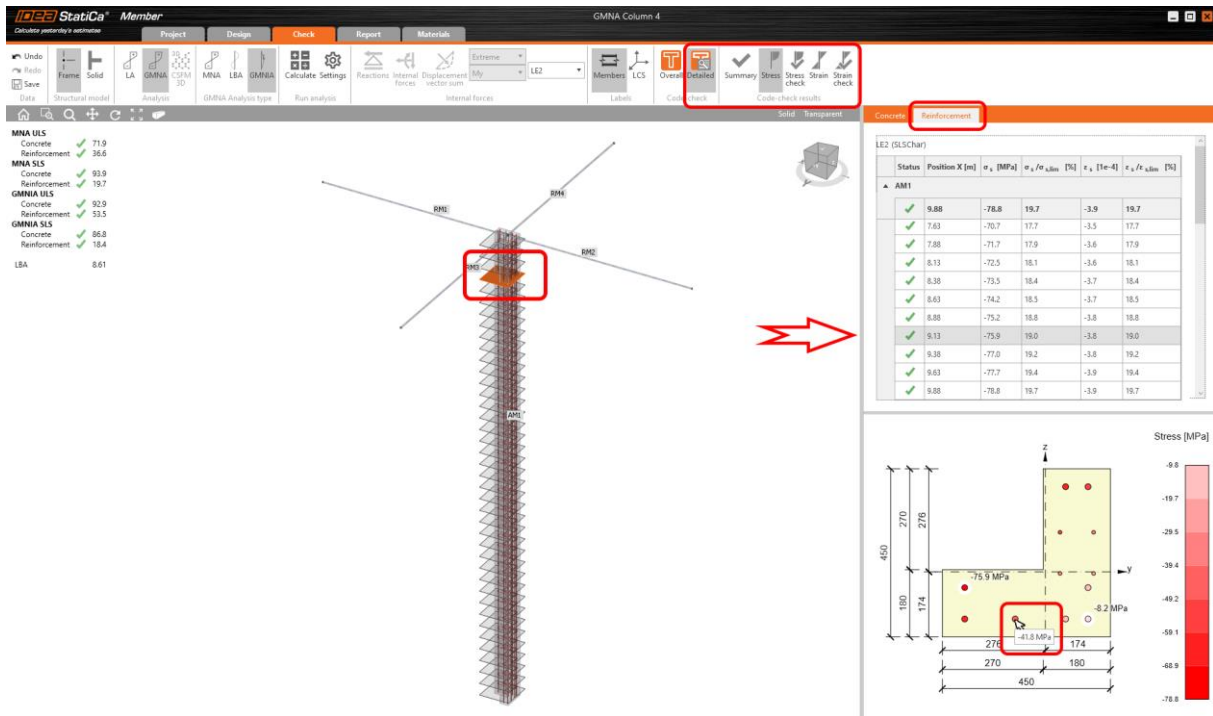
Users can display Overall or Detailed (sectional) results. The Overall code-check shows a summary of all the results available in the project. The user can immediately see whether the structure satisfies or not the particular code-checks.



If the user selects the particular section in the 3D scene then the corresponding results are highlighted in the result table, and the detailed view of the selected section is displayed in the 2D scene. Calculated stresses, strains, and checks are available for each section for concrete and reinforcement parts in particular tabs.





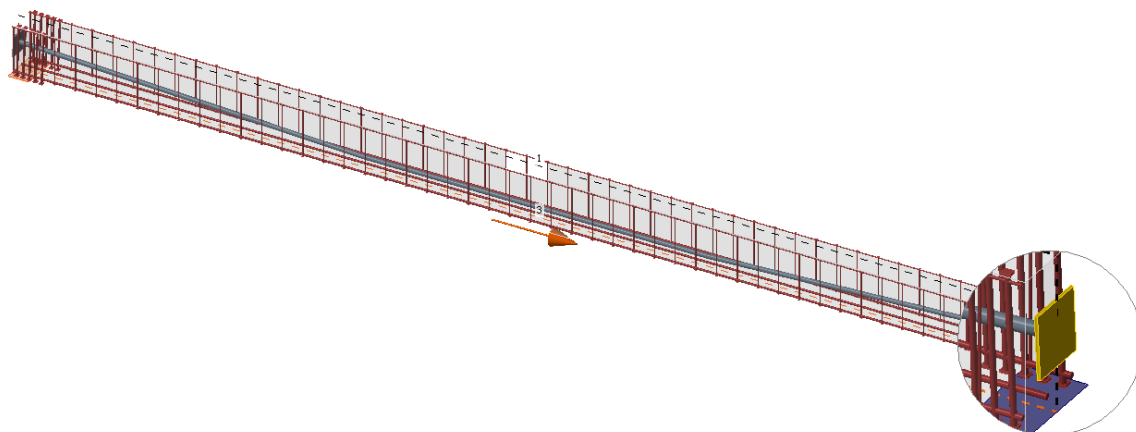


Design and check of slender columns in Member for concrete is available in **Concrete Expert**, **Concrete Enhanced**, **Prestressing Expert** and **Prestressing Enhanced** edition of IDEA StatiCa.

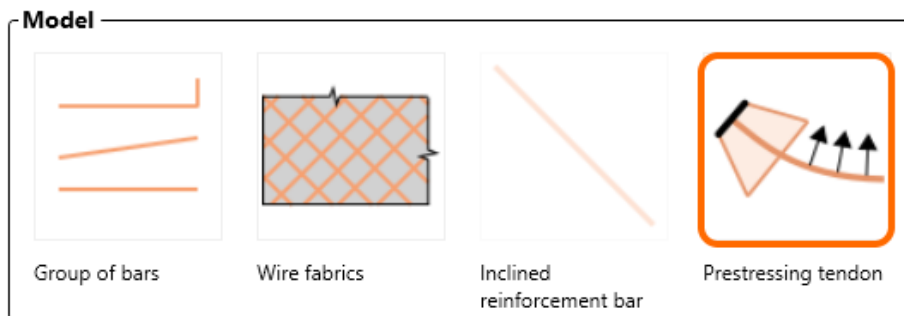
### Structural design of prestressed discontinuity regions

Not only do bridge engineers deal with prestressed concrete and use its advantages to cross over the long spans, deep valleys, or rivers, but civil engineers increasingly encounter the design of airy open spaces with minimum columns. That leads to long-span beams where reinforced concrete is not economical anymore. It is more convenient to use pre-stressed concrete, which allows decreasing the dimensions of the long-span beams.

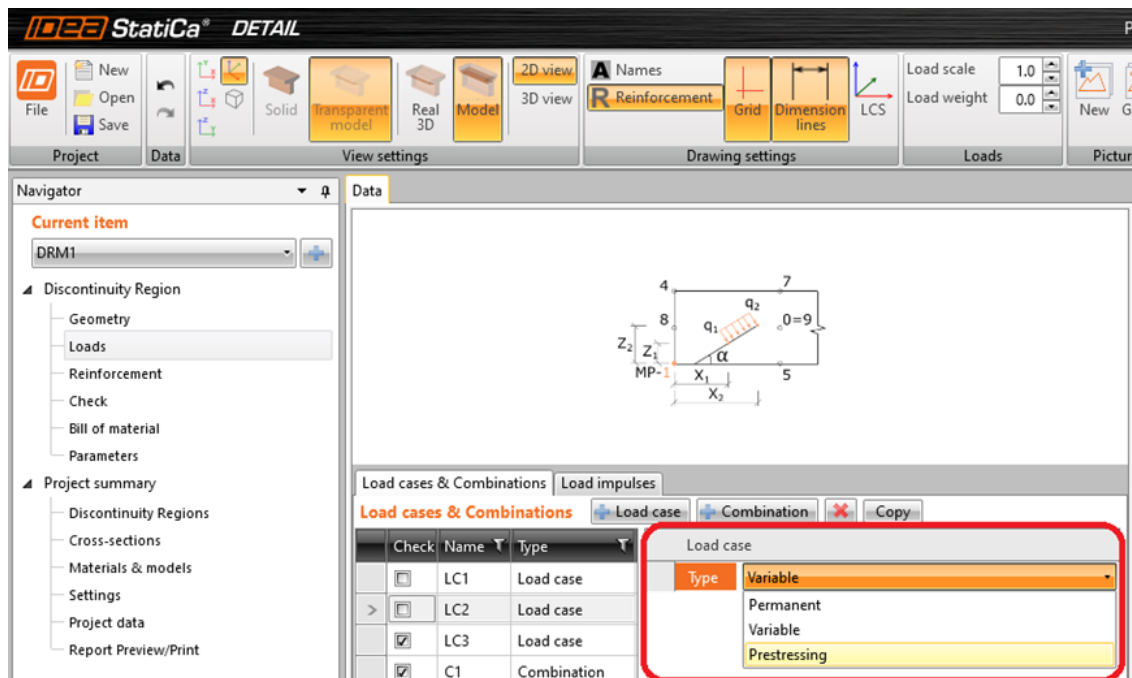
Due to the innovative and more advanced method, CSFM (Compatible Stress Field Method) engineers can now design and code-check pre-stressed concrete members, including the discontinuities like openings, haunches, dapped-ends, or anchorage zones. In the end, you will get the report with the complete ULS checks.



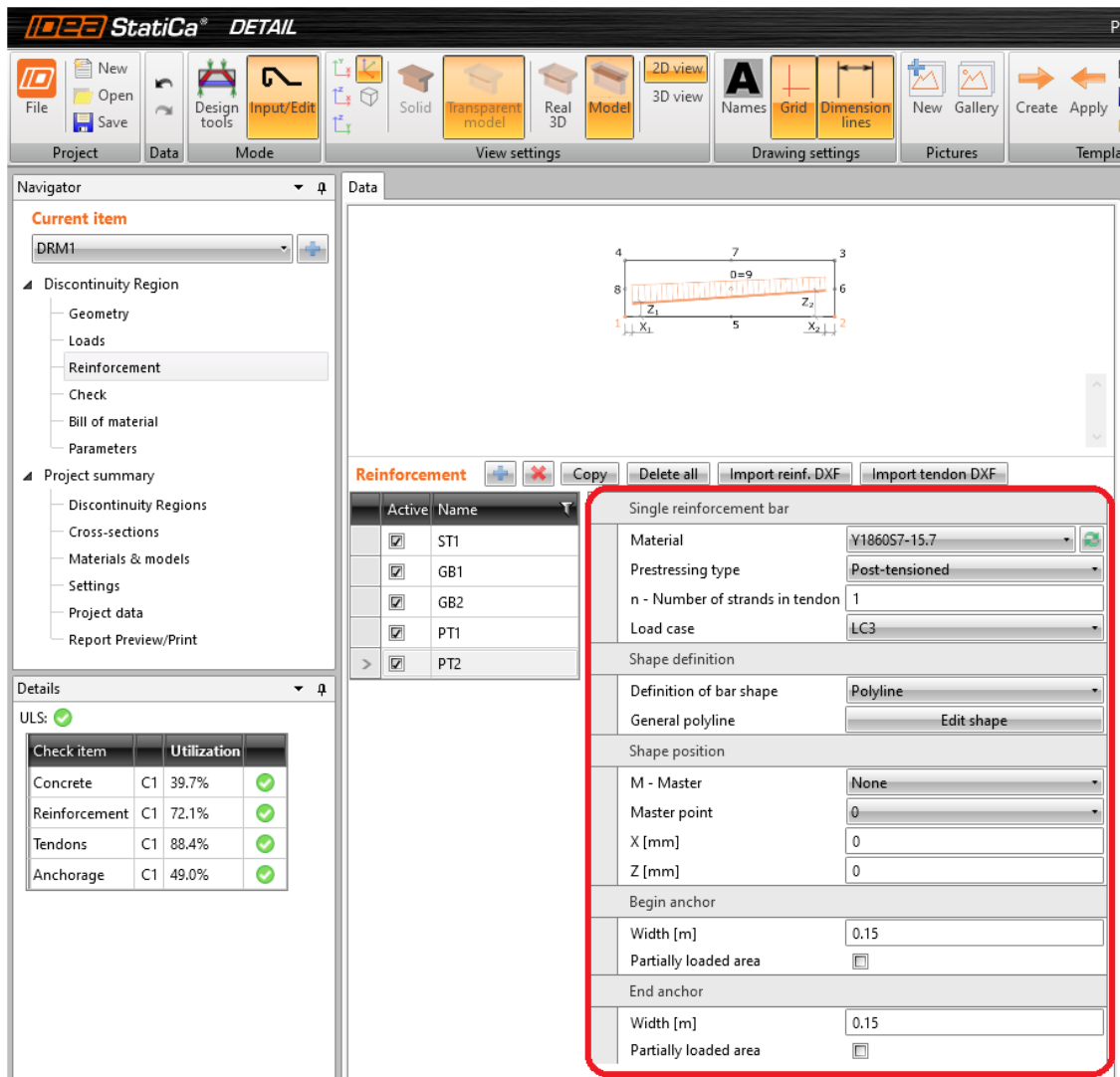
Users can now input pre-stressing tendons into their projects in IDEA StatiCa Detail. This option makes the design of a pre-stressed structural member much more efficient since they do not have to calculate the equivalent load effects as it was until now. Simply input a prestressing load case, and the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



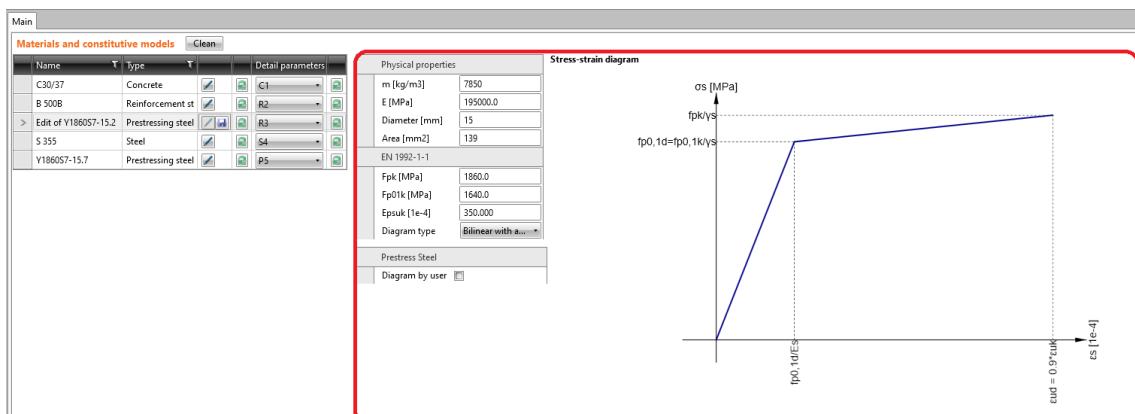
Prestressing type load case must be defined to enable this option.



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



Currently, the bonded tendons and only ultimate limit state (ULS) checks are supported. Completely new material library for prestressing tendons has also been added.



The pre-stressing 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 concrete members is available in **Prestressing Enhanced** edition of IDEA StatiCa.

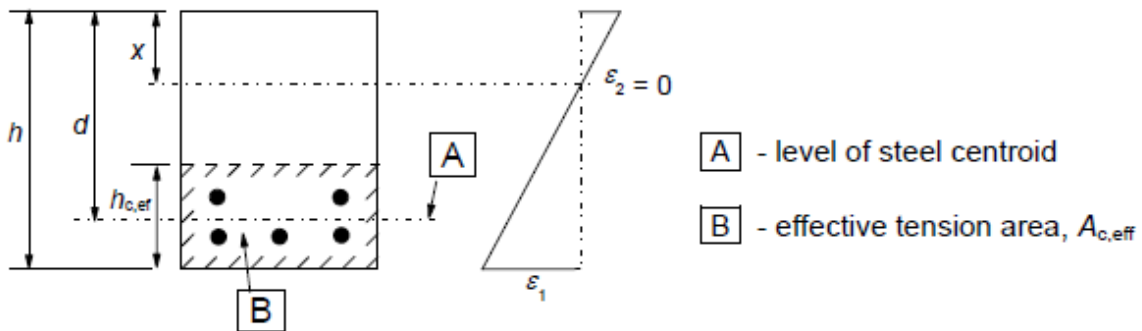
### Crack width check of cross-sections with a large concrete cover

The crack width check is an important and integral part of the structural report. If you code-check reinforced concrete cross-sections with a large concrete cover, you may have got into trouble; specifically, the check cannot be performed.

For the calculation of the crack width according to EN 1992-1-1, chapter 7.3.4, the depth of effective tension area  $h_{c,eff}$  and reinforcement ratio  $\rho_{p,eff}$  need to be evaluated. The depth  $h_{c,eff}$  is determined according to this formula:

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

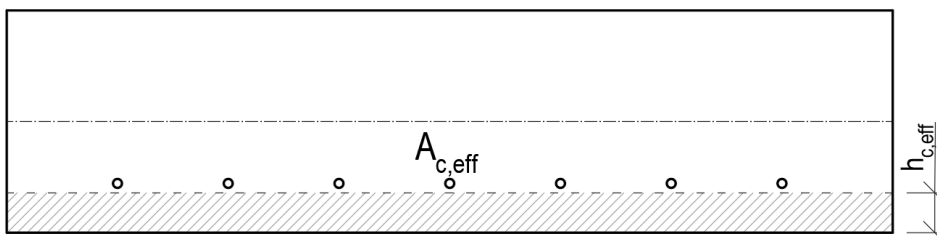
$$\rho_{p,eff} = \frac{A_{s,eff}}{A_{c,eff}}$$



$$w_k = s_{r,max}(\epsilon_{sm} - \epsilon_{cm})$$

$$\epsilon_{sm} - \epsilon_{cm} = \frac{\sigma_s - k_t \cdot \frac{f_{ct,eff}}{\rho_{p,eff}} \cdot (1 + \alpha_e \cdot \rho_{p,eff})}{E_s} \geq 0,6 \frac{\sigma_s}{E_s}$$

There are cases where the cross-section is subjected to a bending moment and a compressive normal force, while at the same time a large reinforcement cover is designed. This situation occurs mainly in bridge substructures, piers. In this case, the depth of effective tension area lead to zero value due to large concrete cover:



And Eurocode methodology could not be applied since no reinforcement was found in the tension area. No reinforcement in the tension zone, no crack width check. But engineers need to have this check done. How did we deal with satisfying both engineer's needs and code requirements?

Since several enhancements have been made to the crack width check over the last four releases, let's re-visit these particular enhancements and present the newest, of course.

### Version 20.0 and older

By version 20.0, the absence of reinforcement in the  $A_{c,ef}$  region was resolved by using the right hand side of the equation (7.3.4) to calculate the difference of the relative strains of concrete and reinforcement:

$$\varepsilon_{sm} - \varepsilon_{cm} = 0.6 \frac{\sigma_s}{E_s}$$

where stress in the most tensioned reinforcement was substituted for  $\sigma_s$ .

And the maximum crack spacing was calculated according to Equation 7.14, i.e.:

$$s_{r,max} = 1.3(h - x)$$

If the issue with no reinforcement in the tension zone came to happen, the above-mentioned workaround was applied.

**Crack width**

Crack width - short-term effect									
Combination	N [kN]	M <sub>y</sub> [kNm]	M <sub>x</sub> [kNm]	W <sub>y</sub> [mm]	W <sub>pl,y</sub> [mm]	Value [%]	Limit [%]	Check	
Quasi	-6478.5	1648.9	-64.3	0.009	0.300	2.9	100.0	OK	
Crack width - long-term effect									
Combination	N [kN]	M <sub>y</sub> [kNm]	M <sub>x</sub> [kNm]	W <sub>y</sub> [mm]	W <sub>pl,y</sub> [mm]	Value [%]	Limit [%]	Check	
Quasi	-6478.5	1648.9	-64.3	0.010	0.300	3.2	100.0	OK	
Intermediate results and coefficients for crack width calculation - short-term effect									
k <sub>s</sub> [-]	x [mm]	d [mm]	ε <sub>sm</sub> -ε <sub>cm</sub> [1e-4]	s <sub>r,max</sub> [mm]	σ <sub>s</sub> [MPa]				
0.40	758	887	0.3	328	8.9				
Intermediate results and coefficients for crack width calculation - long-term effect									
k <sub>s</sub> [-]	x [mm]	d [mm]	ε <sub>sm</sub> -ε <sub>cm</sub> [1e-4]	s <sub>r,max</sub> [mm]	σ <sub>s</sub> [MPa]				
0.40	807	895	0.4	295	12.1				
Creep coefficient									
Way of assessment	h <sub>0</sub> [mm]	A <sub>c</sub> [mm <sup>2</sup> ]	u [mm]	t [d]	t <sub>0</sub> [d]	t <sub>1</sub> [d]	RH [%]	Use V <sub>ic</sub> [-]	φ(t,t <sub>0</sub> ) [-]
Automatic	738	2227462	6033	18250.0	28.0	7.0	65	No	1.70

Nonconformity

Nonconformities

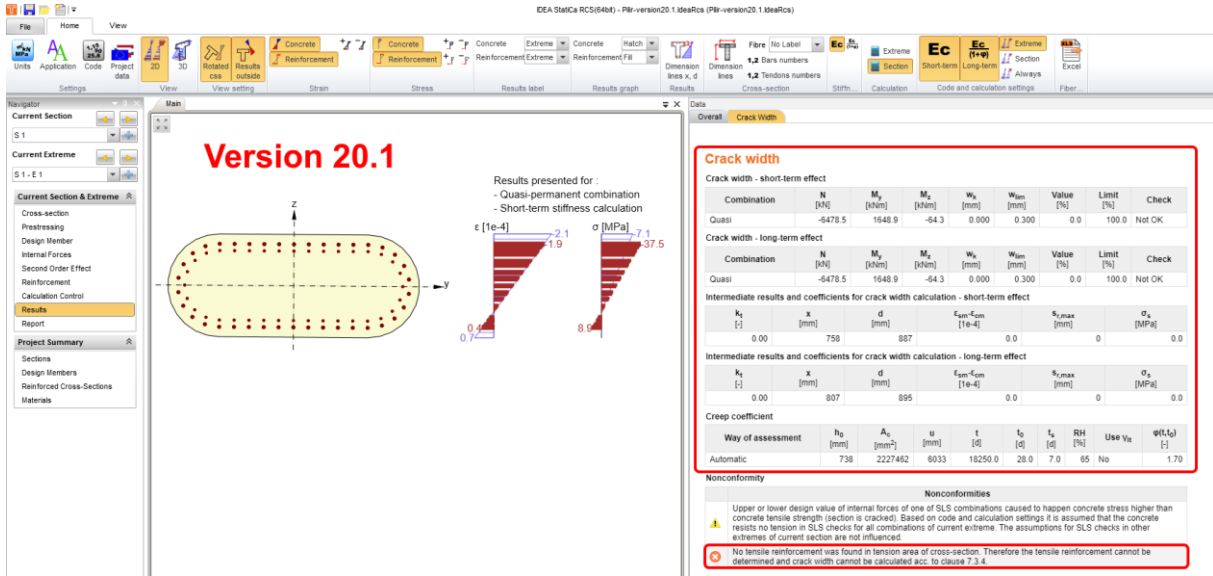
Upper or lower design value of internal forces of one of SLS combinations caused to happen concrete stress higher than concrete tensile strength (section is cracked). Based on code and calculation settings: it is assumed that the concrete resists no tension in SLS checks for all combinations of current extreme. The assumptions for SLS checks in other extremes of current section are not influenced.

The strain is calculated according to EN 1992-1-1 eq. (7.14)

### Version 20.1

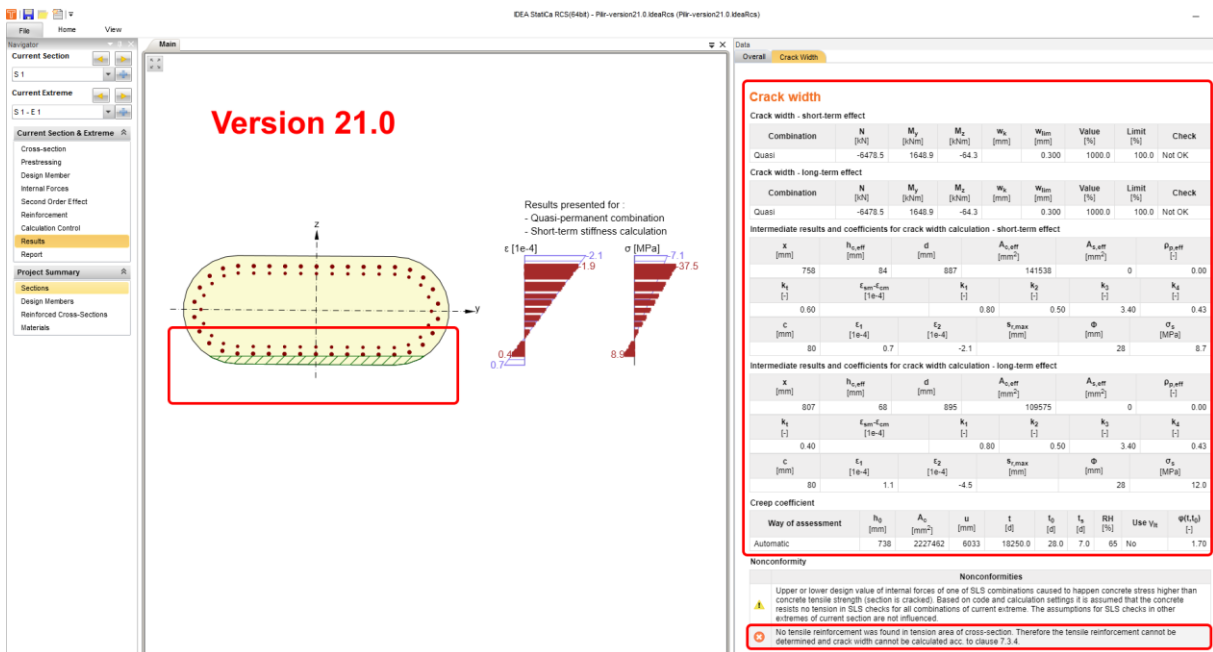
Since the calculation used in Version 20.0 and older does not meet the basic assumptions of the tension chord model (reinforcing bar embedded in concrete), we decided to not use the workaround implemented in version 20.0 and older. And if the reinforcement was out of the tension zone, a user was warned by the nonconformity:

No tensile reinforcement was found in tension area of cross-section. Therefore the tensile reinforcement cannot be determined and crack width cannot be calculated acc. to clause 7.3.4.



### Version 21.0

The only change compared to version 20.1 is that we started to display all the values we are able to count, at the same time we started to display the effective tensile area of the concrete in the scene. This allows the user to see that all the reinforcement is outside the Ac,ef region. Note that the center of gravity of the reinforcement must be inside of the tension area to consider it as embedded reinforcement. At the same time, the nonconformity was displayed: No tensile reinforcement was found in the tension area of cross-section. Therefore the tensile reinforcement cannot be determined and crack width cannot be calculated acc. to clause 7.3.4.



### Version 21.1

When determining the effective tension area, for each case of crack width calculation, the calculation is performed as follows:

- Step 1 – a regular calculation is performed, where the height  $h_{c,ef}$  is determined as:

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

- Step 2 – if the effective depth  $h_{c,ef}$  is so small that all the reinforcement lies outside this region, the depth is determined as described in step 3.
- Step 3 –  $h_{c,eff}$  calculation excluding the critical condition is determined as:

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

At the same time, the following nonconformity is displayed:

The effective area of concrete in tension surrounding the reinforcement or prestressing tendons of depth  $h_{c,ef}$ , where  $h_{c,ef}$  is the lesser of  $2.5(h-d)$  or  $h/2$ . In case of considering of value as  $(h-x)/3$ , the reinforcement is out of the effective area of concrete in tension and therefore it would not be possible to calculate crack width according to clause 7.3.4.

Note that this enhancement was already released in patch of version 21.0, i.e. 21.0.4.

**Crack width**

Crack width - short-term effect											
Combination	N [kN]	M <sub>y</sub> [kNm]	M <sub>x</sub> [kNm]	W <sub>y</sub> [mm]	W <sub>lim</sub> [mm]	Value [%]	Limit [%]	Check			
Quasi	-6478.5	1648.9	-64.3	0.011	0.300	3.7	100.0	OK			
Crack width - long-term effect											
Combination	N [kN]	M <sub>y</sub> [kNm]	M <sub>x</sub> [kNm]	W <sub>y</sub> [mm]	W <sub>lim</sub> [mm]	Value [%]	Limit [%]	Check			
Quasi	-6478.5	1648.9	-64.3	0.015	0.300	5.1	100.0	OK			
Intermediate results and coefficients for crack width calculation - short-term effect											
x [mm]	h <sub>c,eff</sub> [mm]	d [mm]	A <sub>c,eff</sub> [mm <sup>2</sup> ]	A <sub>s,eff</sub> [mm <sup>2</sup> ]	ρ <sub>s,eff</sub> [-]						
758	310	887	642007	20231	0.03	K <sub>1</sub> [-]	K <sub>2</sub> [-]	K <sub>3</sub> [-]	K <sub>4</sub> [-]		
	ε <sub>sp-ε<sub>cm</sub></sub> [1e-4]					0.60	0.3	0.80	0.50	3.40	0.43
c [mm]	ε <sub>1</sub> [1e-4]	ε <sub>2</sub> [1e-4]	s <sub>1,max</sub> [mm]	φ [mm]	σ <sub>s</sub> [MPa]						
80	0.7	-2.1	429	29	8.7						
Intermediate results and coefficients for crack width calculation - long-term effect											
x [mm]	h <sub>c,eff</sub> [mm]	d [mm]	A <sub>c,eff</sub> [mm <sup>2</sup> ]	A <sub>s,eff</sub> [mm <sup>2</sup> ]	ρ <sub>s,eff</sub> [-]						
807	289	895	592185	19427	0.03	K <sub>1</sub> [-]	K <sub>2</sub> [-]	K <sub>3</sub> [-]	K <sub>4</sub> [-]		
	ε <sub>sp-ε<sub>cm</sub></sub> [1e-4]					0.40	0.4	0.80	0.50	3.40	0.43
c [mm]	ε <sub>1</sub> [1e-4]	ε <sub>2</sub> [1e-4]	s <sub>1,max</sub> [mm]	φ [mm]	σ <sub>s</sub> [MPa]						
80	1.1	-4.5	422	29	12.0						
Creep coefficient											
Way of assessment	h <sub>0</sub> [mm]	A <sub>c</sub> [mm <sup>2</sup> ]	u [mm]	t [d]	t <sub>0</sub> [d]	t <sub>e</sub> [d]	RH [%]	Use V <sub>h</sub>	φ(t,t <sub>0</sub> ) [-]		
Automatic	738	2227452	6033	18250.0	28.0	7.0	65	No	1.70		

**Nonconformity**

**Nonconformities**

Upper or lower design value of internal forces of one of SLS combinations caused to happen concrete stress higher than concrete tensile strength (section is cracked). Based on code and calculation settings it is assumed that the concrete resists no tension in SLS checks for all combinations of current extreme. The assumptions for SLS checks in other extremes of current section are not influenced.

Assuming the condition  $h_{c,ef} = (h-x)/3$ , the longitudinal reinforcement is out of the effective area of concrete in tension  $A_{c,ef}$ , therefore it would not be possible to calculate crack width according to clause 7.3.4. The depth of the effective area of concrete in tension  $h_{c,ef}$  is calculated as the lesser value of  $2.5(h-d)$  or  $h/2$ .

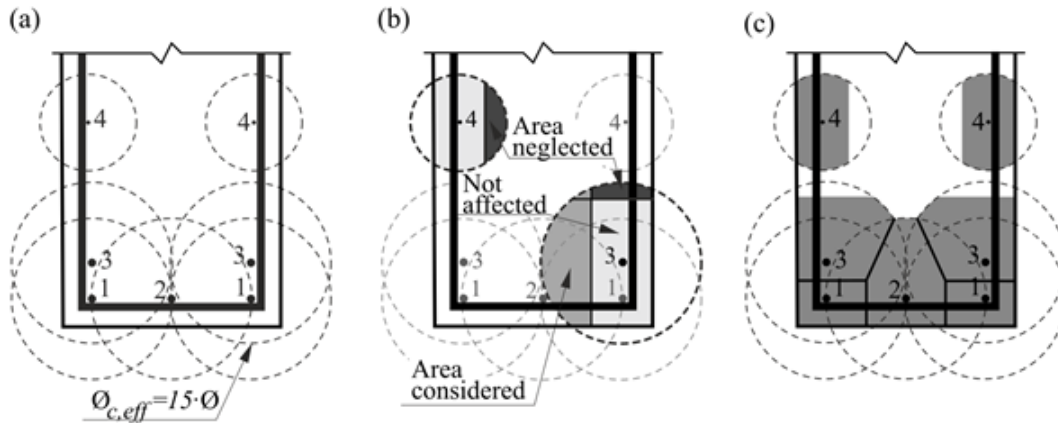
This improvement will lead to more consistent results for crack calculations in IDEA StatiCa RCS, despite the limitations set in the code.

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

## Speeding up the solver of IDEA StatiCa for concrete design

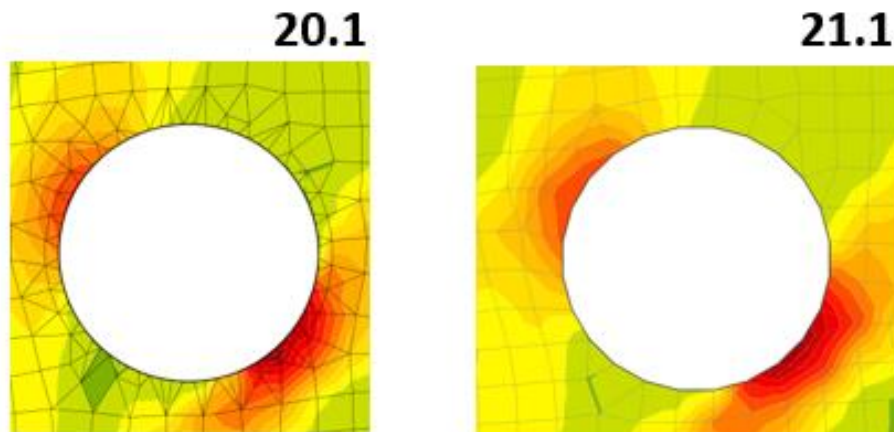
### New reinforcement ratio calculation

A new algorithm behind the calculation of reinforcement ratio in IDEA StatiCa Detail significantly improves 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 faster calculations and better convergence while keeping the results consistent with previous versions.



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

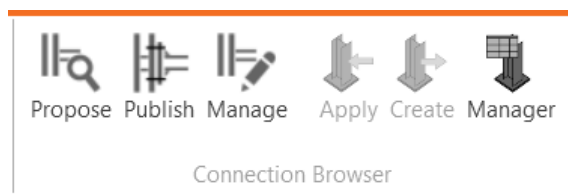


## News for Steel

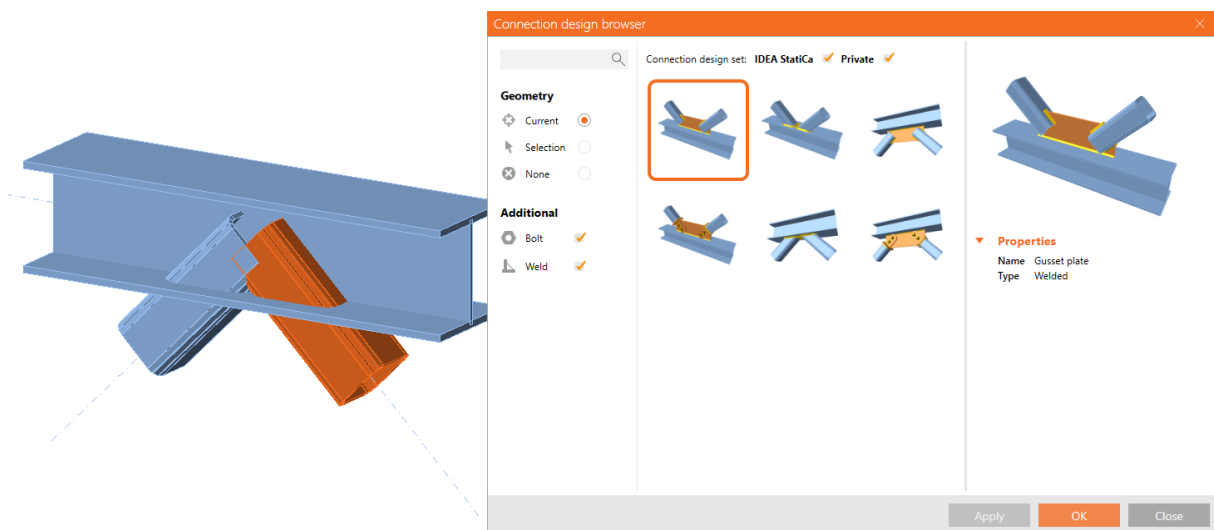
### Connection Browser – new library of connections

Connection Browser is here! This unique tool will help you find a suitable design solution from a library of predefined designs and apply them right away. The Connection Browser will work with three databases of steel connections. The first is the set defined by IDEA StatiCa in every installation. The second is the set of connection designs created and saved by each user. The third will be a company set of connections each of our customers can create and maintain – (this will be released in one of the patches of version 21.1. coming out in a couple of weeks).

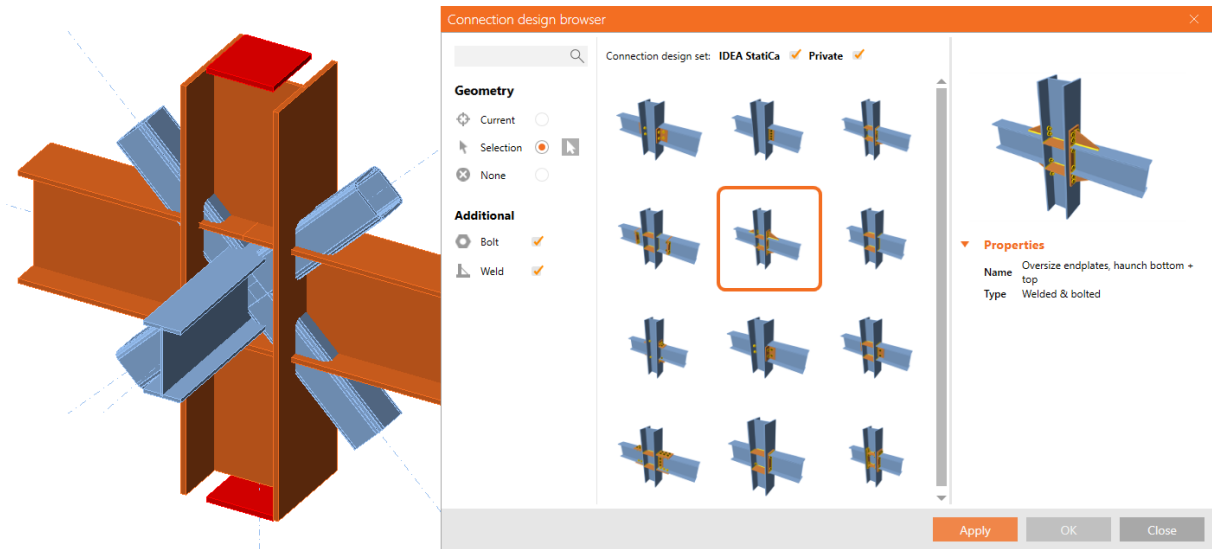
The Connection Browser allows you to find a suitable design solution from a library of predefined designs and directly apply it to your members. Connection Browser is going to replace the old Template manager in the future. Nevertheless, for the time being, you can find both in the top ribbon.



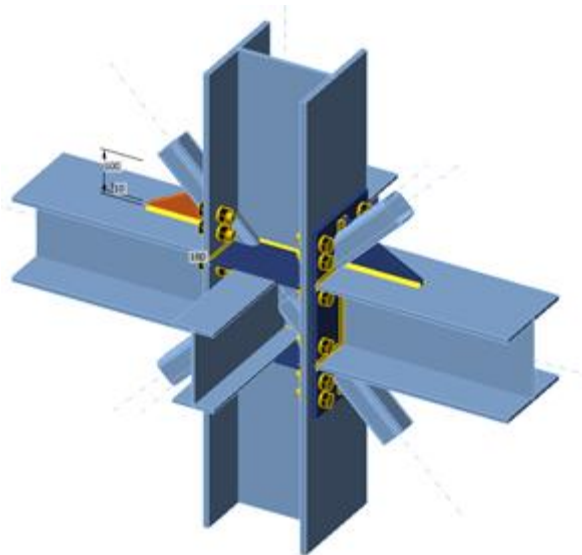
There are three new buttons dedicated to the Connection Browser. The **Propose** button opens the Connection design browser, a general library of saved designs. You can directly go through available designs for all current members in the topology of your connection.



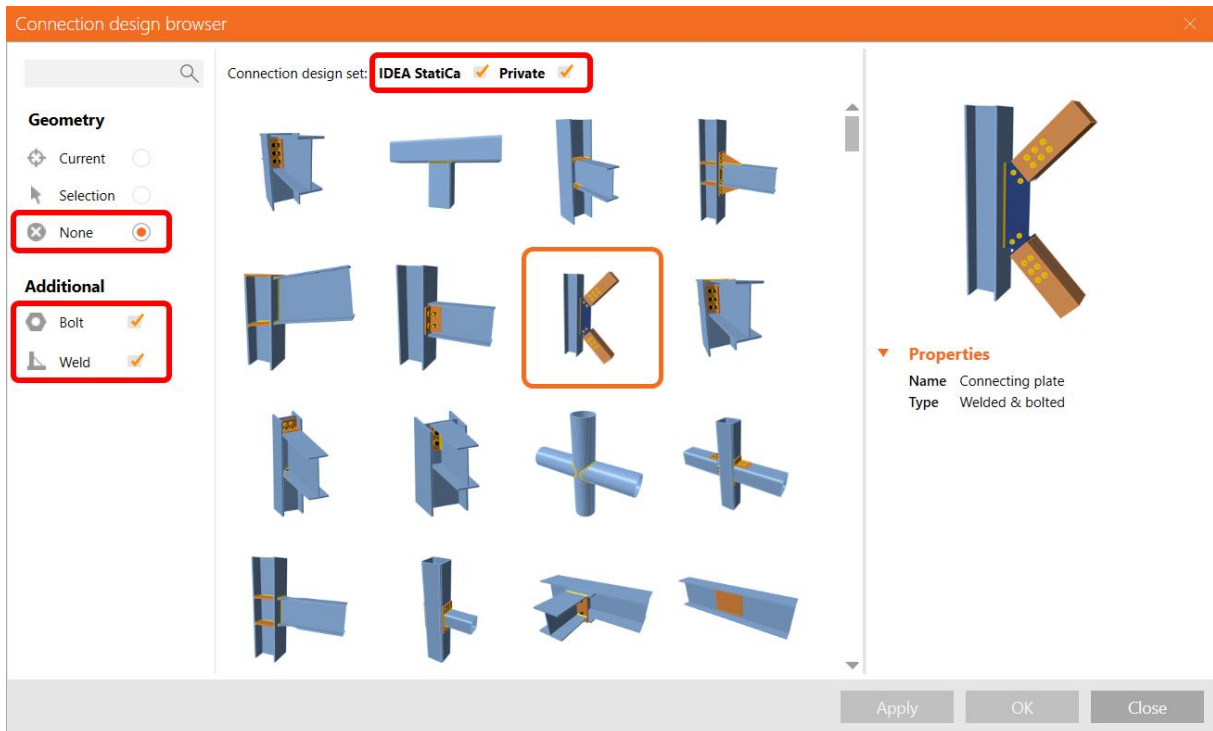
For the nonstandard geometries, try to select only a subset (i.e., two or three) members at a time. Switch the Geometry to Selection and use the pick tool and hold CTRL to select multiple members in the scene. Confirm the selection with Enter/Spacebar or right mouse button. The proposer will filter the topologies based on your selection.



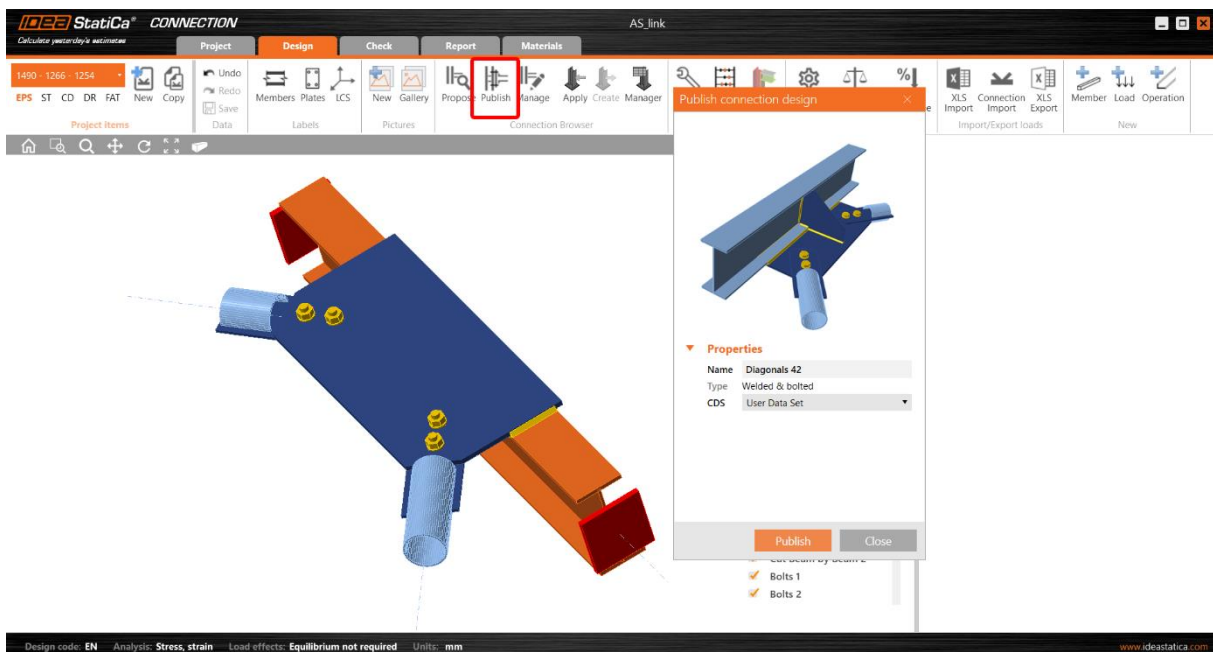
Once you find a suitable design, it is directly applied to the selected part of the joint.



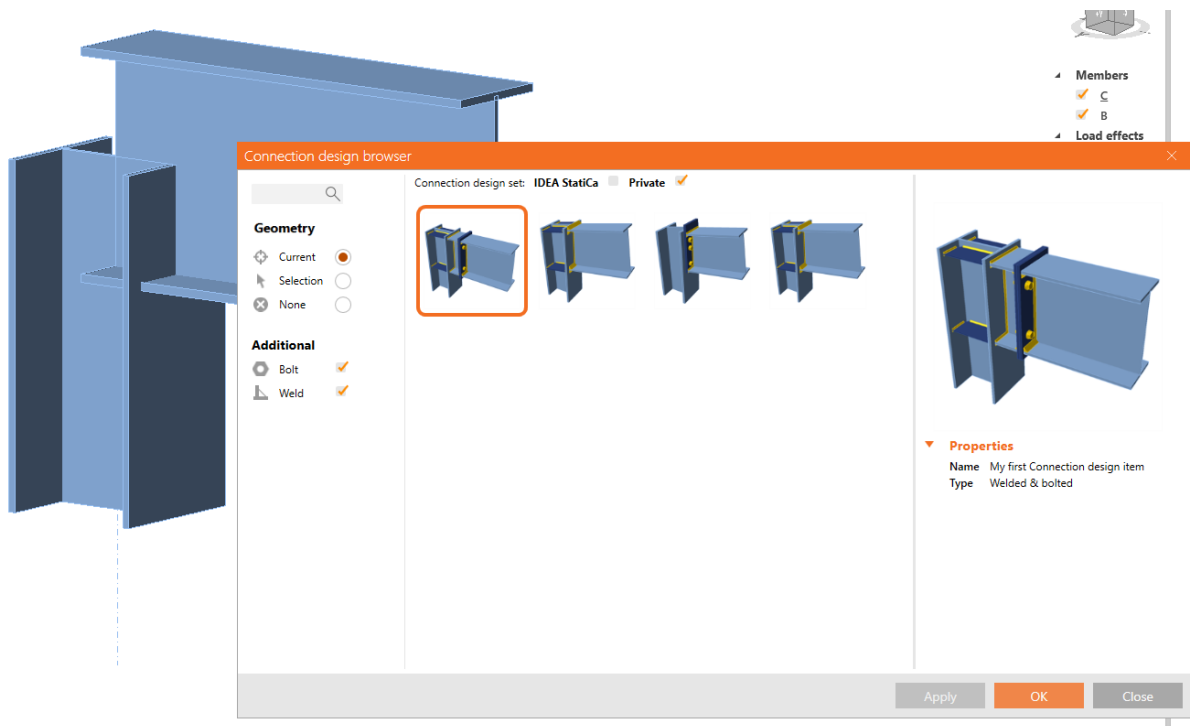
To display the full content of the library, switch the Topology to None. You can also filter the designs based on different criteria: Designs including bolts or welds or switch on/off the IDEA StatiCa default library of designs or your private (User-made) design set.



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).



Migration of current templates – the current feature for templates in IDEA StatiCa Connection will gradually be replaced by the Connection Browser. We advise all users to save their current templates as their personal designs in the Connection Browser.

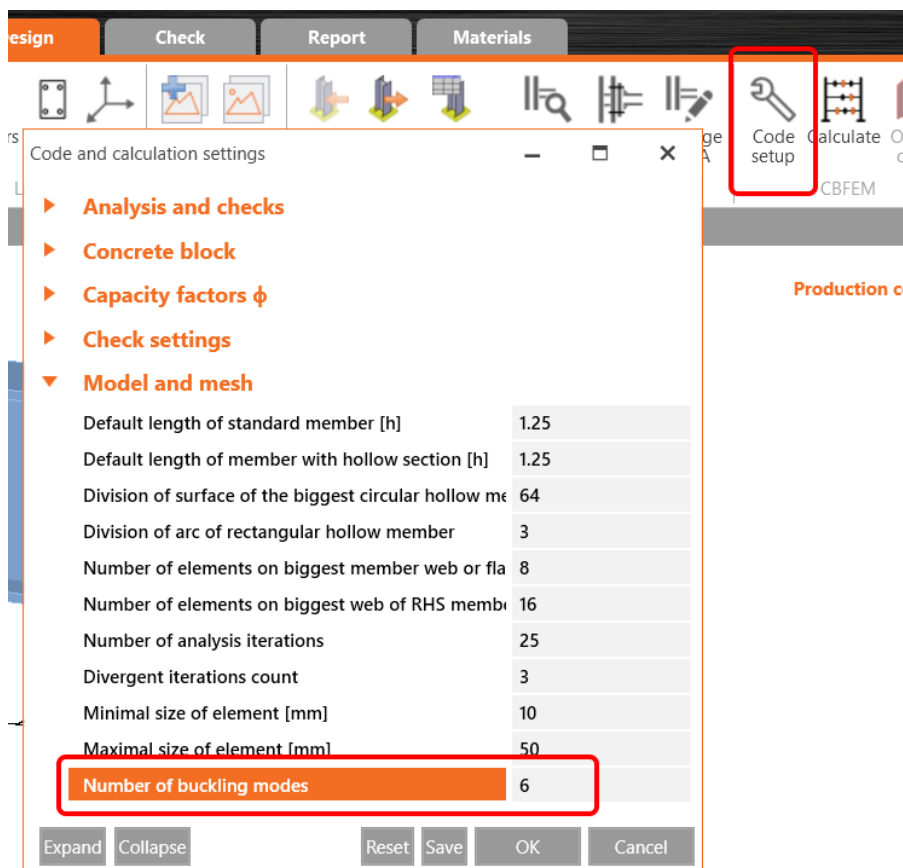
Available in **Expert** and **Enhanced** edition of IDEA StatiCa Steel.

## Number of buckling modes

Before 21.1, 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 placed a restriction on the Engineers ability to complete their checks.

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. Buckling modes numbers can be customized in both, IDEA StatiCa Connection and IDEA StatiCa Member applications.

Available in **Enhanced** edition of IDEA StatiCa Steel.



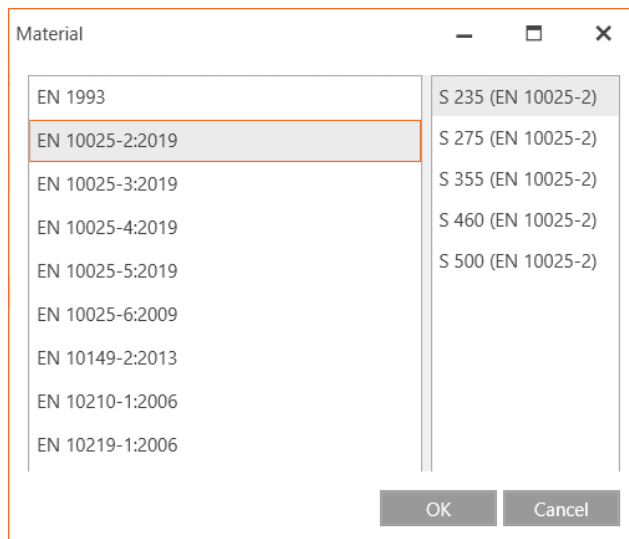
## European steels according to product sheets

IDEA StatiCa material database (MPRL) has been amended 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:

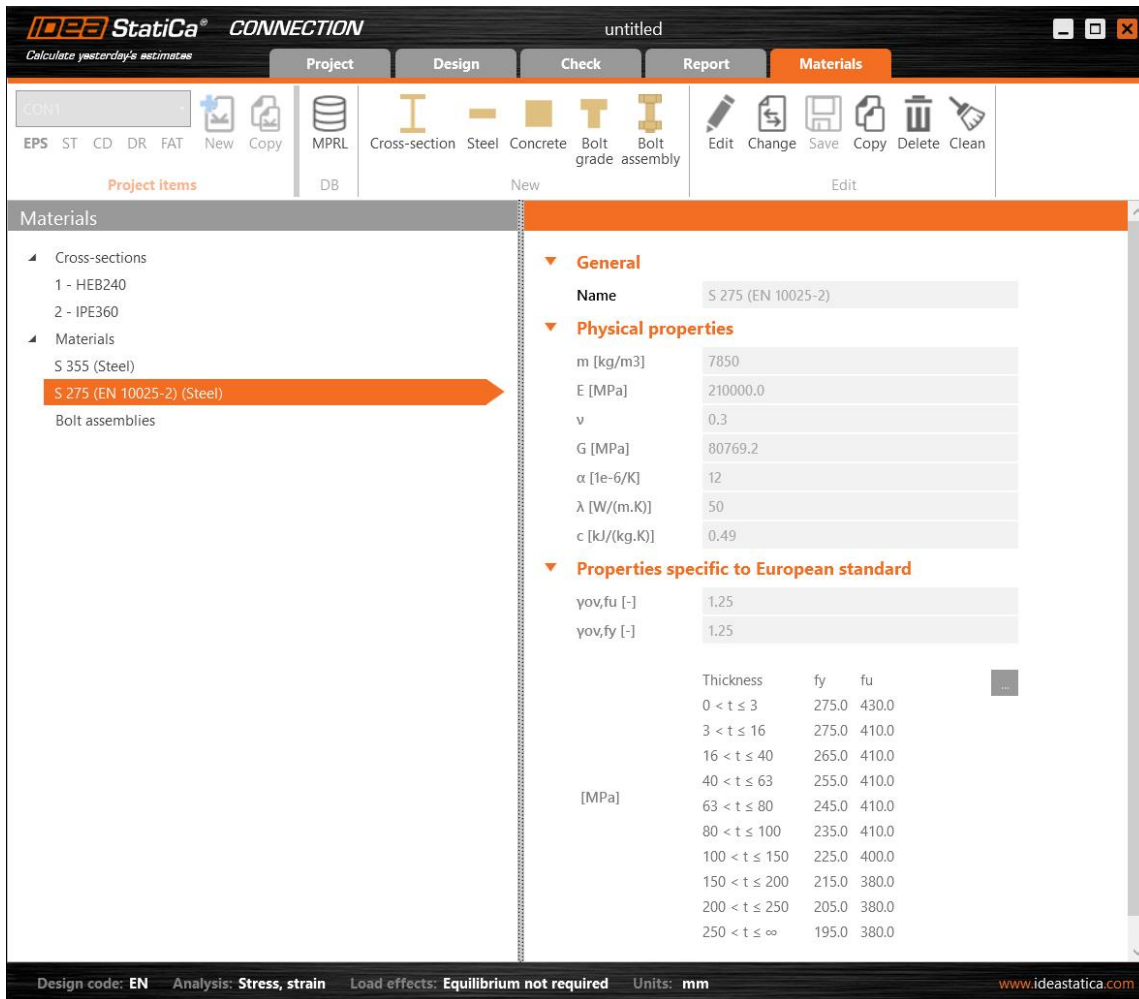
- EN 10025-2
- EN 10025-3
- EN 10025-4
- EN 10025-5
- EN 10025-6
- EN 10149-2
- EN 10210-1
- EN 10219-1

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 of IDEA StatiCa Steel.

### 3D visualization of deformations

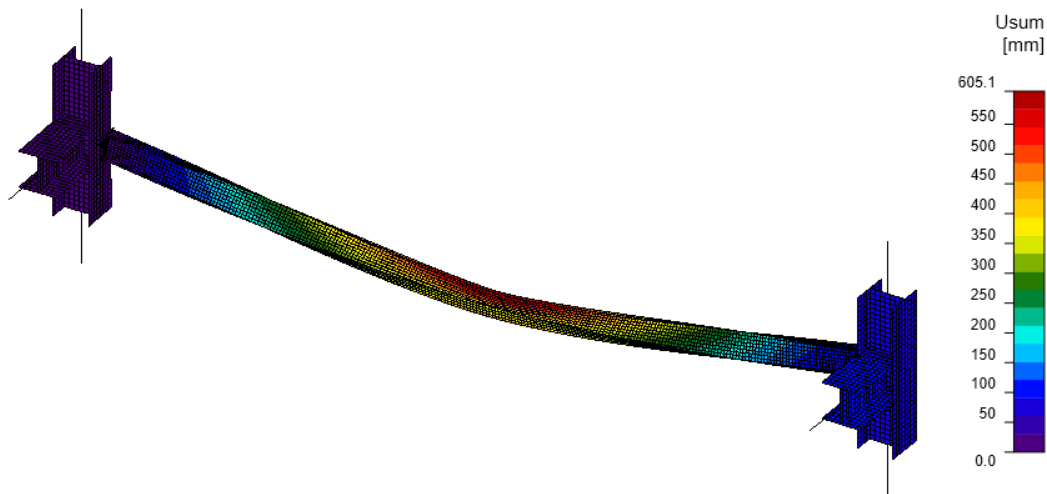
It is critical to visualize real deformations of members. IDEA StatiCa now allows you to display the spatial (real) deformation of your designed steel members.

The summation displacement vector  $U_{sum}$  has been added next to  $U_x$ ,  $U_y$ ,  $U_z$  (deformations in each direction) in results. This gives the engineer a quick insight into where the member is deforming the most or failing. The summation  $U_{sum}$  is set as a default view for steel.

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

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

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



### Use global/local coordinates in supports settings

While defining supports of the model, you will now have 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.

RM4 [Related member]
Delete

**▼ Properties**

Name	RM4		
Cross-section	2. HEB140 (Beam) <span style="float: right;">✎ +</span>		
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  Y  Z  Rx  Ry  Rz

End        X  Y  Z  Rx  Ry  Rz

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

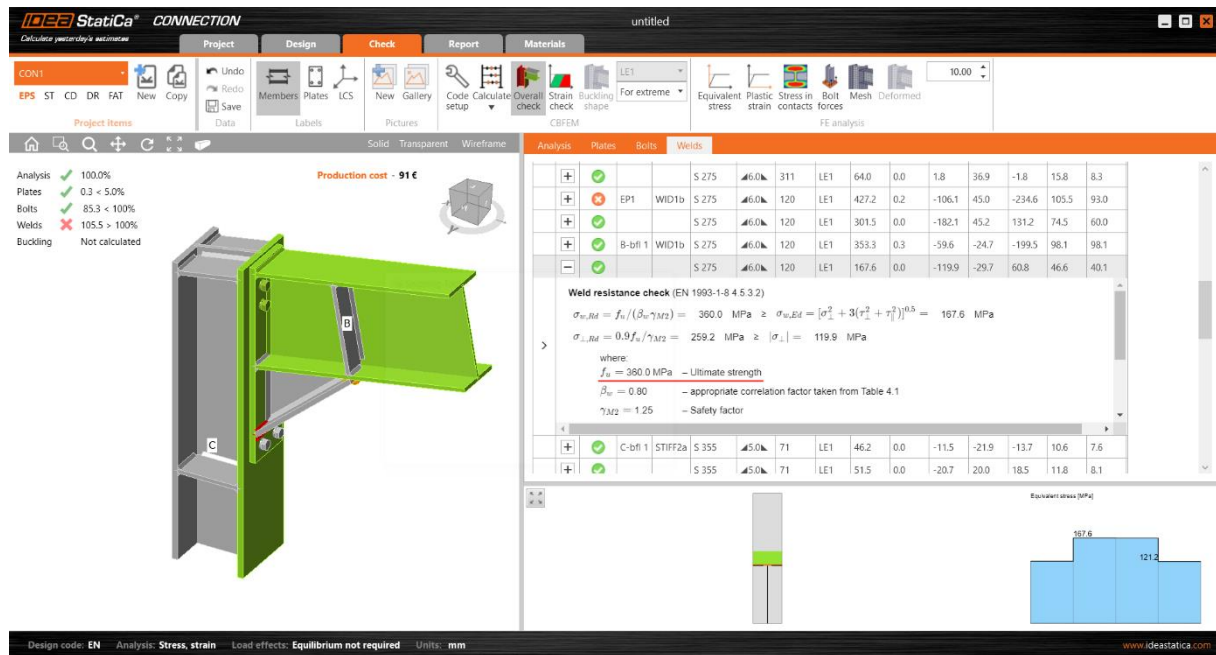


## Weld checks specifics as per Eurocode (EN) and Indian Standard (IS)

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:

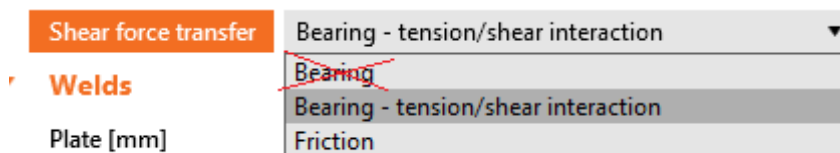
- $f_u$  of connected plate 1
- $f_u$  of connected plate 2
- $f_u$  of welding material selected in manufacturing operation



For other standards (AISC, CSA, AS, SP, GB, HKG), the strength is taken from weld electrode. Updated in **Expert** and **Enhanced** edition of IDEA StatiCa Steel.

## Bearing type options for bolts in version 21.1 and onwards

Since the 21.1 version, we will not consider the Bearing option of the shear force transfer in bolts anymore. For old IDEA StatiCa Connection projects where this type of bolt was set, the type will change from Bearing to Bearing – tension/shear interaction, which may change the results. This change aims the better understandability, simplicity and error-proof model of the application.



Updated in **Expert** and **Enhanced** edition of IDEA StatiCa Steel.

## Licensing

### Obsolete applications from version 21.1

We decided to focus our development capacity on selected applications and discontinue the support of several outdated applications. These will not be included in the 21.1 setup. 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 of IDEA StatiCa Steel.

## Solved incidents

See the [current list of solved incidents](#) reported by our customers and solved with this release of IDEA StatiCa.