



RELEASE NOTES

Release notes IDEA StatiCa 22.0 - Full version

Apr 21, 2022

IDEA StatiCa version 22.0 is alive now and brings you many improvements in both the steel and concrete structural design. Working with IDEA StatiCa will take less time than ever while offering you additional modeling and design options.

What is new? Connection Browser can now create company sets of steel connections. Two new BIM links, improved imports to Checkbot for all existing links. New connections modeling tools, better concrete result analysis, improved UI of our applications, and much more.

Discover all the improvements below and Calculate yesterday's estimates!

News for Steel

Although our steel applications for connection and member design are leading the market already, we keep improving them rapidly. Connection Browser now enables you to create and share your connection designs with colleagues in your company. This means your unique company set of connections, synced automatically to all users of your license, ready to be used!

We have solved the challenging cutting and welding of members going through another member of a hollow section. Moreover, you will be able to simulate the connection of the member to, for example, a slab providing additional support against lateral-torsional buckling. This works both for IDEA StatiCa Connection and IDEA StatiCa Member.

IDEA StatiCa Connection is now also able to check the contact between a baseplate and a concrete block even without installed anchors. Our focus was also on updating our material library and complementing the code checks based on your design code. We have specified the points of consideration when connecting hollow sections with long bolts going through.

Analysis of fatigue was supplemented by the checks of the stress directly in the welds as well as in plates next to them. And for better safety, the setup of loads in equilibrium will be the default from now on.

Last but not least, our Connection Lite free cloud service was updated to comply with the latest desktop application.

Company sets & selection in the Connection Browser

Connection Browser introduced in the previous 21.1 version keeps boosting its functionality. From now, you can share designs within your company, enter the Connection Browser in the scene by the right-mouse button or filter the designs based on several criteria.

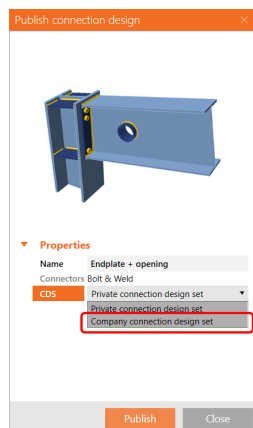
Check all the features in IDEA StatiCa 22.0 and compare them with the [Connection Browser \(v21.1\)](#) introduced in the previous version.

Company set of your designs

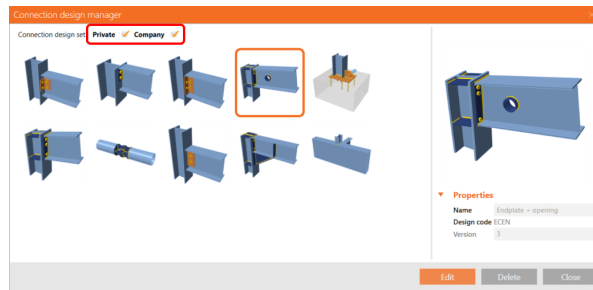
A company connection design set is a set of saved designs (templates) created by you or your colleagues from the same company. The set is accessible for all users from the company (based on the company license group), while users from different companies cannot see, use or access these design items.



Every user from a company can search and apply the saved company designs as well as add a new design to the company set. After creating your custom connection, press the **Publish** button in the top ribbon, specify the description and select the Connection Design Set (CDS) to save your connection as a new template. The Private items will be available only for you, while the Company is shared with your colleagues.



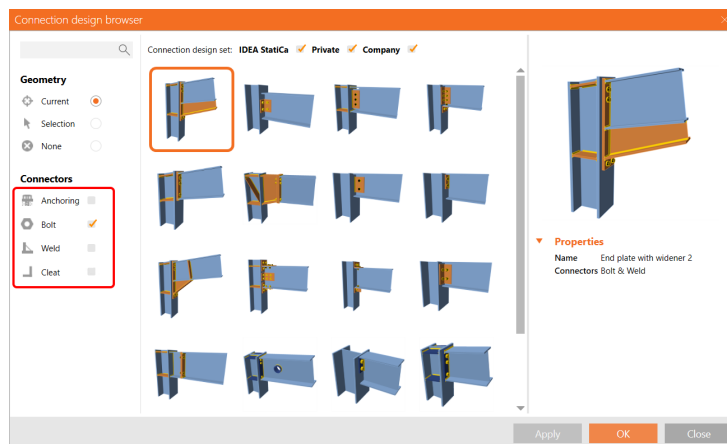
To view and manage your Private and Company design sets, use the **Manage** button in the top ribbon.



Introduction video of the Company sets in the new Connection Browser:

Improved filters to manage designs

In the Connection Browser (opened by the **Propose** button from the top ribbon), you can find a gallery of all saved designs proposed for a given geometry (see [the article from Release notes 21.1](#) to get familiar with the interface and find out how to work with the geometry). To easily find the desired design, you can switch on and off items based on the used connectors (anchors, bolts, weld, cleats) by the filter on left.



Above the gallery preview of the designs, you can turn on and off the three different sets of designs:

IDEA StatiCa connection design set is a set of design items (templates) created for you by IDEA StatiCa team and accessible for all users without limitation.

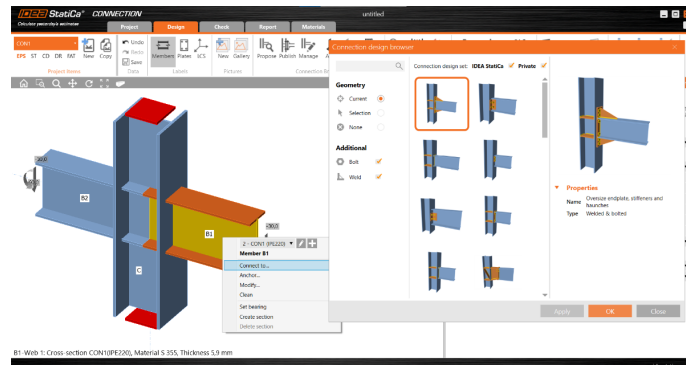
Private connection design set is a set of design items (templates) created by a user and accessible only by the user (based on user account). No other users can see, use or access these design items.

Company connection design set is a set of design items (templates) created by users from a company and accessible by users from a company (based on company account license group). Users from different companies cannot see, use or access these design items.

The right-mouse button opens the Connection Browser

This feature is another step of the Connection Browser integration into IDEA StatiCa Connection interface and replaces the similar old-style feature under the right-mouse button. Shortcut for those of you who like to be super fast - the right-click on a member in the 3D scene automatically proposes designs that fit from your library.

The UI of the right mouse button features **Connect to**, **Anchor**, and **Modify** remains the same. Instead of a solid set of general templates, the Connection Browser proposes designs that fit the number of selected members, their geometry, and cross-sections.



The right-click selection workflow with integrated Connection Browser:

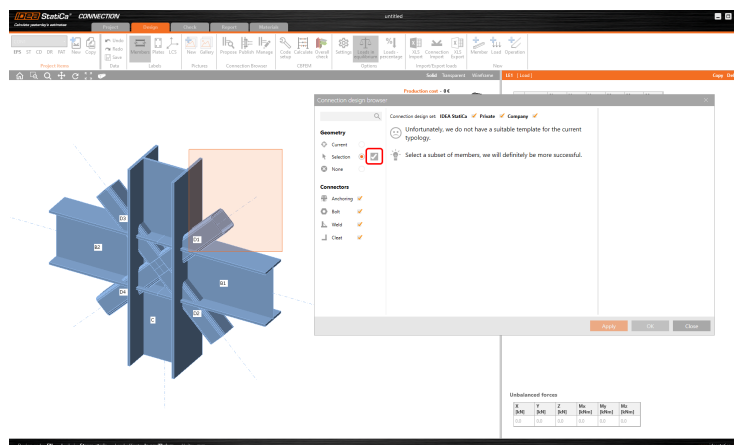
- Click the right mouse button at a member
- Click on **Connect to** (or Anchor or Modify)
- You can select one or multiple members to connect to. Multiple members can be selected either by holding CTRL or SHIFT key or by dragging the mouse in the scene
- Press Spacebar to confirm the selection
- Choose one of the proposed designs in the Connection Browser

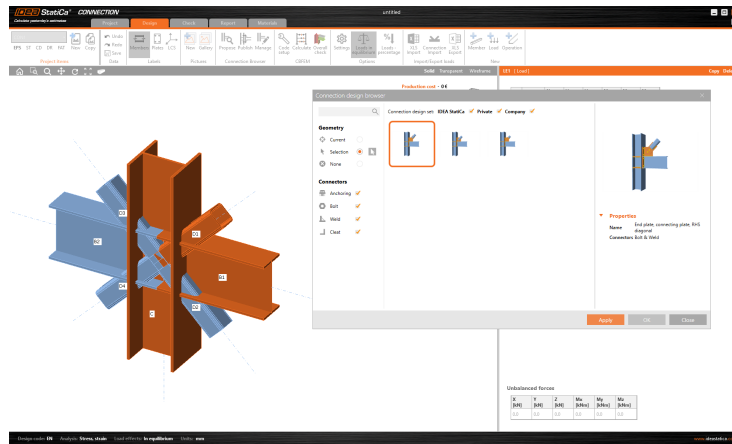
The right-mouse button feature has been available since the 21.1.1 patch.

Selection functionality

As mentioned above, you can use the selection of multiple members by holding the CTRL or SHIFT key and selecting them in the scene. Confirm the selection by the spacebar key or Enter key or by another right-mouse click.

Alternatively, you can select multiple members by clicking and dragging the selection window. In this case, no confirmation is needed.

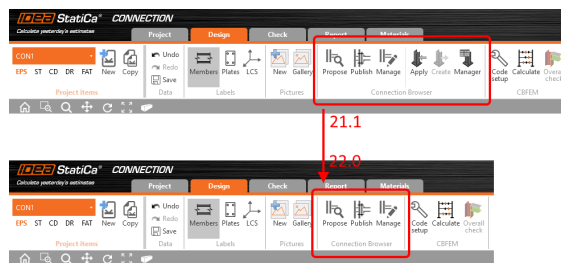




Termination of old Template Manager

As you might have noticed, the old-fashioned and outdated Template Manager has been made obsolete in version 22.0. If you need to transfer your custom designs into the new Connection Browser, use the 21.1 version of IDEA StatiCa where both functions are implemented next to each other.

Create the design you want to transfer by loading it from the Template Manager in 21.1 and Publish it to Connection Browser to your Private or Company set of designs.

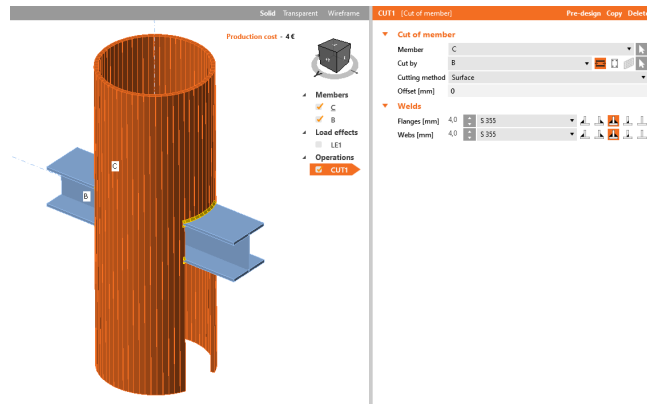


Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

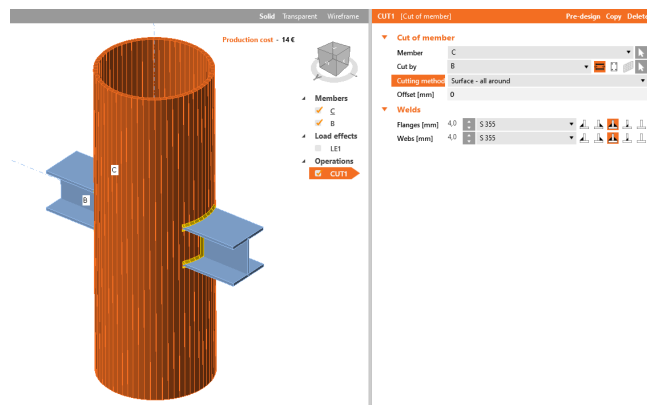
Surface - all around cutting method

The all-around cutting method solves the issue with cuts of the tubular sections. For such a Cut operation it is possible to use a simple "Surface - all-around" feature.

Curved hollow sections in IDEA StatiCa (e.g. CHS) are represented by a polygon of multiple plates composing the shell of the member. When a hollow section is cut by the cutting method **Surface**, the cut plates are divided into two parts and only one part remains, while the other one is cut off. There is no workaround to cover this issue with this cutting method.



For such cases, there is an updated cutting method **Surface - all around**, which doesn't cut off any part of the member shell except the shape that goes thru it.



Note that cutting methods **Surface** and **Surface - all around** may lead to slightly different weld geometries. Especially for CHS cross-sections, where the **Surface** method is specifically modified to avoid bad mesh with very small triangle lengths.

Also, the position of one-sided fillet welds is changed compared to the surface method, which is immediately visible in the 3D model.

This update has been available since the 21.1.2 patch.

Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

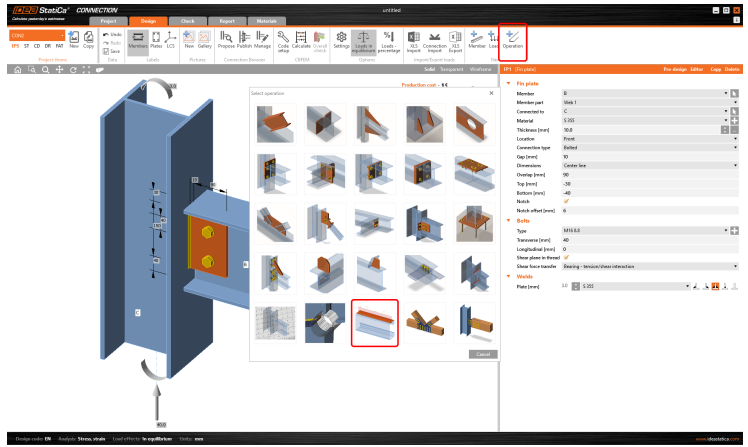
Introduction video of the Surface – all-around feature:

Lateral-Torsional Restraint at the member with defined stiffness

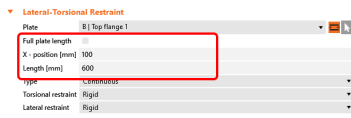
Lateral-torsional restraint is available in IDEA StatiCa Connection and IDEA StatiCa Member. The main application is for members that are restrained by floor, trapezoidal sheeting, or cladding. This feature is useful especially to determine lateral-torsional buckling for members in the Member application and to avoid unwanted torsional deformation and stress of eccentrically loaded members in the Connection application.

Both symmetric or asymmetric members are loaded in the center of gravity but because the deformation of the top flange is typically restrained, torsion is neglected. Such members can now be analyzed correctly.

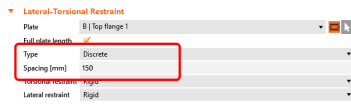
A manufacturing operation has been added to **Connection**.



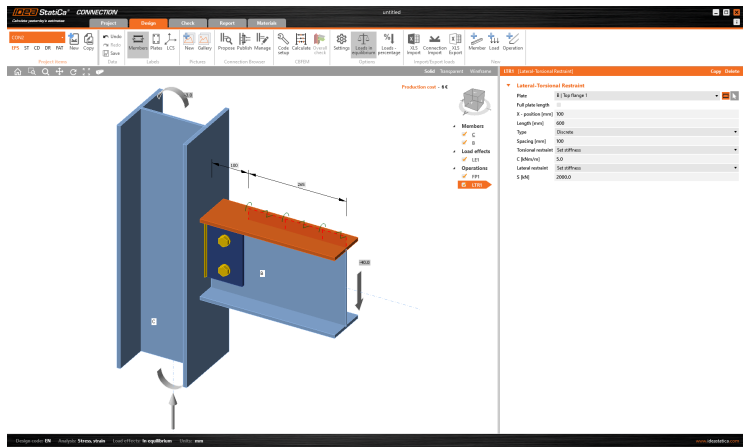
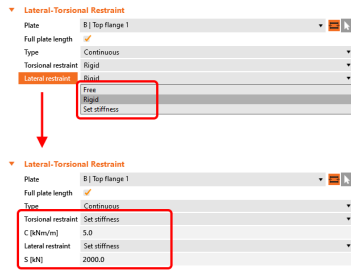
The length of the restraint can be either full or partial.



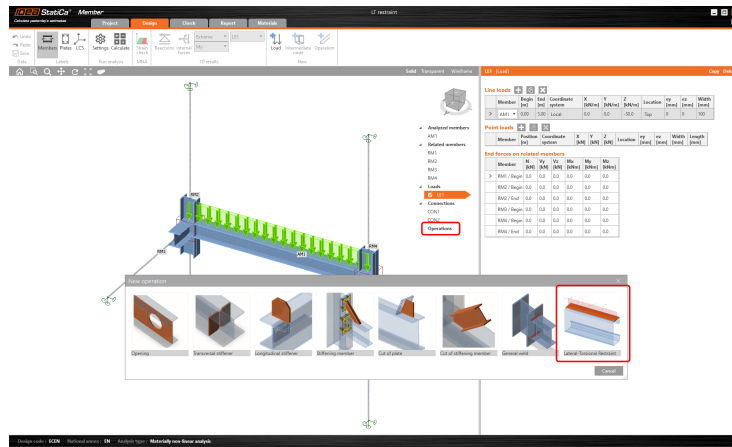
The restraint can be set as continuous or discrete.



And the stiffness of the torsional or/and lateral restraint can be defined independently as free, rigid or with a specific value and this can be set independently for torsion and lateral buckling.



The operation Lateral-Torsional Restraint has been added to the Member application as well.



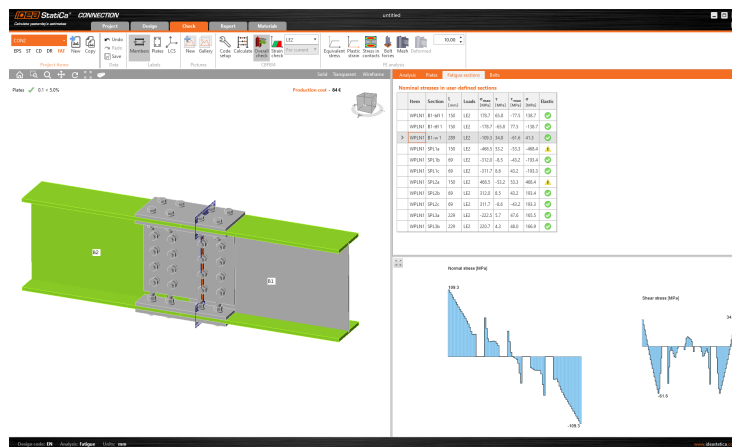
Available in both Expert and Enhanced editions of **IDEA StatiCa Steel**.

Introduction video of the Lateral-Torsional Restraint at the member:

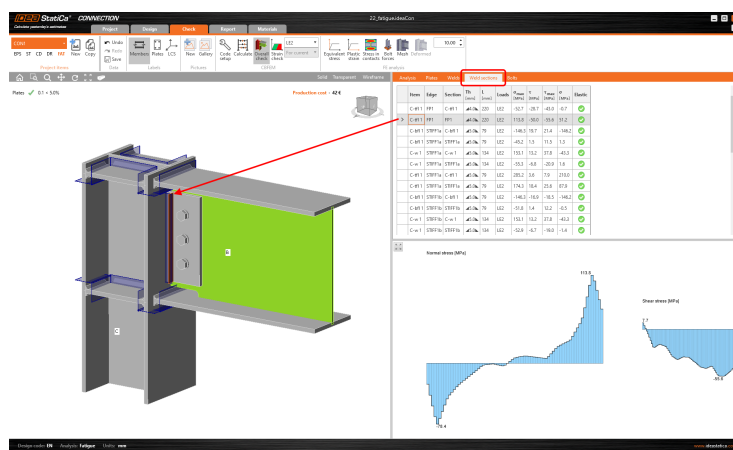
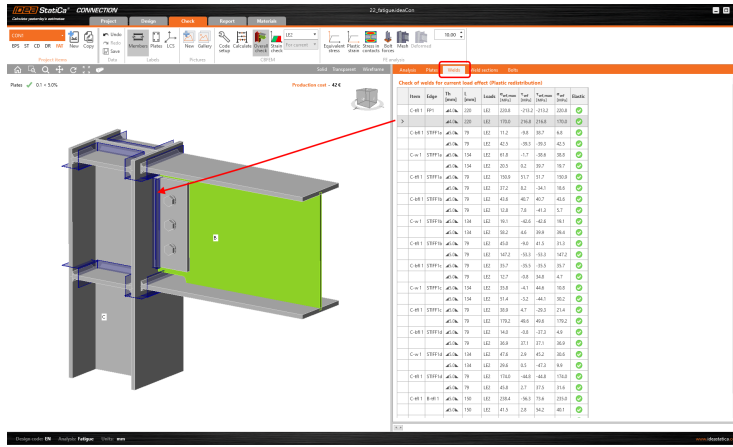
Fatigue analysis improvements

There are several improvements in the analysis of fatigue:

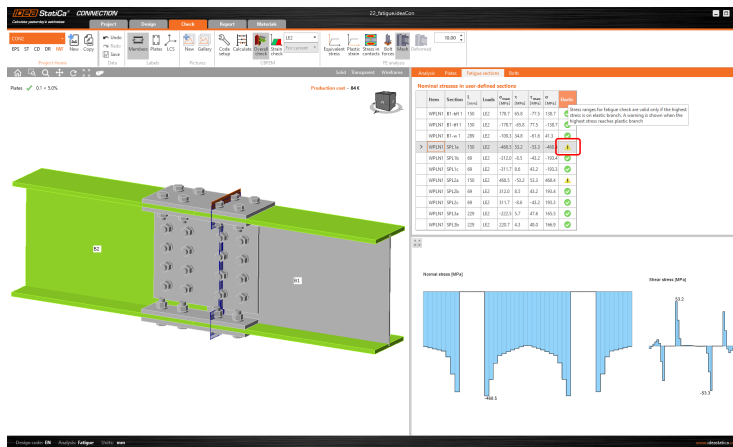
- The evaluation of stress in the fatigue analysis was improved by taking the bolt holes into account. Holes (e.g. bolt holes) are taken into account in the fatigue section. That means the stress concentrations now correctly influence the calculation of the stress unlike in the past when the stress was averaged along the whole section length. The part of the section passing the (bolt) hole has zero stress.



- The fatigue stress can be displayed in welds directly as well as in plate sections near the weld (as it used to be in previous versions). In results, open the Weld tab to see the maximum normal stress ($\sigma_{wf,max}$) and corresponding shear stress (τ_{wf}) and maximum shear stress ($\tau_{wf,max}$) and corresponding normal stress (σ_{wf}). The results for plates at the weld are now available under the Weld sections tab. You can get more details in the [Theoretical Background: Fatigue analysis type](#).



- If the strain reaches the plastic branch of the material diagram, a warning is shown (for plates, sections, bolts, welds, and anchors).



The features have been available since the 21.1.1 patch.

Available in both Expert and Enhanced editions of IDEA StatiCa Steel.

Through long bolts for hollow sections warning

There is a warning regarding the analysis of through bolts for hollow sections (e.g. RHS). Both Bearing and Friction types of bolts in IDEA StatiCa applications are considered in the CBFEM analysis as nonlinear springs.

For this reason, analysis of through bolts is not accurate and those should be avoided in the model for code-checking connections and as member details. Through bolts should be modeled as solid pins, which is not yet a supported feature in IDEA StatiCa. Read more about bolts in the article [Bolts and preloaded bolts](#).

Bolts are considered as through bolts for bolted connections with a gap between any connected plates greater than the tolerance limit of 3 mm.

For bolts with such a gap whose type is **Friction** - analysis can not run and a message is displayed in the 3D window with the name of the operation and the explanation.

For bolts with such a gap whose type is **Bearing - tension/shear** - there is a note in the report stating the results may not be accurate.

Bolts										
Name	Loads	F_{LEd} [kN]	V [kN]	U_{LE} [%]	$F_{b,Rd}$ [kN]	U_{bE} [%]	U_{tE} [%]	Status		
B1	LE1	0.0	6.3	0.0	88.1	7.2	6.7	OK		
B2	LE1	0.0	6.2	0.0	103.9	6.6	6.6	OK		
B3	LE1	0.8	6.2	0.6	89.1	7.0	7.0	OK		
B4	LE1	0.0	6.3	0.0	118.8	6.7	6.7	OK		
B5	LE1	0.0	6.2	0.0	129.2	6.6	6.6	OK		
B6	LE1	0.8	6.2	0.6	89.1	7.0	7.0	OK		
B7	LE1	0.0	6.3	0.0	118.8	6.7	6.7	OK		
B8	LE1	0.0	6.2	0.0	129.2	6.6	6.6	OK		
B9	LE1	0.8	6.2	0.6	89.1	7.0	7.0	OK		
B10	LE1	0.0	6.3	0.0	88.1	7.2	6.7	OK		
B11	LE1	0.0	6.2	0.0	88.1	7.1	6.6	OK		
B12	LE1	0.8	6.2	0.6	89.1	7.0	7.0	OK		

Design data			
Name	F_{LEd} [kN]	$P_{b,Rd}$ [kN]	$F_{t,Rd}$ [kN]
M20 B 8 - 1	141.1	234.7	94.1

Symbol explanation

- $F_{b,Rd}$ Bolt tension resistance EN 1993-1-8 tab. 3.4
- F_{LEd} Tension force
- $P_{b,Rd}$ Punching shear resistance
- V Resultant of shear forces V_1 , V_2 in bolt
- $F_{t,Rd}$ Bolt shear resistance EN 1993-1-8 tab. 3.4
- $F_{b,Rd}$ Plate bearing resistance EN 1993-1-8 tab. 3.4
- U_{LE} Utilization in tension
- U_{bE} Utilization in shear

Bolts: B1, B2, B3, B4, B5, B6, B7, B8, B9, B10, B11, B12 There is a gap between connected plates. Bolts should be designed as pins. Provided resistances of bolts in shear and cables in bearing may be incorrect

This update has been available since the 21.1.2 patch.

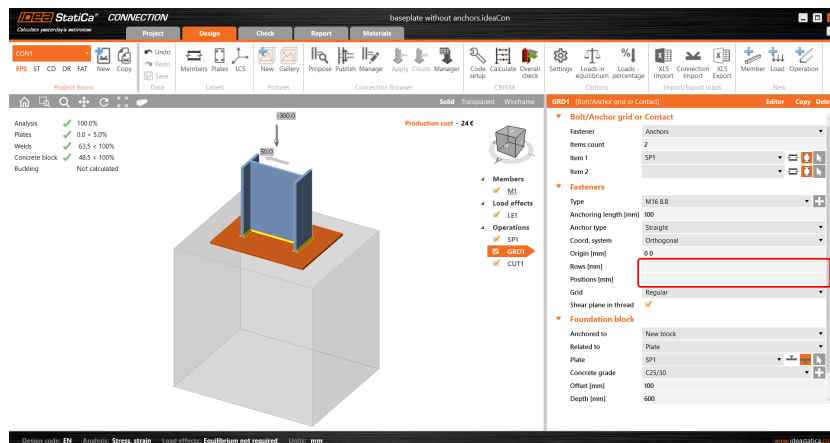
Available in both **Expert** and **Enhanced** editions of [IDEA StatiCa Steel](#).

Baseplate connection without anchors

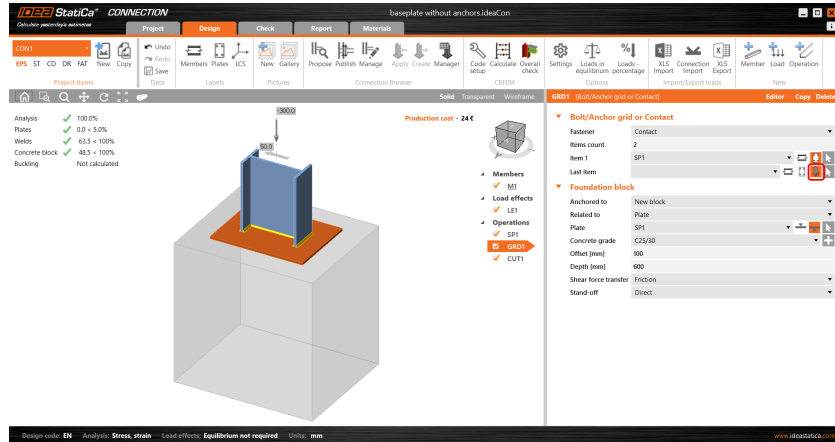
In IDEA Statica you can design and code-check baseplate connections without anchors. Such a feature helps engineers with compression checks in concrete.

In the IDEA StatiCa Connection application, the concrete block is simplified as 2D contact elements. The contact between the concrete and the baseplate works only in compression and the compressive force is transferred via the Winkler-Pasternak subsoil model.

A concrete block can be set up with an anchor grid with empty rows or positions of anchors.



Or you can set the simple Contact option.



This update has been available since the 21.1.4 patch.

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

Loads in equilibrium by default

To provide better safety and reduce the risk of underestimating the load effects on supported members, the load in equilibrium will be set as the default option when starting a new project in version 22.0.

The importance of defining the load on both ends of the supported member is described in many Support Center articles (such as [Equilibrium and supporting member](#)).

From version 22.0 on, the default settings of the load will be "Loads in equilibrium" and the table with unbalanced forces will be displayed below. The user is responsible for the correct input of the internal forces on all members. Of course, the settings can be switched off, but attention must be paid to possible consequences.

All templates in IDEA StatiCa Connection have been adjusted accordingly.

Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

Material library and code checks update

The Material and product range library (MPRL) in Connection and Member as well as some specific code checks have been updated for several design codes.

European standard (EN)

- Change bolt assemblies in EN according to ISO 4014
- Add non-standard bolt assemblies M14, M33, etc. to EN MPRL

Changes may have possibly a very small effect on the punching shear resistance of bolts.

The update has been available since the 21.1.1 patch.

UK cross-sections (BS)

The following sections have been added to MPRL:

Added UB	Added UC
UB 406 x 140 x 53	UC 152 x 152 x 44
UB 406 x 178 x 85	UC 152 x 152 x 51
UB 457 x 191 x 106	UC 203 x 203 x 100
UB 457 x 191 x 133	UC 203 x 203 x 113
UB 457 x 191 x 161	UC 203 x 203 x 127
UB 533 x 165 x 66	
UB 533 x 165 x 75	
UB 533 x 165 x 85	

This update has been available since the 21.1.4 patch.

Cross-sectional properties are taken from: <https://www.steelforlifebluebook.co.uk/ub/ec3-ukna/section-properties-dimensions-properties/>

US standards (AISC)

- ASTM F1554 bolt assemblies were added

The update has been available since the 21.1.1 patch.

- Update of cross-sections - open sections W, S, M according to Steel Design Manual 15.0.
- The following sections have been added:

W40X655
W36X925
W36X853
W36X802
W36X723
W21X275
W21X248
W21X223
W14X873
W14X808

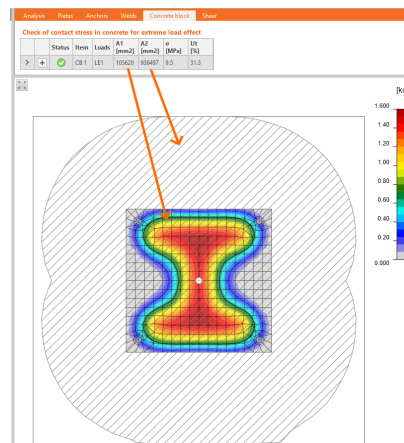
- Update of HSS from 14.1 to 15.0 (new cross-sections were added).
- The division between A1085, A1065 (nominal wall thickness) and A500, A501, A618, A847 (reduced wall thickness).

- HSS (14.1) were updated to HSS (15.0 - A1085, A1065)
- HSS (15.0 - A500, A501, A618, A847) are new cross-section tables
- Pipes (A53) are from now on with reduced thickness - results will change due to this change.

These updates above have been available since 21.1.5 patch.

- **Concrete in compression**

The resistance of concrete in compression may now be reduced by stress cut-off ratio in Code setup. The loaded area, A_1 , is determined as the area, where stress in concrete is higher than the peak stress multiplied by stress cut-off ratio. Loaded area A_1 and Supporting area A_2 are shown in 2D window. The stress cut-off ratio is set by default to 0.4.



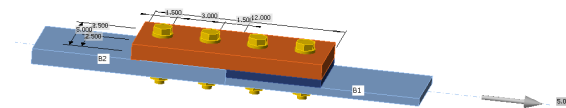
- **Bolts in bearing**

The resistance of bolts in bearing can be now increased by disregarding deformation at the bolt hole at service load as a design consideration according to AISC 360-16: J3.10: Bearing and Tearout Strength at Bolt holes. A checkbox is now available in Code setup.

- **Reduction of bolt shear resistance for connection with fillers**

Bolt shear resistance for connections with fillers is now reduced according to AISC 360-16, J5.2. A filler is recognized by IDEA StatiCa as an inserted plate that is not connected to any other weld or bolt grid. It is not checked whether the filler is properly developed.

The factor for fillers of slip-critical connections, h_f , is now determined properly according to AISC 360-16, J3.8. For two or more filler plates, h_f is reduced to 0,85.



These three updates above are available since version 22.0.0.

Australian standard (AS)

The new Australian steel code was released in 2020 and it replaced the code from 1998. In IDEA StatiCa Connection, we have updated the checks as well as the documentation available on our web or in the application.

The update has been available since the 21.1.1 patch.

Furthermore, we have included several new bolt assemblies according to EN 14399-3:2015:

- HR8.8 bolt assemblies
- HR10.9 bolt assemblies

Regarding the code check, it was complemented for high strength bolt assemblies according to ASI TN001 by:

- Reduction factor 0.5/0.6 for tensile strength, $f_{uf} > 840$ MPa, and shear plane intercepting bolt threads

Shear resistance check (AS 4100-2020 - Cl.9.2.2.1)

$$\phi V_f = \phi \cdot a_v \cdot 0.62 \cdot f_{uf} \cdot A_c = 61.681 \text{ kN} \geq V_f^* = 10.899 \text{ kN}$$

Where:

- $f_{uf} = 1040.0$ MPa – minimum tensile strength of the bolt
- $a_v = 5/6$ – reduction factor due to reduced bolt ductility
- $A_c = 143 \text{ mm}^2$ – minor diameter area of the bolt
- $\phi = 0.80$ – resistance factor

- Minimum bolt tension for preloaded bolts:

For bolts grade 8.8 ($f_{uf} = 830$ MPa) and grade 10.9 ($f_{uf} = 1040$ MPa), the minimum bolt tension is taken from AS 4100:2020, Table 15.2.2.2:

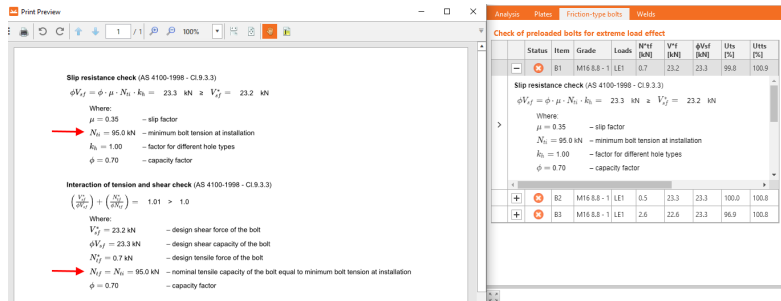
Table 15.2.2.2 – Minimum bolt tension

Nominal diameter of bolt	Minimum bolt tension, kN	
	8.8	10.9
M16	95	130
M20	145	205
M24	210	295
M30	335	465
M36	490	680

NOTE: The minimum bolt tensions given in this Table are approximately equivalent to the minimum proof loads derived from a proof load stress of 600 MPa for grade 8.8 bolts and 830 MPa for grade 10.9 bolts, as specified in AS 4291.1.

For bolt strengths and dimensions missing in the table, following formulas are used:

- for $f_u < 1000$ MPa: $N_{ti} = f_u \cdot 60/83 \cdot A_s$
- for $f_u \geq 1000$ MPa: $N_{ti} = f_u \cdot 83/104 \cdot A_s$



Indian standard (IS)

- Added cross-sections: CHS, RHS, and SHS by Tata Structura.

The update has been available since the 21.1.5 patch.

Russian standard (SP)

- The dimensions unified according to ISO 4014. The update has been available since the 21.1.1 patch.
- The last revision of the Russian code affected the friction-type bolts code check in IDEA StatiCa Connection. The formula for the design tensile strength of a high-strength bolt $R_{bh} = 0.7 \cdot R_{bun}$ is replaced by design tensile strength R_{bt} according to SP 16, Table 5.

The updated formula looks like this:

Slip resistance check (SP16 - Cl.12.3)

$$N_{sf} = Q_{th} \cdot \gamma'_b \cdot \gamma_c = 15.5 \text{ kN} \geq N_s = 10.2 \text{ kN}$$

Where:

$$Q_{th} = 21.1 \text{ kN} \quad \text{-- design slip resistance of one preloaded bolt and one friction plane}$$

- $Q_{th} = \frac{R_p \cdot A_{bol}}{\gamma_b}$, where:
 - $R_p = 448.2 \text{ MPa}$ -- design preload in the preloaded bolt - Table 5
 - $R_{bol} = 830.0 \text{ MPa}$ -- ultimate tensile resistance of the bolt
 - $A_{bol} = 157 \text{ mm}^2$ -- tensile stress area
 - $\mu = 0.35$ -- slip factor
 - $\gamma_b = 1.17$ -- coefficient in case of bolt tightening
- $\gamma'_b = 0.74$ -- friction joint service factor for bolts in a friction-type connections loaded by combined shear and tension
- $\gamma'_b = \max(\gamma_b \cdot (1 - \frac{N_s}{P_b}), 0)$, where:
 - $\gamma_b = 0.80$ -- friction joint service factor
 - $N_s = 5.4 \text{ kN}$ -- tensile force in a bolt
 - $P_b = R_p \cdot A_{bol} = 70.4 \text{ kN}$ -- preload in a bolt
- $\gamma_c = 1.00$ -- service factor

The update has been available since the 21.1.4 patch.

Find out more about all the MPRL possibilities in the [Support Center](#).

Available in both **Expert** and **Enhanced** editions of [IDEA StatiCa Steel](#).

Export of moment-rotation curve to CSV and DXF

The moment-rotation curve of the rotational and axial stiffness in the Stiffness analysis or the load-strain curve in the Joint design resistance are now possible to export into CSV or DXF files for further custom processing. Another option is to export the figure as a bitmap.

Available in both **Expert** and **Enhanced** editions of [IDEA StatiCa Steel](#).

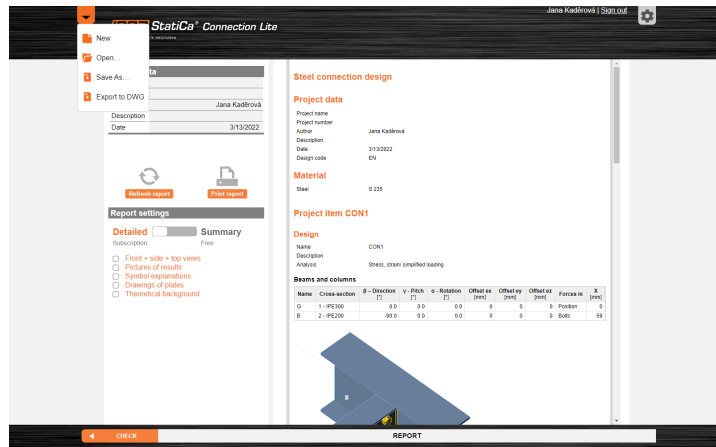
Connection Lite update

Our popular free cloud service for checking of dozens of joint configurations has been updated to comply with the desktop version.

The results of the current version of [Connection Lite](#) comply with the released version of our desktop applications. In the same way, the [IDEA StatiCa Viewer](#) was updated as well.

You can design, analyze and check your connection, export the connection to 3D DWG, save and open the project and print the report under your account.

Logging by Google account is disabled in the current version.



Available at <https://connection.ideastatica.com/>.

News for BIM links

Structural engineers use **IDEA StatiCa Checkbot** a lot - on average, nearly 40,000 steel connections are imported every month, worldwide. Version 22 makes this even more usable for the everyday design process, for all our supported **BIM links**.

IDEA StatiCa Checkbot version 22.0 brings you:

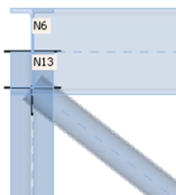
Add or merge nodes of connections and members in Checkbot

Since version 22.0 defining and editing new design items is possible in the Checkbot app. You can simply select and create connections and members directly in the structural model inside the Checkbot.

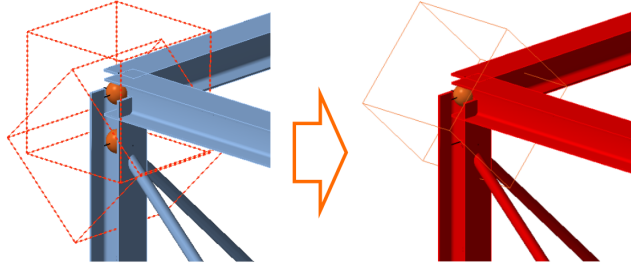
Why did we come up with this feature? In previous versions, the user could face the obstacle that some Connections were not created automatically or had two connection models for one structural joint. This issue can now be solved by manual creation of whatever joint or member needs to be analyzed.



For import from FEA applications both options, Connections, and Member creation is available. For import from CAD applications, only the Connection option is present.



Maybe an even more important issue solved with this manual creation is the connection arrangement with close structural joints which should be analyzed in one common Connection model. Typically with eccentric diagonal bracing members, it's very often the case. Now the user can decide which specific joints should be analyzed in the common Connection model.



The **secondary use-case** for this feature is in the use of IOM/ SAF / RAM Import. When a user uses **Checkbot** as a standalone application, the structural model is brought from 3rd party application automatically, but the Connections and Members to be analyzed need to be selected manually. And that's exactly where this feature helps a lot.

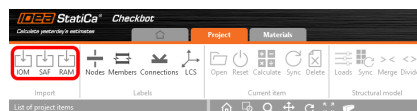
Two possibilities of creating new Connections:

- 1 one connection - the user can go one by one
- 2 multiple connections at once - several Connections can be created at once

Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

A wider range of 3rd party applications

Our list of Import formats for standalone use of Checkbot has grown in version 22.0. Besides already embedded IOM and SAF formats, a new RAM option is here.



Bentley RAM Structural System (RSS) link with RAM interface

We have implemented an **FEA link with Bentley** built on the **.rss format** import. This link is a result of communication with our customers and their requests for linking to this popular FEA software.

After saving the structural model in RAM Structural System software, the **.rss** file is created. Now you have an option to simply import these files into the Checkbot, where the model will be loaded and displayed in the 3D graphic window.



Here you can download a **PDF file with the details** for use of RAM SS BIM-link import.

Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

Tekla Structural Designer linked through Checkbot

Not only the CAD app Tekla Structures is possible to connect with IDEA StatiCa. Also, another Trimble company application is now possible to link - the FEM app called **Tekla Structural Designer** (TSD).

This link is not actively supported by IDEA StatiCa but it is being developed on side of Trimble.



STRAP linked through Checkbot

Another 3rd party application possible to connect with IDEA StatiCa apps from version 22.0 is STRAP. This **STR**uctural **A**nalysis **P**rogram is a solution of Israel Software company ATIR Engineering Software Development. The user can import the structural model into Checkbot app thanks to the integrated plug-in.

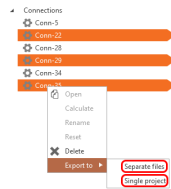
This link is not actively supported by IDEA StatiCa but it is being developed on side of ATIR.



Export of editable connections

We hear very often from our customers, that they like to work with IDEA StatiCa Connection Member models independently from the Synced models inside the Checkbot. For many reasons, this can help in everyday design workflows.

Therefore the possibility of exporting editable files from **Checkbot** was introduced in version 22.0.



The user can now export particular connections into:

- 1 **Separate files** - individual IDEA Connection project files would be created for every Connection selected in Checkbot's list of project items
- 2 **Single project** - one Multi-connection IDEA project file with several connection models will be created from selected Connections listed in Checkbot

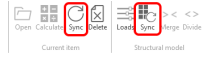
Once the Separate project files are exported, they are no longer linked to the original Checkbot structural model, and therefore no further synchronization with the 3rd party applications or their management in Checkbot is possible.

Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

Syncing of solved connection models

Users of Checkbot app asked for an option of synchronization for specific design items only. This is possible from version 22.0 on.

With a feature of Sync Current Item, the user has an option to select particular Connections or Members which should be updated and therefore save some time with unnecessary synchronization. Furthermore, the user can be sure that already solved Connections won't be affected by the synchronization of the whole structural model.

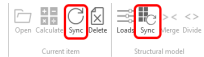


Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

Sync all vs. Sync current item

Users of the Checkbot app asked for an option of synchronization for specific design items only. This is possible from version 22.0 on.

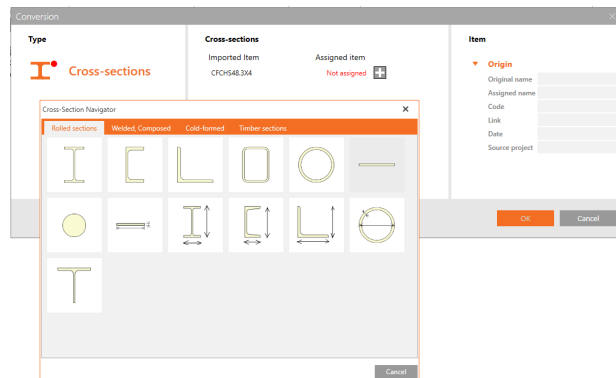
With a feature of Sync Current Item, the user has an option to select particular Connections or Members which should be updated and therefore save some time with unnecessary synchronization. Furthermore, the user can be sure that already solved Connections won't be affected by the synchronization of the whole structural model.



Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

Improved Cross-section Conversion

From version 22.0 on, the Conversion table for cross-section got the more usable skin. In previous versions, looking for your particular cross-section could be slightly unclear as the user had to scroll down the long list of items. As an improvement of usability, we edit this conversion table to be very similar to what is well proven in the Connection app. Now you can select from the set of tiles with distinguished cross-section types.



Available in both **Expert** and **Enhanced** editions of **IDEA StatiCa Steel**.

The **Checkbot** can be started from your **3rd party application** or as a standalone app and enables you to combine inputs from multiple sources.

If you want to check the compatibility with your particular application, just take a look at our list of actively supported versions.

Supported BIM links in version 22.0

The latest two major releases supported

We have made several actions to ensure the consistent update process of our BIM links. In each major release of IDEA StatiCa (this year, it will be 22.0 and 22.1), 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 the 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 be always presented on our web. With the 22.0 release, we will support the versions presented in the first column of the table. The “In development” column represents the newest versions that will start to be supported in a patch of 22.0. The fourth column shows versions that are no longer supported.

IDEA StatiCa - Steel		22.0		
Application	Supported	In development	Obsolete	
Advance Steel	2021, 2022	2023	-	
Revit	2021, 2022	2023	-	
Talis Structures	2021, 2022	-	2020	
Advance Design	2021, 2022	-	-	
AxisVM	X5.4, X6.1	-	-	
ETABS	19, 20	-	18	
midas Civil / Gen	2021, 2022	-	2020	
RAM Structural System	17.02, 17.03	-	-	
RFEM / RSTAB	5.27 / 8.27, 5.28 / 8.28	6.02 / 9.02	5.25 / 8.25, 5.26 / 8.26	
Robot Structural Analysis	2022, 2023	-	2021	
SAP2000	23, 24	-	22	
SCIA Engineer	21, 21.1	-	20	
STAAD.Pro	22	-	-	

IDEA StatiCa - Concrete		22.0		
Application	Supported	In development	Obsolete	
Advance Design	2021, 2022	-	-	
AxisVM	X5.4, X6.1	-	-	
midas Civil / Gen	2021, 2022	-	2020	
RFEM / RSTAB	5.27 / 8.27, 5.28 / 8.28	6.02 / 9.02	5.25 / 8.25, 5.26 / 8.26	
Robot Structural Analysis	2022, 2023	-	2021	
SAP2000	23, 24	-	22	
SCIA Engineer	21, 21.1	-	20	

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

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

RAM Structural System BIM link

With version 22.0 a next 3rd party application is possible to connect with IDEA StatiCa apps through Checkbot. The Structural model from RAM Structural System from Bentley can be imported through .rss file format import.

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

News for Concrete and Prestressing

IDEA StatiCa Detail has earned a leading position in the design and assessment of concrete structures and details, especially in cases of difficult parts of the structures with openings or abrupt changes in geometry where classical beam theory cannot be applied. We have listened to your requests and developed frequently asked features to IDEA StatiCa Detail. Now engineers can minimize the errors and let IDEA StatiCa calculate the self-weight of the structure automatically. No matter how complex the geometry, haunches, complex cross-sections, and discontinuity regions with different thicknesses are, we improved and speed up the CSFM solver and expanded the analysis results. IDEA StatiCa Detail is a powerful, fast, and safe tool for the structural design of concrete structures.

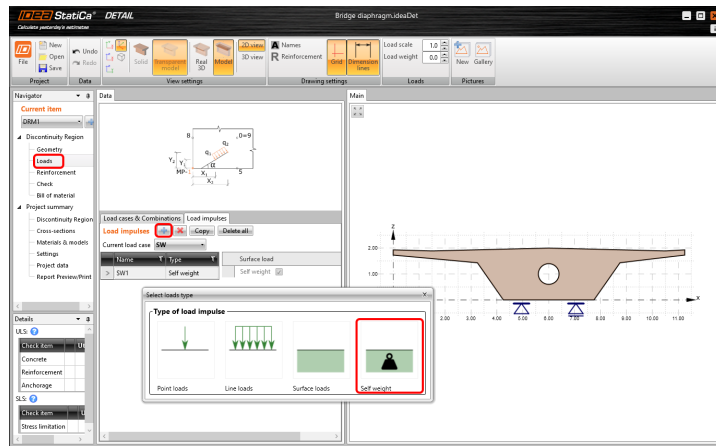
IDEA StatiCa Member - in the previous version, we implemented the GMNIA nonlinear method. In version 22, we focused on user interface improvements, such as facilitating the input of geometry and loads, and especially on the interpretation of the outputs of the nonlinear analysis. These UI improvements speed up the process of modeling, facilitate the inputs and make your work in IDEA StatiCa Member even more efficient.

The improvements in Concrete and Prestressing include:

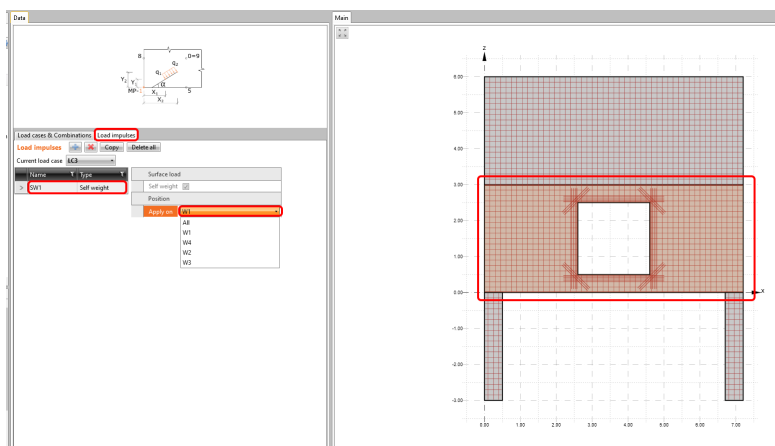
Auto-calculation of self-weight of discontinuity regions

Self-weight of the analyzed structure is from IDEA StatiCa v22.0 considered automatically by the software, so the users don't have to spend extra time with manual calculation.

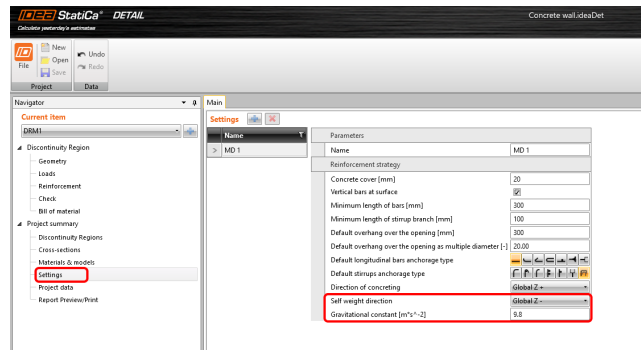
The functionality **Self-weight** is available for all structures modeled as a whole except parts of the beams with trimmed ends. It works similarly to Surface load type, the value of the self-weight is calculated automatically for each part of the structure according to its real thickness. The Self-weight is defined as a Permanent load case.



If the model contains more regions (walls), the self-weight can be defined for each region separately. There are options for assigning the self-weight for a particular region or for all of them at the same time.



Properties such as the direction of gravitation and gravitational constant can be changed in the Settings.



Available in both Enhanced edition of [IDEA StatiCa Concrete](#) and [IDEA StatiCa Prestressing](#).

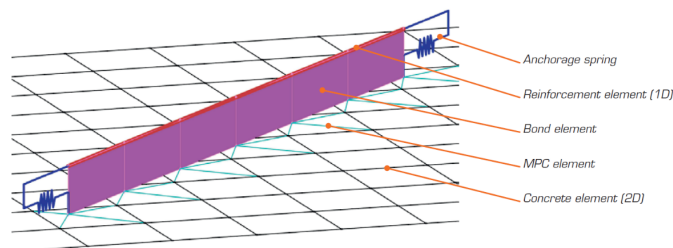
Introduction video of the Self-weight auto-calculation feature:

Up to 4-times faster CSFM solver

Innovative design and code-check of reinforced and prestressed concrete discontinuity regions can be done via CSFM analysis implemented in IDEA StatiCa Detail.

Almost every release of IDEA StatiCa we announce the speeding-up of the CSFM calculation. Let me explain why is that. And why is there still an opportunity to speed up some part of the calculation? After pressing the Calculate button in IDEA StatiCa Detail, several actions are performed:

- Preparation of the CSFM model (meshing concrete region, detecting reinforcement, and connecting the concrete and steel entities with multi-point constraint elements bond elements) - **preprocessor**

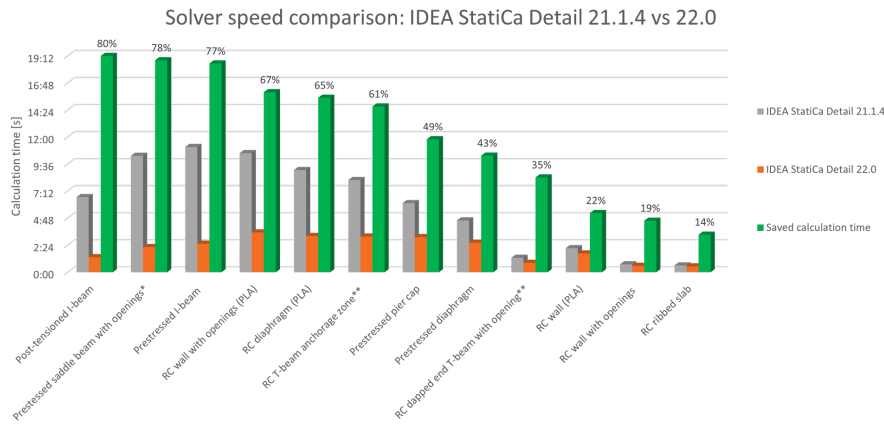


- **CSFM (Compatible stress field method) analysis - solver**
- Result interpretation and code-checks - **postprocessor**

In the last release, we focused on the preprocessor. The data preparation is an essential part of the CSFM analysis. The more complex is the discontinuity region, the more time-consuming is this part. That is why we keep improving the algorithms for meshing and assembling the finite element model. [Read more about it in the previous release of IDEA StatiCa.](#)

This time we focused on the **solver** part. We implemented a new material model which considers the Poisson number. This modified material model makes the calculation more stable and requires fewer iterations, hence faster calculation time. Other improvements like better control over the model divergency and improved algorithms for assembling the stiffness matrices also caused additional speeding-up of the solver.

The most significant solver speeding-up can be observed for the prestressed members or projects containing lots of load cases/combinations. The CSFM solver is now up to 4 times faster than the previous versions! Look at the chart of the comparison of the calculation time between IDEA StatiCa versions 21.1.4 and 22.0.



Available in both Enhanced editions of [IDEA StatiCa Concrete](#) and [IDEA StatiCa Prestressing](#).

Thorough CSFM results available

Thanks to improved post-processing of the CSFM results you can observe all results at any part of the analyzed structure and comprehensive the structure behavior upon defined loads.

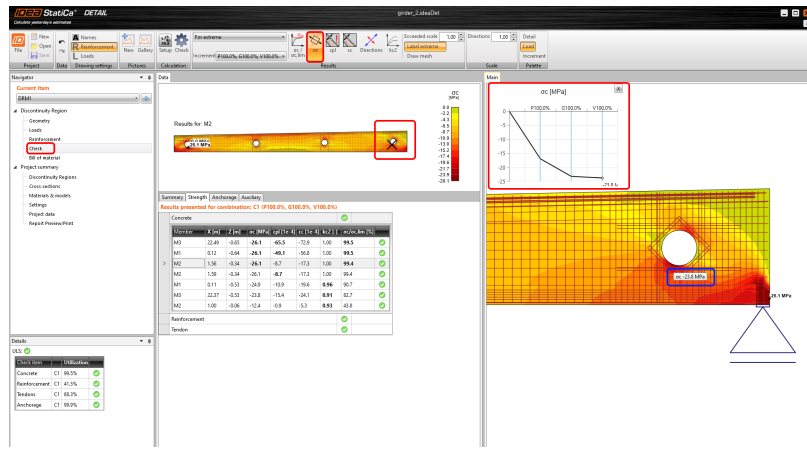
In the older versions of IDEA StatiCa Detail, you obtained only the extreme results from CSFM analysis. The critical spot of the structure was labeled, and the remaining concrete parts or reinforcements possessed the corresponding color based on the color palette. If there was an interest in other values apart from the critical point, you needed to estimate them based on the colors.

All CSFM outputs are available

Now, no estimation is necessary! All CSFM results can be displayed as a tooltip by staying with your mouse pointer at the point of your interest for a few seconds or simply using the left-click button. The displayed value always corresponds to the result of the closest node of the finite element mesh of the reinforced concrete model. This functionality is available for all CSFM outputs and checks for all entities of the finite element model, i.e., concrete, reinforcement and tendon.

Results in a Chart

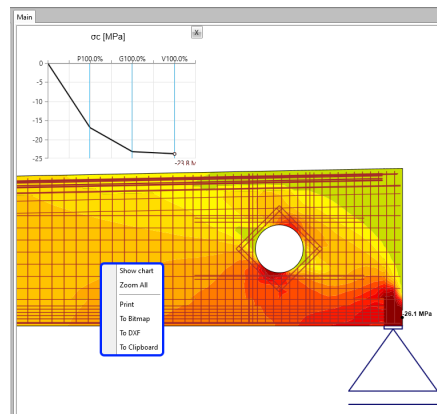
And there is more! Use the right-click button and observe new functionality - result interpretation in a Chart. After clicking on a part of a structure (Shift+ left-click button or running the context menu by right-click button), the load-deformation curve or curve presents the development of the observed variable (stress, strain, crack width, etc.) is drawn in the upper-left corner of the main scene. The curve presents all load increments that were applied in CSFM analysis for the current load case or combination. The value of the load increment selected in the ribbon of Results is highlighted in the chart by the red color. The horizontal axis represents the applied load increment (P = 100% of applied prestressing, G = applied 100% of permanent load, V = applied 100% of variable load). The vertical axis represents the observed CSFM results, such as stresses, strains, deformation, crack width, anchorage forces, and others. The data from the chart can be exported to a bitmap, CSV, or DXF file using the context menu of the chart (right-click button).



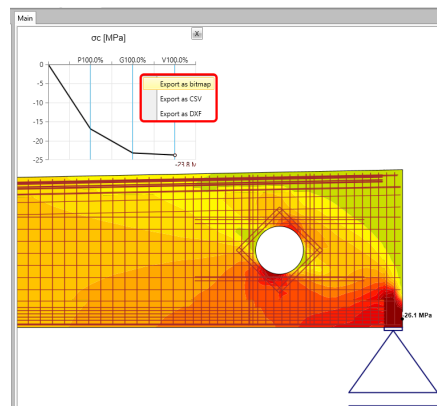
The load-deformation curve is actually the capacity curve (or pushover curve) of the analyzed discontinuity region. So you can use these chart outputs when the structural seismic design is necessary for your project. In that case, you appreciate the data export to CSV file.

Tips on how to work efficiently with thorough results in IDEA StatiCa Detail

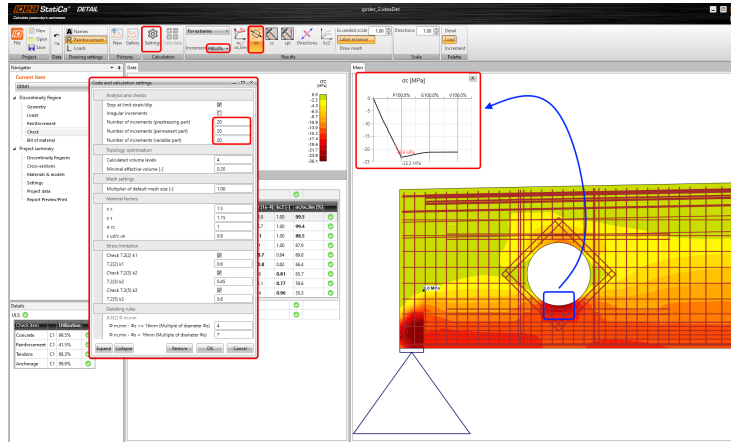
- staying on the point displays the tooltip with the currently observed result
- using left-click button displays the tooltip with the currently observed result
- using right-click context menu within the main scene offers a new possibility to display the detailed results in the chart, apart from the functionalities (Print, export to Bitmap, DXF and clipboard)



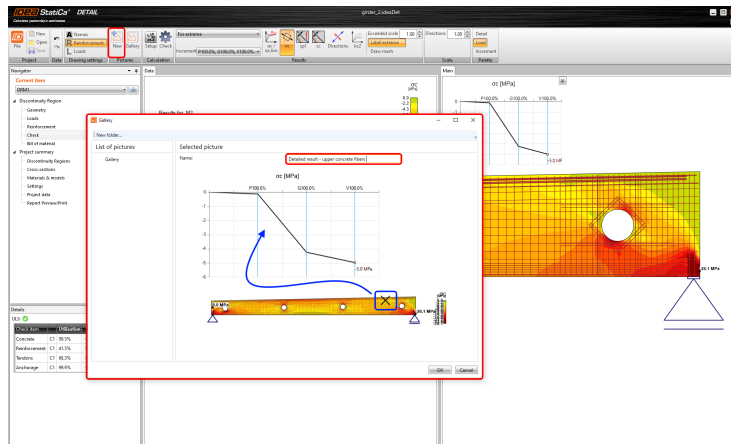
- using right-click context menu within the chart you can export the chart to bitmap, CSV or DXF



- set the number of the load increments to get the smooth capacity curves



- pressing Shift key+left click button within the analyzed structure, you get the corresponding result in the chart
- save the detailed results to the Gallery and attach these snapshots to your report



Available in both Enhanced edition of **IDEA StatiCa Concrete** and **IDEA StatiCa Prestressing**.

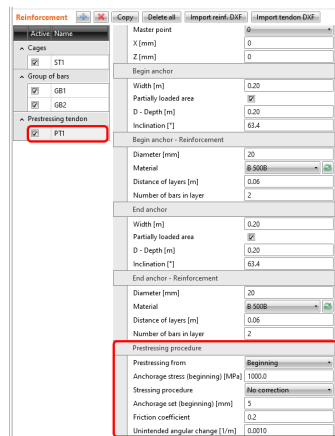
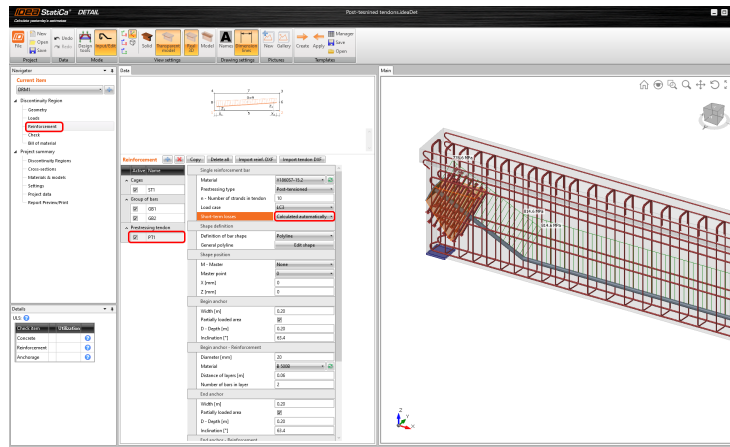
Introduction video of the self-weight auto-calculation feature & thorough CSFM results:

Auto-calculation of short-term losses for post-tensioned tendons

Model, design, and code-check prestressed beams, deep beams via the more advanced method, CSFM. Compatible stress field method.

Since version 21.1, it is possible to model the prestressing tendons and include them in CSFM analysis. Prestressing losses had to be defined by the user. There is an option of auto-calculation of short-term losses for post-tensioned tendons only. You can use the auto-calculation or still define them, it is up to you. We automatically calculate losses due to friction, anchorage set, and short-term relaxation among short-term losses.

If you wish to calculate short-term losses automatically, just select the option Calculate automatically.



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

Bulk import from DXF drawing

Since version 21.1.1, it is possible to use bulk selection in the Detail application when importing a cross-section or reinforcement from a DXF file. Bulk selection is achieved by pressing the Ctrl key and selecting entities one by one or selecting by the selection window.

Bulk import from DXF drawing in Detail application is available in **Concrete Enhanced** and **Prestressing Enhanced** edition of **IDEA StatiCa Concrete**.

Ordering of load cases and combinations in Detail

In previous versions, mixing occurred when a new load case or combination was created. Since version 21.1.1, load cases and combinations have been rearranged according to the ultimate and serviceability limit state.

A great benefit for the users in terms of clarity and load handling.

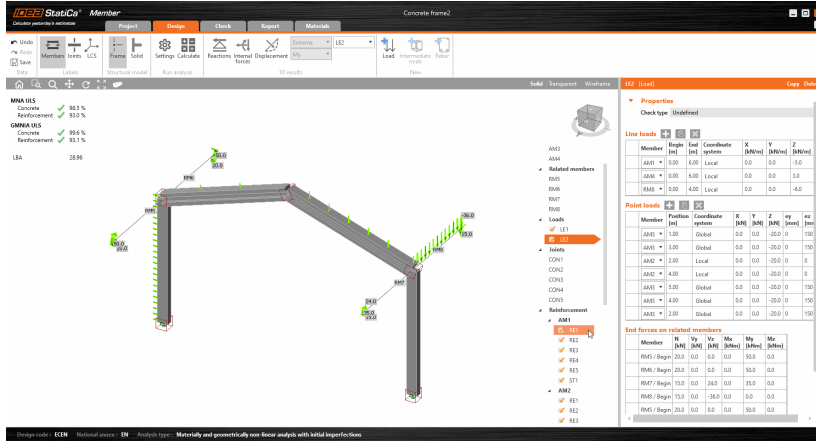
Ordering of load cases is available in **Enhanced** and **Prestressing Enhanced** edition of **IDEA StatiCa Concrete**.

Wireframe view & result interpretation of concrete members

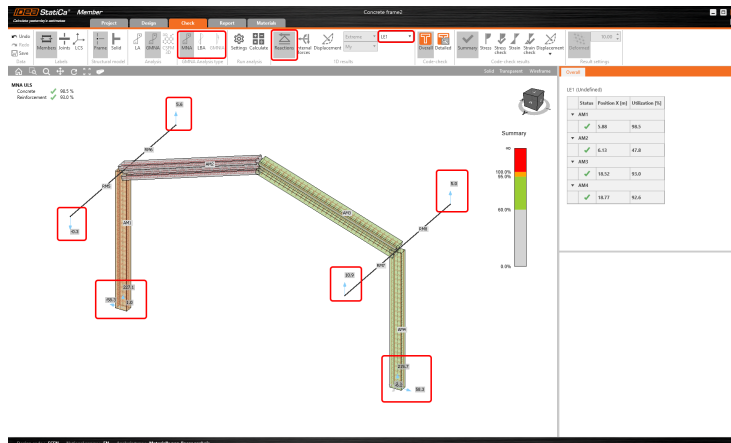
IDEA StatiCa 22.0 Member app is subjected to continuous development. First, the focus was on implementing various types of analyses. As a second we focus on post-processing the results.

Huge improvements were done at result interpretation in **IDEA StatiCa Member** for Concrete

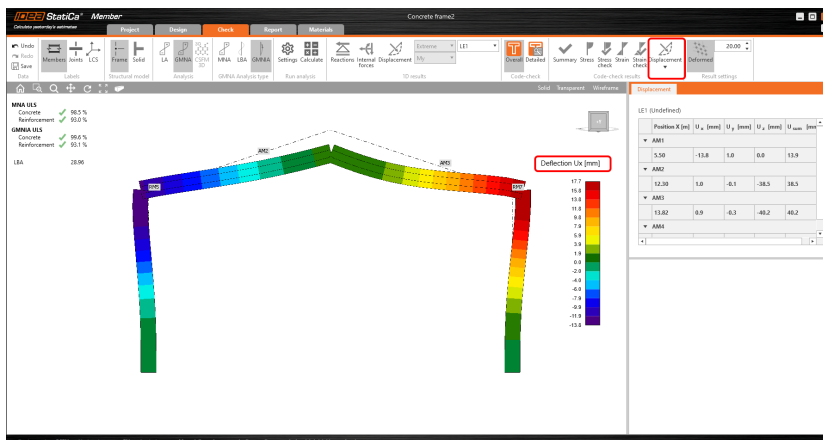
- Structure view has been added - Wireframe view. Now you can observe the structure in a solid, transparent, or wireframe view.

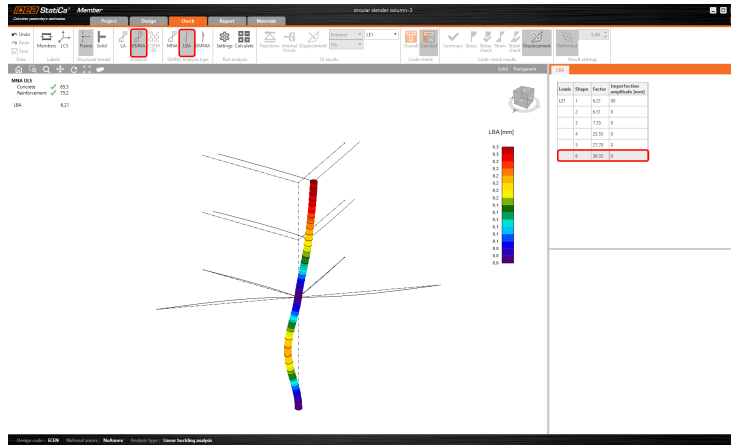


- Reactions displayed in the 3D scene according to selected Load case or Combination

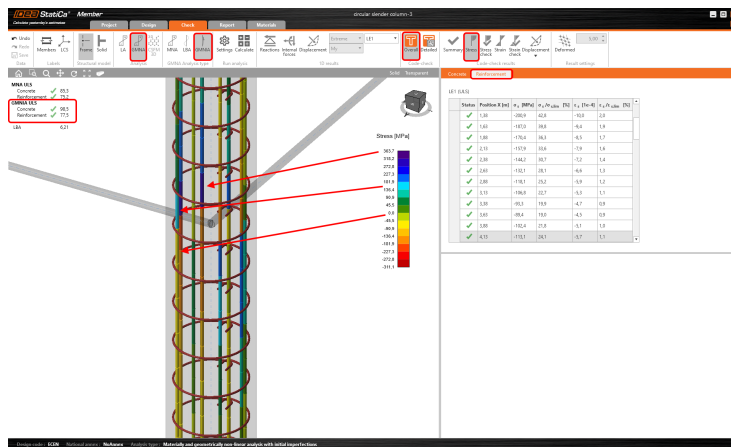
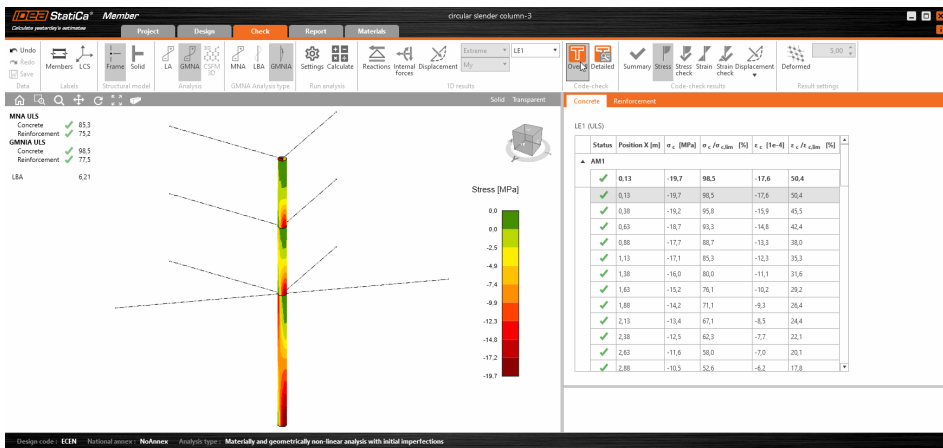


- 3D visualization of the deformed structure. You can observe the total displacement (Usum) or the particular components in directions x,y and z (Ux, Uy, Uz).





- 3D visualization of the overall results from **MNA** and **GMNIA** analysis



Available in all editions of **IDEA StatiCa Concrete and Prestressing**.

Introduction video of the new features in IDEA StatiCa Member:

Model management improved, right-click actions and point loads in Member

IDEA StatiCa Member, the application for concrete design and code-check via standard and more advanced methods. You can use it as standalone or launch it from Checkbot linked with your FEA application.

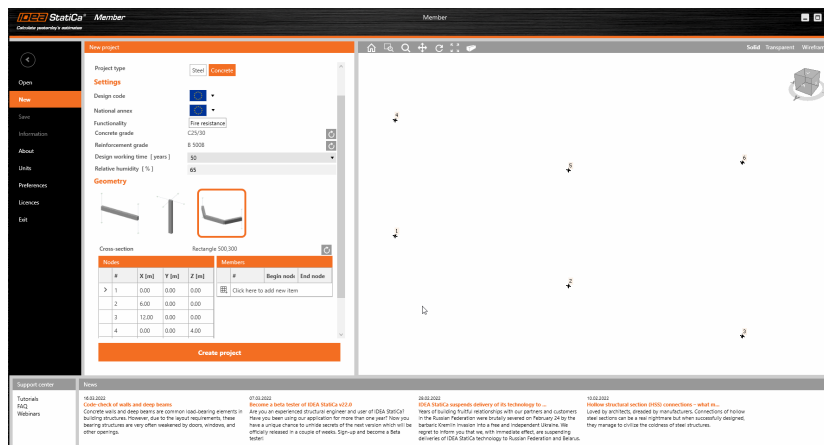
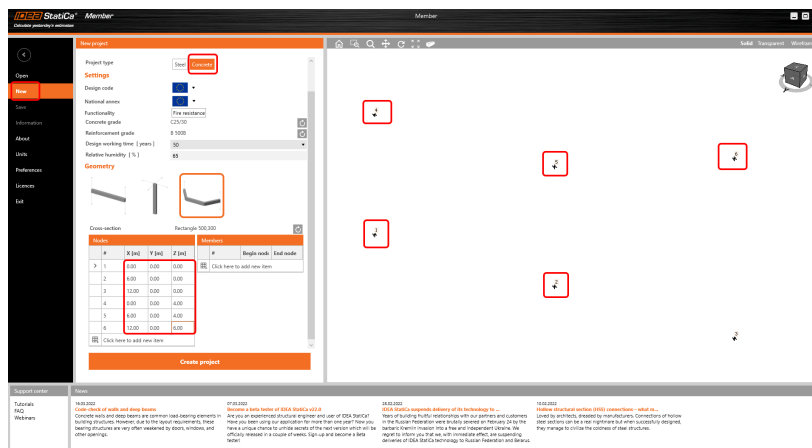
IDEA StatiCa Member has been around for a while. In the previous version, we focused on the implementation of the GMNIA nonlinear method. This time, we focused on user interface improvements that push the Member sky high. There are two ways how to use the Member app.

- Standalone use
- Launching the Member from **Checkbot linked with your FEA application**

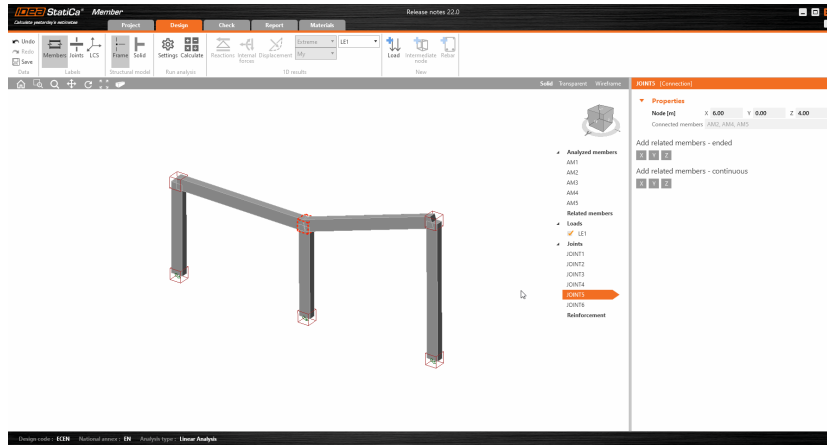
If you use the Member as a standalone application you will appreciate these improvements very much. All these improvements speed up the process of modeling, facilitate the inputs and make your work even more efficient.

List of UI improvements in IDEA StatiCa Member

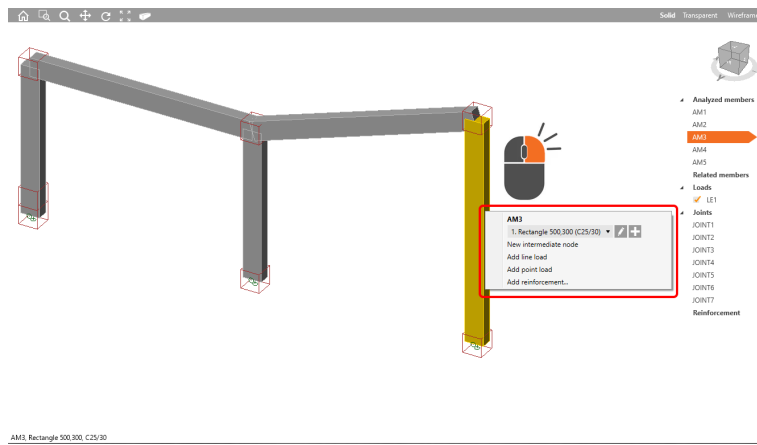
- Display of the defined nodes prior to the member being defined in case of using the general input. It speeds up the process of modeling and minimizes the incorrect inputs.



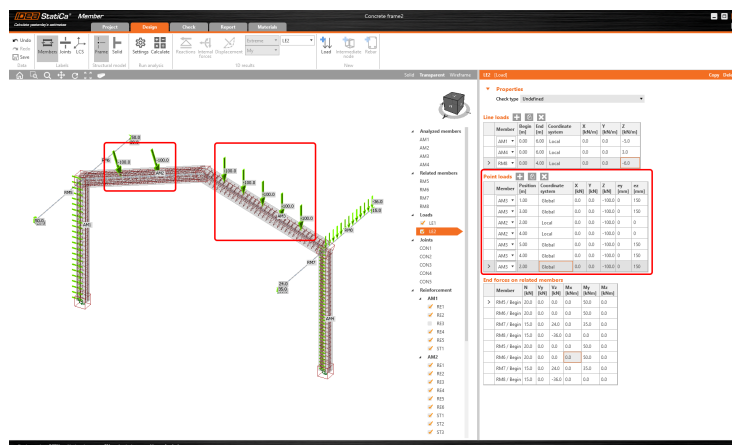
- Work with the well-organized model. In the previous versions, you had to click further to the properties of the member to get to know the name of the joint and its boundary conditions and it all took you precious time. With the new version, the joints are displayed with the cube around that is highlighted when the joint is selected, either in the navigation tree or in the 3D scene. By clicking on Joints in the ribbon the labels can be displayed, too.

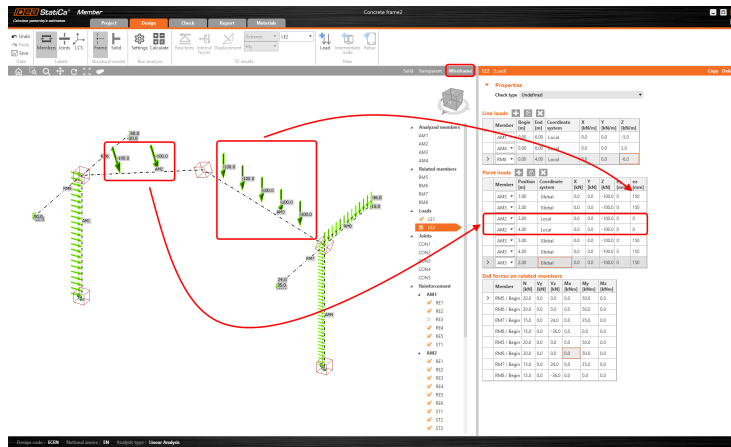


- Use the right-click button to run the context menu offering several speed functionalities, such as change of cross-section, adding of the intermediate joint, line load, point load, or running the reinforcement editor.

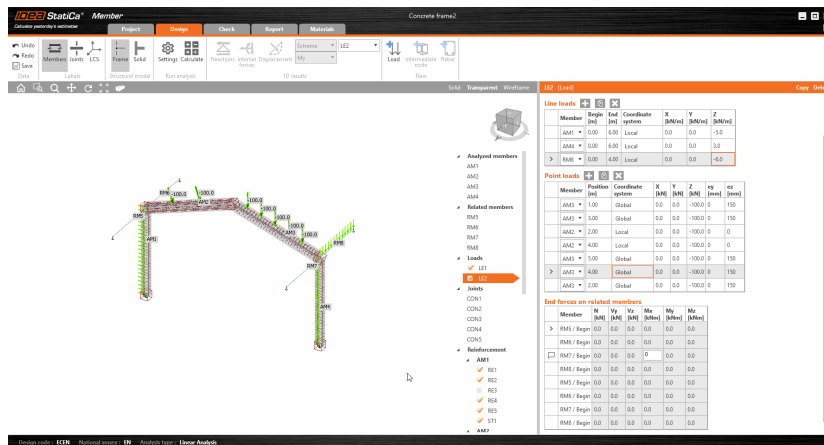


- The load effect definition is more user-friendly than before. Now you can input the point load on the analyzed or related member. The line loads and point loads have different scales to keep the model well-arranged. The eccentricity can be defined for the point load.

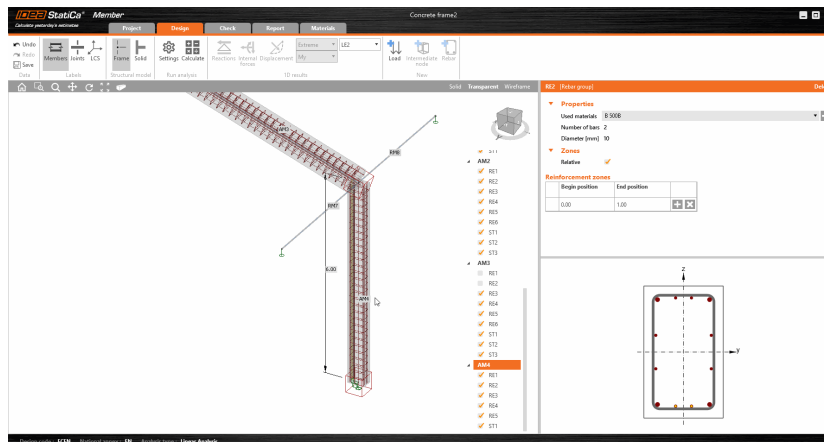




- Faster input via copy&paste values to the table of end forces on the related members



- Reinforcement editing is now available directly from the property window. In the previous version, you needed to launch the reinforcement editor and modify the reinforcement material, diameters, and numbers there. Now you can modify the reinforcement faster.
- You can notice that the checkboxes were added to Loads and Reinforcement. Thanks to them you can input the load effects or reinforcement and do not have to consider them for the analysis. In case of an unsatisfied check, you can simply switch on or off the particular reinforcement group and keep it in the model and thus avoid deleting and defining a new reinforcement group.



Dutch Annex for reinforced and prestressed concrete sections

Implementation of the Dutch Annex NEN 1992-1-1 NB 2020 for code-checks of reinforced and prestressed concrete sections has been done in IDEA StatiCa. Once an update of the annex is released we always implement it to keep your projects safely designed according to the valid code.

Available in all editions of [IDEA StatiCa Concrete and Prestressing](#).

Licensing & all applications

To facilitate the way of reserving or releasing seats in your shared license pool, we [adjusted the user interface of communication dialogs](#).

Other improvements:

New version indicator

The application will notify you when a new patch is released.

The application will inform the user whenever the new patch is available by the small "i" button in the top right corner. If the icon turns orange, your installed version is outdated.

You can open the Download page directly by pressing the "i" button, which opens the About window. At the bottom, the Update button opens a browser where you can get the latest patch. Again, the orange color of the Update button symbolizes a newly available patch.



The feature has been available since the 21.1.1 patch.

Available in both **Expert** and **Enhanced** editions of [IDEA StatiCa Steel](#).

Decimal separator and date format according to Windows Regional Settings

The regional setting of your Windows are now used for the decimal separator as well as for the date format.

In IDEA StatiCa the decimal separator is set based on your Windows settings. No separator of thousands is used. The date format is also set according to the Windows regional settings.

This update has been available since the 21.1.2 patch.

Available in both **Expert** and **Enhanced** editions of [IDEA StatiCa Steel](#).

Solved incidents

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