

IDEA StatiCa 21.0

Release notes

April 2021



Contents

INTRODUCTION	3
UPDATED CBFEM SOLVER	5
CONNECTION DESIGN IMPROVEMENTS	7
RELATIVE POSITIONS OF MEMBERS	7
FATIGUE ANALYSIS, NOMINAL STRESS	9
US-DRIVEN IMPROVEMENTS.....	10
CONNECTION DESIGN PROPOSAL (BETA)	14
ROTATIONAL CAPACITY LIMITED BY BOLT AND WELD FAILURE (STIFFNESS ANALYSIS)	15
TRAFFIC LIGHTS FOR FORCES IN BOLTS.....	16
RUSSIAN STANDARD (SP) ANCHORAGE.....	17
ANCHORAGE FORMULAS, TOOLTIPS.....	17
VIEWER	19
ENHANCED VIEWER PLUGINS FOR CAD SYSTEMS	19
MEMBER FOR STEEL	20
DESIGN ENHANCEMENTS IN RCS AND DETAIL	23
FIRE RESISTANCE OF SLENDER COLUMNS (RCS)	23
IMPROVEMENT OF STIFFNESS CALCULATION (RCS).....	24
IMPROVEMENT OF INTERACTION CHECK (RCS)	25
CRACK WIDTH CHECK (DETAIL).....	27
REBAR MODELING ENHANCEMENT – SUPERELEMENT (DETAIL)	28
MEMBER FOR CONCRETE	29
CODE-CHECK MANAGER (BIM LINKS)	32
THE LATEST TWO MAJOR RELEASES SUPPORTED	32
NEW AXISVM BIM LINK	32
USER PORTAL	33
ACCOUNT	33
LICENSES & USERS.....	33
CASES	34

Introduction

The new IDEA StatiCa version 21.0 aspires to further streamline the engineering design process. We have improved the analytical model for more accurate results. We are speeding up designs by adjusting the position of members relative to each other. Engineers can now slash the time of defining simple and moderate connections because IDEA StatiCa will propose them automatically – a new design proposal feature. Version 21 also brings major AISC update with explanations and tips.

The new 21.0 version of IDEA StatiCa brings another batch of enhancements in Concrete to the engineering community as well. Starting with IDEA StatiCa Member application to be thoroughly verified and taken from BETA, accompanied by a set of improvements to the RCS application, such as fire resistance or the Detail application, where the main attention has been paid to the CSFM solver. This all helps our application to provide faster calculations while improving stability and reliability.

Calculate yesterday's estimates!

News for steel

Updated CBFEM solver

- 30% faster calculation times
- More accurate modeling of hollow sections connection
- A new set of verifications and guidelines to interpret results between versions

Connection design improvements

- Top of steel modeling options; the relative position of members
- Fatigue analysis: nominal stress
- AISC code-compliance update, new seismic connection templates
- Automated selection of connection design templates
- Rotational capacity limited by bolt and weld failure
- Traffic-light results for forces in bolts
- Extended anchorage formulas; specifics for Russian standard
- Anchorage formulas, tooltips

Bulk selection with Viewer

- The Viewer can now export multiple connections from CAD/BIM applications in one go

Steel member design without limits

- New application Member is going LIVE – it designs general steel members including connections (the application Connection is embedded in it)
- Structural engineer no longer has to estimate the effects of the boundary conditions and can analyze and check members of any topology and loading
- Ability to analyze imperfections, large deformations (2nd order), non-linearities, torsion and warping

News for concrete and prestressing

Design enhancements in RCS and Detail

- Fire resistance of slender columns
- Extension of stiffness calculation
- Improvement of interaction check
- Updated crack width check
- CSFM analysis speed improvements

New concrete workflows with IDEA StatiCa Member

- The New application Member is going LIVE. It designs concrete members with complex cross-sections and has the application RCS embedded in it.
- All ULS and SLS checks for critical beams and frames of varying topologies, including capacity, shear, torsion, interaction, stress limitation, and crack width

BIM links

- Update of supported version of 3rd party software

User portal and Licensing

- New portal to manage licenses and submit support cases

Updated CBFEM solver

The numerical solver is the hearth of the CBFEM. It is a computational method that is being continuously improved and refined. With the 21.0 version, we have completed the development in the field of geometrical nonlinearities and large deformations including the initial imperfections (GMNIA) lasting several years. This resulted in the release of the Member application which is not in the BETA regime anymore. Member is there for all engineers in cases when their 3D FEA software is not capable of analyzing and checking the particular beam/column member. The main benefit is that the elements are completely modeled, including the real end connections. This frees the designer from the necessity of estimating the effect of the end connections on the load capacity and the stability loss. The Member can deal with the influence of transverse and longitudinal reinforcements, openings, changes in the cross-section height, but also with the influence of connected secondary members. The effect of torsion and warping is not a problem for the method.

At the same time, the new GMNIA method significantly served to refine the connection models in the Connection app. It is now used in all hollow cross-section connections - circular and rectangular pipes. According to standards, pipe joints are assessed according to empirical formulas, the validity of which is limited to certain geometric conditions. The compliance of these formulas with reality is quite questionable, especially at the edges of the validity intervals. The improved GMNIA method shows very good compliance with the standard formulas, especially in the middle parts of the validity intervals. In the marginal ones, it was thoroughly validated against higher mathematical models (ABAQUS) and physical experiments.

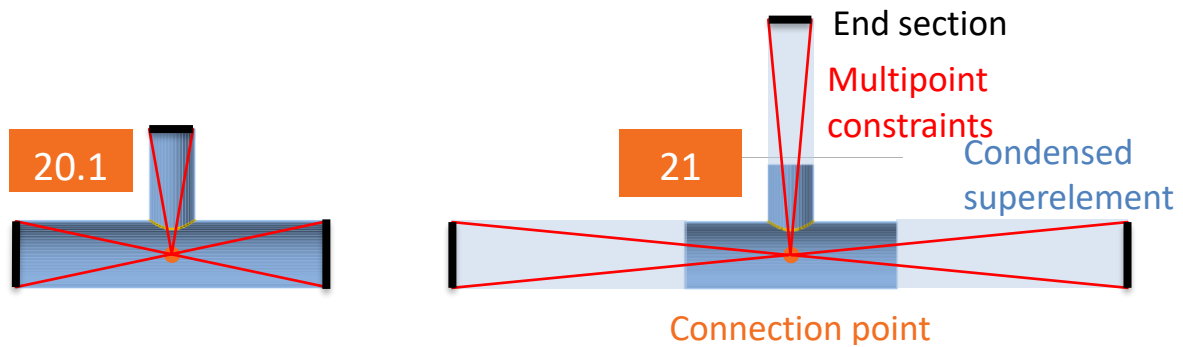
Every change in the numerical model necessarily brings a change in the results. This will also be reflected in version 21, and in the vast majority of cases, it will be in percentage units. Our experts will be happy to help you explain any major deviations.

However, a side effect of the model improvement is a significant acceleration of the calculation in the range of up to 30%.

GMNA solver

The solver used for the hollow section joints in IDEA StatiCa Connection and also for GMNIA in IDEA StatiCa Member was improved. Now, it contains a nonlinear formulation of not only shell elements (which was already present in previous versions) but also of links and constraints used in components, such as bolts or welds.

The model of the connection is greatly improved by the insertion of a condensed element. This element is added behind the member end and has the same properties as the elastic shell model of the member. It is just one element but it allows any elastic deformation and stress to develop in the member ends. Because of this, the part of the member consisting of shell elements may be shorter and still even improve the model behavior.

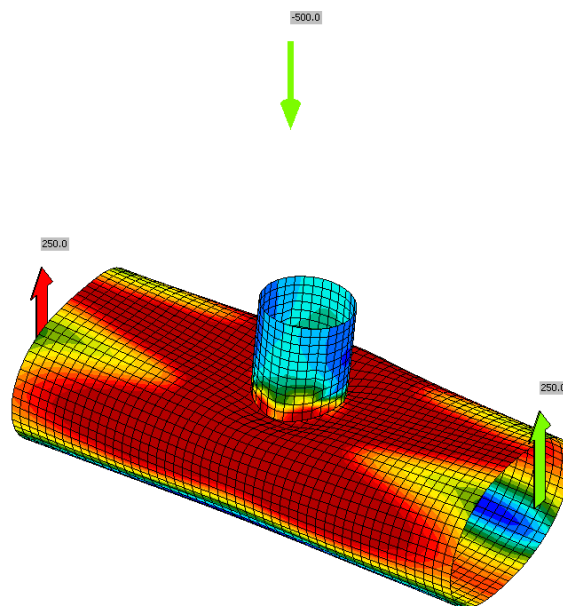


A condensed element has been added at member ends

This allows shortening the part of the model where shell elements are used and still increases the precision of the model. The big improvement is fewer elements in the model leading to faster calculation time and faster visualization of results.

- Cross-section deforms at the end of shell model

This is the main reason the change was made. The cross-section may deform at the ends of model consisting of shell elements. Joints of hollow sections require relatively long members – up to 10 times cross-section diameter. By introducing condensed element behind the part of the model consisting of shell elements, the calculation is much faster with the same precision.

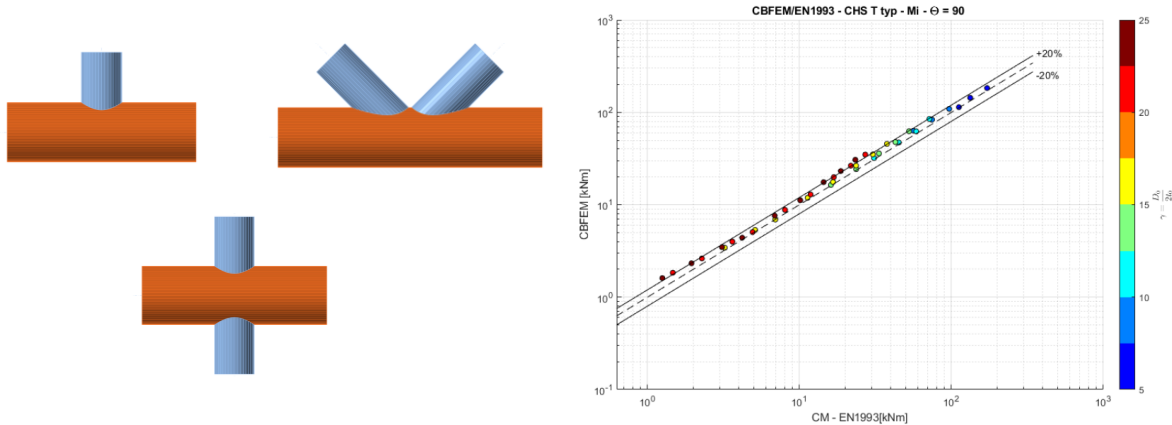


- Shell bending resistance reduced for hollow sections (imperfections)

Load resistances of hollow section joints in the codes are determined by Failure Mode Method that uses curve-fitting models determined from experiments and advanced numerical models. The real structure contains initial imperfections and residual stresses, which are not captured by shell models in IDEA StatiCa Connection. To achieve closer compliance with results of codes,

the influence of residual stress and initial imperfections was introduced in IDEA StatiCa models by reducing the bending resistance of shells of hollow sections with high $D/(2t)$ ratio.

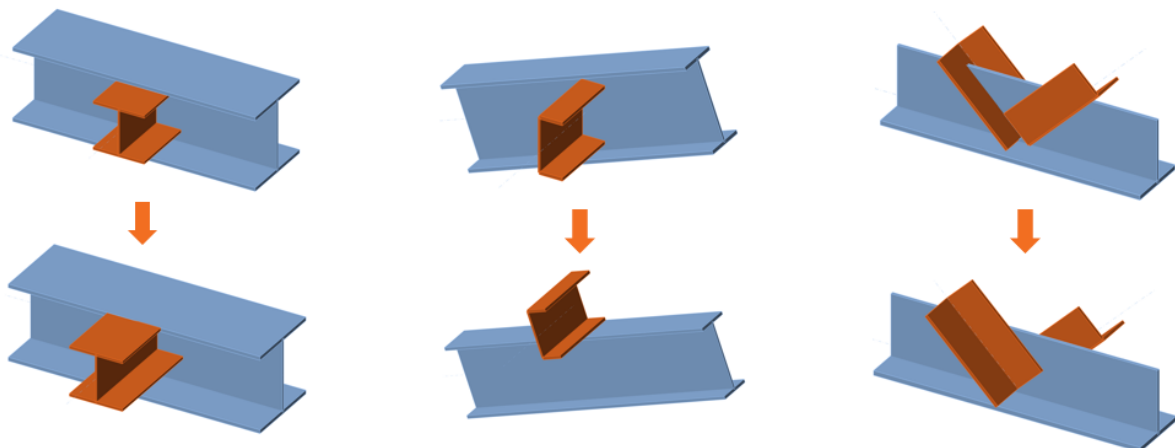
These combined changes allowed to achieve close compliance with the results of Failure Mode Method contained in design codes.



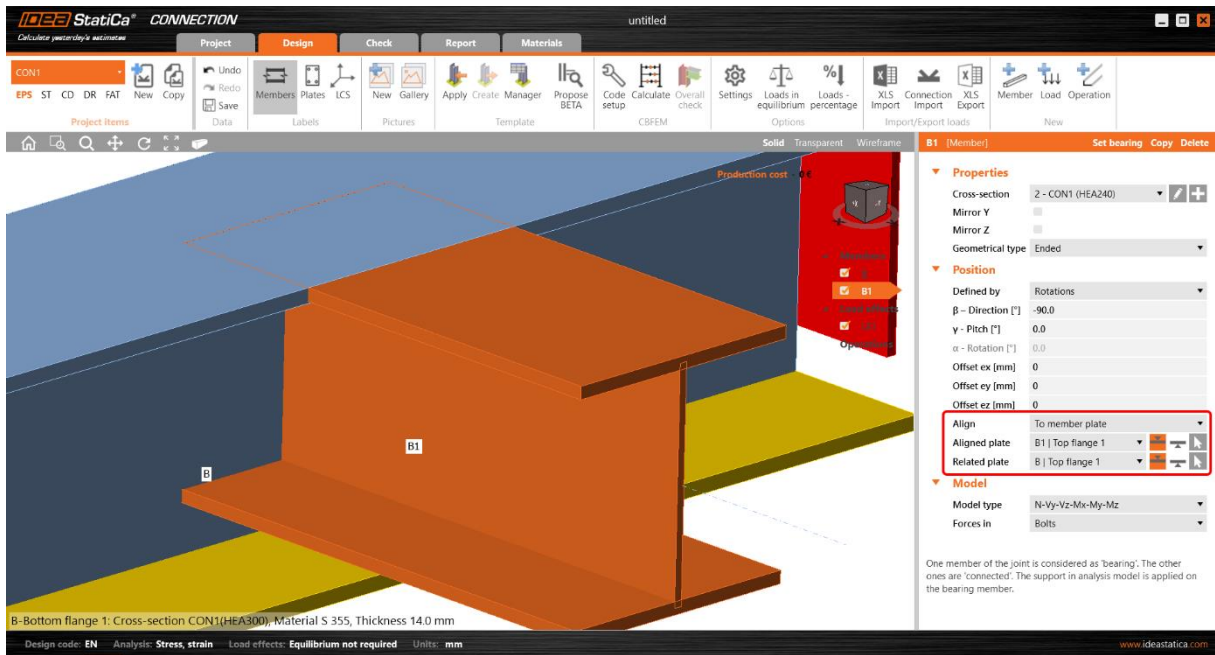
Connection design improvements

Relative positions of members

This new feature enables to easily place members relative to other member plates. Members can be aligned to plates of other members. The plates are moved and rotated to be parallel.



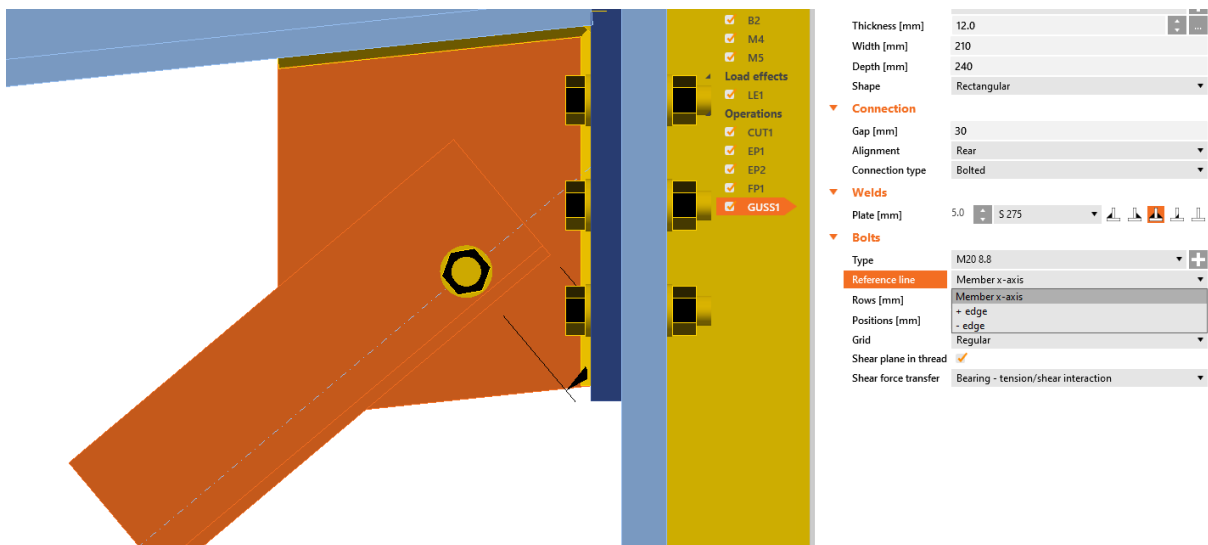
You can change the position of the member in its properties.



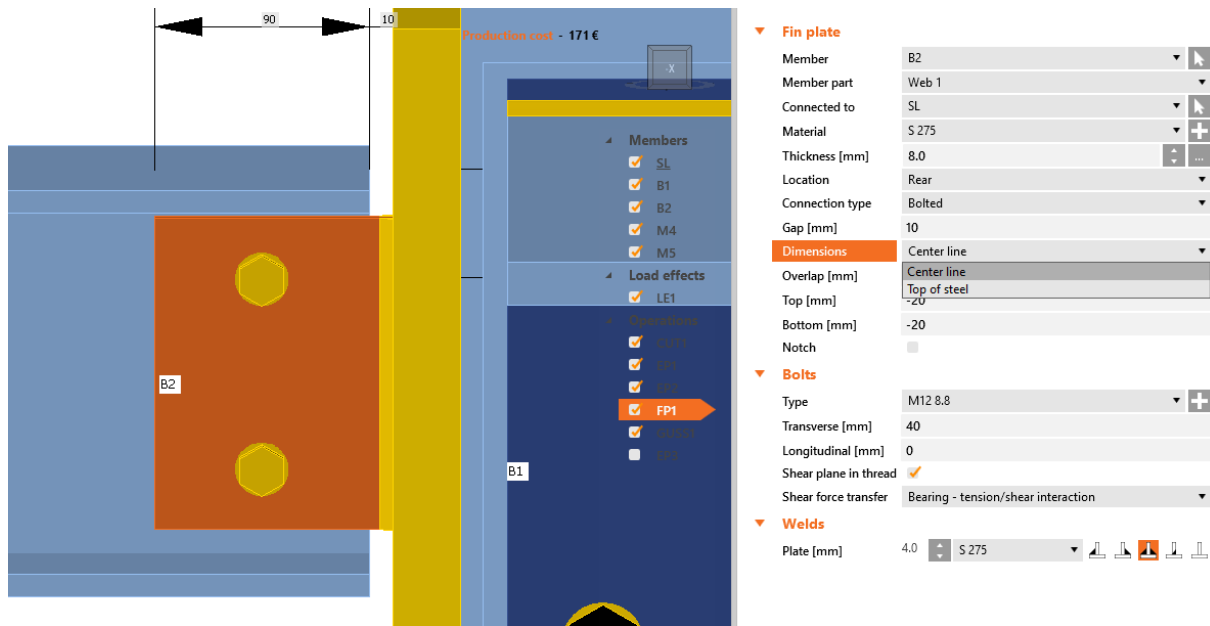
Moreover, the feature is available also for different manufacturing operation: Stiffener can be positioned relative to another plate. The stiffener is placed perpendicular to member axis by default.

Manufacturing operations are consolidated - the property grid is unified and a new setting of Dimension: "Top of steel" is made available for:

- Cleat
- Fin plate
- End plate
- Splice
- Stub
- Plate to plate



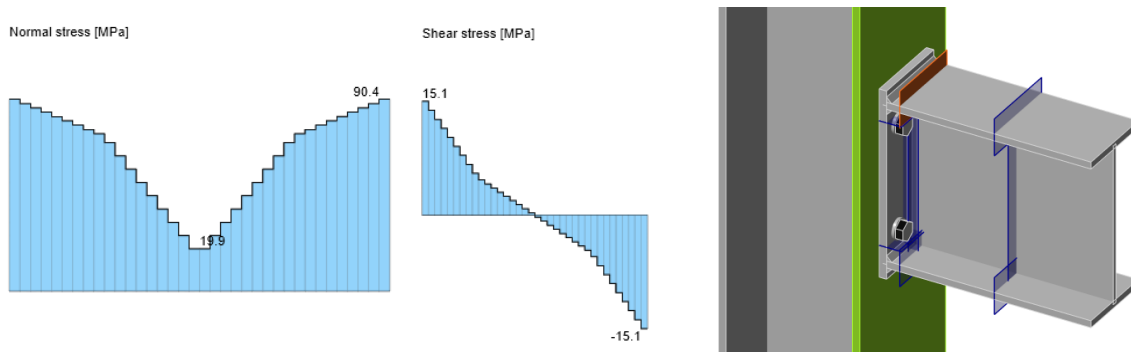
Entering the bolt position: A reference line can be changed



Entering the bolt position: A reference line can be changed

Fatigue analysis, nominal stress

Fatigue analysis type serves to determine normal and shear stress range between two load cases. The stresses correspond to nominal stresses and must be further evaluated using code design methods. It is assumed to be used for the design of high-cycle fatigue details, where no yielding is expected.



Fatigue analysis type does not provide any final resistance or number of cycles the detail can take. It just provides inputs to further calculations according to codes (nominal stresses in user and automatically defined sections).

The nominal stress can be calculated for:

- Bolts – in tension and shear
- Welds – averaged at plates near the weld
- Plates – averaged along selected lines

The nominal stress is determined by subtracting the stresses of reference load case from another load case.

To take a closer look at this type of analysis, please visit our Support Center:

[Theoretical Background: Fatigue analysis type](#)

[Fatigue analysis – Butt welds of I section](#)

US-driven improvements

We have focused on our US users and made their life easier with a set of localization improvements.

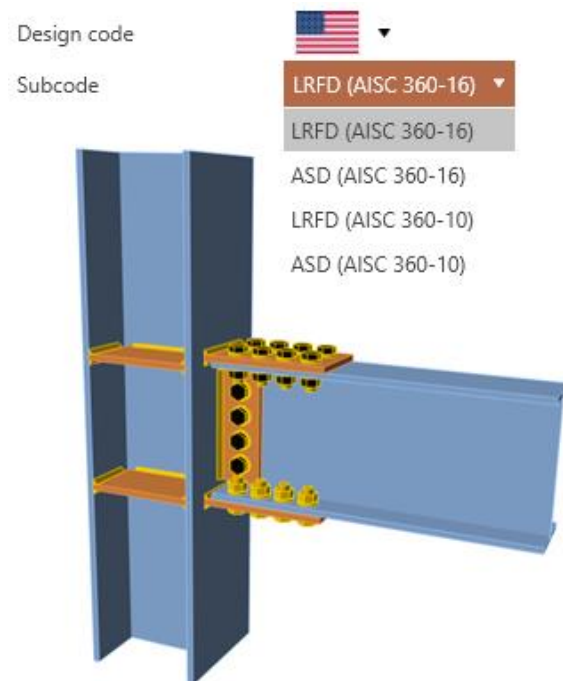
Design codes

Possibility to select desired version of AISC 360 (LRFD & ASD) – bolt assemblies affected.

Available standards:

- LRFD (AISC 360-16)
- ASD (AISC 360-16)
- LRFD (AISC 360-10)
- ASD (AISC 360-10)

The LRFD/ASD differences are already covered.



The differences between AISC 360-10 and 360-16 bolts are marked red:

Table 1: Bolt grade A325

Diameter [in]	AISC 360-10		AISC 360-16	
	Yield strength [ksi]	Tensile strength [ksi]	Yield strength [ksi]	Tensile strength [ksi]
$d \leq 1$	92	120	92	120
$d > 1$	81	105	92	120

Bolt grade A325M is the same in all codes – yield strength = 660 MPa, tensile strength = 830 MPa

Table 2: Bolt grade A325: Minimum bolt pretension (Table J3.1)

	AISC 360-10	AISC 360-16
Diameter [in]	Minimum bolt pretension [kip]	Minimum bolt pretension [kip]
1 1/8	56	64
1 1/4	71	81
1 3/8	85	97
1 1/2	103	118

Table 3: Nominal hole dimensions for standard bolts (Table J3.3)

Bolt diameter [in]	AISC 360-10	AISC 360-16
1/2	9/16	9/16
5/8	11/16	11/16
3/4	13/16	13/16
7/8	15/16	15/16
1	1 1/16	1 1/8
≥1 1/8	d + 1/16	d + 1/8

US-specific materials added

New grade for anchors:

- ASTM F1554

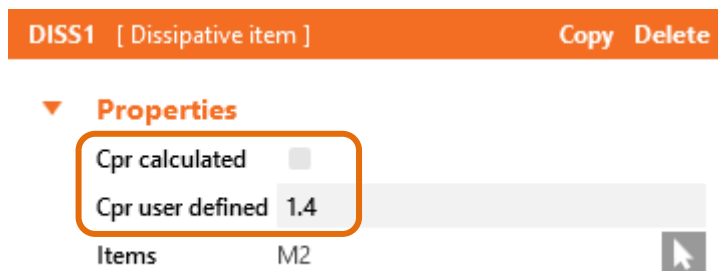
New MPRL items:

- ASTM A1043(M), Gr. 36
- ASTM A1043(M), Gr. 50
- ASTM A1065, Gr. 50
- ASTM A1085(M), Gr. A
- A500, Gr. B, Round
- A500, Gr. B, Shaped
- A500, Gr. C, Round
- A500, Gr. C, Shaped

Capacity design: User-defined C_{pr}

In the analysis of capacity design, the user may set his/her own value of C_{pr} (the factor to account for peak connection strength, including strain hardening, local restraint, additional reinforcement, and other connection conditions.)

To input a custom value of C_{pr} , uncheck the “Cpr calculated” option and define the user value.



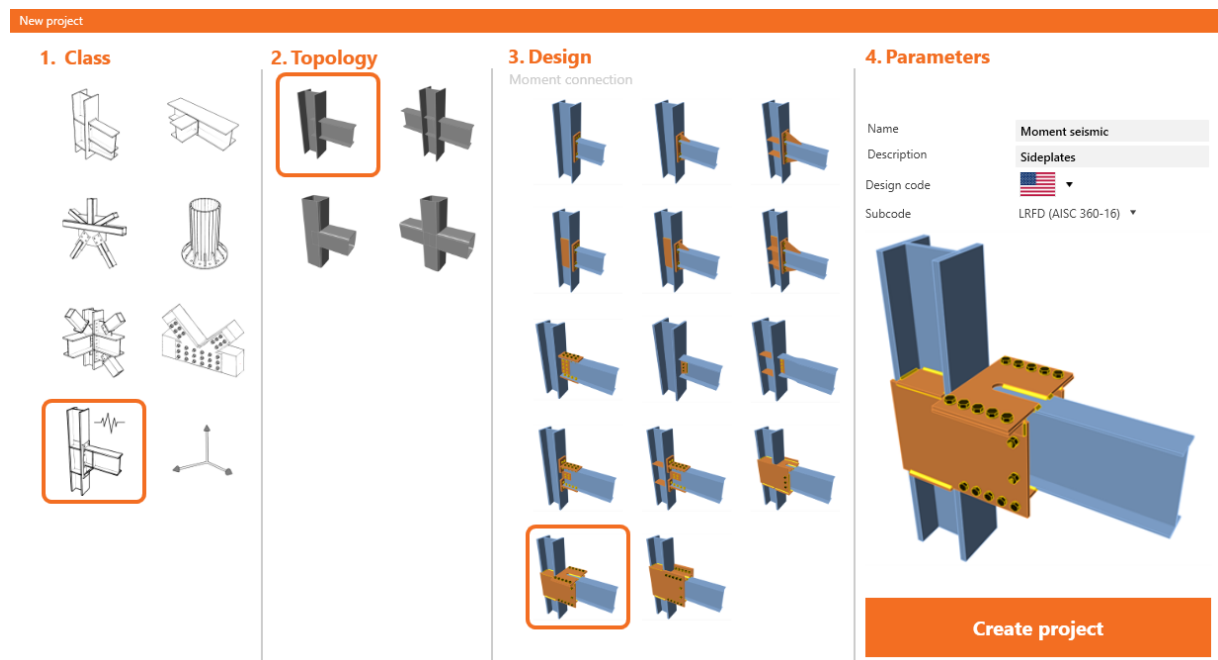
Top of steel input

The new version brings the possibility to define the bolt spacing and dimensions of several operations with a “top of steel” reference: Cleat, Fin plate, End plate, Splice, Stub and Plate to plate.

AISC & EN seismic templates

A brand-new class of connections is in the New project wizard. You can find here the predefined designs designated for capacity design check (seismic templates).

Based on the selected standards, different templates are displayed. There are twenty-nine templates according to ANSI/AISC 358-18 and eight templates according to EC8.



Imperial unit input

Users working in imperial units will appreciate the implementation of the established format of dimensions. From now on, the following number format are accepted:

Examples: 2'-0"3/4, 2-0"3/4, 2-3/4, 3/4 [in,ft] for Imperial format
 1, -1, 15, 0.75 [in,ft] for Decimal format

Units

Unit type	Unit	Precision	Format
▲ Main			
Length - Structure	in	1/16"	Imperial
Length - Cross-section	in	1/16"	Imperial
Plate or weld thickness, bolt hole diameter	in	1/16"	Imperial
Angle	°	1	Decimal
Force	kip	3	Scientific
Moment	kip.in	2	Imperial

For bolts, you are now able to define the transverse and longitudinal spacing, which is more understandable than the previous input method. The numerical format for bolts is

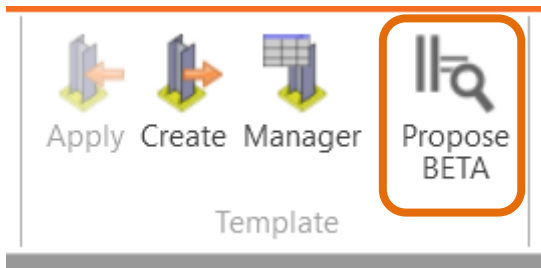
Examples: 2"2/3 3" or 2"2/3;3" [in,ft] for Imperial format

0.15 0.12 or 0.15;0.12 [in,ft] for Decimal format

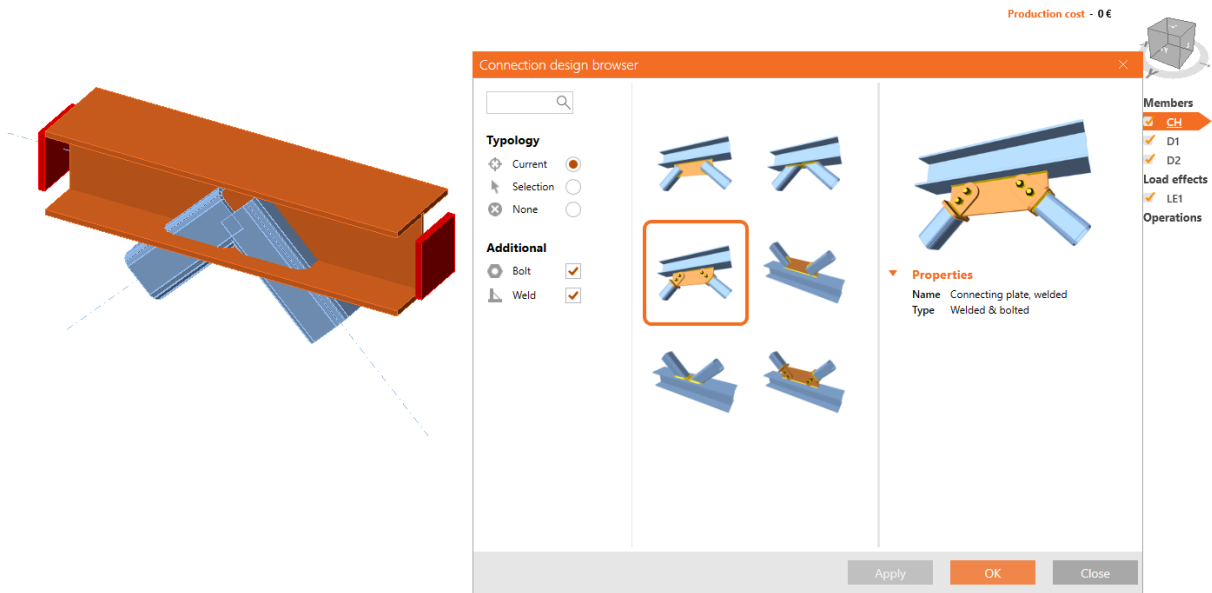
The first coordinate represents the first bolt to be placed, the position of the other can be defined either relative to the first one (space separator) or in an absolute manner (semicolon separator).

Connection design proposal (BETA)

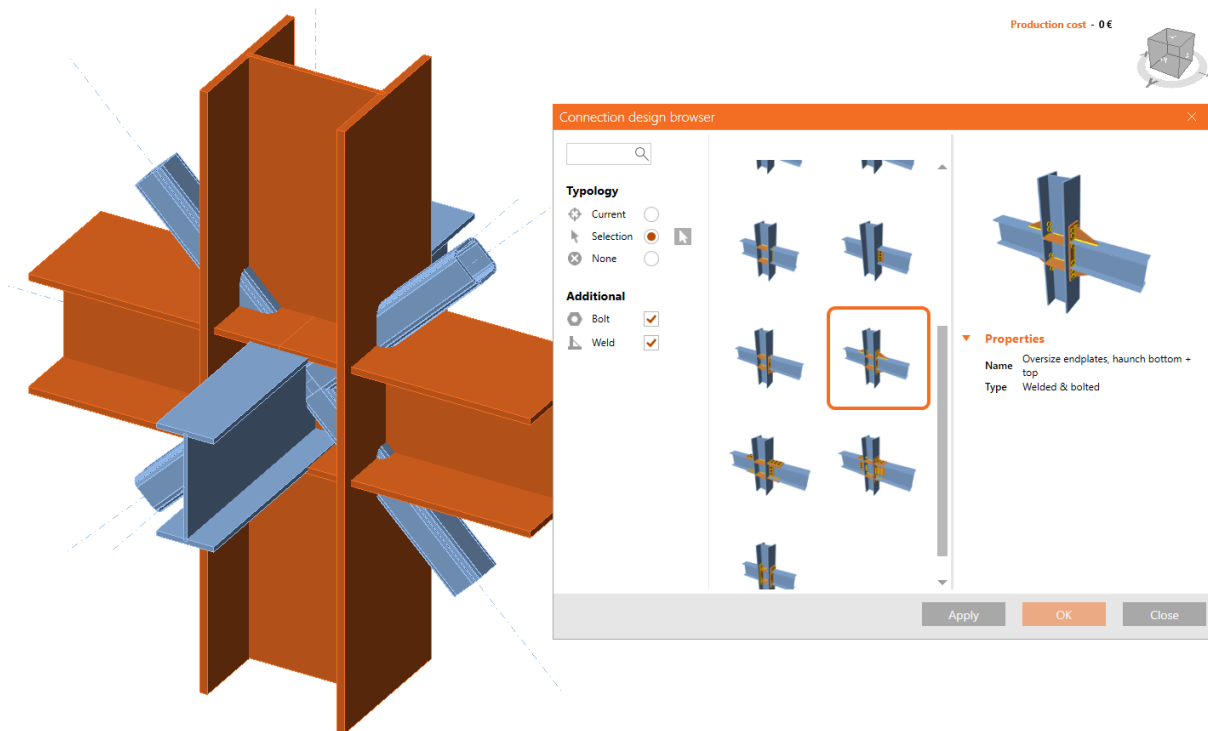
Beta version of the new Connection design proposal helps you find a suitable design solution from a library of predefined designs and directly apply it on your members.



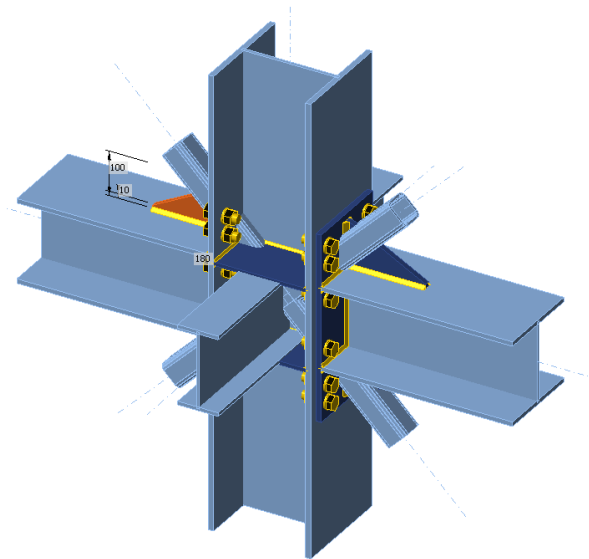
You can open the browser and directly go through available designs for all current members topology of your connection.



Nor the nonstandard topologies, try to select only three members at a time. Switch the Topology to Selection and use the pick tool and hold CTRL to select multiple members. Confirm the selection with Enter/spacebar or right mouse button. The browser will filter the topologies based on your selection.



Selected solution is applied to a 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 bolts and welds or use the search field on the top.

Prospectively, we will implement also the possibility to save custom designs and share them with others.

Rotational capacity limited by bolt and weld failure (Stiffness analysis)

In the older versions, the rotational capacity of the connection in the stiffness analysis was determined by the 15% plastic strain of plates only.

Newly, the limit is defined by these three factors:

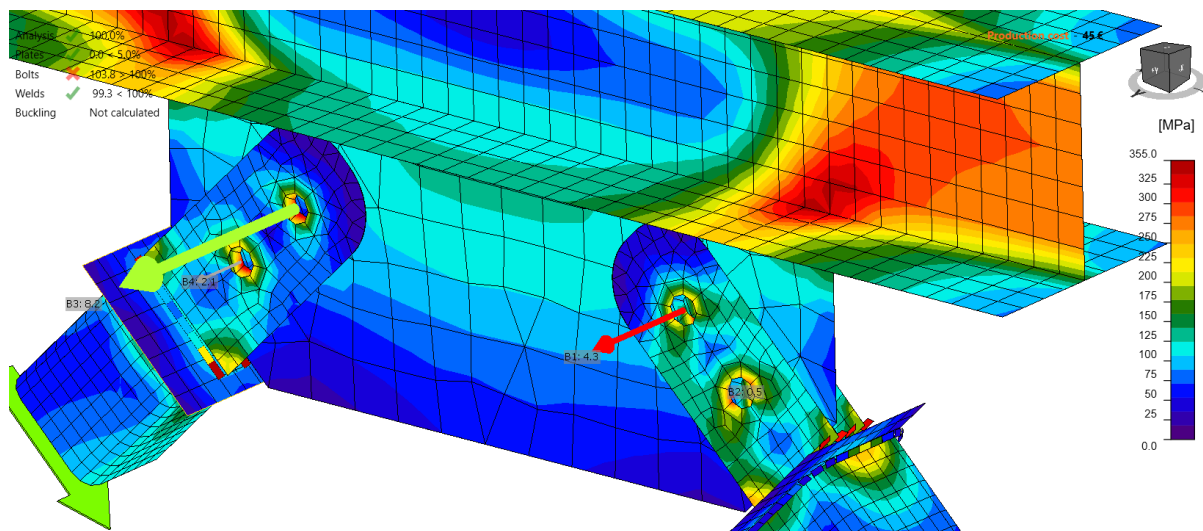
- Failure of bolts (bolt steel failures in tension and shear)
- Failure of welds
- Plates (15% plastic strain)

This affects the Rotational capacity value Φ_c .

Analysis		Rotational stiffness										
Rotational stiffness of joint component												
Item	Comp.	Loads	MEd [kNm]	Mj,Rd [kNm]	Sj,ini [MNm/rad]	Sjs [MNm/rad]	ϕ [mrad]	ϕ_c [mrad]	L [m]	Sj,R [MNm/rad]	Sj,P [MNm/rad]	Class
> B	My	LE1	150.0	23.6	1.6	0.0	1718.7	25.8	Rotational capacity		13.5	Pinned

Traffic lights for forces in bolts

The visual check of the bolts will be even easier with this useful feature. The tensile force in each bolt is now represented with an arrow in different colors. These colors comply with the well-known “traffic light” principle (grey, green, orange, and red). The size of the arrow corresponds to the tensile force magnitude.



You can change the limits for the level checks in Code setup.

▼ Check settings

Limit plastic strain [%]	5.0
Local deformation limit [%]	3.0
Warning plastic strain [%]	3.0
Warning check level [%]	95.0
Optimal check level [%]	60.0

Russian Standard (SP) anchorage

Implementation of all checks for anchoring in Russian code

[Theoretical Background: Check of anchors according to STO](#)

Russian code STO 36554501-048-2016, SP 16, and SP 43

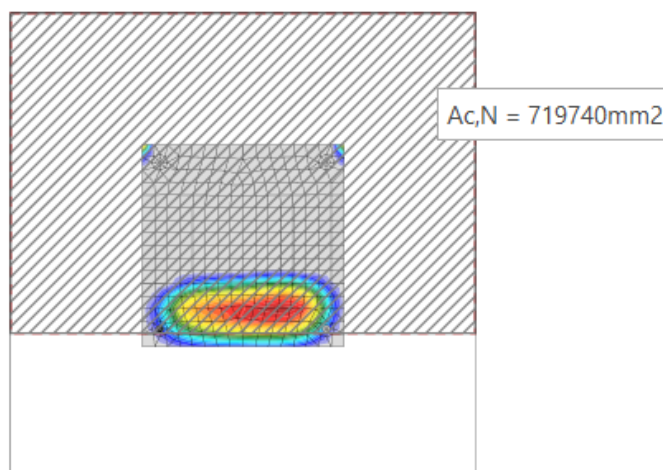
- Tensile steel resistance (SP 43 - Annex G)
- Pull-out resistance (EN 1992-4, Cl. 7.2.1.5)
- Concrete cone failure resistance of anchor or group of anchors (STO - Cl. 6.1.3)
- Anchor shear steel resistance (SP16 - Cl. 14.2.9 and STO - Cl. 6.2.1)
- Concrete pry-out failure (STO - Cl. 6.2.2)
- Concrete edge failure (STO - Cl. 6.2.3)
- Interaction of tensile and shear forces (STO - Cl. 6.3)

Anchorage formulas, tooltips

Slight improvements could be found in footings.

Tooltip for concrete cone breakout area

A tooltip will help you understand the hatched area representing the $A_{c,N}$ (concrete breakout cone area for a group of anchors) in Stress in the concrete preview of results for anchors.



Descriptions for anchor failure modes

The components of the interaction equations are now numerically represented so that it is clear which one is decisive.

Interaction of tensile and shear forces in steel (EN 1992-4 - Table 7.3)

$$\left(\frac{N_{Ed}}{N_{Rd,s}}\right)^2 + \left(\frac{V_{Ed}}{V_{Rd,s}}\right)^2 = 0.21 \leq 1.0$$

Where:

$N_{Ed} = 32.6$ kN – design tension force

$N_{Rd,s} = 71.2$ kN – fastener tensile strength

$V_{Ed} = 0.0$ kN – design shear force

$V_{Rd,s} = 50.2$ kN – fastener shear strength

Interaction of tensile and shear forces in concrete (EN 1992-4 - Table 7.3)

$$\left(\frac{N_{Ed}}{N_{Rd,i}}\right)^{1.5} + \left(\frac{V_{Ed}}{V_{Rd,i}}\right)^{1.5} = 0.63 \leq 1.0$$

Where:

$\frac{N_{Ed}}{N_{Rd,i}}$ – the largest utilization value for tension failure modes

$\frac{V_{Ed}}{V_{Rd,i}}$ – the largest utilization value for shear failure modes

$\frac{N_{Ed}}{N_{Rd,c}} = 64\%$ – concrete breakout failure of anchor in tension

$\frac{N_{Ed}}{N_{Rd,p}} = 29\%$ – concrete pullout failure

$\frac{N_{Ed}}{N_{Rd,cb}} = 0\%$ – concrete blowout failure

$\frac{V_{Ed}}{V_{Rd,c}} = 22\%$ – concrete edge failure

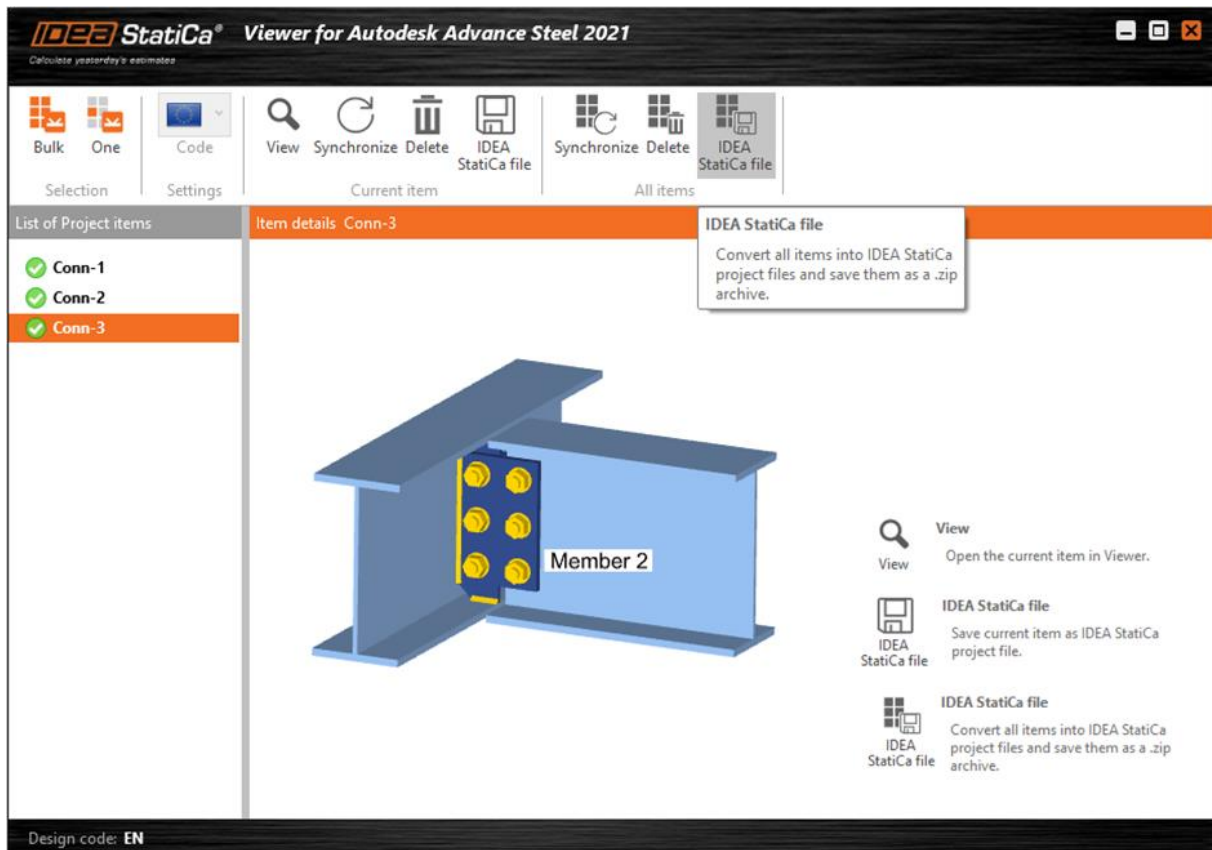
$\frac{V_{Ed}}{V_{Rd,cb}} = 4\%$ – concrete pryout failure

Viewer

Enhanced Viewer plugins for CAD systems

The free Viewer plugins for CAD applications (Tekla Structures, Autodesk Advance Steel and Autodesk Revit) have been enhanced by new functionality. The Viewer button integrated into the CAD will now open a wizard enabling following actions:

- Possibility to select desired standard
- Bulk and One selection mode (known from Code-check manager)
- List of exported nodes from the CAD model
- View button will open the online Project viewer with the current connection
- Synchronize reads the changes in the CAD model for the current connection
- Delete the exported connection, if necessary
- You can directly create an IDEA StatiCa Connection file from here
- Synchronize, Delete and IDEA StatiCa file may be called as a bulk action for all the connections in the list



Member for steel

The CBFEM analysis model of the IDEA StatiCa Member has been improved and verified. The application will no longer be in the BETA version.

Member is now available to structural engineers wherever the build-in checks in their 3D FEA program are not enough to fulfill their needs. The main benefit is that the members are completely modeled, including the end joints, which frees the engineer from having to estimate the effect on the resistance and loss of stability. The engineer always first designs the joints using IDEA Connection. Then he just selects the member between them and has a ready-made model for the check of the member, except for some additional information. Just run the calculation.

The Member can deal with the influence of transverse and longitudinal stiffeners, openings, changes in the height of the cross-section, but also with the impact of connected secondary beams. The effect of torsion and warping is not a problem for the method

The solver in Member for steel structures is completed. Nonlinear behavior (GMNIA) is ensured for all constraints and links in the components – bolts, welds, anchors, concrete subsoil. In the BETA version, nonlinear behavior was implemented only for shell elements of plates.

New solver verifications

IDEA StatiCa Member can model a never-ending variety of models. The verification studies were focused on buckling of members and local buckling of plates.

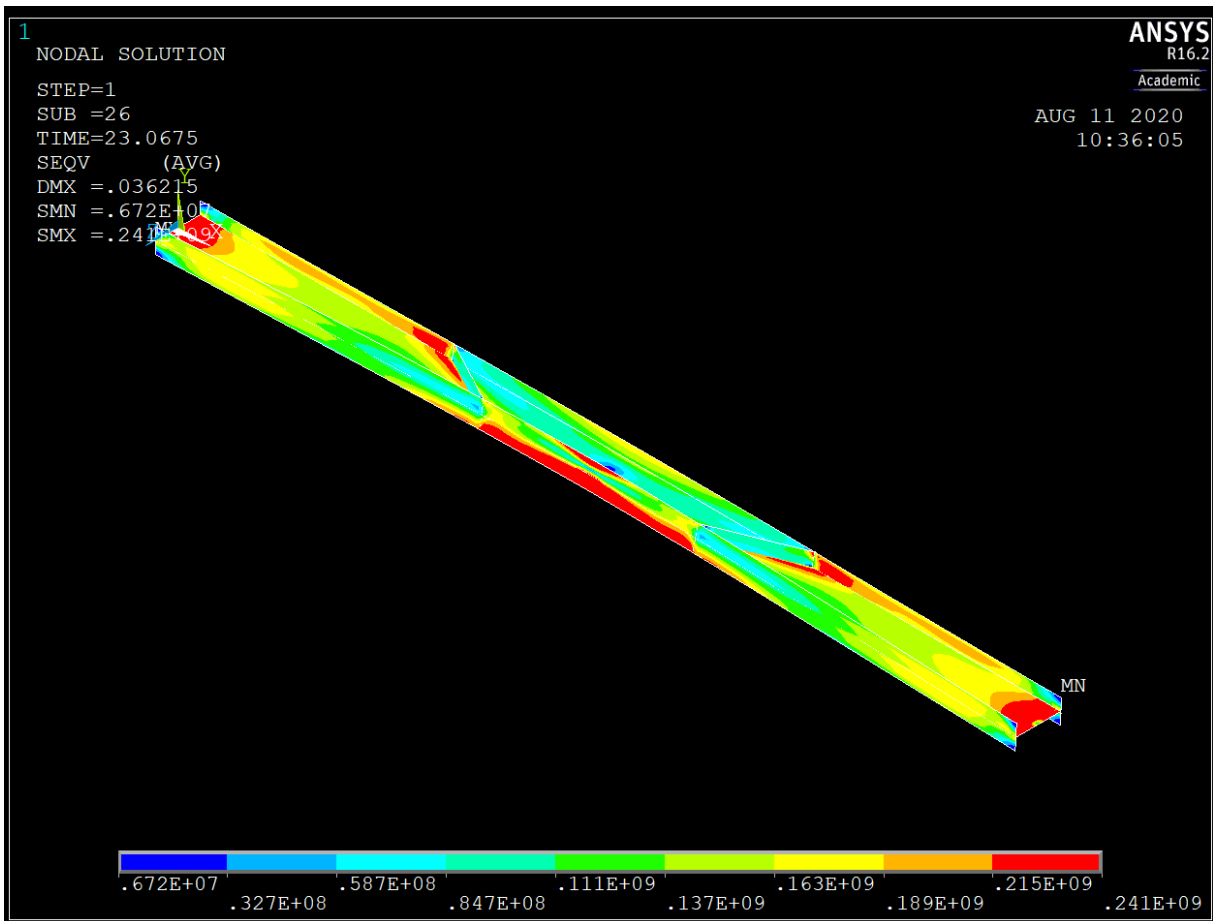
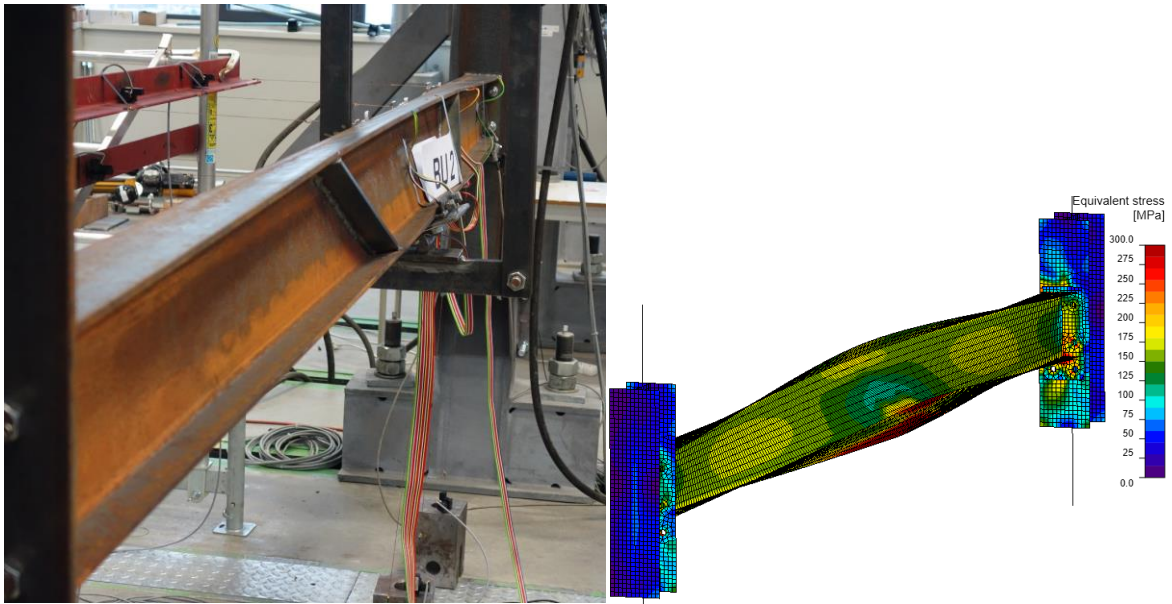
Verification of flexural buckling of columns and lateral-torsional buckling of beams is shown in this link. It contains a comparison to EN design code.

[Member for steel verification examples](#)

The advantage of modeling a member together with connections is shown in this example, where IDEA Member models of compressed symmetrical angles bolted to gusset plates are compared to experiments and advanced numerical models available in the literature.

[Verification article: Symmetrical angles in compression](#)

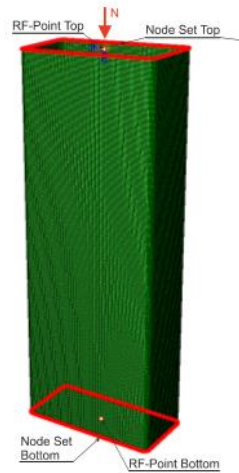
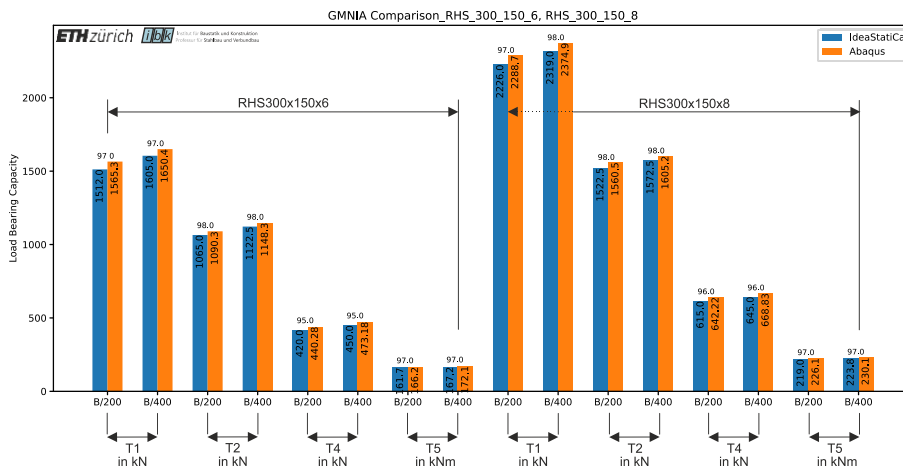
Lateral-torsional buckling of beams with semi-rigid end plate joints was investigated in a series of experiments at the Brno University of Technology. A research paper is submitted for the Eurosteel conference.



The results of stiffened beams were compared also to ANSYS models.

Boundary conditions	Stiffener position	GMNIA				LBA	
		F_{Rd} [kN]		w_y [mm]		F_{cr} [kN]	
		ANSYS	Member	ANSYS	Member	ANSYS	Member
Fixed	1550,2050	56.79	56.97	22.07	23.30		104.4
	1250,2350	61.56	63.27	21.36	21.40		121.5
	550,3050	60.21	60.84	18.46	19.70		122.4
Pinned	1550,2050	38.39	40.77	33.69	36.10	57.04	55.8
	1250,2350	41.52	43.92	32.60	30.40	64.75	63.9
	550,3050	44.01	47.07	22.40	23.60	82.6	81

Local buckling was investigated in cooperation with ETH in Zurich with Prof. Andreas Taras and his team. Rectangular hollow sections are investigated and compared to ABAQUS models used for formulation of 2nd generation Eurocodes. The results show the finite elements in IDEA StatiCa Member provide very similar results to these in ABAQUS. The results of LBA and GMNIA are nearly identical with the biggest difference of 5%. Also, it was shown that class 4 sections may be safely used with correct settings of initial imperfections.



Design enhancements in RCS and Detail

Fire resistance of slender columns (RCS)

A tabulated method providing structural fire design according to EN 1992-1-2 was implemented in version 10.1 of IDEA StatiCa RCS. When assessing the columns according to tabulated data, two possible methods (A, B) are recommended by the code. Both of them were implemented in version 10.1, but method B has been extended to include Annex C. Assessment of fire resistance of slender columns is now available in version 21.0! This solution is suitable for slender columns with slenderness under fire conditions within the range $30 < \lambda_{fi} \leq 80$.

Older versions

Fire resistance

Resistance: R90

$N_{ed,fi}$ [kN]	$M_{ed,y,fi}$ [kNm]	$M_{ed,z,fi}$ [kNm]	Value [%]	Limit [%]	Check
-2400.000	273.3	0.0			Not done

Check provisions for used method B

Conditions

	Value _{calc}	Value _{lim}	Check
$l_{0,fi}$ [m]	4.50	∞	OK
e_y [mm]	83.52	100.00	OK
e_z [mm]	0.00	100.00	OK
λ_{fi} [-]	36.02	30.00	Not OK

Input values and intermediate results for fire resistance

$N_{0,ed,fi}$ [kN]	A_c [mm ²]	f_{cd} [MPa]	A_s [MPa]	f_{yd} [MPa]	ω [-]	n [-]
-2400.000	196100.42	20.000	3926.99	434.783	0.44	0.61

Version 21.0

Fire resistance

Resistance: R90

$N_{ed,fi}$ [kN]	$M_{ed,y,fi}$ [kNm]	$M_{ed,z,fi}$ [kNm]	Value [%]	Limit [%]	Check
-2400.0	273.3	0.0	98.0	100.0	OK

Check provisions for used method B

	Value _{calc}	Value _{lim}	Utilization [%]	Check
$b(h)$ [mm]	500	490	98.0	OK
a_s [mm]	53	32	61.0	OK

Conditions

	Value _{calc}	Value _{lim}	Check
$l_{0,fi}$ [m]	4.50	∞	OK
e_y [mm]	84	100	OK
e_z [mm]	0	100	OK
λ_{fi} [-]	36.00	80.00	OK

Input values and intermediate results for fire resistance

$N_{0,ed,fi}$ [kN]	A_c [mm ²]	f_{cd} [MPa]	A_s [MPa]	f_{yd} [MPa]	ω [-]	n [-]
-2400.0	196100	20.0	3927	434.8	0.44	0.61

Improvement of stiffness calculation (RCS)

A correct determination of reinforced concrete section stiffness is the critical characteristic for serviceability limit states assessments. It is not a trivial task. There are many theories or approaches on how to calculate the resulting RC section stiffness. We slightly changed the calculation principles and, most importantly, cleaned up or rearrange the result presentation. As you know, in the previous versions, some values for stiffness calculation were displayed repeatedly, while others were missing. It was hard to understand it, kind of black box. Now, the users can find straightforward and comprehensive tables listing all necessary stiffness values and inter-result in the Stiffness tab of the RCS app. You can easily find out the stiffness corresponding to a characteristic or quasi-permanent combination, the bending moment at the first occurrence of cracking, etc.

Older versions

Stiffness

Short-term stiffness

Type	N [kN]	M _y [kNm]	M _z [kNm]	EI _y [MNm ²]	EI _z [MNm ²]	EA _x [MN]
Resulting stiffness	0.000	6.5	0.0	4	272	3282
Uncracked cross-section	0.000	6.4	0.0	5	321	3860
Cracked cross-section	0.000	6.5	0.0	1	58	708

Stiffness ratio

Type	EI _y [MNm ²]	EI _y /EI _{y1} [-]	EI _z [MNm ²]	EI _z /EI _{z1} [-]	EA _x [MN]	EA _x /EA _{x1} [-]
Uncracked cross-section	5	1.02	321	1.02	3860	1.02
Cracked cross-section	1	0.12	58	0.18	708	0.19
Linear	5	1.00	315	1.00	3777	1.00

Curvatures

Type	N [kN]	M _y [kNm]	M _z [kNm]	EI _y [MNm ²]	EI _z [MNm ²]	EA _x [MN]	r _y [1e-4]	r _z [1e-4]	ε _x [1e-4]
Resulting stiffness	0.000	6.5	0.0	4	272	3282	18.2	0.0	0.0
Uncracked cross-section	0.000	6.4	0.0	5	321	3860	13.8	0.0	0.0
Cracked cross-section	0.000	6.5	0.0	1	58	708	118.4	0.0	0.0

Cross-section characteristics for short-term stiffness

Type	A [mm ²]	S _y [mm ³]	S _z [mm ³]	I _y [mm ⁴]	I _z [mm ⁴]	t _y [mm]	t _z [mm]	x [mm]
Uncracked cross-section	122626.57	0.00	0.00	147148633.83	10189638543.24	0.00	-0.75	60.75
Cracked cross-section	22493.55	0.00	0.00	17440737.24	1845219934.45	0.00	40.13	19.87

Version 21.0

Stiffness

Short-term

Combination type	N [kN]	M _y [kNm]	M _z [kNm]	β _{st} [-]	σ _s [MPa]	ζ _{st} [-]
Quasi-permanent	0.0	6.5	0.0	1.00	177.9	0.04
Characteristic	0.0	10.0	0.0	1.00	273.7	0.59
Critical load	0.0	6.4	0.0	1.00	174.4	0.00

Short-term cross-section characteristics

Type	α _e [-]	x [mm]	t _y [mm]	t _z [mm]	A [mm ²]	I _y [mm ⁴]	I _z [mm ⁴]
Linear	6.35	60	0	0	120000	144000000	10000000000
Uncracked cross-section	6.35	61	0	-1	122627	147148634	10189638543
Cracked cross-section	6.35	20	0	40	22494	17440737	1845219934

Resulting short-term stiffnesses and curvatures

Type	EI _y [MNm ²]	EI _z [MNm ²]	EA _x [MN]	1/r _y [1e-4]	1/r _z [1e-4]	ε _x [1e-4]
Uncracked cross-section	5	321	3860	13.754	0.000	0.000
Cracked cross-section	1	58	708	118.405	0.000	0.000
Quasi-permanent	4	272	3282	18.164	0.000	0.000
Characteristic	1	87	1059	117.005	0.000	0.000

Improvement of interaction check (RCS)

Interaction check verifies a cross-section capacity to carry additional forces due to shear and torsion in longitudinal reinforcement. Resistance of shear reinforcement is checked for the combination of shear and torsion. The cross-section can be subjected to the normal force, bending moments, shear forces simultaneously. In complex structures such as bridges, even another component, the torsion moment, can act in the cross-section. That is why we need to take all these components into account and check the cross-section capacity. In the previous versions of IDEA StatiCa RCS, the increment of stress/strain calculation was based on the rebar's linear-elastic behavior assumption (we didn't respect the plastic branch of the stress-strain diagram of rebar). With version 21.0, real axial stiffness and tensile strength are respected. Therefore, each rebar's actual stress-state is captured, and increments due to shear and torsion are calculated based on their nonlinear stress-strain diagrams. The deformation compatibility of the group of rebars is respected. Practical application: Combination of rebars with different material properties (such as Young's modulus, tensile strength) – shear reinforcement can be designed as stirrups combined with prestressing bars (initial stress by prestressing is not defined in this version). Interaction check also applies to longitudinal reinforcement. The additional tensile forces due to shear and torsion are added to the stress state caused by normal force and bending moments. In some cases, a limited interaction check according to EN 6.2.3 (7) can be applied. When this limited check was considered, the additional tensile forces were neglected from both shear and torsion in older versions. This approach led to the fact that a limited interaction check cannot be used if the cross-section is subjected to a torsional moment in addition. And thus unsatisfied check for the interaction. Now, the interaction of normal force, bending and torsional moments can be checked. No more unsatisfied interaction checks!

Older versions

Interaction

Results presented for combination : Fundamental ULS

N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	V_{Ed} [kN]	T_{Ed} [kNm]	Value V+T [%]	Value V+T+M [%]	Value [%]	Limit [%]	Check
483.0	-1128.0	-1082.0	2569.2	1508.0	84.2	48.8	84.2	100.0	OK

Interaction check of shear and torsion (concrete)

$V_{Rd,c}$ [kN]	$T_{Rd,c}$ [kNm]	$V_{Rd,max}$ [kN]	$T_{Rd,max}$ [kNm]	Eq. 6.31 [%]	Eq. 6.29 [%]	Value [%]	Limit [%]	Check
1520.2	4288.4	29160.0	16751.5	204.2	17.8	17.8	100.0	OK

Interaction check of shear and torsion (longitudinal reinforcement)

A_{sI} [mm ²]	F_{sI} [kN]	$F_{sI,lim}$ [kN]	Value [%]	Limit [%]	Check
12277	4495.4	5338.0	84.2	100.0	OK

Interaction check of shear and torsion (shear reinforcement)

a_{sw} [mm ² /m]	F_{sw} [kN]	$F_{sw,lim}$ [kN]	Value [%]	Limit [%]	Check
3142	830.7	1365.9	60.8	100.0	OK

Interaction check of shear, torsion, bending and normal force

F_b [kN]	$\Delta F_{td,s}$ [kN]	$\Delta F_{td,t}$ [kN]	$\Delta \epsilon_s$ [1e-4]	$\Delta \epsilon_t$ [1e-4]	Extreme in bar	Value [%]	Limit [%]	Check
984.9	2569.2	1926.2	0.0	0.0	26	48.8	100.0	OK

Version 21.0

Interaction

Results presented for combination : Fundamental ULS

N_{Ed} [kN]	M_{Edy} [kNm]	M_{Edz} [kNm]	V_{Ed} [kN]	T_{Ed} [kNm]	Value V+T [%]	Value V+T+M [%]	Value [%]	Limit [%]	Check
483.0	-1128.0	-1082.0	2569.2	1508.0	84.2	84.8	84.8	100.0	OK

Interaction check of shear and torsion (concrete)

$V_{Rd,c}$ [kN]	$T_{Rd,c}$ [kNm]	$V_{Rd,max}$ [kN]	$T_{Rd,max}$ [kNm]	Eq. 6.31 [%]	Eq. 6.29 [%]	Value [%]	Limit [%]	Check
1520.2	4288.4	29160.0	16751.5	204.2	17.8	17.8	100.0	OK

Interaction check of shear and torsion (longitudinal reinforcement)

A_{sl} [mm ²]	F_{sl} [kN]	$F_{sl,lim}$ [kN]	Value [%]	Limit [%]	Check
12277	4495.4	5338.0	84.2	100.0	OK

Interaction check of shear and torsion (shear reinforcement)

a_{sw} [mm ² /m]	F_{sw} [kN]	$F_{sw,lim}$ [kN]	Value [%]	Limit [%]	Check
3142	830.7	1365.9	60.8	100.0	OK

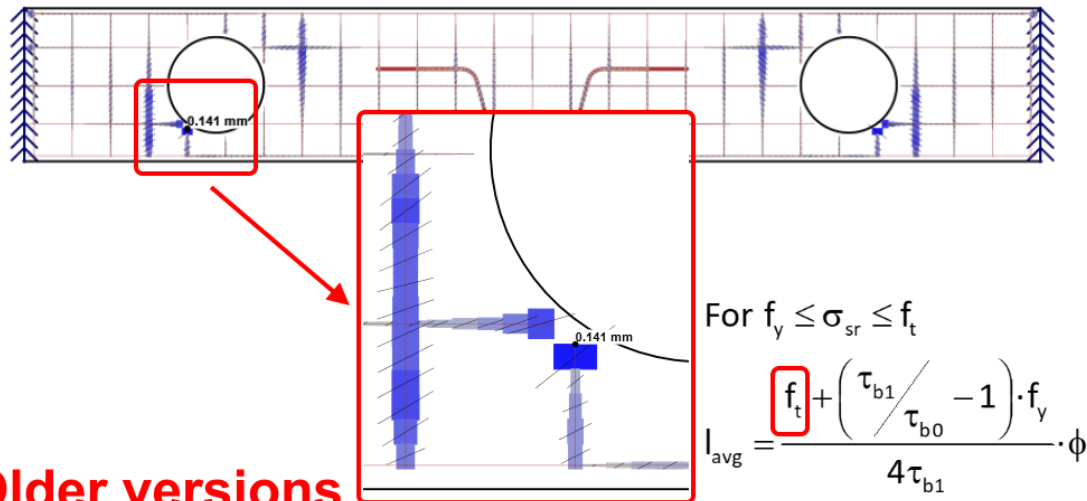
Interaction check of shear, torsion, bending and normal force

F_b [kN]	$\Delta F_{td,s}$ [kN]	$\Delta F_{td,t}$ [kN]	$\Delta \epsilon_s$ [1e-4]	$\Delta \epsilon_t$ [1e-4]	Extreme in bar	Value [%]	Limit [%]	Check
984.9	2569.2	1926.2	-	7.8	26	84.8	100.0	OK

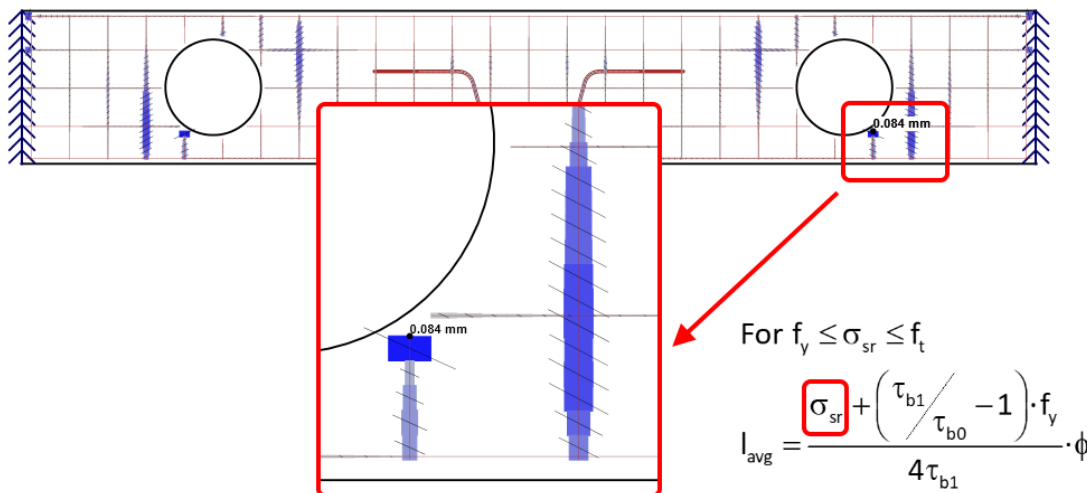
Crack width check (Detail)

An advanced CSFM theory considers two types of cracking, stabilized and non-stabilized. Non-stabilized cracks can occur in regions with small reinforcement ratios. Typically, stirrups or frame corners are the regions where the non-stabilized cracking is considered. A single crack occurs, and the tension stiffening effect is captured by the so-called Pull-Out-Model (POM). According to the latest research done by ETH Zurich, we improved the theory of the Pull-Out-Model, precisely the average length where the crack can occur or is expected. In the older version of IDEA StatiCa Detail, the average length depended on the rebar's actual stress state σ_{sr} . Now, the average length is considered constant with respect to ETH research. That is why the resultant crack width might differ in previous versions.

Version 21.0

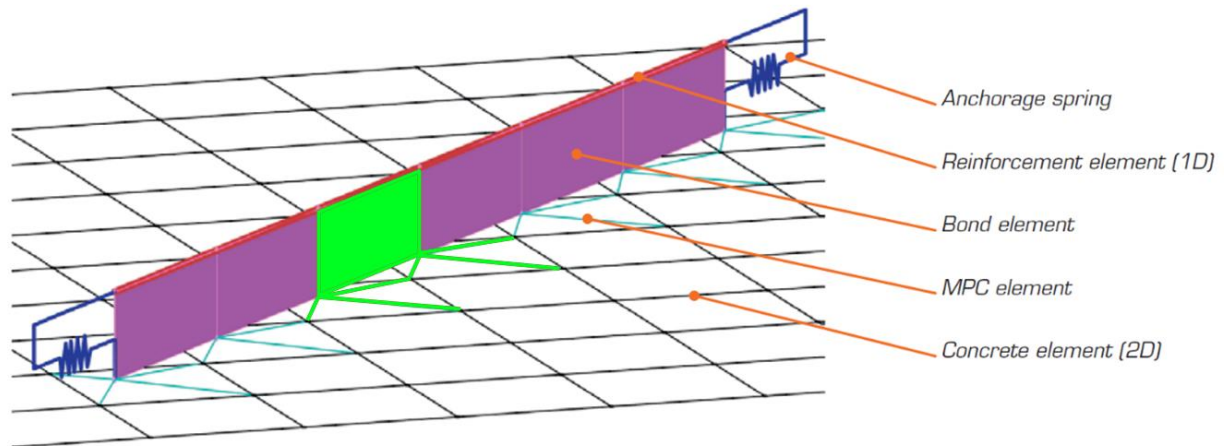


Older versions



Rebar modeling enhancement – superelement (Detail)

CSFM analysis is based on finite element methods. Concrete members and rebars are composed of finite 2D and 1D elements, respectively. At the same time, multi-point constraints (MPC) and bond elements together with springs at the ends of each rebar capture the interaction between concrete and reinforcement. In version 21.0, the MPC and bond elements were joined together and substituted by the new so-called superelement, which covers their functions in the finite element model. As a result, the CSFM analysis is faster, more stable, and may fix some models that did not converge.



MPC elements + Bond elements = Superelement

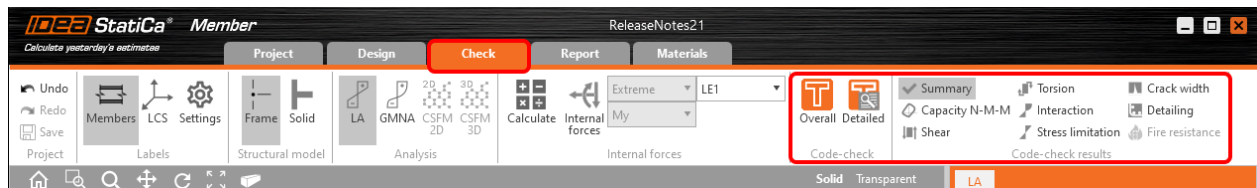
Member for concrete

Another step enables the Member application to emerge from BETA. For concrete, the focus was on fully implementing the RCS app and automatization of reinforced concrete section code-checks.

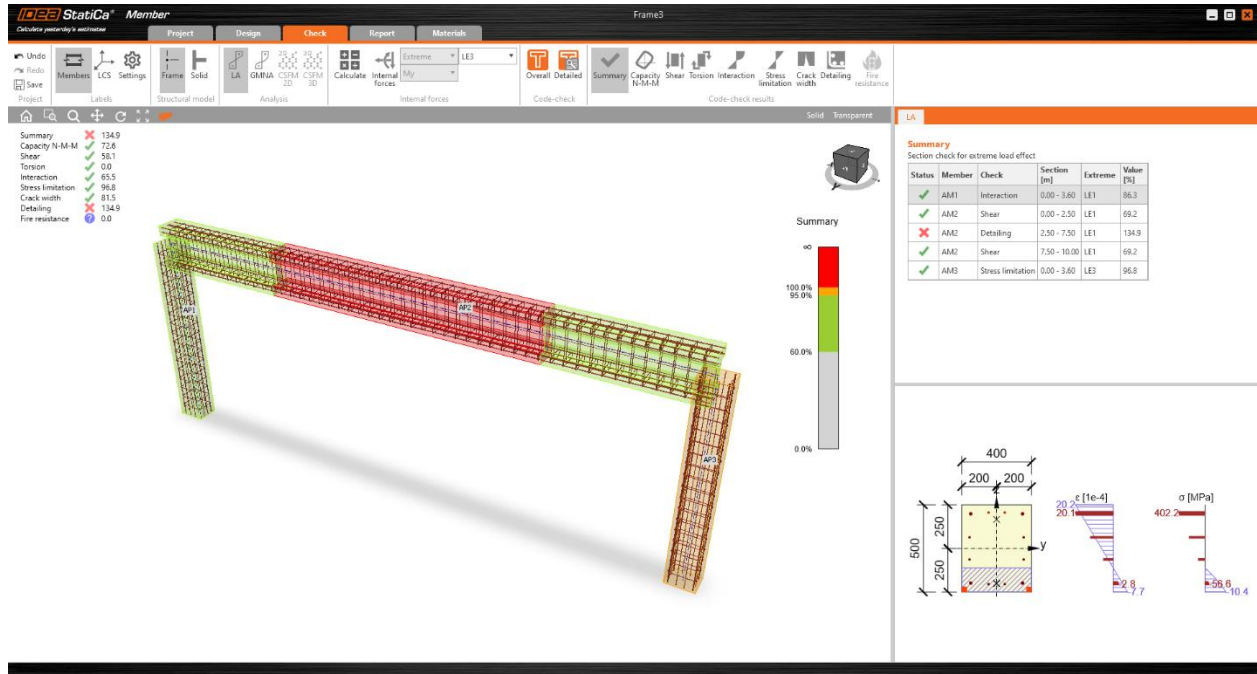
Code-checks

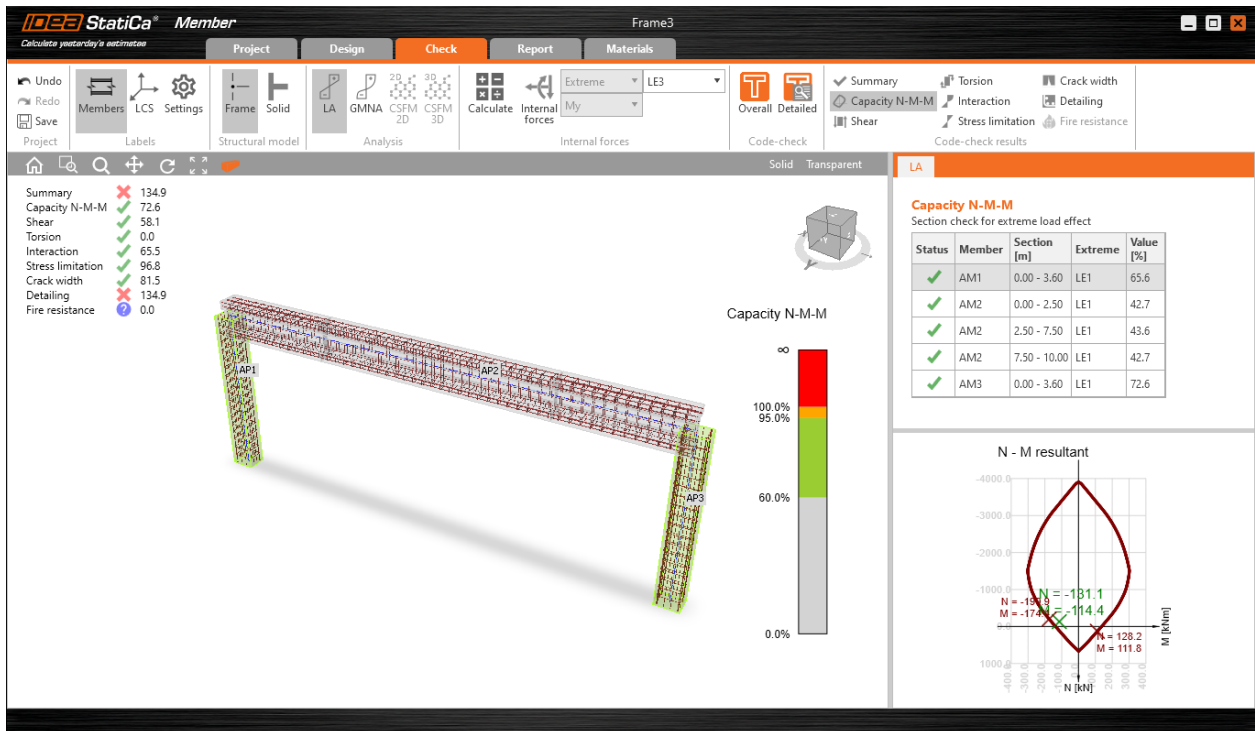
Now, all checks are available directly in the Member app, specifically at the Check tab ribbon; no necessity to go to RCS application.

All eight code-checks as Capacity, Shear, Torsion, Interaction, Stress limitation, Crack width, Detailing, and the Summary are available from the ribbon group Code-check results after clicking on the Overall button on the Code-check ribbon group!

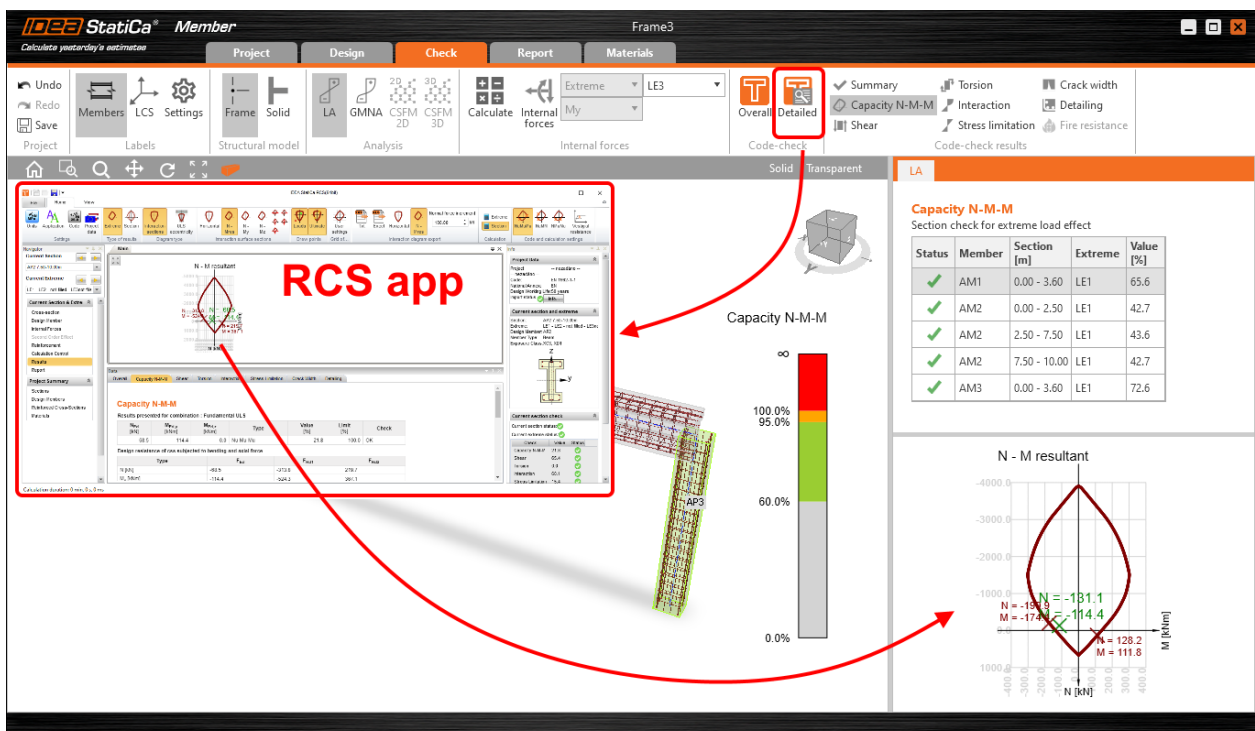


Traffic light visualization signaling pass/fail code-check beloved by our users was integrated into the 3D scene when viewing the result checks. Users can easily find out what part of the structure is ok or not and the utilization of each analyzed member. Plus, a table listing sections of extreme checks and a 2D-view of the particular check result in the property window, the same one as in RCS app.





The RCS application engine always performs Code-checks, and it is up to the user how he/she wants to check out the results. To see overall check results in 3D-scene, you click on the Overall code-check button. If you wish to examine the result code-checks, you can click on the Detailed code-check button, which launches the RCS app.



Report

Generating a report is a crucial part of code-checks. Now, the report provides all input data and results, both overall and detailed. All necessary information from RCS and Member apps is combined and offers a fully customizable report to print out or save in an editable format.

The screenshot displays the IDEA StatiCa Member software interface. The top menu bar includes 'Project', 'Design', 'Check', 'Report', and 'Materials'. The 'Report' tab is active. In the top-left toolbar, the 'DOC' and 'PDF' icons are highlighted with a red box. A red arrow points from these icons to a preview window on the right side of the interface. This preview window shows a report layout with a 3D model of a frame structure, a progress bar indicating 100.0% completion, and a 'Report' settings panel on the right. Below the 3D model, there are detailed tables for section AP1.

Begin [m]	End [m]	Governing type of check	Value [%]	Check
0.00	3.60	LE1 - Interaction	86.3	OK

Section check for zone: 0.00 - 3.60 m

Governing type of check	Combination	N_{Ed} [kN]	$V_{Ed,y}$ [kN]	$V_{Ed,x}$ [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,x}$ [kNm]	Value [%]	Check	
Interaction	LE1 - Fundamental	-131.1	0.0	-68.5	-7.0	-114.4	0.0	86.3	OK

Type of check	Combination	N_{Ed} [kN]	$V_{Ed,y}$ [kN]	$V_{Ed,x}$ [kN]	$M_{Ed,y}$ [kNm]	$M_{Ed,x}$ [kNm]	Value [%]	Check	
Capacity N-M-M	LE1 - Fundamental	-131.1	0.0	-68.5	0.0	-114.4	0.0	65.6	OK

Code-check manager (BIM links)

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

The current state of the supported versions will be always presented on our web. With the 21.0 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 21.0. The third column shows versions which are no longer supported.

IDEA StatiCa Application	21.0		
	Supported	In development	Obsolete
Advance Steel	2020, 2021	2022	-
Revit	2020, 2021	2022	-
Tekla Structures	2019i, 2020	2021	2019
Advance Design	2020, 2021	-	-
AxisVM	X5.4, X6	-	-
ETABS	18, 19	-	17
Midas Civil	2020, 2021	-	2019
Midas Gen	2020, 2021	-	2019
RFEM / RSTAB	5.24 / 8.24, 5.25 / 8.25	-	5.22 / 8.22, 5.23 / 8.23
Robot Structural Analysis	2020, 2021	2022	-
SAP2000	22, 23	-	21
SCIA Engineer	19.1, 20	-	18.1, 19
STAAD.Pro CONNECT	22	-	-

Steel

Steel & Concrete

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

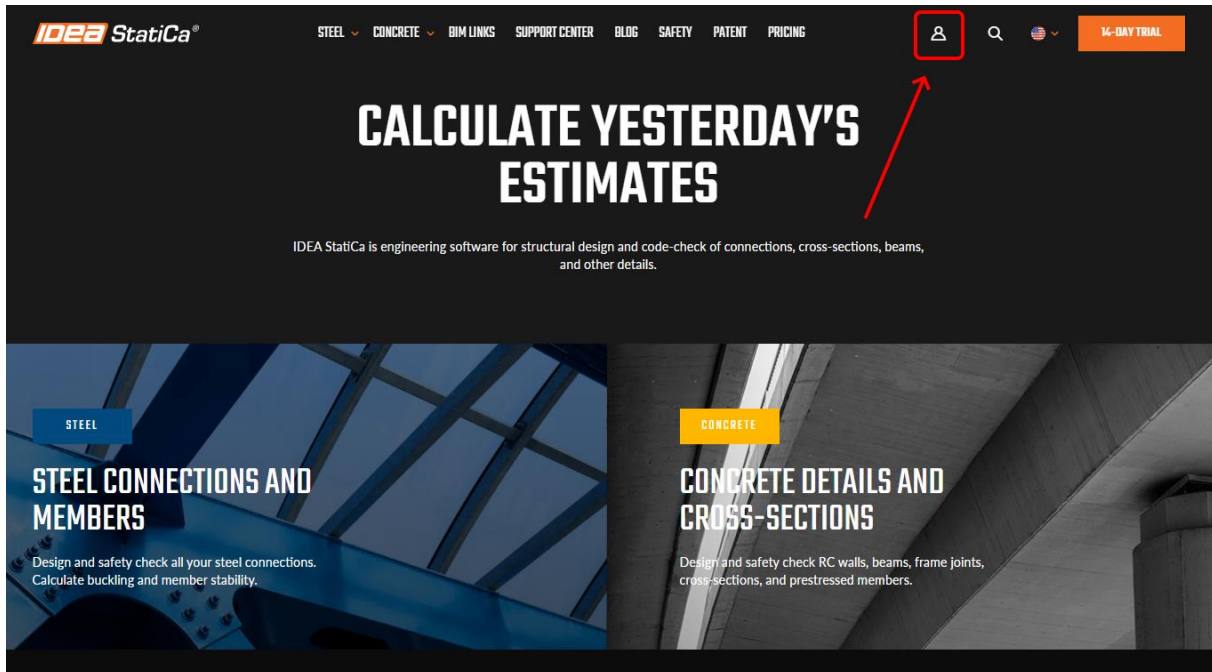
New AxisVM BIM link

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

User portal

A new online portal for managing IDEA StatiCa accounts, licenses, and support cases has been integrated into the website www.ideastatica.com.

To enter the portal, click on the login icon and sign in with your IDEA StatiCa credentials.



The portal is represented by a dashboard containing three sections:

- Account
- Licenses & Users
- Cases

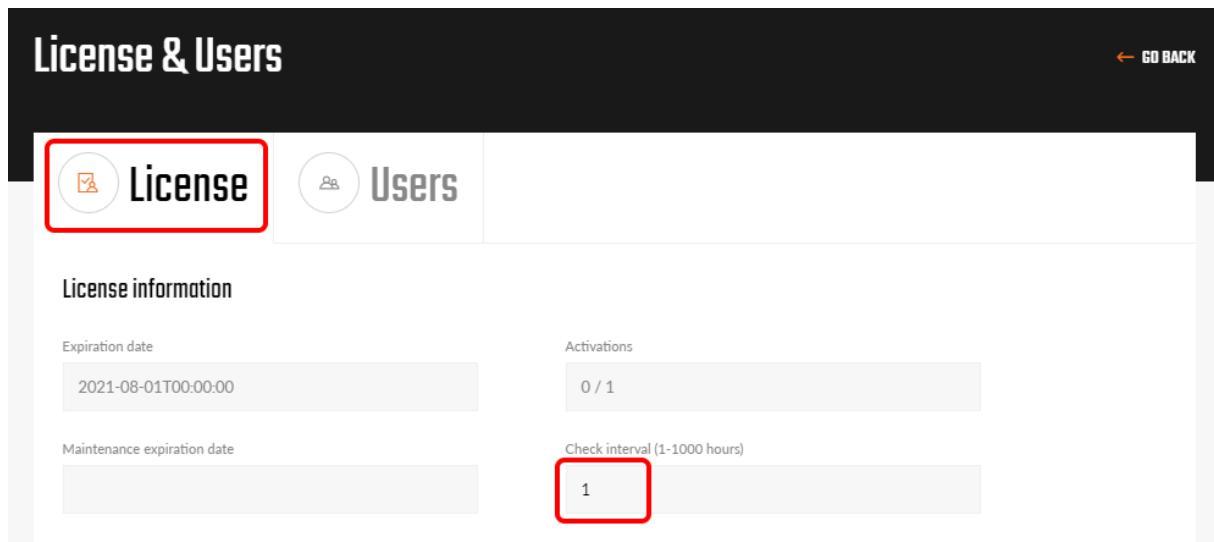
Account

This section contains general information such as company name and address, contact emails, reseller name, etc. Administrator can edit some of the data.

Licenses & Users

The tab Licenses contains information about the software license and its usage. Every user can view purchased products, expiration dates, and current usage of the license seats.

An administrator can change the **check interval**, which can be set as 1-1000 hours (default value 72 hours). The number represents for how long the license will work on a computer, after it is turned offline from the internet.



The tab Users is visible only for the administrator. Here license users can be added, deleted or edited.

Cases

In the section Cases, users can create a new support case. Cases from all users within one account are listed here, visible to each other.

Create a new case sends a support ticket to the IDEA StatiCa helpdesk. By selecting the application and typing the title of the case, the system offers possible answers from the Support center.