

#### RELEASE NOTES Release notes IDEA StatiCa 23.0

Apr 26, 2023

Welcome to the latest version of IDEA StatiCa. We've turned the volume all the way up to 23.0 to bring you super new features that help you to streamline your workflows, win back valuable time and money, and eliminate risk. And for those of you in the US, we have some significant updates... read on!

### **News for Steel**

**Speed**, **precision**, and **clarity** are the order of the day for our Steel applications in IDEA StatiCa 23.0. For those of you looking for **more transparency in your code check outputs**, we have included more features and **improvements to reports**. As one example, not only can you see things such as eccentricities added due to the alignment of one plate to another, you can **also see the load effects**, whether they start at the beginning of the member or at the end. Plus, should you, in your excitement, accidently click the report tab, the report will not auto-generate, which is guaranteed to avoid some serious frustration!

Did we mention **AISC prequalification checks for seismic design**? This allows you to already see limits in the 3D scene, remedy any problems in your models immediately with everything then clearly shown in check tables and the report.

With our **stunning new mesher**, you get even more accurate results with improved reliability and a nicer mesh. Now open sections have a finer mesh by default, helping you achieve results that are safer than ever before.

#### Code check more with transparent inputs

#### Check welds of welded sections

IDEA StatiCa can check the longitudinal welds of members with welded cross-sections now.



You can change the type of the weld from butt to fillet weld and set its parameters as material or weld throat thickness.

The option to change the weld type to such weld profiles to perform crane girder welds and crane runway beam welds is available for welded cross-sections in the library:

Cross-Section I	Navigator				×
Rolled Weld	led, Composed	Cold-formed	Timber		
			$\bigtriangledown$	TT	
					Close

The new fields with the weld parameters can be found in the cross-section definition window:

lw						×
-	lw				7	
	Name	lw400x200			Ā	
	Height [mm]	400		<b>1</b>		
-	Flange					
	Thickness [mm]	16.0				
	Width [mm]	200				
-	Web					
	Thickness [mm]	12.0		- 368	у	
-	Material					
	Steel	s 355 🔹 🖍 🕂				
-	Welds					
	Top flange-web [mm]	5 📫 < default > 🔻 🚣 🚣 🚣				
	Bottom flange-web [mm]	5 🗘 < default > 🔻 🦺 🦺 🔔				
					$4^{94}$ $12^{94}$	
			∧ <sup>z</sup> ,		200	
			<u>└</u> >'			
					OK Cancel	

If you want to review the parameters, you can open the appropriate cross-section in Materials:

/ CONNECT		untitled E
	oject Design Check Report Materials	u and a second
New Copy MPRL Cross-sect	tion Steel Concrete Bolt grade assembly:	
Project items DB	New Edit	
Materials		
✓ Cross-sections 1 - HEB240	▼ Iw 5.5	
2 - IPE360	Height (mm) 400	Z
<ul> <li>Materials</li> </ul>	▼ Flange	
S 355 (Steel) Bolt assemblies	Thickness [mm] 16.0	۲ et
	▼ Web	
	Thickness [mm] 12.0	
	Material     Steel S355	
	Velds	
	Top flange-web (mr S 🗧 < default > 🔻 🔔 🛄 🔔	
	Bottom flange-web 5 Characteristics	
	A [mm2] 10816	
	Ay [mm2] 6400	8 %
	α[ <sup>1</sup> ] 0.0	4 %
	ly [mm4] 2.859E+08	
	lyz [mm4] 2.1386E+07 lyz [mm4] 3.0726E-09	
	lp (mm4) 3.0729E+08	
	It [mm4] 7.4888E+05 Iw [mm6] 7.8839E+11	
	Wely [mm3] 1.4295E+06	
	Wpl,y [mm3] 1.6351E+06 Wel v [mm3] 2.1386E+05	
	Wp[,z [mm3] 3.2E+05	
	ly [mm] 162.6	
	y0 (mm) 0	
	20 [mm] 0	
		94 12 94
	,Z	200
	L_Y	
Design code: EN Analysis: Stress. strain	Load effects: In equilibrium Units: .mm	www.idextbics.com

The welds are checked, and the values of stress, strain, and checks based on code are presented in the Results:



All the results are presented also in the Report.

StatiCa <sup>®</sup> CONNECTION				untitled		
Debuice percentry's enclosed Project Design	Check	Report Materials				۹
CONI · COPY Concrete Print Preview DOC PDF DXF	Brief One Detailed	BOM Current All Selected				
Welds					*	Repet
Item	Edge Tw	L Loads 0 <sub>w,Ed</sub> Epi	σ <sub>⊥</sub> τ <sub>  </sub> τ <sub>⊥</sub>	Ut Uto Status	✓ Report	
0.01	C-w 1 ∡ 5.0 ⊾	(MPa) (%) 966 LE1 297.6 0.0	(MPa) (MPa) (MPa) 122.5 -97.4 122.7	68.3 19.3 OK	- Parag	aph
	⊿ 5.0 ⊾	966 LE1 297.7 0.0	122.6 97.4 -122.6	68.3 19.3 OK	- Mate	ials 🖌
C-bit 1	C-w 1 ∡ 5.0 ⊾	964 LE1 121.7 0.0	0.1 70.2 -0.1	27.9 13.1 OK	- Proje	t items
	<b>⊿</b> 5.0 ⊾	964 LE1 121.7 0.0	0.1 -70.3 0.1	27.9 13.1 OK	Lc	3N1
B-th 1	B-w1 ⊿ 5.0 ⊾	500 LE1 127.6 0.0	35.2 61.4 -35.2	29.3 15.5 OK	— Code	settings 🗸
	45.0 ⊾	500 LE1 127.7 0.0	35.2 -61.5 35.2	29.3 15.5 OK	- Theor	etical background
8-011	B-w1 45.0 L	567 LE1 214.6 0.0	-71.3 -92.7 -71.3	49.3 16.8 OK	- Softw	are info
081	B-H1 47.0 M	199 LE1 194.8 0.0	96.0 0.8 97.9	44.7 31.9 OK		
	47.0 ⊾	199 LE1 173.0 0.0	83.6 25.4 -83.7	39.7 24.0 OK		
C-811	B-bfl1 ∡7.0 ⊾	199 LE1 212.8 0.0	-31.5 32.7 -117.0	48.8 30.9 OK		
		199 LE1 216.0 0.0	-146.7 1.5 91.5	49.6 36.5 OK		
C-811	B-w 1 ▲ 4.0 ⊾	373 LE1 172.0 0.0	-52.2 79.0 -52.2	39.5 21.6 OK		
	<b>⊿</b> 4.0 ⊾	373 LE1 172.0 0.0	-52.1 -79.0 52.2	39.5 21.6 OK		
C-bil 1	STIFF1a 4.0 L	94 LE1 25.7 0.0	8.1 10.9 8.9	5.9 3.6 OK		
Cont.	4.0 k	94 LE1 52.7 0.0	-28.5 14.4 21.2	12.1 8.0 OK		
CW1	440	370 LE1 92.1 0.0	-117 -485 -82	18.8 9.9 OK		
C411	STIFF1a 44.0 M	94 LE1 202.7 0.0	-97.0 -17.1 -101.3	45.5 42.6 OK		
		94 LE1 122.4 0.0	-64.1 12.4 58.9	28.1 23.9 OK		
C-bfl 1	STIFF1b 4.0 L	94 LE1 52.7 0.0	-28.5 -14.4 -21.2	12.1 8.0 OK		
	<b>⊿</b> 4.0 ⊾	94 LE1 25.7 0.0	8.1 -10.9 -8.9	5.9 3.6 OK		
C-w 1	STIFF1b 4.0 L	370 LE1 82.1 0.0	-11.7 46.5 -6.2	18.8 9.9 OK		
	4.0 ⊾	370 LE1 79.9 0.0	21.8 -41.3 -16.3	18.3 10.0 OK		
041	STIFF16 44.0 K	94 LE1 122.4 0.0	-64.1 -12.3 -58.9	28.1 23.9 OK		
971002	0.01 440	100 LE1 202.6 0.0	-97.0 17.1 101.3	40.5 42.0 UK		
STIFF2	C-bfl 1 44.0	199 LE1 30.2 0.0	18.8 -6.7 -11.9	6.9 3.6 OK		
STIFF2	C-w 1 🖌 4.0	366 LE1 253.5 0.0	-117.5 54.9 -117.5	58.2 47.5 OK		
Design da	ata				Project item	settings
	Material	β_	σ <sub>w,Rd</sub>	0.9 0	Drawings - mo	del JA JA JA
8 355		(-)	(MPa) 435.6	(ser-a) 352.8		I 🖾 🗷
Symbol	volapation				Drawings - res	alts 😫 📶 🔛
Junior e	Throat thicknes				Bill of material	× 📰 🗷
L	Length				Cost estimation	× 🗉
σ <sub>wEd</sub>	Equivalent stre	15			Formulas	× :- :=
Ser.	Strain				Explanations	× 📰 📰
σ_	Perpendicular s	itress			Picture colors	4 4
T <sub>1</sub>	Shear stress pa	anallel to weld axis				

This feature is available in IDEA StatiCa Connection, Member, and Checkbot. In Connection, the full functionality is provided. For projects in Member, the welds of the welded cross-sections can be defined, nevertheless, the codechecks of the full member's welds are not present. The welded cross-sections can be defined in Checkbot as well. Available in the **Enhanced** edition of IDEA StatiCa Steel.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Detailed model and loads reporting

To deliver the project, a proper report is necessary. We ensure you have all the detailed information and calculation details about the model and loads setup included. You are able to generate a more detailed report with the following information:

#### Supports and forces

The chosen model type (the applied supports, respectively) are presented in the table and in the figure below. For example, the supported end of the bearing member will have all six reactions listed, while the free end will have none of them. For the N-Vy-Vz model type, the complementary forces, i.e., Mx-My-Mz, will be presented. The calculation report also includes boundary conditions.

The table also presents the position of the applied forces.



#### Alignment added to eccentricities

The chosen alignment type is now taken into account when displaying the table of members' offsets. The geometry table in the report calculates the total offset in the global coordinates.



#### Load effects

The loaded part of members (Begin/End), as well as the table of unbalanced forces, has been added to the table of applied load effects.

Name	Member	N [kN]	<b>Vy</b> [kN]	Vz [kN]	Mx [kNm]	<b>My</b> [kNm]	<b>Mz</b> [kNm]
LE1	B / Begin	0.0	0.0	50.0	-0.3	0.0	0.0
	B / End	0.0	0.0	50.0	-0.3	0.0	0.0
	B1 / End	0.0	0.0	-100.0	0.0	0.0	0.0
LE2	B / Begin	0.0	0.0	50.0	-0.3	0.0	0.0
	B / End	0.0	0.0	50.0	-0.3	0.0	0.0
	B1 / End	0.0	0.0	0.0	0.0	0.0	0.0

#### Load effects (forces in equilibrium)

#### **Unbalanced forces**

Name	<b>X</b> [kN]	Y [kN]	<b>Z</b> [kN]	<b>Mx</b> [kNm]	<b>My</b> [kNm]	<b>Mz</b> [kNm]
LE1	0.0	0.0	0.0	0.0	0.0	0.0
LE2	-17.4	0.0	98.5	-0.5	0.0	-0.1

#### **Buckling shape**

A figure with the most probable buckling shape (lowest factor) has been added to the report for when the buckling analysis is performed.



First buckling mode shape, LE2

#### Report not generated automatically

Generating the report may take some time, and it is not always needed. From now on, the report is generated manually by pressing the Generate button.

Columna StatiCa <sup>®</sup> CONNECTION Columna project	Design Check Report Materials	untitled	
CONS • 20 Copy New Copy Project Items	CF DIS Berl Or Detailed SUM Current All Selected Type of report Type of report		
	Set Project News settings for dealed report and clock Generation	bion	Properties       Image and the setting of

Currently, this functionality works properly just the first time the Report tab is opened. The next time, the report is automatically regenerated. This may be a part of future development.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Qualification checks of seismic prequalified connections for AISC

IDEA StatiCa provides the feature in the Connection application – Check of Prequalified Connection for Seismic Applications according to the code ANSI/AISC 358-16. The results of the checks are listed in a new table in the report with all the necessary information to prove that the requirements of prequalified connections design of the code ANSI/AISC 358-16 have been fulfilled.

The checking of the code requirements is done in real-time. Short messages displayed directly on the 3D scene help the designer create the connection design in a more efficient way with significant time savings.

This new feature is intended as a check of scope of prequalification for all engineers who perform structure designs and analysis of structures according to the AISC codes for seismic design.

### Theoretical background

Standard ANSI/AISC 358-16 specifies the design, detailing, fabrication and quality criteria for connections that are prequalified in accordance with the AISC Seismic Provisions for Structural Steel Buildings (herein referred to as AISC Seismic Provisions) for use with special moment frames (SMF) and intermediate moment frames (IMF). The connections contained in this standard are prequalified to meet the requirements in AISC Seismic Provisions only when designed and constructed in accordance with the requirements of this standard.

### First setting

There is a new property grid where it is necessary to set the Analysis type to Capacity design and enable the option for Prequalified connection.

I

Solid	Transparent Wireframe	09		
		•	Project item	
			Name	09
			Description	
	K		Design code edition	AISC 360-16 (LRFD) 🔻
	09		Analysis type	Capacity design 🔹 🔻
	Members		Prequalified connection	×
	✓ <u>M1</u>		System	Intermediate moment frames (IMF)
	🖌 M2		Connection type	Reduced beam section (RBS)
	▲ Load effects		Report	You can add text and pictures
	🖌 LE1			
	LE-CD2			
	<ul> <li>Operations</li> </ul>			
	FP1			
	CUT1			
	OPN1			
	OPN2			
	OPN3			
	STIFF1			
	<ul> <li>Dissipative items</li> </ul>			
	JISS1			

After that, it is possible to select the System and Connection type according to the one in code ANSI/AISC 358-16. There are two possible systems, Special moment frame (SMF) and Intermediate moment frame (IMF)

The connection types incorporated into the Connection app were selected based on their frequency of use and usability for analysis in the Connection app. A list of them is shown in the figure below.

### 09 Project item Name 09 Description 09 Losign code edition AISC 360-16 (LRFD)

# Analysis type Capacity design Prequalified connection Intermediate moment frames (IMF) System Intermediate moment frame (SMF) Connection type Special moment frames (IMF) Benort Intermediate moment frames (IMF)

▼ Pr Na De De	roject item ame escription esign code edition	09 AISC 360-16 (LRFD)
Na De De	ame escription esign code edition	09 AISC 360-16 (LRFD)
De De	escription esign code edition	AISC 360-16 (LRFD)
De	esign code edition	AISC 360-16 (LRFD) *
An	nalysis type	Capacity design 🔹
Pre	requalified connection	×
Sv	vstem	Intermediate moment frames (IMF)
Co	onnection type	Reduced beam section (RBS)
Re	eport	Reduced beam section (RBS) Bolted unstiffened extended end plate (BUEEP - 4E) Bolted stiffened extended end plate (BSEEP - 4ES) Bolted stiffened extended end plate (BSEEP - 8ES) Bolted flange plate (BFP) Welded flange-welded web (WUF-W) Double-tee

Models for seismic design are proposed in the wizard. The user can choose a single-sided or double-sided beam connected to the column. All these models are set to the proper connection type by default and are set up with appropriate load effects.



#### Usage

A message regarding the design of the prequalified connection is always displayed to the user.

If the design of the prequalified connection meets the code requirements, a green-OK-status message is displayed.

If any of the requirements is not fulfilled, the status of the message is changed to a yellow-exclamation-mark warning and a short message with the specific limitation is displayed. This message provides an immediate response to the designer's action as well as a specific value that needs attention for correction. Most of the checked values are related to IDEA StatiCa operations. If the design consists of only basic items such as stiffening plate, bolt grip, etc., a generic message can appear directing the user to use IDEA StatiCa's operations instead.



The results of the successful analysis of the detail are presented in the Report. There is a new table where all provided checks are stated. Every check has information about the item, the current value and requested value limit, a reference to the chapter of the code and the status of the check.

The used system and Connection type are also stated.

.,	Intermediate	e moment frames	(IMF)						
Connection type:	Reduced be	am section (RBS	)						
imit checks									
Iter	m	Value		Requirement	Reference	Status			
M1 – depth		393.70	≤ 1095.38	mm	[1] 5.3.2(3)	ОК			
M1 – flange slenderness	s	5.45	≤ 9.19		[1] 5.3.2(6)	ОК			
M1 – web slenderness		14.18	≤ 36.06		[1] 5.3.2(6)	OK			
M2 – depth		414.02	≤ 1095.38	mm	[1] 5.3.1(2)	ОК			
M2 – weight		75	≤ 449 kg/m	13	[1] 5.3.1(3)	OK			
M2 – flange thickness		16.00	≤ 44.45 mr	n	[1] 5.3.1(4)	ОК			
M2 – flange slenderness	s	5.61	≤ 9.19		[1] 5.3.1(6)	OK			
M2 – web slenderness		39.58	≤ 90.94		[1] 5.3.1(6)	ОК			
OPN2 – distance "a"		98.43	≥ 89.79 mr	m ≤ 134.68 mm	[1] (5.8-1)	OK			
OPN2 – length "b"		304.80	≥ 269.11 m	nm ≤ 351.92 mm	[1] (5.8-2)	ОК			
OPN2 – depth "c"		38.10	≥ 17.96 mr						
OPN3 – distance "a"		98.43	≥ 89.79 mi	Frequaimeu co	requaimed connection				
ODNO Leasth Th				Contains	the first second s		- (IME)		
OPINS – lerigth "b"		304.80	≥ 269.11 n	system:	Intermediate	e moment trame	is (IIVIF)		
OPN3 – length "b" OPN3 – depth "c"		304.80 38.10	≥ 269.11 n ≥ 17.96 mi	Connection type:	Reduced be	am section (RE	s (IMP) S)		
OPN3 – length "b" OPN3 – depth "c" M1-bfl 1 : M2-bfl 1 – wel	ld type	304.80 38.10 CJP	≥ 269.11 n ≥ 17.96 mi CJP	Connection type:	Reduced be	am section (RE	s (INF) S)		
OPN3 - length "b" OPN3 - depth "c" M1-bfl 1 : M2-bfl 1 - wel M1-bfl 1 : M2-bfl 1 - wel	ld type Id material	304.80 38.10 CJP E70xx	≥ 269.11 n ≥ 17.96 m CJP E70xx/E80	Connection type:	Reduced be	am section (RE	S (INF) S)	Deference	Statu
OPN3 - length "b" OPN3 - depth "c" M1-bfi 1 : M2-bfi 1 - wei M1-bfi 1 : M2-bfi 1 - wei M1-bfi 1 : M2-tfi 1 - weic	ld type Id material d type	304.80 38.10 CJP E70xx CJP	≥ 269.11 n ≥ 17.96 m CJP E70xx/E80 CJP	Limit checks	Reduced be	Value	S (IMF) S) Requirement	Reference	Statu
OPN3 – length "b" OPN3 – depth "c" M1-bfi 1 : M2-bfi 1 – wel M1-bfi 1 : M2-bfi 1 – wel M1-bfi 1 : M2-tfi 1 – wel M1-bfi 1 : M2-tfi 1 – welc	ld type Id material d type d material	304.80 38.10 CJP E70xx CJP E70xx	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8(	Connection type: Limit checks	Reduced be	Value	Requirement	Reference [1] 5.3.2(3)	Statu
OFN3 - length "c" OPN3 - depth "c" M1-bft 1 : M2-bft 1 - wel M1-bft 1 : M2-bft 1 - wel M1-bft 1 : M2-tft 1 - wel M1-bft 1 : M2-tft 1 - wel M1-bft 1 : M2-tft 1 - welc	ld type Id material d type d material d type	304.80 38.10 CJP E70xx CJP E70xx CJP	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP CJP	Connection type: Limit checks M1 – depth M1 – flange slenderne	Reduced be	Value 15.500 5.45	Requirement           ≤ 43,125 in           ≤ 9,19           > 26 05	Reference [1] 5.3.2(3) [1] 5.3.2(6)	Statu: OK OK
OPN3 - length "b" OPN3 - depth "c" M1-bfi 1: M2-bfi 1 - wel M1-bfi 1: M2-bfi 1 - wel M1-bfi 1: M2-tfi 1 - welc M1-bfi 1: M2-tfi 1 - welc M1-bfi 1: M2-tfi 1 - welc M1-bfi 1: M2-tfi 1 - welc	ld type Id material d type d material d type	304.80 38.10 CJP E70xx CJP E70xx CJP Fulfilled	≥ 269.11 n ≥ 17.96 mi CJP E70xx/E8( CJP E70xx/E8( CJP Free from	Limit checks M1 – depth M1 – liange slendernes M2 – depth	Reduced be	Value           15.500           5.45           14.18	Requirement < 43.125 in < 9.19 < 33.06 < 10.15 in	Reference [1] 5.3.2(3) [1] 5.3.2(6) [1] 5.3.2(6) [1] 5.3.2(7)	Status OK OK
OPN3 - length "b" OPN3 - depth "c" M1-bf1 1: M2-bf1 1 - wel M1-bf1 1: M2-bf1 1 - wel M1-bf1 1: M2-tf1 1 - welc M1-bf1 1: M2-tf1 1 - welc M1-bf1 1: M2-tf1 1 - welc M2 - protected zone M2 - fin plate	ld type Id material d type d material d type	304.80 38.10 CJP E70xx CJP E70xx CJP Fulfilled 9.53	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 mm	Connection type: Limit checks M1 – depth M1 – flange slendernes M2 – depth M3 – web slendernes	Reduced be	Value           15.500         5.45           14.18         16.300	Requirement \$43.125 in \$9.19 \$36.06 \$43.225 in \$29.19 \$36.06 \$43.225 in \$29.06 \$43.25 in \$29.06 \$43.25 in \$29.06 \$45.06	Reference [1] 5.3.2(3) [1] 5.3.2(6) [1] 5.3.2(6) [1] 5.3.2(6) [1] 5.3.1(2)	Status OK OK OK
VPTV3 - identifin "D" OPN3 - depth "c" M1-bfi 1: M2-bfi 1 - wel M1-bfi 1: M2-bfi 1 - wel M1-bfi 1: M2-tfi 1 - wel M1-bfi 1: M2-tfi 1 - wel M1-bfi 1: M2-wi 1 - welc M2 - protected zone M2 - fin plate M1 - continuity plates	id type Id material d type d material d type	304.80 38:10 CJP E70xx CJP E70xx CJP Fulfilled 9.53 Fulfilled		Connection type: Limit checks M1 – depth M1 – flange slendernes M2 – depth M2 – depth M2 – gange thickness	Reduced be	Value           15.500         5.45           14.18         16.300           5         0.630	Requirement           ≤ 43.125 in           ≤ 9.19           ≥ 30.06           ≤ 43.125 in           ≤ 28.06           ≤ 1.25 in	Reference           [1] 5.3.2(3)           [1] 5.3.2(6)           [1] 5.3.2(6)           [1] 5.3.1(2)           [1] 5.3.1(3)           [1] 5.3.1(2)	Status OK OK OK OK
OFN3 - depth "0" OFN3 - depth "0" M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-tfl 1 - wel M1-bfl 1: M2-wfl 1 - wel M2 - protected zone M2 - fin plate M1 - continuity plates M2 - weld access holes	Id type Id material d type d material d type	304.80 38.10 CJP E70xx CJP E70xx CJP Fuffiled 9.53 Fuffiled Fuffiled		Limit checks the second secon	Reduced be	Value           15.500         5.45           14.18         16.300           5         0.630           5.61         5.61	Requirement           < 43.125 in	Reference           [1] 5.3.2(3)           [1] 5.3.2(6)           [1] 5.3.2(6)           [1] 5.3.1(2)           [1] 5.3.1(3)           [1] 5.3.1(4)           [1] 5.3.1(6)	Status OK OK OK OK
OPN3 – depth "C" OPN3 – depth "C" M1-bfl 1: M2-bfl 1 – wel M1-bfl 1: M2-bfl 1 – wel M1-bfl 1: M2-tfl 1 – welc M1-bfl 1: M2-tfl 1 – welc M1-bfl 1: M2-tfl 1 – welc M2 – protected zone M2 – fin plate M1 – continuity plates M2 – weld access holes derence	id type Id material d type d material d type	304.80 38.10 CJP E70xx CJP E70xx CJP Fuifilled 9.53 Fuifilled Fuifilled	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 mm Geometry Geometry	System: Connection type: Limit checks M1 – depth M1 – depth M1 – depth M2 – web slendernes: M2 – depth M2 – meight M2 – flange thickness M2 – flange slenderne M2 – web slenderne	Intermediate Reduced be	Value           15.500           5.45           14.18           16.300           5           0.630           5.61           30.58	Requirement           < 43.125 in	Reference           [1] 5.3.2(3)           [1] 5.3.2(6)           [1] 5.3.2(6)           [1] 5.3.1(2)           [1] 5.3.1(3)           [1] 5.3.1(4)           [1] 5.3.1(6)           [1] 5.3.1(6)	Status OK OK OK OK OK
DFN3 - depth "c" M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-bfl 1 - wel M1-bfl 1: M2-tfl 1 - weld M1-bfl 1: M2-tfl 1 - weld M1-bfl 1: M2-tfl 1 - weld M2 - tin plate M2 - continuity plates M2 - weld access holes effence 11 _ ANSI/AISC	id type id material d type d material d type 35 358-16 ANSI/AISE	304.80 38.10 CJP E70xx CJP Fuffiled 9.53 Fuffiled C 358e1-18	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 mm Geometry Geometry	Connection type: Limit checks It depth M1 – depth M1 – flange slenderne: M2 – depth M2 – depth M2 – flange thickness M2 – flange slenderne M2 – flange slenderne M2 – flange slenderne M2 – web slendernes	Intermediate Reduced be am	Woment training           warm section (RE           15.500           5.45           14.18           16.300           5           0.630           5.61           39.58           4.000	Requirement           < 43.125 in	Reference           [1] 5.3.2(3)           [1] 5.3.2(6)           [1] 5.3.2(6)           [1] 5.3.1(2)           [1] 15.3.1(3)           [1] 15.3.1(4)           [1] 15.3.1(6)           [1] 15.3.1(6)           [1] 15.3.1(6)	Status OK OK OK OK OK
UPR3 - energin To OPR3 - depth "c" M1-bf1 1: M2-bf1 - wel M1-bf1 1: M2-bf1 - wel M1-bf1 1: M2-bf1 - wel M1-bf1 1: M2-bf1 - wel M1-bf1 1: M2-bf1 - welch M1-bf1 1: M2-wf1 - welch M2 - protected zone M2 - fin plate M2 - weld access holes <b>eference</b> 11 ANSI/AISC 21 ANSI/AISC 21 ANSI/AISC	id type id material d material d type d material d type s c 358-16, ANSI/AISI c 318-16	304.80 38.10 CJP E70xx CJP E70xx CJP Fulfilled 9.53 Fulfilled C 358s1-18	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 mm Geometry	System: Connection type: Limit checks It depth M1 - depth M1 - depth M1 - depth M1 - web slenderns M2 - depth M2 - weight M2 - meight M2	intermediate Reduced be em ess a a a a a a a a a a a a a a a	Woment trame           value           15.500           5.45           14.18           16.300           5           0.630           5.61           39.58           4.000           14.000	Requirement           < 43.125 in	Reference [1] 5 3 2(3) [1] 5 3 2(6) [1] 5 3 2(6) [1] 5 3 1(2) [1] 5 3 1(2) [1] 5 3 1(3) [1] 5 3 1(4) [1] 5 3 1(6) [1] (5 8-1) [1] (5 8-2)	Status OK OK OK OK OK OK
Um3 - migm 10 DPN3 - depth "c" M1-bit 1: M2-bit 1 - wel M1-bit 1: M2-bit 1 - wel M2 - protected zone M2 - protected zone M2 - continuity plates M2 - wel access holes M2 - well access holes M2 - M3/M3/M3C 21 ANSI/M3C	id type Id material d material d material d type 3 3 3 3 3 3 3 3 4 1 6 3 6 3 6 - - - - - - - - - - - - -	304.80 38.10 CJP E70xx CJP E70xx CJP Fuffiled 9.53 Fuffiled Fuffiled C 358s1-18	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 mm Geometry Geometry	Connection type: Limit checks IIII - depth M1 - depth M1 - web stenderness M2 - depth M2 - weight M2 - mange stendernes M2 - mange stendernes M2 - weight M2 - weigh	intermediate Reduced be em iss iss iss iss iss	Woment training           Value           15.500           5.45           14.18           16.300           5           0.630           5.61           39.58           4.000           15.00	Requirement           < 43.125 in	Reference (15 5.2(3) (15 5.2(6) (15 5.3(6) (15 5.3(6) (15 5.3(3) (15 5.3(4) (15 5.3(6) (15 5.3(6) (11 (5.8-1) (1) (5.8-2)	Status OK OK OK OK OK OK OK OK
Umx3 - migm 10           OPN3 - depth "c"           M1-bit 11         M2-bit 1 - well           M1-bit 12         M2-bit 1 - well           M1-bit 14         M2-bit 1 - well           M1 - bit 14         M1-bit 14           M2 - protected zone         M2 - protected zone           M2 - min plate         M1 - continuity plates           M1 - continuity plates         M2 - axis/AISC           Q1         ANSI/AISC           Q2         ANSI/AISC           M3         ANSI/AISC	id type id material d type d material d type 3 358-16, ANSI/AISI 341-16 360-16	304.80 38.10 CJP E70xx CJP Fufiled 9.53 Fufiled Fufiled C 358s1-18	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 m Geometry Geometry	System:           Connection type:           Limit checks           It           M1 – depth           M1 – flange slenderne           M2 – depth           M2 – depth           M2 – nange slenderne           M2 – flange slenderne           OPN2 – slength "b"           OPN2 – slength "b"           OPN2 – depth "b"	Intermediate Reduced be em	Walue           15,500           5,45           14,18           16,300           5           0,630           5,61           39,58           4,000           14,000           15,000	Requirement           ≤ 43.125 in           ≤ 9.19           ≤ 30.66           ≤ 43.125 in           ≤ 28.06           ≤ 43.125 in           ≤ 28.06           ≤ 1.750 in           ≤ 9.19           ≤ 30.30 in           ≤ 1.750 in           ≤ 9.55 in ≤ 5.303 in           ≥ 10.595 in ≤ 13.855 in           ≥ 10.70 fm = 1.768 in           ≥ 15.50 in ≤ 5.303 in	Reference           [1] 5 3.2(3)           [1] 5 3.2(6)           [1] 5 3.2(6)           [1] 5 3.2(6)           [1] 5 3.3(2)           [1] 5 3.3(2)           [1] 5 3.3(4)           [1] 5 3.3(4)           [1] 5 3.3(6)           [1] 5 3.3(6)           [1] 5 3.3(6)           [1] (5 8-1)           [1] (5 8-2)           [1] (5 8-3)           [1] (5 8-3)	Statu: OK OK OK OK OK OK Not OI
Um3 - migm D DPN3 - depth "c" MI-b01 1: M2-b11 - wel MI-b01 1: M2-b11 - wel MI-b01 1: M2-b11 - wel MI-b01 1: M2-t11 - wel MI-b01 1: M2-t11 - wel MI-b01 1: M2-t11 - wel MI-b01 1: M2-t11 - wel MI - continuity plates M2 - weld access holes eference (1) ANSI/AISC (2) ANSI/AISC (2) ANSI/AISC	Id type Id material d type d material d type 2 358-16, ANSI/AISI 3 341-16 2 360-16	304.80 38.10 CJP E70xx CJP E70x CJP Fulfilled 9.53 Fulfilled Fulfilled Fulfilled	≥ 269.11 n ≥ 17.96 m CJP E70xx/E8( CJP E70xx/E8( CJP Free from ≥ 9.53 m Geometry Geometry	System:           Connection type:           Limit checks           M1 – depth           M1 – flange sienderms           M2 – depth           M2 – web slendernes           M2 – flange sienderms           OPN2 – distance "a"           OPN2 – distance "a"           OPN2 – distance "a"           OPN3 – death "b"	intermediate Reduced be	moment trame am section (RE 15.500 5.45 14.18 16.300 5 5 0.630 5.61 39.58 4.000 14.000 1.500 4.000 1.500 4.000	Requirement           4 3.125 in           9.19           \$ 50.06           4 3.125 in           2.8 pcf           1.750 in           9.19           \$ 50.55 in ≤ 5.303 in           > 10.595 in ≤ 5.303 in           > 10.595 in ≤ 5.303 in           > 10.595 in ≤ 1.3855 in           > 20.707 ms.1768 in           > 3.535 in ≤ 5.303 in	Reference (1 5 3 2(3) (1 5 3 2(4)) (1 5 3 2(4)) (1 5 3 2(4)) (1 5 3 1(4)) (1 5 3 1(4)) (1 5 3 1(4)) (1 5 3 1(6)) (1 5 3 1(6)) (1 5 8 -1) (1 (5 8 -2)) (1 (5 8 -2)) (1 (5 8 -2)) (1 (5 8 -2))	Statu OK OK OK OK OK OK Not O OK

#### Specification of the checked requirements

Connection type specific checks and the general checks that are not related to a specific connection type are provided.

#### **General checks**

A check of **Limiting Width-to-Thickness Ratios** is performed according to AISC 341-16: Chapter D.1.1b and Table D1.1. The minimum requested slenderness of the flange and the web of the cross-section is checked.

A check of **Type of member's section** according to the ANSI/AISC 358-16: Chapter 2.3 is performed. The types of the cross-section used in the members for prequalified design are limited according to the code. In the Connection app, the limitations of the supported cross sections are used for prequalified connection design.

Types of cross-sections used for beams:

Cross-Sectio	n Navig	ator				×	
Rolled W	elded, Cor	mposed	Cold-formed Timb	er			
	1.6		D.				
		Cross-Sec	tion Navigator				×
		Rolled	Welded, Composed	Cold-formed	Timber		
Ţ							
					$\bigtriangledown$	TT	
							Close

Types of cross-sections usable for columns:

Cross-Section	n Navigator ×	
Rolled We	elded, Composed Cold-formed Timber	
	Cross-Section Navigator	×
	Rolled Welded, Composed Cold-formed Timber	
Ţ		I
		Close

**Fastening assemblies** are checked according to ANSI/AISC 358-16: Chapter 4.1. There is a list of bolt assemblies dedicated to a prequalified connection.

Protected zones are checked according to ANSI/AISC 341-16: Chapter D1.3.

The geometry of the **Continuity plates** is checked according to AISC 341-16: Chapter E2.6f.2 and Chapter E3.6f.2. The geometry of the **Doubler plates** is checked according to AISC 341-16: Chapter E3.6e.

#### **Specific checks**

**Reduced Beam Section (RBS)** 



This Connection type is checked according to Chapter 5 in code ANSI/AISC 358-16. The following checks are provided:

- Checks of the beam limitations according to Chapter 5.3.1.
- Checks of the column limitations according to Chapter 5.3.2.
- Checks of the beam-flange to column weld limitations according to Chapter 5.5 and a related check of the weld access hole according to AISC 360-16: Chapter J1.6.
- A check of the beam-web to column weld limitations according to Chapter 5.6.
- A check of the geometry of the RBS according to Chapter 5.8 Step 1.

The shear strength according to Chapter 5.6 (1) and the weld of the fin plate, according to Chapter 5.6 (2), are stated as out of scope of the check in Connection.

A new feature is prepared for the pre-design of the weld access hole, which is performed in the beam web according to the AISC Seismic Design Manual – Table 1-1. This feature can be used to define the optimal geometry of the weld access hole by software that fulfills the code's requirements.

OPN	l [Opening, notch]		Pre-design 🔹	Copy Delete
_	On an in a state		AISC Seismic	design manual
	Opening, notch		AWS D1.8 - 6.	11.1.2
	Cross-section part	M2   Web 1		🔹 🗖 🔖
	Shape	Notch		•
	Location	Both		•
	B, B1 - width [in]	2 -1.15625		
	H, H1 - depth [in]	1.59375 0		
	Rounding radius [in]	0.500		

#### Bolted Unstiffened/Stiffened Extended End Plate (BUEEP - 4E / BSEEP - 4ES / BSEEP - 8ES)



This connection type is checked according to Chapter 6 in code ANSI/AISC 358-16. The following checks are provided:

- Checks of the beam limitations according to Chapter 6.3.1.
- Checks of the column limitations according to Chapter 6.3.2.
- Checks of the connection detailing limitations according to Chapter 6.7 and Table 6.1. This check contains a check of the bolt and end plate geometry, if stiffened, also the stiffener of the end plate.

The finger shims are not supported in the operation and are stated as out of scope of the check in Connection. Also, the check of the welds for the full strength of the beam web in tension, according to the Chapter 6.7.6 (3), is stated as out of scope of the check in Connection.

#### **Bolted Flange Plate (BFP)**



This connection type is checked according to Chapter 7 in code ANSI/AISC 358-16. The following checks are provided:

- Checks of the beam limitations according to Chapter 7.3.1.
- Checks of the column limitations according to Chapter 7.3.2.
- Checks of the connection detailing limitations according to Chapter 7.5. This check contains a check of the plate's material, welds and bolt grade and diameter.



#### Welded Unreinforced Flange-welded Web

This connection type is checked according to Chapter 8 in code ANSI/AISC 358-16. The following checks are provided:

- Checks of the beam limitations according to Chapter 8.3.1.
- Checks of the column limitations according to Chapter 8.3.2.

- Checks of the beam-flange to column weld limitations according to Chapter 8.5. and a related check of the weld access hole according to AWS D1.8/D1.8M: Chapter 6.11.1.2.
- Check of the beam-web to column weld limitations according to Chapter 8.6.

The weld of the single plate to the column according to the Chapter 5.6 (2) is stated as out of scope of the check in Connection.

A new feature is for the pre-designing of the weld access hole, which is performed in the beam web according to AWS D1.8/D1.8M: Chapter 6.11.1.2. This feature can be used to define the optimal geometry of the weld access hole using the software, which fulfills the code requirements.

OPN1	[Opening, notch]		Pre-design 🔹	Copy I	Delete			
		_	AISC Seismic o	design ma	anual			
•	Opening, notch		AWS D1.8 - 6.11.1.2					
	Cross-section part	M2   Web 1						
	Shape	Notch			•			
	Location	Both			•			
	B, B1 - width [in]	2 -1.15625						
	H, H1 - depth [in]	1.59375 0						
	Rounding radius [in]	0.500						

#### Double-tee



This connection type is checked according to Chapter 13 in code ANSI/AISC 358-16. The following checks are provided:

- Checks of the beam limitations according to Chapter 13.3.1.
- Checks of the column limitations according to Chapter 13.3.2.
- Checks of the connection detailing limitations according to Chapter 13.5. This check contains a check of the continuity plates', welds' and bolts' geometry and grade.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Update of ANSI/AISC 360-22, CSA S16:19 standards, and Taiwan sections

We have implemented the new editions of the structural design standards for North American users. The library of cross-sections was enhanced by Taiwanese profiles.

#### No directional strength increase for HSS (AISC)

For the welds of the rectangular hollow sections, we have implemented the requirement given by the Specification Edition 2022 of AISC. No directional strength increase is assumed in the weld resistance check.

This will lead to better safety of the design.



#### New codes

American users may now select the new editions of the design standards:

- CSA S16:19 (Canada, 2022)
- ANSI/AISC 360-22 (USA, 2022)

CON	1		CON	н				
•	Project item		•	Project item				
	Name	CON1	•	Name	CON1			
	Description			Description				
1	Design code edition	CSA S16-14 🗸		Design code edition	AISC 360-22 (LRFD)			
	Analysis type	CSA 516-14 CSA 516:19		ľ	Analysis type	AISC 360-22 (LRFD)		
1	Report			Report	AISC 360-22 (ASD) AISC 360-16 (LRFD) AISC 360-16 (ASD) AISC 360-10 (LRFD)			
					AISC 360-10 (ASD)			

#### **Cross-section library update**

The library of cross-sections (MPRL) was enhanced by local sections from Taiwan. These are:

- Channel parallel
- Channel tapered
- Wide flat plate

The update has been available in Connection, Member, and Checkbot since patch 22.1.1.



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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Calculation of anchors with stand-off

In version 23.0, IDEA StatiCa provides a new feature regarding RN 23.0: Anchors with stand-off. An anchor with stand-off is designed as a bar element loaded by compressive or tensile force, shear force, and **bending moment**. The calculation of the bending moment has been improved to respect better the real behavior of stand-off anchors.

The value of the bending moment depends on the ratio between base plate stiffness and anchor stiffness. For a very weak plate and stiff anchors, the bending moment in the anchor will be as follows:



For stiff plate and weak anchors, the bending moment will look like this:



In previous versions, the bending moment in stand-off anchoring was always calculated as in the first case, assuming a weak plate and stiff anchors. The lever arm for moment calculation was the distance between the mid-plane of the plate and the concrete plus one-half of the anchor diameter. This was quite an over-conservative approach.

Since release 22.1.5, the calculation of the bending moment has been updated. The value is determined directly by the finite element model. The anchor is modelled as a beam element and moments in it respect the actual stiffness of the system. The anchor is fixed on both sides, one side is  $0.5 \times d$  below the concrete level, and the other side is in the midplane of the plate. The extreme of the calculated moment is taken into account in the code check then.

More information regarding the code checking of anchors can be found in the theoretical background. Individual codes can be found here: EC, AISC, AS, STO.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Eurocode updates to thin-walled members and anchors

The calculations of checks by Eurocode are refactored in the Connection application. This also features warnings when the use-case lies out of the validity of the Eurocode.

### Eurocode refactoring

To comply with the structure of the application, additional slight changes in the code calculations are present in the following:

- Improved formulas in Eurocode
- Improved Check tables with unified subscripts, and units format

• Improved Report tables

Check	k of	bolts fo	or extr	eme loa	d effect			_	_		_			
		Status	ltem	Grade	Loads	F <sub>t,Ed</sub> [kN]	F <sub>v,Ed</sub> [kN]	F <sub>b,Rd</sub> [kN]	Ut <sub>t</sub> [%]	Ut <sub>s</sub> [%]	Ut <sub>ts</sub> [%]			
	-	0	B1	M16 8.8	- 1 LE1	31.3	3.0	156.8	34.6	5.0	29.7			
	Tens	sion res	istance	e check	(EN 1993-1	-8 – Tab.	3.4)							
	$F_{t}$	$Rd = \frac{k_2}{k_2}$	$f_{ab} A_{a}$	= 90.4	kN a	$F_{t,Ed} =$	31.3	kN			- 1			
			1142											
Bolts														
		hana		Itom	Crad		Londo	F <sub>t,Ed</sub>	F <sub>v,Ed</sub>	F <sub>b,Rd</sub>	Utt	Uts	Utts	Status
	5	nape		Item	Grad	6	Loads	[kN]	[kN]	[kN]	[96]	[96]	[%]	Status
	2			B1	M16 8.8 -	1 L	E1	73.7	3.2	99.1	81.5	5.3	63.5	OK
	+ -	+  15		<b>D</b> D	M16 0 0	4 J	E 4	70 7	3.5	00.4	01 E	6.0	82.8	OK
tress	utilia	zation												
$U_i =$	may	$\left(\frac{\sigma_{v,Ed}}{2}\right)$	$ \sigma_1 $	= 09	9 < 10									
Ĩ.	Mibe	•\ <i>0<sub>0,El</sub></i>	$\sigma_{1,Bd}$											
- 1	$\sigma_{w,k}$	ਮਾਰ. ਅ = 43	0.7 MP	a – Ma	ki <b>mum norn</b>	nal stress	transver	rse to the a	axis of the	weld				
- 1	$\sigma_w$	M = 43	5.6 MP	a – Equ	uvalent stre	ss resista	ance							
- 1		= -165	5.7 MPa	- Nor	mal stress	perpendi	cular to the	he throat		- 1				
	σ	w = 35	2.8 MP	a – Per	pendicular	stress res	istance							

This EC refactoring is in both Expert and Enhanced editions of IDEA StatiCa Steel.

### Range of EC validity

To increase the perceived safety of use, we have added information about the limitation and validity of the checks. These warnings are displayed at the top of the 3D scene.

#### Thin-walled members

The warning will tell you when the model stays out of the presumptions given by the Eurocode. This is to improve the safety of the use of application for thin-walled members.

Available in Connection since patch 22.1.5.



#### Anchor checks

The warning will show information about why anchors are failing. Anchor design codes are severely limited, and often we cannot perform anchor checks. Information about the limitation is given on top of the formulas.

Available in Connection since patch 22.1.1.



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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

Intuitive and fast modeling

### Working plane reference in LCS

The working plane manufacturing operation can be defined in reference to the local coordinates system of members and plates in the model. This will lead to a smoother workflow when modeling your connection.

In the past, defining the position and rotation of the working plane could be a little bit difficult. From now on, the origin of Working plane operation may be selected at the Member or Plate local coordinate system.

•	Work plane						
	Method	By angles 🔹					
•	Origin						
	Origin	Member 🔹					
	Member	Joint					
	X [mm]	Plate					
	Y [mm]	0					
	Z [mm]	0					
•	Rotation						
	Rotation Y [°]	0.0					
	Rotation Z [°]	0.0					



That is especially useful for, e.g., cuts of gusset plates according to members and all general cuts in the plane of model item.



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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Changes in the tree of entities

We like our UI to be well-organized and easy to understand, as you can see in the tree of entities presented in the 3D scene of Connection projects.

#### **Project settings**

For each project, you can set the project name, description, analysis type and also the field with a detailed description to be presented in the report. For the capacity design under AISC, additional parameters are also set here.

Transparent Wireframe	CON1							
USS	<ul> <li>Project item</li> </ul>	Project item						
	Name	CON1						
	Description							
×	Design code edition	AISC 360-16 (LRFD)	•					
CON1	Analysis type	Capacity design	•					
4 Members	Prequalified connection	×						
<u>✓ c</u>	System	Special moment frame (SMF)	¥					
🖌 В	Connection type	Reduced beam section (RBS)	•					
▲ Load effects	Report	You can add text and pictures	+					
LE-MC1								

#### Groups of the manufacturing operations

You are able to group the used manufacturing operations based on:

- Operation type
- Connected member

The function can be accessed using by right-mouse clicking Operations. There are three types of view.





The default (Sequential) refers to the original way of organization of the tree – the order is identical to the order of the addition of each operation.

Please note that the grouping of operations does not change the principle of the application – the operations are always organized by order of appearance.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### **Cross-section searchbox**

As the library of cross-section (MPLR) is getting bigger, we have employed a new tool to make searching the desired section easier.

Since previous versions, you can set the specific class of cross-sections as favorite.

Now, you can use the search box within the library to find the desired section fast and easily. The search always works under the selected tab, i.e., if you are in Favorites, the search will go through your favorite sections only.

Together, it will speed up your work with cross-sections management.

Cross-section ×								
Favorites All		W	×					
w	*	W200X22.5	<b>A</b>					
W(ARC)	*	W200X26.6						
W(Imp)	*	W200X31.3						
W (CISC TSS 9.2)	*	W200X35.9						
WWF (CISC TSS 9.2)	*	W200X41.7						
HW (China)	*	W200X46.1						
WB (IS 808)	*	W200X52						
W - imp (Singapore)	*	W200X59						
W - met (Singapore)	*	W200X71						
W (AISC 15.0)	*	W200X86						
		W200X100						
		W250X17.9						
		W250X22.3						
		W250X25.3						
		W250X28.4						
		W250X32.7						
		W250X38.5						
		W250X44.8						
		W250X49.1						
		W250X58						
		W250X67						
		W250X73	-					
		ОК	Cancel					
		UK	Cancer					

The feature has been possible to use in Connection, Member, and Checkbot since patch 22.1.5. Available in both **Expert** and **Enhanced** editions of IDEA StatiCa Steel.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Edge indexing in Member and Connection models

The indexing of plate edges means there are no discrepancies between Connection and Member data sharing and it prevents any model problems (e.g., missing welds).

The indexing of plate edges is synchronized for both untouched and shaped plates (via the plate Editor). New weld/contact operations added either in the Member app directly or in its Connection module now have the same numbering to prevent accidental mismatches and resolve bugs in the Member application.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### **Empty connections in Member**

In previous versions of the Member application, it was not possible to leave a connection (a node) empty without a design otherwise the calculation could not be started.

This problem is now solved with the release of version 23.0, enabling the Member to model, e.g., cantilevers or leave empty intermediate nodes.

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

#### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Load on surface warning in Member

A new warning message has been added in the Member app when applying a line load on a member surface.

When the line load is set to a member surface, such as the top flange of a beam with an I section, it is possible to input the load width. If the applied load area is wider than the plate, there is a warning displayed, and only the portion of the load that touches the surface is applied.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Supports with offset in Member

Supports for related members in the Member application models can now be shifted with eccentricity to provide models with higher precision.

By default, the related member end is supported in alignment with the **Node**. In case there is an **ey** or **ez offset** for the related member, the support stays aligned to the original position on the same level as the theoretical node.

To align the support with the end of the member, you can now switch to the **Member axis**, which deletes the eccentricities for the support location.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### **3D presentation: Scene options**

Draw results in 3s or 10s? The performance of 3D presentation in a graphical window highly depends on user display settings. The user is encouraged to test the optional settings for the computer. Correct settings may increase the speed of redrawing of 3D scene.

The 3D presentation depends on both, hardware and software. Unfortunately, it is not possible to suggest one setting for high performance and another for beautiful screenshots. Processor performance, graphics card performance, producer and drivers – they all affect the resulting performance of 3D window in IDEA StatiCa.

We recommend testing ideal settings for each personal computer. Pick a large model (you can find many of them in our Sample projects) and time how long it takes to switch, e.g., from stress plot to strain plot with various settings.

Note that 3D presentation settings are applied after starting a new project or opening an old one. You do not need to close the whole application.



#### Use hardware acceleration

Hardware acceleration, or GPU rendering, lets the application move all graphics and text rendering from the CPU to the GPU. For high-performance GPU, this should reduce latency and speed-up 3D scene.

When compatibility issues occur, please install the latest GPU drivers or turn hardware acceleration off.

From our testing, the hardware acceleration has a big impact on 3D scene performance. However, this impact is not always positive. It actually seems that on computers with with weak, integrated graphics cards, but also with powerful AMD graphics cards, the rendering is about 3x slower with hardware acceleration turned on.

#### Antialiasing

Antialiasing smooths the edges of inclined lines. Theoretically, antialiasing turned on would slow down performance. However, not a significant impact was measured on our computers, even with a high sample number.

#### Show shade

Again, theoretically, shade should increase computational effort but no high impact was measured on our computers.

#### **Use 1D textures**

1D textures are suitable for maximizing performance. However, some graphic cards may have troubles, causing the rendered pictures to be black. If that happens, change the Standard for 3D graphics. Moreover, the stress patterns may change slightly for using 1D textures or not on one computer and even for computers using a different graphic card (NVidia vs AMD). The stress peaks are not affected and shown stress heatmaps are averaged in nodes, so it does not need to be a big concern.



Figure: 1D textures activated on the left; deactivated on the right

#### Mode for incompatible graphic drivers

This mode should be activated if unwanted black rectangles appear in reports.

#### Standard for 3D graphics

Options are Direct3D and OpenGL. This setting should be changed, especially if rendered pictures are black when using 1D textures. The standard also affects the performance and optimal settings should be tested for a particular computer.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

**Speed and precision** 

#### Load Extreme Selection

This new feature of the 22.1.3 patch will help all users get the relevant result faster. There is no need to evaluate all of the load effects as most of them do not influence the final result. These depend on the particular extremes of loads only and now you are able to filter them easily. As designers of steel connections, you need to iterate to find the final design quickly. To do that, you don't have to evaluate the full set of load effects on the model. You only need to use the critical load effects to get this done.

There are different ways to define load effects in the IDEA StatiCa Connection model. For several load effects, the easiest way is to import the load via a BIM link or use the XLS import option.

You may also receive the load inputs in an Excel spreadsheet, which can contain hundreds of load combinations per connection. Since it is very ineffective to calculate that amount of load effects (time-wise) in IDEA StatiCa Connection, you need to select the decisive ones only, thus reducing the total number of load effects (and time!).

IDEA StatiCa Connection now has a useful function to filter the extreme load effects in an automated and easy way. But first...

#### What load is the extreme one?

The Load Extreme Selection evaluates the load components of **every connected member** in the connection and selects the most extreme ones based on the following criteria:

- Positive or negative maximum of each load component (N, Vy, Vz, Mx, My, Mz) 12x
- Maximum shear force resultant 1x
- Maximum bending moment resultant 1x
- Maximum positive and negative normal resultants 2x
- Maximum shear resultant 1x
- Maximum total resultant 1x



A theoretical maximum of 18 load extremes can be selected for each connected member. The total number of selected extremes is therefore *n* x 18, where *n* is the number of members in a connection. Nevertheless, the number of selected load sets can be lower as the extremes of members overlap.

Now, let's have a look at how you can filter extreme values.

#### Select the extremes and perform the analysis

To select the extremes to be analyzed, use the **right mouse button** on the Load effects and select the **Calculate load extremes** option.



This unselects the irrelevant (non-extreme) load effects from the list. The reason for each selected extreme load effect is presented in the description under the table.



#### **XLS import of extremes**

You can use the extremes selection also when importing the load values from an Excel sheet. Check the **Import only** extremes box. You are instantly shown how many load effects are considered extremes.

<u> = =   S</u>	tatiCa° (	CONNECTION	v					noc LEesDEce	k - Copy.ideaCo	on							-
culata yesterday	le estimates	Project	t De:	sign	Check	Report	Materials										Q
ST CD	DR FAT FIR Project items	HT New Copy	► Undo Redo Save Data	Members Plate	s LCS Ne	w Gallery	Propose Publish Ma	nage Code setup	Calculate CBFEM	Settings	ads - entage	(LS port Imp Import/Ex	ection > port Ex	LS Me	ember Loa	, 뉟 Operatio	n
· 🗟 (	∽ <b>+</b> . ~ Import of loa	d effects								/ X	LE1 [Lo	ad ]					Сор
[	Name	Member	Position	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]	Order of loads		Member	N (kN)	Vy [kN]	Vz N	lx My (Nm) [kN	Mz [m] [kNm]
	LE3	19	Begin	0.0	0.0	0.2	0.0	-0.1	0.0	19 - Begin	>	19 / Begin	0.0	0.0	0.0 0	0 0.0	0.0
		19	End	0.4	0.0	0.2	0.0	0.1	0.0	19 - End		19 / End	0.0	0.0	0.0 0	0.0	0.0
		41	Begin	0.1	0.0	0.0	0.0	0.0	0.0	41 - Begin 42 - Begin		41 / Begin	0.0	0.0	0.0 0	0.0	0.0
		42	Begin	0.7	0.0	0.0	0.0	0.0	0.0	in begin		42 / Begin	0.0	0.0	0.0 0	0 0.0	0.0
	LE4	19	Begin	0.0	0.0	0.0	0.0	0.0	0.0								
		19	End	0.3	0.0	0.0	0.0	0.0	0.0		Values i	in disabled	cells are r	not taken i	nto account	in CBFEM	
		41	Begin	-0.2	0.0	0.0	0.0	0.0	0.0		internal	s. Members I forces whi	can be lo ch are del	aded only 'ined in m	by that con ember "Moo	ponents of el type".	
		42	Begin	0.2	0.0	0.0	0.0	0.0	0.0								
	LES	19	Begin	0.0	0.0	0.0	0.0	0.0	0.0								
		19	End	0.9	0.0	0.0	0.0	0.0	0.0								
		41	Begin	-0.6	0.0	0.0	0.0	0.0	0.0	Total load effects: 258							
		42	Begin	0.6	0.0	0.0	0.0	0.0	0.0	Extreme load effects:							
	LE6	19	Begin	-5.1	0.0	0.0	0.0	-0.1	0.0								
		19	End	36.9	0.0	0.0	0.0	0.1	0.0	Import only extremes							
		41	Begin	-22.4	0.0	0.0	0.0	0.0	0.0	Replace existing	Unbal	anced fo	rces				
	1									▼ loads	X	Y	Z	Mx	My ml [kNm	Mz	
											[KN]	[KN]	[KN	IKN			

The extreme load effects are imported, and the rest are ignored.



#### Setting the tolerances

To neglect the irrelevant (small) values of extremes, you can set the limits/tolerances for each component of the internal forces. The force value (an absolute value) lower than the set limit will not be considered to evaluate extremes.



When you change these values to higher ones, a warning and the reasons for selecting the given load effect are presented.



You can save your custom values among the templates and use them in another project by loading it or setting the template as the default one.



#### Selecting extreme loads via Checkbot (BIM link)

There is also another possibility to filter the load extremes in case you are using the BIM link (IDEA StatiCa Checkbot). Open the **Load configurator** in Checkbot and go to **Result classes for checks** in the middle column. Select the desired result class and in the right field, specify whether the loads should be filtered. When the **Evaluate critical effects** checkbox is selected, the force extremes are filtered. (You need to call the **Load update** after closing the Load configurator window.)

Please note that the filtering algorithm differs from the above-mentioned procedure: Read more in the Evaluate critical effects for loads imported via BIM links article.

Calculars pactority's antifuced	a* Checkbot	Project Materials			Checkbot				_ <b>_ X</b>
Connections Member	Lodes Members Connections	Open Reset Calculate S	nc Delete Loads ymc Merge Divide	Settings Conversion	I Mx Results LC1LC1 ▼ Scale y My → Global J→ Local Extreme Local es z Mz	tre •			
List of project items	Labels	+	C 12 P	Options	Member ID Forces	iolid Transparent Wireframe	4 [Connection]		Open Load update Reset
✓ Connections ♦ 2 ♦ 4		Load configurat Imported items Load cases Combinations CO1 CO2 CO1011 CO1021	or Rec (	ult classes for checks AII ULS Fund CO1 CO2 CO1011 CO1021 CO1028	Properties - All ULS Fund     Result Class     Name All ULS for     Cookate collocal effects	×	Properties     Name     Node     Connected members     Connection point     Loads     Result class	4 4 2, 11, 12, 16, 18 [0.00; 1.40; 2.20] All ULS Fund	5
		C01028 C01032		C01692					
					Save	Cancel			
Support center Tutorials FAQ Webinars	News 26:10:2022 University Webinas: University of TT This webins will introduce the ress studies completed at the University verifications compared results from calculations used in practice in the the process as well as discuss the re-	nnessee Verifica its of a series of verification of Tennessee. These IDEA StatiCa to traditional USS, You will learn more about suits	21092022 How IDEA StatiCa works with your other In this webinar the process of linking dff StatiCa connection through Checkbot wi	software? - US 31.08.2022 erent software with IDEA In this well I be demonstrated. about bot	while in IDEA StatiCa (ASC) iner we will explain and answer the frequent questions is and welds in IDEA StatiCa, from modeling to design.	16082022 How to code-check a deep beam Deep beans are nowadays a com industrial buildings, or bridge str the beam theory or deep beam? V Navieror's hypothesis is still valid	non part of civil structures, uctures. When we can consider shat are the limits, where Bernor	ulli	
Design code: EN									RFEM -@-

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Smooth results with precise meshing

As a result of the continuous research of our solver team, we are always bringing a big improvement in mesh generation for the finite element model in the Connection and Member applications. The updated mesher now gives smoother and, therefore, more precise results. We use the mesher CM2 MeshTools SDK. Using the latest meshing algorithms, it provides a smooth mesh structure.





We have also programmed the newest rules for mesh generation to ensure better distribution in all plates. Connections composed of splice and backing plates, intermediate stiffening members, or stiffening plates used as custom end plates are tuned for even higher precision of analysis and match better with other parts of the connection.

## Version 22.1.5 (Splice vs backing plate) Version 23.0 (Splice vs backing plate)

Also, the default values for the mesh settings, such as the minimal size of elements and the number of elements on plates, are refined in the Code setup.



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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Fast calculation time and redrawing of results

We have improved the speed of the IDEA StatiCa Connection and Member apps on multiple levels. Both calculation time and redrawing of the results are now faster.

Our mathematical team has improved the performance of the analysis solver. According to internal tests, there is a 35% average decrease in solver run time (calculation time may vary depending on connection models and CPU configuration).

Redrawing results in the 3D scene is now faster as well. Benchmark tests on large projects in the Member app show a speed up by 20-25%. 3D scene refreshing was simplified as well, instead of clearing entities one by one, they are now cleared as a whole.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Base plate extended over concrete block

Since patch 22.1.1, there has been an improvement in the analysis of concrete blocks under steel base plates to deliver more precise results in less calculation time.

In previous versions of IDEA StatiCa Connection, the whole concrete block used for anchoring a base plate used to be meshed for the analysis, meaning increasing the calculation time, especially in models with larger concrete blocks and a higher number of load effects.

This improvement ensures that only the necessary area of the concrete block is meshed and the untouched remaining surface is ignored.

This leads to cutting the calculation time dramatically for some models while the same safe and precise results are still provided.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### **News for Concrete and Prestressing**

Our Concrete improvements are of a truly practical and international nature. And for those more demanding Concrete users of you out there, you will be excited to learn that we have implemented **automized calculation of equivalent time for long term deflection** in the Beam application. And what about those international updates?

Although metric is commonplace in Europe, the US and Middle East markets still use imperial. Previously, in the Detail application, when you applied new entities to the model that were in metric units but presented everything in imperial units, you got dimensions containing many decimal places. Not any longer. From 23.0 onwards, the dimensions can be **rounded according to your preferred unit system**.

It is now also possible to run a limited stress check to **ignore any irrelevant stress peaks for SLS checks** of concrete, which is fully in **compliance with Eurocode**. You get a 100% code check with a warning about neglected parts, while still remaining completely safe. This and the other enhancements to Detail mean it is an impressive **25% faster, saving you both time and money**!

Member also enjoys a serious mesh update in the form of **triangular mesh**. This brings the advantage of a much bigger mesh and the results being less mesh sensitive. Plus, due to the lower amount of finite elements, the **calculation is faster, the presentation of the results is faster and the application's response is better**, too.

#### The improvements in Concrete and Prestressing include:

### Limited stress check feature in Detail

Have you ever had an unsatisfactory SLS check due to a stress peak in concrete? We have come up with a new functionality, Limited Stress Check, that can solve this problem. Read the following article to learn how you can use the feature effectively and safely.

Stress peaks (singularities) can now be neglected from the stress limitation check! It was a big obstacle that has finally vanished thanks to the Limited Stress Check feature.

Before we explain this new option in detail, let's recall how the singularity, which can be neglected, and stress concentration, which cannot be neglected, differ. In the figure below, you can see a typical stress peak (singularity) in a sharp corner and a typical stress concentration around an opening.



It looks very similar, so how can you recognize the singularity from the stress concentration? When can you use the limited stress check? Well, the fact is that the decision is always up to the responsible engineer. We can offer you advice and name some typical singularities to help you.

- When there is a rapid change in stress in one element of the mesh there is a singularity.
- If the stress is high above the limit only in one node of the mesh there is a singularity.
- Only the very small areas of unsatisfactory stress should be neglected.
- Typical singularities **sharp corners**, the point of the mesh that is near to the anchorage or bearing plate and is not included in the partially loaded area.

Now, let's explain how the feature works. There is a wall with an opening with an unsatisfactory stress limitation check in the right sharp corner.



As you can see, the area is very small according to the whole structure. From version 23.0, there is a new option in the top ribbon that allows you to neglect the unsatisfactory areas.



The utilization is then 100%, and nonconformity appears.



You can also display the neglected areas.



All of these checks and images can be added to the report. So it will be shown that some areas were neglected, or how large these areas are. You can choose if you want to report original values or not, etc.

Available in Enhanced editions of IDEA StatiCa Concrete and Prestressing.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Implementation of long-term losses in Detail

Every prestressed structure has to be checked at least at the beginning and end of its service life. That is the reason why we added the option to set the stress in the prestressing reinforcement at the end of its working life. Read the following article to learn how to do it.

From version 23.0, you can set the **estimation for long-term losses** for pre-tensioned and post-tensioned tendons. It means that you won't have to complicatedly create special combinations with different prestressing coefficients to assess the structure at the beginning and end of its service life. A new entity in the properties of prestressing tendon called **Long-term losses for SLS check [%]** is now available, which allows you to check the short-term and long-term effects within one combination.



After calculation, you can switch between long-term and short-term to see the different results affected by the different material models as well as different initial stresses in the tendon.



How does it work? Read the following text to explore all the consequences of this new functionality.

For the calculation of the long-term effect (prestressing and permanent increment) in SLS checks, the values of stresses in the prestressing reinforcement are reduced by the defined value of the long-term losses. In other words, reduced stress by the estimation of long-term losses is used for prestressing and permanent increment where the  $E_{c,eff,press}$  and  $E_{c,eff,perm}$  are used. For a variable increment (and for short-term effects), where the  $E_{cm}$  is used, the reduction is no longer taken into account.



The value of long-term losses for SLS check [%], which should be set for pre-tensioned and post-tensioned tendons, differs. Here's why.

#### Pre-tensioned tendons

For pre-tensioned tendons, the input stress is the **stress just before the release from abutments** (stress after short-term losses due to anchorage set -> slip, deformation of abutments, short-term relaxation, etc.). The elastic strain in concrete is automatically calculated with the corresponding material model ( $E_{cm}$  is used for short-term effects, and  $E_{c,eff}$  is used for long-term effects). It follows that the long-term losses for the SLS check [%] is the value defining long-term losses caused by **shrinkage and long-term relaxation**. The default value is 10%.

As you can see in the figure below, there are now three types of stress.

- Input stress just before the release
- Stress after short-term losses
- Stress after long-term losses



Note that the self-weight is defined as a prestressing load type. The first increment (including self-weight) prestressing is shown.

#### Post-tensioned tendons

For post-tensioned tendons, the input stress is the **stress after short-term losses**, or you can set the **anchorage stress** and get the program to **calculate the short-term losses automatically**. Short-term losses, in this case, are – friction, immediate elastic strain in concrete, and anchorage set -> slip. It follows that the long-term losses for SLS check [%] is the value defining long-term losses caused by **creep, shrinkage, and long-term relaxation**. The default value is 10%.

As you can see in the figure below, there are now two types of stress.

- Input stress = Stress after short-term losses
- Stress after long-term losses



Note that the self-weight is defined as a prestressing load type. The first increment (including self-weight) prestressing is shown.

Available in Enhanced editions of IDEA StatiCa Prestressing.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Improvements for ACI 318-19 in Detail

Are you a structural engineer who works with ACI code? This article is for you because it describes how we made the application friendlier for you.

The application environment for users who work with ACI code was adapted to imperial units, materials defined by the code, detailing rules, code-checks, nomenclature, etc. These implemented improvements will make Detail more user-friendly and understandable for ACI users.

Let's go through the new features one by one!

#### New Templates for ACI code

A brand new set of templates for ACI code was created concerning the detailing rules and materials defined by ACI 318-19.

In a New project window, you can see three sets of new templates - Beams, Frame joints, and Walls.



Once you choose and select the template, the new model, including reinforcement, loads, and combinations, is created.



#### The proper grid for imperial units

The grid in the scene in the main graphical window is now presented correctly rounded in feet and inches if imperial units are set.



#### **Renamed load types**

Since the ACI code has different terminology, we redefined the load case types. **Dead** load and **Live** load replaced Permanent and Variable.



#### Anchorage type

Compared to the previous version, now, we allow defining reinforcement without any modification at the beginning (end), such as a hook, perfect bond, etc.



#### The new set of materials defined in ACI 318-19

ACI 318-19 defines materials of normal-weight concrete and reinforcement (non-prestressed deformed bars), and they are now implemented in the Detail app. The materials are defined in imperial units as well as in metric units. All material properties (for concrete and steel reinforcement) defined by ACI 318-19 (and relative ASTM docs) are specified below.

#### Materials defined by ACI 318-19

Legend

given value
calculated value

#### **Concrete (normalweight concrete)**

#### **Concrete (imperial)**

	Cylinder strength f´ <sub>c</sub> = f <sub>ck</sub> [psi]	Cube strength f <sub>ck</sub> [psi]	Tensile strength f <sub>ct</sub> [psi]	Young´s modulus E <sub>c</sub> [ksi]	ε <sub>c3</sub> [-]
3000	3000	3750	367	3122	0.002
4000	4000	5000	424	3605	0.002
5000	5000	6250	474	4031	0.002
6000	6000	7500	519	4415	0.002
8000	8000	10000	599	5098	0.002
10000	10000	12500	670	5700	0.002

#### **Steel reinforcement (nonprestressed deformend bars)**

#### **Steel reinforcement (imperial)**

	Minimum yield strength f <sub>y</sub> [ksi]	Min. tensile strength f <sub>t</sub> [ksi]	Young´s modulus E <sub>c</sub> [ksi]	ε <sub>y</sub> [-]	ε <sub>t</sub> [-]
Grade 40	40	60	29000	0.00138	0.0500
Grade 60	60	80	29000	0.00207	0.0500
Grade 75	75	100	29000	0.00259	0.0500
Grade 80	80	100	29000	0.00276	0.0500
Grade 100	100	115	29000	0.00345	0.0500

#### **Reinforcement Profiles (imperial)**

Bar size	Nominal Diameter [in]	Nominal Area [in <sup>2</sup> ]	Weight [lb/ft]
#3	0.375	0.110	0.376
#4	0.500	0.200	0.688
#5	0.625	0.310	1.043
#6	0.750	0.440	1.502
#7	0.875	0.600	2.044
#8	1.000	0.790	2.670
<b>#9</b>	1.128	1.000	3.400
#10	1.270	1.270	4.303
#11	1.140	1.560	5.313
#14	1.693	2.250	7.650
#18	2.257	4.000	13.600

Each imperial bar diameter increases by 1/8 inch. You can multiply the bar size by 1/8 to get the nominal diameter in inches. For example, #8 rebar = 8/8 inches (or 1 inch) in diameter. Available in Enhanced editions of IDEA StatiCa Concrete.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Imperial rounding improvements in Detail

A brand new feature that allows you to define preferred application units (imperial or metric) for new entities like geometry, cross-sections, loads, etc. All of the values of new entities will be now nicely rounded according to the selected application units.

We distinguish two types (groups) of units:

- Units for presentation purposes
- Units for the applying of new entities into the model

These groups are independent and can be arbitrarily combined (for example, metric units for presentation but imperial units for application of the new entities).

For this purpose, the new feature that defined the type of preferred dimension for new entities was implemented. When defining a new project, the user can set if the dimension of all entities (geometry, load, reinforcement) input into the model will be rounded for metric or imperial units.

New project				×
Design code Concrete Reinforcement Concrete cover	ACI	Name Author Description Model Type	2D •	
New entities Templates —	Imperial Metric Imperial			General input —
<u>a</u> 0				
Beams	Frame joints	Wa	lls	General shape
				Cancel

When you decide to change the setting of application units in an existing project, it can be done in Settings.

Calculate yesterday'a estimate	a* DETAIL							
Image: Save         Image: Save	A Names	New Gallery Settings	For extreme to the second seco	eme  Stres flow	Reactions and loads			
Project Data	Drawing setting	s Pictures Calcul	ation		Results			
Navigator	▼ 4 Code	and calculation settings						
Current item		lew entities and operations		•	^			
DRIVIT		Preferred dimensions for new ent	tities	Imperial	-			
▲ Discontinuity Region	A	nalysis and checks						
Geometry	S	Stop at limit strain/slip						
Reinforcement		rregular increments						
Check		Number of increments (permanent part)						
Bill of material	N	Number of increments (variable part)						
Project summary	Т	opology optimization						
— Discontinuity Regi	ons	Calculated volume levels		4				
- Cross-sections	N	Minimal effective volume [-]						
Materials & model	S N	lesh settings						
Project data								
Report Preview/Pri	Expan	d Collapse	Restore	OK	Cancel			

Let's also recall where you can set the presentation units. Go to File -> Units and make changes.

Ш	<b></b> St	🔜 Units setup					×
Calc	uleta yaetarday'a	Main		Unit type 🍸 🔨	Unit	Precision	Format
T	New 🗎	Material	>	Length - Structure	in •	2	Decimal 🔹
File	📄 📄 Open	Results		Length - Cross-section	in •	3	Decimal •
	Save			Angle	• •	1	Decimal 🔹
	New			Force	kip •	3	Decimal 🔹
	Open			Moment	kip.in 🔹	2	Decimal 🔹
	Save			Stress	ksi 🔹	1	Decimal 🔹
	Save as			Coefficient	No symb 🔹	2	Decimal 🔹
				Curvature	1/in •	5	Decimal 🔹
	About			Time (short-term)	s •	0	Decimal 🔹
	Units						
	Preferences						
	Licences						
1	Close						
•]	Exit	Default - metric	)efau	lt - imperial Import Export		OK	Cancel

For example, an I-shaped cross-section is presented with a different combination of application and presentation units.



Available in Enhanced editions of IDEA StatiCa Concrete and Prestressing.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### **Realistic reinforcement layouts in Detail**

Do you want to have nicer images of stirrups reinforcement in your report? You can have them now because we improved the 3D realistic layout. Hence, it is not just about images. The option of considering the stirrups' diameter in cover calculation has also been added. **Concrete cover** improvement (for ACI and EN) – when the beam element is modelled, the concrete cover for

longitudinal reinforcement can be calculated considering the biggest stirrups' diameter. The reinforcement layout is also closer to reality, and longitudinal rebars do not collide with stirrups.



This option can be turned ON or OFF in Settings.



Available in Enhanced editions of IDEA StatiCa Concrete and Prestressing.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Equivalent time for deflection evaluation

To determine the non-linear long-term deflection of staged elements, the time for the application of long-term load has to be defined. Since version 22.1.3, it is possible to calculate effective time automatically using new functionality in the Beam application.

The calculation is based on a linear deflection from permanent loads (or prestressing) applied in construction stages. The calculation will be performed after clicking the Auto button (the value can still be defined manually).



The effective time *d* is calculated according to the formula:

$$d = rac{\sum\limits_{n=1}^{i} t_i \cdot \, u_{st,i}}{\sum\limits_{n=1}^{i} u_{st,i}}$$

#### where

 $t_i$  time when the load (or prestressing) for each phase is applied

*u*<sub>st,i</sub> the absolute value of the maximum short-term linear deflection from the applied load (or prestressing)

The particular times used when the load is applied can be found in the construction stage tab, whereby the Superimposed dead load is the last phase taken into account. The used value of the linear deflection has been calculated by the software. For the construction stage in which we apply prestressing, we consider both values in the calculation - from the permanent load and from prestressing.

Available in both Expert and Enhanced editions of IDEA StatiCa Concrete and Prestressing.

### Triangular mesh in concrete Member

To get more accurate results, we have implemented triangular meshing for the Member application. Read more about this meshing.

A **triangular mesh** for cross-sections has been implemented in the Member application for concrete members. The rectangular mesh that was used previously was sufficient for members in bending only if the very fine mesh was defined. The reason is that the results are calculated in the integration points located in the center of gravity of each finite element, so we don't calculate the actual stress at the edge of the cross-section. Results are then very sensitive to mesh size, especially for members in bending, for which the compressive zone in concrete could be just a few centimeters high. Hence, many finite elements cause a slowing down of the calculation as well as the presentation of the results in the 3D scene.

All these problems have been solved by implementing the **triangular mesh** for sections calculated by the MNA and GMNIA analysis. Integration points are located in nodes (finite element vertexes), so we get results at the edge of the cross-section to capture stress peaks in concrete due to bending. This brings many advantages to users. **The results are not sensitive to mesh size**, so by default, the mesh is rough (in comparison with rectangular mesh), leading to **decreasing the calculation time and faster application response** when working with results along with **more precise and safer results**.



Available in the Enhanced edition of IDEA StatiCa Concrete

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Minor improvements

- Faster calculation by up to 25 % in Detail (again!)
- Transition length added to SLS for pre-tensioned tendons in Detail

• GMNIA analysis in Member now includes the code-checking of shear and torsion

### **News for BIM links**

BIM interconnectivity of IDEA StatiCa applications with the third-party software has been here for some time. The coverage of the brands and products is really wide, with thirteen in-the-house solutions and several more developed on side of our partners. So, the focus for version 23.0 was not about the number of links, but about the **quality and reliability**. What has been changed is the way in which we read and use data from other applications. This means our users shouldn't experience any surprises in special cases and workflows.

Our new BimApi solution is continuously spread within the BIM portfolio, and the Load mapping algorithm has got to the new generation of connection recognition.

### BIM link with RFEM 6 and RSTAB 9

Since patch 22.1.4, IDEA StatiCa Checkbot is compatible with new versions of Dlubal RFEM 6 and RSTAB 9. The BIM link provides the usual data transfer between the FEA applications and IDEA StatiCa with some exceptions.

The list of supported third-party applications was once more extended in patch 22.1.4, enabling RFEM 6 and RSTAB 9 to be a part of the standard Checkbot workflows within the IDEA StatiCa software package.

#### How to activate the BIM link with RFEM 6 and RSTAB 9

Download and install the latest version of IDEA StatiCa. Open IDEA StatiCa and navigate to the **BIM** tab and open the **BIM link installer** (Activate your BIM link...).



A notification "Do you want to allow this app to make changes to your device?" may appear. If so, please confirm with the **Yes** button.

The BIM link for **RFEM 6/RSTAB 9** is installed. The screen also tells you the status of other BIM links that are installed or may be installed later.

	<b>=</b> StatiCa <sup>®</sup> Calculate yesterday's el	stimates	×
	RFEM 5	Installed	>
<b>Z</b>	RSTAB 8	Installed	
	RFEM 6	Installed	
4	RSTAB 9	Target application not found	
R	Robot Structural Analysis	Installed	
<u>s</u>	SAP2000 v24	Please install our add-in for your SAP2000 version 24	
<u>s</u>	SAP2000 v23	Please install our add-in for your SAP2000 version 23	
4	SCIA Engineer	Installed	
	STAAD.Pro CONNECT Edition	Target application not found	
4	ConSteel	Please install our add-in for your ConSteel	~
-			-

Open RFEM 6/RSTAB 9 and navigate to the menu **Options** and **Program options**. Here, scroll down and turn on the **WebService** feature to enable the data transfer of your RFEM 6/RSTAB 9 and other applications.

Dlubal RFEM   6.02.0050   New model.rf6*							- 0	×
File Edit View Insert Assign Calcul	ate Results Tools Options Window CAD-BIM Help	p	<b>.</b>	101 . @ 101 III	8 MI - S 104 - 1 (X 111 - 1		License 06   Adam Kožoušek   IDEA StatiCa s.r.o. =	6º X
						1 17 1 + X + Y + Z - X + Y .		
Nucleoter Data	Model Tolerances	1.4.1.11 (A ≤ (B ± ≤ )		I - Gibbar Xiz	* <del></del>	·····································		_
Navigator - Data B' X	A second second						~	
> P IDEA-REEM EN.ITS	🤤 Import GUI Settings	Regarder Options & Settions				×		
✓	Sport GUI Settings	a riogium opuuna di settinga			0	~		
✓	Program Options	Options Settings						
> 😜 Materials	😤 Customize Menus and Toolbars	Options						
Thicknesses		Work Plane and Grid						
Nodes		Use identical grid for all windo	ws					i .
Lines		Y Connect Lines/Members						i .
T Members		Copy properties to created line	s/members			in the second		l i
Openings	la l	Auto connect anes/members		•				i .
Solids		Y Display						i .
Line Sets		Information about position of Small pumbers in exponential f	structure objects					ļ
A Member Sets		Full section description	on the second seco					
Solid Sets		Full material description						
> i Special Objects	$\leq$	Warning message before delet	ing mesh or results	•				
> Types for Nodes		Display whole model in 3D pres	/ew	0				
> Types for Lines		Y Swine Ontions						
> Types for Surfaces		Automatic save to backup copy		0				
> 🦲 Types for Solids	~~_ <i>/</i>	Save after calculation						
> Dypes for Special Objects		Y Miscellaneous						1
> Imperfections	$\sim$	Load last opened models when	the program restarts					
<ul> <li>Load Cases</li> </ul>		Delete useless nodes during in Delete useless nodes during in	iport					i .
> 🔁 Actions		Validate objects when selecting	in graphics					i .
Design Situations		Calculate object information fr	om generated mesh					i .
Action Combinations		Automatic generation of repres	ientatives	•				i .
Static Analysis Settings		Y Diagnostic data						i .
> 💭 Combination Wizards		Send feedback & diagnostic da	ita to Diubal Software					i .
Relationship Between Load Cases		✓ WebService						i .
> Load Wizards		Start the server automatically w	ith the application	•				i .
> LC1 - Self-weight						_		
Calculation Diagrams		0						
> Results					Ok Cancel Ap	ev		i .
Guide Objects								
	ž							
	Materials							8 ×
	Go To Edit Selection View Settings							
	🔝 Structure 🗸 🔸 🏒 Basic Obj	jects 🗸 🔹 🕄 🧲 🔛	in 🖻 🗏 🔳 🐹 💼	s 🖮 🗣 🏓 🖂 🞜	fx fŽ fŽ 🗐 🗂 🐻 J	9		
	Material	Material	Modulus of Elast. Shear M	Iodulus Poisson's Ratio	Specific Weight Mass Density	Coeff. of Th. Exp.		
	No. Material Name	Type Material Model	E [N/mm <sup>2</sup> ] G [N/m	mm²] v []	γ [kN/m <sup>2</sup> ] ρ [kg/m <sup>2</sup> ]	a [1/*C] Option	s Comment	
	1 \$235	Steel Isotropic   Linear Elastic	210000.0	80769.2 0.300	78.50 7850.00	0.000012		-11
	3							-11
	4							- 1
	6							
	7							
S .	H 4 1 of 13 + H Materials Sections Thicknesses	Nodes Lines Members Surfaces Openings	Solids Line Sets Member	r Sets Surface Sets Solid	Sets			
Edits various program options				SNAP GRID LGR	ID OSNAP	CS: Global XIZ	Plane: XY	

#### How to use Checkbot with RFEM 6 and RSTAB 9

Open an RFEM 6/RSTAB 9 project and run the calculation. To run the **Checkbot**, open the shortcut icon **IdeaRFEMLink/IdeaRSTABLink**, by default, located on your desktop (original file in path C:\Program Files\IDEA StatiCa\StatiCa 22.1\net6.0\IdeaRFEMLink.exe).



Checkbot for the current RFEM 6/RSTAB 9 project opens, and you can create a new project and start importing connections and members.

#### Limitations

- Missing a command directly in the application RFEM 6/RSTAB 9 instead of a desktop shortcut icon. Waiting for Dlubal development to add the commands in their menu/interface.
- Result combinations are not supported.

Waiting for Dlubal development to provide the necessary data provided in the API.



 Eccentricities specified relatively in RFEM 6/RSTAB 9 are not supported. Only absolute eccentricities can be imported.

🔗 New Member Eccentricity			
List 1 Absolute   Local xyz	No. Name Absolute   Local xyz		<u>N</u>
	Eccentricity Type Absolute Relative to Section Absolute Relative and Absolute	Absolute Ordinates Coordinate system Local xyz ~ ex 0.0 + [mm] ey 0.0 + [mm] ez 0.0 + [mm]	1

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### Unified BimApi solution – Advance Steel, SAP2000, ETABS

The unified BimApi solution for Advance Steel, SAP2000, and ETABS is available. We have developed a unified BimApi solution for faster integration and better cross-section recognition.

This affects other third-party software and covers, namely, Advance Steel, SAP2000, and ETABS.

It can also deal with eccentricities, concrete cross-sections and materials.

Note: Older projects can be opened but not updated (the member links are broken)

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

### Download and try features of IDEA StatiCa 23.0

DOWNLOAD IDEA STATICA 23.0 ightarrow

### Load mapping algorithm for importing steel connections

We developed an algorithm for recognizing members' geometry for load imports. This tool assigns imported load effects according to the real connection space arrangement. The algorithm is based

on topological code, which is independent from LCS or the order of the members' selection in a joint.

This helps mostly in cases with this typical workflow. The connection designer wants to load the structural model of a connection from a detailer (exported from CAD) using loads in a model provided by an engineer (exported from FEA). This workflow saves a lot of time and provides effective use of the data exchange between offices and applications.

#### **Fewer limitations:**

- LCSs of the current model and the model being imported don't have to be identical
- The order in the project member list doesn't need to be the same

The algorithm automatically recognizes and identifies the corresponding members and reliably assigns the load effects to them.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### **Load Extreme Selection**

This new feature of the 22.1.3 patch will help all users get the relevant result faster. There is no need to evaluate all of the load effects as most of them do not influence the final result. These depend on the particular extremes of loads only and now you are able to filter them easily. As designers of steel connections, you need to iterate to find the final design quickly. To do that, you don't have to evaluate the full set of load effects on the model. You only need to use the critical load effects to get this done.

There are different ways to define load effects in the IDEA StatiCa Connection model. For several load effects, the easiest way is to import the load via a BIM link or use the XLS import option.

You may also receive the load inputs in an Excel spreadsheet, which can contain hundreds of load combinations per connection. Since it is very ineffective to calculate that amount of load effects (time-wise) in IDEA StatiCa Connection, you need to select the decisive ones only, thus reducing the total number of load effects (and time!).

IDEA StatiCa Connection now has a useful function to filter the extreme load effects in an automated and easy way. But first...

#### What load is the extreme one?

The Load Extreme Selection evaluates the load components of **every connected member** in the connection and selects the most extreme ones based on the following criteria:

- Positive or negative maximum of each load component (N, Vy, Vz, Mx, My, Mz) 12x
- Maximum shear force resultant 1x
- Maximum bending moment resultant 1x

- Maximum positive and negative normal resultants 2x
- Maximum shear resultant 1x
- Maximum total resultant 1x



Shear force and Bending moment resultants (2x)







A theoretical maximum of 18 load extremes can be selected for each connected member. The total number of selected extremes is therefore *n* x 18, where *n* is the number of members in a connection. Nevertheless, the number of selected load sets can be lower as the extremes of members overlap.

Now, let's have a look at how you can filter extreme values.

#### Select the extremes and perform the analysis

To select the extremes to be analyzed, use the **right mouse button** on the Load effects and select the **Calculate load extremes** option.



This unselects the irrelevant (non-extreme) load effects from the list. The reason for each selected extreme load effect is presented in the description under the table.



#### **XLS import of extremes**

You can use the extremes selection also when importing the load values from an Excel sheet. Check the **Import only** extremes box. You are instantly shown how many load effects are considered extremes.

<b>==) S</b> t	tatiCa° (	CONNECTION	v				"	noc LEesDEce	k - Copy.ideaCo	n								
aca yestanaay-s	e excimates	Project	t De	sign	Check	Report	Materials											Q
ST CD D	DR FAT FIR	HT New Copy	Redo	Members Plate		Gallery	Propose Publish Man	hage Code setup	Calculate Overall check	Settings	ids - Intage	KLS Ioni iport In	nection in port	XLS P	Aember I	.oad Op	eration	
ি বি	Project items	R 7	Data	Labels		ictures	Connection Brows	21	- ···		LE1 [ L	ad ]	cport load	15		New		Сору [
Ir	mport of load	d effects								× .								
	Name	Member	Position	N [kN]	Vy [kN]	Vz [kN]	Mx [kNm]	My [kNm]	Mz [kNm]	Order of loads		Member	N (kN)	Vy (kN)	Vz (kN)	Mx [kNm]	My [kNm]	Mz (kNm)
	LE3	19	Begin	0.0	0.0	0.2	0.0	-0.1	0.0	19 - Begin	>	19 / Begi	n 0.0	0.0	0.0	0.0	0.0	0.0
		19	End	0.4	0.0	0.2	0.0	0.1	0.0	19 - End		19 / End	0.0	0.0	0.0	0.0	0.0	0.0
		41	Begin	0.1	0.0	0.0	0.0	0.0	0.0	41 - Begin 42 - Begin		41 / Begi	n 0.0	0.0	0.0	0.0	0.0	0.0
		42	Begin	0.7	0.0	0.0	0.0	0.0	0.0	in begin		42 / Begi	n 0.0	0.0	0.0	0.0	0.0	0.0
	LE4	19	Begin	0.0	0.0	0.0	0.0	0.0	0.0									
		19	End	0.3	0.0	0.0	0.0	0.0	0.0		Values	in disabled	cells are	not taker	into acco	unt in CB	FEM	
		41	Begin	-0.2	0.0	0.0	0.0	0.0	0.0		analysi interna	s. Member I forces wh	s can be lo ich are de	baded on fined in r	ly by that nember "I	compone vlodel typ	nts of e".	
		42	Begin	0.2	0.0	0.0	0.0	0.0	0.0									
	LES	19	Begin	0.0	0.0	0.0	0.0	0.0	0.0									
		19	End	0.9	0.0	0.0	0.0	0.0	0.0									
		41	Begin	-0.6	0.0	0.0	0.0	0.0	0.0	Total load effects: 258								
		42	Begin	0.6	0.0	0.0	0.0	0.0	0.0	Extreme load effects:								
	LE6	19	Begin	-5.1	0.0	0.0	0.0	-0.1	0.0	¥								
		19	End	36.9	0.0	0.0	0.0	0.1	0.0	Import only extremes								
		41	Begin	-22.4	0.0	0.0	0.0	0.0	0.0	Replace existing	Unba	lanced f	orces					
										- Ioads	X	Y	Z	M	K M	y M	/z	
									ОК	Cancel	0.0	0.0	0.0	0.0	0. 0.	) (	.0	

The extreme load effects are imported, and the rest are ignored.



#### Setting the tolerances

To neglect the irrelevant (small) values of extremes, you can set the limits/tolerances for each component of the internal forces. The force value (an absolute value) lower than the set limit will not be considered to evaluate extremes.



When you change these values to higher ones, a warning and the reasons for selecting the given load effect are presented.



You can save your custom values among the templates and use them in another project by loading it or setting the template as the default one.



#### Selecting extreme loads via Checkbot (BIM link)

There is also another possibility to filter the load extremes in case you are using the BIM link (IDEA StatiCa Checkbot). Open the **Load configurator** in Checkbot and go to **Result classes for checks** in the middle column. Select the desired result class and in the right field, specify whether the loads should be filtered. When the **Evaluate critical effects** checkbox is selected, the force extremes are filtered. (You need to call the **Load update** after closing the Load configurator window.)

Please note that the filtering algorithm differs from the above-mentioned procedure: Read more in the Evaluate critical effects for loads imported via BIM links article.

Calculates particular de satisfactor	a* Checkbot	Project Materials			Ch	eckbot				_ <b>_ X</b>
Connections Member	Vodes Members Connections LCS	Open Reset Calculate S	mc Delete Loads ymc Merge Divide Structural model	Settings Conversion Dre	D N Mx R Vy My Vz Mz	esuits LC 1 LC1 V Scale J= Global J= Local Extreme Local extr Member 1D Forces	1.00 Connection Member			
List of project items		Load configurat Imported items Load cases Combinations C01 C02	C II -	It classes for checks NULS Fund CO1 CO2 CO1011	Ð	roperties - All ULS Fund ▼ Result Class Name All ULS Fund Evaluate ontical effects ♂	lid Transparent Weeframe	4 (Connection)      Properties Name Node Connection point Connection point Consection point Result class	4 4 2, 11, 12, 16, 18 [0.00; 1.40; 2.20] All ULS Fund	Open landupdate Reset
		C01011 C01021 C01028 C01032		CD1021 CD1028 CD1032						
						Save	Cancel			
Support center	Mear		1							
Tutorials FAQ Webinars	26:50202 University Webinas: University of Tr This webinas will introduce the res- version for a constraint of the results for version for the results for version of the results for version of the results for version of the results for version of the results for version of the results for version of the results for version of the results for version of the results for version of the results for version of the results for v	nnessee Verifica its of a series of verification of Tennessee. These DEA Static a to traditional US. You will learn more about suits	2109.2022 How IDEA StatiCa works with your other In this webmar the process of Inning diff StatiCa connection through Checkbot wil	31.00 software? - US Wel erent software with IDEA In th be demonstrated. abo	063022 elds & Bolts in IDE this webiner we w out bolts and wel	A Stablice (AISC) off explain and answer the frequent questions of an IDEA StatiCe, from modeling to design.	16/08/2022 How to code-check a deep beam Deep beams are nowaday a com- industrial buildings, or bridge str industrial buildings, or bridge str inthe beam theory or deep beam? V Navierov's hypothesis is still valid	non part of chil structures, uctures. When we can conside that are the limits, where Berno	suffi	
Design code: EN										RFEM -

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

#### Setting up parametric design with Developer Tab

Many users prefer modeling and analyzing connections through parameters and commands instead of by manual input. It is easy to deal with parametric design in IDEA StatiCa, giving users greater freedom to explore additional functionality.

It is not necessary to manually modify the IdeaConnection.exe.config file in the root folder, which would be timeconsuming and inconvenient.

Thanks to the raising numbers of advanced users of IDEA tools, it is enough to just enable the Developer mode in **Project tab/Preferences**.

	Application p	eferences	
		StatiCa* Customers logo Default logo	
n			
	( ) Ca	lors	
as	Cole		
mation	Con	Item	Color
ut	>	Background - top	
ts		Background - bottom	<b>•</b>
		Members	<b>•</b>
erences		Plates	
ices		Bolts	<b>•</b>
		Welds	<b>•</b>
		Concrete	
		Timber	<b>•</b>
		Current	•
		Supports	•
		Load	•

This brings the user a new tab in the main toolbar, with additional features.

//=/=== StatiCa* CONNECTION											
Calculata yastarday'a astimatas		Project	Design	Check	Report	Materials	Developer				
CON1 - Kan Copy	🖍 Undo 🎮 Redo 🔛 Save	Model Conn Data	Save Apply Simple								
Project items	Data	IOM	Templates	J							
🖞 🖓 🖓	C S	1									

#### **Export IOM:**

- Model export the model data through IOM (Idea Open Model) in .xml format
- Conn Data export the model data through IOM in .json format

#### Template:

- Save as the user can save the connection model properties in the Idea Template (.IdeaTemp) format
- Apply the user can apply Idea Template format to the whole Connection model (all members are involved)
- Simple the user can apply Idea Template format to a selected part of the Connection model (only selected members are involved)

There is also a property grid window part-displayed with two separate tables:



**Parameters table** – this provides the user with the set of IOM parameters that are available to use for parametric design

**Model parameters** – this allows joining the specific IOM parameter with the specific Model parameter of the current Connection project

For more information about these features, please visit our GitHub page

https://github.com/idea-statica/ideastatica-public/wiki/Developer-Mode-Parameters#parameter-input https://github.com/idea-statica/ideastatica-public/wiki/Reference-Guide-Expression-Parameters

This feature can be found in the applications **Connection** and **Member**.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

If you want to check the compatibility with your particular application, just take a look at our list of actively supported versions in 23.0.

### **Cloud & Licensing & All applications**

The biggest news in the Cloud is that **Web Connection Browser has been renamed Connection Library**. This was the decision after a lot of market research into how we can make the library more user-friendly, intuitive and optimize its workflow.

**User Portal has also been improved** to report the product version, country analytics, the reporting of license clashes and the licensing system has increased stability, **eliminating any related support cases**.

### **Connection Browser renamed Connection Library**

The clever design template feature called Connection Browser is renamed Connection Library. This is due to further development of this feature and similar web-based services for a broader community of engineers.

The **Connection Library** functionality will not be touched by this change, it still uses the Cloud-based technology and provides design templates to be **directly used** for new projects, **saved** in the database from current projects, and **shared** with your colleagues.

The design sets will be slightly renamed as well following this pattern:

- Predefined (before IDEA StatiCa) the default design set maintained by IDEA StatiCa, not possible to manage.
- Personal (before Private) a design set visible only to the current user, possible to add/delete/manage templates.
- **Company** (before Company) a design set visible only to all users of the same license, possible to add/delete/manage templates.

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

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### User Portal – last time active, version tracking

We continue to improve and develop the User Portal to make it more applicable for users, especially in the area of its monitoring and reporting capabilities.

One of the main purposes of the User Portal is to see clearly where and when potential issues with the license are happening on the user side. Also, to provide users and admins **more control over the license and usage of the application**.

You can find the following data in the section License & Users in the User Portal:

All users, including admins

- Last time active
- User setting of the tickbox Keep products reserved
- Used version

#### Admins only

• More information about **usages and clashes** – An admin is able to see the details of license clashes, especially the daily instances of clashes and also the users who were colliding.

We have also implemented measures (such as the proper release of licenses) to reduce clashing and licensing issues.

Available for all editions of IDEA StatiCa.

### Download the new version of IDEA StatiCa and try all the features

DOWNLOAD NEW VERSION ightarrow

### IDEA StatiCa Basic plan

The option to sign up for an IDEA StatiCa account without a license

A valid license is not required to create an account and to sign into **IDEA StatiCa User Portal**. This allows all subscribers to take advantage of the following:

- Connection Lite
- Viewer

- Viewer plugins
- Connection Library
- Campus E-learning

One of the main advantages of this option is to speed up the communication between interested users and IDEA StatiCa without the need for requesting a license when using only the Cloud tools and E-learning.

### **Solved incidents**

See the current list of solved incidents reported by our customers.