IDEA StatiCa tutorial – Revit link

Welcome to IDEA StatiCa tutorial. This tutorial will show how to activate and use the link between Revit and IDEA StatiCa Connection.



Activate the link

Use the link

Known limitations

1 How to activate the link

We have to follow these steps:

- Install Revit first make sure you are using a supported version of Revit updates are published in the <u>BIM section</u>
- Install Robot Structural Analysis
- Install Structural Analysis Toolkit for Autodesk Revit (2017 2018)
- Install the Steel Cross-section database

IDEA StatiCa automatically integrates the BIM link into your CAD/CAE software during its installation. You can check the status and activate more BIM links for later installed software in the BIM link installer.

Open IDEA StatiCa and navigate to the panel **BIM** and open the **BIM** link installer. A notification "*Run as administrator*" may appear, please confirm with the **Yes** button.



Select the software to integrate the IDEA StatiCa BIM link into, click the Install button and check the Installed status.

	∃StatiCa° Calculate yesterday's es	timates
G	Advance Design 2018	Install
-	Tekla 2017i	Target application not found
-	Tekla 2016i	Target application not found
A	Advance steel 2018	Install
A	Advance steel 2017	Target application not found
	ETABS 2016	Please add our Add-in into your ETABS 2016
X	SAP 2000 v 20	Install
X	SAP 2000 v 19	Install
X	SAP 2000 v 18	Install
<u>X4</u>	Axis VM	Install
R	Robot Structural Analysis	Install
R	Revit 2017	Install
R	Revit 2018	Install
see.	SCIA Engineer	Install
	STAAD.Pro v8i	Installed
	RFEM 5	Install
	RSTAB 8	Install

2 How to use the link

Open the attached project in Revit.



https://resources.ideastatica.com/Content/02_Steel/Tutorials/BIM/Revit/Revit_link.htm

We have to select the right node and command **IDEA StatiCa Connection** command from the ribbon.



The import wizard automatically appears. We proceed with the **Next** command button.

	® CONNECTION		
0000000 90000 009 0 000000000			
Design code:			
EN			
Select project file:			
D:\SAS_SVN\01 - Steel\Tutorials\v9\+-BI	M-+\Revit\EN\Model_Revit_tutorial[Noc	leMiddle 04].ld	Browse
Overwrite existing project, all additional data	will be deleted.		
IDEA Open Model processing 0%			
		Next >	Cancel

We continue with the **Next** command button.

Calculate yeste	StatiCa [®] CONNECTION
Connection design need define them in this wiza	ls more data to be able to provide a proper design according to national codes. You can use default settings or rd.
Design code:	EN
Type of structure:	General structure
Default setting:	M534630
All load combinations ar	re used for the design.
Load combinations are s	sorted into classes ULS, SLS etc.
2-Node N2	
Connected members:	M534700
Cross-section > M534618 (W(Imp)10X) M534630 (W(Imp)10X) M534630 (W(Imp)10X) M534700 (W(Imp)10X) M534700 (W(Imp)10X)	Role Type 33: Image: Searing in the searing in the searing in the searing in the search in the sear
	M534618
	Connection design < Previous Next > Cancel

We can see imported Load Groups. Let's continue with the **Next** command button.

Calculate yesterday's est	iCa [®] Load Groups imates	
Load Groups	T Last Gran Departies	
Not grouped cases	Load Group Properties	
▲ Dead Loads	Name	Not grouped cases
DL1		
▲ Live Loads		
LL1		
▲ Wind Loads		
WIND1		
▲ Snow Loads		
SNOW1		
▲ Roof Live Loads		
LR1		
▲ Accidental Loads		
ACC1		
 Temperature Loads 		
TEMP1		
▲ Seismic Loads		
SEIS1		
Expand all items	When extreme values in th (a) permanent group are a (b) standard group are ad negative extreme if they a (c) exclusive group are ev positive extreme and only (d) fatigue group are con	he combination are searched, the results of the load cases from : simply added up, dded to the positive extreme if they are positive and to the are negative, valuated in the way that only one most positive is added to the one most negative to the negative extreme, sidered as Qfat load in fatigue combinations, considered as Qfat load in fatigue combinations,
New Delete	(e) accidental group are c	considered as Ad load in accidental combinations.
	Connection design < Previous	Next > Cancel

Now we can manage the Combinations. Let's add a new one by the **New** command button. We have to select the DL1, LL1 and WIND1 load case and add it to our combination by **Right arrow** button command.

Paratination	Combination P	roperties				
All ULS Combinations ULS Fundamental 1	Name	ULS Fundamen	tal 1			
	Evaluation Type	Code, (6.10) ULS Fundament	al T		Load Cases Combinations	
	Load Cases in (Combination T	Coeff	ļ	Load Cases in Project	۲
	Dead Load DL1	S	0.00		Not grouped cases	
	Live Loads		0.00		DL1	
	▲ Wind Load	5	0.00		LL1	
	WIND1		1.00	>	Wind Loads WIND1 Snow Loads Show Ka	
					Roof Live Loads	
				~~~	<ul> <li>Accidental Loads</li> <li>ACC1</li> </ul>	
					Temperature Loads     TEMP1	
					Seismic Loads     SEIS1	
] Expand all items			•		Expand all items	
Export all linear combination						

We can see imported Result Classes. We have to remove the SNOW1, LR1, ACC1, TEMP1 and SEIS1 from the class. Let's continue with the **Finish** command button.

Calculate vesterday's es	ti <b>Ca[®]</b> Results Classes	×
Classes All ULS Results Classes All ULS Fund All SLS Char. Results Classes All SLS Char (deflection) All SLS Freq. Results Classes All SLS Quasi. Results Classes All SLS Quasi	Result Class Properties         Name       All ULS Fund         Type       ULS Fundamental         Items in Result Class       Items in Result Class         Dead Loads       DL1         Live Loads       LL1         Wind Loads       WIND1         Snow Loads       SNOW1         Roof Live Loads       LR1         Accidental Loads       Acc1         Temperature Loads       TEMP1         Seismic Loads       SEIS1	Items in Project       Image: Constraint of the image: Constraintof the image: Constraint of the image: Constraint of the image: Co
Expand all items      New      Delete	Connection design	Expand all items  Finish Cancel

# 3 Design

Automatic data transfer is finished and IDEA StatiCa Connection with the generated project is launched. All members and load effects were added automatically.



We have to define the right manufacturing operation. Let's start with the **right mouse button** click on the **Operations** and select the **New operation**.



The first manufacturing operation will be the **End plate** of member.

Revit link | IDEA StatiCa



We have to define the bolt group and diameter.



We change the properties of the operation CUT1 according to the picture below.

In the next step, we use again **right-click** on **Operation** and add the manufacturing operation **Cleat**.



Let's change the properties for CLEAT1 according to the picture below.

Solid Transparent Wireframe	CLEAT1 [Cleat]	Editor Copy Delete
A Members	<ul> <li>Cleat</li> <li>Member</li> <li>Mot specified</li> <li>Connected to</li> <li>M534618</li> <li>Profile</li> <li>3 - L80X60X7</li> </ul>	<ul> <li>■</li> <li>■</li></ul>
<ul> <li>✓ <u>M534618</u></li> <li>✓ M534630</li> <li>✓ M534700</li> <li>✓ Load effects</li> <li>✓ DL1</li> </ul>	Profile conversely Cleat length [mm] 150 Cleat shift [mm] 0 Location Both Connection type Bolted	:
VIL1 VIND1 Operations FP1 CLEAT1	Gap [mm] 10 Notch ■ ▼ Bolts Type M16 8.8	• +
CLEATI b	Rows [mm] 0 Positions [mm] -50; 0; 50 Grid Regular Shear plane in thread Shear force transfer Friction	•
	Same on both legs 🖌	

## **5 Check**

We can calculate the analysis right in the Design tab by the Calculate command.



After a while, the results summary will appear at the top left corner of the 3D scene and the **Overall check** model view is displayed. We can quickly fine-tune the model in case we are not satisfied.



We will choose the **Check** tab and turn on the **Equivalent stress**, **Mesh** and **Deformed** model view. We can explore the detailed results for the Pre-loaded bolts also, let's expand the results for the bolt B14.



## **6 Report**

At last, we go to the tab **Report**. IDEA StatiCa offers a fully customizable report to print out or save in editable format.

StatiCa* CONNECTION	RevitBIMtutorial.IdeaCon	- I X
Colculate yestenday's estimates Project	Design Check Report Materials	
2 Copy Project items	New DOC PDF DXF Find Core Detailed BDM Correct All Selected Type of report	
	Project data       Project name     Revit Tudotial       Project number     06       Auhor     UEA Statica Team       Description     Revit BitM report Natorial       Date     3262018       Design code     EN       Material     Statica Team       Steril     S 275       Project item 2     En       Design     E       Design     E       Design     S 275       Description     Revit National       Design     S 2018       Design     S 2018	<ul> <li>A Report</li> <li>Projet dia</li> <li>Projet dia</li> <li>Projet dia</li> <li>Maraiah</li> <li>Projet Hems</li> <li>Destringing</li> <li>Code settinging</li> <li>Code settinging</li> <li>Software info</li> </ul>
	Name         Cross-section         P-Direction         V-Phila         a-Rotation         Offset as         Offset as         Offset as         Direction         Pacing         Direction         Direction <thdirection< th="">         Direc</thdirection<>	Project items settings         Dawings - model       If       If       If       If         Drawings - results       If       If       If       If       If         Drawings - results       If       If       If       If       If       If       If         Drawings - results       If       If </th

## 7. Revit known limitations

Pay attention to the maximal possible correspondence between the geometry of the Revit model to the analytical model (used to calculate the numerical analysis in the Robot). Due to the differences the results may vary.

#### Keywords:

connection, joint, EN, Eurocode, analysis, end plate, cleat, Revit, BIM, BIM link