New approach to design and check steel connections with Revit, Advance Steel or Robot

Juraj Šabatka
IDEA StatiCa

Hugo Michaud
Graitec Inc.

Learning Objectives

- Learn how to design and check any steel joint according to the code in minutes
- Learn how to work with a model from Advance Steel, Revit Structure, and Robot to increase productivity of steel joint design
- Understand current methods and differences between the new CBFEM method for design and code check of steel joints
- See this new approach in real life projects all around the world

Description

This class will introduce a novel approach to structural design and code check of steel joints and connections of any topology. It is based on a new method called Component based Finite Element Model (CBFEM) and has been implemented in Advance Design Connection software. Structural engineers and fabricators can use models created in Advance Steel, Revit Structure and Robot Structural Analysis to quickly export all data necessary and perform pass/fail check of any joint according to the US, CA and EU codes. We will examine this newly extended workflow for Autodesk users, and show its relevance on real life projects from all around the world. Join this class to learn how to deal with any steel joint your projects can bring, in minutes. We’ll explain the whole calculation process and show tools to optimize the joint. This session features Advance Steel, Revit Structure and Robot Structural Analysis.

Speaker(s)

Juraj Šabatka, CSO&CFO IDEA StatiCa

Juraj is in charge of sales, support, and marketing of IDEA StatiCa. He is also responsible for interoperability of IDEA StatiCa with Autodesk Advance Steel, Revit, Robot and FORGE platform. Juraj is a big enthusiast about technology and entrepreneurship. After years in corporate banking, he joined IDEA to mix his business skills with a unique Czech technology for structural engineers. He has an academic background from three continents – Prague University of Economics, Harvard University and Singapore Management University.
Hugo Michaud, Managing Director Graitec Inc.

Hugo is a passionate software executive with extensive experience in the construction, engineering, aeronautics, and fashion industry. With more than 15 years’ in managing CAD, BIM and PLM implementation projects, Hugo has helped hundreds of companies successfully achieve their corporate goals.

He was a key actor in the success of Advance Steel for the Graitec Americas division, before Autodesk’s acquisition of the software. Since then, Hugo has been very involved in the Autodesk world by helping engineering firms and steel fabricators in their BIM evolutions. Hugo holds an Information Technology Management Bachelor degree from the Hautes Études Commerciales in Montreal, Canada.
A new method for the weld stress evaluation – plastic welds

Method

Advance Design Connection uses a novel Component Based Finite Element Method to model structural connections. Plates are evaluated using shell finite elements and bilinear material diagram. On the other hand, for example bolts and welds are simplified and assessed as a component using steel structural code, i.e. AISC 360-16.

To express the weld behavior, an improved weld model is applied. A special elastoplastic element is added between the plates. The element respects the weld throat thickness, position and orientation. The equivalent weld solid with bilinear material properties is inserted with the corresponding weld dimensions. The nonlinear material analysis is applied and elastoplastic behavior in equivalent weld solid is determined. The plastic strain in the weld is limited to 5 % as in the plate (e.g. EN1993-1-5 App. C, Par. C.8, Note 1). The stress peaks are redistributed along the longer part of the weld length.

General welds, while using plastic redistribution, can be set as continuous, partial and intermittent. Continuous welds are over the whole length of the edge, partial allows a user to set offsets from both sides of the edge, and intermittent welds can be additionally set with a set length and a gap.

Weld stress evaluation for plastic welds

While using plastic redistribution in welds, there is no need to average or interpolate the stress. Calculated values are used directly for checks. The most stressed weld finite element is evaluated using normative checks, i.e.

\[ F_{nw} = 0.6 F_{kxx}(1.0 + 0.5 \sin(1.5\theta)) \]

The magnitude and direction of the force vector in the weld are therefore determined using finite element method but due to simplifications, e.g. neglecting of residual stresses, the normative check is used for final evaluation.
Workflow description – Code-check of a complex anchoring

We will go through the design and check of the anchoring used in a propane/butane storage sphere in a high seismic area. This is a real example from our customer in Canada.

Geometry

We have to define New project by command button from the ribbon. The wizard will pop-up. Let's start to define the code AISC, name and description is optional and we will choose the figure of Anchoring.

Another wizard window will pop-up.
To make the design process easier we will choose the anchoring of the circular column.

Advance Design Connection will define the first member for us.
Let’s change the properties of the cross-section of member COL (Column) by green arrows. We will choose the **parametric circular hollow section** from the **Rolled section** tab.

Input data according the figure below:
Let’s define a new member by **plus** button command. We will define the properties of the new member **M2** according to the picture below:

The important properties set:
- Cross-section: HSS(Imp) 12x12x1/2
- \( \beta \) - Direction: 15°
- \( \gamma \) – Pitch: 45°

Now we will add another cross-section, choose the plus button command and define properties of the new member **M3**.
Load effects

We have to load the members with internal forces. We can input the manually or import them. In this example we will use import function to show it’s capabilities.

We will proceed with the ASD code check. Then choose Import command button from the Data window.

We will use the Excel sheet to define the loads (prepare the table according the figure below). Select the area and copy data by Ctrl+C.
We will switch windows back to our project, choose the cell for N values for column COL and paste the data from excel spread sheet file by Ctrl+V. It’s better to check the check box Replace existing loads.
Design

The design of connection is based on the definition of manufacturing operations. We will define a several of them. Let’s start with the Base plate – BP1.

You can see the changing model in a 3D window.
Let’s proceed with another manufacturing operation with plus button command. Choose the **Cut** (first one) and define the properties as shown in figure below.

We have to cut the same way the other diagonal M3, let’s use the function **Copy** at the top ribbon. Then only redefine the **Member** property to **M3**. We connected the diagonals M2 and M3 to the column COL.
We have to add ribs to the column and base plate. We will use the manufacturing operation **Rib** (fourth one in the first row).
Let’s define the properties of Rib operation according the figure below:

![Manufacturing operations](image1)

And we will add another rib as a plate to support the diagonals. Let’s define another **Rib** with different properties.

![Manufacturing operations](image2)
And we can copy this RIB2 operation with Copy from the top ribbon. We will proceed by setting the properties of the RIB3 operation.

Only one thing is missing, the cut of the members M2 and M3 by their ribs. Let’s choose another Cut and define the properties according to the figure below.

We can explore the cut in the model. There is a notch thru the walls of the HSS member, all welds should be present.
Let's use the Copy function and copy the CUT3 and redefine the Member to M3 and Cut by to RIB2

We finished our design of the model.
Check

Let’s **Calculate** the project by command in the top Ribbon. The process is iterative and after a while the results will be shown.

We can see the overall results in Details window and in the Summary tab. In the 3D window is visualized the Overall check, which means the „Traffic light“ function. Gray zones are utilized less than 60%, green zones are utilized less than 95%, orange ones less than 100% and red ones did not pass the code check.

Let’s explore the results. We can see the formulas and values used for the code check by clicking **Plus** button in the row BP1. Let’s see the **Equivalent stress** and **Mesh**.
The last step is to create the report. Let’s choose the Preview/Print command in the Navigator window.

We can modify the content of the report with the command at the bottom of the Data window, for example switch on the **Formulas** and **Bill of material**. Then **Refresh** the document and the report is ready.
Workflow description – Export from Advance Steel

To enable this workflow, we need to integrate Advance Design Connection into Advance Steel. We do this easily using BIM links tool.

Import

Let’s open the model in Advance Steel. Then we type in command CONCHECKAISC for export to Advance Design Connection.
Let’s keep it simple, we have to choose:

1. The node (which will be representing the node in the ADC model).
2. All members (first chosen member will be the bearing one), choose first the main beam.
3. All other entities (bolts, plates, welds)

All needed data is exported from Advance Steel - cross-sections, geometry, bolts, welds, etc.

**Load effects**

We have to load the members with internal forces. We can input the manually or import them. In this example we will use the import function to show it's capabilities.

We will proceed with the **LRFD** code check. Then choose the **Check equilibrium** command and **Import** command button from the Data window
We will use the Excel sheet to define the loads (prepare the table according to the figure below).

Select the area and copy data by Ctrl+C.

![Excel sheet with loads](image1)

We will switch windows back to our project, choose the cell for N values for column COL and paste the data from excel spreadsheet file by Ctrl+V. It's better to check the checkbox Replace existing loads.

![Project window with loads](image2)
Let's proceed to the Check item and **Calculate** the project. After a while the results are shown, we can see the summary and the Overall check in the 3D view.

We want to see the **Equivalent stress**, **Mesh** and **Bolt forces**. If we pick up the flange, one can see the detailed view in the Data window and in the result tables.

The steel truss joint was imported from Advance Steel, loaded and checked by Advance Design Connection.