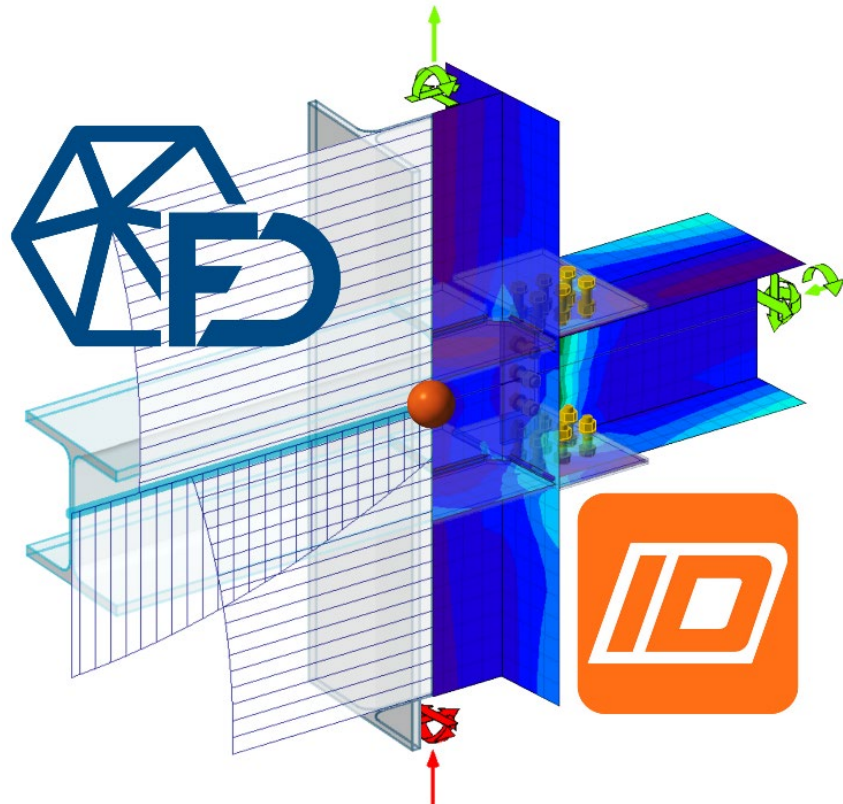


# FEM-Design to IDEA StatiCa Workflow



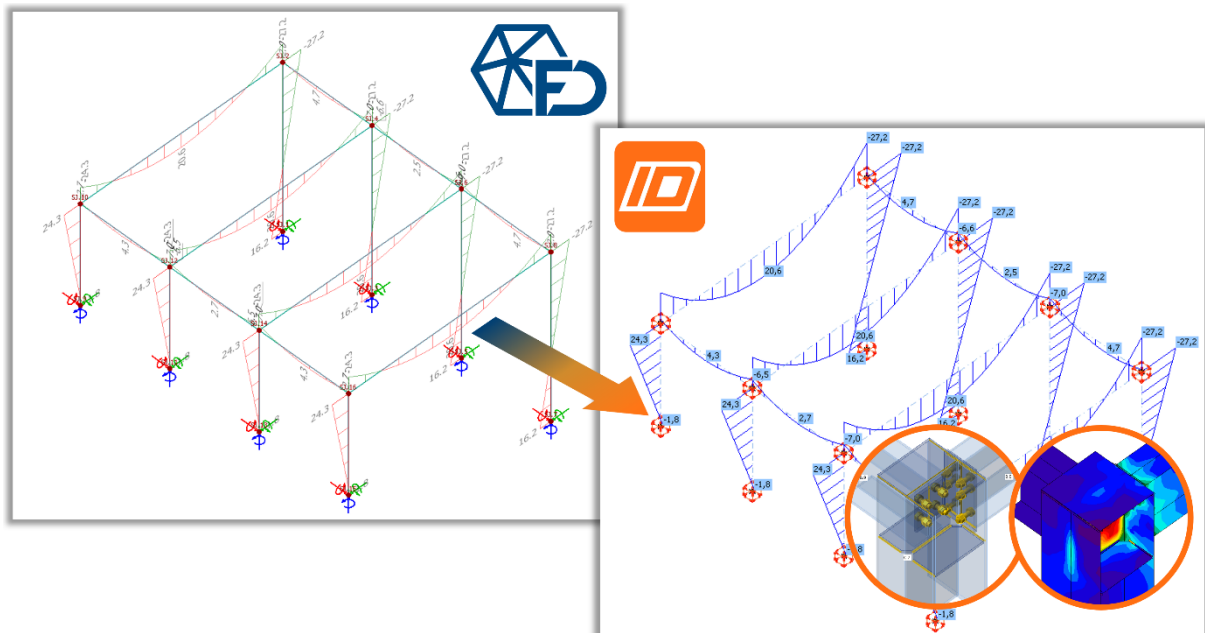
StruSoft offers a free tool that enables a smart connection between FEM-Design and IDEA StatiCa. This tool allows us to feed IDEA StatiCa with 3D steel structures (or even just selected connected members) and their internal forces as intelligent input for detailed design and verification of steel connections. In this way, this dedicated tool accelerates the finalization, detailing and documentation of steel connections thanks to IDEA StatiCa's 250 ready connection templates and more than 10,000 configurations, as well as the optimization of fabrication and raw material costs.

## Contents

|   |    |
|---|----|
| Key features.....                                 | 2  |
| Limitations .....                                 | 2  |
| Compatibility .....                               | 2  |
| Workflow steps.....                               | 3  |
| FEM-Design model-preparation for sharing.....     | 3  |
| Transfer of FEM-Design model to IDEA StatiCa..... | 4  |
| Detailed connection design (briefly) .....        | 8  |
| Change management .....                           | 11 |

## Key features

- Sharing FEM-Design data with IDEA StatiCa as an input for steel connection design including:
  - positions, cross-sections and materials of steel bar members
  - internal forces (normal, shear forces and moments) calculated in the members
- Possibility to select which connected steel members we want to transfer to IDEA StatiCa
- Data transfer can be done in two simple ways:
  - via a live direct connection if both programs are available on the computer used
  - via a dedicated file if the two softwares are running on separate computers
- In case of the live connection, a single click updates the input of the IDEA StatiCa connection design based on changes in the linked FEM-Design model
- Automatic but editable data matching (cross-section and material) between the two softwares
- The connection between the two programs does not require the installation of any additional programs on either side



## Limitations

Connection design in IDEA StatiCa works only for

- straight and uniform bars
- bar internal forces are calculated for load cases only

## Compatibility

- FEM-Design 22 and later version
- IDEA StatiCa 21.0 and later version

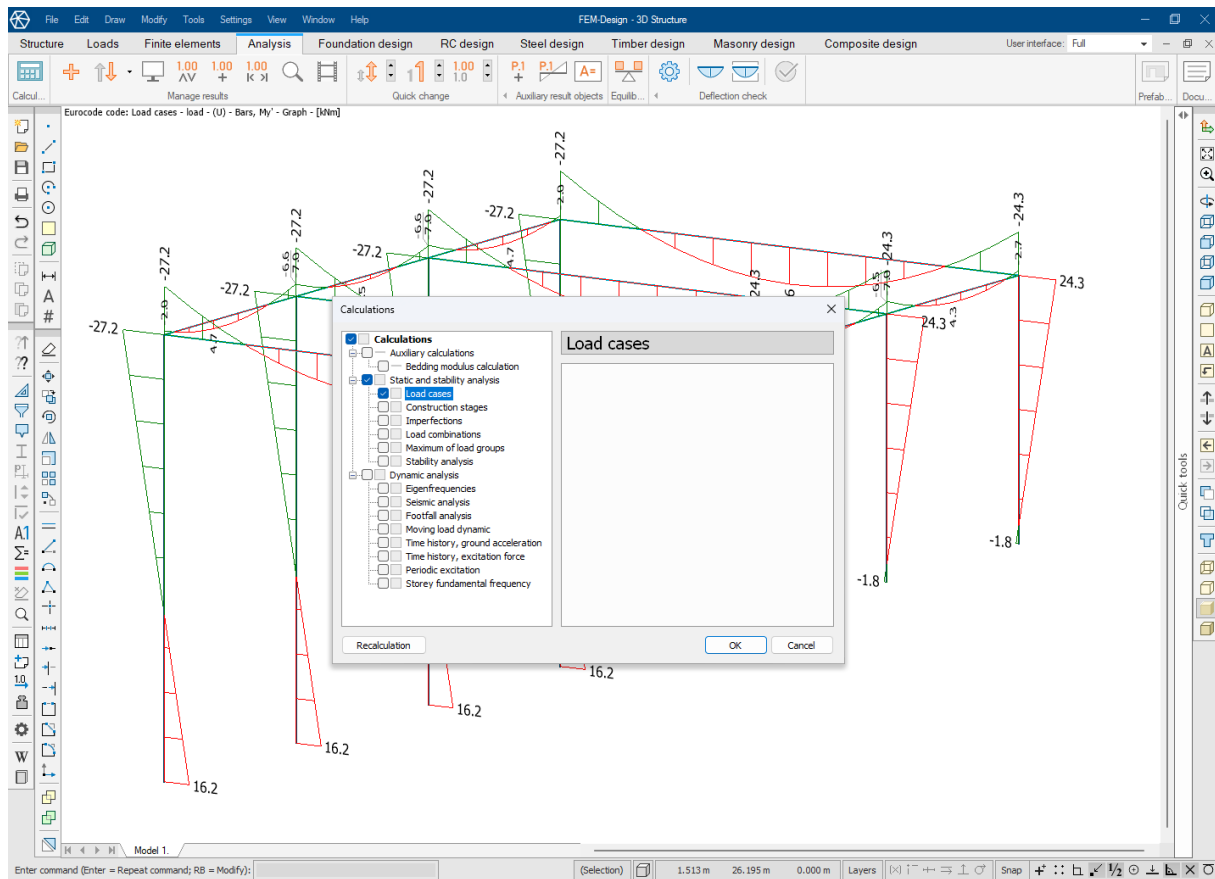
## Workflow steps

### FEM-Design model-preparation for sharing

Since the main task and advantage of the collaboration workflow is the intelligent sharing of the steel bars designed in FEM-Design and their internal forces from Load cases with IDEA StatiCa for the subsequent detailed connection design, the following preparations must be made in FEM-Design:

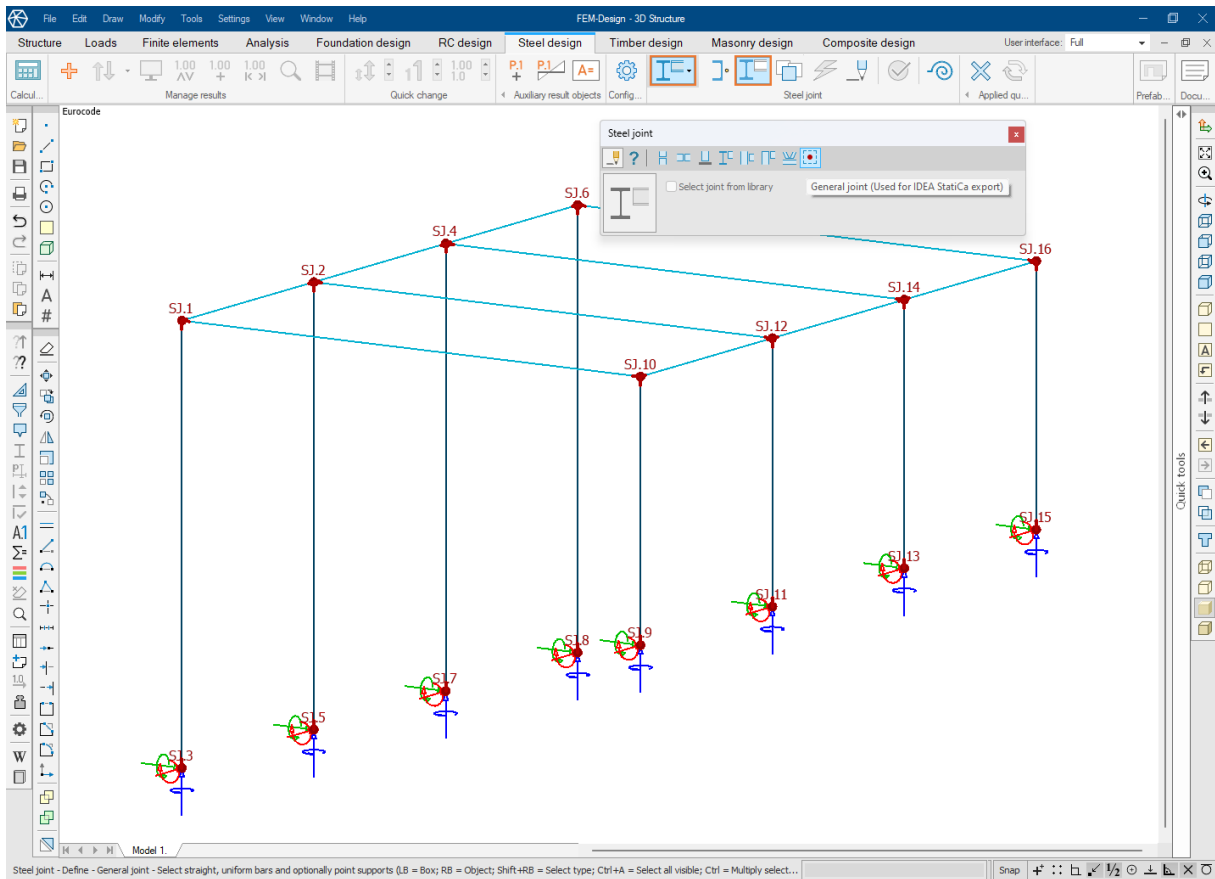
#### 1. Run analysis for Load cases

In order to obtain the bar internal force results and transfer them later, we need to run analysis on the current model containing steel structure elements. This can be done with *Calculate* in *Analysis* mode or any design mode (e.g. *Steel design*): *Static and stability analysis / Load cases*. (After export, we can browse and display the results of the individual Load cases in IDEA StatiCa, who will automatically combine them according to applied code required for connection planning.)



#### 2. Definition of joints

We need to specify the bar connections/nodes that we want to design in IDEA StatiCa. In *Steel design / Steel joint* mode, use the *Define joint* command and create the connections with the *General joint* function specialized for the IDEA StatiCa connection. The image shows an example where joints have been defined at all bar ends, so we can design connections not only between bars, but also between bars and supports. When exporting, we will be able to choose from among these joints, either sending all of them or just a few for the design with IDEA StatiCa.



## Notes

- Since IDEA StatiCa's connection design only works for straight and uniform bars, if we change the geometry of a bar to a curved axis or varying cross-section, the joints belonging to the bar ends are automatically deleted.
- Regular FEM-Design Steel joints can also be exported to IDEA StatiCa.

## Transfer of FEM-Design model to IDEA StatiCa

Sharing joints, steel members and results for IDEA StatiCa can be done in two simple ways:

- via a direct, so called "Live" connection if FEM-Design and IDEA StatiCa are both available on our computer; or
- via a dedicated file exchange if the two softwares are running on separate computers.

## Notes

- The following export processes must be started from *Steel design* mode, where steel joints are visible and ready for selection by default. In other modes, the "Steel joint, symbol" Object layer must be active for the joints to be selected.
- In both cases, with a selection step, we can specify whether we want to send all joints or only a few selected ones from FEM-Design to IDEA StatiCa.
- Steel material and cross-section mapping between the programs is based on an editable template file:

C:\Users\USERNAME\AppData\Roaming\StruSoft\FEM-Design  
VERSION\FEMData\saf\_export\_mappings.xlsx

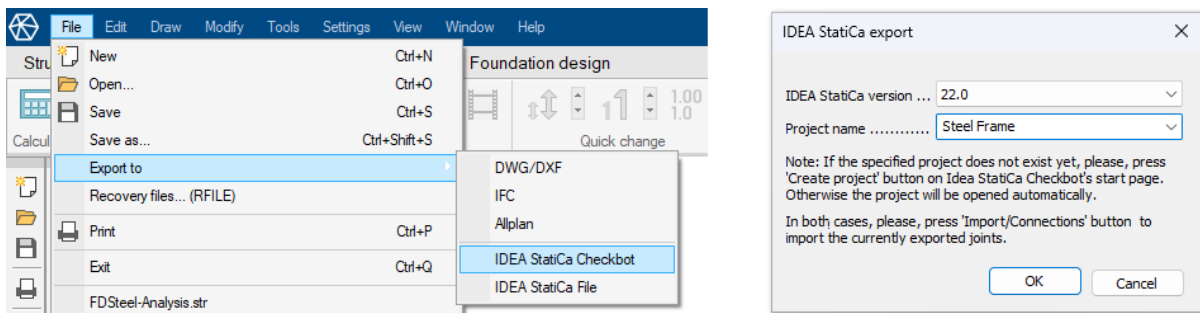
- Unmatched FEM-Design cross-sections are output with a parametric description, based on which IDEA StatiCa creates the sections.

## A. Live connection

In this method, the FEM-Design export creates an IDEA StatiCa project ready to start designing transferred joints. The steps of data exchange are the following.

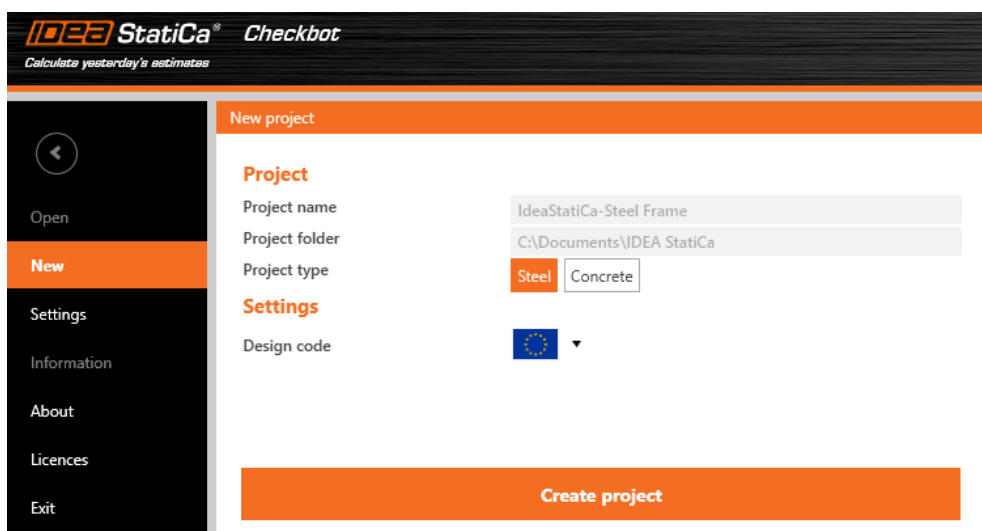
Within FEM-Design:

1. Save the current project before data transfer.
2. Run the command *Export to / IDEA StatiCa Checkbot*.
3. Select the Joints to be designed with IDEA StatiCa.
4. Select the version of your IDEA StatiCa (21.0 or later).
5. Set an IDEA StatiCa project name, too. By default, the name of the current FEM-Design project (saved in the first step) is offered, but it is not mandatory to use it. We can enter a completely new name or choose from existing IDEA StatiCa projects. The data structure of a new IDEA StatiCa project is created in the subfolder with its name within the folder of the current FD project. By clicking the *OK* button, IDEA StatiCa Checkbot will start.

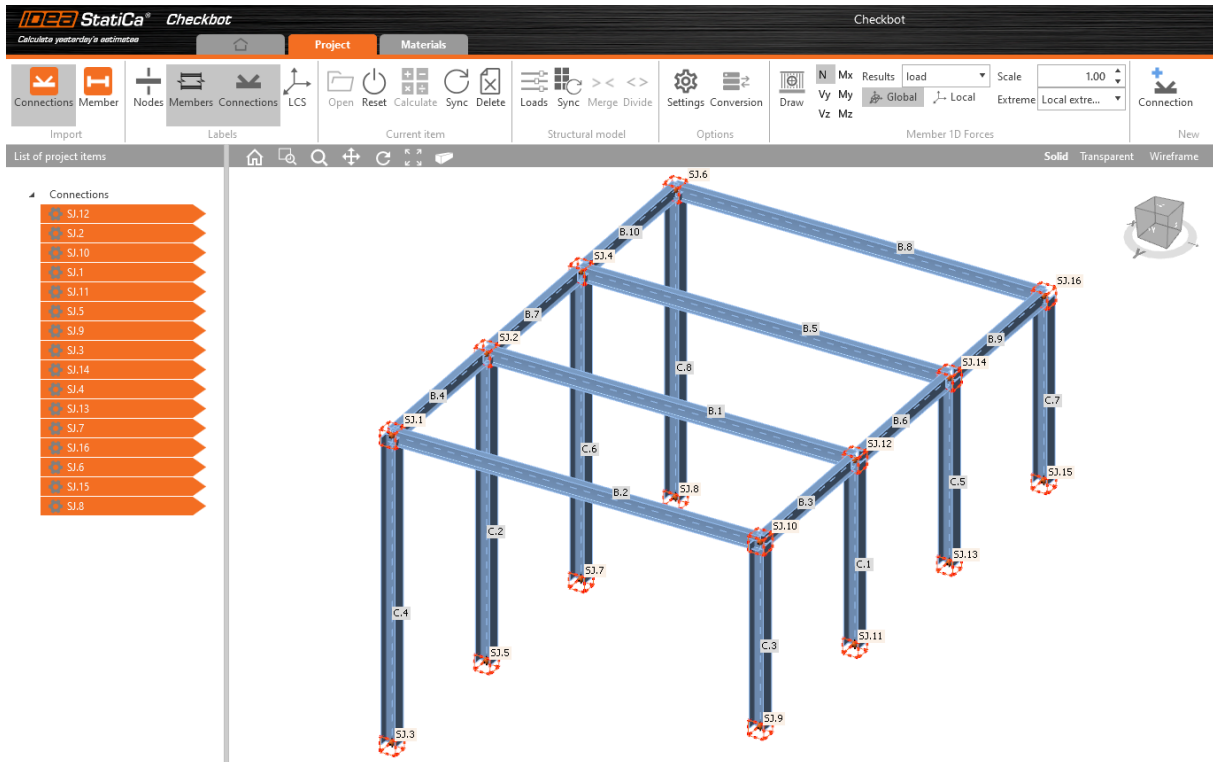


Within IDEA StatiCa Checkbot:

6. If the specified IDEA StatiCa project does not exist yet, we must approve its creation by pressing the *Create project* button in Checkbot's *New project* dialog. Otherwise, an already existing project will be opened automatically.



7. Press the *Connections* button (Import) to load data exported from FEM-Design.

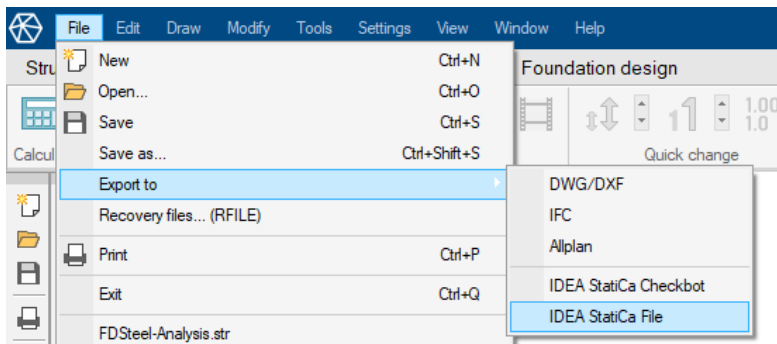


## B. File exchange

In this method, FEM-Design saves the data required for the design of selected joints in a SAF (.xlsx) file dedicated to IDEA StatiCa, which IDEA StatiCa Checkbot can read and easily use to build the connection design project. The steps of data exchange are the following.

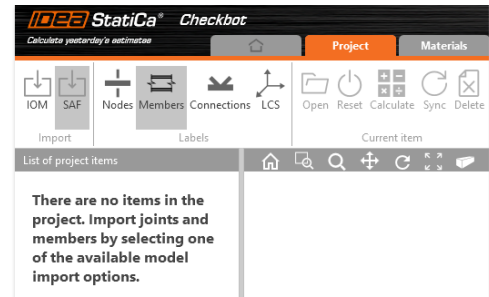
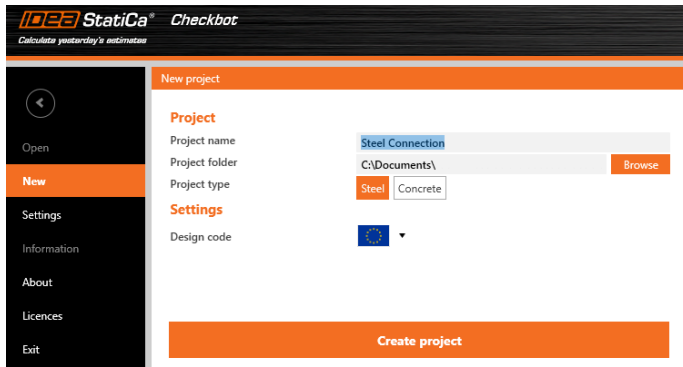
Within FEM-Design:

1. Run the command *Export to / IDEA StatiCa File*.
2. Select the *Joints* to be designed with IDEA StatiCa.
3. Save the file with a specific name and location.

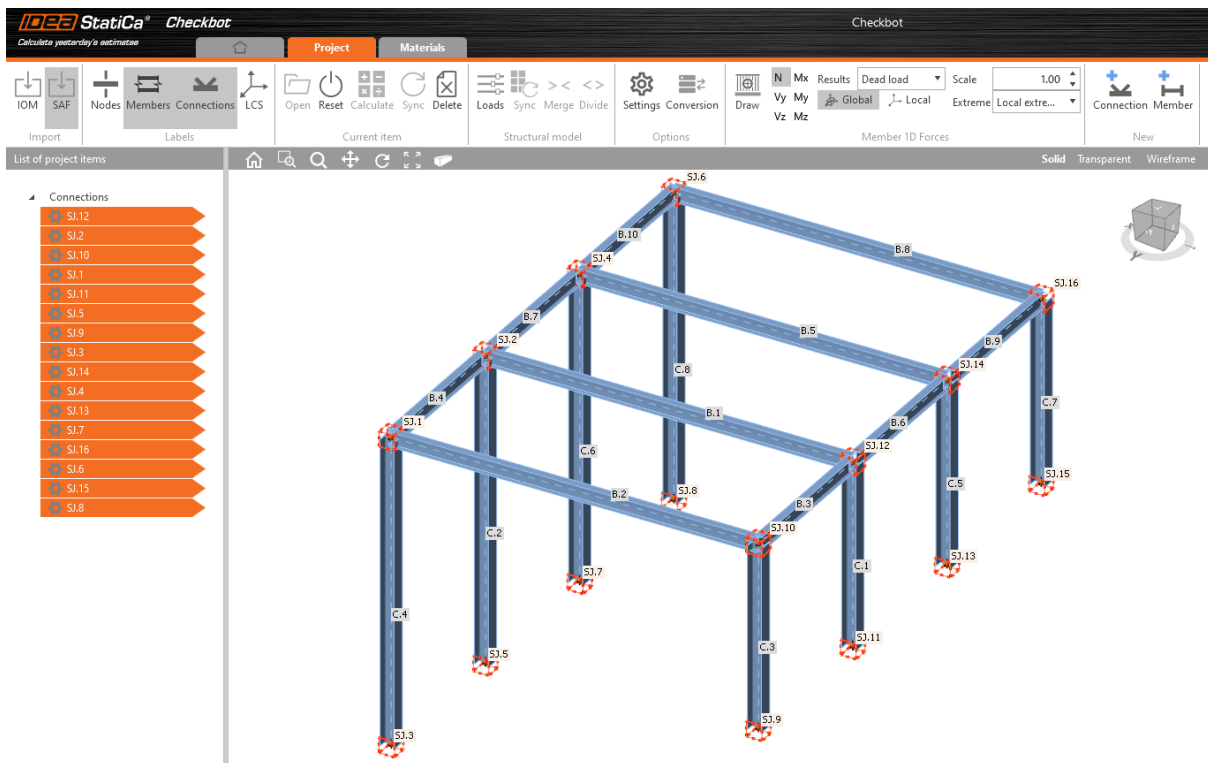


Within IDEA StatiCa Checkbot:

4. Create a new IDEA StatiCa project at New.



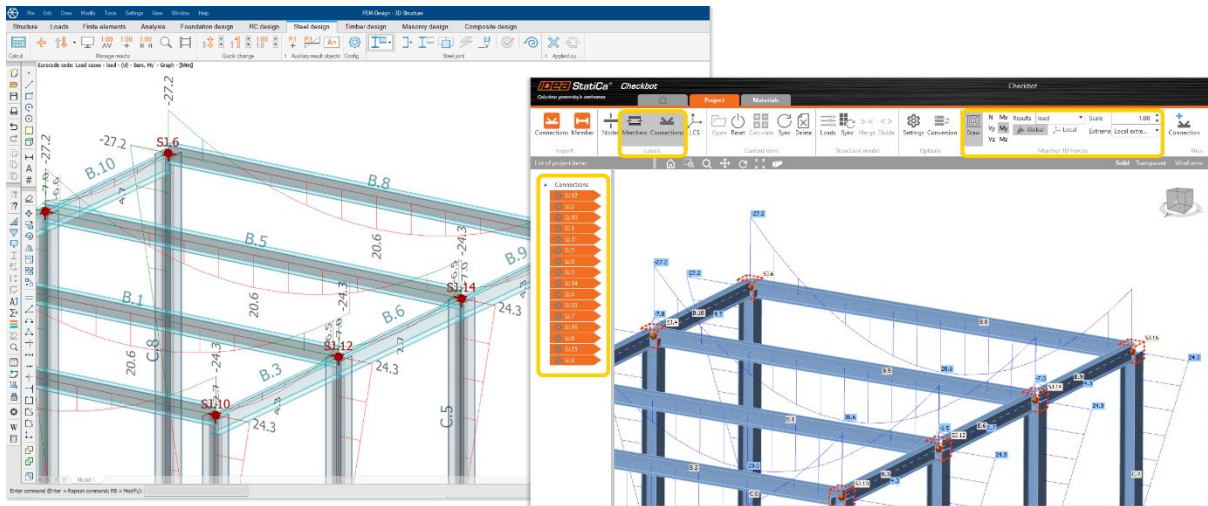
5. Open the file saved with FEM-Design with the SAF command (*Import*).



## Detailed connection design (briefly)

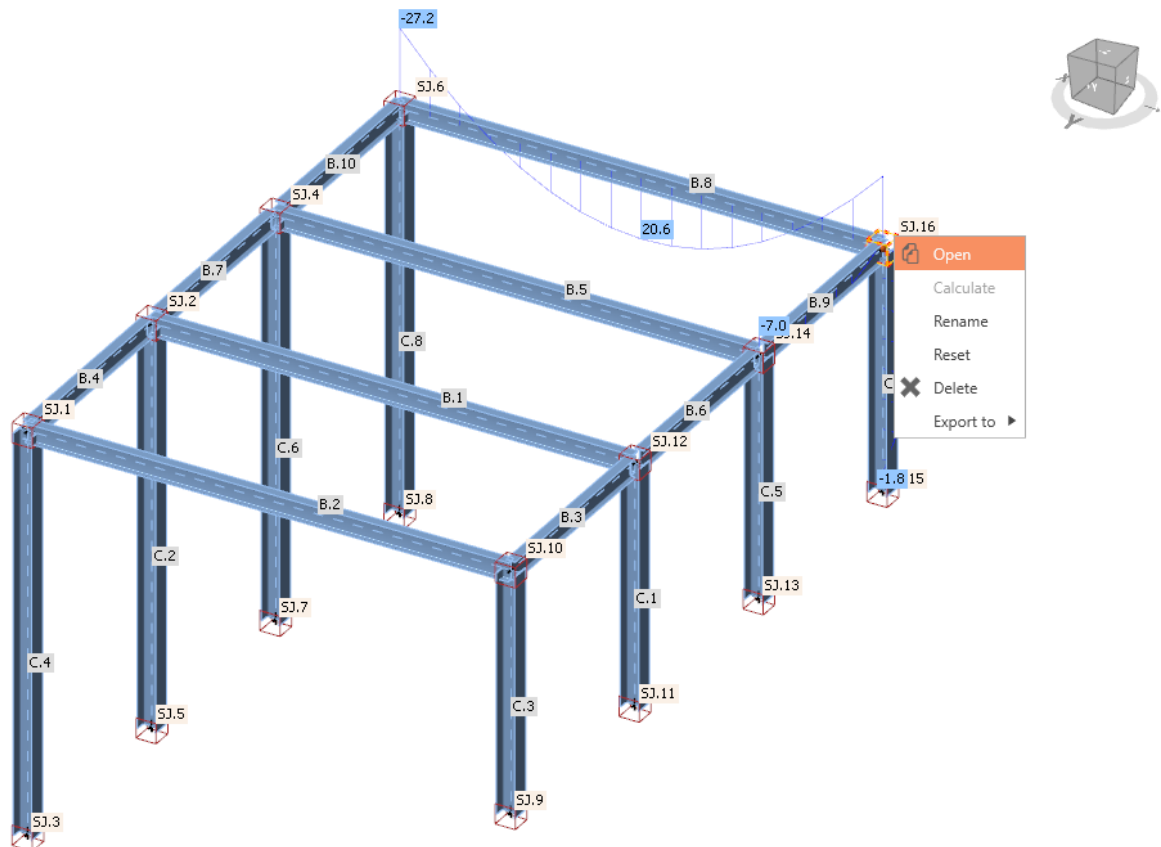
Thanks to the fast import, the model part selected in FEM-Design is immediately displayed on the Checkbot interface. Here we can display:

- the identifiers of the objects specified in FEM-Design with the *Label* tools, and
- the internal force diagrams of the transferred individual load cases with the *Member 1D Forces* tool.



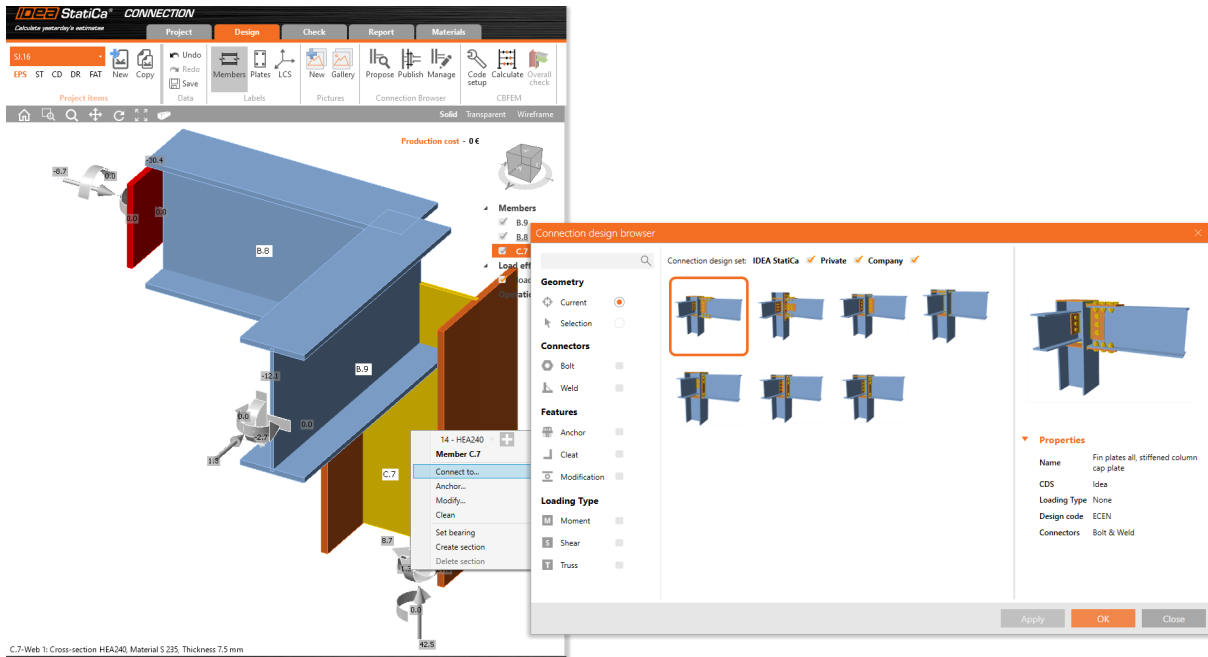
Then in IDEA StatiCa we design the connections one by one as usual with IDEA StatiCa Connection:

1. Send the connection to be designed to the *Connection* module with the *Open* command.

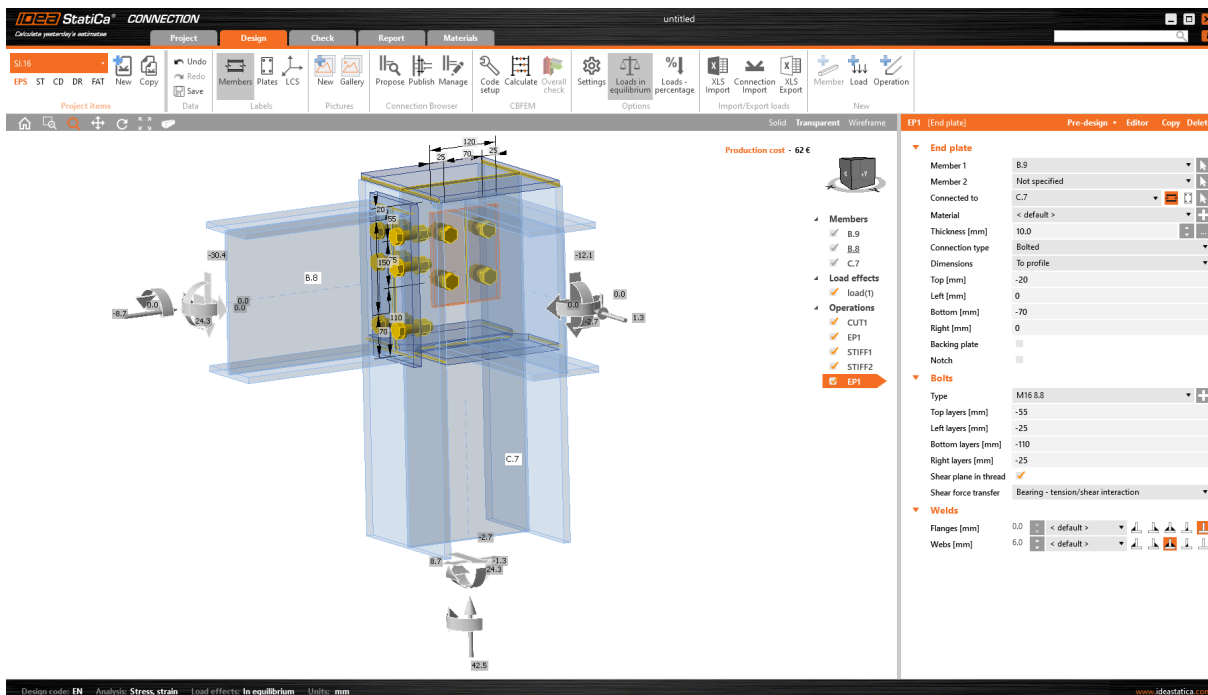




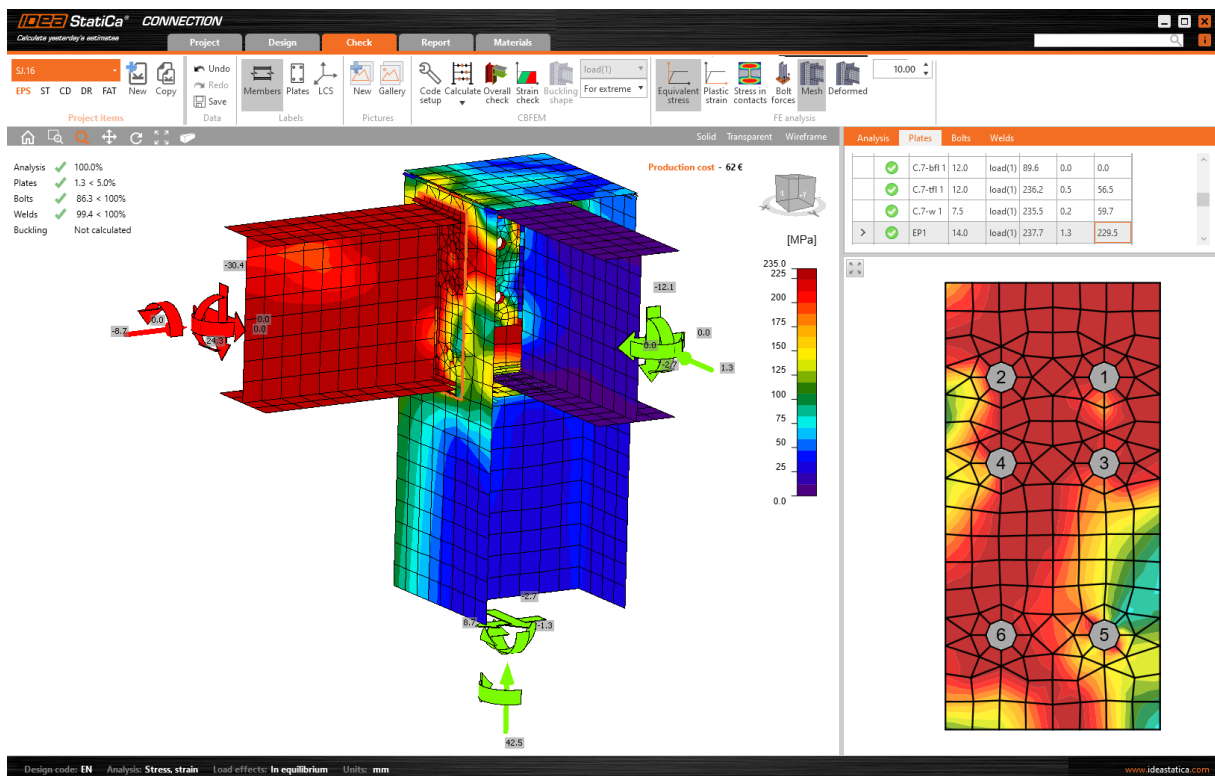
2. Specify how each member is connected to each other and with what design type.



3. Set the parameters, the sizes, the positions, and the behaviors of the assembly components such as plates, bolts and welds.



- Analyze, check and optimize the connection until it meets the requirements. Finally, we can document the finalized connection type in detail.



## Change management

In case of a live connection - running the two programs together - we have the opportunity to update the changes of the FEM-Design model in the IDEA StatiCa connection design project. We must approve the structural change by re-running the analysis for the load cases (in order to update the internal forces to be transferred), then click the *Sync* button (*Structural model*) in IDEA StatiCa to transfer and track the changes.

