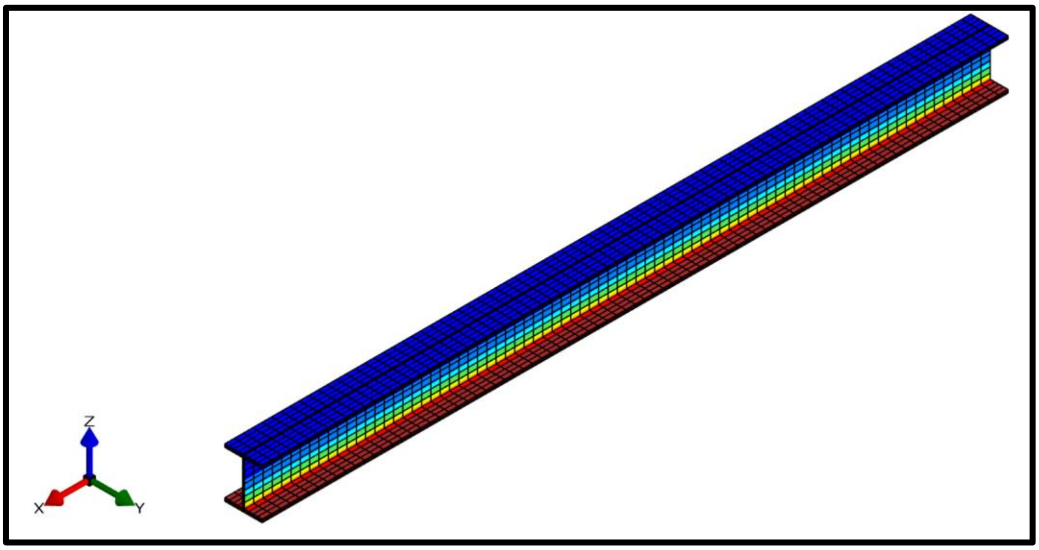
SAFIR® training session
Johns Hopkins University, Baltimore

Example: steel beam with 3D solid elements

“Loaded steel beam heated from below, modeled with solid FE”

T. Gernay & J.M. Franssen

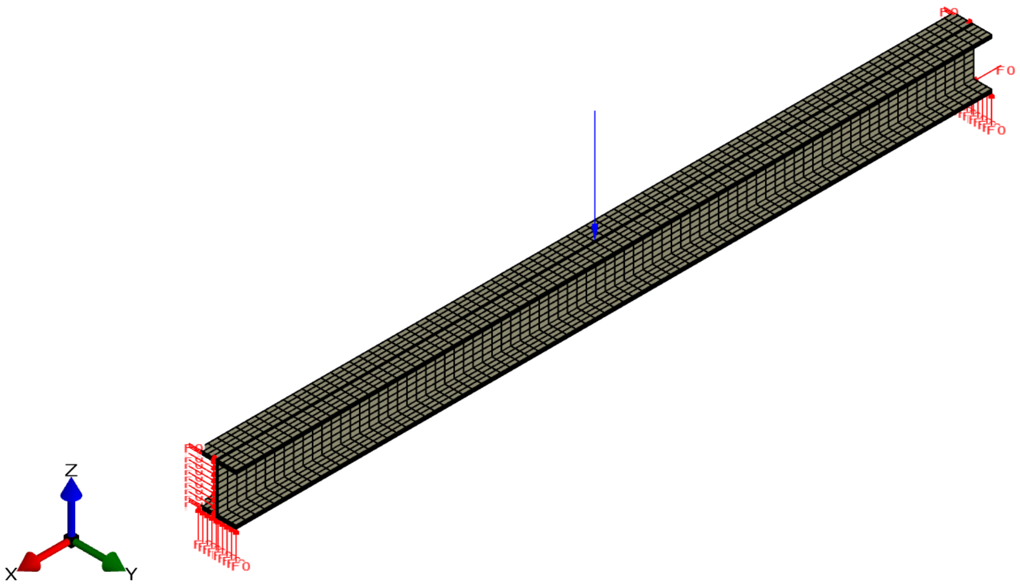


1. General description

This example deals with a steel beam exposed to fire on its lower face. The beam will be modeled using 3D solid finite elements.

General data:

- Simply supported beam with 6 m span
- Steel profile HEB 400
- Yield strength 355 MPa
- Fire: ISO fire on the lower face
- Loads: point load applied at mid-span



The dimensions of the HEB 400 section are as follows:

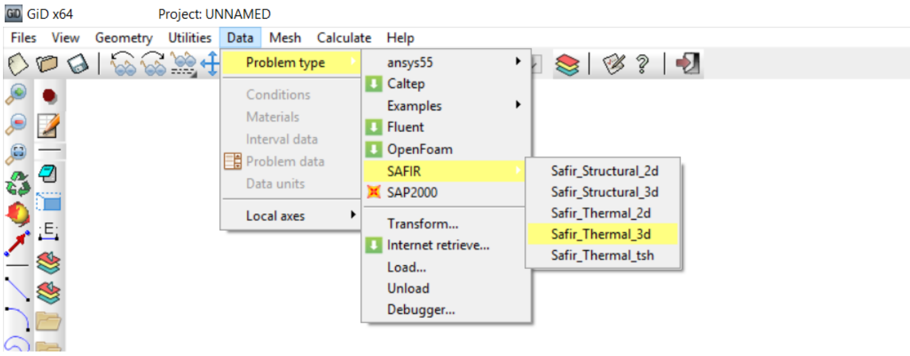
- Width 300 mm
- Height 400 mm
- Web thickness 13.5 mm
- Flange thickness 24 mm

2. Thermal Model

The objective is to create the 3D geometry of the beam and to analyze it under fire exposure. The same geometry and discretization will be used for the subsequent 3D structural analysis.

2.1. Create a project in 3D for Thermal Analysis

From the pull down menu select:
Data -> Problem type -> SAFIR -> Safir_Thermal_3d



To save the project select (or use icons on the left):
Files->Save or or [Ctrl + s]

Enter a file name, e.g.: ***3Dtherm_steel***
GiD creates a directory with the name ***3Dtherm_steel.gid***
GiD creates a number of system files in this directory.
When you start the SAFIR calculation the Safir .IN and .OUT files will be created in this directory.
Note: the ***3Dtherm_steel.OUT*** file will be needed for the 3D structural model.

2.2. Create the geometry of the beam

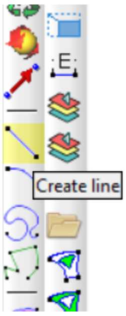
We will create the beam with the following coordinate convention:
- Length along the global x-axis
- Cross-section in the y-z plane with the height along the global z-axis

The view is by default in the x-y plane.
We will start by drawing the cross-section. To switch the view to the y-z plane, select:
View->Rotate->Plane YZ

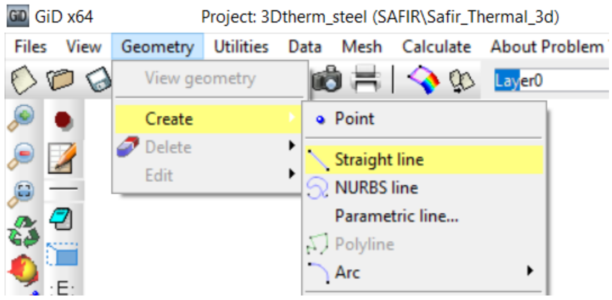
<u>Important note</u>: 3D solid elements in SAFIR are hexahedra (version 2019 and earlier).
To mesh with tetrahedral in GiD, a *Structured Volume* mesh should be specified. To ensure that such mesh condition can be applied to the volumes, it is recommended to define a regular structure based on parallelepiped volumes.

We will first create the geometry of the cross-section in the YZ plane, centered at (0,0,0), using Lines and NURBS surfaces.

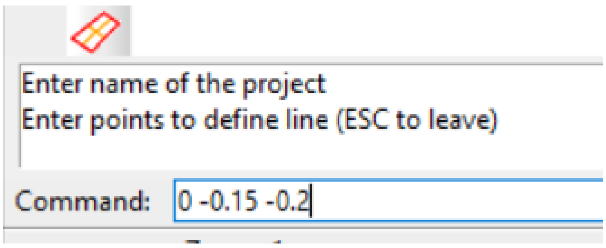
Select the create line command.



or



We start by defining the left half of the lower flange. The first point defining the lines is:
0 -0.150 -0.200
Type these coordinates in the command box, then validate with enter.

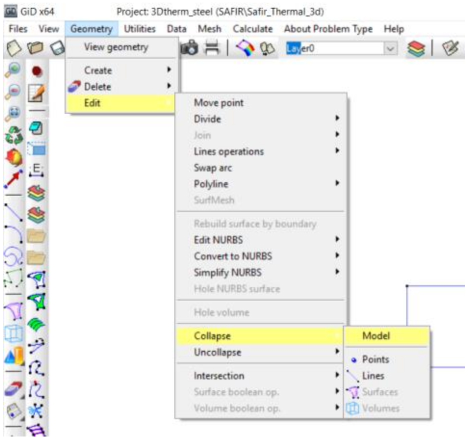


Next, type the coordinates:
0 -0.00675 -0.200
0 0. -0.200
0 0. -0.176
0 -0.00675 -0.176
0 -0.150 -0.176
0 -0.150 -0.200
When validating the latter, press *Join* to close the rectangle (if prompted).

Press *[Esc]* then select again the create line command.
Press *Ctrl+A* to switch the mouse mode to *Pick an existing point*. Pick the points (0,-0.00675, -0.200) then (0,-0.00675, -0.176) to draw a line between these two.
At the end, it should look like that (left half of lower flange):

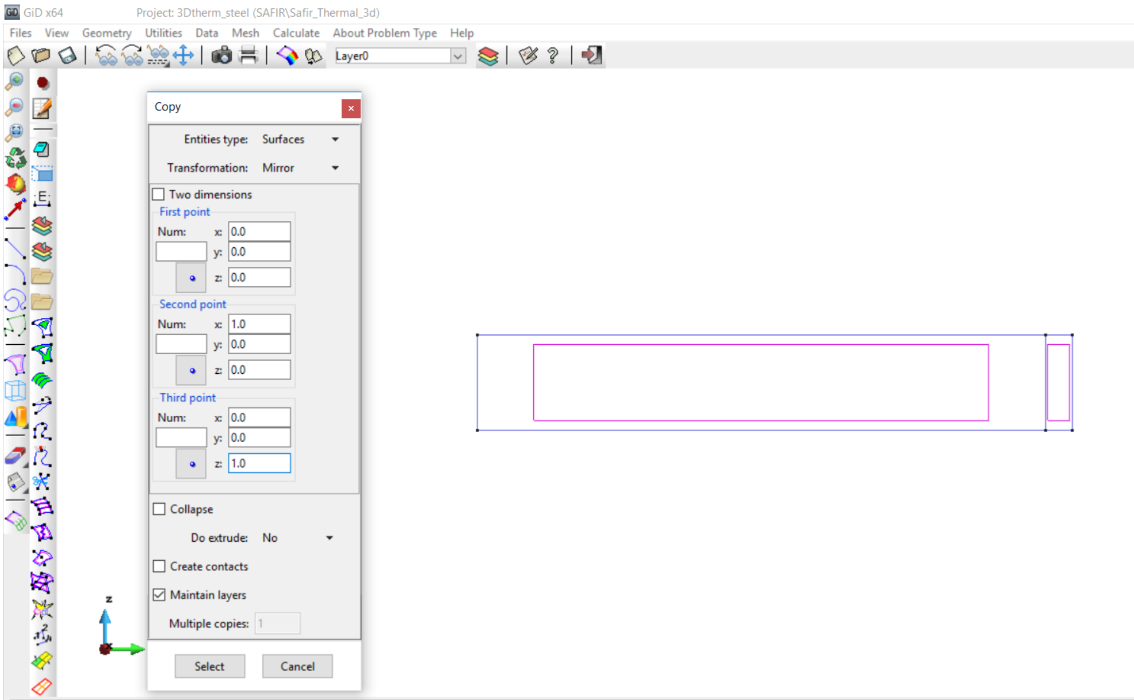
Note: if two points with the same coordinates were created, use the command *Geometry->Edit->Collapse->Model*

This command will merge points or lines which are on top of each other (the command box will indicate if nodes have been deleted).



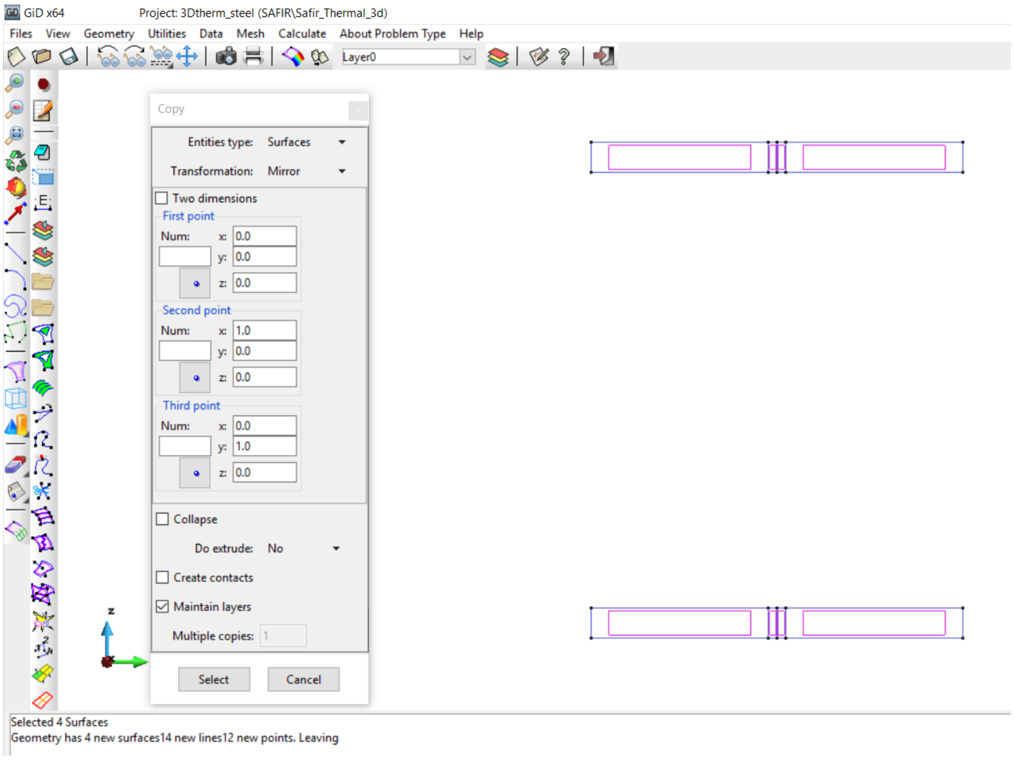
Next, create 2 NURBS surfaces for the 2 rectangles.

We will now duplicate these surfaces to create the right half of the lower flange.
Select *Utilities->Copy*
and apply a mirror transformation to the surfaces as shown below. The plane defining the mirror is the plane XZ (1st point 0,0,0; 2nd point 1,0,0; 3rd point 0,0,1):



Select the 2 surfaces and press *Finish*. This resulted in the creation of the right half of the lower flange.
Now, we can apply the same technique to copy the 4 surfaces of the lower flange to create the upper flange.
Select *Utilities->Copy*

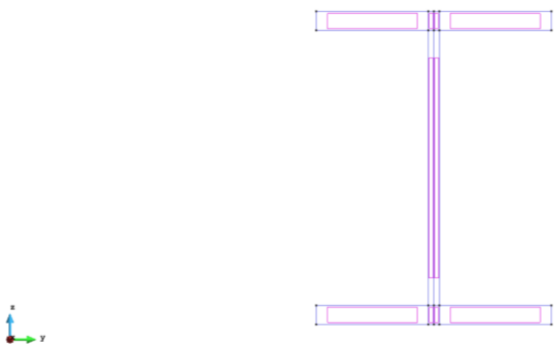
and apply a mirror transformation to the 4 surfaces where the plane defining the mirror is the plane XY (1st point 0,0,0; 2nd point 1,0,0; 3rd point 0,1,0).
Select the 4 surfaces and press *Finish.* This resulted in the creation of the upper flange.



Next, we will create the web.
Select the *create line* command. Press *Ctrl+A* to switch the mouse mode to *Pick an existing point*. Pick the points defining the web and join them by lines. Then, create the 2 NURBS surfaces for the left and right part of the web.

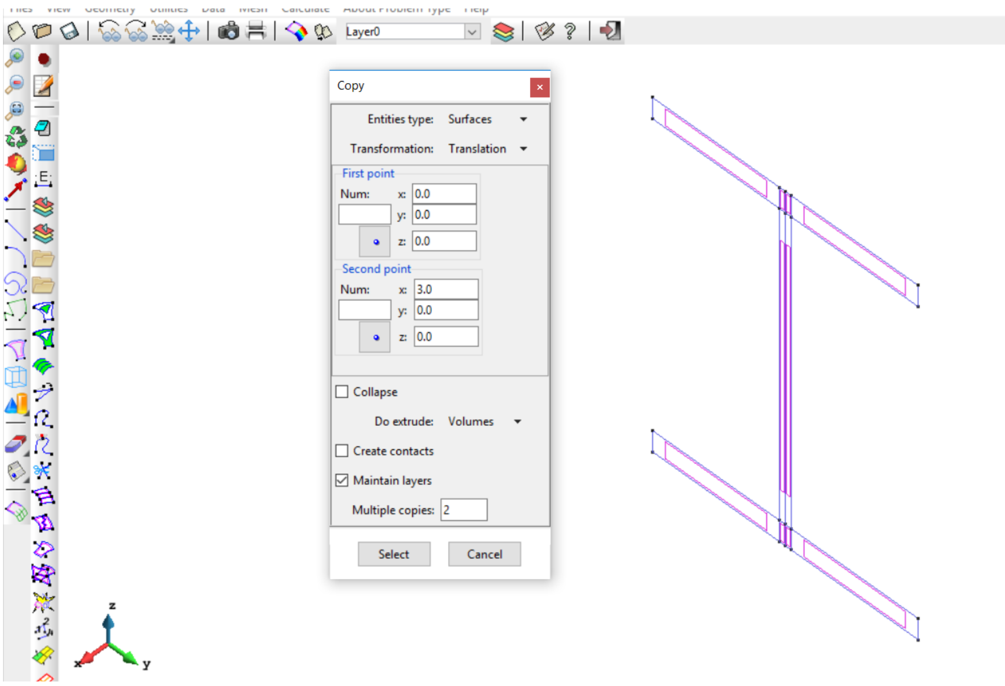
Note: if a NURBS surface fails to be generated (with the command box indicating a message such as "error, bad surface, contour lines not closed"), the most likely reason is that there are several nodes at the same coordinates. This can be solved again using the command: *Geometry->Edit->Collapse->Model*

The cross-section is now complete.

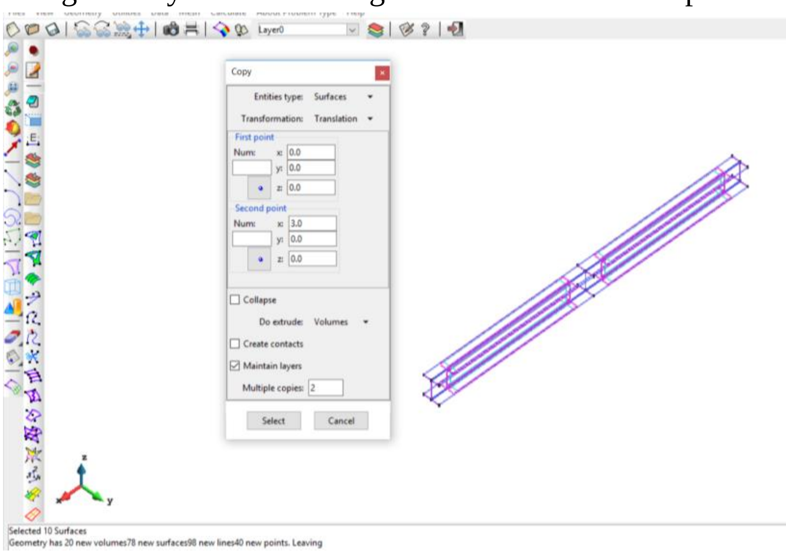


Now, we will extrude this cross section along the length of the profile.
Change to the 3d isometric view by selecting from the pull down menu:
View->Rotate->isometric

Go into the Copy functionality:
Utilities->Copy
And apply *translation* to the *surfaces*, with *Do extrude: Volume*, as shown below.
Note that we proceed in two steps of 3 m along x in order to have a point defined at mid-span. Therefore, we apply the transformation twice (multiple copies: 2).
Select all surfaces and click on *Finish*.



The geometry of the 6 m long steel beam is now completed.



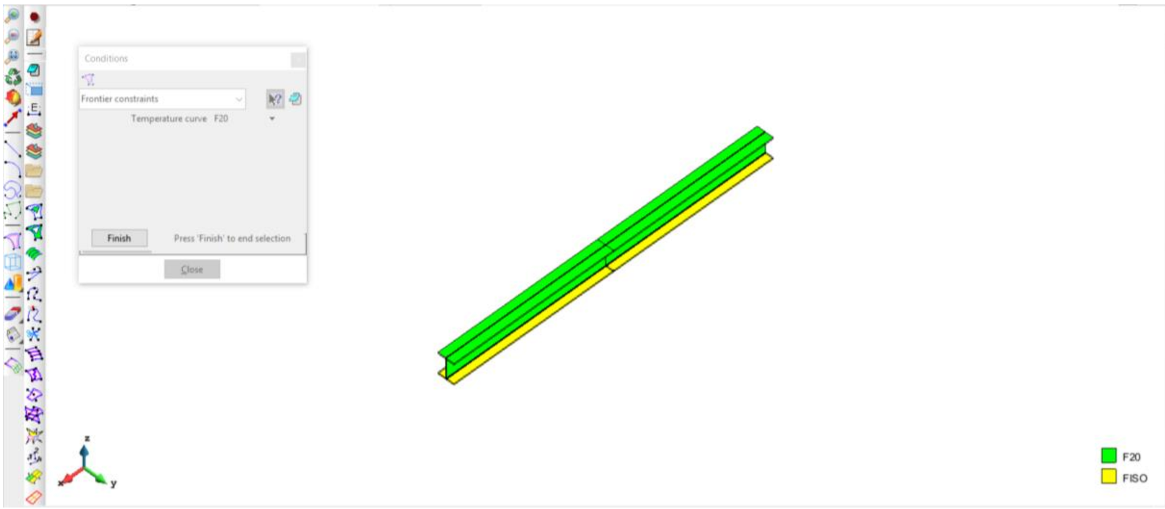
2.3. Assign the thermal conditions

In GiD, from the pull down menu select:
Data->Conditions

Note: With 3D solid thermal analyses, frontier constraints are applied on surfaces.

The ISO fire is selected by default.
Click on the ***Assign*** button and assign the fire on the lower face (8 surfaces in total).
Press ***[Esc]*** or click on ***Finish*** to confirm

Next, select ***F20*** as temperature curve
And assign it to the web and upper flange external faces. Press ***[Esc]*** to confirm.



2.4. Assign the materials

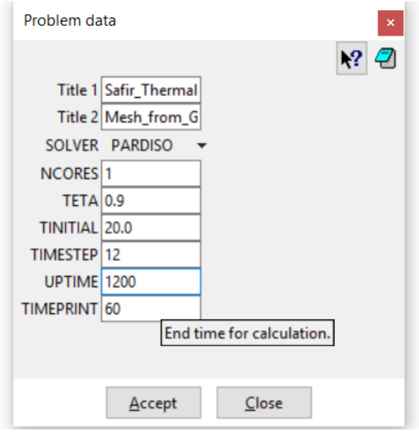
From the pull down menu select:
Data->Materials

STEEL is selected by default in the dialog box pull down list
The ***Thermal*** tab is active. Leave the parameters as they are by default, with ***STEELEC3EN*** as the material type.
Click on ***Assign-> VOLUMES*** and assign it to all volumes (there should be 20 volumes).
Press ***[Esc]*** or ***Finish*** to confirm

2.5. Assign the general data

From the pull down menu select:
Data->Problem Data

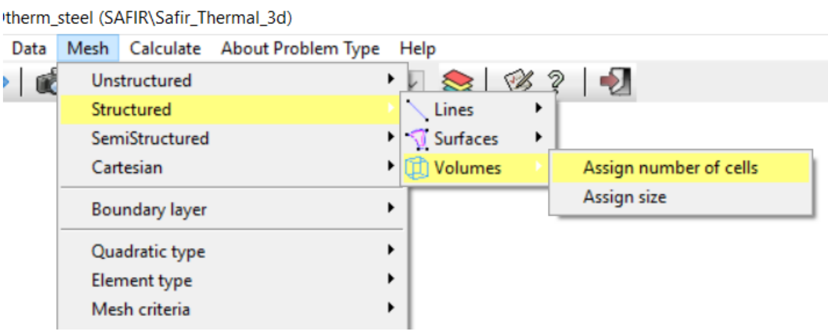
In the Problem Data dialog mask enter: TIMESTEP, UPTIME, TIMEPRINT as needed
In particular, change the UPTIME to 1200 sec



Click on the ***Accept*** data button

2.6. Create the mesh

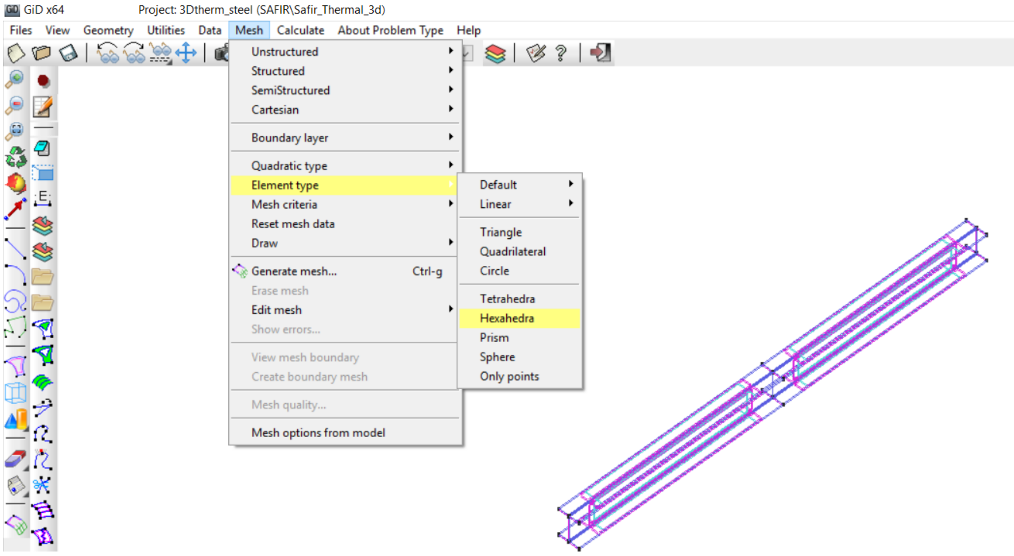
We want to use hexahedra as the FE type. This requires structuring the mesh.
Select ***Mesh->Structured->Volumes->Assign number of cells***



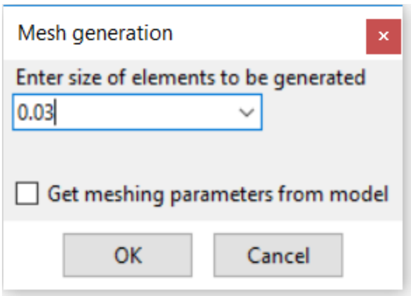
Select all volumes in the model, press ***[Esc]*** to validate. This prompts a dialog box which requires to ***Enter number of cells to assign to lines***.
Type ***40*** as the number of cells to assign to lines
Select the lines along the length of the beam (parallel to the x global axis)
Press ***[Esc]*** to validate
The text in the command box indicates: ***Assigned 40 number of cells to 40 lines***

Next, switch the view back to the YZ plane.
Assign *2* as the number of cells to assign to the thickness of the flanges, *4* to the half flange width, *1* to each half-web width, *8* to the web height.

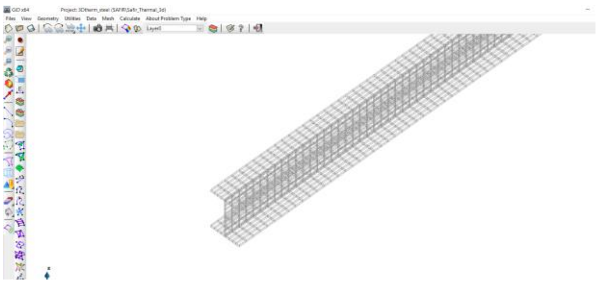
Finally, select ***Mesh->Element type->Hexahedra***
Apply to all volumes (select all volumes and press *[Esc]* to validate)



Select ***Mesh->Generate mesh or use [Ctrl + g]***
The size of elements to be generated is irrelevant since a structured mesh has been defined with specified number of element.
Validate with ***OK***



Click on ***View mesh*** to visualize the mesh



2.7. Start the calculation

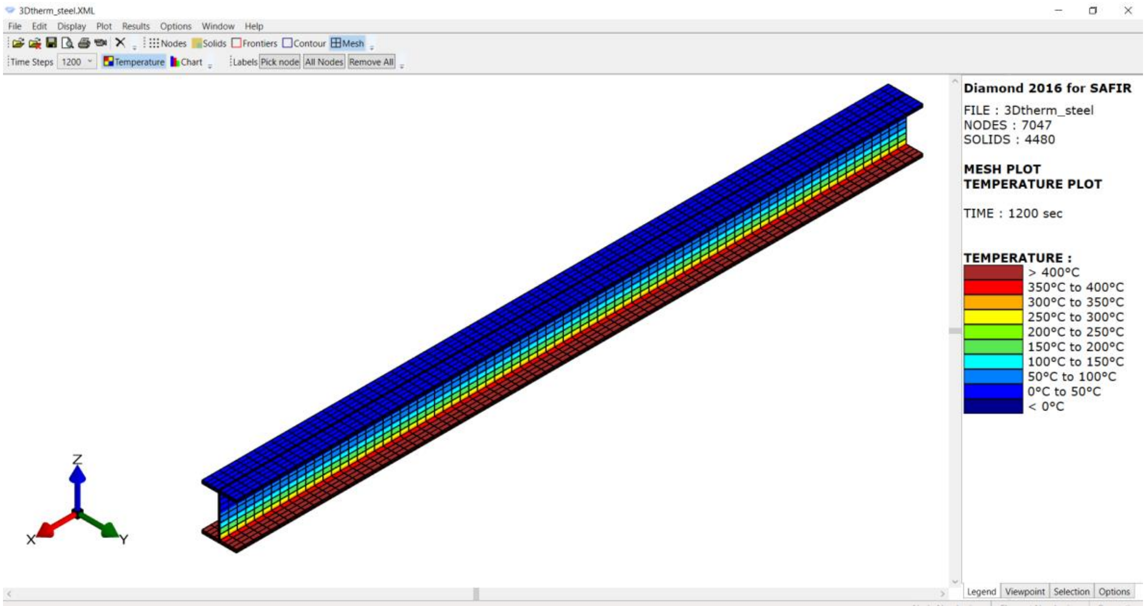
From the pull down menu select:
Calculate->Calculate window
Click the ***Start*** button
(Or select directly ***Calculate->Calculate***)

Select ***Calculate->View process info*** to follow the step-by-step calculation

GiD creates a .IN file in the project directory and starts the calculation.

2.8. Check the results

Once the calculation is done, you can open the .XML file with DIAMOND.



3. Structural model

When using 3D solid FE, the geometry and mesh is the same in the structural model as in the thermal model.
As a result, the easiest way to work on the structural model is to start from the GiD model of the thermal model.

3.1. Translate the existing thermal project into a new project for structural 3D analysis

Starting from the thermal model ***3Dtherm_steel.gid*** opened in GiD:

Save the project in its current location

Then, save it as a new project:

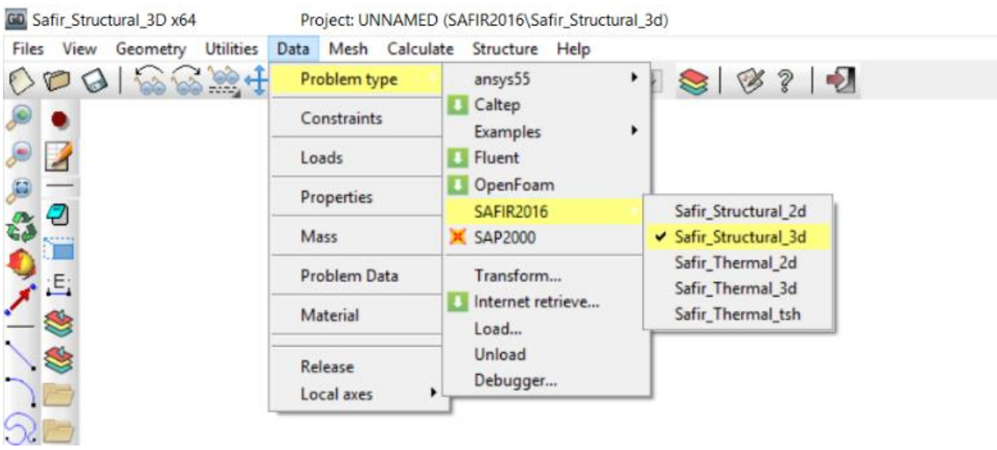
Files->Save as

Enter a file name, e.g.: ***3Dstruc_steel***

GiD creates a directory with the name ***3Dstruc_steel.gid***

From the pull down menu select:
Data->Problem type->SAFIR->Safir_Structural_3d

A dialog box open. Select *Update to new problemtype*



3.2. Copy-Paste the thermal file .OUT in the structural analysis directory

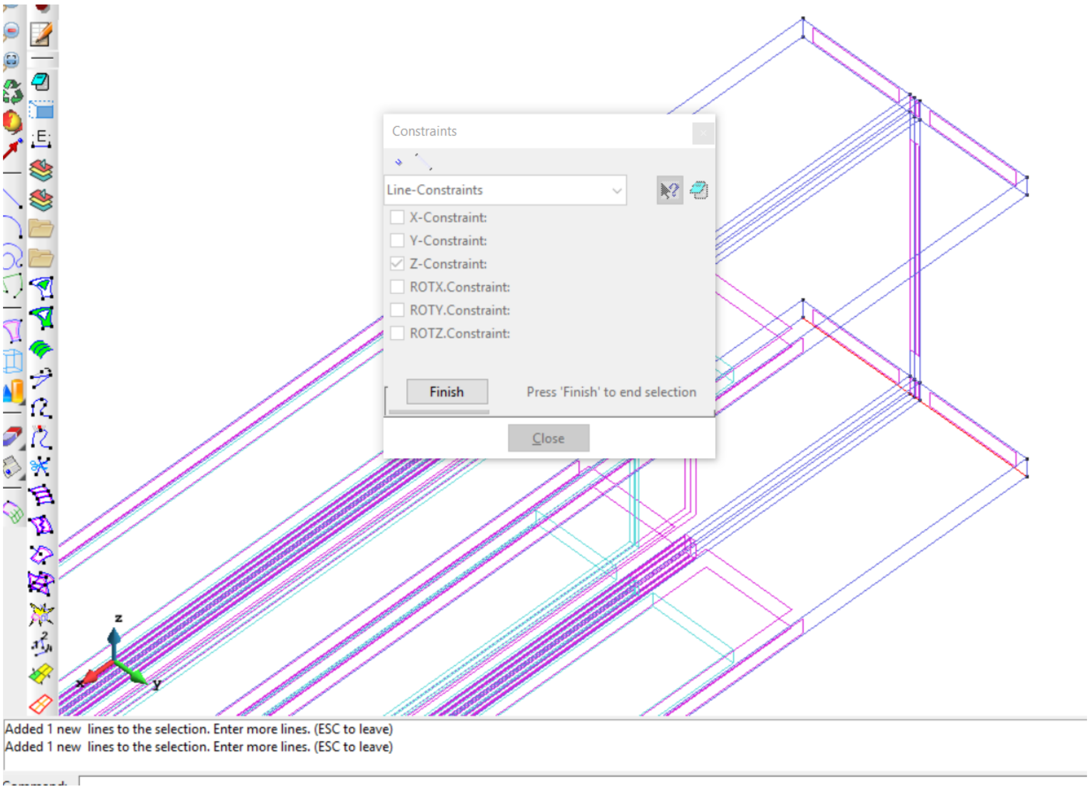
GiD has created the directory ***3Dstruc_steel.gid***
The structural input file, which will be created in this directory, will require the information from the thermal output file (which contains the temperatures). Therefore, thermal .OUT needs to be copy-pasted in the ***3Dstruc_steel.gid*** directory.

Copy and paste the files ***3Dtherm_steel.out*** in the directory ***3Dstruc_steel.gid***

3.3. Define constraints for the supports

From the pull down menu select
Mesh->Erase mesh
To come back to the model without mesh, for defining the properties

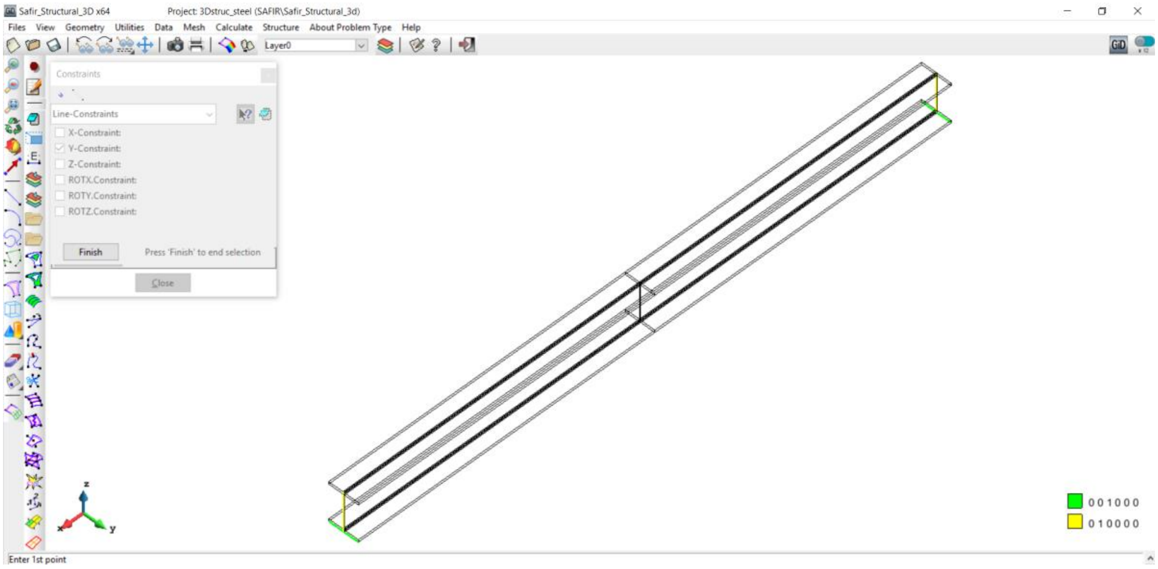
The beam is simply supported
From the pull down menu select
Data->Constraints
Select *Line Constraints*
Select Z-Constraint and assign to the lower flange lines at the two ends of the beam. Press *[Esc]*.



Select Y-Constraint and assign to the section center (vertical) lines at the two ends of the beam. Press *[Esc]*.

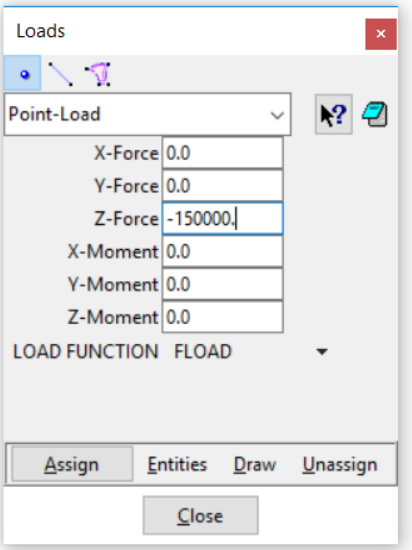
In addition, the DoF x should be blocked somewhere in the model.
Select *Point Constraint*
Select X-Constraint and assign to the lower center point of the section at the x=0 end of the beam. Press *[Esc]*.

In the dial box, with *Draw->Colors* you can display the constraints.
Press *Finish* or *[Esc]* to leave this view mode.



3.4. Assign the loads

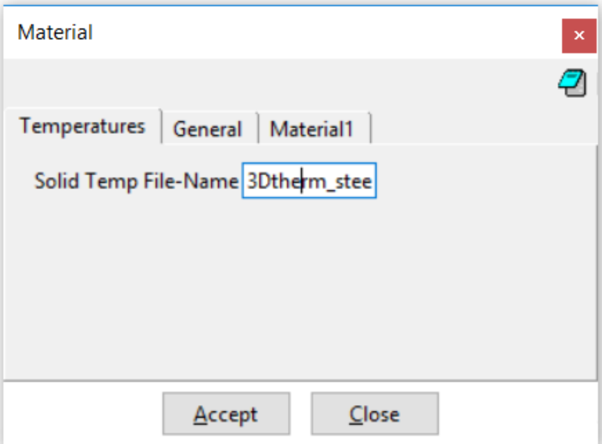
From the pull down menu select
Data->Loads
In the dial box, select *Point Load* and enter a Z-Force of -150000 N.
Select the load function FLOAD to apply the load in 20 sec.
Assign the load to the top center node at mid-span.



3.5. Define the global materials

From the pull down menu select:
Data->Material

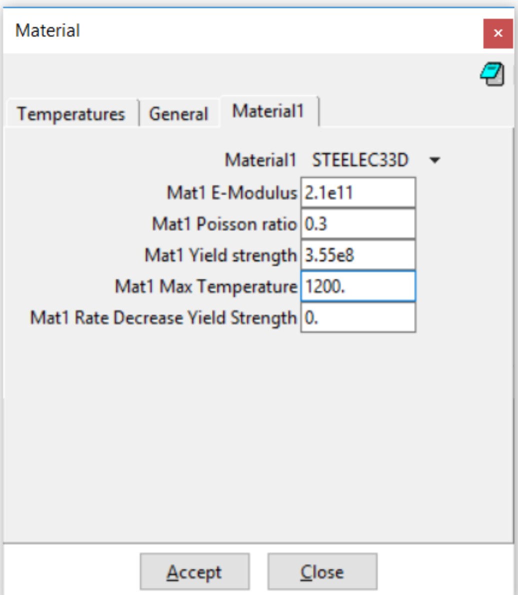
In the Temperatures tab is where the thermal .out file is associated to the structural model.
In the dialog box, type the name of the Solid Temp File: *3Dtherm_steel.out*



Next go into the General tab.
There is only 1 material.

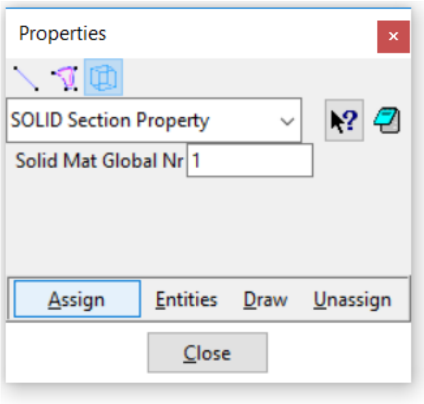
Finally, go into the Material1 tab.
Select STEELEC33D as the material model. Change e yield strength to 355 MPa. Click Accept.

<u>Note</u>: it is required to assign a tridimensional material model when using solid FE. The use of a uniaxial material model (e.g. STEELEC3) with a shell or a solid FE leads to immediate termination of the computation, this is a common source of error in the inputs.



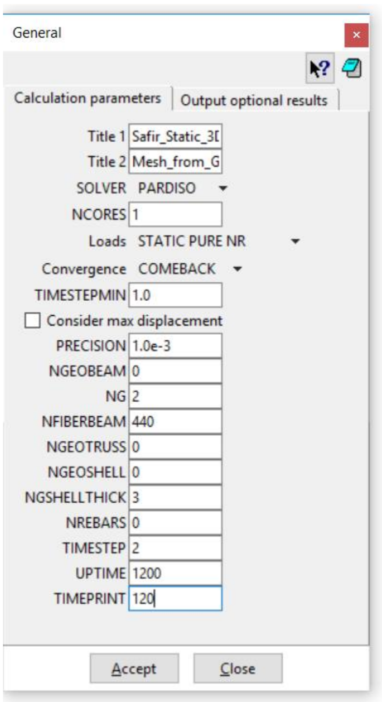
3.6. Define the properties (i.e. assign temperature files)

From the pull down menu select:
Data->Properties
In the dial box, select the volume icon to assign SOLID Section Property.
There is only 1 material in the model, so the Solid Mat Global Nr stays at 1.
Assign to all volumes in the model.



3.7. Define the general problem data

Select from the pull down menu: *Data->Problem Data*
And fill as shown below. Notably, change NGEOBEAM to 0. Click Accept.



3.8. Define the mesh

Select *Mesh -> Generate Mesh* and validate (OK). The mesh size is irrelevant since the structured mesh has previously been specified.
Select *View Mesh*

Note: the structured mesh information should have been kept from the thermal model. If it is not the case, repeat the procedure of Section 2.6. Eventually, the mesh needs to be the same as in the thermal model.

3.9. Start the calculation

From the pull down menu select:
Calculate->Calculate window
Click the *Start* button

You can follow the progress of the calculation by selecting *Calculate->View process info*

3.10. Check the results

Open the .XML file in Diamond to check the model. Plot the support conditions, applied loads, deflected shape, membrane forces, etc.