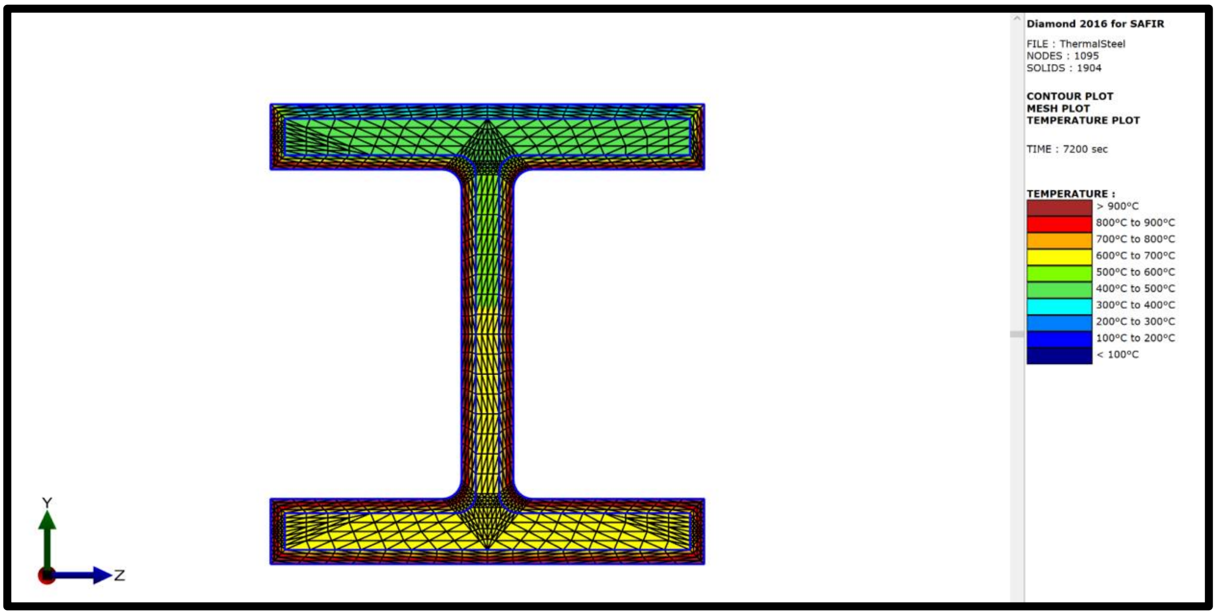
SAFIR® training session

Example: 2D thermal model of a protected steel beam

T. Gernay



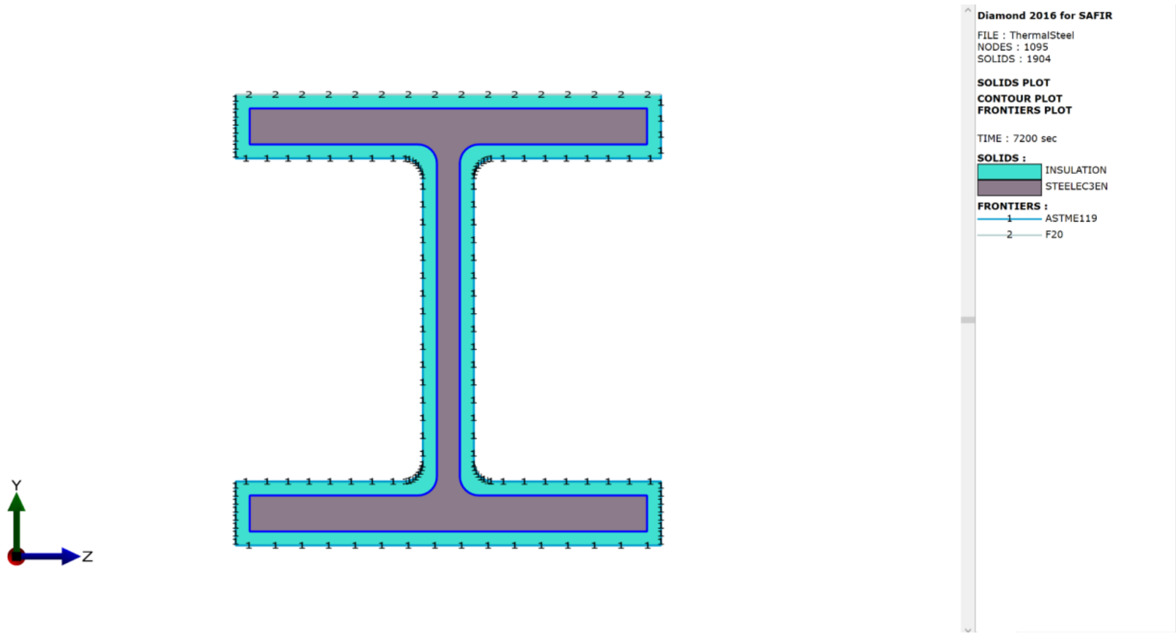
1. General description

This example deals with the thermal analysis of a protected steel cross-section.

The thermal analysis is completed in SAFIR®. The input file for the analysis is prepared using the GUI preprocessor GmSAFIR.

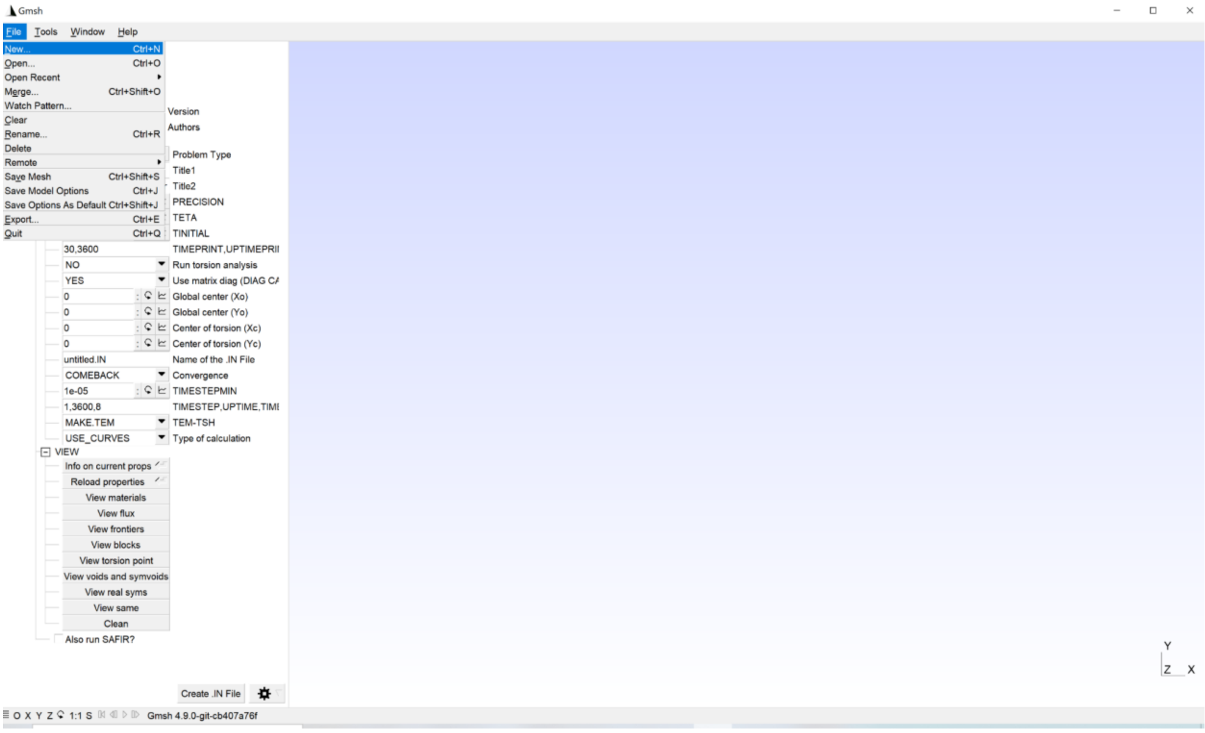
General data:

- Section W12x120
- Protected with 11 mm thickness of Sprayed Fire Resistive Material (SFRM)
- Steel material model according to Eurocode EN1993-1-2
- Section exposed to ASTM E119 fire on 3 sides
- Section in contact with atmosphere at 20°C on the side external to the fire compartment



2. Create a project in GmSAFIR

Start GmSAFIR. Select:
File -> New
Select the existing folder in which you will locate your project files.
Type the name of the project, e.g., ThermalSteel.
Select ***Save***
A window appears asking if you want to continue as-is. Select ***Use '.geo. extension***
A window appears asking which geometry kernel you want to use. Select ***Open CASCADE***



We have now created a project in GmSAFIR. In the selected folder, a file ***ThermalSteel.geo*** was created.

To build the model in GmSAFIR, we will follow a 3-steps procedure:

- Create the geometry
- Assign the SAFIR-specific properties (e.g., materials, fire curves, …)
- Build the mesh

After this procedure, we will use the command ***Create .IN file*** to generate an input file for SAFIR, which will be located in the same folder as the ***.geo*** file.

3. Create Geometry

The cross-section is a W12x120.
Dimensions are:

- d = 332 mm (13.1 in)
- bf = 312 mm (12.3 in)
- tf = 28.2 mm (1.11 in)
- tw = 18.0 mm (0.71 in)
- k = 43.2 mm (1.70 in)

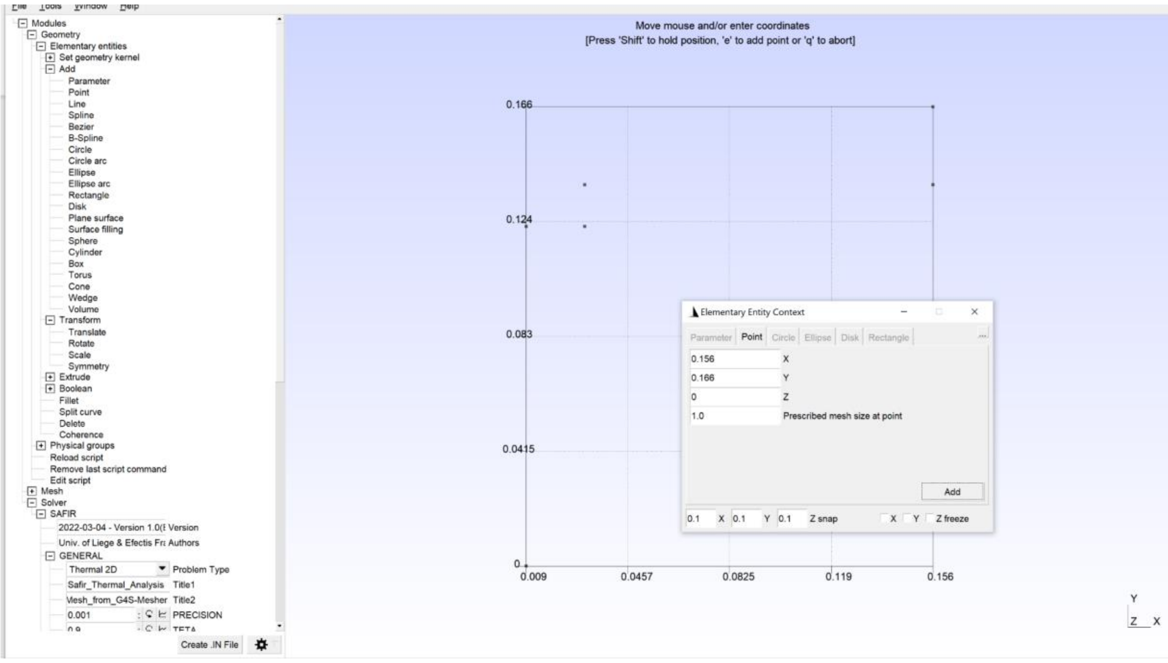
The thickness of thermal insulation is 11 mm (7/16 in).

We will create ¼ of the section and use symmetry. We will use translation to add the SFRM.

3.1. Create Points

In GmSAFIR, select ***Modules -> Geometry -> Elementary entities -> Add -> Point***
Type the coordinates of the points in meters. Select ***Add*** after typing each point.

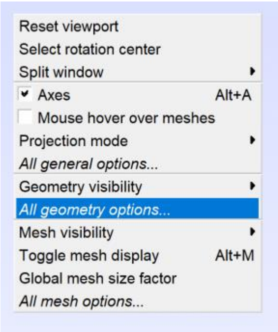
- 1st point: (0.009, 0, 0) (*tw/2*, 0, 0)
- 2nd point: (0.009, 0.1228, 0) (*tw/2*, *d/2* – *k*, 0)
- 3rd point: (0.024, 0.1228, 0) (*tw/2* + *k* – *tf*, *d/2* – *k*, 0)
- 4th point: (0.024, 0.1378, 0) (*tw/2* + *k* – *tf*, *d/2* – *tf*, 0)
- 5th point: (0.156, 0.1378, 0) (*bf/2*, *d/2* – *tf*, 0)
- 6th point: (0.156, 0.166, 0) (*bf/2*, *d/2*, 0)
- 7th point (0.000, 0.166, 0) (*0*, *d/2*, 0)

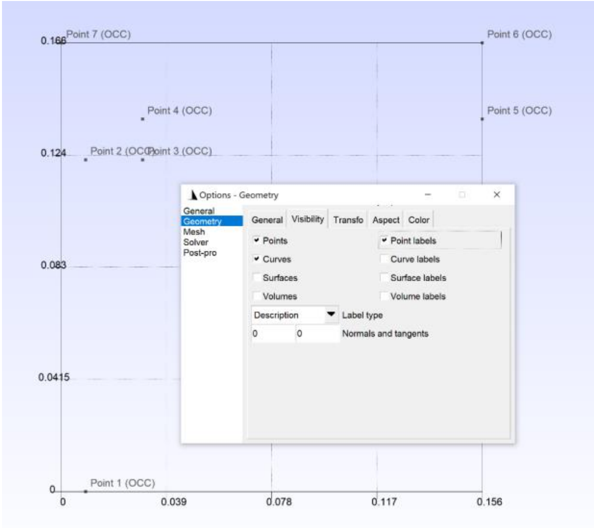


When done, type q to abort.

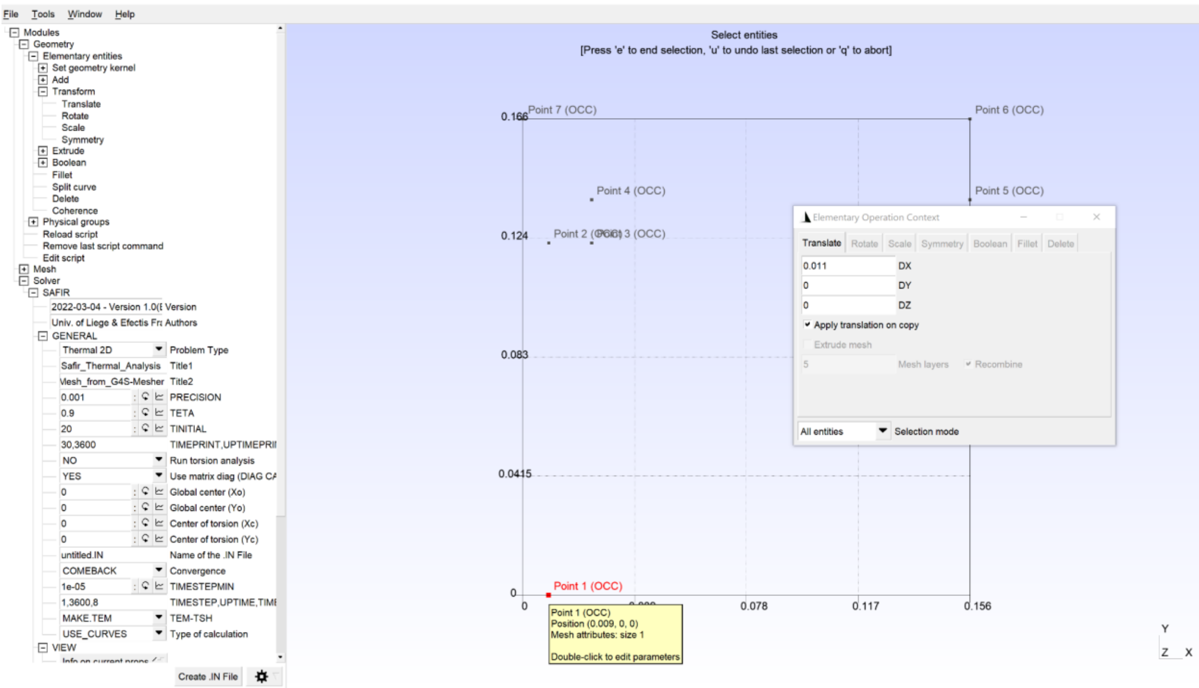
These points will be used to create the upper right corner of the steel section.

Double click on the window and select ***All geometry options*** (alternatively: ***Tools -> Options*** in the menu at the top). Tick the box to show Point Labels.





Now, we will create the points used to define the SFRM. We will use translations.
In GmSAFIR, select ***Modules -> Geometry -> Elementary entities -> Transform -> Translate***
Apply the translation:
DX 0.011, DY 0, DZ 0
Tick the box '***Apply Translation on Copy'***
Select Point 1. Type 'e'.
A new node is created from ea translation of Points by 11 mm in the X direction.



Apply the translation: ***DX 0.011, DY -0.011, DZ 0.*** Tick the box '***Apply Translation on Copy'***
Select Point 2, Point 3, Point 4 and Point 5. Type 'e'. 4 new nodes are created.

Apply the translation: ***DX 0.011, DY 0.011, DZ 0.*** Tick the box '***Apply Translation on Copy'***
Select Point 6. Type 'e'. 1 new node is created.

Apply the translation: ***DX 0., DY 0.011, DZ 0.*** Tick the box '***Apply Translation on Copy'***
Select Point 7. Type 'e'. 1 new node is created.

When done, type q to abort.

These new points will be used to create the SFRM layer on the upper right corner of the steel section.

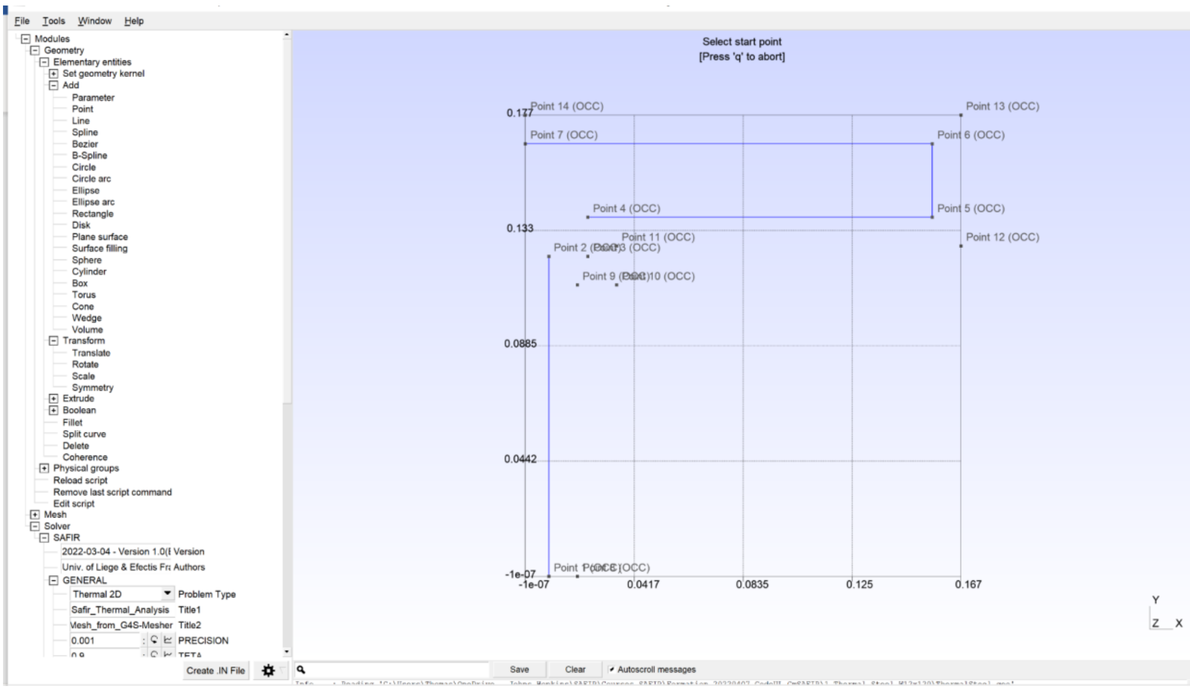
3.2. Create Lines

In GmSAFIR, select ***Modules -> Geometry -> Elementary entities -> Add -> Line***
Select start point of the line.

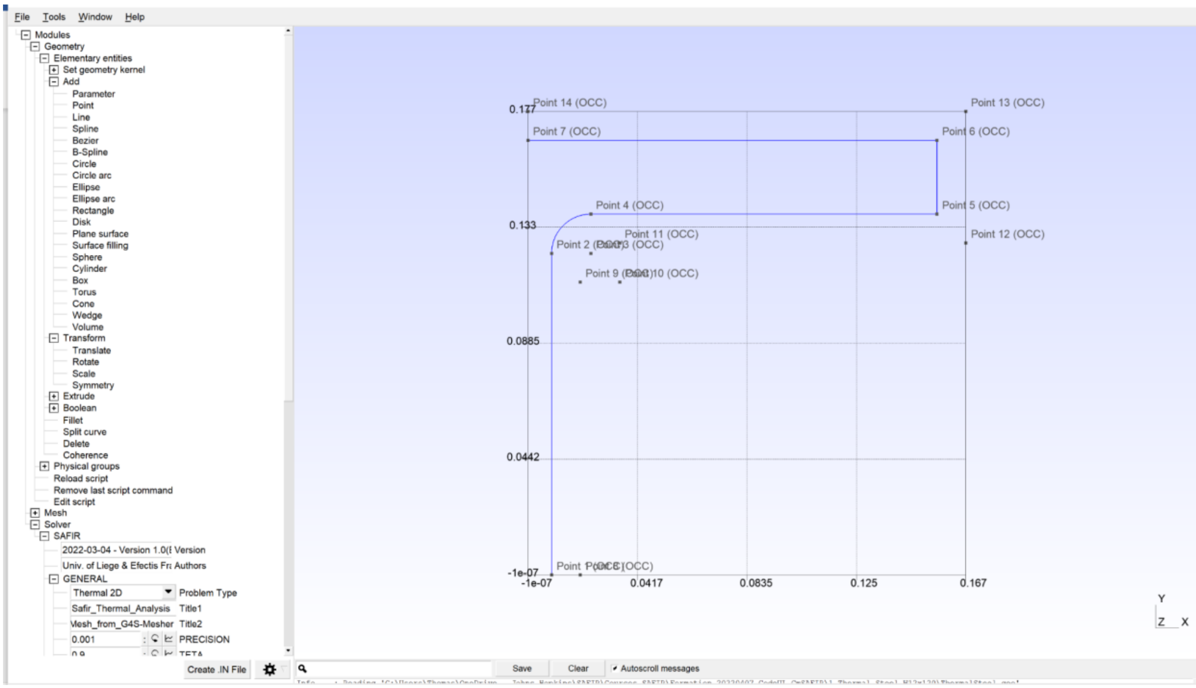
> Lines must be defined such that, going from the first point to the second point defining the line, the surface bounded by the line is located on the left side of the line. Lines must therefore be defined by turning counterclockwise around the surface.

Select Point 1 as start point. Select Point 2 as end point. A line was created joining Points 1-2.

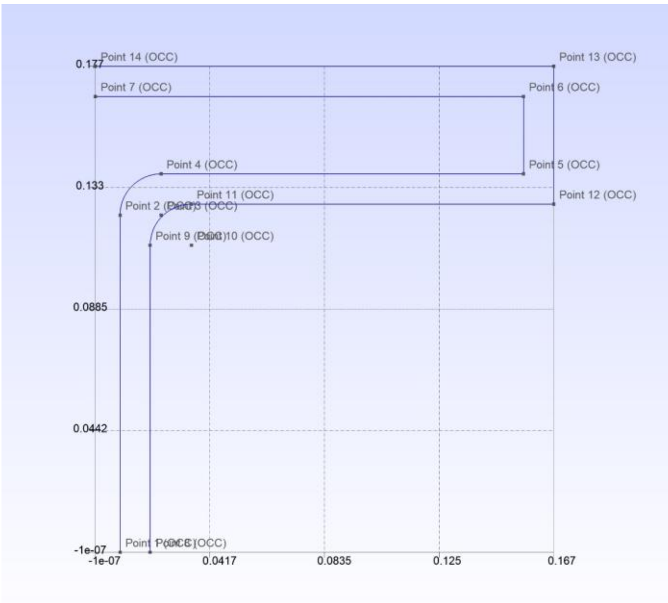
Repeat with Points 4 and 5. Then Points 5 and 6. Then Points 6 and 7. Press q to abort.



Now, we will create the root fillet.
In GmSAFIR, select *Modules -> Geometry -> Elementary entities -> Add -> Circle arc*
Select start point: Point 2. Select center point: Point 3. Select end point: Point 4.
The root fillet of the steel section has been created. Type q to abort.



We repeat the procedure for line creation and circle arc creation for the SFRM.
Starting with the lines. Select *Modules -> Geometry -> Elementary entities -> Add -> Line*
We create a line from Point 8 to 9, from Point 11 to Point 12, from 12 to 13, from 13 to 14. Type q to abort.
Then we select *Modules -> Geometry -> Elementary entities -> Add -> Circle arc*
We select Point 9, 10, and 11 to create the SFRM layer on the root fillet. Press q.

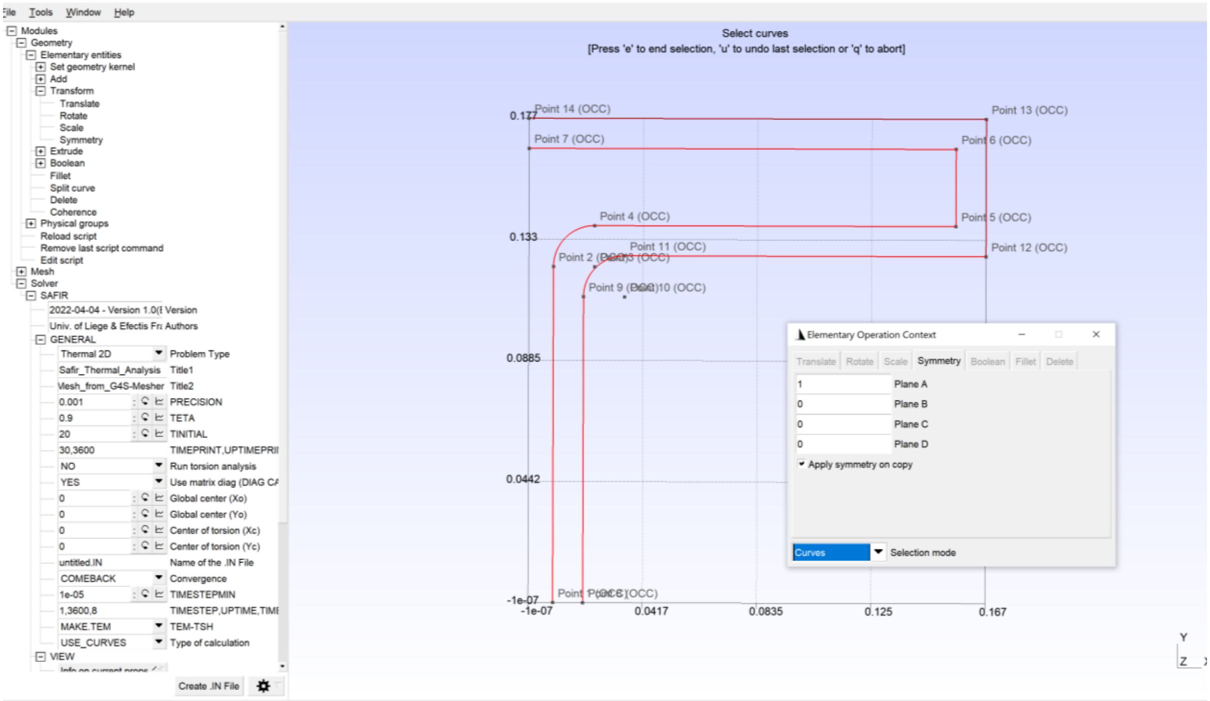


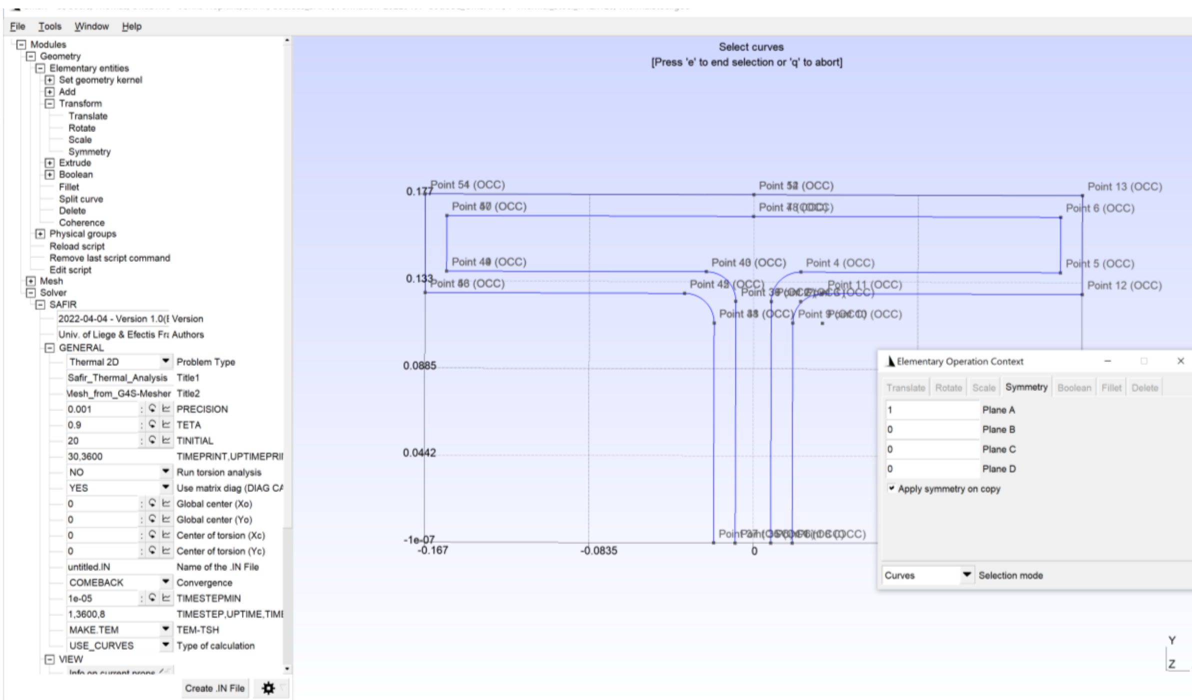
3.3. Use Symmetry

In GmSAFIR, select *Modules -> Geometry -> Elementary entities -> Transform -> Symmetry*

The X direction corresponds to Plane A.
Type ‘1’ for Plane A. ‘0’ for Planes B, C, D.
Tick the box ‘*Apply Translation on Copy’*
Select ‘*Curves’* for the selection mode

To select all the points and lines, press ‘ctrl’ + left click on one corner, drag the mouse to the other corner, then again ‘ctrl’ + left click. Everything should be selected. Then type ‘e’.



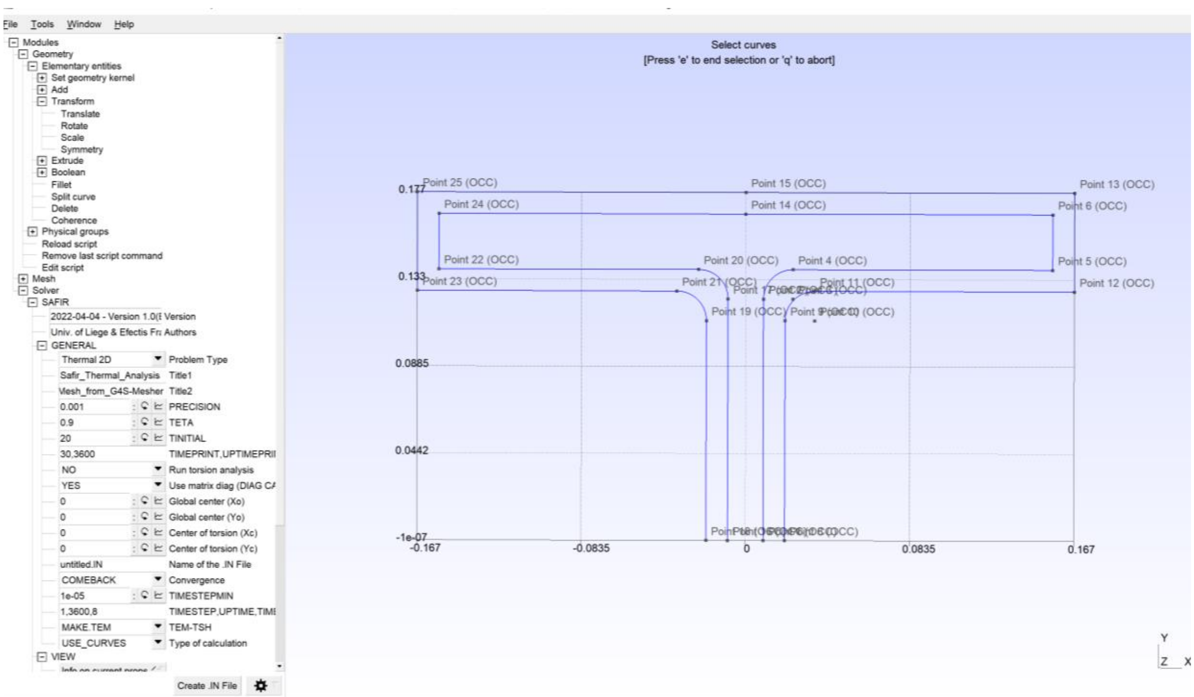


We now have half of the section. However, the *Symmetry* action has resulted in duplication of nodes. It is observed by showing node labels. To address this, there is a specific command in GmSAFIR to 'merge' duplicated entities.

In GmSAFIR, select *Modules -> Geometry -> Elementary entities -> Boolean -> Coherence*

Then, select *Reload script*

This has the effect of merging the duplicated points. It is verified through the node labels.



Now, we apply a symmetry about the Y-axis (Plane B) to create the bottom half of the section.

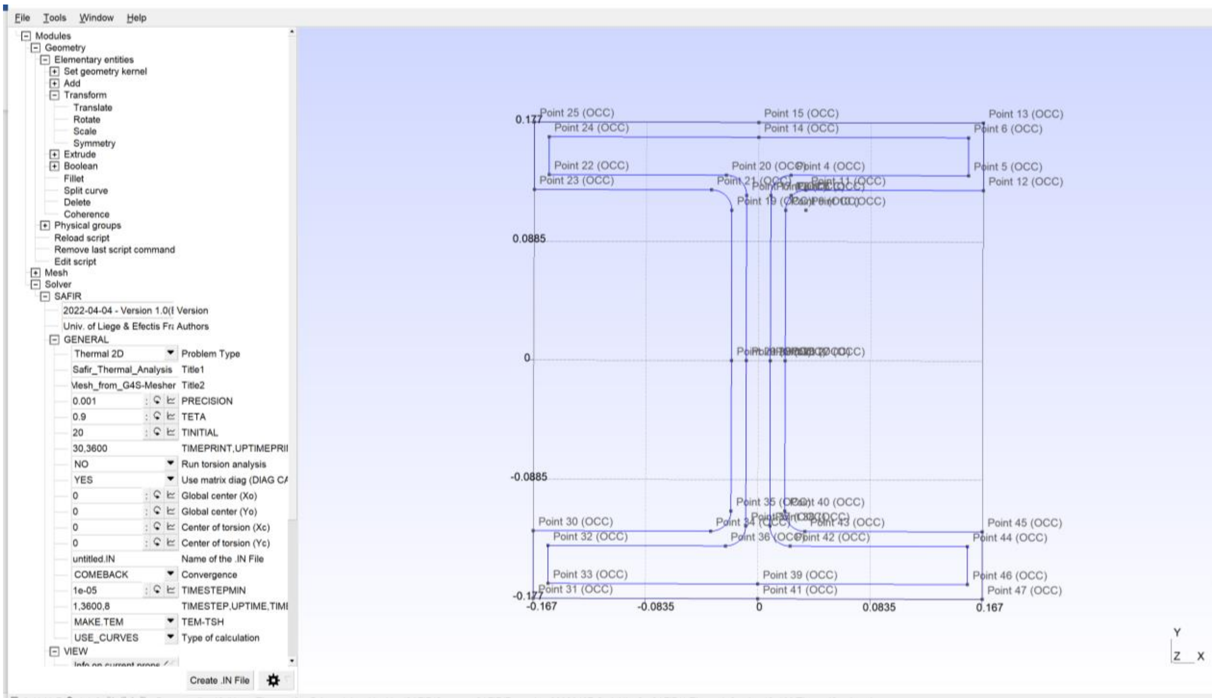
In GmSAFIR, select ***Modules -> Geometry -> Elementary entities -> Transform -> Symmetry***

Type '0' for Plane A, '1' for Plane B, '0' for Plane C, '0' for Plane D.
Tick the box '***Apply Translation on Copy'***. Select '***Curves'*** for the selection mode
Select everything. Type 'e' to end selection. The symmetry is applied and the bottom half of the section appears.
Close the symmetry window and type 'q'.

Then, select ***Modules -> Geometry -> Elementary entities -> Boolean -> Coherence***

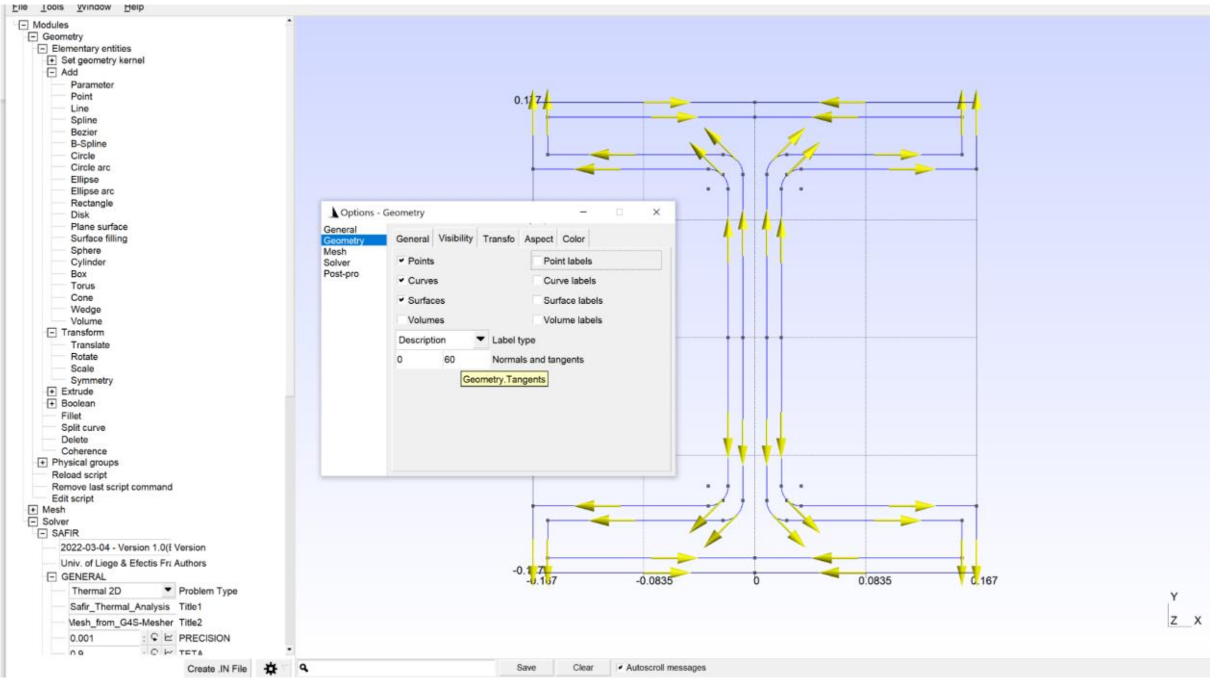
Then, select ***Reload script***

This has the effect of merging the duplicated points. Show the node labels to check that nodes are not in duplicate.



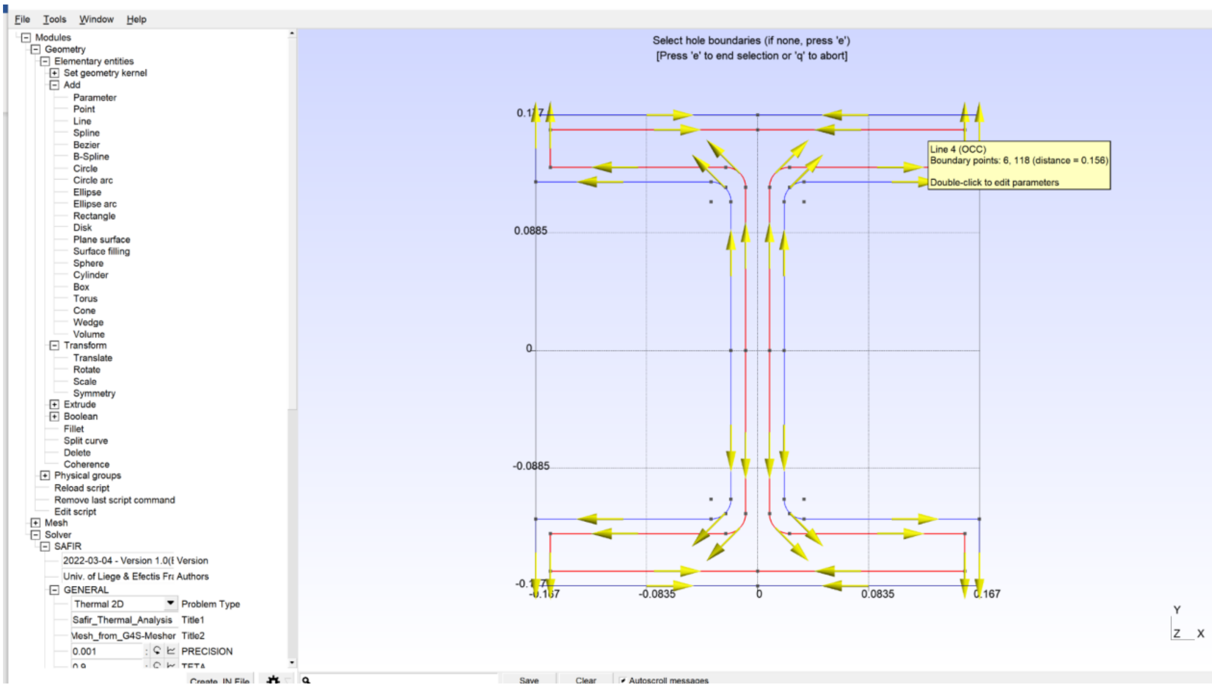
3.4. Create Surfaces

The first line that is selected must be oriented in the appropriate direction, i.e., with the surface on its left (counterclockwise rotation). It is possible to draw the line directions.
Select ***Tools -> Options*** in the menu at the top.
Type a value in the right box ***Normals and tangents*** (or click in the box and drag to the right) to make the line directions appear.
Now, when creating the surface, we can select as the first line of the surface a line which would rotate in a counterclockwise direction around the surface.



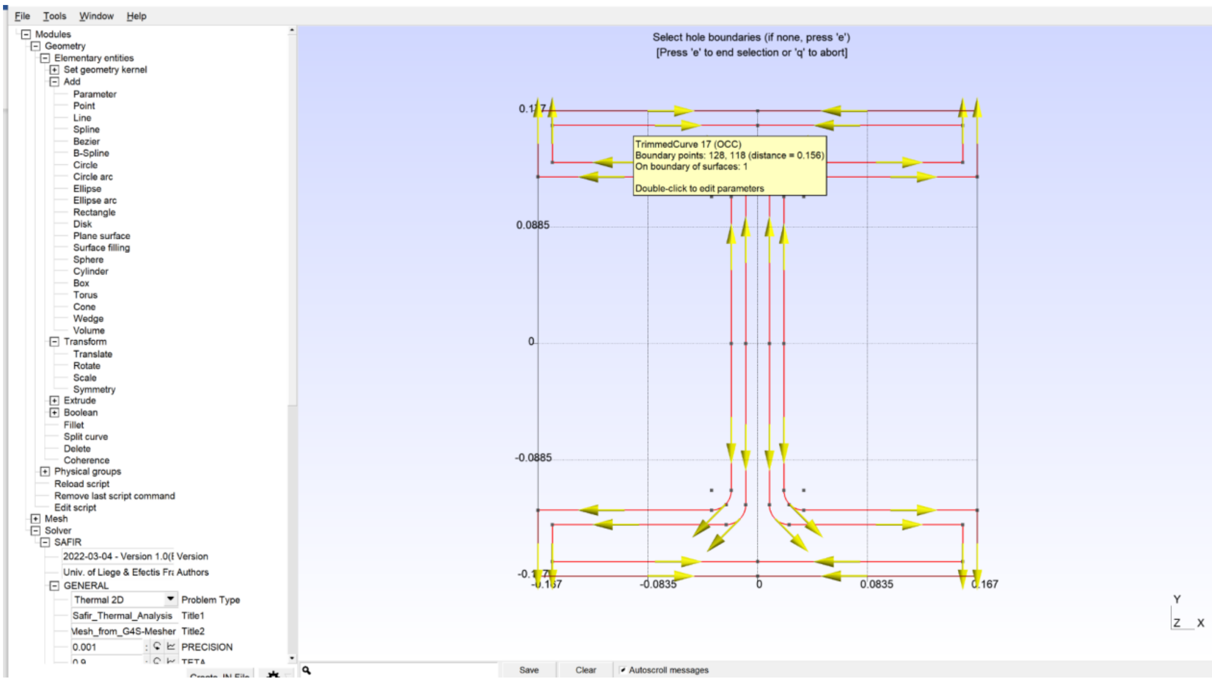
In GmSAFIR, select ***Modules -> Geometry -> Elementary entities -> Add -> Plane surface***

Starting with the steel section, select for example Line 4.
GmSAFIR automatically selects the other boundaries of the surface because, in this case, there is only one possible surface bounded by that line (using the counterclockwise rule).
Press 'e'. The surface was created.



Now for the SFRM, select Line 9 at the top right. The external boundary is selected automatically, but the lines that form the interior boundary must also be selected. Select Line 17 of the top left flange to satisfy the counterclockwise rule for the interior boundary of the SFRM surface.

Press ‘e’. The surface was created. Press ‘q’ to abort.



Surface labels can be visualized in ***All geometry options.*** The two surface labels lie on top of each other, but it can be checked that two surfaces have been created.

This concludes the creation of the geometry.

4. Assign Properties

In this step, we will define the properties of the model and the inputs necessary for the SAFIR calculation. We will use physical groups to assign the properties of the model to the lines and surfaces.

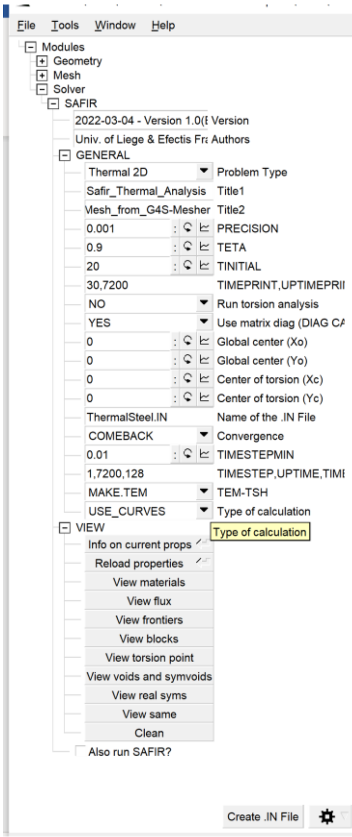
.

4.1. Define the SAFIR General Input Data

In GmSAFIR, select ***Modules -> Solver -> SAFIR -> General***

Here we can define the inputs of the SAFIR calculation.
Set the ***Problem Type*** to Thermal 2D
Set the ***UPTIMEPRINT*** to 7200 sec
Set the ***Name of the .IN File*** as ThermalSteel.IN
Set the ***TIMESTEPMIN*** to 0.01 sec
Set the ***UPTIME*** to 7200 sec and the ***TIMESTEPMAX*** to 128 sec

The ***Type of calculation*** gives different options for interfacing a fire with the thermal analysis. Here, we will use a time-temperature curve, but it is also possible to use localized fires and CFD outputs.

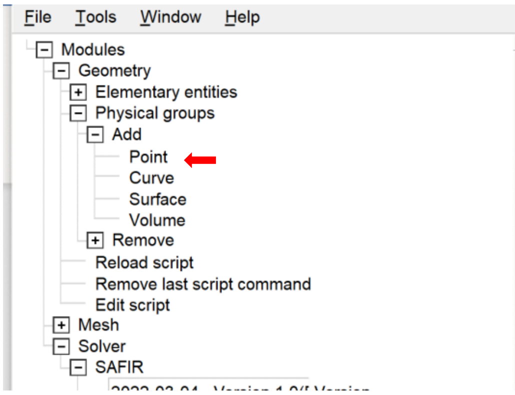


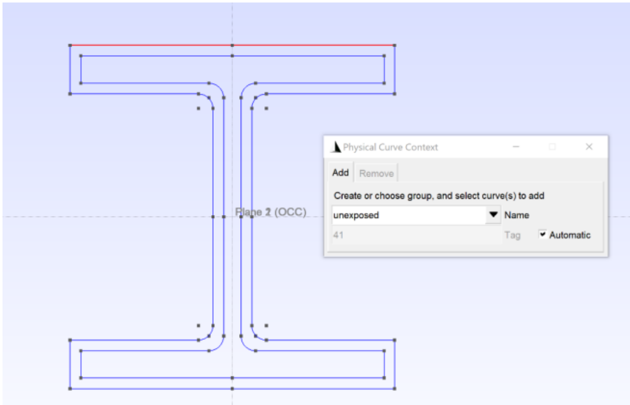
.

4.2. Define the Physical Group Properties: Fire Boundary Conditions

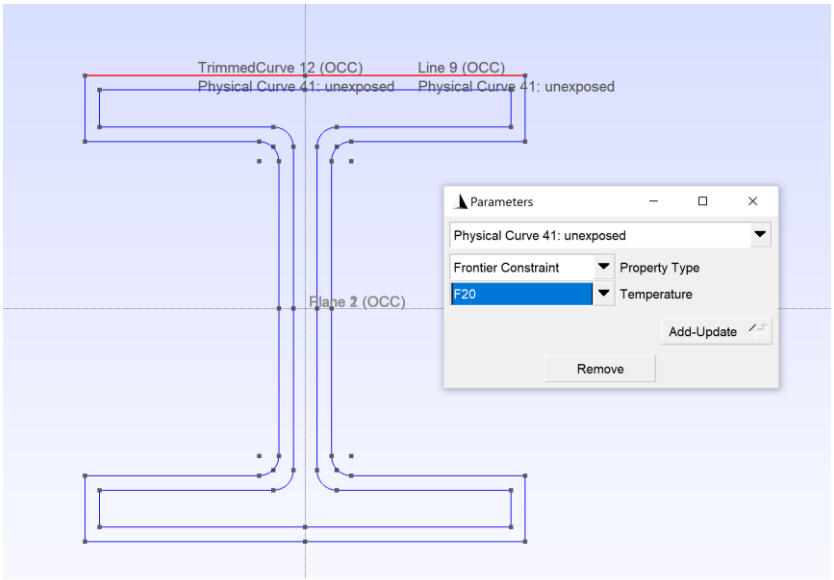
We will start by applying the thermal boundary conditions (e.g., fire curves).

In GmSAFIR, select ***Modules -> Geometry -> Physical Groups -> Add -> Curve***



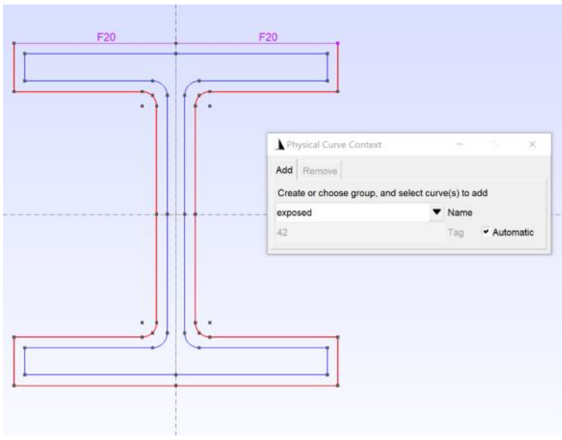


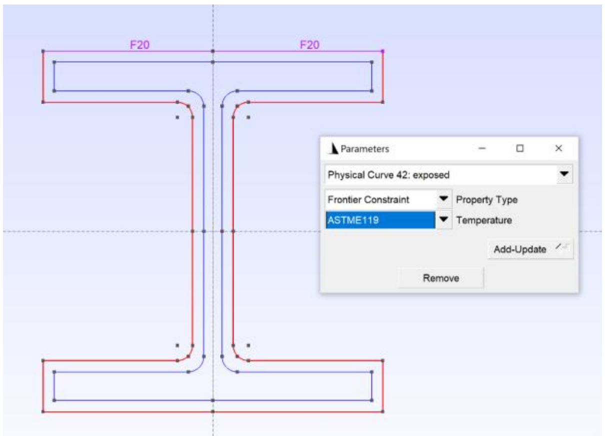
Type the name ‘unexposed’.
Select the two curves at the top, which are not exposed to the fire.
The new physical group ‘unexposed’ which we created now contains two elementary entities, Lines 9 and 12.
Type ‘e’.
The window changes and enables selecting the properties for the ‘unexposed’ group.
Select ‘frontier constraint’ and ‘F20’.
Click ‘Add-Update’ to confirm.



We can close the window.
To check, select ***Modules -> Solver -> SAFIR -> View -> View frontiers***
The frontier ‘F20’ should be displayed on the two top lines.

We repeat the procedure to create a second physical group, for the ‘exposed’ curves.
In GmSAFIR, select ***Modules -> Geometry -> Physical Groups -> Add -> Curve***
Type the name ‘exposed’.
Select the curves at the boundary of the section, except Lines 9 and 12 which are not exposed to the fire.
The new physical group ‘exposed’ now contains the relevant elementary entities.
Type ‘e’.
The window changes and enables selecting the properties for the ‘unexposed’ group.
Select ‘frontier constraint’ and ‘ASTME119’.
Click ‘Add-Update’ to confirm.





4.3. Define the Physical Group Properties: Materials

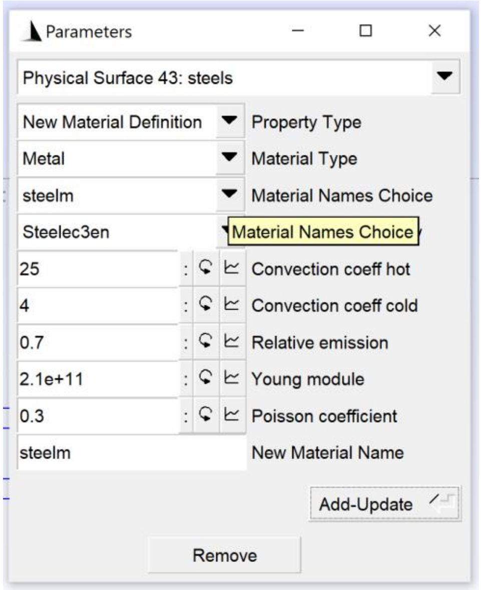
We will now define surface physical groups to assign the material properties.

In GmSAFIR, select ***Modules -> Geometry -> Physical Groups -> Add -> Surface***

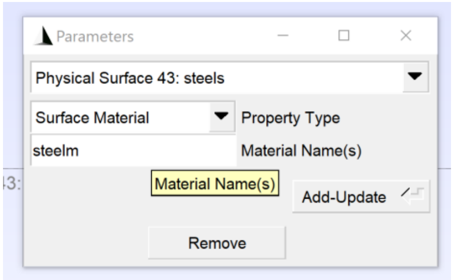
Type the name ‘steels’.
Select the steel surface.
The new physical group ‘steels’ which we created now contains one surface.
Type ‘e’.
The window changes and enables selecting the properties for the ‘steels’ group.

Materials are not defined yet. In ***Property Type***, select ***New Material Definition***.
Material Type: select ‘metal’
Material sub-category: select ‘steelec3en’
The thermal properties can be left to their default value.
New material name: type ‘steelm’

Click ‘add update’. As a result, a new entry appears, ‘Material Names Choice’. The material ‘steelm’ is now available to be attributed to a physical group.

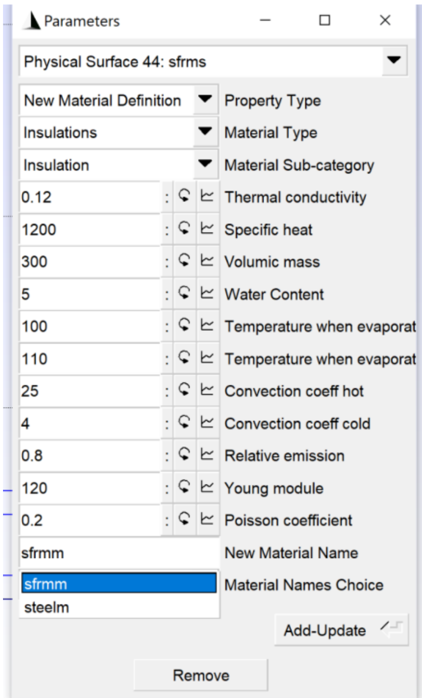


In Property Type, select ‘Surface Material’.
In Material Name, type ‘steelm’
Click ‘Add-Update’.
This material has now been allocated to the physical group that contains Surface 1.
This can be checked by selecting ***Modules -> Solver -> SAFIR -> View -> View materials***.



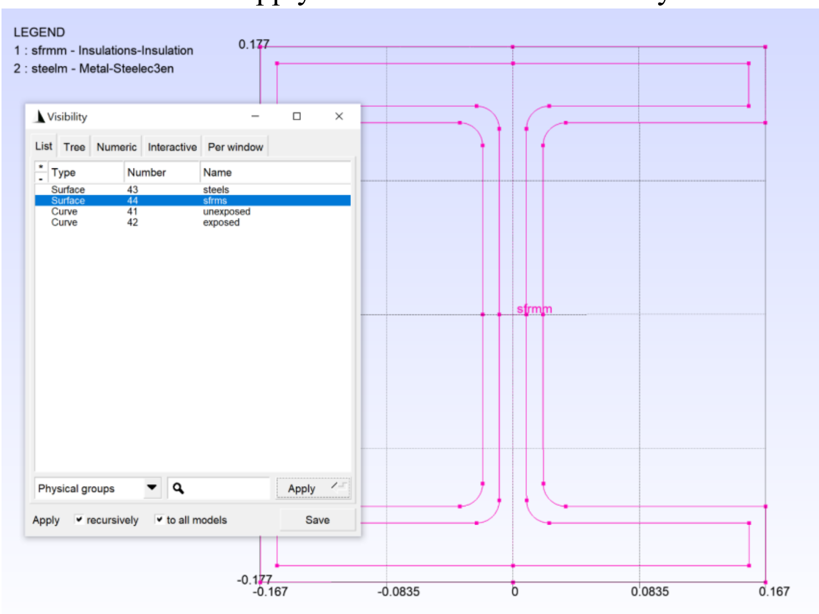
We will now define the physical group for the SFRM.
In GmSAFIR, select ***Modules -> Geometry -> Physical Groups -> Add -> Surface***
Type the name ‘sfrms’.
Select the SFRM surface.
The new physical group ‘sfrms’ which we created now contains one surface.
Type ‘e’.
The window changes and enables selecting the properties for the ‘sfrms’ group.
The material is not defined yet. In ***Property Type***, select ***New Material Definition***.
Material Type: select ‘insulations’
Material sub-category: select ‘insulation’
The thermal properties can be input as desired.
Thermal conductivity: 0.12 W/mK.
Specific heat: 1200 J/kgK
Density: 300 kg/m^3
New material name: type ‘sfrmm’
Click ‘Add-Update’.

The material has been added in the 'Material Names Choice'.



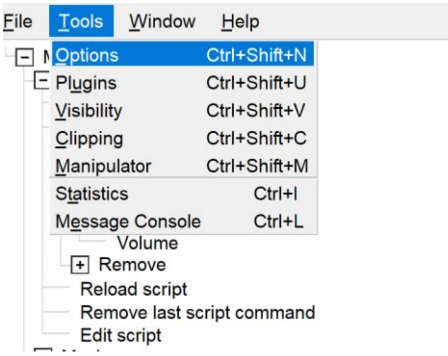
As Property Type, select 'Surface Material'
In Material Name, type 'sfrmm'
Click 'Add-Update'.
This material has now been allocated to the physical group that contains Surface 1.
This can be checked by selecting ***Modules -> Solver -> SAFIR -> View -> View materials***

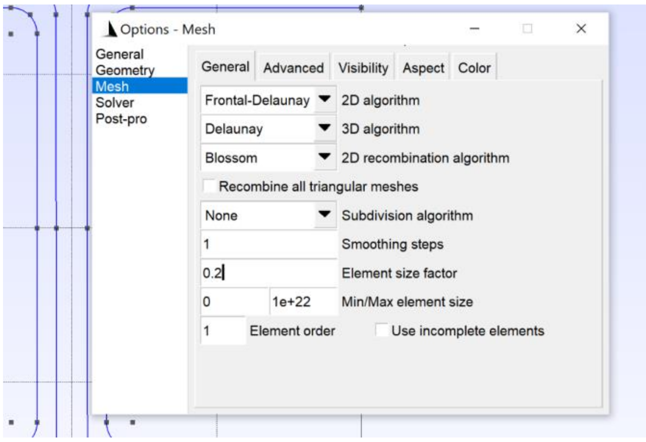
Another useful visualization is in ***Tools -> Visibility*** (or ctrl + shift + v)
In 'List' the physical groups are shown. There are two for surfaces and two for curves. We can select the surface 'sfrms' and apply to visualize the SFRM only.



5. Create the Mesh

In GmSAFIR, select ***Tools -> Options***
In the window, select ***Mesh -> General*** and change the element size factor to 0.2.
Close the window.



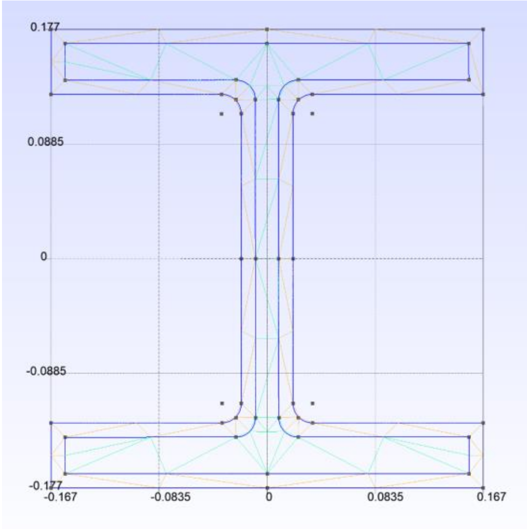


Next, select ***Modules -> Mesh -> 2D***

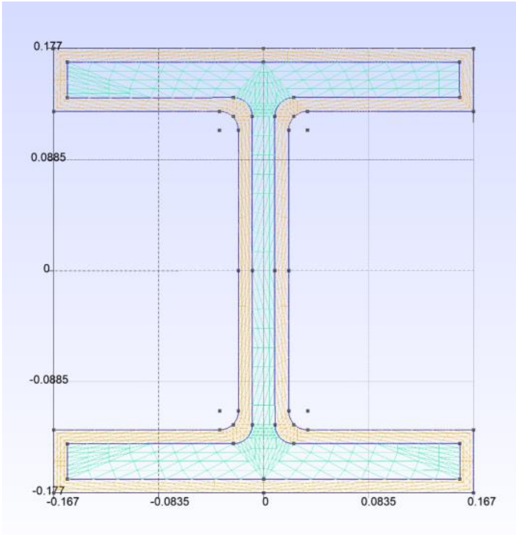
A mesh is automatically generated.
As this mesh is coarse, select ***Modules -> Mesh -> Refine by splitting***
We can refine twice.
We now have a mesh based on triangular elements.

Modules -> Mesh -> 2D

(with element size factor 0.2)



Modules -> Mesh -> Refine by splitting (x2)



GmSAFIR offers various options to define and optimize the mesh, which are not covered here. For example, the command ***Modules -> Mesh -> Recombine 2D*** transforms triangular elements into quadrangular elements, but we will not use it here.

6. Create SAFIR input file and run the file

In GmSAFIR, select *Modules -> Solver -> Create .IN File* (at the bottom of the menu)

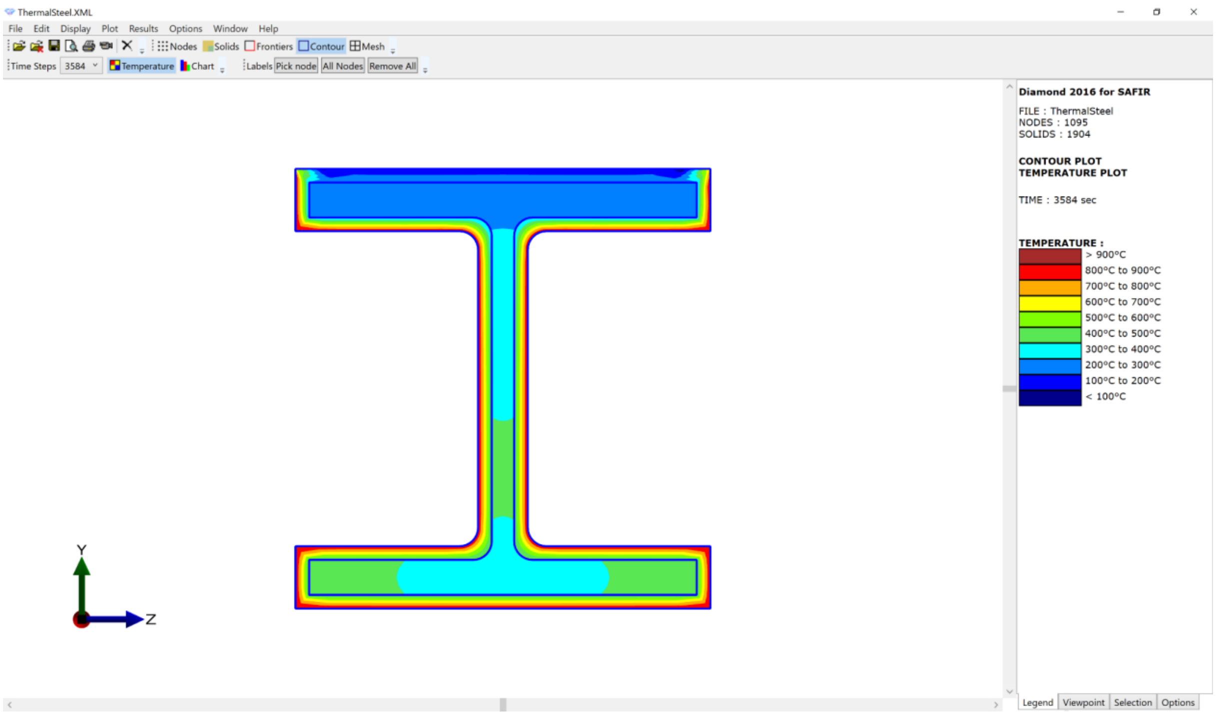
The input file ThermalSteel.IN has been generated in the folder.

Name	Status	Date modified	Type	Size
ThermalSteel.g4s	✓	3/31/2022 9:47 PM	G4S File	3 KB
ThermalSteel.geo	✓	3/31/2022 9:36 PM	GEO File	3 KB
ThermalSteel.IN	✓	3/31/2022 9:55 PM	IN File	60 KB

Run this input file with SAFIR. This can be done by using the command window of the executable, or the SAFIR Shell tool.

Name	Status	Date modified	Type	Size
SAFIR2022a0.exe		3/30/2022 9:22 AM	Application	108,901 KB
ThermalSteel.g4s	✓	3/31/2022 9:47 PM	G4S File	3 KB
ThermalSteel.geo	✓	3/31/2022 9:36 PM	GEO File	3 KB
ThermalSteel.IN	✓	3/31/2022 9:55 PM	IN File	60 KB
ThermalSteel.OUT		3/31/2022 9:58 PM	spice file	1,295 KB
ThermalSteel.TEM		3/31/2022 9:58 PM	TEM File	867 KB
ThermalSteel.XML		3/31/2022 9:58 PM	XML Document	1,282 KB

Open the postprocessor DIAMOND to visualize the results.



DIAMOND can be used to check the geometry, boundary conditions, materials, mesh, etc. as well as to plot charts of the temperature at the nodes and draw temperatures on the section.

7. Creating a configurable GmSAFIR script

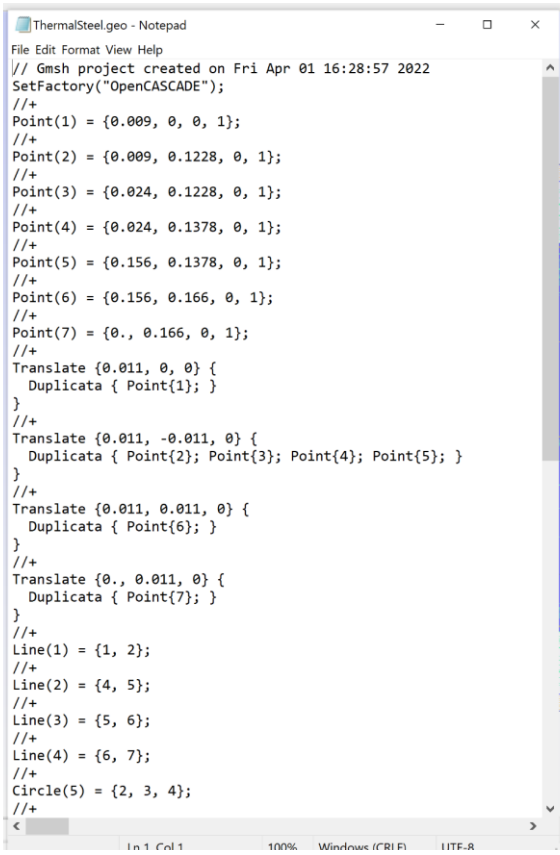
GmSAFIR allows reading and modifying the script at each step of the procedure of creating the model.

In GmSAFIR, select ***Modules -> Geometry -> Edit script*** to visualize the script in a text editor.

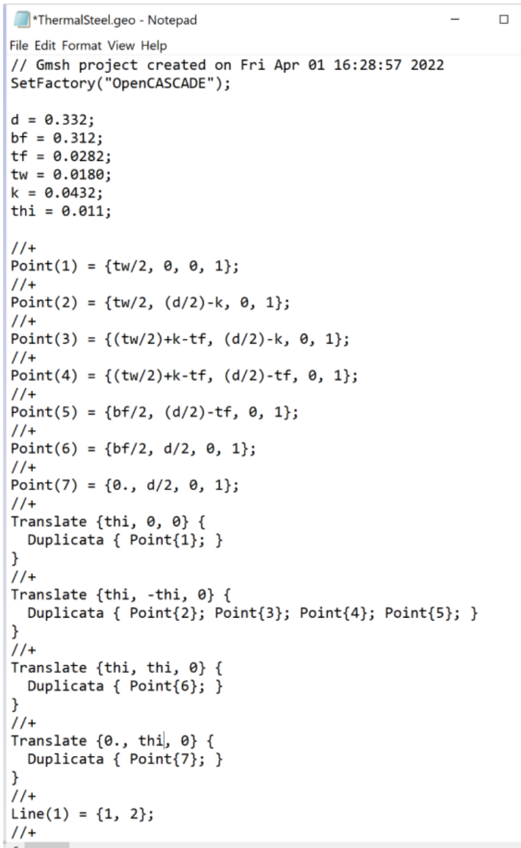
If a mistake is made at any step of the procedure (creation of geometry, physical groups), **the script can be edited in the text editor**. It can then be reloaded using:
Modules -> Geometry -> Reload script
and the modification will take effect.

Besides the correction of mistake, another interesting functionality is that **the model can be made parametric using the script**.

In GmSAFIR, select ***Modules -> Geometry -> Edit script***
We create variables and make the points definition parametric. The original script is on the left. The modified one is on the right. Both the 'Points' and 'Translate' commands now are defined as functions of the variables, which are the W12x120 dimensions and the thickness of SFRM.


ThermalSteel.geo - Notepad
File Edit Format View Help

```
// Gmsh project created on Fri Apr 01 16:28:57 2022
SetFactory("OpenCASCADE");
//+
Point(1) = {0.009, 0, 0, 1};
//+
Point(2) = {0.009, 0.1228, 0, 1};
//+
Point(3) = {0.024, 0.1228, 0, 1};
//+
Point(4) = {0.024, 0.1378, 0, 1};
//+
Point(5) = {0.156, 0.1378, 0, 1};
//+
Point(6) = {0.156, 0.166, 0, 1};
//+
Point(7) = {0., 0.166, 0, 1};
//+
Translate {0.011, 0, 0} {
  Duplicata { Point{1}; }
}
//+
Translate {0.011, -0.011, 0} {
  Duplicata { Point{2}; Point{3}; Point{4}; Point{5}; }
}
//+
Translate {0.011, 0.011, 0} {
  Duplicata { Point{6}; }
}
//+
Translate {0., 0.011, 0} {
  Duplicata { Point{7}; }
}
//+
Line(1) = {1, 2};
//+
Line(2) = {4, 5};
//+
Line(3) = {5, 6};
//+
Line(4) = {6, 7};
//+
Circle(5) = {2, 3, 4};
//+
```

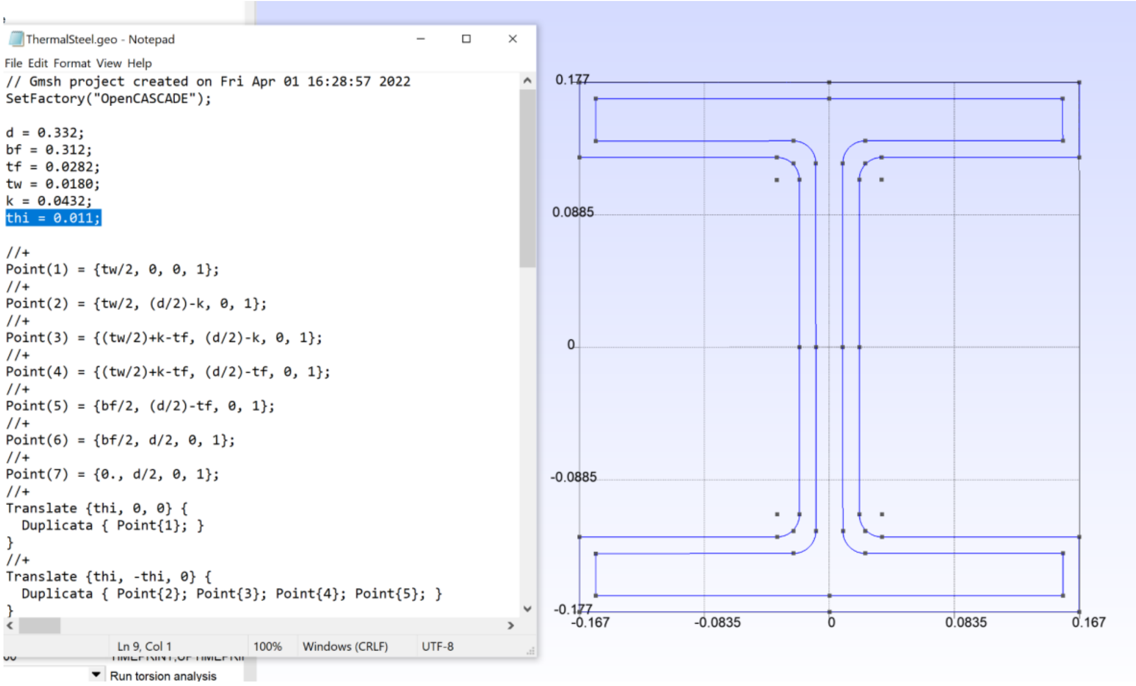

*ThermalSteel.geo - Notepad
File Edit Format View Help

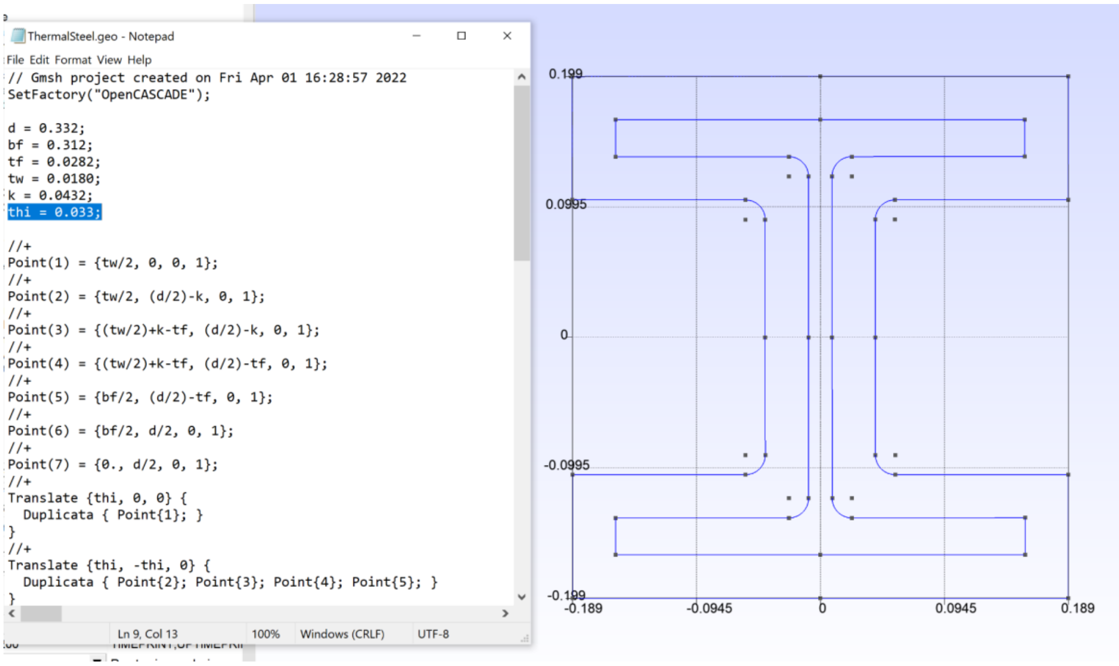
```
// Gmsh project created on Fri Apr 01 16:28:57 2022
SetFactory("OpenCASCADE");

d = 0.332;
bf = 0.312;
tf = 0.0282;
tw = 0.0180;
k = 0.0432;
thi = 0.011;

//+
Point(1) = {tw/2, 0, 0, 1};
//+
Point(2) = {tw/2, (d/2)-k, 0, 1};
//+
Point(3) = {(tw/2)+k-tf, (d/2)-k, 0, 1};
//+
Point(4) = {(tw/2)+k-tf, (d/2)-tf, 0, 1};
//+
Point(5) = {bf/2, (d/2)-tf, 0, 1};
//+
Point(6) = {bf/2, d/2, 0, 1};
//+
Point(7) = {0., d/2, 0, 1};
//+
Translate {thi, 0, 0} {
  Duplicata { Point{1}; }
}
//+
Translate {thi, -thi, 0} {
  Duplicata { Point{2}; Point{3}; Point{4}; Point{5}; }
}
//+
Translate {thi, thi, 0} {
  Duplicata { Point{6}; }
}
//+
Translate {0., thi, 0} {
  Duplicata { Point{7}; }
}
//+
Line(1) = {1, 2};
//+
```

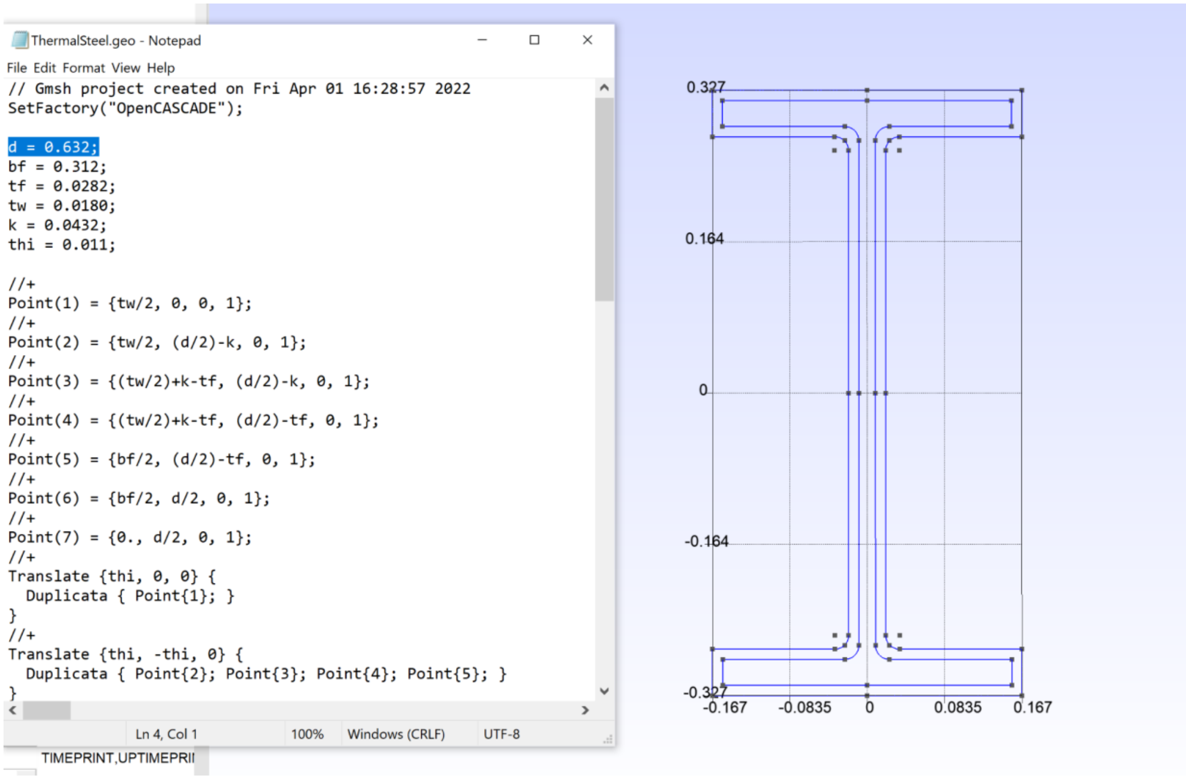
After modification, save and close the script.
Select ***Modules -> Geometry -> Reload script***
The model does not change.

Note that this action (editing the script, reloading the script) has conserved the geometry and the physical group definitions. It cannot be applied on the mesh, though. The mesh needs to be recreated each time the script is reloaded.

Now, we can change the values of the variables. For example, the thickness of insulation is changed from 11 mm to 33 mm. Dimensions of the steel section can also be changed.
It suffices to change one line in the script, then reload it, and the model is adjusted.







For each case:

- Edit the script, and modify the required input (e.g., depth d of the profile)
- Reload the script with ***Modules -> Geometry -> Reload script***
- Recreate the mesh (see Section 5)
- Create SAFIR input file and run the file (See Section 6)