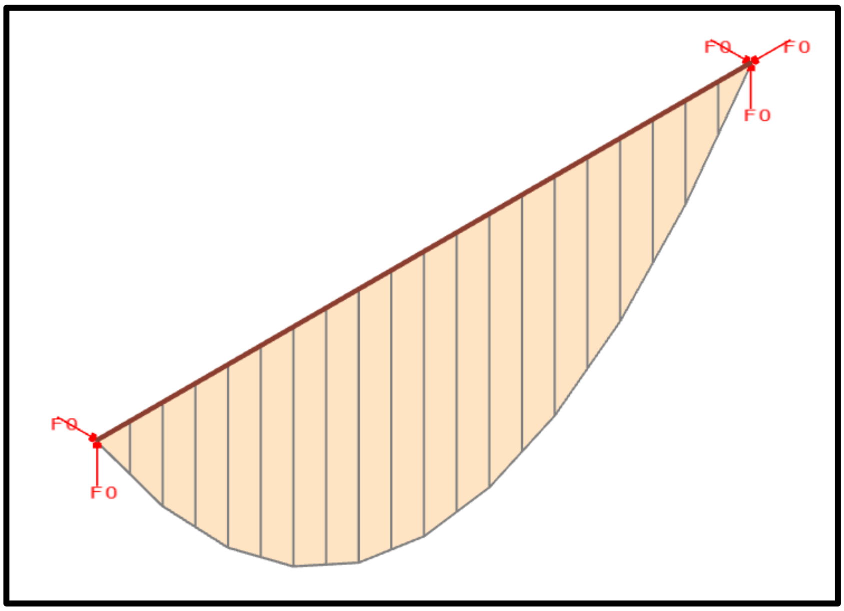
SAFIR® training session – level 1
Johns Hopkins University, Baltimore

Example: 3D structural model of a beam

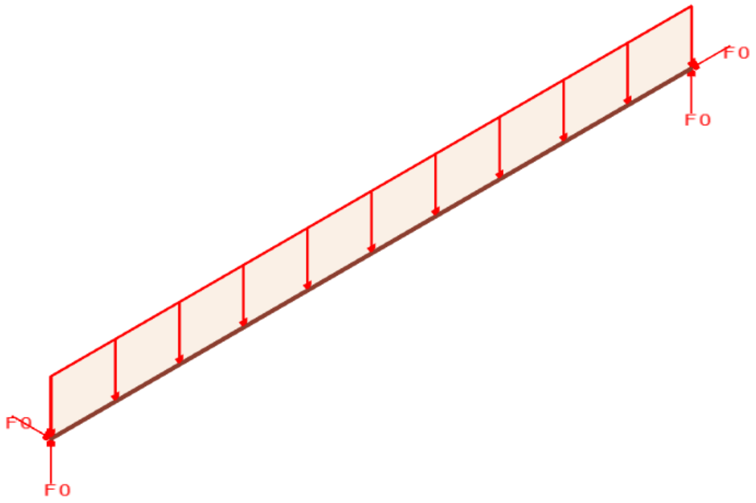
“3D steel beam with thermal insulation”

T. Gernay & J.M. Franssen



1. General description

This example deals with a 3D beam of 6 m length. The beam is a W21x44 steel profile protected with 25.4 (1 inch) of sprayed fire resistive material (SFRM). It is exposed to the ASTM E119 fire on its four sides. The beam is simply supported. It is subjected to a uniformly distributed load of 50 kN/m.



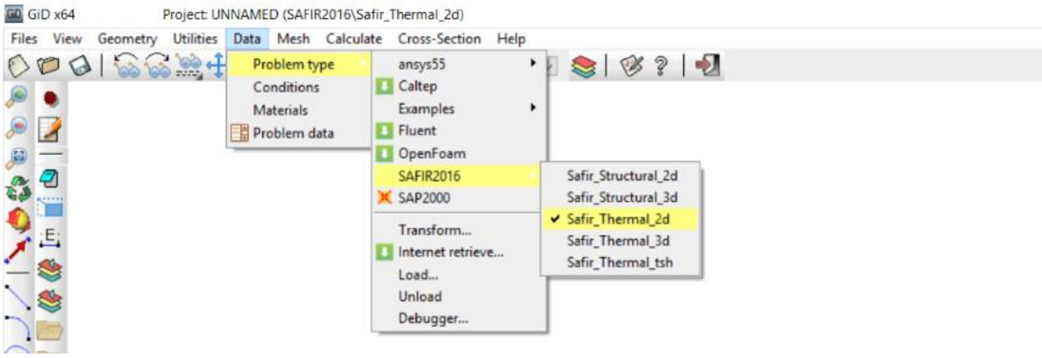
Steel for W21x44:
- Yield strength 355 MPa
- Material model from Eurocode 3 part 1-2

Thermal insulation (SFRM)
- Thermal properties constant with temperature

2. Section for W21x44 steel beam

2.1. Create a project for 2D thermal analysis

From the pull down menu select:
Data -> Problem type -> SAFIR2016 -> Safir_Thermal_2d



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

Enter a file name, e.g.: *W21x44*
GiD creates a directory with the name *W21x44.gid*
GiD creates a number of system files in this directory.
When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created in this directory.

2.2. Create the geometry of the cross-section

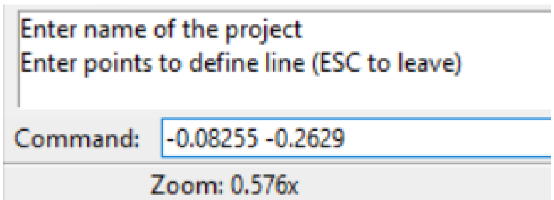
The profile is a W21x44 with dimensions given here below.

Depth	525.8	mm
Width	165.1	mm
Web thickness	8.9	mm
Flange thickness	11.4	mm
SFRM thickness	25.4	mm

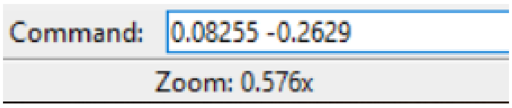
We will create the steel profile section first. The profile is centered on (0, 0). Select the option to create system lines:

Geometry->Create->Straight Line or 

Enter in the command line (at the bottom of the widows) successively the coordinates of the 2 nodes that define the lines. After typing the coordinates of a node, click *[Enter]* to validate.

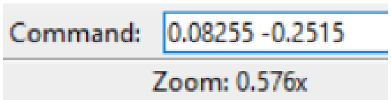


Press *[Enter]*

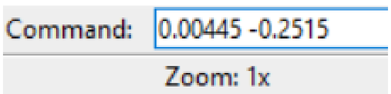


Press *[Enter]*

This creates a first line based on these two points, which is the lower face of the lower flange. Now adding a point to create the right face of the lower flange:

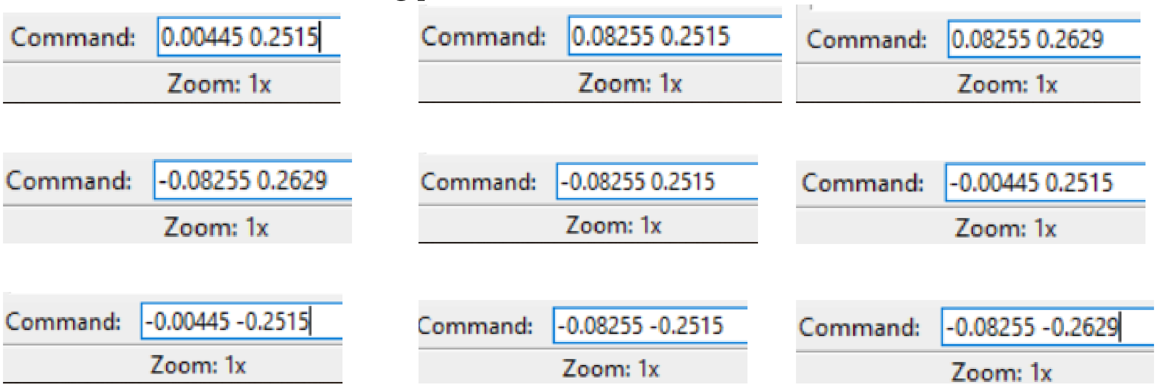


Press *[Enter]*. Then adding a point on the right side of the web:



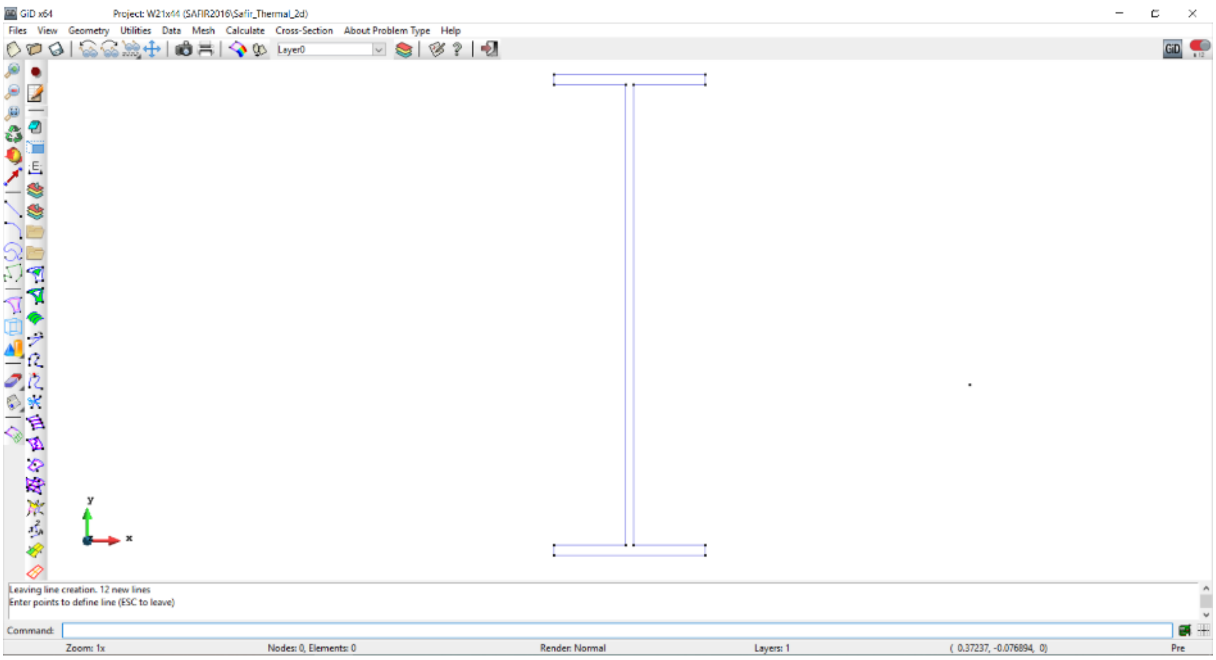
Press *[Enter]*.

And so forth with the following points:



When entering this last point, select *[Join]* when asked by GiD to connect with the existing point at these coordinates. Then press *[Esc]* to exit the line creation command.

GiD displays the following geometry:



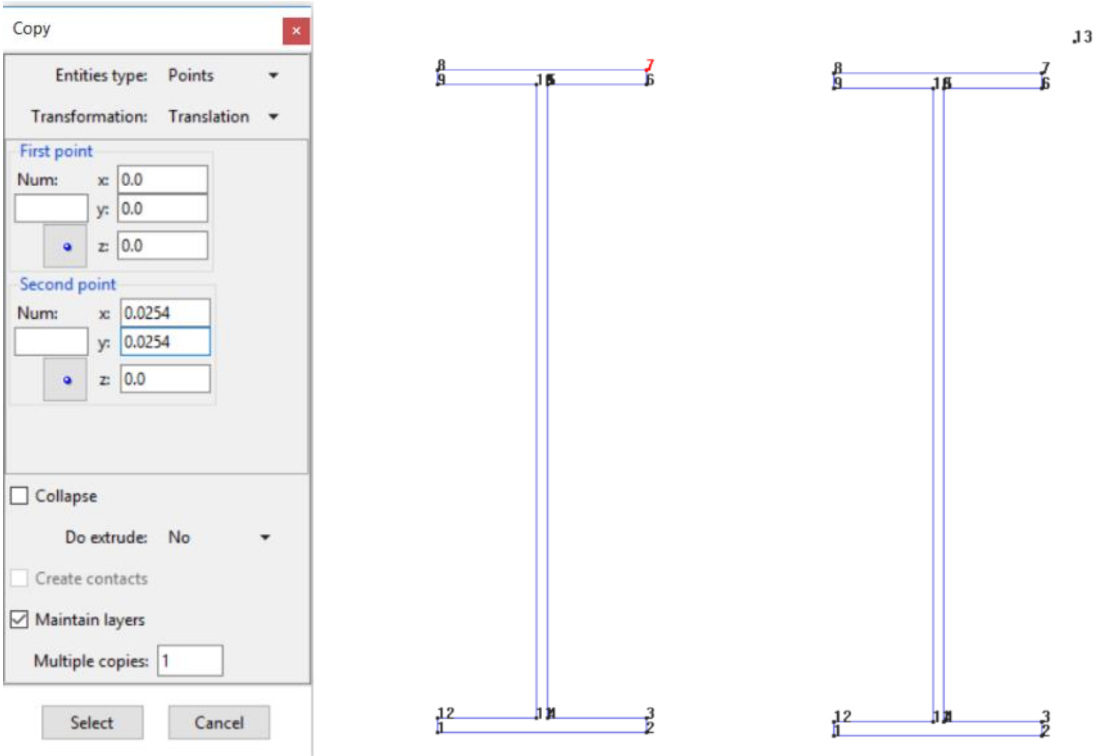
Note: GiD offers a number of tools to create the geometry efficiently, such as the ones that can be accessed in *Utilities->Copy…* as well as the possibility to directly create objects. The objective here above was to show the process to create a profile line by line. Now, the *Copy* tool will be used to create the SFRM.

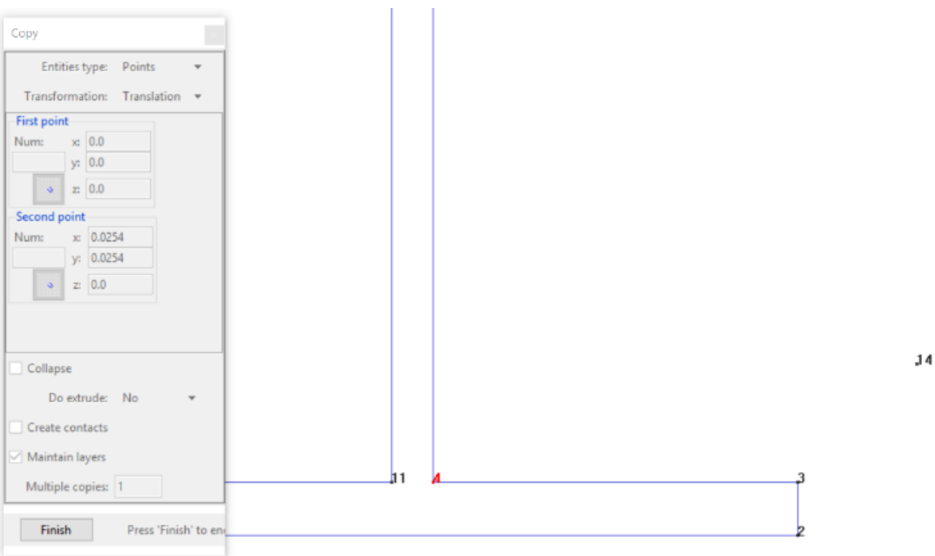
First, select *View->Label->All in->Points* to display the points numbers.

Select the option to copy existing model entities:
Utilities->Copy…

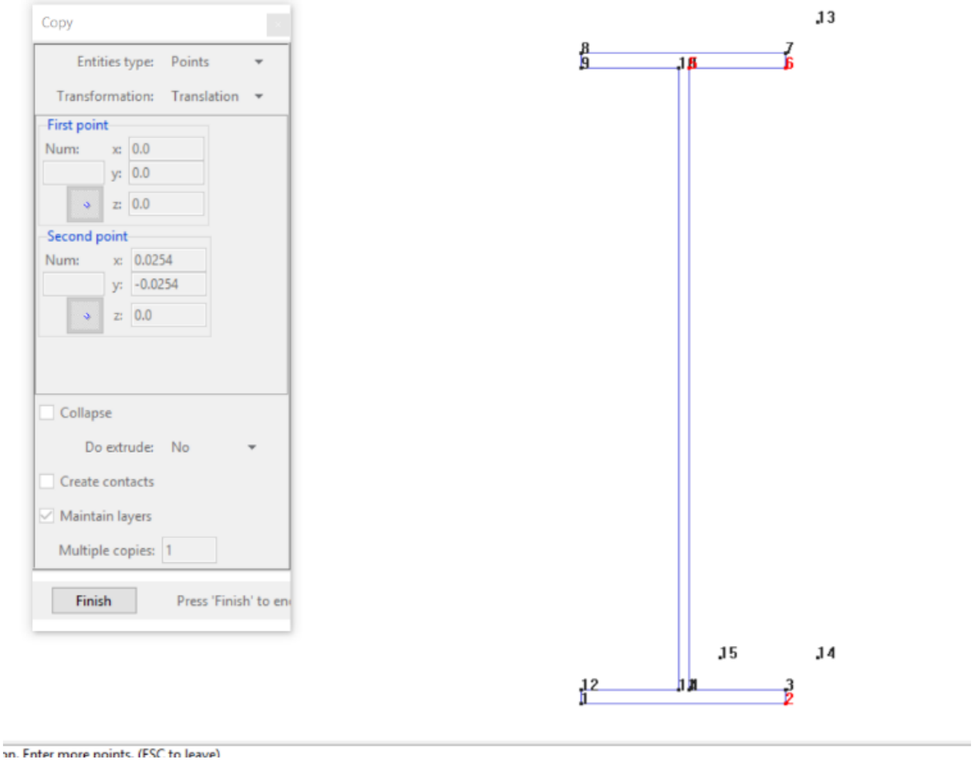
Select *Entities types: Points* and *Transformation: Translation*.
As first point, leave the coordinates: *0.0, 0.0*.
As second point, enter: *0.0254, 0.0254* (i.e. the thickness of the SFRM)
Then, select the top right point of the profile (no. 7). Click on *Finish* or *Esc* to validate.
This creates a new point (13) that is at a distance of 25.4 mm of the profile.

The same translation is applied to nodes 3 and 4.





Then, a new translation is defined with the second point being: *0.0254, -0.0254*.
This translation is applied to nodes 2, 5 and 6.



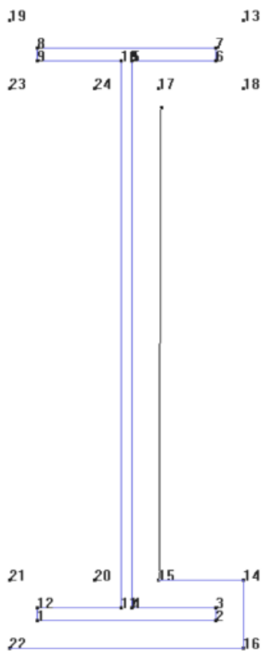
A third translation is defined with the second point being: *-0.0254, 0.0254*.
This translation is applied to nodes 8, 11 and 12.

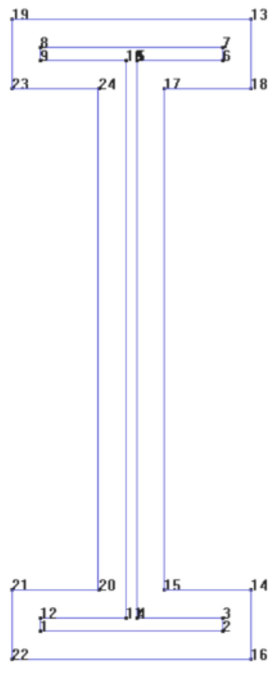
Finally, translation *-0.0254, -0.0254* is applied to nodes 1, 9 and 10.

This procedure has created 12 new points which define the contour of the SFRM. Now, select the option to create system lines:

Geometry->Create->Straight Line or 

Use *ctrl + A* to pick an existing point with the mouse. Click successively on the points to create the contour of the SFRM. (For instance: 22 – 16 – 14 – 15 – 17 …).
When it is done, click *[Esc]*.

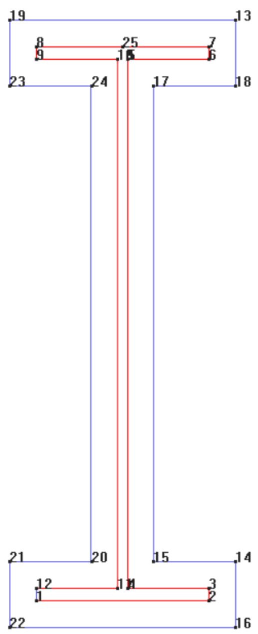


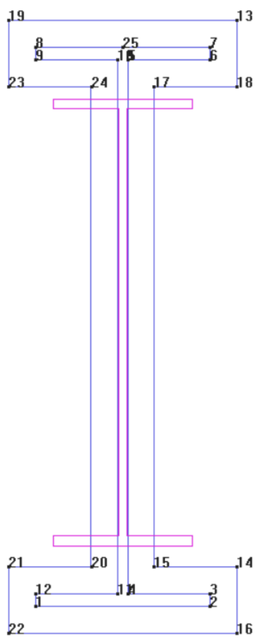


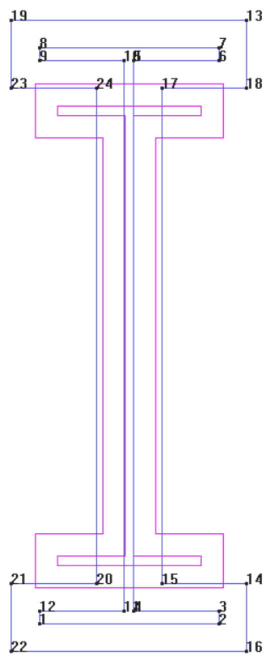
There remains to create the surfaces.

Select *Geometry->Create->NURBS surface -> By contour* or 

Select the lines that form the contour of the steel profile and click *Esc* to validate.







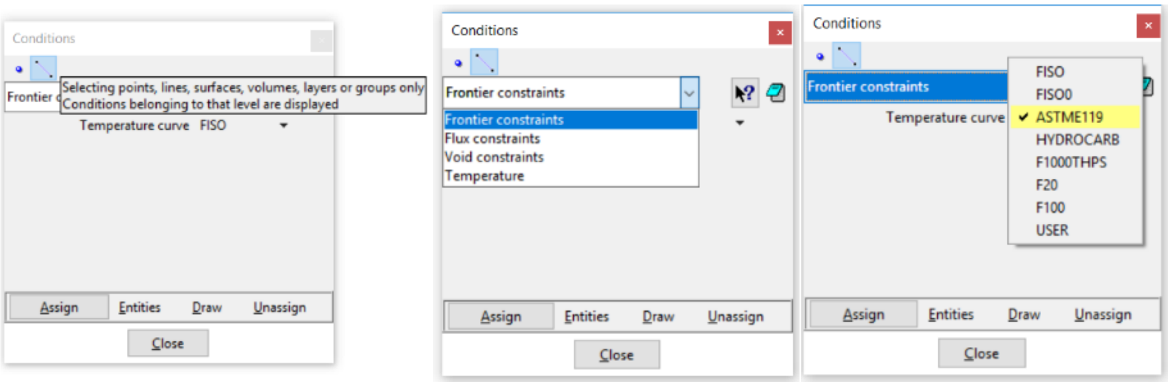
Then select the lines that form the contour of the SFRM (including the internal contour, i.e. actually all the line in the model). Click *Esc* to validate.

2.3. Assign the thermal boundary conditions

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

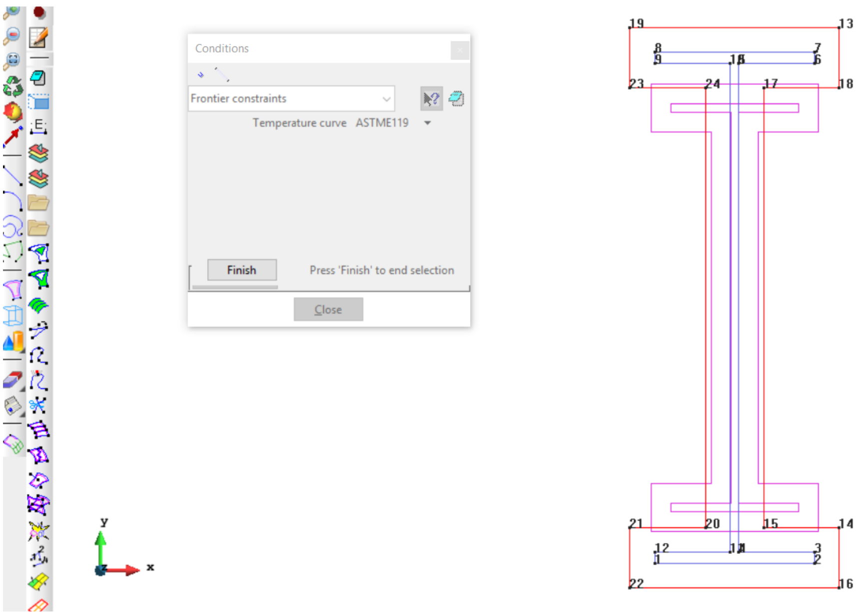
A window opens up in GiD. Select the  button (“Line”). On the first pull down list, select: *Frontier constraints*

Different time-temperature curves are predefined. Select *ASTME119*.



Click on the *Assign* button and assign it to the contour of the SFRM as shown below.

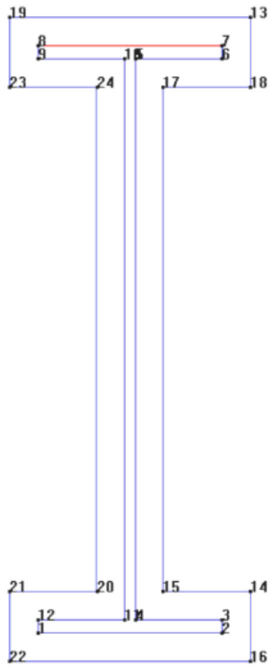
Press *[Esc]* or click on *Finish* to confirm.
Select *DRAW->Colors* in the Conditions dialog box to display the frontier constraints.
Press *[Esc]* or click on *Finish* to leave this view mode.

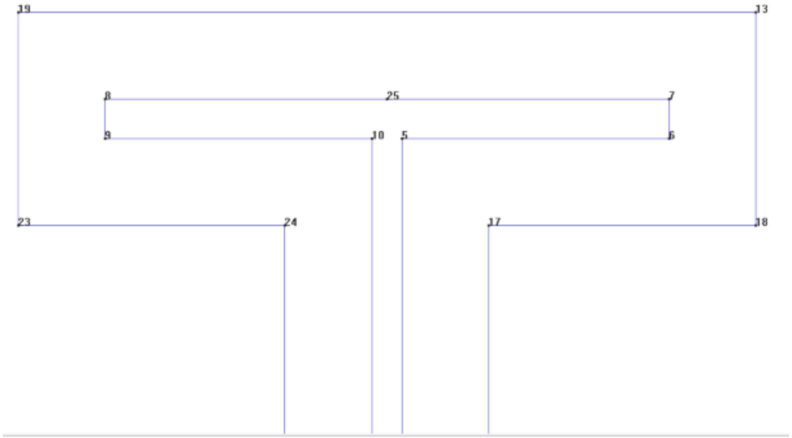


2.4. Assign a torsion constraint

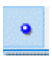
The torsion constraint needs to be applied on a node that is on an axis of symmetry of the section. To create such a node, from the pull down menu select:
Geometry -> Edit -> Divide -> Lines -> Num divisions

Enter number of divisions: 2. Select the line at the top of the top flange of the profile. Validate with ***Esc***. A node 25 is created.





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

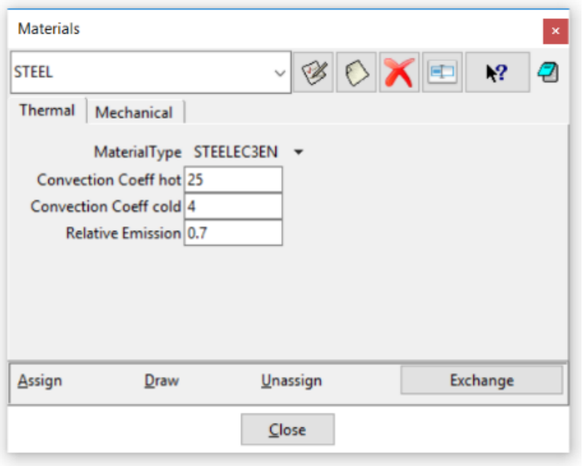
Select the  button
On the pull down list: ***Torsion constraints***
Tick the box ***Constraint*** (only in GiD problem types versions prior to 1.4)

Select the node 25 on the vertical axis of symmetry of the steel profile. Validate with ***Finish***.

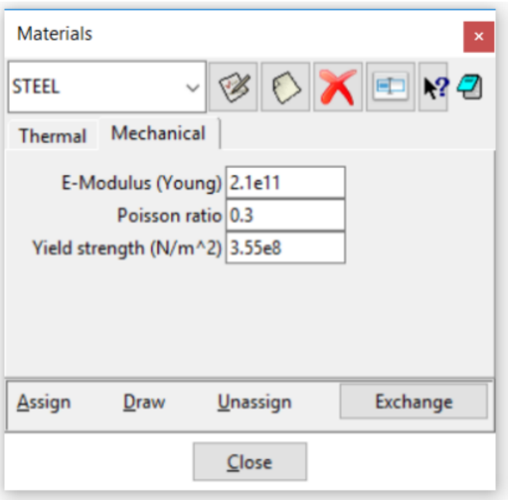
2.5. Assign the materials

From the pull down menu select:
Data->Materials
Select ***STEEL*** from the dialog box pull down list
The ***Thermal*** tab is active.
Then select:
STEELEC3EN as Material Type

A Convection Coeff hot of *25*
A Convection Coeff cold of *4*
A Relative Emission of *0.7*

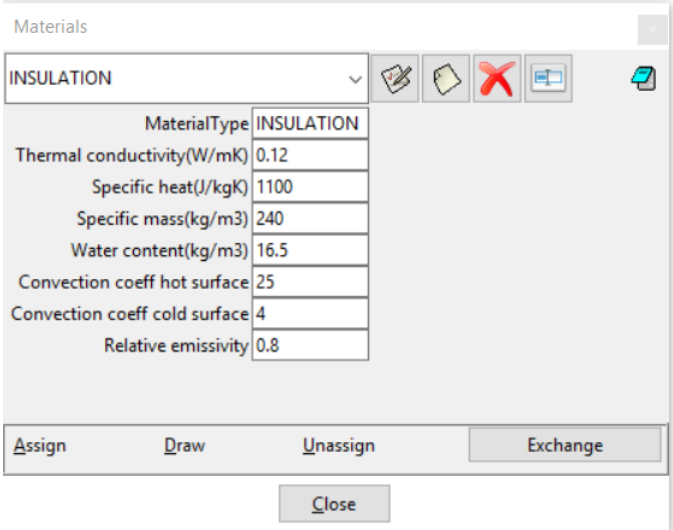


Then select the *Mechanical* tab. Input:
A Young modulus of *210 000 MPa*
A Poison ratio of *0.3*
A Yield strength of *355 MPa*



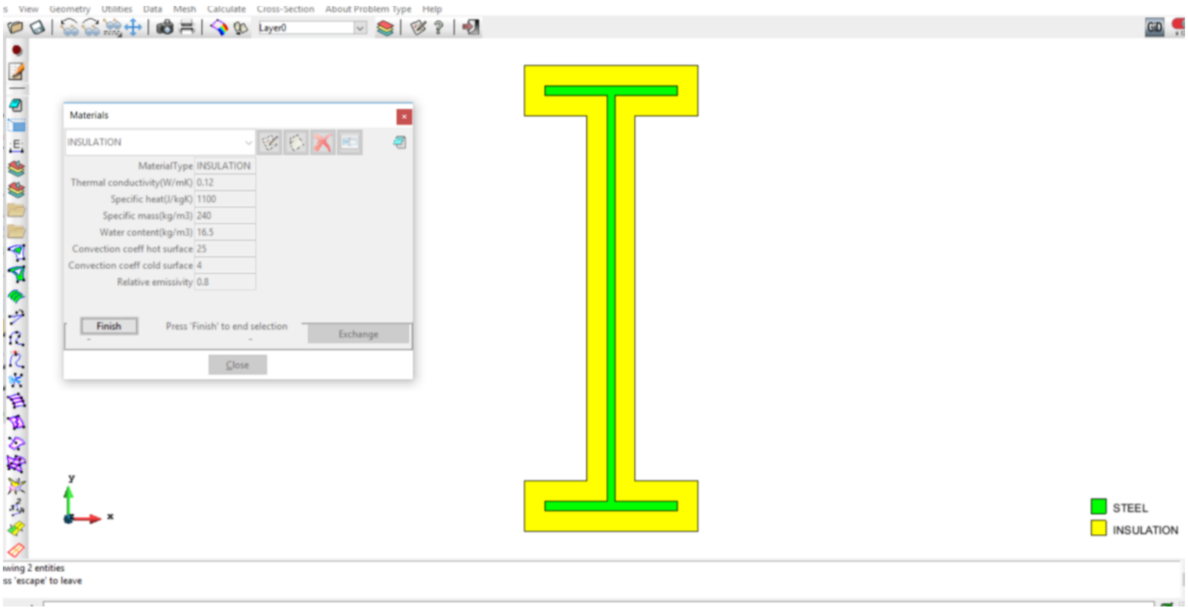
Click on *Assign-> Surfaces* and assign it to the W21x44 surface
Press *[Esc]* or *Finish* to confirm.

Next, select *INSULATION* from the dialog box pull down list
Fill in with the following properties:



Click on *Assign-> Surfaces* and assign it to the SFRM surface.
Press *[Esc]* or *Finish* to confirm.

Select *DRAW->all materials* in the Material dialog box to display Materials
Press *[Esc]* or *Finish* to leave

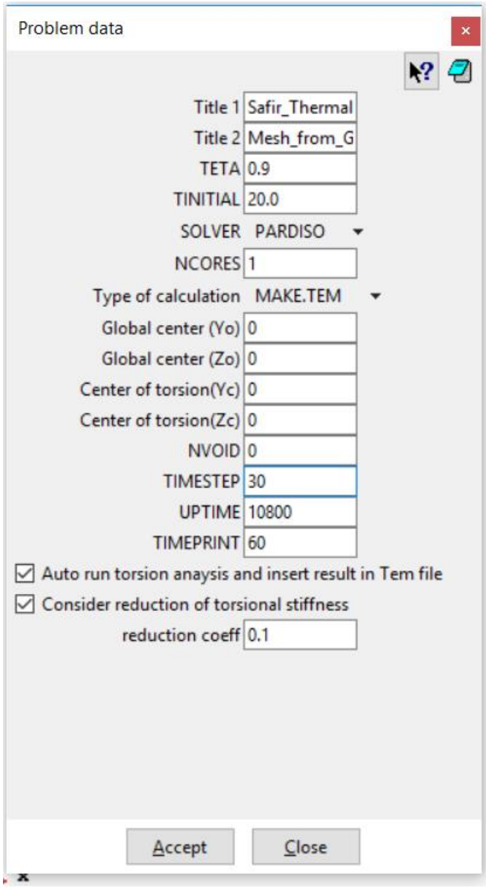


2.6. Assign the general data

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

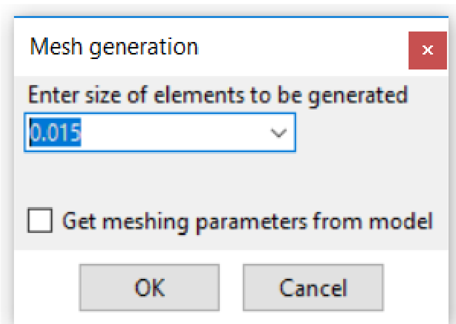
In the Problem Data dialog mask enter:
TIMESTEP, UPTIME, TIMEPRINT as needed
Do not forget to tick the box *Autorun Torsion Analysis*
Also tick the box *Consider reduction of torsional stiffness* and leave the value as 0.1

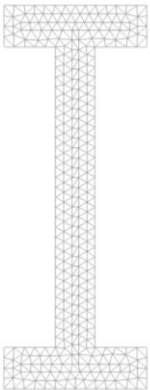
Click on the *Accept* data button



2.7. Create the mesh

Select *Mesh->Generate mesh or use [Ctrl + g]*
Enter 0.015 as size of elements to be generated (note: a coarser mesh may be required with the demonstration version of SAFIR. If GiD does not run, try coming back to the mesh definition and using 0.018 as the size).
Validate with *OK.* Click on *View mesh* to visualize the mesh



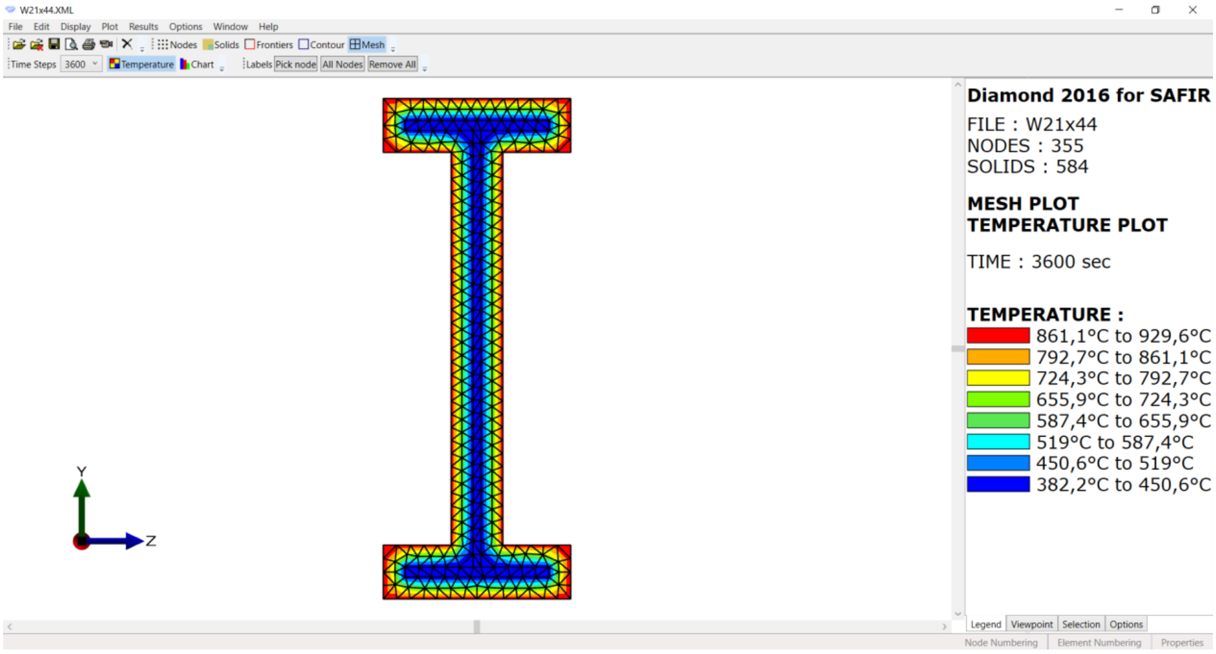


2.8. Start the calculation

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

GiD creates a .IN file in the project directory and starts the calculation.
In the output window you can see the calculation progress from SAFIR and the GiD interface program which generates GiD postprocessor files from the .OUT file.

Click on "Ok", save, and open the postprocessor Diamond to visualize the results.

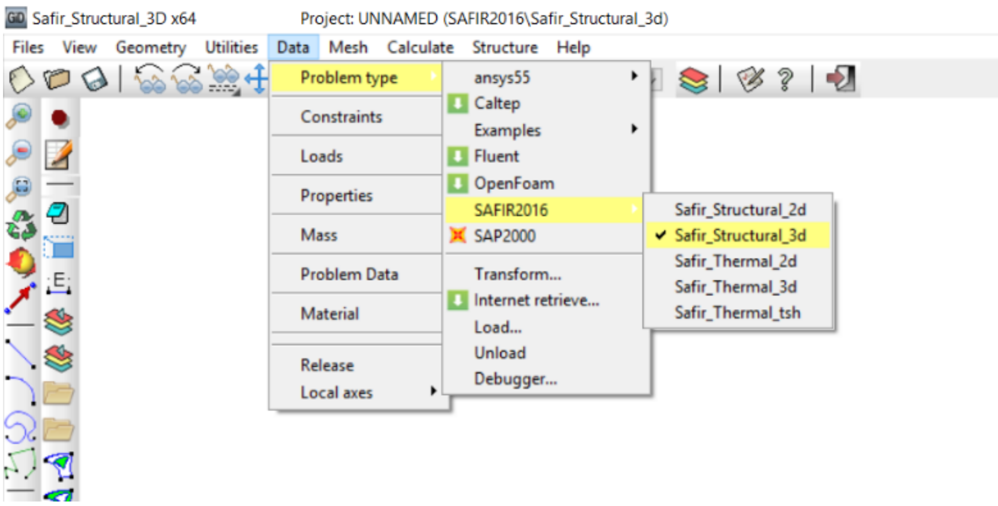


The .tem file of this model (*'w21x44.tem'*) will need to be copy-pasted in the folder with the structural model.

3. Create model for the 3D structure

3.1. Create a new project for structural 3D analysis

From the pull down menu select:
Data->Problem type->SAFIR2016->Safir_Structural_3d



To save the project select (or use icons on the left):

Files->Save *or*  *or [Ctrl + s]*

Note: If Caps lock is active on your keyboard, shortcut do not work

Enter a file name, e.g.: *3DBeam*
GiD creates a directory with the name *3DBeam.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.

3.2. Copy-Paste the section file in the structural analysis directory

GiD has created the directory *3DBeam.gid*

The structural input file, which will be created in this directory, will require the information from the section files. Therefore, these sections files need to be located in the same directory.

Copy and paste the files *'W21x44.tem'* in the directory *3DBeam.gid*

3.3. Create the system geometry (3D beam)

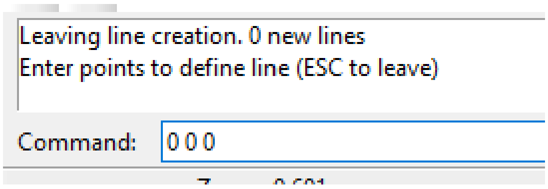
The view is by default in the x-y plane. Here, the plane of the frame will be defined in the x-z plane. To change to the 3d isometric view select from the pull down menu:
View->Rotate->isometric
Or if you want to define a point of view by your own use:

View->Rotate->Trackball or *[F7]* or 

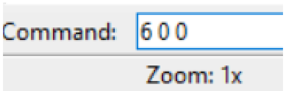
Create the system lines:

Geometry->Create->Straight Line or 

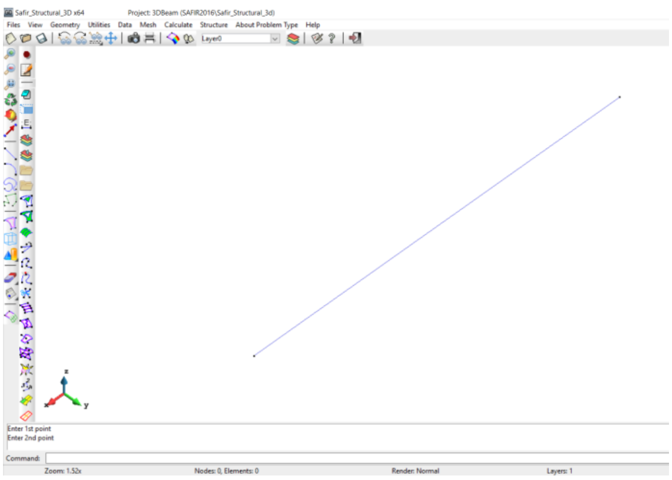
Enter in the command line (at the bottom of the widows) successively the coordinates of the 2 nodes that define the beam. After typing the coordinates of a node, click *[Enter]* to validate.



Press *[Enter]*



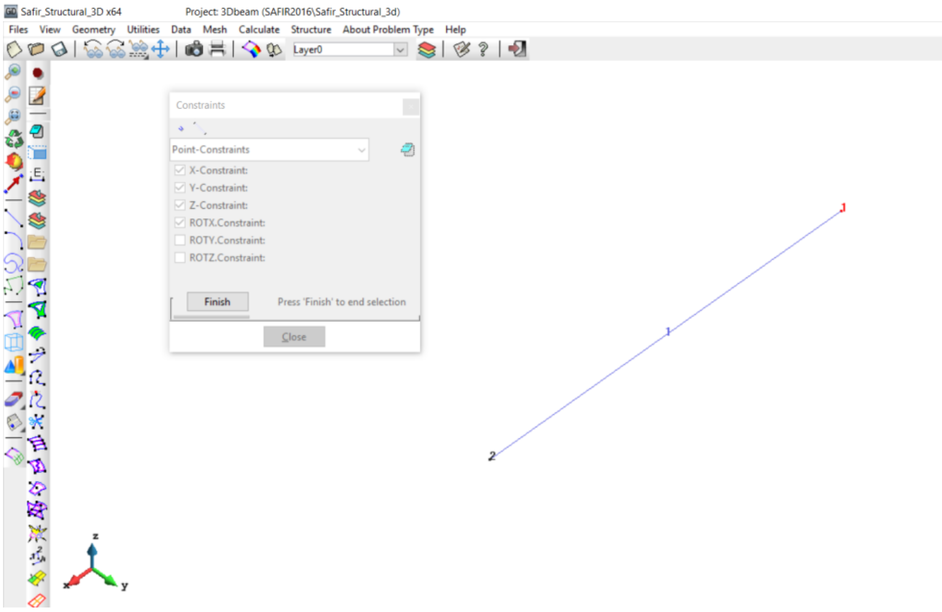
Press *[Enter]*
Then press *[Esc]* to leave the line creation menu. You should see this in GiD:



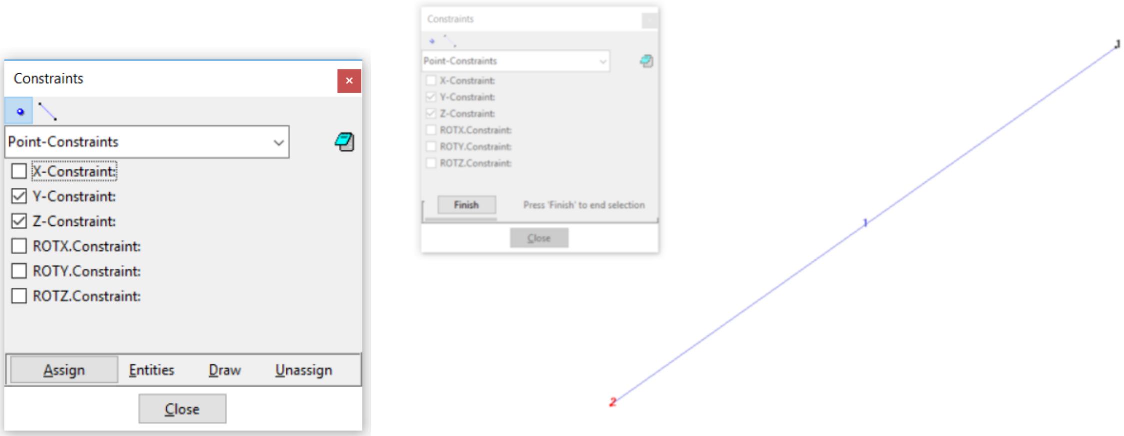
To see nodes and beams numbers select:
View->Label->All

3.4. Define constraints for the supports

From the pull down menu select
Data->Constraints
Select *Point Constraints*
Tick the boxes for X-constraint, Y-constraint, Z-constraint, and ROTX constraint. Assign these constraints to *POINT 1* and press *[Esc]*.



Then, tick the boxes for Y-constraint and Z-constraint and assign these constraints to *POINT 2* and press *[Esc]*.

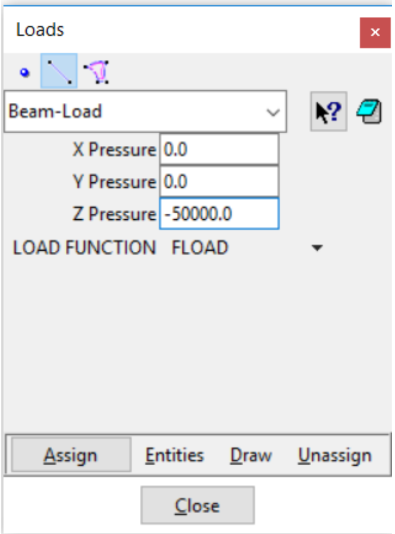


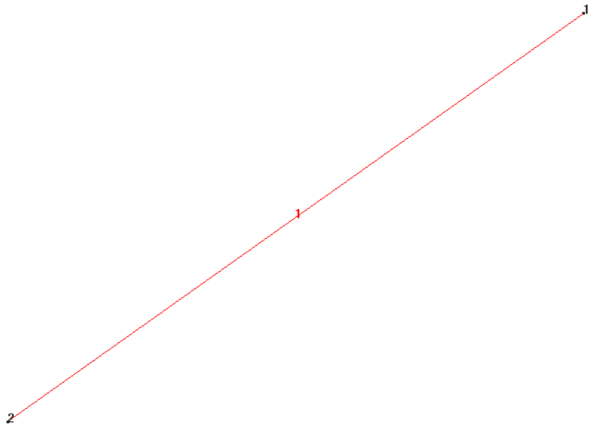
In the dial box, with *Draw->Colors* you can display the constraints.
Press *Finish* or *[Esc]* to leave this view mode.

3.5. Assign the loads

From the pull down menu select
Data->Loads

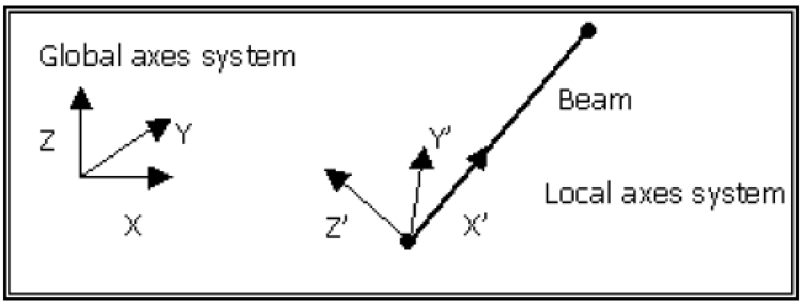
Assign a distributed load of -50 000 N/m in Z direction on the beam.
Use the function FLOAD.
Press *Finish* or *[Esc]* to validate.





4.6. Create Local Axes

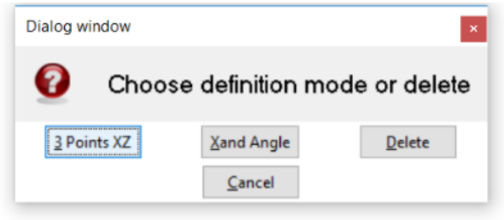
Local Axes: The orientation of the cross-section is controlled by defining a local axes X'Y'Z' –system.



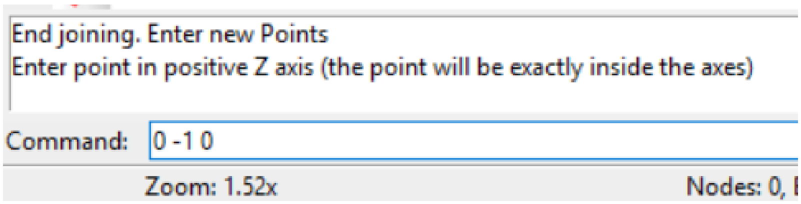
Unlike SAFIR which needs a 4th node to describe the orientation of a cross section on a beam, the GiD-SAFIR interface uses a local X'Y'Z' axes system. When you start the SAFIR calculation the GiD-SAFIR Interface creates the 4th node in the X'Y' plane. If the center of the local axes is not located on the system line of the beam, the direction vector of the Y'-axis is used together with the starting point of the beam to define the 4th node. However the GiD-SAFIR interface will issue a warning message in the Viewoutput window of the calculation run.

From the pull down menu select:
Data->Local Axes->Define

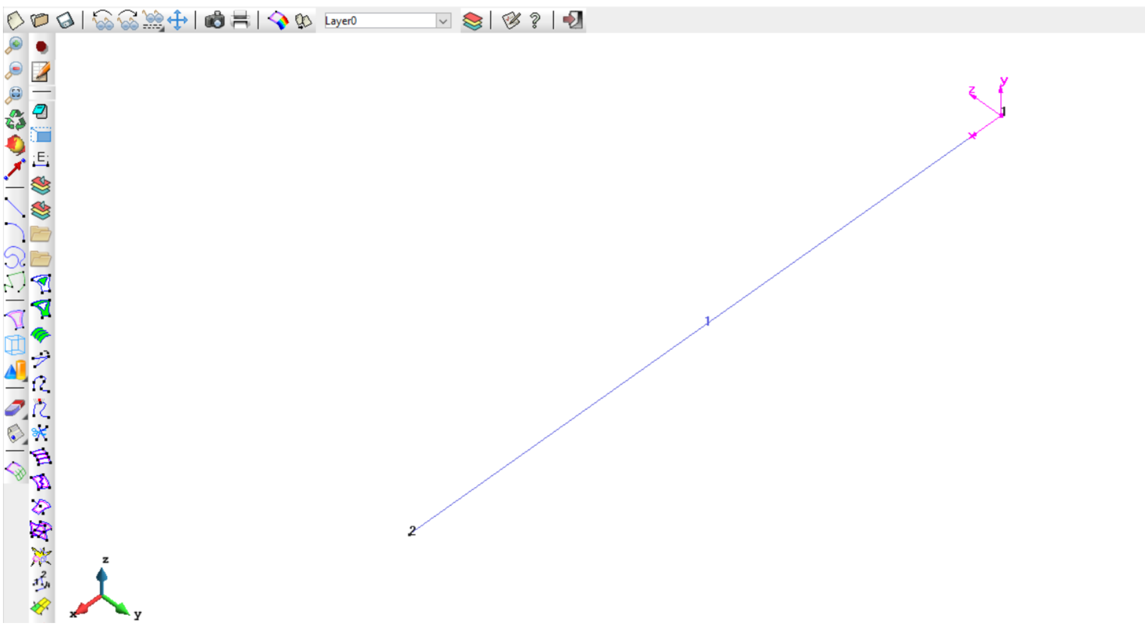
Enter the name of the local axis *LAX*
Select *3 points XZ*



Select *Point 1* as the local axis center. *Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.*
Select *Point 2* as the point in positive x axis.
The third node points to the positive direction along the z axis. There is no such node available in the model. You have to enter coordinates manually.



Press *Enter*

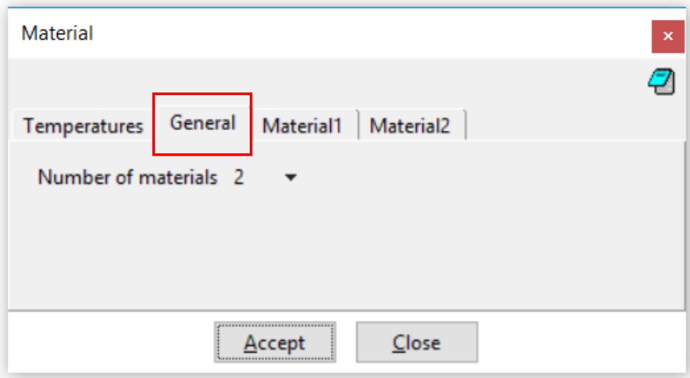


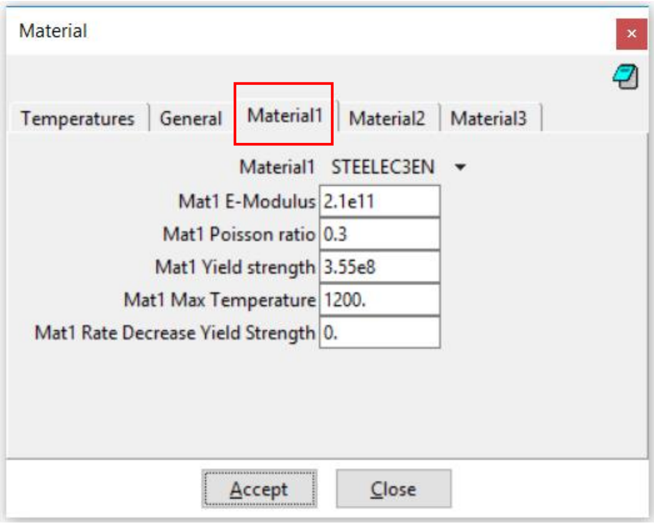
To draw local axes select:
Data->Local Axes->Draw all

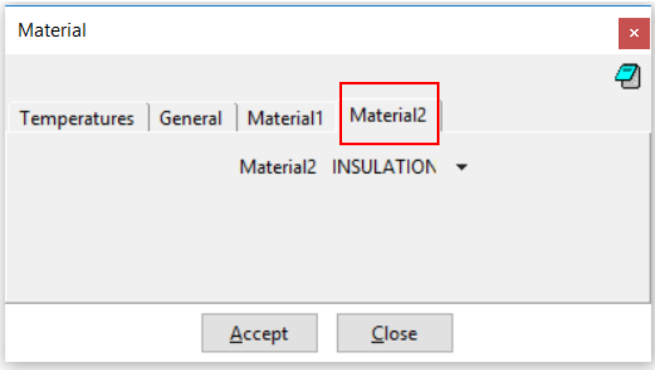
4.7. Define the global materials

From the pull down menu select
Data->Material

There are 2 materials in the model:
- The steel for the profile W21x44: STEELEC3EN
- The insulation for the SFRM: INSULATION





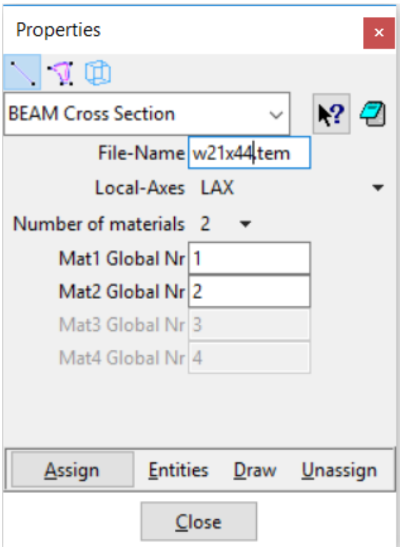


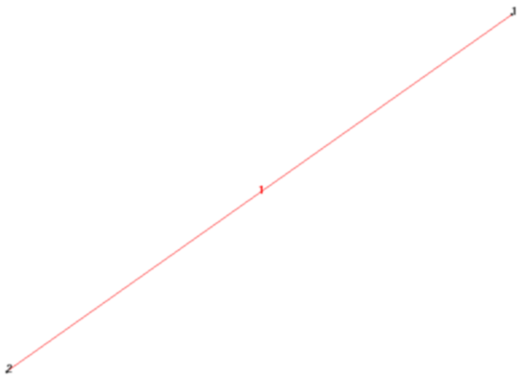
4.8. Define the properties (i.e. assign temperature files)

From the pull down menu select
Data->Properties

The objective is to assign the .tem file named *w21x44* to the model line. In the dial box of *Data->Properties*, change the File-Name: ***safir.tem*** to the temperature file (.TEM file) of the cross-section, in this case ***w21x44.tem***.

Change ***Local-Axes*** from ***-Automatic-*** to ***LAX***
Change the number of materials to 2.
Assign the ***w21x44.tem*** section to the beam.

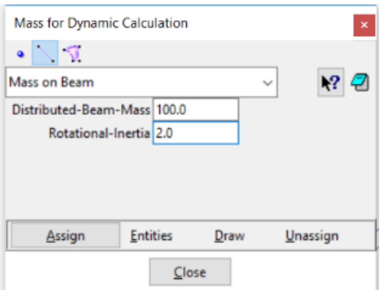




You can draw the local axes of the beams to check the model.
Select ***Draw -> All Conditions -> Include Local Axes***

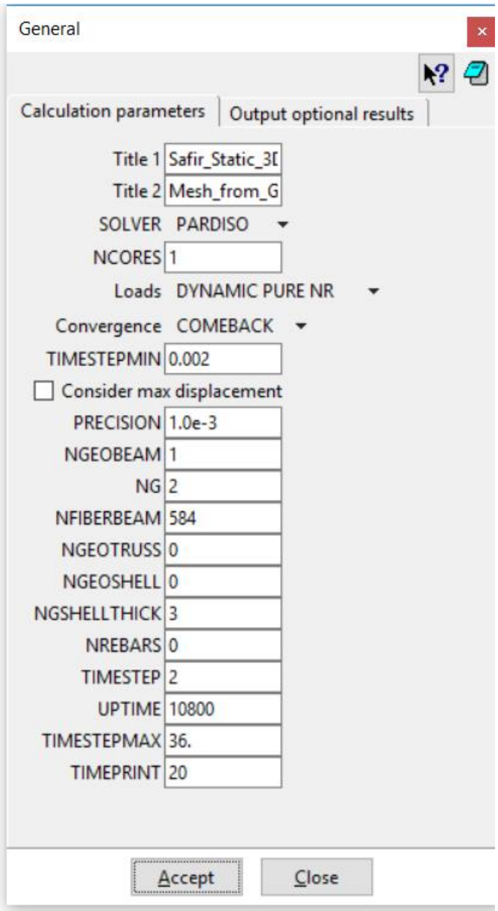
4.9. Assign the mass

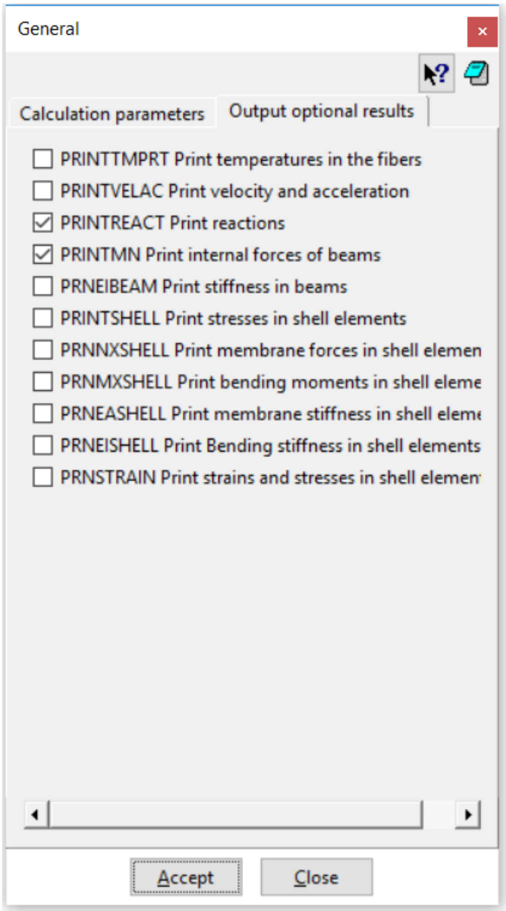
To define the mass for dynamic calculation, select from the pull down menu:
Data-> Mass
Select ***Mass on Beam*** and put 100 kg/m as Distributed-Beam-Mass and 2 as Rotational-Inertia. Assign to the beam.



4.10. Define the general problem data

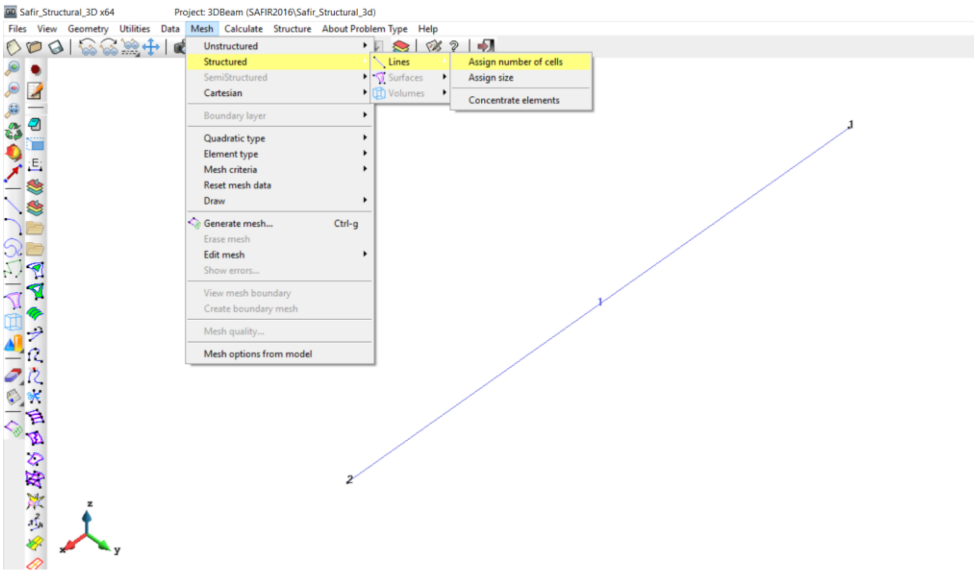
Select from the pull down menu: ***Data->Problem Data***
And fill as shown below



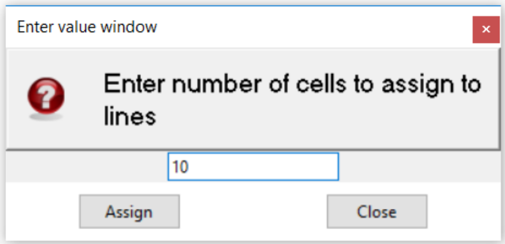


4.11. Define the mesh

Select ***Mesh -> Structured -> Lines -> Assign number of cells***



Assign 10 elements. Select the line. Press *[Esc]* to validate.



Select ***Generate Mesh*** and then ***View Mesh***

4.12. 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 clicking on *Output view* or by selecting *Calculate->View process info*

4.13. Check the results

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