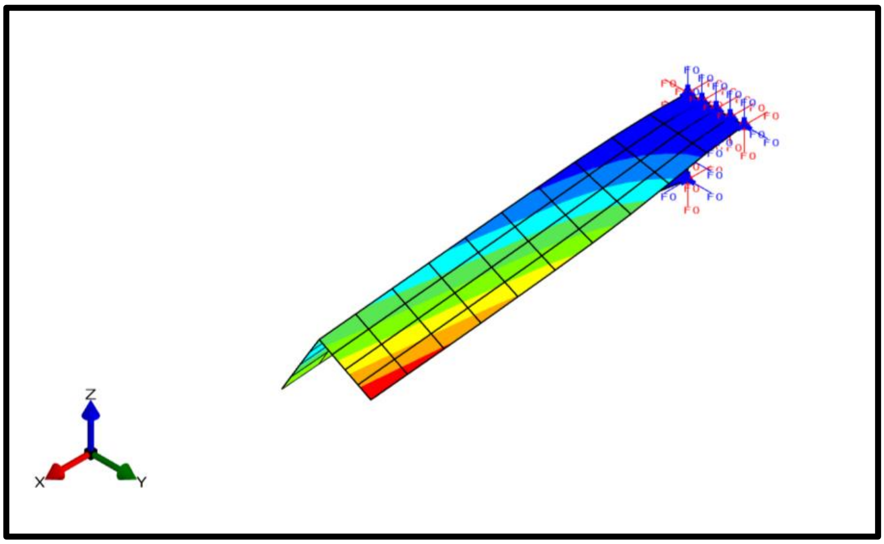
SAFIR® training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model with shell

“Steel angle in cantilever modeled with shell finite elements”

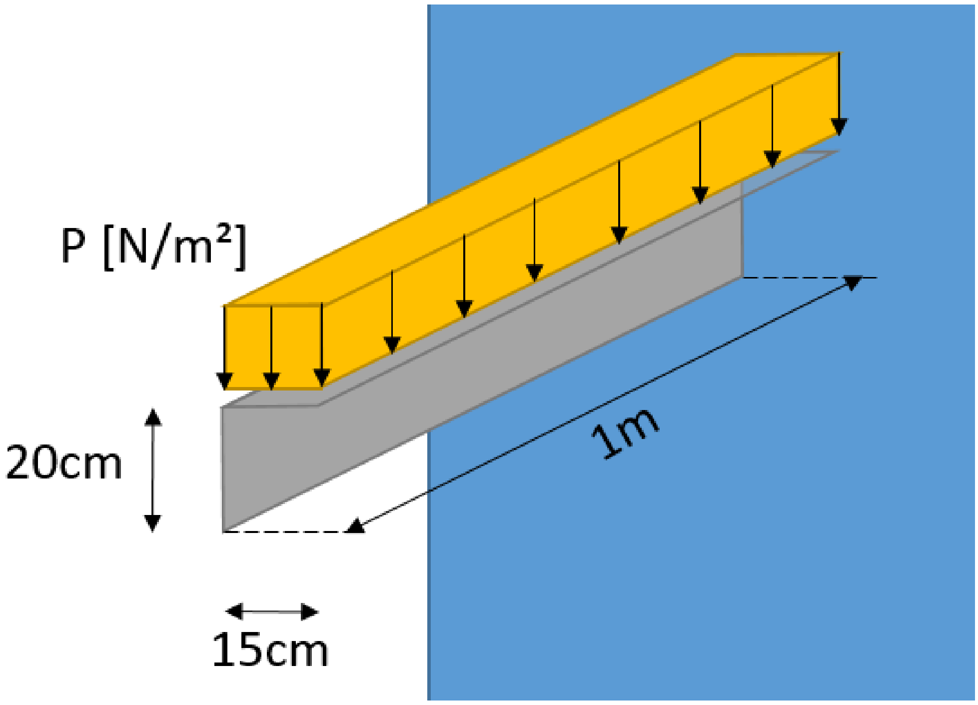
T. Gernay & J.M. Franssen



1. General description

This example deals with a cantilever beam made of an angle profile. The beam is 1 m length and is subjected to a uniformly distributed load. The plates are 6 mm thick. The steel has a yield strength of 275 MPa at ambient temperature. The steel beam is at uniform temperature of 600°C and the objective is to determine the failure load P.

The beam is modeled with shell finite elements.



2. Section for the plates

2.1. Create a project in 2D for TSH thermal analysis

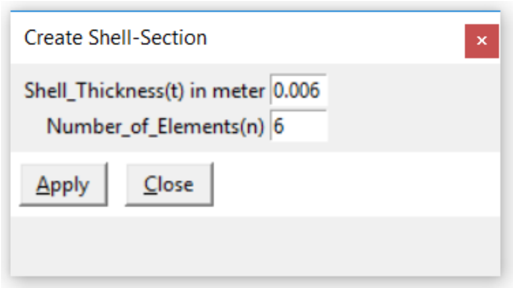
From the pull down menu select:
Data -> Problem type -> SAFIR2016 -> Safir_Thermal_tsh

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

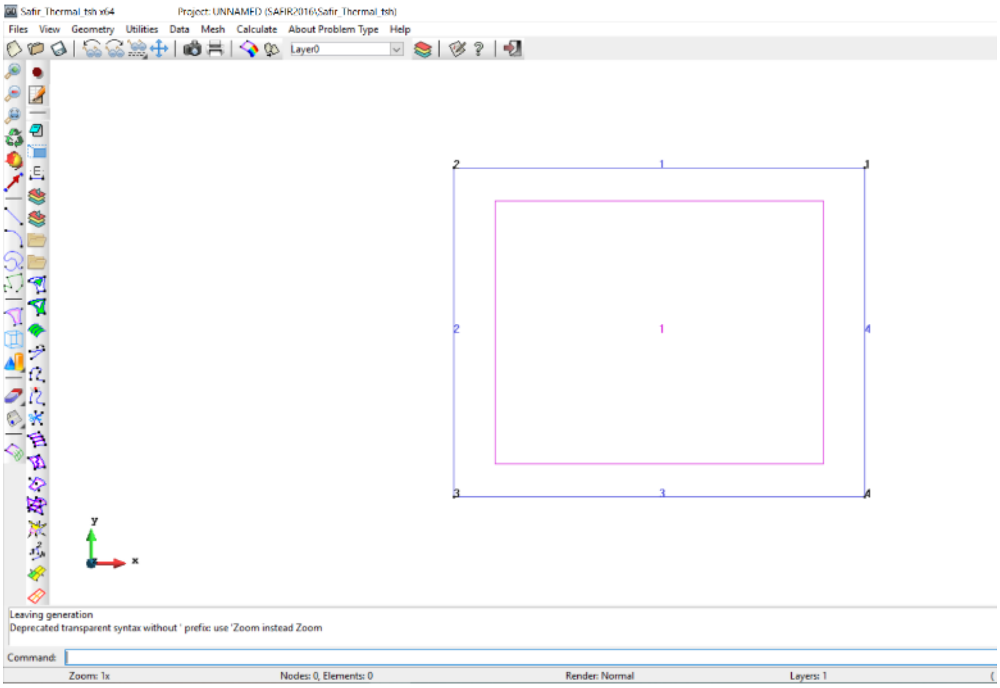
Enter a file name, e.g.: ***plate***
GiD creates a directory with the name ***plate.gid***
GiD creates a number of system files in this directory.
When you start the SAFIR calculation the Safir . IN, .OUT and .TSH files will be created in this directory.

2.2. Create the geometry of the cross-section

GiD will open automatically a new window.
Put *0.006* m as shell thickness and *6* as number of elements, as shown below:



Click on *Apply*



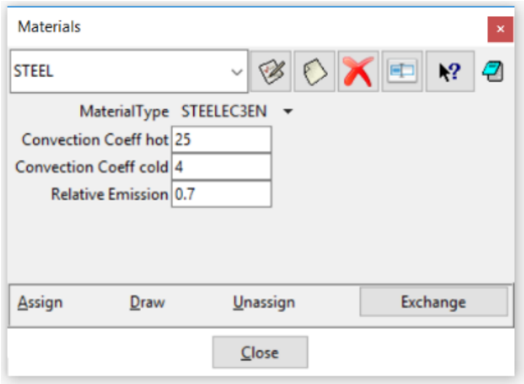
Note that the section is centered with respect to the y-axis.
It means that in the structural model, the nodeline of the shell elements is located at mid-height of the section (if no modification of the nodeline is made by the user).

2.3. Assign the thermal boundary conditions

No thermal boundary conditions will be assigned in this model. The temperatures will be modified manually in the .tsh file later on to impose a uniform temperature of 600°C constant over time.

2.4. 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*



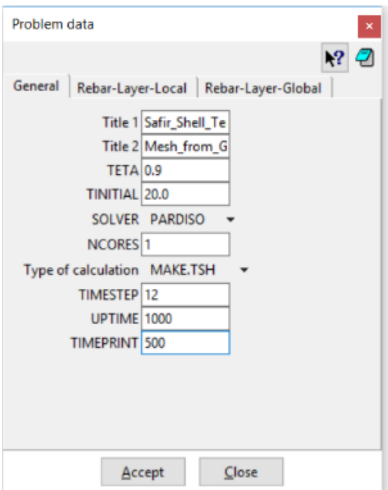
Click on *Assign-> Surfaces* and assign it to the surface. Press *[Esc]* or *Finish* to confirm.

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

2.5. Assign the general data

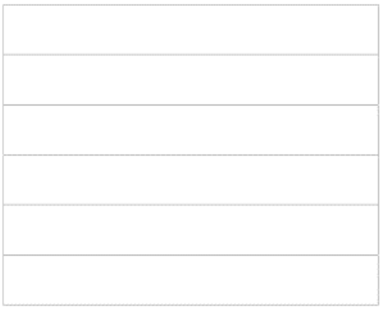
From the pull down menu select: *Data->Problem Data*

In the Problem Data dialog mask enter TIMESTEP, UPTIME, TIMEPRINT as needed. Click on the *Accept* data button



2.6. Create the mesh

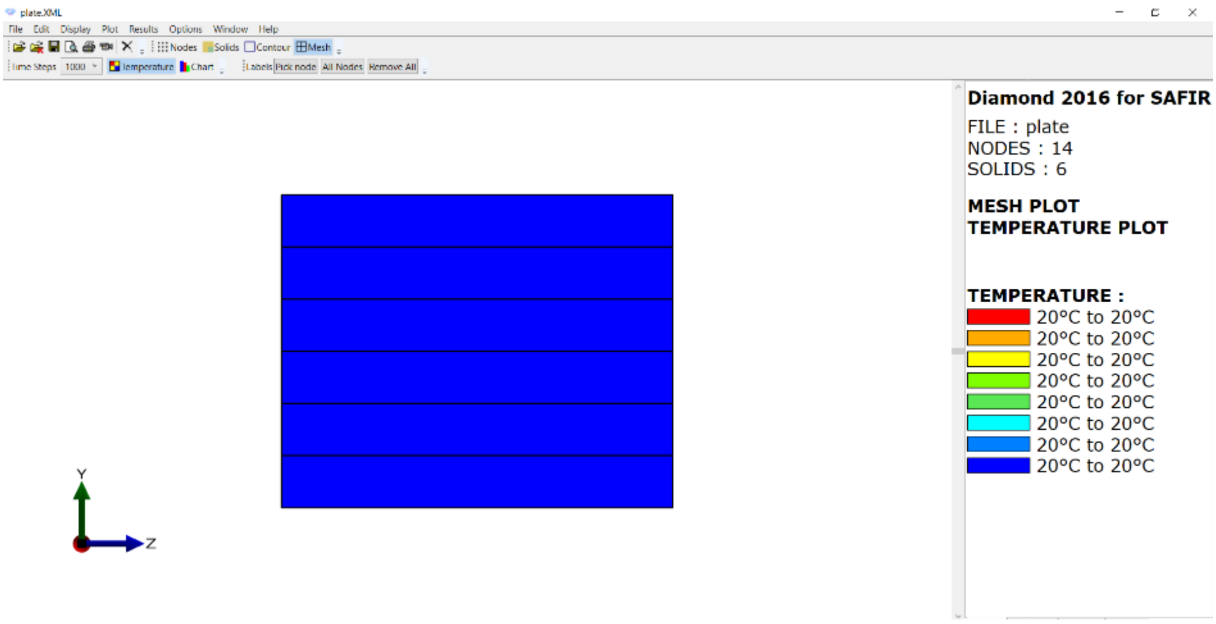
Select ***Mesh->Generate mesh or use [Ctrl + g]***
The size of elements is irrelevant since the number of elements has been chosen in 2.2. Just validate with ***OK.*** Click on ***View mesh*** to visualize the mesh



2.7. Start the calculation

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



2.8. Modify the .tsh file

The .tsh file of this model (*'plate.tsh'*) will need to be copy-pasted in the folder with the structural model. But first it needs to be modified. Go into the folder *plate.gid* and open the file *plate.tsh*.

D:\SAFIR\Courses_SAFIR\Formation-20180621\GiD-Examples\SAFIR-training-3-Shell_profile\plate.gid\plate.TSH - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

frame.in plate.TSH

```
Safir_Shell_Temperature_Analysis
Mesh_from_GID-Mesher

 THICKNESS     0.006
  MATERIAL     1
  REBARS     0

       HOT
 POSITIONS OF THE NODES.
 ======================
 NUMBER OF POSITIONS:    7
 -0.3000E-02 -0.2000E-02 -0.1000E-02  0.0000E+00  0.1000E-02  0.2000E-02  0.3000E-02

 TIME=   1000.0000 SECONDS    OR  16 MIN. 40 SEC.
 ===========================================
   -0.0030      20.00
   -0.0020      20.00
   -0.0010      20.00
    0.0000      20.00
    0.0010      20.00
    0.0020      20.00
    0.0030      20.00

 TIME=   1000.0000 SECONDS    OR  16 MIN. 40 SEC.
 ===========================================
   -0.0030      20.00
   -0.0020      20.00
   -0.0010      20.00
    0.0000      20.00
    0.0010      20.00
    0.0020      20.00
    0.0030      20.00
```

The temperature will uniformly be increased to 600°C in 100 sec, then it will be maintained constant. To do that, modify the file as shown below.

D:\SAFIR\Courses_SAFIR\Formation-20180621\GiD-Examples\SAFIR-training-3-Shell_profile\shell.gid\plate.TSH - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?

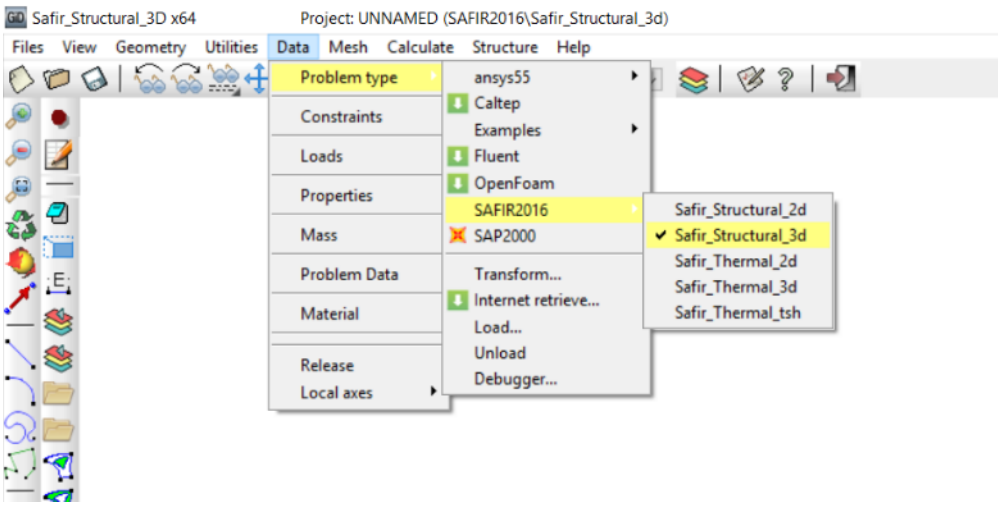
frame.in plate.TSH

```

 THICKNESS     0.006
  MATERIAL     1
  REBARS     0

       HOT
 POSITIONS OF THE NODES.
 ======================
 NUMBER OF POSITIONS:    7
 -0.3000E-02 -0.2000E-02 -0.1000E-02  0.0000E+00  0.1000E-02  0.2000E-02  0.3000E-02

 TIME=   0.0000 SECONDS    OR  16 MIN. 40 SEC.
 ===========================================
   -0.0030      20.00
   -0.0020      20.00
   -0.0010      20.00
    0.0000      20.00
    0.0010      20.00
    0.0020      20.00
    0.0030      20.00

 TIME=   100.0000 SECONDS    OR  16 MIN. 40 SEC.
 ===========================================
   -0.0030      600.00
   -0.0020      600.00
   -0.0010      600.00
    0.0000      600.00
    0.0010      600.00
    0.0020      600.00
    0.0030      600.00

 TIME=   10000.0000 SECONDS    OR  16 MIN. 40 SEC.
 ===========================================
   -0.0030      600.00
   -0.0020      600.00
   -0.0010      600.00
    0.0000      600.00
    0.0010      600.00
    0.0020      600.00
    0.0030      600.00

```

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]*

Enter a file name, e.g.: *shell*
GiD creates a directory with the name *shell.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 *shell.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 *'plate.tsh'* in the directory *shell.gid*

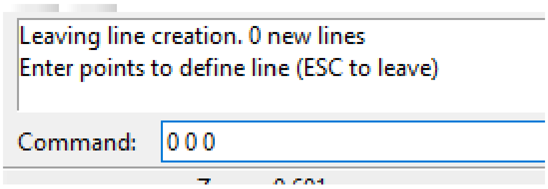
3.3. Create the system geometry

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 

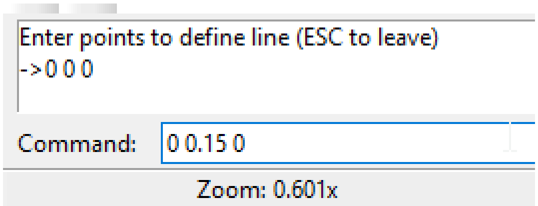
Create the system lines:

Geometry->Create->Straight Line or 

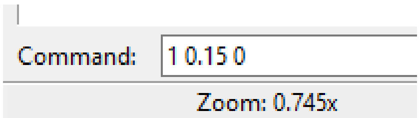
Enter in the command line (at the bottom of the widows) successively the coordinates of the nodes that define the lines. After typing the coordinates of a node, click *[Enter]* to validate.



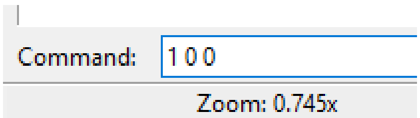
Press *[Enter]*



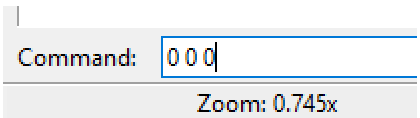
Press *[Enter]*



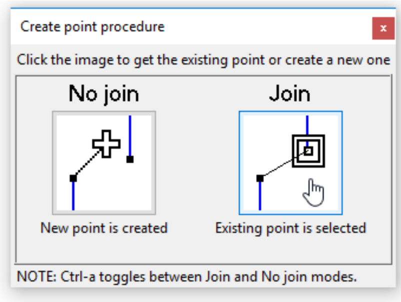
Press *[Enter]*



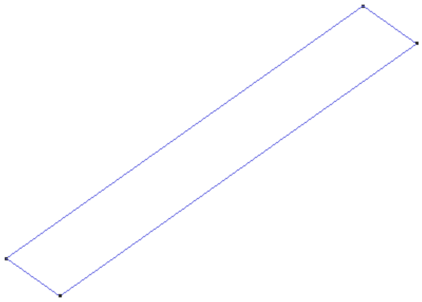
Press *[Enter]*



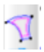
Press *[Enter]* and select *Join*, then press *[Esc].*



You should see this in GiD:

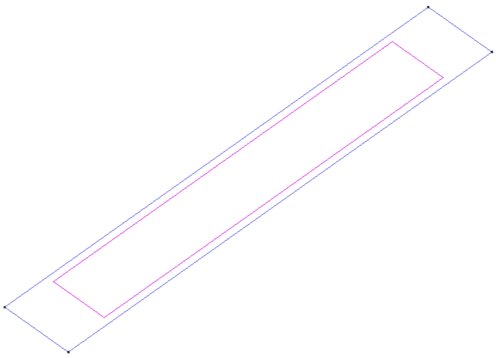


Create the first surface:

Geometry->Create->Surface or 

Select the lines that define the contour of the surface.

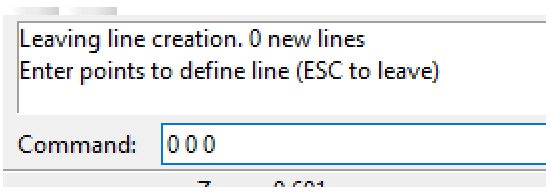
Then, press the *[Esc]* key to validate. You should see this in GiD:



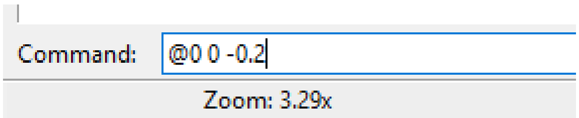
Proceed similarly to create the second plate. First create the system lines:

Geometry->Create->Straight Line or 

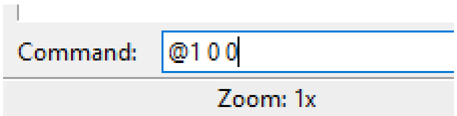
Enter in the command line:



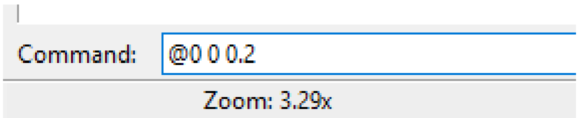
Press *[Enter]* and select *Join*



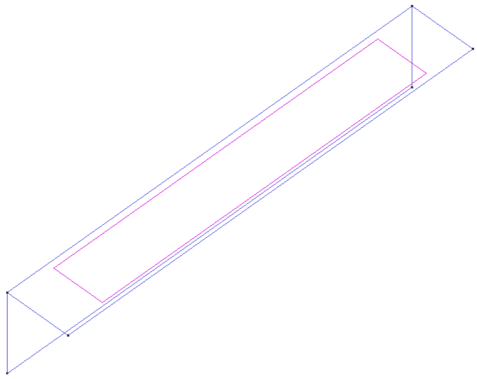
Press *[Enter]*



Press *[Enter]*

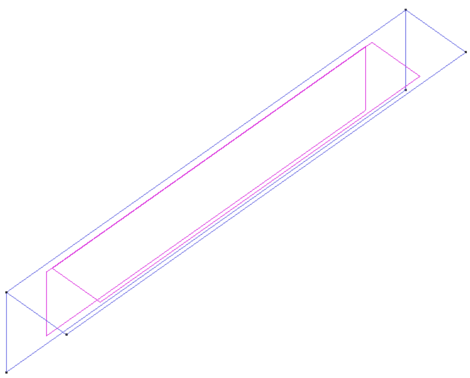


Press *[Enter]* and select *Join,* then press *[Esc]*.
You should see this in GiD:



Then create the second surface:

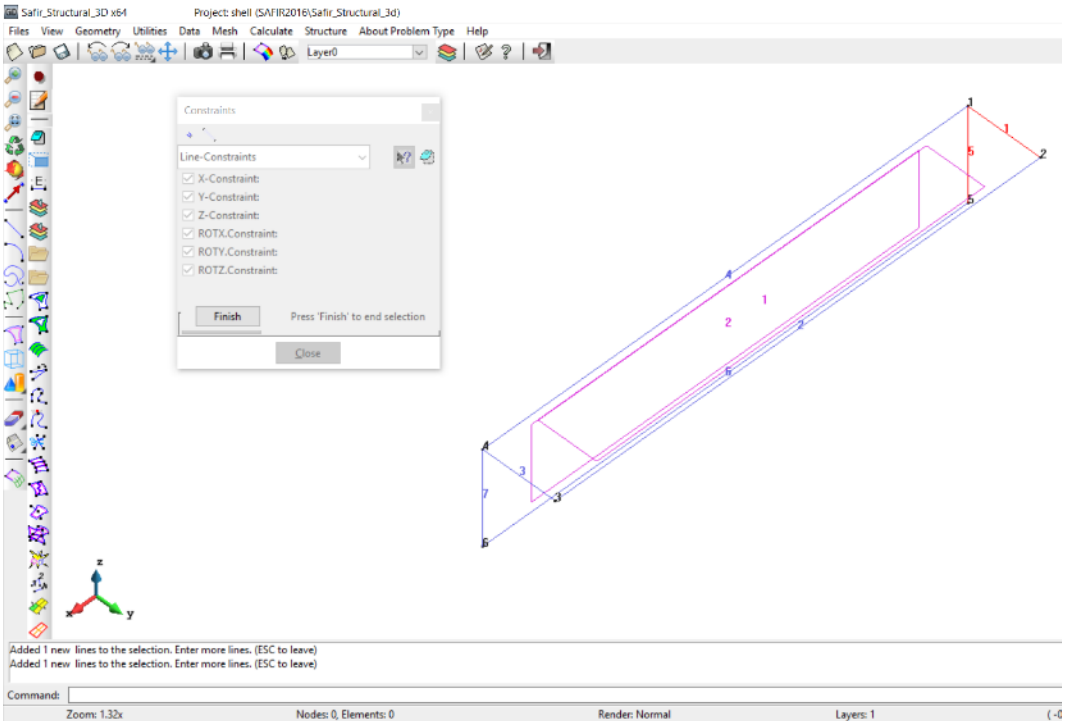
Geometry->Create->Surface or 
Select the lines that define the contour of the second surface.
Then, press the *[Esc]* key to validate. You should see this in GiD:



To see nodes, lines and surfaces numbers select:
View->Label->All

3.4. Define constraints for the supports

The angle beam is fully fixed at one end.
From the pull down menu select
Data->Constraints
Select *Line Constraints*
Tick all the boxes.
Assign these constraints to *Line 1* and *Line 5* and press *[Esc]* or *Finish* to validate.

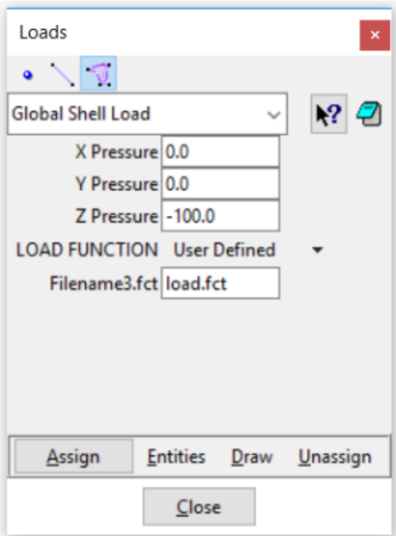


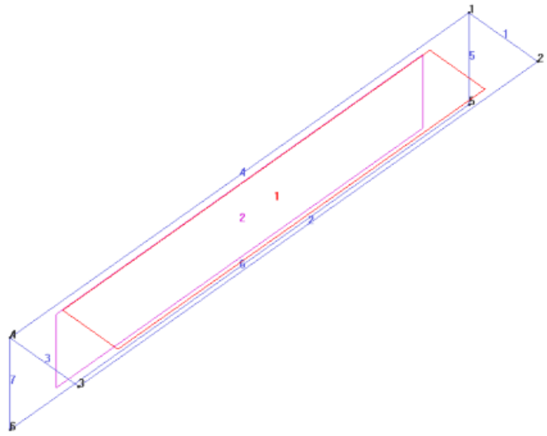
In the dial box, with *Draw->Colors* you can display the constraints.
Press *Finish* or *[Esc]* to leave this view mode.

3.5. Assign the loads

From the pull down menu select
Data->Loads
Select *Global Shell Load*

Specify a distributed load of *-100 N/m²* in *Z direction*
Use the function *User Defined*
As Filename, write *load.fct*
Assign this load on the S*urface 1*
Press *Finish* or *[Esc]* to validate

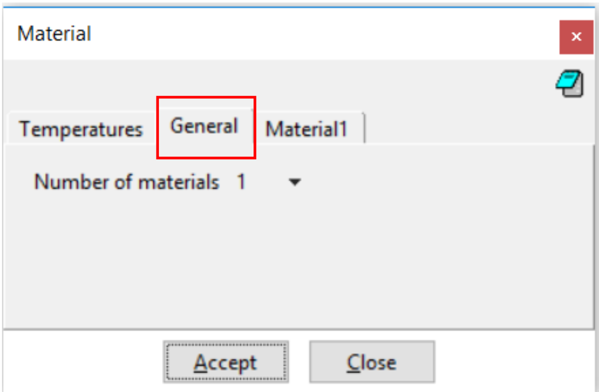


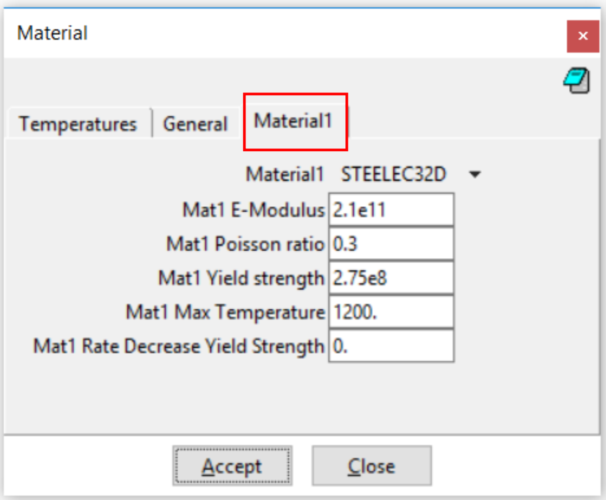


3.6. Define the global materials

From the pull down menu select
Data->Material

There is only 1 material in the model: STEELEC32D (note that it is a plane stress model)

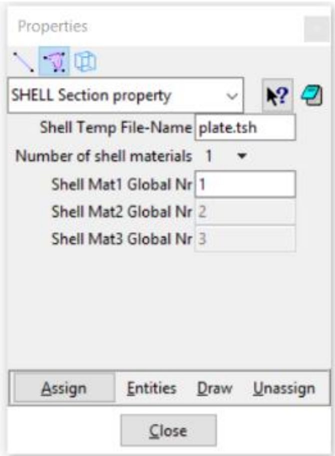


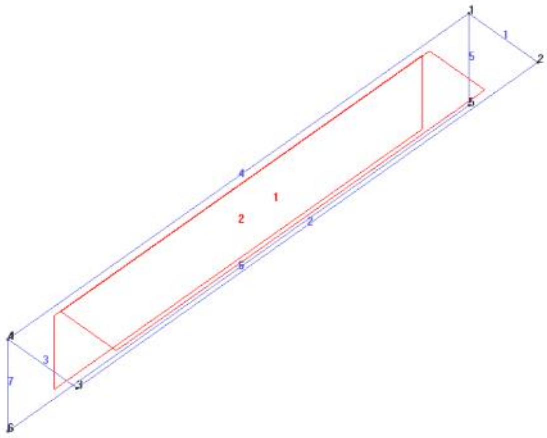


3.7. Define the properties (i.e. assign temperature files)

From the pull down menu select
Data->Properties

The objective is to assign the .tsh file named *plate.tsh* to the model surfaces.
In the dial box of *Data->Properties*, select the *SHELL Section Property*
Change the File-Name to ***plate.tsh***
Keep the number of materials to 1
Assign the ***plate.tsh*** section to the surfaces.

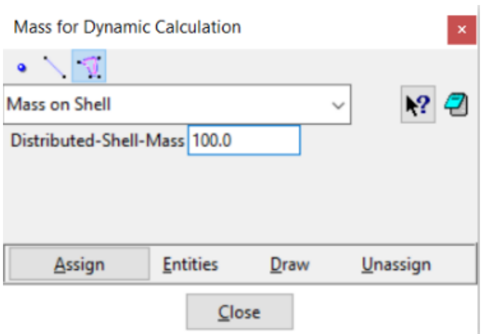




You can draw the properties to check the model.
Select ***Draw -> Colors***

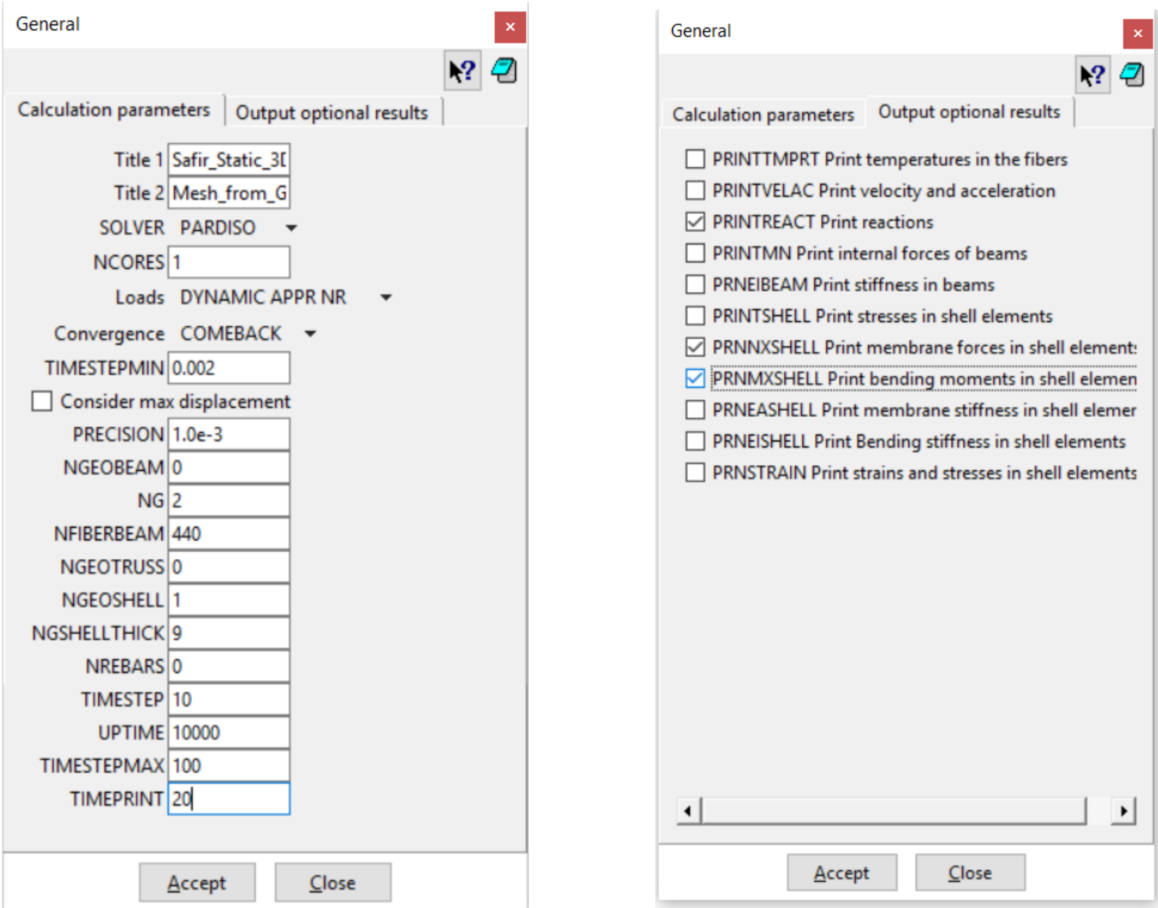
3.8. Assign the mass

To define the mass for dynamic calculation, select from the pull down menu:
Data-> Mass
Select ***Mass on Shell*** and put 100 kg/m² as Distributed-Shell-Mass.
Assign the mass to the two surfaces and validate.



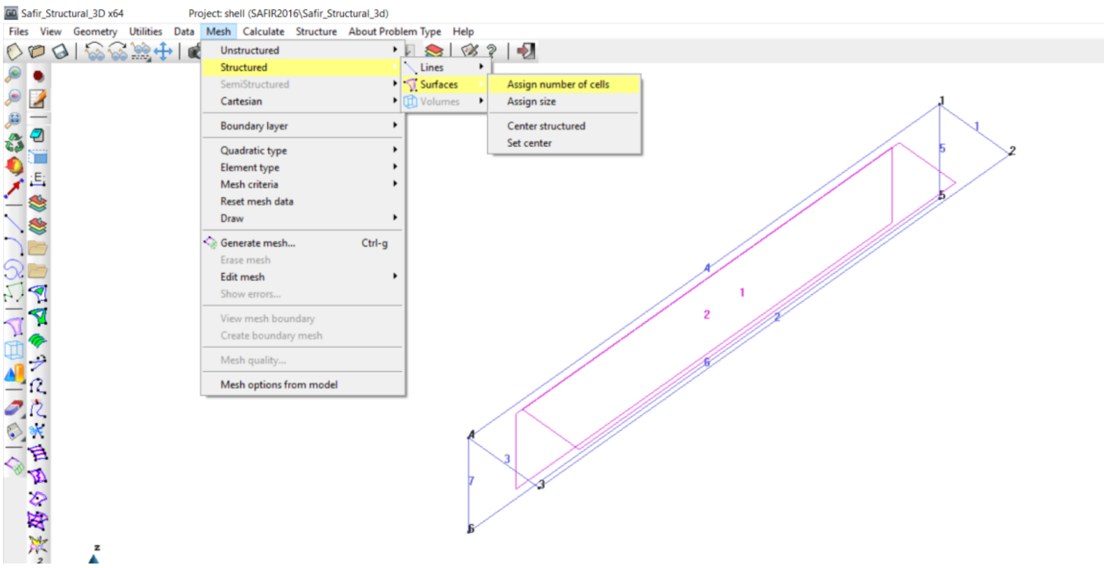
3.9. Define the general problem data

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



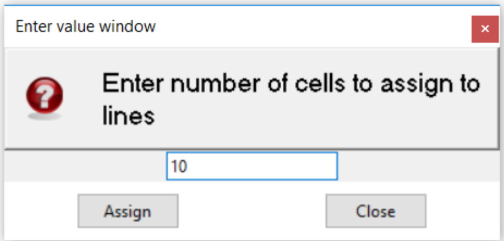
3.10. Define the mesh

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

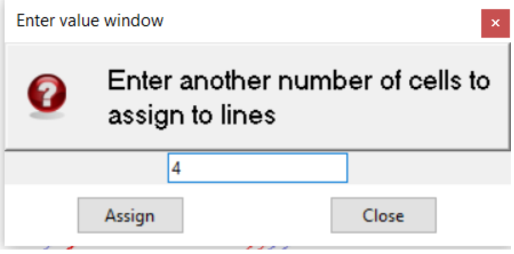


Select the two surfaces, for which a structured mesh will be applied. Press *[Esc]* to validate.

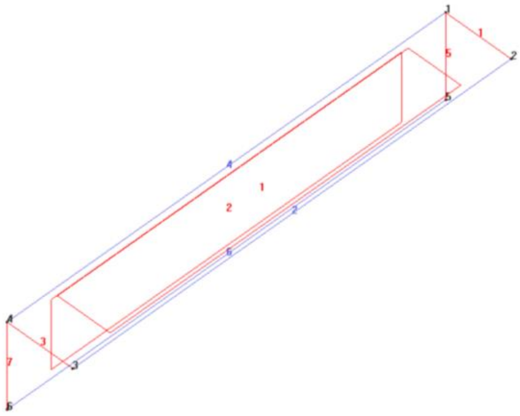
Enter 10 as the number of cells to assign to lines. Select the lines 2, 4 and 6 (1 m long lines). Press *[Esc]* to validate.



Then enter 4 as another number of cells to assign to lines.

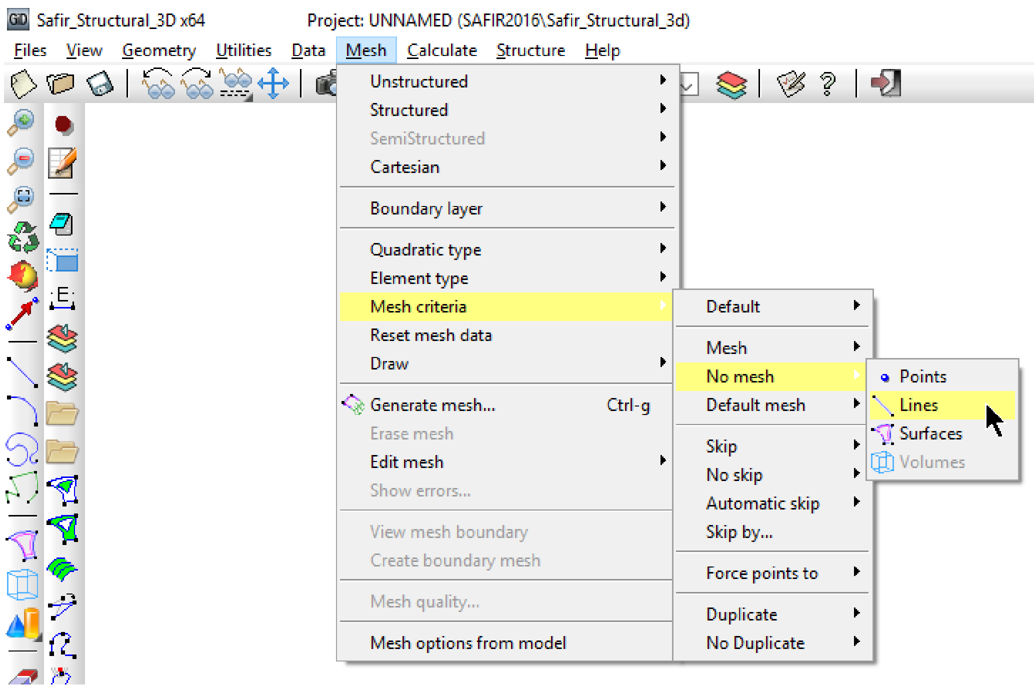


Select the lines 1, 3, 5 and 7 (short lines). Press *[Esc]* to validate.

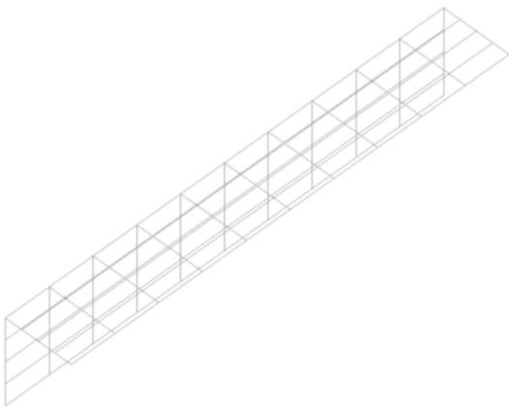


Click on ***Close***.

Next, it must be specified that the lines in the model need not be meshed as beam finite elements.
Select ***Mesh -> Mesh criteria -> No mesh -> Lines***
Select all the lines in the model. Press *[Esc]* to validate.



Select *Generate Mesh, OK* and then *View Mesh*

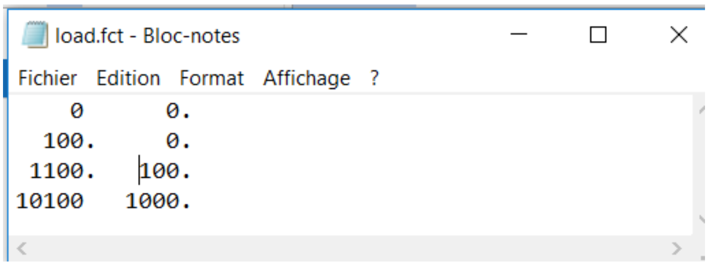


3.11. Create the loading file

In this model a user-defined load function was used, named *load.fct* (see 3.5). This file needs to be created and located in the directory *shell.gid*.

The text file *load.fct* can be edited in a text editor. It is structured as shown below, where the first column is the time (in sec) and the second column is the load (in N). You can define as many time-load pairs as needed. SAFIR will interpolate linearly between the given values.
Here, it is decided to keep the load to zero while the structure is heated (from 0 to 100 sec) then to increase the load linearly as time/10.

Note that here this function will be multiplied by -100 N/m² as defined in Section 3.5. As a result, we increase the load by 1 kN/m² every 100 sec in the vertical direction pointing downwards, starting after 100 sec.

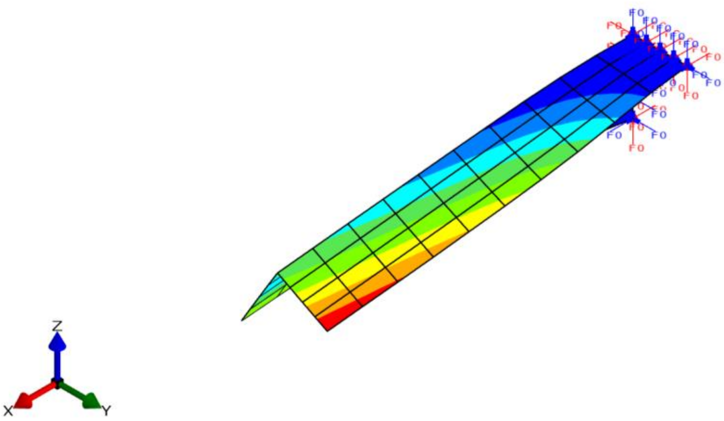


3.12. Start the calculation

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*

3.13. Check the results

Open the .XML file in Diamond to check the model.



Here below is the result with a finer mesh (not suitable for the demonstration version).