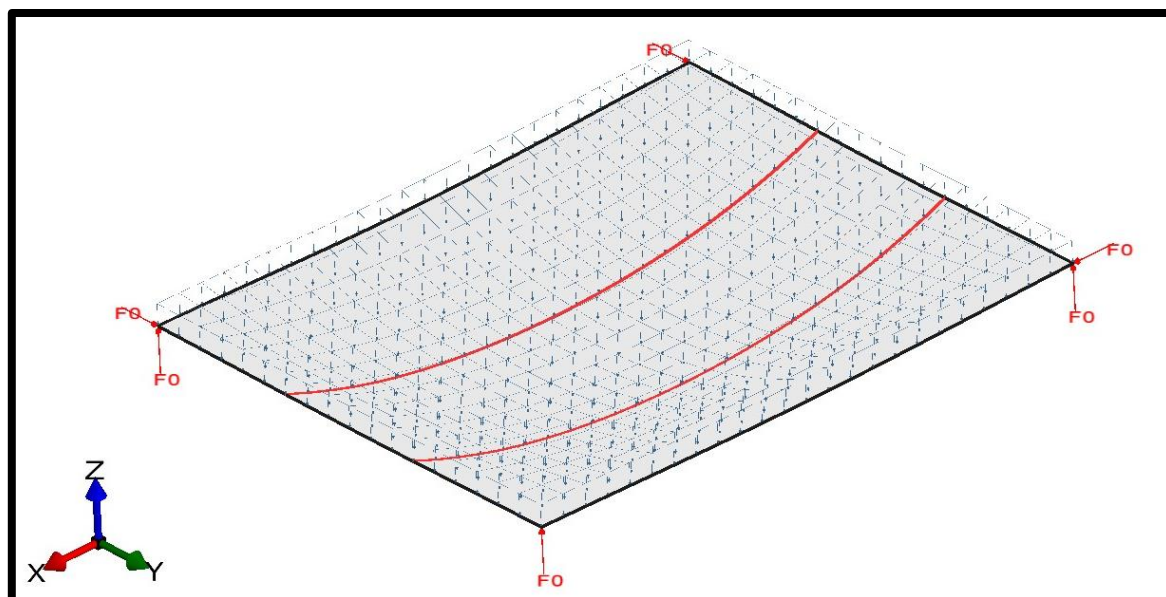# SAFIR® training session – level 1

Johns Hopkins University, Baltimore

Example: 3D structural model with shell and beam elements

"Membrane effect in a reinforced concrete slab supported by steel beams"

T. Gernay & J.M. Franssen

1. General description

This example deals with a steel-concrete structure. A reinforced concrete slab is supported by peripheral and central steel beams. The structure is simply supported at its four corners.

General data:
- 15 m by 12 m plane dimensions
- Reinforced concrete flat slab of 0.16 m thickness
    - Two layers of orthogonal steel mesh (393 mm²/m per layer per direction)
- Protected (cold) peripheral steel beams IPE550
- Two unprotected (hot) central steel beams IPE550
    - The central beams span 15 m and are spaced by 4 m
- Simply supported
- Fire: natural fire curve with a cooling phase
- Loads: uniform distributed load
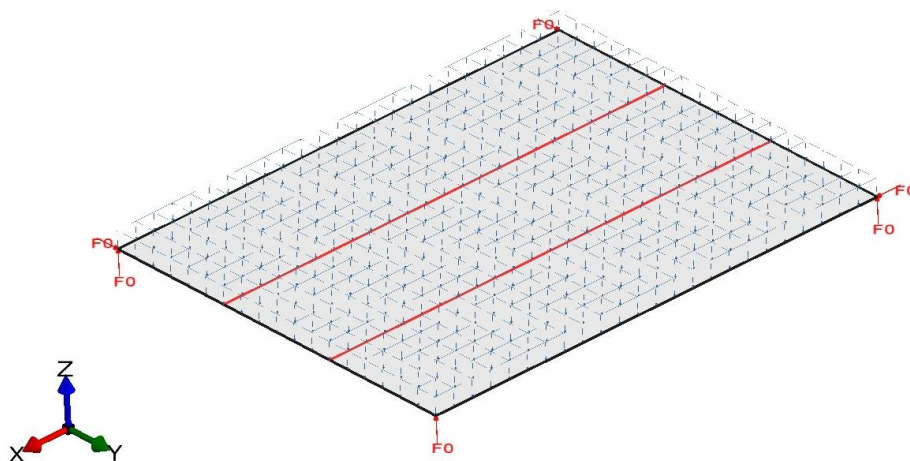
Concrete:
- Compressive strength 30 MPa
- Tensile strength 0.5 MPa

Steel for IPE550:
- Yield strength 355 MPa

Steel for reinforcement bars:
- Yield strength 500 MPa

2. Create section for IPE550 steel beam

The objective is to create a IPE550 section with a slab, exposed to the natural fire curve.

The section file has to be created for a subsequent 3D structural analysis. Therefore, it will also include a torsional analysis.

2.1. Create a project in 2D for Thermal Analysis

From the pull down menu select:
Data -> Problem type -> SAFIR2016 -> Safir_Thermal_2d
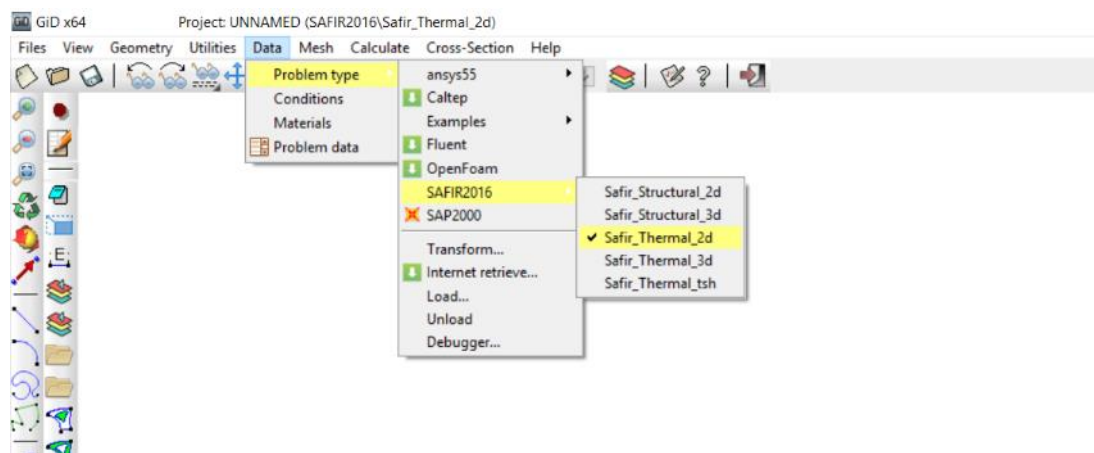
To save the project select (or use icons on the left):
Files->Save or or [Ctrl + s]

Enter a file name, e.g.: *IPE550*
GiD creates a directory with the name *IPE550.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.
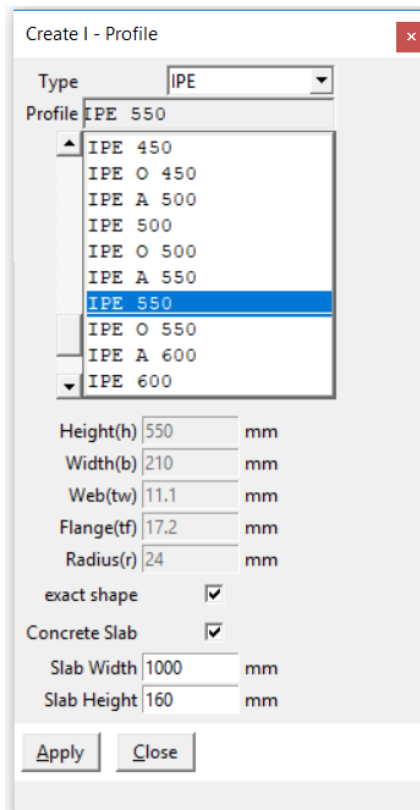
2.2. Create the geometry in the xy-plane

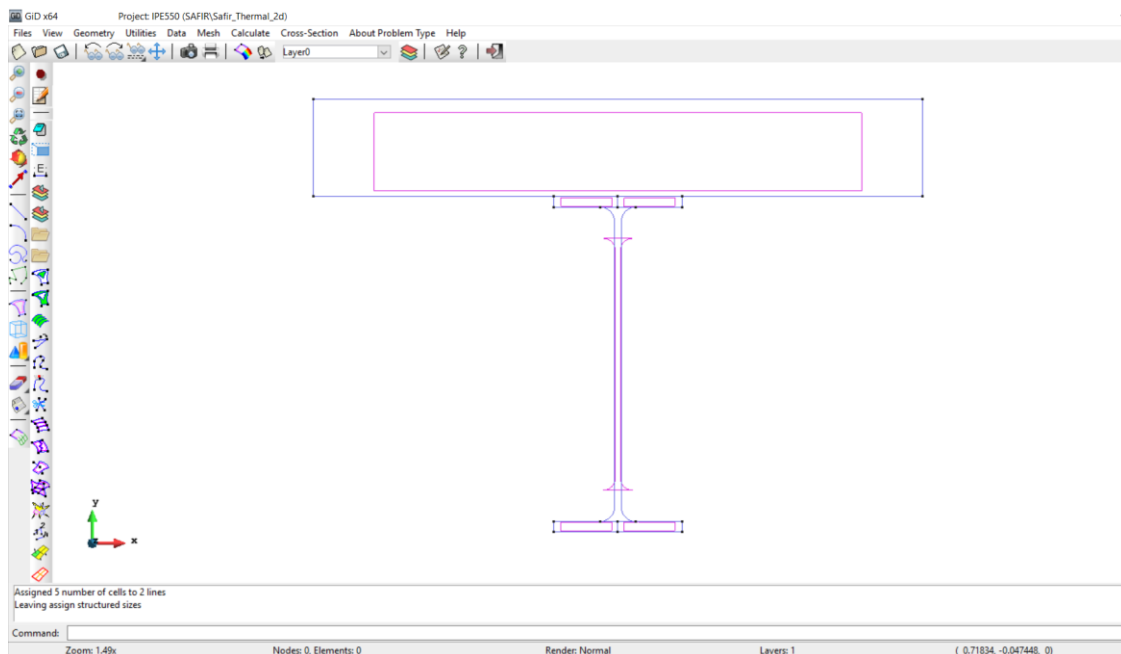From the pull down menu select:
Cross-Section->I-Profile

Select *IPE* as type, *IPE 550* as Profile, tick *exact shape*, put *1000* mm as slab width and a slab height of *160* mm
Click on *Apply*

Note: GiD-Safir will create an IPE550 profile. The center of this profile will be automatically centered on the 0,0 point of the xy-plan.

2.3. Create and assign a temperature curve

In this exercise, we will suppose that you want to import your own fire curve of temperature. In order to permit to GiD-Safir to use your data, you have to first create a .txt document. Open your Notepad and wrote as many line as necessary, each line being, in a free format, a pair of values in the form:

<p align="center">*TIME   TEMPERATURE*</p>

Save the file created. Here, we are going to save under the name: *Fire.txt*
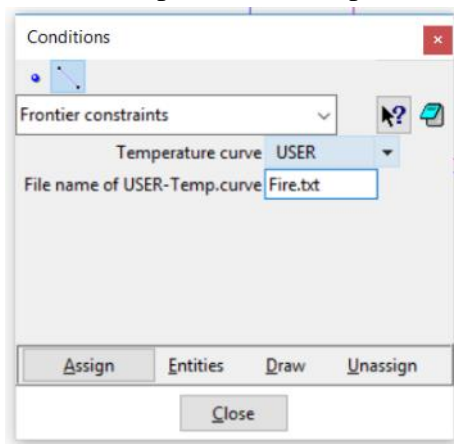Then copy the *Fire.txt* file in the *IPE550.GiD* file you just created.
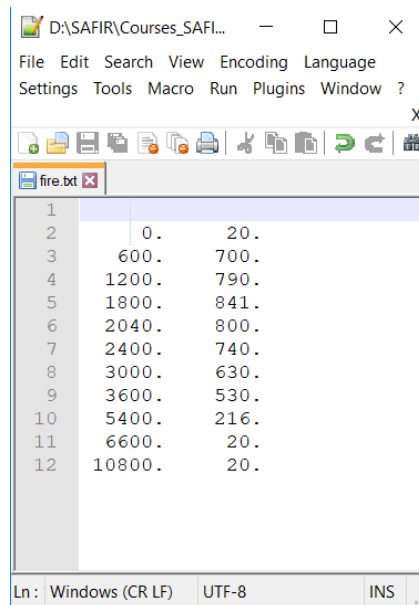
In GiD, from the pull down menu select:
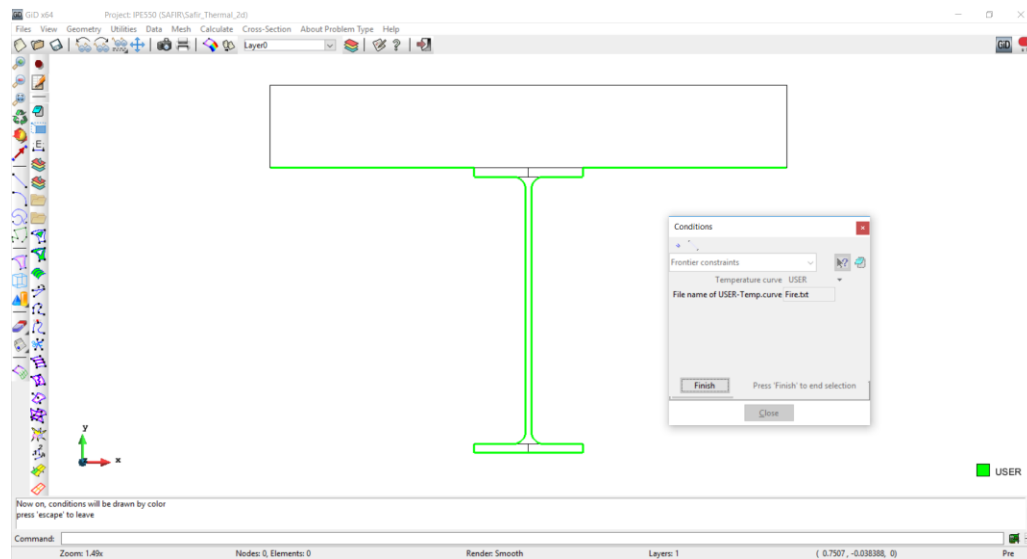*Data->Conditions*

Select the ⬒ button
On the first pull down list: *Frontier constraints*
On the Temperature curve pull down list *USER*

Write the File name of user temperature curve in the dial box (*Fire.txt* in this example).

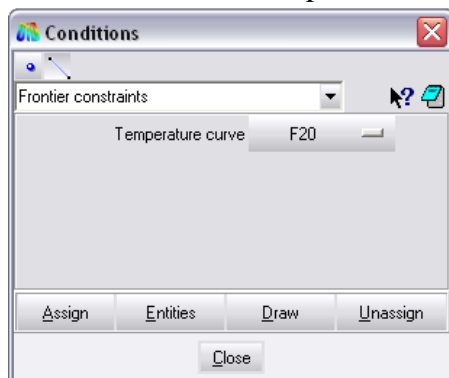Click on the *Assign* button and assign it to profile and slab lines as shown below.
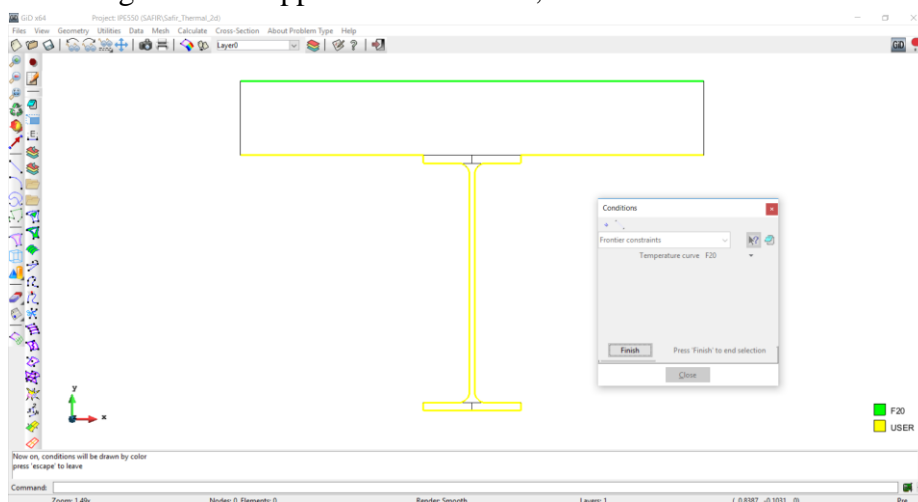

Press *[Esc]* or click on *Finish* to confirm
Select *DRAW->Colors* in the Conditions dialog box to display the frontier constraints
Press *[Esc]* or click on *Finish* to leave this view mode

Then select *F20* as temperature curve

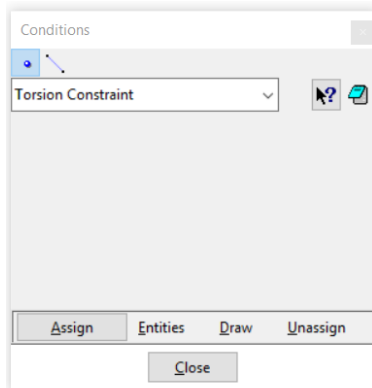And assign it to the upper line of the slab, as shown below:

2.4. Assign a torsion constraint (for the torsional analysis)

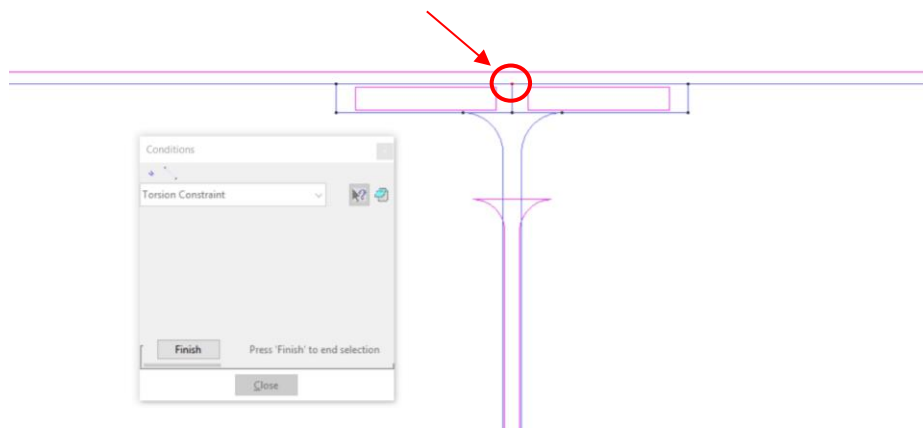In GiD, from the pull down menu select:
Data->Conditions

Select the ⬜ button
On the pull down list: *Torsion constraints*

Select the node on the vertical axis of symmetry of the steel profile, at the connection with the slab (see below). 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
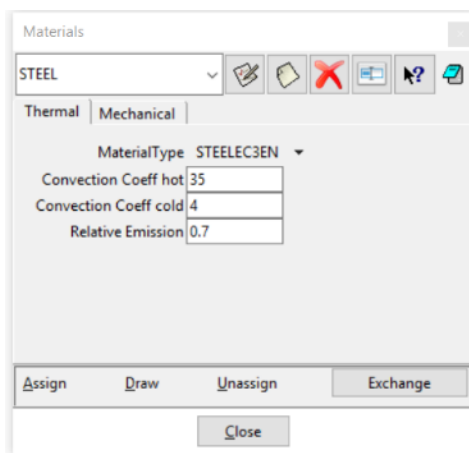The *Thermal* tab is active.
Then select:
STEELEC3EN as Material Type
    A Convection Coeff hot of *35*
    A Convection Coeff cold of *4*
    A Relative Emission of *0.7*
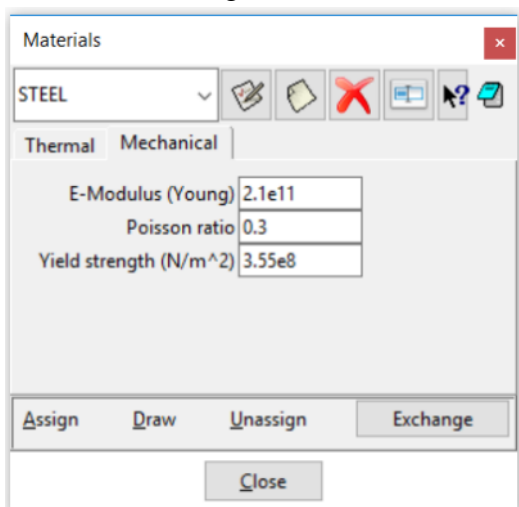

Then select the *Mechanical* tab. Input:
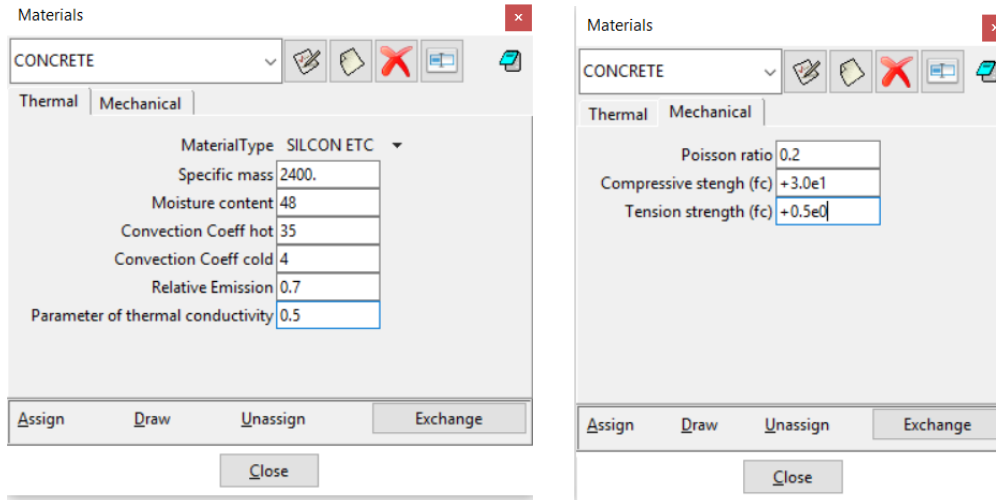   A Young modulus of *210 000 MPa*
    A Poison ratio of *0.3*
    A Yield strength of *355 MPa*

Click on *Assign-> Surfaces* and assign it to the IPE550 surface
Press *[Esc]* or *Finish* to confirm

Select *CONCRETE* from the dialog box pull down list. Then fill as below:

<u>Note</u>: The compressive strength has been divided by 10^6 (value: 30 N/m²). This is because the mechanical properties input here are used for the torsional analysis. In this torsional analysis, we want to consider the stiffness of the beam only (not the slab), because, in the structural model, the slab will be modeled by shells. In the concrete model, the Young modulus is calculated from the compressive strength; hence, reducing the strength automatically reduces the stiffness.

Click on *Assign-> Surfaces* and assign it to the slab 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
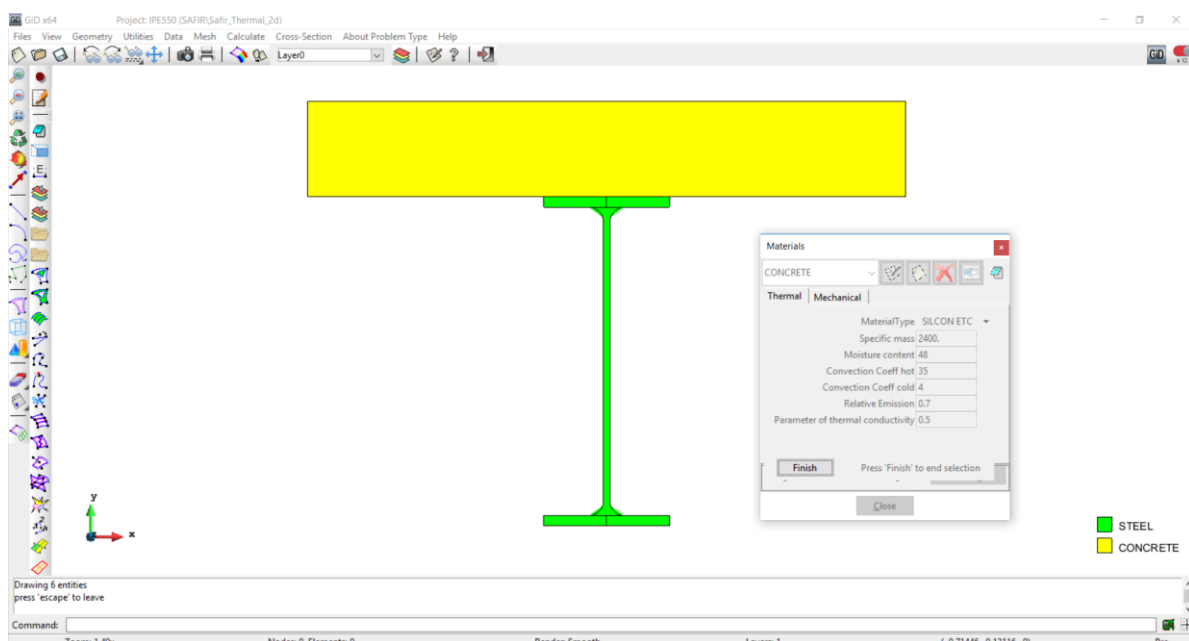
2.6. Assign the general data
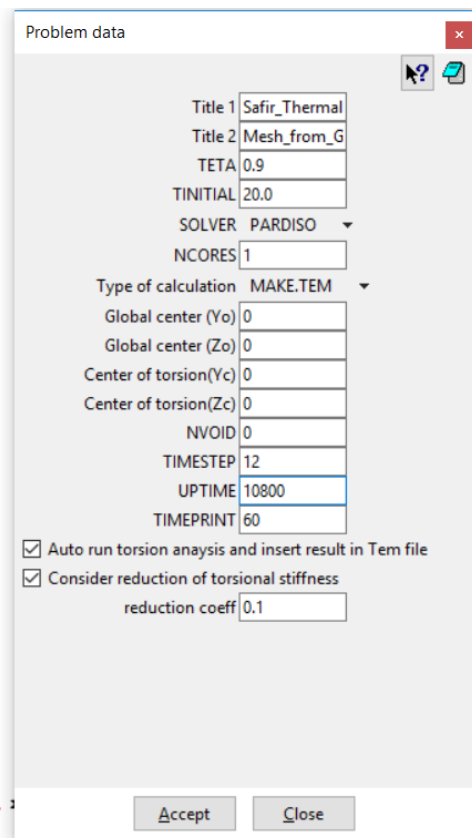
From the pull down menu select:
Data->Problem Data

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

Note: The global center of coordinates (0,0) is by default positioned at the center of the steel profile. This position can be adjusted here to consider the relative position of the steel profile with respect to the rest of the structure in the structural model. This will be discussed later when defining the structural model.

Note2: In this case we are using a temperature curve with user's values. As you can see in the Fire.txt file, the last time step is at 10800s. The UPTIME have to be lower or equal to this value.
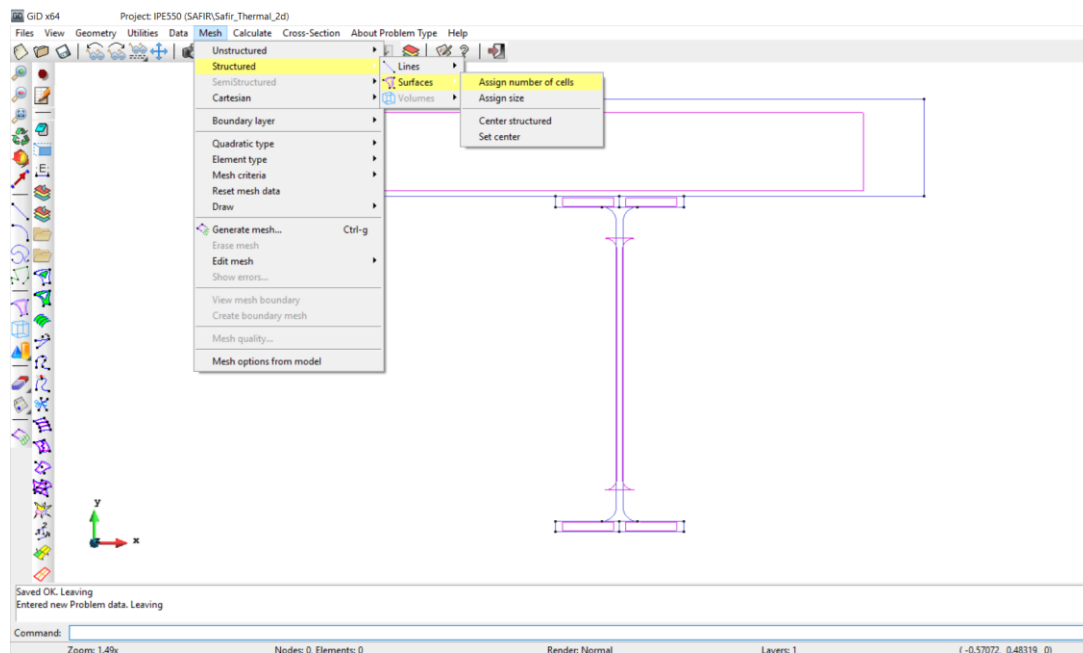
2.7. Create the mesh

We will assign 10 elements over the slab thickness to reduce the skin effect.

Select *Mesh->Structured->Surfaces->Assign number of cells*
Select the slab surface, press *[Esc]* to validate
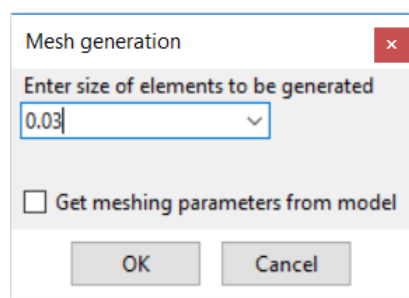Enter *10* as the number of cells to assign to lines
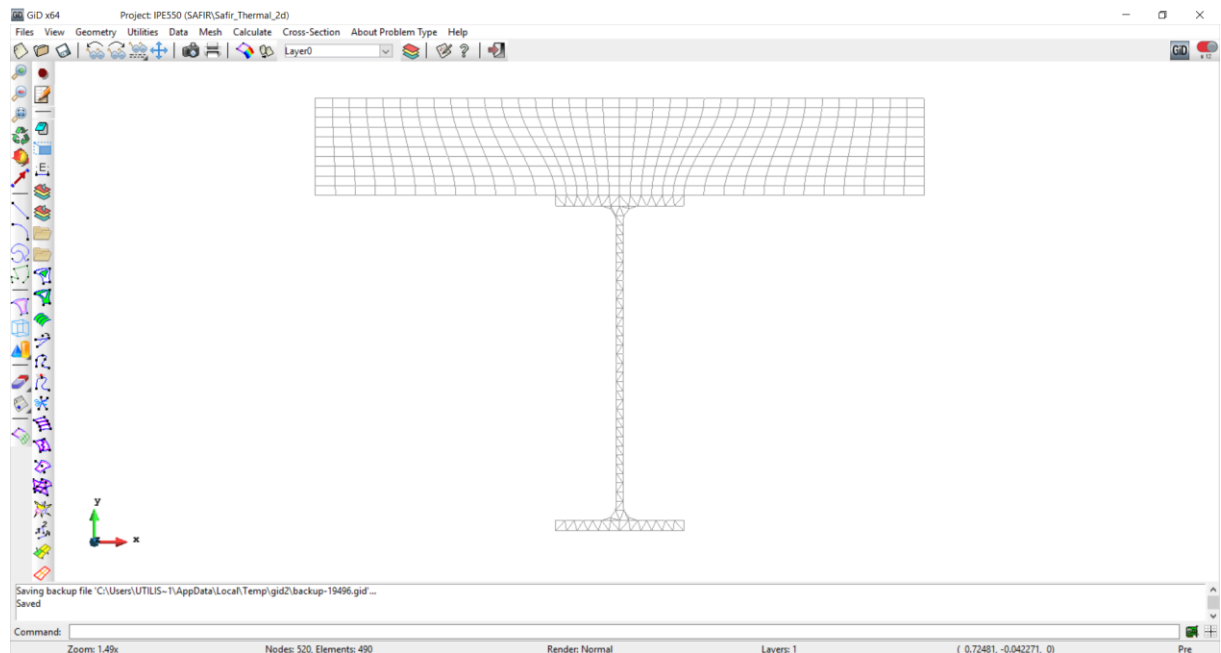Select the left vertical line of the slab and press *[Esc]* to validate

Select *Mesh->Generate mesh or use [Ctrl + g]*
Enter 0.03 as size of elements to be generated
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.
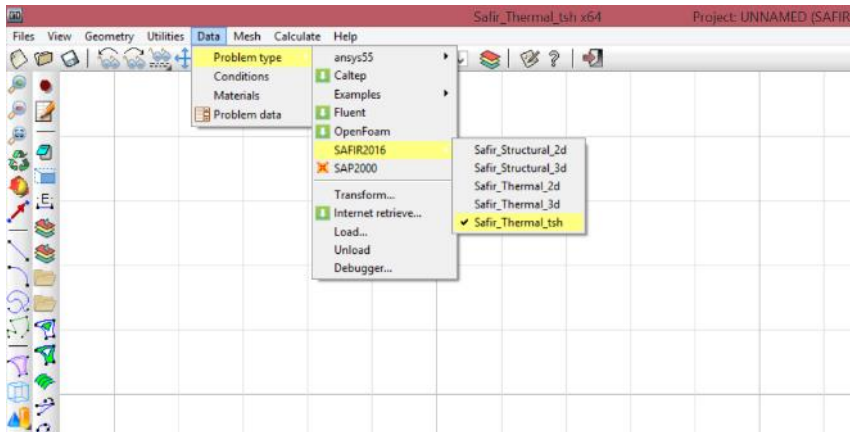Once the calculation is done, you can open the .XML file with DIAMOND.
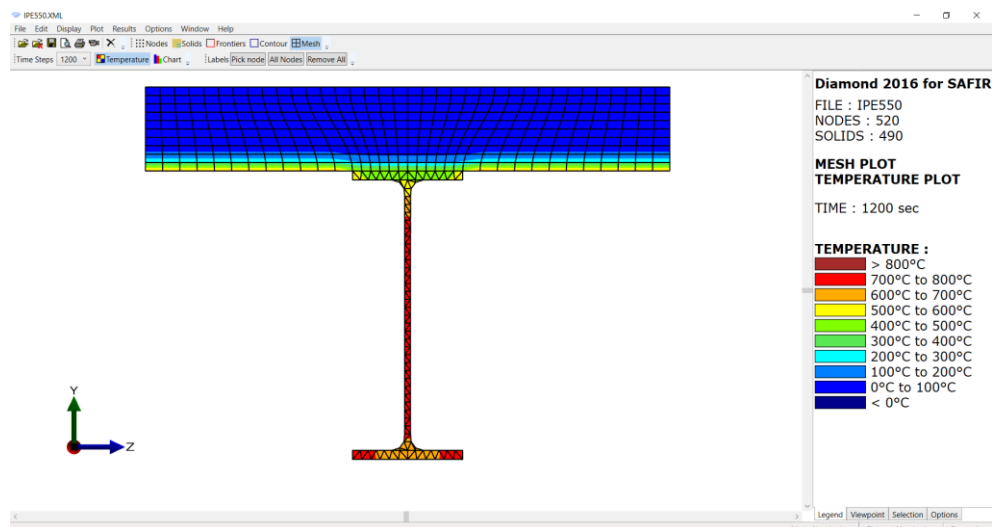
3. Create section for concrete slab

3.1. 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 🖫   or [Ctrl + s]

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

Enter a file name, e.g.: Slab16
GiD creates a directory with the name *Slab16.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.

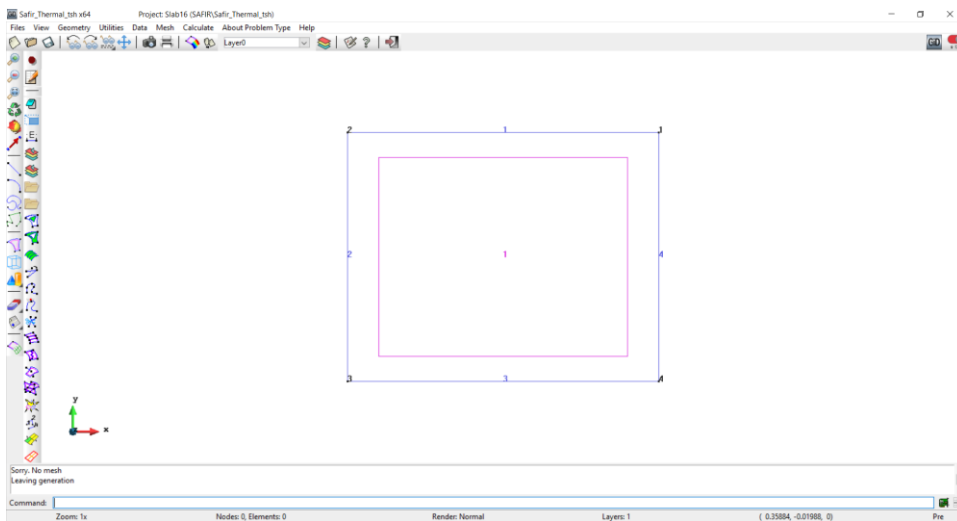Copy the file *Fire.txt* file in the *Slab16.GiD* directory you just created.

3.2. Create the geometry (slab cross section) in the xy-plane

GiD will open automatically a new window.
Put *0.16* m as shell thickness and *20* as number of elements, as shown below:
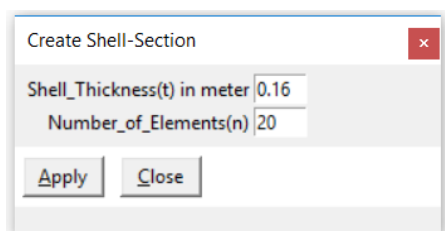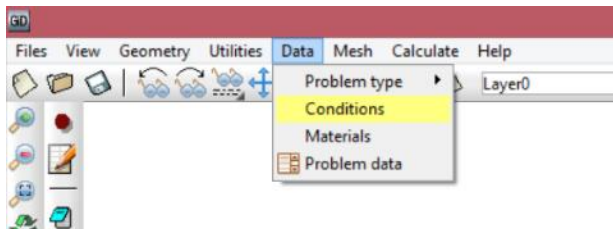
Click on *Apply*

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).

3.3. Assign the thermal boundary conditions

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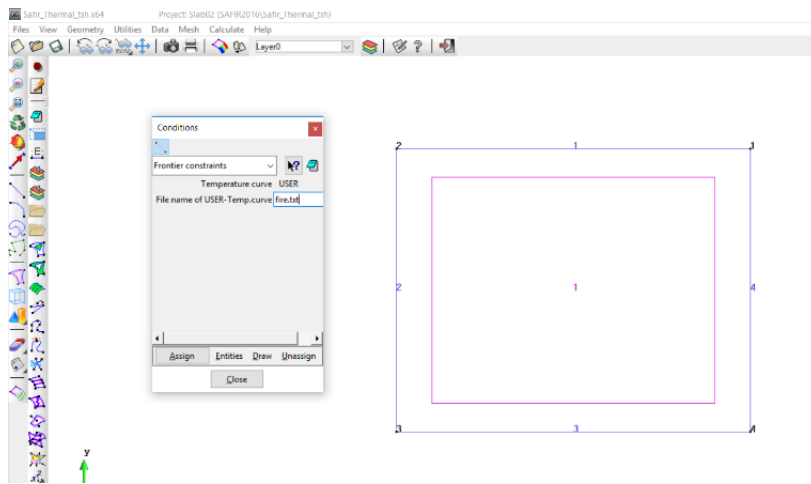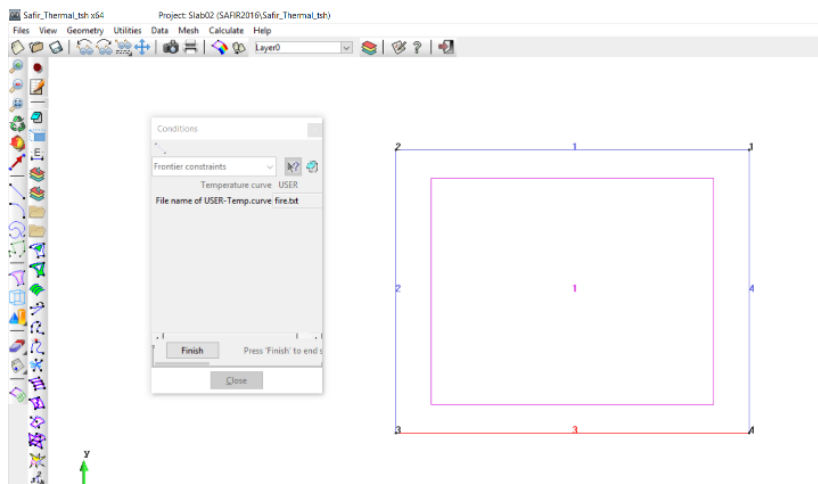*Data->Conditions*

Select:
On the first pull down list: *Frontier constraints*
On the Temperature curve pull down list *USER*
Write the File name: *Fire.txt*

Click on the *Assign* button and assign it to the lower line as shown below.
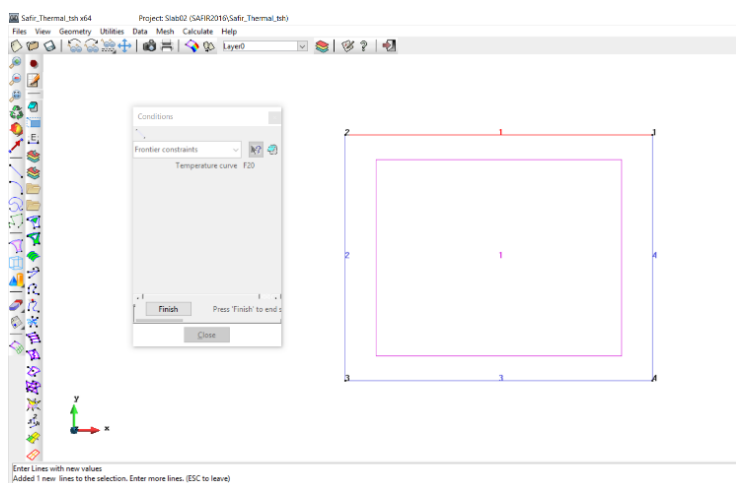

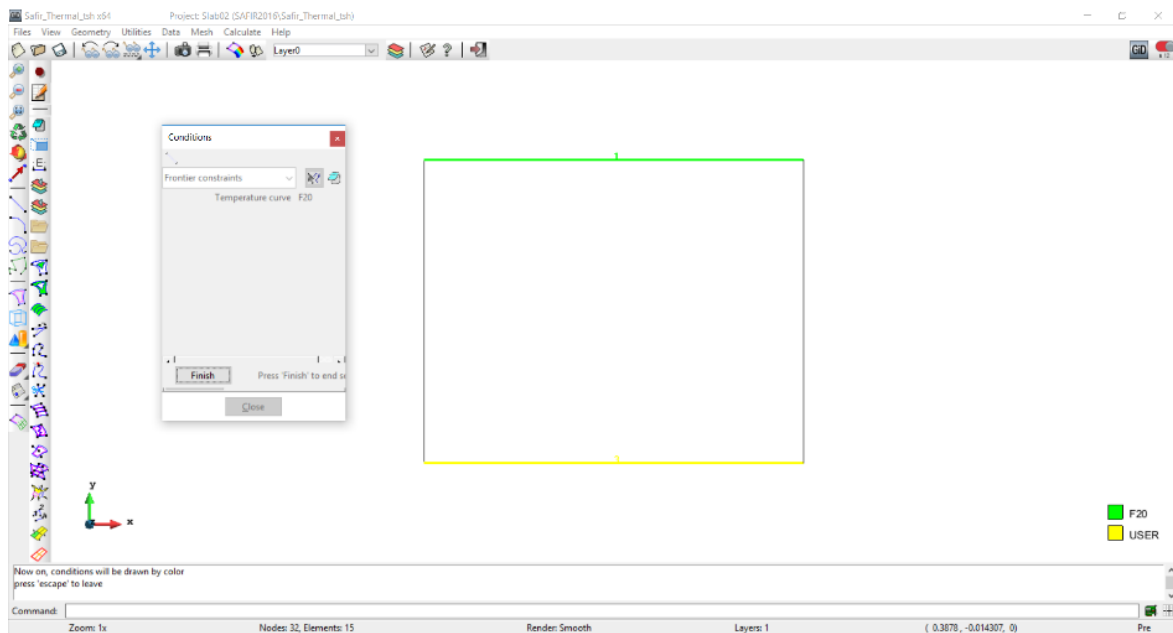Press *[Esc]* or click on *Finish* to confirm

Next, select:
On the Temperature curve pull down list *F20*
Click on the *Assign* button and assign it the upper line as shown below

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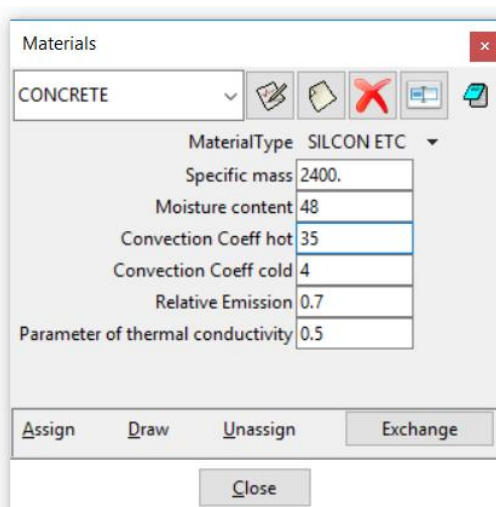Press *[Esc]* or click on *Finish* to leave this view mode

You will need to have the file *Fire.txt* file in the *Slab16.GiD* directory for the analysis to run.

3.4. Assign materials

From the pull down menu select:
Data->Materials
Select *CONCRETE* from the dialog box pull down list

Then select:
SILCON ETC as Material Type
A specific mass of *2400*
A moisture content of *48*
A Convection Coeff hot of *35*
A Convection Coeff cold of *4*
A Relative Emission of *0.7*
A Parameter of thermal conductivity of *0.5*

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

Note: a description of the material *SILCON ETC* is given in the following paper (for the mechanical behavior). Please cite this paper when using *SILCON ETC* or *CALCON ETC*.

> Gernay, T., Franssen, J.-M. (2012). A formulation of the Eurocode 2 concrete model at elevated temperature that includes an explicit term for transient creep. *Fire Safety Journal*, 51, 1-9. http://hdl.handle.net/2268/114050

Select *DRAW->all materials* in the Material dialog box to display Materials
Press *[Esc]* or *Finish* to leave
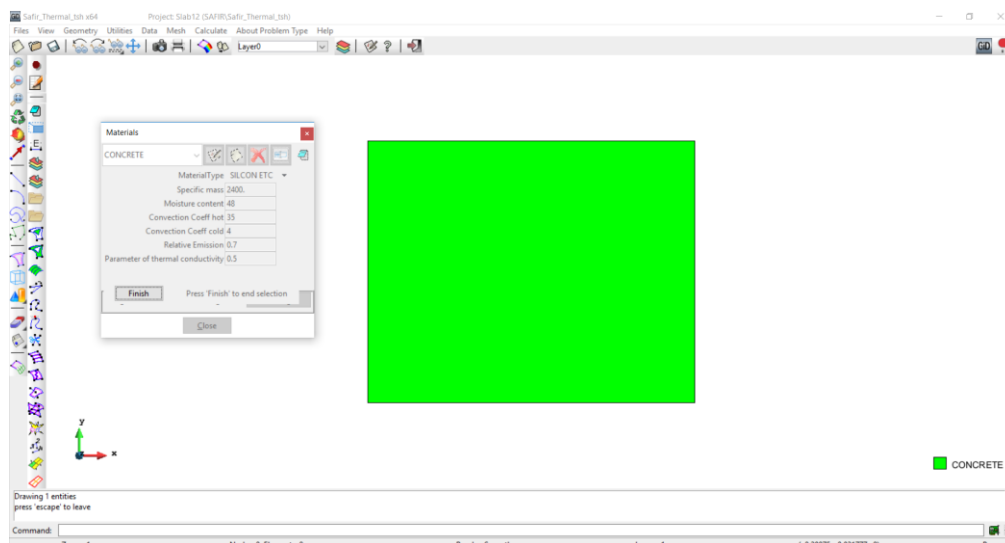
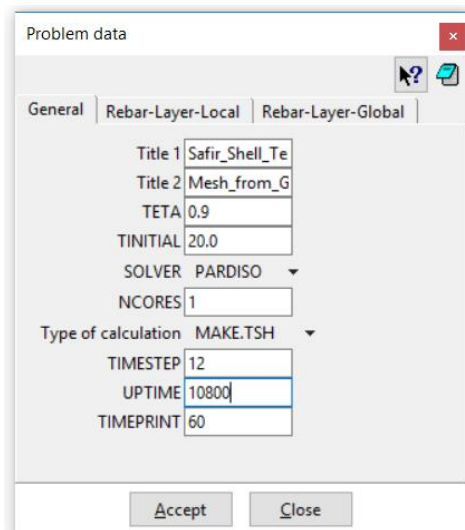### 3.5. Define the general problem data

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

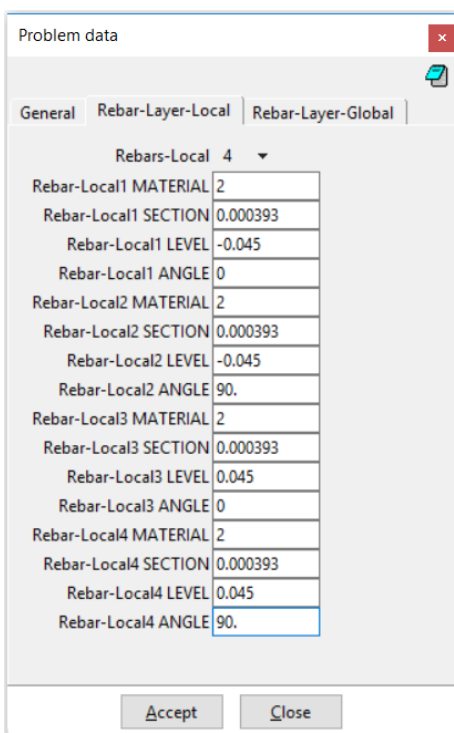In the Problem Data dialog mask enter:
TIMESTEP, UPTIME, TIMEPRINT as needed

Click on the *Accept* data button

Note: When you click with the left button on one of the variables, GiD will display a short description of the variable. The variables are also described in more detail in the SAFIR manual.

Click on the *Rebar-Layer-Local* tab and fill as shown below:

Rebars-Local is the number of rebar layers in the section type. Here it is equal to 4, because the mesh consists in one upper layer and one lower layer in each orthogonal direction.

Rebar-Local1 MATERIAL is the local material number of the layer 1. In this case it is equal to 2 (concrete is material 1, steel is material 2). The rebar material type will have to be defined later for the mechanical calculation.

Rebar-Local1 SECTION is the cross sectional area of the rebars in this layer 1 in [m²/m]. In this case, it is equal to 393 mm²/m, assuming a 200mm x 200mm grid with a rebar diameter of 10mm:

$$\text{Section} = \pi * (0.010^2/4) / 0.200 = 393 \text{ e-6 m}^2/\text{m}$$

Rebar-Local1 LEVEL is the position of this layer in y axis with respect to the thickness (the TSH section is by default centered in y-axis). Here the cover from lower face to rebar axis is 35 mm; therefore the *LEVEL* is –(0.080-0.035) = -0.045

The same data has to be input for the other layer: *Rebar-Local2-3-4* (because here the mesh is equal in both directions and both upper and lower layers), but with an angle of 90° (orthogonal) and a position of 0.045 for the upper layer.

Click *Accept* and then *Close*.

Note that alternatively, you can define the rebars in the global axes.
In that case, click on the *Rebar-Layer-Global* tab and fill as shown below.

Rebar-Global1 N1-N3 defines a vector in the global system of coordinates of the structure. The rebars have the orientation of the line which is the intersection between the shell element and a plane that is perpendicular to the vector (see SAFIR manual).

If the norm of the vector is 0, then the orientation of this bar layer is perpendicular, in each element, to the previous bar layer (not possible for bar layer 1).

Here, considering that the shell elements will be defined in the plane x-y in the global system of coordinates, the first vector is given by <1,0,0>. As a result, the rebar layer will be oriented in the direction parallel to the global y-axis.

3.6. Define the mesh

To create meshes select from the pull down menu:
Mesh->Generate mesh   or use *[Ctrl + g]*

The number of elements will be as already defined above.

Note: In .TSH thermal analysis, the heat transfer is unidirectional across the thickness of the shell element. Therefore, only 1 FE is used on the width (i.e. in x-direction). Also, the width is irrelevant for the calculation (whether it is 1 m or 0.1 m will have no impact on the calculation).
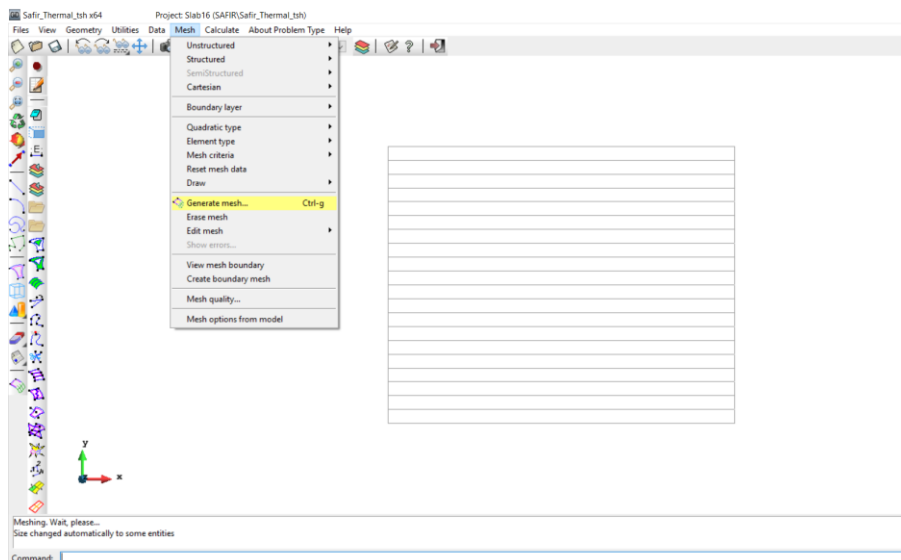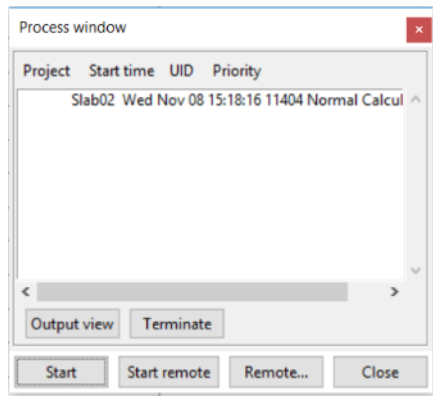

3.7. Start the calculation

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

3.8. Debug - Check the results

If there is an error message, or the calculation did not run properly:
- Check that GiD has created a .IN file in the project directory
- Check that a .OUT file is also present in the project directory
- Open the .OUT file, go to the end of the file and check whether there is an error message
- It is also interesting to look at the .IN file line by line, with the help of the SAFIR manual, to verify whether the model is correct

When the calculation has run properly, a .OUT and a .XML files have been created which contain the results. You can open the .XML file using DIAMOND. There, you can check the geometry, the mesh, the thermal boundary conditions, the distribution of temperature across the section as a function of time, etc. *It is important to have a look at the results with a critical mind before proceeding to the next step, which will make use of the thermal results*.
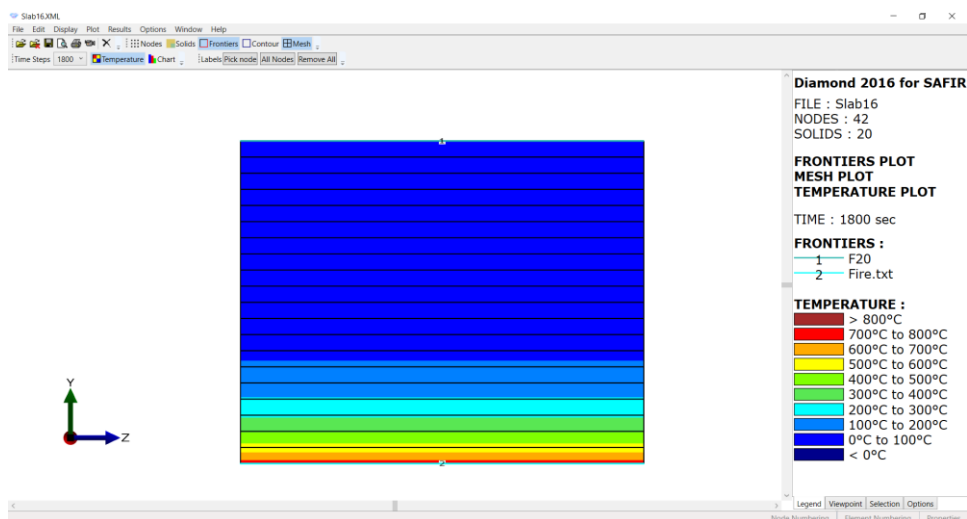
4. Create model for the 3D structure

4.1. Modify the node line in the .TEM file

As a general rule in GiD:

- If you are using the "Cross-Section" option to create a profile in thermal 2D GiD-Safir always center the profile on the 0,0 point of the plan who is located at the middle of the section height as shown below.
- If you are using the thermal TSH mode, GiD-Safir will automatically center the shell element node line on 0,0.

As a result, currently, the nodelines of our beam (IPE550.tem) and our slab (Slab16.tsh) are not consistent. We have to adjust them for the structural model.
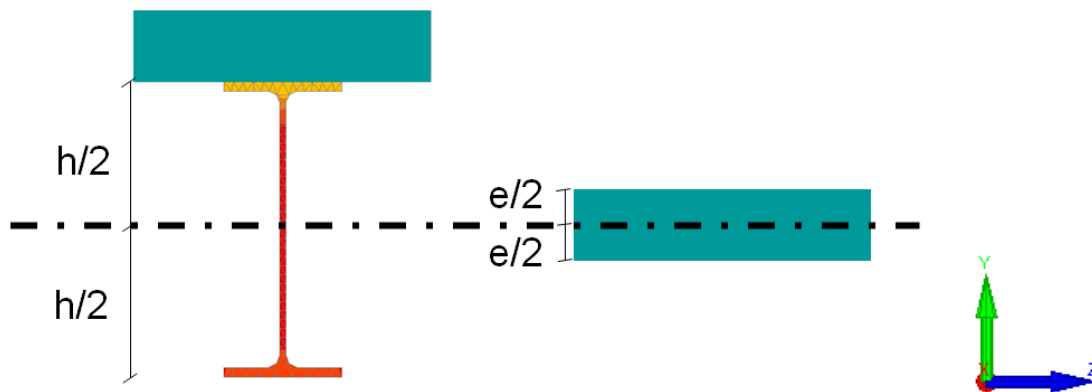
Open the ipe550.tem file.
On the 3rd line you can see the line "NODELINE". The two 0.0000E+00 numbers are respectively the Y0 and the Z0 coordinate in meter of the global center.

To displace the local center as show below, write Y0 = 0.3550E+00
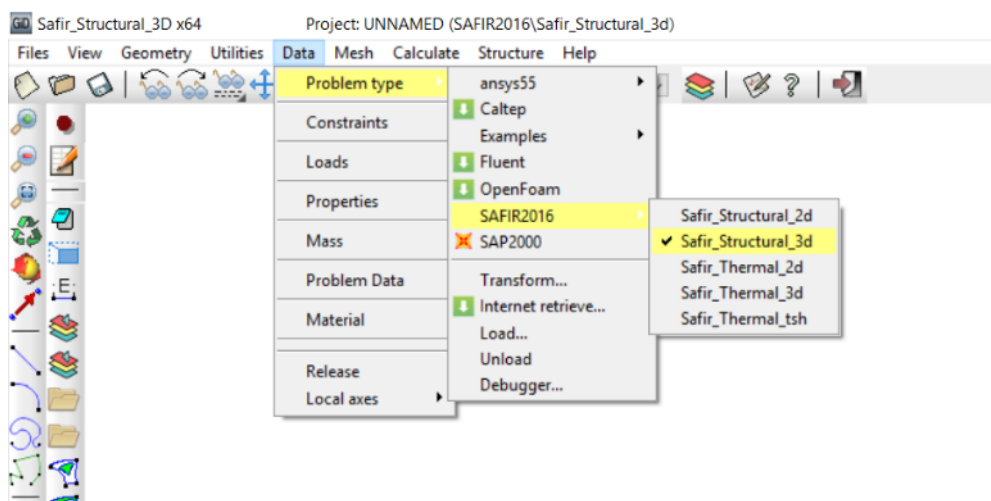(calculated as: h/2 + e/2 = 275 mm + 80 mm = 0.355 m)

Save this file and use it in the subsequent structural analysis.

4.2. 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.: *3DMembrane*
GiD creates a directory with the name *3DMembrane.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.

4.3. Copy-Paste the section files in the structural analysis directory

GiD has created the directory *3DMembrane.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 *'ipe550.tem'* and *'Slab16.tsh'* in the directory *3DMembrane.gid*

In the model, the peripheral beams are assumed to remain at ambient temperature (considering that there are sufficiently protected from the fire). We will thus create a new section, which is the same as *'ipe550.tem'*, but remains at ambient temperature.

Create a copy of the file *'ipe550.tem'*. Rename it *'ipe550c.tem'*.
Open the file *'ipe550c.tem'*. Change the line *HOT* into *COLD*.
Save and close the file *'ipe550c.tem'*.
Copy and paste the file *'ipe550c.tem'* in the directory *3DMembrane.gid*

4.4. Create the system geometry (3D structure)

The view is by default in the x-y plane. Here, the plane of the shells will be defined in the x-y plane, thus the view can be kept as is.
Yet if you want to change the view:
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
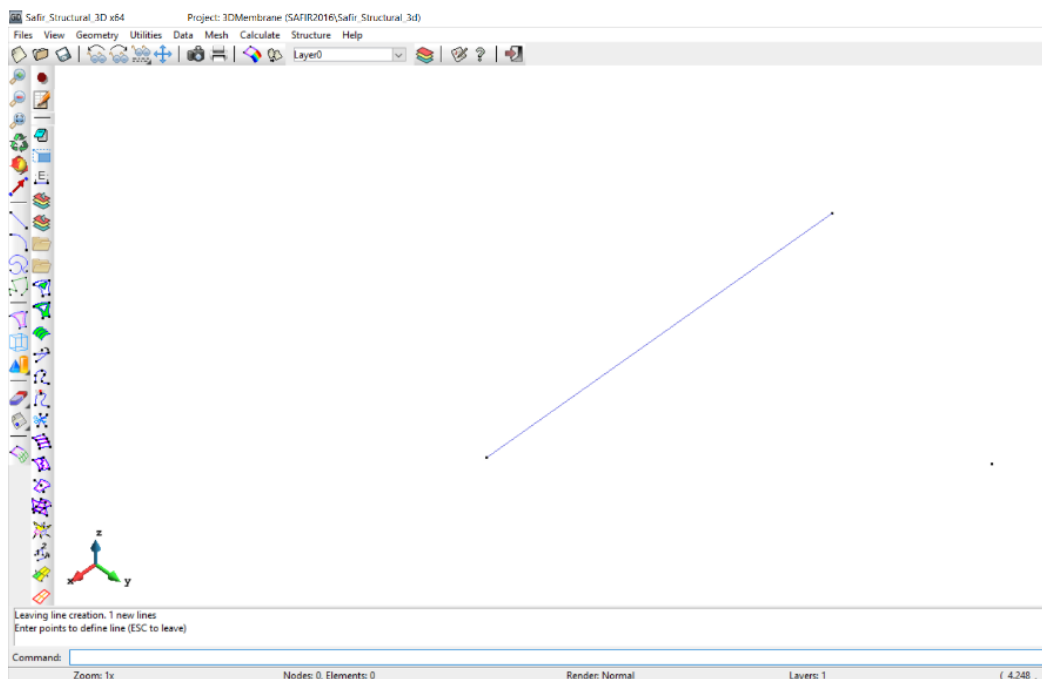
Create the system lines:

Geometry->Create->Straight Line   or

Enter in the command line (at the bottom of the widows):
0,0,0 15,0,0   and press *[Enter]*
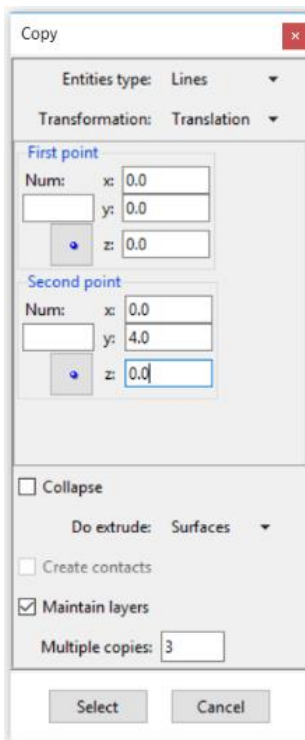Then press *[Esc]* to leave
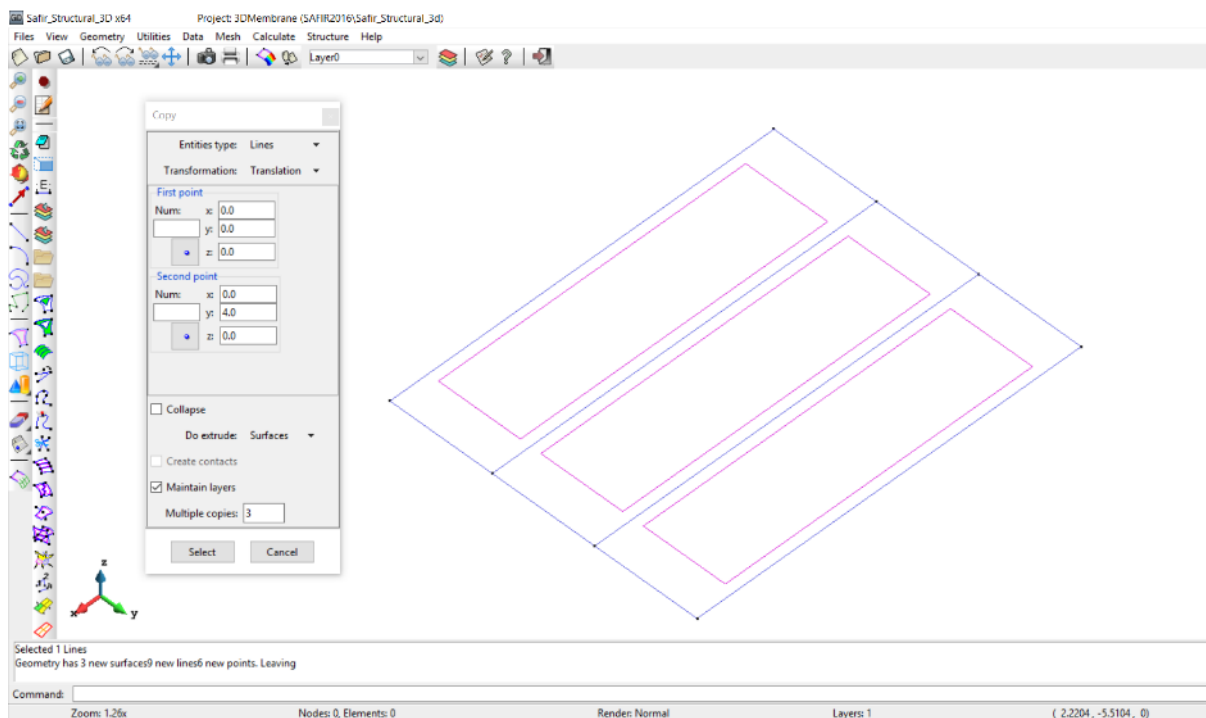

To create the 3 other lines and the corresponding surfaces:
Utilities->Copy   or [Ctrl + c]
And fill as shown below:

Then click on *select* and select the line and press *[Esc]*

To see nodes and beams numbers select:

View->Label->All

4.5. Define constraints for the supports

From the pull down menu select
Data->Constraints
Select *Point Constraints*
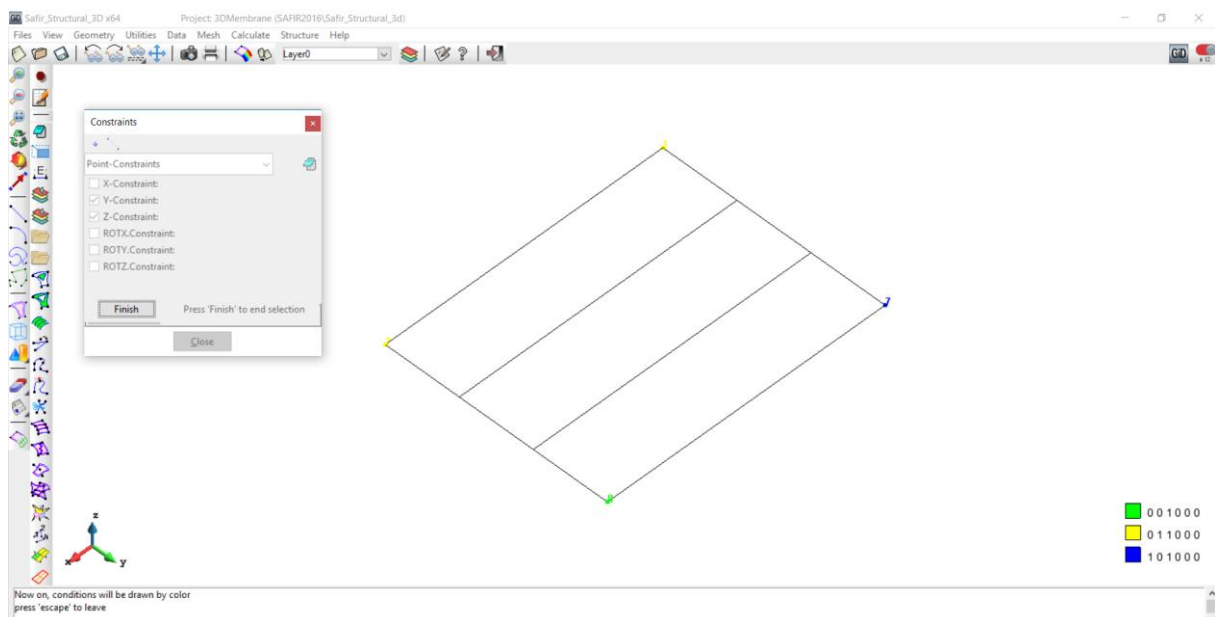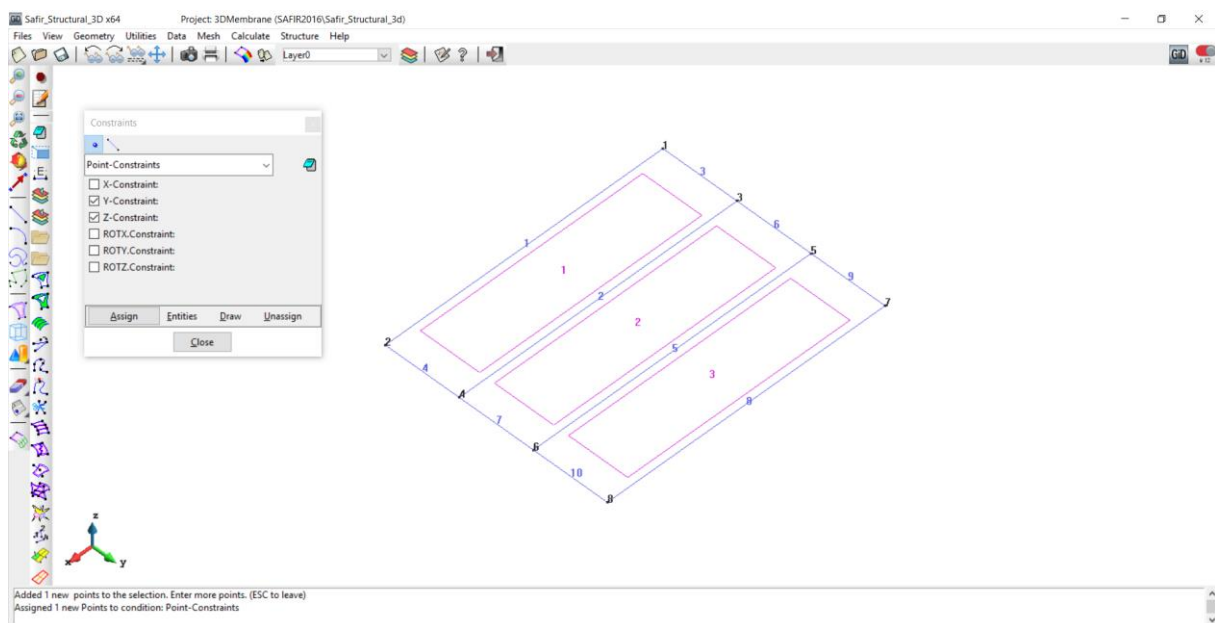Select y and z constraints and assign them to *POINT 1* and press *[Esc]*.
Select y and z constraints and assign them to *POINT 2* and press *[Esc]*.
Select x and z constraints and assign them to *POINT 7* and press *[Esc]*.
Select z constraint and assign them to *POINT 8* and press *[Esc]*.

In the dial box, with *Draw->Colors* you can display the constraints.
Press *Finish* or *[Esc]* to leave this view mode.
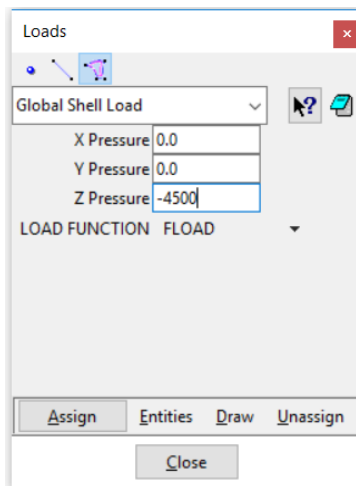
4.6. Assign the loads
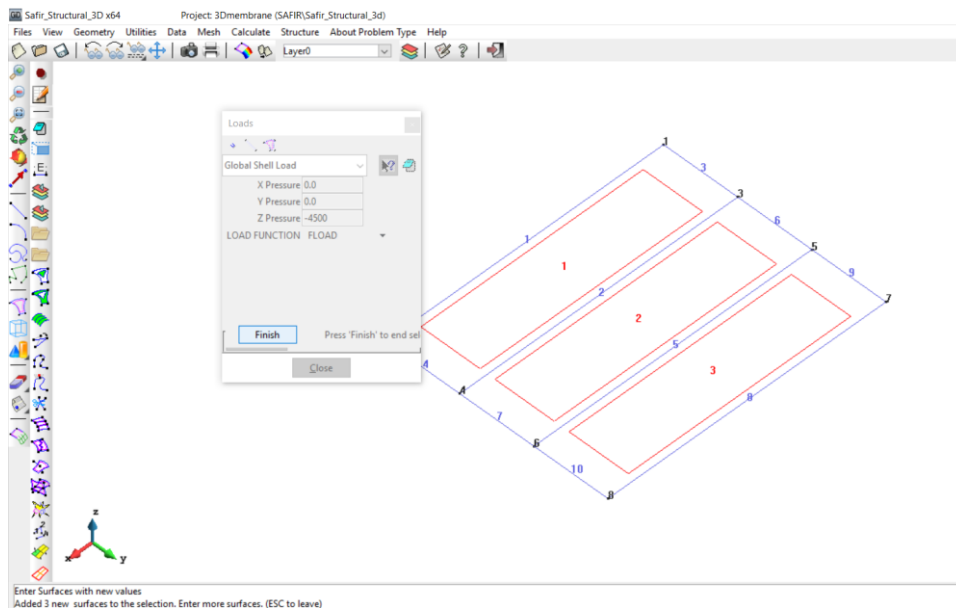
From the pull down menu select
Data->Loads
In the dial box, select *Global Shell Load* and enter a Z-Pressure of -4500 N/m².
Select the load function FLOAD to apply the load in 20 sec.
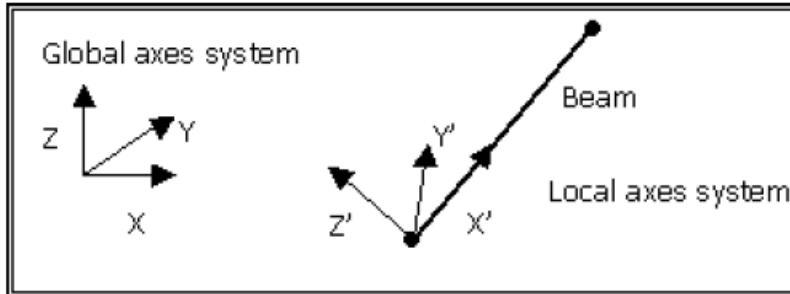Assign the load to all Shells.


Apply to surfaces 1, 2 and 3.

4.7. 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.

The objective is to create 2 local axes (LAX, LAY) as shown in the figure below.
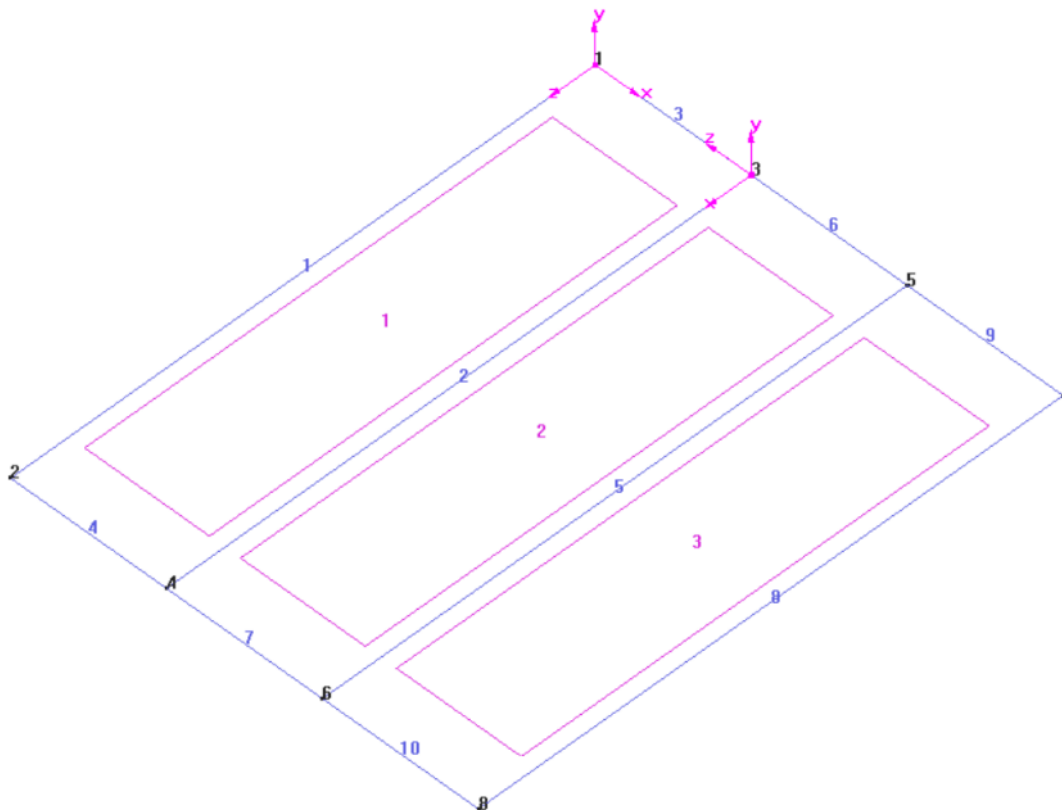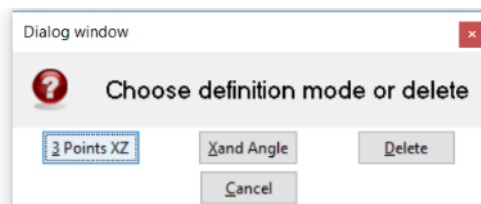
From the pull down menu select:
Data->Local Axes->Define
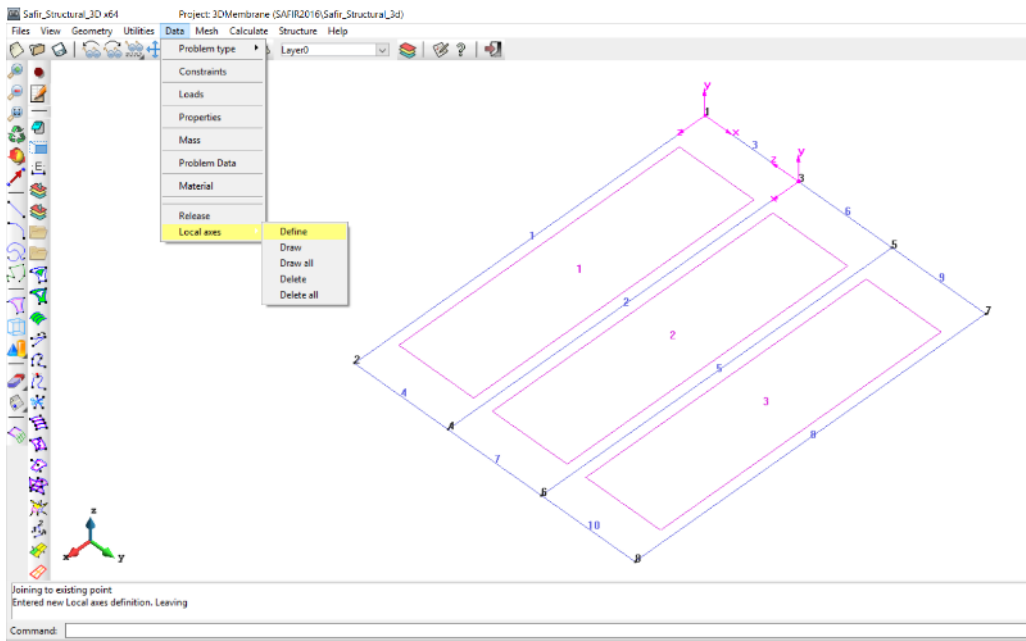
Enter the name of the new local axe *LAX*
Select *3 points XZ*

Select *Point 3* as the local axis center.
Select *Point 4* as the point in positive x axis.
Select *Point 1* as the point in positive z axis.

Repeat the operations to create the local axe *LAY*
Select *Point 1* as the local axis center.
Select *Point 5* as the point in positive x axis.
Select *Point 2* as the point in positive z axis.

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

4.8. Define the global materials

There are 4 materials:
- The steel for the profile IPE550: STEELEC3EN
- The material for the slab modeled in the steel thermal analysis: INSULATION
- The concrete slab in the shell element: SILCOETC2D
- The steel rebars in the shell element: STEELEC2EN
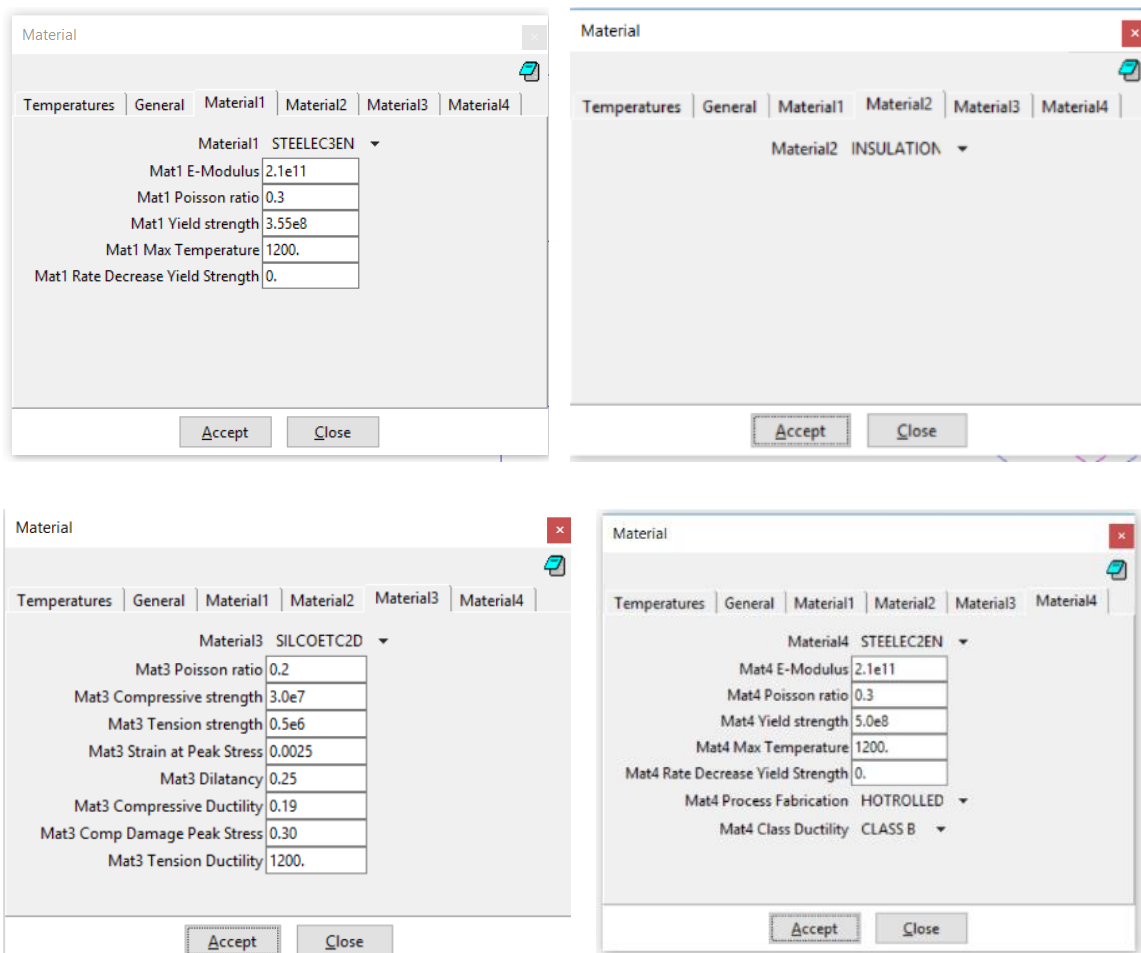
Note: a description of the material *SILCOETC2D* is given in the following paper (for the mechanical behavior). Please cite this paper when using *SILCOETC2D*.

Gernay, T., Millard, A., & Franssen, J. M. (2013). A multiaxial constitutive model for concrete in the fire situation: Theoretical formulation. *International Journal of Solids and Structures*, *50*(22), 3659-3673.

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

The objective is to assign the .tem file named *ipe550* and *ipe550c* to the system lines; as well as the .tsh file named *Slab16.tsh* to the system surfaces.
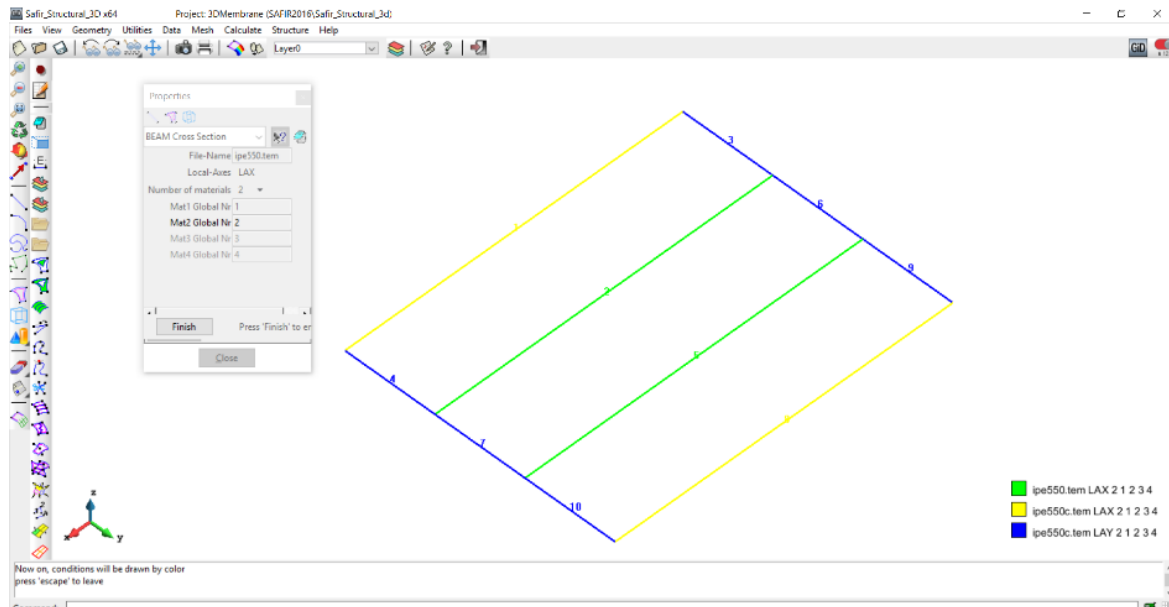

From the pull down menu select:

Data->Properties

In the dial box change the File-Name: *safir.tem* to the temperature file (.TEM file) of the cross-section, in this case *ipe550c.tem*.

Change *Local-Axes* from *-Automatic-* to *LAX*

Change the number of materials to 2. The considered beam section properties have 2 materials: one material for the profile (steel) and one for the slab (which will be replaced by a material with no structural properties, because the slab is modeled apart by the shell element).

Assign the *ipe550c.tem* section to beams in IPE550 who have a local axe *LAX* (in this case beams 1 and 8).
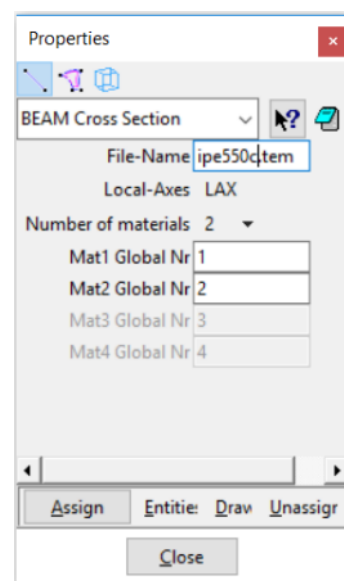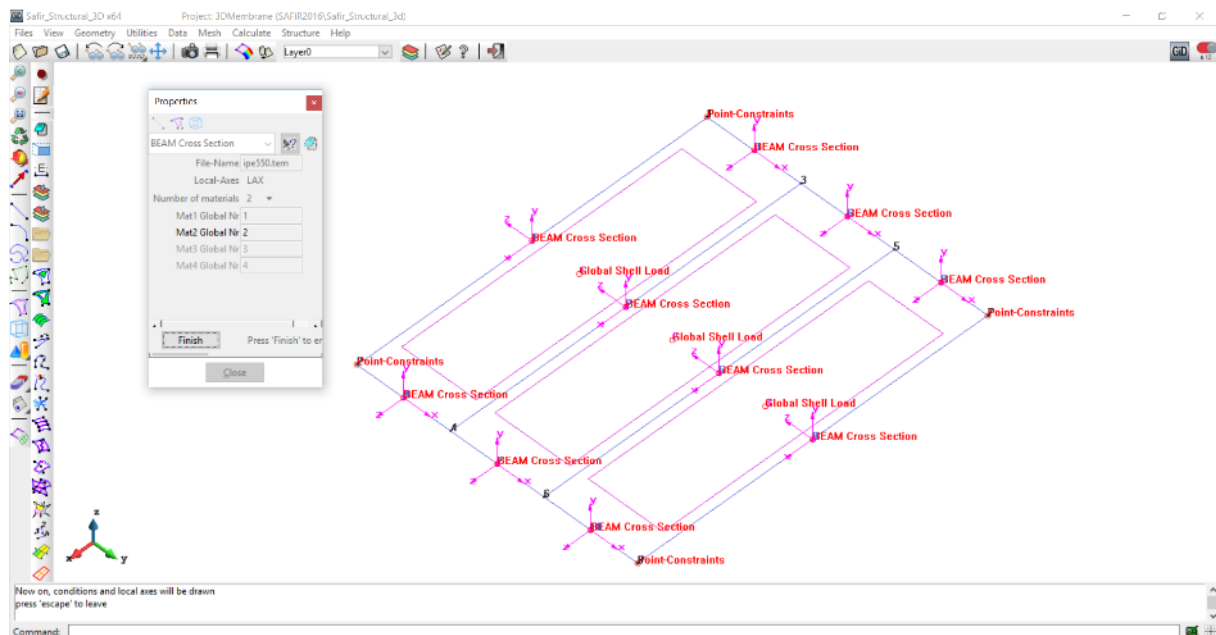
Then, change the local axis to *LAY* and assign the property to beams 3, 4, 6, 7, 9 and 10.
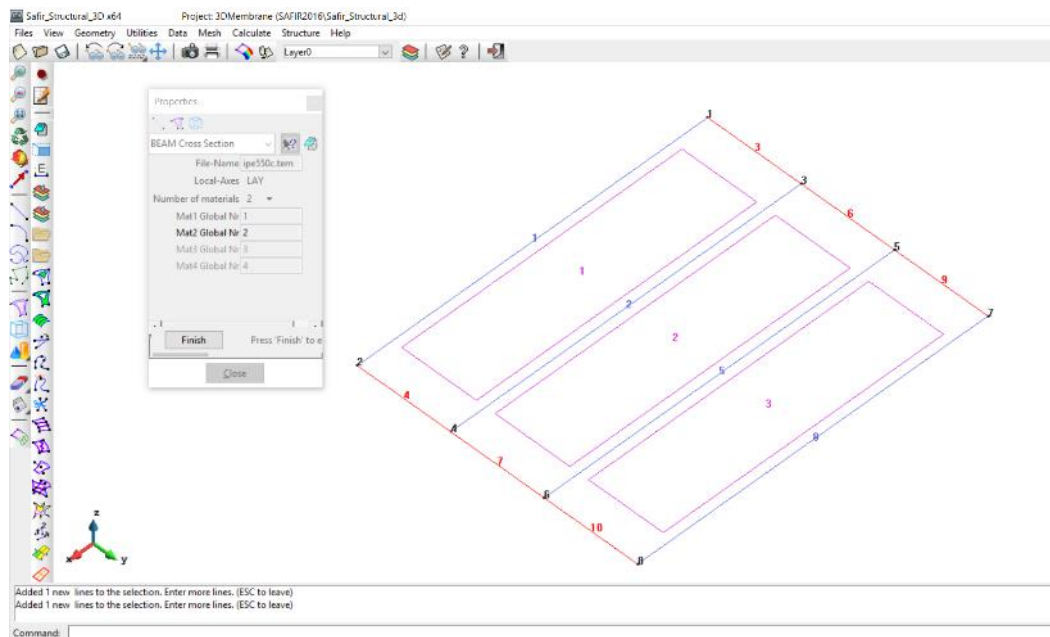
Then, change the file name to *ipe550.tem*. Change *Local-Axes* to *LAX*. Keep 2 materials.
Assign to beams 2 and 5.

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

Finally, select the surface (shell section property). Fill as indicated below.
The shell section property has 2 materials. The materials are concrete from the slab and steel
from the reinforcement bars. These are materials 3 and 4 in the global material system.
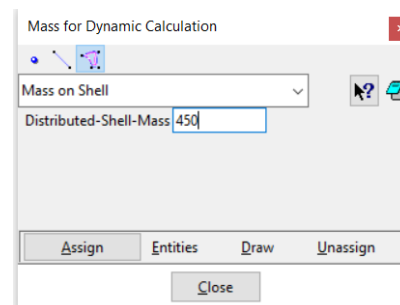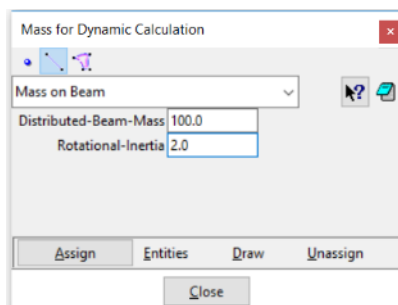
Assign this shell section property to the 3 surfaces.
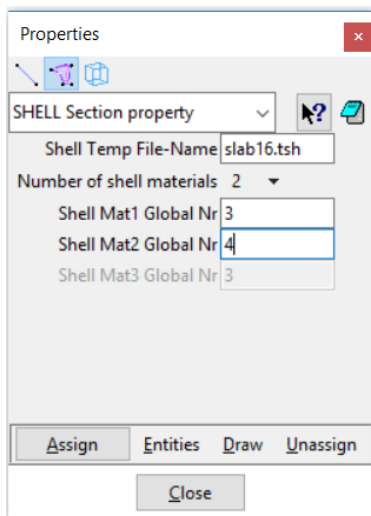
4.10.   Assign the mass

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.
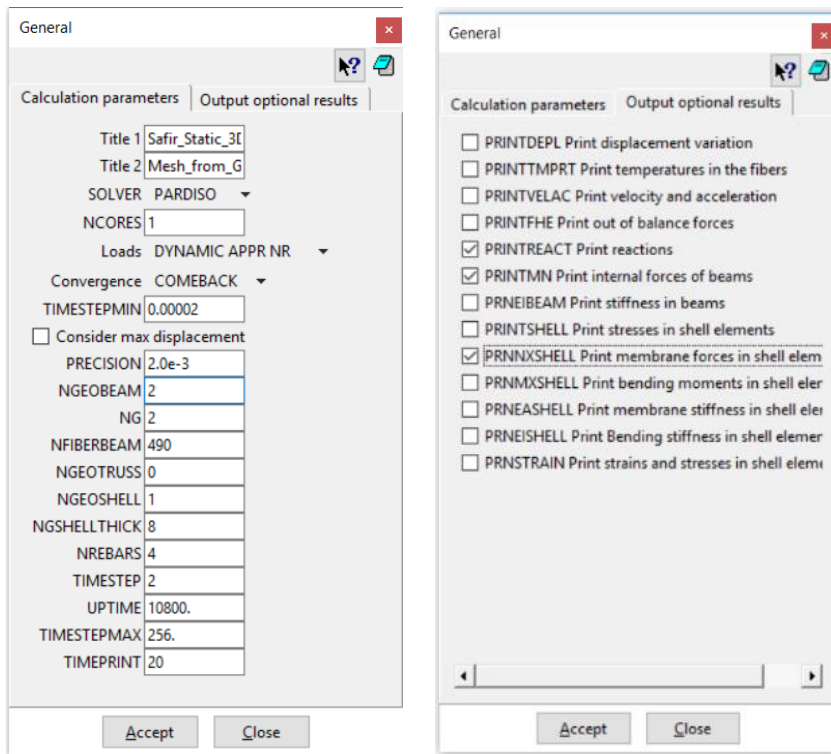Assign to all the beam elements.

Then, select *Mass on Shell* and put 450 kg/m². Assign the mass to all the surfaces.

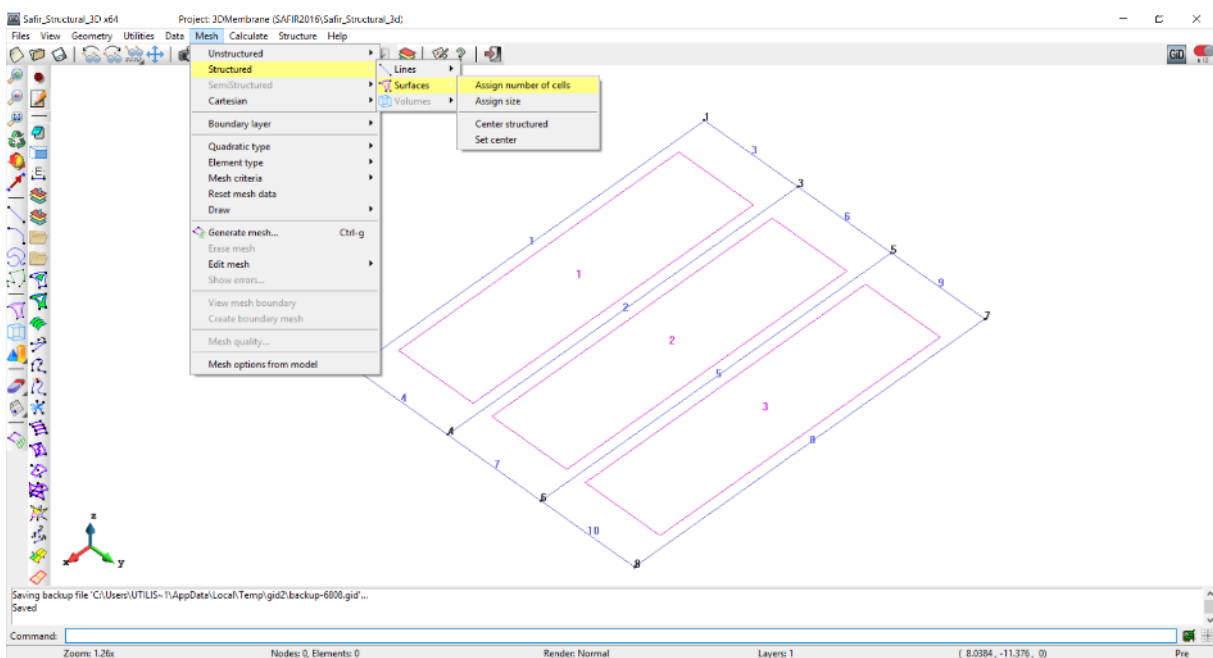## 4.11.   Define the general problem data

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

4.12.   Define the mesh

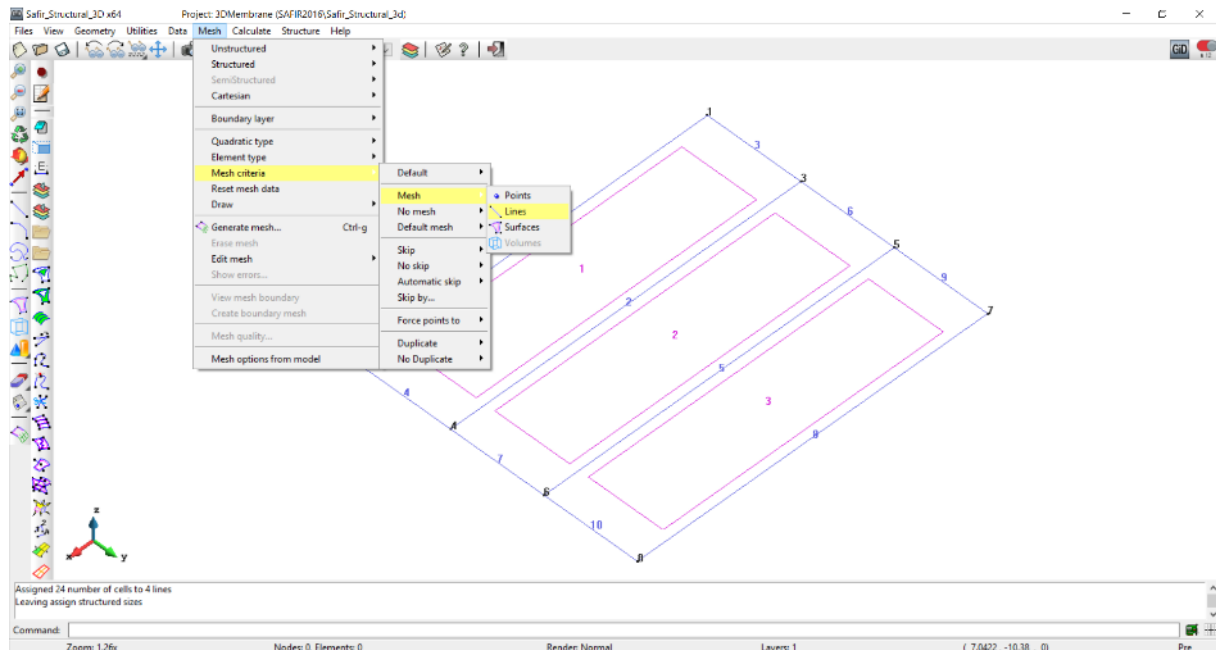Select *Mesh -> Structured -> Surfaces -> Assign number of cells*

Select the 3 surfaces.

Assign 6 elements to lines 3, 6 and 9 (lines 4, 7 and 10 are automatically selected because the mesh is structured).
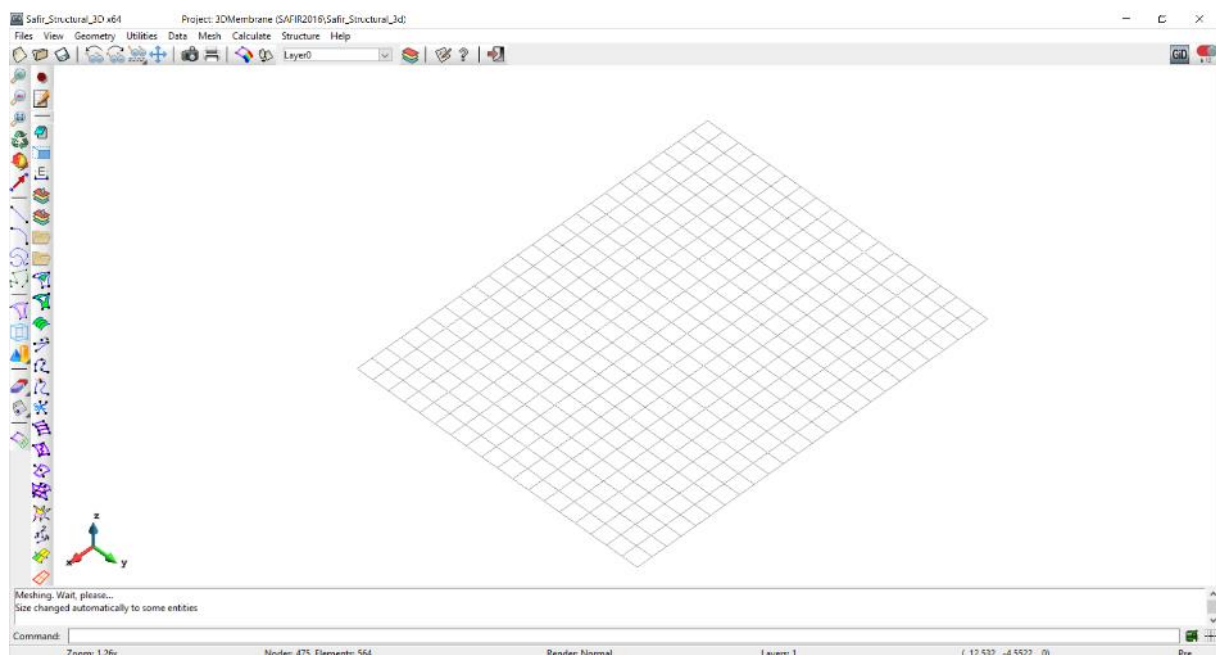Assign 24 elements to line 1 (lines 2, 5 and 8 are automatically selected).

Select *Mesh -> Mesh criteria -> Mesh -> Lines*
Select all the lines. Press *[Esc]* to validate.

Select *Generate Mesh* and then *View Mesh*

4.13.   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*

4.14.   Check the results

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