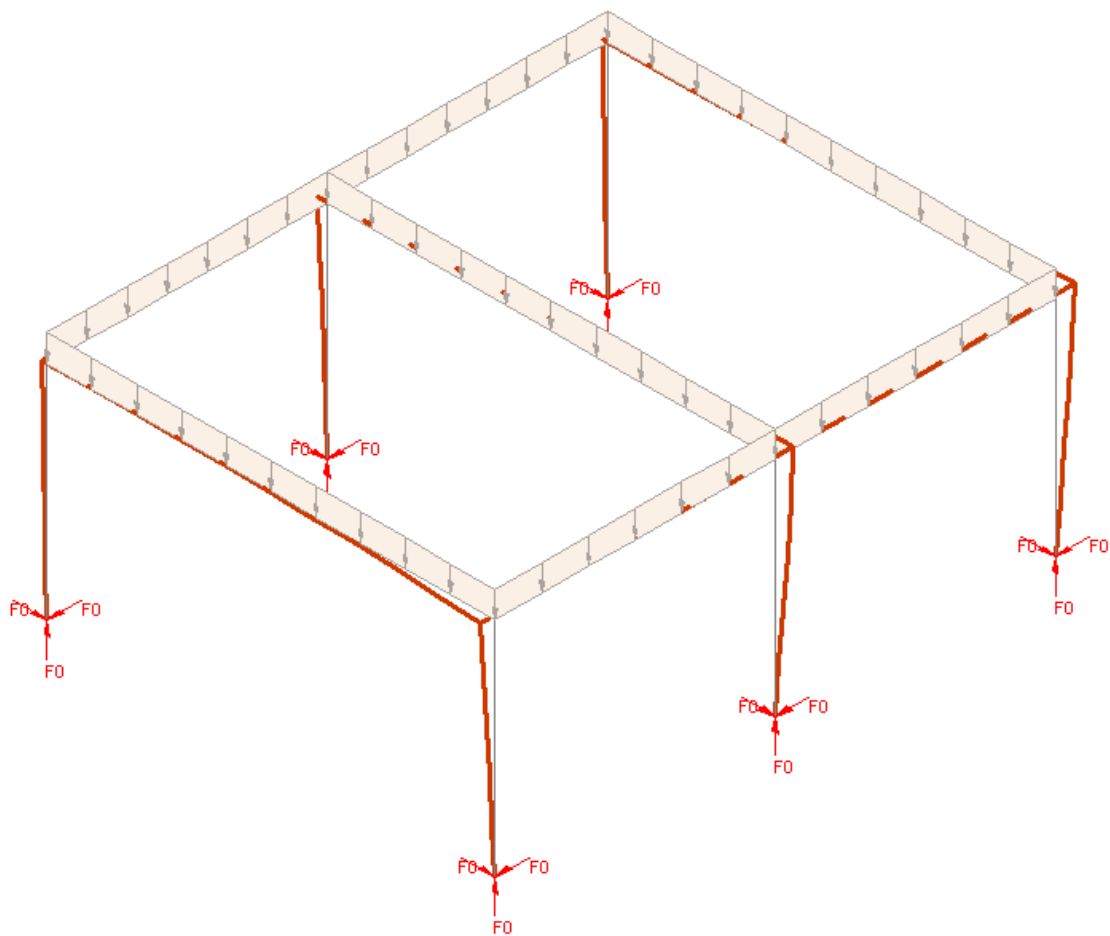


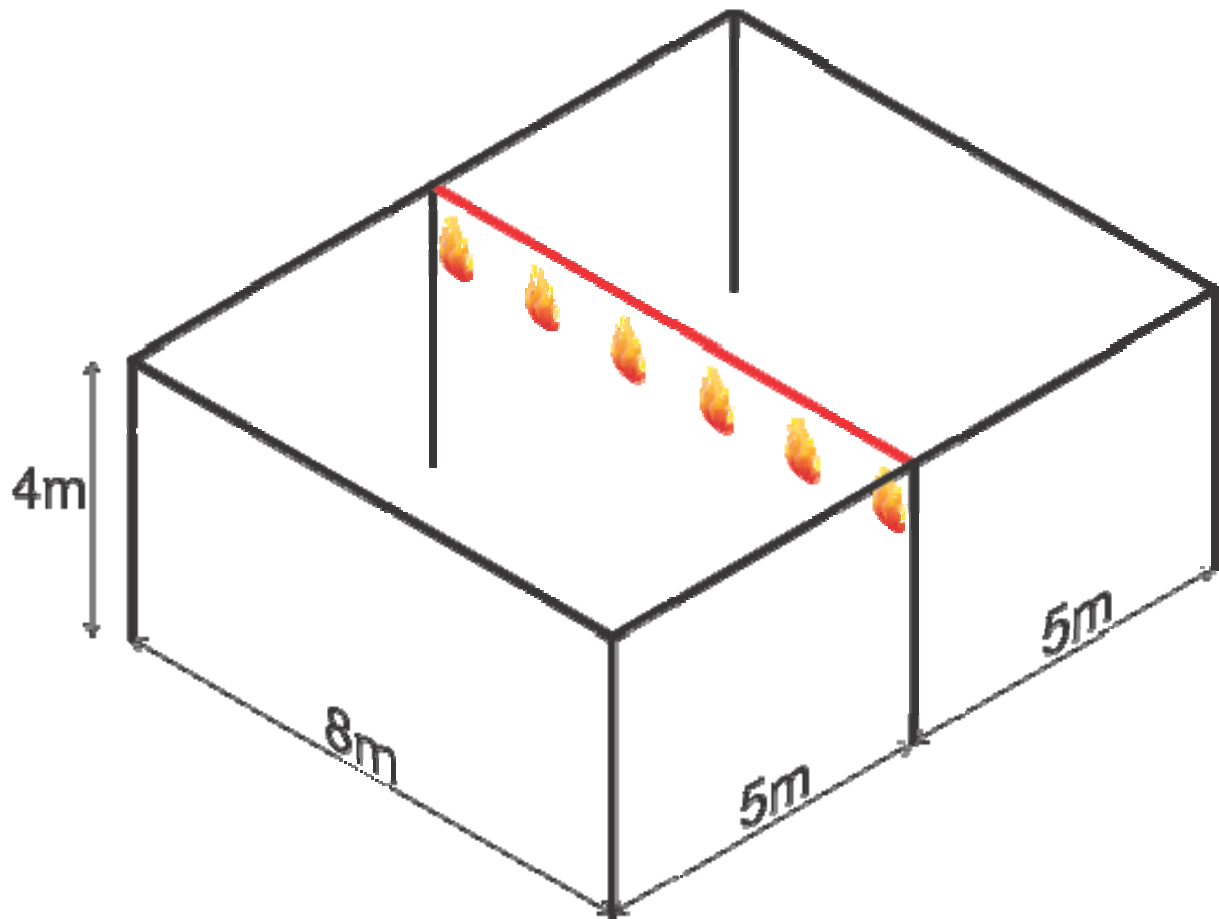
# Example of GID-SAFIR

## Structural Analysis

### Exercise n°8 – 3D Hall



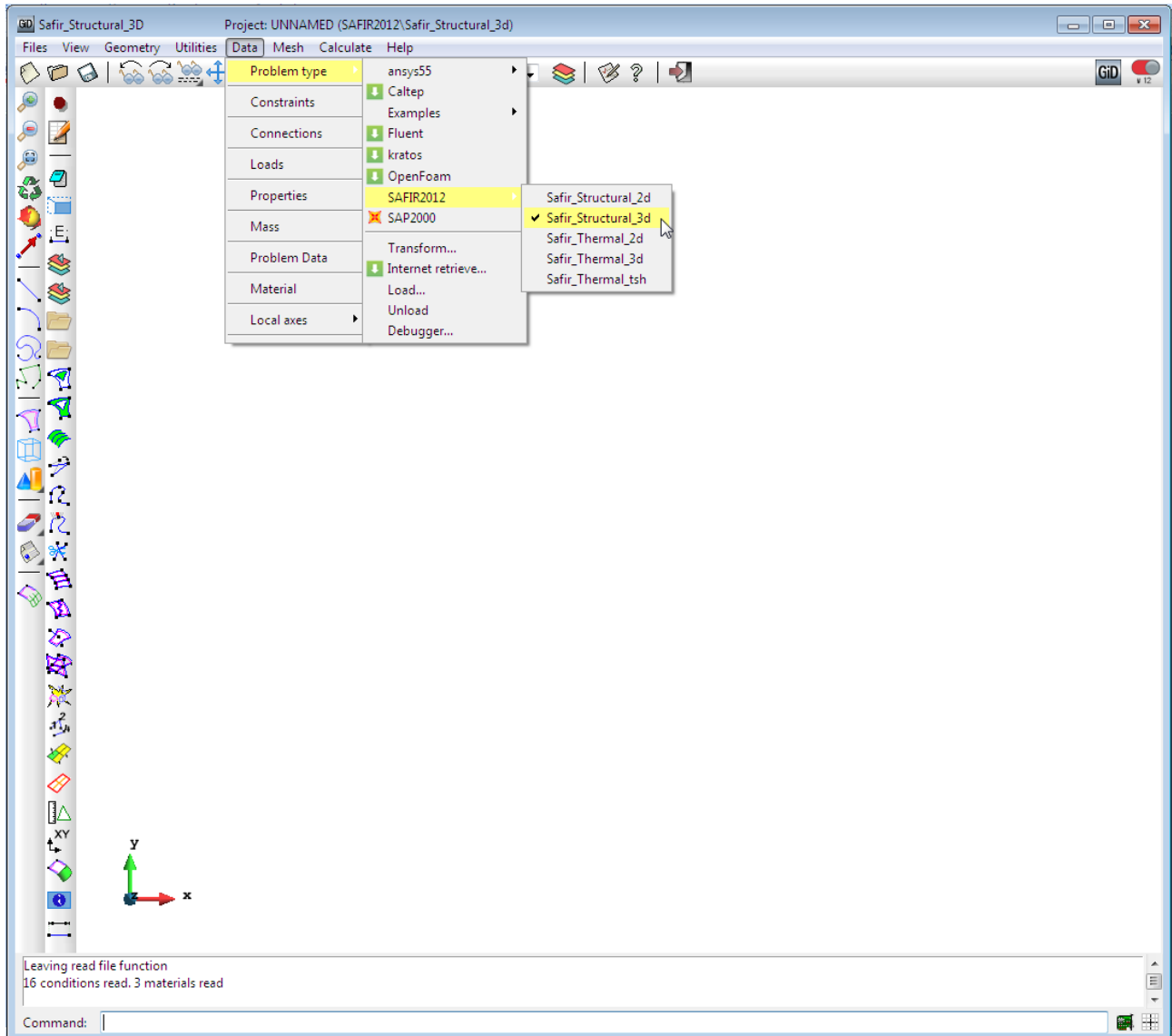
This example is a 3D frame composed of IPE550 beams and HEB220 columns. Only the central IPE550 beam is exposed to ISO fire.



## 1. Create a project for a 3D structural analysis

From the pull down menu select:

➤ **Data->Problem type->SAFIRxxx->Safir\_Structural\_3d**



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

➤ **Files->Save**

or  or [Ctrl + s]

Enter a file name, eg.: **Hall1\_3D**

## 2. Create the system geometry

Change to the 3D isometric view :

select from the pull down menu

➤ **View->Rotate->isometric**

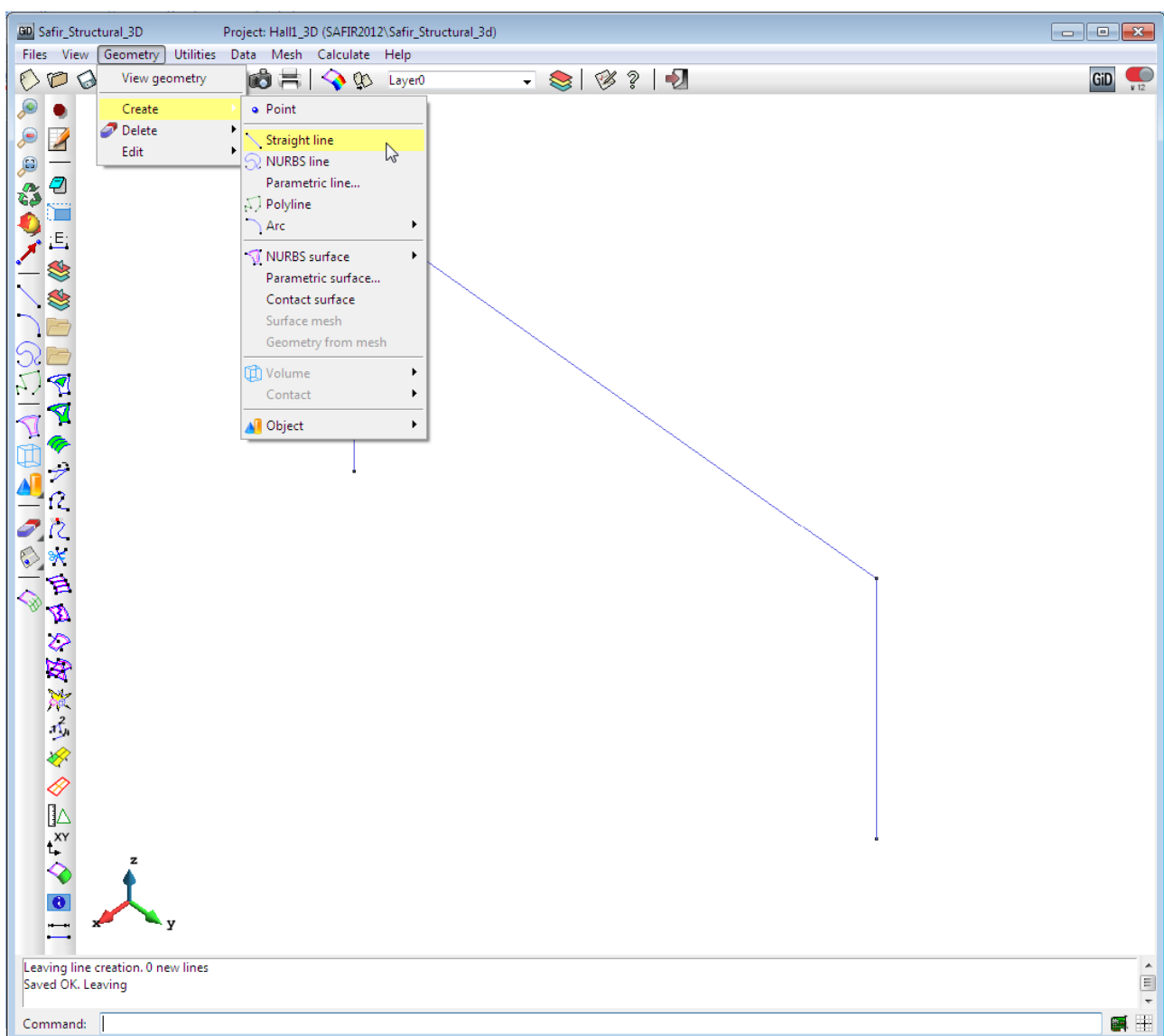
Create the system lines:

➤ **Geometry->Create->Straight Line**



In the command line (at the bottom of the window) enter the coordinates in [m] of the line points (with a whitespace between each coordinates):

**0,0,0 0,0,4 @0,8,0 @0,0,-4** and press **[Enter]**, then twice **Esc** to quite this line mode.



The first frame is created.

To create the 2 other frames Select:

► **Utilities->Copy**

or [Ctrl + c]

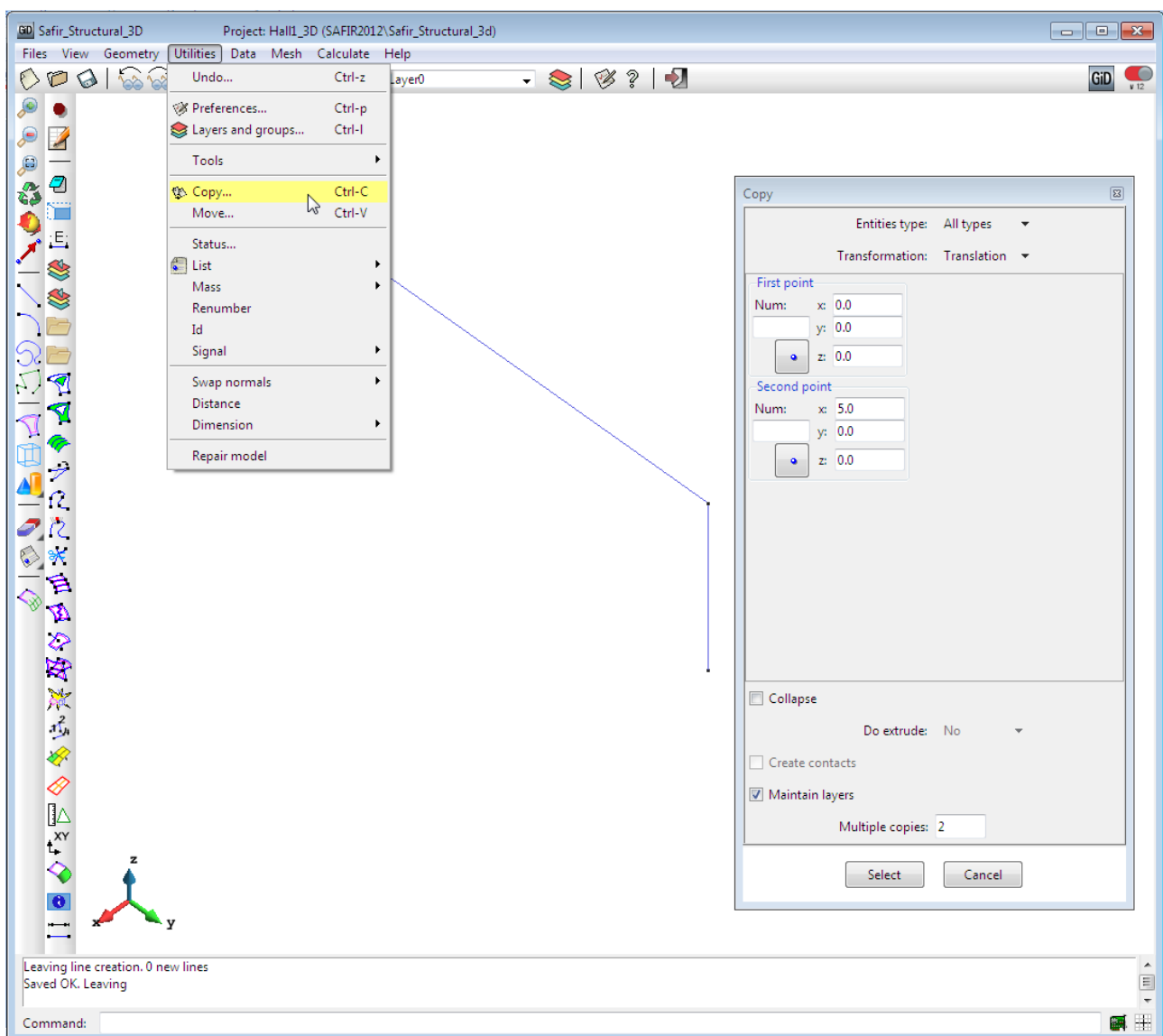
In the **Copy** dialog box, fill the data as follows :

For **Entities type**, use : **All types**

For **Second point**, type : **x = 5.0**

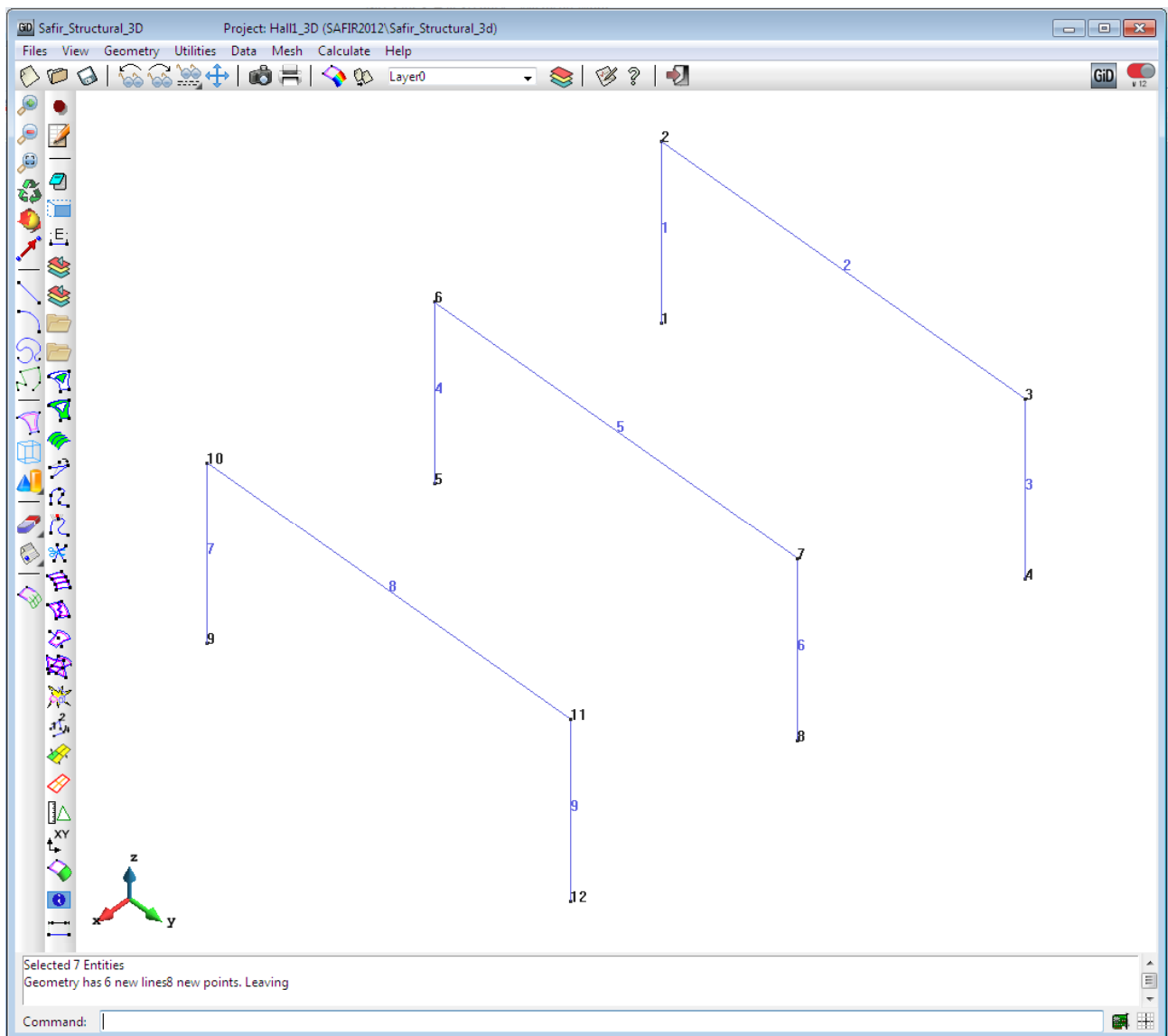
For **Multiple copies**, type : **2**

Select all the frame and press **Finish** , then **Cancel**.



To see nodes and beams numbers, select:

➤ **View->Label->All**



Then connect the top of the columns in x direction.

Select:

➤ **Geometry->Create-> Straight Line**

Press **[Ctrl-a]** and pick points 2, 6 and 10 and press **Esc**, then pick points 3, 7 and 11 and press **Esc** twice to quit the line command.

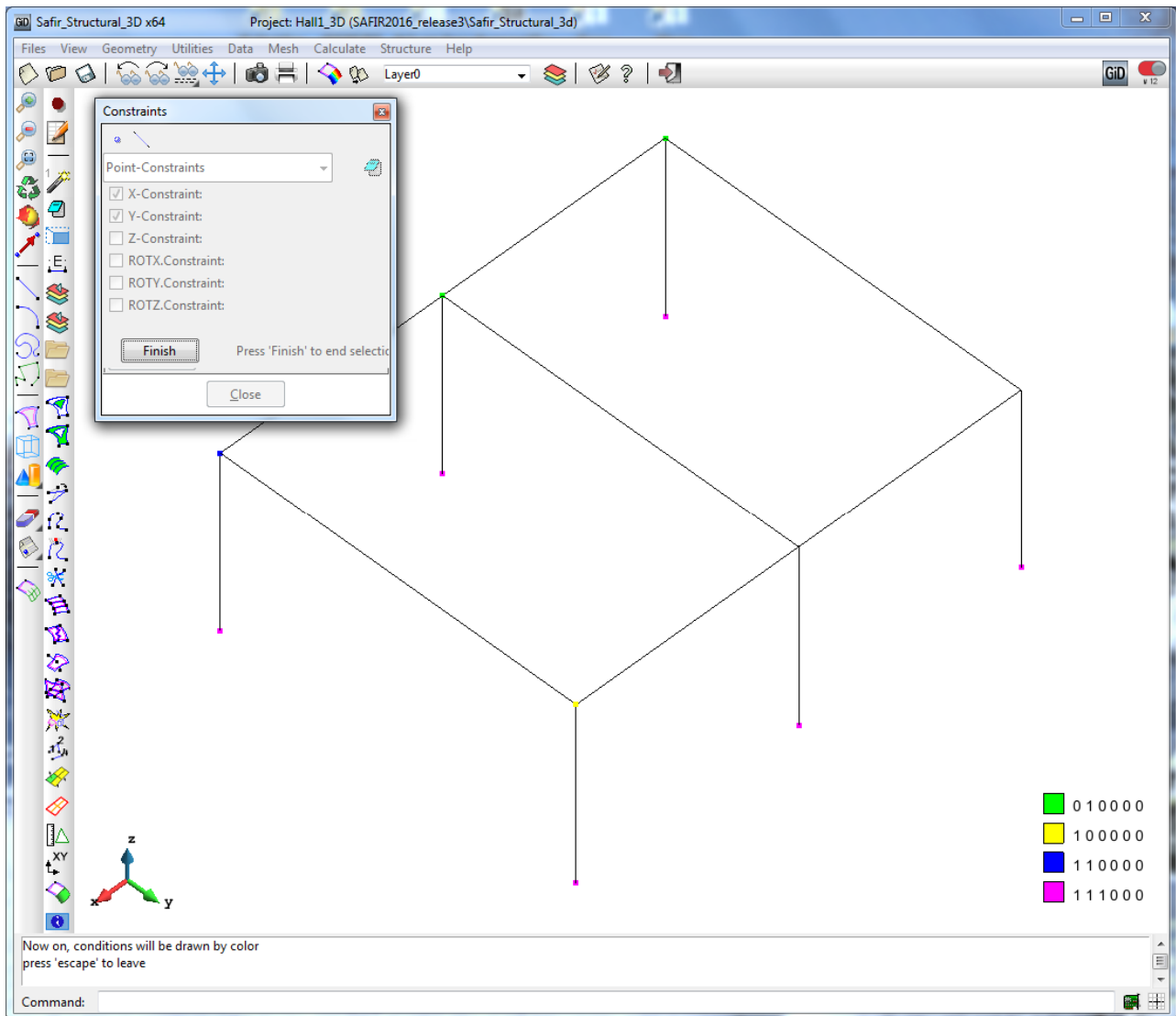
### 3. Define constraints for the supports

From the pull down menu select

➤ **Data->Constraints**

Select X,Y and Z constraint and assign them to the base points of all columns.

In the *Constraints* dialog box, with **Draw->Colors** you can display the constraints.

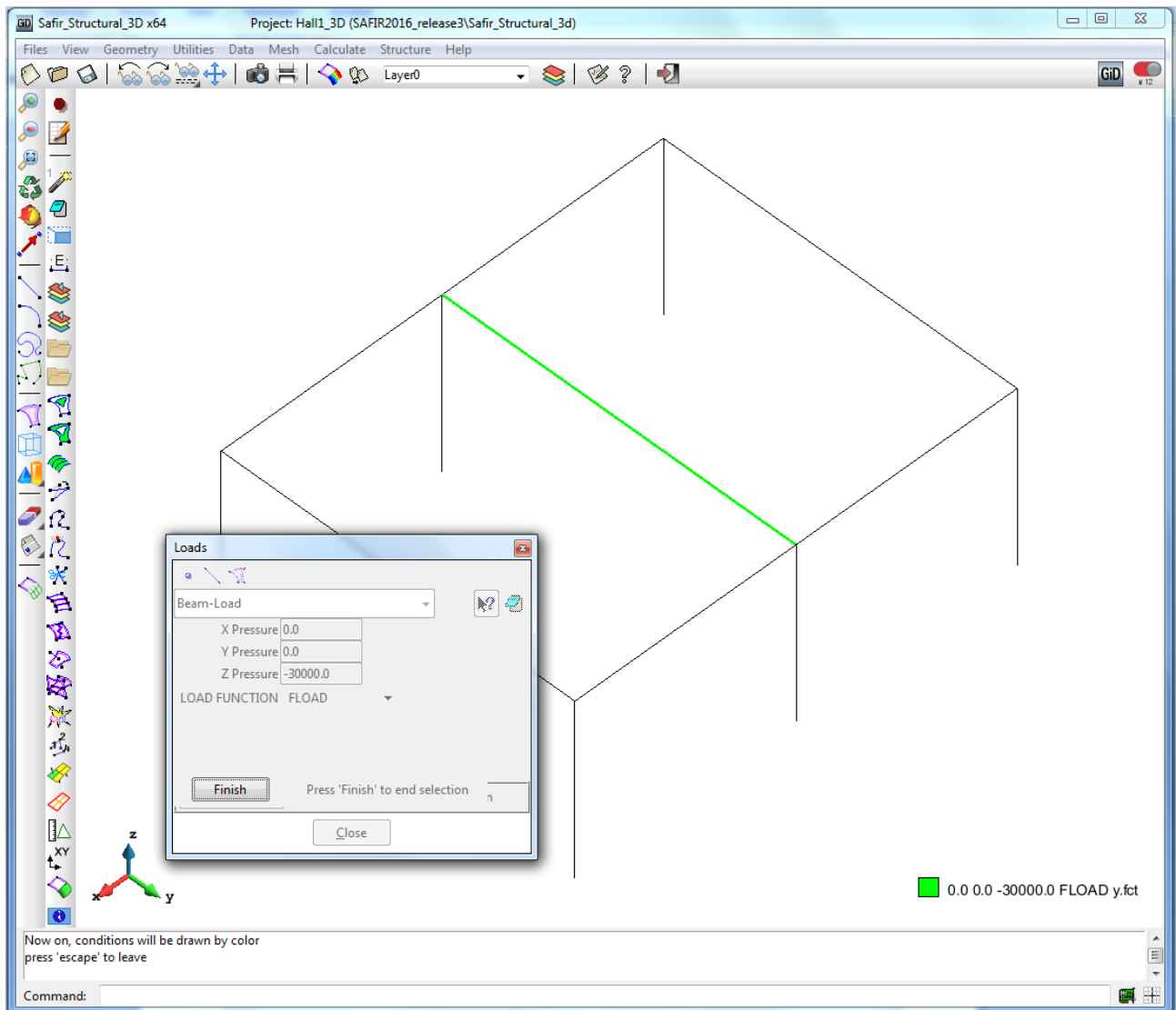



Press **Finish**, then **Close** to leave this view mode.

## 4. Define loads

From the pull down menu select:

➤ **Data->Loads**



In the *Loads* dialog box, select **Beam-Load** tab , put a Z-Pressure of **-30000N/m**, select **FLOAD** in the **LOAD FUNCTION** dialog box and assign to the central beam.

To display the load, select **Draw->Colors** in the *Loads* dialog box.

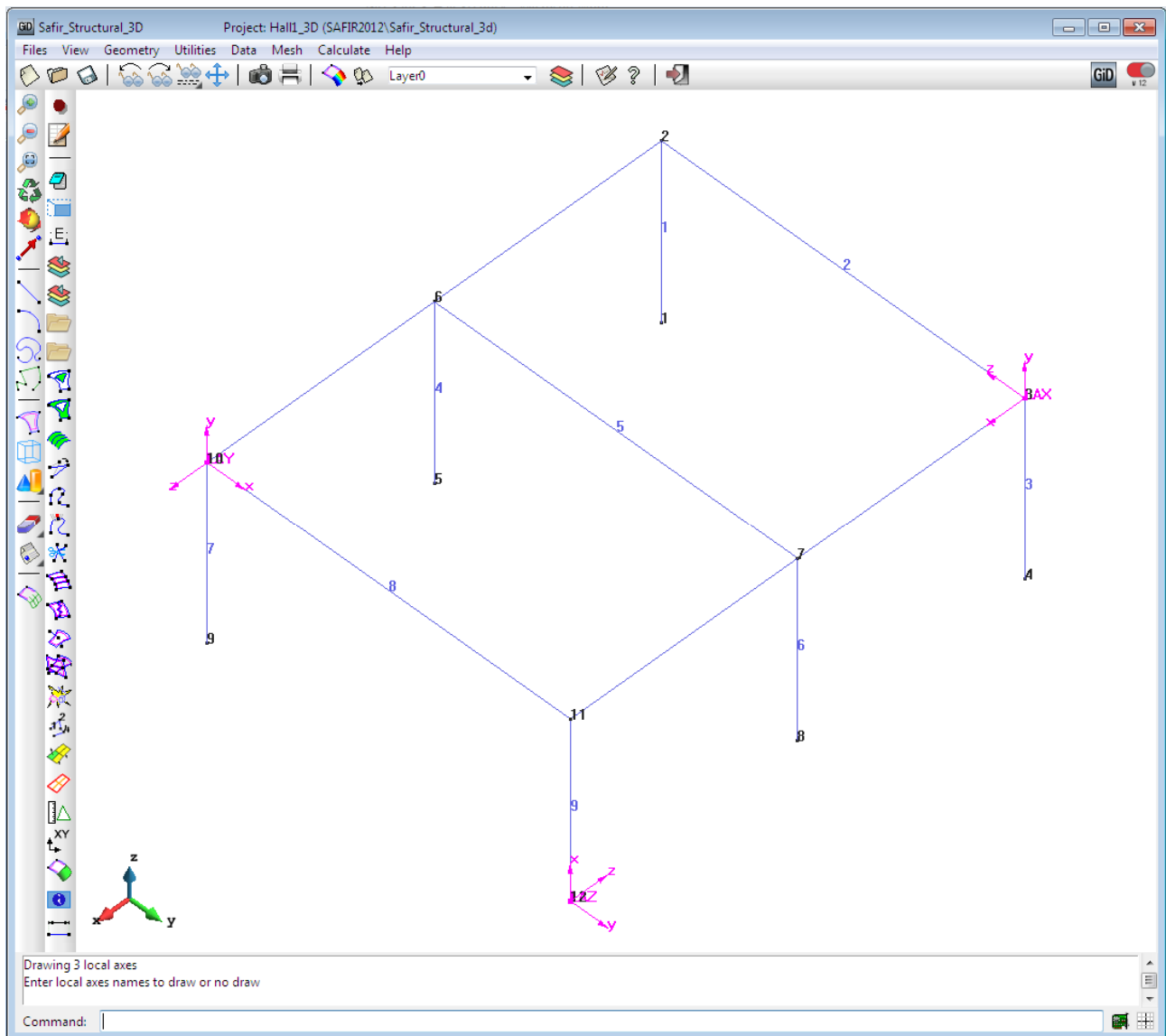
Press **Finish**, then **Close** to leave this view mode.



## 5. Create Local axes

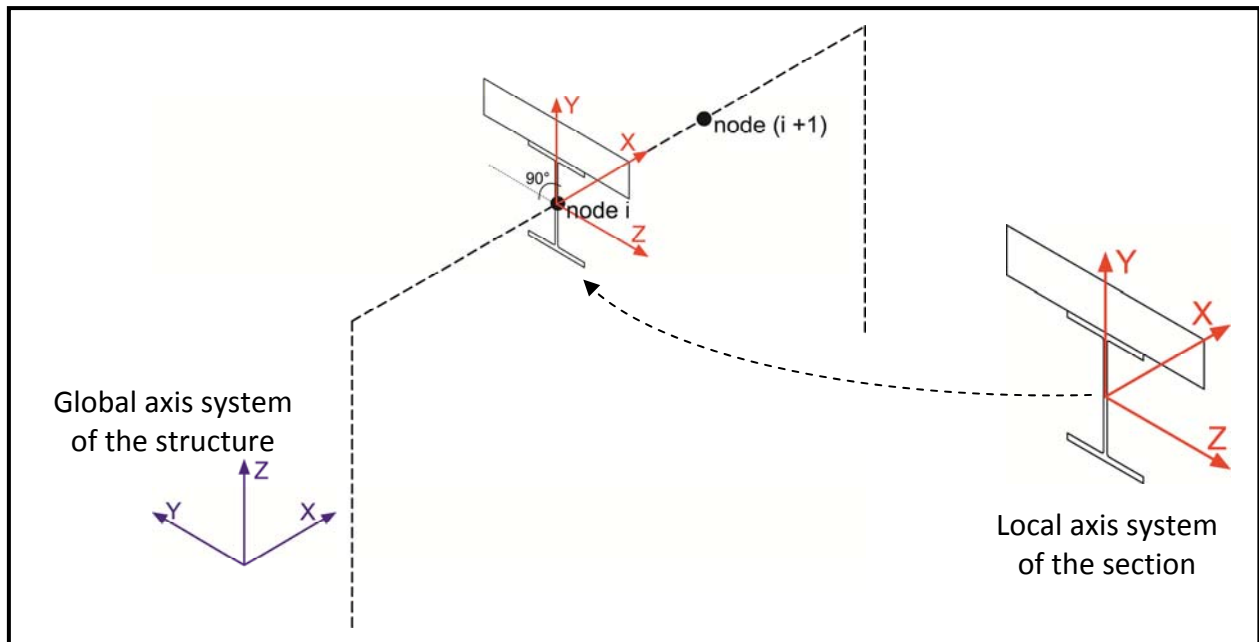
The orientation of the cross-section is controlled by defining his local axis in the global XYZ plan of the structure. The objective is to create a local axis for each different section orientations.

Create 3 local axis (LAX, LAY, LAZ) as shown in the figure below.



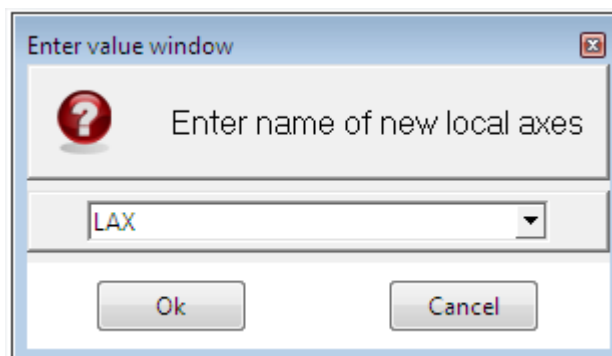
The most used procedure to create a local axis is

- to fix the X direction by given 2 existing nodes of the structure (node i and node i+1)
- to fix the Y direction by given an angle ( $90^\circ$ )



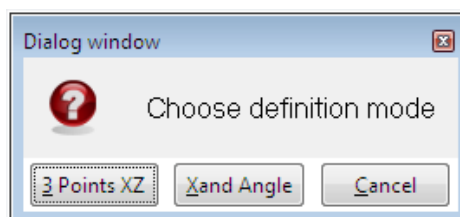
To define the local axis LAX, from the pull down menu select:

➤ *Data->Local Axes->Define*



Enter the local axis name **LAX** and click **OK**.

Then choose the definition mode *X and Angle*.

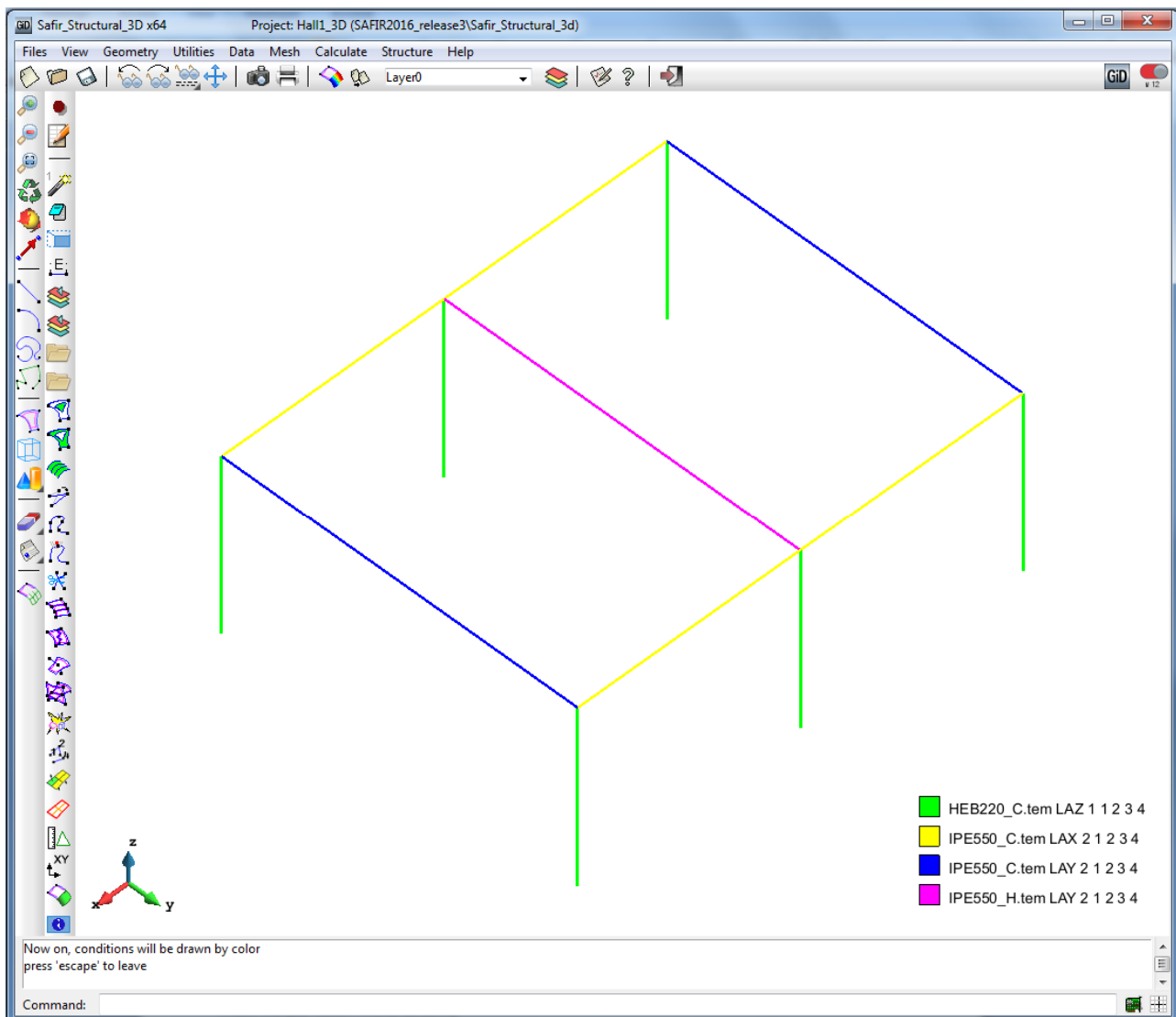


Press **[Ctrl+a]** to select the center of the local axis LAX (left point of the beam). Then press **[Ctrl+a]** to select a point in the positive orientation of the local axis LAX (right point of the beam). Now enter an angle (in this example put 90°) to get the orientation of the Y axis of the local axis LAX, and therefore the orientation of the section.

Use the same procedure to create the local axis LAY (with an angle of 90°) and LAZ (with an angle of 90°).

## 6. Assign the sections (.TEM files)

The objective is to assign the .tem files as shown in the following figure:



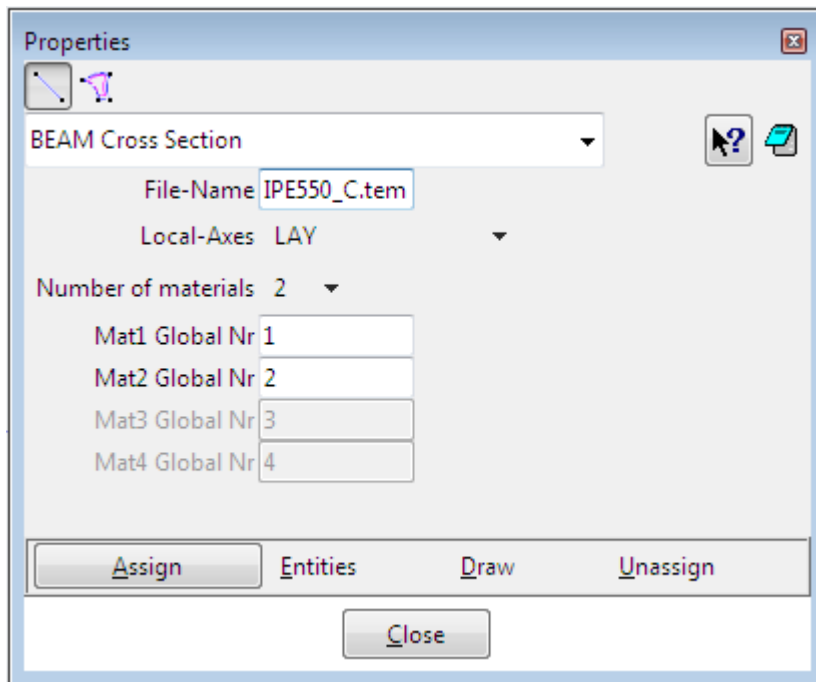
From the pull down menu select:

➤ **Data-> Properties**

In the File-Name dialog box, give your section file name **IPE550\_C.tem**.

As the section is steel and concrete materials, select **2** for the Number of materials.

Change **Local-Axes** from **-Automatic-** to **LAY**.



- ⚠ **Number of materials**     The number of material should be the same as defined for the thermal analysis of the considered section (.tem file).
- Mat $x$  Global Nr  $y$**      This is the rank  $y$  in the "structure.in" file (Global Nr  $y$ ) of the Material  $x$  (Mat $x$ ) defined in the .tem file.

Click on **Assign** and select the corresponding beam element line.

Click on **Finish** , then **Close**.

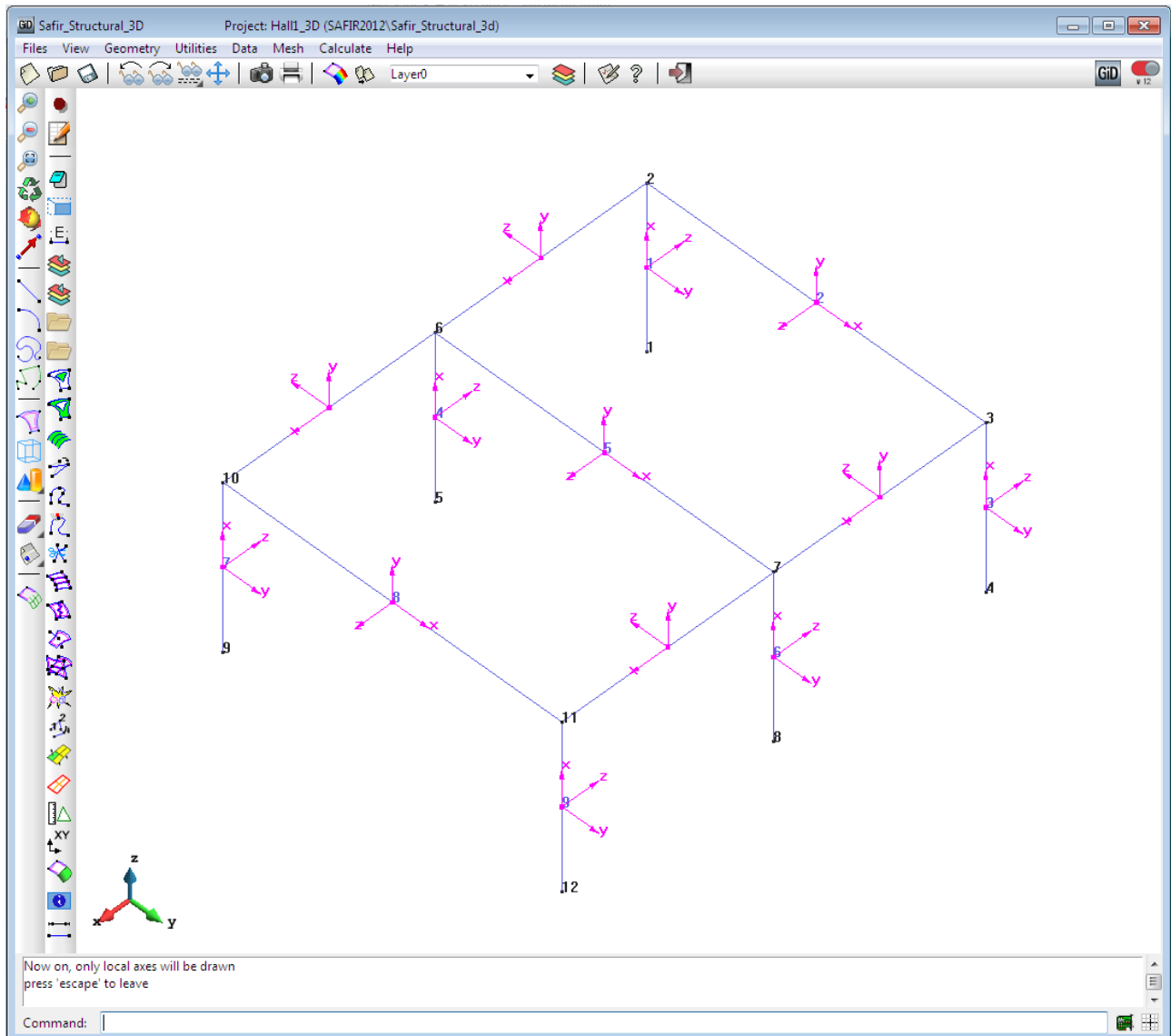
Do the same procedure to assign the following sections :

- IPE550\_H.tem with local axis LAY
  - IPE550\_C.tem with local axis LAX
  - HEB220\_H.tem with local axis LAZ
  - HEB220\_C.tem with local axis LAZ
- } ⚠ For these 2 sections, Number of materials = 1

To display the local axis, select in the *Properties* dialog box:

**Draw->All conditions->Only local axes**

Press **Finish** then **Close** to leave this view mode.

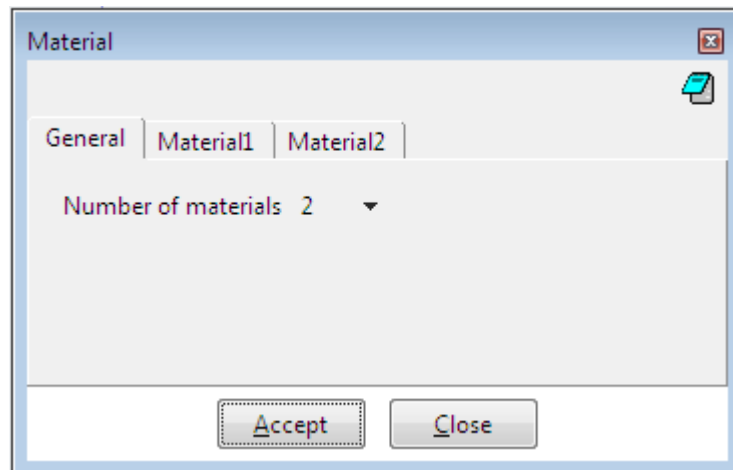


## 7. Define global materials

To define material select from the pull down menu:

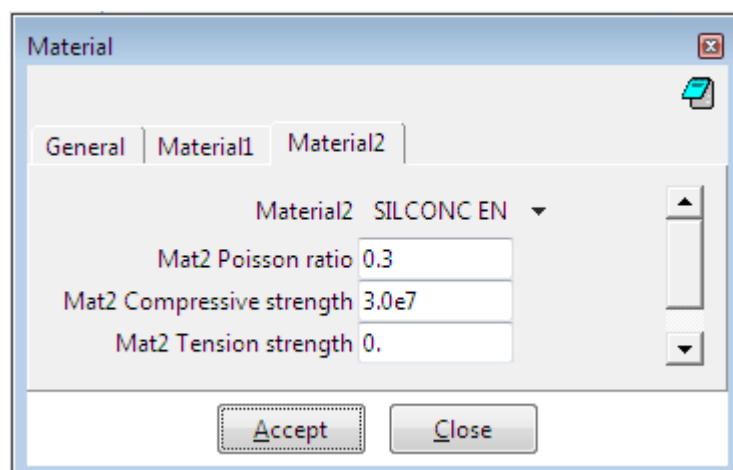
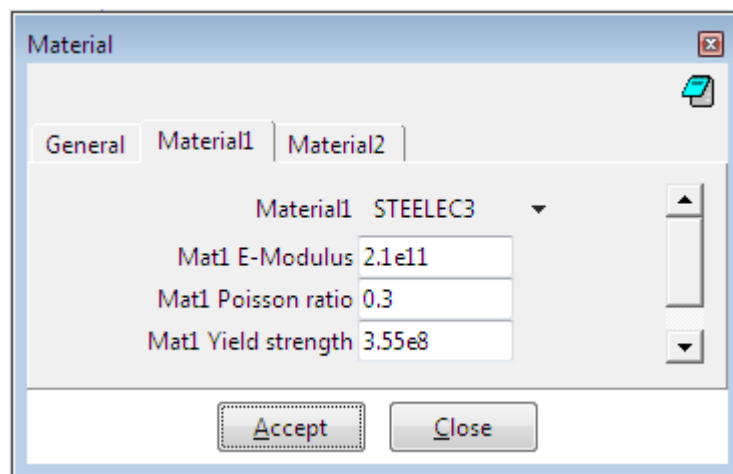
➤ *Data->Material*

In the *General* tab, put 2 materials.



Then fill the *Material1* and *Material2* tabs as shown below :

Click on **Accept** to confirm.

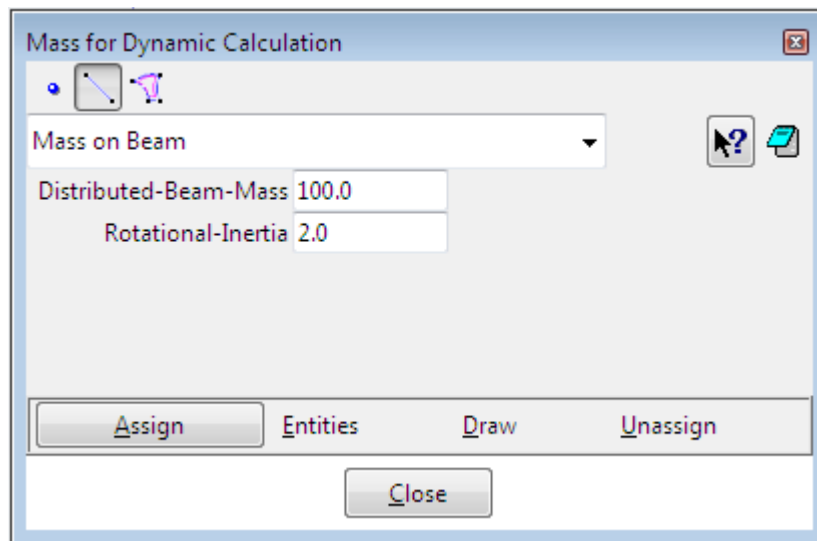


## 8. Define the Mass

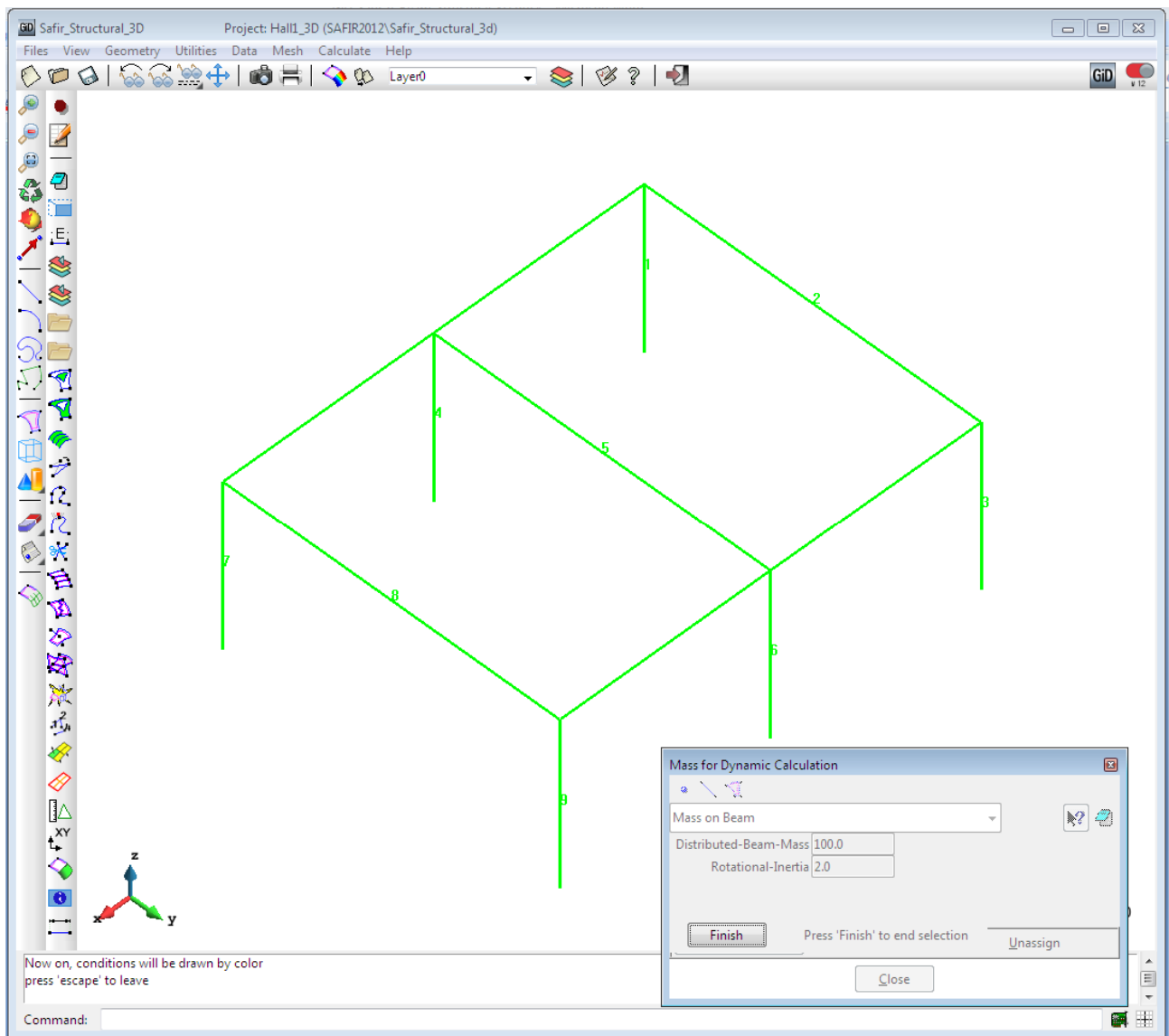
For the dynamic calculation, you need to give a mass to the beam elements. Select from the pull down menu:

➤ *Data-> Mass*

To put Mass on Beam, select the  tab and put **100kg/m** as Distributed-Beam-Mass and **2** as Rotational-Inertia.



Assign the mass to all the beam elements, and click on *Finish* then *Close*.



To display Property select in the dialog box:

➤ **Draw->Colors**

Press on **Finish** then **Close** to leave this view mode.



## 9. Define general data

Select from the pull down menu:

➤ **Data->Problem Data**

Fill the dialog box as below :

**General**

Calculation parameters | Output optional results

Title 1 Safir\_Static\_3I  
Title 2 Mesh\_from\_G

SOLVER PARDISO ▾  
NCORES 1

Loads DYNAMIC APPR NR ▾

Convergence COMEBACK ▾

TIMESTEPMIN 1.0e-5

☐ Consider max displacement

PRECISION 1.0e-3

NGEOBEAM 3

NG 2

NFIBERBEAM 550

NGEOTRUSS 0

NGEOSHELL 0

NGSHELLTHICK 0

NREBARS 0

TIMESTEP 12

UPTIME 3600

TIMESTEPMAX 32

TIMEPRINT 60

Accept Close

⚠ **NGEOBEAM** is the number of different sections which constitute the structure (.tem files).

**NFIBERBEAM** is the number of fibers (meshes) in the section. This value is given in the first line of the .TEM file (in case of several .TEM files, give the maximum value).

Change **TIMESTEPMIN**, **PRECISION**, **TIMESTEP**, **UPTIME**, **TIMESTEPMAX** and **TIMEPRINT** as needed.

Click on **Accept** to save your change, then **Close**.

## 10. Generate the mesh

To create meshes select from the pull down menu:

➤ **Mesh->Generate mesh**

or use [Ctrl + g]

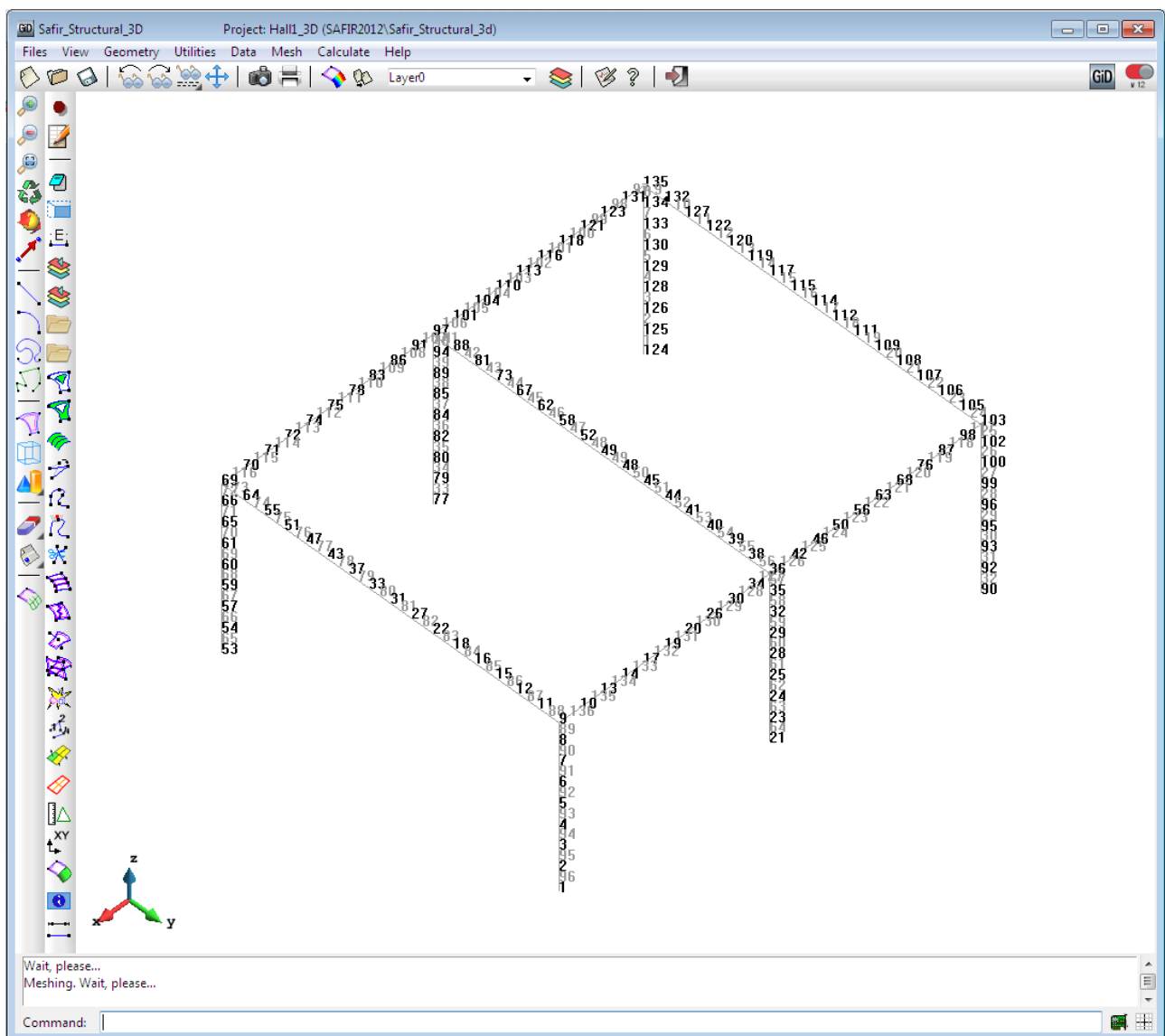
Enter **0.5m** as size of elements to be generated.

Select from the pull down menu:

➤ **Mesh->Generate**

or [Ctrl + g]

Enter the element size of 0.5 m



## 11. Create sections

Before starting the calculation, don't forget to copy all the .tem files in the directory "Hall1\_3D.gid".

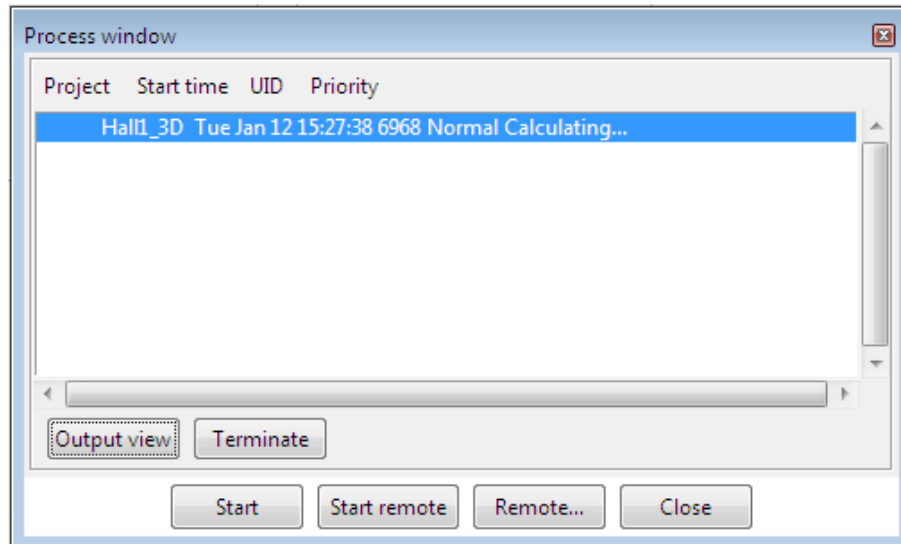
The sections are distributed as follows :

- IPE550\_H.tem : this section has been created in the Exercise 3 – Part 2, but is exposed to a user-defined fire. Calculate this section with ISO fire, copy the file .tem and rename it to IPE550\_H.tem.
- IPE550\_C.tem : this section is not heated. Copy the file IPE550\_3D.tem and rename it to IPE550\_C.tem and replace the command "HOT" by "COLD".
- HEB220\_C.tem : this section is not heated. Copy the file HEB220\_3D.tem and rename it to HEB220\_C.tem and replace the command "HOT" by "COLD".

## 12. Start the calculation

To start the calculation, select from the pull down menu:

➤ *Calculate->Calculate window*



Click on the **Start** button then on the Output view button.

