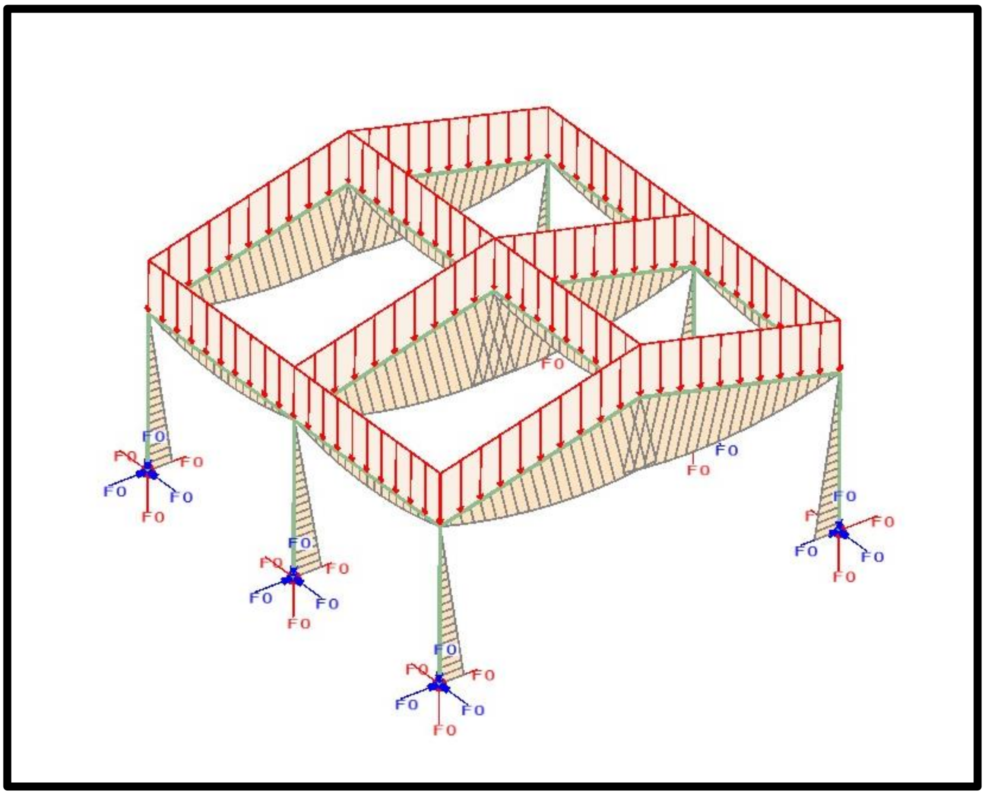
SAFIR® training session – level 1
Johns Hopkins University, Baltimore

Example: 3D structural model of a building frame with pinned connections

“3D building frame with concrete columns and steel beams”

T. Gernay & J.M. Franssen



1. General description

This example deals with a 3D frame structure of 8 m by 8 m by 4 m. The columns are reinforced concrete section of 30 cm x 30 cm, with 4 φ20 steel bars with an axis cover of 30mm, heated on 3 faces. The beams are IPE 300 steel profiles, heated on 3 faces. The columns are fixed, while the beams are pinned. The beams are subjected to uniformly distributed loads of 10 kN/m.

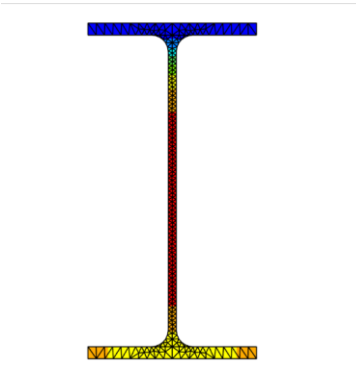
Steel for IPE300: yield strength 355 MPa

Steel for reinforcement bars: yield strength 500 MPa

Concrete: compressive strength 30 MPa; tensile strength 1 MPa

2. Section for IPE300 steel beam

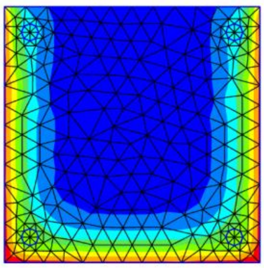
The model created in a previous exercise will be used here.



The .tem file of this model (*’ipe300.tem’*) will need to be copy-pasted in the folder with the structural model.

3. Section for concrete column

The model created in a previous exercise will be used here.

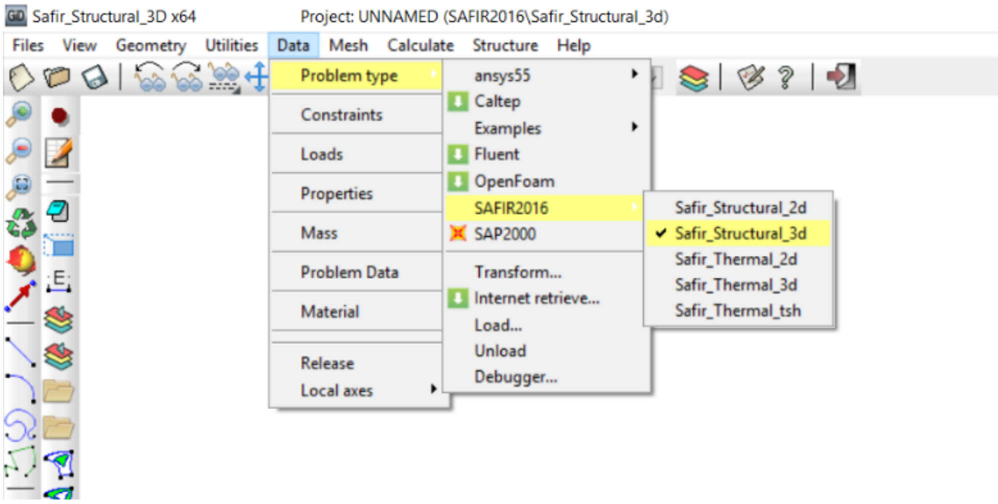


The .tem file of this model (*’rc30x30.tem’*) will need to be copy-pasted in the folder with the structural model.

4. Create model for the 3D structure

4.1. 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.: *3DBuild*
GiD creates a directory with the name *3DBuild.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.

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

GiD has created the directory *3DFrame.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 *'ipe300.tem'* and *'rc30x30.tem'* in the directory *3DFrame.gid*

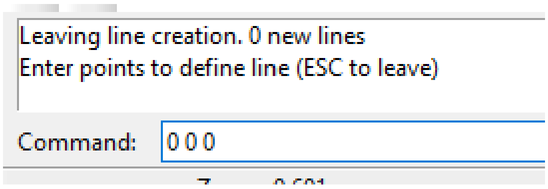
4.3. Create the system geometry (3D structure)

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:

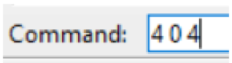
Geometry->Create->Straight Line or

Enter in the command line (at the bottom of the widows) successively the coordinates of the 5 nodes that define the first frame in the command box. After typing the coordinates of a node, click *[Enter]* to validate.

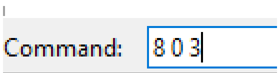


Press *[Enter]*

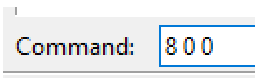
Press *[Enter]*



Press *[Enter]*



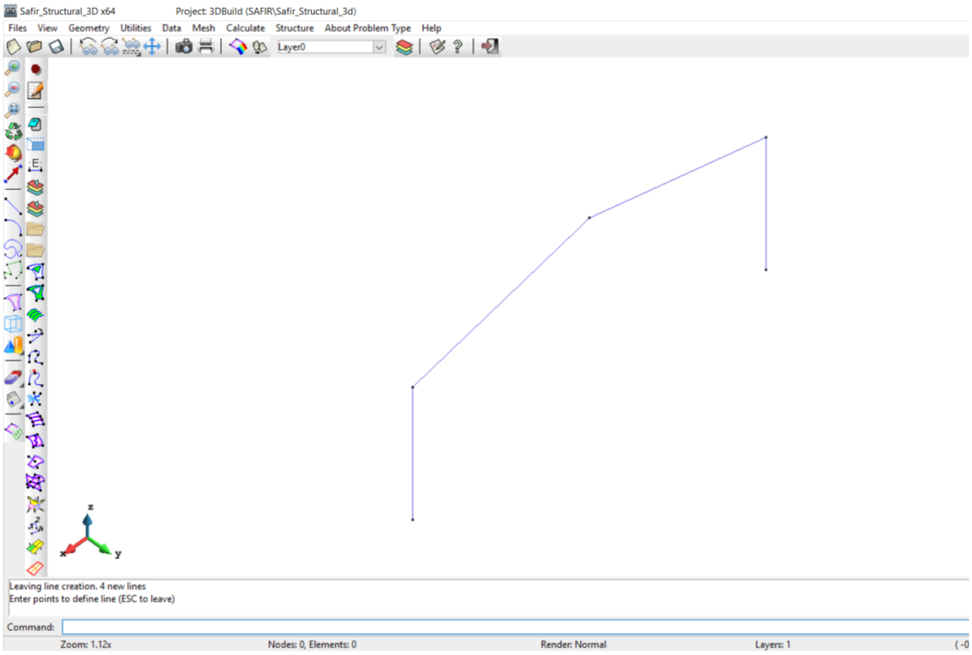
Press *[Enter]*



Press *[Enter]*

Then press *[Esc]* to leave the line creation menu.

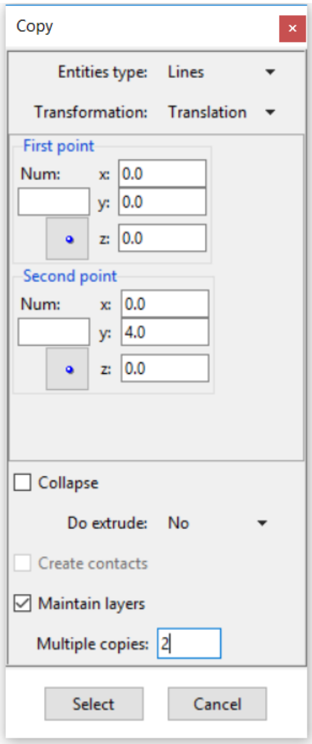
You should see this in GiD:



To create the 2 other frames:

Utilities->Copy *or [Ctrl + c]*

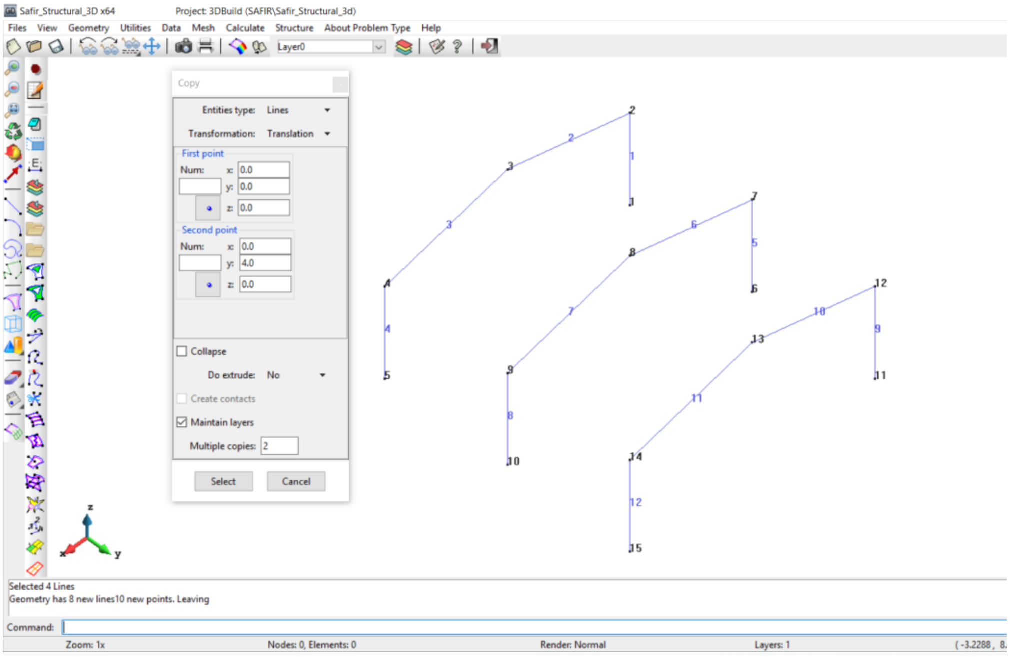
And fill as shown below:



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

To see nodes and beams numbers select:

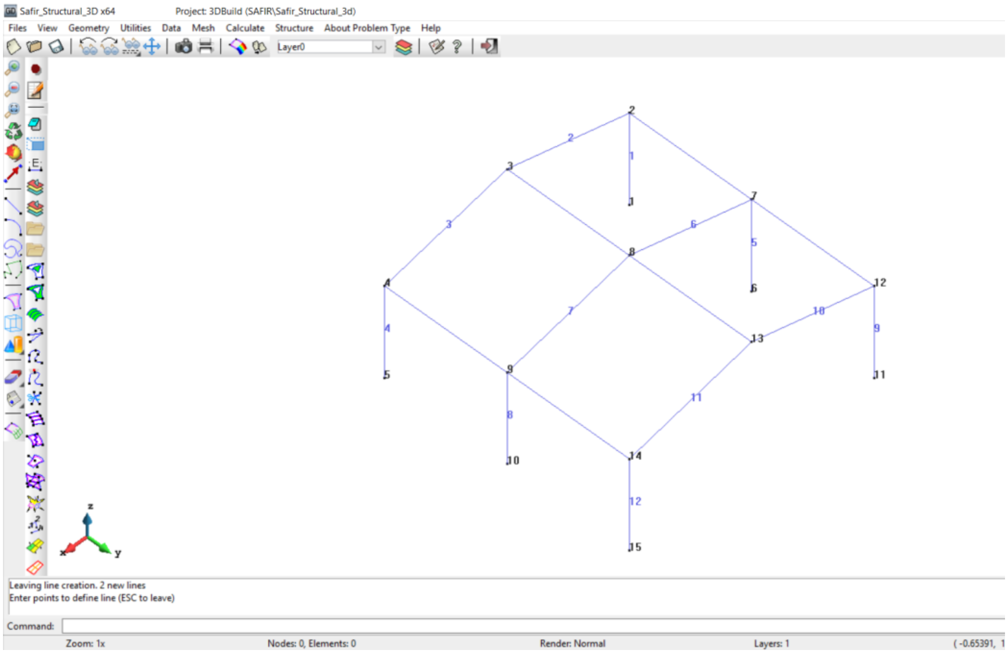
View->Label->All



Finally, create the last system lines:

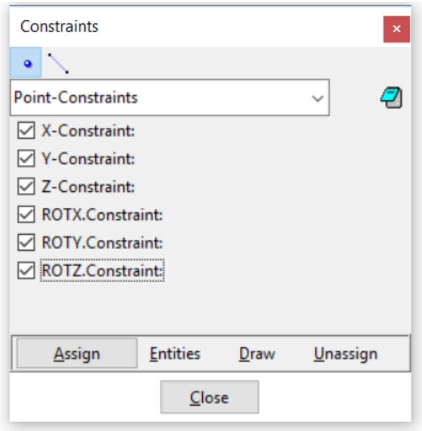
Geometry->Create->Straight Line or 

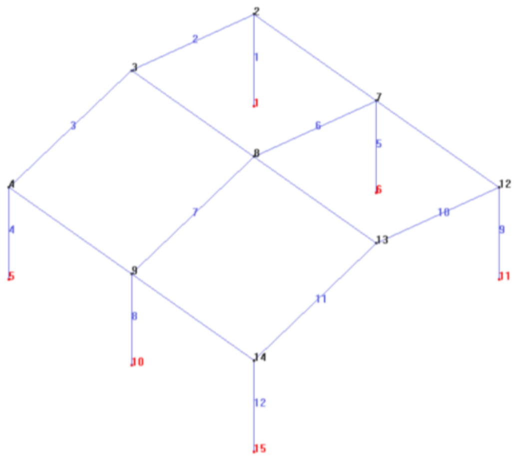
Press *Ctrl + A* to switch the pointer to the mode: '*pick an existing point*'.
Click on Node 2, then Node 7, then Node 12. Press *[Esc]*.
Repeat with Node 3, then Node 8, then Node 13. Press *[Esc]*.
Repeat with Node 4, then Node 9, then Node 14. Press *[Esc]*.
(Note: do not forget to press *Ctrl + A*, otherwise the line would not be connected to the nodes although it might look like it goes through the nodes).



4.4. Define constraints for the supports

From the pull down menu select
Data->Constraints
Select *Point Constraints*
Tick all the boxes for a fully fixed condition. Assign these constraints to *POINT 1, POINT 5*, *POINT 6, POINT 10*, *POINT 11*, *POINT 15*, and press *[Esc].*



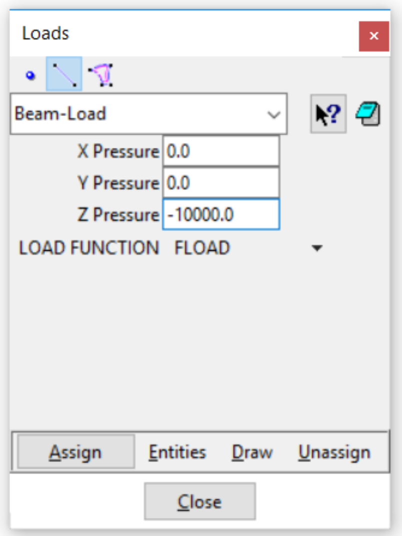


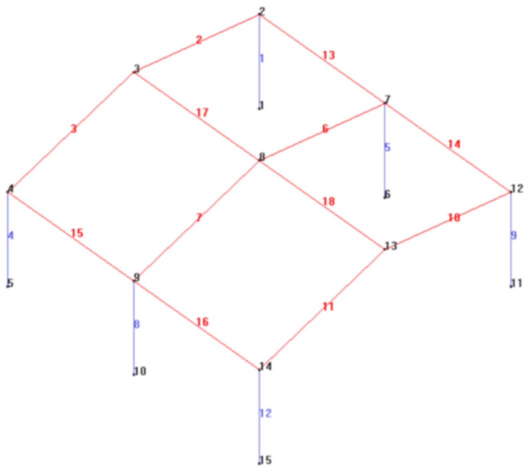
In the dial box, with *Draw->Colors* you can display the constraints.
Press *Finish* or *[Esc]* to leave this view mode.

4.5. Assign the loads

From the pull down menu select
Data->Loads

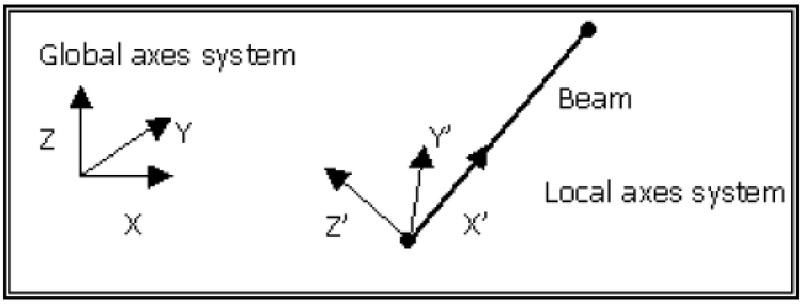
Assign a distributed load of -10 000 N/m in Z direction on the beams, using the load function *FLOAD*. Press *Finish* or *[Esc]* to validate.





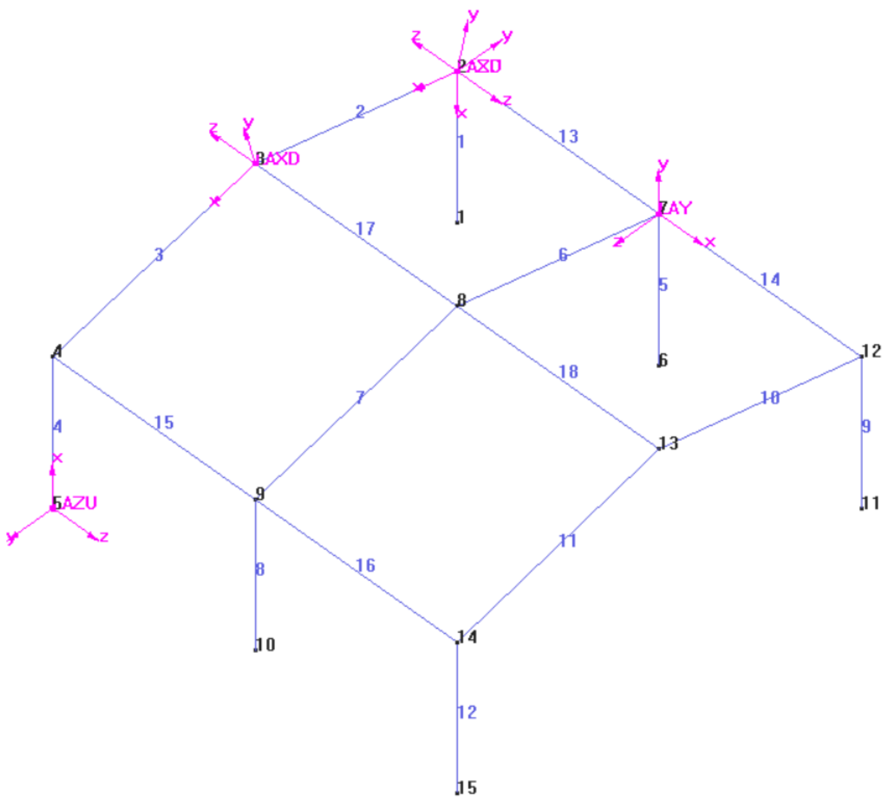
4.6. Create Local Axes

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



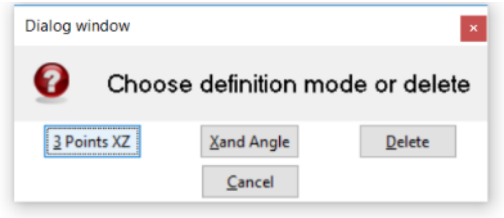
Unlike SAFIR which needs a 4th node to describe the orientation of a cross section on a beam, the GiD-SAFIR interface uses a local X'Y'Z' axes system. When you start the SAFIR calculation the GiD-SAFIR Interface creates the 4th node in the X'Y' plane. If the center of the local axes is not located on 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.

The objective is to create 5 local axes (LAXU, LAXD, LAY, LAZU, LAZD) as shown in the figure below.

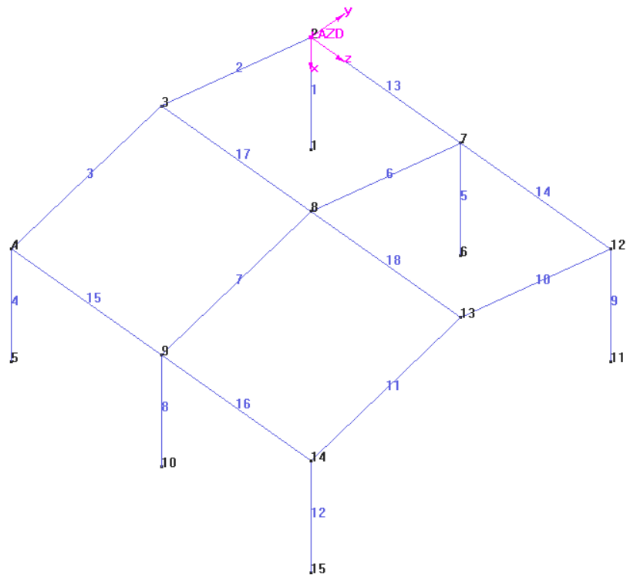


From the pull down menu select:
Data->Local Axes->Define

Enter the name of the new local axe *LAZD*
Select *3 points XZ*



Select *Point 2* as the local axis center. *Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.*
Select *Point 1* as the point in positive x axis.
Select *Point 7* as the third node points to the positive direction along the z axis.
Press *Enter*
You should see this in GiD. This local axis *LAZD* will be used for Columns 1, 5 and 9.

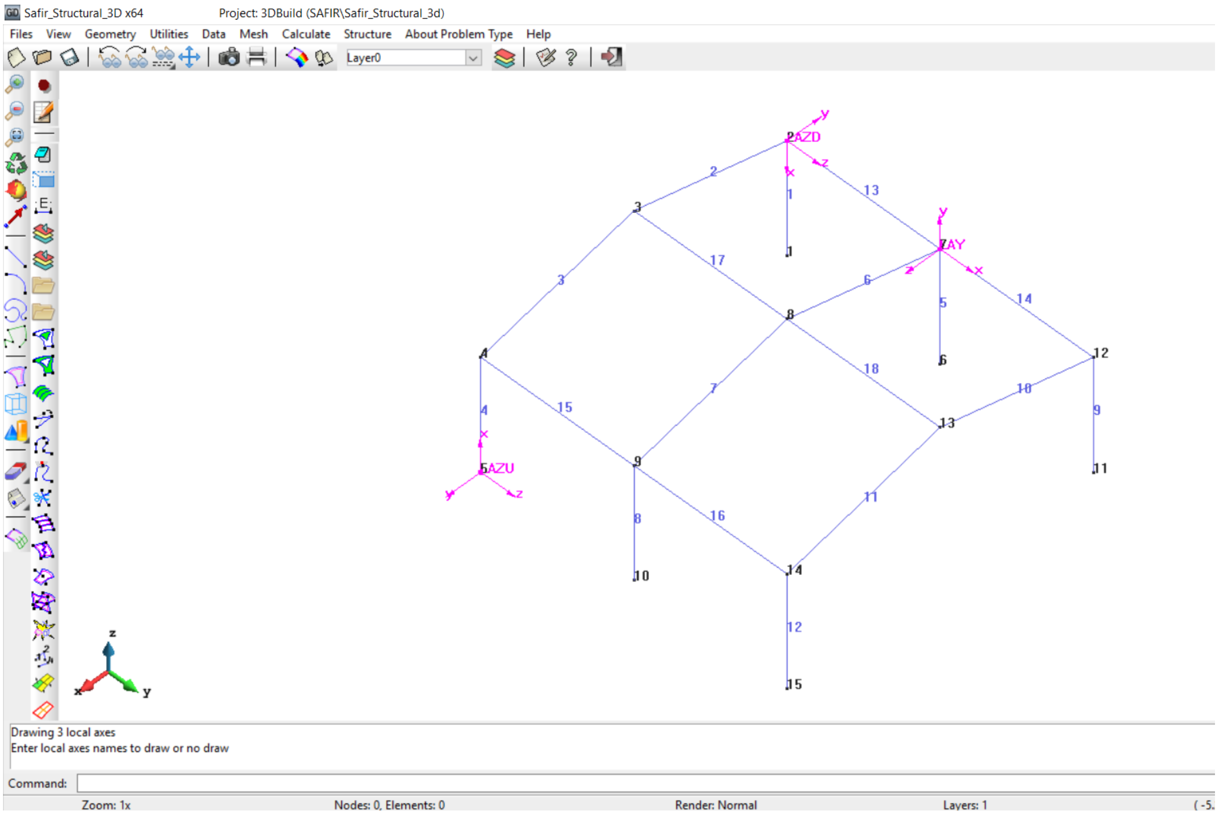


Repeat the operations to create the local axe *LAZU*

Select *Point 5* as the local axis center.
Select *Point 4* as the point in positive x axis.
Select *Point 10* as the point in positive z axis.
Press *Enter*
This local axis *LAZU* will be used for Columns 4, 8, 12.

Repeat the operations to create the local axe *LAY*

Select *Point 7* as the local axis center.
Select *Point 12* as the point in positive x axis.
Select *Point 9* as the point in positive z axis.
Press *Enter*
This local axis *LAY* will be used for Beams 13, 14, 15, 16, 17, 18.



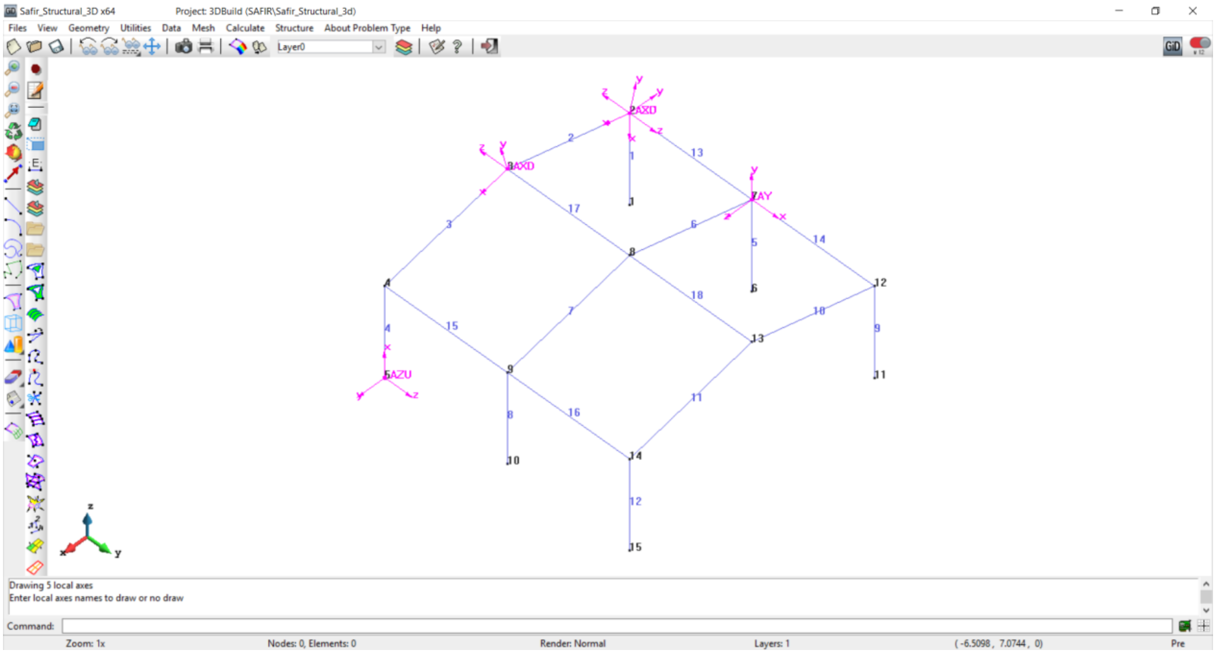
From the pull down menu select:
Data->Local Axes->Define
Enter the name of the new local axe *LAXU*
Select *X and Angle*

Select *Point 2* as the local axis center. *Note: Press "CLTR + A" to allow the selection of an existing point with the mouse.*
Select *Point 3* as the point in positive x axis.
Enter *90* in the command box as the angle.
Press *Enter*

From the pull down menu select:
Data->Local Axes->Define
Enter the name of the new local axe *LAXD*
Select *X and Angle*

Select *Point 3* as the local axis center.
Select *Point 4* as the point in positive x axis.
Enter *90* in the command box as the angle.
Press *Enter*

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

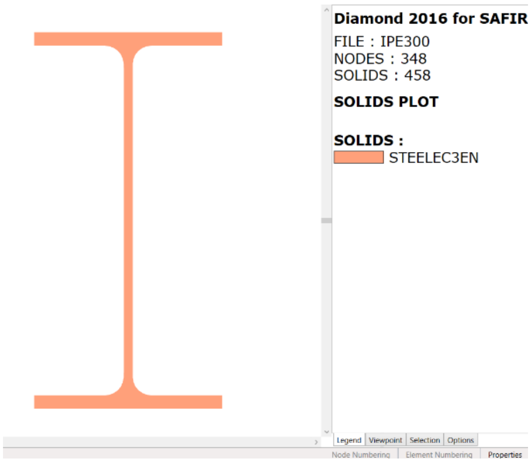


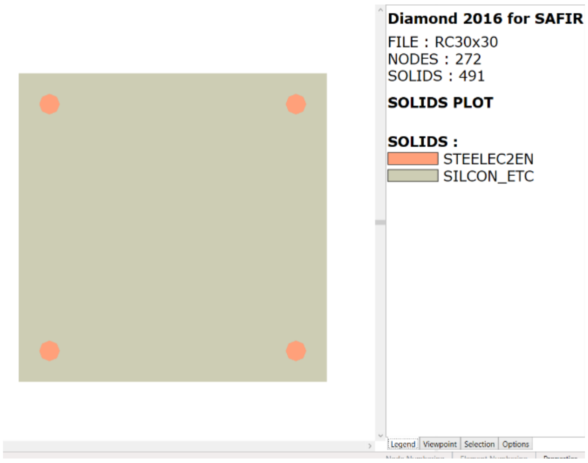
4.7. Define the global materials

From the pull down menu select
Data->Material

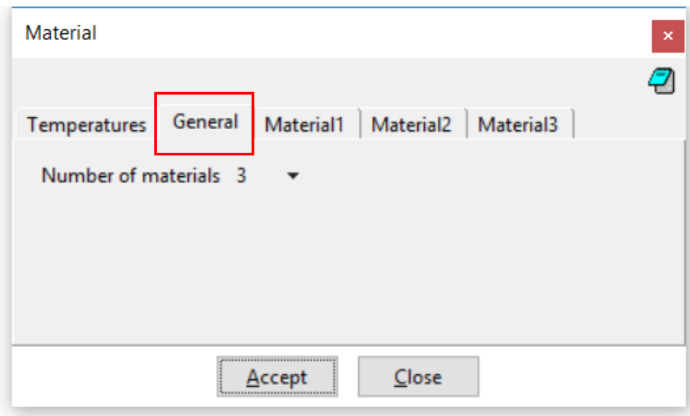
There are 3 materials in the model:
- The steel for the profile IPE300: STEELEC3EN
- The steel for the reinforcement of the concrete columns: STEELEC2EN
- The concrete for the columns: SILCON_ETC

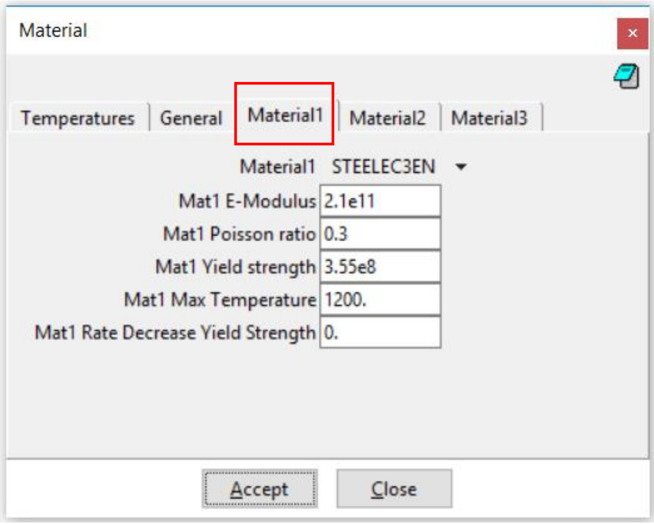
In the section IPE300: MAT 1 = STEELEC3EN
In the section RC30x30: MAT 1 = STEELEC2EN, MAT 2 = SILCON_ETC

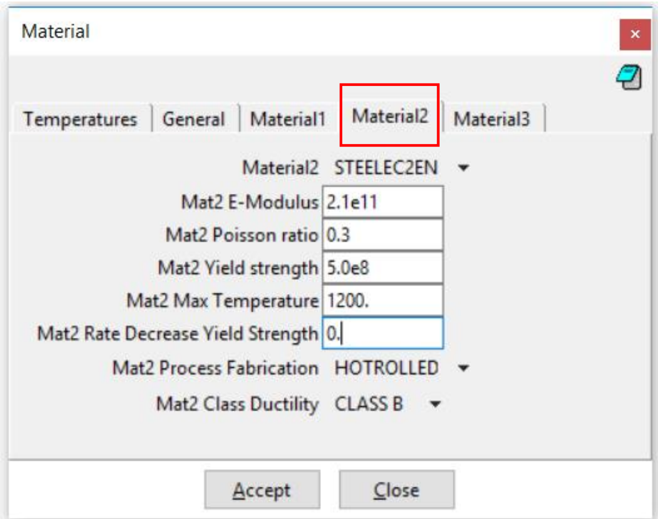


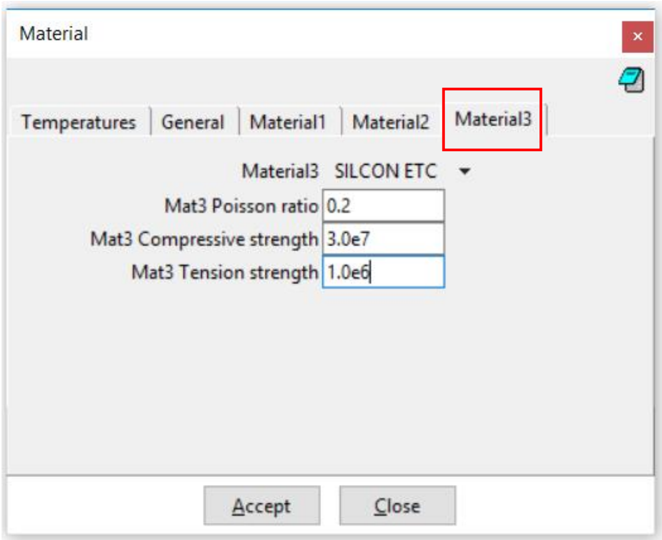


In the structural model, we decide that the order is:
MAT 1 = STEELEC3EN
MAT 2 = STEELEC2EN
MAT 3 = SILCON_ETC









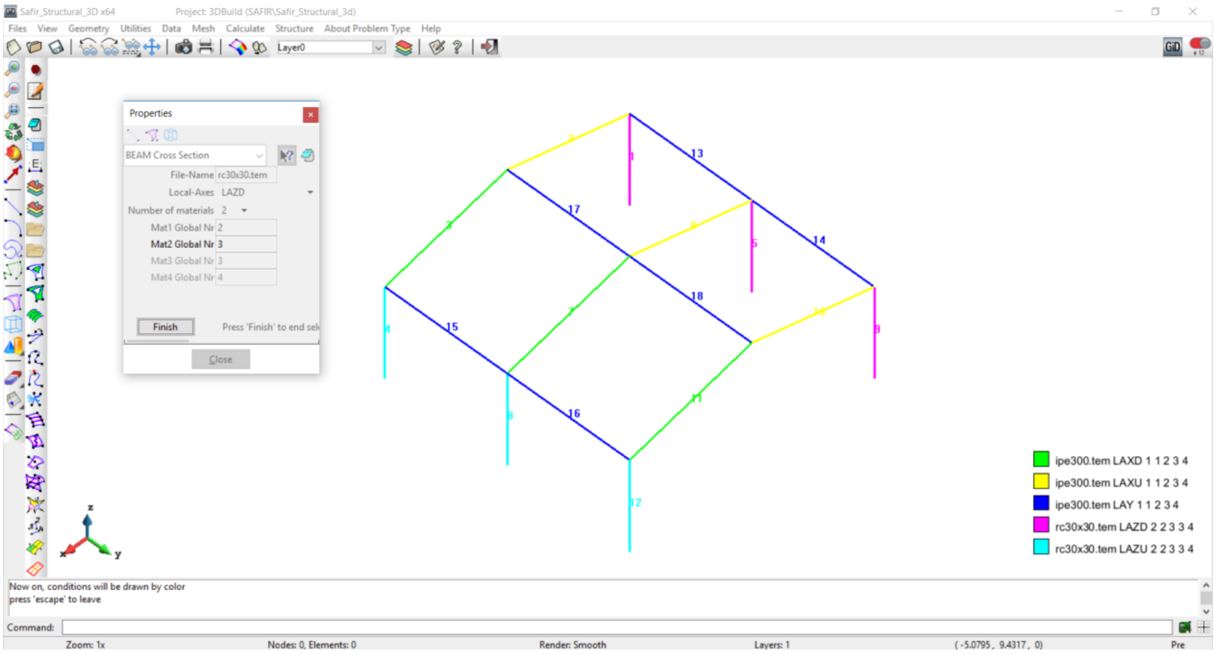
Note: a description of the material *SILCON_ETC* is given in the following paper:

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.

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

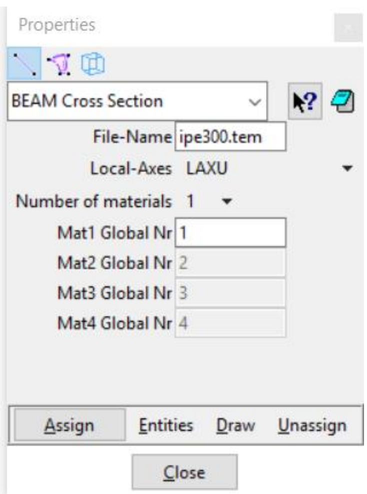
From the pull down menu select
Data->Properties

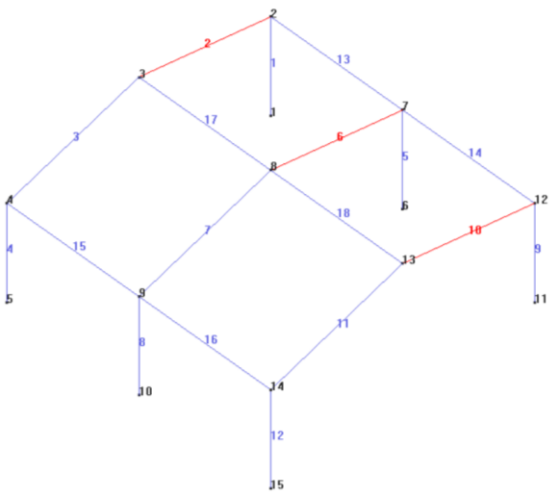
The objective is to assign the .tem file named *ipe300* and *rc30x30* to the model lines.



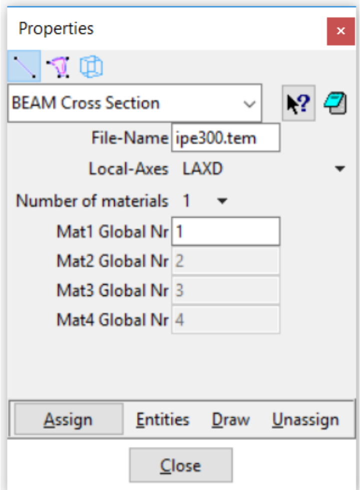
In the dial box of *Data->Properties*, change the File-Name: ***safir.tem*** to the temperature file (.TEM file) of the cross-section, in this case ***ipe300.tem***.

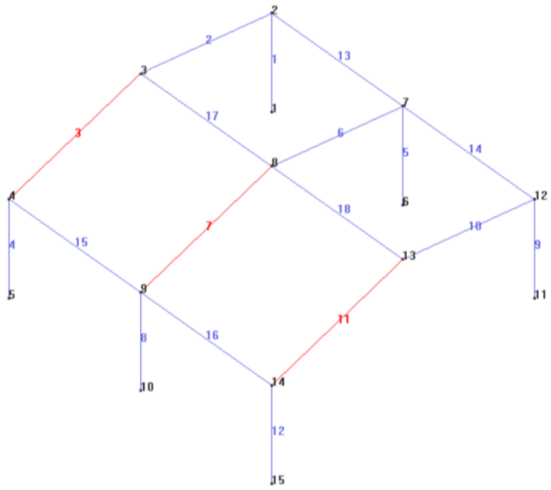
Change ***Local-Axes*** from ***-Automatic-*** to ***LAXU***
Keep the number of materials to 1. The Mat1 Global Nr is 1 (it is the STEELEC3EN).
Assign the ***ipe300.tem*** section to the beams 2, 6 and 10.



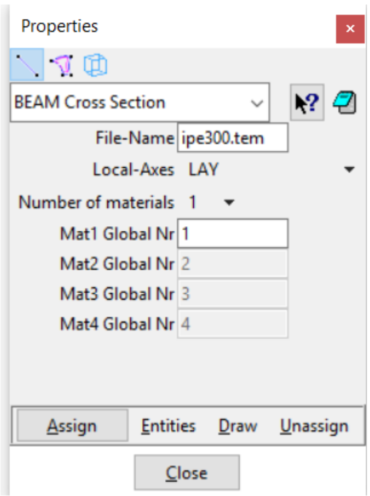


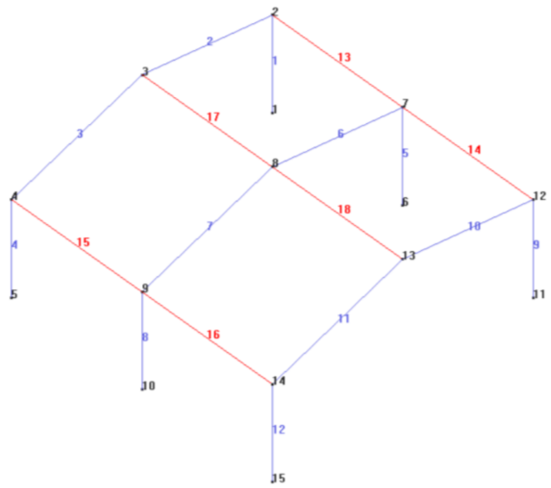
Then, keep the file name to *ipe300.tem* but change *Local-Axes* to *LAXD*. Keep 1 material with the Mat1 Global Nr equal to 1.
Assign to the beams 3, 7 and 11.



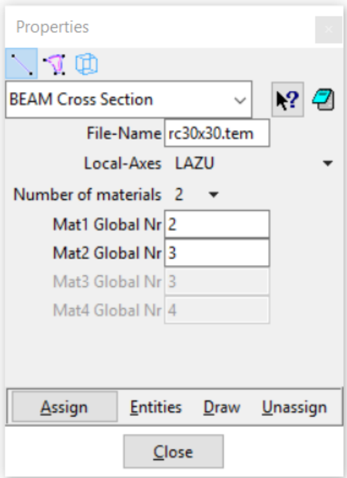


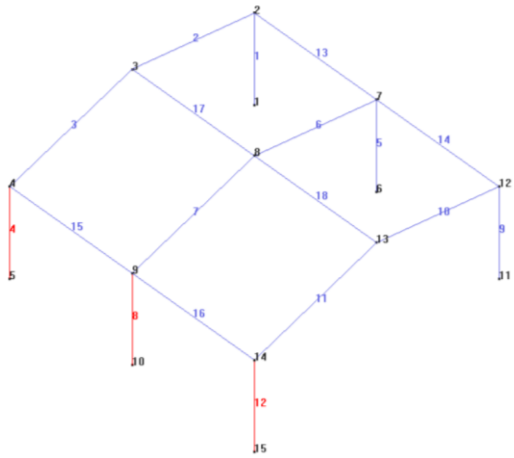
Then, keep the file name to *ipe300.tem* but change *Local-Axes* to *LAY*. Keep 1 material with the Mat1 Global Nr equal to 1.
Assign to the beams 13, 14, 15, 16, 17, 18.



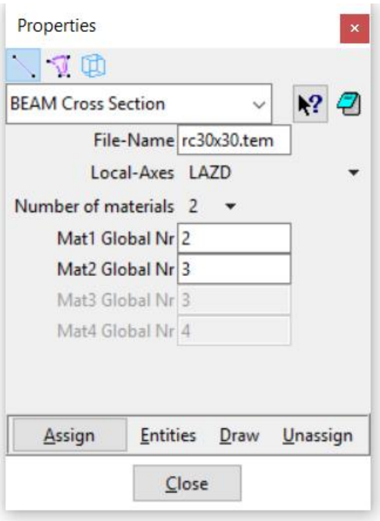


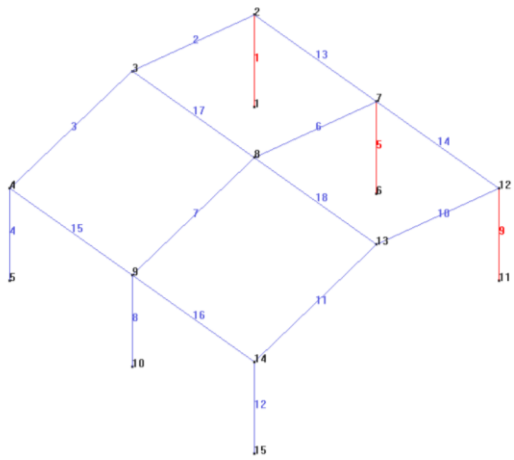
Then, change the file name to *rc30x30.tem*. Change *Local-Axes* to *LAZU*. Specify 2 materials. The Mat1 Global Nr is 2 (it is STEELEC2EN). The Mat2 Global Nr is 3 (SILCON_ETC). Assign to columns 4, 8, 12.





Finally, keep the file name to *rc30x30.tem* but change *Local-Axes* to *LAZD*. Keep 2 materials with Mat1 Global Nr is 2 and Mat2 Global Nr is 3. Assign to columns 1, 5 and 9.



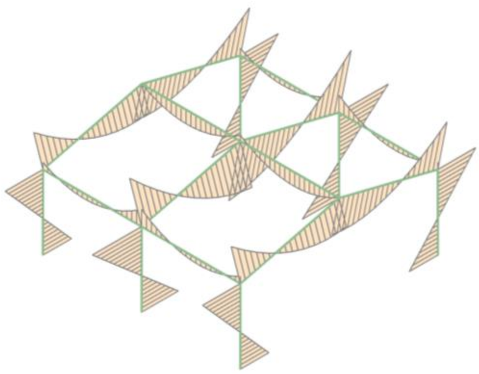
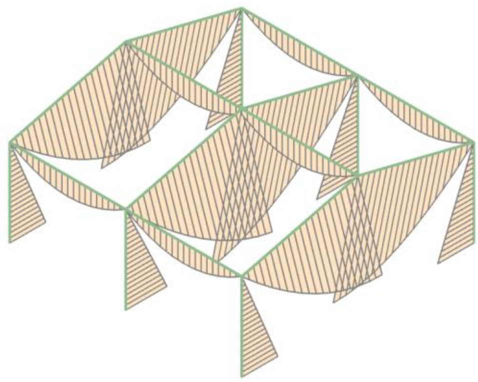


You can draw the local axes of the beams to check the model.
Select *Draw -> All Conditions -> Include Local Axes*

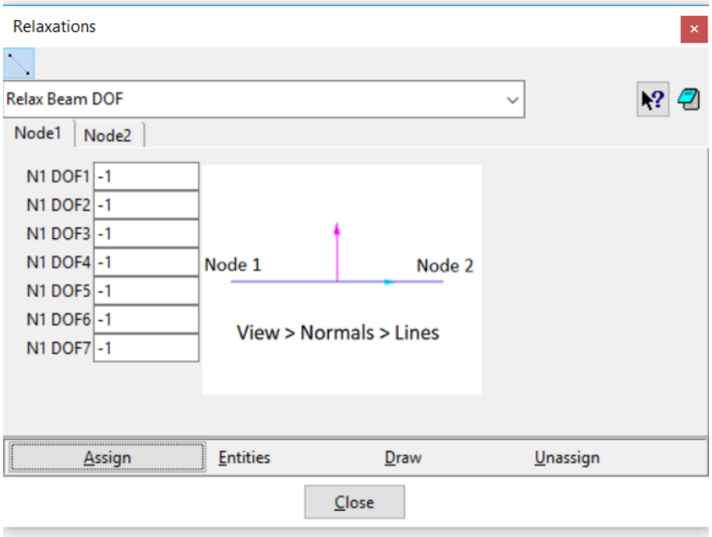
4.9. Assign the RELAX conditions

From the pull down menu select
Data->Relaxations

The objective is to assign pinned conditions at the end of all the beams in the model (i.e. no transfer of strong axis bending moment between the beams and the columns).

Before (without relaxations)	After (with relaxations)
	

After selecting *Data->Relaxations*, this window appears:

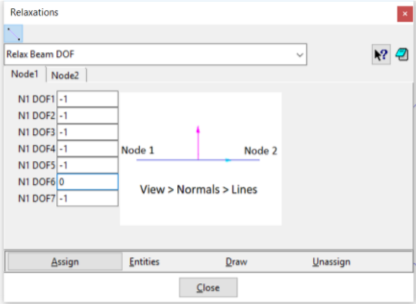


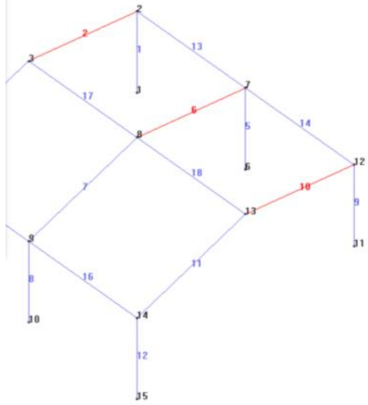
For each *Beam*, the user can apply *Relaxations* conditions to either extremity (*Node1* or *Node2*), according to any of the 7 degree of freedom at this end node.

The degrees of freedom are to be considered in ***Local Axes*** coordinates!

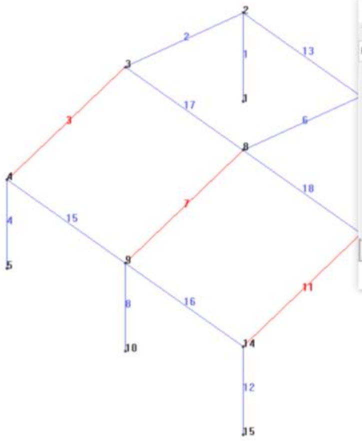
To identify *Node1* from *Node2*, arrows are drawn on the beam elements of the structure, as indicated on the picture in the window here above.

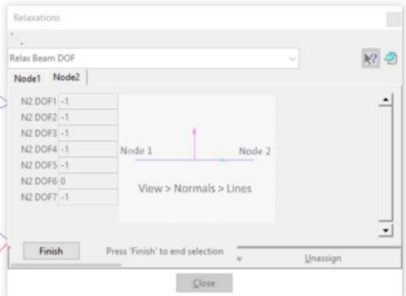
Assign a value of “0” to N1 DOF6 and assign to Beams 2, 6 and 10.
Press *Finish* or *[Esc]* to validate.



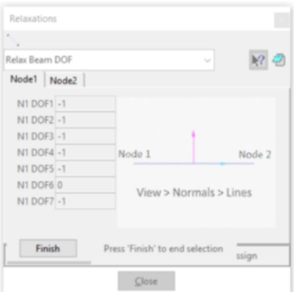


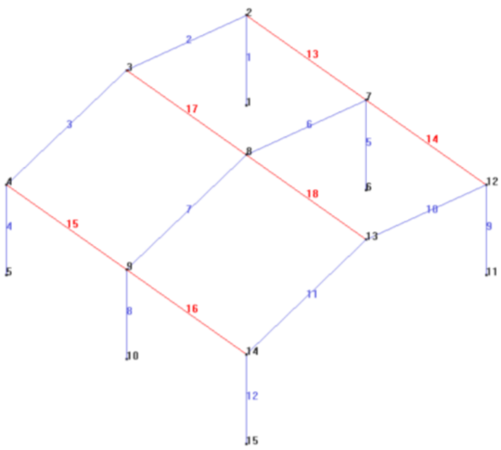
Assign a value of “0” to N2 DOF6 and assign to Beams 3, 7 and 11.
Press *Finish* or *[Esc]* to validate.



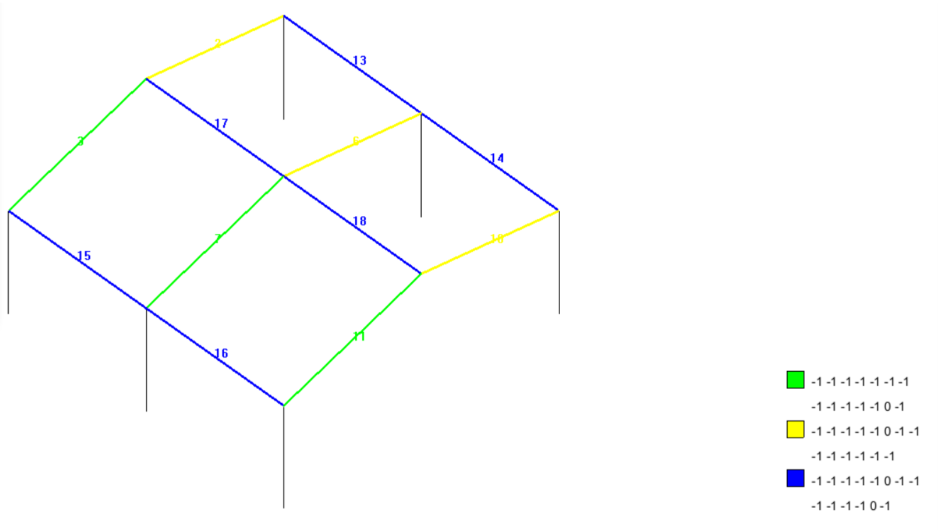


Assign a value of “0” to both N1 and N2 DOF6. Assign to Beams 13, 14, 15, 16, 17, 18.
Press *Finish* or *[Esc]* to validate.



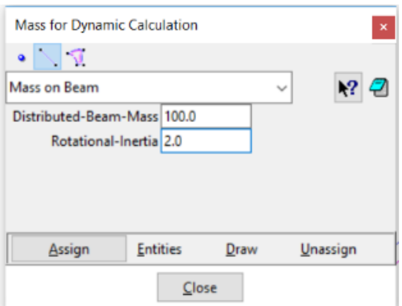


You can draw the *Relaxations* to check the model.
Select ***Draw -> Colors***



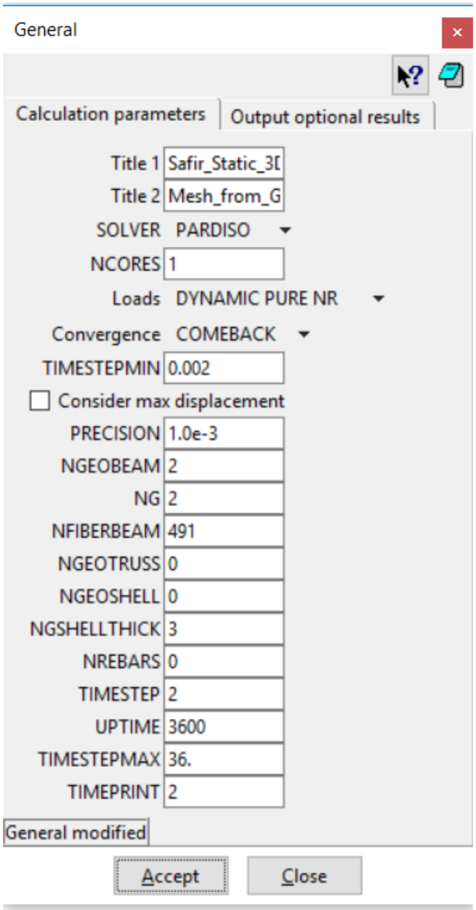
4.10. Assign the mass

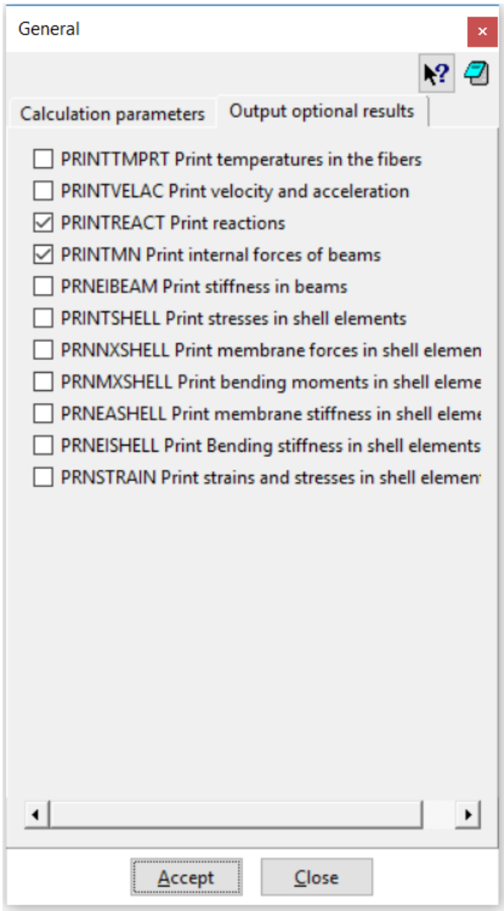
To define the mass for dynamic calculation, select from the pull down menu:
Data-> Mass
Select ***Mass on Beam*** and put 100 kg/m as Distributed-Beam-Mass and 2 as Rotational-Inertia. Assign to all the beam elements.



4.11. Define the general problem data

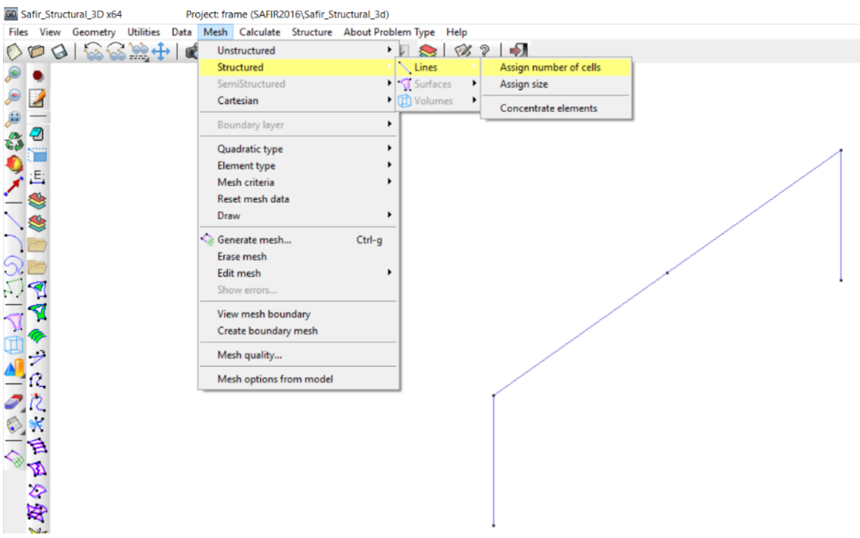
Select from the pull down menu: ***Data->Problem Data***
And fill as shown below



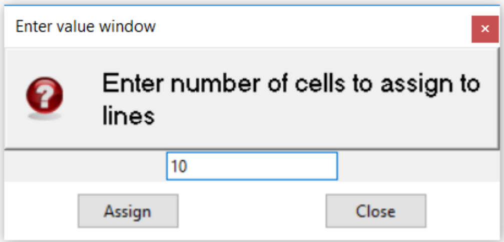


4.12. Define the mesh

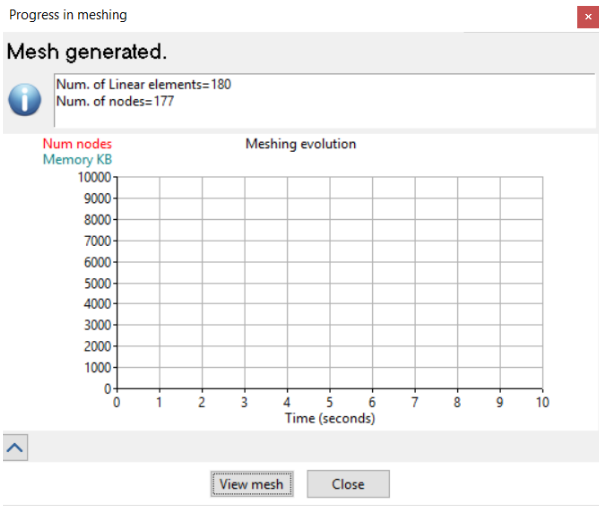
Select ***Mesh -> Structured -> Lines -> Assign number of cells***



Assign 10 elements. 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.