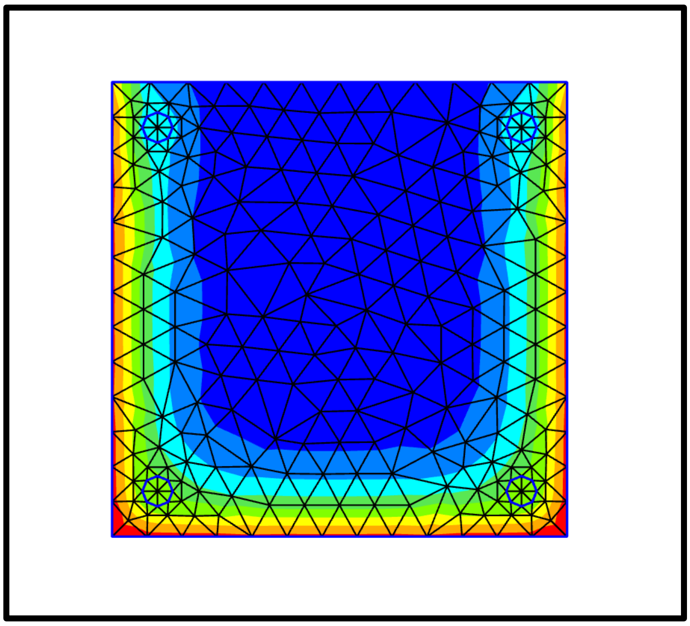
SAFIR® training session – level 1
Johns Hopkins University, Baltimore

Example: 2D thermal analysis of a beam

“Reinforced concrete section heated on 3 sides”

T. Gernay & J.M. Franssen



1. General description

This example deals with a 2D thermal analysis of a reinforced concrete section.

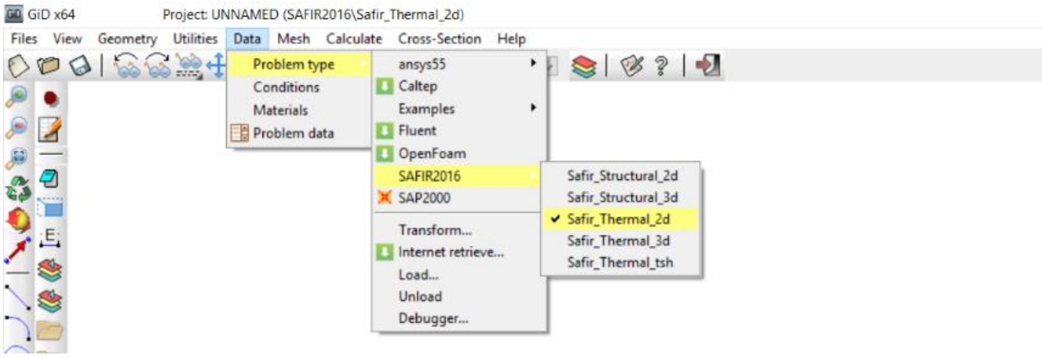
General data:

- Section 30 cm x 30 cm in concrete
- 4 steel reinforcement bars of 20 mm diameter and with 30 mm axis cover
- Material model from Eurocode 2 part 1-2
- Exposed to ISO fire on 3 sides
- In contact with atmosphere at 20°C on the fourth side

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

2. 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]

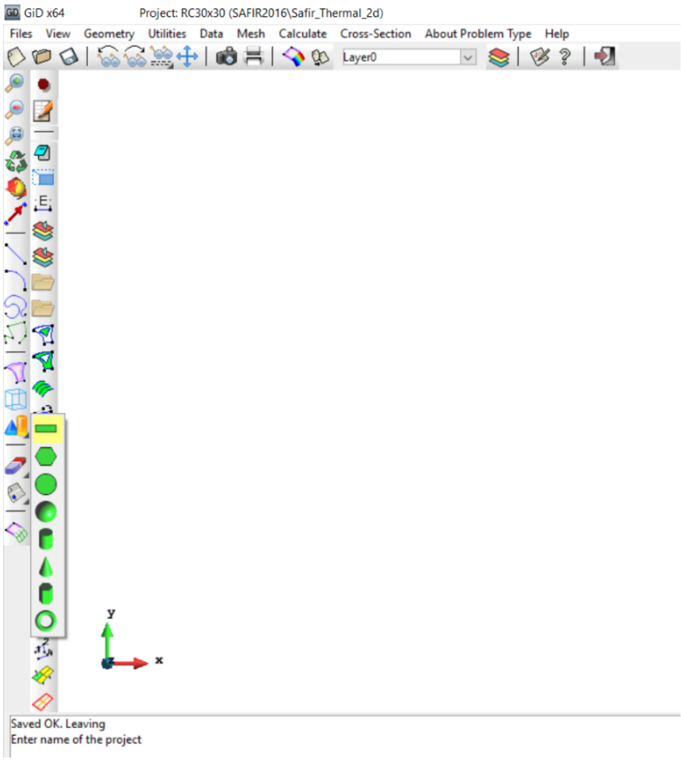
Enter a file name, e.g.: ***RC30x30***
GiD creates a directory with the name ***RC30x30.gid***
GiD creates a number of system files in this directory.
When you start the SAFIR calculation the Safir . IN, .OUT and .TEM files will be created in this directory.

Note: the project's name cannot contain spaces or special characters. Regarding the names of the files, SAFIR is not case sensitive.

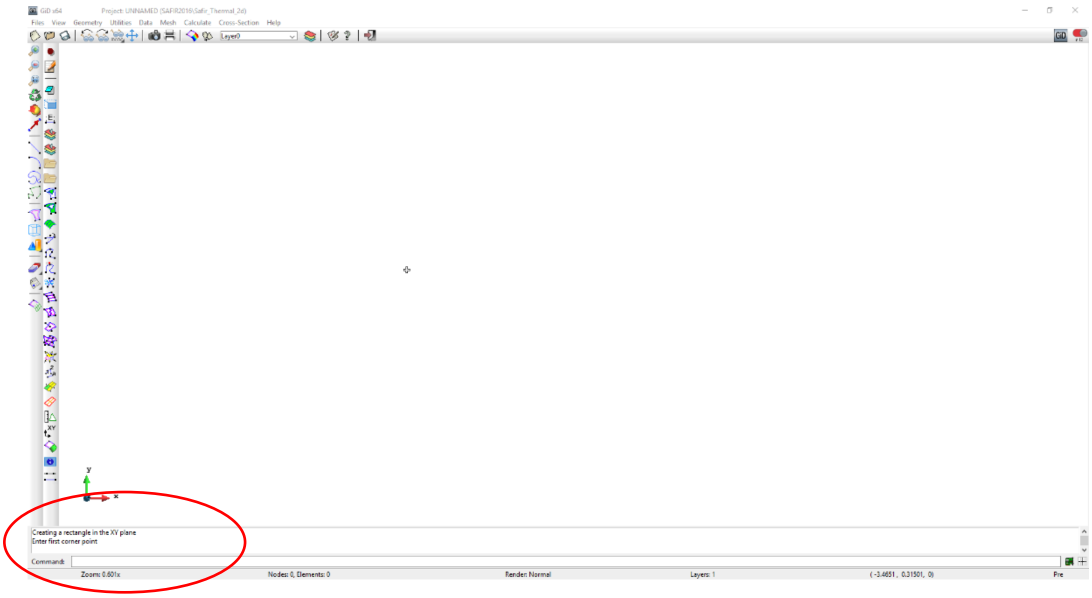
3. Create the geometry in the xy-plane

3.1. The concrete section

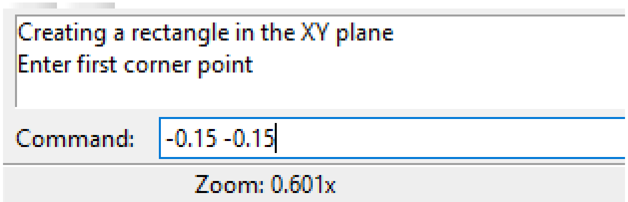
From the left bar menu, select: ***Create object -> rectangle***



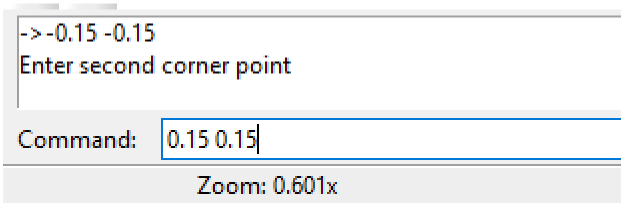
Follow the instructions in the command box: ***Enter first corner point***



Zoom on the command box:

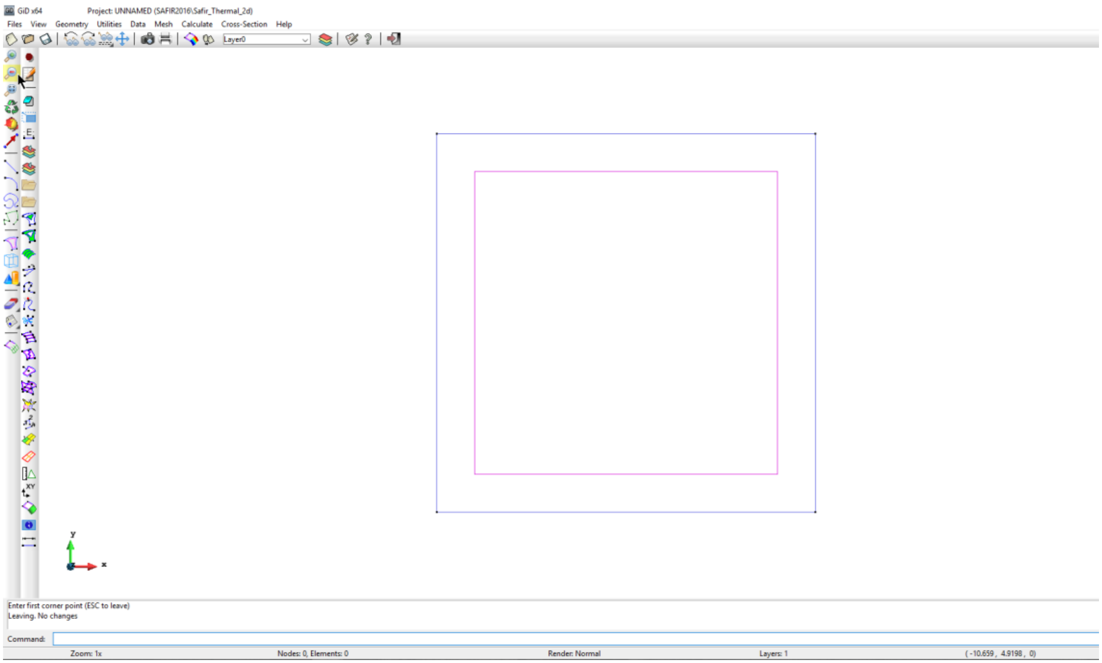


Press « Enter »



Press « Enter »

You can Adjust the zoom to center the section on the screen. GiD displays this section:

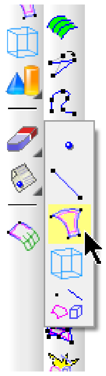


Note: the blue lines represent the contour of the section, while the pink lines represent the surfaces delimited by the blue lines. For instance, blue lines are used to assign thermal boundary conditions to the contour of a cross-section, while the pink surfaces are used to assign thermal properties to an area of the cross-section.

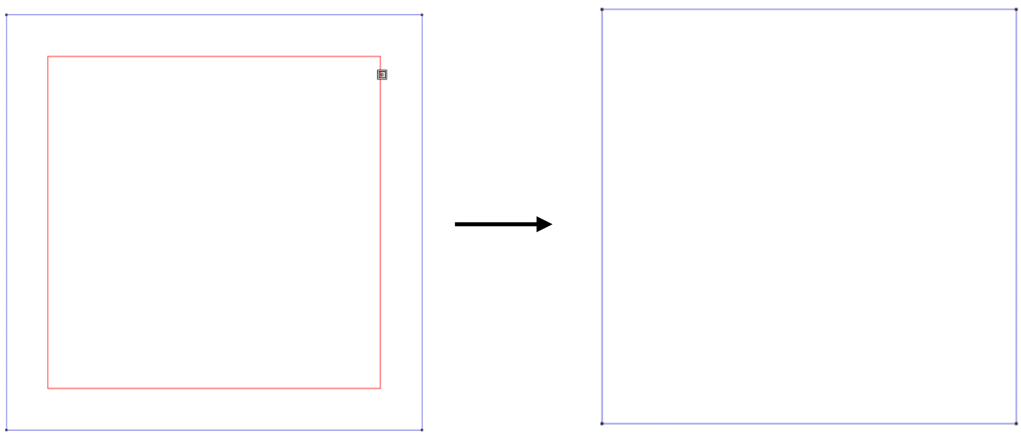
3.2. The rebars

Four reinforcement bars have to be introduced in the section.

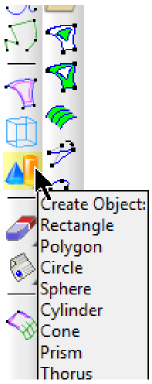
First, delete the concrete surface (in pink). From the left bar menu: ***Delete -> surface***

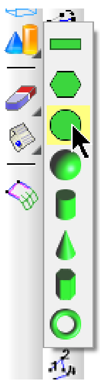


Select the surface and press ***Esc*** to validate.

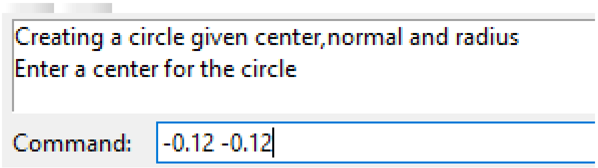


Then, from the left bar menu, select: ***Create object -> circle***

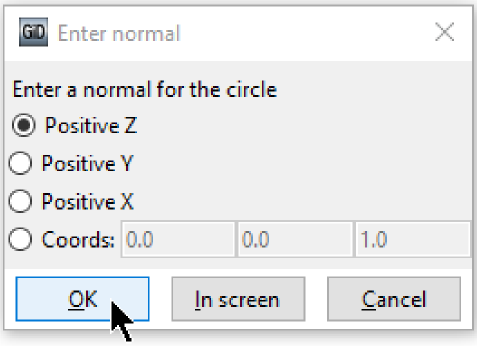




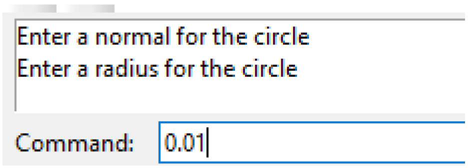
Following the instructions in the command box, introduce the coordinates of the center of the first rebar.



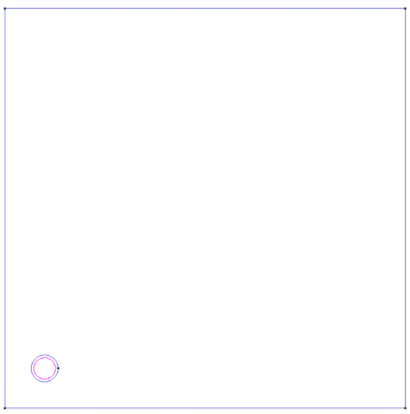
Press ***Enter.*** This window appears in GiD. Select ***Ok***.



Specify the radius of the first rebar.



Press ***Enter.*** This window appears in GiD.



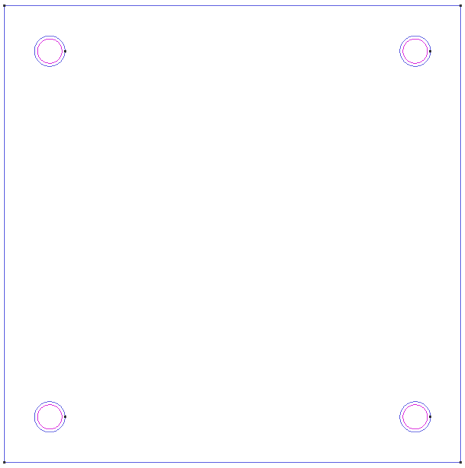
Then repeat 3 times this procedure in order to create the 3 other rebars with the following coordinates for the center of the circles:

(-0.12 0.12)

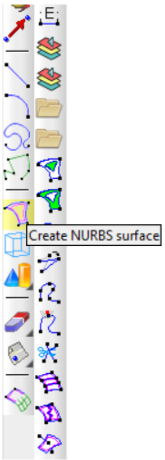
(0.12 -0.12)

(0.12 0.12)

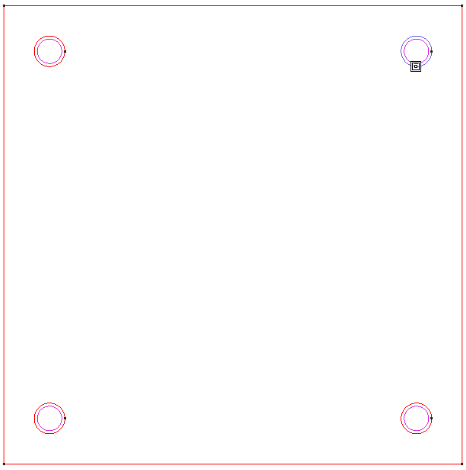
At the end, you should see this in GiD:

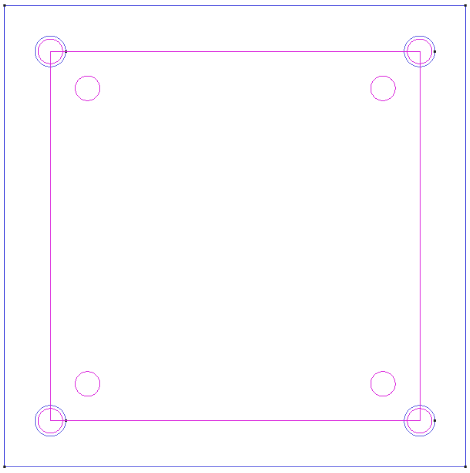


Now, we need to create the concrete surface that contains the 4 holes. From the left bar menu, select: ***Create NURBS surface***.



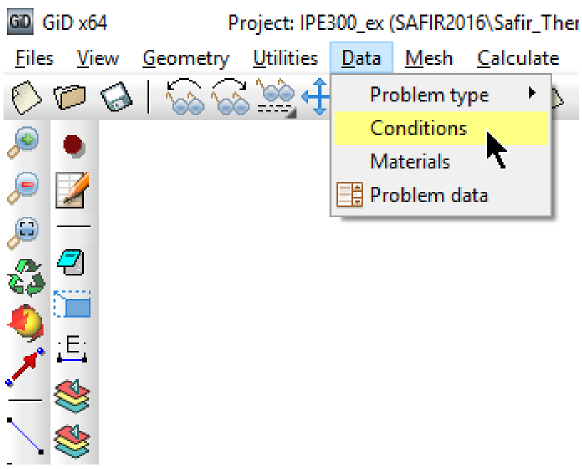
Then, select all the lines that define the contour of the concrete area (including the rebars contour). Press the ***Esc*** key to validate. GiD displays the surface that includes the holes.



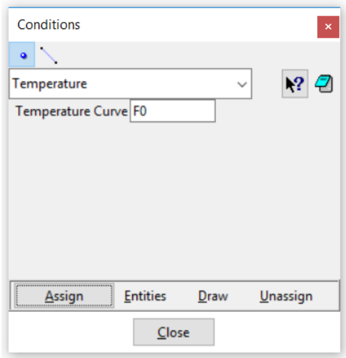


4. Assign the thermal boundary conditions

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

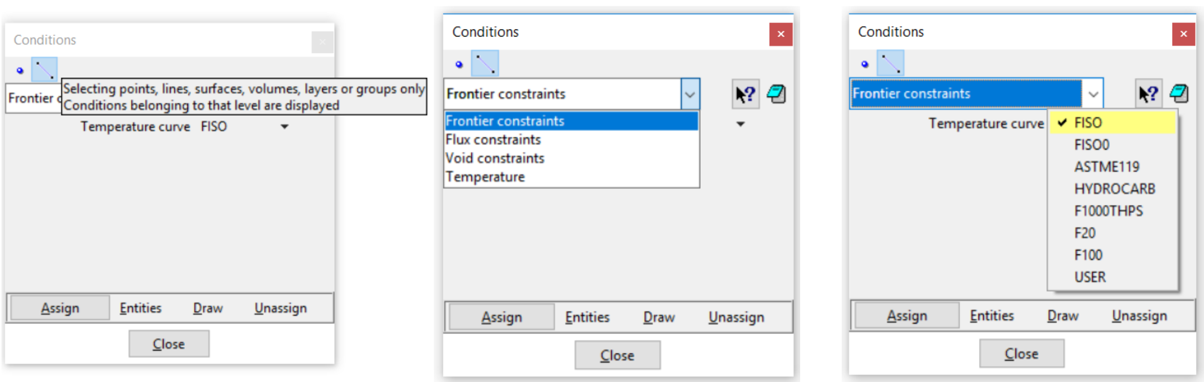


This window appears in GiD:

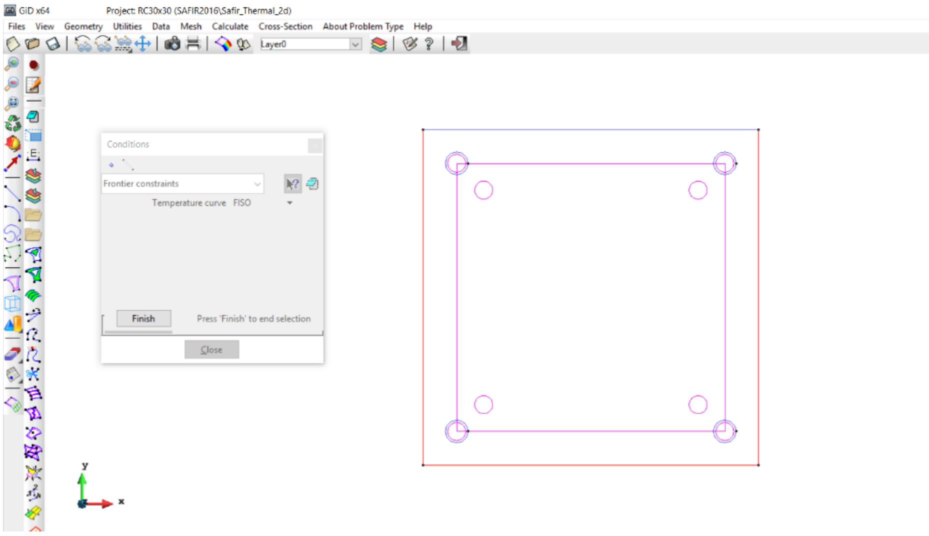


Select the  button ("Line"). On the first pull down list, select: ***Frontier constraints***

Different time-temperature curves are predefined. Select ***FISO*** for the ISO 834 fire curve.

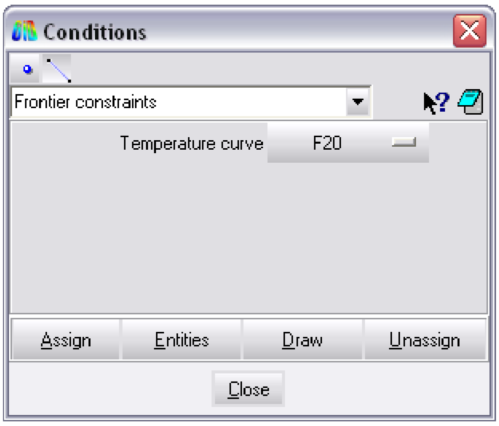


Click on the ***Assign*** button and assign it to the section as shown below.

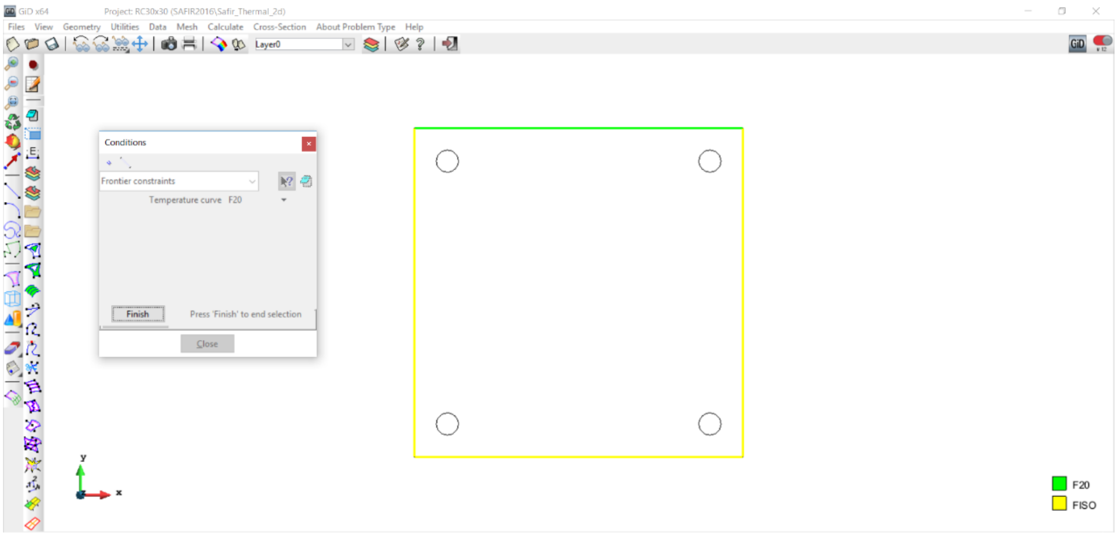


Press *[Esc]* or click on ***Finish*** to confirm

Then select ***F20*** as temperature curve and assign it to the upper side of the section.

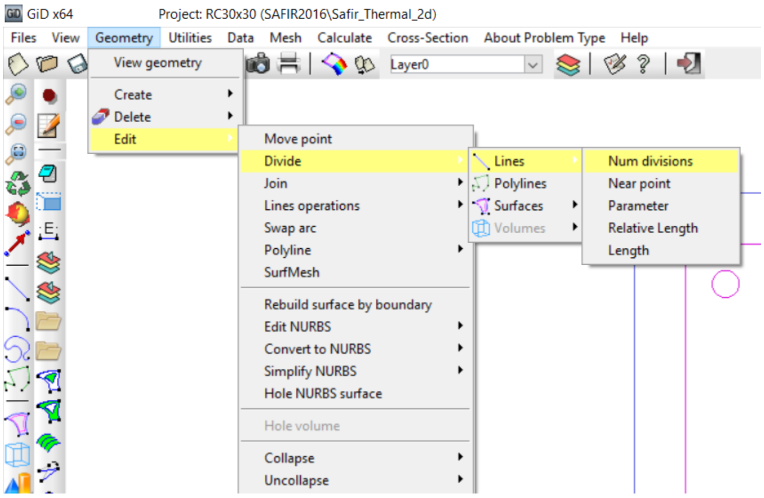


Select ***DRAW->Colors*** in the Conditions dialog box to display the frontier constraints
Press *[Esc]* or click on ***Finish*** to leave this view mode.

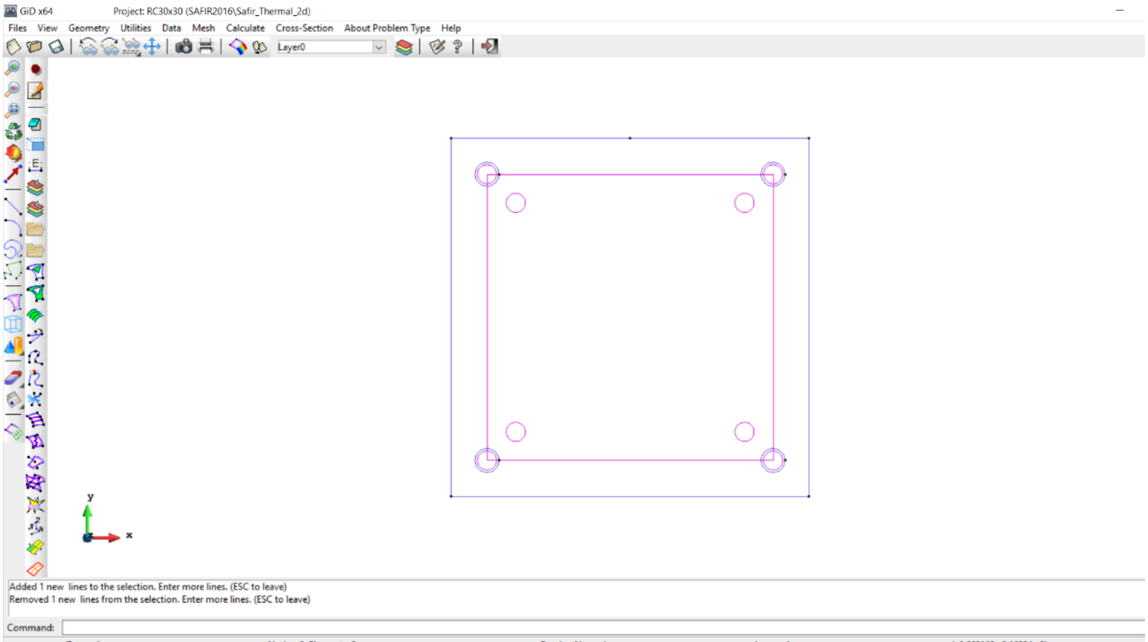


5. Assign a torsion constraint (for the torsional analysis)

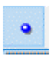
The torsion constraint needs to be applied on a node that is on an axis of symmetry of the section. To create such a node, from the pull down menu select:
Geometry -> Edit -> Divide -> Lines -> Num divisions

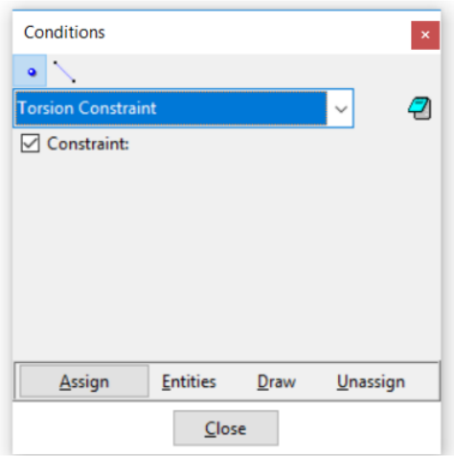


Enter number of divisions: 2. Select the line at the top of the section. Validate with ***Esc***.

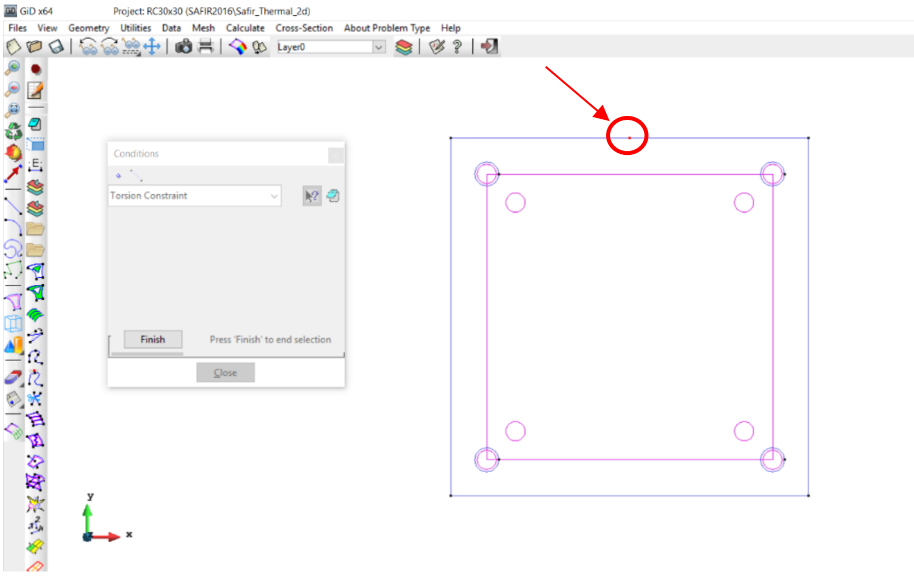


Then, from the pull down menu select:
Data->Conditions

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

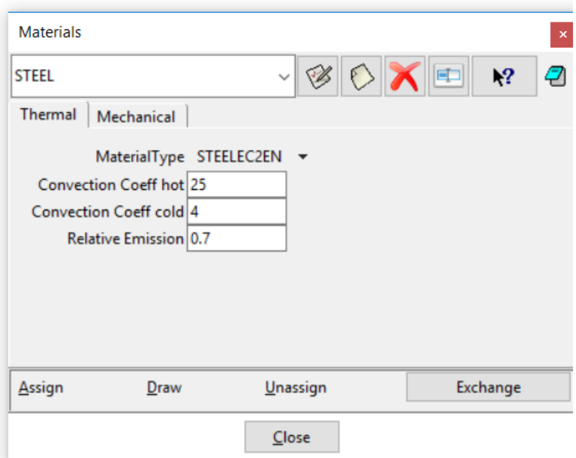


Select the node on the axis of symmetry and validate with ***Finish.***

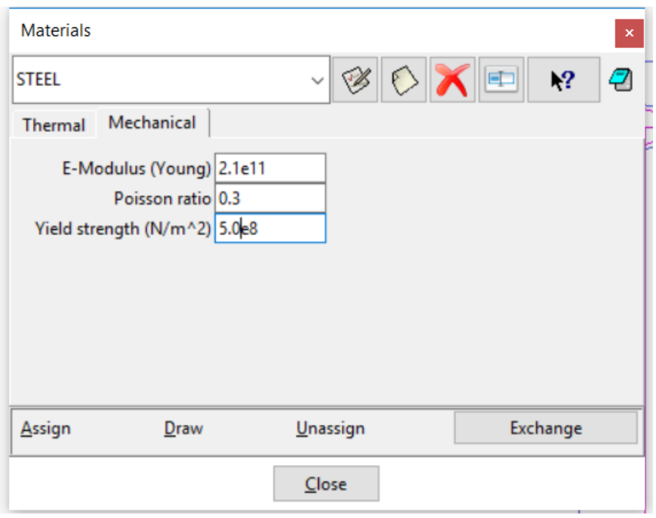


6. Assign the materials

From the pull down menu select:
Data->Materials
Select *STEEL* from the dialog box pull down list
The *Thermal* tab is active.
Then select:
STEELEC2EN as Material Type
A Convection Coeff hot of *25*
A Convection Coeff cold of *4*
A Relative Emission of *0.7*

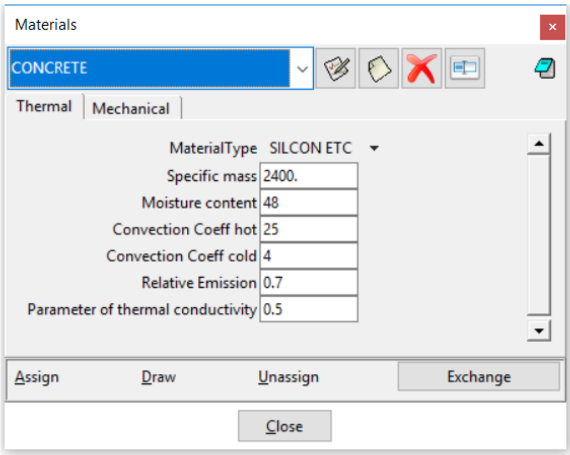


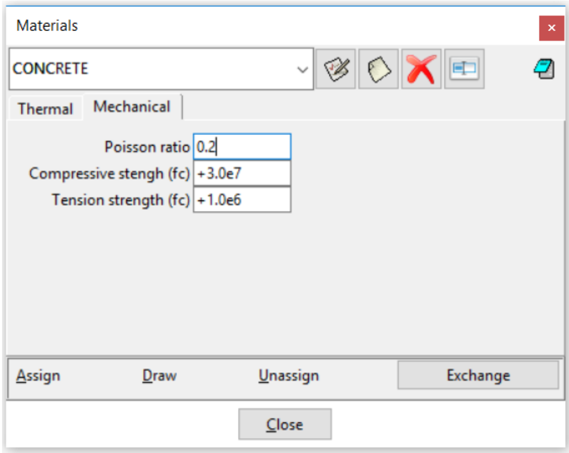
Then select the *Mechanical* tab. Input:
A Young modulus of *210 000 MPa*
A Poison ratio of *0.3*
A Yield strength of *500 MPa*



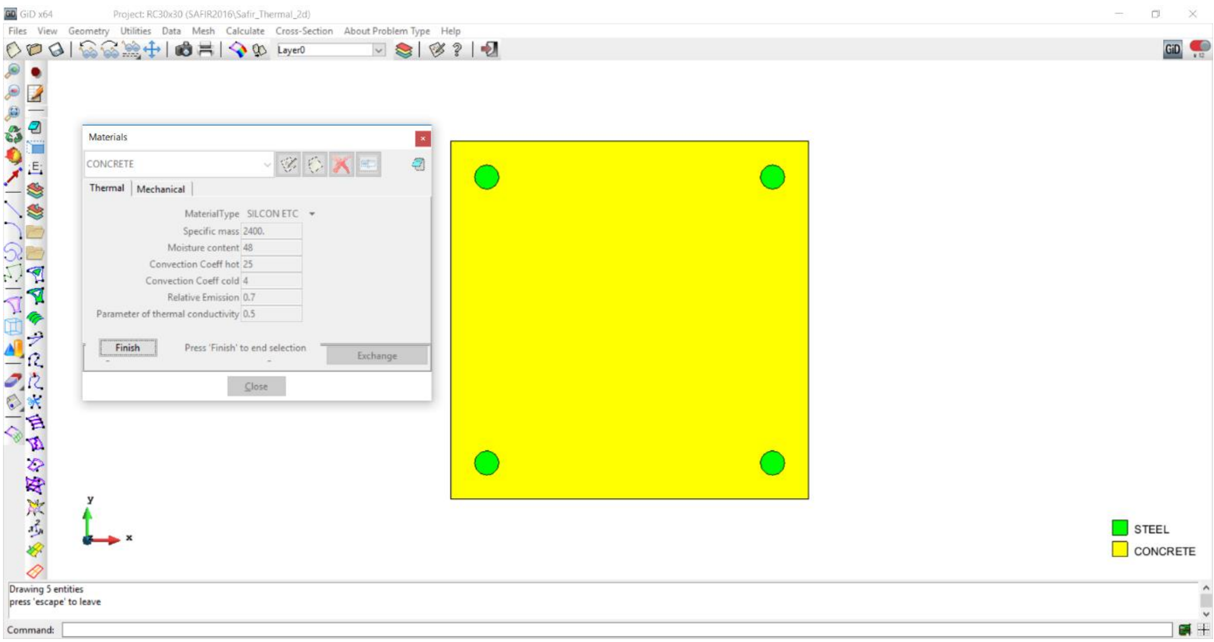
Click on *Assign-> Surfaces* and assign it to the steel rebars surfaces.
Press *[Esc]* or *Finish* to confirm.

Then, select ***CONCRETE*** from the dialog box pull down list. The ***Thermal*** tab is active.
Select:
SILCON_ETC as Material Type
Modify the thermal properties of the material if needed. Modify the mechanical properties if needed in the ***Mechanical*** tab. Then assign the material to the concrete surface.





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

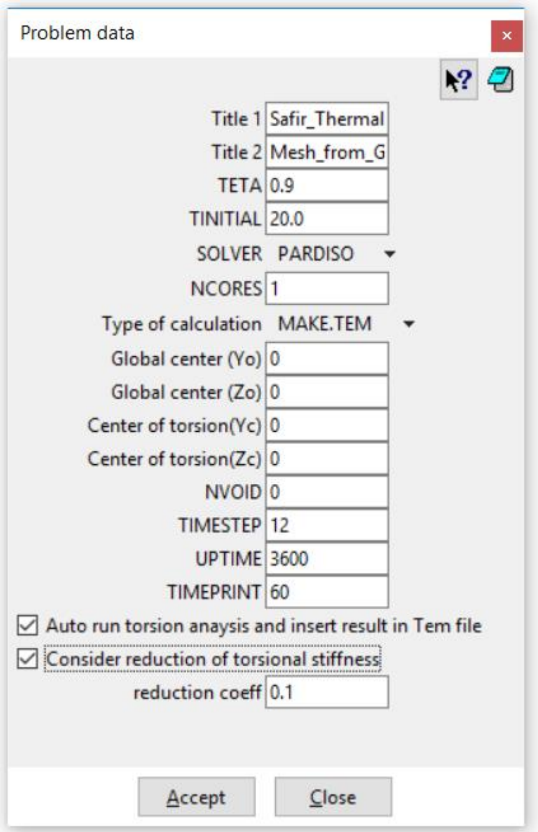


7. Assign the general data

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

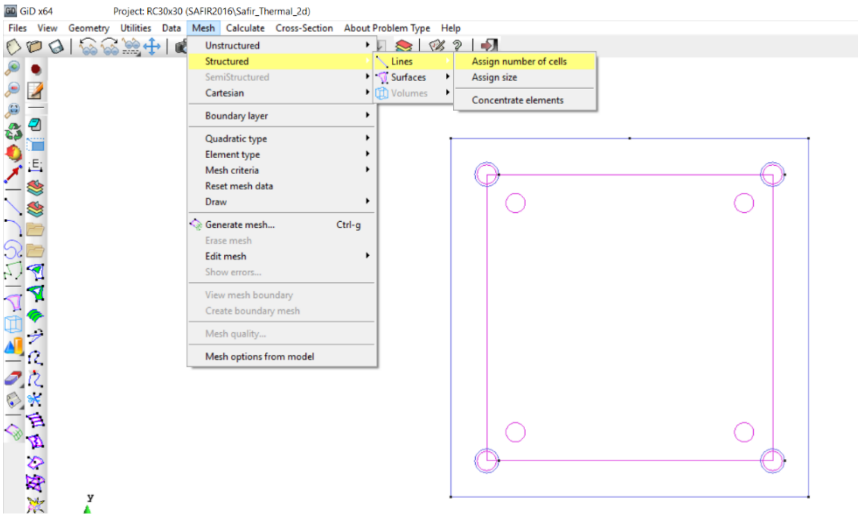
In the Problem Data dialog mask enter:
TIMESTEP, UPTIME, TIMEPRINT as needed
Do not forget to tick the box ***Autorun Torsion Analysis***
Also tick the box ***Consider reduction of torsional stiffness*** and leave the value as 0.1

Click on the ***Accept*** data button

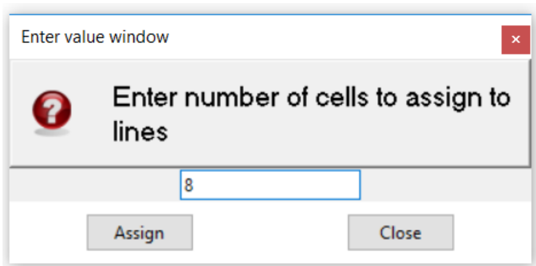


8. Create the mesh

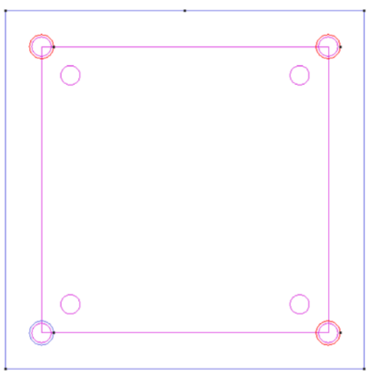
Select *Mesh -> Structured -> Lines -> Assign number of cells*



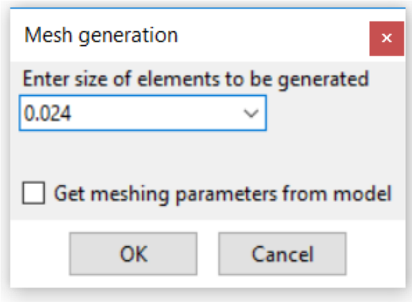
Enter 8 as the number of cells



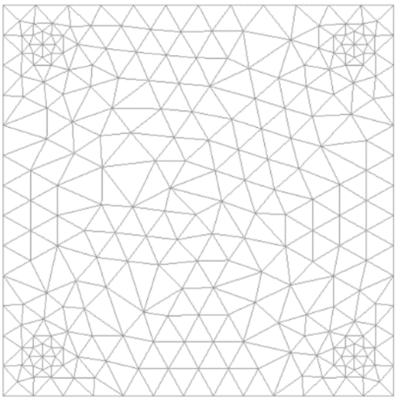
Assign to the lines that form the contour of the rebars.



Select *Mesh->Generate mesh or use [Ctrl + g]*
Enter 0.024 as size of elements to be generated
Validate with *OK*



Click on ***View mesh*** to visualize the mesh

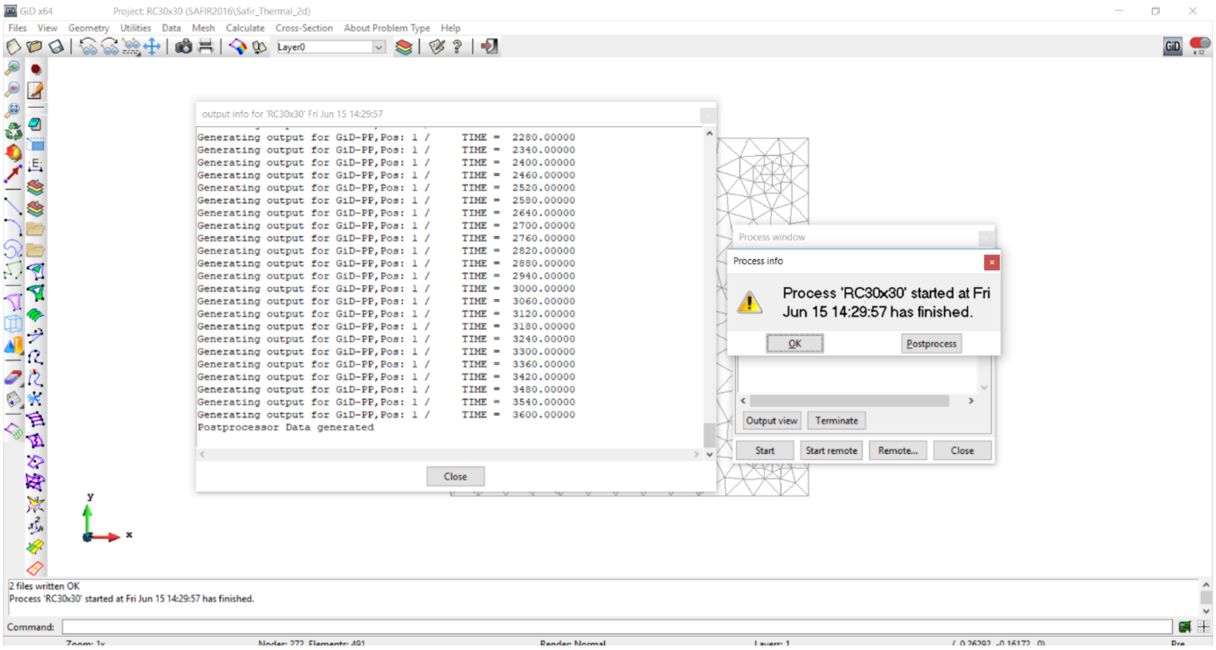


Note: the number of elements is limited here in order to be suitable for the demonstration version of SAFIR. For users of the full version, a smaller size of elements should be preferred for a reinforced concrete section.

9. 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.