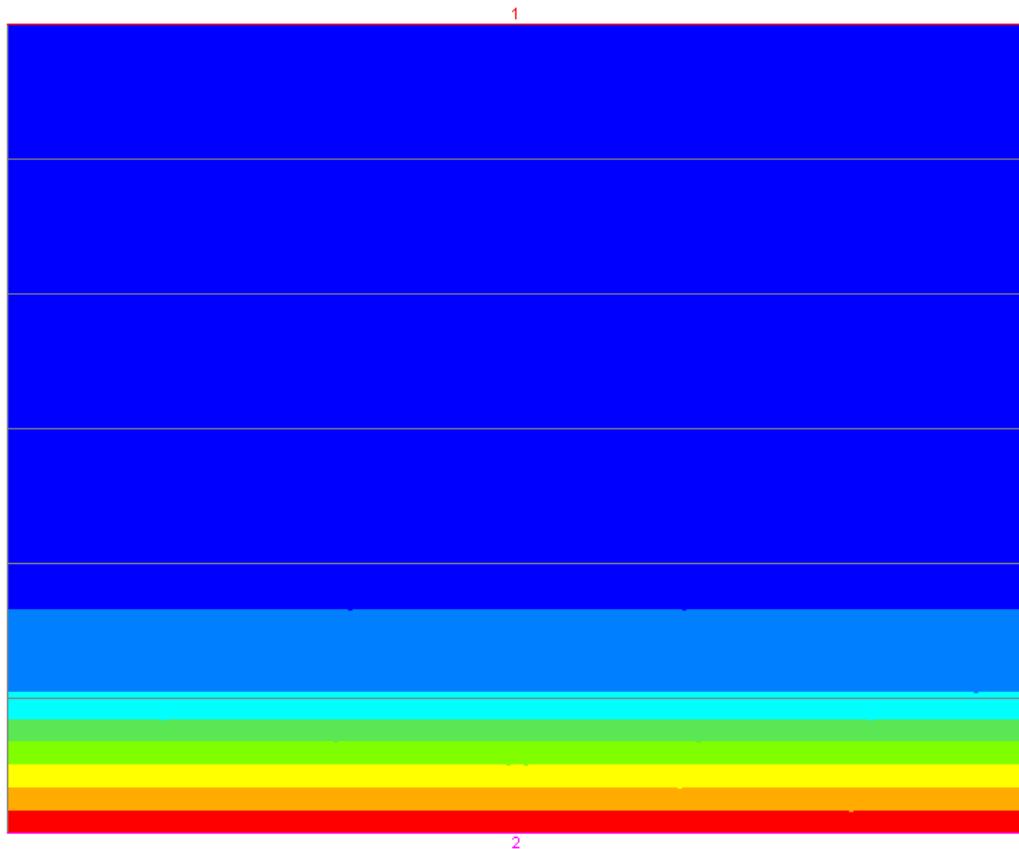


Example of GID-SAFIR

Thermal Analysis for 3D structure

Exercise n°4 – Slab in SHELL element

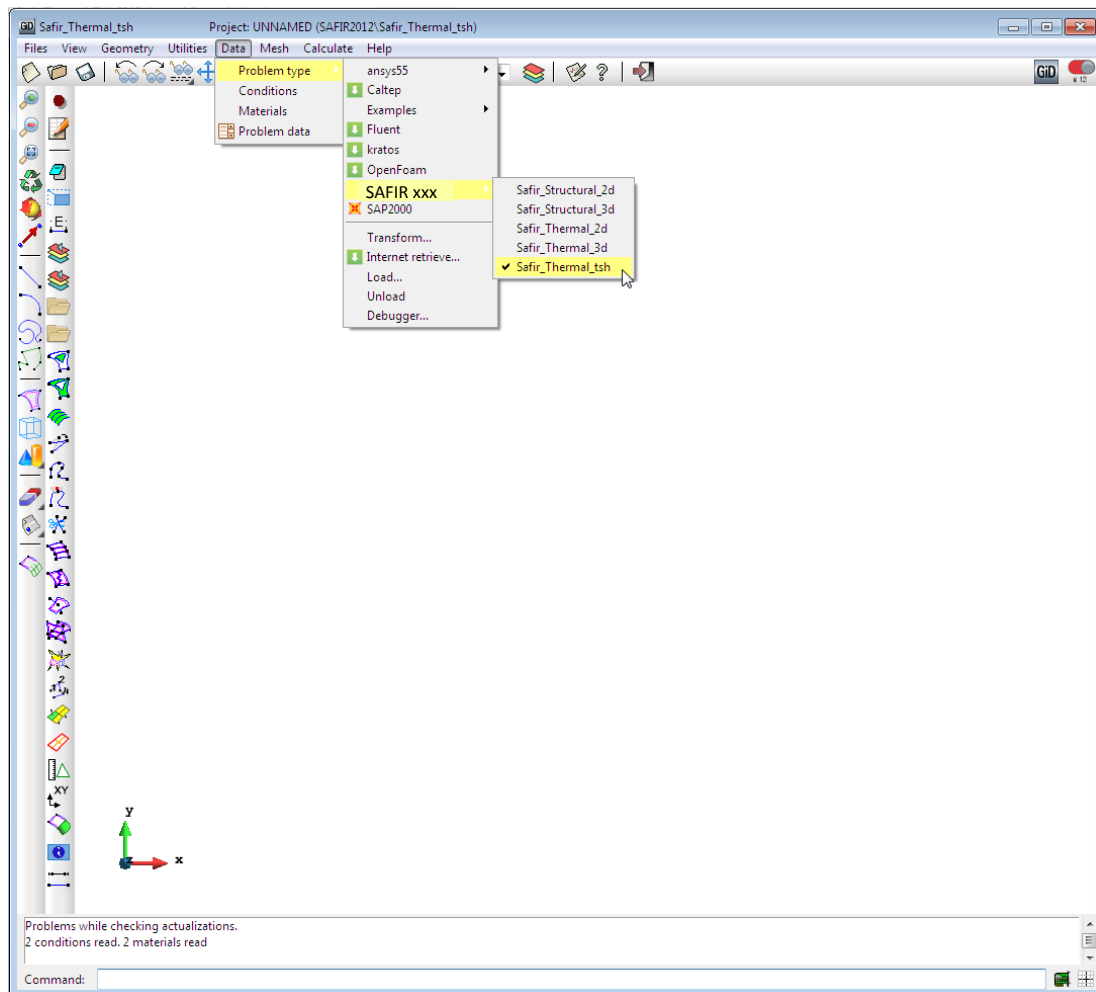


This example explains how to create a slab (thickness = 130mm) for a 3D calculation in shell elements.

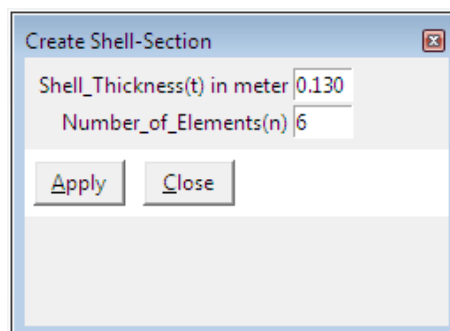
1. Create a project for the Thermal Analysis

From the pull down menu select:

➤ *Data->Problem type->SAFIRxxx->Safir_Thermal_TSH*



GiD will open automatically a new window where you have to give the parameters of the slab.

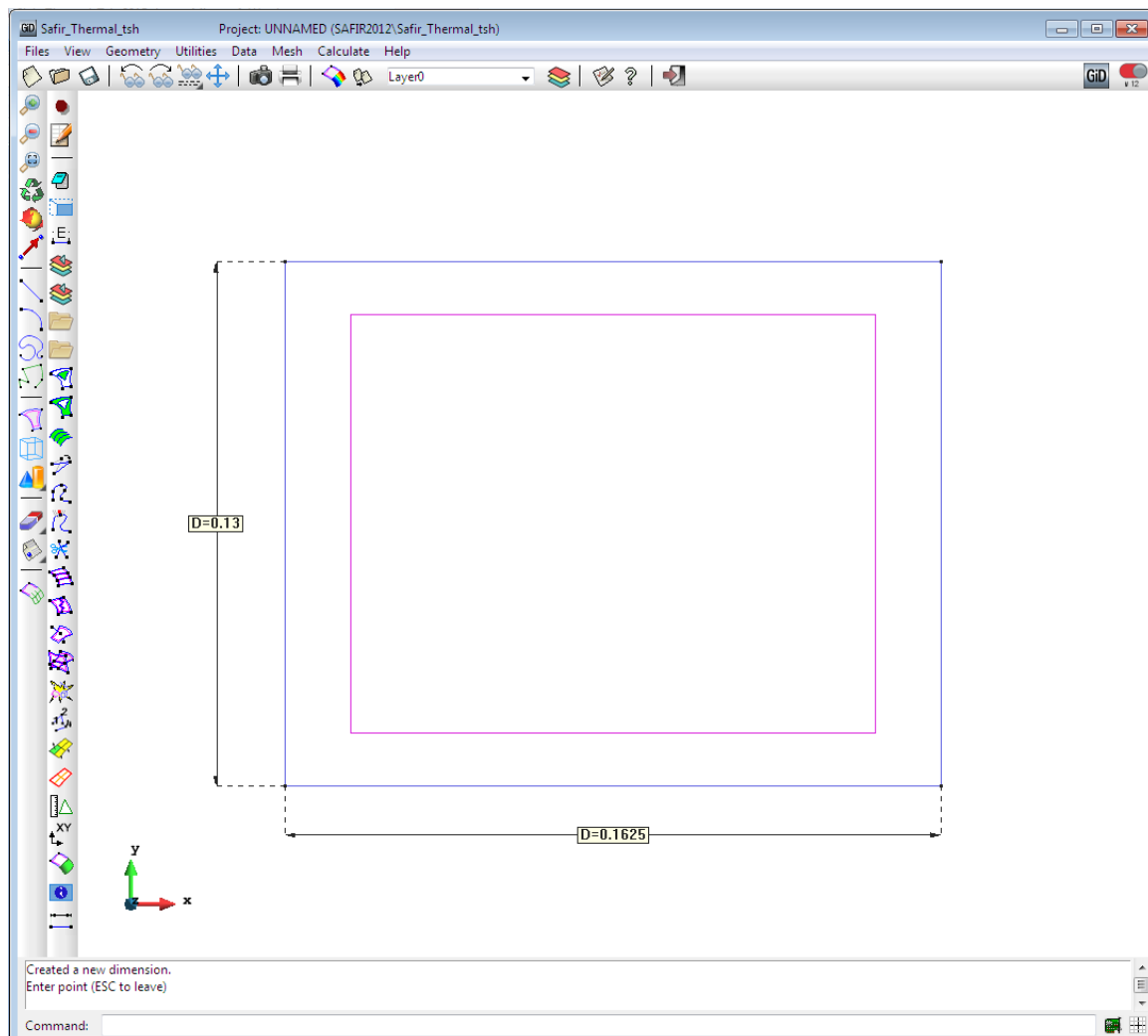


Put **0.130m** as *Shell thickness (t)* and **6** as *Number of Elements (n)* ($n=6$ mean that GiD will generate meshes by cutting the element height in 6 horizontal parts).

⚠ Give the thickness of the slab in **meter** !!

⚠ In case of a slab with a steel sheet deck, the thickness t should be calculated according to EN1994-1-2 (see Exercise 11).

GiD will create a section with a thickness $t = 0.130\text{m}$, and a width $= (5/4)t = 0.1625\text{m}$



GiD put the nodal axis of the slab at mid-height of the section thickness t ; SAFIR calculates the static values from $+(t/2)$ to $-(t/2)$.

To save the project select:

➤ **Files->Save**

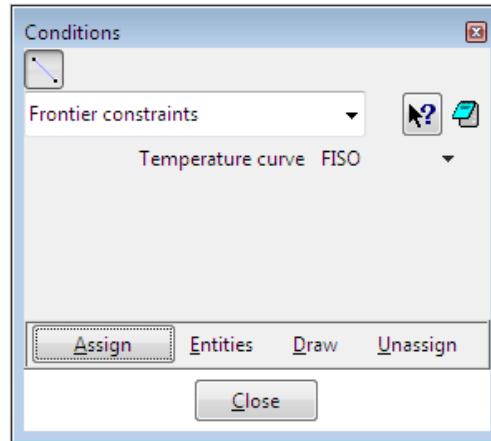


Then enter a file name, eg.: **Slab130** .

2. Assign a temperature curve

From the pull down menu select:

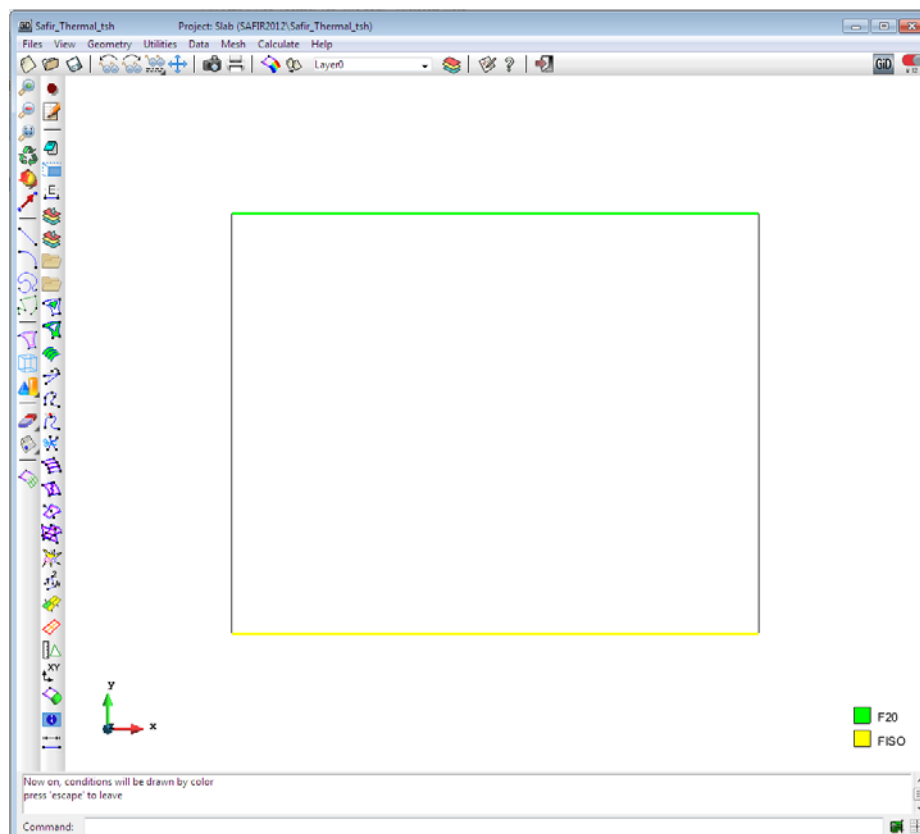
➤ **Data->Conditions**



Select **Frontier constraints** on the first pull down list.

Select **FISO** on the **Temperature curve** pull down list, then click on the **Assign** button and assign the FISO temperature curve to the lower part of the slab.

Do the same operation to assign the F20 temperature curve to the upper part of the slab as shown hereafter.



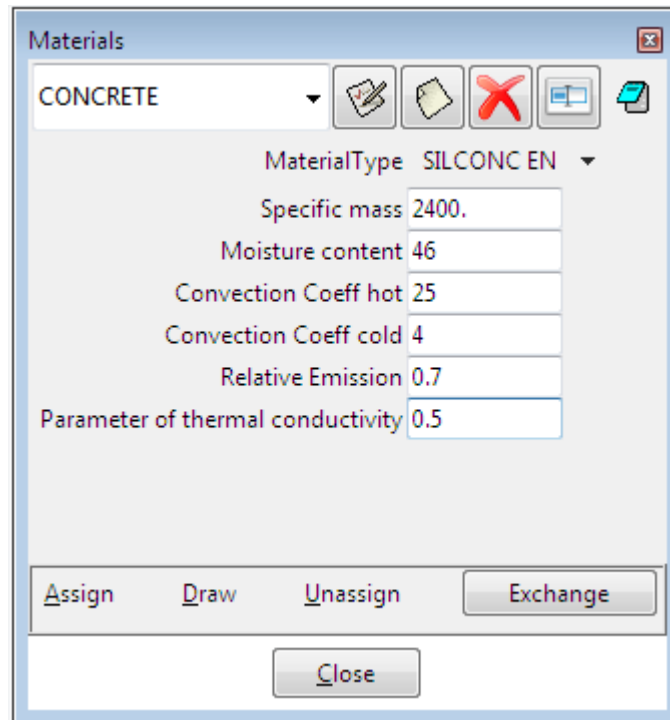
Click on **Finish** to confirm, then **Close**.

3. Assign Material

From the pull down menu select:

➤ *Data->Materials*

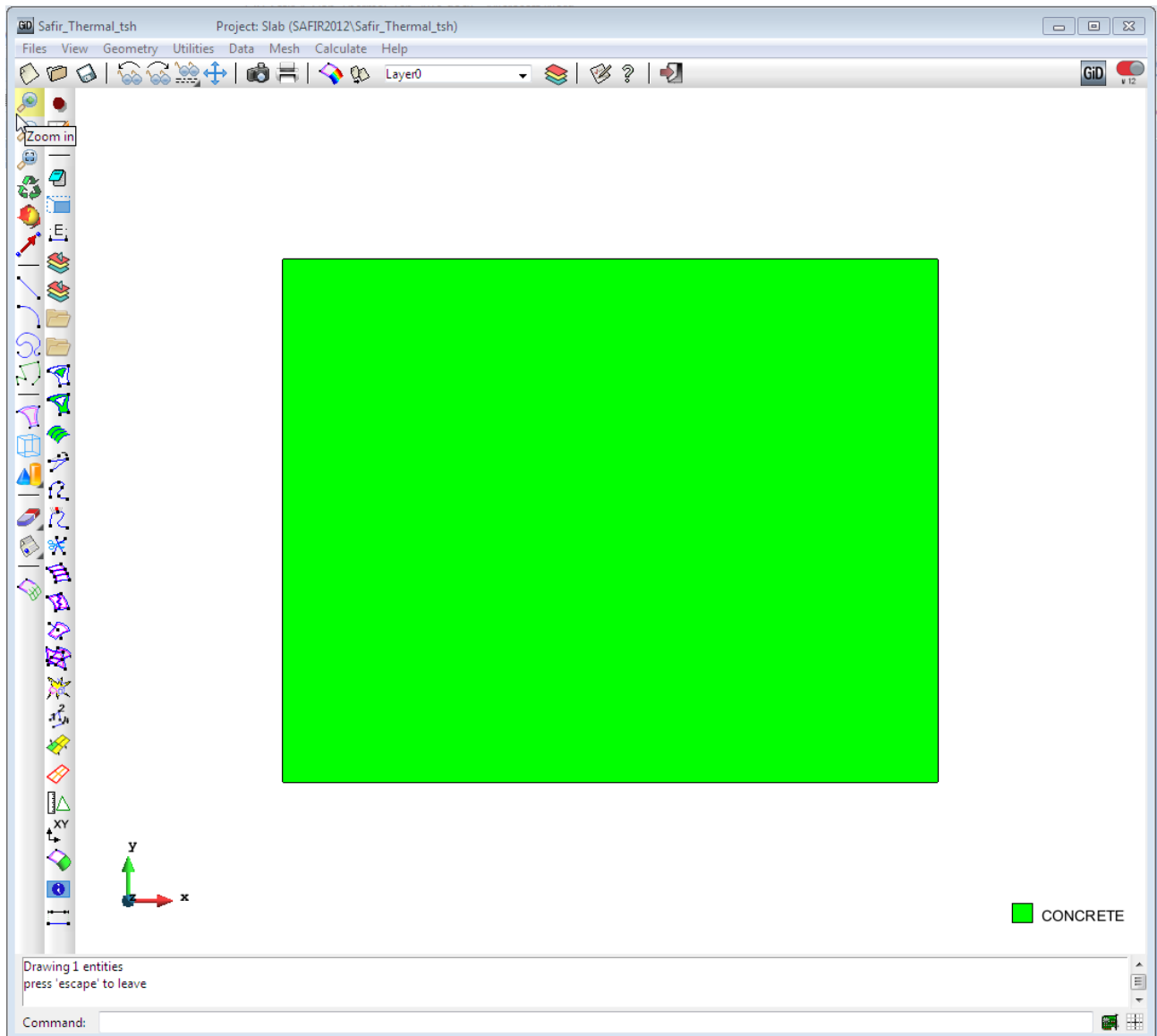
Select **CONCRETE** from the dialog box pull down list



Then select:

- **SILCONC EN** as Material Type
- A specific mass of **2400** [kg/m³]
- A moisture content of **46** [kg/m³]
- A Convection Coeff hot of **25** [W/m²K]
- A Convection Coeff cold of **4** [W/m²K]
- A Relative Emission of **0.7** [-]
- A Parameter of thermal conductivity of **0.5** [W/mK]

Click on **Assign-> Surfaces** and assign it to the surface, press **Finish** to confirm then **Close**.



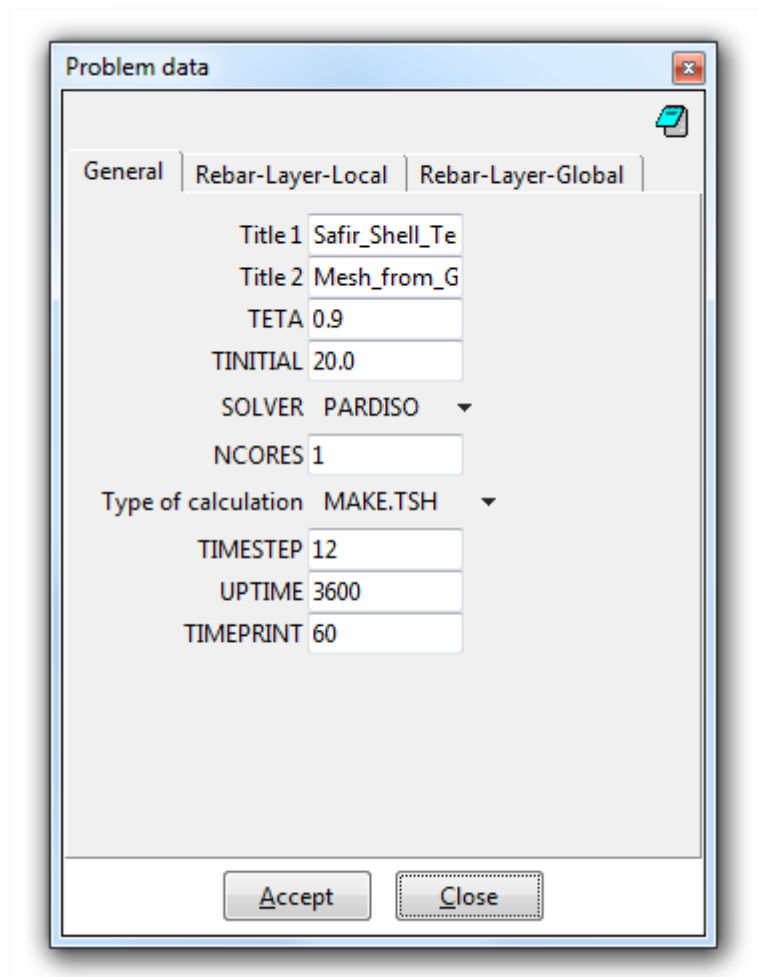
Select **DRAW->all materials** in the Material dialog box to display Materials

Press **Finish** then **Close** to leave.

4. Assign General Data

From the pull down menu select:

➤ *Data->Problem Data*



In the *Problem Data* dialog box, enter TIMESTEP, UPTIME, TIMEPRINT as needed.

a. In case of **Rebar-Layer-Local** rebars definition, fill as shown below:

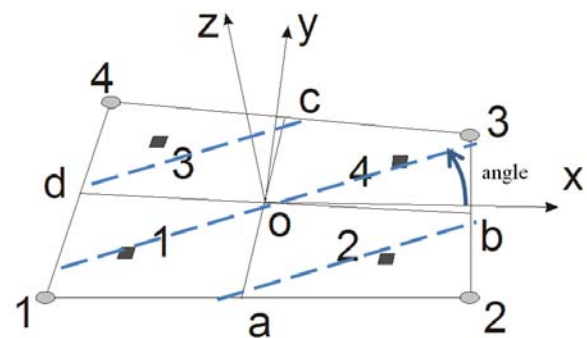
Problem data

General Rebar-Layer-Local Rebar-Layer-Global

Rebars-Local 4

Rebar-Local1 MATERIAL	2	Rebar layer // Y	Lower layer
Rebar-Local1 SECTION	393e-6		
Rebar-Local1 LEVEL	-0.035		
Rebar-Local1 ANGLE	90.0		
Rebar-Local2 MATERIAL	2	Rebar layer // X	Upper layer
Rebar-Local2 SECTION	393e-6		
Rebar-Local2 LEVEL	-0.035		
Rebar-Local2 ANGLE	0.0		
Rebar-Local3 MATERIAL	2	Rebar layer // Y	Upper layer
Rebar-Local3 SECTION	393e-6		
Rebar-Local3 LEVEL	0.035		
Rebar-Local3 ANGLE	90.0		
Rebar-Local4 MATERIAL	2	Rebar layer // X	Upper layer
Rebar-Local4 SECTION	393e-6		
Rebar-Local4 LEVEL	0.035		
Rebar-Local4 ANGLE	0.0		

Accept Close



Nodes

Points of integration

Click on the **Accept** data button.

- ⚠ **Rebars** is equal to the number of rebar layers in the section type
- ⚠ **Rebar x MATERIAL** is the local number of the layer x (rebars are not taken into account for the thermal analysis)
- ⚠ **Rebar x SECTION** is the cross sectional area of rebar x in the layer x [m²/m].
In this exercise : $1\phi 10/20\text{cm} = 5\phi 10/\text{m} = 5 \cdot \left(\frac{\pi \cdot 10^2}{4}\right) = 393 \cdot 10^{-6} \text{ m}^2/\text{m}$
- ⚠ **Rebar x LEVEL** is the position of the layer x with respect to the thickness symmetry axis
- ⚠ **Rebar x ANGLE** is the angle in degrees between the local X axis and the layer of rebars

b. In case of **Rebar-Layer-Global** rebars definition, fill as shown below:

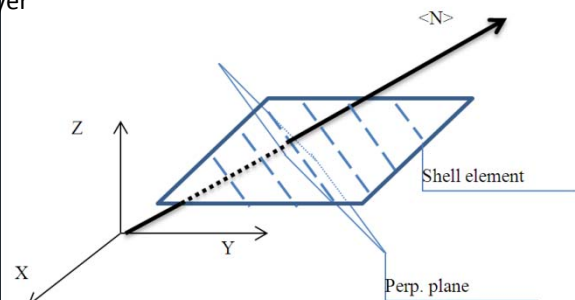
Problem data

General | Rebar-Layer-Local | **Rebar-Layer-Global**

Rebars-Global 4 ▾

Rebar-Global1 MATERIAL	2	} Rebar layer // Y	} Lower layer
Rebar-Global1 SECTION	393e-6		
Rebar-Global1 LEVEL	-0.035		
Rebar-Global1 N1	1		
Rebar-Global1 N2	0	} Rebar layer // X	
Rebar-Global1 N3	0		
Rebar-Global2 MATERIAL	2		
Rebar-Global2 SECTION	393e-6		
Rebar-Global2 LEVEL	-0.035	} Rebar layer // Y	} Upper layer
Rebar-Global2 N1	0		
Rebar-Global2 N2	1		
Rebar-Global2 N3	0		
Rebar-Global3 MATERIAL	2	} Rebar layer // X	
Rebar-Global3 SECTION	393e-6		
Rebar-Global3 LEVEL	0.035		
Rebar-Global3 N1	1		
Rebar-Global3 N2	0	} Rebar layer // Y	
Rebar-Global3 N3	0		
Rebar-Global4 MATERIAL	2		
Rebar-Global4 SECTION	393e-6		
Rebar-Global4 LEVEL	0.035	} Rebar layer // X	
Rebar-Global4 N1	0		
Rebar-Global4 N2	1		
Rebar-Global4 N3	0		



Accept Close



Click on the **Accept** data button.

- ⚠ **Rebars-Global** is the number of rebar layers in the section type
- ⚠ **Rebar x MATERIAL** is the local number of the layer x (rebars are not taken into account for the thermal analysis)
- ⚠ **Rebar x SECTION** is the cross sectional area of rebar x in the layer x [m^2/m].

$$\text{In this exercise : } 1\phi 10/20\text{cm} = 5\phi 10/\text{m} = 5 \cdot \left(\frac{\pi \cdot 10^2}{4} \right) = 393 \cdot 10^{-6} \text{ m}^2/\text{m}$$

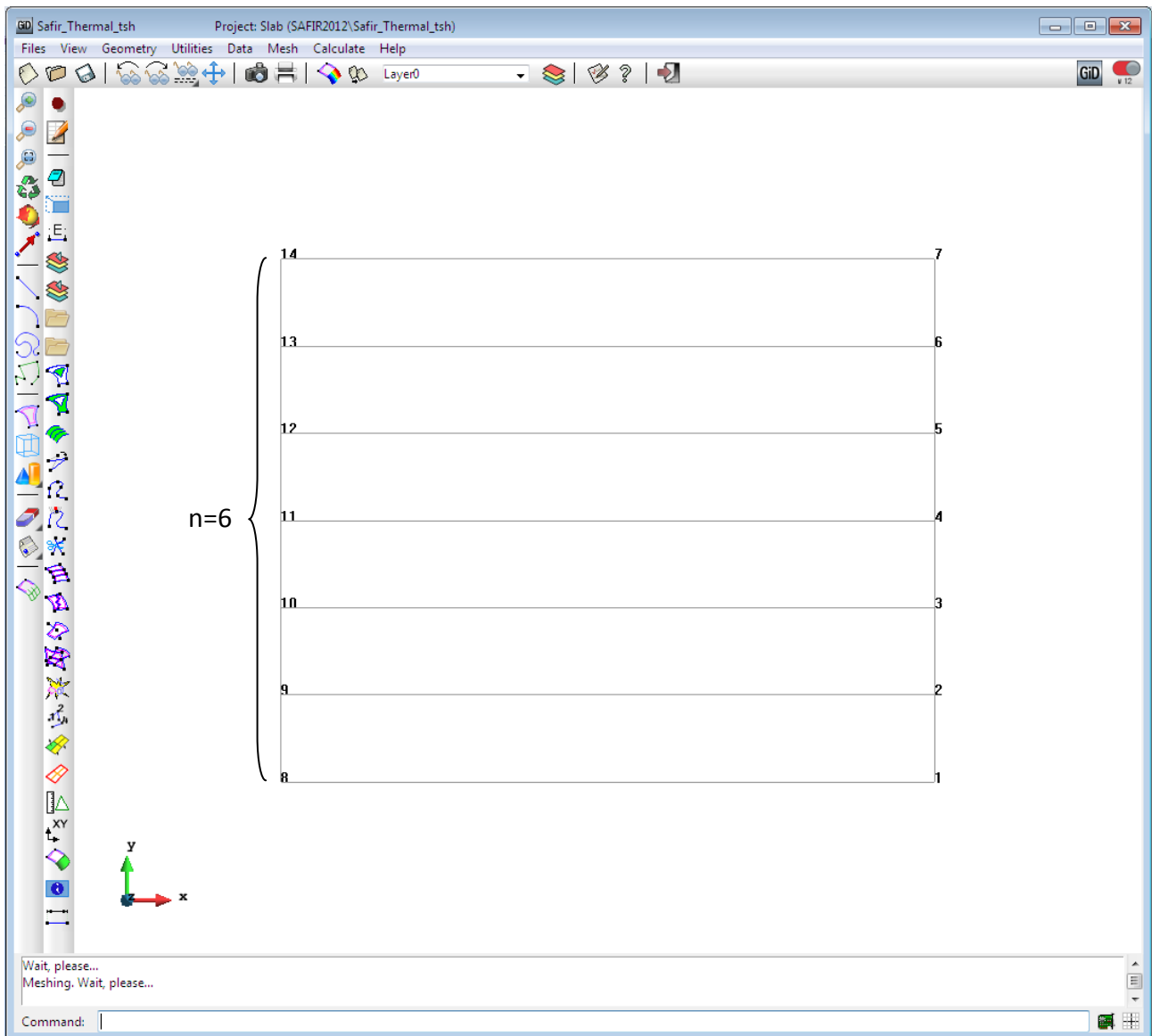
-  **Rebar x LEVEL** *is the position of the layer x with respect to the thickness symmetry axis*
-  **Rebar x N1,N2,N3** is a vector in the global system of coordinates of the structure. The norm of the vector does not have to be 1. This vector is used to define the position of the bar layers in the shell elements with respect to the global system of coordinates. The bars have the orientation of the line which is the intersection between the shell element and a plane that is perpendicular to the normal. 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).

5. Meshes

To create meshes select from the pull down menu:

 **Mesh->Generate mesh**

or use [Ctrl + g]



6. Start the calculation

From the pull down menu select:

► *Calculate->Calculate window*

Click the **Start** button.

