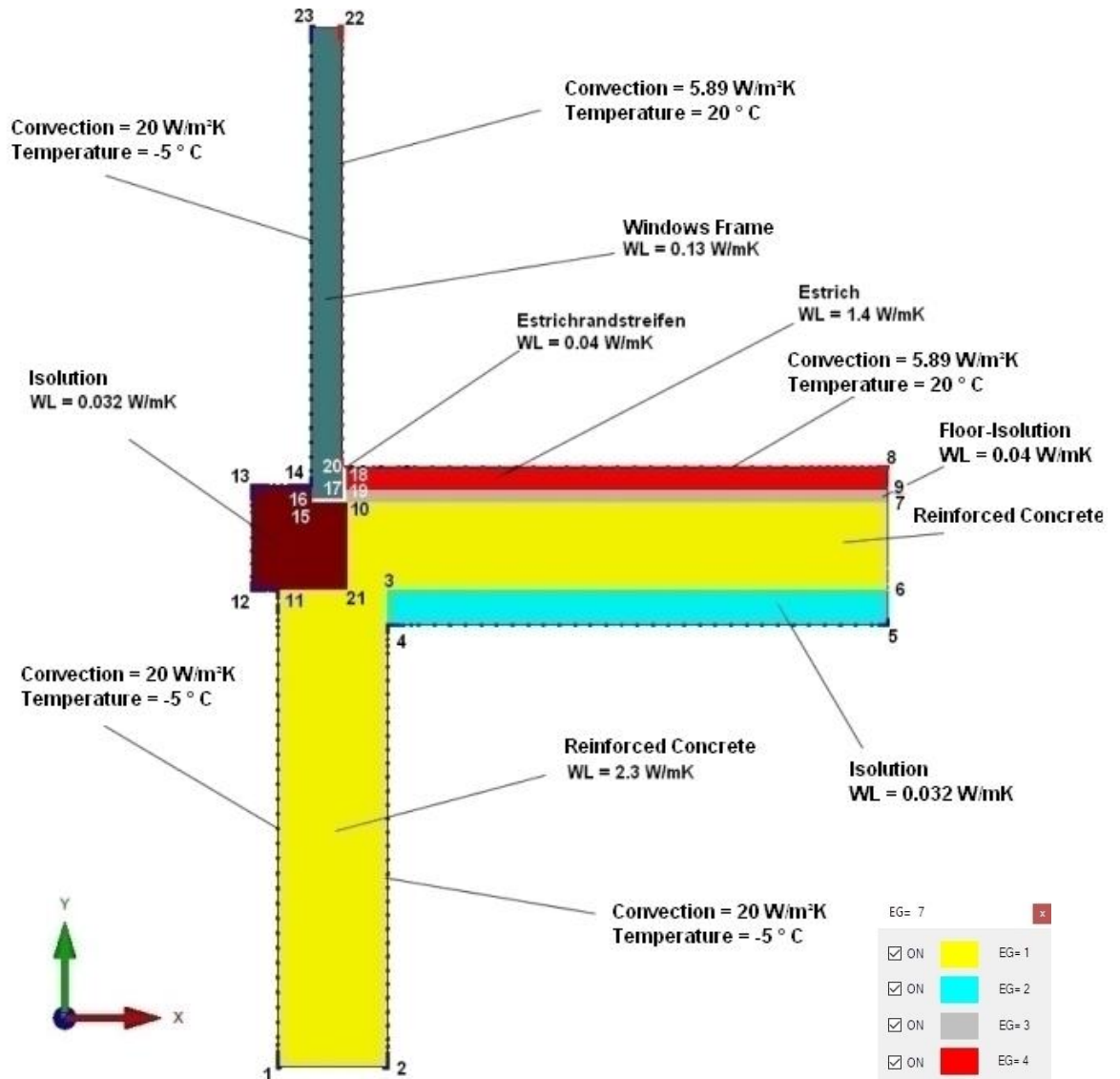
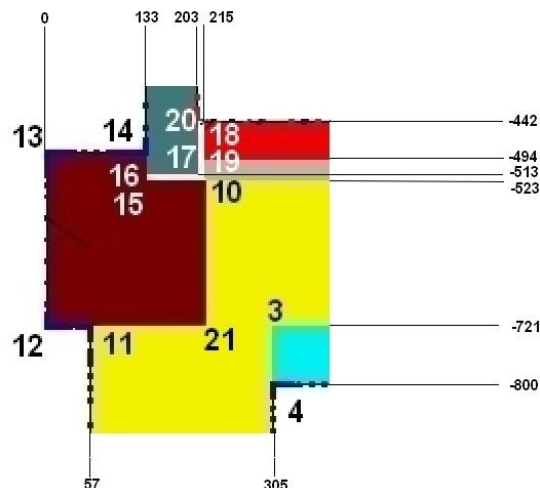


## Part 31: Temperature Distribution and Heat Flux of a Windows Profile

The following window profile from „Porenbeton-Wärmebrückenkatolog“ Side 908 is recalculated with the following element groups, thermal conductivities and convections:



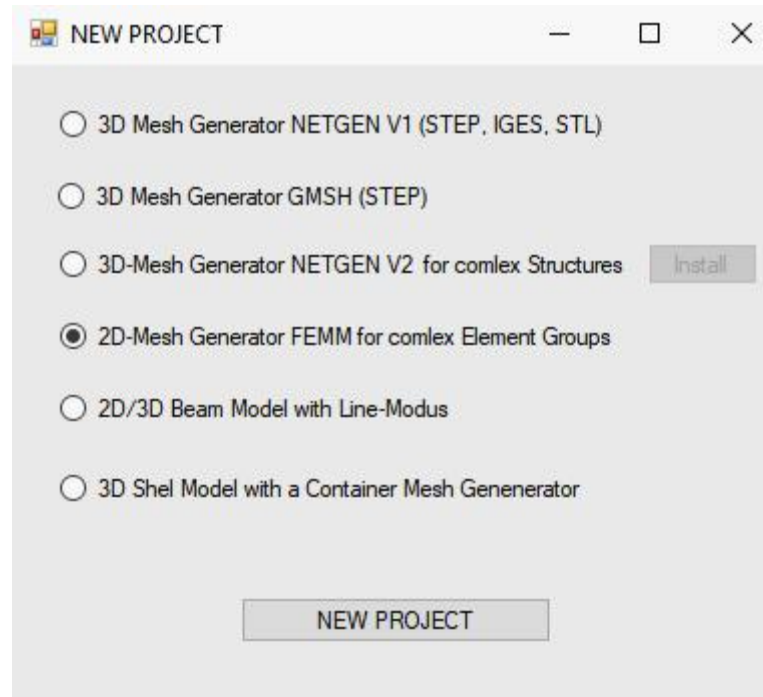
XY-Zoom:



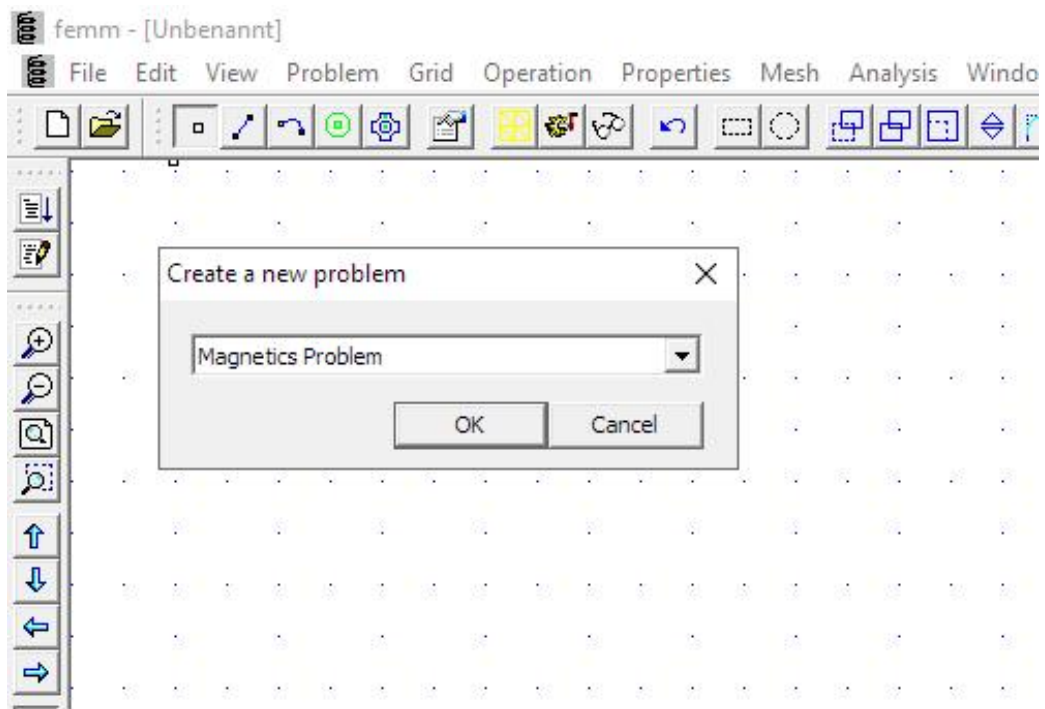


Now start the FEM system MEANS V12 via the desktop icon

Select the "File" tab and the "New" menu and select "2D mesh generator FEMM for complex element groups" in the project menu to create a 2D FEM model with the 2D mesh generator FEMM.

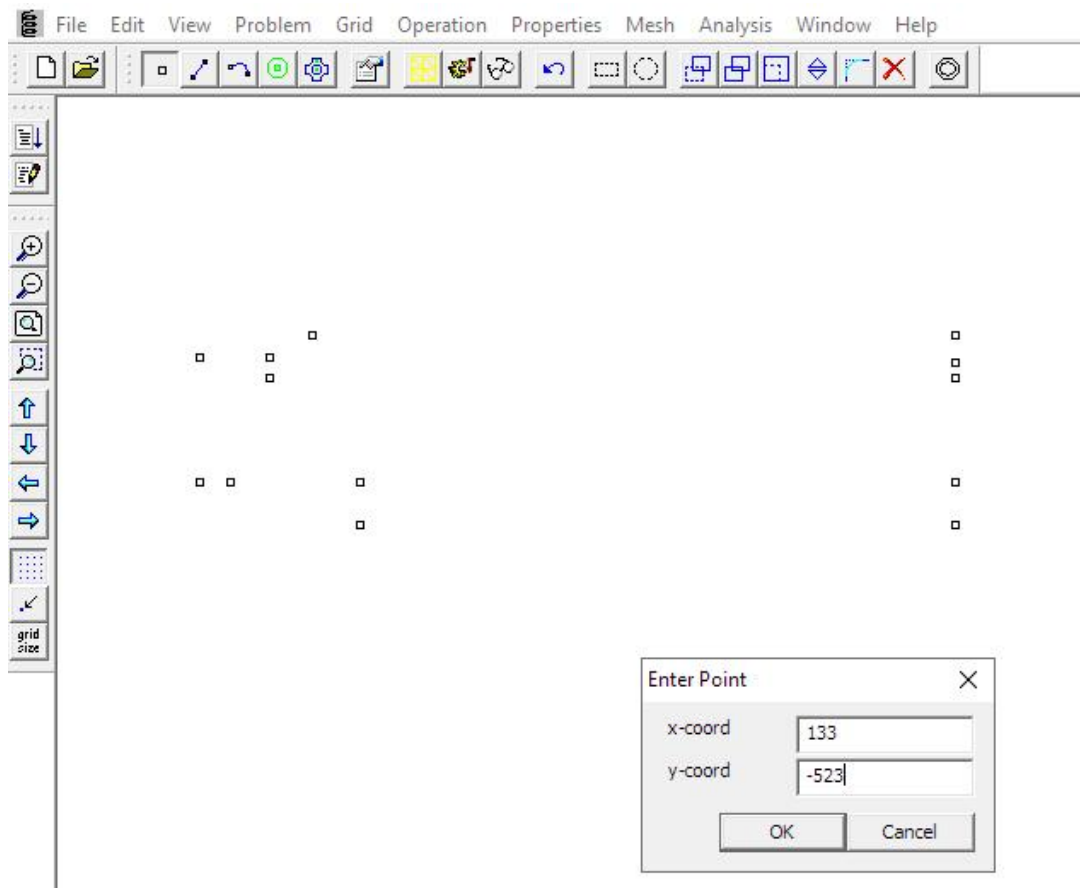


The FEMM user interface now appears. Choose menu "File" and "New" and create a new project with "Magnetics Problem".



## TAB key

With the Tab key and the icon, all 23 corner nodes can now be entered one after the other in a dialog box. The coordinates are entered in millimeters but must later be converted to meters in MEANS to match the thermal conductivities.



## DXF import

The nodes can also be created first with AutoCAD or MEANS and imported into FEMM via the DXF interface. Here, however, care must be taken that no overlapping or double nodes or lines are created.

## Edit FEM file with editor

If you load the FEM file into any text editor, e.g. Notepad, you can quickly edit or delete the node coordinates or lines.

For a correction with the editor you should save the FEM file and exit FEMM. Then edit the FEM file with the editor and start FEMM again and reload the FEM file.

[NumPoints] = 23

```
57   -1800 0    0
305  -1800 0    0
305  -721  0    0
305  -800  0    0
1435 -800  0    0
```

```

1435 -721 0 0
1435 -523 0 0
1435 -442 0 0
1435 -494 0 0
215 -523 0 0
57 -721 0 0
0 -721 0 0
0 -485 0 0
133 -485 0 0
133 -523 0 0
133 -513 0 0
203 -513 0 0
215 -442 0 0
215 -494 0 0
203 -442 0 0
215 -721 0 0
203 552 0 0
133 552 0 0

```


[NumSegments] = 0

[NumArcSegments] = 0

[NumHoles] = 0

[NumBlockLabels] = 0

### Definition of the 7 element groups

After all 23 nodes have been entered, switch to the icon  to define the lines for the 7 element groups by clicking on the following pairs of nodes one after the other.

#### Elementgruppe 1

1 - 2, 2 - 3, 3 - 6, 6 - 7, 7 - 10, 10 - 21, 21 - 11, 11 - 1

#### Elementgruppe 2

4 - 5, 5 - 6, 6 - 3, 3 - 4

#### Elementgruppe 3

10 - 7, 7 - 9, 9 - 19, 19 - 10

#### Elementgruppe 4

19 - 9, 9 - 8, 8 - 18, 18 - 19

#### Elementgruppe 5

12 - 21, 21 - 10, 10 - 15, 15 - 14, 14 - 13, 13 - 12

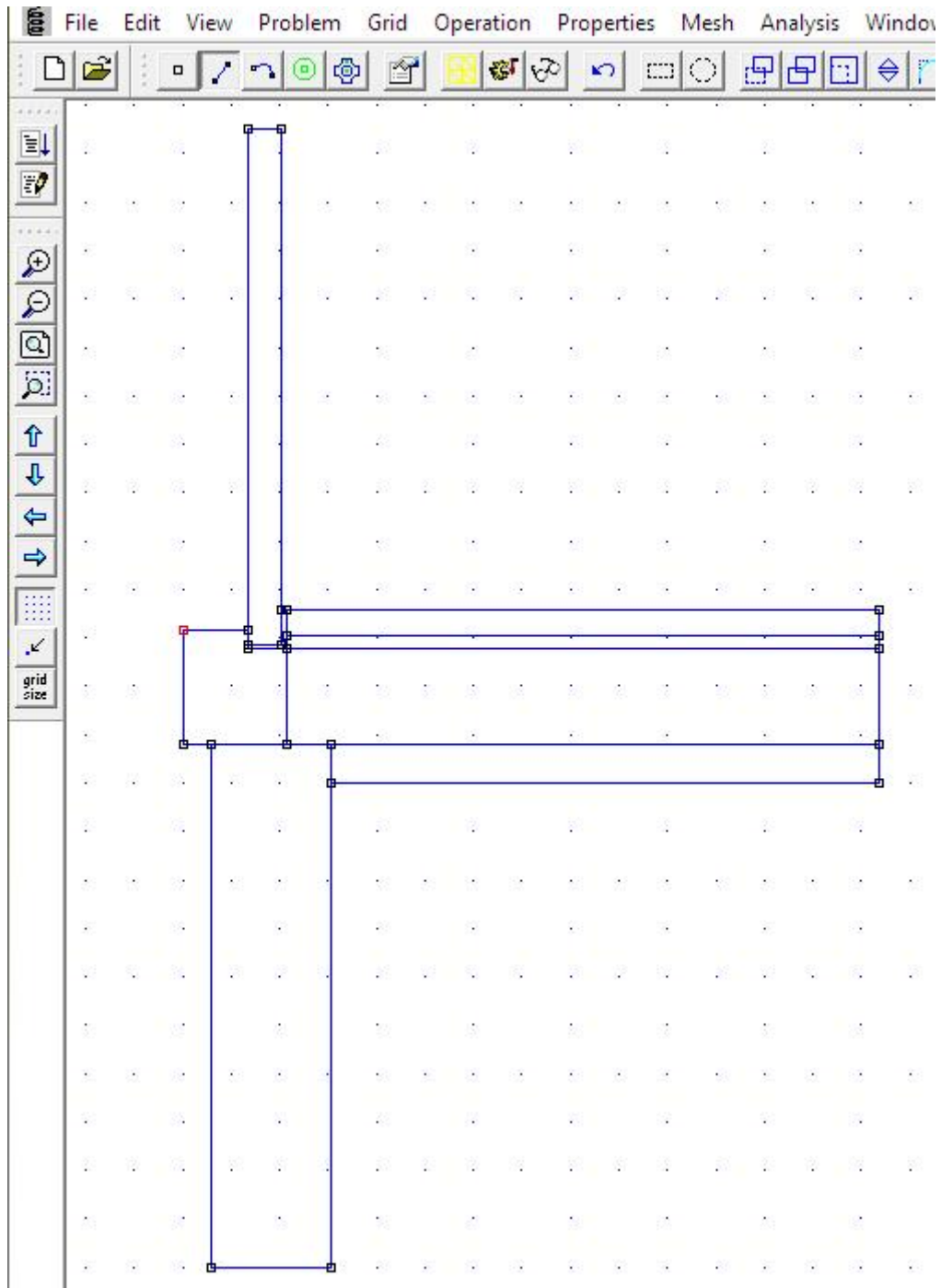
#### Elementgruppe 6

16 - 17, 17 - 22, 22 - 23, 23 - 16

#### Elementgruppe 7

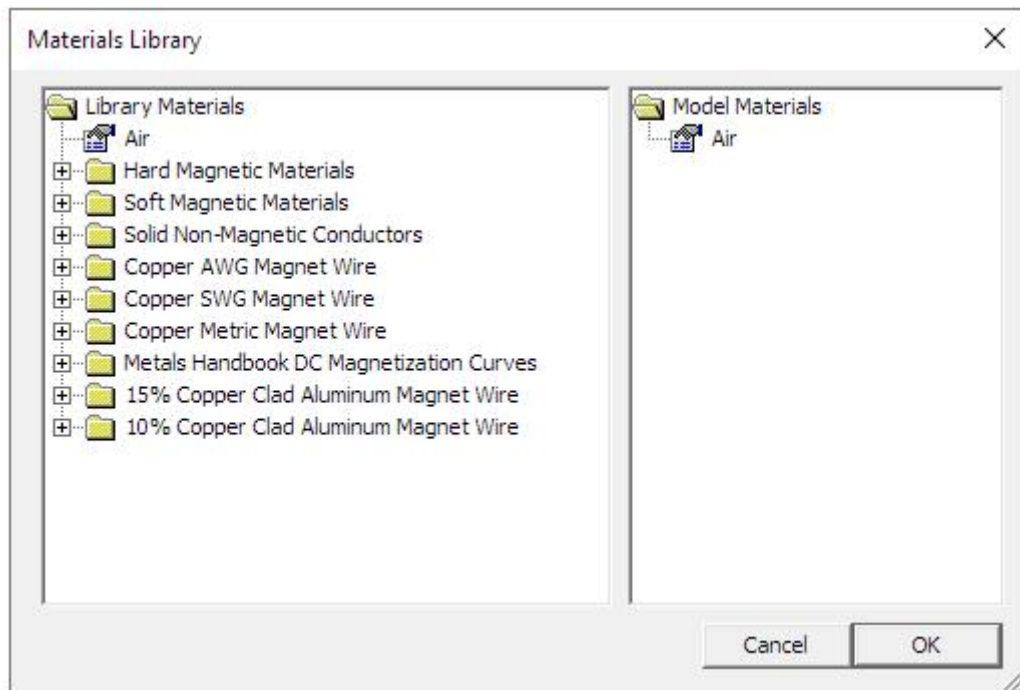
15 - 10, 10 - 18, 18 - 20, 20 - 17, 17 - 16, 16 - 15







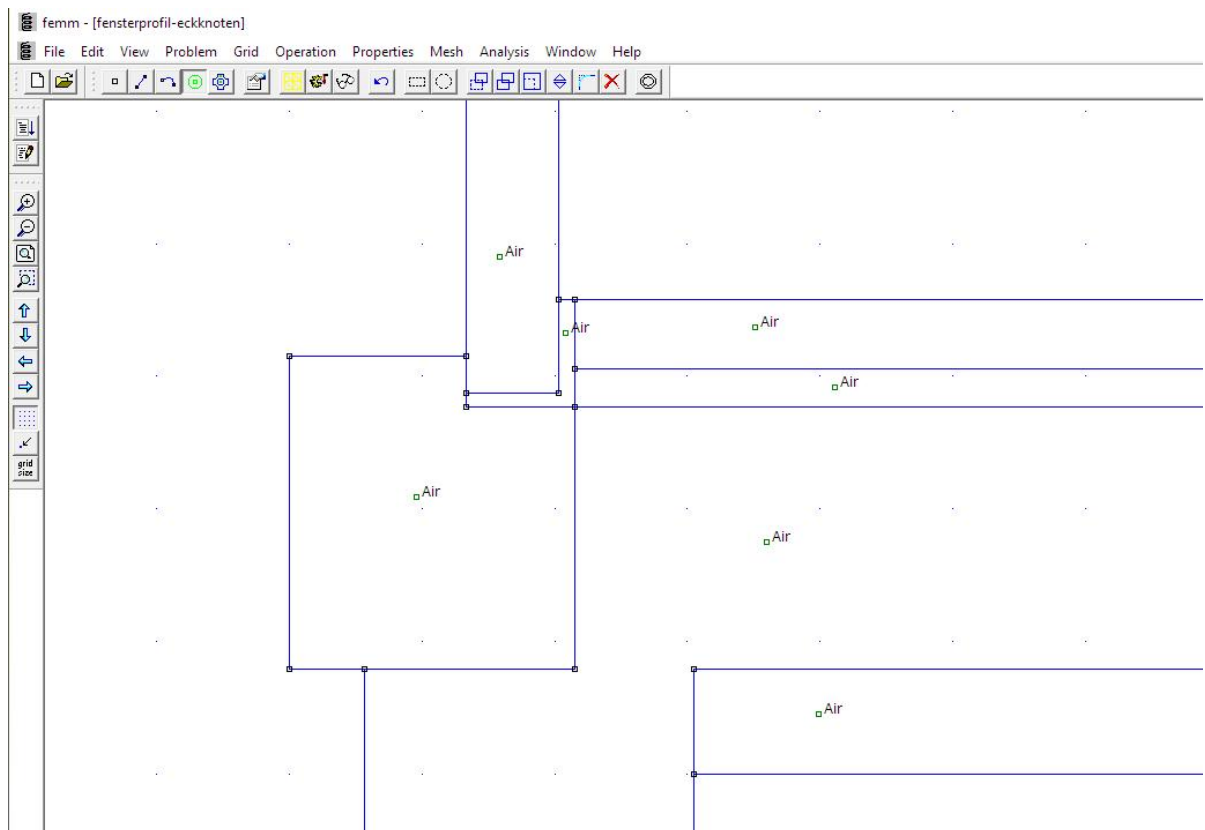
## Define Materials

Only the FEM mesh is required to be saved to an ANS file, but the FEM mesh must first be generated and an analysis performed. To do this, simply select the material "Air" for analysis from the "Properties/Material Library" menu.



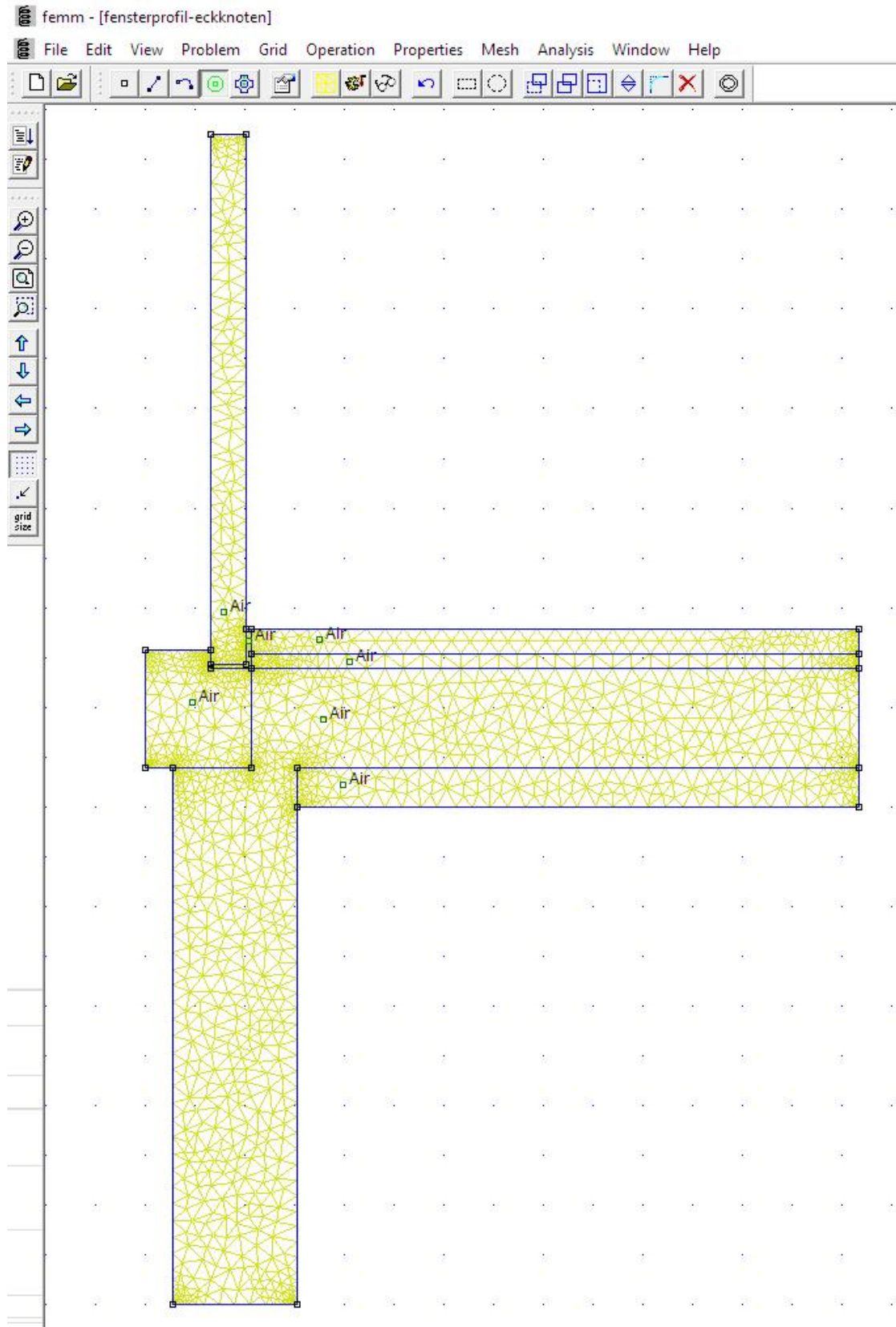
## Move Materials

The material must be moved in each element group with the icon  by first placing the material with the left mouse button and activating it red with the right mouse button. The material "Air" can now be selected with the icon .



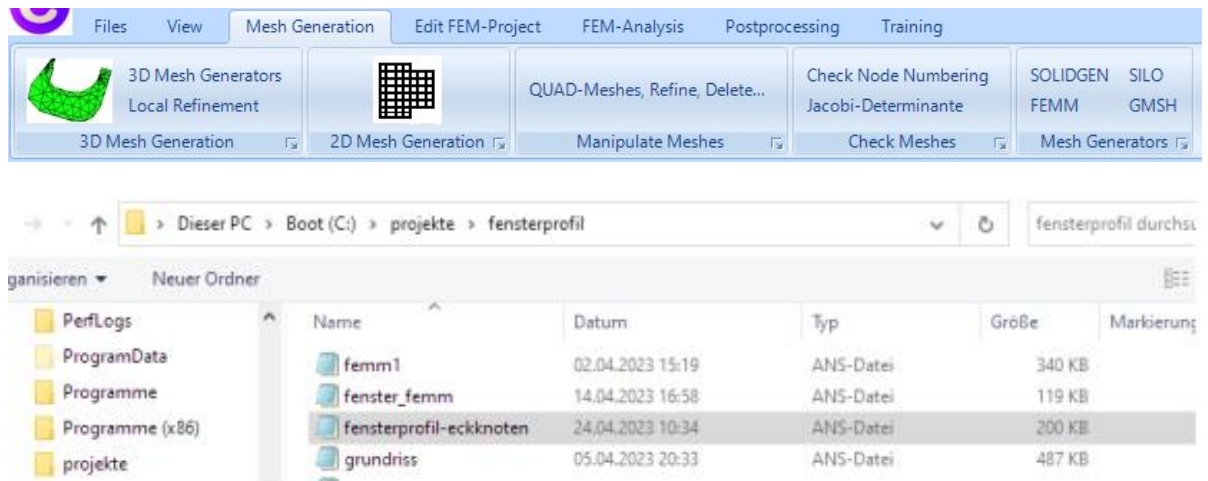
## Analysis in FEMM

Finally, the FEM mesh is generated with the icon and the analysis is carried out with the icon. If no error message is displayed, there is now a new file with the extension .ANS in the project directory, which can be imported and processed with MEANS. FEMM can be closed and is no longer needed.

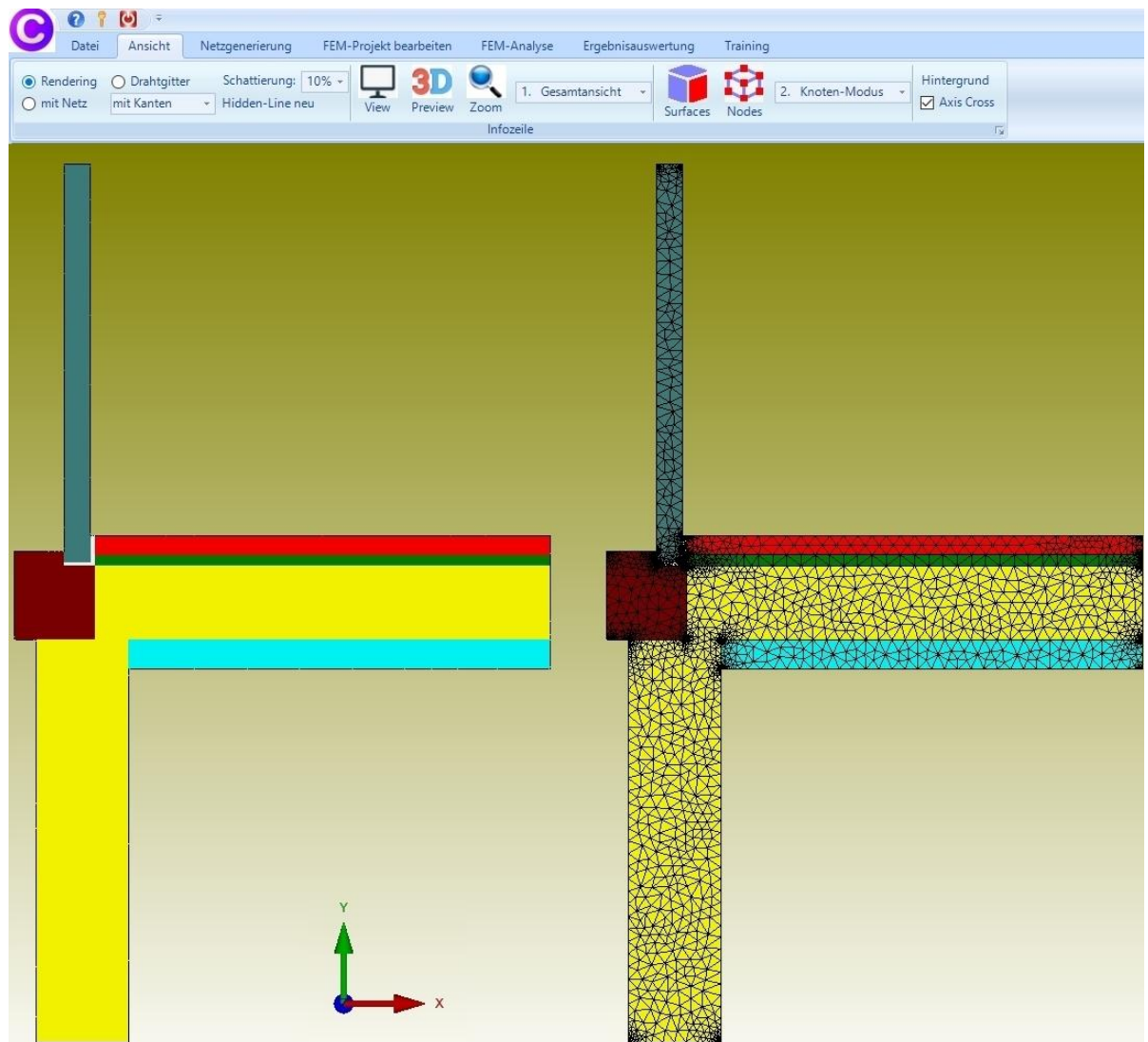


## FEMM-Interface

In the MEANS user interface, select the "Mesh Generation" tab and the right button "FEMM" to import the ANS file.



A 2D model consisting of 4046 TRI3S, 2232 nodes and 7 element groups is created.





## Enter the Thermal Conductivities

The Thermal Conductivities in W/mK for all 7 element groups are entered with the "Edit FEM-Project" tab and the "Material-Datas" menu:

Element group 1 - Reinforced Concrete WL = 2.3 W/mK

Element group 2 - Isulation-WL = 0.032 W/mK

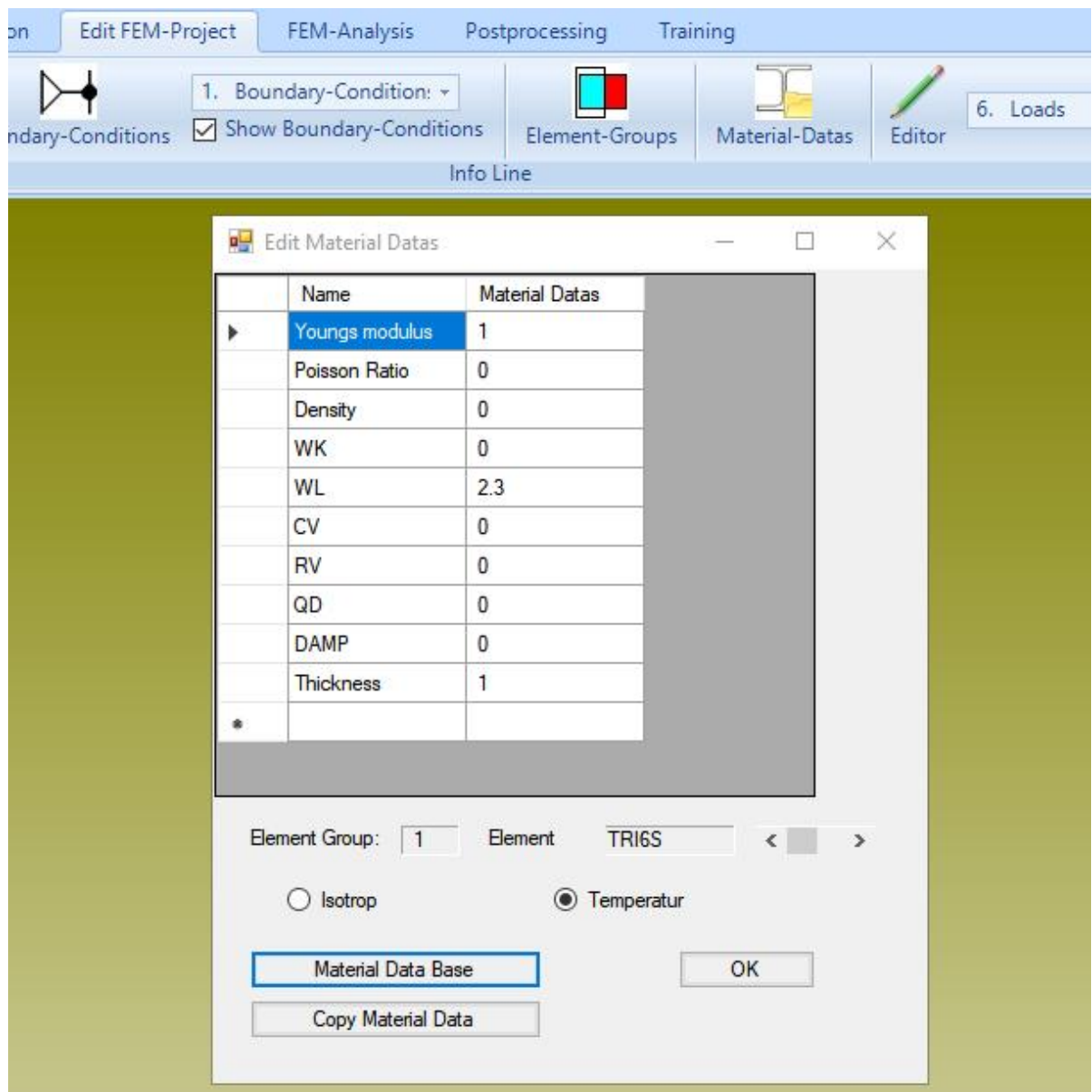
Element group 3 - Floor-Isulation-WL = 0.04 W/mK

Element group 4 - Estrich WL = 2.3 W/mK

Element group 5 - Window Frame WL = 0.13 W/mK

Element group 6 - Estrich Strip WL = 0.04 W/mK

Element group 7 - Isulation-WL = 0.032 W/mK



## Entering the Convections

The window profile is loaded with an outside and inside convection.

### Outside Convection

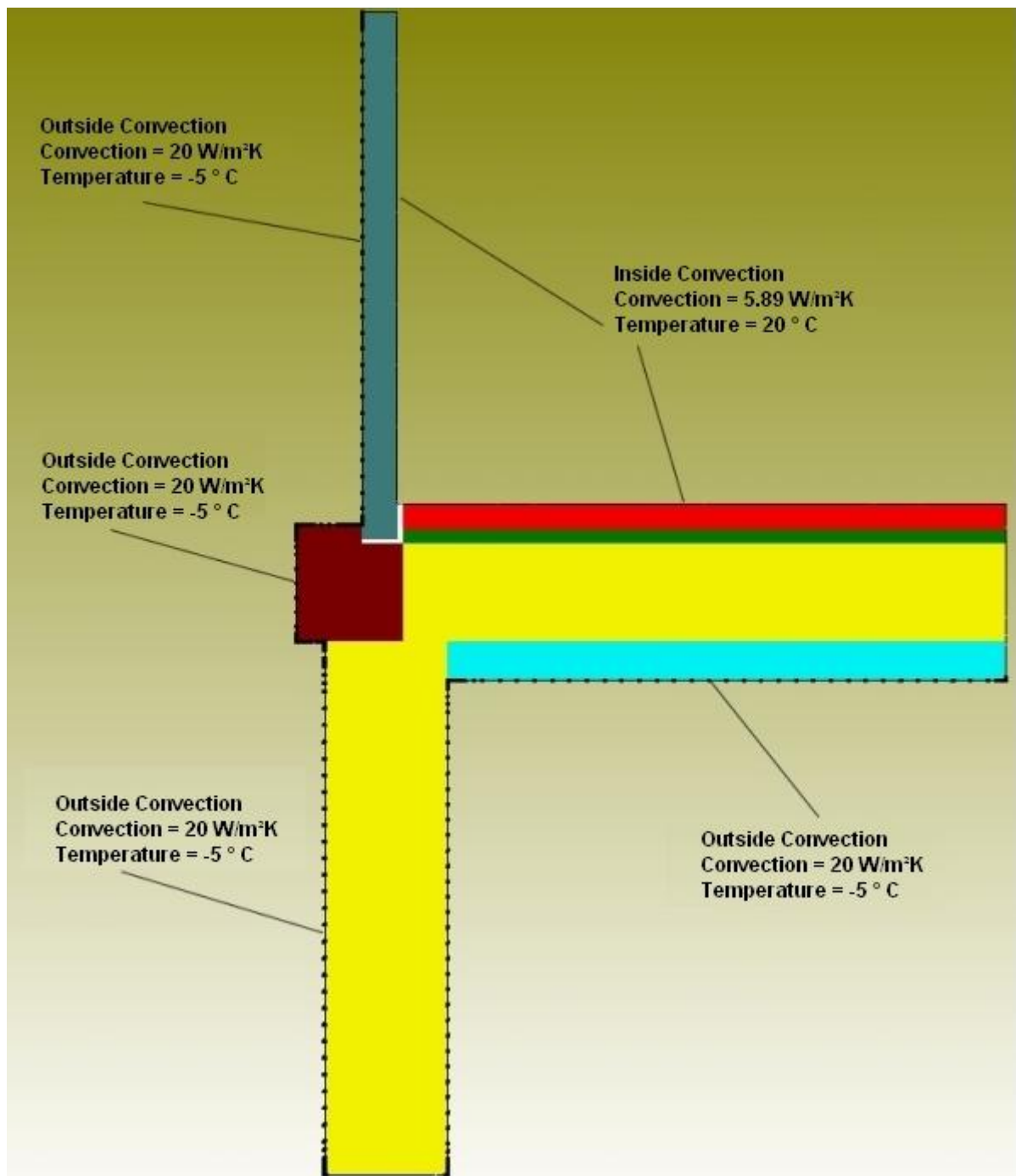
Convection =  $20 \text{ W/m}^2\text{K}$

Temperature =  $-5^\circ\text{C}$

### Inside Convection

Convection =  $5.89 \text{ W/m}^2\text{K}$

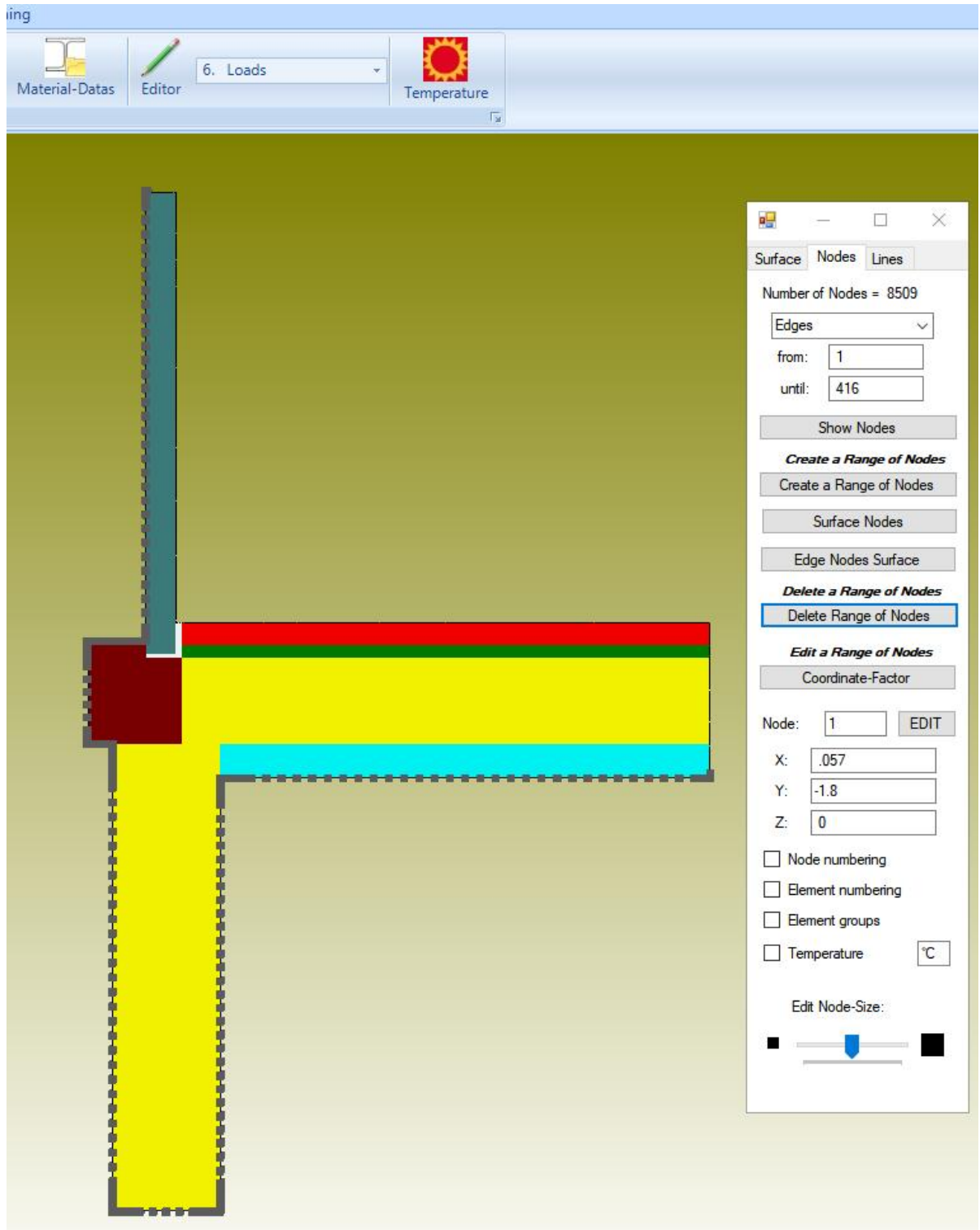
Temperature =  $20^\circ\text{C}$




## Create a Range of Nodes

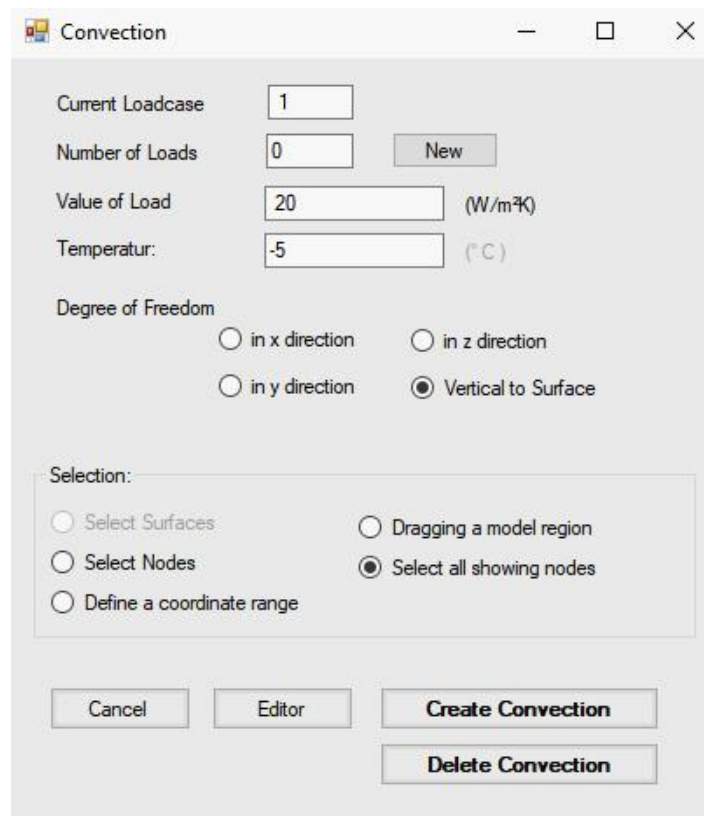
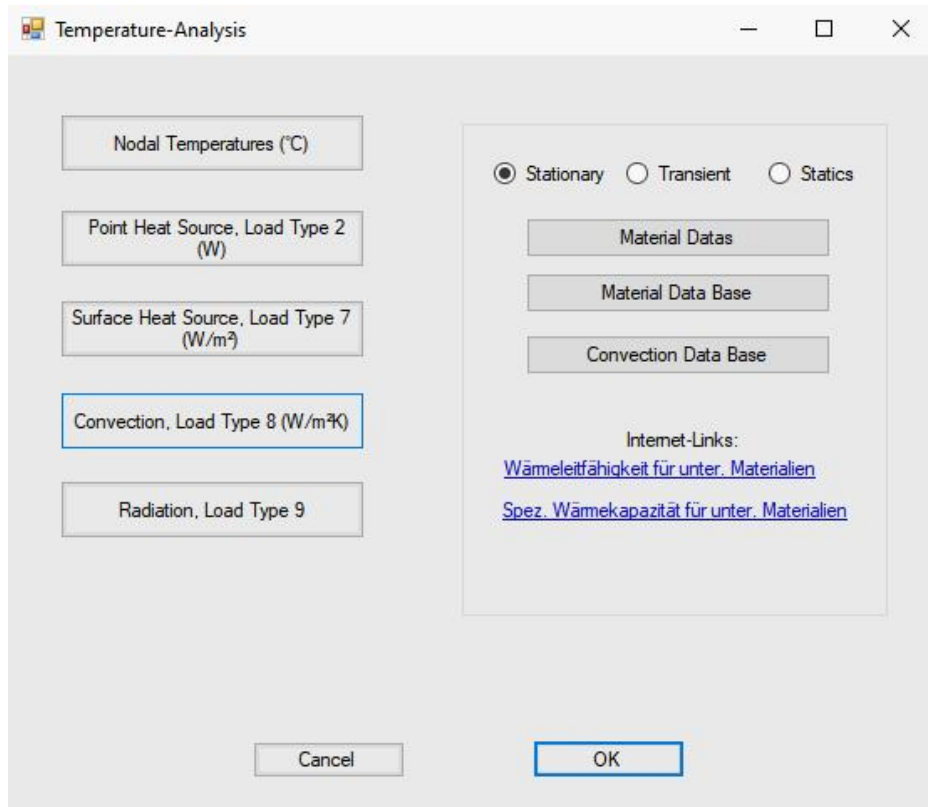
Select the "View" and "Node-Modus" tabs and first create all edge nodes with "Edges" and "Show nodes".

Select "Delete Range of Nodes" and delete all nodes of the inner area for the Outside Convection with a rectangle and all nodes of the outer area for the Inside Convection.



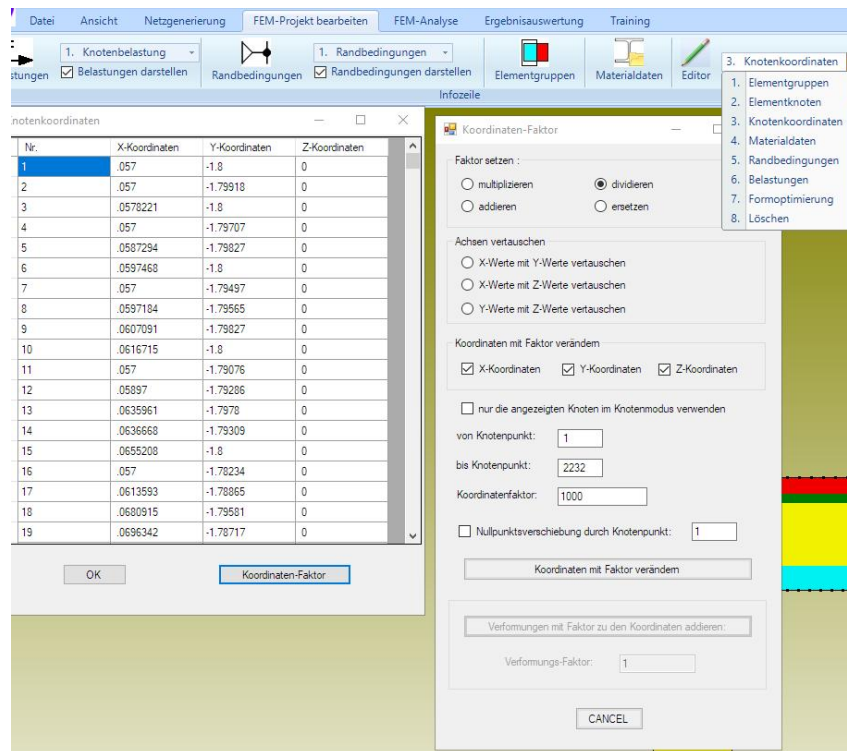


Use the temperature icon  from the "Edit FEM project" tab to generate the two convections with the "Convection, load type 8 (W/m<sup>2</sup>K)" menu with the selection "Select all showing nodes" in load case 1.



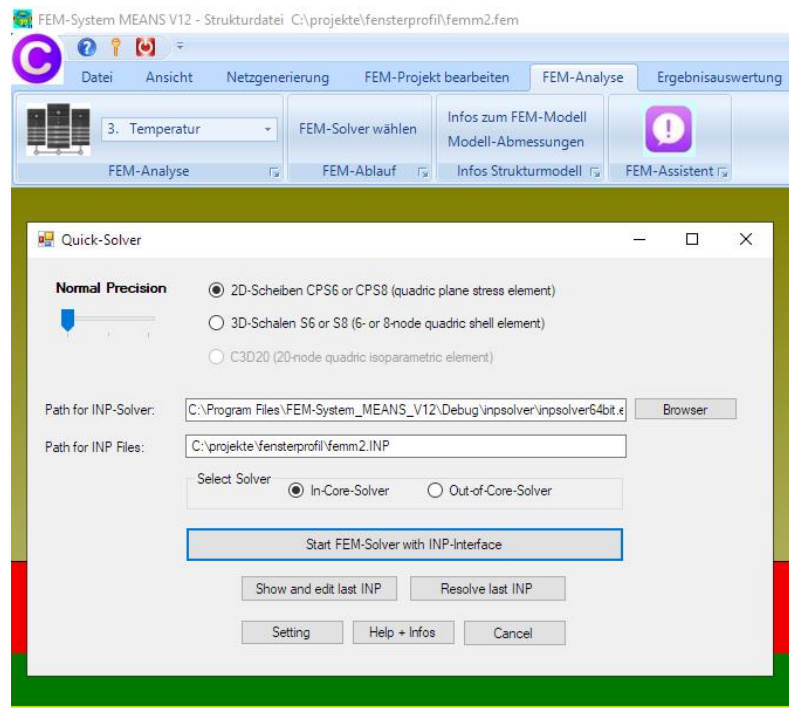
## Change node coordinates from millimeters to meters

WLs and convection were entered in meters but the coordinates in millimeters, so the coordinates must now be divided by "1000" using the "Edit FEM project" and "Editor" and "Node coordinates" tabs.



## Temperature Analysis

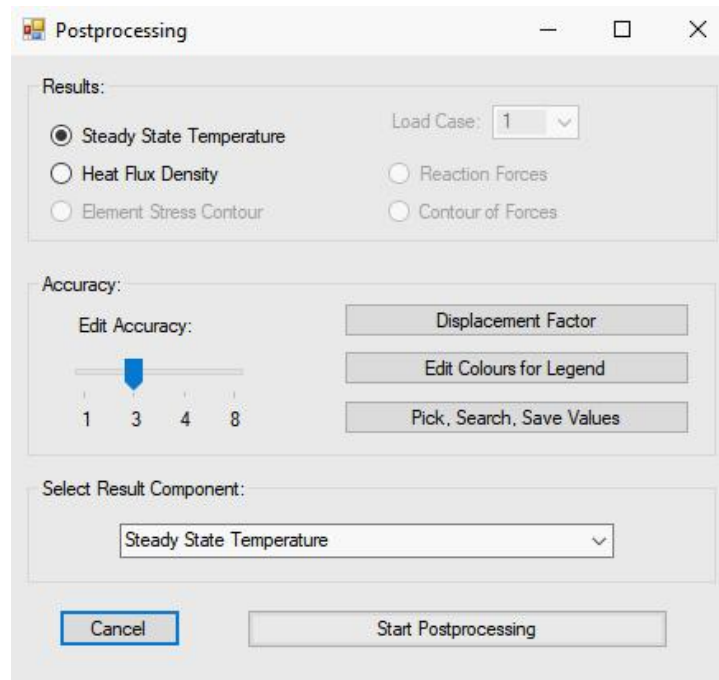
Save the FEM model under any name in the project directory and select the "FEM Analysis" tab to carry out a temperature analysis with the triangles TRI6S or CPS6.



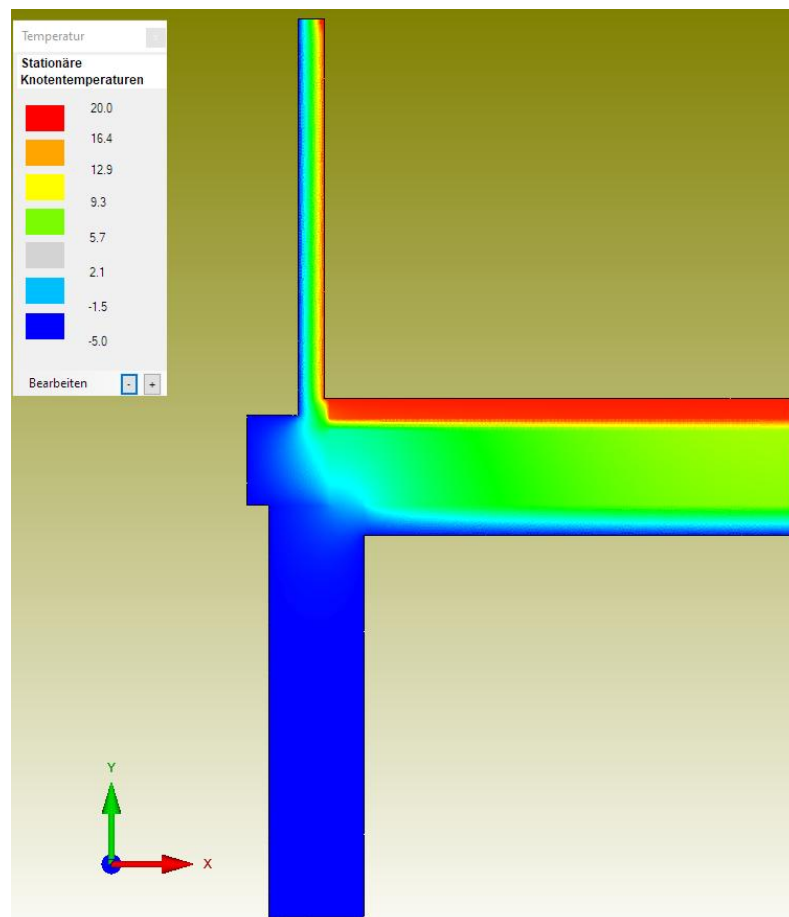
## Postprocessing



The temperatures and the heat flow density are displayed with the icon and the "Postprocessing" tab.



## Steady State Temperature Distribution



## Heat Flow Density

The heat flow density must be set to the value "20" W/m<sup>2</sup> using the "Deformation factor" menu be scaled, otherwise too high values will be seen.

