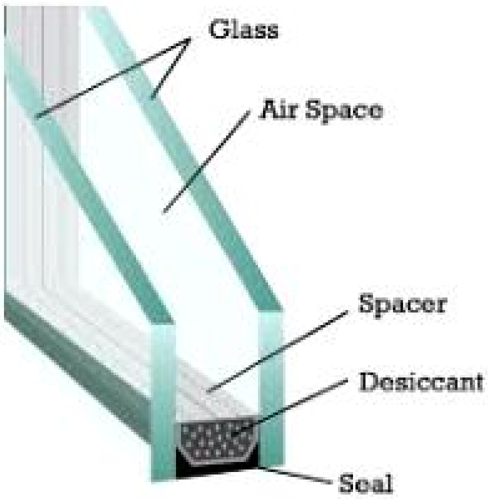
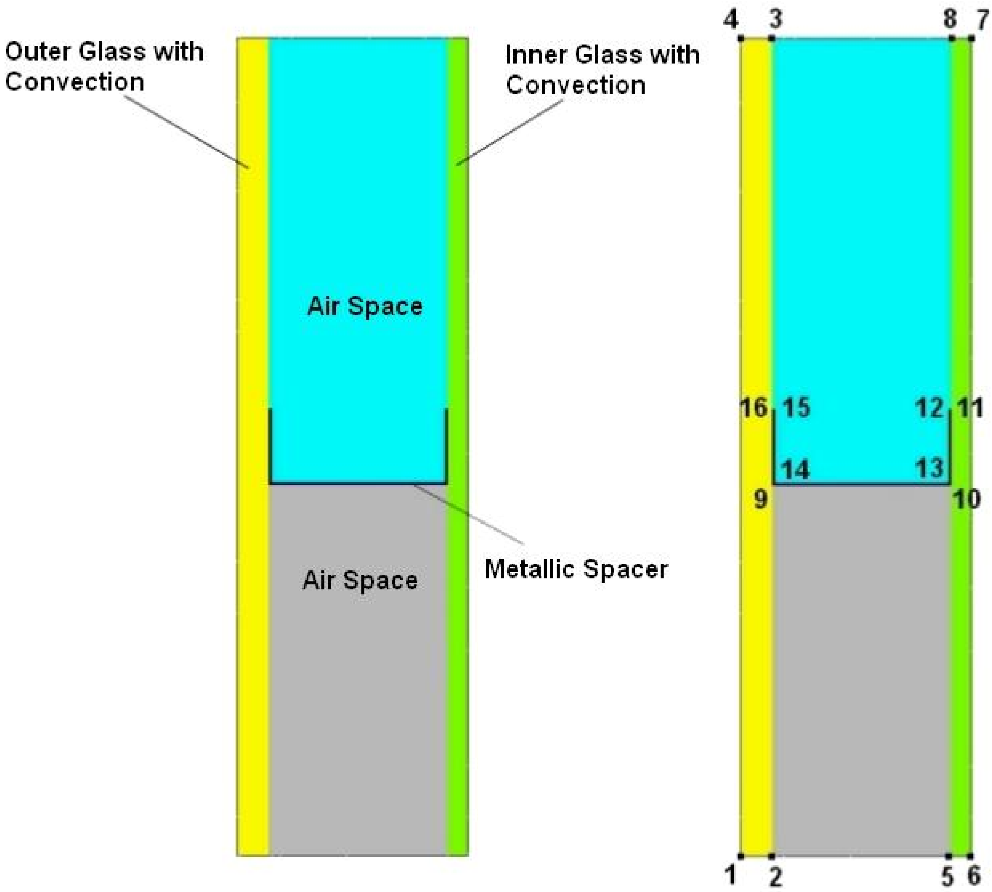
Part 15: Stationary Temperature-Analysis in a Double-Glazing with MEANS V11

Calculation of stationary temperature distribution and heat flux density in a double glazing with thermal convection in a outer and inner glass, air space and a metallic spacer.

Double-Glazing



FEM Model



16 Nodal Coordinates:

	X (m)	Y (m)		X (m)	Y (m)
1.	0.00	0.00	9.	0.000625	0.00725
2.	0.000625	0.00	10.	0.004125	0.00725
3.	0.000625	0.016	11.	0.004125	0.00875
4.	0.00	0.016	12.	0.004075	0.00875
5.	0.004125	0.00	13.	0.004075	0.0073
6.	0.004525	0.00	14.	0.000675	0.0073
7.	0.004525	0.016	15.	0.000675	0.00875
8.	0.004125	0.016	16.	0.000625	0.00875

FEMM Mesh Generator

With the new FEMM-Interface for MEANS V11 it is now possible to generate the double glazing FEM model with the 2D mesh generator of the FEMM program. FEMM is very useful and powerful FEM freeware for solving electromagnetic problems.

Download FEMM

Please download and install the FEMM 2D freeware program from the website **http://www.femm.info/wiki/Download.**

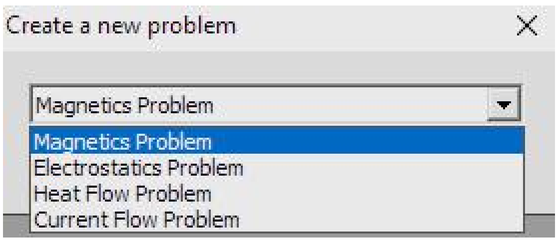
After installing FEMM, it will start with the desktop icon 

.

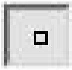
Now follow these steps to generate a FEM mesh with FEMM:

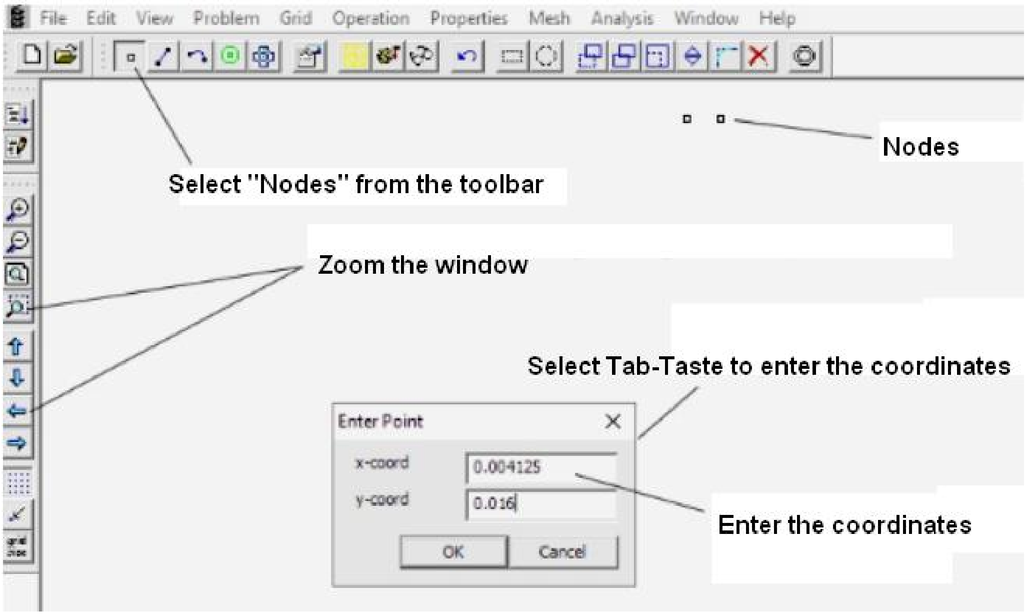
Magnetics Problem

Select "Magnetics Problem" from 4 different calculation options. However, only mesh generation is needed

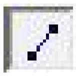


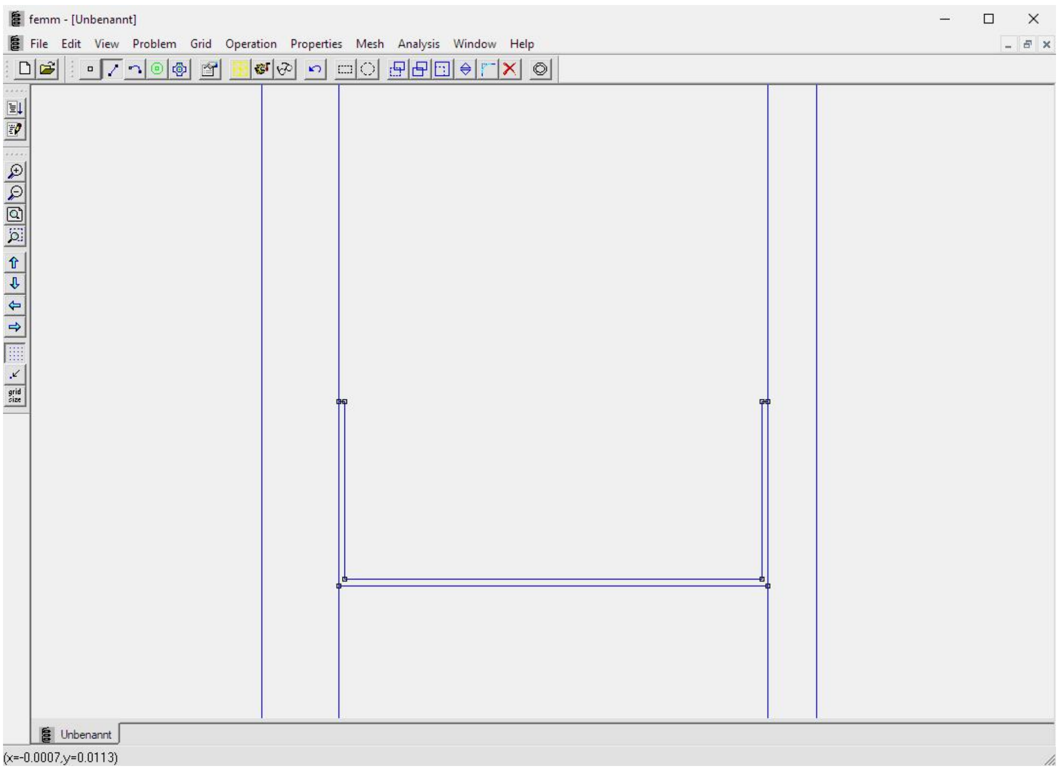
Enter the Nodal Coordinates

First, select “nodes”  in the toolbar and press the tab key each time to enter the coordinates in a dialogbox. With the icon  the nodes can be zoomed onto the screen.



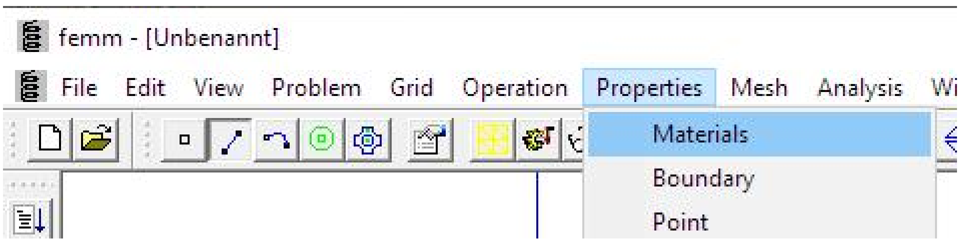
Enter the Lines

Connect the nodes with the icon  by clicking with the left mouse button on the first and then on the second node of the line.

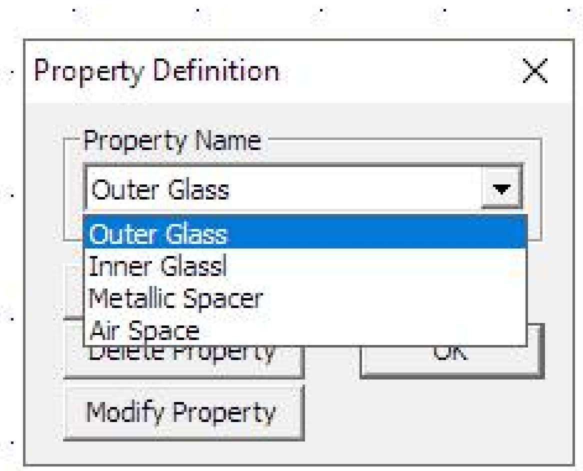


Define Properties

Now define the properties with the "Properties" and "Materials" menu and create the 4 properties "Outer Glass", "Inner Glass", "Metallic Spacer" and "Air Space".



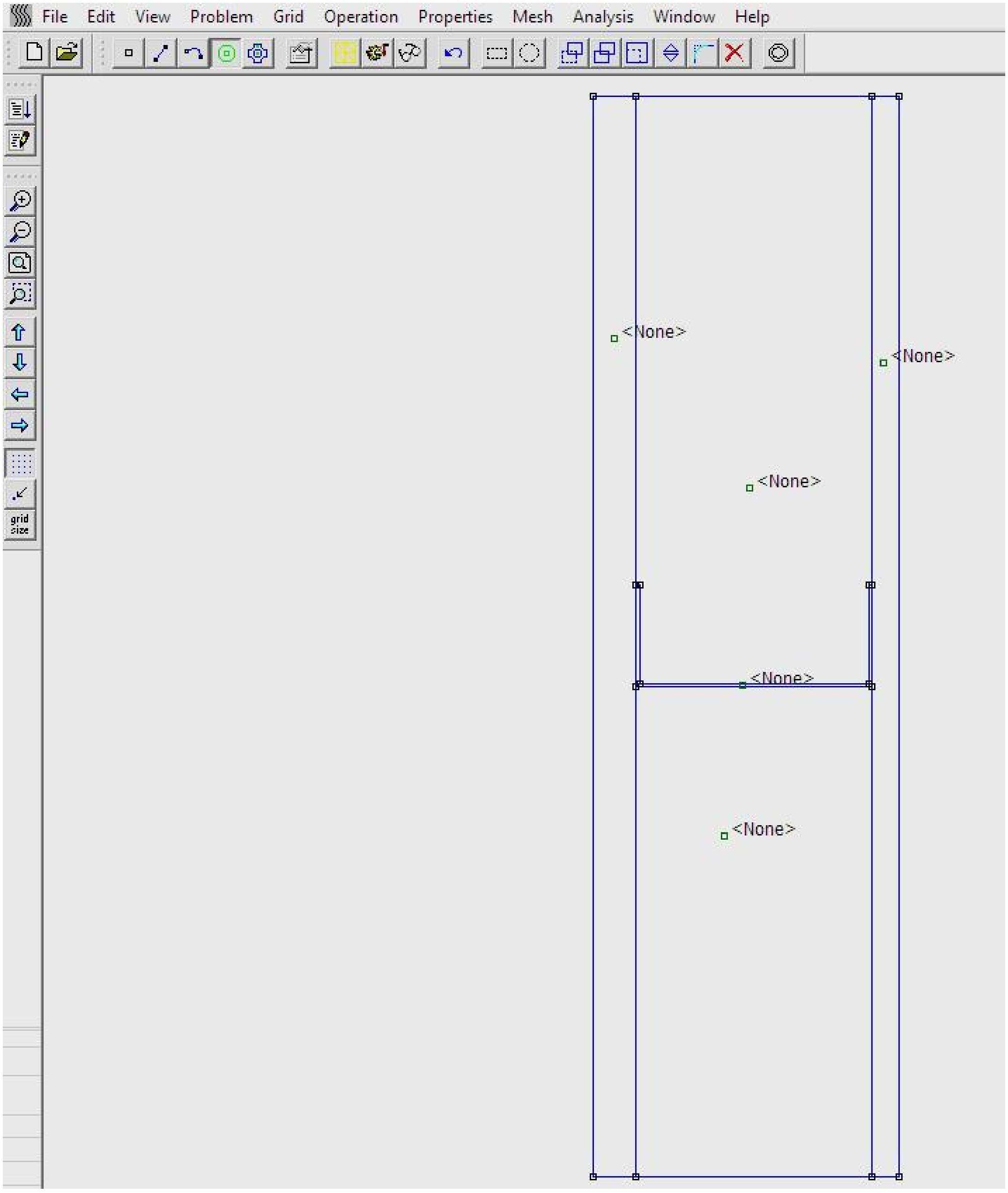
Accept the preset material data as only the FEM mesh is interested in the electromagnetic calculation.



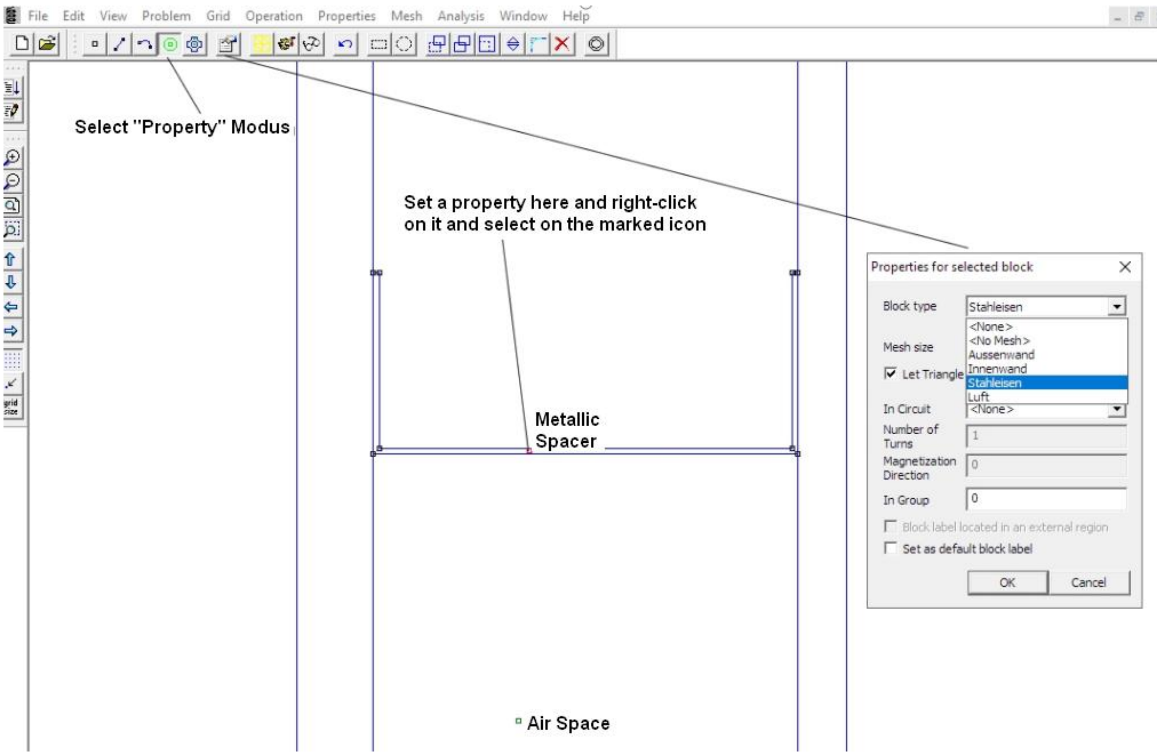
Place and select properties

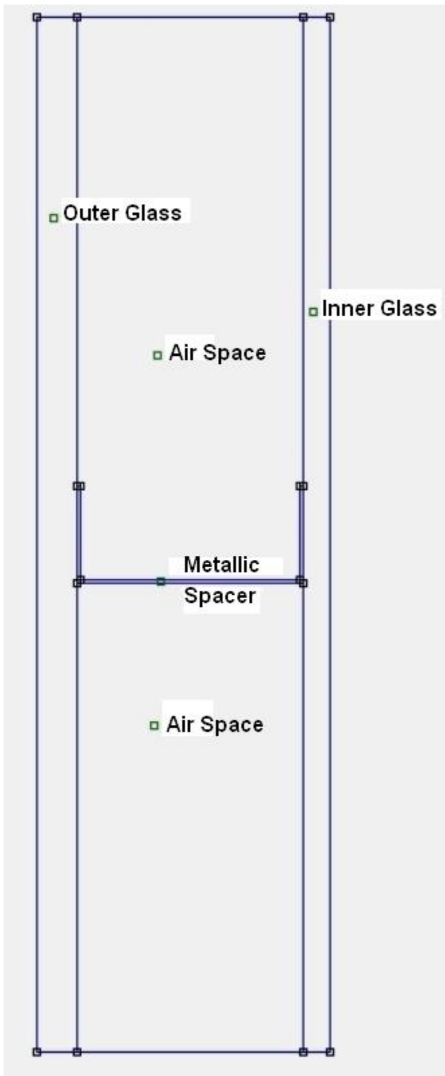
Place now with the icon  5 new properties at the respective places.

Select the property with the right mouse button and click on the icon  and select one of the 4 properties from the list.



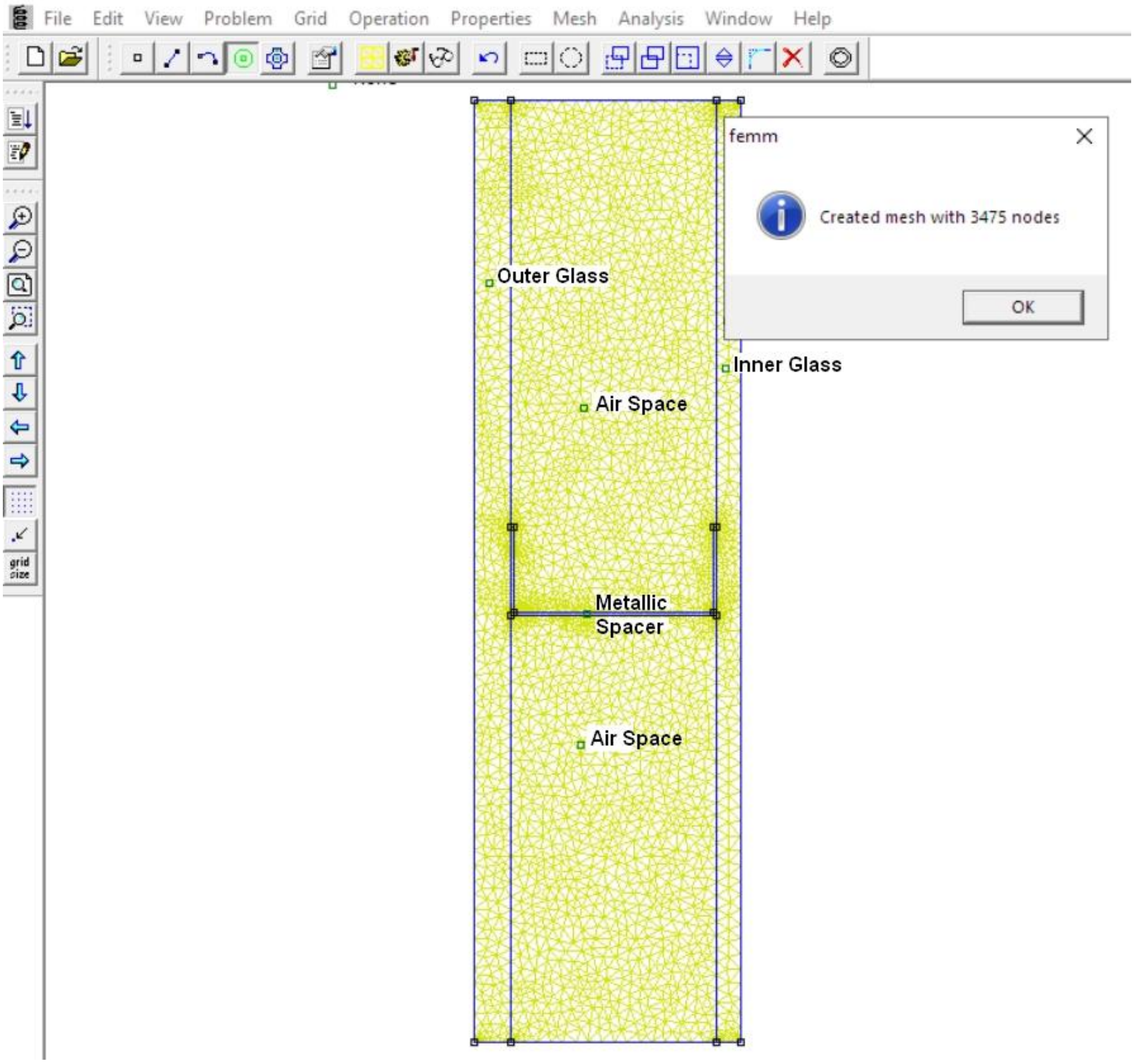
Zoom in with the icon  to the middle area so that the property "Metallic Spacer" can be placed.


File Edit View Problem Grid Operation Properties Mesh Analysis Window Help
Select "Property" Modus
Set a property here and right-click on it and select on the marked icon
Properties for selected block
Block type
Stahleisen
<None>
<No Mesh>
Aussenwand
Innenwand
Stahleisen
Luft
Mesh size
Let Triangle
In Circuit
Number of Turns
Magnetization Direction
In Group
Block label located in an external region
Set as default block label
OK
Cancel
Metallic Spacer
Air Space


Outer Glass
Inner Glass
Air Space
Metallic Spacer
Air Space

Generate FEM Mesh

Now save the nodes, lines and properties under FEMM1.FEM in the project directory and use the menu "Create Mesh" to generate a FEM Mesh of 3475 nodes.

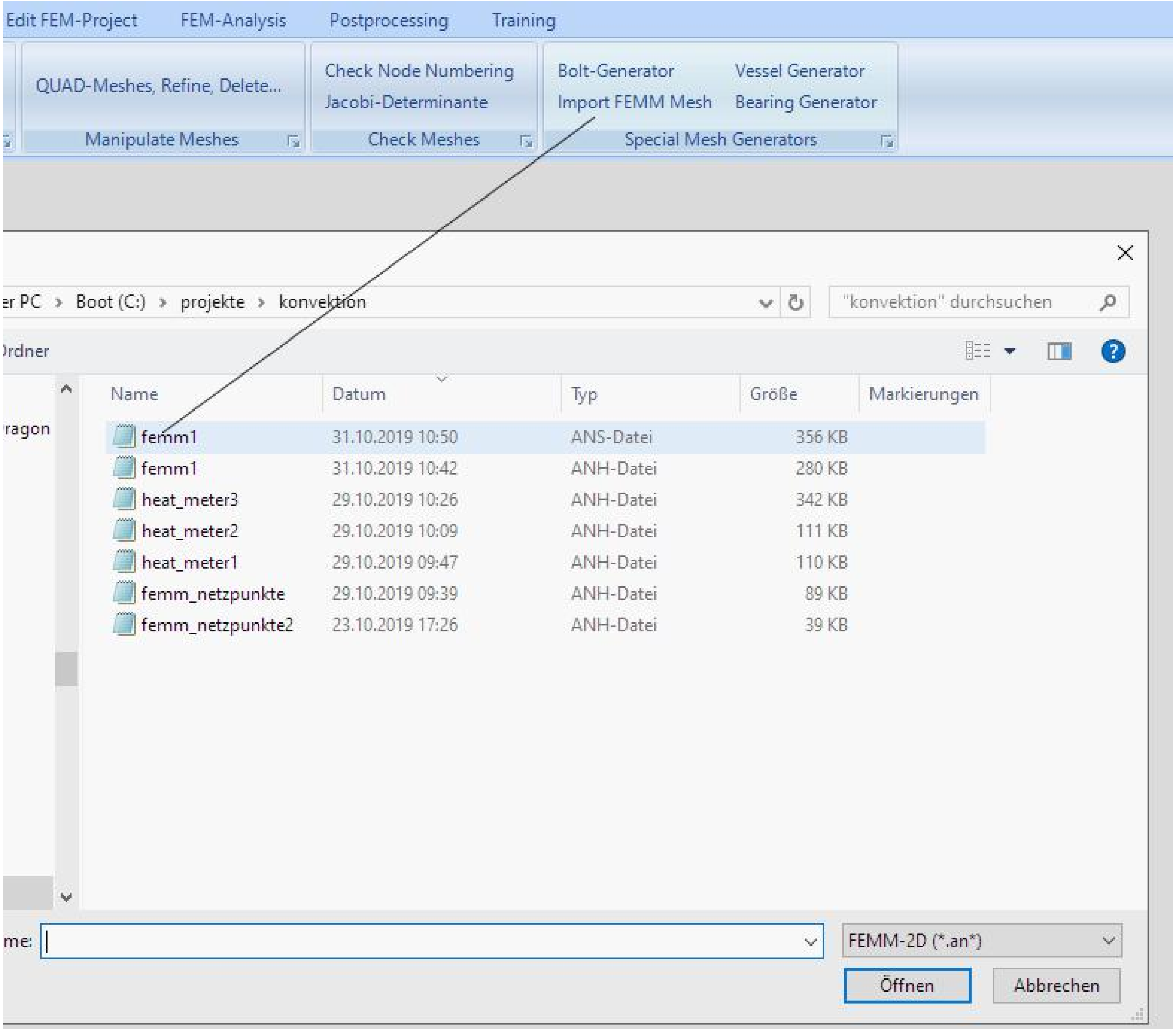


Select Analyze

Select the menu "Analysis" and "Analyze" and carry out a short calculation to generate the required ANS file FEMM1.ANS. FEMM is no longer needed and can be closed.

Import the FEMM Mesh

Now start the FEM system MEANS V11 with the desktop icon  and select the tab "Mesh generation" as well as "Import FEMM Mesh" to load the FEMM mesh **FEMM1.ANS**.

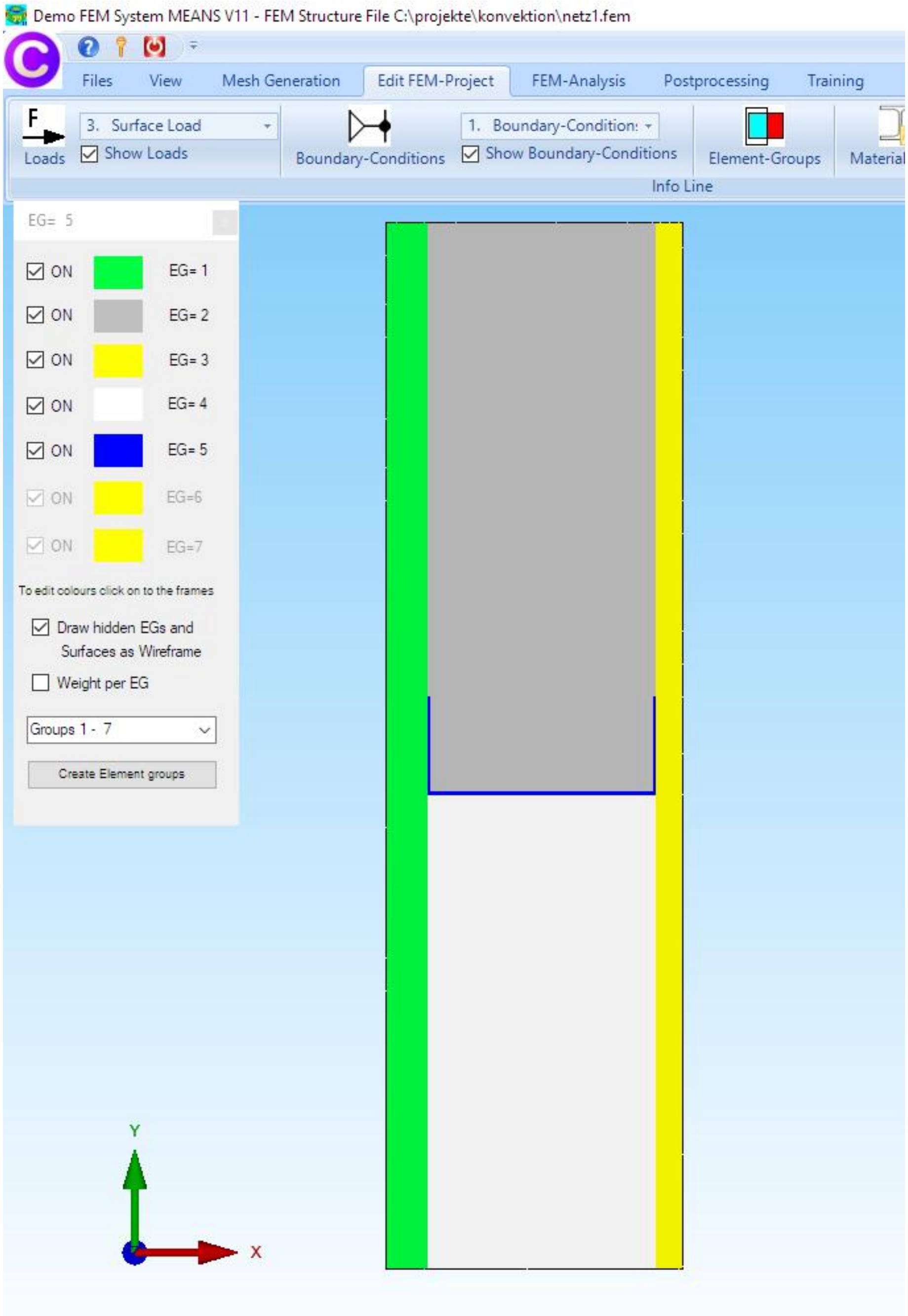


After the import a FEM Mesh with 6693 TRI3S elements, 3475 nodes and 5 element groups is created.

This FEM Mesh can now be used for all statics, dynamics or temperature calculations of MEANS V11.

Edit Element groups

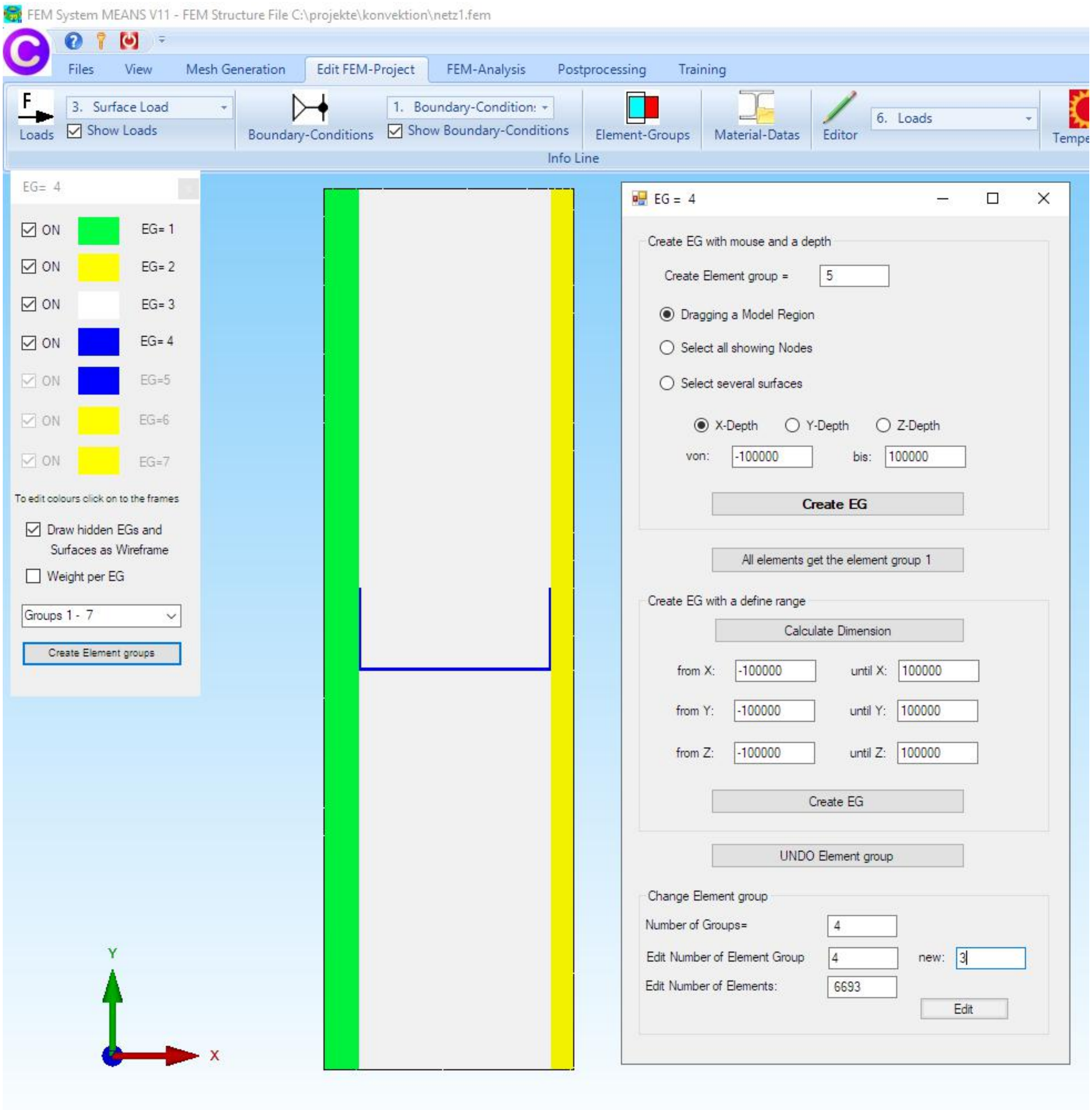
Select the "Edit FEM Project" and "Element Groups" tabs and give each of the five element groups a color by clicking on the left color frame.
You can also switch the element groups on and off to edit one or to evaluate a specific or inner group



Change Element Groups:

If the two groups of elements 2 + 4 are combined for air, an element group can be saved. In the legend, select the "Create Element groups" menu and change the element groups as follows:

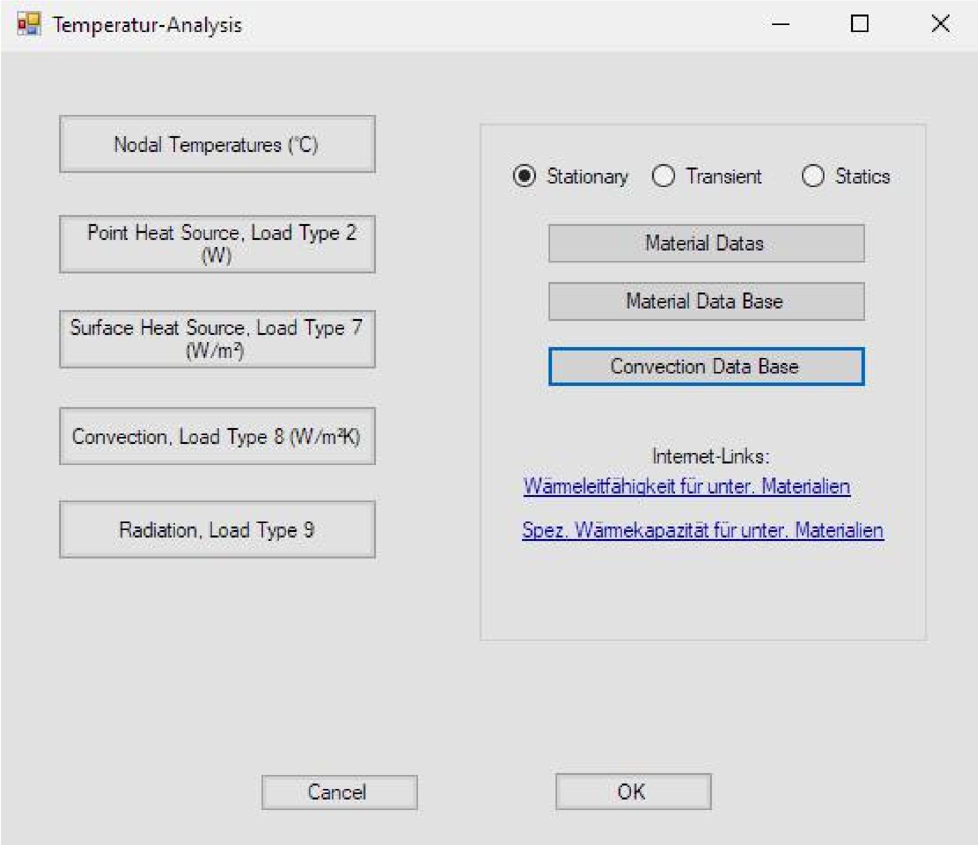
Increase number of element groups from 5 to 6
Change element group 2 to 6
Change element group 3 to 2
Change element group 6 to 3
Change element group 4 to 3
Change element group 5 to 4
Change number of element groups to 4



Stationary Temperature

Select register “Edit FEM-Project” and the icon 

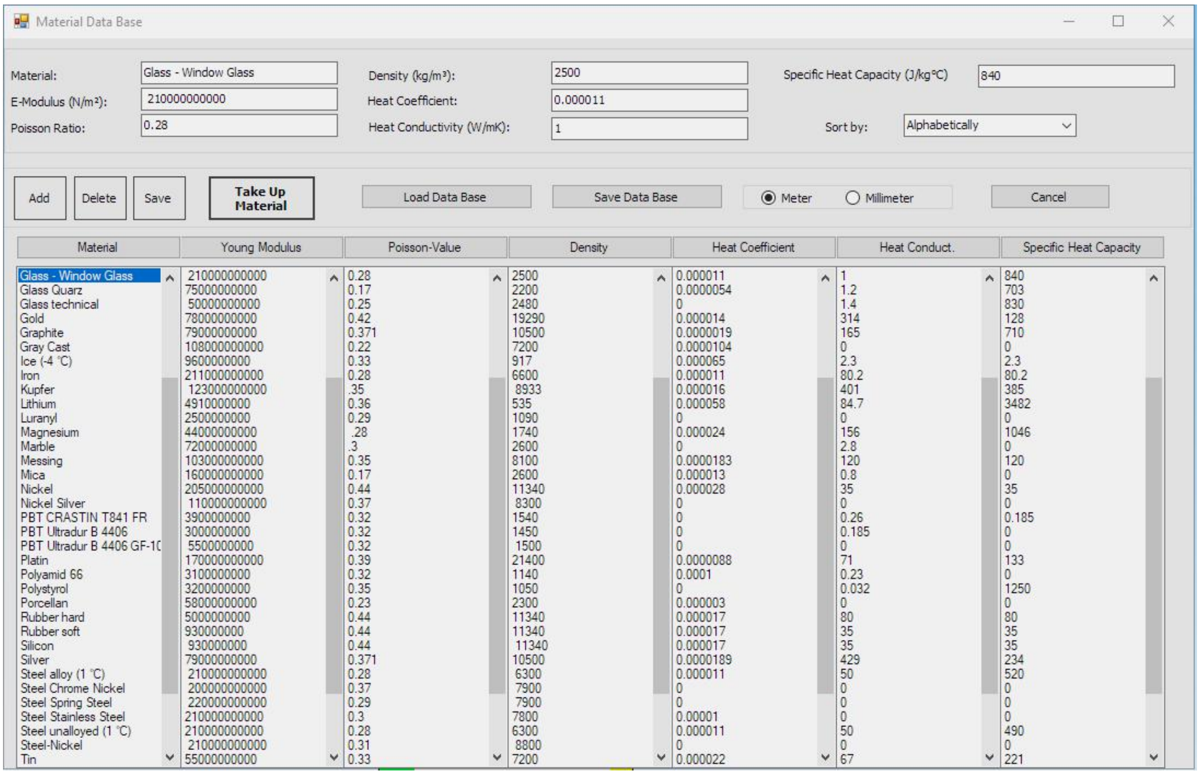
to enter the stationary temperature as well as the thermal conductivity and convection.



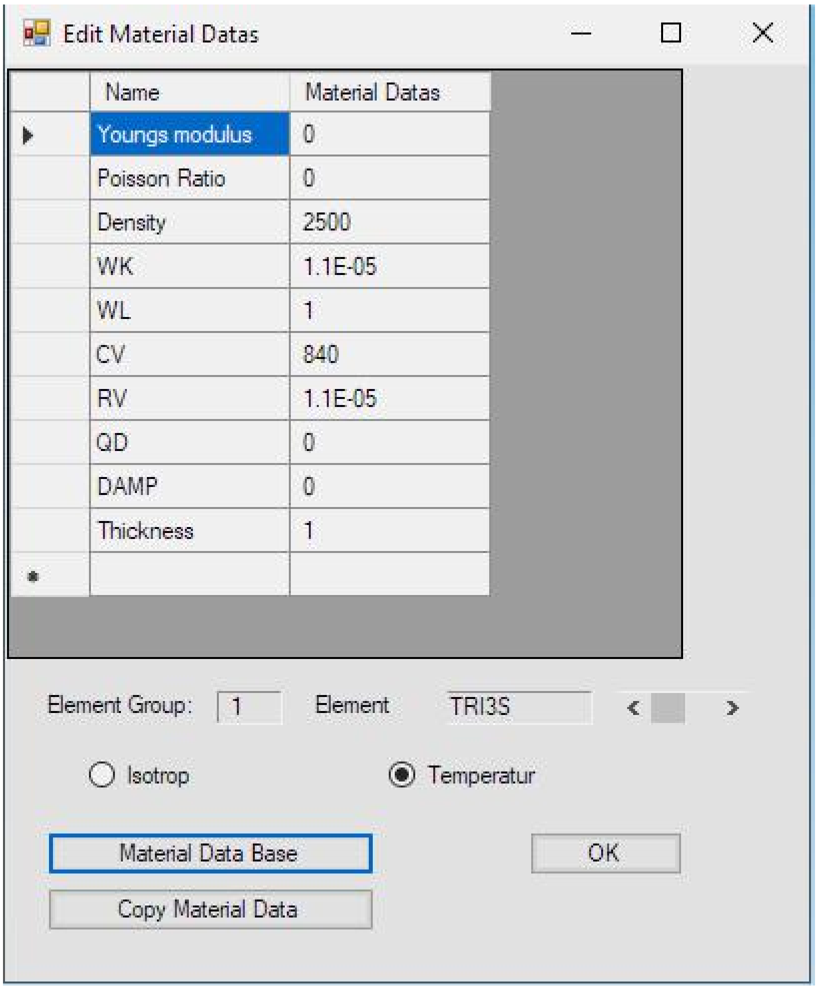
Create the Thermal Conductivities

Select “Material Datas” to take over the Thermal Conductivities of Glass, Steel and Air from the self-expandable Material Data Base of MEANS V11.

Element Group 1	Outer Glass:	WL = 1 W/mK
Element Group 2	Inner Glass:	WL = 1 W/mK
Element Group 3	Air-Space:	WL = 0.0181 W/mK
Element Group 4	Metallic Spacer :	WL = 10 W/mK



Select “Glass Window Glass” and “Take Up Material” to create the Material Datas:

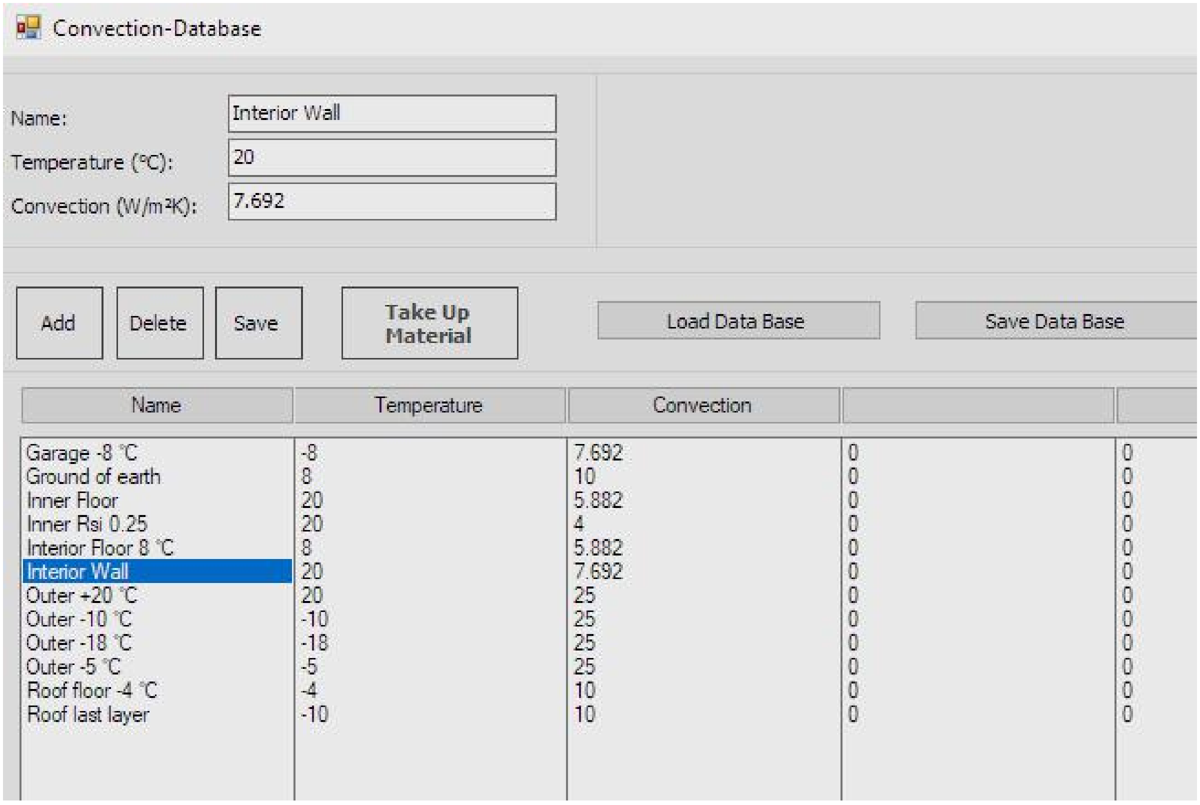


Create Convection

Select the icon 

again to enter the convection of the outer and inner glass. First select the following convection values from the self-expandable Convection Database:

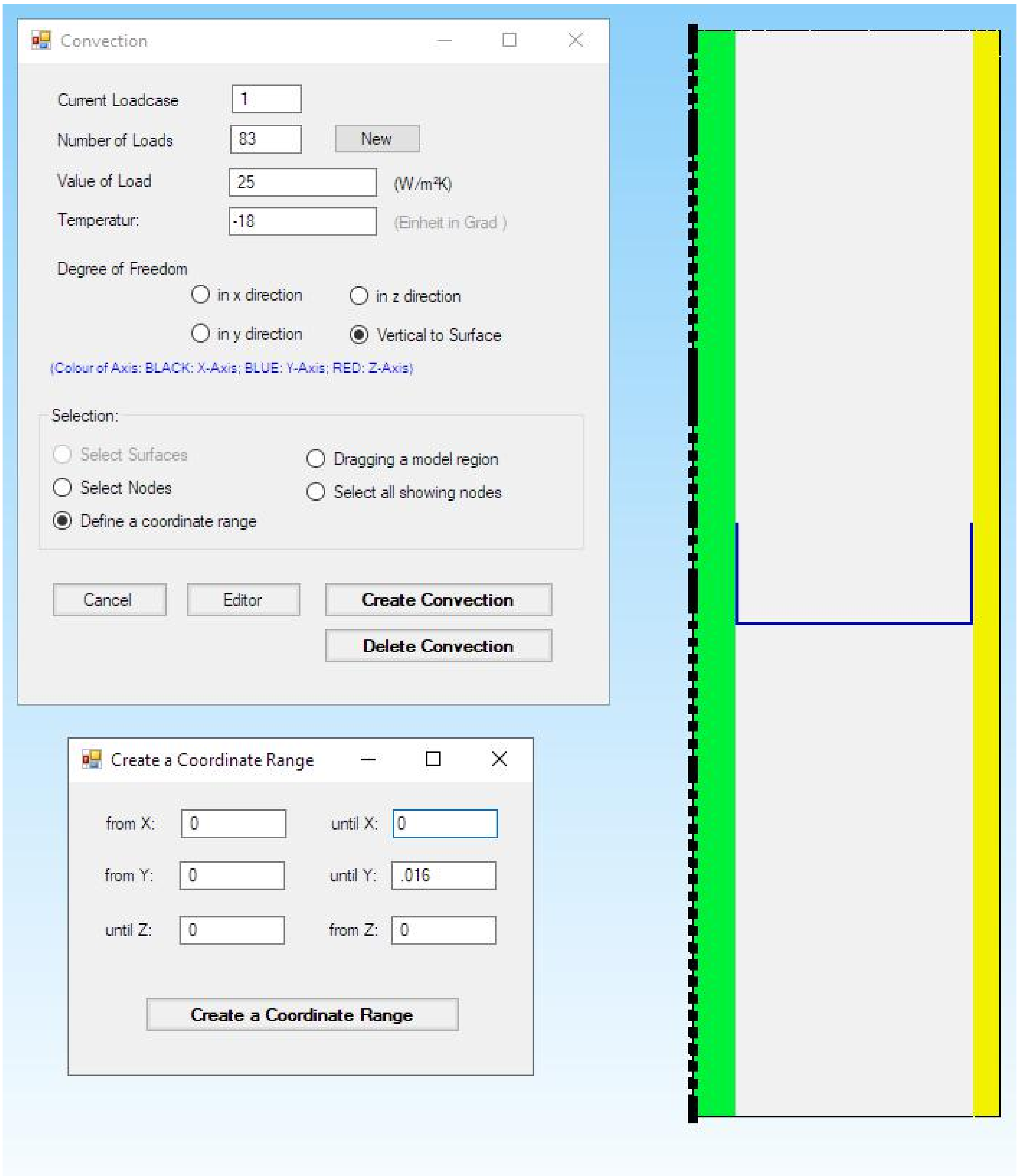
Convection on outer Glass = 25 W / m²K and temperature = -18 ° C
Convection on Inner Glass = 7,692 W / m²K and temperature = 20 ° C



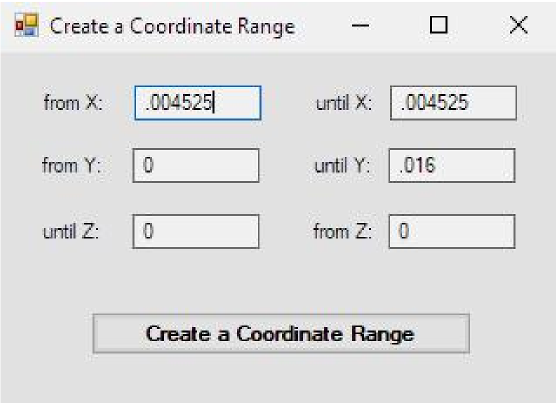
Coordinate Range

Now select the menu “Convection, Load Type 8 (W/(m²K)) and enter the value of convection = 25 W/m²K and the temperature of -18 ° C with the selection “Define a coordinate range” and select “Create Convection” to enter in the next dialogbox the coordinate range to create the convection on the outer glass:

from x = 0 until x = 0
from y = 0 until y = 0.016
from z = 0 until z = 0



Create with the same steps for the inner glass with the convection = 7,692 W / m²K and the temperature of 20 ° C also with the cooordinate range from x = 0.004525 until X = 0.004525 or with the selection “Dragging a model region”.



Save the model

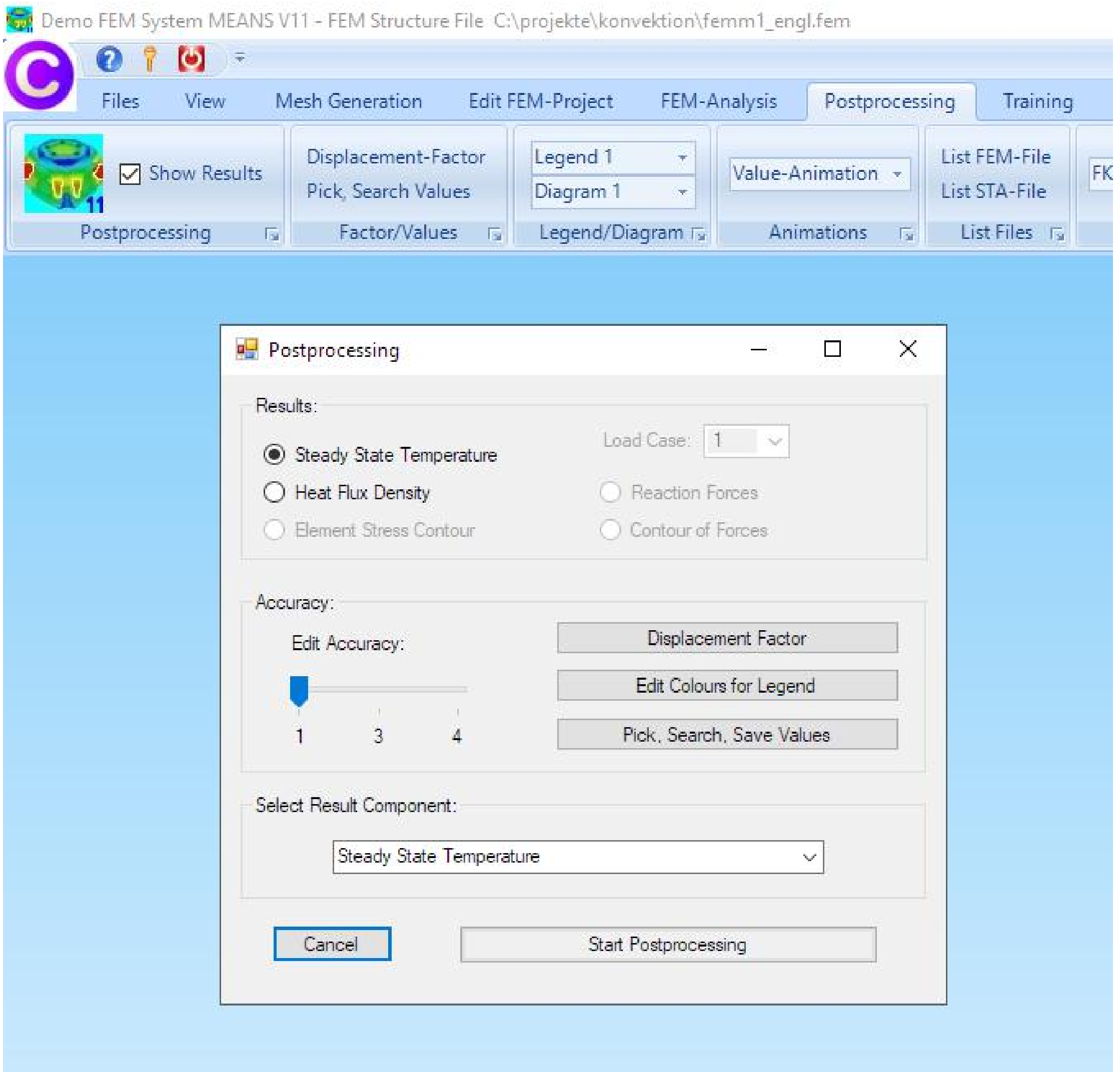
The FEM model is now complete and can be calculated. First save it with the “File” and “Save” tabs under the name “FEMM2.FEM” in the project directory.

FEM Analysis

Select register FEM-Analysis and menu "3. Temperature” to calculate the node temperatures.

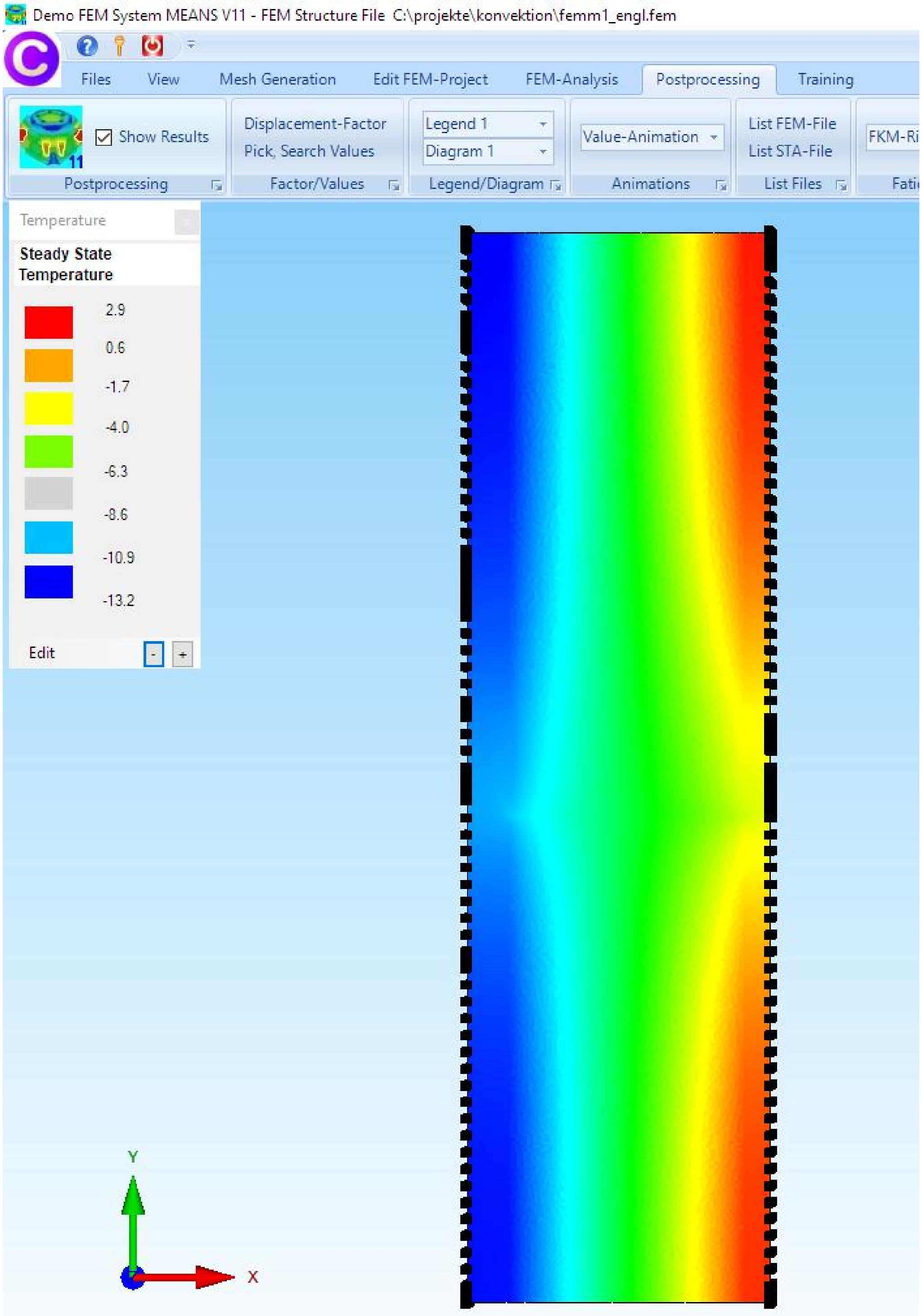
Postprocessing

Use the register Postprocessing and the icon  to show the stationary temperature distribution and the heat flux density.



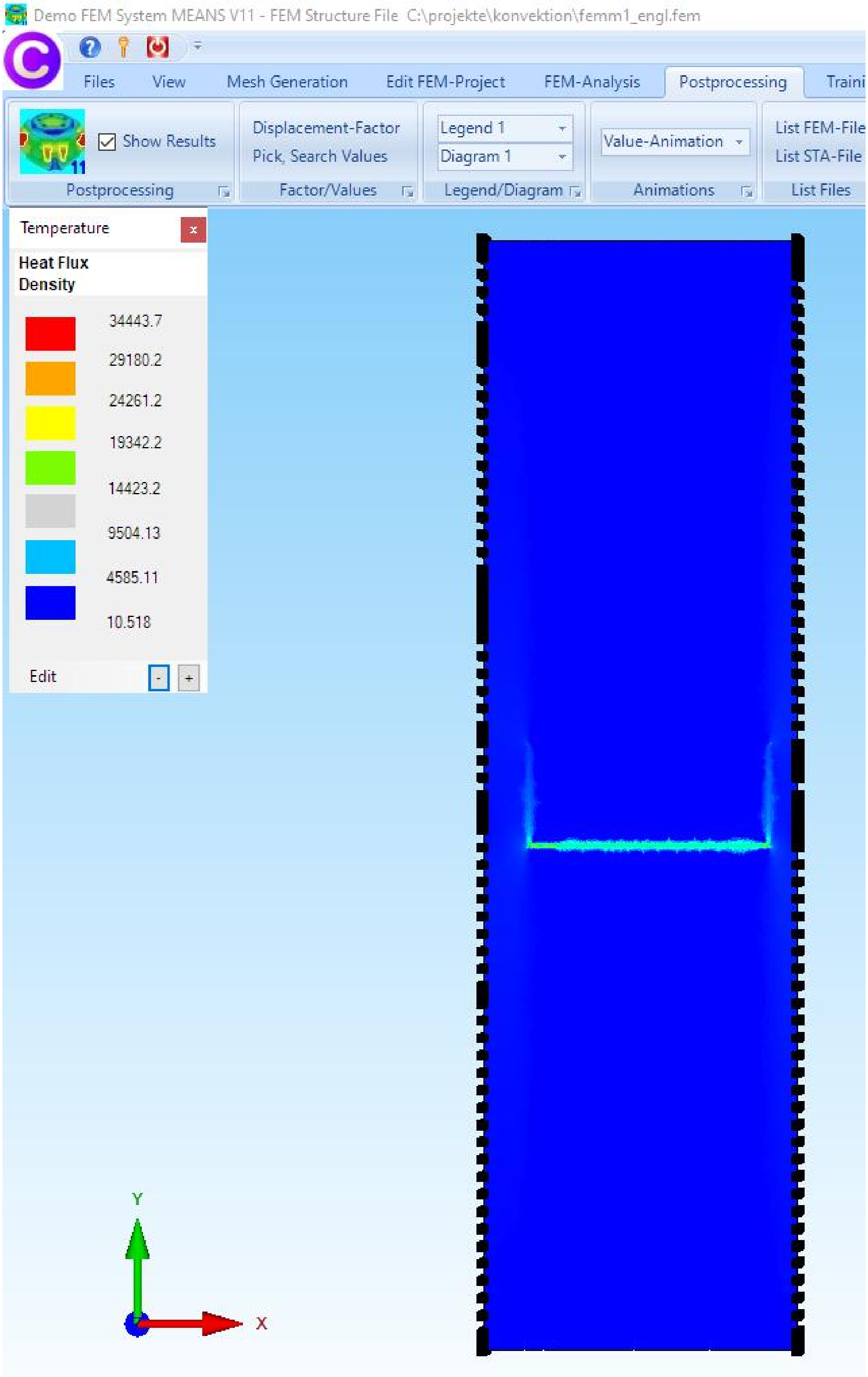
Temperature Distribution

Max. Temperature = 2.9 °C
Min. Temperature = -13.2 °C



Heat Flux Density

Middle Heat Flux Density = 10500 W/m² = 10.5 kW/m²

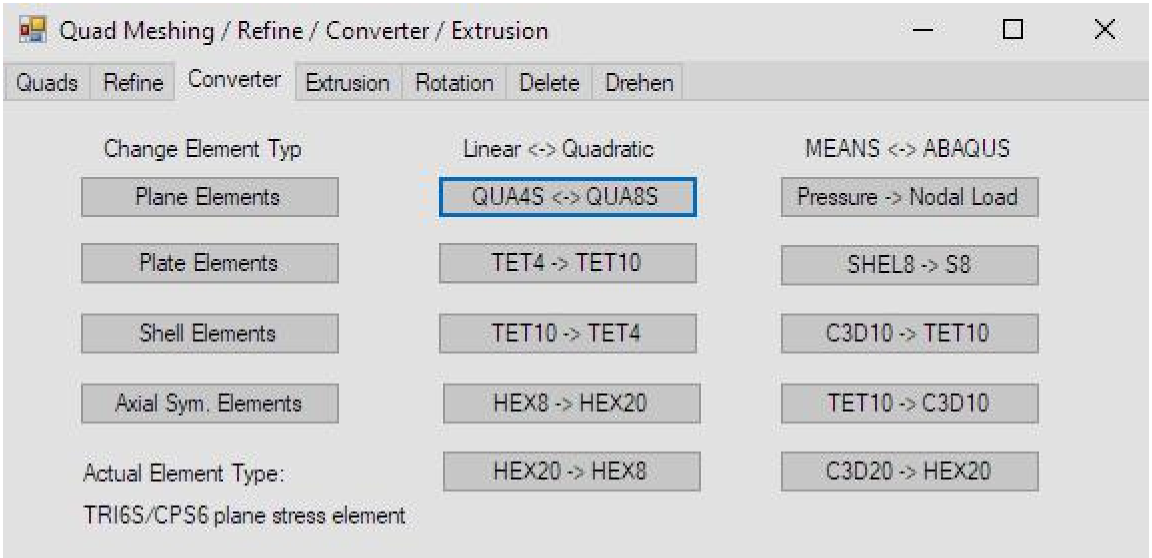


3D-Thermal Bridge

A 3D calculation follows by extruding the 2D thermal bridge with a Z depth into a 3D solid model.

Convert TRI6S to TRI3S

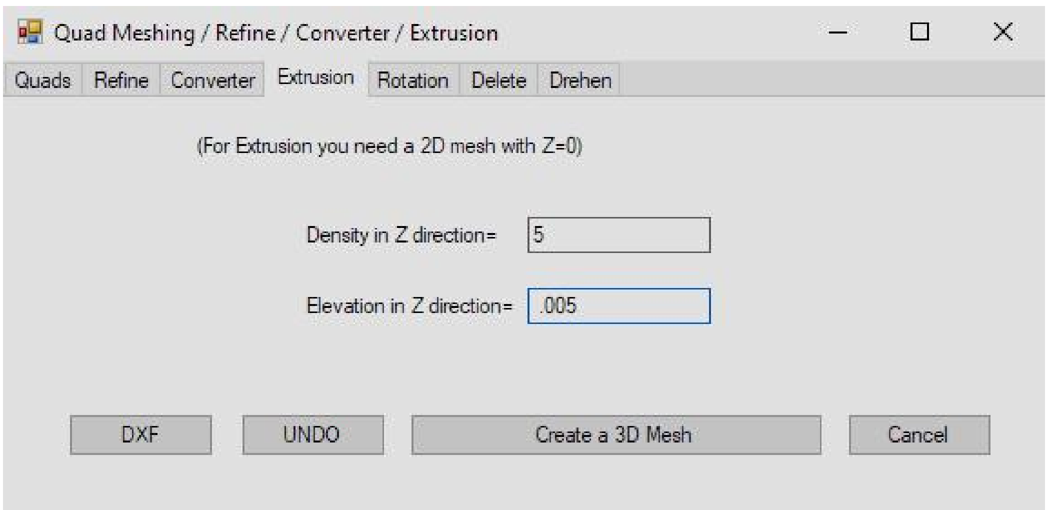
First, the quadratic TRI6S model must be converted to a TRI3S model without middle nodes. Select register "Mesh Generation" and menu "Quad-Meshes, Refine, Delete ..." and in the next dialogbox select register "Converter" and menu "QUA4S <-> QUA8S". After the conversion, the linear model named “QUA4S.FEM” is obtained.



Now select register “Extrusion” in the same dialog box and extrude the 2D mesh with the following setting

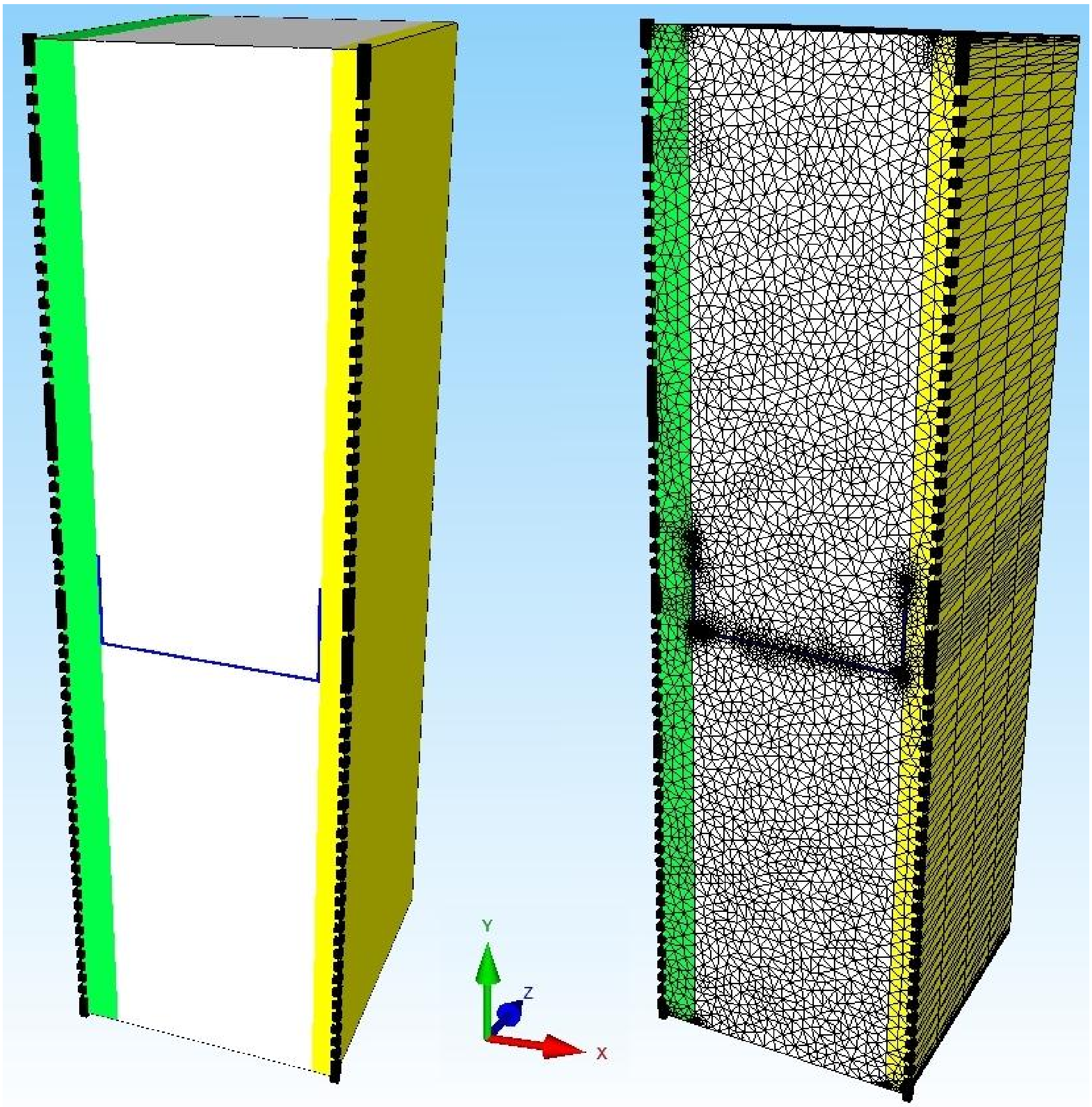
Number of nodes in the Z direction = 5
Z object height = 0.005 m

The new 3D mesh consisting of 60 237 PEN6 solid elements, 34 750 nodes and 4 element groups.



After the extrusion, the numbering is automatically checked and the hidden line and area model with 5 surface areas is generated.

Here you can see the extruded pentahedron model with and without mesh:

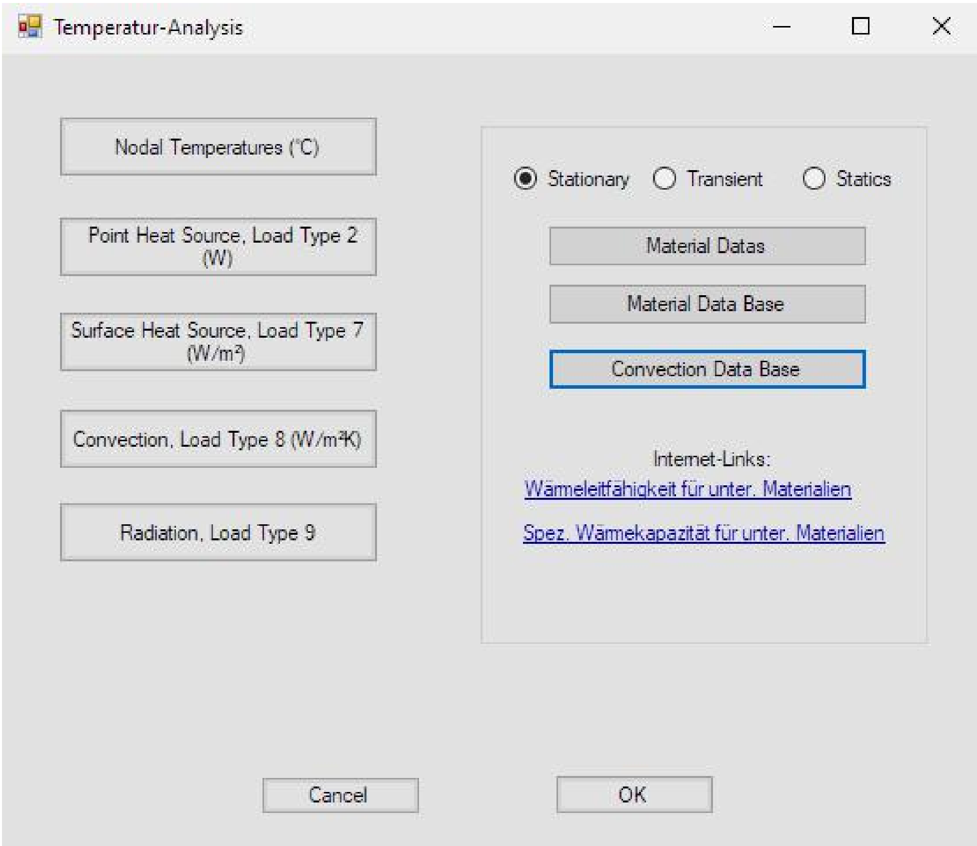


3D Convection

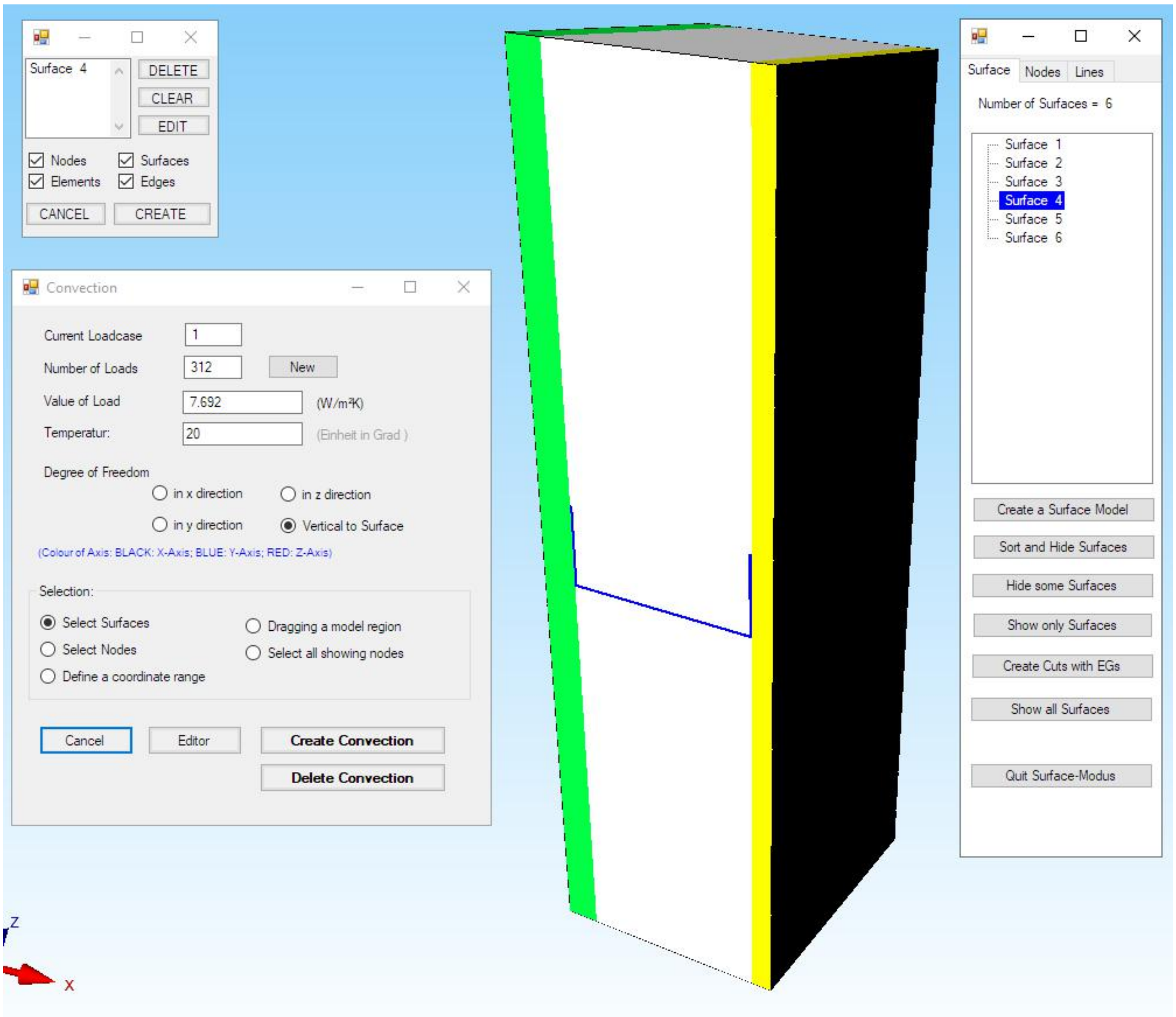
Select the icon 

again to enter the convection of the outer and inner glass. First select the following convection values from the self-expandable Convection Database:

Convection on outer Glass = 25 W / m²K and temperature = -18 ° C
Convection on Inner Glass = 7,692 W / m²K and temperature = 20 ° C



Select menu "Convection, Load Type 8 (W/m²K) and enter the 3D convection on the outer glass with the surface 1 and the convection on the inner glass with the surface 4.



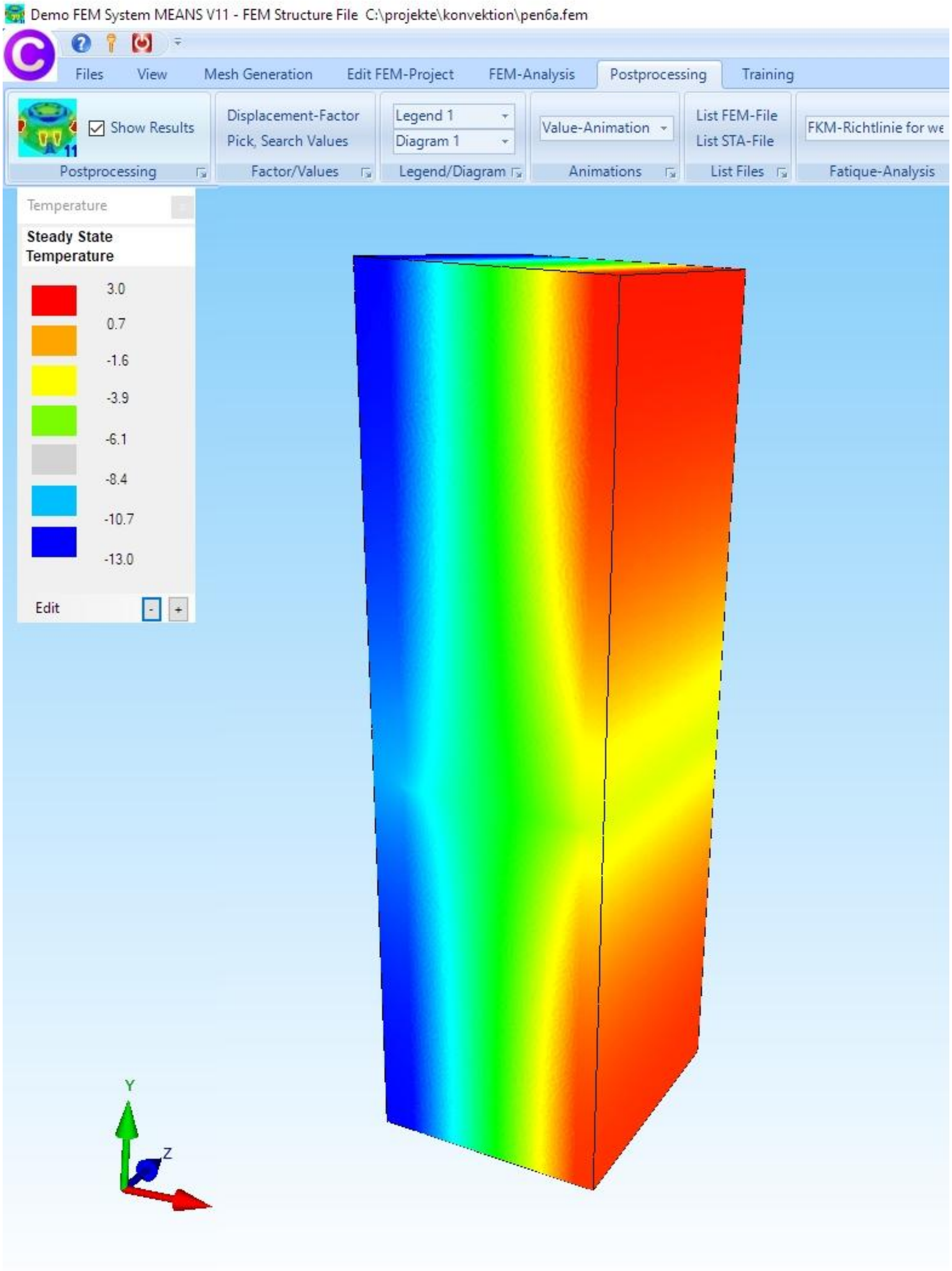
First save the model under the name “PEN6.FEM” and select FEM-Analysis to calculate the temperatures and heat flux density.

Postprocessing

Use the register Postprocessing and the icon  to show the stationary temperature distribution and the heat flux density.

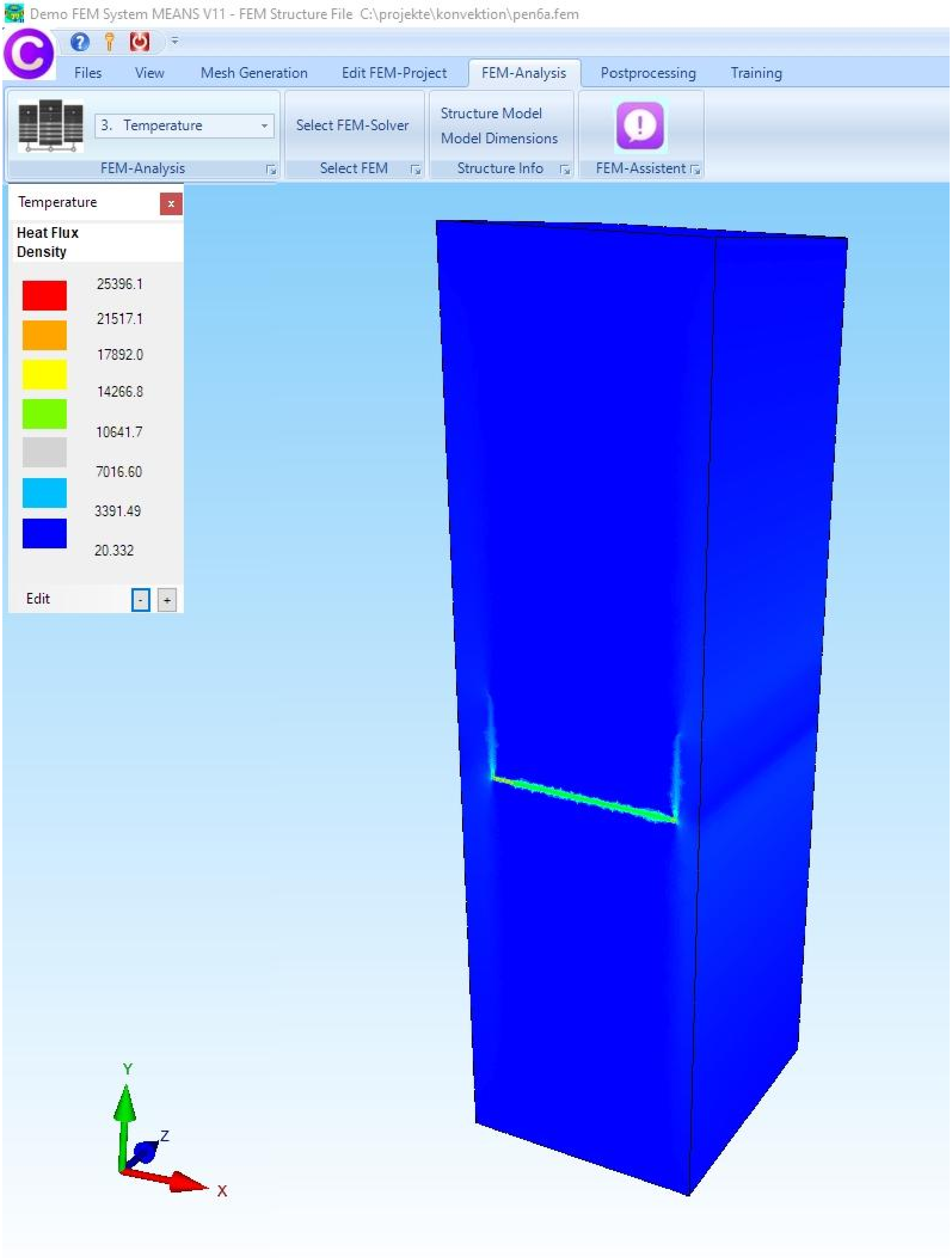
Temperature Distribution

Max. Temperature = 3 °C
Min. Temperature = -13 °C



Heat Flux Density

Middle Heat Flux Density = 11000 W/m² = 11 kW/m²

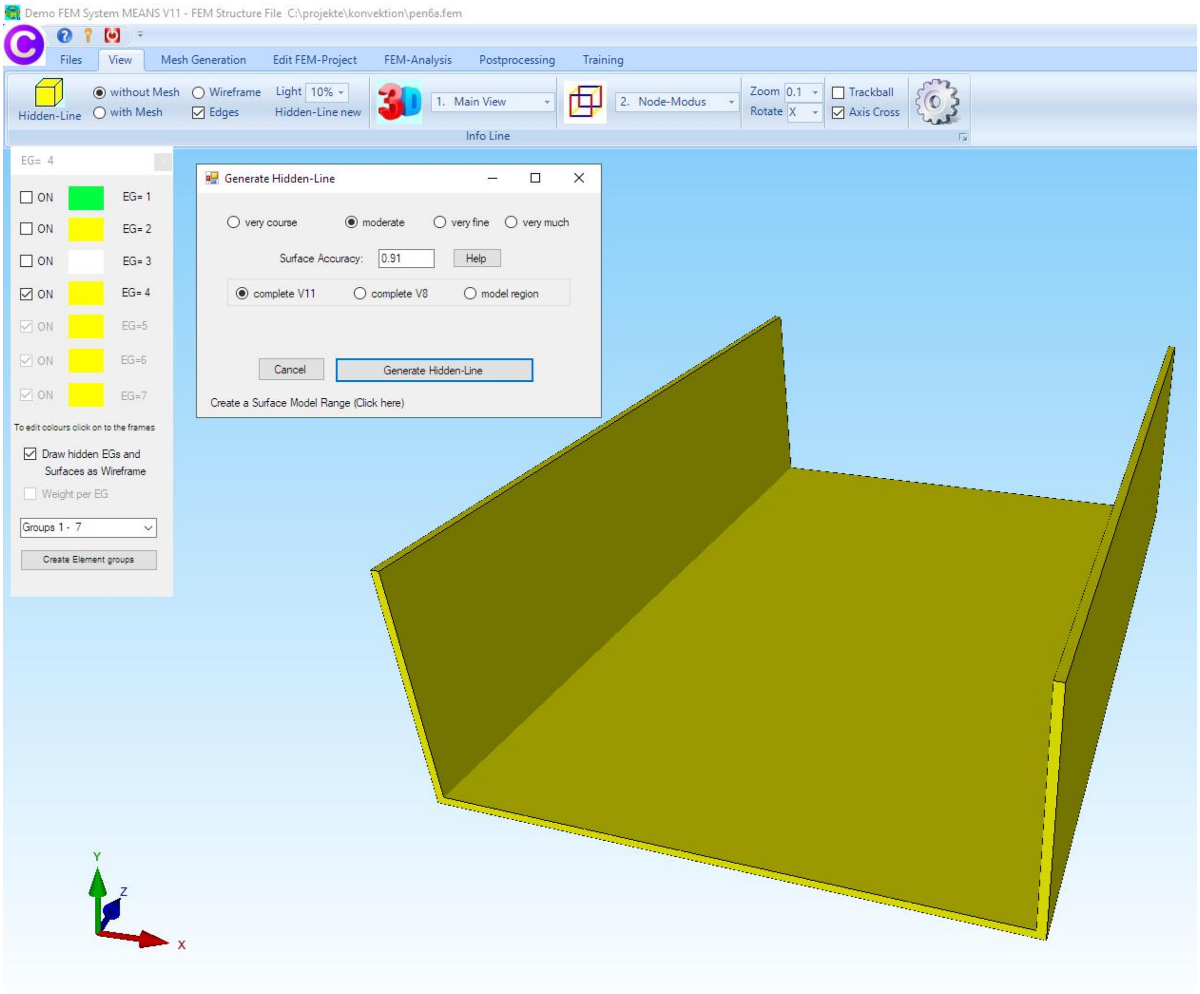


3D Point Heat Source

Now a 3D Heat Source Simulation is calculated with MEANS V11, which is not possible with a 2D program.
What temperature is set on the outer glazing when there is a heat source of 0.1 W in the middle of the metallic spacer.

Show Metallic Spacer

Select register "Edit FEM Project" and "Element Group" and activate only element group 4 so that the spacer is shown alone. Use the “View” and “Hidden Line New” tabs to recalculate the hidden line.



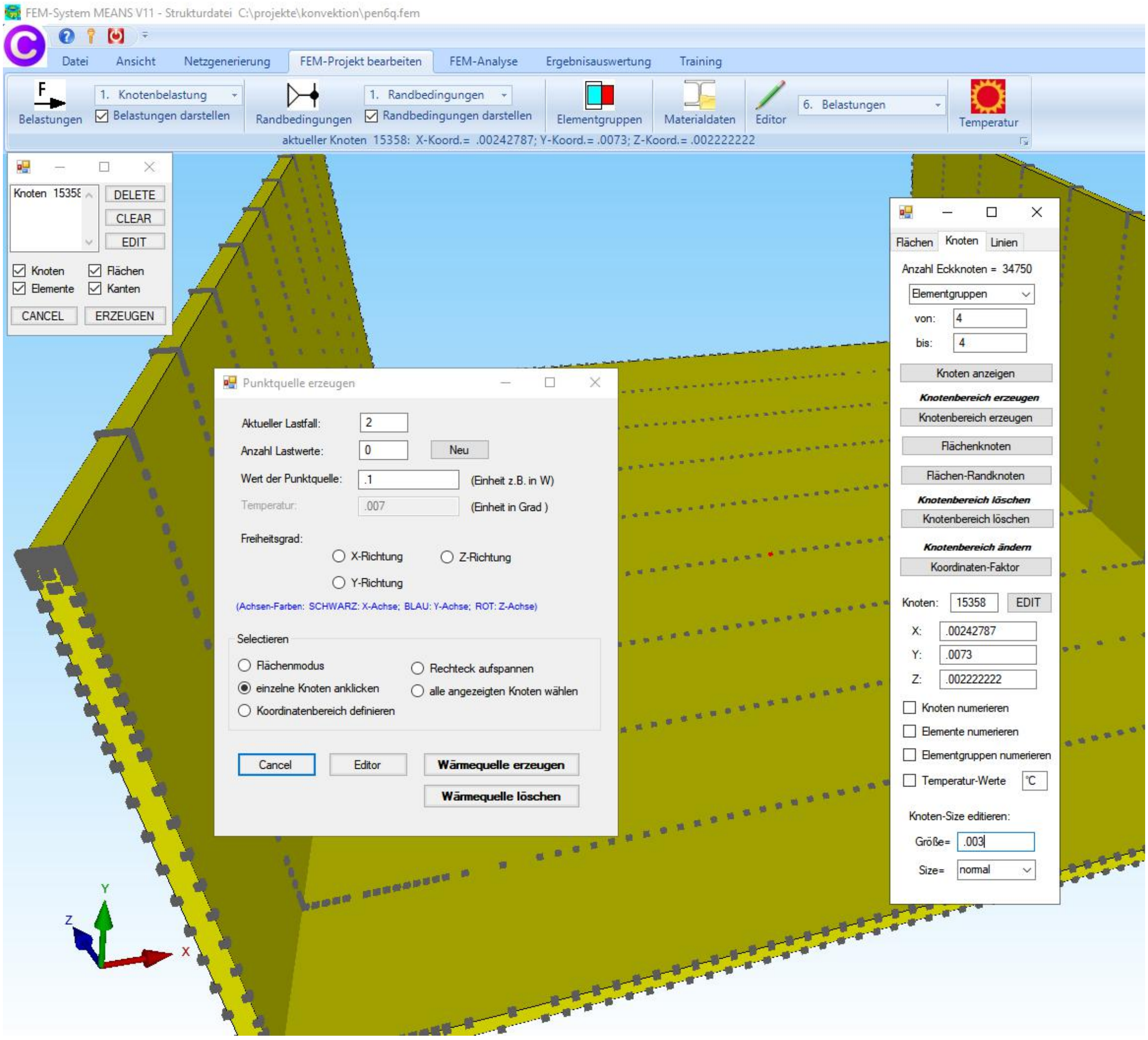
Create a Range of Nodes

Now create a Point Heat Source of 0.1 W in the middle of the metallic spacer as follows:

Select Node-Modus and create a selectable range of nodes of element group 4.

With the icon 

, select the menu “Point Heat Source, Load Type 2 (W)” and with load case 2 and the value = 0.1 and the selection “Select Nodes”, double-click on node 15358 so that it is displayed in the select box, there with “Create” generate the point source in load case 2 and the convection in load case 1.



Then calculated the new temperature distribution with element groups 1 and 4

Temperatures with Element Group 1 and 4