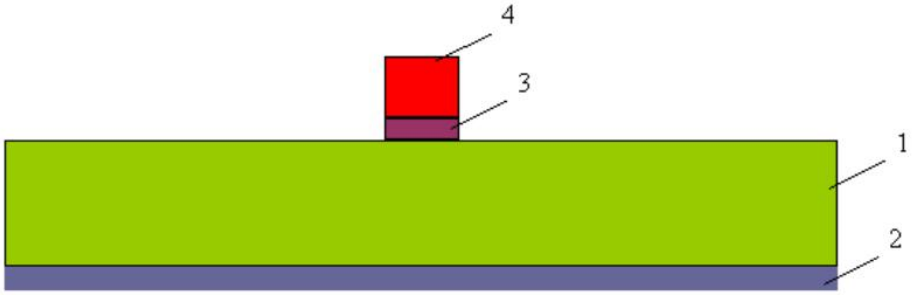
Part 6 - Temperature-Analysis with FEM-System MEANS V11

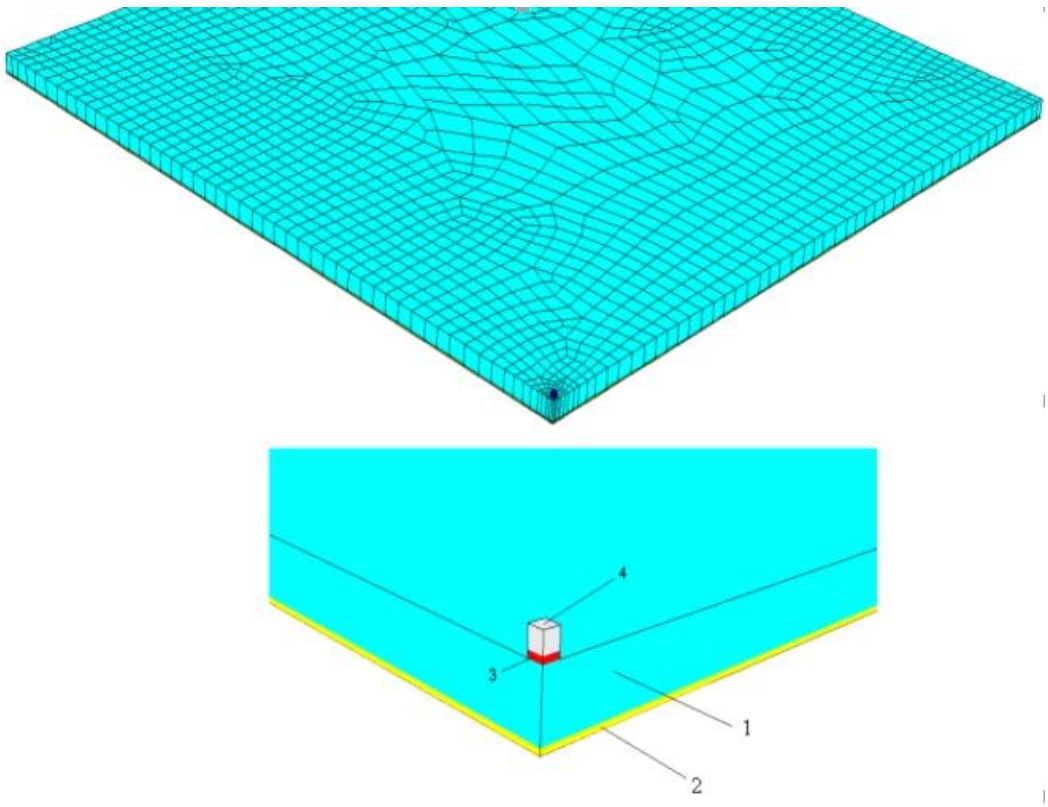
Chip-Assembly:

Element Group 1 **Board:** X = 100 mm; Y = 100 mm; Z = 1 mm; ʎ = 1 W/m*K
Element Group 2 **Leiter:** X = 100 mm; Y = 100 mm; Z = 0.1 mm; ʎ = 30 W/m*K
Element Group 3 **Chipkleber:** X = 0.5 mm;Y = 0.5 mm; Z= 0.1 mm; ʎ = 40 W/m*K
Element Group 4 **Chip:** X = 0.5 mm; Y = 0.5 mm; Z = 0.3 mm; ʎ = 300 W/m*K



How big is the temperature distribution at a source of heat = 1.5W in Element Group 4 (Chip) when the board is cooled with a convection of 5.5W / m² * K and an ambient temperature of 20 ° C. Since the model is symmetrical, it is sufficient to meshes only a quarter.

3D-Model



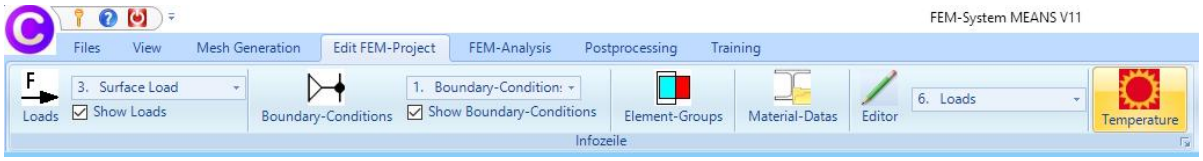
Additional module TEMPERATURE

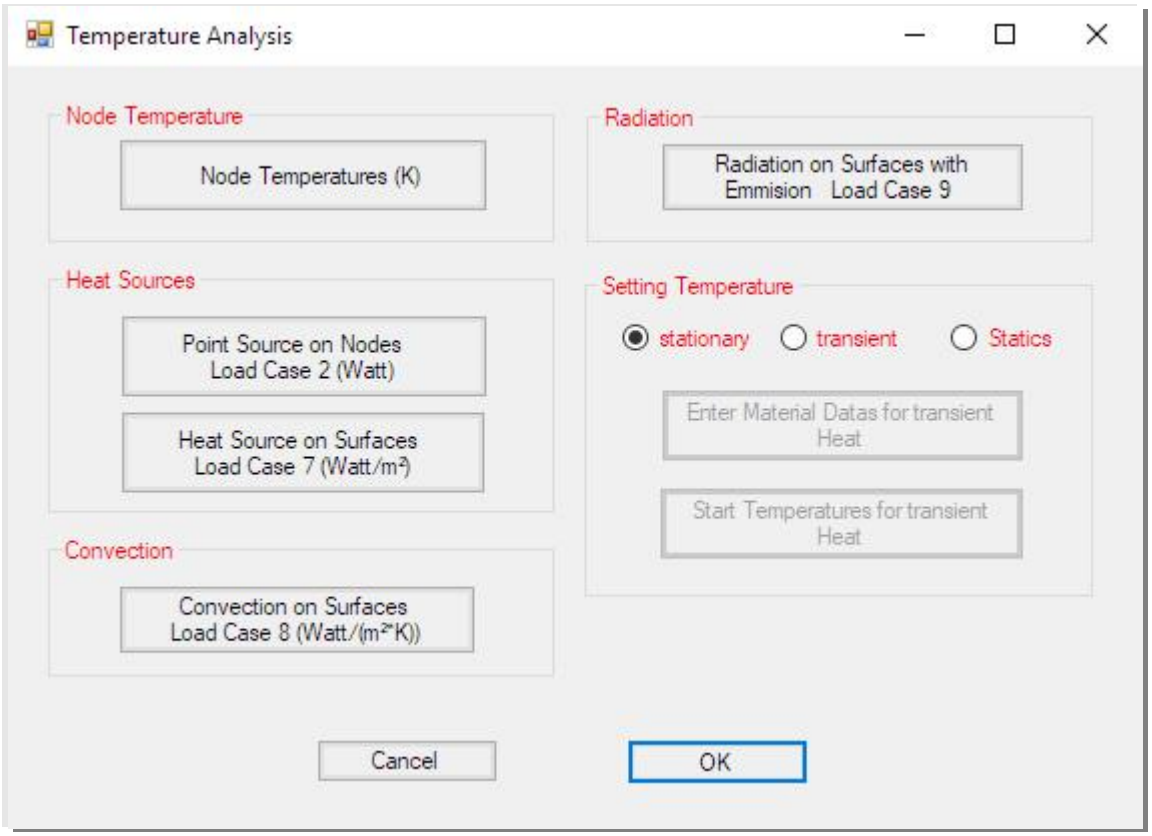
The additional module MEANS-TEMPERATURE contains a module for stationary (steady state) and unsteady (transient) temperature field calculation.

All finite elements used can process the following boundary conditions:

- Node Temperatures [K]
- Convection on all surfaces [Watt / (m² * K)]
- Point Source on Nodes [Watt]
- Heat Source on Surfaces [Watt / m²]
- Radiation with Emissions

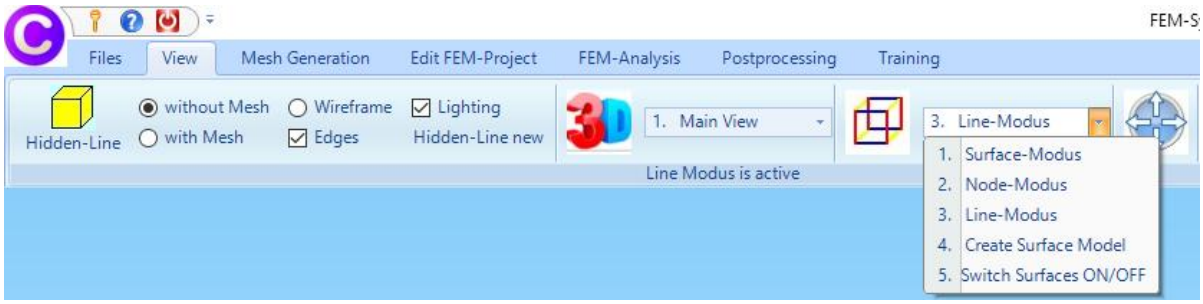
For a Temperature-Analysis select the „Edit FEM-Project“ tab and 





Create 2D template with Line-Modus

First, create a 2D template for hexahedral extrusion by selecting the „View“ tab and drop-down menu "3. Line Modus" and enter the following 4 nodes with the new side menu.



Node 1:

Select "New" and enter node 1 with X = 0, Y = 0 and Z = 0 and select "Create Nodes".

Node 2:

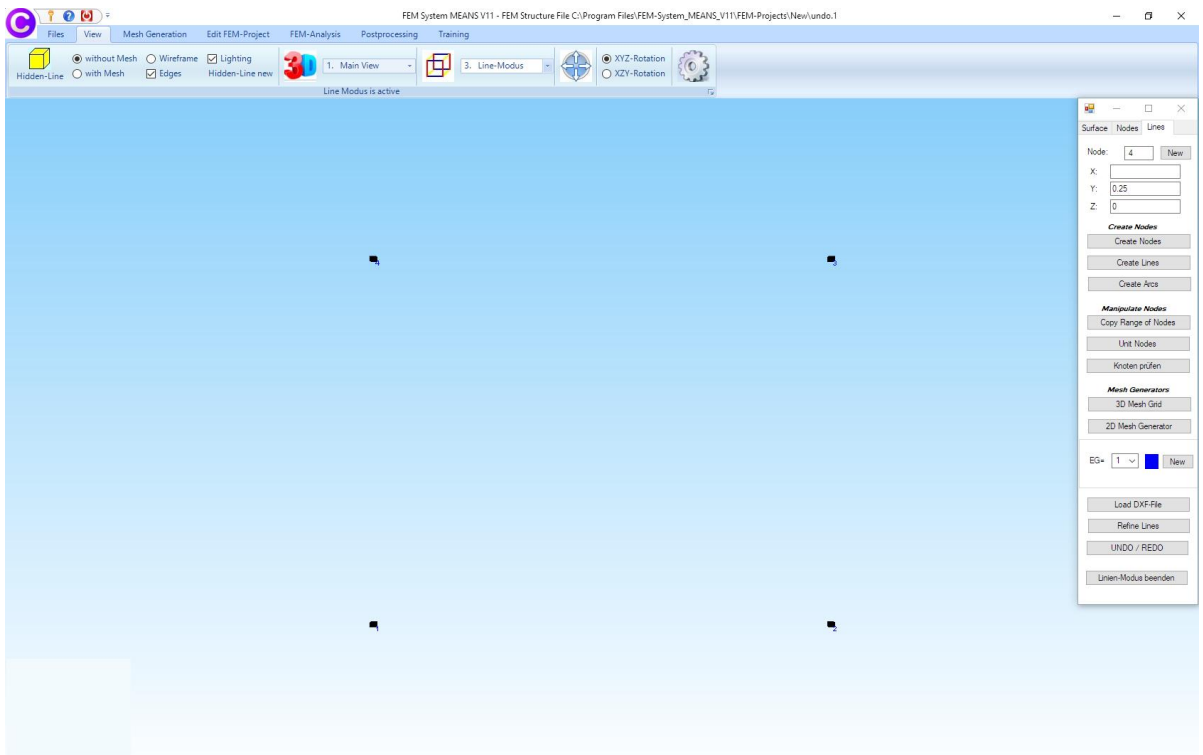
Select "New" and enter as above node 2 with X = 0.25, Y = 0 and Z = 0

Node 3:

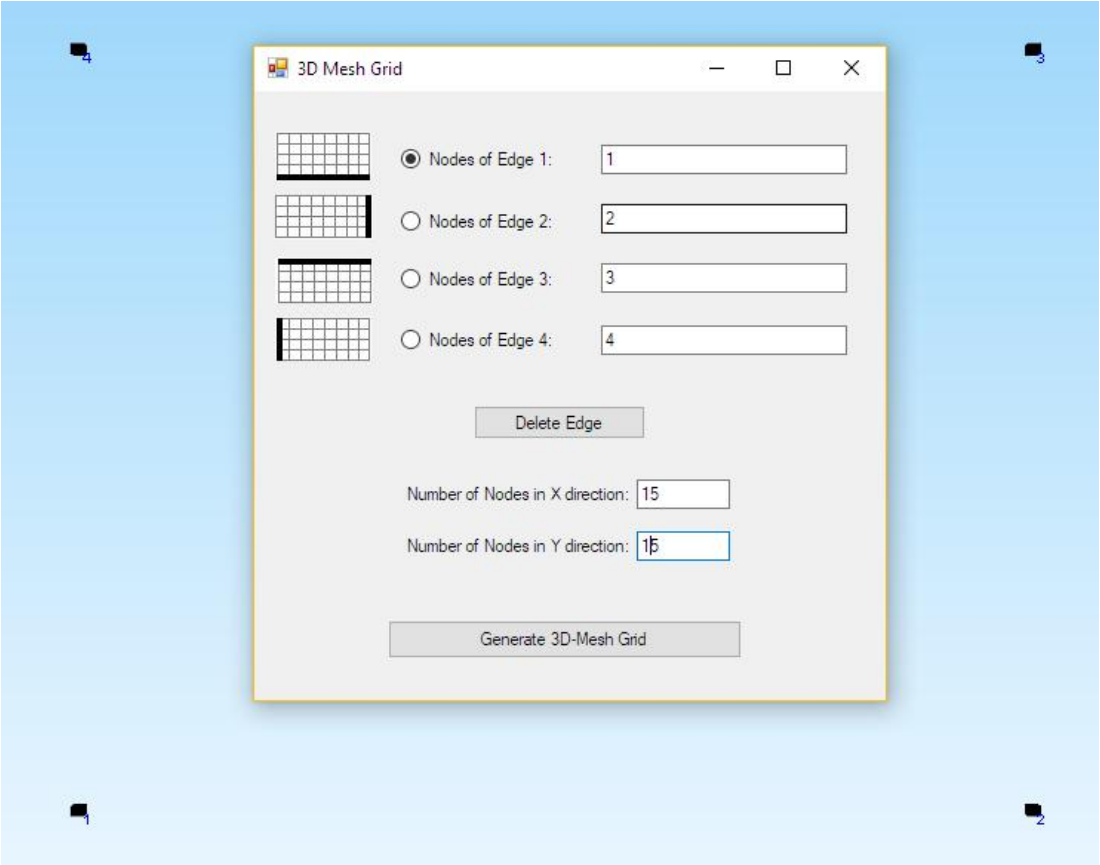
Select "New" and enter, as above, Node 3 with X = 0.25, Y = 0.25 and Z = 0

Node 4:

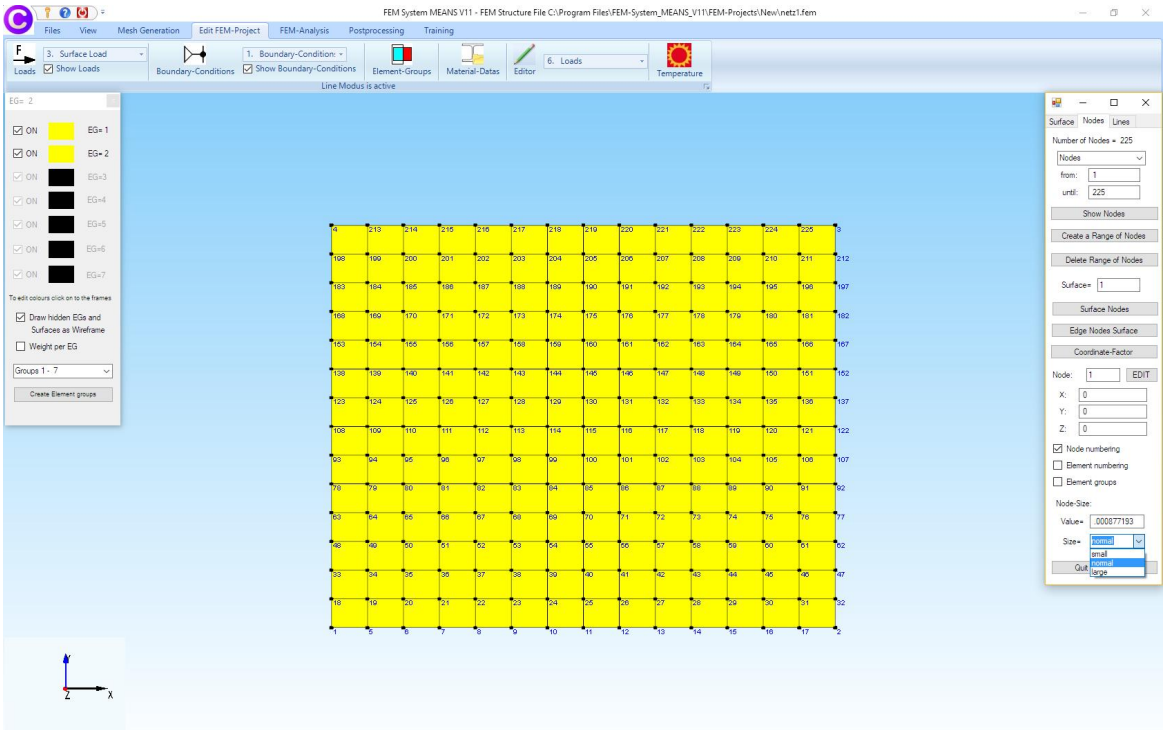
Select "New" and enter, as above, Node 4 with X = 0, Y = 0.25 and Z = 0



Select the menu "3D Mesh Grid" and enter Node 1, 2, 3, 4 as Edge 1,2, 3, 4, then define the Number of Nodes in X direction = 15 and in Y direction = 15 and select "Generate 3D-Mesh Grid".



A FEM model of QUA4S-Elements und 225 Nodes will be generated.



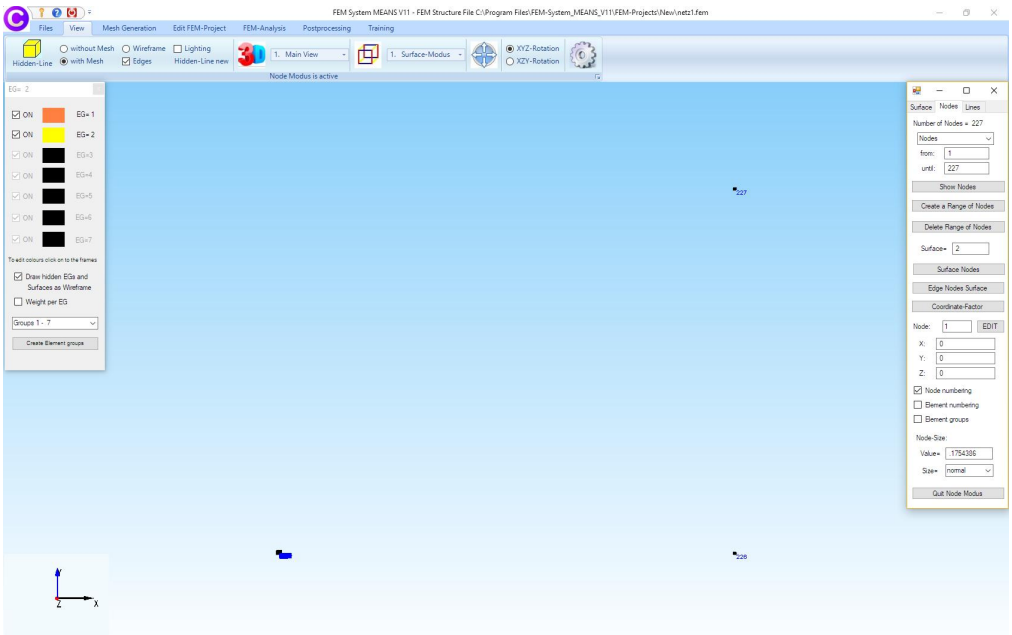
First, switch on the Node-Modus with the "Nodes" tab and show all nodes with a Node numbering, then with "Lines" tab expand the mesh with two new nodes:

Node 226:

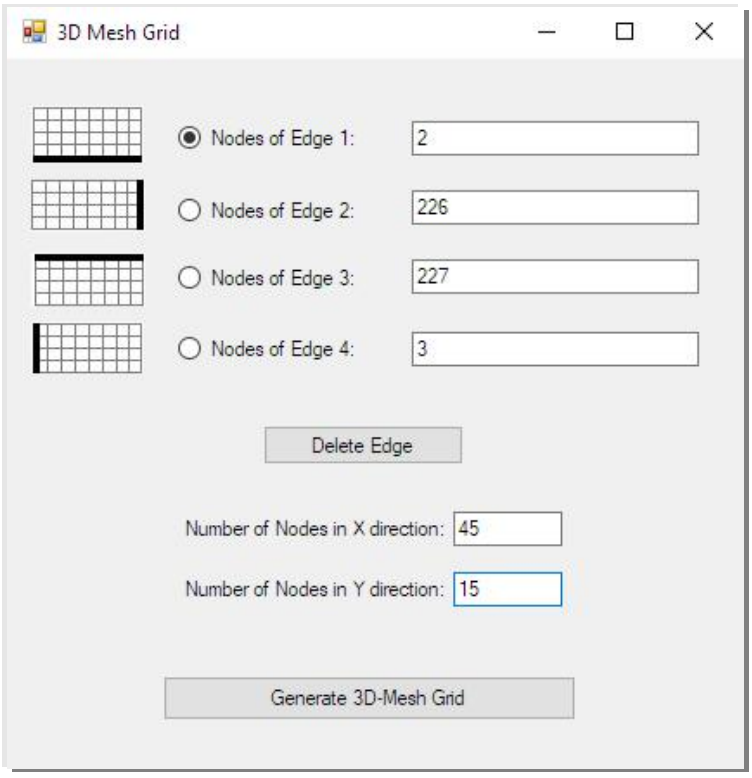
Select "New" and enter node 226 with X = 50, Y = 0 and Z = 0 and select "Create Nodes".

Node 227:

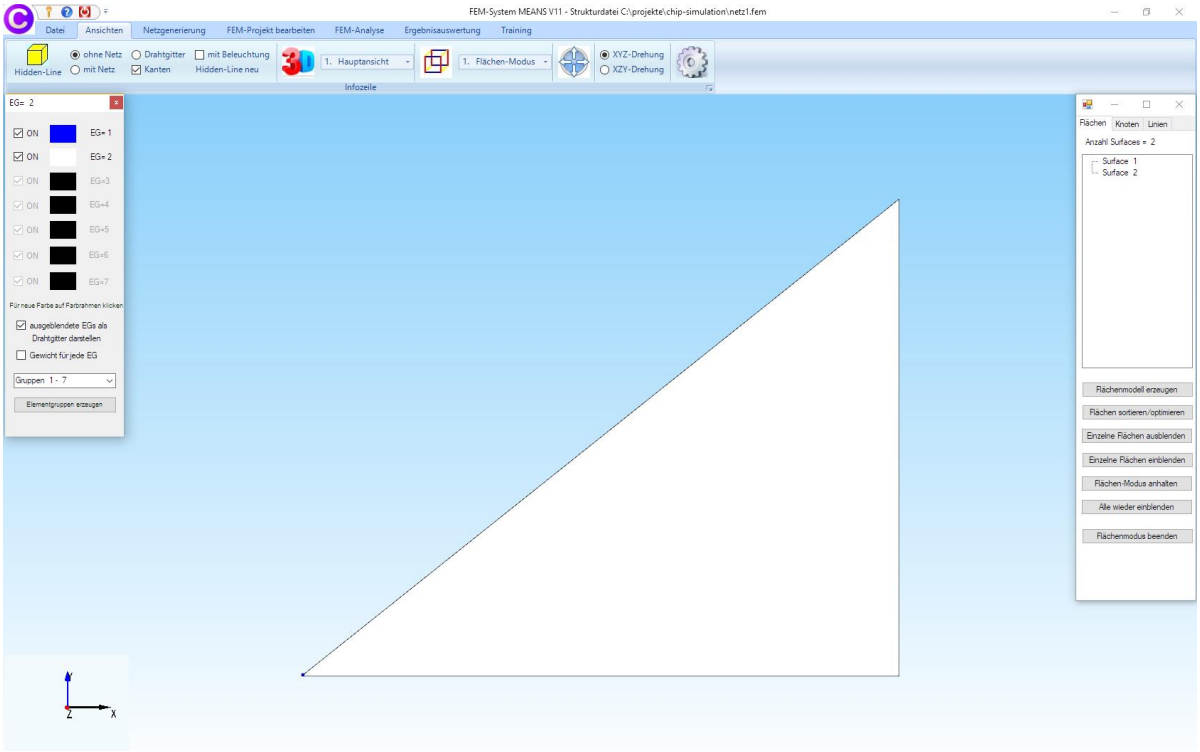
Select "New" and enter node 227 with X = 50, Y = 50 and Z = 0 as above



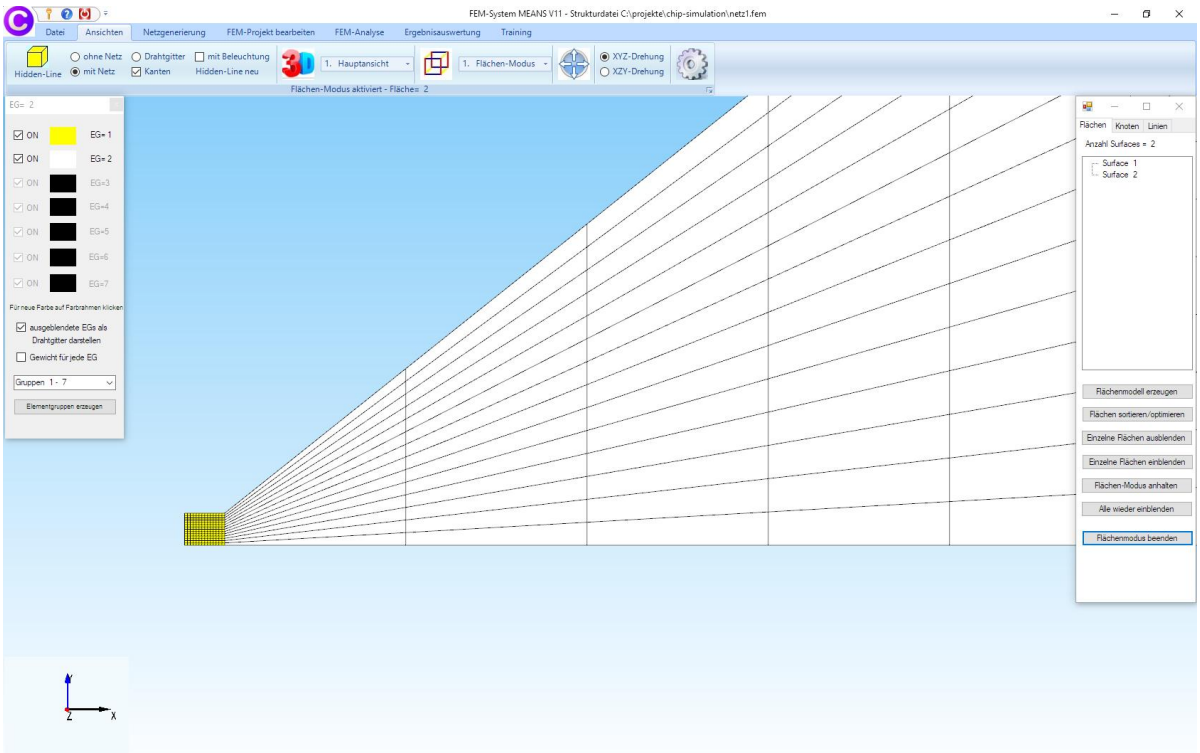
Select the menu "3D Mesh Grid" and enter Node 2, 226, 227, 3 as Edge 1,2, 3, 4



then define the Number of Nodes in X direction = 45 and in Y direction = 15 and select "Generate 3D-Mesh Grid" and generate a FEM model with 812 QUA4S elements, 887 nodes and 2 element groups.
Select the "Edit FEM Project" and "Element Groups" tabs and give element group 2 the color "white" by clicking on the color frame.

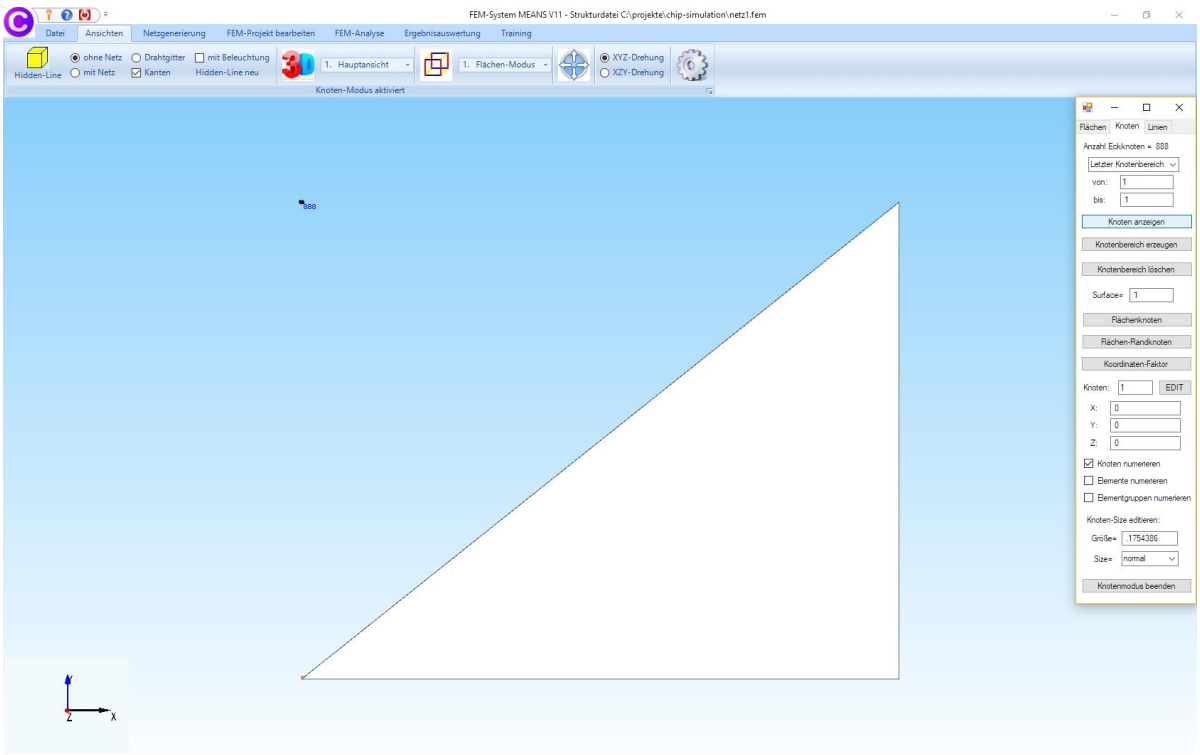


Zoom and position the FEM model in the middle and check the nodes between element group 1 to 2 with "Hidden Line with mesh" and without lighting.

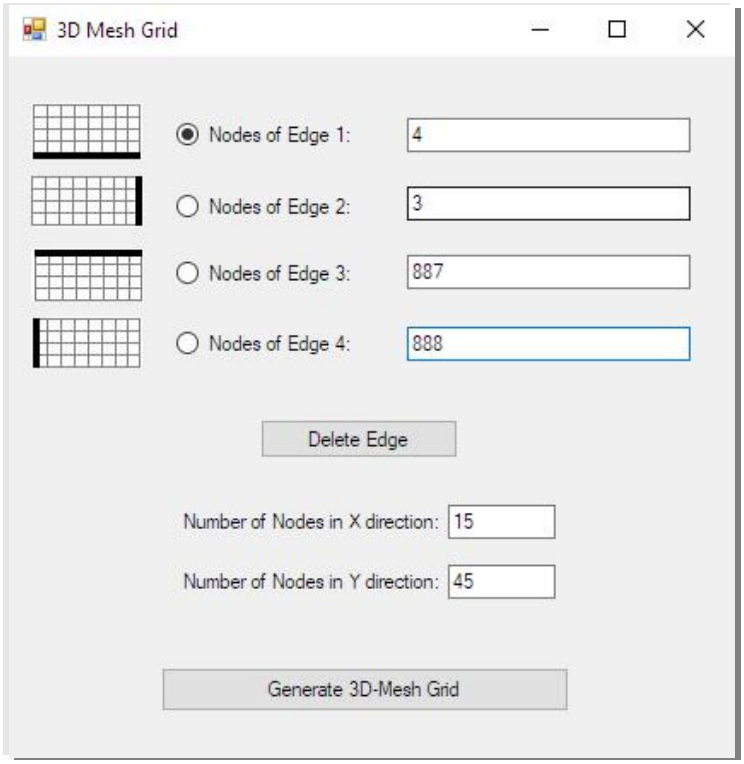


Node 888:

Select „New“ and enter node 888 with X=0, Y=50 and select „Create Nodes“.



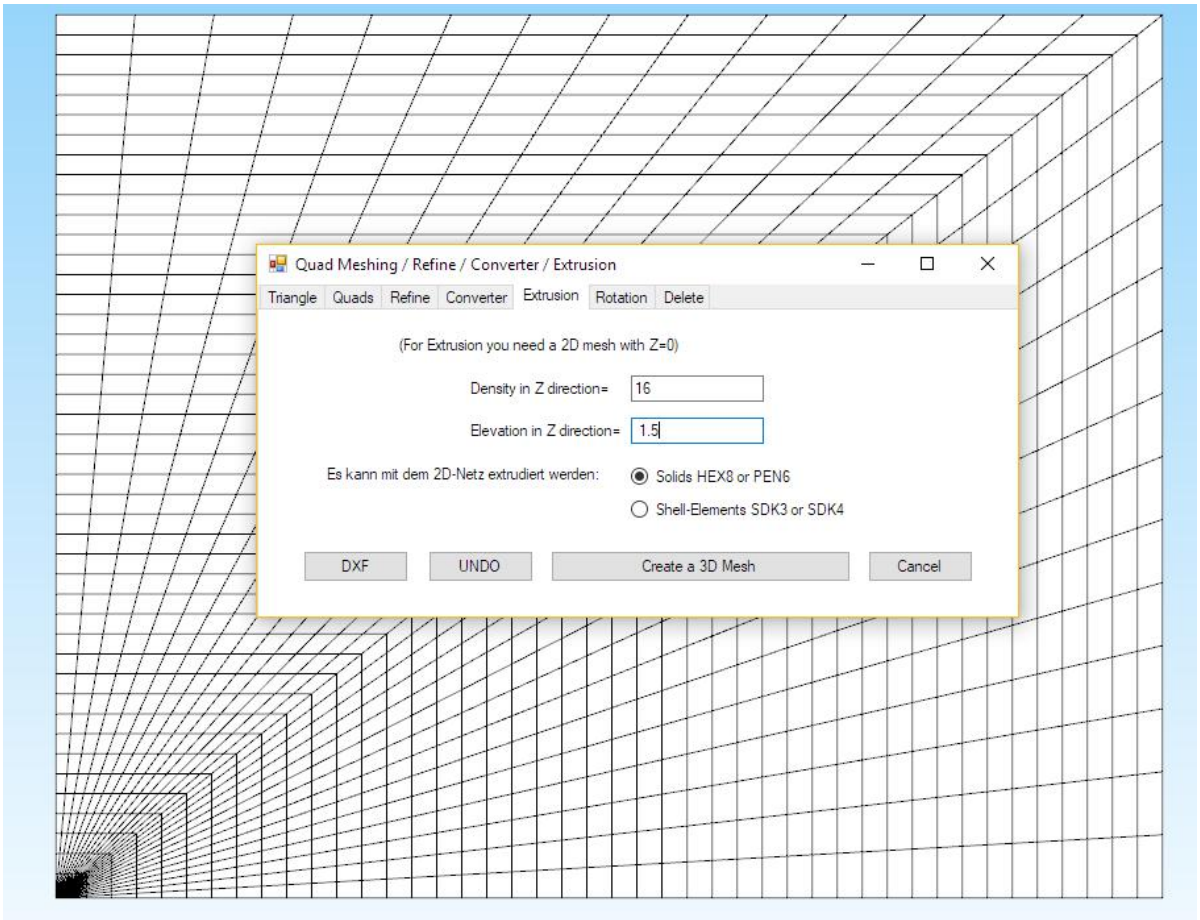
Select „3D-Mesh Grid“ and enter the Nodes 4, 3, 887, 888 as the Edges 1, 2, 3, 4 and define the Number of Nodes in X direction = 15 and in Y direction = 45 and and generate a mesh with 1428 QUA4S elements, 1501 nodes and 3 element groups.



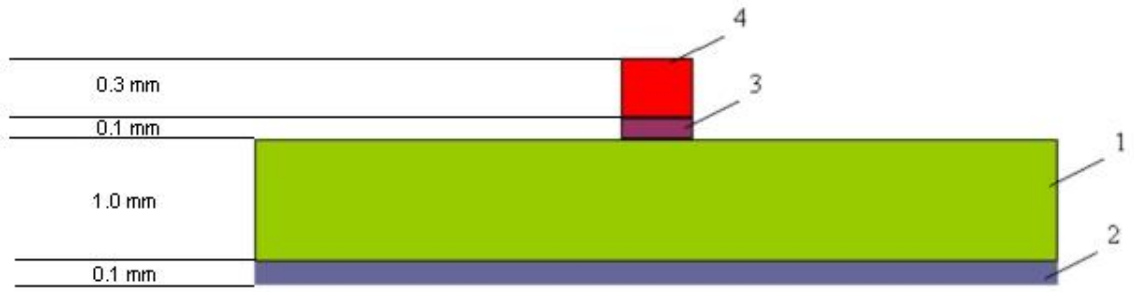
Select "Create element groups" and change element group 3 to 2 and the number of element groups from 3 to 2.

Extrude a Hexahedral FEM-Model

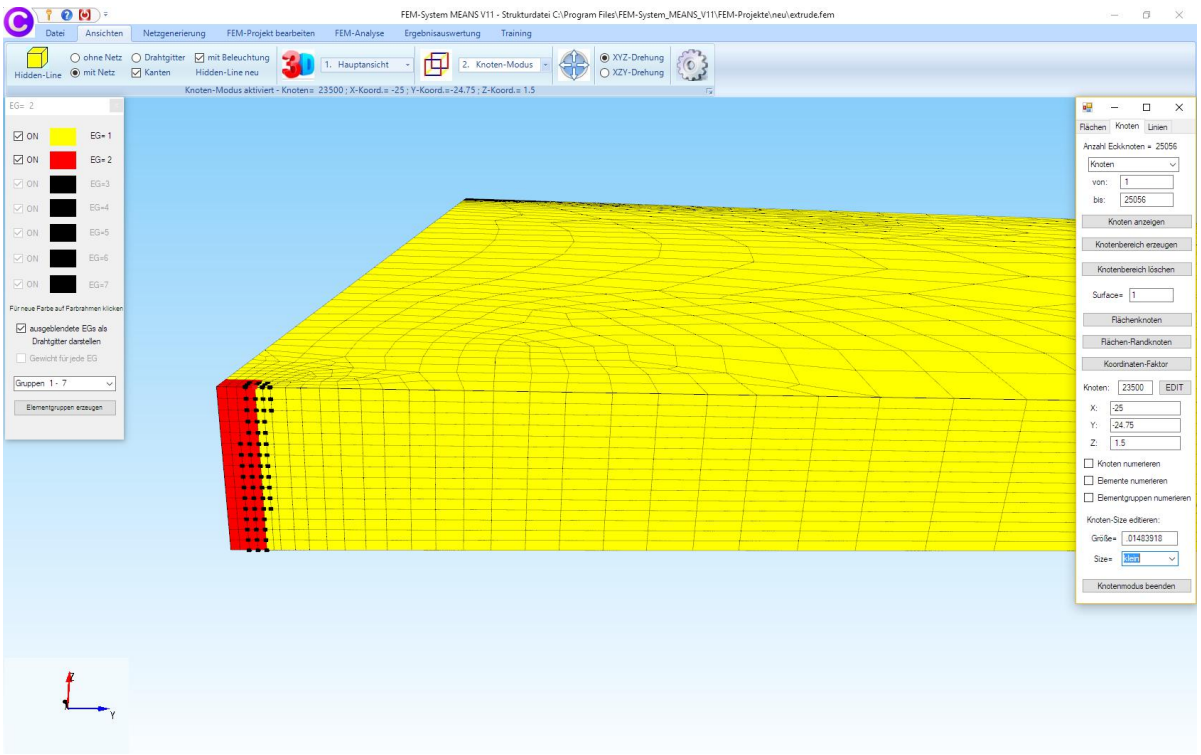
The 2D quadrilateral mesh will now be used as a template for 3D extrusion. Select the "Mesh Generation" tab and the "QUAD Meshes, Refine, Delete" menu and the tab in the new "Extrude" dialog box.



and extrude with the following settings "Density in Z direction = 16" and with "Elevation in Z direction = 1.5 mm" so that every 0.1 mm a row of nodes in the Z direction is created.



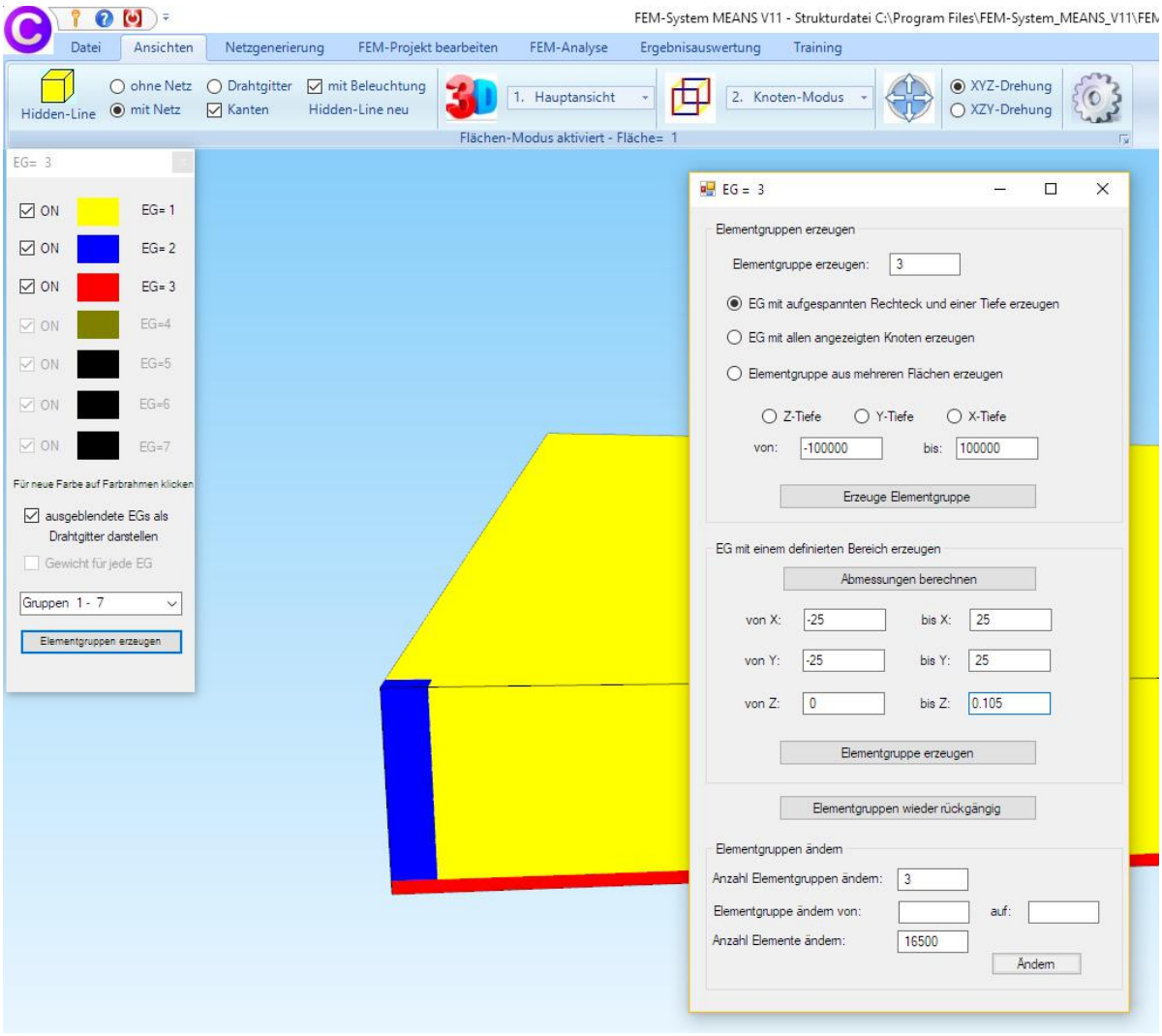
Select „Create a 3D-Mesh“ and a FEM-Model with 18700 HEX8 elements, 21618 nodes and 2 element groups is created.



Adjust the view above by rotating the z-axis with the right mouse button (only with DX9) and zooming the model with the mouse wheel. Set the Node Modus and create a range of nodes and check the Z coordinates of the 16 rows of nodes. Select the "Edit FEM Project" and "Element Groups" tab and click on the color frames to assign any color to the two element groups. You can also show and hide both element groups for practice later.

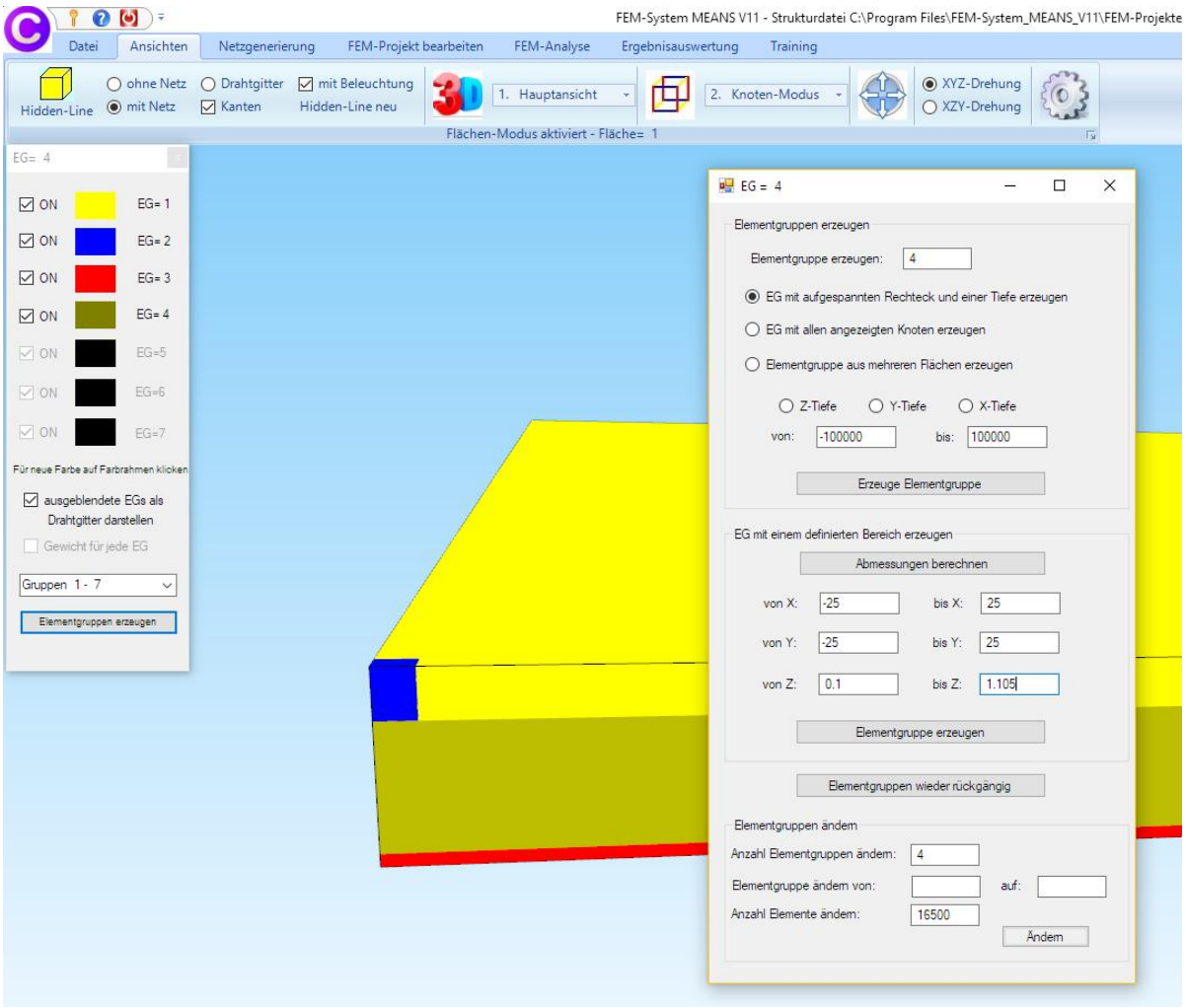
Generation of element group 3

Select the Element Group menu "Create Element Groups" and enter the Element Group 3 first and select "Calculate Dimensions" and enter a Z-Range from 0.0 to 0.105 to create the Element Group 3. Give the element group the color "red".



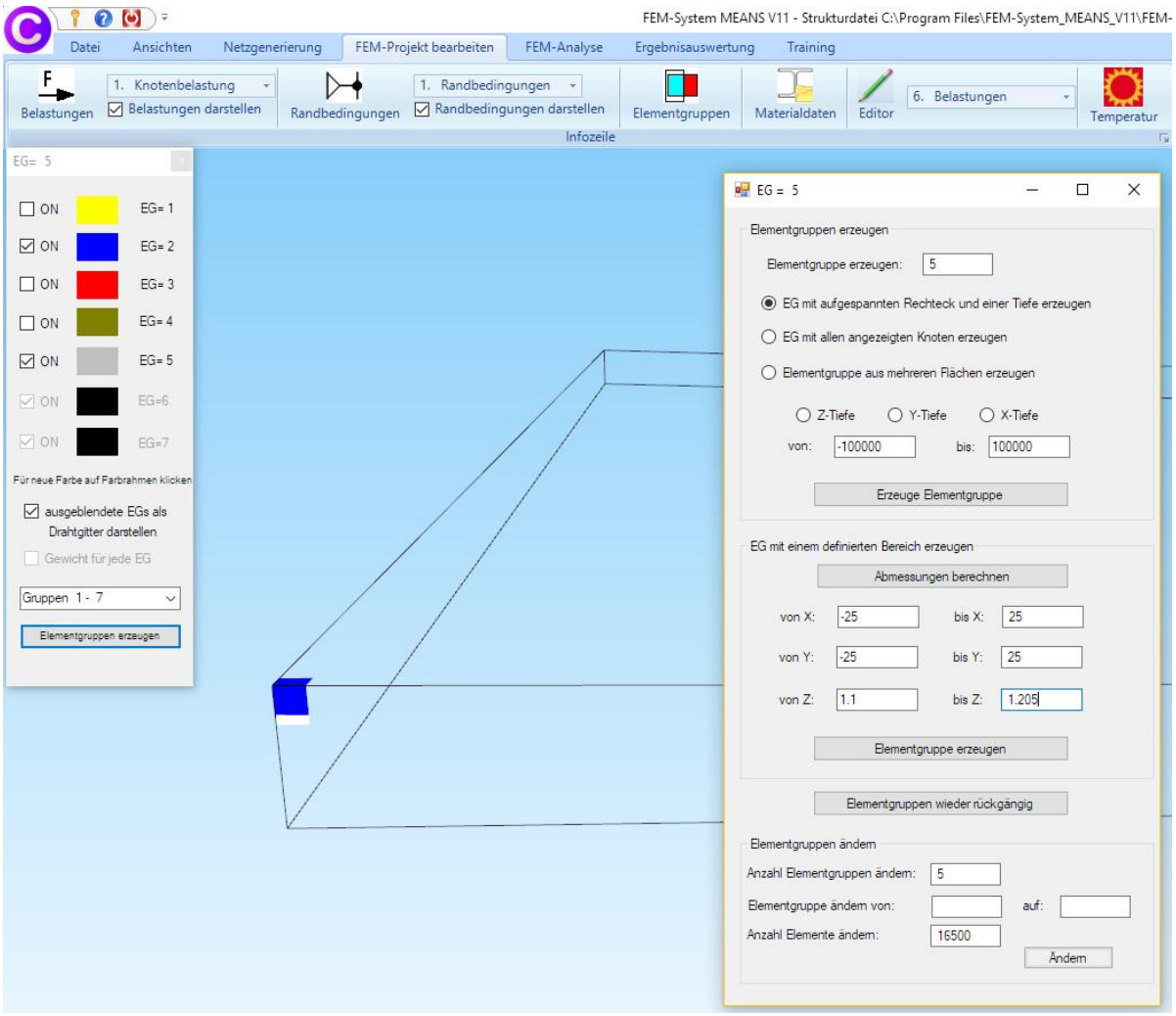
Generation of element group 4

Select the element group menu "Create element groups" and enter the element group 4 first and select "Calculate dimensions" and enter a z-range from 0.1 to 1.105 to create the element group 4. Give the element group the color "green".



Generation of element group 5

It remains the element group for the Chipkleber, show element group 2 and hide all the others 1, 3 and 4 and enter the element group 5 and select again "Calculate dimensions" and enter a Z-range from 1.1 to 1.205 to create the element group 5. Give the element group the color "gray" and show all element groups again.

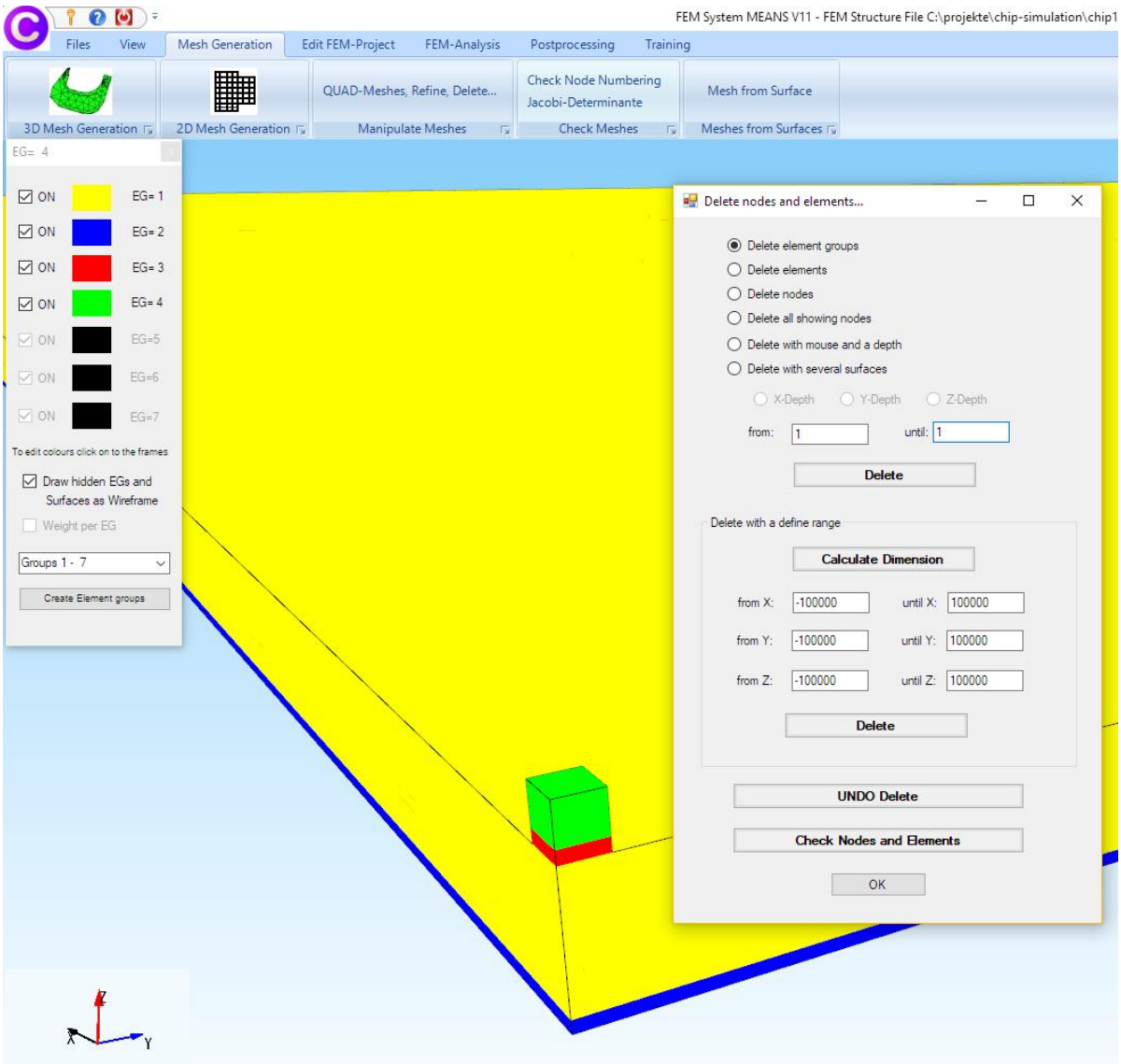


Delete element group 1

Finally, element group 1 is left which still has to be deleted. Select the "Mesh Generation" tab and "QUAD Meshes, Refine ..." and delete the Element Group 1 with the next dialog box.

Alternatively, you can hide the element group 2 and 5 and delete the element group 1 with a defined range of Z = 1.1 to Z = 1.5.

Then a model check is performed automatically and the hidden line is recreated.



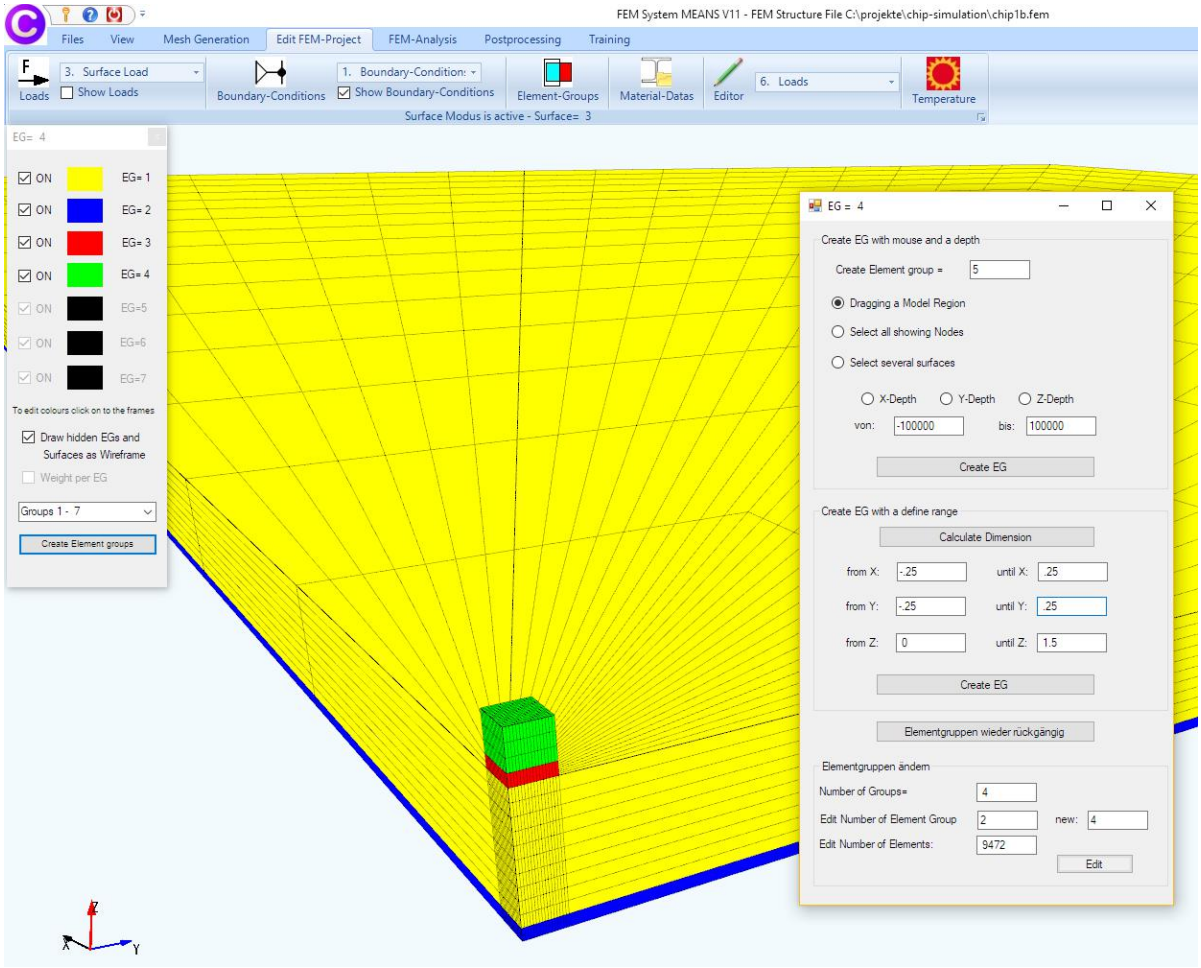
Change element groups

Select „Create Element Group“ and change the element groups as follows:

Element group 4 -> Element group 1
Element group 2 -> element group 4
Element group 3 -> Element group 2
Element group 5 -> Element group 4

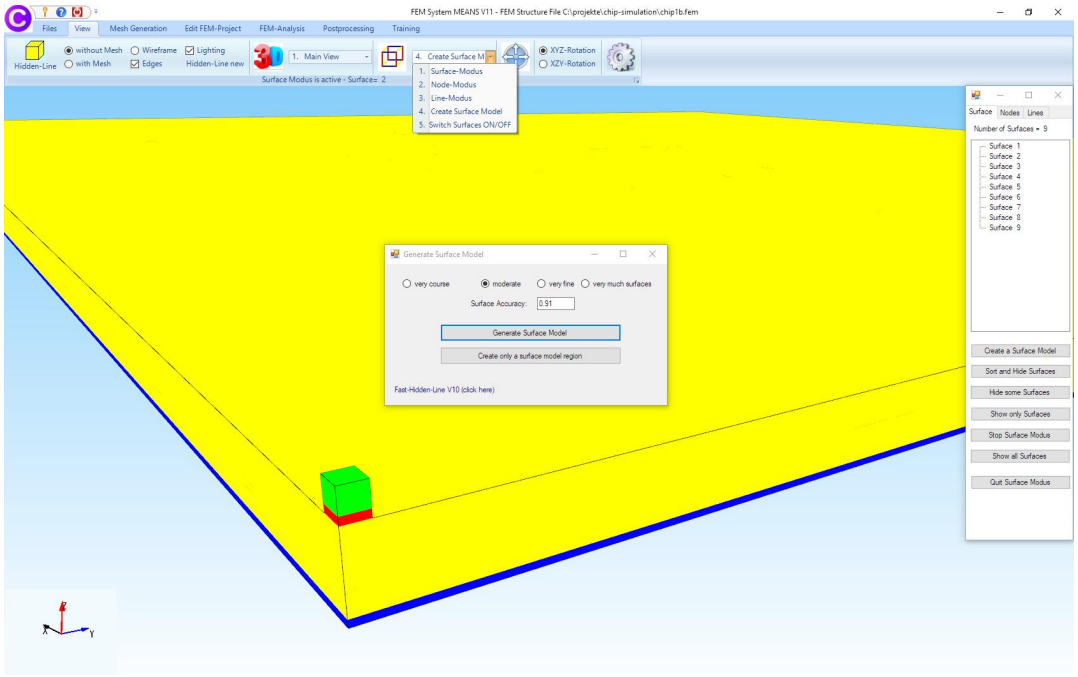
Finally, enter the number of element groups = 4.

The model is now ready and can be saved under the name "Chip-Simulation.fem.
Then you have to enter the heat conductivities, the convections and the point source.


FEM System MEANS V11 - FEM Structure File C:\projekte\chip-simulation\chip1b.fem
Files
View
Mesh Generation
Edit FEM-Project
FEM-Analysis
Postprocessing
Training
Loads
3. Surface Load
Show Loads
Boundary-Conditions
1. Boundary-Condition:
Show Boundary-Conditions
Element-Groups
Material-Datas
Editor
6. Loads
Temperature
Surface Modus is active - Surface= 3
EG= 4
ON EG= 1
ON EG= 2
ON EG= 3
ON EG= 4
ON EG=5
ON EG=6
ON EG=7
To edit colours click on to the frames
Draw hidden EGs and Surfaces as Wireframe
Weight per EG
Groups 1 - 7
Create Element groups
EG = 4
Create EG with mouse and a depth
Create Element group = 5
Dragging a Model Region
Select all showing Nodes
Select several surfaces
X-Depth
Y-Depth
Z-Depth
von: -100000
bis: 100000
Create EG
Create EG with a define range
Calculate Dimension
from X: -.25
until X: .25
from Y: -.25
until Y: .25
from Z: 0
until Z: 1.5
Create EG
Elementgruppen wieder rückgängig
Elementgruppen ändern
Number of Groups= 4
Edit Number of Element Group 2
new: 4
Edit Number of Elements: 9472
Edit
Z
X
Y

Create Surface Model

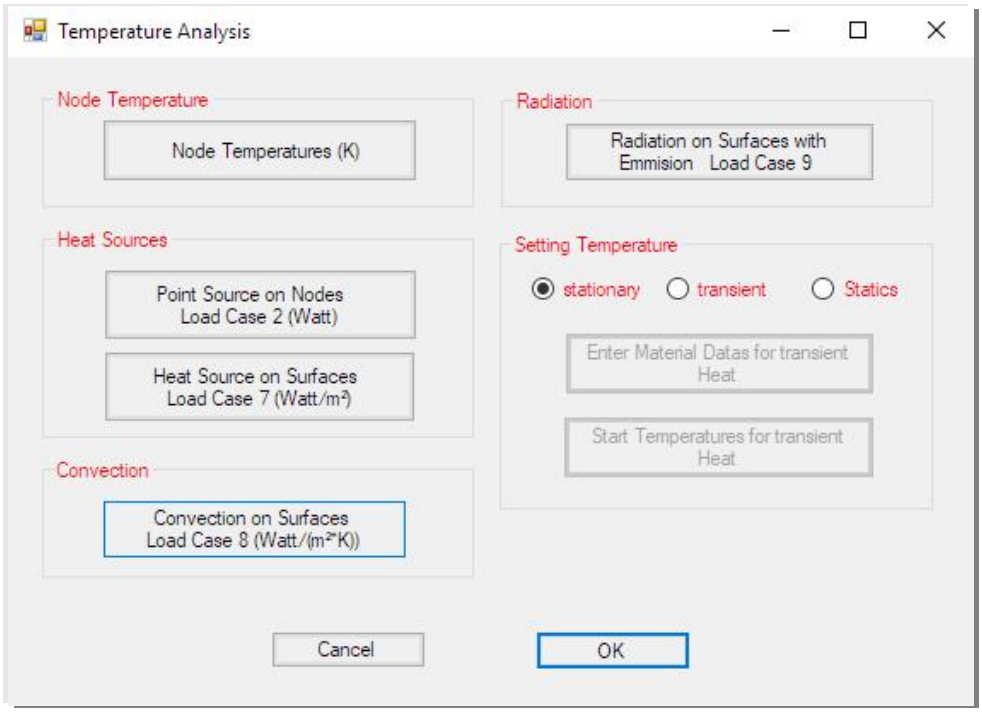
Select the "View" tab and "Create Surface Model" to create a surface model with 9 surfaces, the surface turn in red as soon as you move the mouse over the surface.



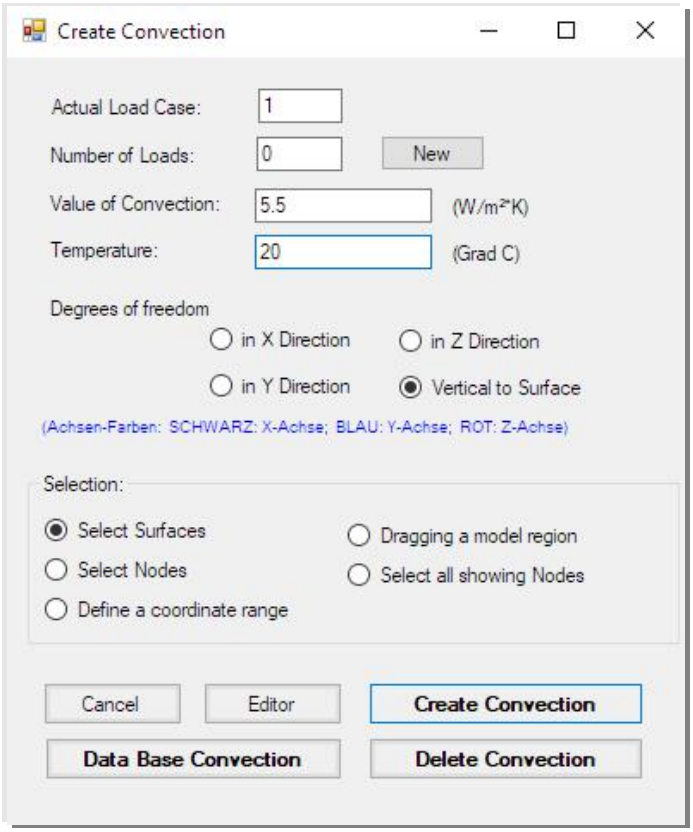
Create Convection

Select the "Edit FEM project" tab and 

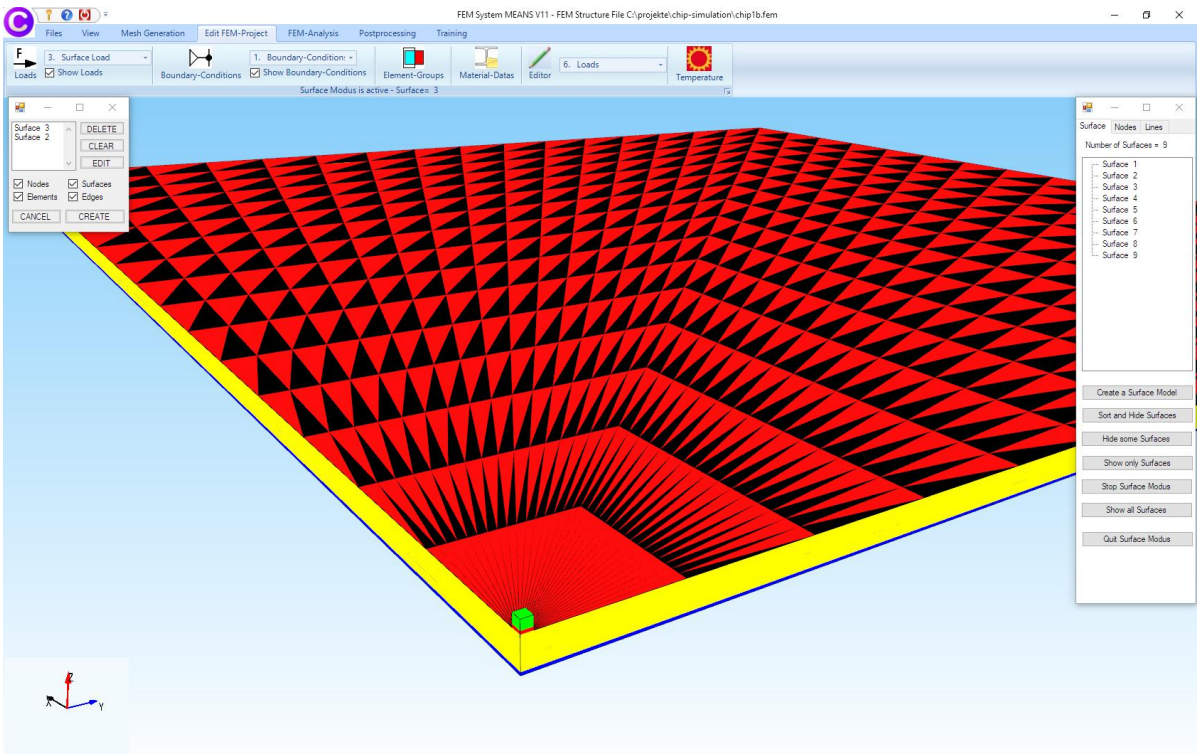
and choose in the next dialogbox menu "Convection on Surfaces Load Case 8 (Watt/(m²K)" and enter the Value of Convection of 5.5 W/(m²K) and the Temperature of 20 Grad C on the top side of the board with surface 3 and on the underside with surface 2.



Select „Create Convection“ and double-click on Surface 3 and Surface 2.



The surfaces are listed in the selectbox, there select „Create“ to generate the convection.

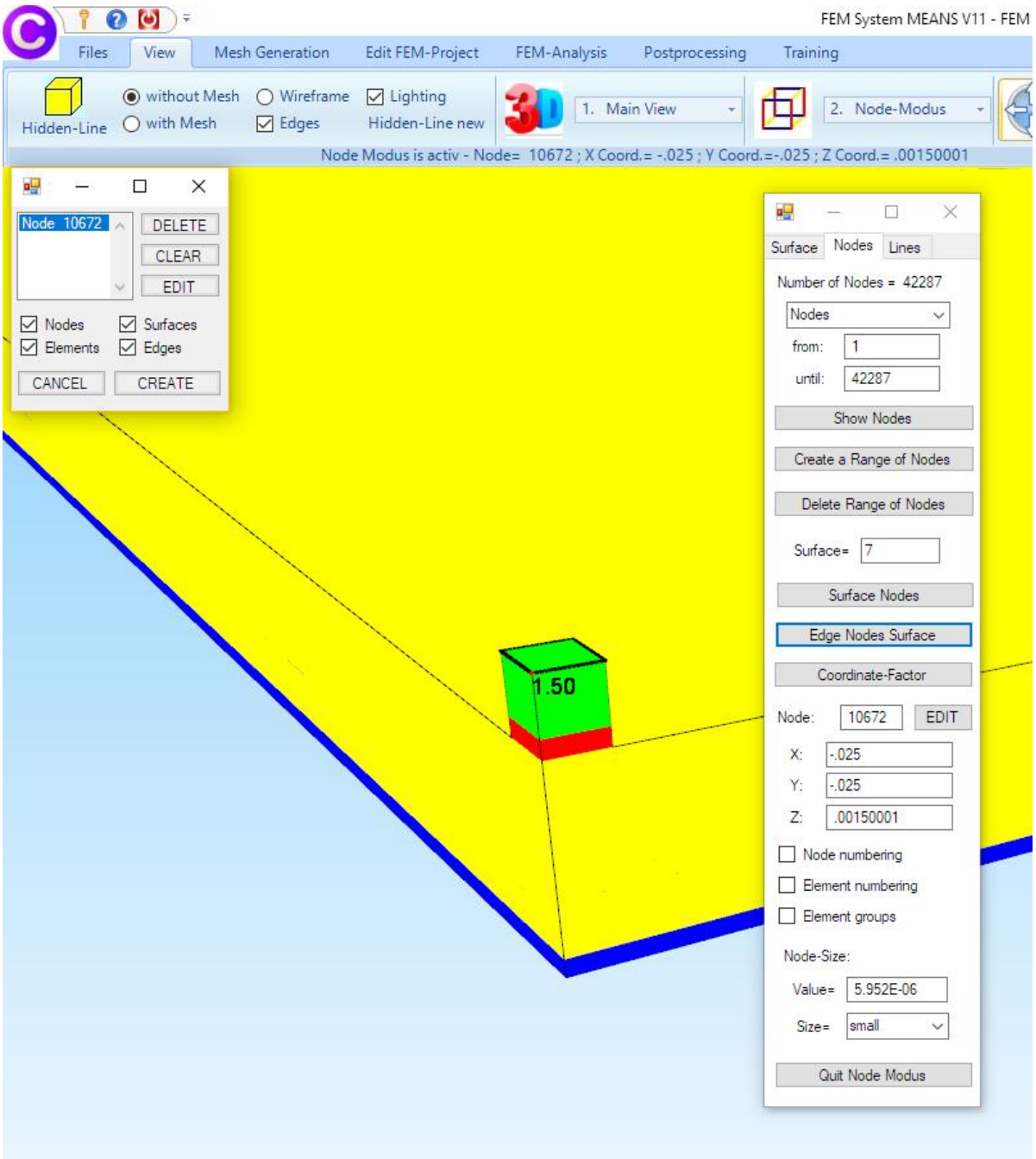


Create a Point Source

Hide the convection with "Edit FEM Project" tab and "Show Loads". Create with the „View“ tab and „Node-Modus“ a Range of Nodes of the Chip Surface 7. If the nodes are displayed too large, they can be zoomed down with node size and "small".

Select the "Edit FEM project" tab and 

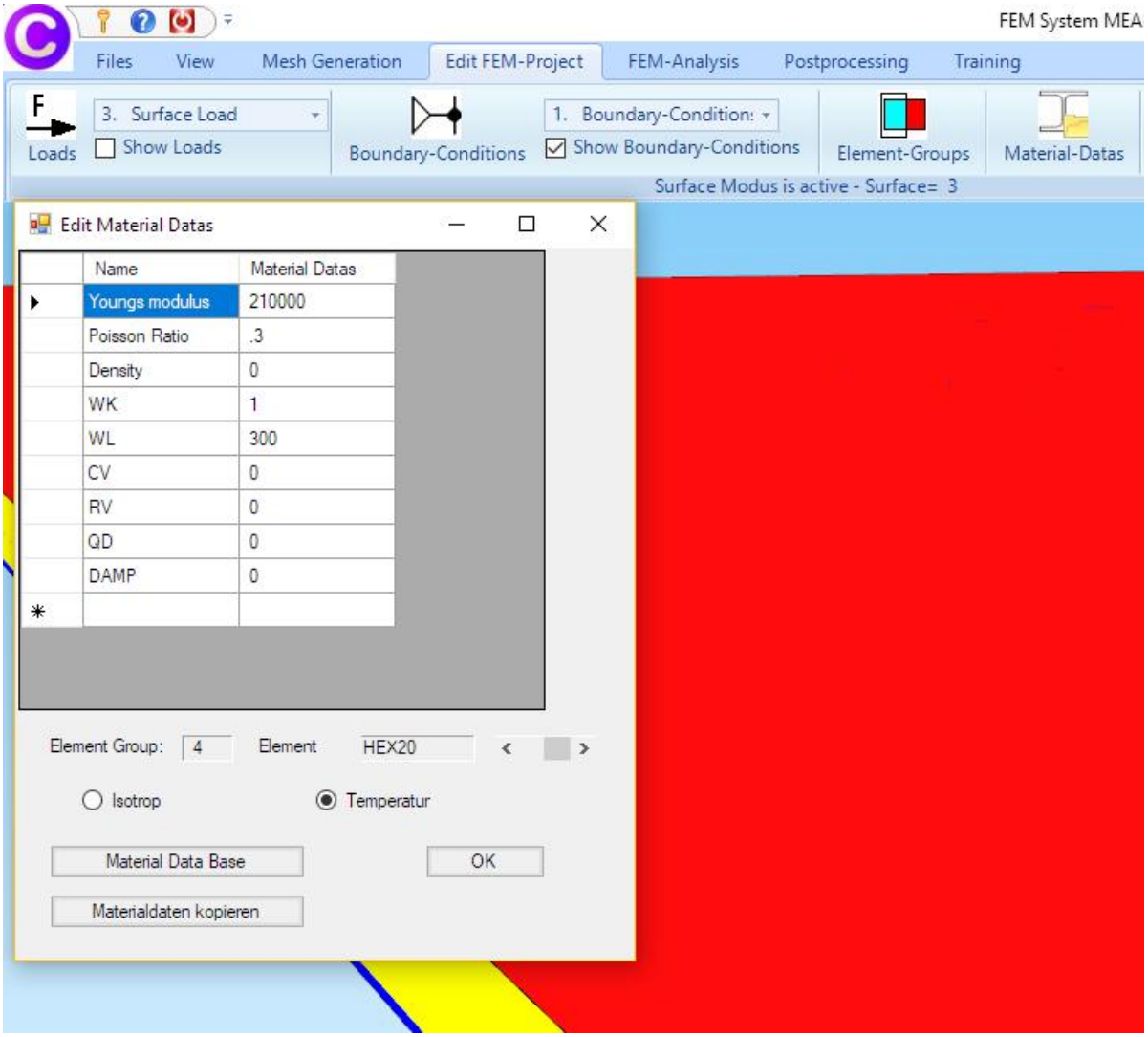
and choose in the next dialogbox menu "Point Source on Nodes Load Case 2 (Watt)“ and enter the Load Case 2 and enter the value of point source 1.5.
Than select „Create a Point Source“ and double-click on Node 10672. The Node is listed in the selectbox, there select „Create“ to generate the point source.



Enter Heat Conductivities

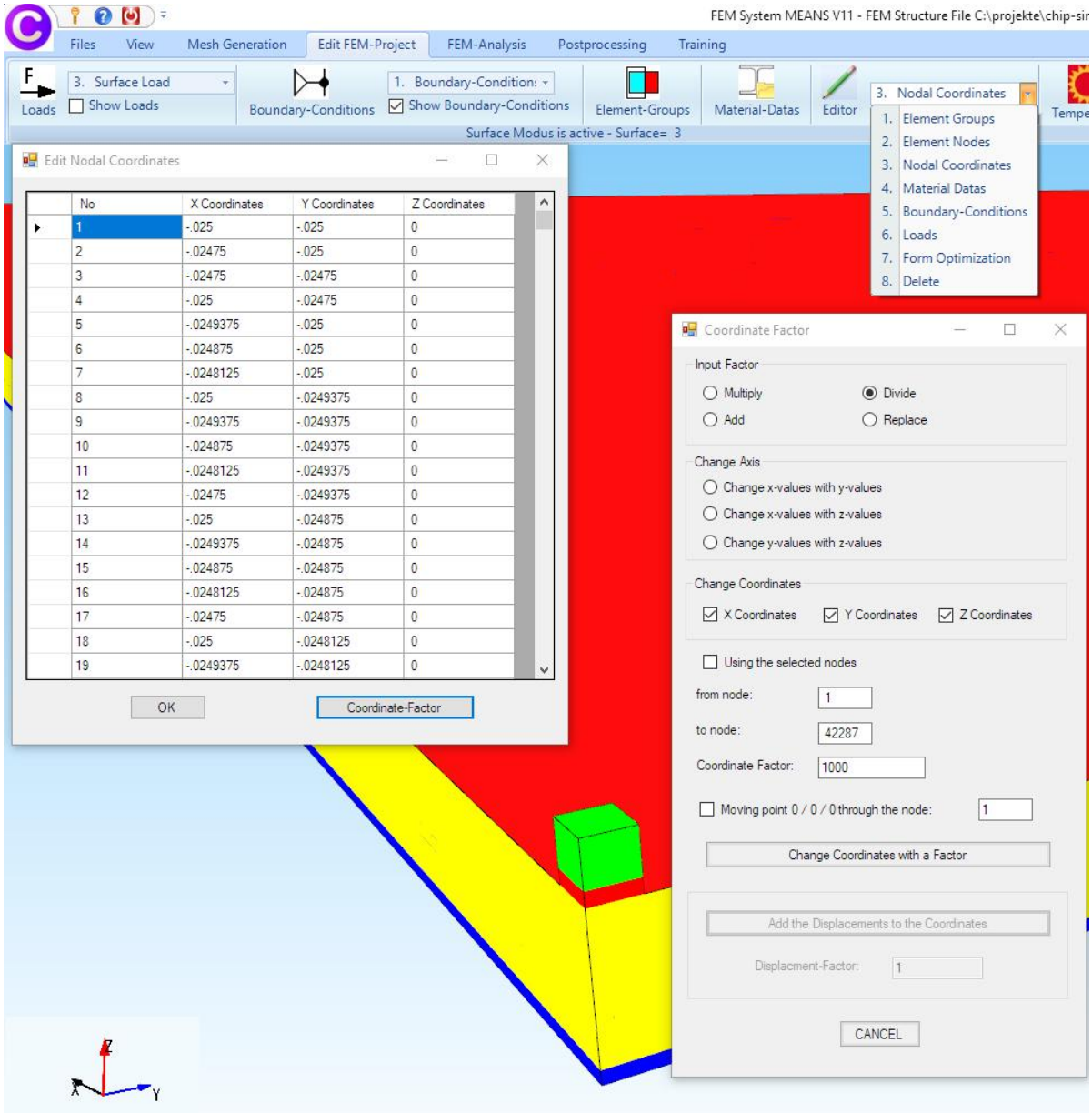
Select „Edit FEM-Project“ tab and „Material-Datas“ and enter the heat conductivities for the element groups 1 - 4:

λ_1 = 1 W / m*K (Board)
λ_2 = 30 W / m*K (Leiter)
λ_3 = 40 W / m*K (Chipkleber)
λ_4 = 300 W / m*K (Chip)



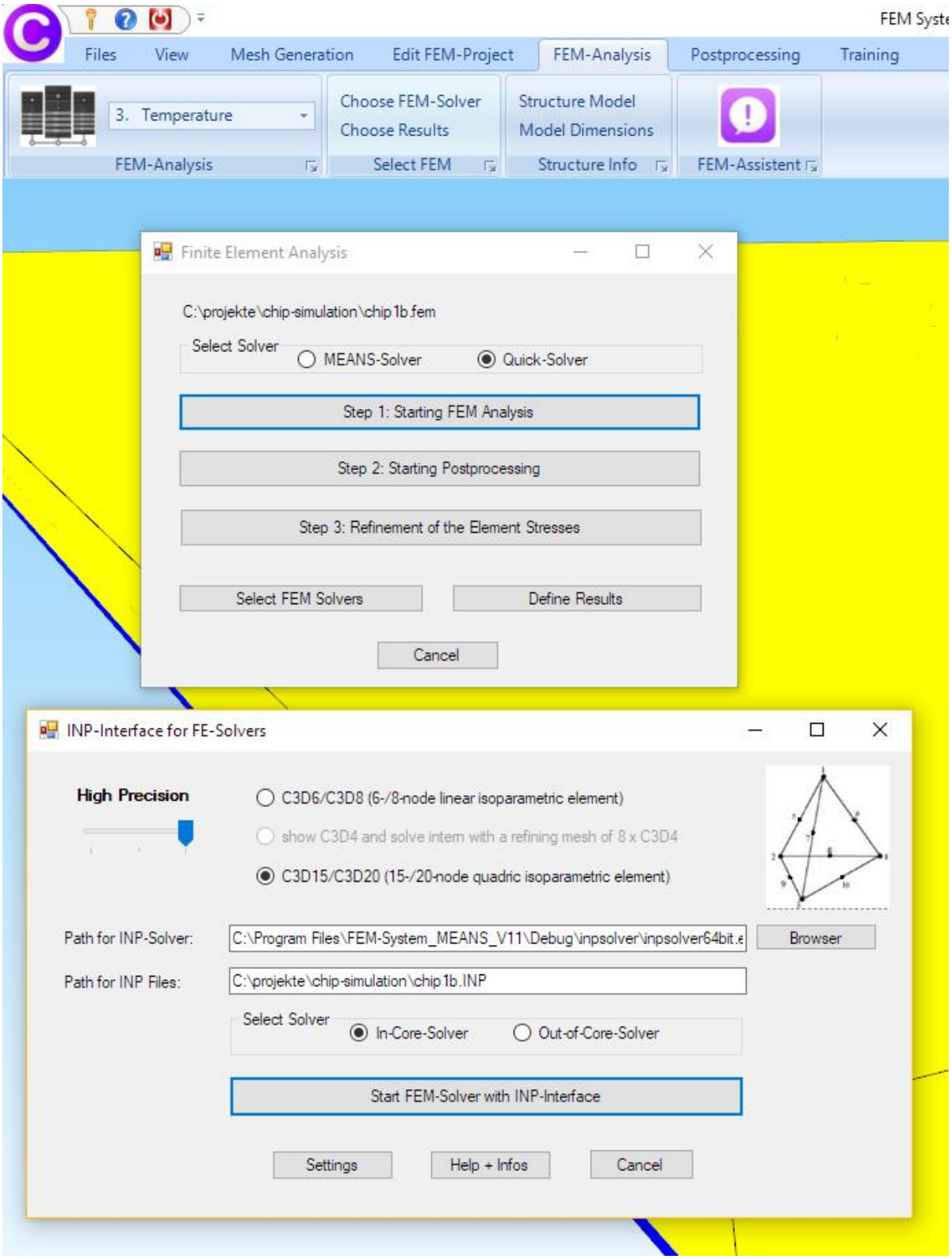
Change Nodal Coordinates

Finally, the coordinates have to be changed from millimeters to meters by dividing the coordinates by a coordinate factor of 1000. Select "Edit FEM Project" tab with menu „3. Nodal Coordinates“ and select in the next dialogbox "Coordinate-Factor" and set "Divide" and enter a factor of "1000" and select "Change Coordinates with a Factor“.



FEM-Analysis

Select „FEM-Analysis“ tab and „Quick-Solver“ to start the FEM-Solver.



Select „C3D15/C3D20 (15-/20-node quadratic....) to calculate the node temperature with HEX20-Solid elements.

Postprocessing

Select „Postprocessing“ and the Icon  to evaluate the node temperature.