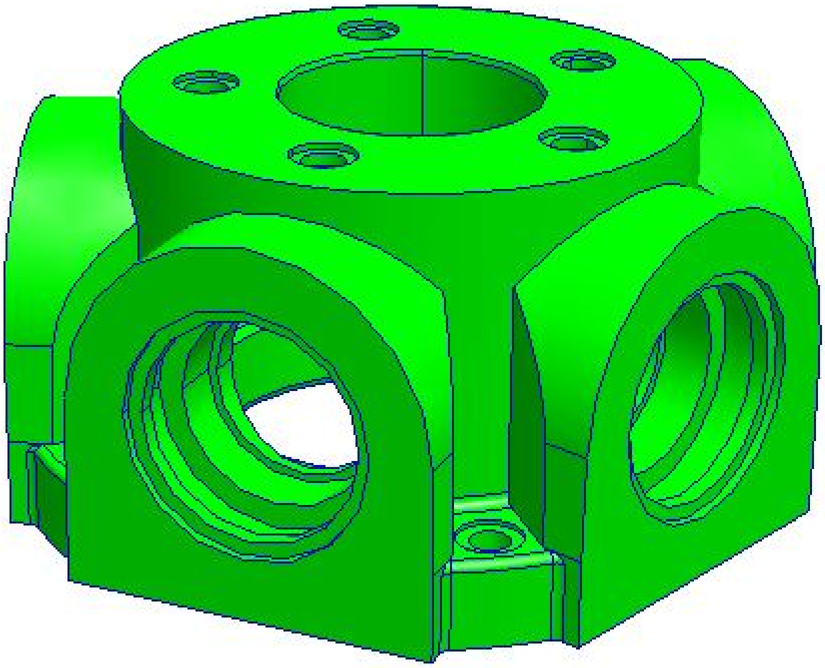
Part 14: FEM-Simulation of a Hydro-Casing with MEANS V10

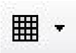
A Hydro-Casing made of steel is loaded with a pressure load of 30 tons on each flange surface. Calculate the displacements and the v.Mises-stresses in the five bolt holes?

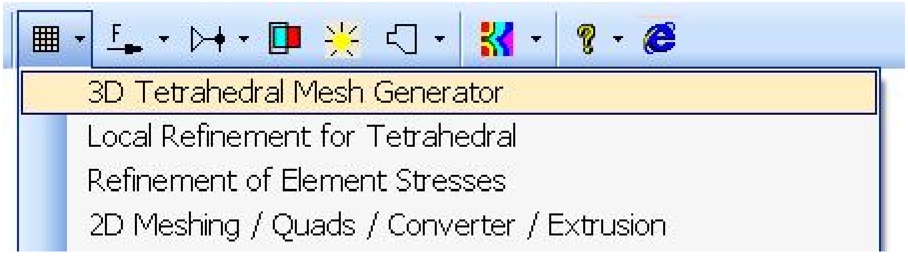


Because the STEP model is available, the Hydro-Casing can be meshing with the automatic 3D mesh generator and be calculating with the Quick-Solver very fast.

Start by double-clicking on the desktop icon to start “MEANS V10 for DirectX9”. Select "New Project" and "3D Unstructured Tetrahedral Meshing” to load the CAD file into the 3D mesh generator.

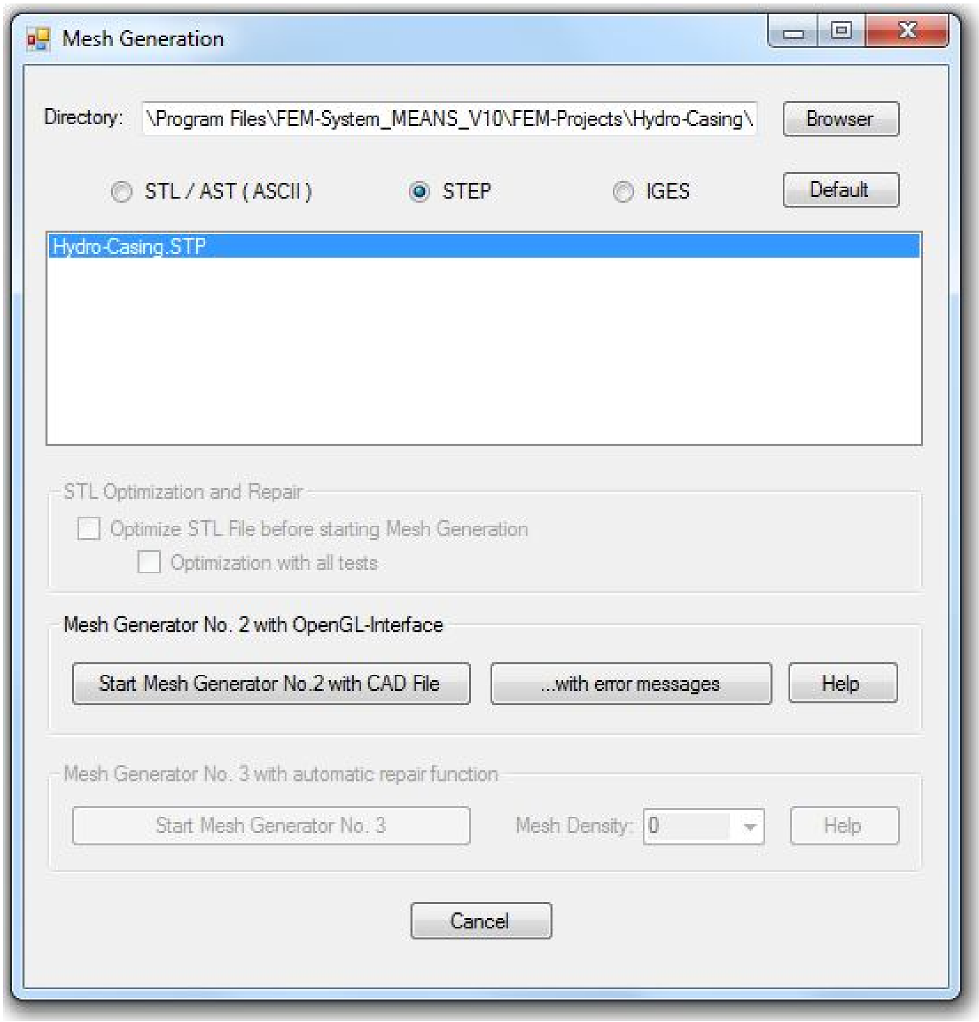
Generate Mesh

Or select in the view bar the icon  and the drop-down menu „3D Tetrahedral Mesh Generator".



to show a new dialogbox for mesh generation with the supported CAD formats:

DXF mainly for 2D mesh generation or for 3D beam elements
STL consists of triangle facets for 3D mesh generation
STEP consists of solid elements and is the most appropriate 3D format
IGES older CAD format but still in use

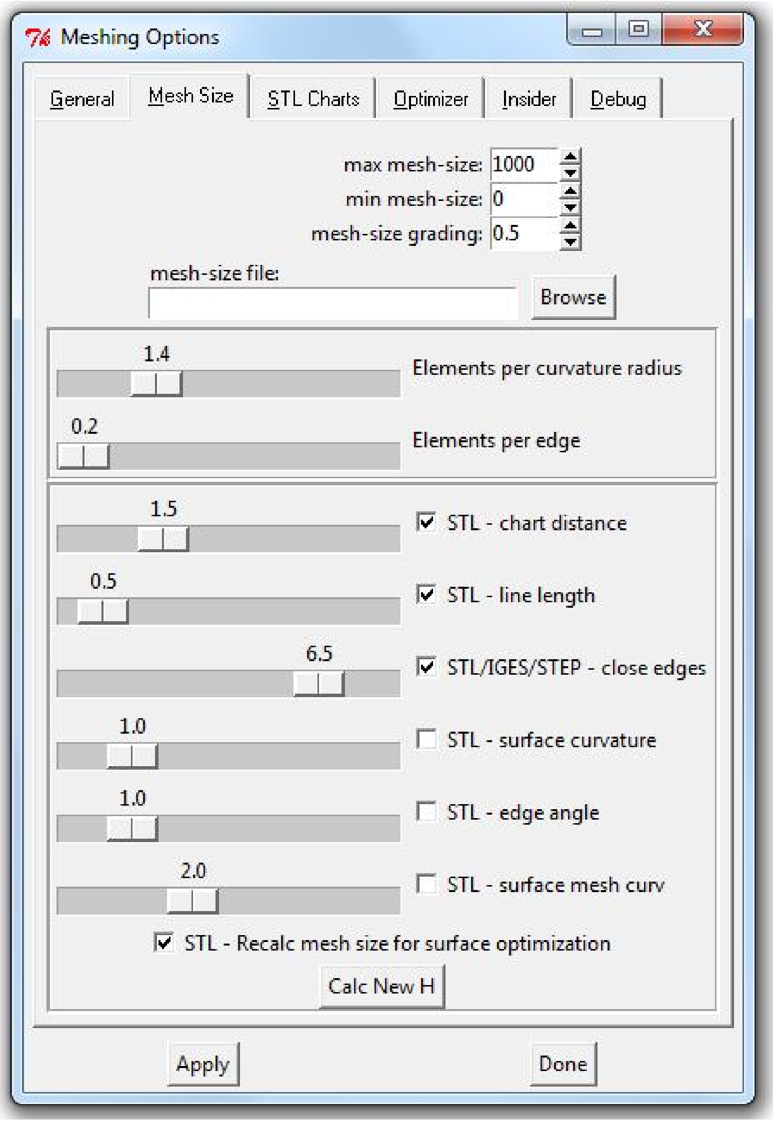


Mesh Generator No. 2

Choose “STEP” and set with menu “Browser” the the step file path:
C:\Programs\FEM-System_MEANS_V10\FEM_Projects\Hydro-Casing
and select the STEP file "hydro-casing.stp and menu "Start Mesh generator No. 2 start with CAD file" to start Netgen.

Mesh granularity

Selct the register “**Mesh-Size**” with menu „**Mesh**“ and „**Meshing Options**“ in order to make following setting:

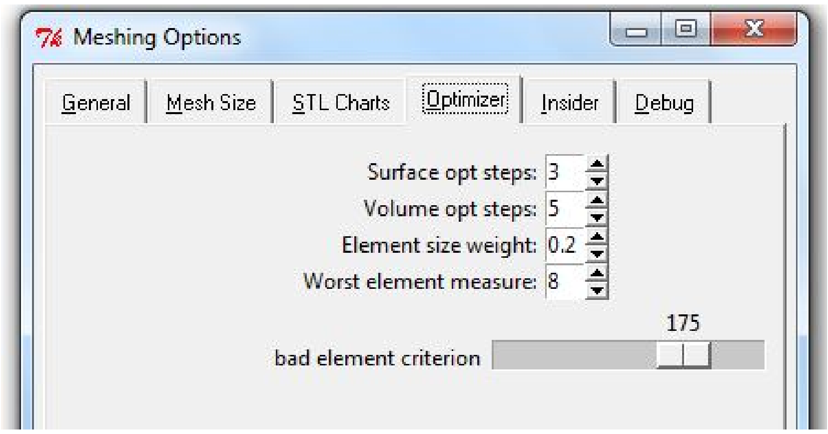


Generate Mesh

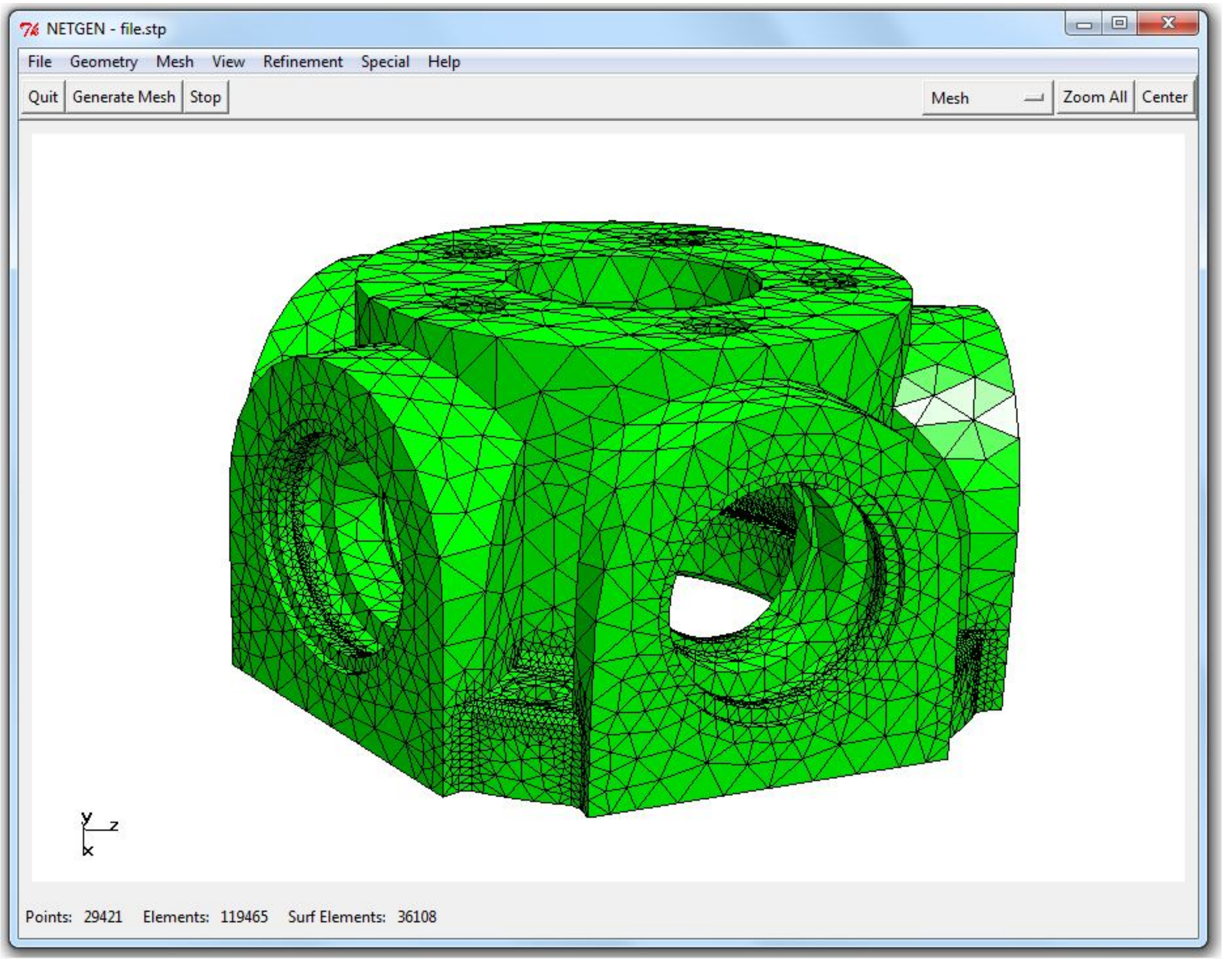
After setting choose „Generate Mesh“ to create a FEM model with 29 887 nodes and 121 564 tetrahedral elements.

EasyFEM-Limit

In order to reach the allowable limit for the MEANS-Light Version EasyFEM until 120 000 elements we must selected additionally the register “Optimizer” to edit the “Worst element measure” from “2” to “8”.

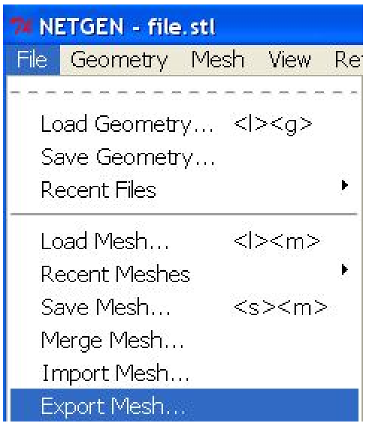


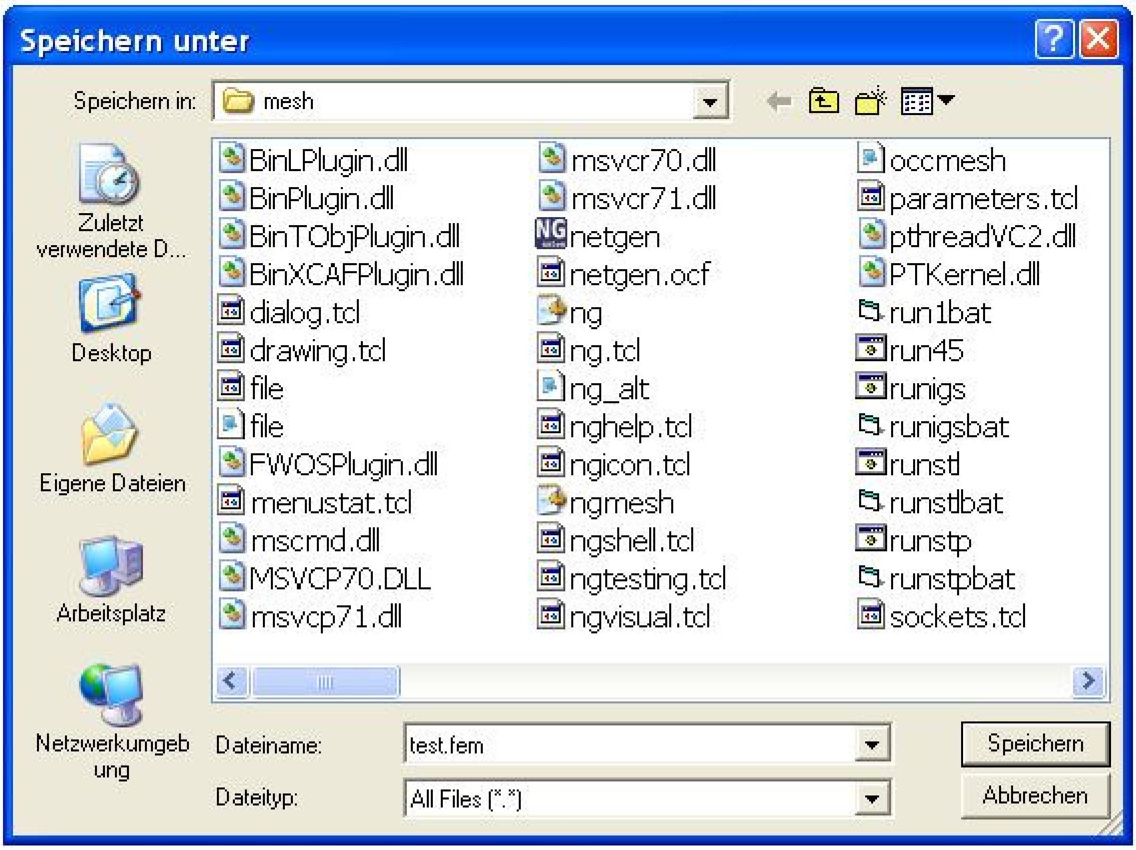
After a new generation we reach the limit with 119 465 tetrahedral elemens just below.



Export with "test.fem"

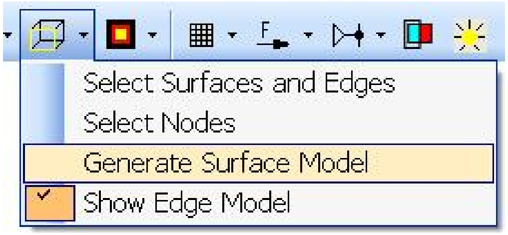
Now export the mesh under the name "test.fem" with the menu "File" and "Export Mesh". The path name must not be changed, otherwise the automatic loading of the mesh into MEANS does not work. The mesh generator window can be closed.



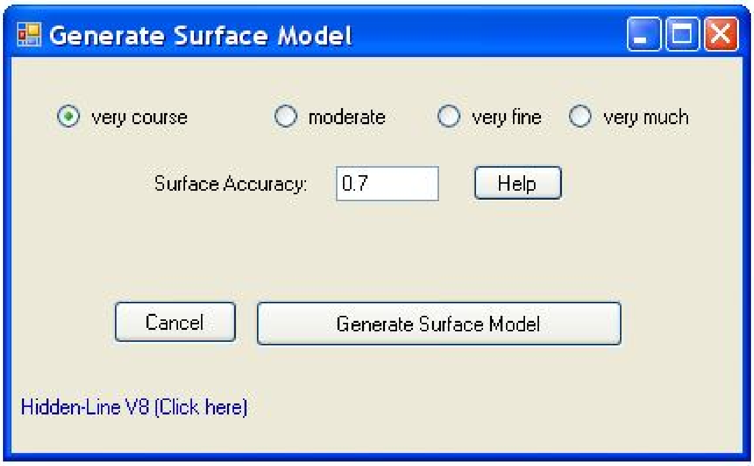


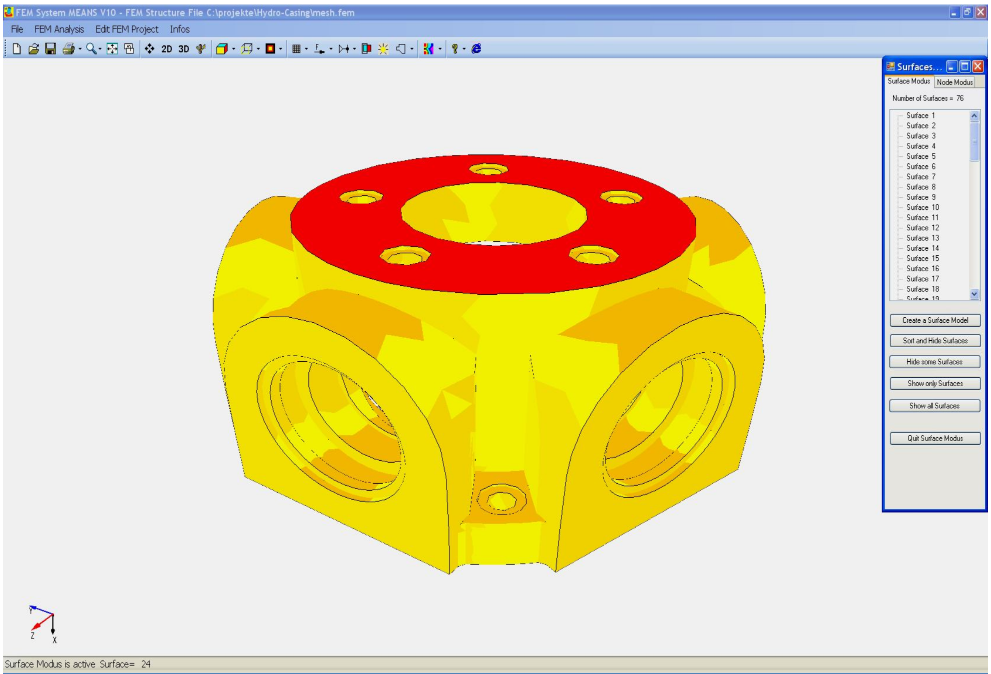
Create a Surface Model

Choose menu „Generate Surface Model“ in order to create a surface model for selecting single surfaces for loads and boundary conditions. Also you can show or hide surfaces from the model so it is possible to select the inside surfaces.



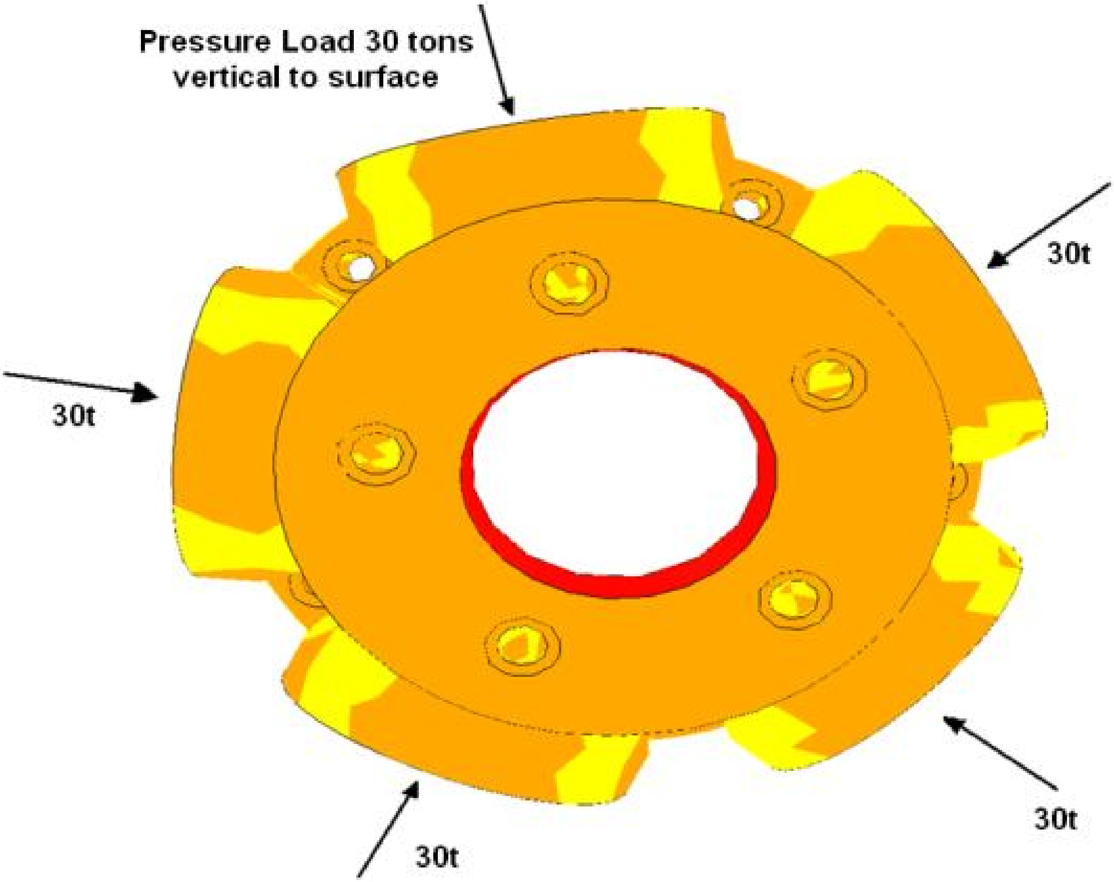
For create the surface model select „very course“ in order to create a surface model with 76 surfaces.





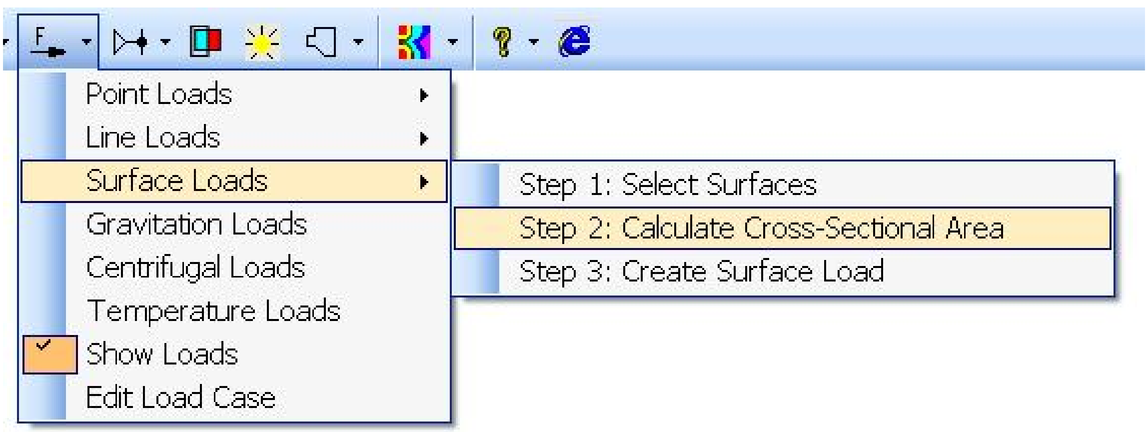
Create surface load

Each of the 5 flange surface is loaded with a vertical pressure load of 30 tons.

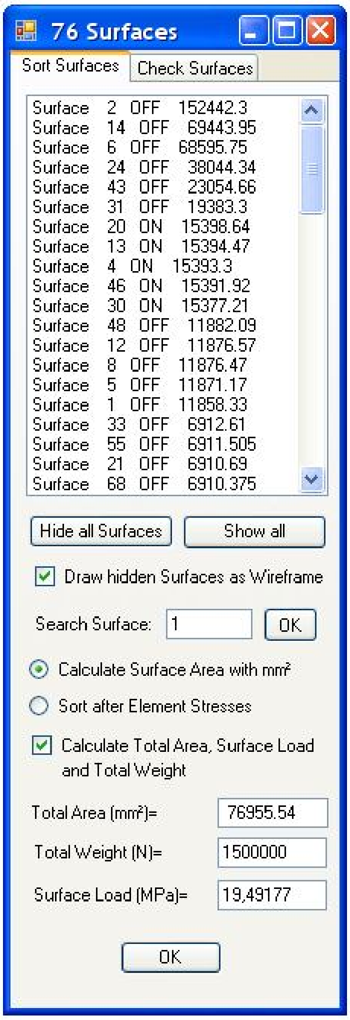


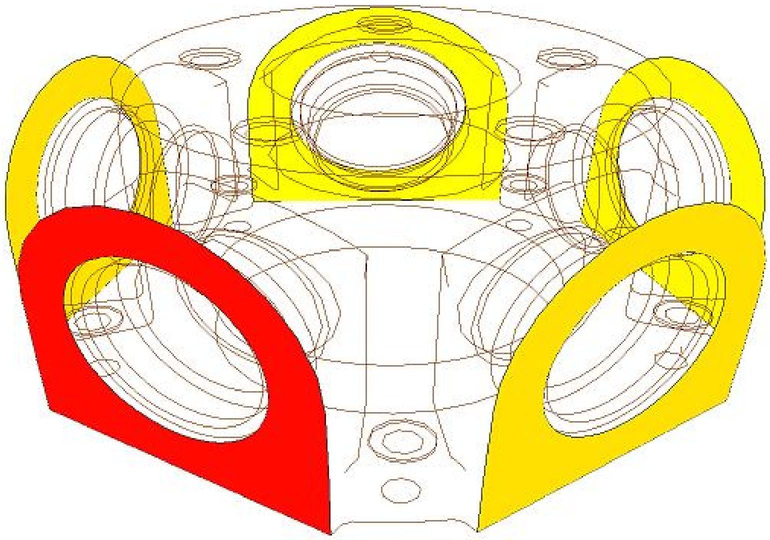
Calculate Cross-Sectional Area and Surface Load

At first we must calculate the surface load with the difference of 300 000 N and the cross-sectional area in mm². Select menu “Surface Loads / Step 2: Calculate Cross-Sectionsal Area”



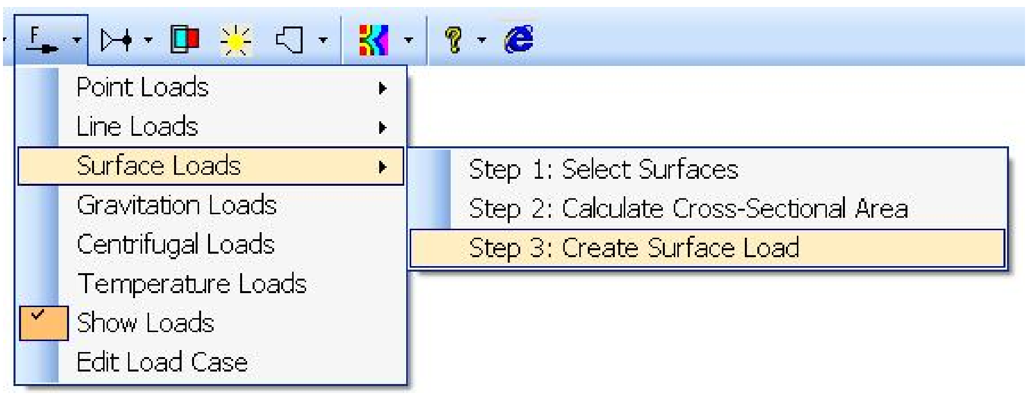
On the left side menu bar choose menu “Hide all Surfaces” and then click on the surfaces 4, 13, 20, 30 und 46 in the list box. Now you can see only the 5 flange surfaces on the screen. Activate “Calculate Total Area, Surface Load and Total Weight” and input the Total Weight of 1 500 000 N in order to obtain a result for the Surface Load of 19.492 MPa.

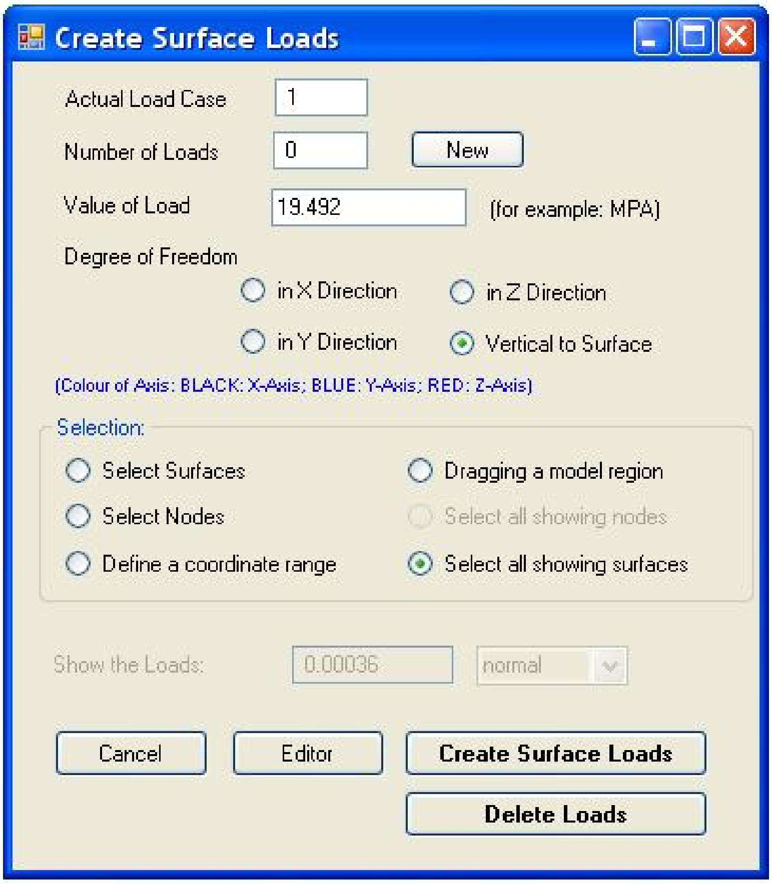




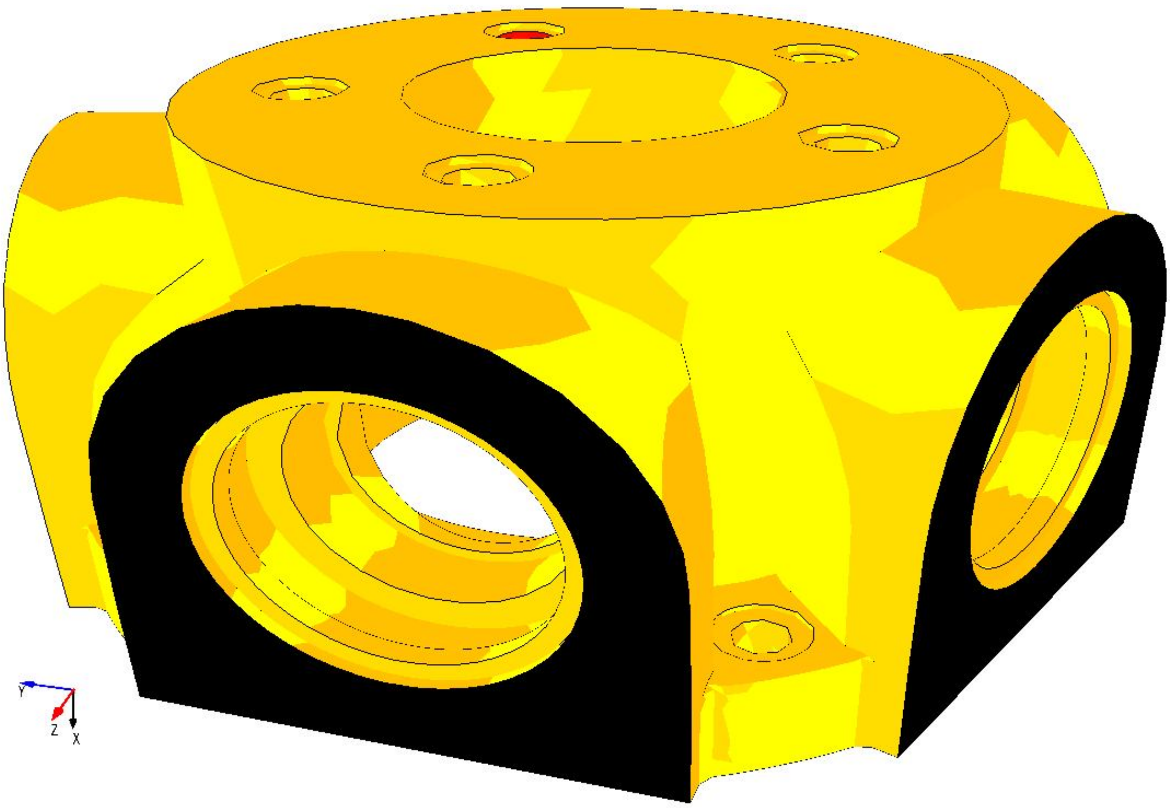
Create Surface Load

Now select menu “Surface Loads” and “Step 3: Create Surface” and input a surface load of 19.492 MPa in the direction “Vertikal to Surface”.





Choose the new Selection-Option “Select all showing surfaces” and select “Create Surface Load” to create the load. Now we can see the the 5 black surface loads:

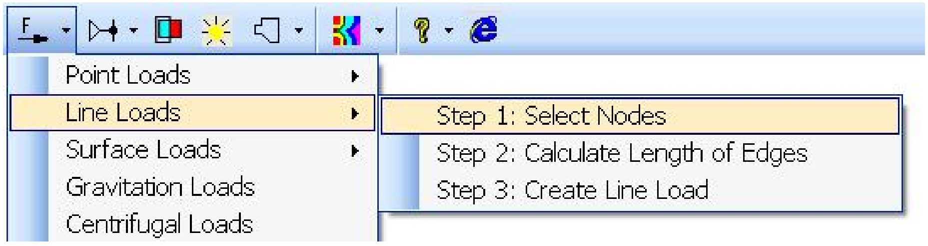


Create a Line Load

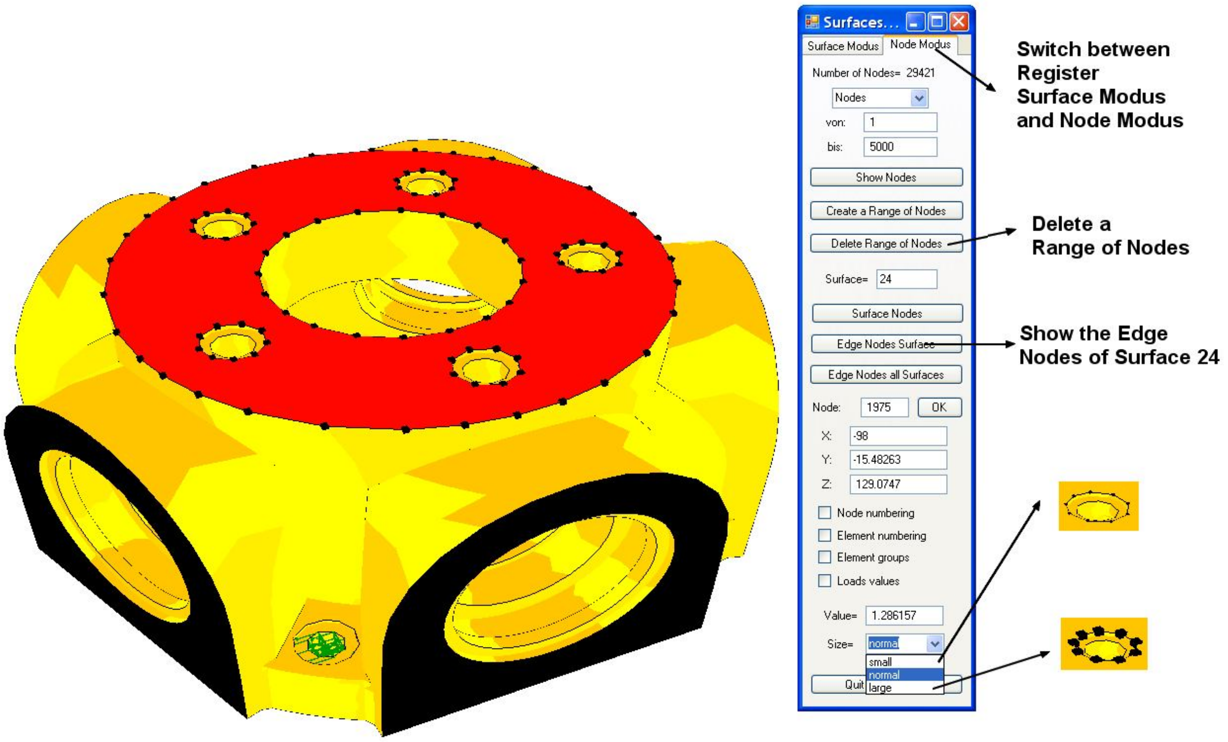
The Load Case 2 is a line load of 10 tons which is loaded on the inner and outer edge line of surface 24.

Step 1: Select Nodes

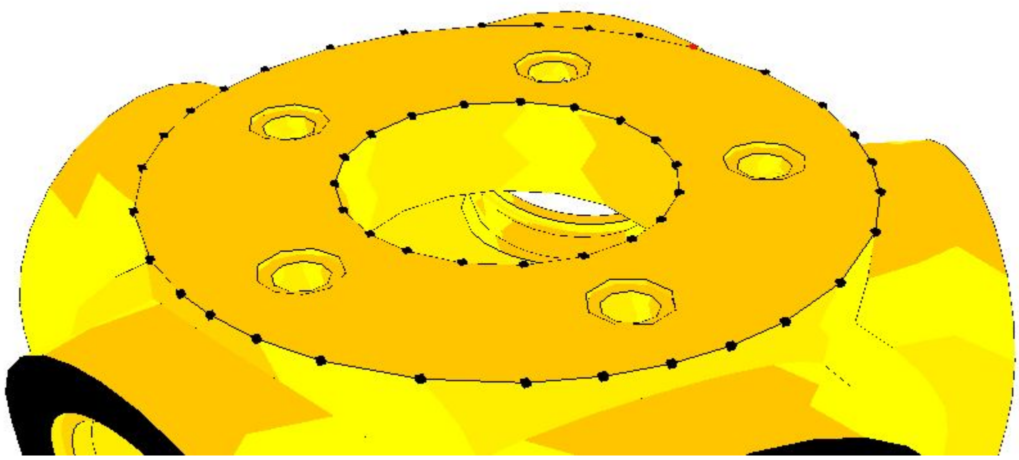
Select menu "Line Loads / Step 1: Select Nodes" or choose in the right menu bar



the register "Node Modus" and select "Edge Nodes Surface" in order to show the edge nodes of surface 24.

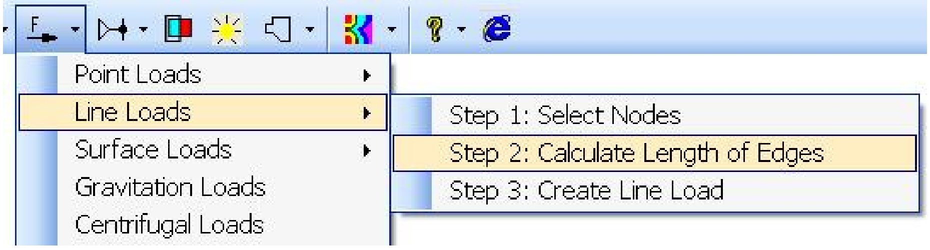


Then choose menu "Delete Range of Nodes" and delete the edge nodes of the 5 holes which are not loaded.

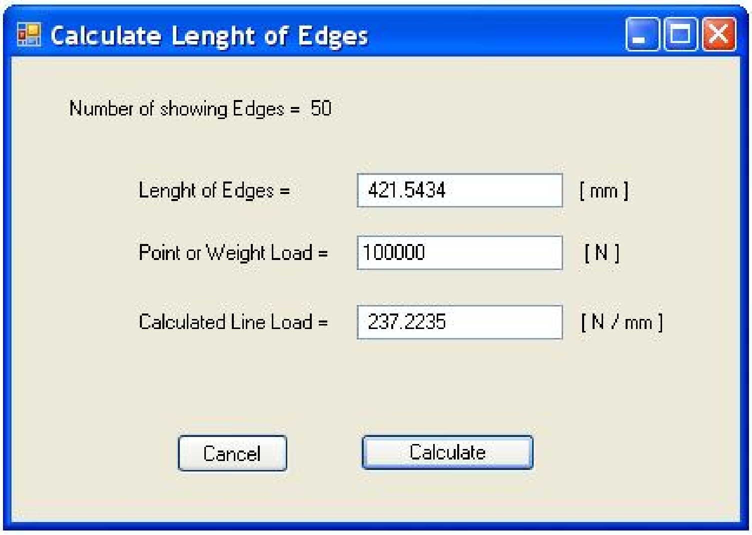


Step 2: Calculate Length of Edges

The next step is to obtain the value of line load by dividing the weight load of 10 tons by the length of edges.

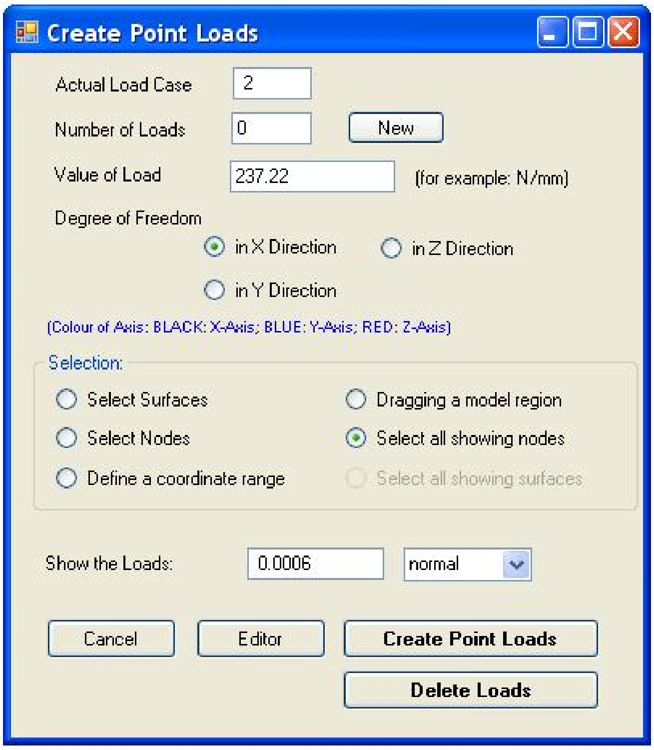


Select menu “Step 2: Calculate Length of Edges” and we can see the length of the edges is 421.54 mm. Now input the load of 100000 N and select “Calculate” in order to obtain a line load of 237.22 N/mm.

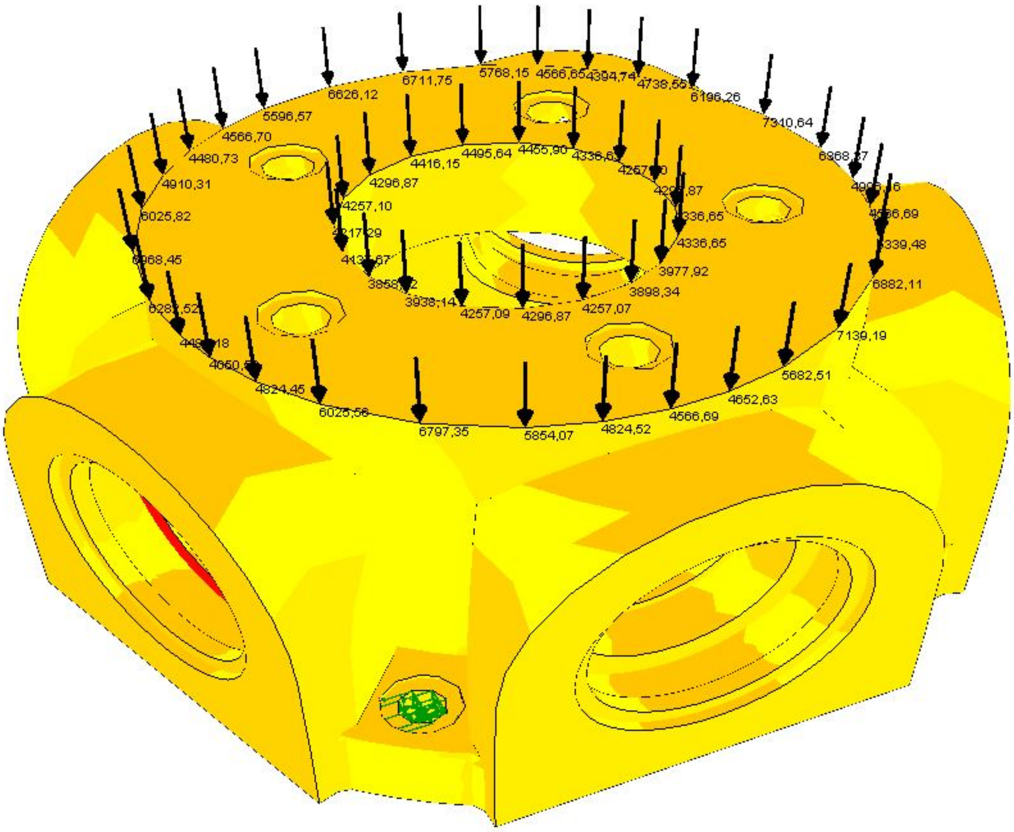


Step 3: Create Line Load

Select menu “Step 3: Create a Line Load” with Load Case 2 and input the load value “237.22” in X-direction.



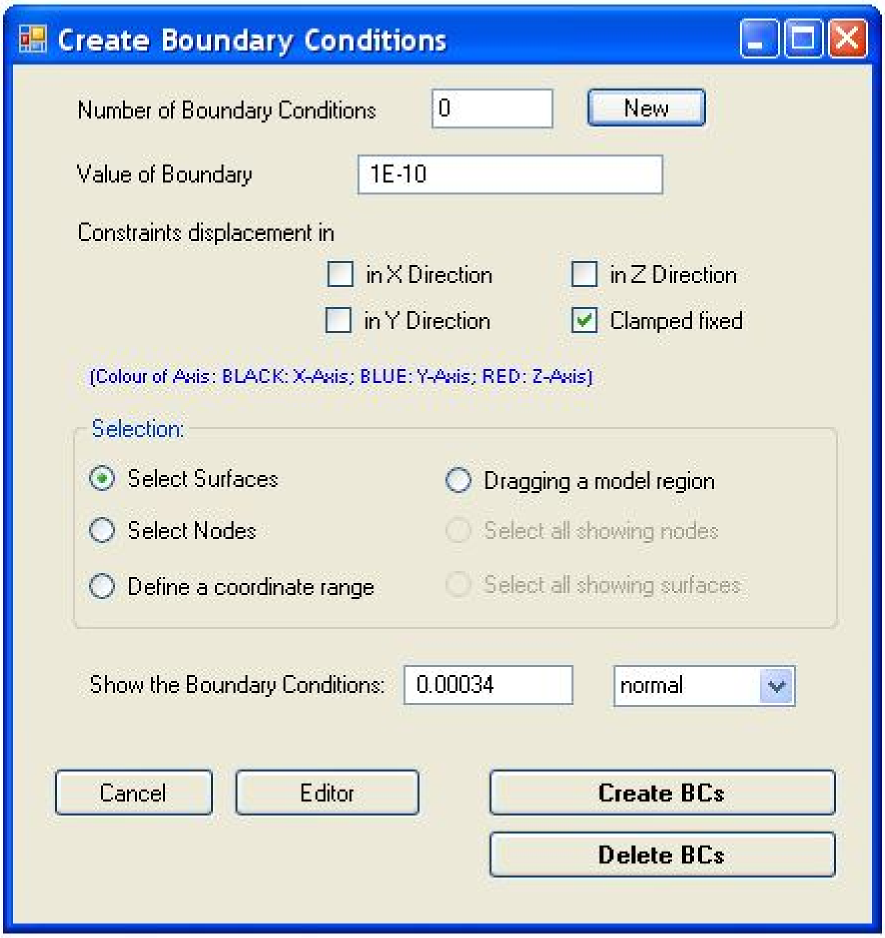
Choose “Select all showing nodes” and we obtain 50 different single point loads.



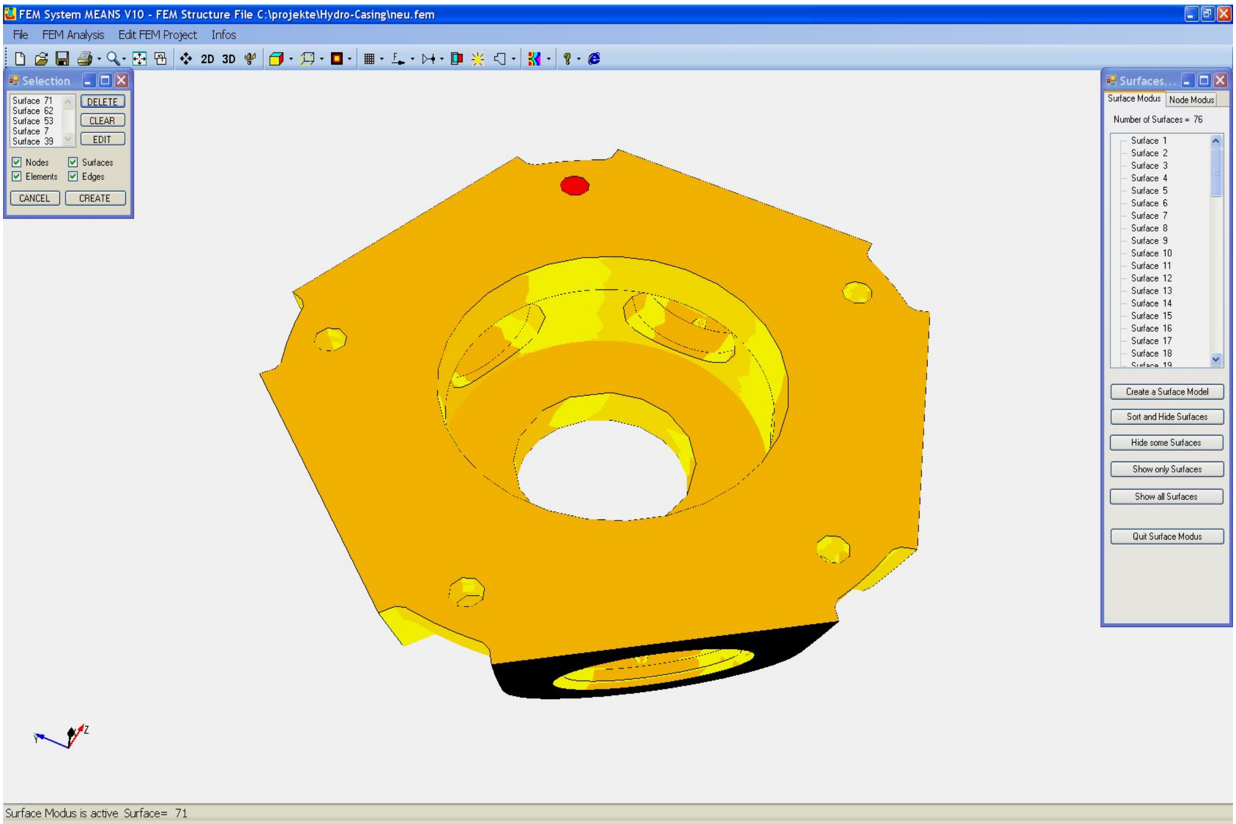
Create Bounday Conditions

The casing is clamped fix on the bolt holes around the 5 flange surfaces. Select menu in the view bar “Boundary Conditions” and “Step 2:Create Boundary Conditions”.

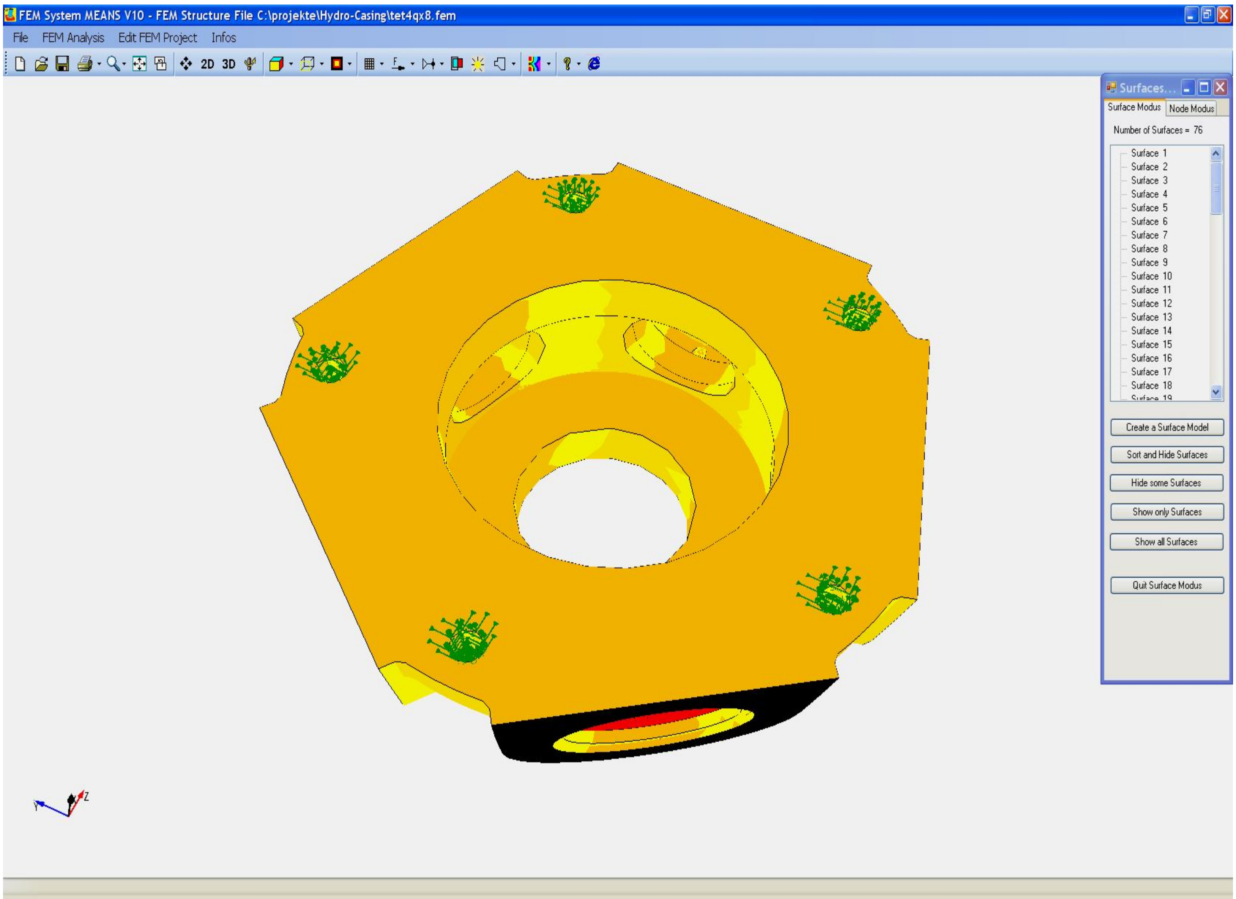
Select “Clamped fixed” and the selection option “Select Surfaces” and choose the button “Create BCs” and double click on the surfaces 7, 39, 53, 62 and 71.



In the Select Box on the left side above select menu “Create” to create the boundary conditions or you can edit here a wrong input.

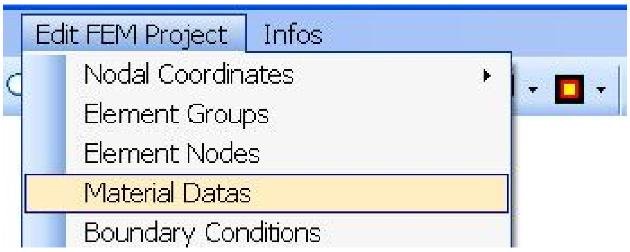


After this step you can see the Boundary Conditions on the bolt holes:

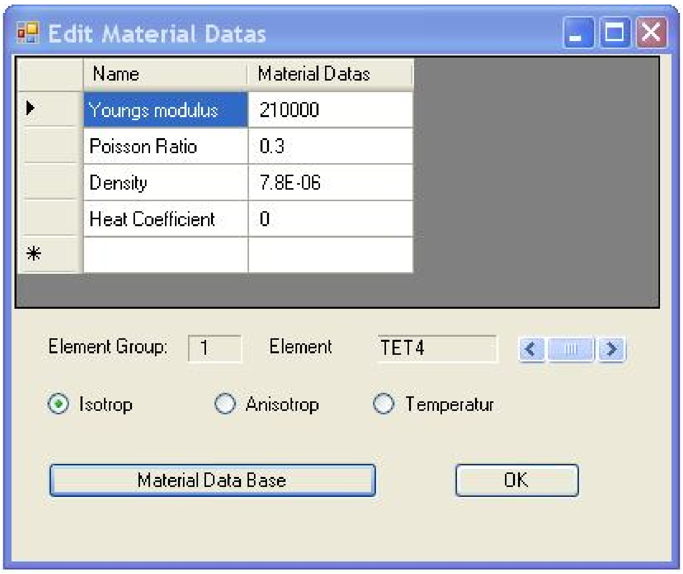


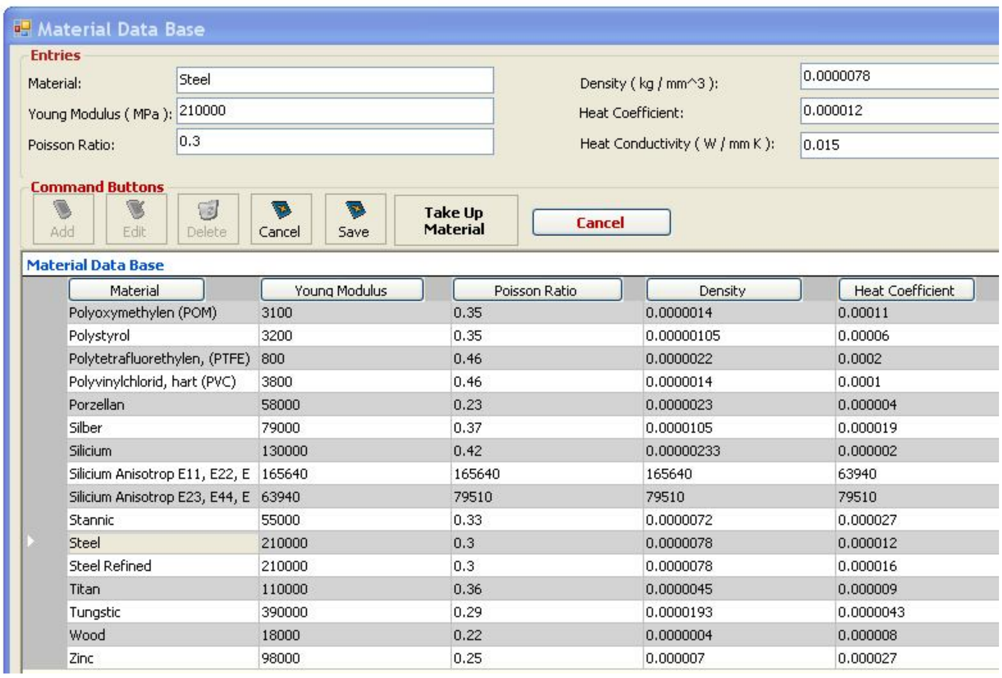
Input Material Datas

Select the menu „Edit FEM Project“ and „Material Datas“ and input the material datas for steel.



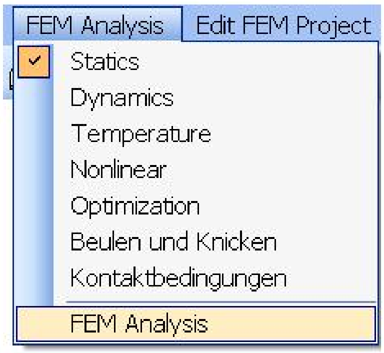
Select “Material Date Base” and take up the material “steel” in the material data box.

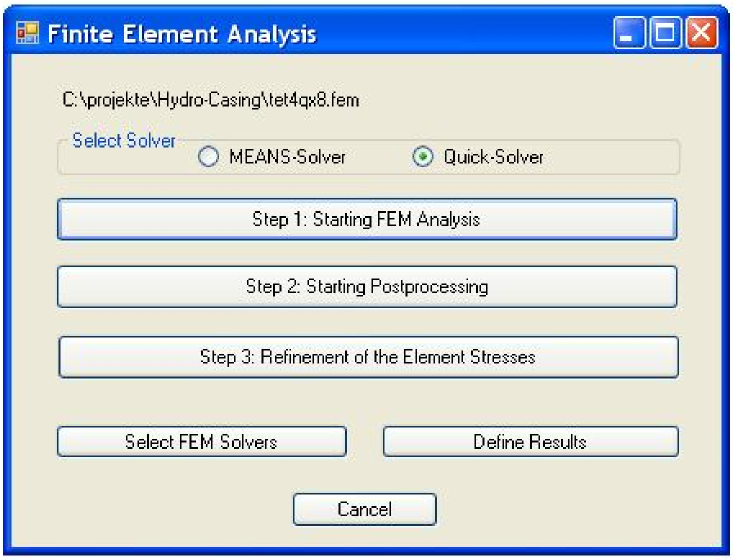




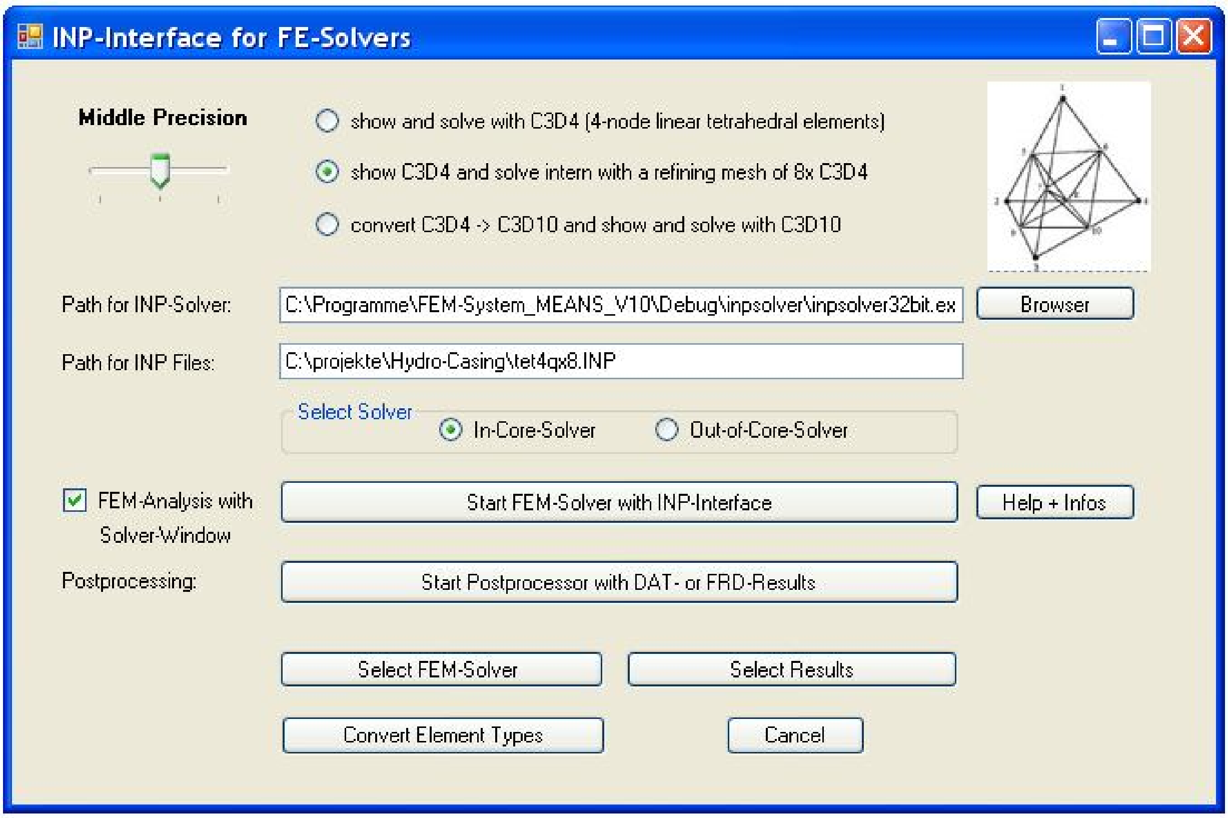
FEM-Analyse

First save the FEM file and then choose menu “FEM Analysis” and “FEM Analysis” and in the next box select “Quick-Solver” and menu “Step 1: Starting FEM Analysis“.



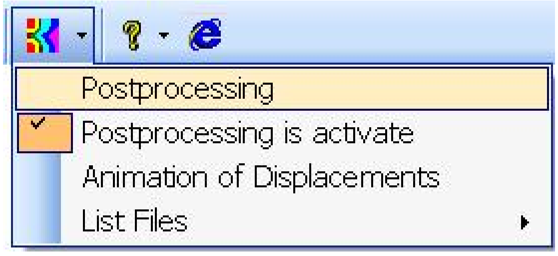


Then choose “**convert C3D4->C3D10 and show and solver with C3D10**” or for EasyFEM-User with the 120000 element limit menu “**show C3D4 and solve intern with a refining mesh of 8x C3D4**” and select “Start FEM-Solver with INP-Interface” to calculate the results. After the FEM Analysis evaluate the results with menu “Start Postprocessor”.



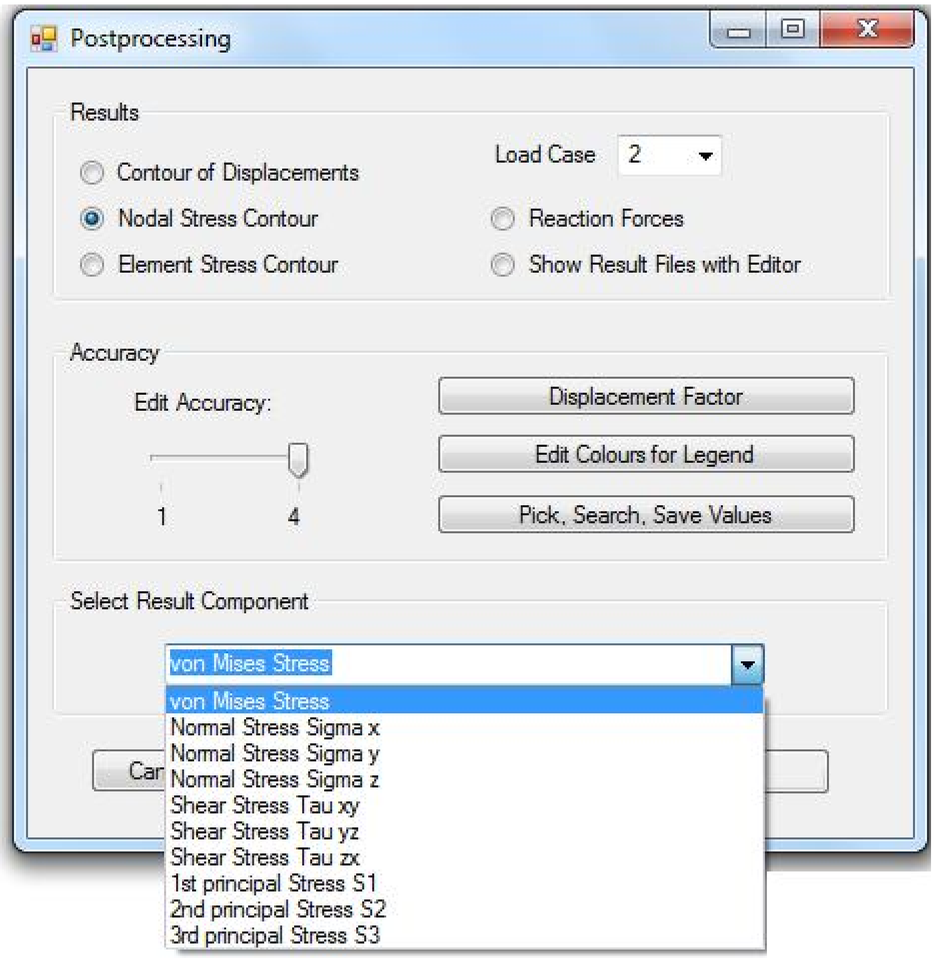
Postprocessing

Choose the menu „Postprocessing“ or select in the FEM Assistance in the row of Postprocessing the link “goto” to evaluated the results.



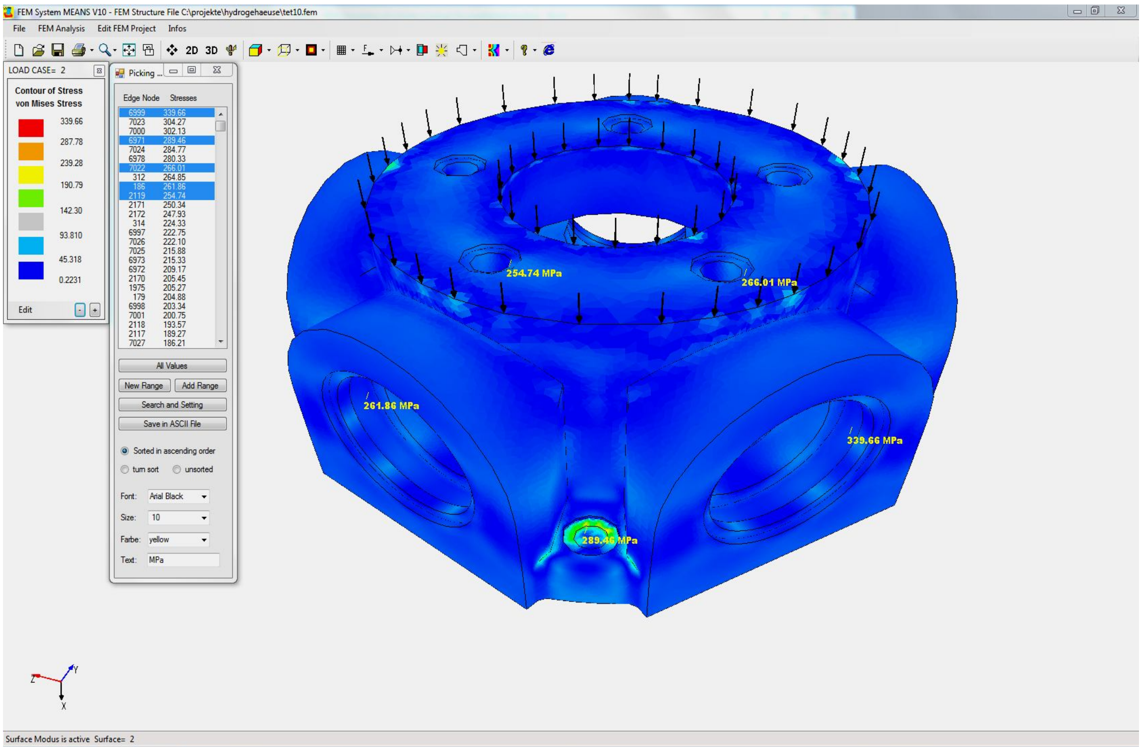
Please select following Postprocessing:

- Results: **Nodal or Element Stress Contour**
- Load Case: 2
- Edit Accuracy: **4**
- Result Component: **von Mises Stress**



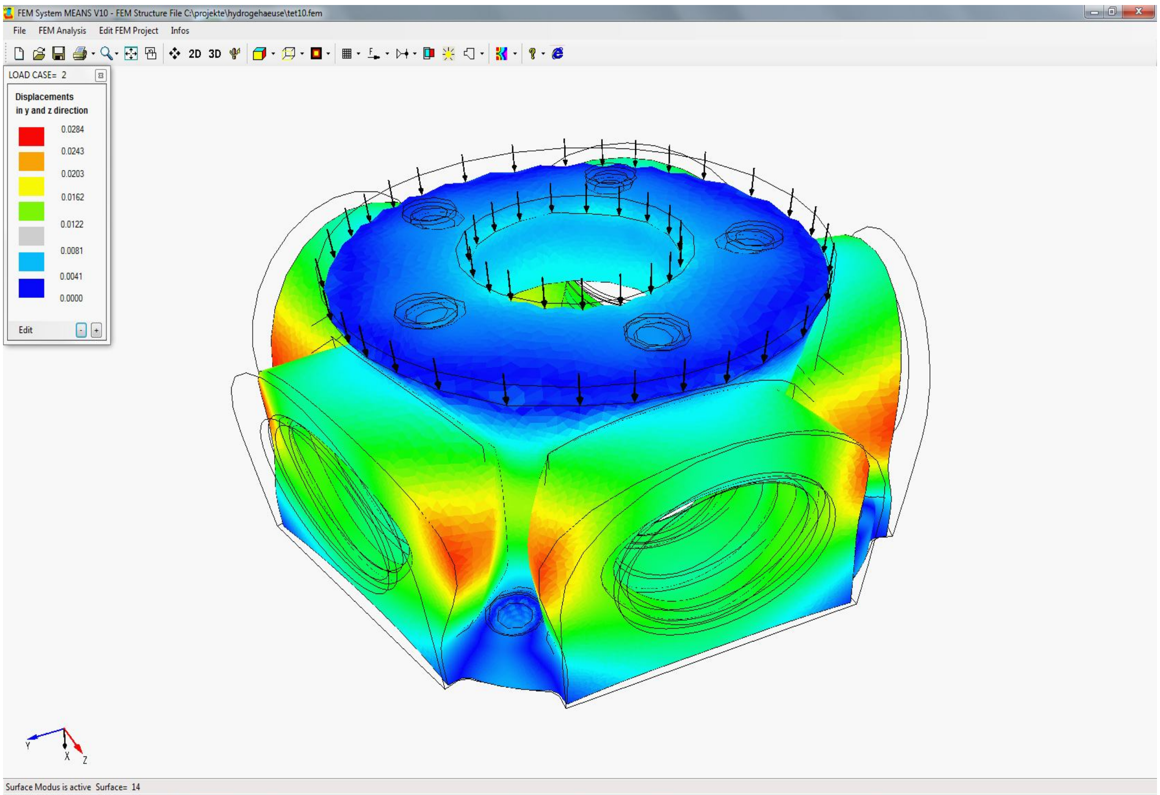
v.Mises-Stresses

Max Value = 339.66 Mpa



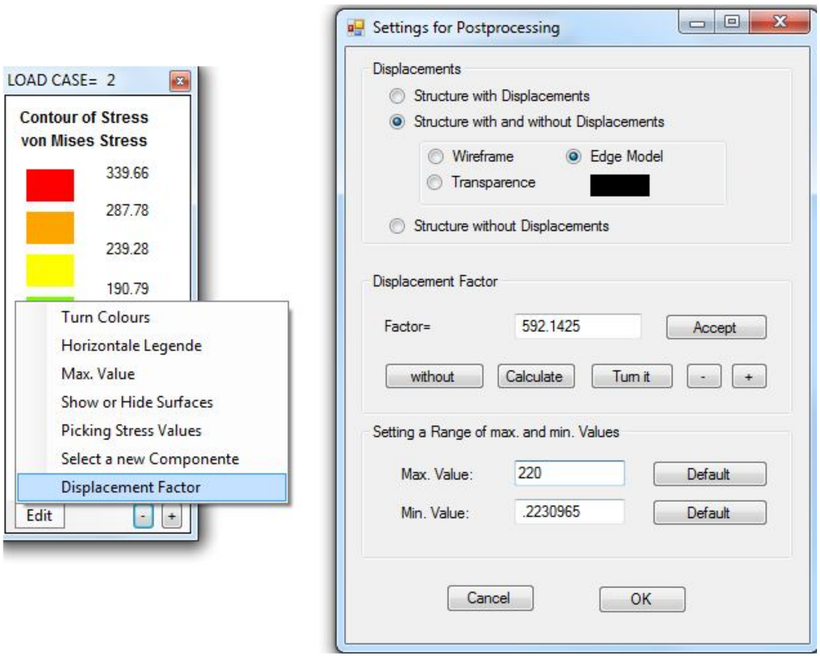
Displacements

Max Value = -0.0284 mm



Displacement Factor

- Select Structure with and without Displacements
- Calculate a Displacement Factor of 592.14
- Set down the Range of Stresses from 339 to 220 Mpa, because 339 is a "Secundary Point Stress" caused by a badly tetrahedral element.



Further select „Picking Stress Values", „Search and Setting" and menu "Hide and Show Surfaces" and display only the bolt surfaces stresses.