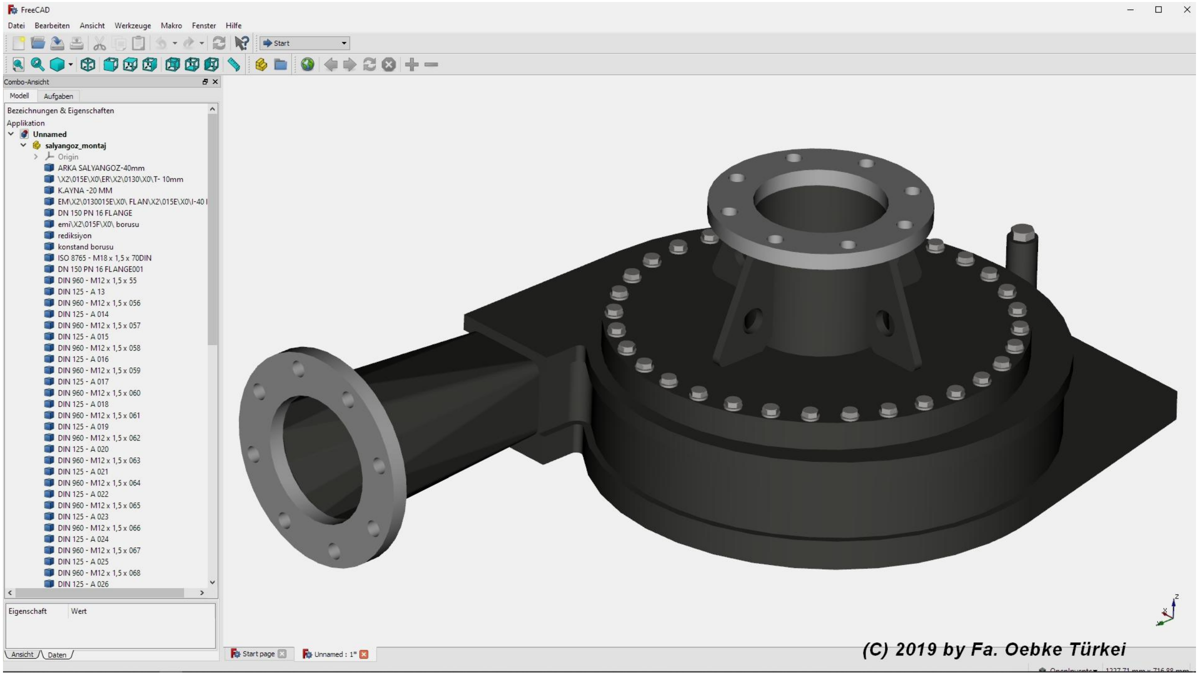
Part 14: FEM-Analysis of a CAD Assembly with MEANS V11

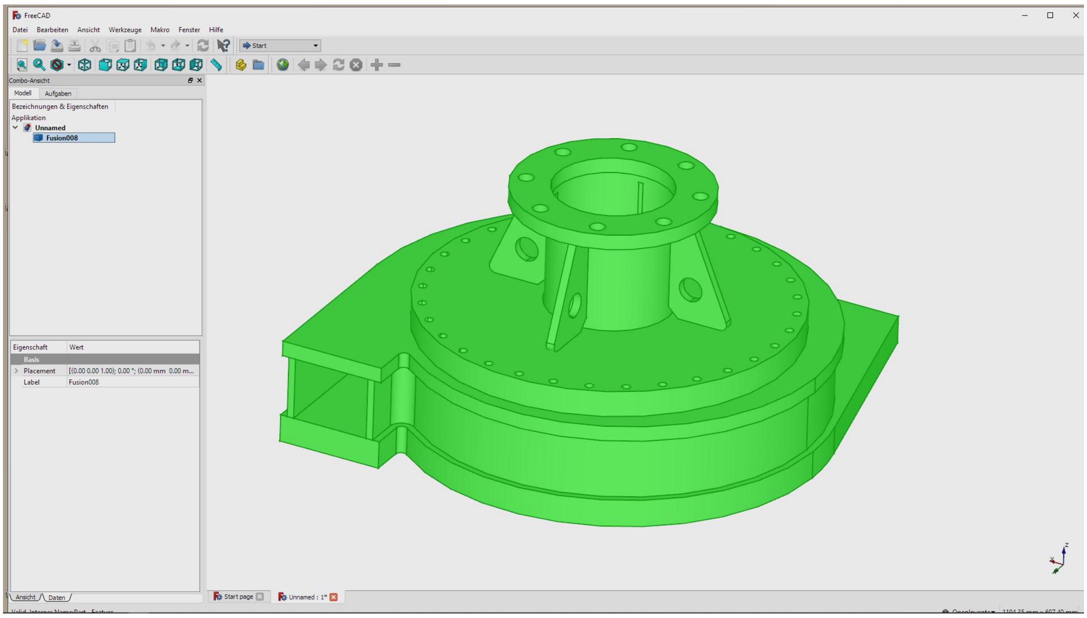
A CAD Assembly from Inventor is loaded with 7 bar internal pressure, how high are the displacements and stresses with FEM System MEANS V11.

CAD-Assembly from Inventor



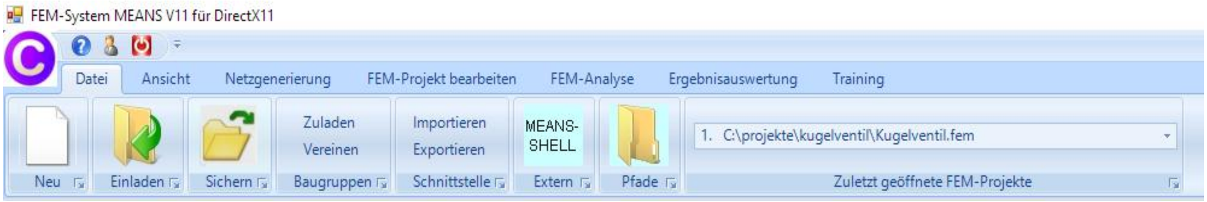
Create a Model Part

Now delete all unimportant components such as screws, left flange, sockets in the CAD system and combine the model into a large part and export it as a STEP file.

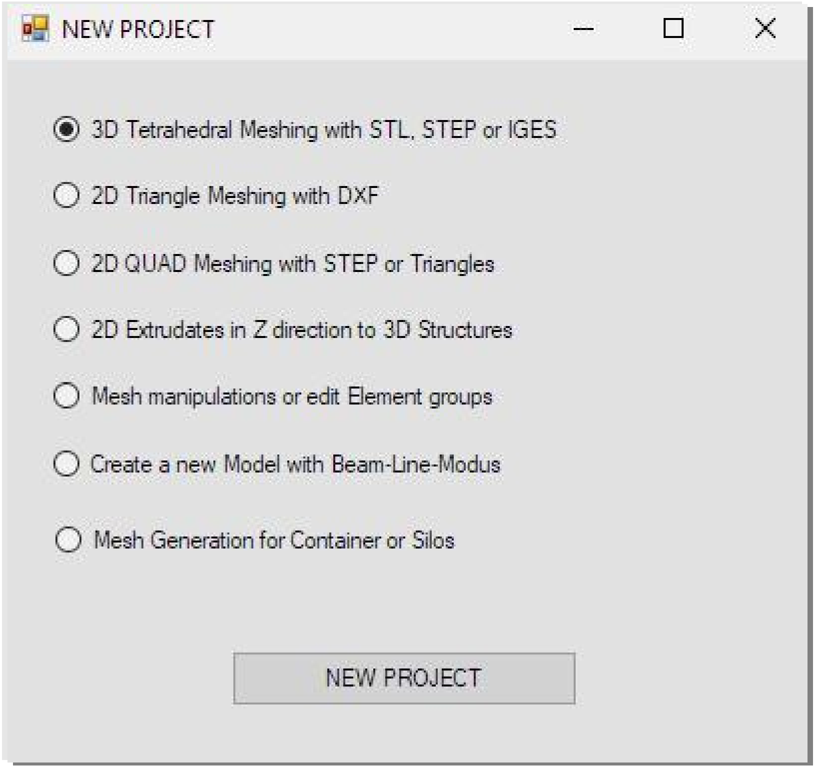


STEP Interface

Now close the CAD System and start the FEM System MEANS V11 to generate a FEM Mesh with the STEP-Model.



Select the „Mesh Generation“ tab and „New“ to create a new FEM project.

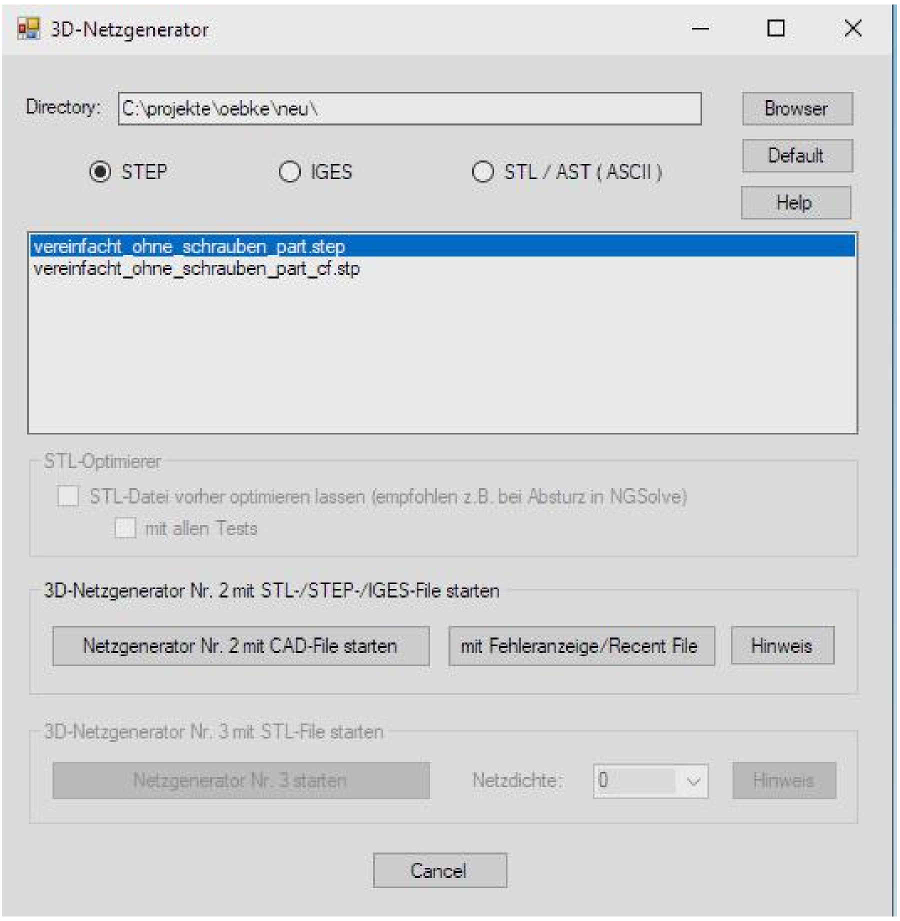


Select "3D Tetrahedral Meshing with STL, STEP or IGES

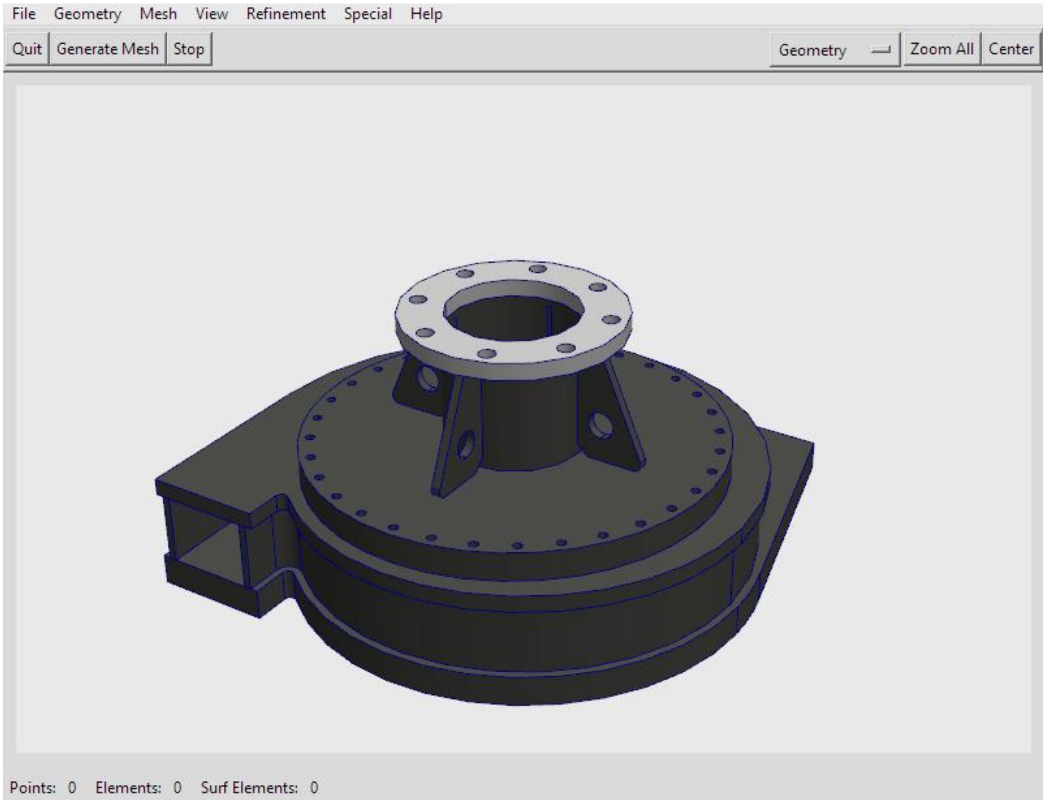
A dialog box appears, showing the following CAD formats:

STL consists of a triangular outer shell for 3D mesh generation
STEP consists of solid elements and is the most suitable 3D format
IGES like STEP format but not so common anymore

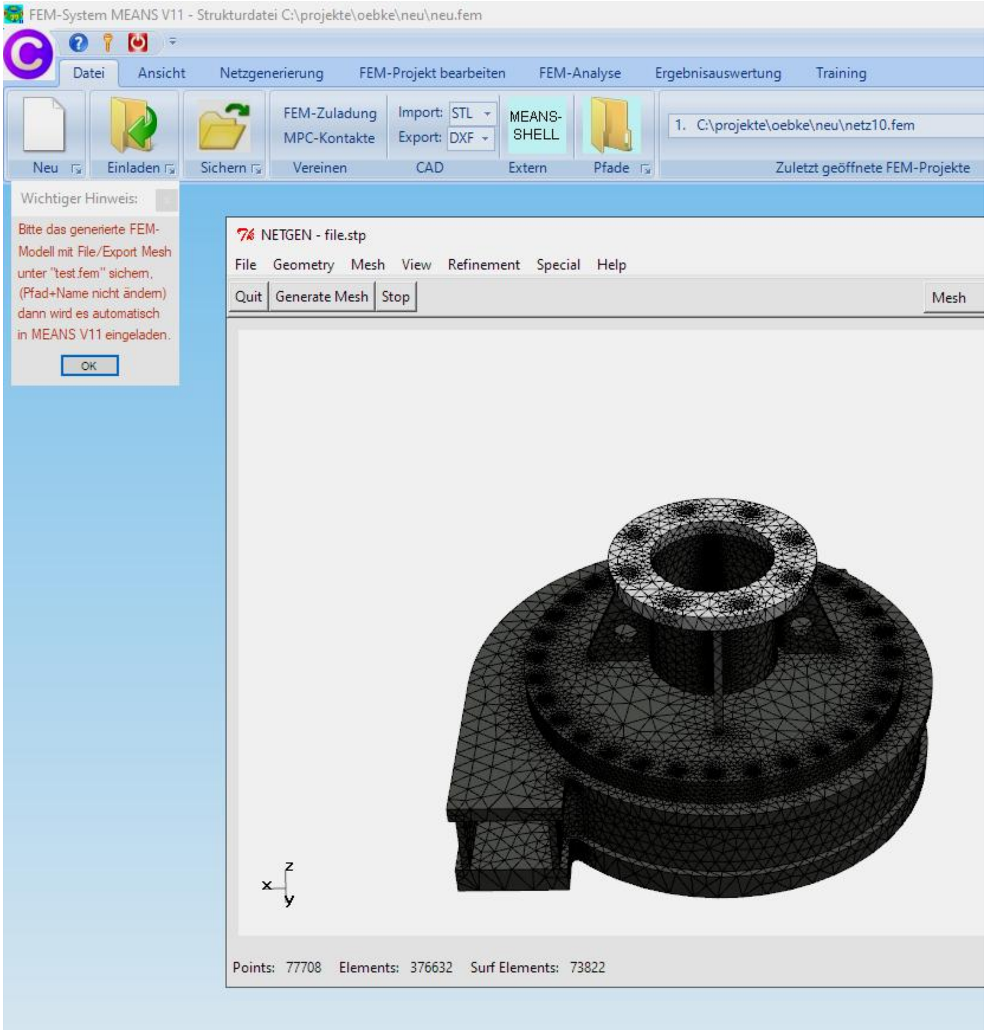
Select the STEP file from the FEM-Projects/New-Directory or with "Browser" and click on the button "Start Mesh generator No. 2 with CAD File" to display it in the mesh generator.



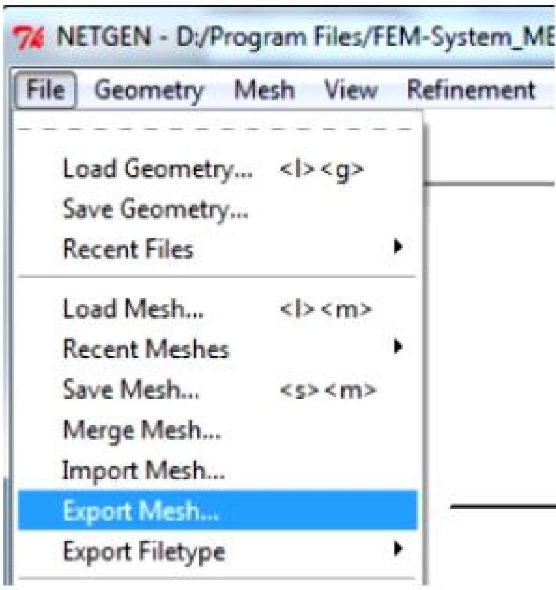
The STEP File can be seen in the mesh generator and can be rotated as required.



Select the menu "Mesh" and "Meshing Options" and generate with the mesh density "moderate" and the menu "Generate Mesh" a FEM Model with tetrahedral elements.



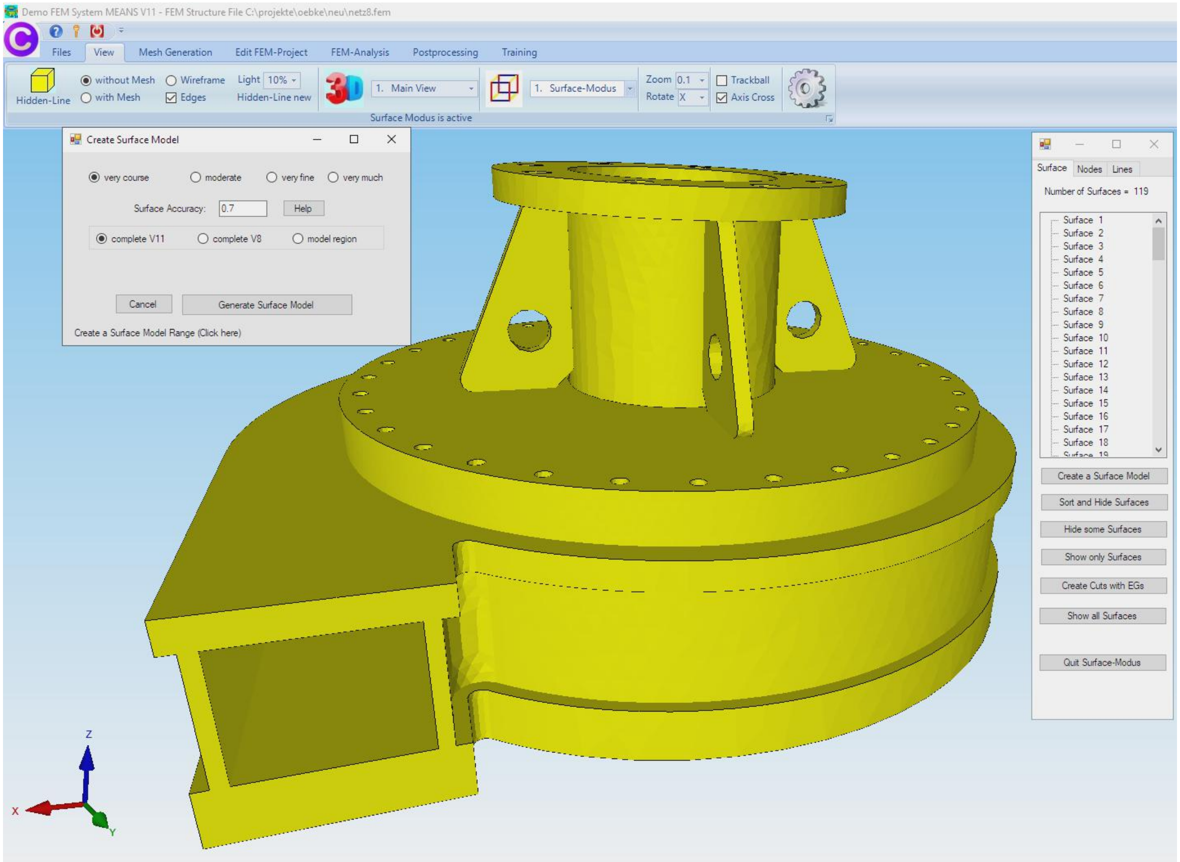
After the mesh generation, the mesh must be exported to MEANS V11 with name **"test.fem"**. Select "File" and "Export Mesh" and save it into the Debug/Mesh Path.



please export the generated model with the name "test.fem" into the default Mesh-Debug-Path.

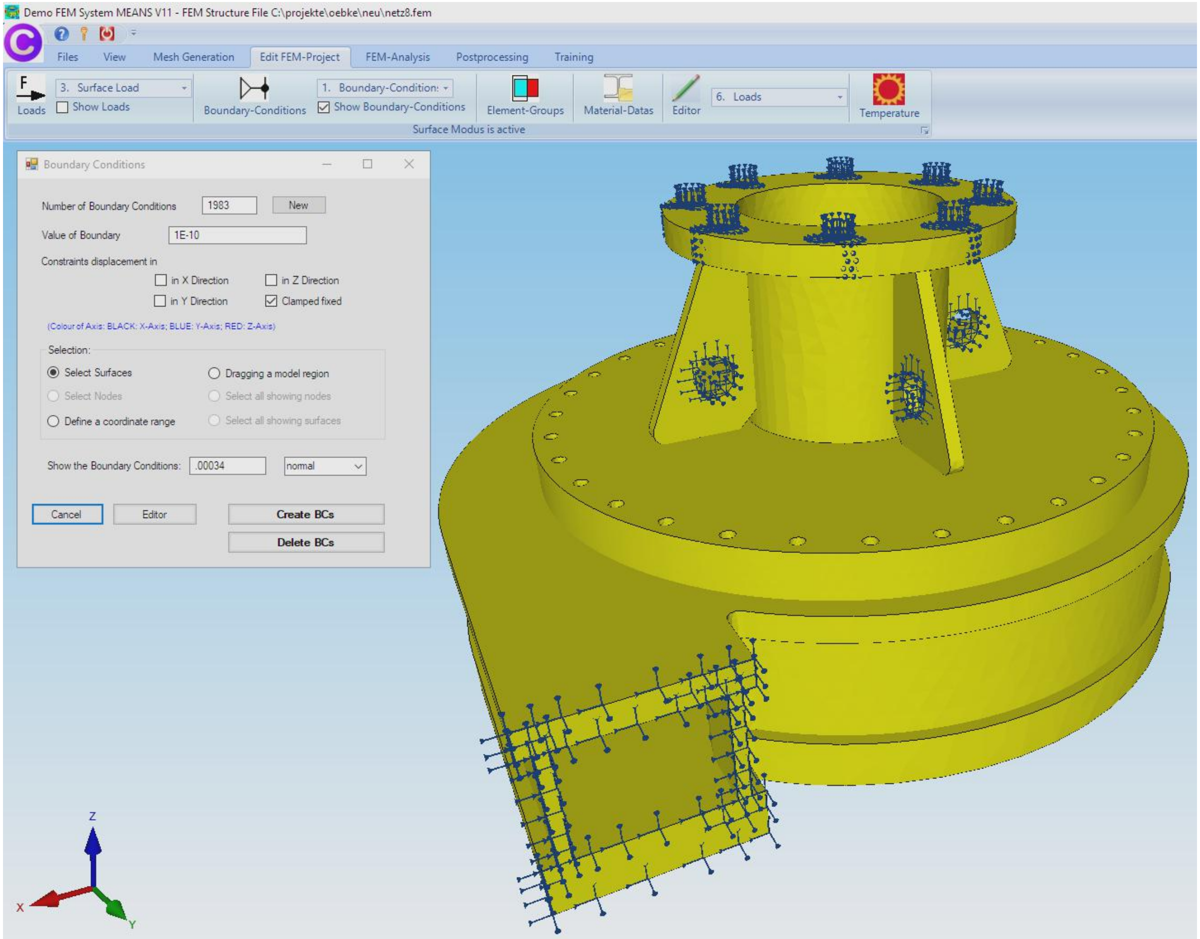
Surface model

After mesh generation, the surface model is created with the option "very course" with 119 faces. The surface model now makes it possible to select the surfaces for the boundary conditions and surface loads or to view and edit the model from the inside.



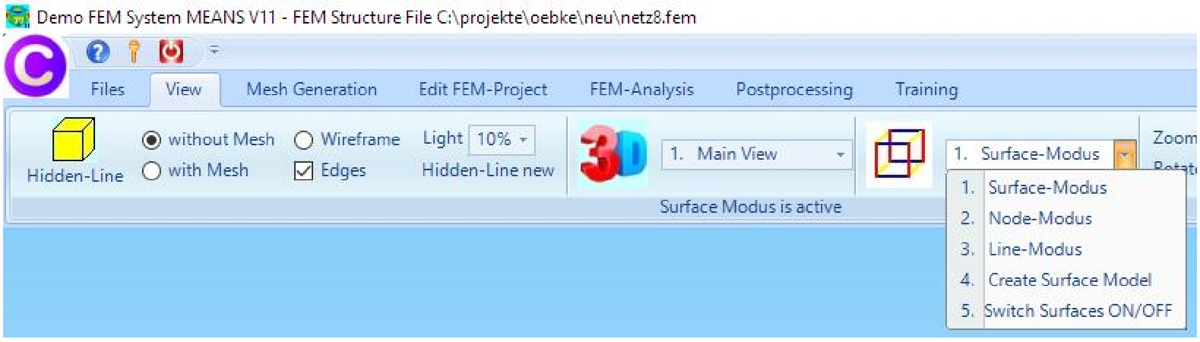
Create Boundary Conditions

Select register "Edit FEM Project" and click on the "Boundary Condition" icon to clamped fix the model to his flanges.
Select "Create BC's" and click on the clamped surfaces. Finally, select "Create" in the Selectbox to create the RBs

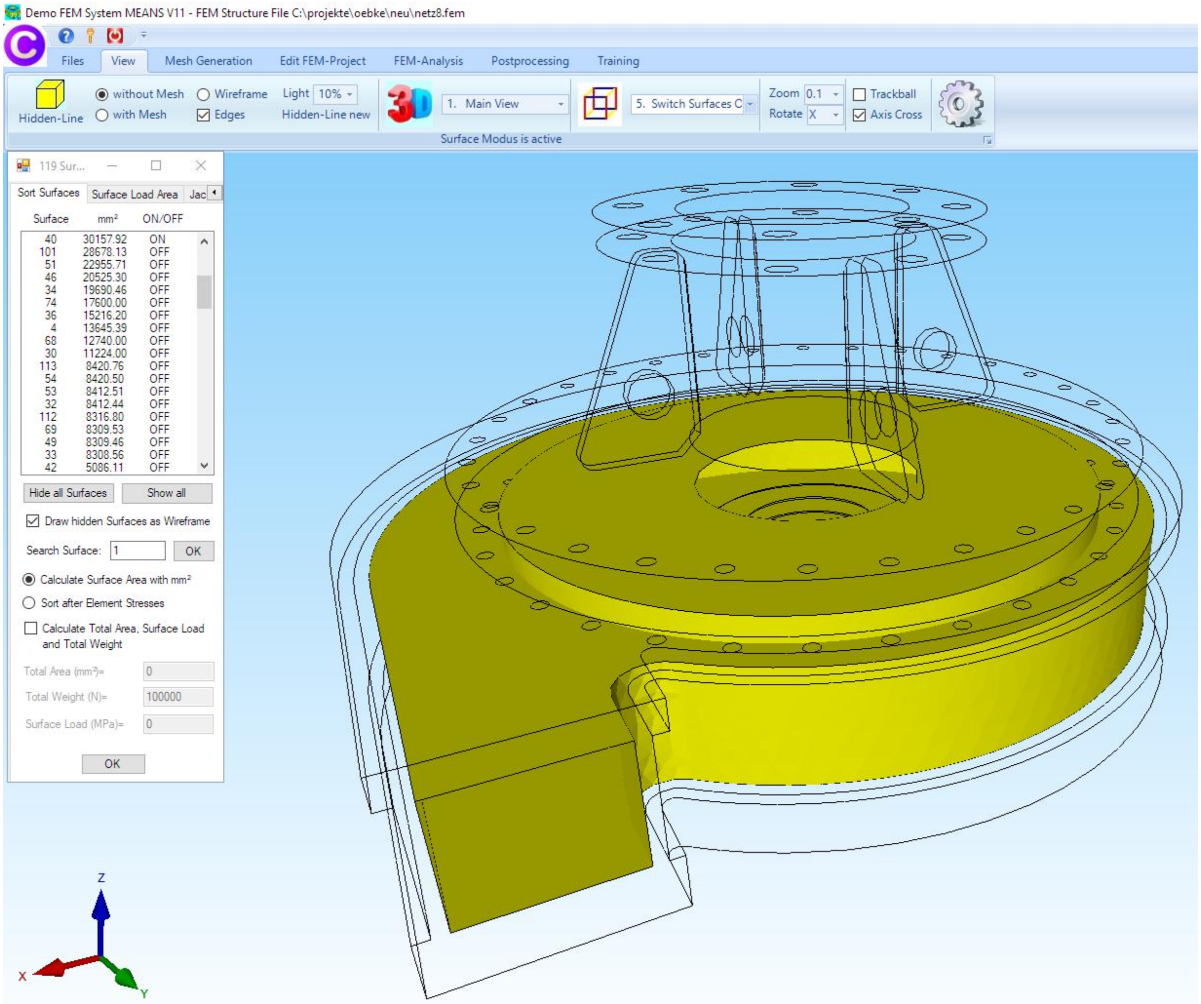


Create the Pressure Loads

The casting is loaded with a pressure load of 7 bar or 0.7 MPa in the inside. First, the surfaces for the pressure must be determined. Select the register “View” and menu “Switch Surfaces ON/OFF”

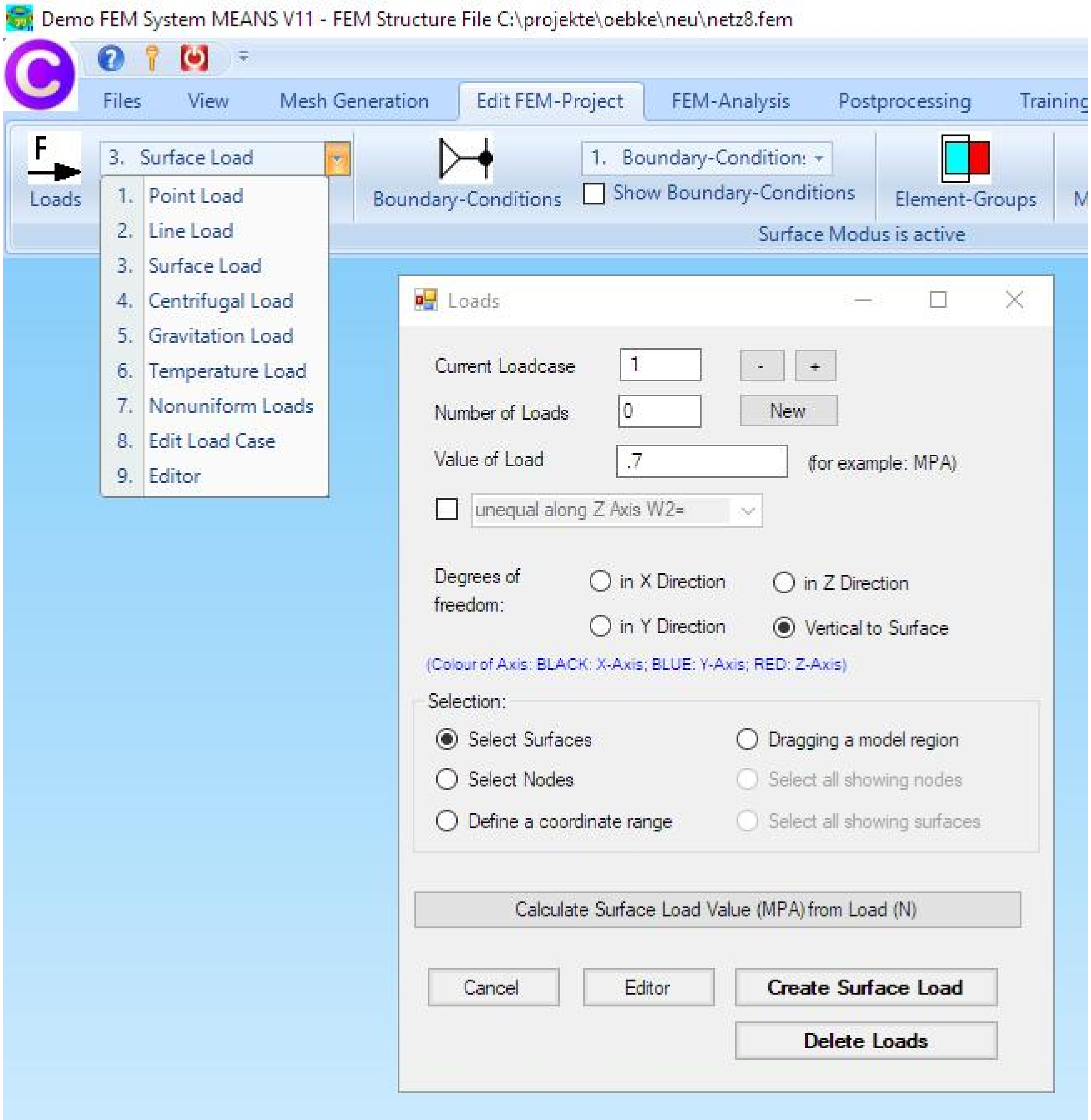


In the new left dialogbox select "Hide all Surfaces" to switch OFF all surfaces. Then switch ON the surfaces 3, 10, 20, 26 and 40 and note this 5 surfaces for the next step. Then select again "Show all surfaces".

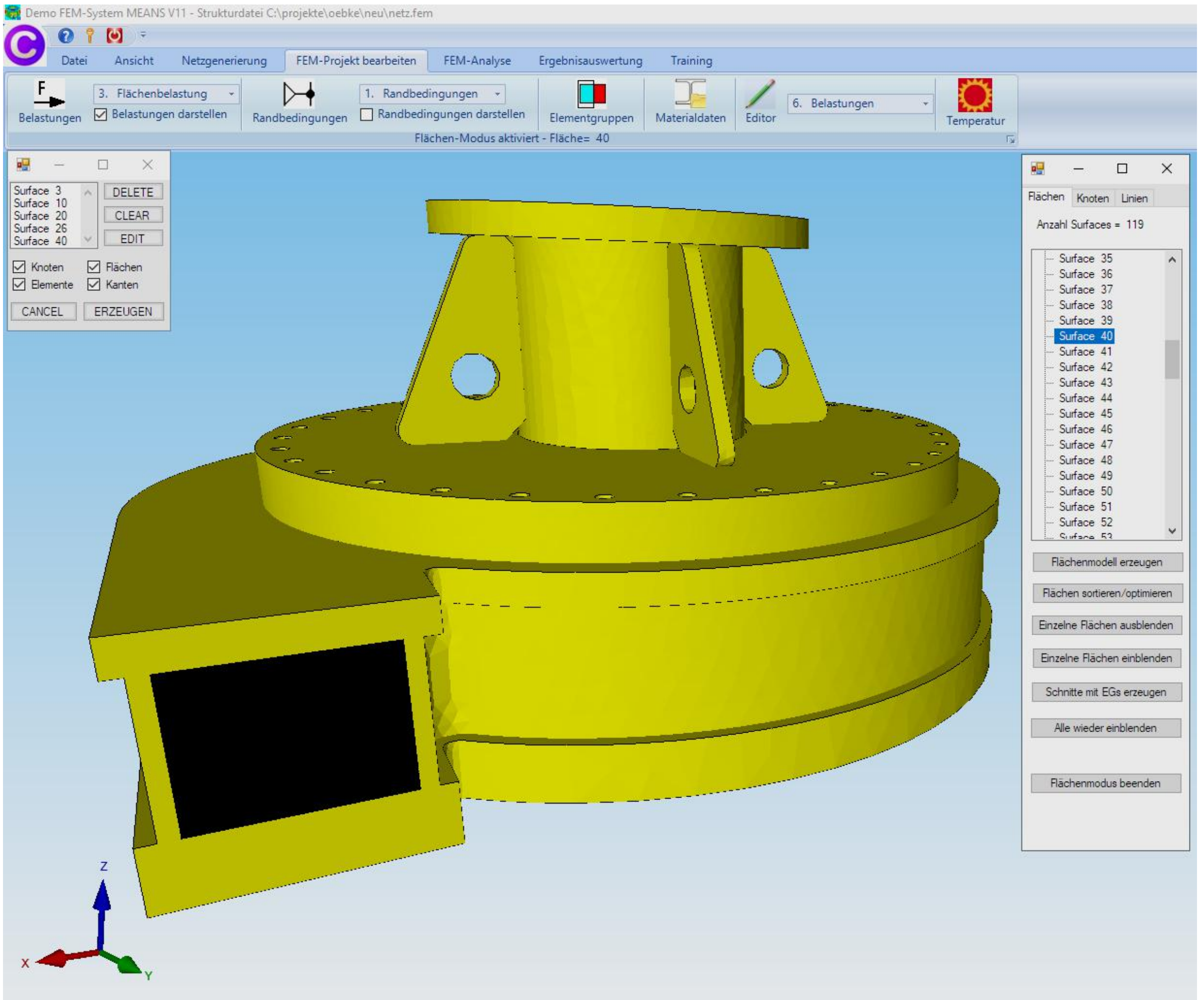


Create the Surface Load

Select register "Edit FEM-Project" and "Surface Load" to create the surface load in the inside.



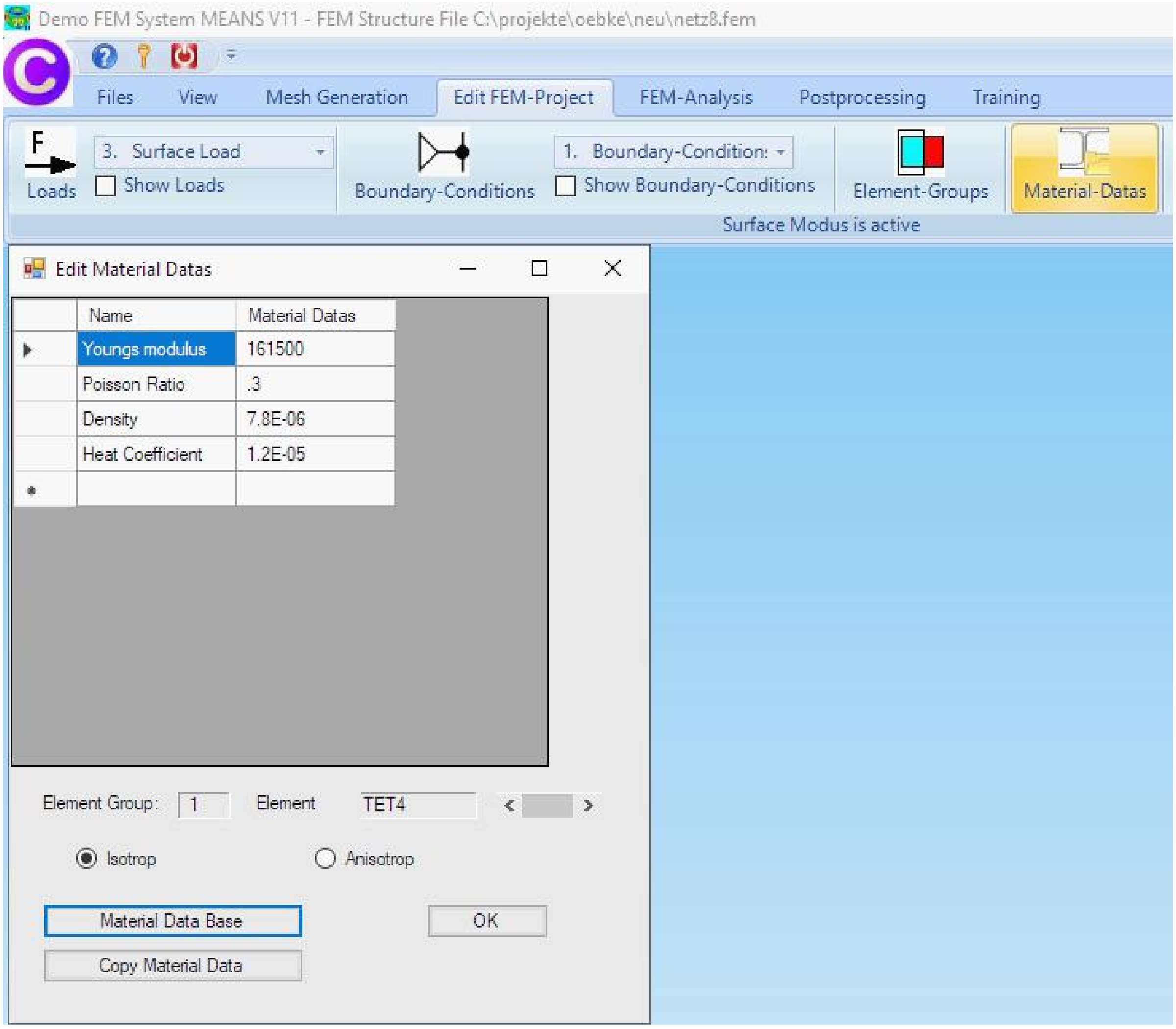
Enter the value "0.7" and select "Create Surface Load"
Now click on the surfaces in the right window one by one on 3, 10, 20, 26 and 40.
These are displayed in the selectbox, where "Generate" generates the surface load.


Demo FEM-System MEANS V11 - Strukturdatei C:\projekte\oebke\neu\netz.fem
Datei
Ansicht
Netzgenerierung
FEM-Projekt bearbeiten
FEM-Analyse
Ergebnisauswertung
Training
Belastungen
3. Flächenbelastung
Belastungen darstellen
Randbedingungen
1. Randbedingungen
Randbedingungen darstellen
Elementgruppen
Materialdaten
Editor
6. Belastungen
Temperatur
Flächen-Modus aktiviert - Fläche= 40
Surface 3
Surface 10
Surface 20
Surface 26
Surface 40
DELETE
CLEAR
EDIT
Knoten
Flächen
Elemente
Kanten
CANCEL
ERZEUGEN
Flächen
Knoten
Linien
Anzahl Surfaces = 119
Surface 35
Surface 36
Surface 37
Surface 38
Surface 39
Surface 40
Surface 41
Surface 42
Surface 43
Surface 44
Surface 45
Surface 46
Surface 47
Surface 48
Surface 49
Surface 50
Surface 51
Surface 52
Flächenmodell erzeugen
Flächen sortieren/optimieren
Einzelne Flächen ausblenden
Einzelne Flächen einblenden
Schnitte mit EGs erzeugen
Alle wieder einblenden
Flächenmodus beenden
Z
X
Y

Material Datas

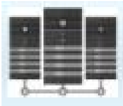
Select register "Edit FEM-Project" and select the icon 

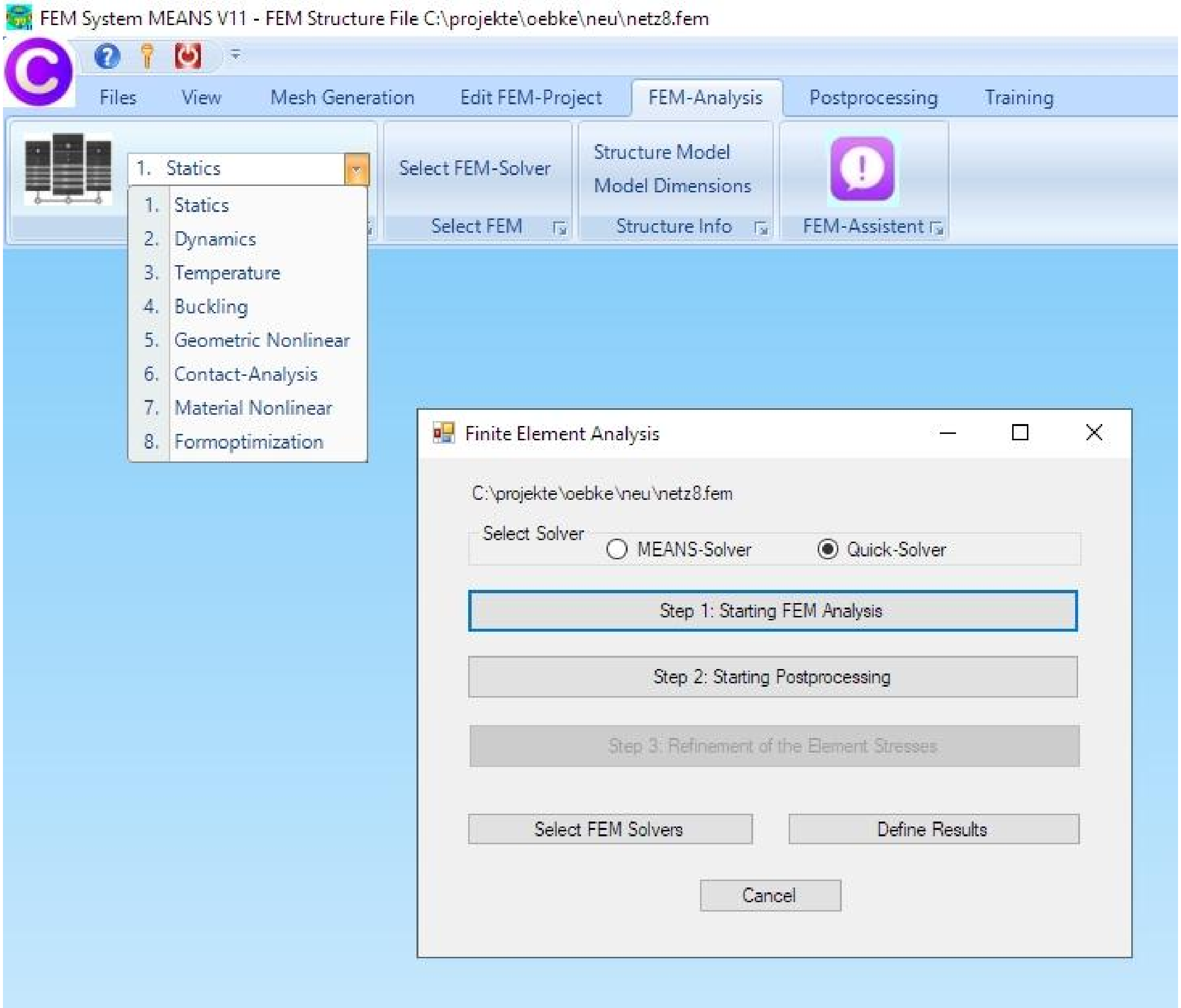
to enter the Material Datas such as Young's modulus and Poisson's Ratio for Steel is always preset.



FEM-Analysis

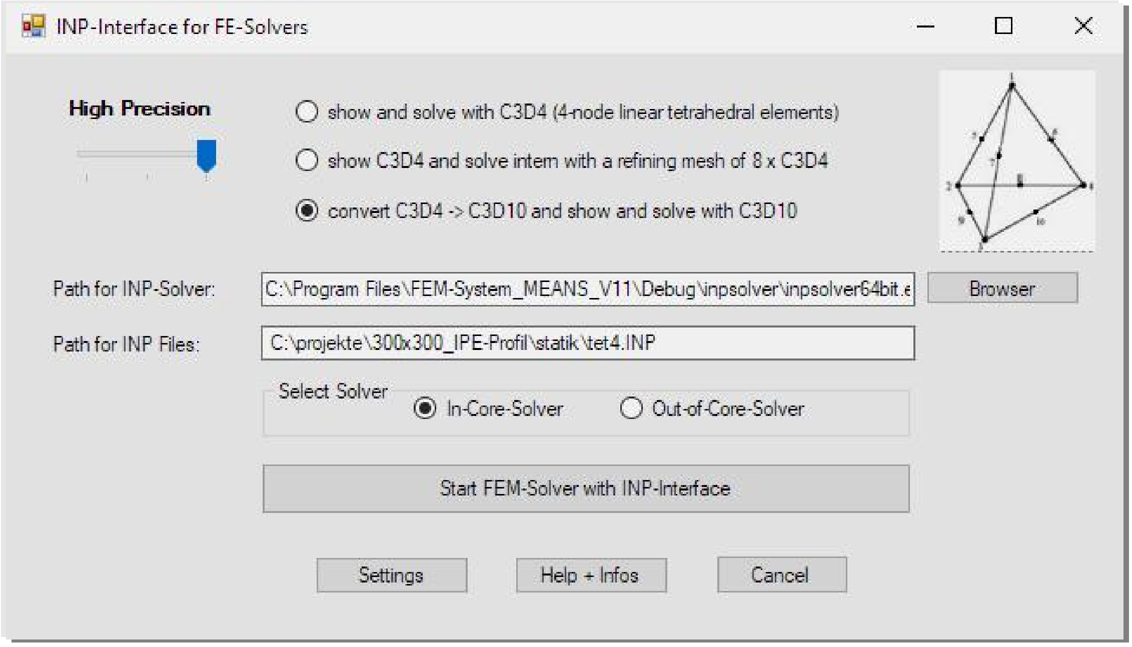
Before the FEM Analysis, always save the model with "Files" and "Save" under a name in the project directory.

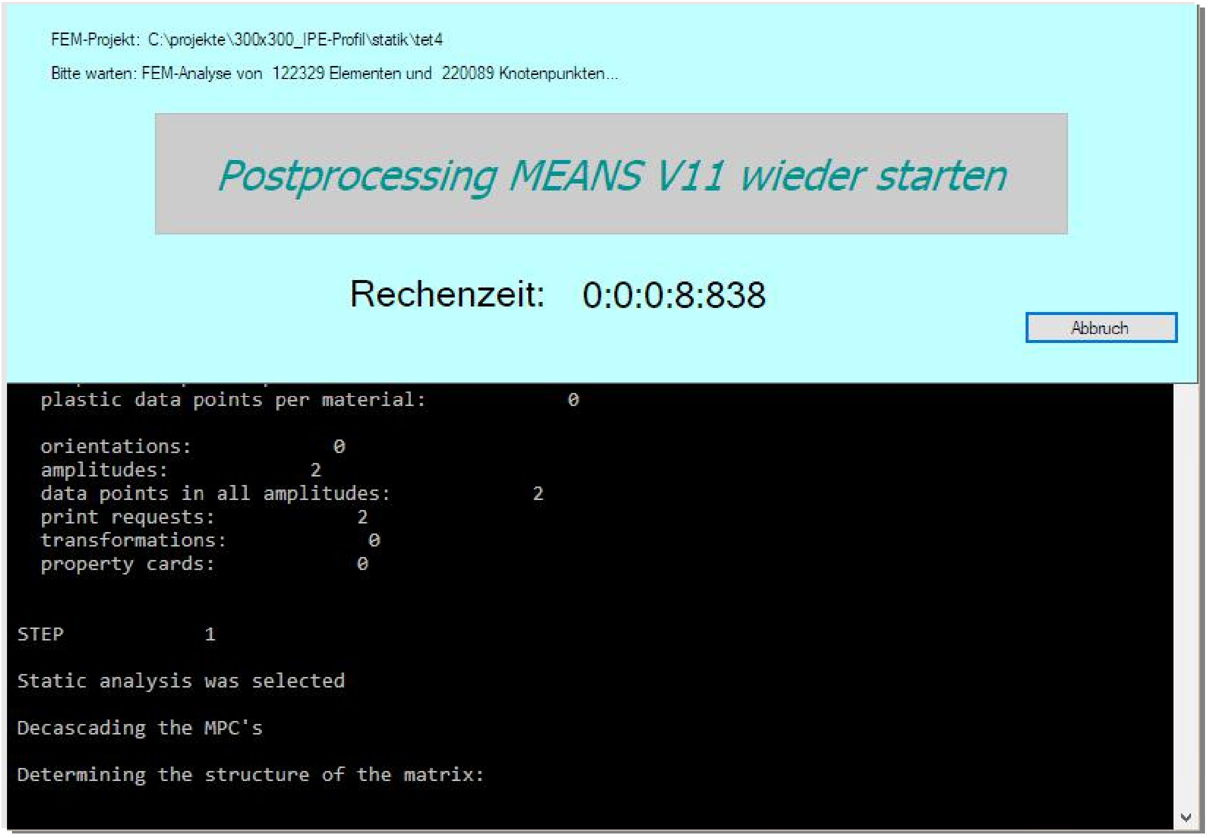
Select FEM-Analysis and “1. Statics” and click on the solver icon 
to start the FEM-Analysis with the Dr.Kühn-Solver or with the Quick-Solver.



For larger FEM structures over 100,000 elements, choose the faster Quick-Solver with three options for precision.

In the Quick-Solver, select "convert C3D4-> C3D10 ..." to perform the FEM analysis with the very accurate quadratic TET10 tetrahedral elements.

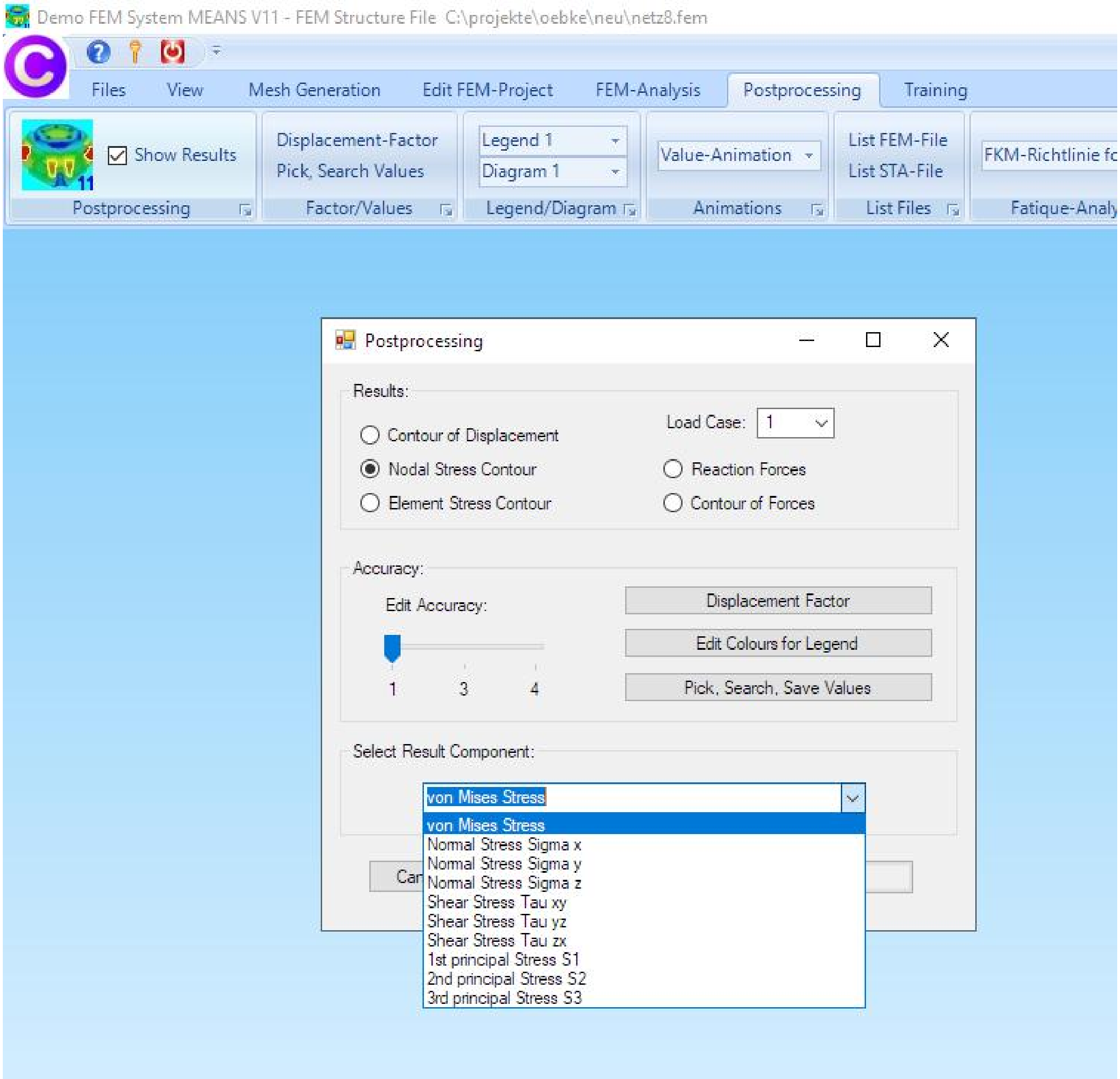




After a calculation time of a few minutes or even longer with the "Out-of-Core-Solver" a short tone signal can be heard, now the menu "Start Postprocessing MEANS V11" is enabled again and you can start the postprocessor for the result evaluation.

Postprocessing

Select the tab "Postprocessing" and click on the Icon  to start the Postprocessor for the result evaluation automatically.

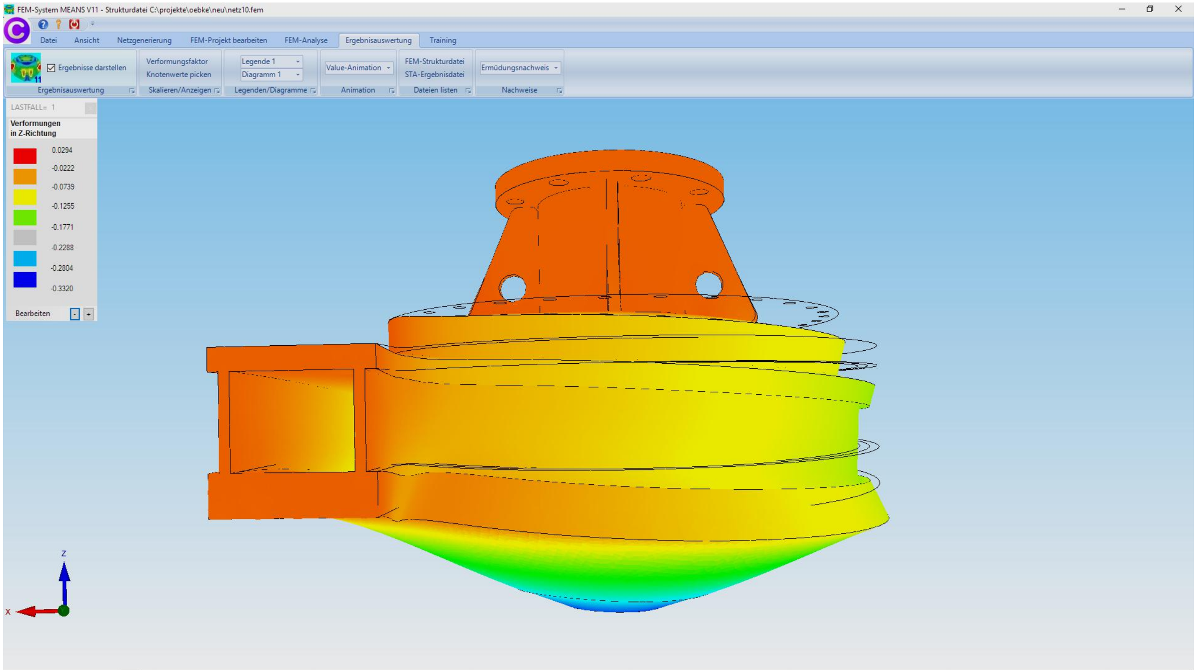


The following Result Evaluations are available:

- Contour of Displacments
- Nodal Stress Contour
- Element Stress Contour
- Reaction Forces
- Contour of Forces

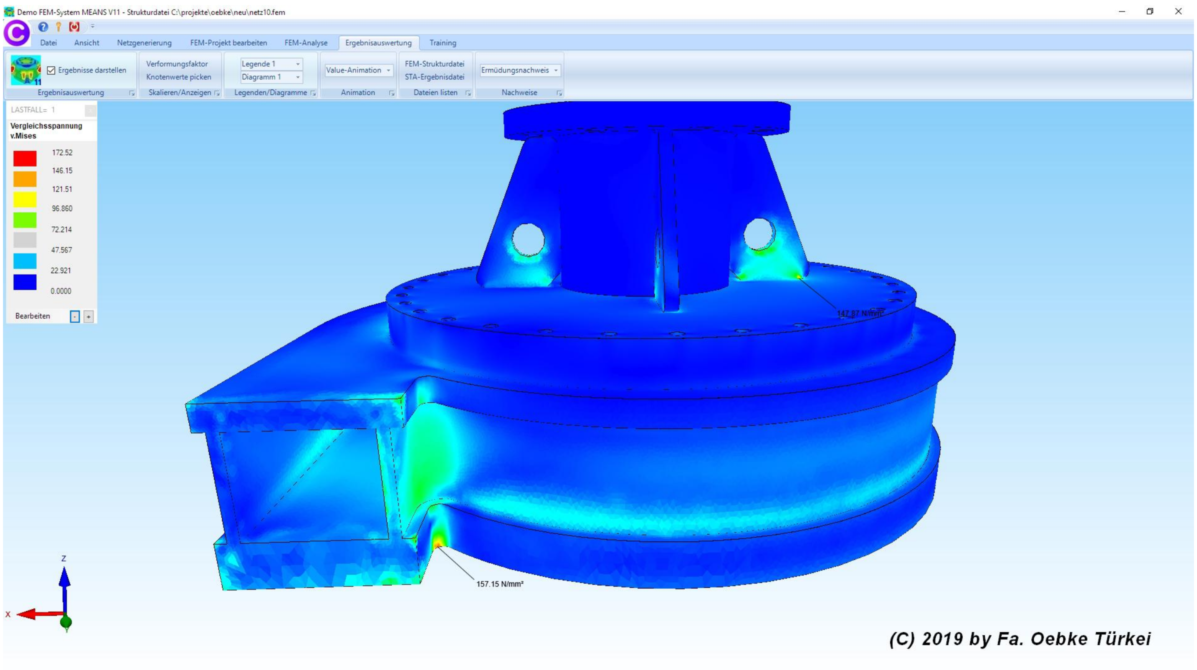
Displacements in Z-Direction

Max. Displacments in Z-Direction = 0.33 mm (with a Displacement Factor = 500)



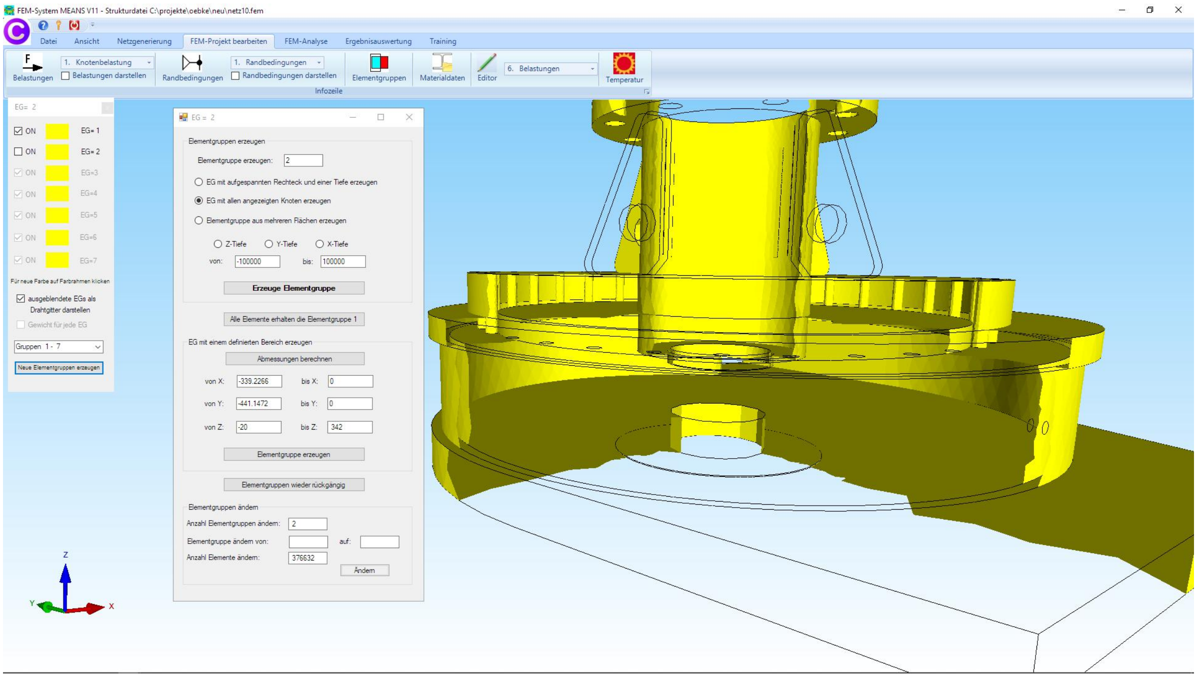
v.Mises Stresses

Max. v.Mises-Stresses = 157 MPa



Model-Cut

Create a Model-Cut with a new Element Group 2 and with a define range.



v.Mises Stresses

Max. v.Mises-Stresses = 157 MPa