Part 1: FEM-System MEANS V12 with a Ribbon-Interface

Hardware requirements for MEANS V12

- Windows 10 with 64-Bit is preloaded
- DirectX11 for Windows 10 is installed
- Working memory with 16 GB
- Hard disk with over 500 GB and more

Licence agreement

Before MEANS can be installed, the licence arrangements must be accepted by HTA software. If you do not agree with it, you may not use MEANS V12 as a full version install only as a limited demo version on your PC.

Installation in different directories

If the installation is started normally, MEANS V12 will be in the current directory

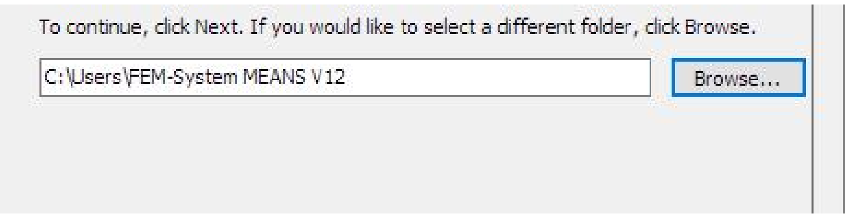
C: \ Programs \ FEM-System MEANS V12

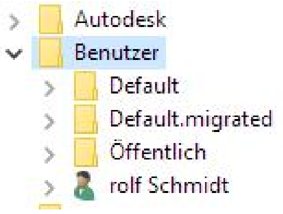
installed. However, MEANS must always be started there with "Administrator Rights".

However, if this directory is write-protected, it can also be installed in the non-write-protected User directory using "Browse ...":

C: \ Users \ FEM-System MEANS V12

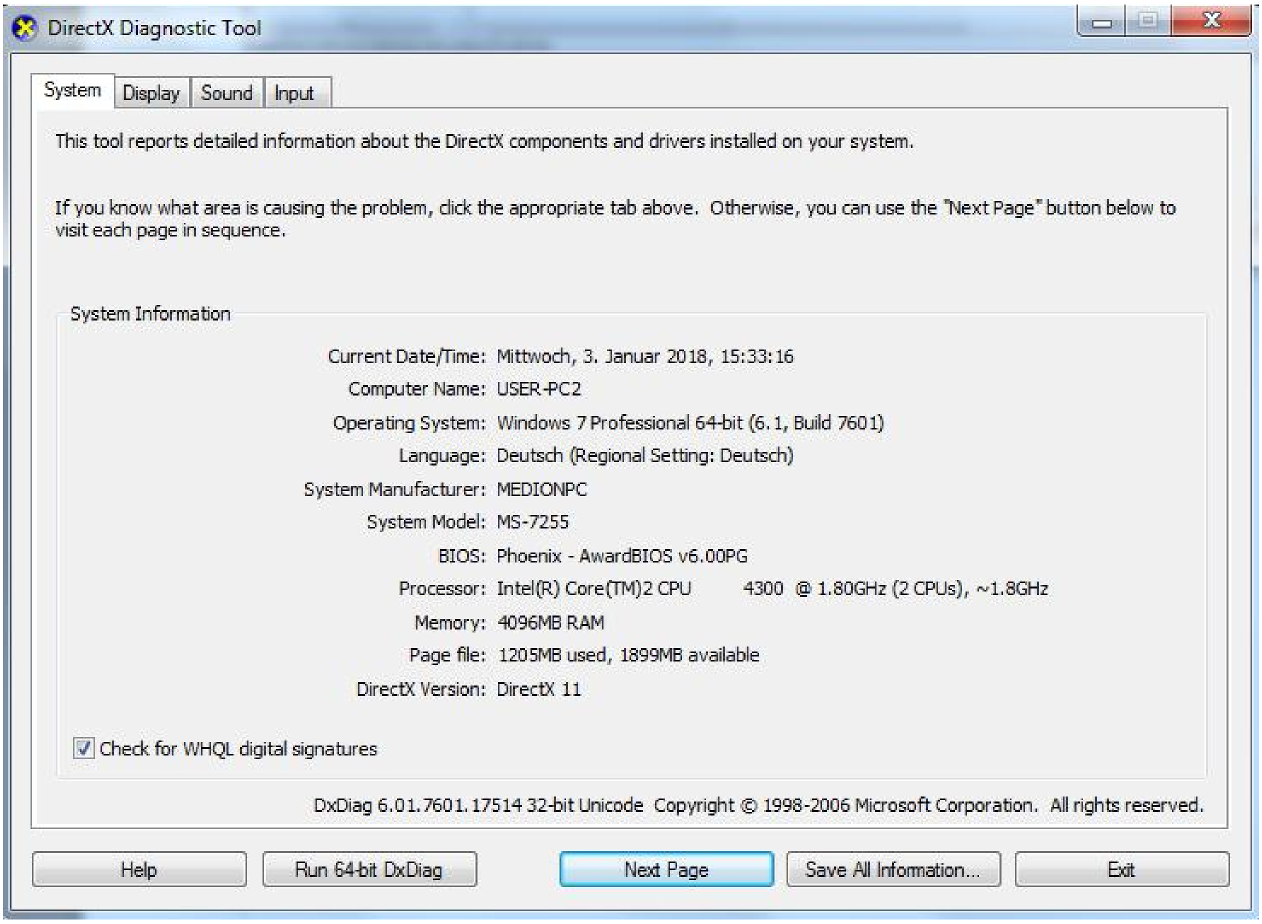
However, the system-relevant DirectX11 must also be installed in the write-protected program directory.





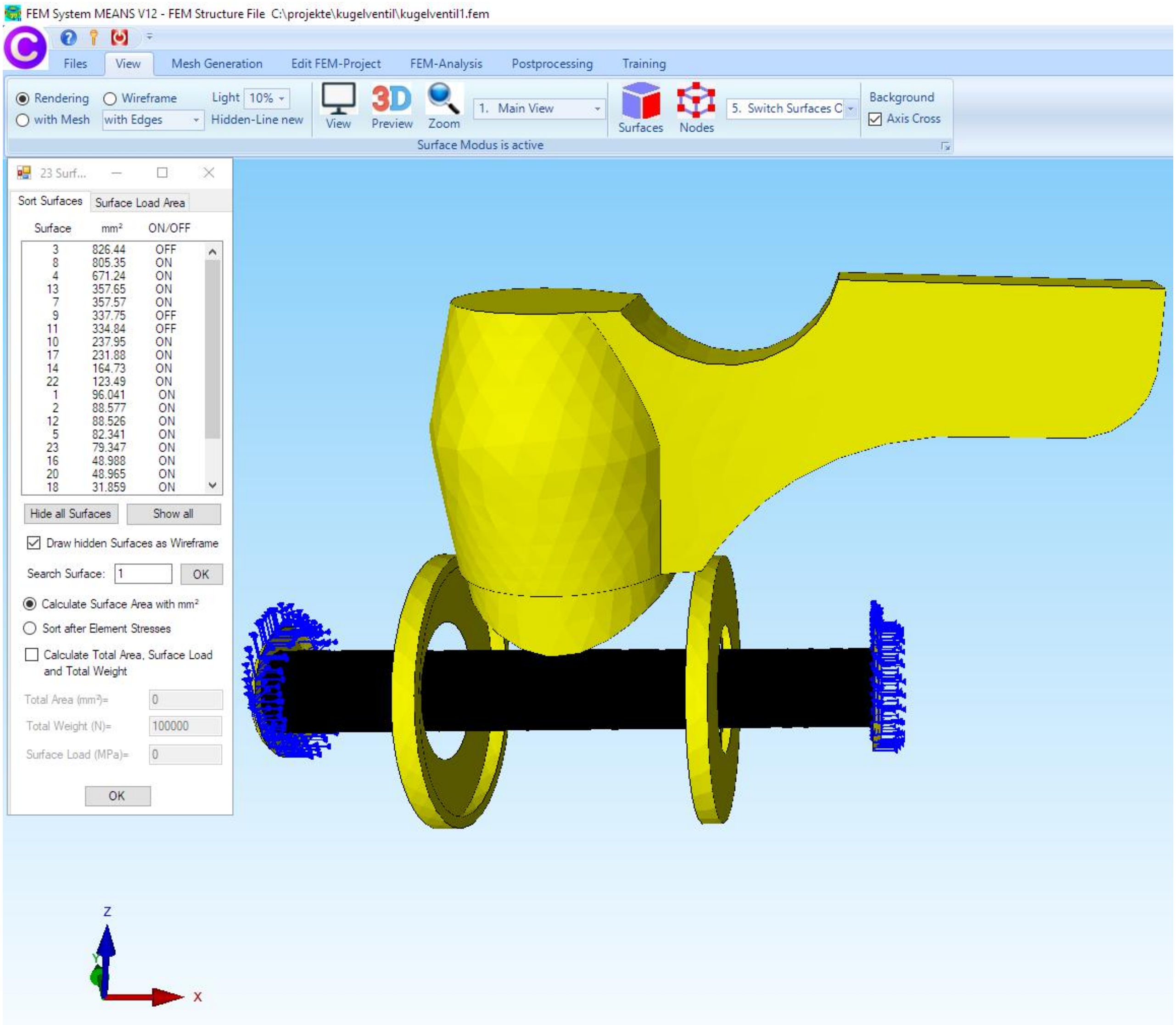
Check DirectX11 installation

In Run, enter the command "DXDIAG" to check the DirectX11. A prerequisite for DirectX11 is that the current DirectX11-capable graphics card driver is installed.



Ribbon user interface of MEANS V12

The new ribbon interface of MEANS V12 consists of a ribbon or Ribbon that links the menu control, toolbars, and dialogs elements. Instead of using a menu item, you now call up the complete ribbon via a tab, for example "File".
Thus, the new MEANS V12 user interface is much more clearly arranged and faster to operate than the old interface of MEANS V10 with the many small icon bars and dialog boxes.

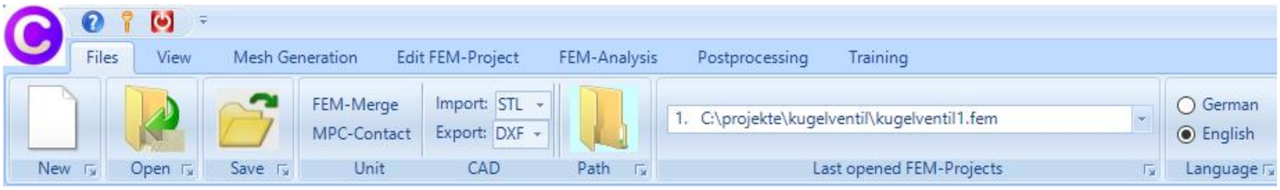


Above, the FEM model of the Ball Valve with Hidden-Line in the Surface-Modus with the surface load and the boundary conditions are shown in different colors.

MEANS V12 has the following 7 Ribbon Tabs:

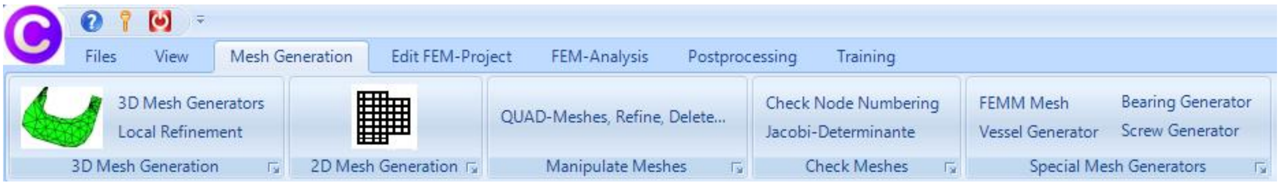
- **File**
- **Views**
- **Mesh Generation**
- **Edit FEM Project**
- **FEM-Analysis**
- **Postprocessing**
- **Training**

Files

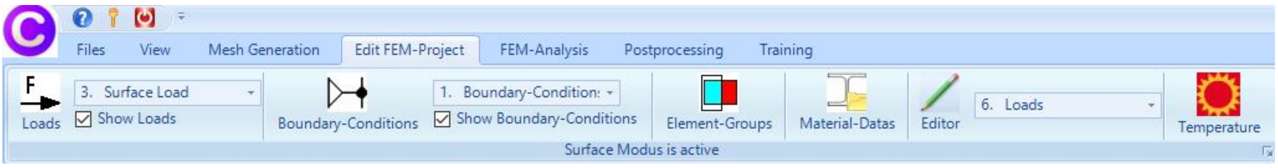


View

Mesh Generation

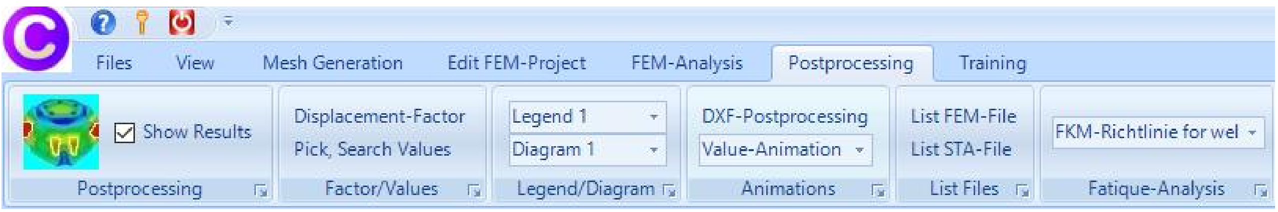


Edit FEM-Project

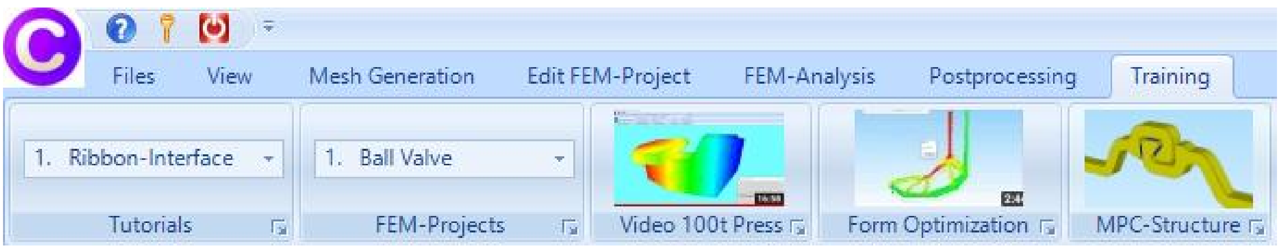


FEM-Analysis

Postprocessing

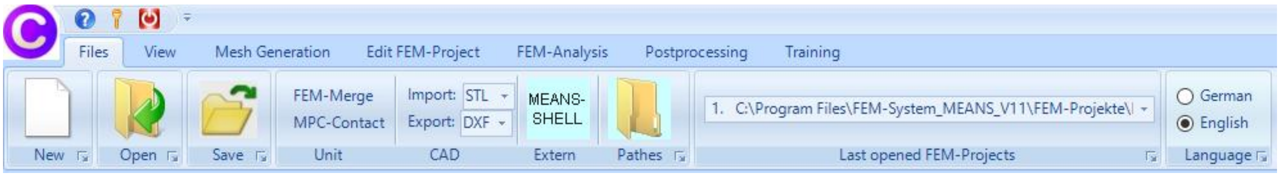


Training

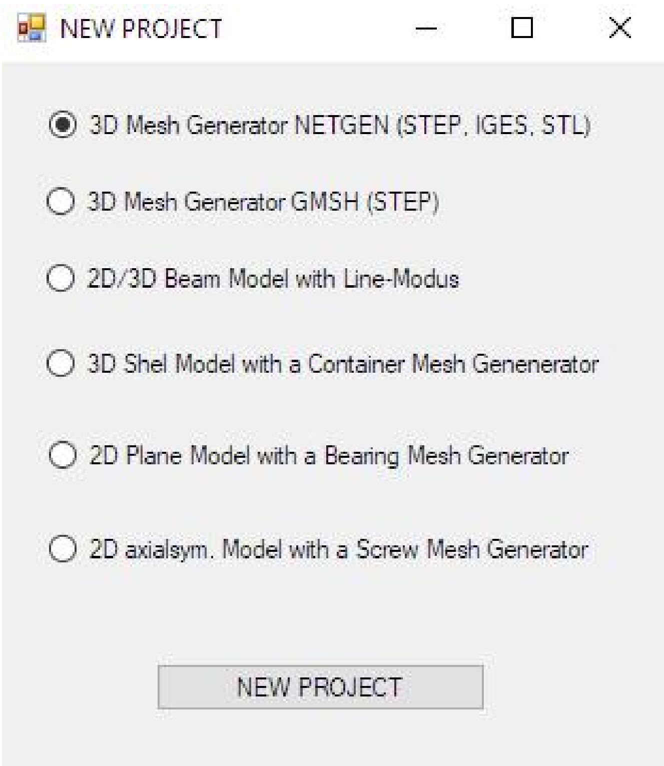


Mesh Generation

Based on the simple FEM example Ball Valve, the Ribbon Interface is now described.



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



Select menu "3D Mesh Generator NETGEN (STEP, IGES, STL)“ to appear

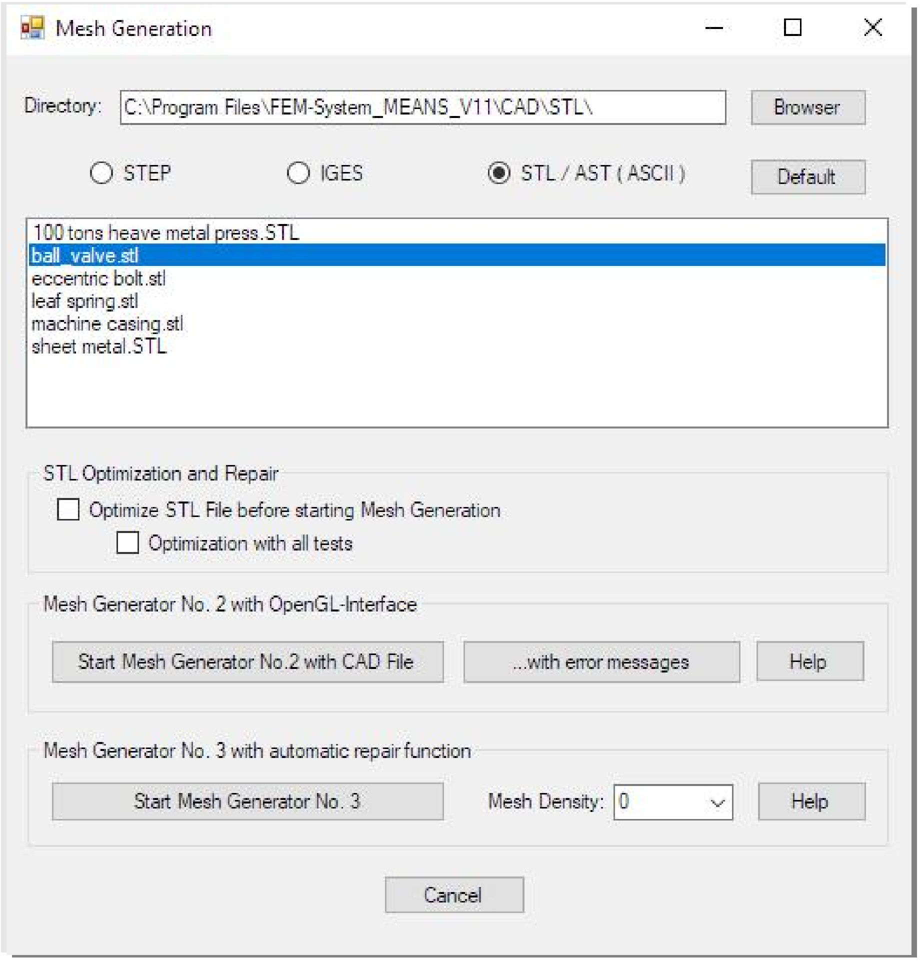
a dialog box, with following CAD formats:

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

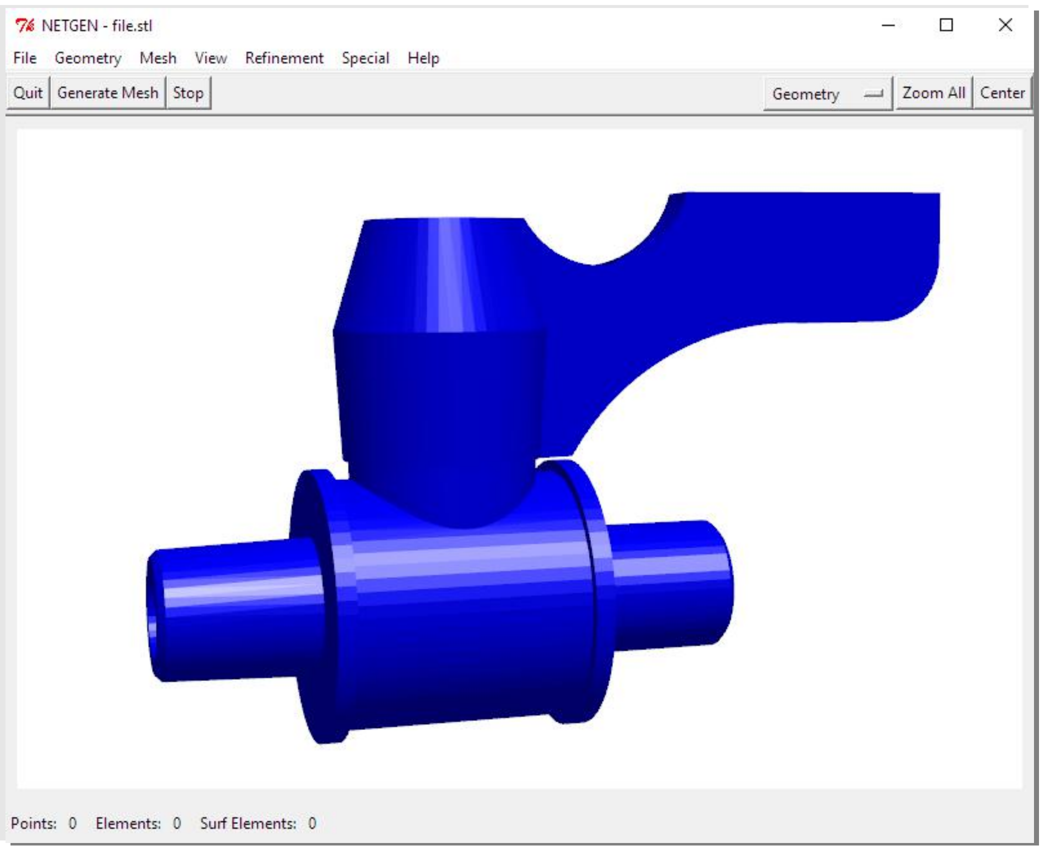
as well as other CAD formats:

DXF is now in the Line mode for reading circles, lines or polygons
3DS can converted with the external tool MEANS-SHELL or with
the Visual Basic CAD Viewer 3DVBVIEW.

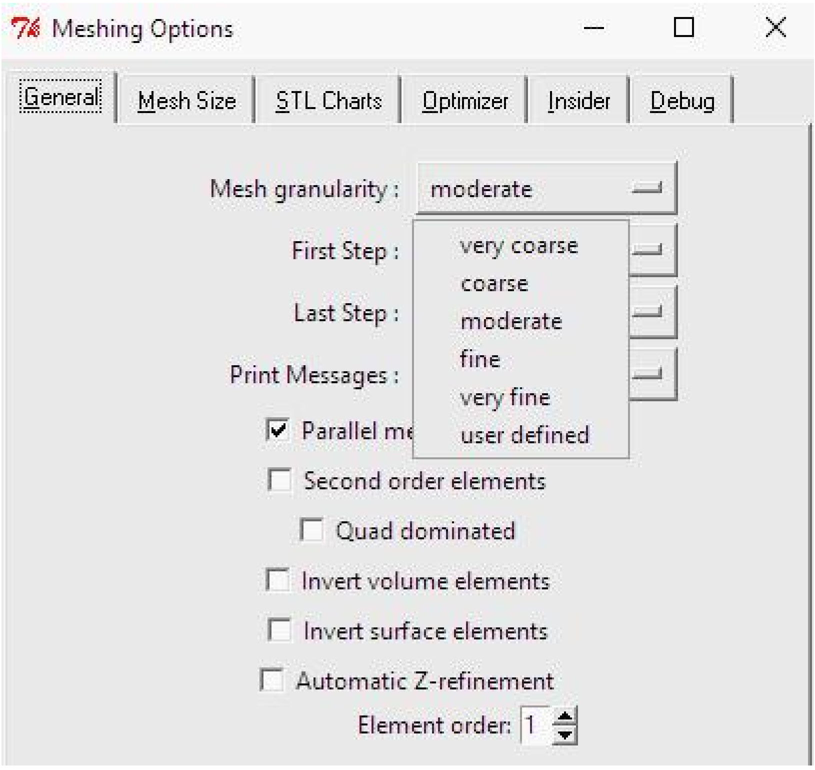
Select the STL file "ball valve.stl" with "Browser" and click on the button "Start Mesh generator No. 2 with CAD File" to display it in the mesh generator.



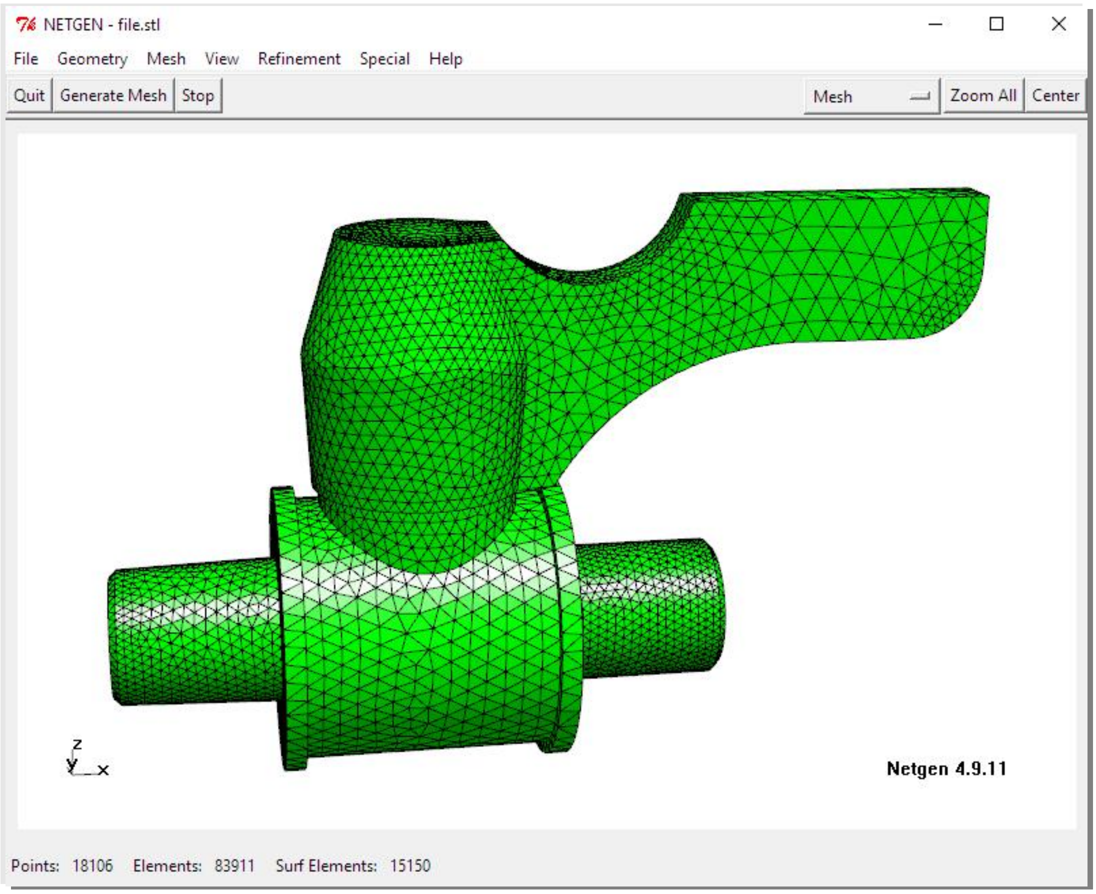
The STL model can now be seen in the mesh generator and can be rotated as required.



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

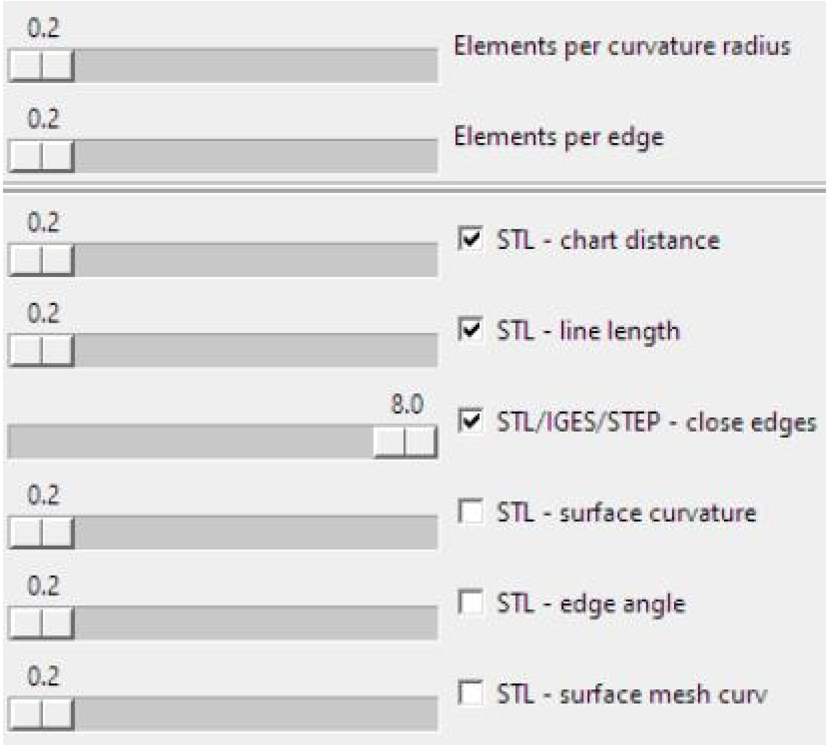


The generated FEM Mesh now consists of 18106 nodes and 83911 tetrahedral elements.

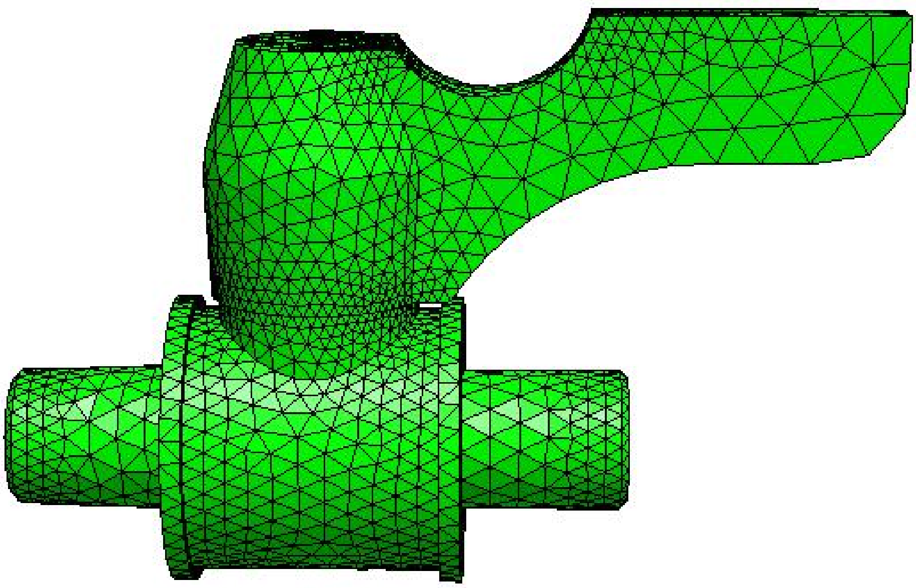


Coarsing

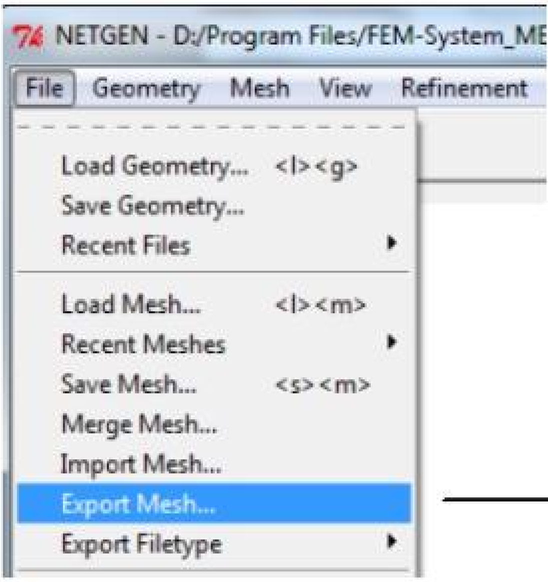
With the option mesh size "STL / IGES / STEP - close edges" and the following setting, "coarse" tetrahedral meshes can be generated, for example, to be able to reduce the number of elements for MEANS-LITE or is often use for complex or thin Structures is the only setting to obtain a useable FEM Mesh.



With the above setting, the generated FEM Mesh now only consists of 6 610 nodes and 29 249 elements.



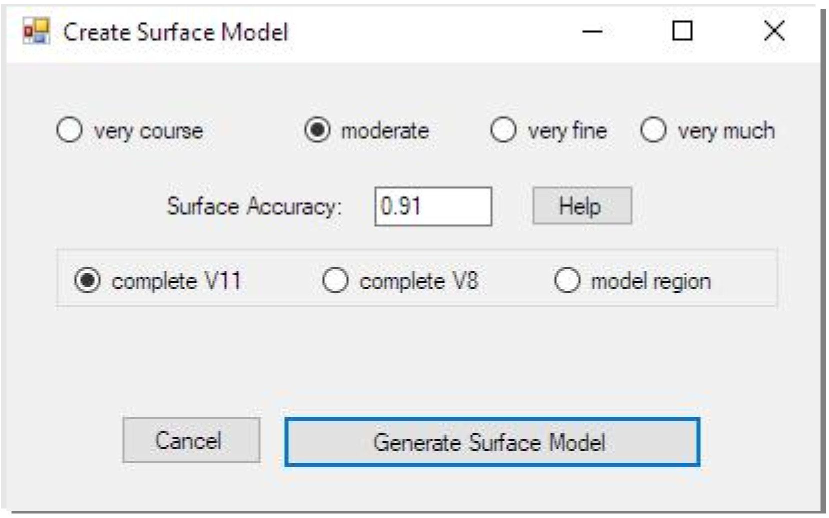
After Generation, the FEM Mesh named "test.fem" must be exported to MEANS V12. Select the "File" and "Export Mesh" menu and save the mesh "test.fem" into the default Debug / Mesh directory.



export here with the
name "test.fem" into the
debug / mesh directory
than MEANS V11 will
be started automatically

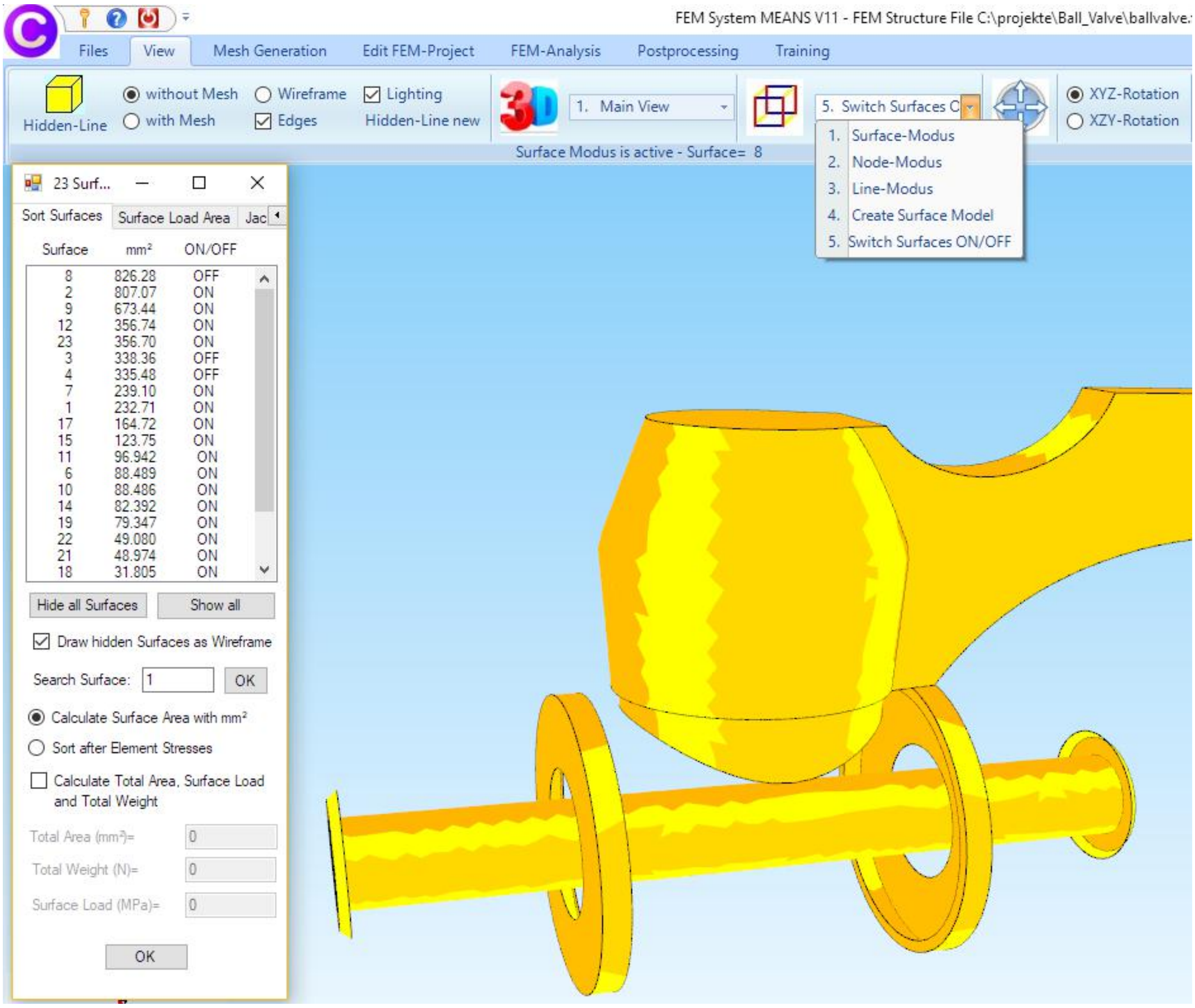
After exporting "test.fem", MEANS V12 will be started automatically and will first create the surface model so that surfaces, edges and nodes for loads, boundary conditions or element groups can be selected.

.



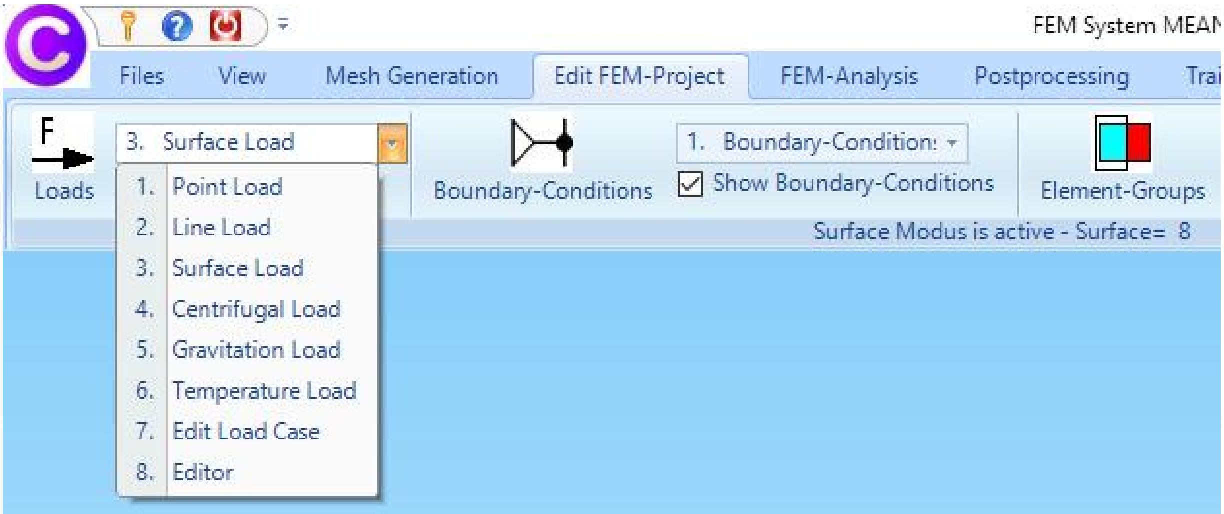
Show and hide surfaces

For the pressure load on the inner tube, first the outer surfaces 3, 4 and 8 must be hidden. Select the "View" tab and the drop-down menu "5. Switch Surfaces ON/OFF“ to hide these surfaces by clicking on the "ON / OFF" column.



Create Load Case 1

Select the „Edit FEM Project“ tab and the drop-down menu "3. Surface load " to enter load case 1 with a pressure load of 5 bar loaded on the inner tube with Surface 2.



You can enter 6 different load types:

Load type 1: node loads for all element types
Value input e.g. 10 000 N or 10 kN

Load type 2: Line loads for all element types
Value input e.g. 1000 N / mm or 1 N / m

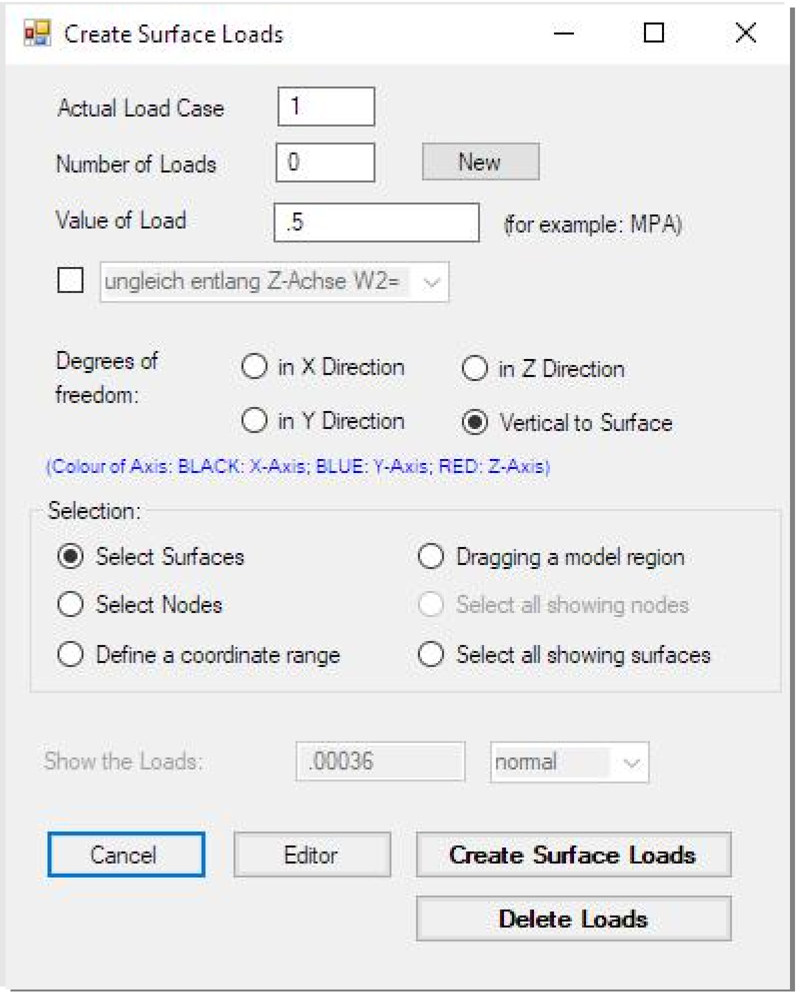
Load type 3: Surface or pressure load for plates, shells and volume elements
Value input in N / mm² or N / m² (1 bar -> 0.1 N / mm²)

Load type 4: Temperature load for all element types
Value input node temperature in degrees Celsius and
Heat transfer coefficients in material data

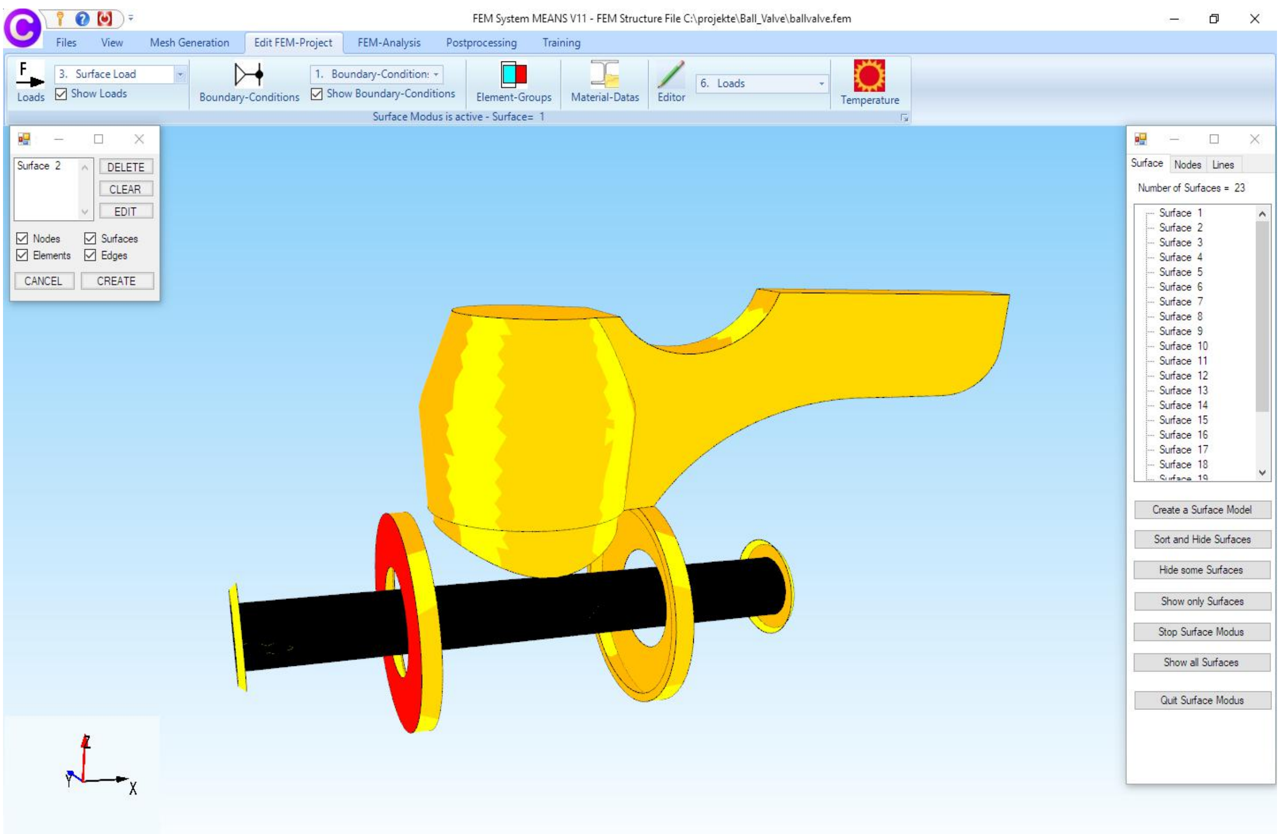
Load type 5: Centrifugal load for all element types
Value input in 1/s or 1/min and density in material data

Load type 6: Gravity load for all element types
Value input e.g. 9.81 m / s² and density in material data

In the next dialog box, enter Load case 1 with the value 0.5 N / mm² (= 5 bar) and with the degree of freedom "Vertical to Surface" and with the Selection "Select Surfaces" and click on the button "Create Surface Loads".

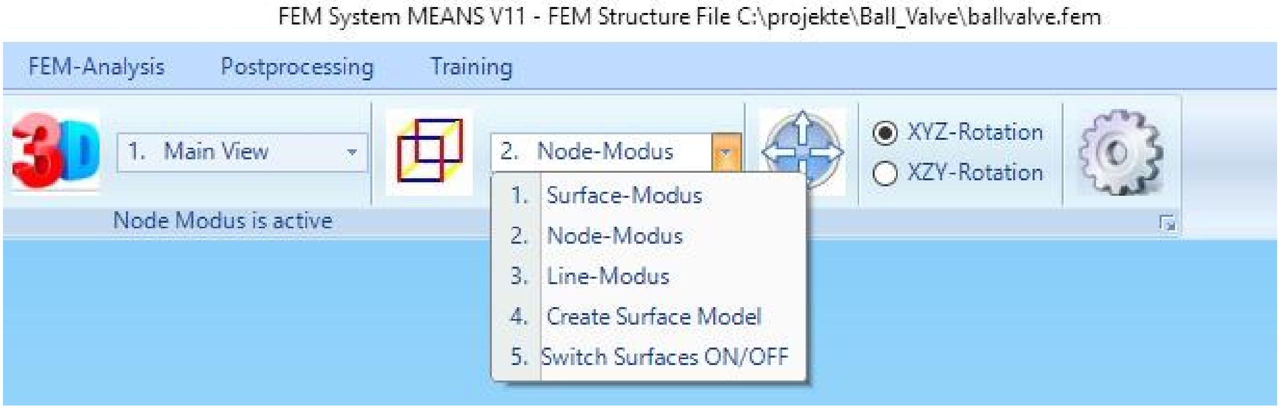


With a double-click on Surface 2 and choose menu "CREATE" in the Selectbox, the Surface Load is created.

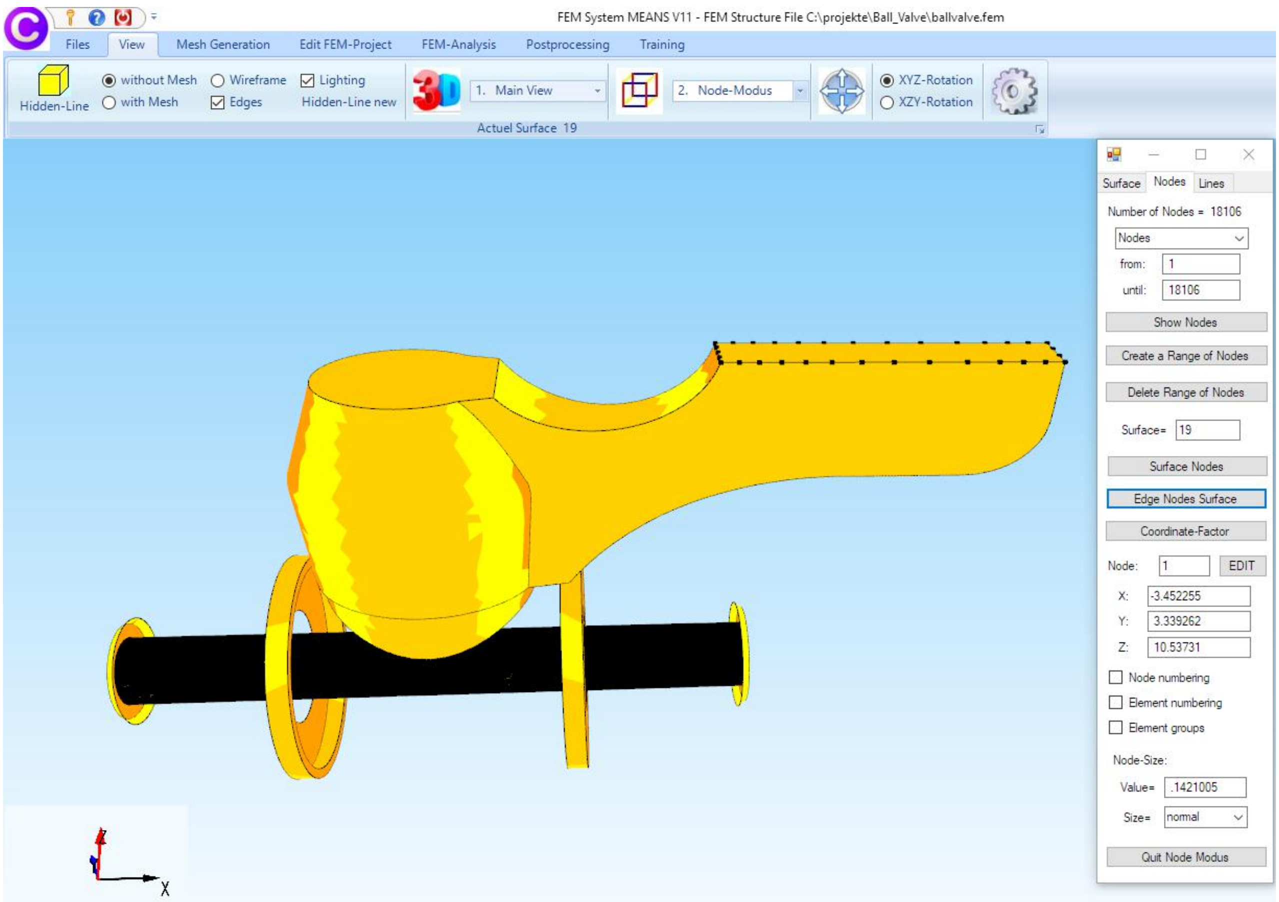


Create Load Case 2

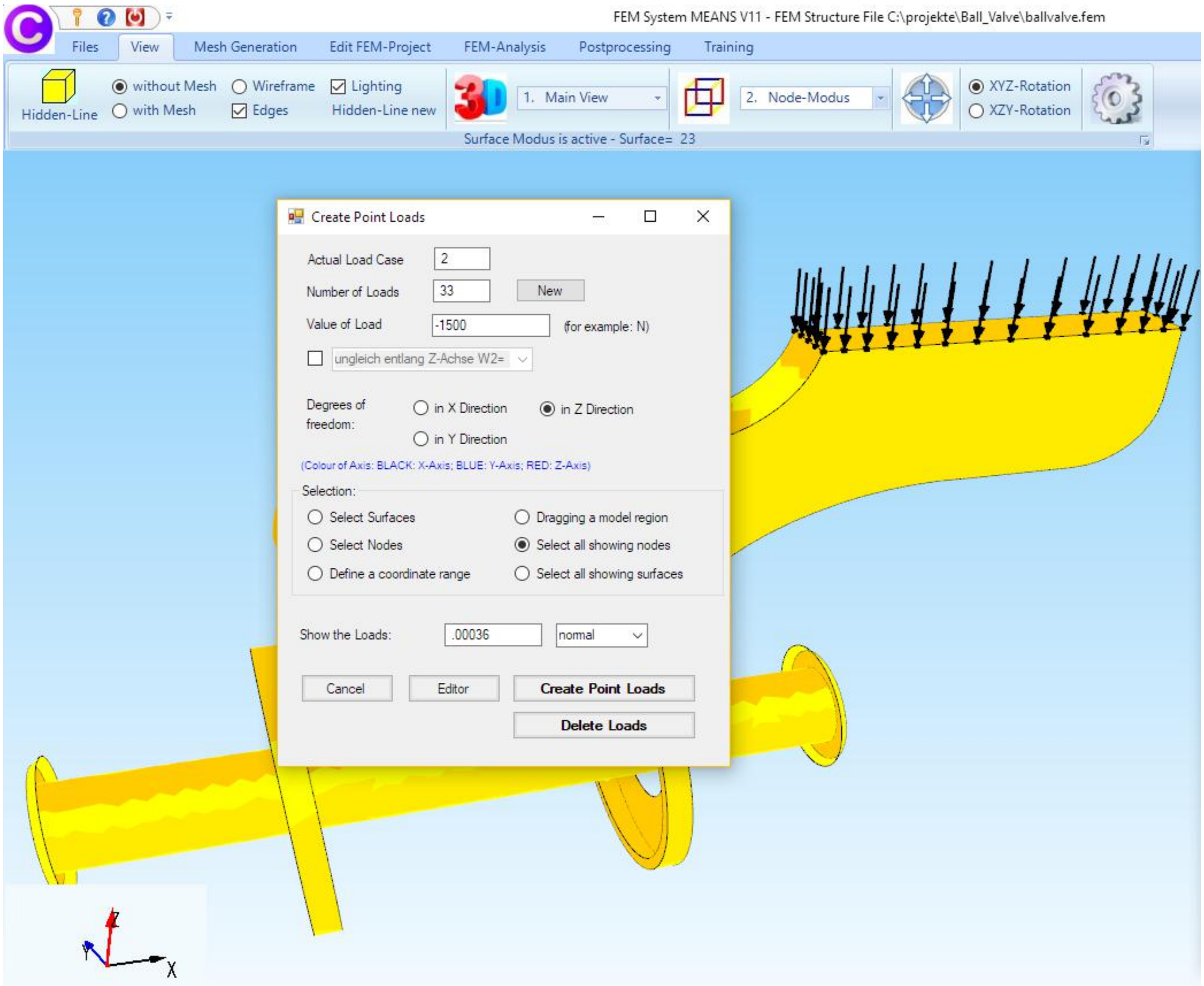
For the point load, a selectable Range of Nodes must first be created. To do this, switch in the "View" tab from the drop-down-menu „1. Surface-Modus“ to the „2. Node-Modus“.



In the right menu field of Node-Modus, enter Surface = 19 and select "Edge Node Surface“ to display the edge nodes of Surface 19 with the Node-Size = 0.142

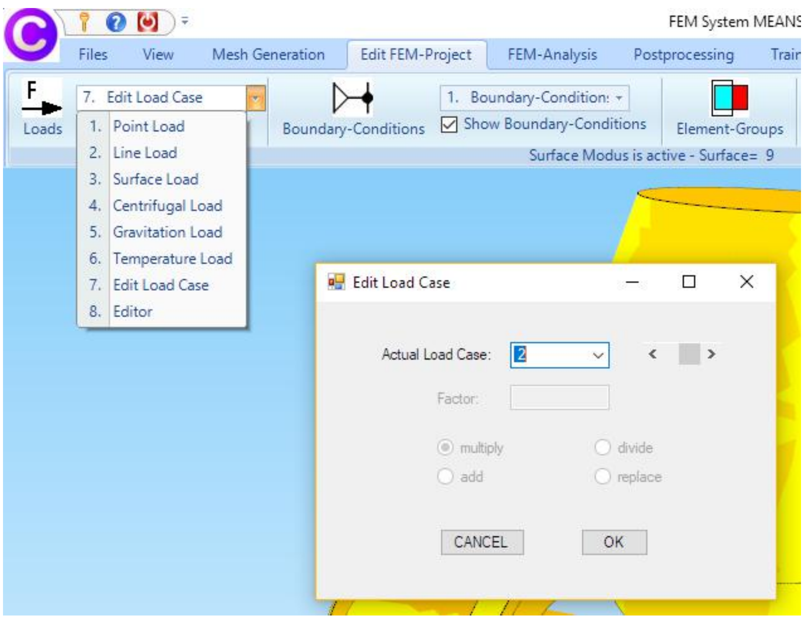


Select the "Edit FEM Project" tab and the "1. Point Load" drop-down menu to create Load Case 2 with a point load in Z-direction.
In the next dialog box, enter Load Case 2 with the value -1500 N and with the degree of freedom "in Z direction" and with the Selection "Select all showing nodes" and select "Create Point Load" to create a point load with 33 nodes.



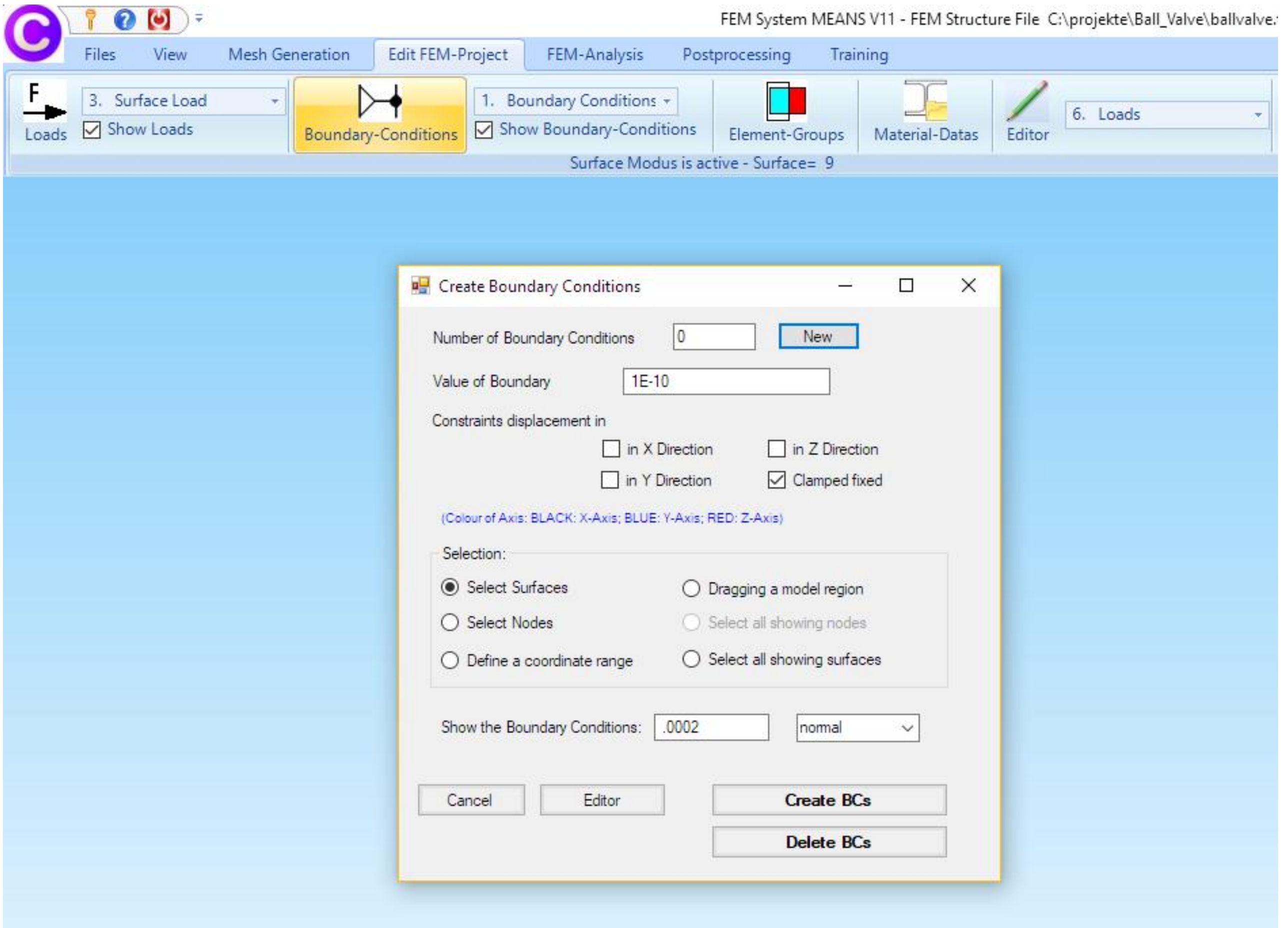
Set load cases

With the dropdown menu "7. Select load case ", load case 1 and load case 2 can be set. Also, the loads can be edited with menu „8.Editor“ to delete the loads, copy them or change them with a load case factor.



Create Boundary Conditions

To clamp the model, select the "Edit FEM Project" tab and click on "boundary conditions".



The boundary conditions are defined by the node and the degree of freedom. An additional value specification indicates how large the displacement or the rotation of this boundary condition is. This value is almost always zero or very small, since in practice solid bearings or clamps predominate. An exception are the spring constants and the elastic bedding.

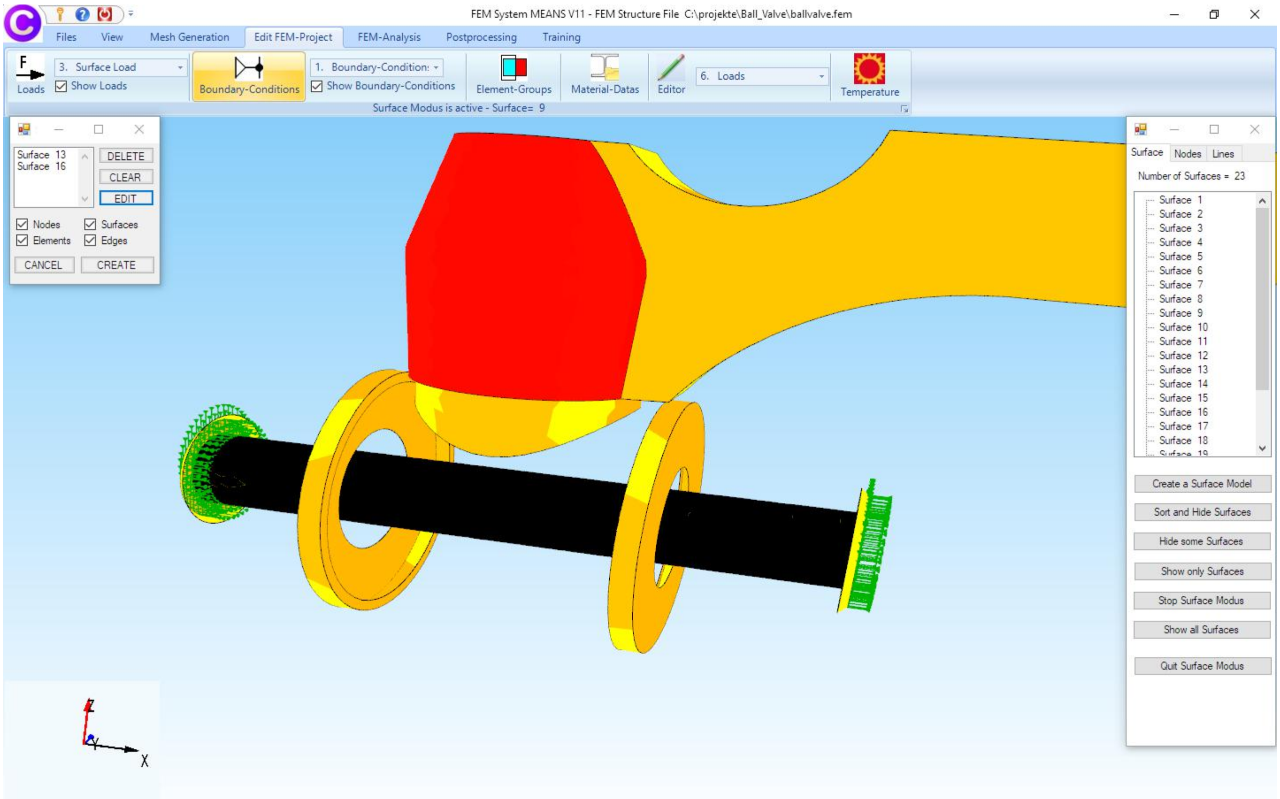
Degrees of freedom

FHG = 1 Constrain the displacement in X direction
FHG = 2 Constrain the displacement in Y direction
FHG = 3 Constrain the displacement in Z direction

Additional degrees of freedom for BEAM2 and Shell elements:

FHG = 4 Constrain the rotation about X axis
FHG = 5 Constrain the rotation about Y axis
FHG = 6 Constrain the rotation about Z axis

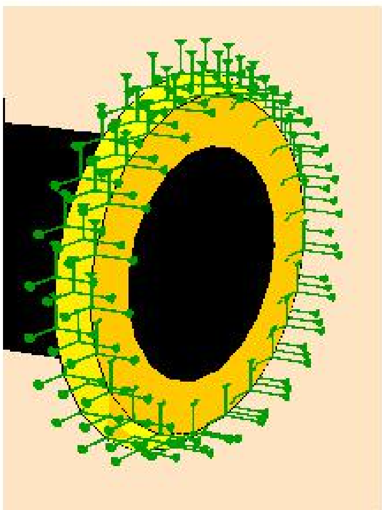
Select in the next dialog box "Clamped fixed" and the Selection "Select Surfaces" and click on the button "Create RBs" and double click on the surfaces 16 and 13 and confirm in the Selectbox the input with "Create".



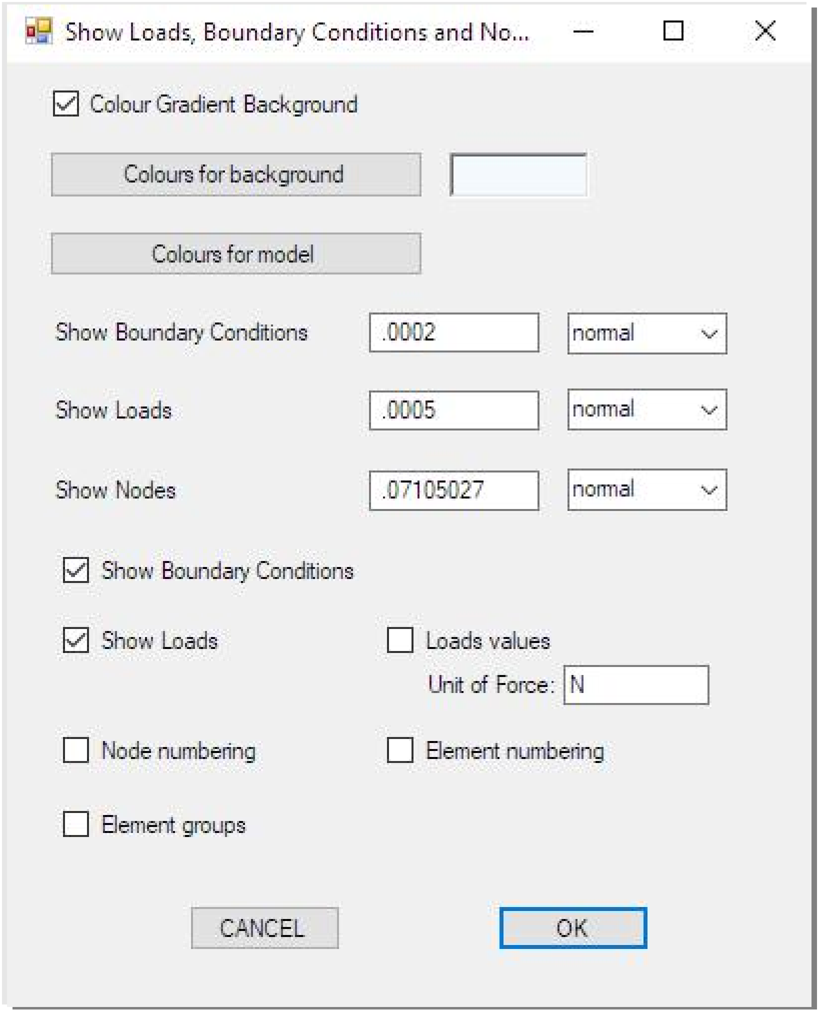
Select the "View" tab and the menu  to select the color for model, intersection, edges, background, loads or boundary conditions by clicking on the color frame.

The following dialogue box for the settings appears in the DirectX9 interface:

Randbedingungen "klein"



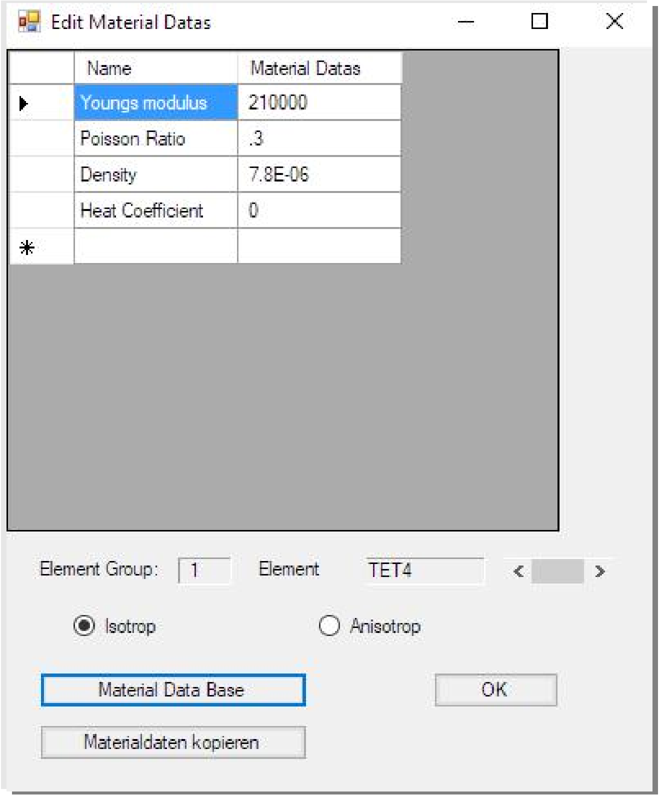
Randbedingungen "gross"


Show Loads, Boundary Conditions and No...
Colour Gradient Background
Colours for background
Colours for model
Show Boundary Conditions
.0002
normal
Show Loads
.0005
normal
Show Nodes
.07105027
normal
Show Boundary Conditions
Show Loads
Loads values
Unit of Force:
N
Node numbering
Element numbering
Element groups
CANCEL
OK

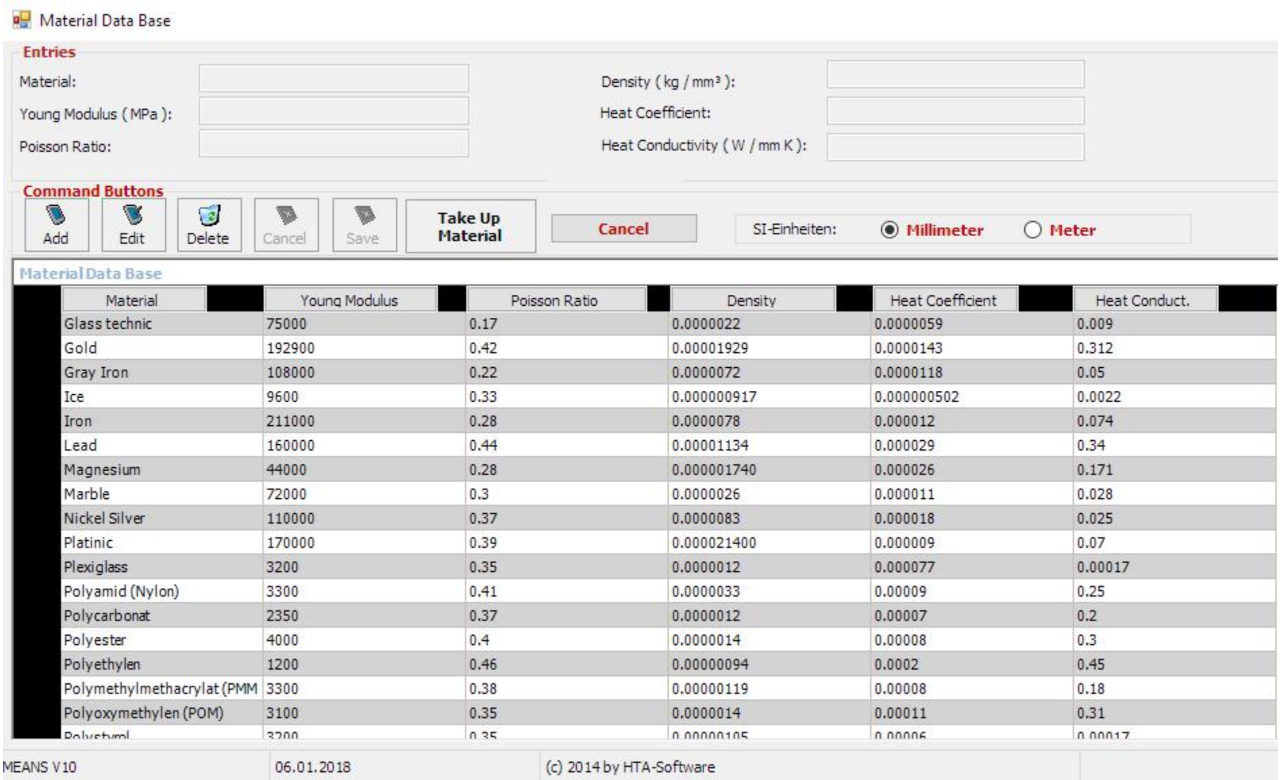
Create Material Datas

Select the "Edit FEM Project" tab and the icon 

to enter the material data such as Young's modulus and Poisson Ratio where steel is always preset.

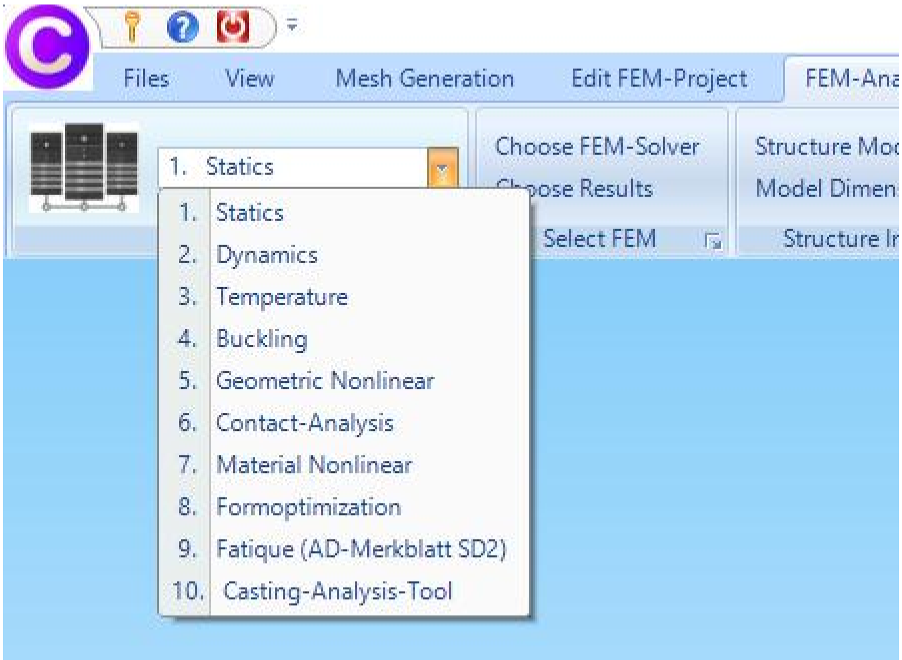


To call up an extensible material database, use the "Material Database" menu.

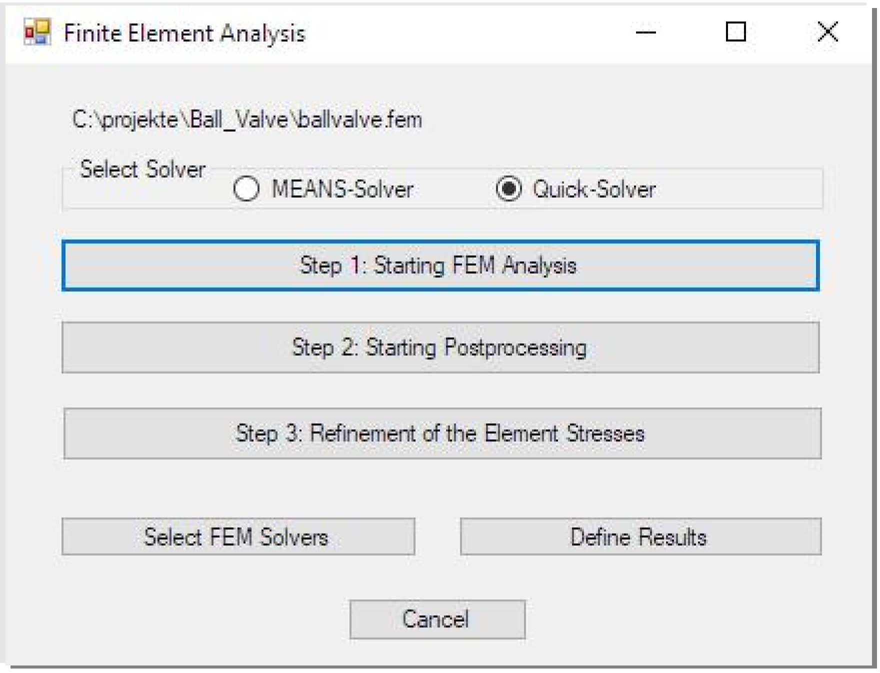


FEM Analysis

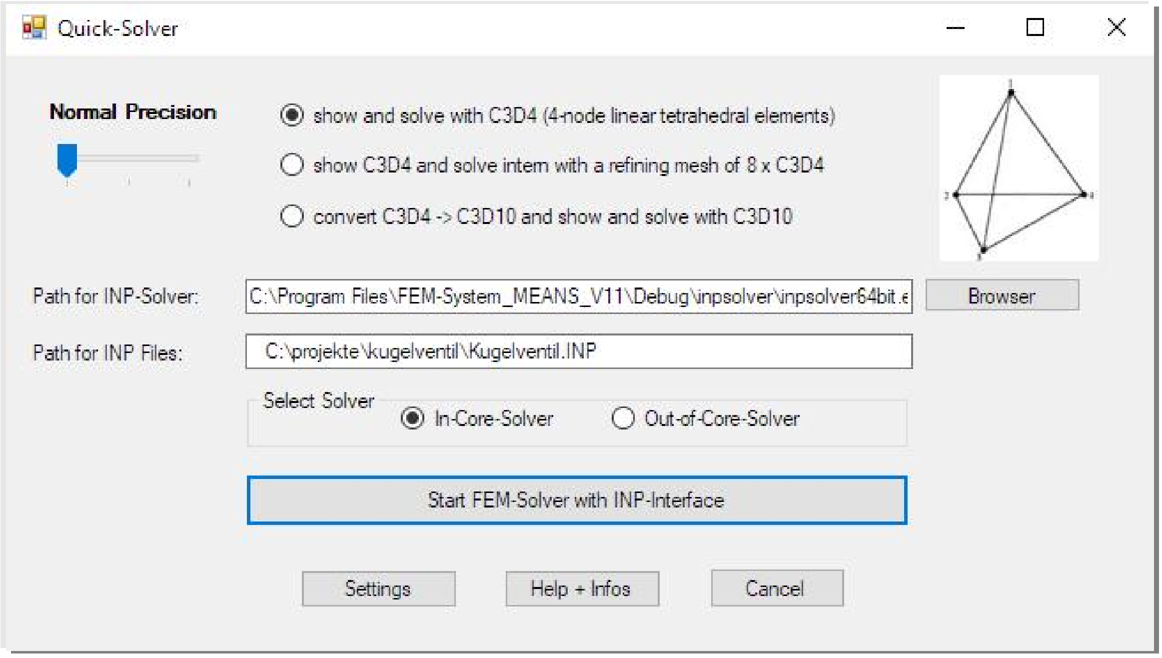
Following is a FEM Analysis, select the FEM Analysis tab. Here, the various FEM solvers for Statics, Dynamics, Temperature, Buckling, Nonlinear and Fatigue Analysis are called.

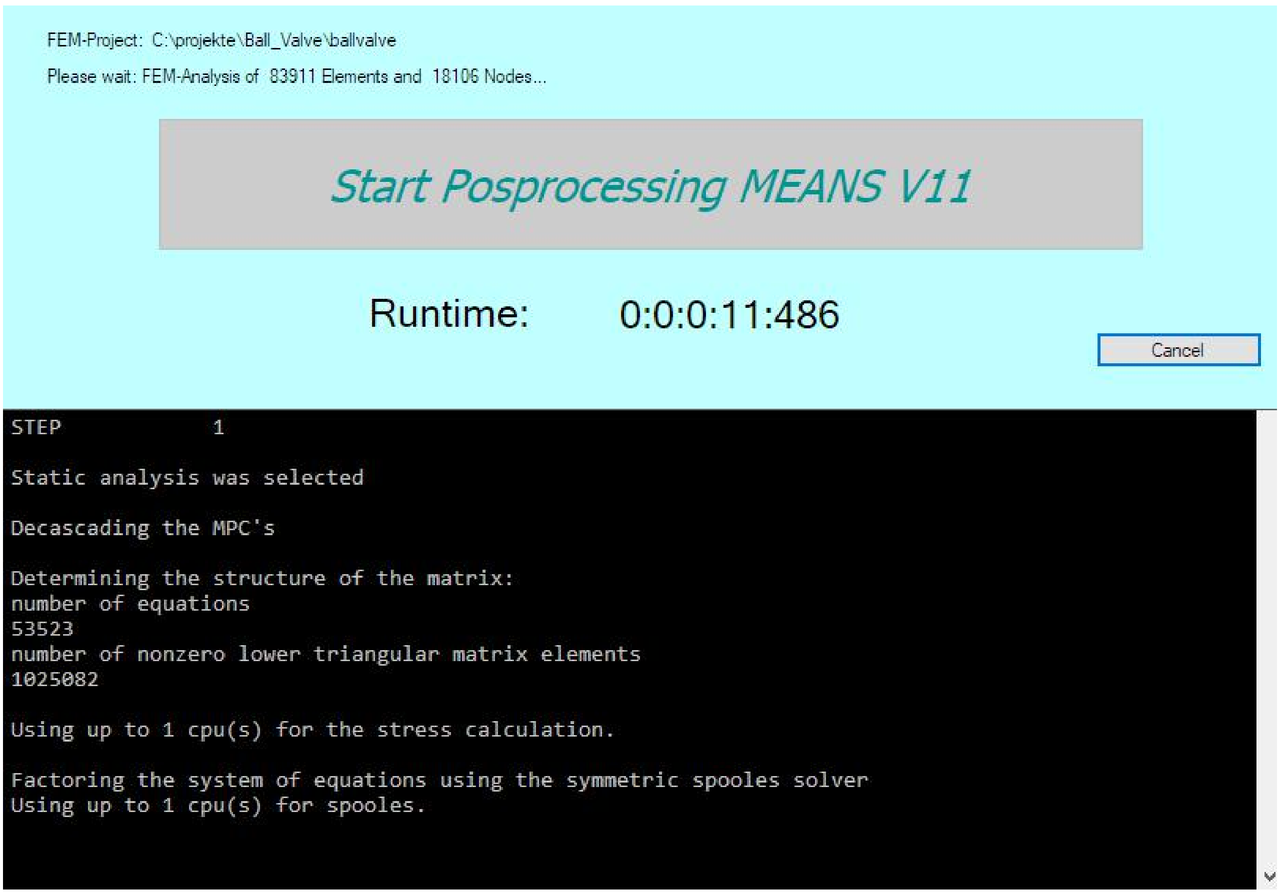


Select "1. Statics" for either the MEANS-Solver from HTA-Software or



or the quick "Quick-Solver" to calculate the deformations and stresses.

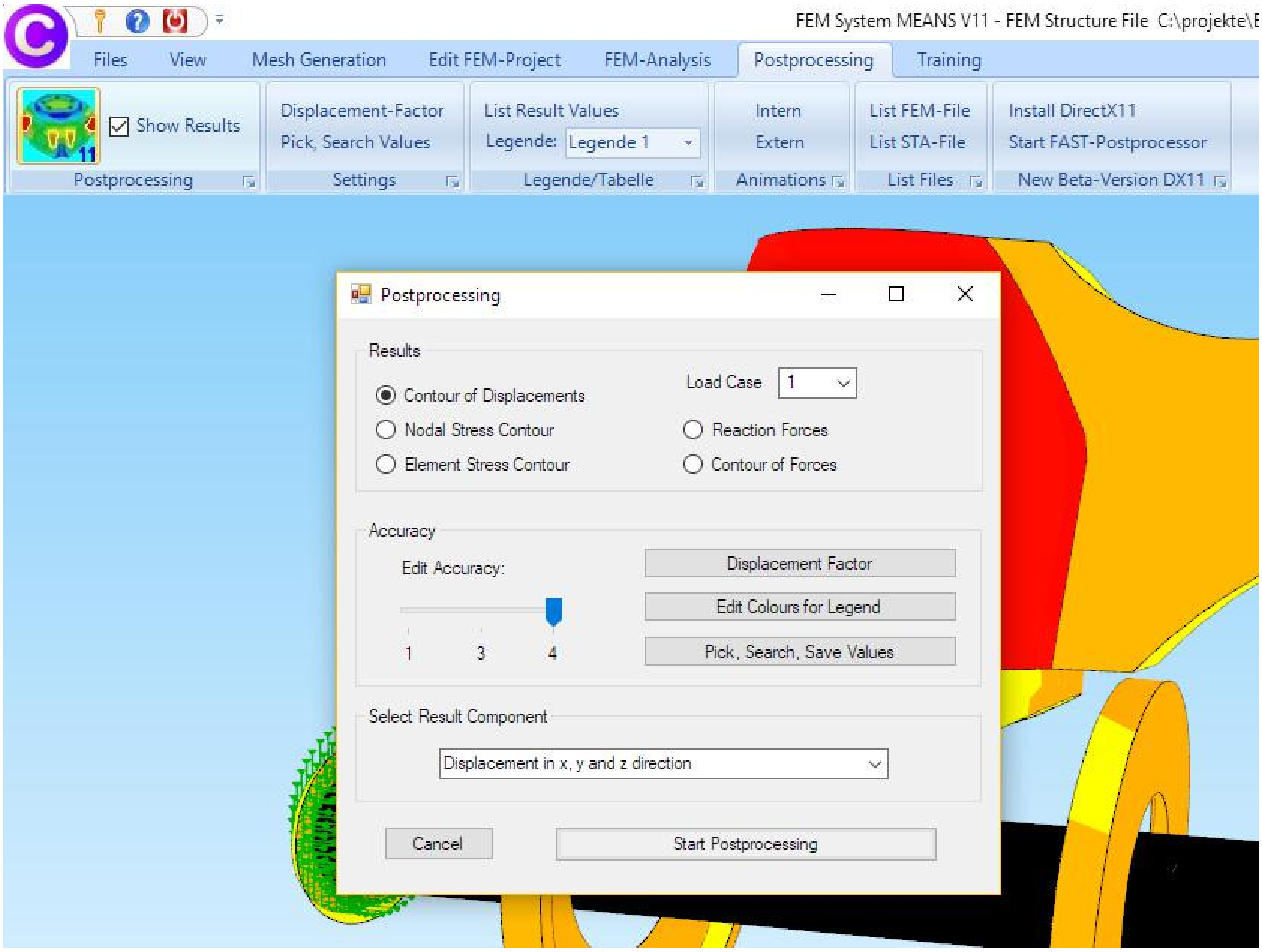




After the FEM analysis a short tone signal is heard, now the menu "Start Postprocessing MEANS V12" is active again and you can start the postprocessor for the result evaluation.

Postprocessing

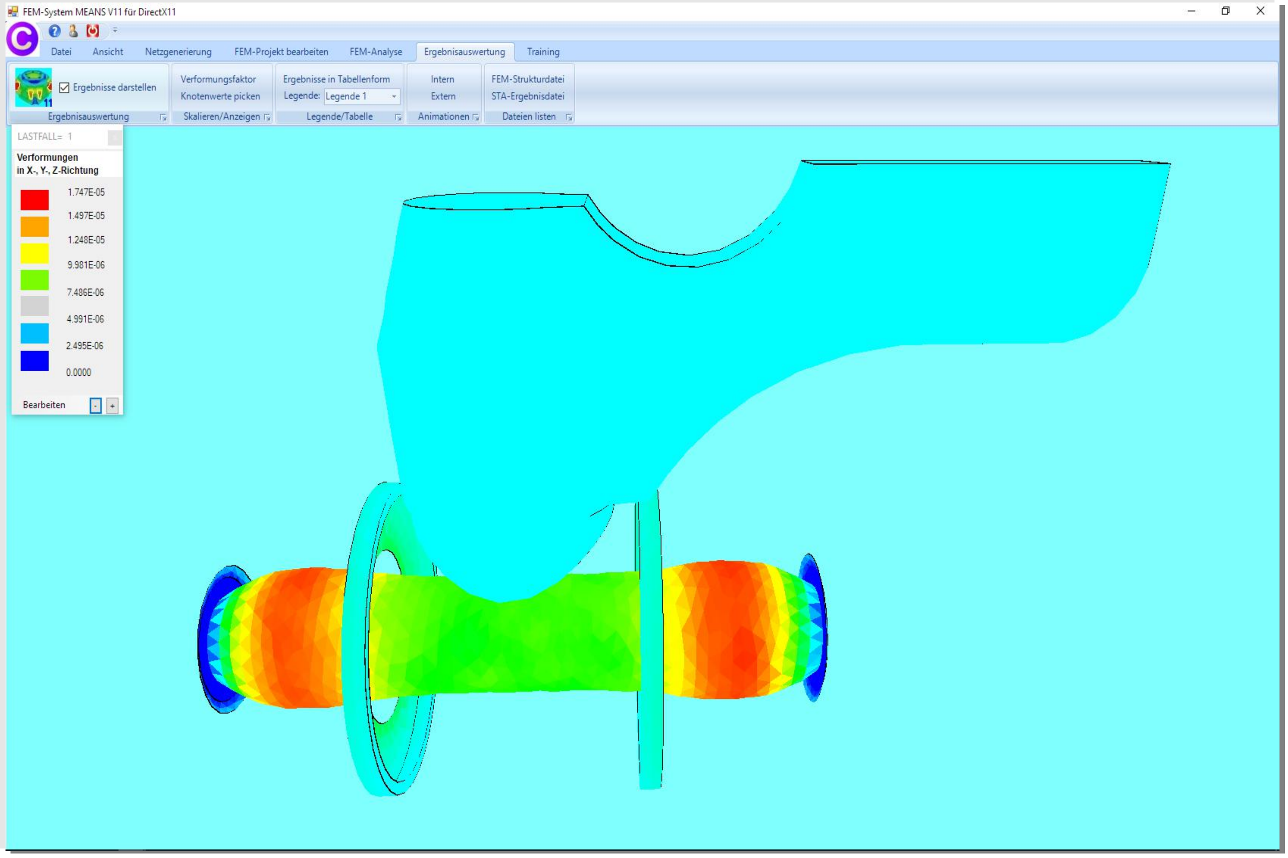
The postprocessor for the result evaluation is started automatically, select the tab "Postprocessing".



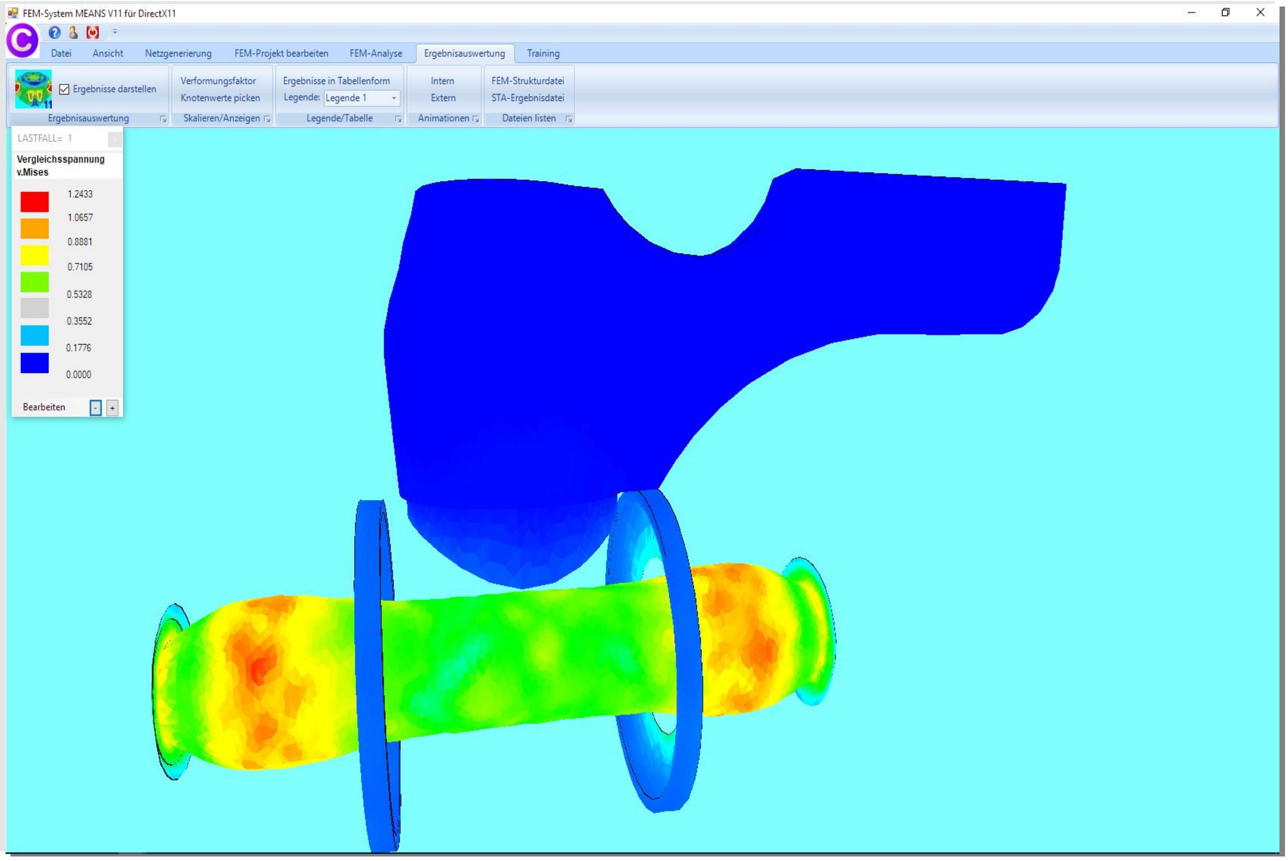
The following Result Evaluations are available:

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

Load Case 1: Displacments in X-, Y- and Z-Direction



Load Case 1: v.Mises-Stress



The Displacements are shown with a Displacement Factor of -3178872