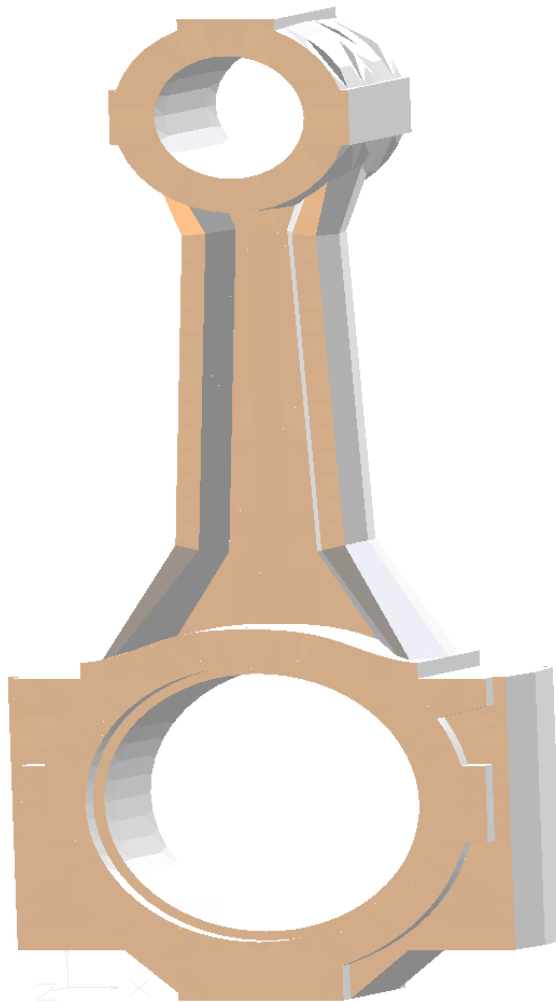


# Z88 AURORA® EXAMPLE MANUAL

## Example 3: Piston rod

(Tetrahedron No. 17 with 4 nodes)




### **3. Example: STL-import (Tetrahedron No. 17 with 4 nodes)**



This example should show how to import geometry data which is stored as a STL-format (surface mesh) into Z88 Aurora. The geometry data representing a connecting rod is designed in the CAD-system Pro/ENGINEER WF4.0 and exported as a STL-format. STL-files can be created with nearly every CAD-system.

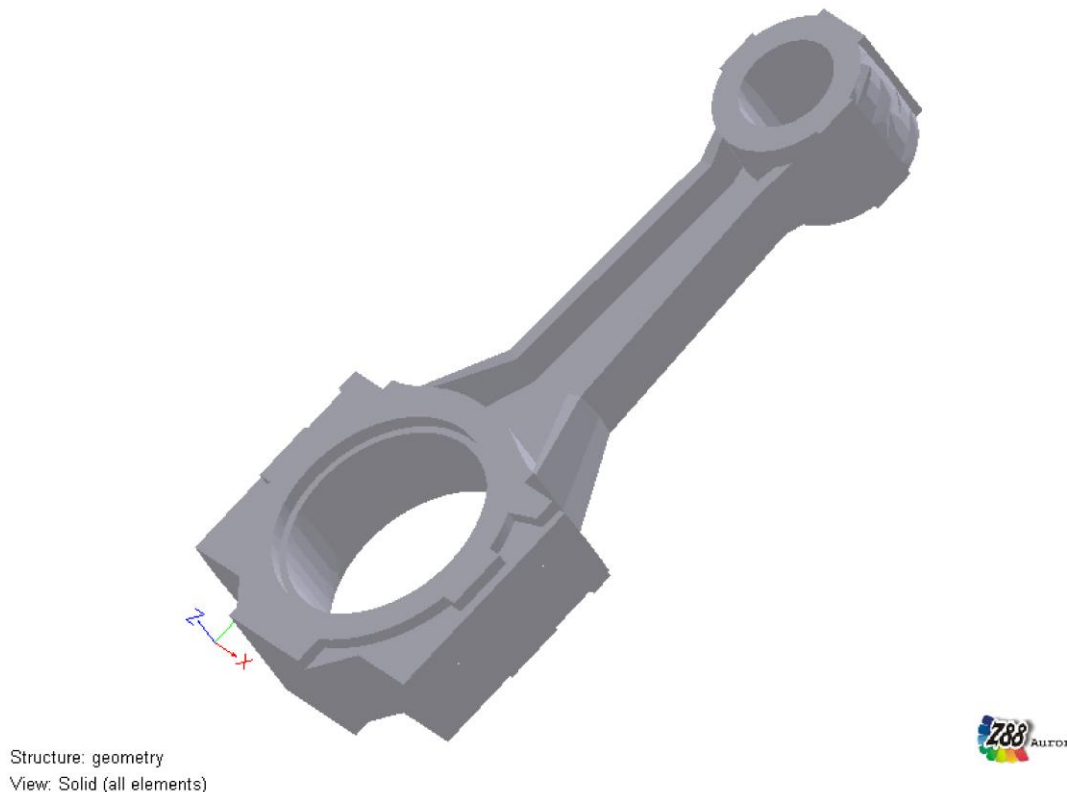
These STL-files representing the geometry by using a surface mesh, but it is not possible to simulate with these meshes. By using Z88 Aurora and the integrated mesher, the generation of a mesh for the simulation can be done.

#### **Input file:**



b25.stl → CAD-geometry-data saved as a STL-format

In the beginning a new project must be created by using  and **Create Folder** in this case for example *Example3*. To finish this topic, press *Enter* and close the dialog with *OK*.

The import of the geometry starts with  **Import/Export**. At this moment a new context menu will be opened on the right side. By using this menu it is possible to open the *STL-File*  **STL-File** and import the data into Z88 Aurora (*Figure 1*).



*Figure 1: Imported geometry which is stored in STL-format*

With the  button the context menu switches to the preprocessor. To create a FE-mesh from the imported geometry, use the free-mesher  **tetrahedron** for tetrahedron meshes. Now on the bottom of the main window a new dialogue opens. The dialogue contains the main parameters for free-meshing. On the left side of this dialogue it is possible to choose between two different free-meshers. The first is named *Tetgen* and the second called *Netgen* (see also Z88 Aurora User Manual). During this example the *Netgen* is used with following parameters (*Figure 2*). The element type is 17 (linear tetrahedron) and 3 is the size of the elements, that are created.



If it is not possible to create a FE mesh from the geometry please check the path to the chosen free-mesher. The corresponding allocations are finding: "*Help* → *Options* → *slider: Paths* → *tetgen*" respectively "*netgen*". The standard path is in the Z88 Aurora installation directory and following subdirectories:  
 „*addons/tetgen/win32*“ respectively „*/win64*“ respectively „*/mac*“  
 „*addons/netgen/win32*“ respectively „*/win64*“ respectively „*/mac*“.

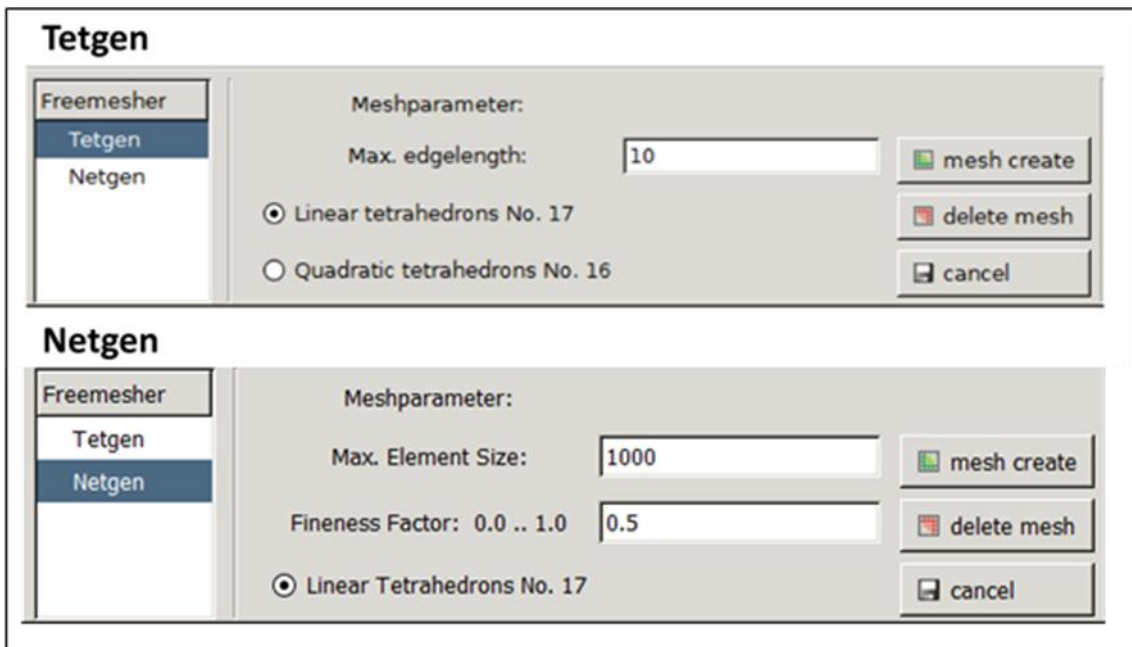



Figure 2: Free-mesher dialogue

To allocate the material, use the  **Define** button. A new window will be opened. On the left side the materials of the material database are showed. Select the material "*cast*"

steel" and allocate the material to the FE mesh. The next step is the creation of a *load case*. To create a new case you have to name it and finish with *OK* (Figure 3).

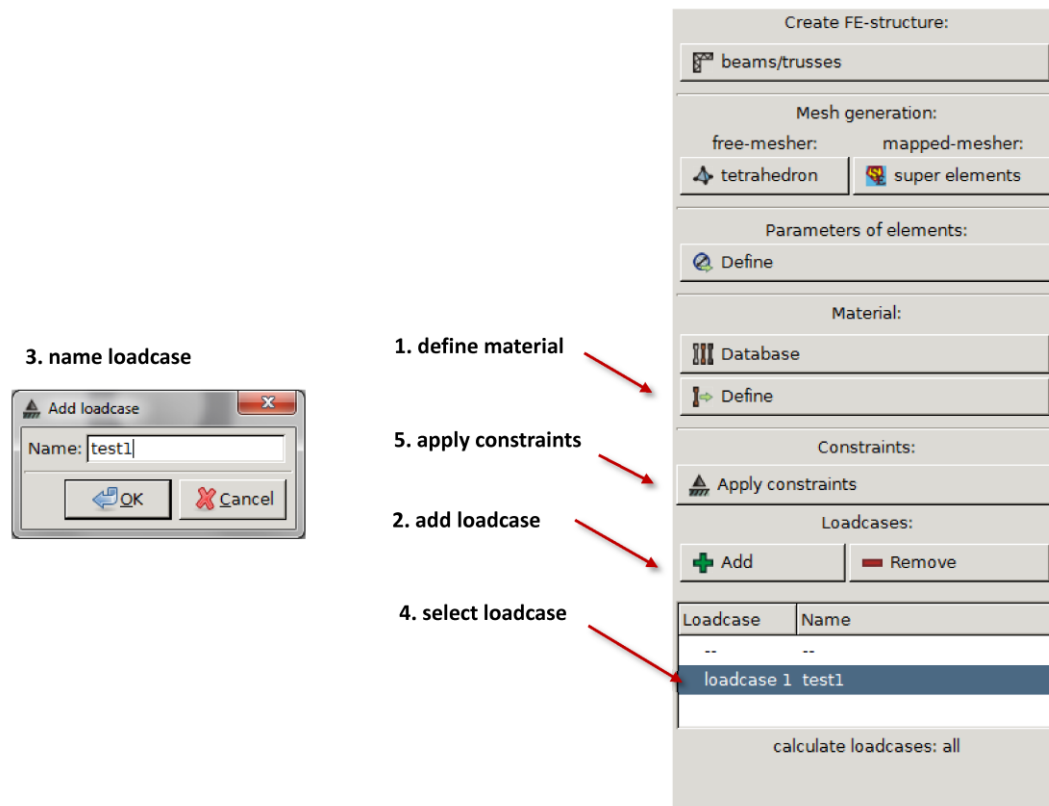


Figure 3: Z88 Aurora preprocessor

To simulate the correct constraints the big connecting rod eye must be fixed and at the surface of the small connecting rod eye a pressure of  $10 \text{ N/mm}^2$  must be assigned (Figure 4). This part is designed in the SI-Unit *mm*, thus it can be simulated with *N/mm<sup>2</sup>* as constraint-unit. It is possible to simulate with every unit, but beware of different units. In one simulation every parameter (geometry, constraints, material,...) must have the same unit.

To create a fixation you can fix all nodes on the surface of the big connecting rod eye. It is possible to select all nodes in this area by selecting one node and then use the area selection mode with an angle of  $10^\circ$  (see Z88 Aurora Theory Manual). After that, choose *Displacement* with the value of *0*. To assign the fixation press *Apply*. After this the view switches to *surface mesh*.

Press *Switch* to switch back to the *picking-view*. To create the pressure load select the surface of the other connecting rod eye and choose *Pressure* with a value of  $10 \text{ N/mm}^2$ . After pressing *Apply*, save the constraints with *Save*. The dialogue should be closed now.

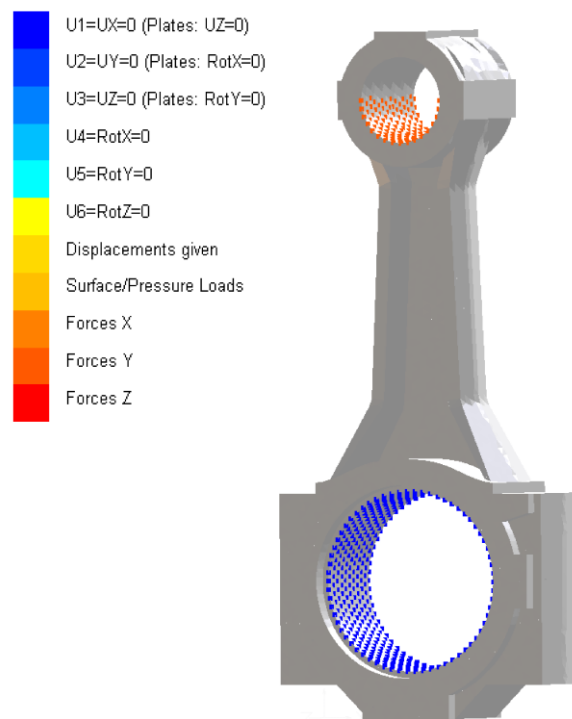



Figure 4: Connecting rod with constraints

With the  button you can switch to solver context menu. In this menu it is possible to choose the numerical solver. For example you can use the iterative SORCG-Solver (*sparse, iterative*). Furthermore the parameters for the stress calculation (Gauss points, in this example 5) and extended options can be found. This example should be simulated by using the standard parameters. For more interests see also Z88 Aurora Theory Manual. The flowchart of this menu is shown in Figure 5

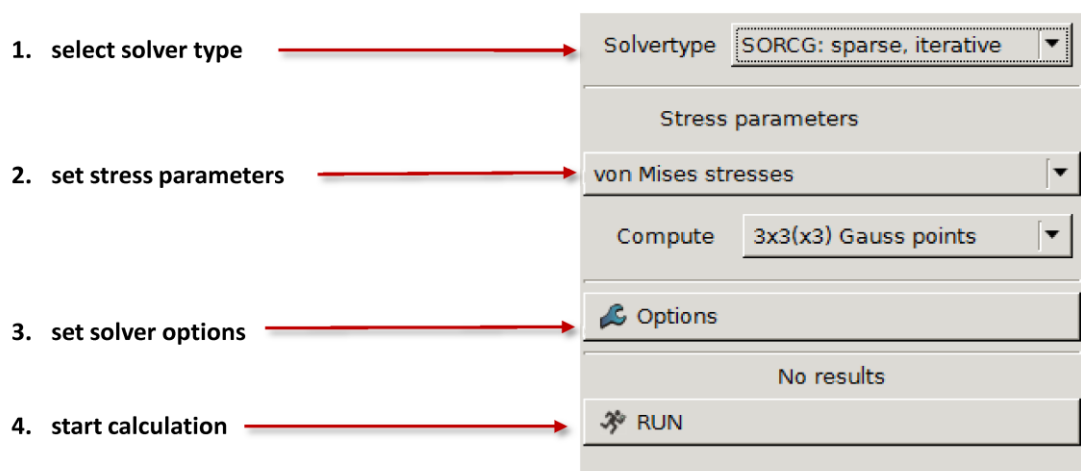


Figure 5: Settings for solver parameters

After pressing the *RUN* button a new window is opening, which starts the calculation if the *OK* button has been clicked (*Figure 6*).

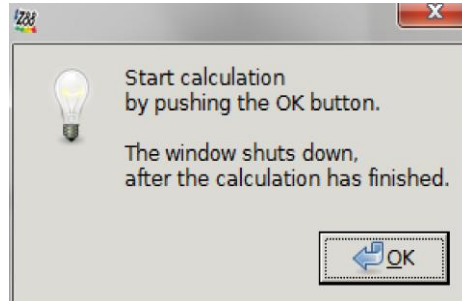



Figure 6: Start calculation

If the calculation was successfully, *Figure 6* disappears and you can select the postprocessor with the  button.

On the right side the context menu switches to the post processing mode. If you want to see the results, you have to select the load case. Afterwards you can choose which kind of result you want to see (e.g. displacements, stresses) and if the part should be deflected or not (*Figure 7*).

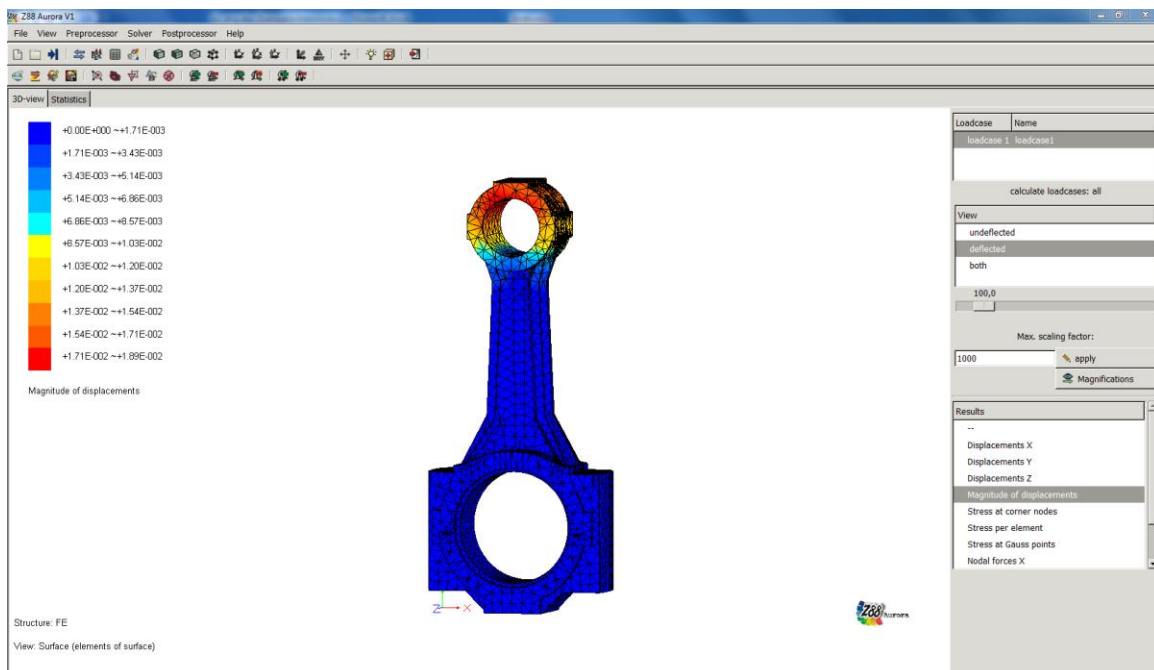


Figure 7: Z88 Aurora post processor (Magnitude of displacements; deflected)