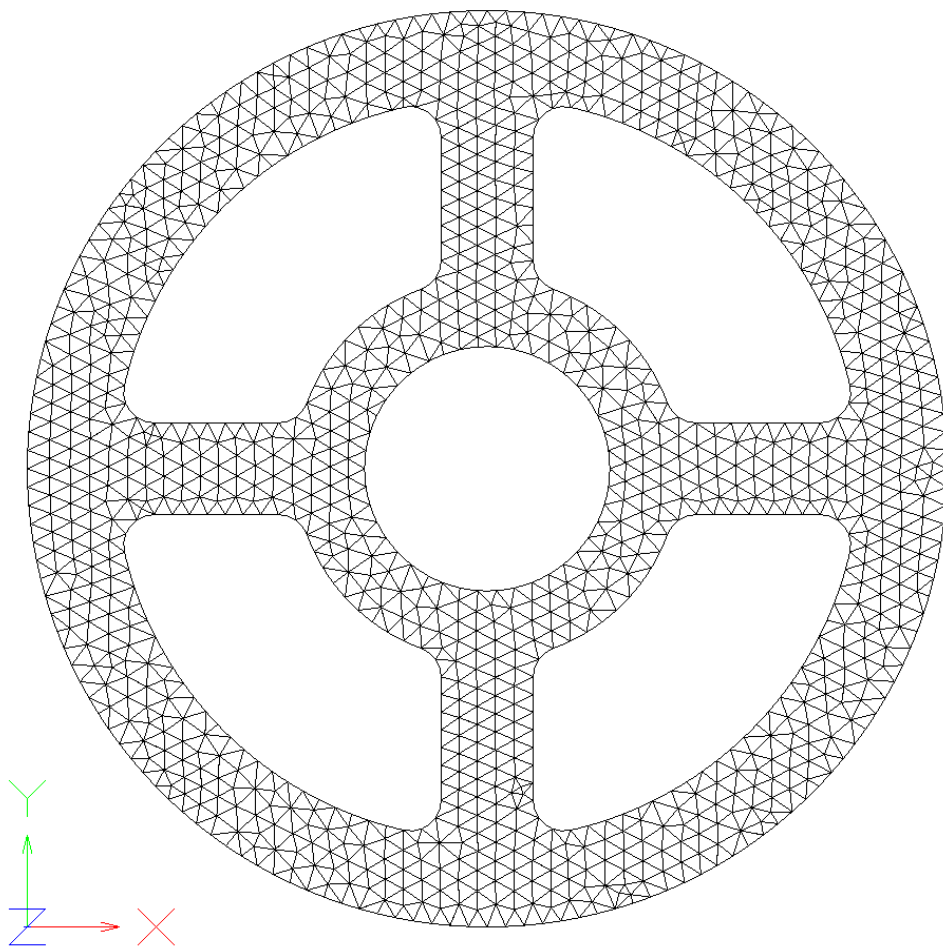


Z88 AURORA® EXAMPLE MANUAL

Example 10: Pinion

(Plane stress element No. 14 with 6 nodes)






10. Example: Pinion (Plane stress element No. 14 with 6 nodes)

In this example we examine a pinion, which hub is pressed on a shaft. The amount of the gap pressure of the interference fit assembly is 100 N/mm^2 . Because of the expansion of the hub into the tooth system, the deflection is to be analyzed. The tooth system on the outside is omitted.

input file:

b14_1.cos → CAD-data

At first it is necessary to create a new project with  and **Create Folder**. In this case, e.g. *Example10*, you have to confirm the dialogue with *Enter* and close the window with *OK*. Now, you have to import the COSMOS file, which is named above.

With  **Import/Export** the example file *b14_1.cos* can be imported. On the right hand a context menu is opened (*Figure 1*), with which it is possible to load the Cosmos file  **Cosmos-File**. The element type, which is to choose, is *plane stress elements* (*Figure 1*).

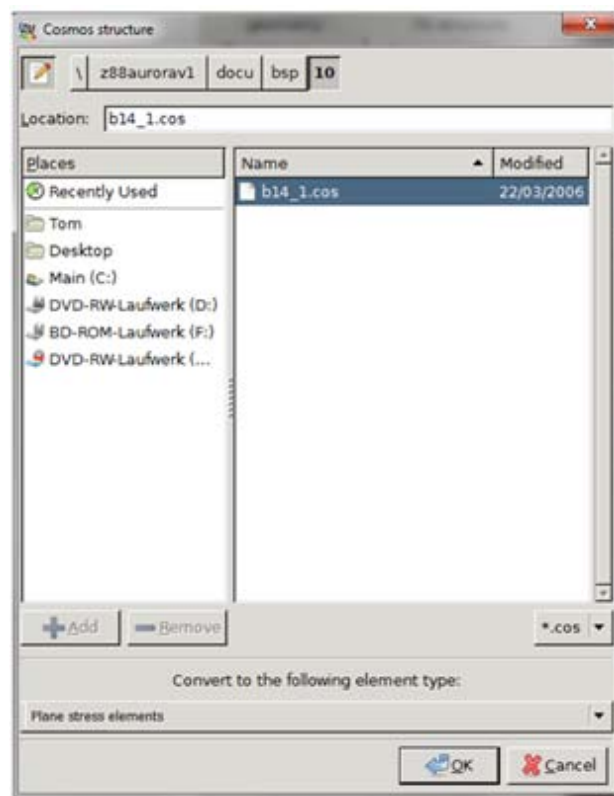



Figure 1: Import of a COSMOS file with disc elements

With the  button you switch to the preprocessor. On the right side of the window you can see the load cases and that already a load case containing boundary conditions exists. If you click there, the loads and the fixings are shown (Figure 2). To blank the display with the boundary conditions deselect the load case by clicking "--".

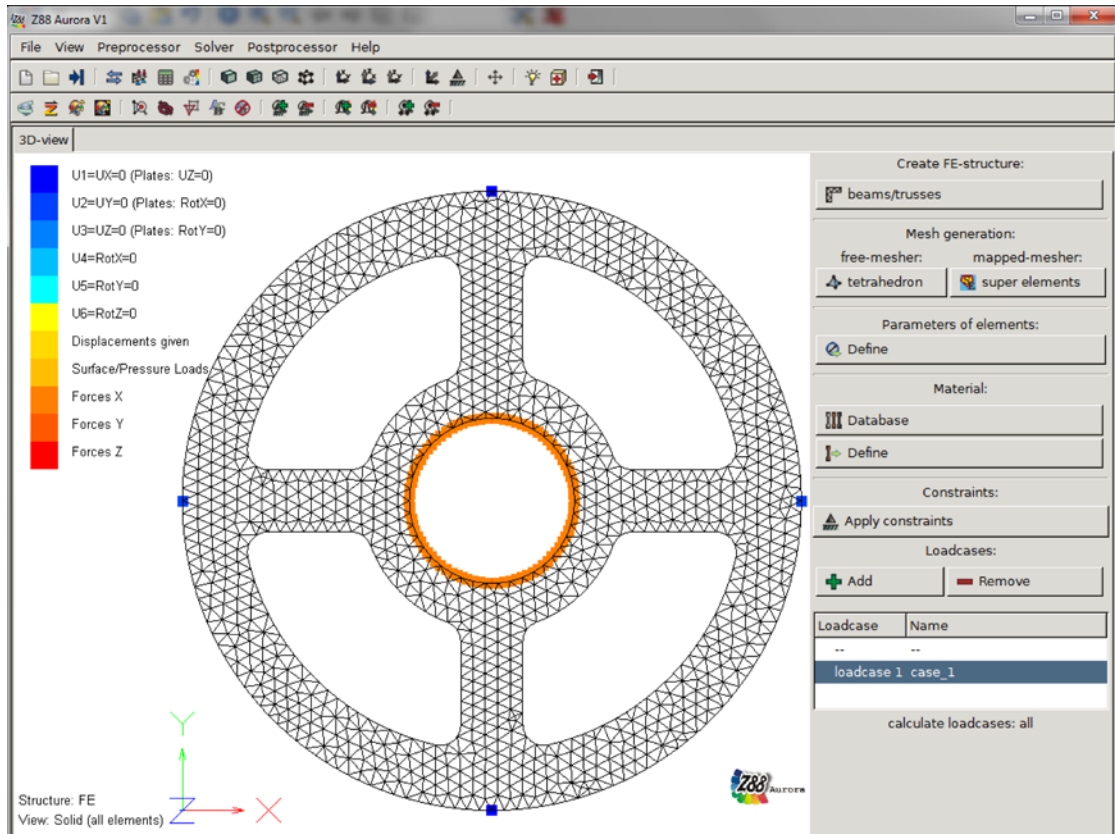


Figure 2: Imported structure with constraints

The legend on the left hand indicates what kind of boundary condition is displayed: displacements, pressures, surface loads, loads, etc. With View → Size constraints you can minimize or maximize the single nodes.

The structure itself was designed in Pro/ENGINEER WF4. In the module Pro/MECHANICA the material (steel: $E = 206,000 \text{ N/mm}^2$ and $\nu = 0.3$), constraints, mesh seeds (global max. 6, global min. 3) and the pressure $p = 100 \text{ N/mm}^2$ have been allocated. Afterwards the disc values had been defined and the mesh was created as plane stress elements out of triangles. The next step was to emit as a NASTRAN file (formerly also COSMOS files were possible, which are used here) parabolic with coordinate system CS0 as b14_1.cos.

The choice of the constraints is important: The allocation of the gap pressure is no problem, but the right choice of the bearing to hold on the one hand the structure in the area and not to constrain on the other hand the expected deformation. Here, the concept of the “virtual fixed-point” had been applied (*Figure 3*):

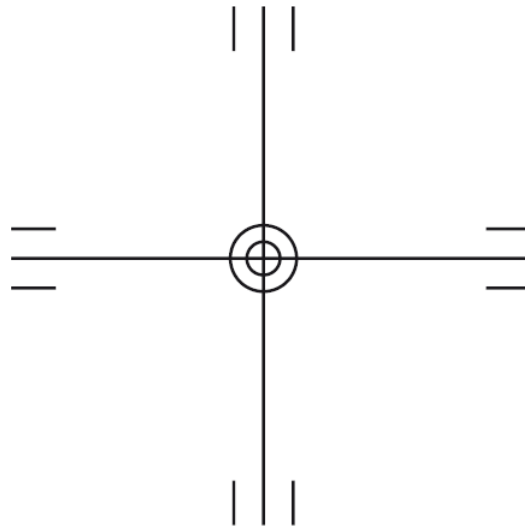

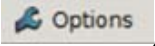




Figure 3: Description of the bearing with the “virtual fixed-point”

Thus, four points of the structure were defined. The two points at 3 and 9 o’clock are constrained into Y direction; in X direction they can be moved. The two points at 6 and 12 o’clock are fixed into X direction; in Y direction they can be moved. Thus a “virtual fixed point” occurs in the middle.

To compute the structure, you have to change to the solver menu using the  button. It is supposed that an inappropriate numbering of nodes exist because of the circular closed structure. The last nodes contact the first nodes and therefore a bad conditioned total stiffness matrix is produced (compare Z88 Aurora Theory Manual). Because of that, it is recommended at the solver menu to select “*node sort*” (Cuthill-McKee-algorithm) in , before you use the Cholesky solver. By clicking the  button, the calculation starts.

If the calculation was completed successfully, you can select the postprocessor with the  button.

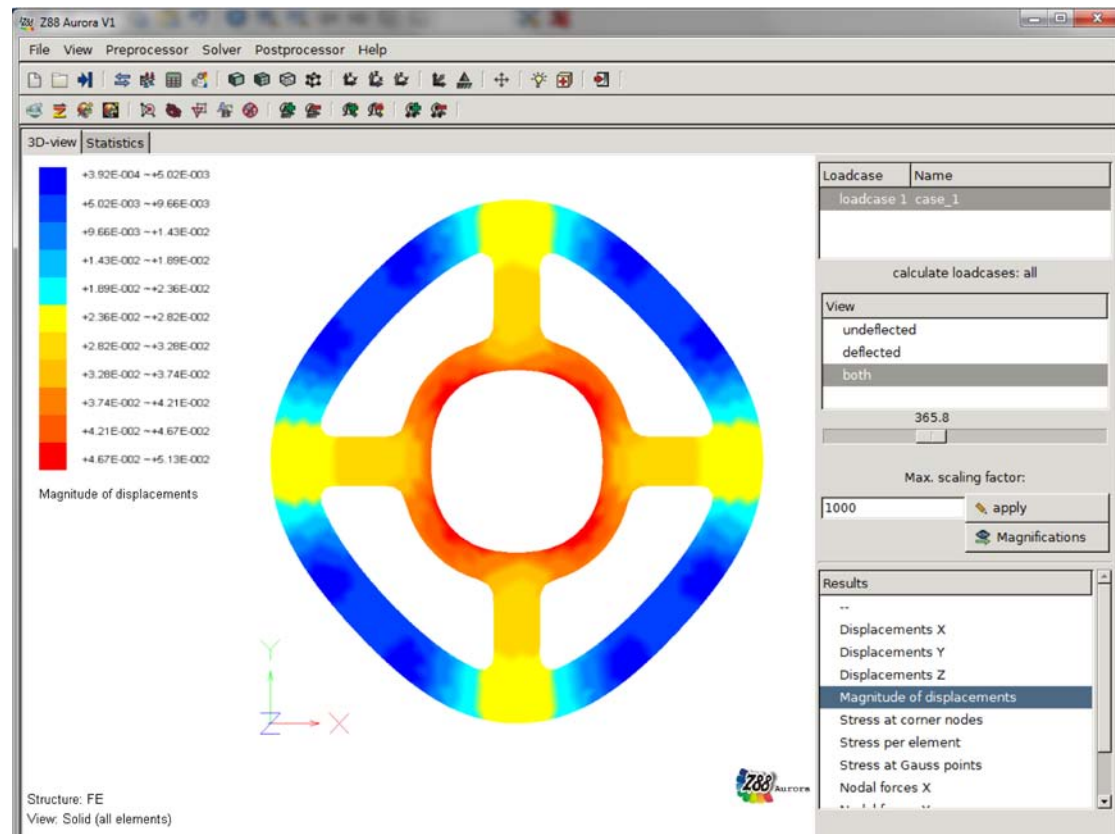


Figure 4: Magnitude of displacements in the postprocessor

As expected the highest von Mises stresses exist in the interior edge, i.e. at the hub borehole. The three Gauss points at every finite element are distinguishable (Figure 5).

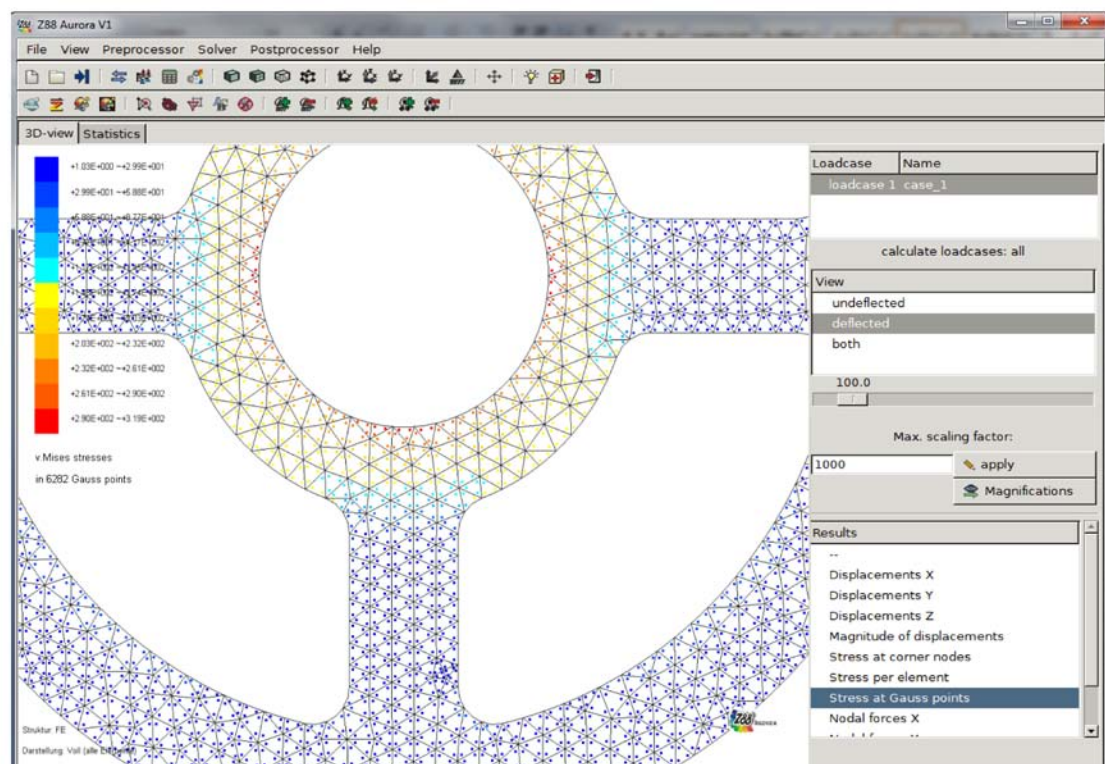


Figure 5: Stress at Gauss points