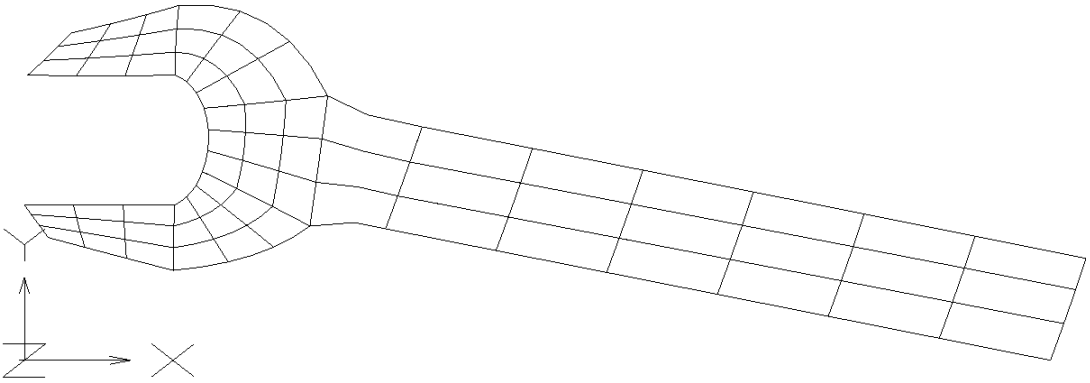
Z88 AURORA® EXAMPLE MANUAL

Example 2: Fork Wrench

(Plane stress element No. 7 with 8 nodes)



Z88 Aurora® is a registered trademark (No. 30 2009 064 238) of Univ. Prof. Dr.-Ing. Frank Rieg

2. Example: DXF import and super structure (plane stress element No. 7 with 8 nodes)

The chosen example is a DXF file – a fork wrench built up by plane stress elements – created in AutoCAD. With the help of this part the export and import procedure from CAD system to Z88 Aurora is explained. In addition the refinement of super structures and the accomplishment and interpretation of a strength determination is expounded.

Input files:

b1_x.dxf → Input file for CAD converter Z88X

b1_2.txt → Boundary conditions from Z88 V13

b1_3.txt → Header parameters for stress processor from Z88 V13

This example will be calculated twice to depict the functional range of Z88 Aurora (*Figure 1*):

1) Import of ready-to-run structure file and constraint file
2) Import of the super structure and manual meshing and application of constraint

The ready-to-run examples are in the sub file folder "*2v1*" (model 1) und "*2v2*" (model 2).

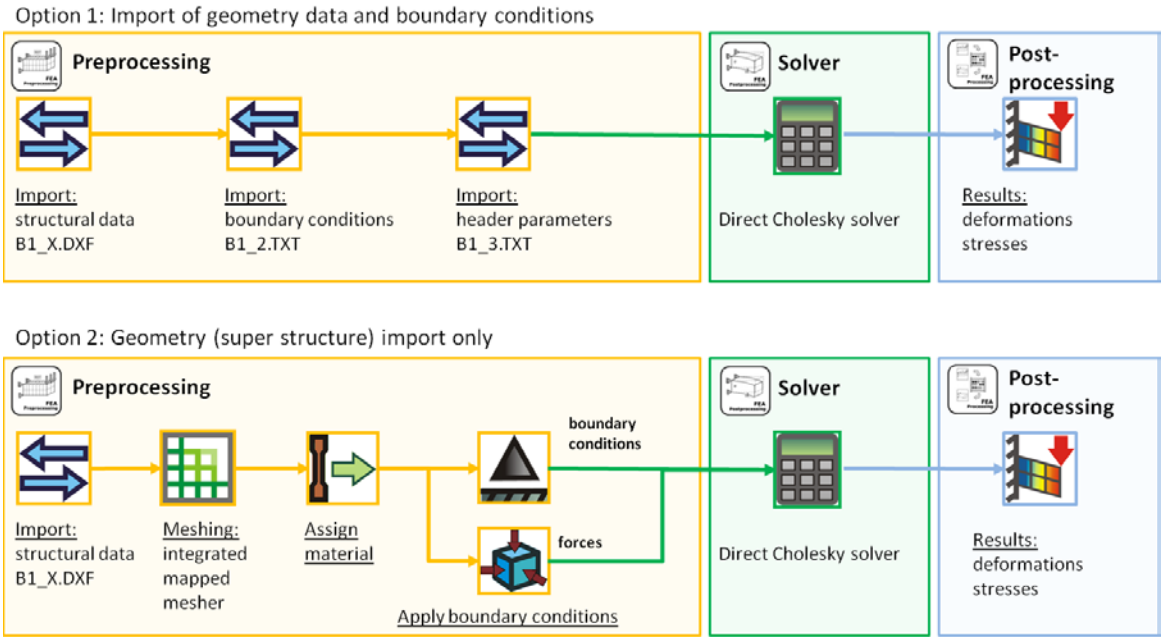


Figure 1: DXF import in Z88 Aurora

To create a new project, use , Create Folder, e.g. *Example2*, confirm with *Enter* and exit the dialogue with *OK* (*Figure 2*). The first step is equal for both analyses.

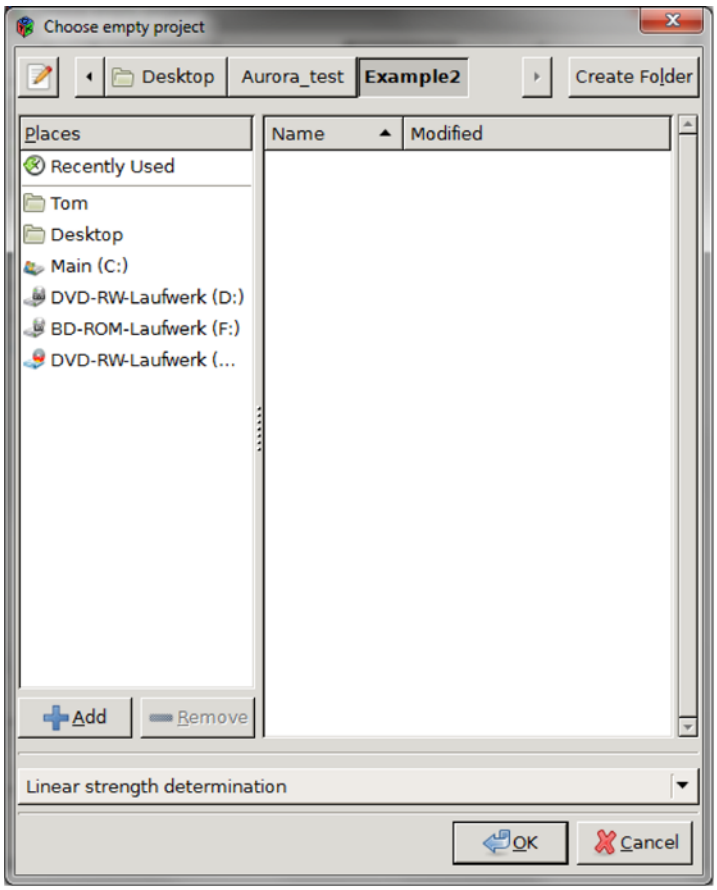


Figure 2: Create new folder

Option 1: Ready-to-run structure file and constraint file

The necessary files are *b1_x.dxf* (structure file), *b1_2.dxf* (constraint file) and *b1_3.dxf* (header parameters for stress processor).

To import the example file *b1_x.dxf,* use the Import/Export function of Z88 Aurora. A new menu will appear at the right side. Use DXF-File to import the file (Figure 3). Choose the option "*conversion of DXF super structure to Z88i1.txt*" in the lower combo box of the file chooser dialogue to import and net the super structure file. The FE mesh will be created automatically by the mapped mesher Z88N. The element type is the plane stress element No. 7 (see Theory Manual) (*Figure 3*). Use the button Z88-File of the import menu to import the boundary conditions *B1_2.TXT* and the header parameters for stress processor *b1_3.dxf*.

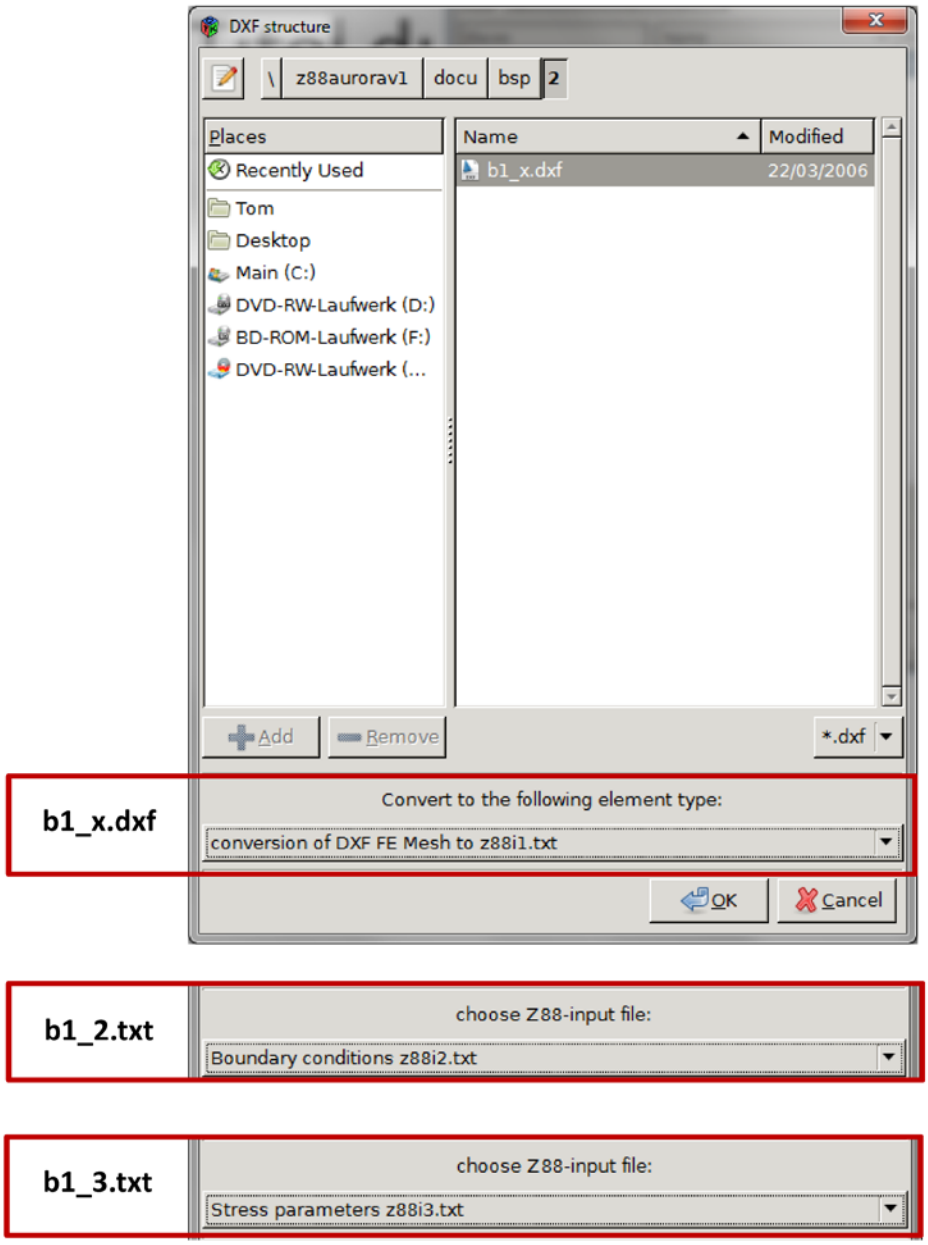


Figure 3: Import the DXF data (b1_x.dxf), boundary conditions (b1_2.txt) and header parameters for stress processor (b1_3.txt)

Switch to the preprocessor using the button. By pressing the button you can change the view to see the FE mesh. On the right hand side you can see that a load case already exists. If you click on the load case, the boundary conditions will be displayed. The handle bar is fixed, the U-shaped opening is loaded (*Figure 4*).

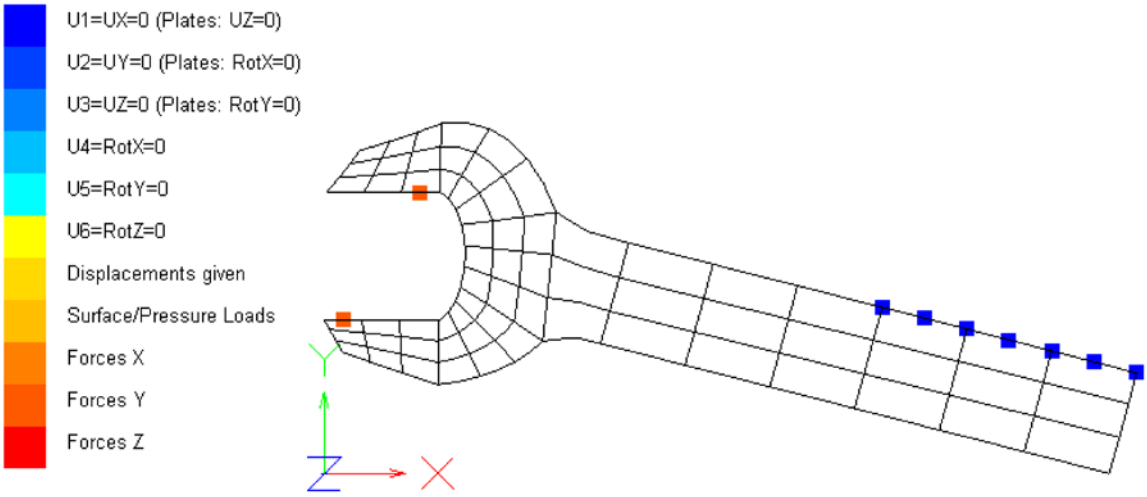


Figure 4: FE mesh with boundary conditions and load

To calculate the example, you have to switch to the solver menu using the button. Four different solvers are provided. For this example you can use the Cholesky solver. Start the solver by pressing the button RUN (*Figure 5*).

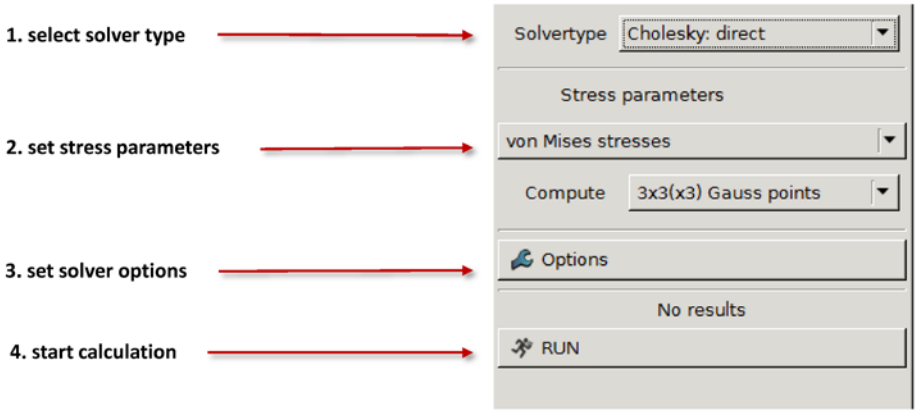


Figure 5: Settings Cholesky solver

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

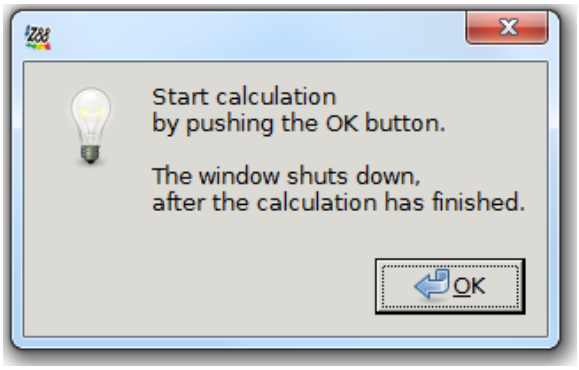


Figure 6: Start calculation

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

Therefore choose *loadcase_1* at the right hand side of the upper tree. You have the possibility to view the part *undeflected*, *deflected* or you can view a combination of *both* states (*Figure 7*).

The display of the results shows the Gauss points only not deflected

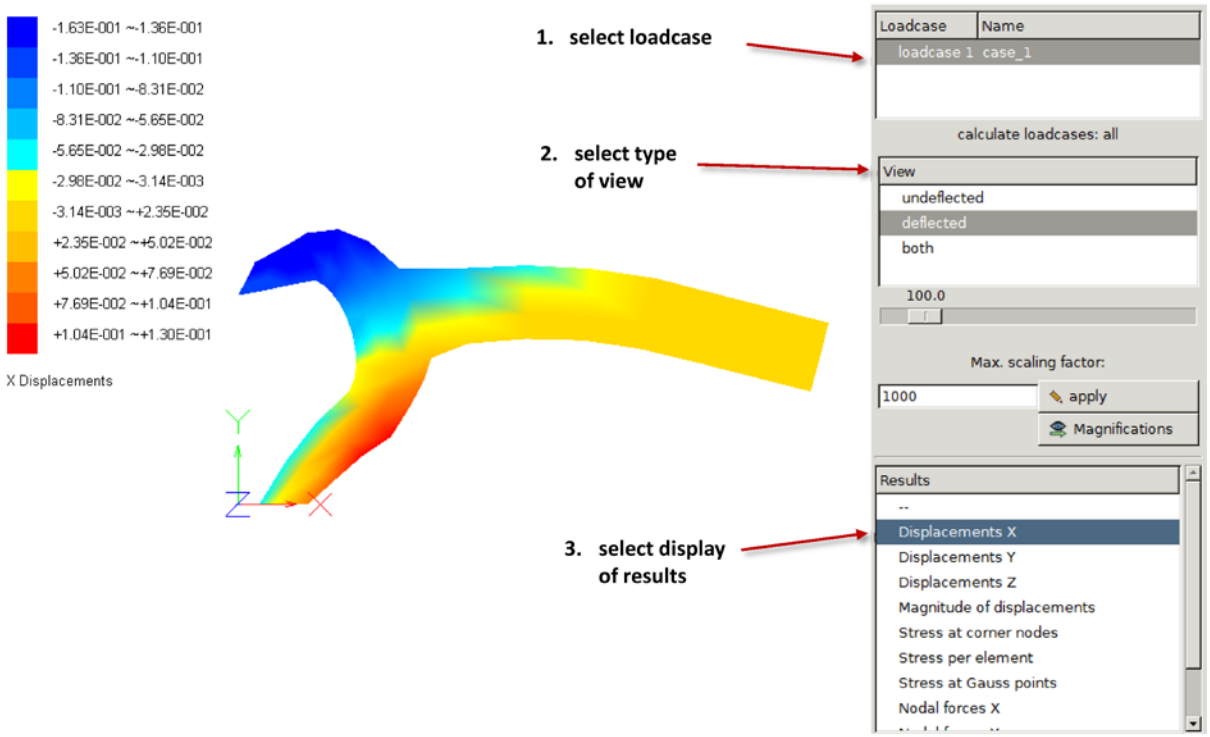


Figure 7: Postprocessor

Option 2: Geometry import (super structure) and manual meshing and assigning boundary conditions

Option 2 wants the user to net the super structure and to assign the boundary conditions manually. Option 2 is a tutorial how to refine FE meshes and assign boundary conditions for different cases.

With Import/Export the structural data *b1_x.dxf* can be imported. A context menu will appear at the right side of the screen and using DXF-File, the DXF file can be added. Choose the import setting "*conversion of DXF super structure mesh to Z88NI.TXT*" (*Figure 8*).

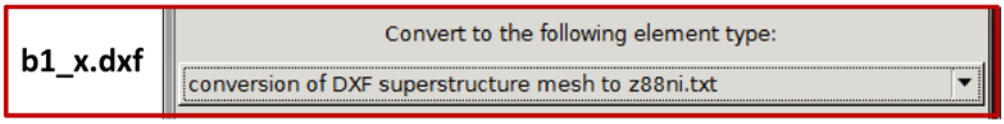


Figure 8: Import settings super structure mesh files

No other data will be imported. Change the view with the button *mesh view*. The displayed mesh is a super structure with seven elements, which were designed in a CAD system. The element labels can be displayed with "*View → Labels → Elements*" (*Figure 9*).

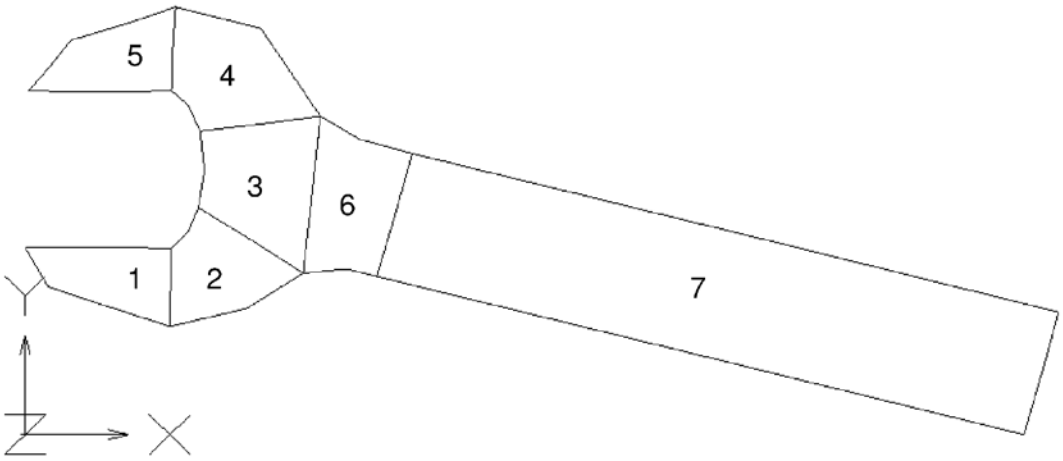


Figure 9: Superelements without mesh refinement

Choosing the button will open the preprocessor. Now the user is able to net the part manually with the mapped mesher and to assign material parameters and boundary conditions (*Figure 10*).

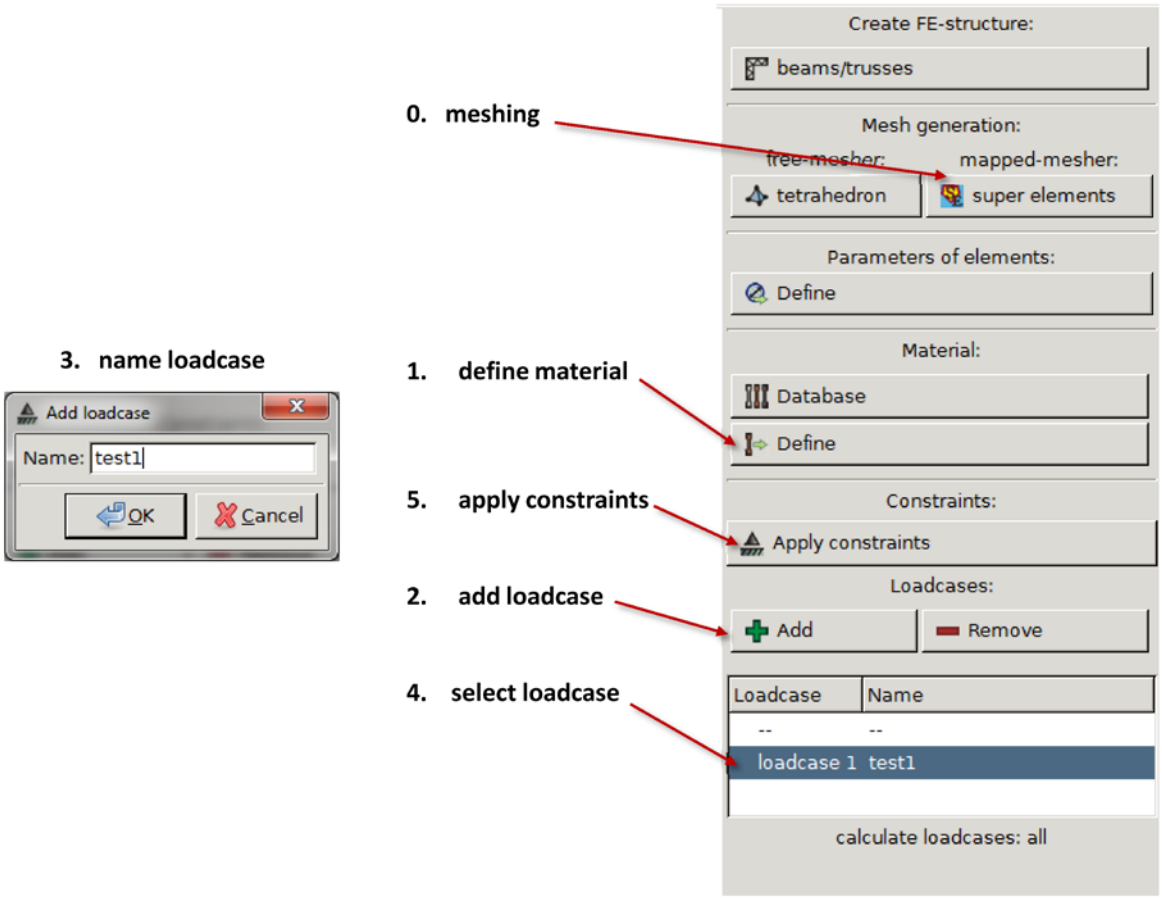


Figure 10: Z88 preprocessor

By pressing the button super elements the mapped mesher menu will be displayed. First choose the super element(s) to be refined, then the element type and at last the fragmentation rule (*Figure 11*).

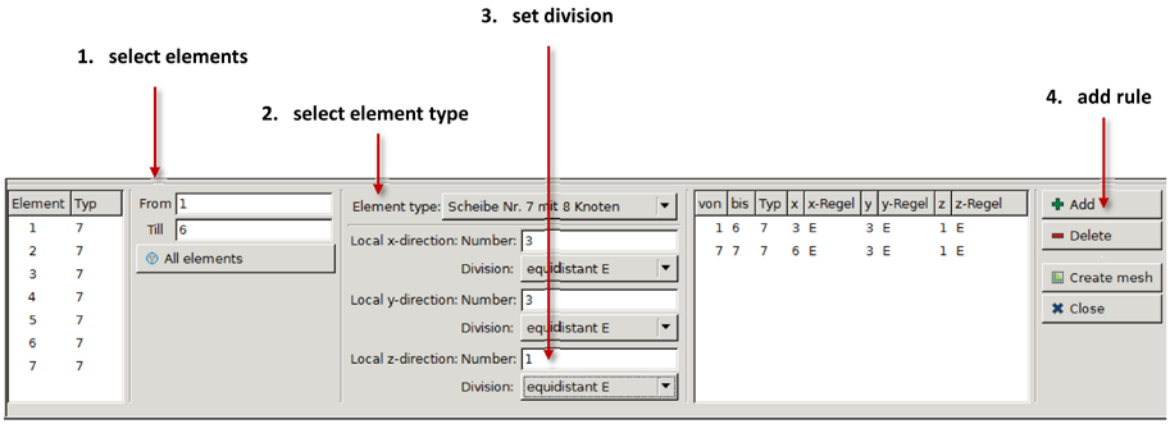


Figure 11: Mapped mesher

The fragmentation of the super elements is corresponding with the local coordinates of the elements (see Z88 Theory Manual). For this example subdivide elements No. 1 to 5 three times, equidistant in local X- and Y- direction. Element No. 6 is subdivided one time

in local X- and three times in local Y-direction and element No. 7 six times in local X-direction and three times in local Y-direction. The fragmentation rule for local Z-direction has no influence for plane elements, but has to be greater than *0*. Add all rules with the button ✚ to the table on the right side and generate the FE mesh by pressing the button Create mesh. *Close* the mapped mesher dialog box with ✖ and change the view to see the refined FE mesh (see *Figure 4*).

To assign a material you can use Define to open the material definition tab. For this example the material definition is done.

Now the constraints have to be applied. You have to add a load case using Add and insert a name for that case, like "*case1*".

Please choose the setting "*Picking the middle nodes*" in the option chart "*Help → Options → View*" to pick all to pick all required nodes.

Open the constraint menu with the button Apply constraints. Pick the nodes and define the necessary loads and fixations (see also Z88 Aurora User Manual). It will be helpful to label the nodes "*View → Labels → Nodes*". Choose the nodes shown in *Figure 12* and assign the following boundary conditions (Please be ahead of differences in the numbering of the nodes!):

- Node 11: Force in negative Y-direction: 7143 N
- Node 143: Force in positive Y-direction: 7143 N
- Nodes 216, 220, 227, 231, 238, 242 und 249: Fixation

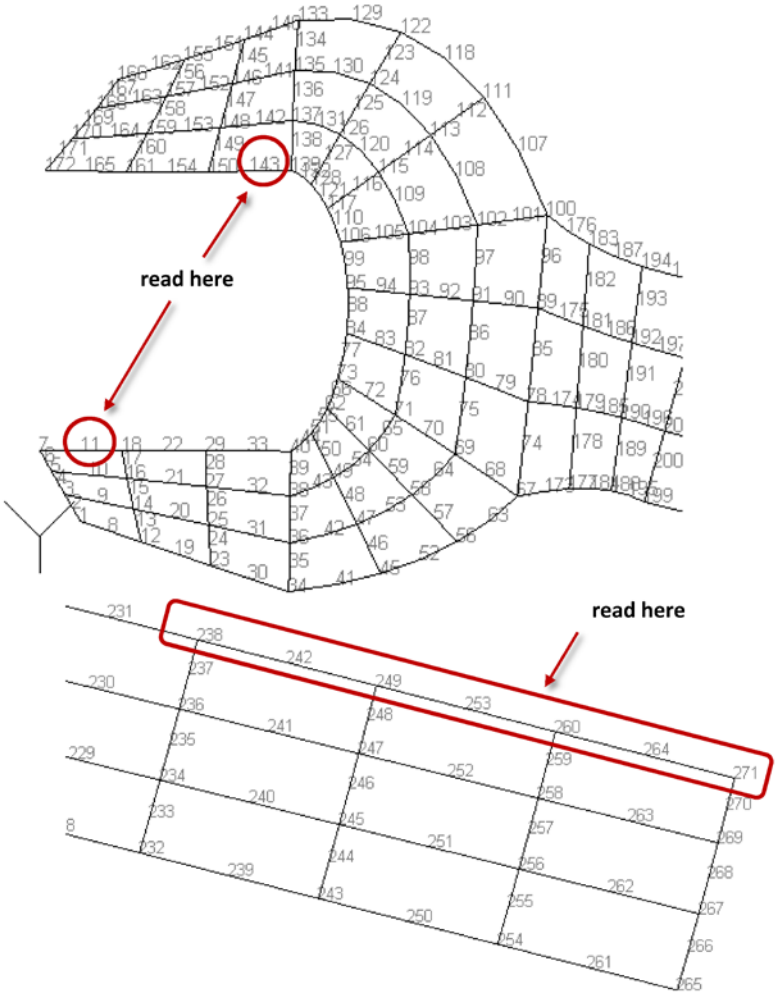


Figure 12: Numbering of nodes for the assignment of the boundary conditions

Finishing the assignment of the boundary conditions please start the calculation. To have your model computed, you need to use the button to switch to the solver menu. The Cholesky solver, von Mises stresses and 3 Gauss points are an appropriate choice. The RUN button will open a notification popup (*Figure 6*). Pressing *OK* will launch the solver.

You can start the post-processor by clicking the button. Select the load case in the post processor menu on the right hand side of the window. Now you can have the model displayed *undeflected*, *deflected*, or with *both* states combined (*Figure 7*).