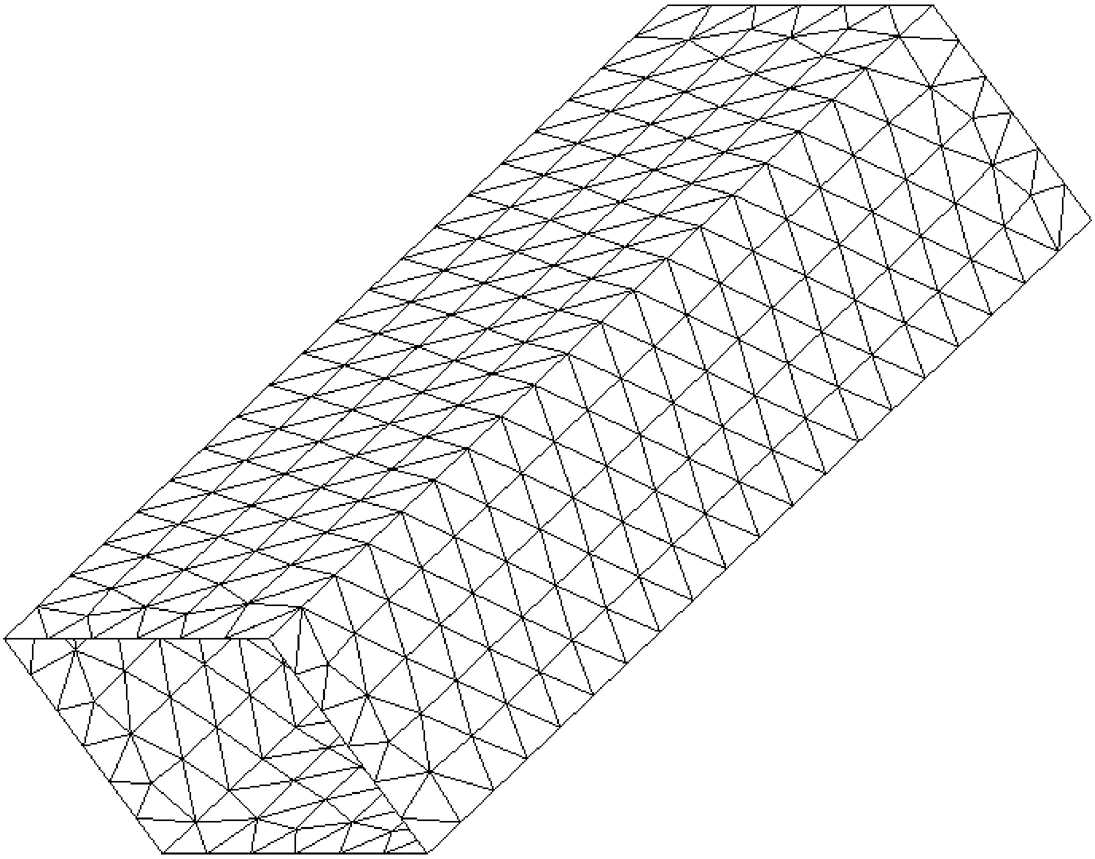
Z88 AURORA® EXAMPLE MANUAL

Example 5: Square end liner

(Shell No. 24 with 6 nodes)



5. Example: Shell-model (Shell No. 24 with 6 nodes)

To represent thin structures like bent sheets, shell-models can be used.

The component handled in this example is a square tube, which was created from shells with the help of an extern FE-program. The information about the geometry and all boundary conditions are saved in a NASTRAN-file. This example demonstrates how shell models can be imported and calculated in Z88 Aurora.

Input file:

b26_g.nas → structure and boundary conditions information from FE-program

At first a new project is created by using and Create Folder - in this case for example *Example5.* The dialogue is confirmed with *Enter* and completed with *OK* (*Figure 1*).

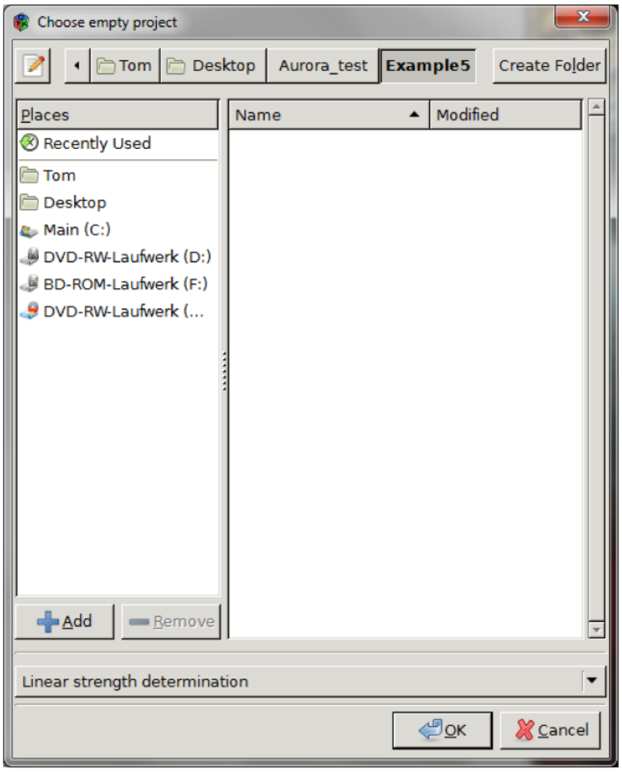


Figure 1: *Create the new project called Example5*

With the button Import/Export the appending file *b26_g.nas* can be imported. A context menu on the right side appears it can be used to load the NASTRAN-file Nastran-File (*Figure 2*).

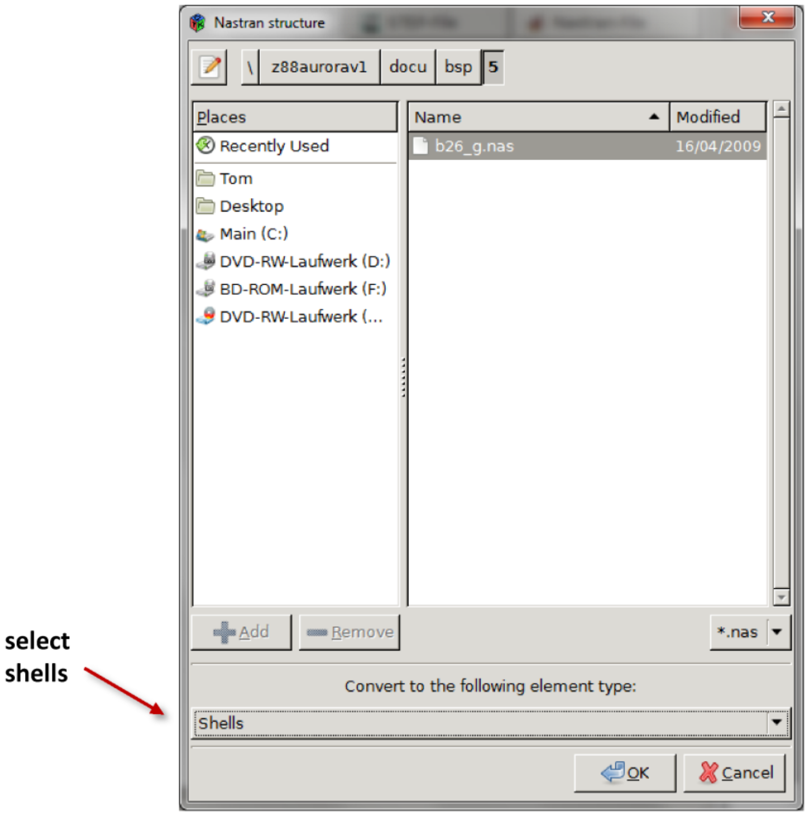


Figure 2: Import NASTRAN-files including shell elements

To get a better view to the structure it is recommended to switch the view-option to the surface mesh with . With the right mouse button you can rotate the model; it consists of triangular shell elements (element type 24, see also Z88 Aurora Theory Manual). With you can get into the preprocessor. In the load case-window on the right side already a load case exists (*Figure 3*). If you click on it, the included boundary conditions are displayed.

The legend on the left side explains the type of every single constraint: displacements, pressures, area loads, forces, etc. With View → *Size constraints* you can zoom the visualization of the single points in and out.

In the left lower screen you can see, that the structure is about a FE-mesh, that is ready to calculate. Furthermore all elements are displayed.

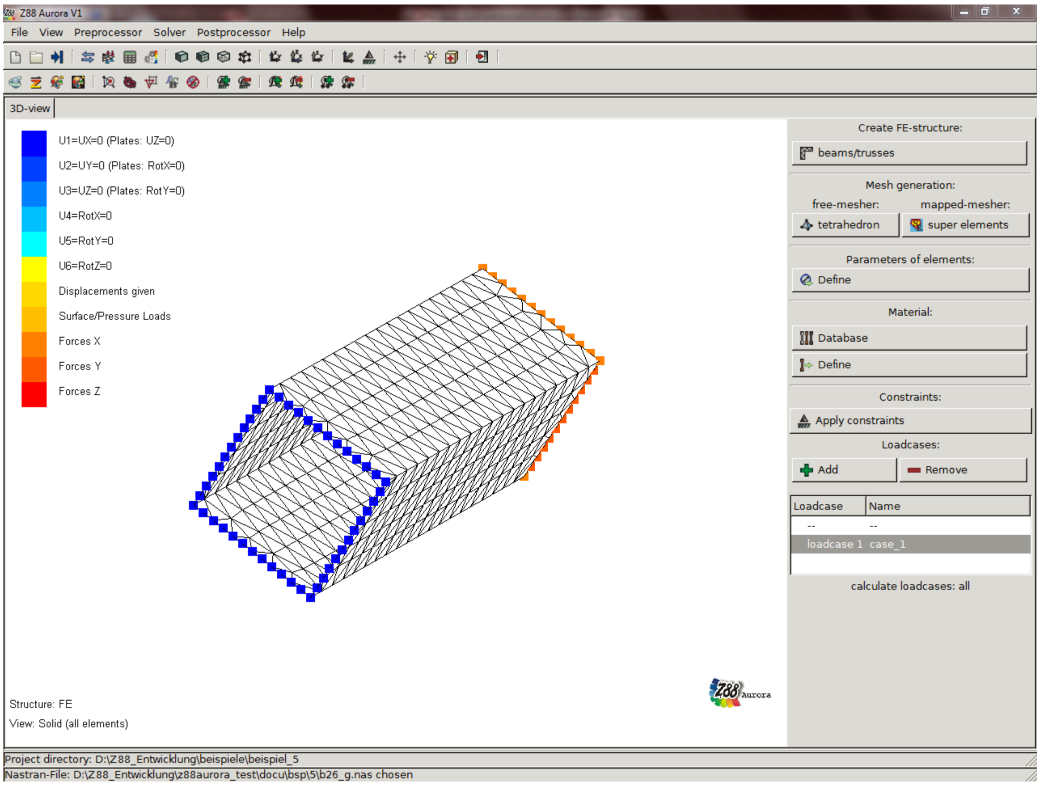


Figure 1: Meshed component with displayed constraints

With the button you switch to the solver menu. Here you can choose between several solvers (see also Theory Manual). Here we want to select the SICCG (sparse, iterative) (*Figure 4*).

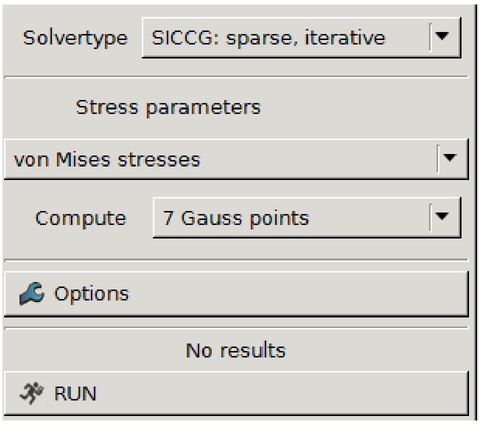


Figure 2: Solver menu

After pressing the *RUN* button a new window is opening, which starts the calculation if the *OK* button has been clicked (*Figure 5*).

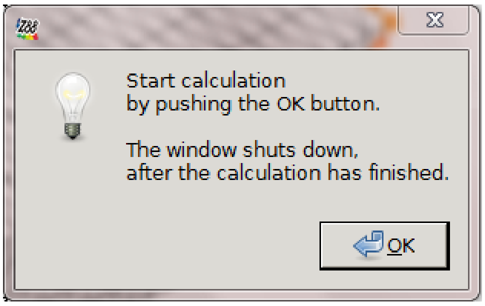


Figure 3: Start calculation

If the calculation was successfully, *Figure 5* disappears and you can select the postprocessor with the button.

On the right side a context menu is opened (*Figure 6*). Here you have to select the first load case before you can take a look at the *deflected*, *undeflected* structure. It is also possible to display *both* at the same time.

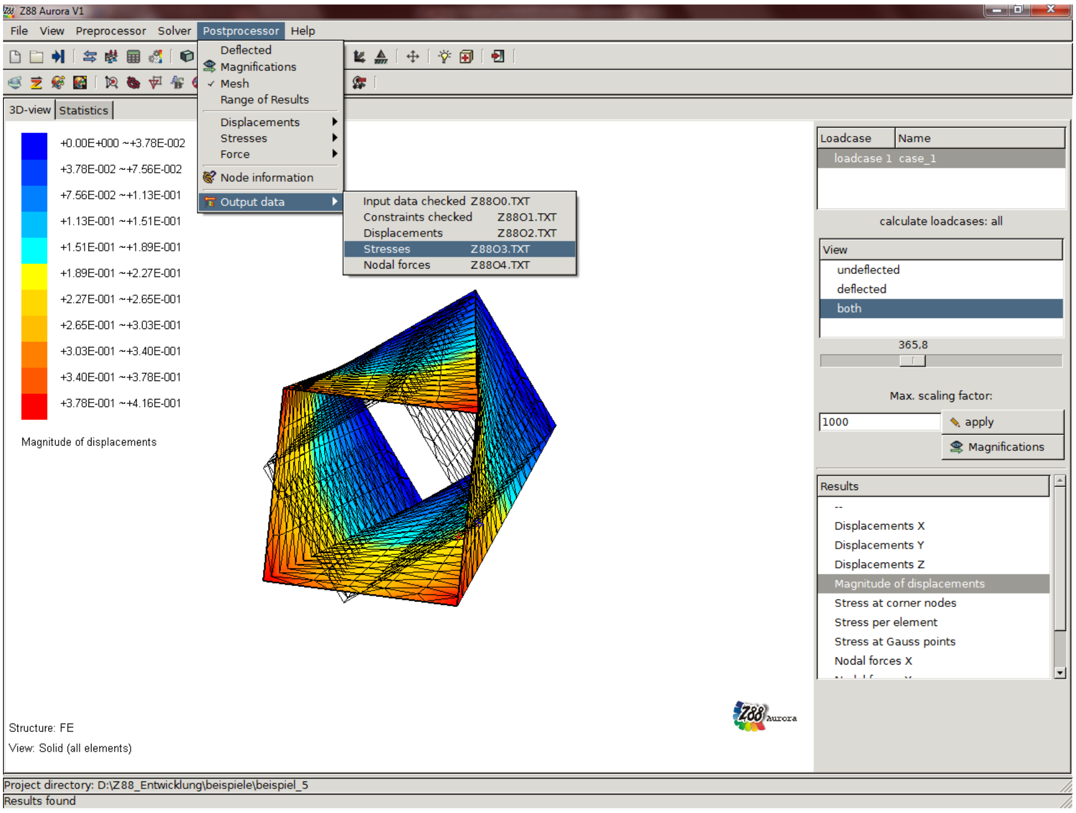


Figure 6: Z88 Aurora postprocessor

There you can find the result menu: The colours can represent: displacements (per coordinate or magnitude) and stresses (at corner nodes, average per element or average per element). For easy result traceability the calculated values are written into ordinary text files. (*Postprocessor → Output data → Z88i0.txt to Z88i5.txt*)

Additionally there is a tool to select single nodes even in the 3D-model itself to get informations about its nodal displacements. By following the menu "*Postprocessor → Node information*" another context menu is opened and the model view turns to the picking mode (*Figure 7*).

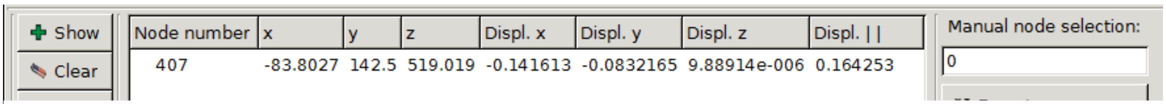


Figure 4: Node informations

With the *CTRL-key* and the *left mouse button* you can select single nodes. If you want to select some more nodes at once, it is also possible to hold *the shift-key* while pull a rectangular with the *left mouse button*. (For more information see the Z88 Aurora User Manual). When you click on Show, the list of node information will be refreshed, it can be exported as an .csv-file via Export->csv. This information consists of node number, the current position in the coordinate system and the displacements (in all three axis directions and magnitude). With Close the dialogue is closed.