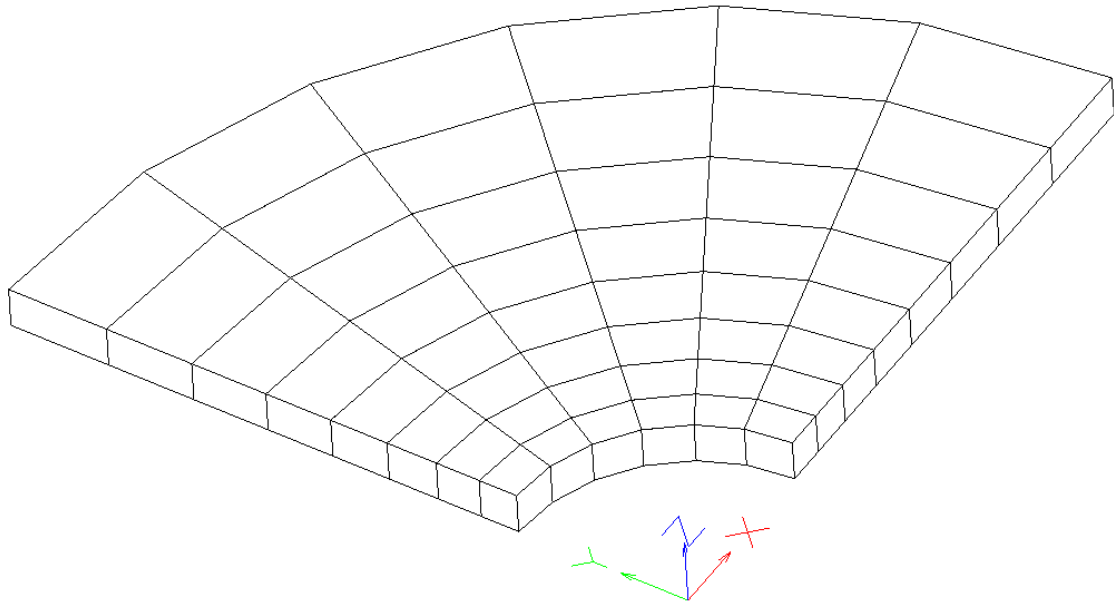


# Z88 AURORA® EXAMPLE MANUAL

## Example 7: Plate segment

(Hexahedron No. 1 and No. 10)



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

## **7. Example: Plate segment (Hexahedron number 1 and number. 10)**

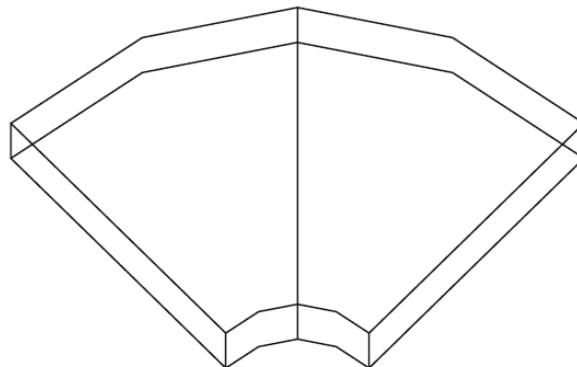
This example demonstrates the calculation of a 3-dimensional plate segment with curvy-linear hexahedrons.

### Input data:

b5\_x.dxf → structure data from a CAD-program




b5\_2.txt → boundary conditions

We consider a 90-degree plate segment. It is fixed at the outer edge and the load at the inner edge is 7,000 N. The best way to describe these structures is a cylindrical coordinate system with two super elements number 10 (*Figure 1*). These super elements should be divided in 48 hexahedrons number 1.




*Figure 1: Super elements, existing of two hexahedrons No. 10, each with 20 nodes*

With this example you can test the mapped mesher. If you do this, maybe you must set new boundary conditions in the CAD-program or in Z88. This will be explained later. The stress will be displayed in Gaussian points. These points are inside the element not on the surface. The stress on the surface of the element can be calculated by using extrapolation.

To create a new project, use , **Create Folder**, prompt e.g. *Example7*, confirm with *Enter* and exit the dialog with *OK*. To import the example file *b5\_x.dxf*, use the  **Import/Export** function of Z88 Aurora. There are many possibilities to import data from another program. Here you have to use  **DXF-File**. This file must be converted from a DXF super structure to a Z88 Aurora input file called *z88i1.txt*. Therefore you have to choose in the

## Example Manual

import dialog the option: “conversion of DXF FE mesh to z88i1.txt”. The boundary conditions will be loaded with  **Z88-File**, choose the file b5\_2.txt and the import option “Boundary conditions z88i2.txt” (Figure 2).

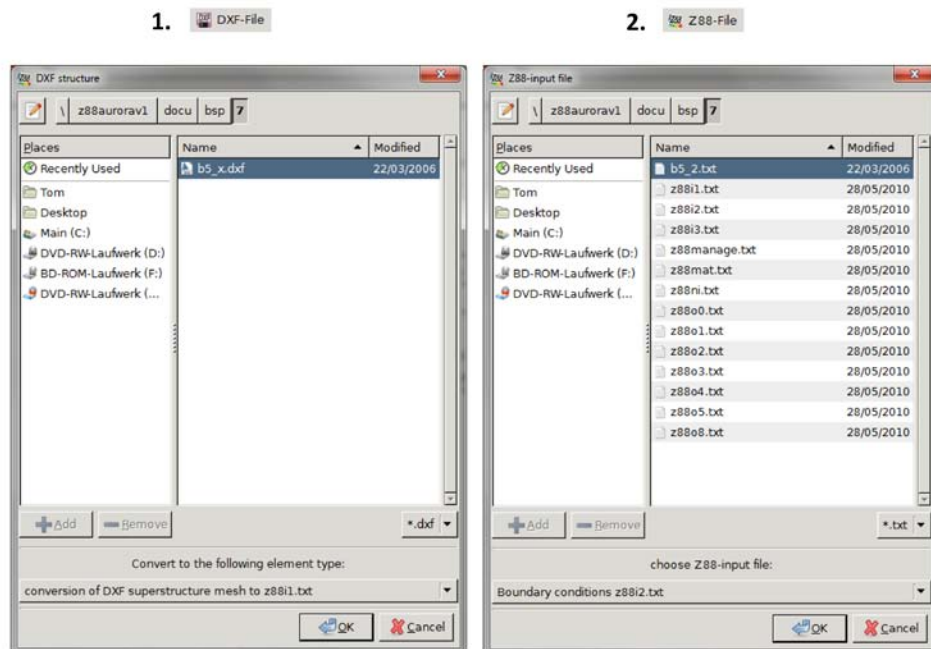






Figure 2: Import of DXF-structure and boundary conditions

Switch to the preprocessor using the  button. There you can see that a load case already exists. If you choose it, the boundary conditions are displayed and on the right side of the window the kind of condition is shown.

To calculate the example, you have to switch to the solver view using the  button. Here all solvers are listed that can be used to calculate the example. For this tiny little structure you can take the Cholesky solver. This example is calculated with  $2 \times 2 (x2)$  Gauss points and Mises-stress. To start the solver use  **RUN**.

When the solver had finished you can look at the results in the postprocessor using  (Figure 3).

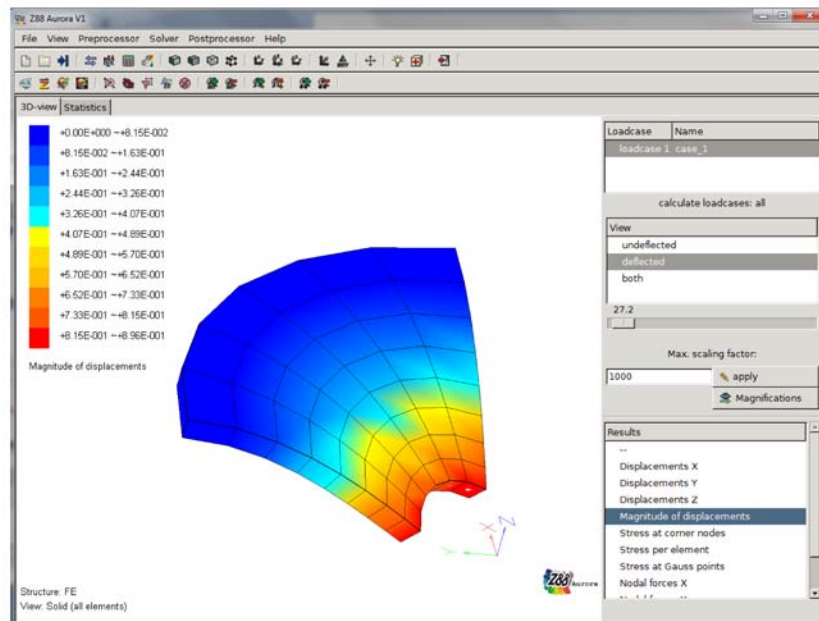


Figure 3: Displacements