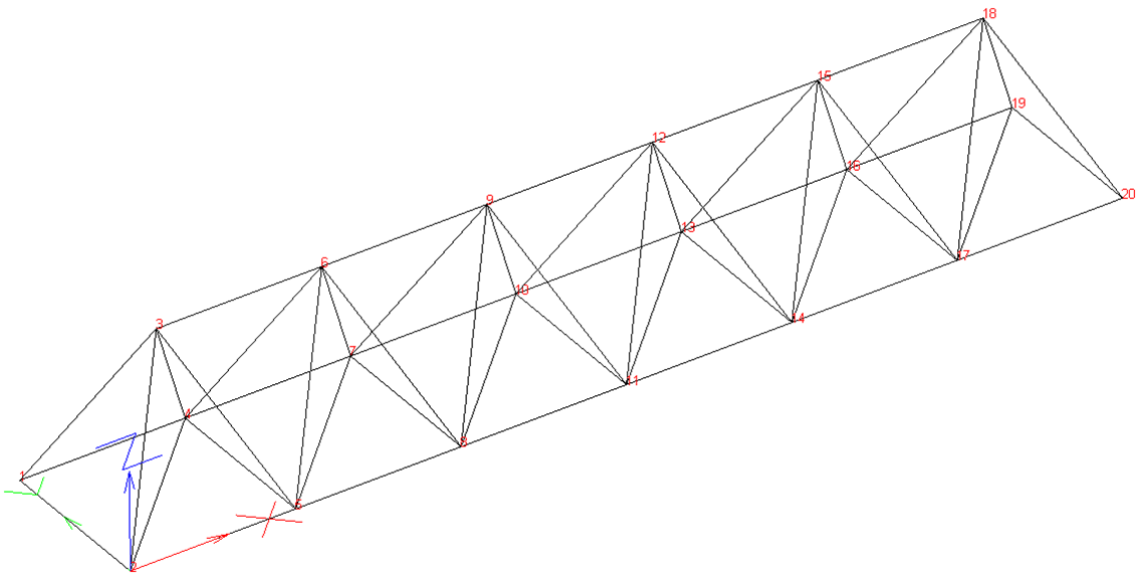
Z88 AURORA® EXAMPLE MANUAL

Example 6: Crane girder

(truss No. 4)



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

6. Example: Crane truss (truss No. 4)

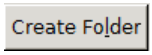
This example is very easy a straight forward: 20 Nodes and 54 trusses constitute a 3D framework. The manual creation of these structures is actual almost the simplest way, CAD-programs are often not very helpful. But let us take a look to the ready model first.

Input file:

b2_x.dxf → structure information's from CAD-program

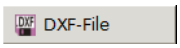
The exercise:

A crane truss consists of 54 trusses (No. 4), 20 nodes and is a 3D-framework. The nodes 1, 2 and 19, 20 are mounted, the nodes 7 and 8 are exposed to a force of 30,000 N. The total length is 12 *m*. The values in the example file are in mm but it is also admissible to use *m*, if the other units like E-modulus and cross section area are adequate to *m* too. The E-modulus is 200,000 N/mm², Poisson ratio v is 0.3. The cross section area of every FE-truss is 500 mm². This example is token from *Schwarz, H.R.: Methoden der finiten Elemente (1991)*.

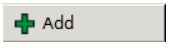
At first a new project is created by using  and 

- in this case for example *Example6* - the dialogue is to be confirmed with *Enter* and completed with *OK*.

With the button 

the appending file *b2_x.dxf* can be imported. A context menu on the right side appears, it can be used to load the DXF-file 

(*Figure 1*). Because the FE-structure is created out of a CAD-system, you have to choose the item *conversion of DXF FE Mesh to z88i1.txt* as import option.

By clicking the button  you switch to the preprocessor. To apply constraints define a new load case has to be added with 

on the right side. Naming it and confirming creates the load case.

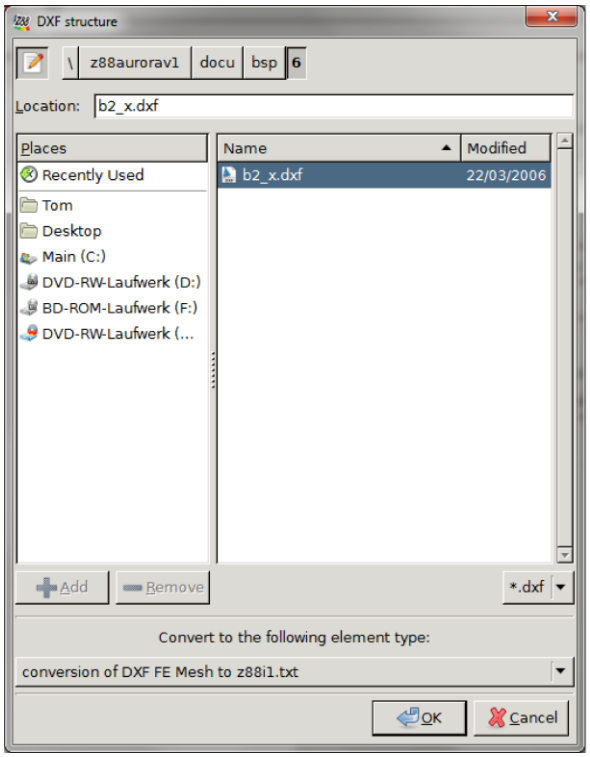


Figure 1: Import the DXF-structure

By clicking Apply constraints, the constraint menu is opening and at the same time the picking view. With F3 you can switch to the mesh view, where it is possible to label nodes. Use *View* → *Labels* → *Nodes* for this purpose, F4 switches back to the picking view. With the help of the CTRL-Key you can choose the nodes 1, 2, 19 and 20, and force its displacements to 0 in every direction. Now deselect them with *Unselect* and pick the nodes 7 and 8 instead. They should be loaded by a force of -30,000 in z-direction. (*Figure 2*) Save saves all these constraints.

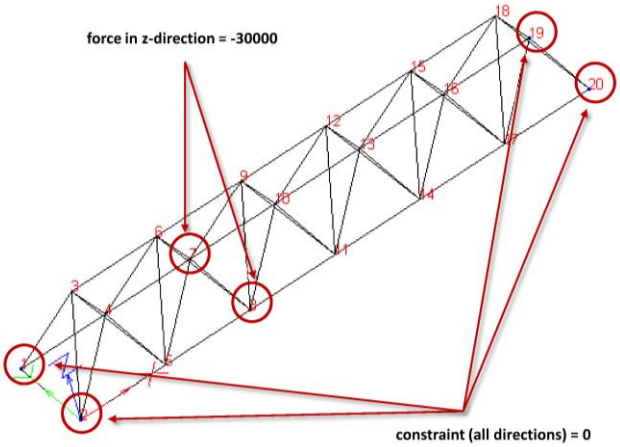


Figure 2: Constraints for example 6

With Define you switch to the file card *"Apply material"*. There already exists a material on the right side (E=200,000 v=0.3). The button Close quits the dialogue.

To calculate the model, switch with to the solver menu. Because in this case we have a very small structure, the direct Cholesky-Solver is a good choice. The stresses are evaluated by von Mises stresses. RUN starts the solver.

After successful calculation the results can be displayed in the postprocessor () (*Figure 3*). To view the tensile and compression stresses choose *Stress per element*. Because in trusses there are just tensile and compression stresses the displayed effective stresses do only represent them.

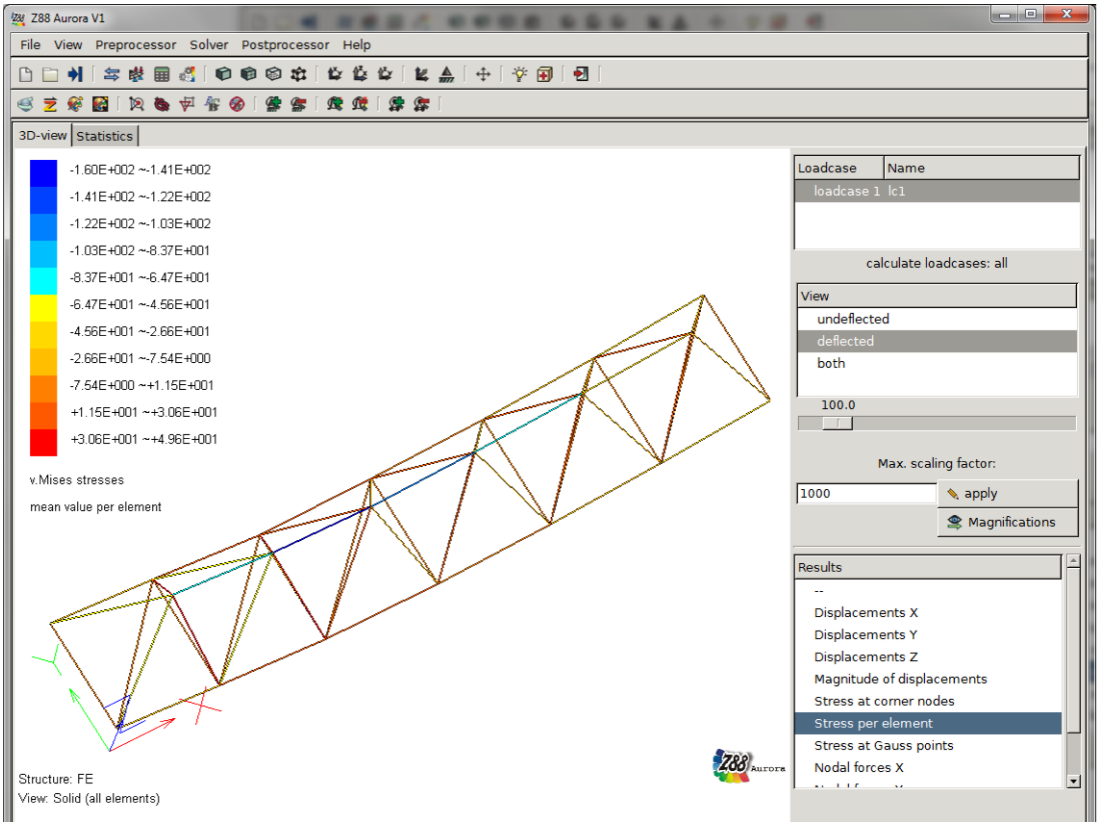


Figure 3: Result screen