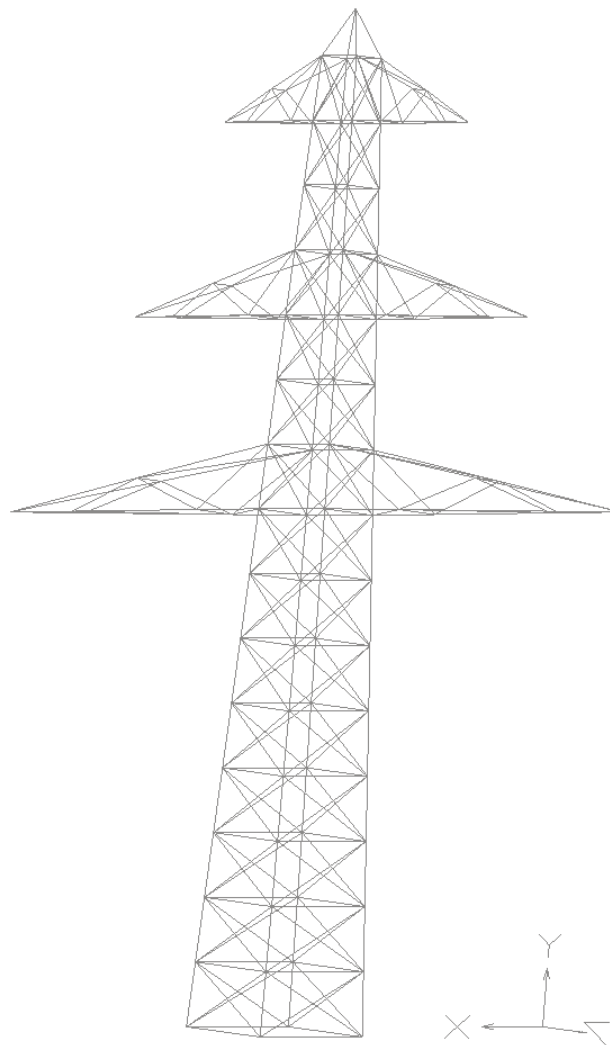


# Z88 AURORA® EXAMPLE MANUAL

## Example 1: Electrical tower

(Truss No. 4, spatial, in 3D)



is a registered trademark (Nr. 30 2009 064 238) of Univ. Prof. Dr.-Ing. Frank Rieg


### Example 1: Structural element (Truss No. 4 in 3D)

Z88 Aurora can handle structural as well as continuum elements. This example will show you how to import data from Z88 V13 and the way to compute truss-based frameworks and skeleton framings.

#### Input files:

b24\_1.txt → Structural information generated with Z88 V13

b24\_2.txt → Boundary conditions created with Z88 V13

Use  and **Create Folder** to create a new project folder, in this case “Example\_1”, then confirm by pressing *Enter* and close the dialog with *OK*.

Now the Z88 V13 input files mentioned above have to be imported as shown in *Figure 1*.

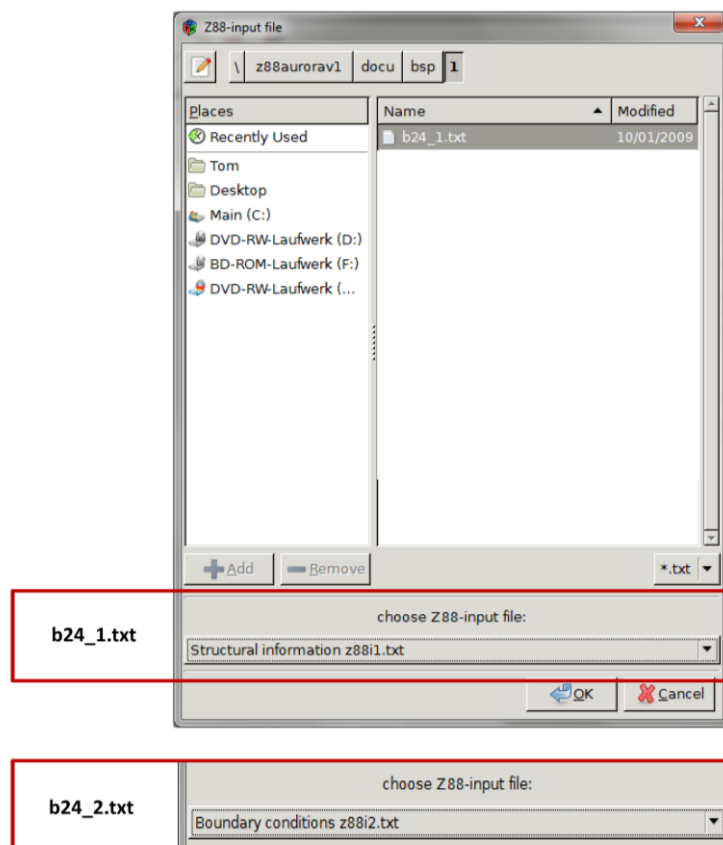





Figure 1: Importing structural data (z88i1.txt) and boundary conditions (z88i2.txt)

With  **Import/Export** the structural data, b24\_1.txt, can be imported. A context menu will appear at the right side of the screen and using  **Z88-File** the Z88 V 13 data can be added to the project. Set the option in the pull-down menu to *Structural information*

## Example Manual

*z88i1.txt*. The structural framework of an electrical pole consisting of truss No. 4 (see Z88 Aurora Theory Manual) will appear.

Now the loads and boundaries have to be provided. With *File* → *Import* → *Z88-files* (\*.TXT) you can open the same dialog again, choose *b24\_2.txt* and add it as the boundary condition data in *z88i2.txt*.

Choosing the  button will open the preprocessor (*Figure 2*). On the right side, you can see that a default load case (*case\_1*) has been created, and by selecting the load case you can view the applied loads and boundaries. The legend on the left side of the viewport can be used to comprehend which type of constraint is used: This example uses fixed displacements and forces only. With *View* → *Size constraints*, you can adjust the display size of the rectangles representing the different types.

In the lower left corner of the viewport, you can see a hint, that you are viewing a *FE* (finite element) *structure*.

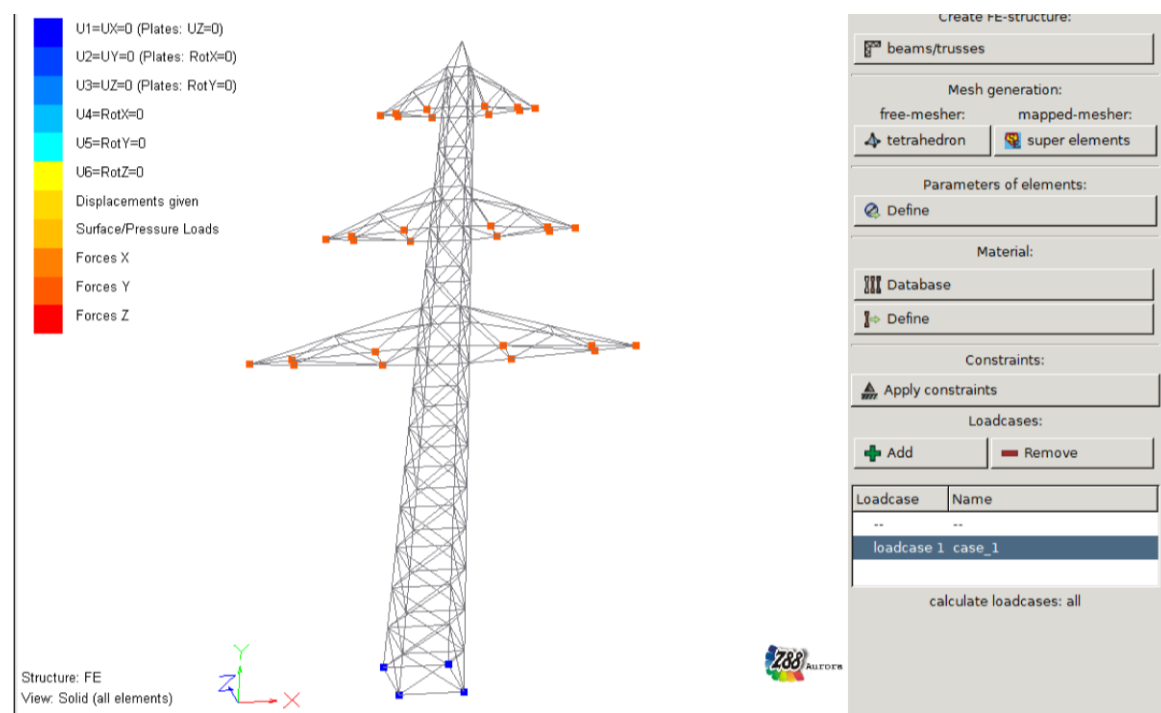




Figure 2: Finite element structure with constraints shown

To get your model computed, you need to use the  button to switch to the solver menu. You can choose from different solvers (see Z88 Aurora Theory Manual). Since in this example we are dealing with a small model, the Cholesky solver is an appropriate choice. The  button will open a notification popup (*Figure 4*). Pressing *OK* will launch the solver.

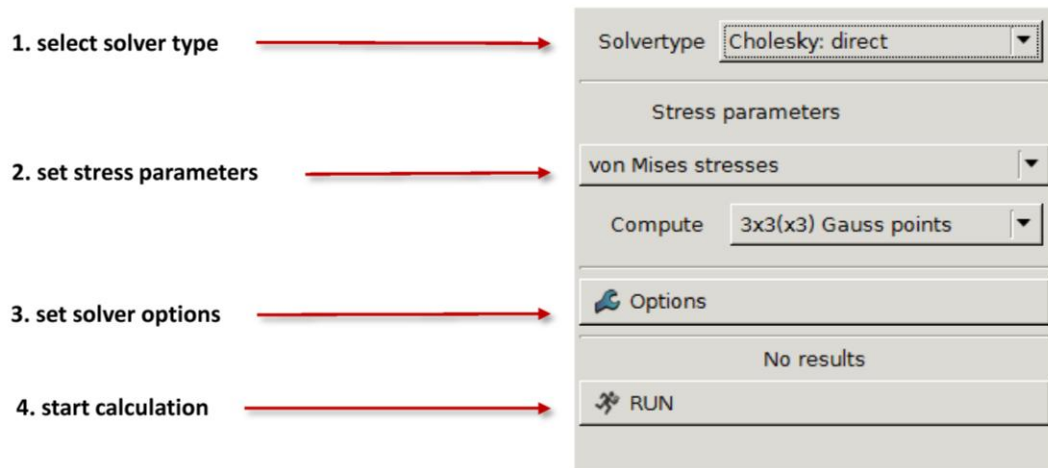



Figure 3: Choosing the options for the Cholesky Solver

When the solver has finished the calculations, the popup will disappear, and if the solver-run was successful, you can start the postprocessor by clicking the  button.

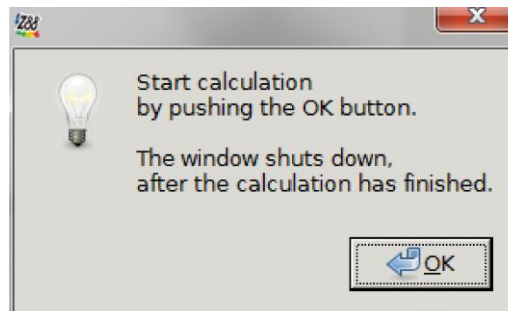


Figure 4: Popup menu

Select the load case in the post menu on the right side of the window (Figure 5). Now you can have the model displayed undeflected, deflected, or with both states combined. With the results tree in the lower part of the menu you can display the displacements (a single component or the magnitude) and the stresses. The use of rods limits the reasonable options to the display of the stresses per element, in this case also representing the reduced stresses since rods can only handle tensile and compressive stress (see Z88 Aurora Theory Manual).

The exact numerical values of displacements and stresses can be viewed directly in output files (*Postprocessor* → *Output data* → *Displacements / Stresses / Nodal forces*).

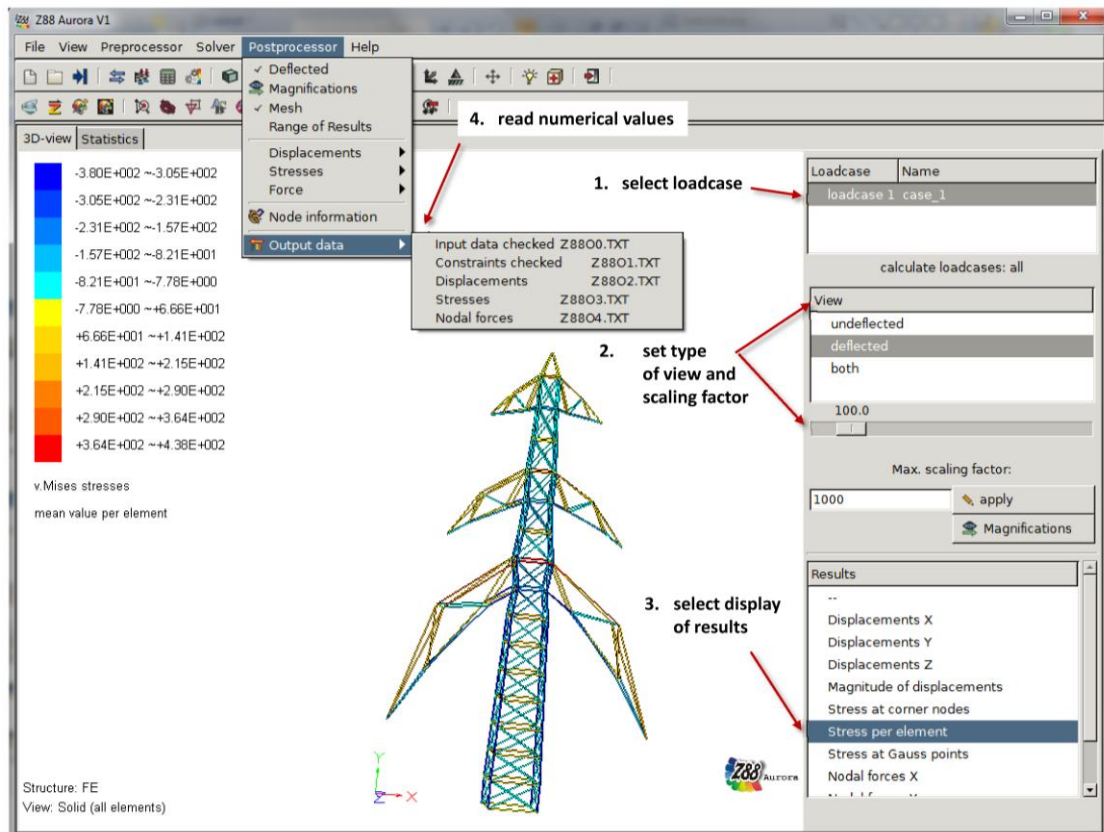



Figure 5: Viewing the results

In addition, Z88 Aurora has an integrated module for statistical evaluation, which can be selected using the tab above the viewport (Figure 6). To use it, select one of the results displaying stress values and select the tab *Statistics*. Either relative or absolute frequencies of the corresponding stress intervals can be selected. If you use the standard value for intervals (11), the colours used in the statistics tab are the same as in the 3D viewport. If you want to save the values for further use in a spreadsheet, use the  csv-Export option and insert an appropriate filename.

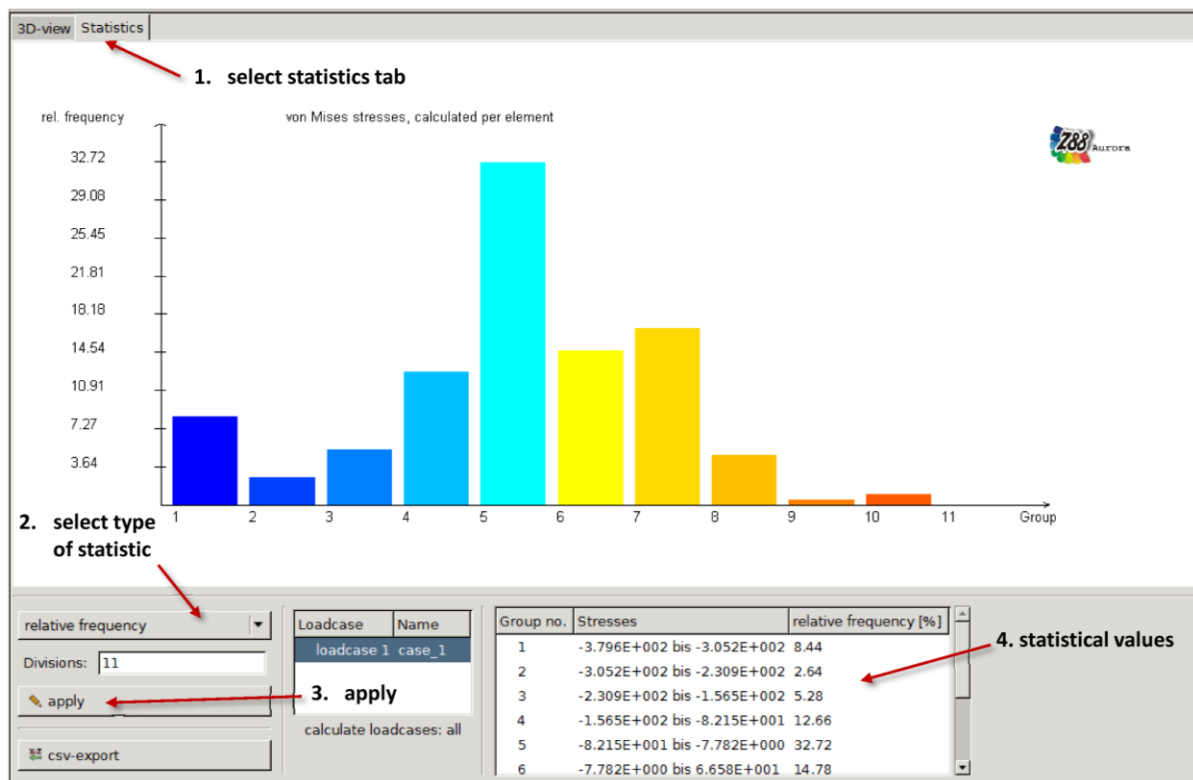


Figure 6: The statistics tab