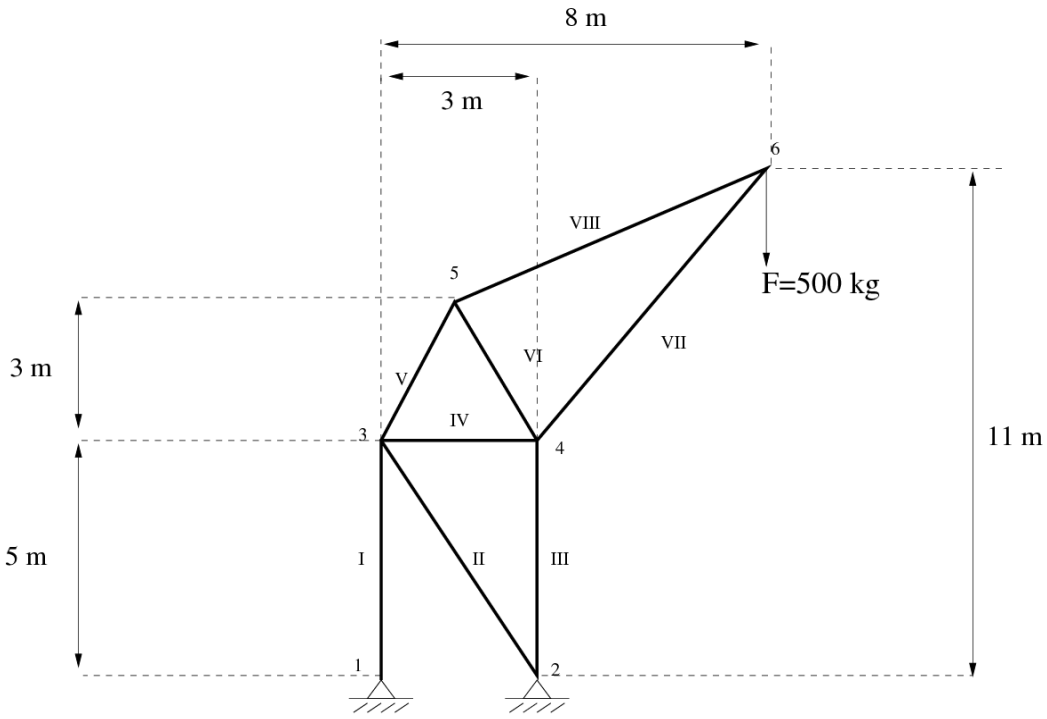
Z88 AURORA® EXAMPLE MANUAL

Example 13: Crane

(Truss No. 9, plane, in 2D)



Cross section: = 8 mm

Example 13: Structures and element parameters (Truss No. 9, plane, in 2D)

Z88 Aurora features an integrated editor that helps you to create structural frameworks made of trusses and beams. You can declare nodes by stating their coordinates, while the elements can be created directly via the viewport (picking) or within the graphical user interface. In this example the procedure is shown for a simplified independent crane. For your convenience the readymade input files are also available.

Input files:

b27_i1.txt → Structural information

b27_i2.txt → Boundary conditions

Use and Create Folder to create a new project folder, in this case "Example_13", then confirm by pressing *Enter* and close the dialog with *OK*. Now the framework shown in *Figure 1* can be created.

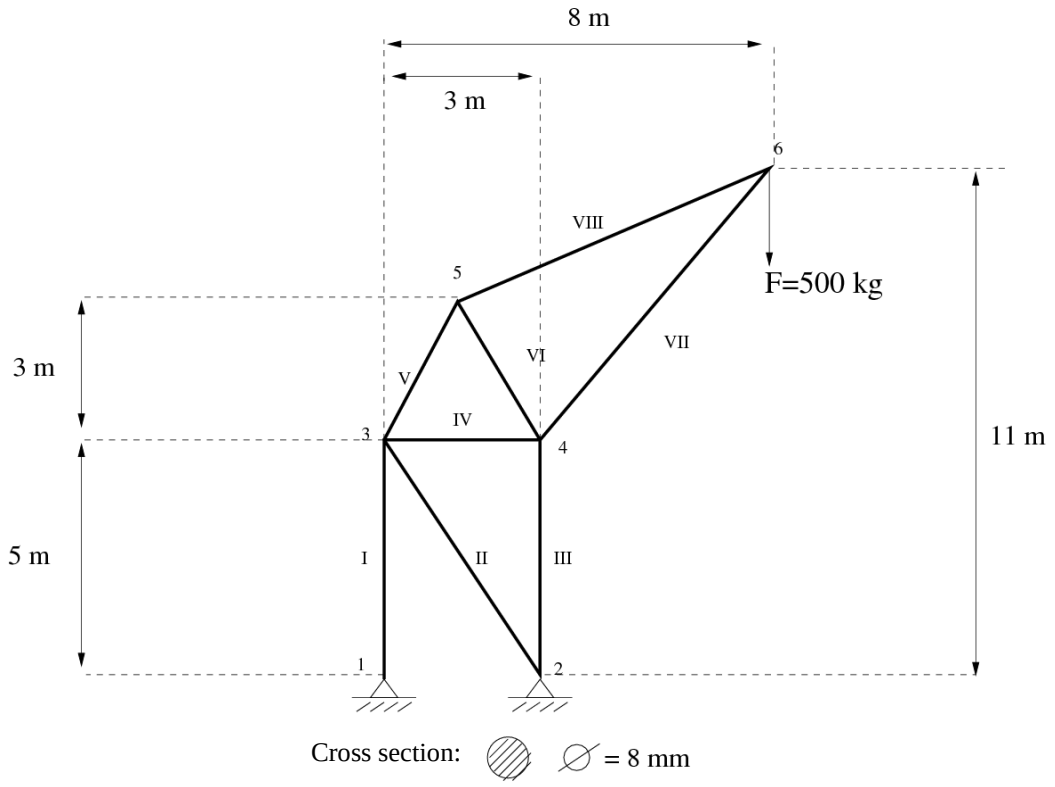


Figure 1: Model of a simplified crane

The shown node (Arabic numerals) and element numbers (Roman numerals) have been chosen arbitrarily and may be replaced at will. Mind that in this case the constraints have to be adjusted as well.

Switch to the preprocessor using . Select beams/trusses to create a new finite element structural framework and a new menu will appear at the bottom of the window (*Figure 2*).

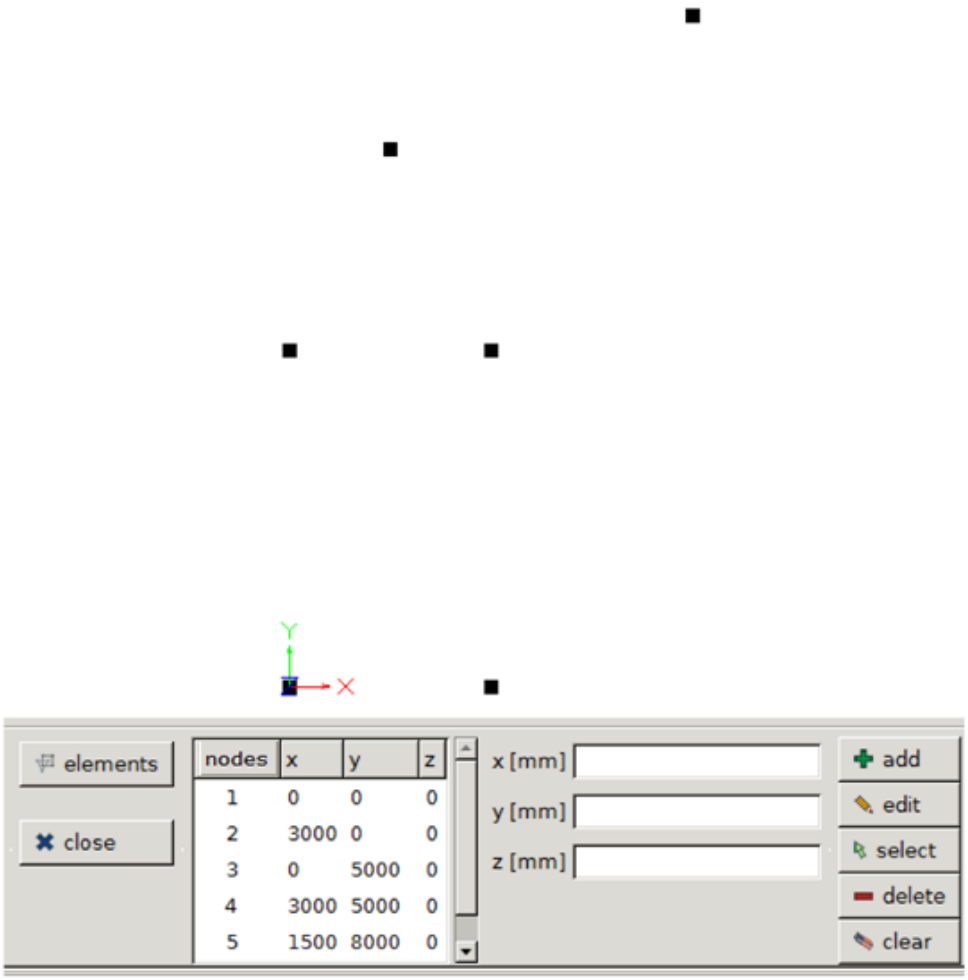


Figure 2: Creating new nodes

In order to add a node, supply the corresponding x-, y- und z-coordinates and use Add. This must be repeated for each node. After selecting a node in the list the values can be edited with edit or removed with delete. For the example, as shown in *Figure 1*, the following coordinates must be used:

Node	x-coordinate	y-coordinate	z-coordinate
1	0	0	0
2	3000	0	0
3	0	5000	0
4	3000	5000	0
5	1500	8000	0
6	8000	11000	0

Now change the menu clicking elements to add the element coincidence. Since this is a model in a plane, choose truss No. 9, plane, in 2D as element type (*Figure 3*).

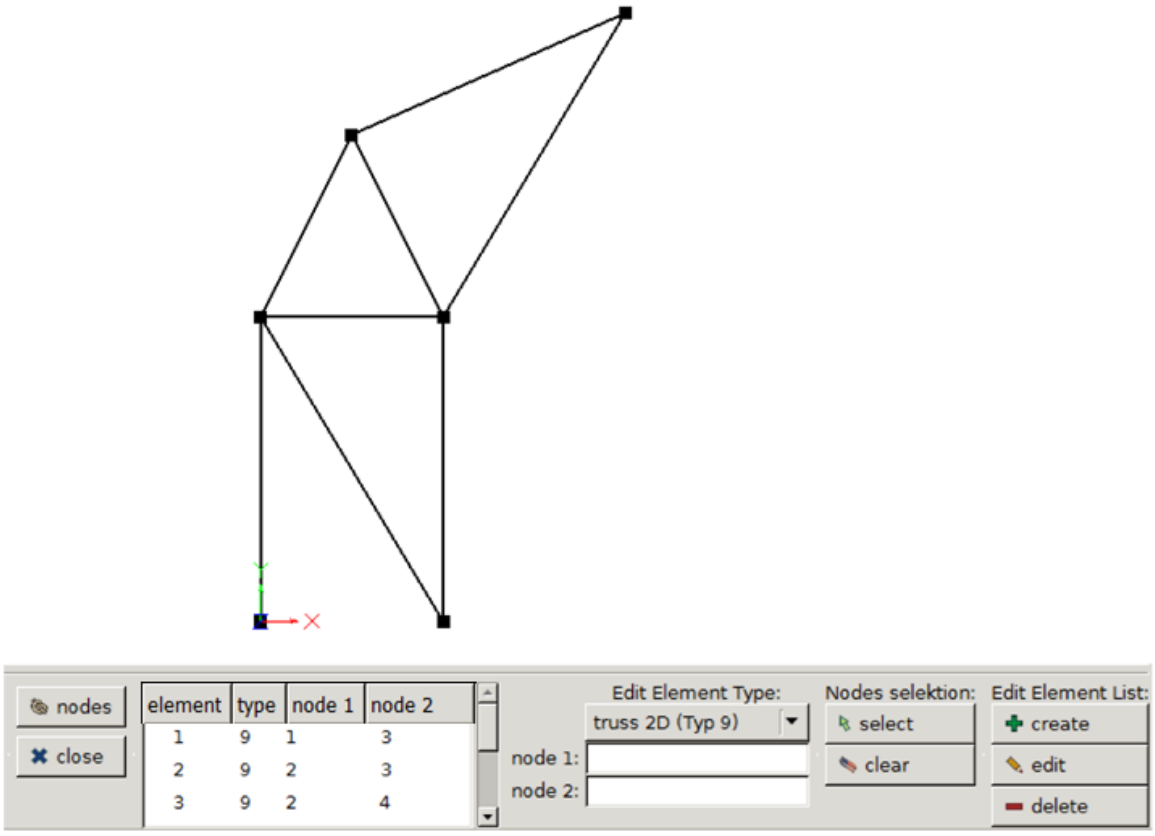


Figure 3: Adding the coincidence

Pick two nodes in the viewport pressing Strg and using the left mouse button and use select, to confirm your choice. Clicking Add will add the element to the list.

Corresponding to *Figure 1* the following elements have to be added:

Element	Type	Node 1	Node 2
1	9	1	3
2	9	2	3
3	9	2	4
4	9	3	4
5	9	3	5
6	9	4	5
7	9	4	6
8	9	5	6

The buttons close and save the newly created structure and close the menu. To assign a material you can use Define to open the material definition tab. Select the material "*Structural Steel S235JR*" and assign it with Add. Save the material definition with the save button and close the tab using close.

Now go to the section "*Parameters for elements*" and open the parameter menu (*Figure 4*) with Define.

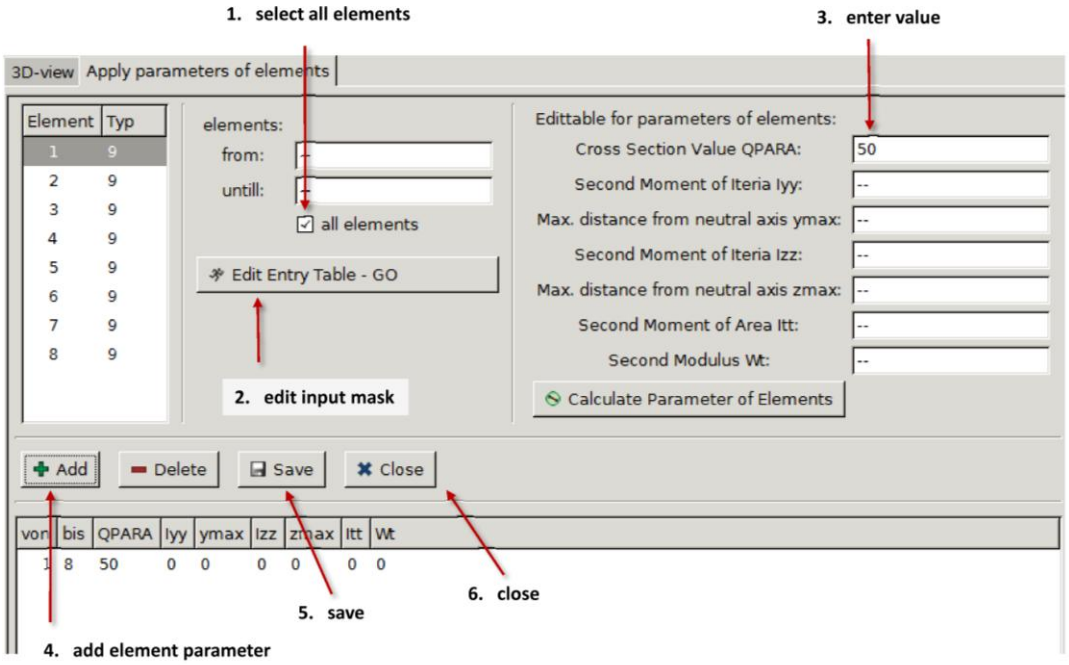


Figure 4: Menu for the declaration of element parameters

Be sure to check the "*all elements*" box, so that your parameters will be applied to the whole model. You can use "*Edit entry table*" to deactivate all entries not needed for your

specific element selection. Trusses only need a value to define the cross sectional area, which in this case is roundabout 50. The "*Add*" button Add will write the parameters to the memory and show them in the list at the bottom of the tab. Now you can save them to a file with save and exit the tab with close.

Now the constraints have to be applied. You have to add a loadcase using Add and insert a name for that case, like "*case1*". Open the constraint menu with the button Apply constraints. Pick the nodes in the viewport pressing Strg and using the left mouse button (see Z88 Aurora User´s Manual).

The crane will be pinned at the two nodes at the base of the structure (nodes 1 and 2) with the option "*Constraint (all directions)*". The crane boom (node 6) will be statically loaded with -5000 in the y-direction (*Figure 5*).

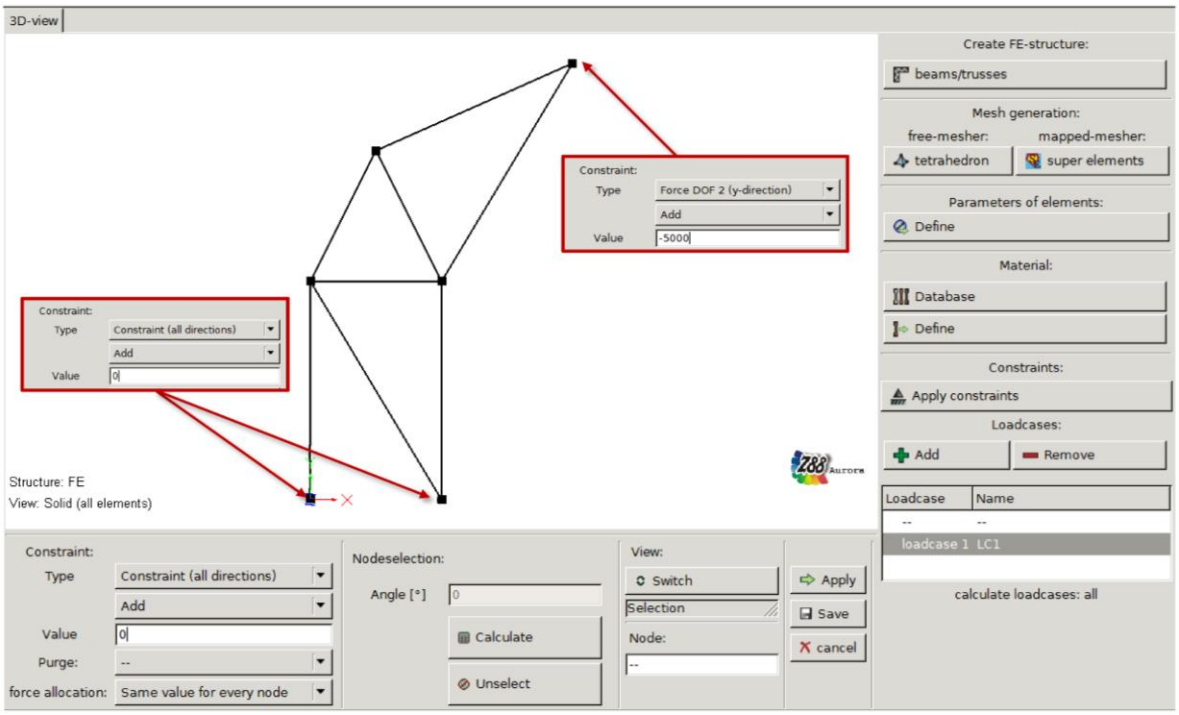


Figure 5: Applying constraints to the crane

save will save the loads and boundary conditions and close the menu.

To have your model computed, you need to use the button to switch to the solver menu. You should use the direct Cholesky solver. Clicking RUN will start the computations.

The postprocessor can be used to view the results (*Figure 6*).

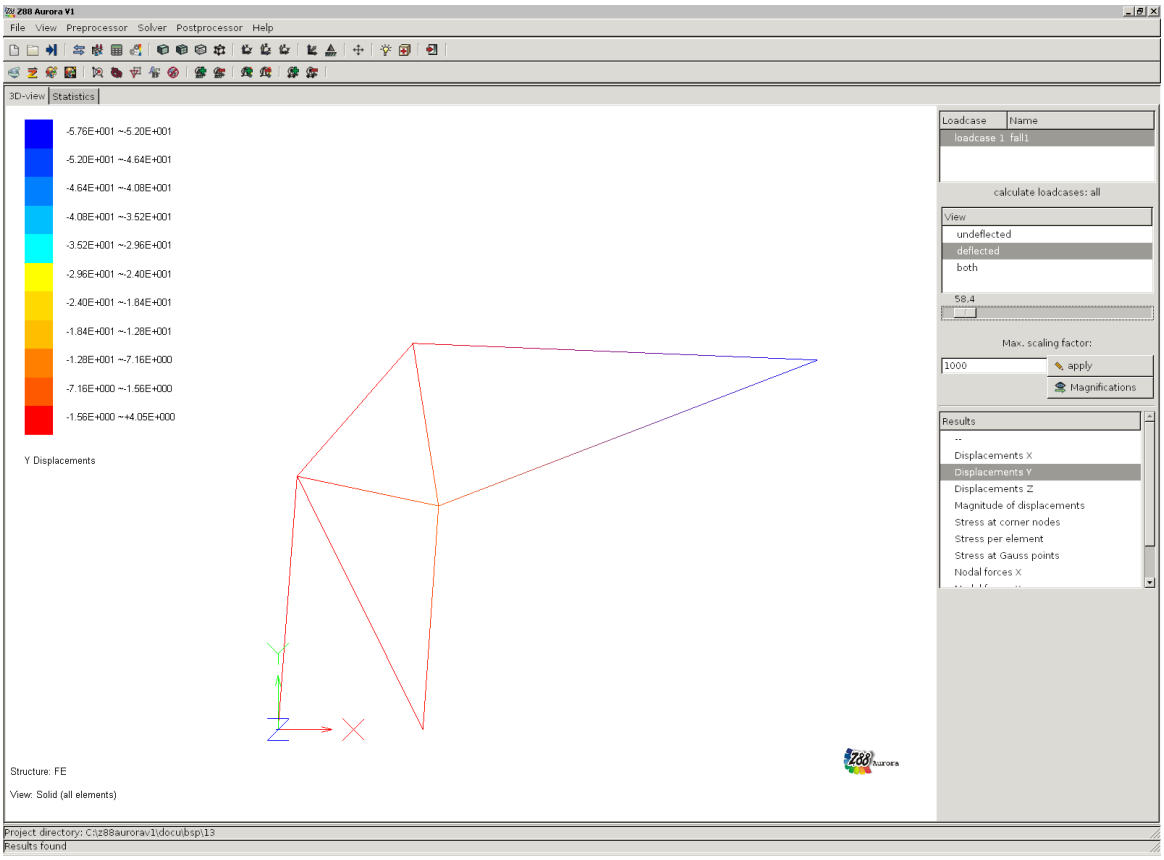


Figure 6: Deflected crane structure