5.1 Crane Modeled by 2D–Trusses

In this section we describe the usage of the general purpose 2d– or 3d–truss element as described before in Sec. 4.6.4 for small strain regime by a real–sized engineering example. Obviously, the following model of a building site crane is very simplified, but it shows quite impressively the capacity of the finite element method in order to obtain numerical results for realistic requests. We assume the model of a crane as given in Fig. 5.1, where the structure

Fig. 5.1 Crane Model by Truss Elements
and the geometrical sizes are given. Further on, we use a Youngs modulus of \( E = 210000 \) MPa for steel and a cross section of the diagonal trusses of \( A = 5 \text{ cm}^2 \) and of \( 2A \) for the main trusses, so that the truss stiffness \( EA \) for each segment is given. The given structure is loaded by a force of \( F = 8000 \) N, while the displacements of point \( P \) as given in Fig. 5.1 are requested.

**Solution**

We realize a numerical solution of the given problem firstly by a discretization of the structure by 2–noded truss elements, where the junctions of the segments represent our finite element nodes and the segments itself the finite (truss) elements. Additionally, due to the different cross sections given above, we define two material sets in the model, which respect for the two different cross sections, see input files `mat1.inp` and `mat2.inp`. The proposed input files for the model are given at \texttt{www.DAEdalon.org/Examples/Crane}. In that model, the marked point \( P \) is identified with `node 11`, where we want to compute the displacement results. Doing so, following the same procedure as described in the very first example in Sec. 1.2, we are able to have a look at the deformed structure by `meshx` as also given in Fig. 5.2. Getting more details of the nodal solutions, we can type e.g. `dis(11)` resulting in \( u_x^P = 23.57 \) mm and \( u_y^P = 23.11 \) mm as the nodal displacements for `node 11`.

![Fig. 5.2 FE model and displacements by `meshx`. Deformations scaled by factor 10 (`defo_scal=10`).](image)
5.2 Axisymmetric Applications

In order to show an application for the axisymmetric element formulation of Sec. 4.6.2, we consider a dome structure of reinforced concrete, assumed to be homogeneous, isotropic and linearly elastic with Young's modulus $E = 20$ GPa and Poisson's ratio $\nu = 0$ as given in Fig. 5.3. Again, a proposal for the node and element information is given at www.DAEdalon.org/Examples/Dome, so that a solution of that task can easily be obtained.

Further on, we want to focus here on the determination of the reaction forces at the mounting of the structure, which is given in more detail including node numbers in Fig. 5.4. As given, we have to add the computed reaction forces in axial–/2–direction of nodes 10, 17–20 and 109–112 in a postprocess after solving the linear elastic mechanical problem following the standard procedure shown in Sec. 1.5. At that point, we have a look on the structure of the used (internal) fields, e.g. rnode in this case. The matrix rnode is of size [numnp × 3], where numnp gives the number of nodes, while the columns are reserved for 3 dimensions. So, the rows in rnode contain the (reaction) forces of each node after solving the given boundary value problem. Consequently, not restricted degrees of freedom are filled with 0.0 in rnode. In our case, we can obtain the requested mounting loads by adding the second column values of rnode of the bounded nodes 10, 17–20 and 109–112, or just by summing up the second column of rnode and subtracting the applied load of 2000 N at node 35:

$$\text{sum(rnode(:,2))} - \text{rnode(35,2)}$$
The resulting sum is according to amount the same as the applied load at the top of the structure, which additionally indicates, that the system is in mechanical equilibrium.

5.3 Crack Tip Simulation

A further, more advanced application of the FEM is given by the analysis of a crack tip field, whose order of magnitude is described by a crack intensity factor $K_I$. We want to address the computation of a given $K_I$–field in comparison to the well discussed analytical solution, see e.g. GROSS & SEELIG [2006].

5.3.1 $K_I$–Field

We describe a crack tip in a plane strain configuration as given in Fig. 5.5 with a radius of $r = 0.1$ mm. As known, this region is loaded by boundary conditions from a $K_I$–dominated far–field of the form
5.3 Crack Tip Simulation

\[ u_x = \frac{K_I}{2G} \sqrt{\frac{r}{2\pi}} \left(3 - 4\nu - \cos \varphi\right) \cos \frac{\varphi}{2} \]

\[ u_y = \frac{K_I}{2G} \sqrt{\frac{r}{2\pi}} \left(3 - 4\nu - \cos \varphi\right) \sin \frac{\varphi}{2}, \]

(5.1)

where we choose a linear elastic material behavior through \( E = 210000 \text{ MPa} \) and \( \nu = 0.3 \) and a loading intensity of \( K_I = 10 \text{ MPa}\sqrt{\text{m}} \). The solution of the given problem computed with D\( \text{AEdal} \)on gives a \( \sigma_y \) stress contour as given in Fig. 5.6. As additionally given in textbooks on fracture mechanics depicting the \( K \)-concept, one obtain a typical opening

\[ u_y^\pm = \pm \frac{2K_I}{G} \sqrt{\frac{r}{2\pi}} (1 - \nu) \]

(5.2)

of the crack edges for the chosen situation. In Fig. 5.7 we give that analytical result (5.2) of the crack tip opening displacement in comparison to the FE solution.
Fig. 5.6 $\sigma_y$ stress distribution on scaled configuration by $\text{meshx}$. Deformations scaled by factor 1000 ($\text{defo}\_\text{scal}=1000$).

Fig. 5.7 Comparison of Analytical Solution (line) and FEM Results ($\times$) for (5.2)
5.3.2 Analysis of Plastic Zone within $K_I$–Field

Furthermore, a straight forward extension of the above linear elastic treatment is the application of a material model incorporating plastic behavior in order to analyze the plastic zone near the crack tip ("small scale yielding"). Exemplarily, one may use simple VON MISES plasticity with a hardening rule as given in (3.26) and implemented in Sec. 4.6.5 of the form

$$\sigma_{\text{yield}} = \sigma_{\text{yield}_0} \left[ \frac{\alpha}{\varepsilon_0} + 1 \right]^\frac{1}{N}$$  \hspace{1cm} (5.3)

with $\varepsilon_0 = \sigma_{\text{yield}_0}/E$ and $\sigma_{\text{yield}_0}$, $E$ and $N$ as material parameters.

For this example we use the elastic parameters as before and $\sigma_{\text{yield}_0} = 210$ MPa as initial yield stress and $N = 6$ as hardening coefficient, respectively. Again, the example input files can be obtained from [www.DAEdalon.org/Examples/KI-field/plastic](http://www.DAEdalon.org/Examples/KI-field/plastic) to follow the descriptions here. The applied loading is now given by the loading factor $K_I = 100 \text{ MPa}\sqrt{\text{m}}$, which we increase by

```matlab
loop(140)
defo_scal=10
cont(13)
```

up to 140 times and observe the result by the (accumulated) equivalent plastic strain $\alpha$. The resulting contour plot is given in a zoom–out in front of the crack tip in Fig. 5.8.

![Fig. 5.8 Plastic zone indicated by the equivalent plastic strain $\alpha$](image-url)