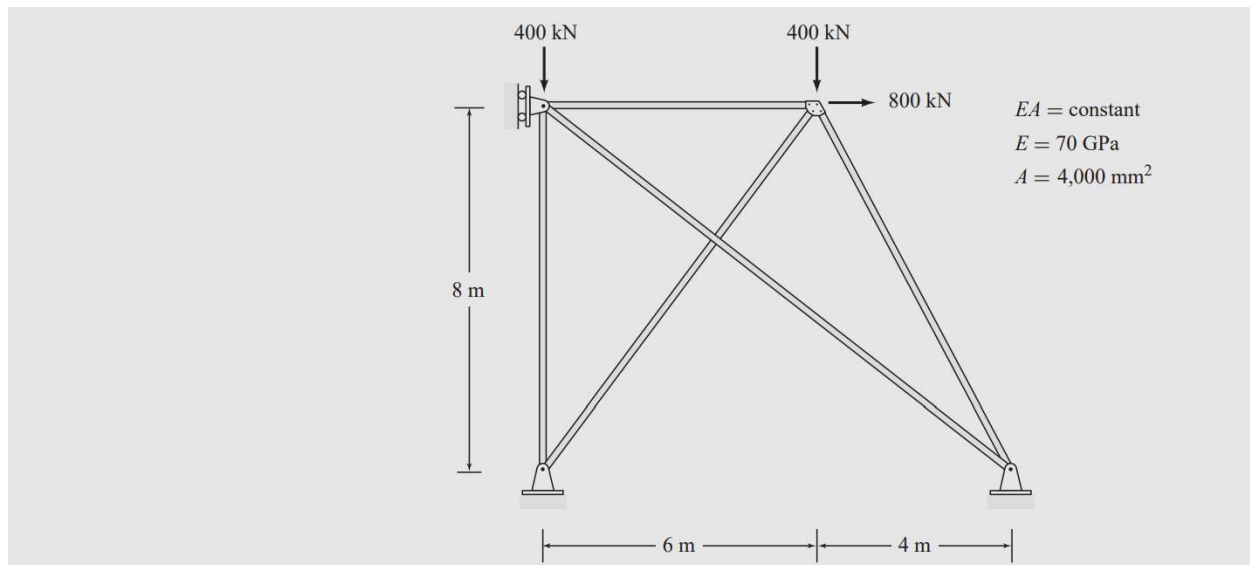# Example Walkthrough

The following example demonstrates how to analyze the truss shown in the diagram below using the provided inputs.

## 1. The Problem

We are analyzing a truss with specific loading conditions (400kN vertical loads, 800kN horizontal load) and geometric constraints (E = 70GPa, A = 0.004m$^2$)

*Figure 1: The engineering problem statement.*



## 2. Defining Geometry (Nodes)

First, define the number of nodes and their coordinates $[x, y]$.

- **Node 1:** (0, 0)
- **Node 2:** (10, 0)
- **Node 3:** (0, 8)
- **Node 4:** (6, 8)

*Figure 2: Entering node coordinates and initial setup.*

## 3. Defining Elements & Properties

Next, define how nodes are connected (Elements), material properties, and external forces.

- **Connectivity:** Connect Node 1 to 3, 3 to 4, etc.
- **Forces:** Enter force vectors [F_x, F_y] for loaded nodes.
    - *Example:* Node 4 has a force of [800 -400] (representing 800kN right and 400kN down).
- **Boundary Conditions:** Define which nodes are fixed (1) or free (0).
    - *Example:* Node 1 is a pinned support, so it is fixed in X and Y [1 1].

*Figure 3: Defining connectivity, loads, and boundary conditions.*

```
Command Window
Enter the number of elements: 5
Enter element connectivity as [Node1 Node2]:
  Element 1: [1 3]
  Element 2: [3 4]
  Element 3: [1 4]
  Element 4: [2 3]
  Element 5: [2 4]
How many different Young's moduli are there? 1
  Young's Modulus 1 (Pa): 70e9
How many different cross-sectional areas are there? 1
  Cross-sectional area 1 (m^2): 0.004
Assign properties to each element:

Element 1 (Nodes 1-3):

Element 2 (Nodes 3-4):

Element 3 (Nodes 1-4):

Element 4 (Nodes 2-3):

Element 5 (Nodes 2-4):
\nEnter forces at each node as [Fx Fy] (enter 0 if no force):
  Force at Node 1: 0
  Force at Node 2: 0
  Force at Node 3: [0 -400]
  Force at Node 4: [800 -400]
\nEnter boundary conditions as [Ux Uy] (1 = fixed, 0 = free):
  Node 1 (Fixed/Free): [1 1]
  Node 2 (Fixed/Free): [1 1]
  Node 3 (Fixed/Free): [1 0]
  Node 4 (Fixed/Free): [0 0]
```

## 4. Numerical Results

Once inputs are complete, the program calculates the stiffness matrix and solves the system. The results are displayed in the command window, showing exact displacement values and stress in Pascals.

*Figure 4: Calculated displacements, reactions, and member stresses.*

```
\n=== ANALYSIS RESULTS ===
\nNodal Displacements (m):
Node 1: Ux = 0.0000e+00, Uy = 0.0000e+00
Node 2: Ux = 0.0000e+00, Uy = 0.0000e+00
Node 3: Ux = 0.0000e+00, Uy = -9.1886e-06
Node 4: Ux = 1.2837e-05, Uy = -9.5844e-06
\nReactions at fixed nodes (N):
Node 1, Fx = -0.58 N
Node 1, Fy = 320.83 N
Node 2, Fx = -298.39 N
Node 2, Fy = 479.17 N
Node 3, Fx = -501.04 N
\nMember Forces (N) and Stresses (Pa):
Element 1 (Nodes 1-3): Force = -321.60 N, Stress = -8.04e+04 Pa
Element 2 (Nodes 3-4): Force = 599.04 N, Stress = 1.50e+05 Pa
Element 3 (Nodes 1-4): Force = 0.96 N, Stress = 2.41e+02 Pa
Element 4 (Nodes 2-3): Force = -125.50 N, Stress = -3.14e+04 Pa
Element 5 (Nodes 2-4): Force = -448.07 N, Stress = -1.12e+05 Pa
```

## 5. Visualization

Finally, a plot is generated.

- **Dotted Black Line:** Original shape.
- **Solid Red Line:** Deformed shape (exaggerated for visibility).
- **Red Circles:** Members in **Tension**.
- **Blue Circles:** Members in **Compression**.
- **Blue Arrows:** Reaction forces at supports.

`Figure 5: Graphical representation of the finite element analysis.