

# MATLAB 2D Truss Analysis Program

## Overview

This MATLAB program (`gai_truss_anaylsis.m`) performs a Finite Element Analysis (FEA) on 2D plane trusses using the Direct Stiffness Method. It calculates nodal displacements, support reactions, member forces, and axial stresses.

The program features an interactive text-based interface for inputs and generates a comprehensive graphical visualization of the deformed structure, including color-coded stress states (Tension/Compression) and reaction vectors.

## Features

- **Interactive Input:** Guided command-line prompts for geometry, connectivity, and properties.
- **Multiple Materials:** Support for varying Young's Modulus (E) and Cross-sectional Areas (A) across different elements.
- **Detailed Results:** Outputs numerical tables for displacements, reactions, and member forces.
- **Visualization:**
  - Plots Undeformed (dotted) vs. Deformed (solid red) shapes.
  - Color-codes members based on stress state (Red = Tension, Blue = Compression).
  - Visualizes reaction forces with scaled arrows.

## Usage

1. Ensure you have MATLAB installed.
2. Save `gai_truss_anaylsis.m` to your working directory.
3. Type `gai_truss_anaylsis` in the MATLAB Command Window and press **Enter**.
4. Follow the prompts to enter your truss data.

---

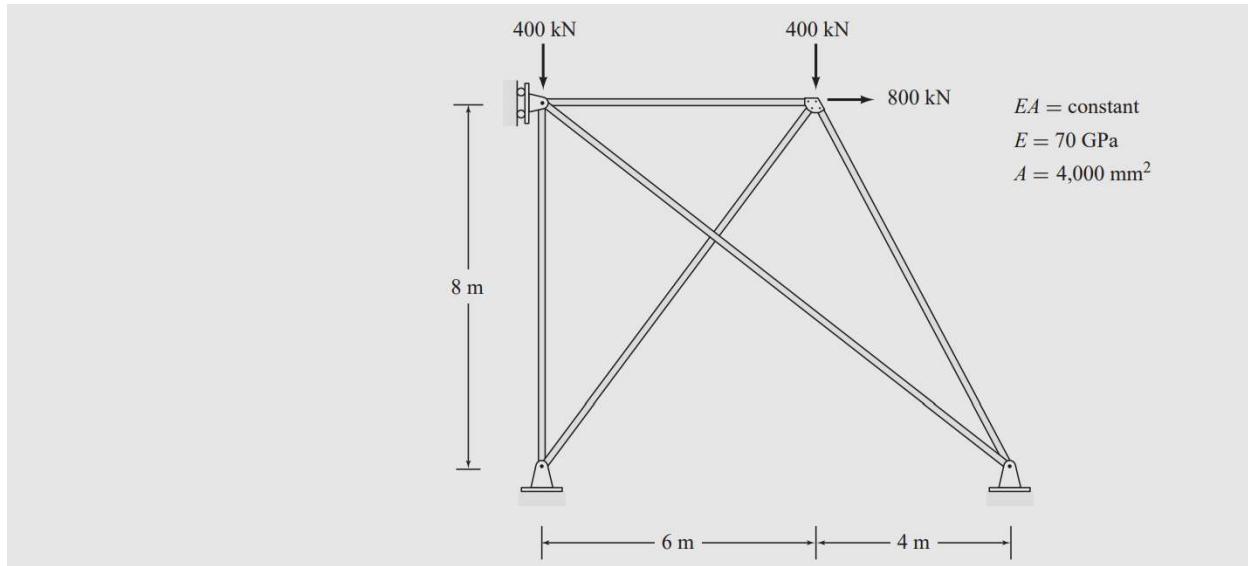
## Example Walkthrough

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 ( $E = 70\text{GPa}$ ,  $A = 0.004\text{m}^2$ )

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.

```
Command Window
==== TRUSS ANALYSIS PROGRAM ====
Enter the number of nodes: 4
Enter node coordinates as [x y]:
Node 1: [0 0]
Node 2: [10 0]
Node 3: [0 8]
Node 4: [6 8]
```

## 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.*

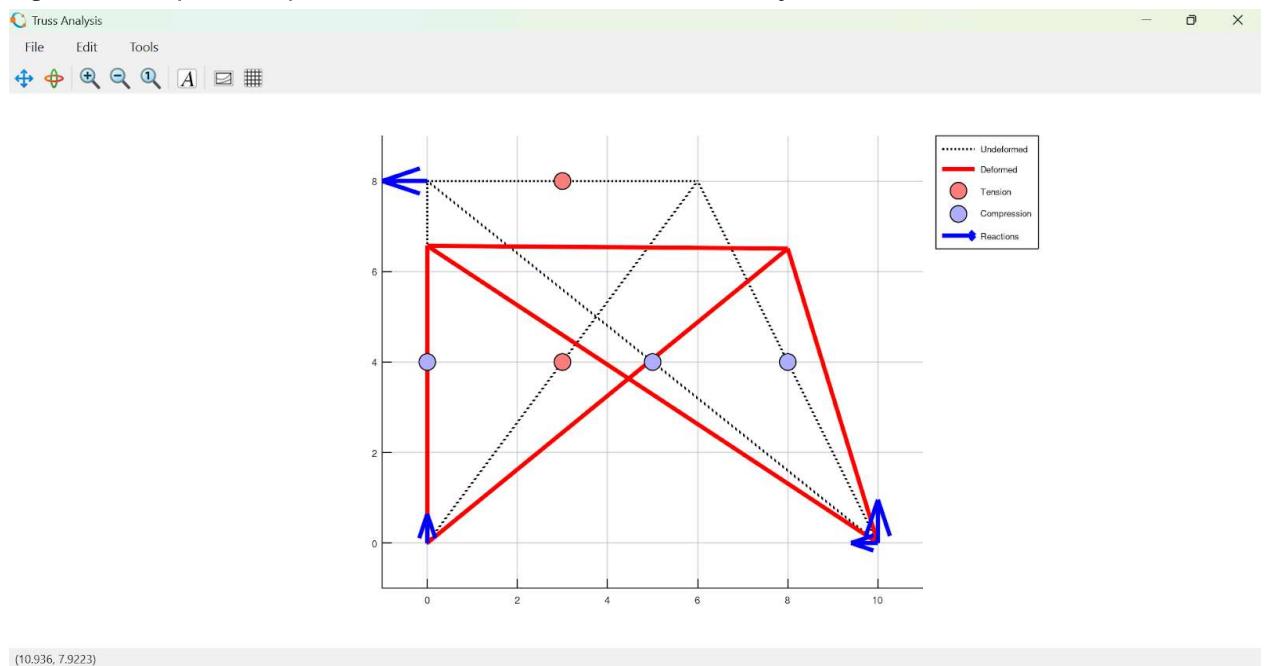
```
\n==== ANALYSIS RESULTS ====\n\nNodal Displacements (m):\nNode 1: Ux = 0.0000e+00, Uy = 0.0000e+00\nNode 2: Ux = 0.0000e+00, Uy = 0.0000e+00\nNode 3: Ux = 0.0000e+00, Uy = -9.1886e-06\nNode 4: Ux = 1.2837e-05, Uy = -9.5844e-06\n\nReactions at fixed nodes (N):\nNode 1, Fx = -0.58 N\nNode 1, Fy = 320.83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



## Input Guidelines

When running the program, use the following formats:

- **Coordinates:** Enter space-separated numbers (e.g., 0 0).
- **Connectivity:** Enter NodeStart NodeEnd (e.g., 1 3).
- **Forces:** Enter Fx Fy in Newtons (e.g., 0 -400 for -400kN downward).
- **Boundary Conditions:** 1 = Fixed, 0 = Free.
  - Pin Support: 1 1
  - Roller (Vertical Wall): 1 0 (Fixed X, Free Y)
  - Free Node: 0 0

## Technical Details

- **Method:** Matrix Structural Analysis (Direct Stiffness Method).
- **Files:** gai\_truss\_anaylsis.m (Main script).
- **Dependencies:** Standard MATLAB installation (no toolboxes required).