1. Overview:

This documentation presents a Python-based tool designed for analyzing 2D trusses using the Finite Element Method (FEM). The tool comprises two main components: the Truss Builder and the Truss Analysis class. The Truss Builder allows users to interactively construct truss structures, including defining nodes, members, supports, and forces. The Truss Analysis class, on the other hand, performs the computational analysis of the truss, calculating displacements, stresses, and reaction forces based on the user-defined structure. The main function of the tool ties these components together, facilitating the loading, analysis, and visualization of the truss analysis results. This tool aims to provide an intuitive and efficient way for engineers and students to analyze truss structures for educational and professional purposes.

2. Components:
    a. Truss Builder:

    The TrussBuilder is a graphical interface developed using Tkinter, enabling users to design truss structures. It provides functionality to place nodes, add members (connecting the nodes), specify supports, and apply forces. This component translates the user's inputs into a structural model that can be analyzed.

    Key Features:

    - Grid system for precise placement of nodes.
    - Options to create members by connecting nodes.
    - Ability to specify different types of supports.
    - Functionality to apply forces at various nodes..

    b. Truss Analysis:

    The TrussAnalysis class is the core computational component. It utilizes the structure defined in the Truss Builder to perform FEM analysis. This class calculates the displacements at each node, the stresses in each member, and the reaction forces at the supports.

    Core Functions:

    - Assembly of the global stiffness matrix for the structure.
    - Calculation of nodal displacements under applied loads.
    - Determination of internal stresses in truss members.
    - Computation of reaction forces at support points.

c. Main Function:

The main function serves as the bridge between the Truss Builder and Truss Analysis. It loads the truss structure from a saved file, executes the analysis, and displays the results.

Workflow:

- Loading the truss structure from a saved file (typically a pickle file).
- Performing the truss analysis to calculate displacements, stresses, and reaction forces.
- Visualizing the truss along with the calculated results to provide a comprehensive understanding of the structural behavior.

3. Usage Guide:
   This section provides a step-by-step guide on how to use the truss analysis tool:

   a. Building a Truss Structure:

   - Open the Truss Builder interface.
   - Use the grid system to place nodes at desired locations by clicking on the canvas.
   - Connect nodes by selecting the 'Add Members' mode, creating members between nodes.
   - Specify supports by switching to the 'Add Supports' mode and clicking on the appropriate nodes.
   - Apply forces by selecting the 'Apply Forces' mode and specifying the magnitude and direction at the relevant nodes.
   - Save the designed truss structure for analysis.

   Running the Analysis:

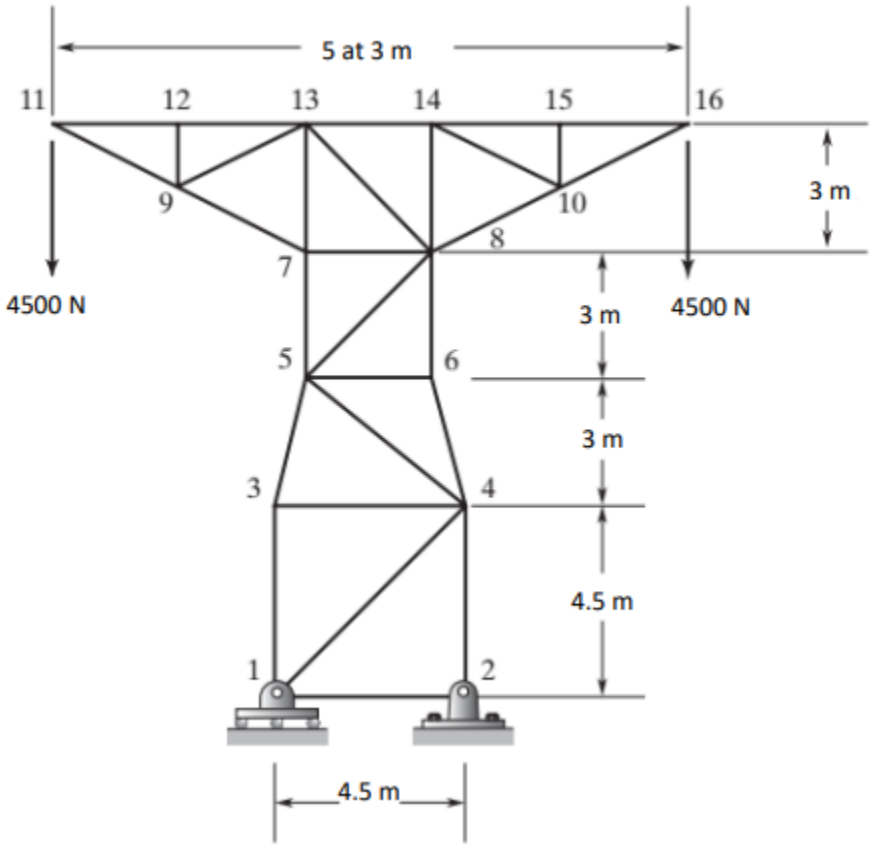
   - Use the main function to load the saved truss structure.
   - The program automatically calls the analyze method from the TrussAnalysis class, which performs the FEM analysis.
   - The analysis results in the computation of nodal displacements, member stresses, and reaction forces at supports.

   Viewing the Results:

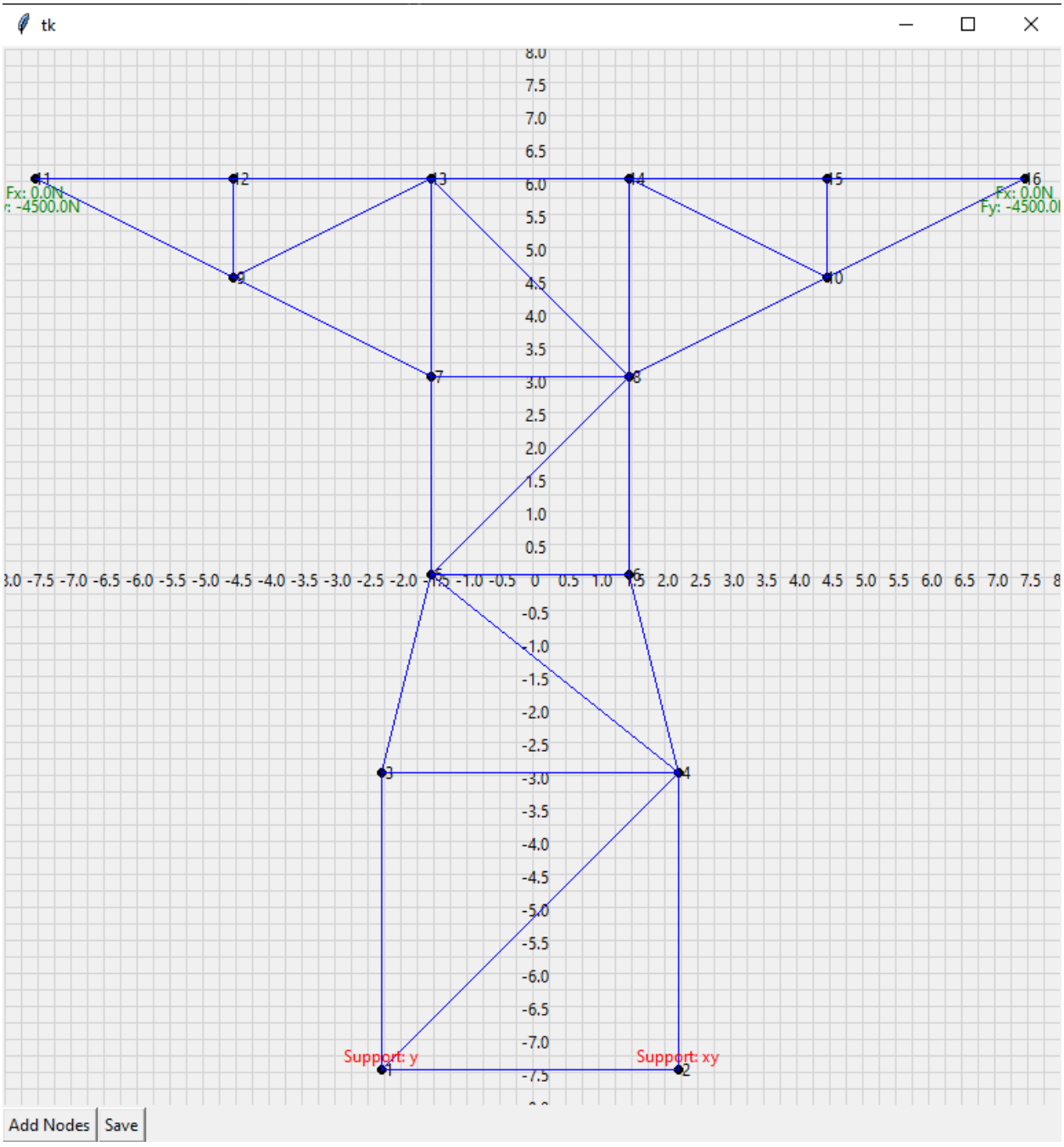
   After analysis, the results (displacements, stresses, and reaction forces) are displayed in the console.
   A visual representation of the truss is also generated, showing the structure along with annotated results.
   The visual output includes:
   - Node positions and connections (members).
   - Displacement vectors at each node.

- Stress values for each member.
- Reaction forces at the supports.

4. Solving the truss given:
    a. Truss given



b. Creating the truss with truss_builder.py

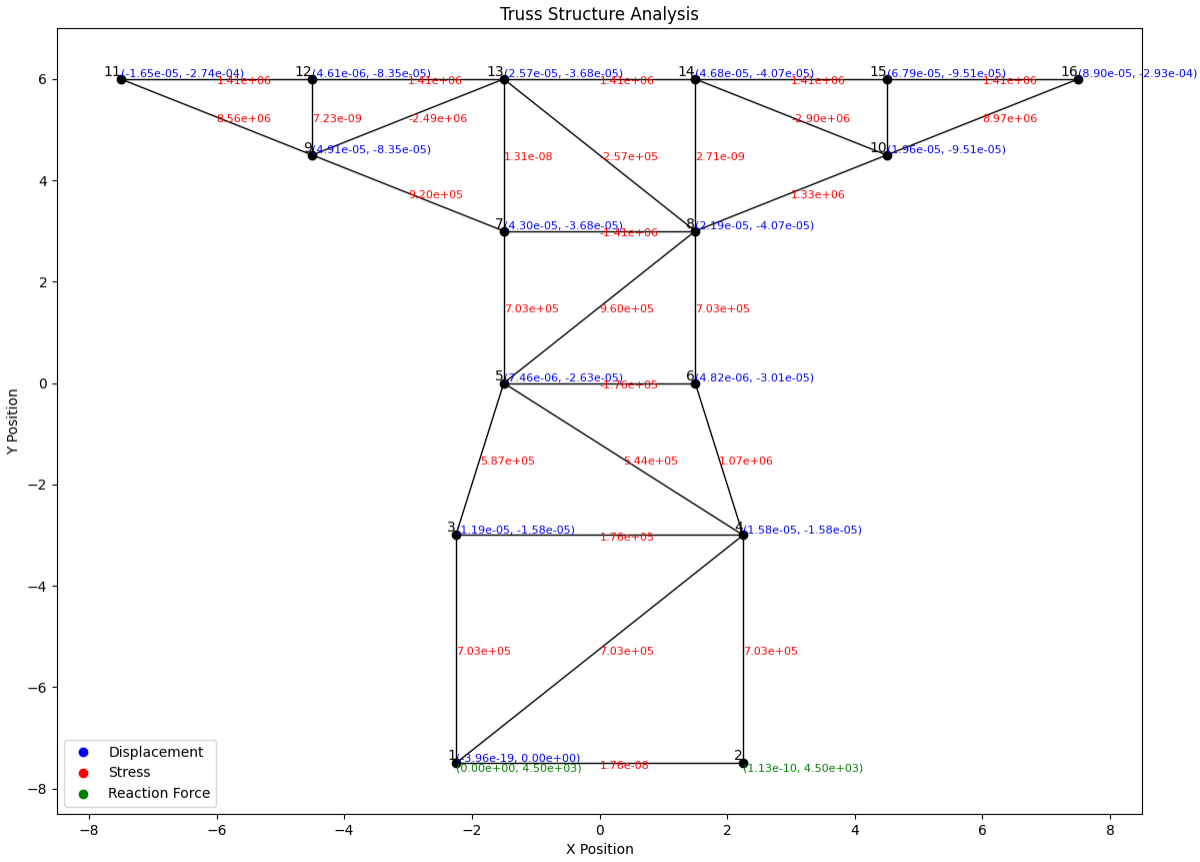


Building the truss requires:

- Adding nodes
- Connecting nodes to make elements
- Adding supports
- Adding loads
- Save the truss

c. Results

After loading truss in the main.py, the program calculates the displacements, stress and reactions. Shows the truss and add the result to the figure



5. Conclusion

The Python-based truss analysis tool effectively simplifies the process of evaluating 2D truss structures using the Finite Element Method. Its integration of a graphical interface enables both educational and professional applications in structural engineering.