2D Truss Module – User Guide, Solution Methodology, and References

1. Introduction

This document describes the usage, numerical solution methods, and key references for the 2D Truss Analysis Module. The tool enables users to interactively draw, edit, solve, and visualize truss structures, including reactions, displacements, stresses, self-weight effects, and modal behavior. It is designed as a general-purpose engineering analysis platform for axially loaded 2D truss systems with optional nodal springs, distributed load visualization, and modal analysis using lumped masses.

2. User Interface Overview

2.1 Drawing Members

• Select 'Draw Members' to begin sketching.
• Click and drag to create a member between two node points.
• If the start or end point is close to an existing node, the member attaches to that node; otherwise a new node is created automatically.

2.2 Selecting and Editing

• Switch to 'Select/Edit' mode to modify the geometry.
• Drag a node to reposition it; the geometry and governing matrices update automatically.
• Double-click a node to edit boundary conditions, loads, springs, and node weights.
• Double-click a member to edit E, A, I, S, weight density (wd), and label.
• Press Delete to remove the selected node or member.

2.3 Viewing Modes

• Geometry view: Shows the undeformed truss.
• Deflection view: Shows the displaced shape, scaled by a user-defined or automatically computed deflection scale.
• Stress view: Displays σL and σR along a selected member, including axial and bending components.

2.4 Solving

Press 'Solve' to compute displacements, reactions, axial forces, and stresses. Nodal results appear in the sidebar. The displaced geometry is available in the Deflection view.

2.5 Modal Analysis

Press 'Modes' to compute the first three natural frequencies and mode shapes of the truss. Modes require valid A and wd for each member, and at least one support condition. The mode selector allows switching between static deflection and modal deformed shapes.

2.6 Animation

The 'Animate' button enables time-varying visualization of the selected mode shape using a harmonic deflection envelope. Frequency may be controlled by an optional slider if present.

2.7 Saving, Loading, and Exporting

• The 'Save Input' button exports the entire model as JSON.
• The 'Load Input' button restores a previously saved configuration.
• CSV export options include nodal displacements and per-member stress distributions.

3. Numerical Formulation

3.1 Degrees of Freedom

Each node has two translational DOFs (ux, uy). Rotational DOFs are not included because the formulation is for classical pin-jointed trusses. Elements carry only axial force; bending behavior appears only in the self-weight superposition used for stress visualization.

3.2 Element Stiffness

For each bar element, the standard 2D truss stiffness matrix is assembled:
[k] = (EA/L) * [[c^2, c s, -c^2, -c s], [c s, s^2, -c s, -s^2], ...]
where c and s are the direction cosines, E is Young's modulus, A is area, and L is length.

3.3 Boundary Conditions

Boundary conditions (free, simple/pinned, roller-X, roller-Y, fixed) are applied by constraining the relevant DOFs. Constrained DOFs are removed from the reduced stiffness system (Kff).

3.4 Loads

The solver includes:
• User-defined nodal Fx, Fy loads.
• Translational nodal springs Kx, Ky.
• Nodal weights Wn, converted to mass for modal analysis.
• Member self-weight: w = wd × A. Equivalent nodal forces (downward) are applied as -wL/2 at each end node.
Distributed q-loads are displayed but explicitly deactivated in the stiffness assembly.

3.5 Displacement Solution

The reduced system Kff * Uf = Ff is solved using numeric.js. Reactions are computed by R = KU − F. Primary nodal displacements are then mapped back into node order and displayed.

3.6 Stress Computation

Axial stress: σ_axial = N / A.
For bending from self-weight, a simply supported beam approximation is used:
$y(x) = w\,x\,(L^3 - 2Lx^2 + x^3) / (24\,E\,I)$
Moments: $M(x) = R\,L/2 - w\,x^2 / 2$ (where R = wL/2).

Fiber bending stress: $\sigma_b = M / S$.
Total stresses at left/right fibers: $\sigma L = \sigma_{axial} - \sigma_b$, $\sigma R = \sigma_{axial} + \sigma_b$.

3.7 Modal Analysis

Modal analysis forms the generalized eigenproblem:
$Kff\ \varphi = \omega^2\ Mff\ \varphi$
Mass matrix: lumped, diagonal, with contributions from member self-weight (converted to mass) and optional nodal weights.
Mass scaling is applied via the transformation $D{-}1\ Kff\ D{-}1$ before eigenvalue extraction.
numeric.eig() is used to compute eigenvalues $\lambda = \omega^2$ and eigenvectors.

4. Limitations

- Truss elements carry axial forces only—no bending stiffness.
- Self-weight bending stresses are approximate (simply supported beam assumption).
- Distributed q-loads are displayed but not incorporated into the stiffness or load vector.
- Modal analysis requires valid A and wd for all members and at least one support.
- Mechanisms (unstable structures) will result in singular stiffness matrices.

5. References

- Gere & Timoshenko, *Mechanics of Materials*.
- Hibbeler, *Structural Analysis* (Truss stiffness formulation).
- Cook, Malkus, Plesha, *Concepts and Applications of Finite Element Analysis*.
- Chopra, *Dynamics of Structures* (modal analysis fundamentals).
- Numerical library: numeric.js.