## redAnTS 2 - Assemble and Solve Global System

Author: Rajesh Bhaskaran, Cornell University

**Problem Specification** 

- 1. Start-Up & Preliminary Set-Up
- 2. Generate Finite-Element Model
- 3. Specify Inputs
- 4. Assemble and Solve Global System
- 5. Post-Process the Solution

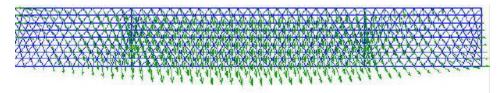
Comments

## Assemble and Solve Global System

Select Static under the Solver menu. This assembles and solves the global matrix. Verify that under Current Settings, the software reports Displ. done. Let's take a look at the nodal displacement values to check that they look plausible.

## **Nodal Displacements**

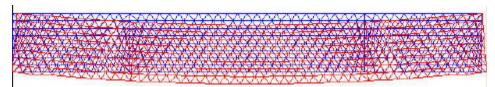
Under Plotting, select Displacement. The nodal displacements are shown below.



Are the displacement boundary conditions at A and B satisfied? The displacement is largest in the middle section of the beam. Is this what you'd expect?

## **Deformed Mesh**

Let's take a peek at how the elements have deformed under the applied vertical loads. Under Plotting, select Deformed mesh. Enter 1000 for the magnification factor and click OK.



Since the nodal displacements look plausible, let's take a peek at stresses and isochromatic fringes. This will occupy us in step 5.

Go to Step 5: Post-Process the Solution

Go to all MATLAB Learning Modules