

# redAnTS 2 - Assemble and Solve Global System

Author: Rajesh Bhaskaran, Cornell University

## Problem Specification

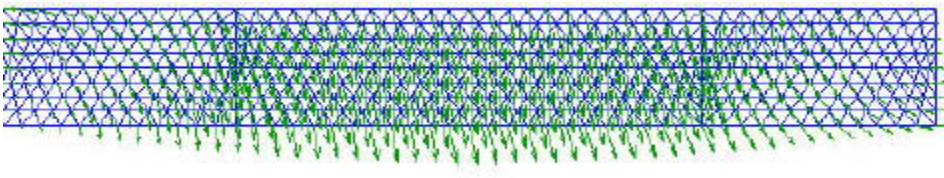
1. Start-Up & Preliminary Set-Up
  2. Generate Finite-Element Model
  3. Specify Inputs
  4. Assemble and Solve Global System
  5. Post-Process the Solution
- [Comments](#)

## 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

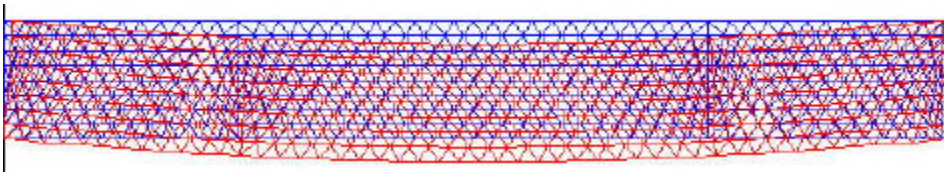
Under **Plotting**, select **Displacement**. The nodal displacements are shown below.



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](#)