

redAnTS 2 - Specify Inputs

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](#)

Specify Inputs

Specify Material Properties

Under **Input Data**, click **Set Properties**.

Set Youngs modulus **E**: $73e9$

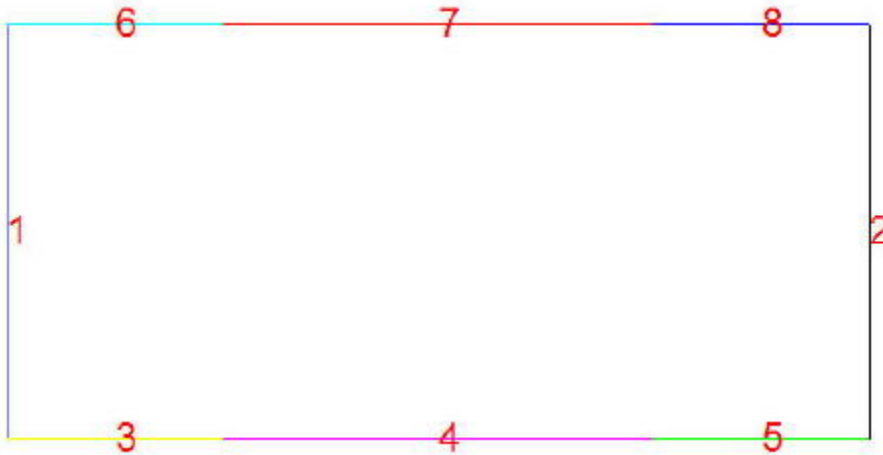
The default **Poisson ratio** matches our value.

Click **OK**.

Under **Current Settings**, you should now see **Properties set**.

Specify Surface Boundary Conditions

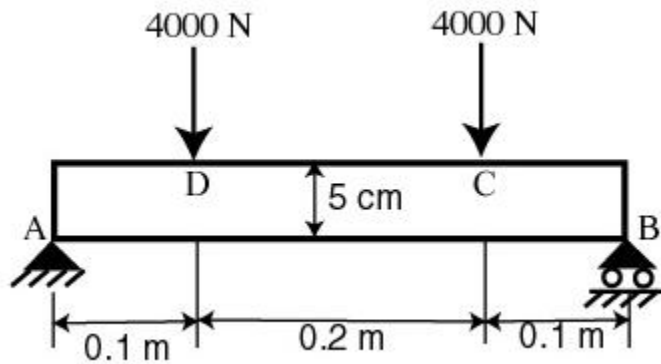
Under **Input Data**, click **Set Surface BC's**. This displays the edges for which we have to specify surface BC's and brings up the dialog box for setting the BC's.



The normal and tangential tractions are zero for all these boundaries. This is the default in *redAnTS*. For each of the eight boundaries, click **OK** to accept the default surface BC's.

Specify Point Boundary Conditions

We need to specify point BC's at *A*, *B*, *C* and *D*.



Specify BC at point A: Under **Input Data**, click **Set Point BC's**.

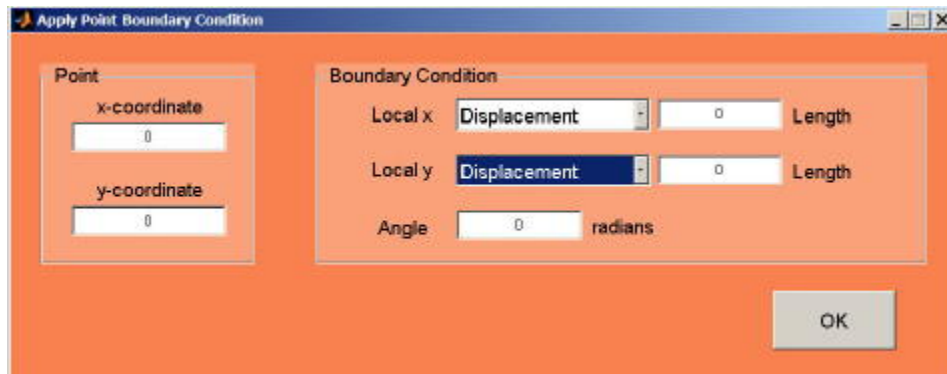
x-coordinate: 0

y-coordinate: 0

Local x: Select **Displacement** and retain 0 for value. This constrains the node at A in the x-direction

Local y: Select **Displacement** and retain 0 for value. This constrains the node at A in the y-direction

Click **OK**.



In the display, you should see an "X" at the node at point A; this indicates that the point BC has been set for this node.

Specify BC at point B: Under **Input Data**, click **Set Point BC's**.

x-coordinate: 0.4

y-coordinate: 0

Local x: Leave default values in place. This means node B will be free to move in the x-direction.

Local y: Select **Displacement** and retain 0 for value.

Click **OK**.

In the display, you should see an "X" for the node at B.

Specify BC at point D: Under **Input Data**, click **Set Point BC's**.

x-coordinate: 0.1

y-coordinate: 0.05

Local x: Leave default values in place.

Local y: Set **Force** to -4000.

Click **OK**.

In the display, you should see a vertical line extending downwards from point D. This is actually a downward arrow that doesn't completely fit in the display window. Does this force act exactly at (0.1,0.05) as we had planned in step 2?

Specify BC at point C: Under **Input Data**, click **Set Point BC's**.

x-coordinate: 0.3

y-coordinate: 0.05

Local x: Leave default values in place.

Local y: Set **Force** to -4000.

Click **OK**.

Does this force act exactly at (0.3,0.05)? Check in the display.

Check that the software reports **BC's set** under **Current Settings**.

Specify Body Forces

Click **Set Body Forces** and **OK** to accept the defaults of zero body force components. Under **Current Settings**, you should see the message **Body forces set**.

Specify Plane Stress

Select [Set Options](#) under [Input Data](#) Input 2 for plane stress. There is no temperature change in this problem, so we can use the default of 0. Click [OK](#). Under [Current Settings](#), you should see the message [Plane Stress](#).

Specify Input Data

Let's save the input settings so that, if necessary, we can read them back in. Under [Input Data](#), select [Write Input File](#). Type filename as `beam.in` and click [Save](#). Check that the file *beam.in* has been saved in your working folder. This is a text file. You can use a text editor such as Notepad to modify entries in the input file in order to change the material properties, boundary conditions, etc.

[Go to Step 4: Assemble and Solve Global System](#)

[Go to all MATLAB Learning Modules](#)