redAnTS 2 - Specify Inputs

Author: Rajesh Bhaskaran, Cornell University

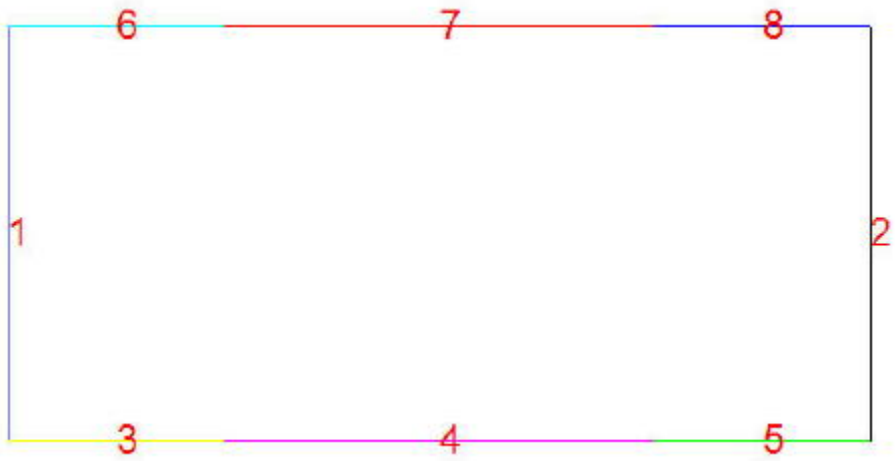
Specify Inputs

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

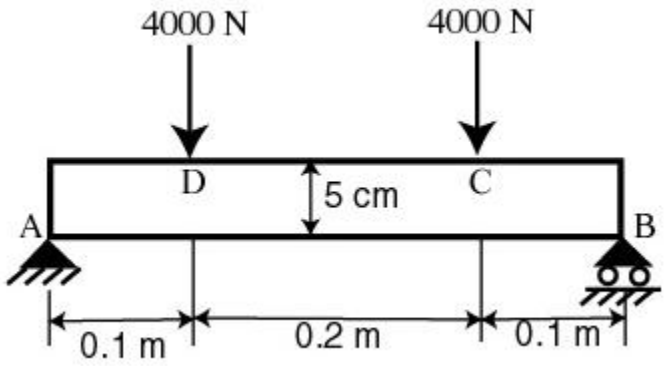
Under Input Data, click Set Surface BC's. This displays the edges for which we have to specify surface BC's and brings up the dialog box for setting the BC's.



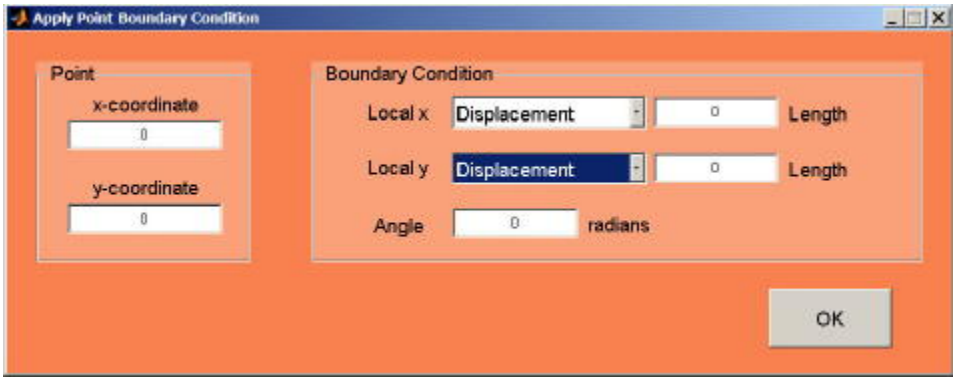
The normal and tangential tractions are zero for all these boundaries. This is the default in *redAnTS*. For each of the eight boundaries, click OK to accept the default surface BC's.

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 *D*: 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