

redAnTS 1 - Physics Setup

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Comments](#)

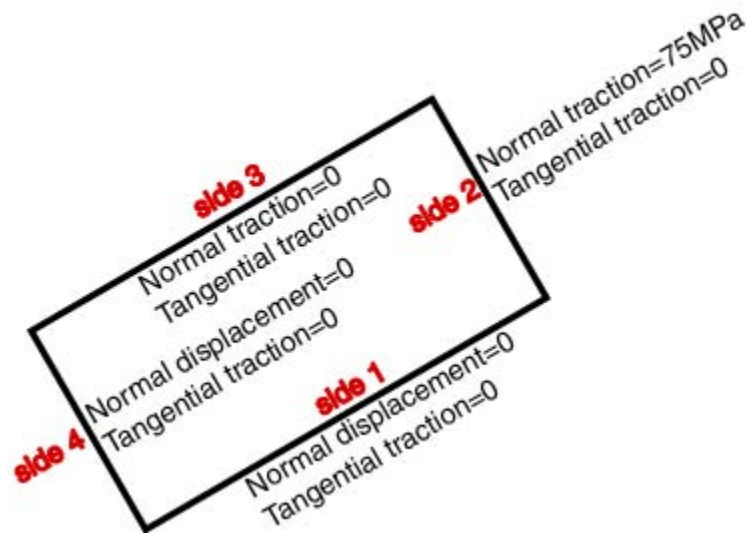
Physics Setup

Specify Material Properties

Under **Input Data**, click **Set Properties**. The default values for the Young's modulus **E** and the **Poisson ratio** match our values. So click OK to accept the default values for material properties. We see under Current Settings the update: **Properties set**.

Specify Surface Boundary Conditions

The surface boundary conditions we'll apply are shown below. Setting the normal displacement for sides 1 and 4 to zero prevents rigid body translations in y' and x' directions, respectively, and arbitrary rotation within the x' - y' plane.



Under **Input Data**, click **Set Surface BC's**. We'll be prompted for BCs for sides 1 through 4.

For side (or edge) 1, select **Essential** under **Normal**. Retain the defaults for the other fields. Verify that this gives the above BC's for side 1. Click **OK**.

For side 2, keep **Natural BCs** under **Normal** and enter $75e6$ as the traction. Verify that this gives the above BC's for side 2. Click **OK**.

For side 3, defaults are valid. Click **OK**.

Side 4 settings are same as side 1: select **Essential** under **Normal**. Retain the defaults for the other fields. Click **OK**.

Check that the software reports **BC's set** under **Current Settings**. This completes the boundary condition specification. Click on the drop-down list under **Input Data**: you'll see that the next item on the list is **Set Point BC's**. Since we don't need to apply any BC's to *points* for this problem, we can skip this item.

Specify Body Forces

The next item on the **Input Data** list is **Set Body Forces**. There are no body forces to be applied. However, *redAnTS* is fastidious and doesn't allow you to skip this item; it requires that you specify a zero body force. 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**... We see that *redAnTS* allows us to solve the problem as plane stress or plane strain. 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**.

Save Input Data

Let's save the input settings so that, if necessary, we can read them back in, make changes and re-run the FEA solution. Under **Input Data**, select **Write Input File**. Type filename as `rect.in` and click **Save**. Check that the file *rect.in* has been saved in your working folder.

Note that there is **Help** available for the **Input Data** menu. Click on **Help** and scan through it to get an idea of the online information that is available. Click **OK** to dismiss the help window.

Let's ask the computer to do something a little more mathematically challenging. So let's move on to [Step 5](#) and prod the beast to assemble and solve the global system.

[Go to Step 5: Numerical Solution](#)

[Go to all MATLAB Learning Modules](#)