redAnTS 1 - Numerical Solution

Author: Rajesh Bhaskaran, Cornell University

Numerical Solution

The only solution option available in *redAnTS* is static deformation. Advanced FEA packages will offer solution options such as modal, buckling, etc. **Select Static** under the **Solver** menu. This assembles and solves the global matrix. Since this is a small problem with less than 100 elements, this takes very little time. Bigger problems will take much longer for the computer to crunch through. Verify that under **Current Settings**, the software reports **Displacements 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.
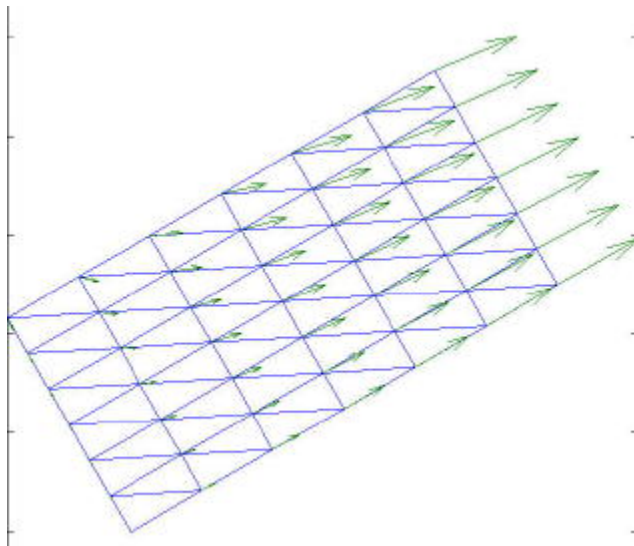

(Click for enlarged image)

Does the displacement field satisfy the imposed boundary conditions shown in this figure? Are the largest displacements where you expect them to be? Are the normal displacements zero for the sides for which this condition is imposed?

Deformed Mesh

Let's take a peek at how the elements have deformed under the applied uniaxial tension. Under **Plotting**, select **Deformed mesh**. Since the deformations are usually very small, the program asks for a maginification factor to make them more visible in the plot. Enter `500` for the magnification factor and click **OK**. You can move the legend out of the way by dragging it with the mouse.

Original and deformed meshes
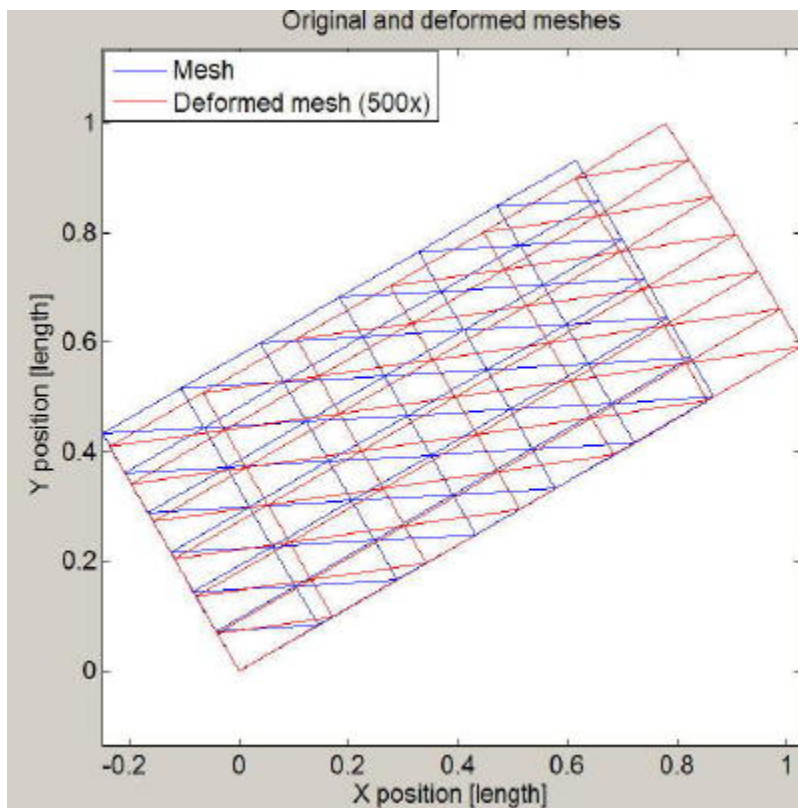
We see that elements have been stretched in the *x'* direction which is the direction along which the tension has been applied. The elements have also shrunk in the *y'* direction. This behavior is as expected.

To save a copy of the plot, click on **Plot** in the **Export** menu. Save the plot as `deformed_mesh.fig`. To convert from MATLAB's *fig* format to a more portable format, say, *jpeg*, open *deformed_mesh.fig* in MATLAB as follows: at the MATLAB prompt, type `open('deformed_mesh.fig')`. This will display the plot in a MATLAB window. You should always include a plot legend to tell the reader what the different lines correspond to. Take a few seconds to get frustrated that MATLAB loses the legend (aarrgggh!!bang!!bang!!). Clever engineers that we are, we'll add the legend back manually. Go to the MATLAB command prompt and type in the appropriate legend command: `legend('Mesh','Deformed mesh (500x)')`. You can move the legend around by dragging it. In the figure window, select **File** -> **Save as**. Under **Save as type**, select **JPEG image** and click **Save**. Open the jpeg image from your working folder to verify that it has been created properly.

To get an idea of the information available in the help page for the **Solver** menu, click on **Help** for this menu and scan through it. Click **OK**.

Since the nodal displacements look plausible, let's prod the beast to display various stress and strain components i.e. post-process the results. This will occupy us in step 6. **Note that the student version doesn't come with the post-processing module.** You'll have to wait until you have developed this module before you can go through step 6. When you are finished with your post-processing module, return to step 6. This will also help you validate your module.

Go to Step 6: Numerical Results

Go to all MATLAB Learning Modules