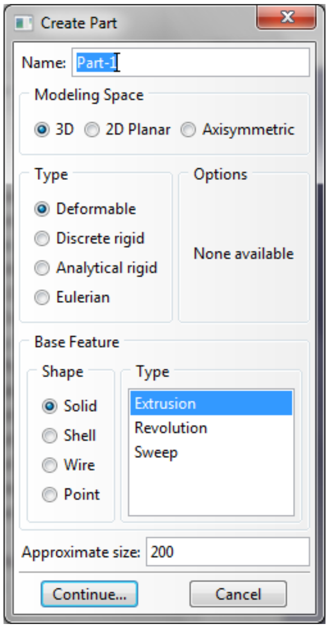
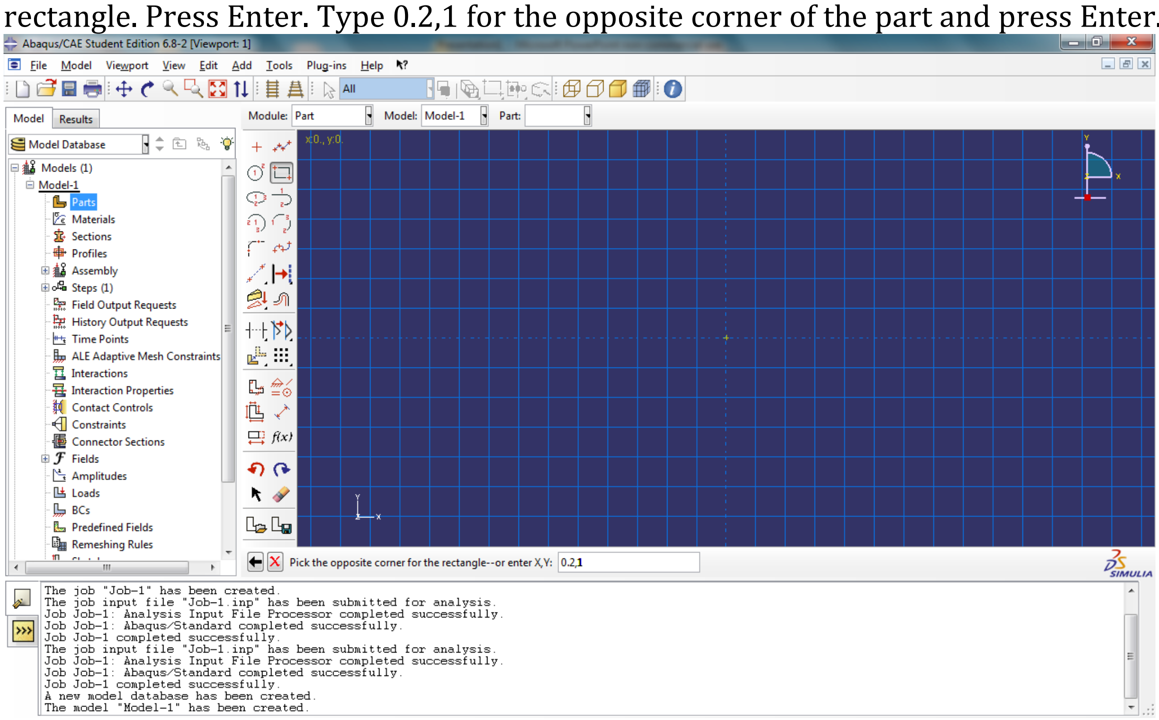
ABAQUS CAE Tutorial – Beam in bending

1. Creating a Part

Double-click on Parts in the model database tree on the left-hand side of the screen. The create part window will appear. We will be performing stress analysis on a 3D deformable solid extrusion of approximate size 1. Click Continue.



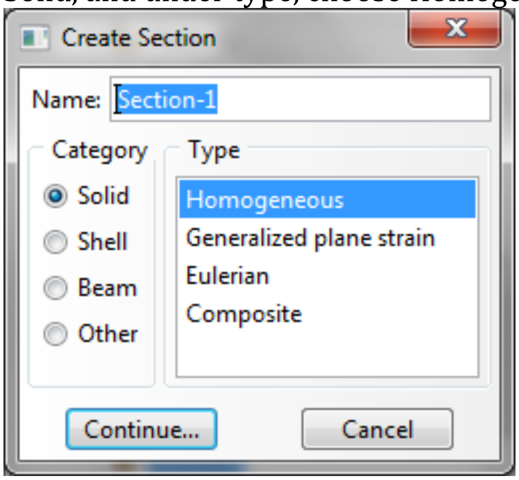
We will make a rectangular extrusion. Click the rectangle in the sketch tools. In the dialog box at the bottom of the window, type 0,0 for the starting corner of the rectangle. Press Enter. Type 0.2,1 for the opposite corner of the part and press Enter.



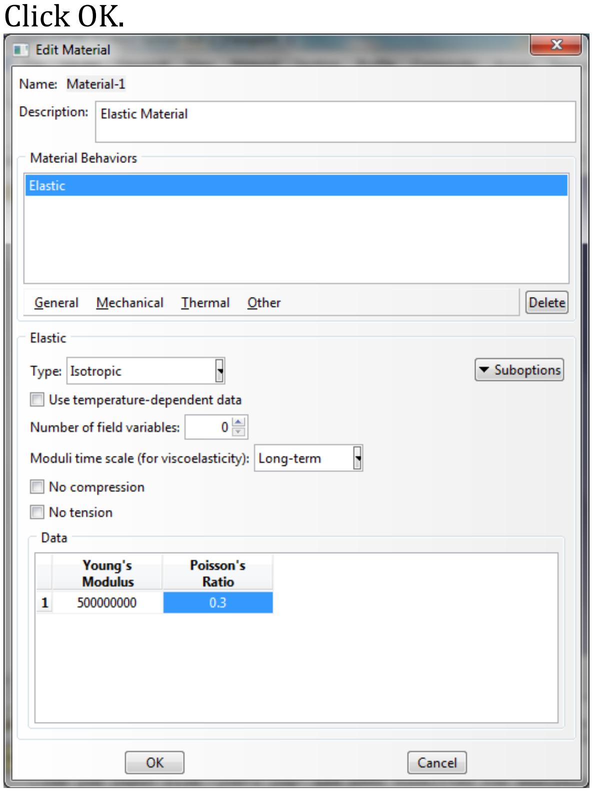
Click on the X button. Fill out the desired depth for the extrusion. In this case we will give the bar a square cross-sectional area so choose 0.2. Press Enter. A rectangular bar should now appear in the ABAQUS CAE window.

In the model database tree, under Assembly, double click on Instances. Select Independent.

Next, in the model database tree, Double-click on Sections. Under category, choose Solid, and under type, choose Homogeneous. Click continue.



In the material assignment window that pops up, click Create. Under the Mechanical tab, select Elasticity, Elastic. Enter values for Young's modulus and Poisson's ratio. Click OK.

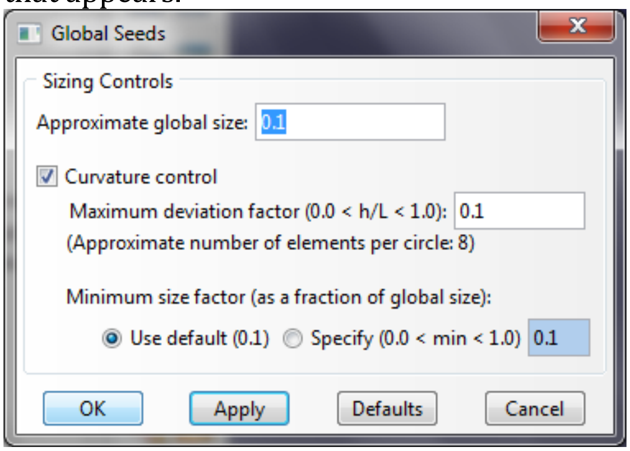


Click OK in the Section window.

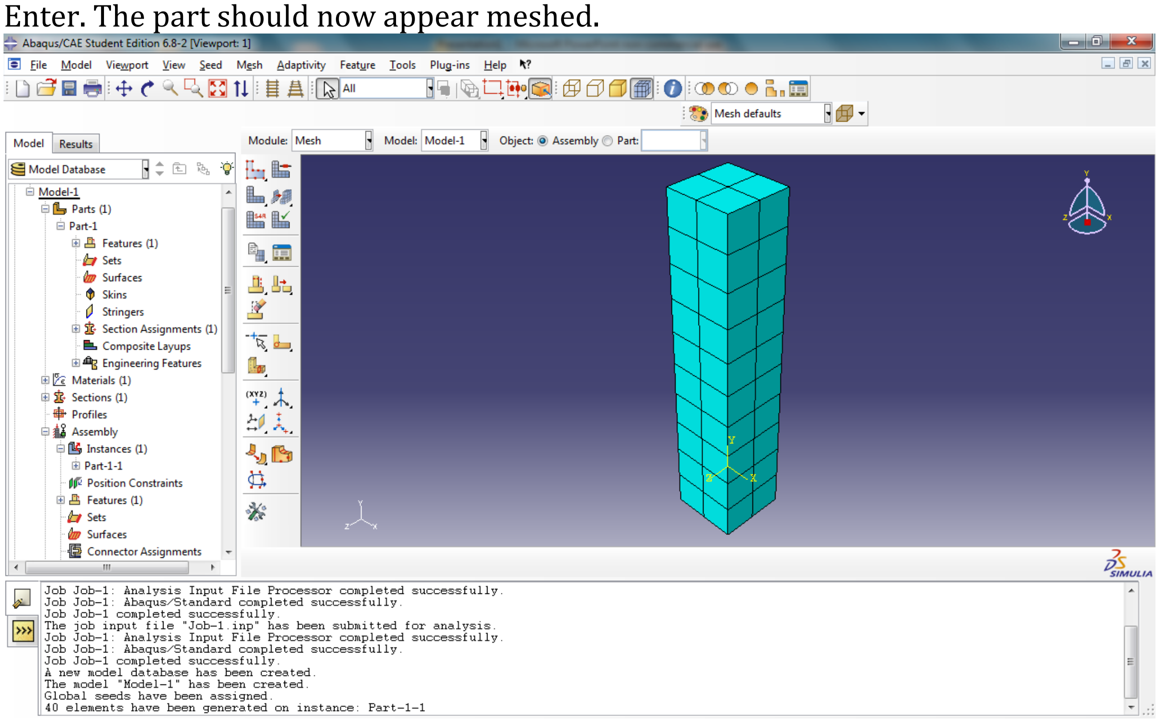
Next under, Parts in the model database tree, expand Part-1 and Double-click on Section Assignment. Click on the part drawing and click Done (or press Enter). Click OK in the Section Assignment window that pops up.

2. Meshing a Part Instance

We will now mesh the part. Under Assembly>Instances>Part-1-1 double click on Mesh. The first step in creating a mesh is to seed the part instance. To do this, click on the Seed Part Instance button on the toolbar to the left of the part drawing. Choose the approximate size for the mesh and click OK in the Global Seeds window that appears.

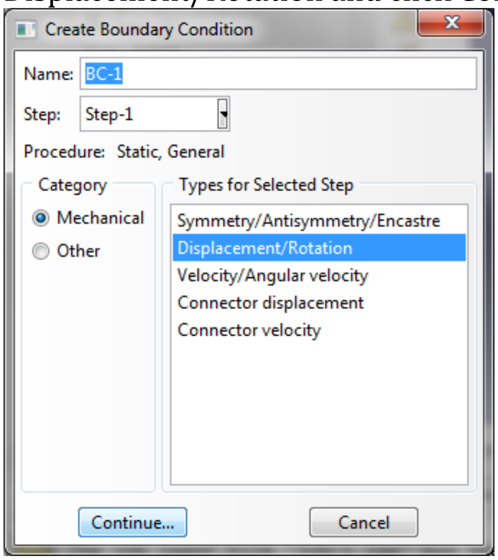


The seed-points for the mesh should appear on the part. Next click the Mesh Part Instance button in the toolbar (directly below Seed Part Instance button). Press Enter. The part should now appear meshed.

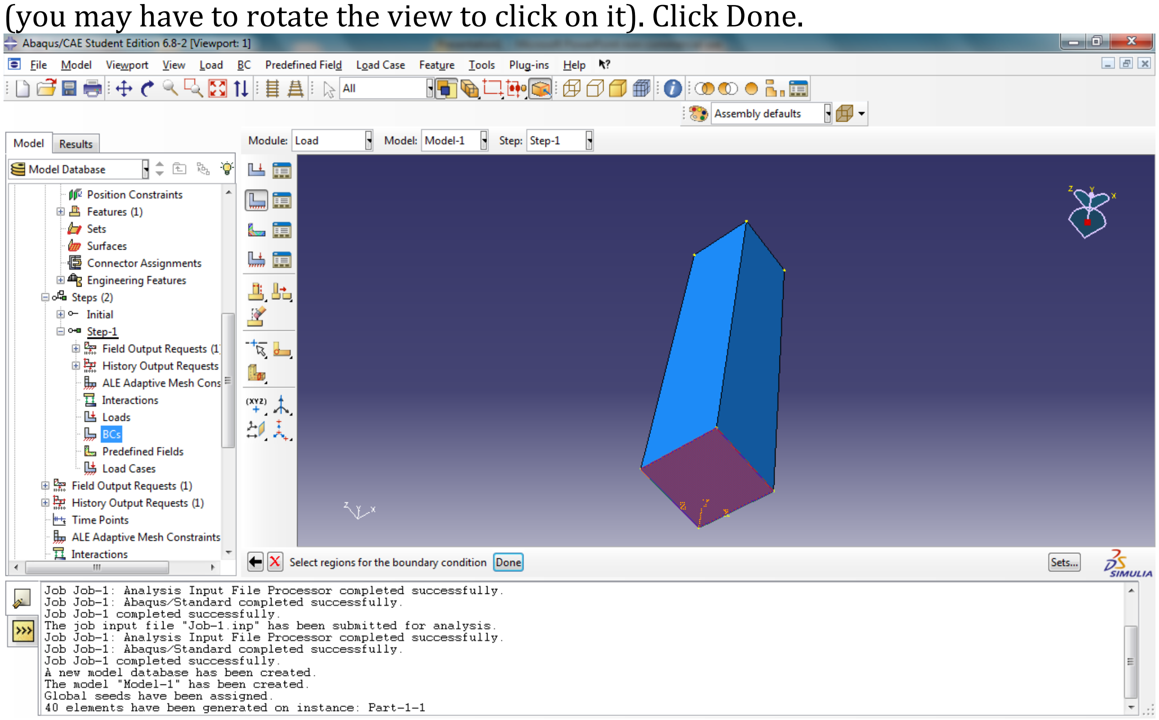


3. Applying Boundary Conditions and Loads

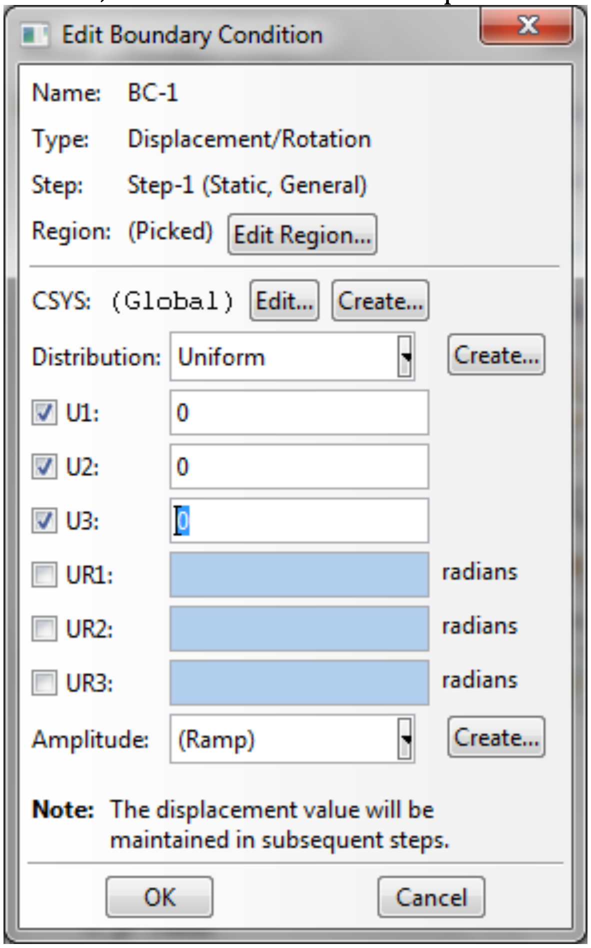
Next we will specify the loading conditions we wish to apply to the bar. In this example, we will analyze the static solution to the bar in bending. First, double-click on Step in the model database tree. We will create a step called Step-1, which will come after the initial step, and will be of type Static, General. With these settings selected, click continue. In the Edit Step dialogue window, enter a description if you choose and click OK. We will be using the default incrementation parameters provided by ABAQUS CAE. Now, under the Steps branch of the model database tree, expand Step-1 and double-click on BCs (Boundary Conditions). For type, select Displacement/Rotation and click Continue.



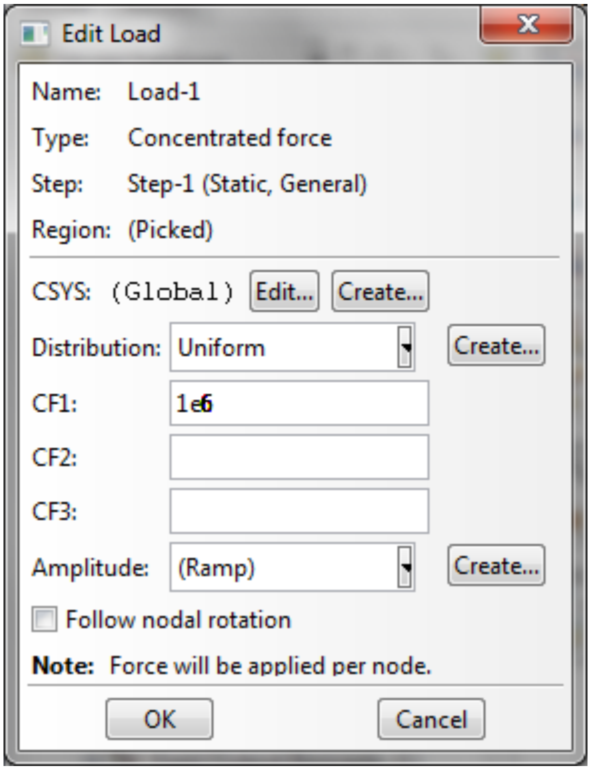
We will fix one end of the bar at y=0. To do this, select the face corresponding to y=0 (you may have to rotate the view to click on it). Click Done.



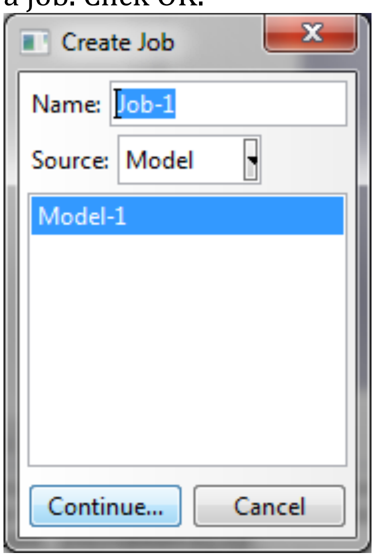
In the Edit Boundary Conditions window that appears, check the boxes for U1,U2 and U3, and enter a value for displacement of 0 for all. Click OK.



Next, we want to apply the natural boundary condition at the other end of the bar (y=L). Again under Steps>Step-1, double-click on Loads. In the Create Loads dialogue box, select Concentrated Load for Types. Select the four nodes at y=L by holding shift while clicking on the nodes and click Done. We will apply a concentrated force of 1e6 force units at each selected point in the x-direction. Under the Edit Load window that appears, enter a magnitude of 1e6 for the CF1 and click OK.



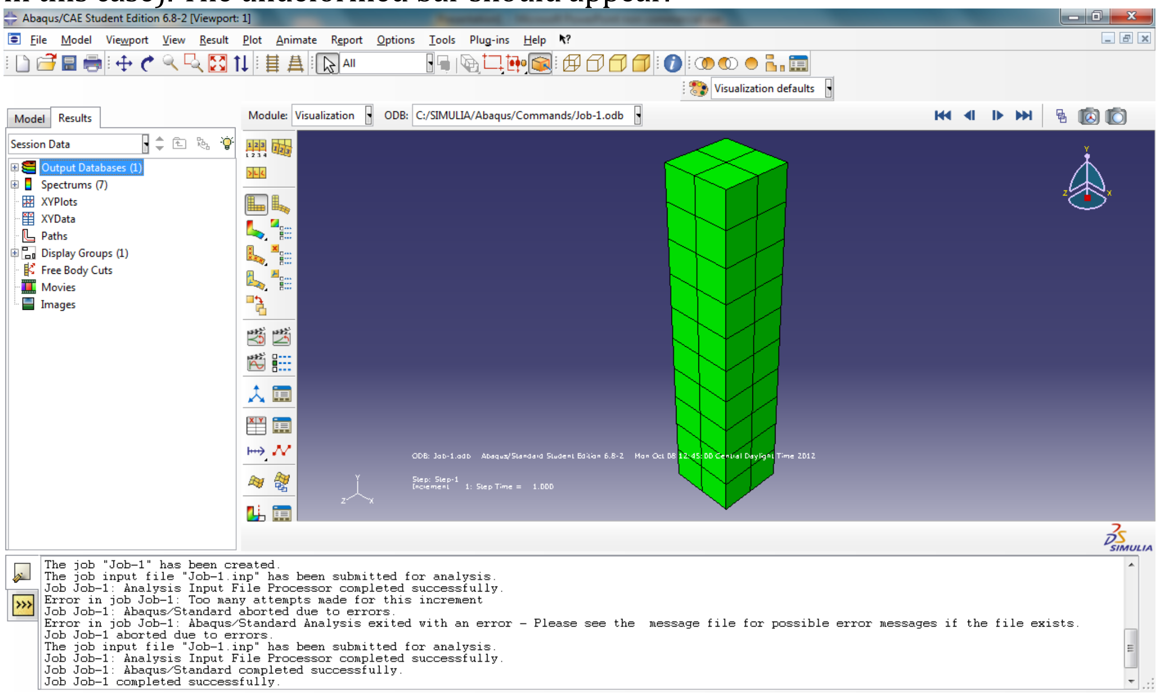
4. Running a Simulation
We have now defined the problem we wish to solve and are ready to run the simulation. Under Analysis in the model database tree, double click on Jobs to create a job. Click OK.



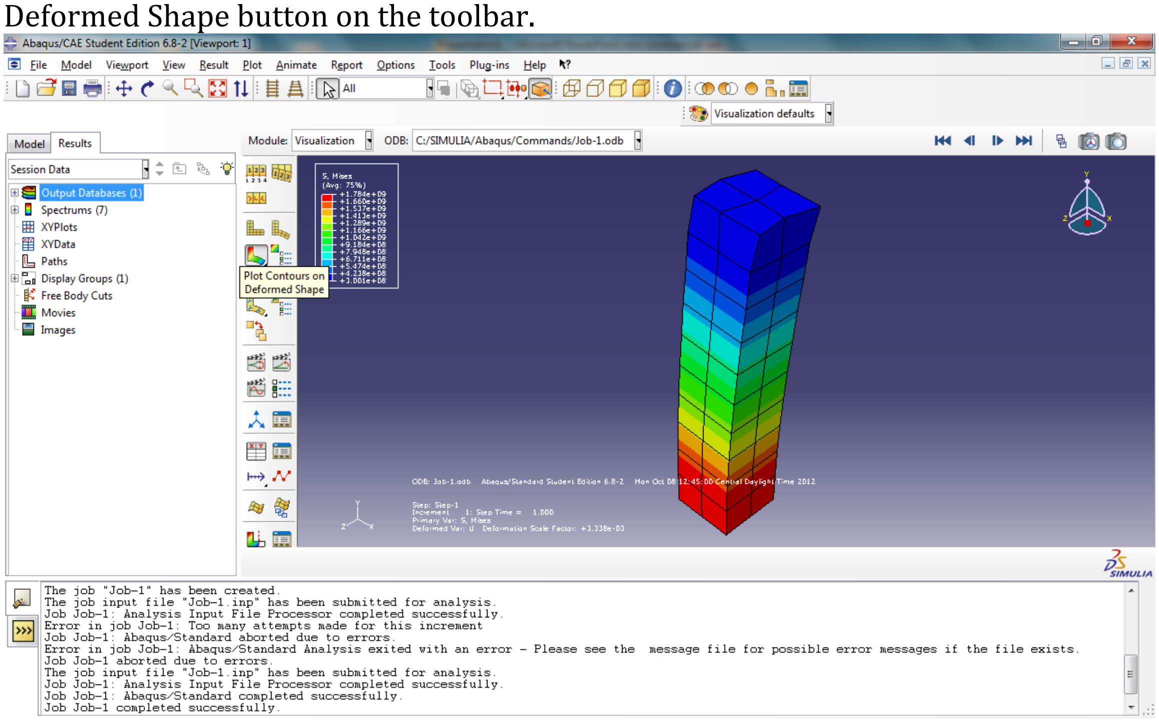
In the Edit Job window, select OK, as we will be using default values for this example. Now under Analysis>Jobs, right-click on Job-1 and select Submit. After a few seconds, "Job Job-1 completed successfully" will appear in the terminal at the bottom of the window. We are now ready to view the results.

5. Viewing Results

In the top-left-hand corner, switch from the Model tab to the Results tab. Double click on Output Databases and open the .odb file corresponding to the job (Job-1.obd in this case). The undeformed bar should appear.



To view the deformed shape with stress contours, click the Plot Contours on Deformed Shape button on the toolbar.



Now say we want to plot the axial component of stress along the bottom face of the bar. At the top of the screen, under the Tools menu, select Path>Create Path. Select

Node List and click Continue. In the window that appears, click the Add Before button under Viewpoint selections. Select the nodes you wish the path to follow. Click Done and OK

Under the Tools menu, select XY Data>Create XY Data and select Path. Click Continue. In the window that appears, click Field Outputs. In the window that appears select “S Stress components at integration points” and under Component, select S22. Click OK. Click Plot in the XY Data from Path window. A plot of the stress at the selected nodes should now appear.