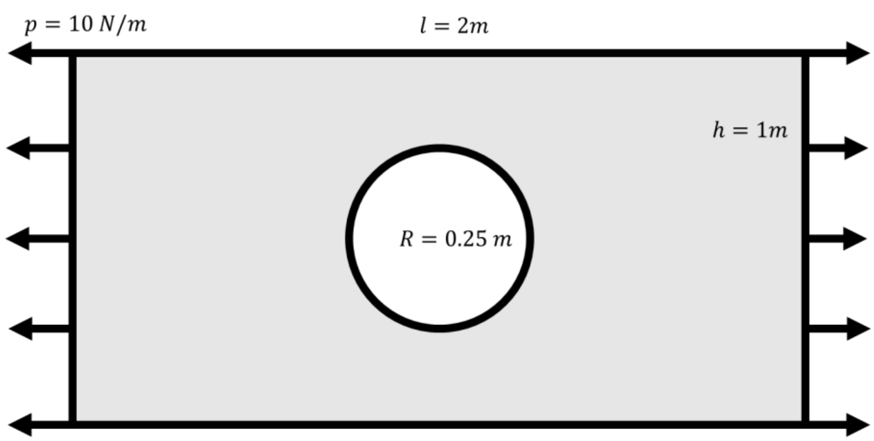


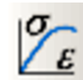
These step-by-step examples are broken up by module. Make sure you have the correct module selected for each part. Any text that is bold, indicates the prompt in the Prompt Area.

Part Module

1) Click Icon, Create Part (*A dialog box will pop up*) 
    a) **Fill out the Create Part dialog**
        i) Modeling Space: 2D Planar
        ii) Approximate Size: 5
        iii) Click Continue…
    b) **Sketch the section for the planar shell**
        i) *Notice that there is now a grid on the screen*
        ii) Click Icon, Create Arc: Center and Two Endpoints 
            (1) **Pick a center point for the arc – or enter X,Y:** Either click at origin or enter 0, 0 in the prompt bar
            (2) **Pick a start point for the arc – or enter X,Y:** 0, 0.25
            (3) **Pick an end point for the arc – or enter X,Y:** 0.25, 0
            (4) Click the Red X, only once! *Notice in the Prompt area that Abaqus is allowing us to perform that same procedure again. But, we don't want to perform that procedure again so we are canceling the procedure. If you click the Red X twice you will also exit out of the Create Part procedure.*
        iii) Click Icon, Create Lines: Connected 
            (1) **Pick a starting point for the line – or enter X,Y:** 0, 0.25
            (2) **Pick an end point for the line – or enter X,Y:** 0, 0.5
            (3) Abaqus automatically repeats the procedure with the previous endpoint the starting point for the next line
            (4) **Pick an end point for the line – or enter X,Y:** 1, 0.5
            (5) **Pick an end point for the line – or enter X,Y:** 1, 0
            (6) **Pick an end point for the line – or enter X,Y**: 0.25, 0
            (7) Click the Red X to cancel the procedure
        iv) Click Done in the prompt area since we are done with our sketch
2) *Our part has now been created. In the model tree you will now see "Parts (1)" indicating that we have created 1 part. If you expand that line you will see the part we created.*
3) *Try rotating part by using Ctrl + Alt + Left Mouse Button*

Property Module

1) Click Icon, Create Material (*An Edit Material dialog box will pop up*)



   a) Name: Aluminum
   b) Click Mechanical > Elasticity > Elastic
   c) Young's Modulus: 70e9
   d) Poisson's Ratio: 0.3
   e) Click OK

2) Click Icon, Create Section

   a) Click Continue…
   b) Click OK

3) Click Icon, Assign Section

   a) Select the regions to be assigned a section
      i) Click on the part
      ii) Click Done
   b) Fill out the Edit Section Assignment dialog
      i) Click OK

4) Notice that the part turned a teal color, that means the part has been assigned a section

Assembly Module

1) Click Icon, Create Instance (a dialog box will appear)

   a) Click OK

Step Module: The step defines the type of analysis you will run

1) Click Icon, Create Step

   a) A "Create Step" dialog box will appear
      i) Ensure "Static, General" is selected
      ii) Click Continue…
   b) A *"Edit Step" dialog box will appear*
      i) Click OK

Skip the Interaction Module (we only have one part so it does not interact with other parts)

Load Module

1) Click Icon: Create Load
   a) Fill out the Create Load dialog
      i) Types for Selected Step: Pressure
      ii) Click Continue...
   b) Select surfaces for the load
      i) Click on the far right edge of the part
      ii) Click Done in prompt area (or click middle/scroll button)
   c) Fill out the Edit Load dialog
      i) Magnitude: 10
      ii) Click OK

2) Notice the Load is pointing in the wrong direction
   a) On the Model tree, expand "Loads (1)" and double-click Load-1 to edit

3) Click Icon: Create Boundary Condition

   a) Fill out the Create Boundary Condition dialog

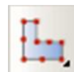
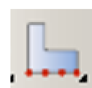
i) Types for Selected Step: Displacement/ Rotation
ii) Click Continue…

b) Select regions for the boundary condition
i) Click the left vertical edge
ii) Click Done in prompt area (or click middle/scroll button)

c) Fill out the Edit Boundary Condition dialog
i) Check U1 box (this mean there will be no displacement in the x-direction)
ii) Click OK

4) Apply the Boundary Condition the Horizontal Edge

Mesh Module

1) Click Icon, Seed Part Instance, an error should pop up 
2) To the right of the Module, next to Object, select Part (instead of Assembly)
3) Click Icon, Seed Part
   a) Set the data using the Global Seeds dialog
      i) Approximate global size: 0.1
      ii) Click OK
   b) Seeding definition complete
      i) Click Done in prompt area (or click middle/scroll button)
4) Click Icon, Mesh Part 
   a) OK to mesh the part? Click Yes
5) Try different meshes
   a) Vary the seed size and also change the seed size for specific edges 
   b) Mesh > Controls to change the element shape
   c) Change element type using element type icon, linear vs quadratic 

Skip the Optimization Module

Job Module

1) Click Icon, Create Job 
   a) Click Continue…
   b) Click OK
2) Click Icon, Job Manager  (Top one)
   a) Click Submit
3) Wait for job to finish running, in the Message area you will see “Job Job-1 completed successfully.”
4) Click Results to transition to Visualization Module

Visualization Module

1) Click Icon, Plot Contours on Deformed Shape 
2) Change size of Legend Text
   a) Viewport > Viewport Annotation Options
   b) Legend tab > Set Font… > Size 14
3) Plot Displacements
4) Find the Values at a Specific Point
   a) Tools > Query… > Element
   b) Select an element to query its attributes