



Autodesk Inventor

- Launching Inventor
 - VLAB
 - Local
- Introduction
- Basic use

Using VLAB



Welcome to VLab



User name

flastXX

Password

.....

Log On

If you have any issues logging into the system please contact the TSC at 1-404-894-7173

To submit a Helpdesk ticket click [here](#)

FAQ [here](#)

Terms & Conditions [here](#).

Log on to mycloud.gatech.edu



COB-Labs

[Details](#)



COE-2018

[Details](#)



Library-2017

[Details](#)



ME-2018

[Details](#)

Select “COE-2019” Desktop and launch Inventor

Installation

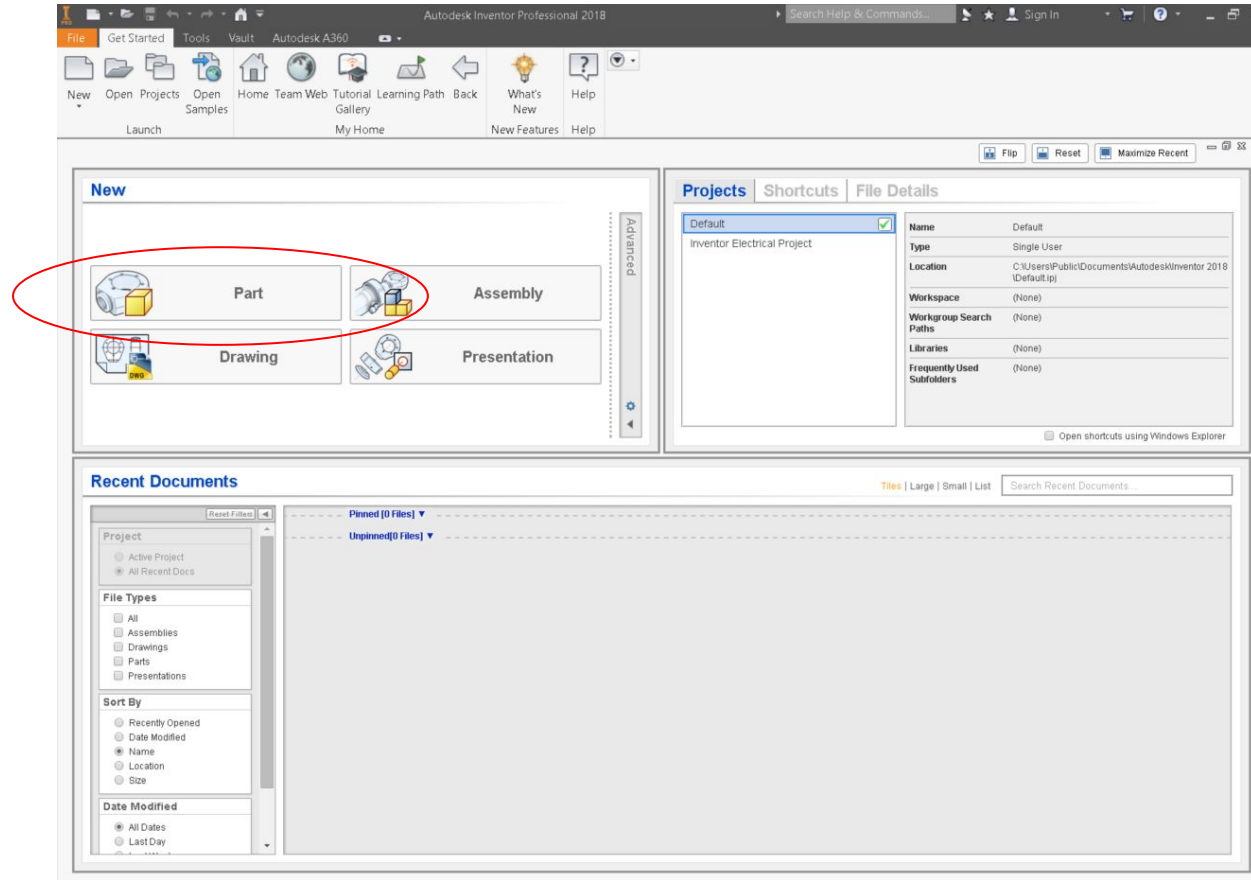
Installing Inventor

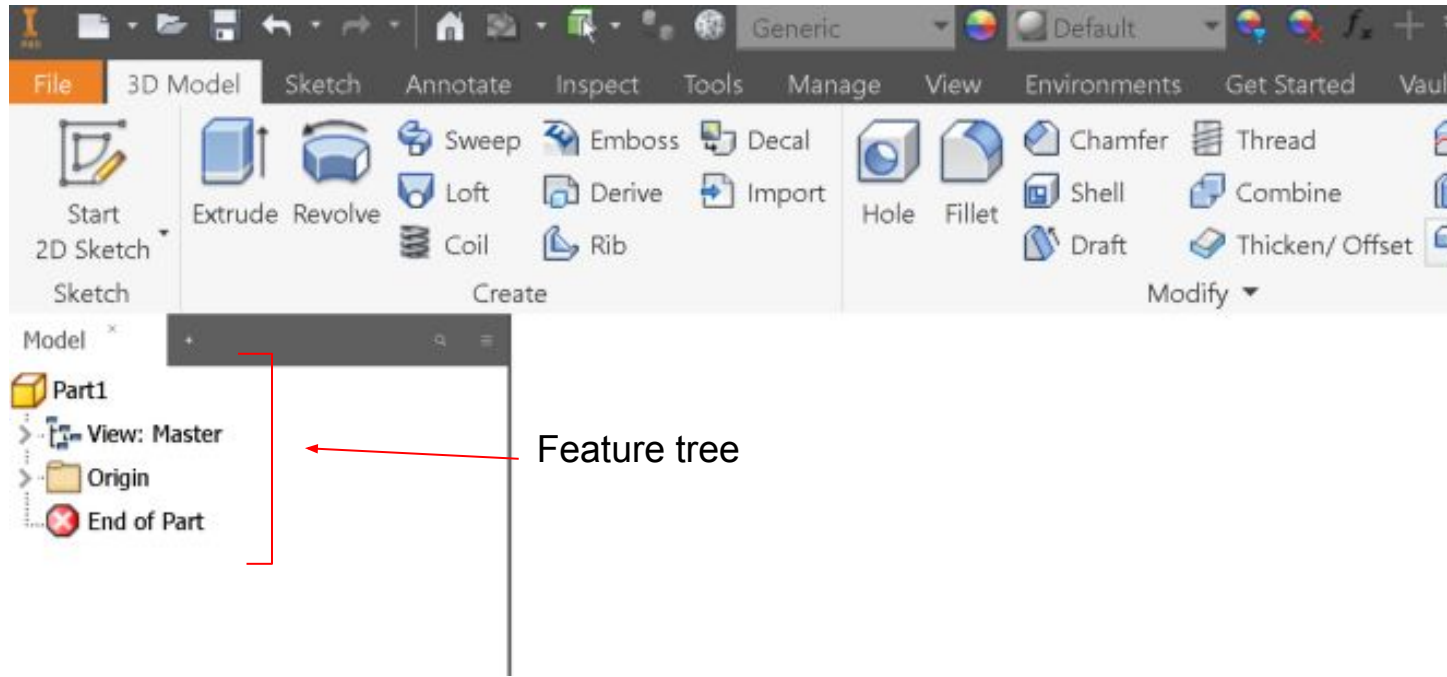
- Visit <https://www.autodesk.com/education/free-software/inventor-professional>
- Create an account (can be your personal email, doesn't have to be .edu)
- Download educational license per your system
 - We'll be working out of Inventor 2019
- Allow several hours to install

For Help: contact #it-helpdesk on RoboJackets Slack

Introduction to Inventor

- Default launch screen
- For now, we'll be working out of the Part environment, which can be opened by clicking Part

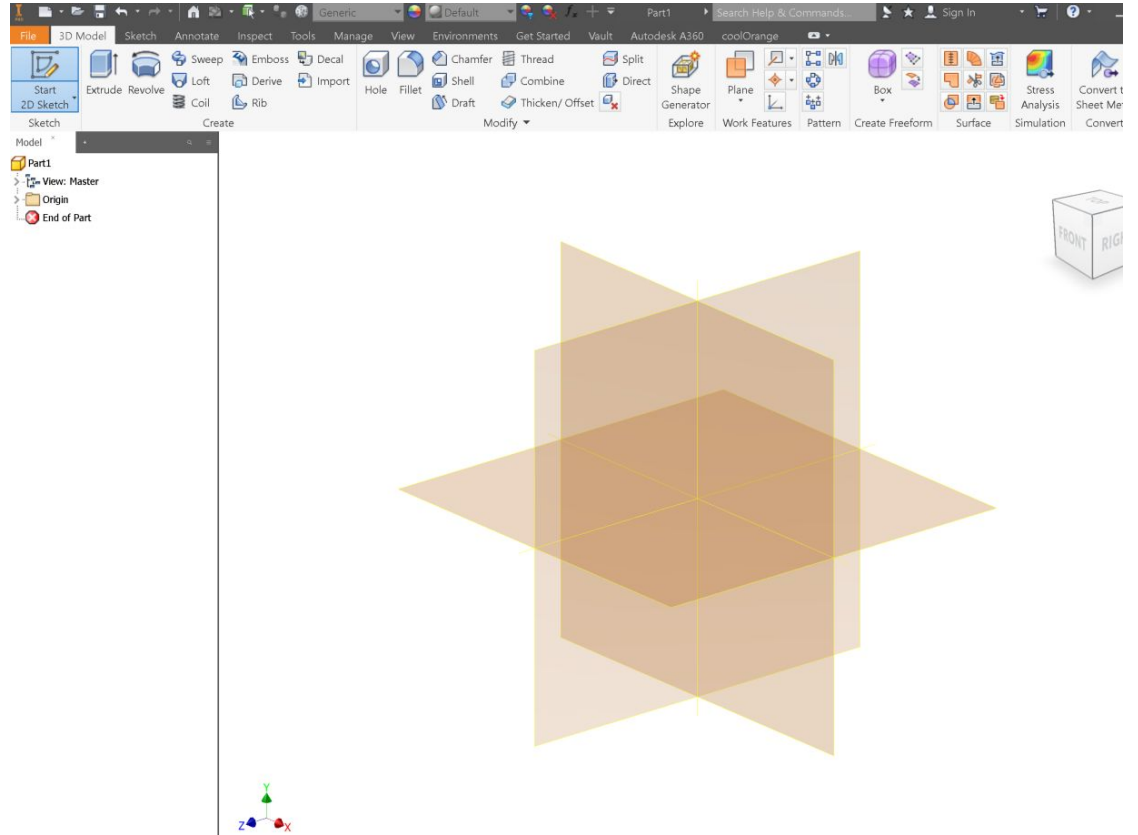


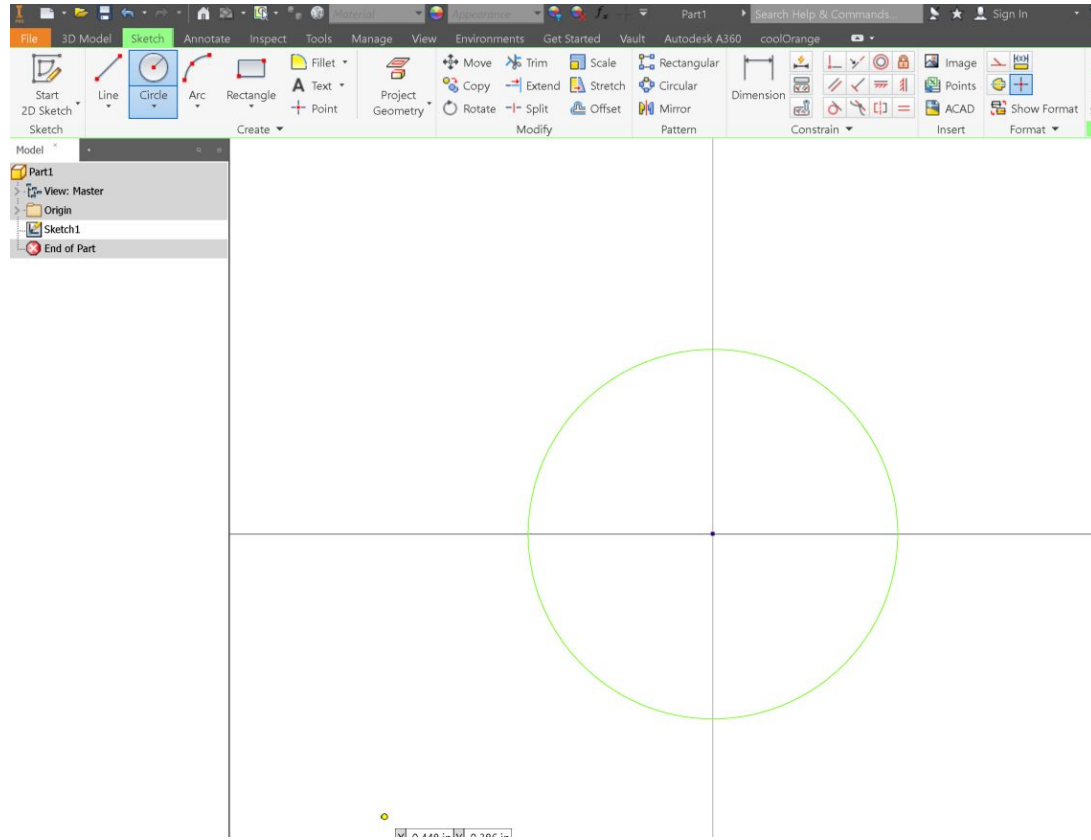


The feature tree displays created objects, allowing easier manipulation and management

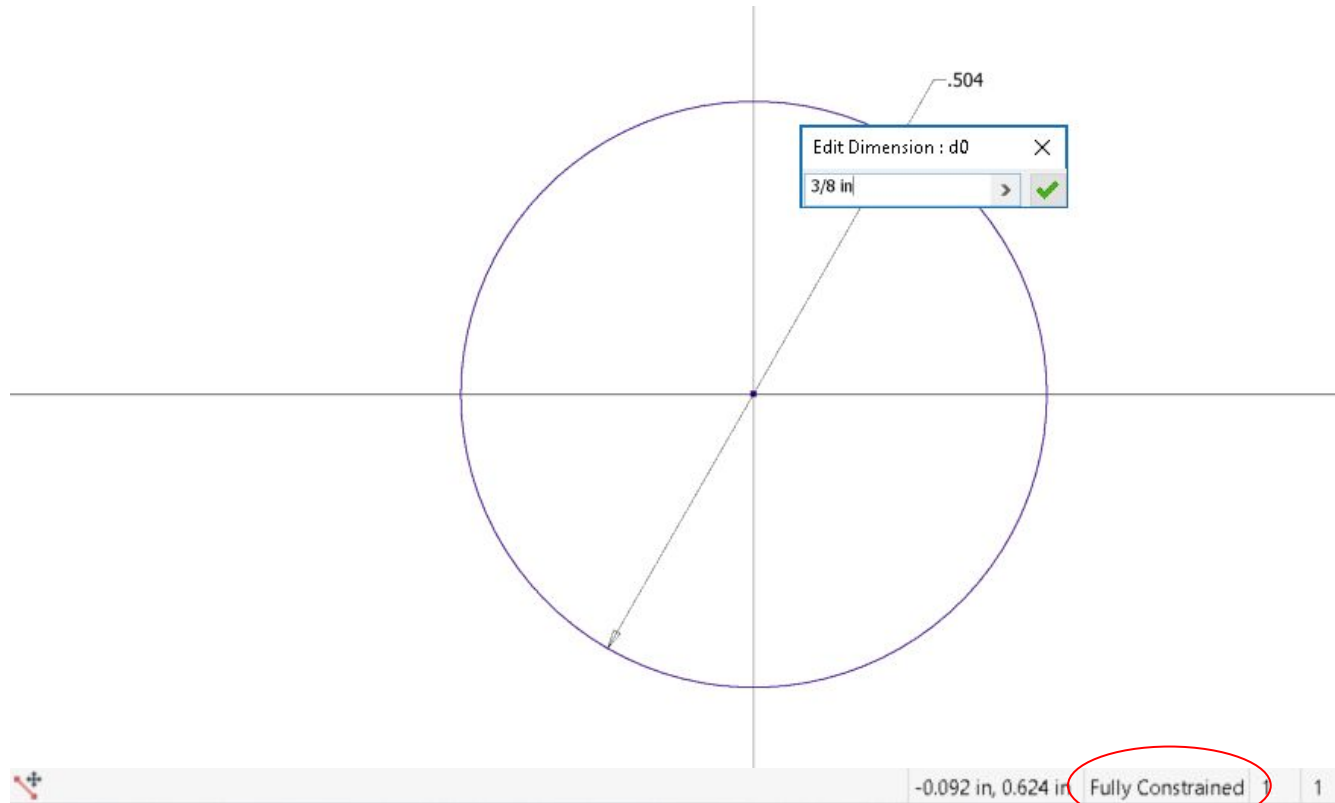
Starting a Sketch

- Select Start 2D Sketch
- Select a plane to begin sketch (XY Plane used in example)





Draw a circle by selecting the circle tool



Select dimension tool and click circle to add dimension.
Set the diameter to $\frac{3}{8}$ in

The circle turns blue when properly dimensioned (fully constrained)

- Inventor must have all information to completely describe an object
 - Note how Fully Constrained is displayed in the status bar (see previous slide)
- Things that are green are not fully constrained
- Do not use “Auto-Constrain” or “Fix”
 - Fix - makes object’s current position permanent
 - Auto-Constrain - makes some assumptions for what it thinks you want and fixes everything else

0.000 in, 0.000 in

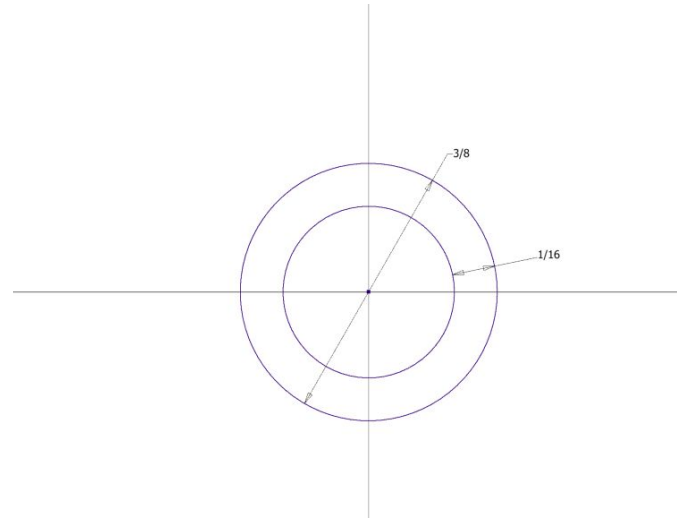
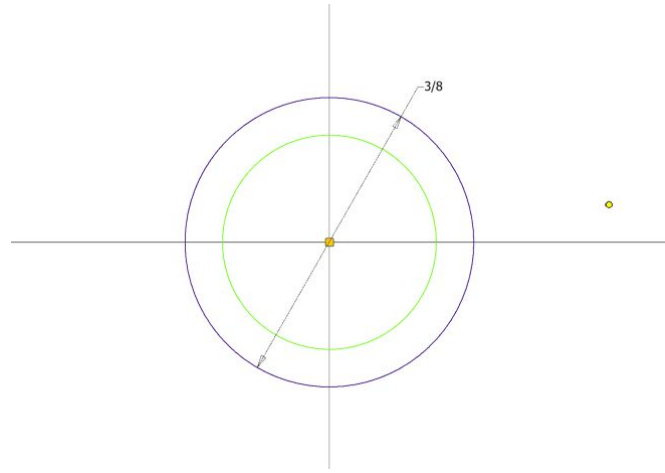
1 dimensions needed

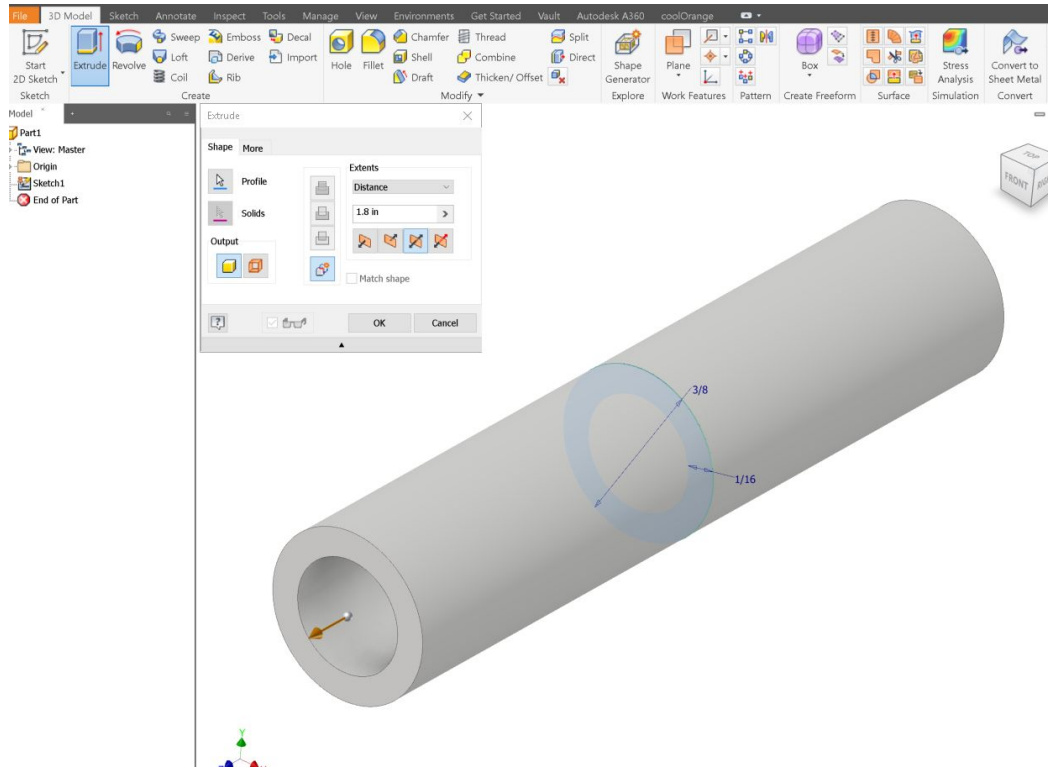
1

56

Create another circle within the previous circle

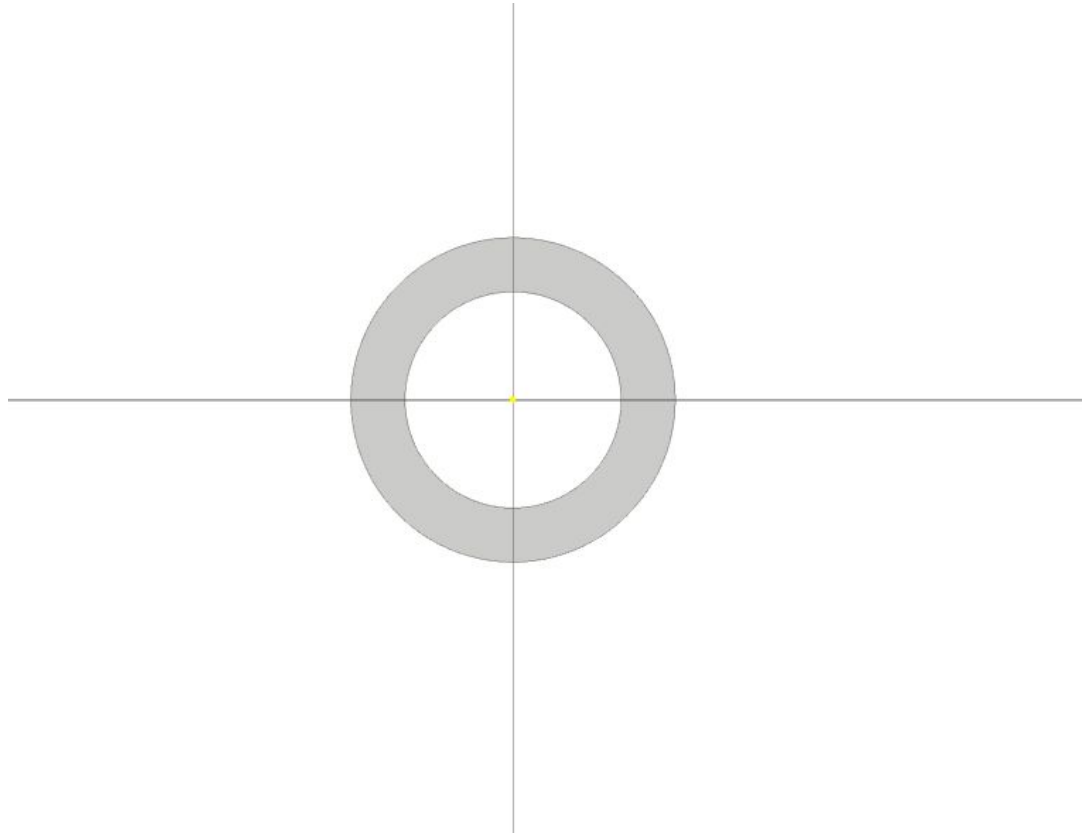
- Dimension this circle as an “offset” by clicking the larger circle followed by the smaller circle
- Enter an offset distance of $1/16$ in





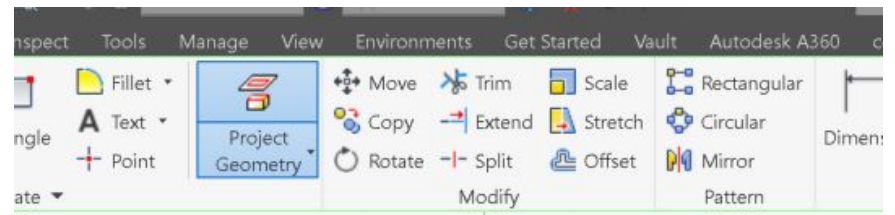
Select *Finish Sketch* and click *Extrude* under the create palette. In the dialog, enter 1.8 in

- Sometimes Inventor will use your last sketch and make a coherent extrusion. A sketch must have an enclosed area for extrusions.



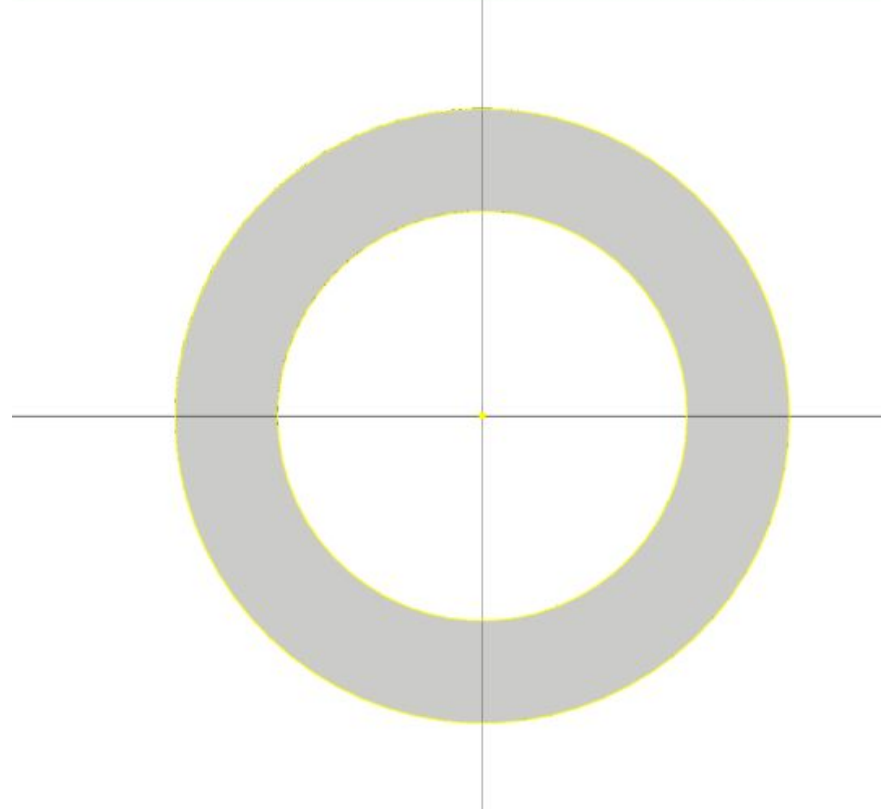
Start a 2D Sketch once again, but this time, click on an end face of the extrusion

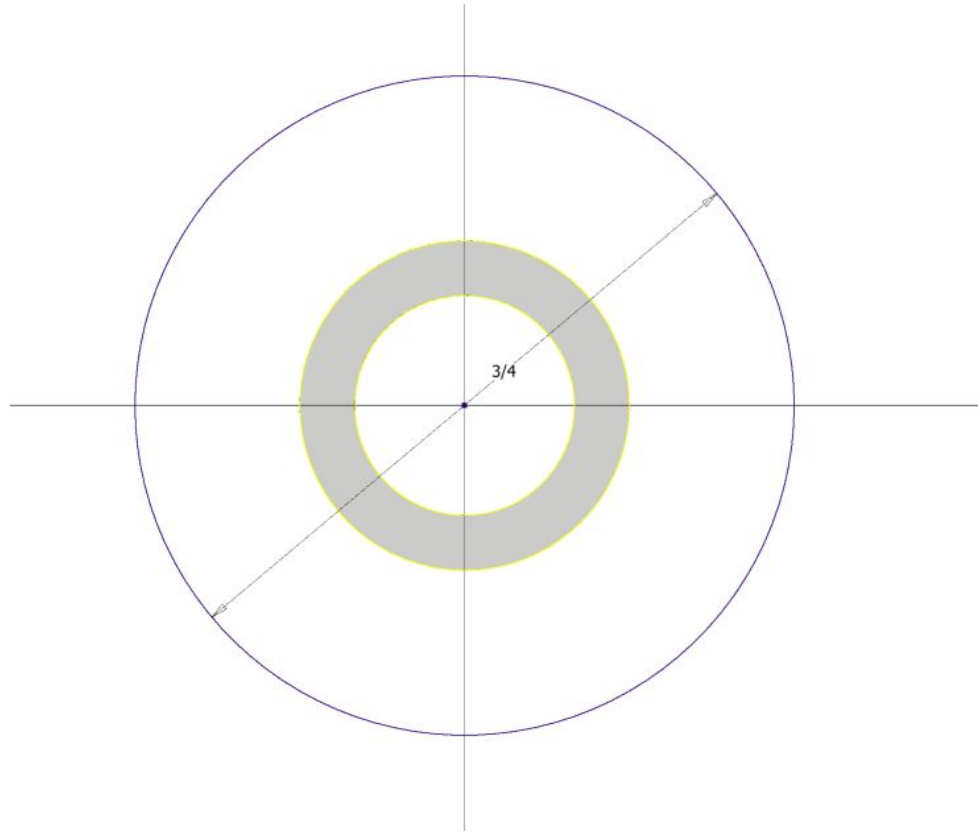
- You can start sketches on any plane, including flat faces on bodies



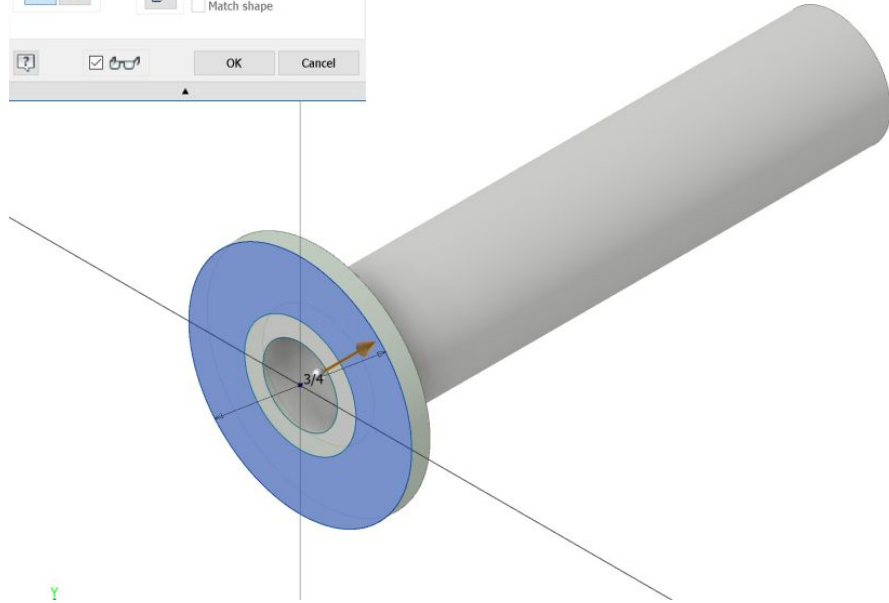
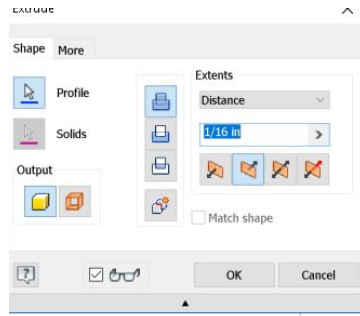
Select project geometry and click the outer edge of the face.

- This allows you to access entities outside the sketch's scope

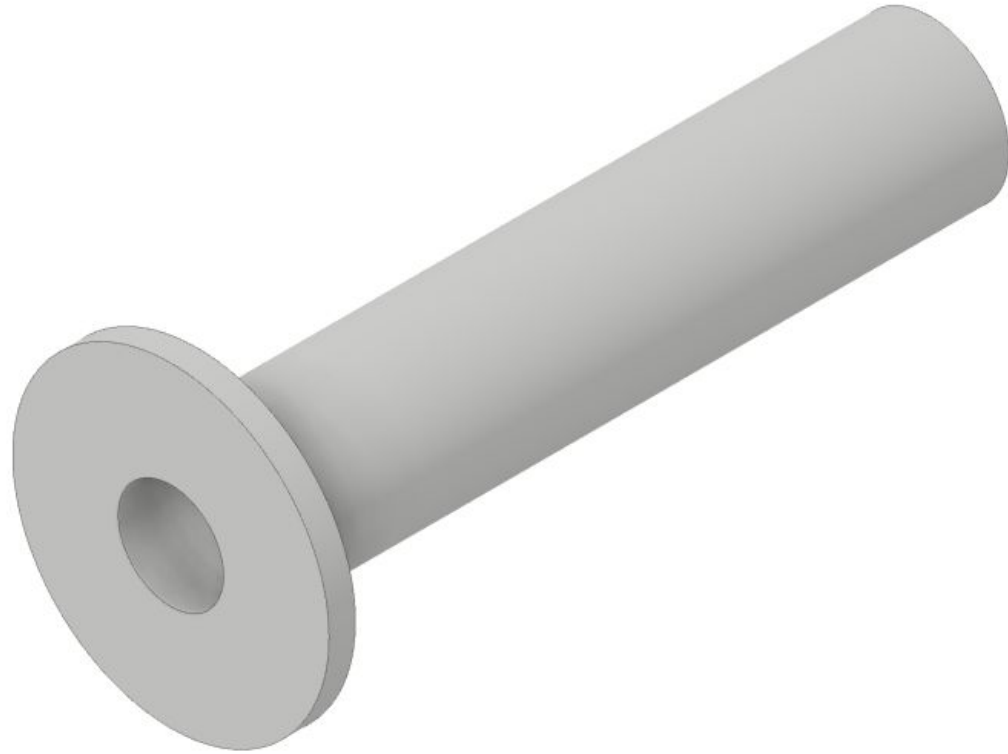




Create a new circle with $\frac{3}{4}$ inch diameter.



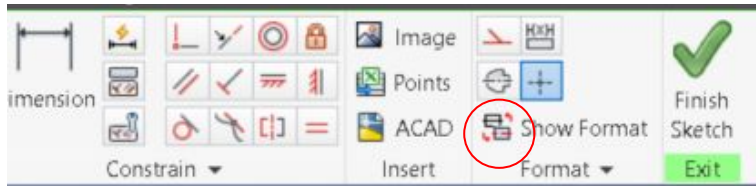
Finish the Sketch and Extrude the outer profile. (Check that your dialog settings match the example)



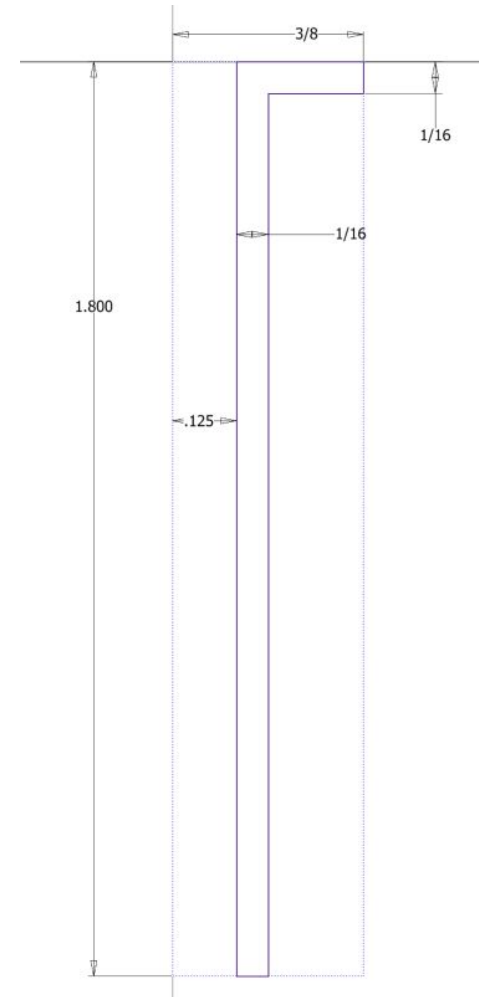
Great job! You've made a simplified rivet. Now, let's make it another way...

Create a new part, and draw a new sketch according to the picture.

- The dashed line is a construction line (these will be elaborated on in a later session)
 - Any sketch entity can be converted into a construction line by clicking the construction button

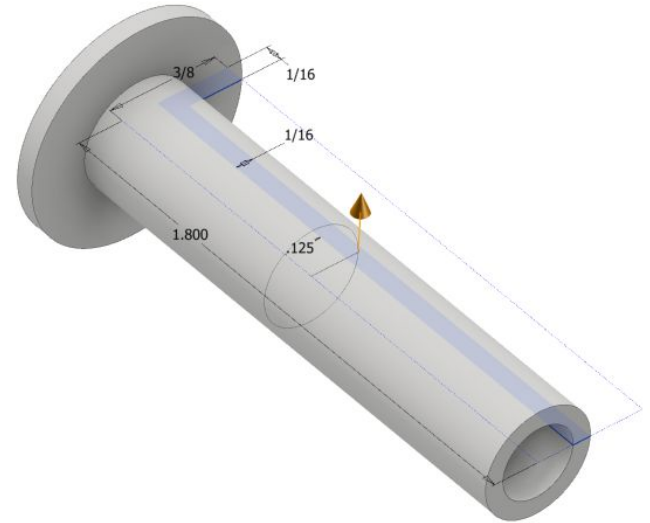
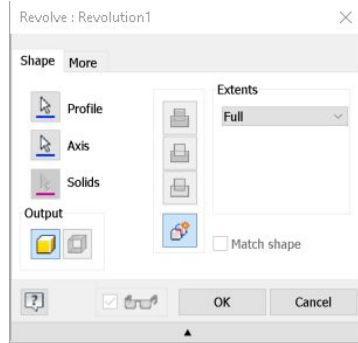


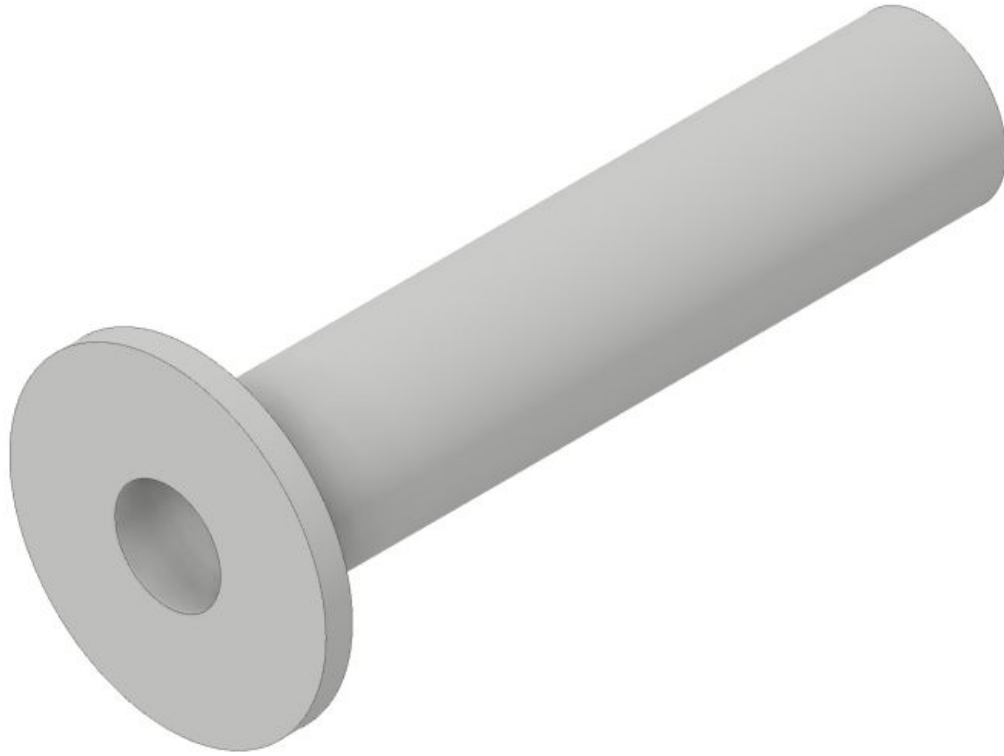
- For this example, draw a 1.8" x .375" rectangle and after selecting the rectangle, click the construction button
- Inventor will automatically snap to horizontal and vertical lines if you let it do so



Finish the sketch, and select *Revolve* (Next to *Extrude*).

- You will be prompted to select a profile to **revolve** around an axis, creating an axially symmetric part
- Select the profile shown in blue in the picture and use the white line as your axis





Surprise! It's the same part, created in two steps.

- This is why it's important to think about your design beforehand