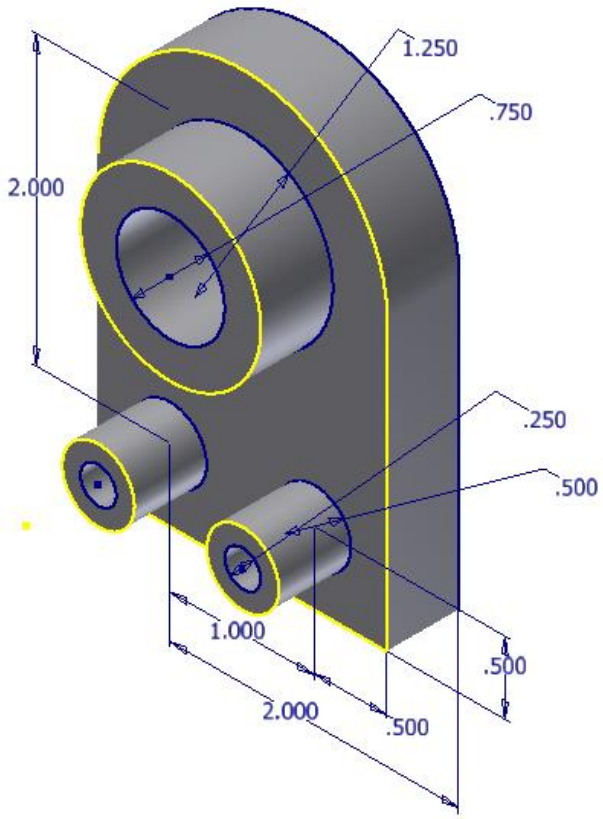
Part Modeling Intro

Using Autodesk Inventor

- Basic Dimensions

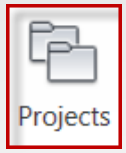


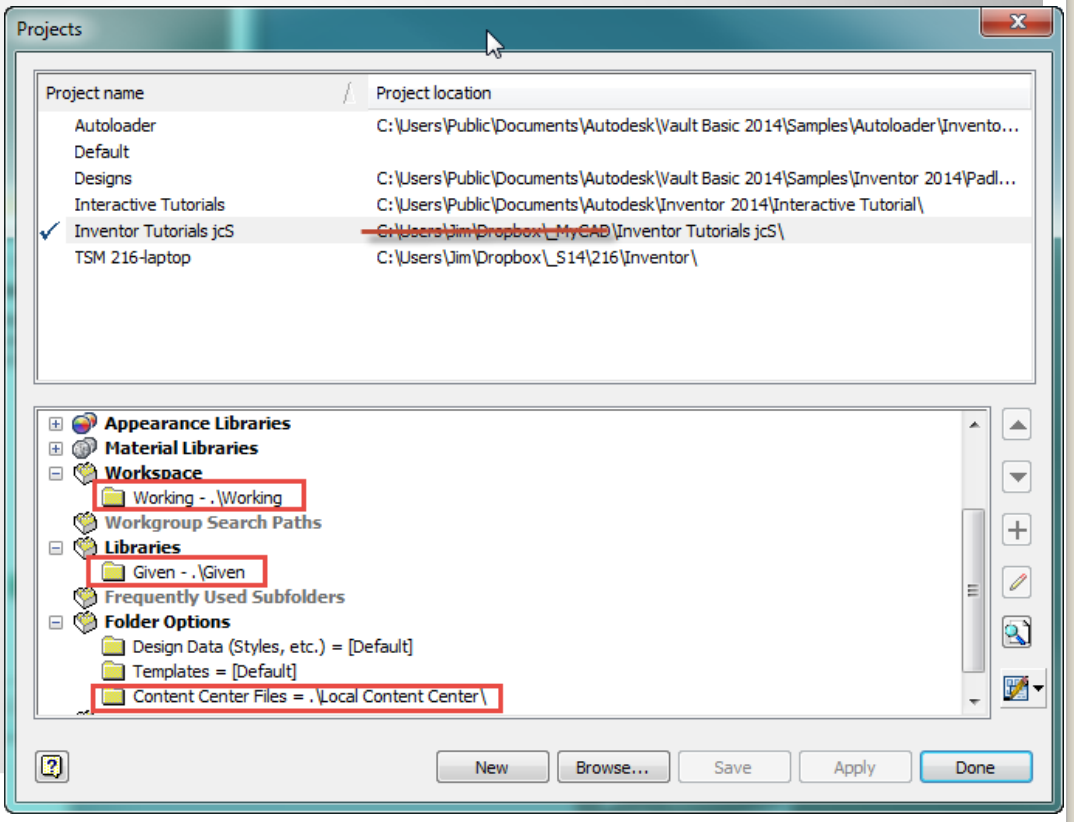
Final Model

- General things to consider
- Follow specific instruction for the class

1.INITIALIZATION

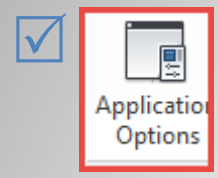
- Start Inventor
- Verify project
  - ☑Selected Project
  - ☑Workspace
  - ☑Libraries
  - ☑Folder Options

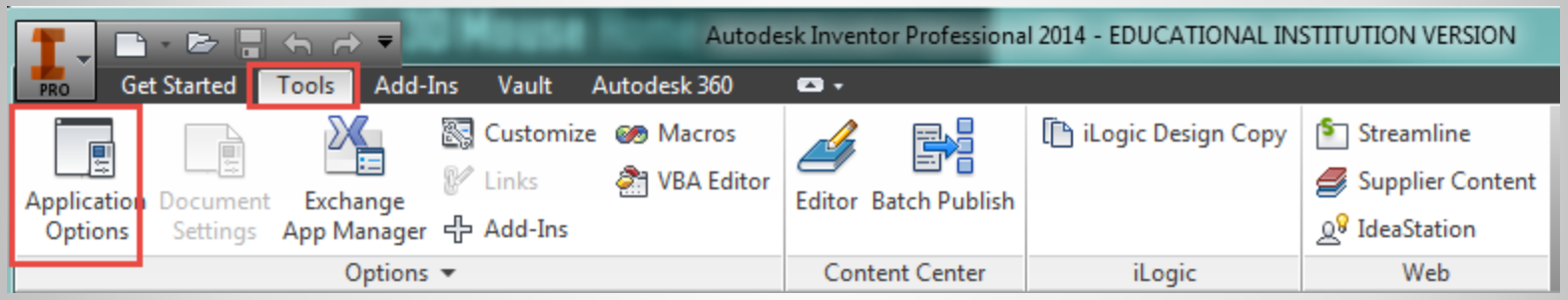




a. Startup

- Tools





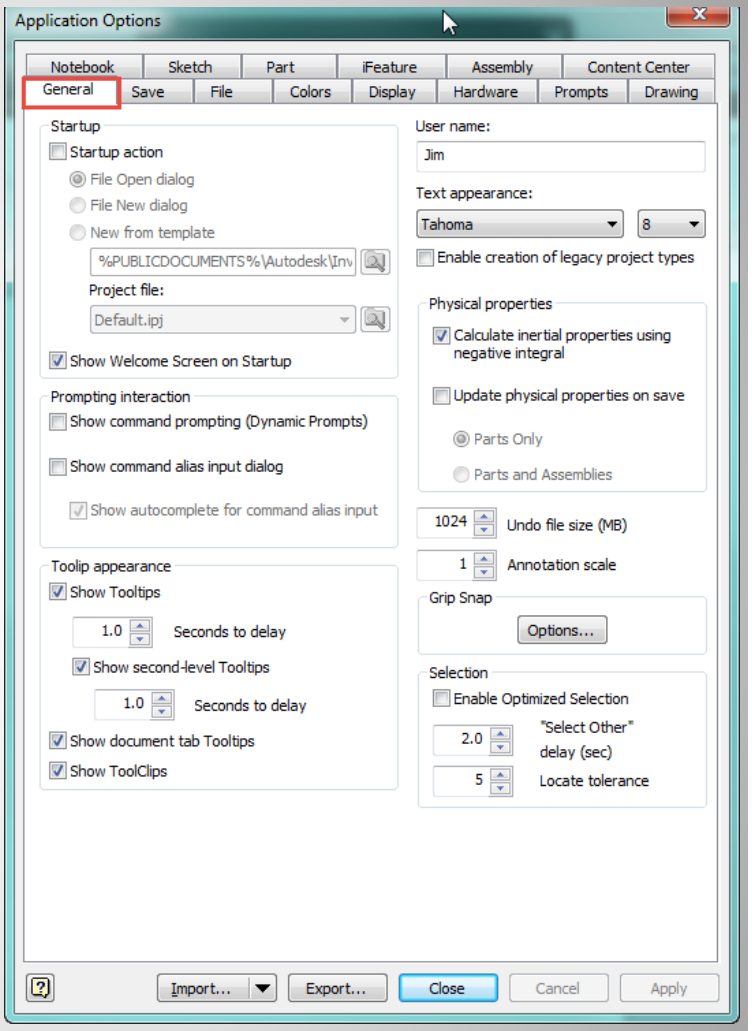
b. Application "Options"

- Application Options:

☑ General

Note:

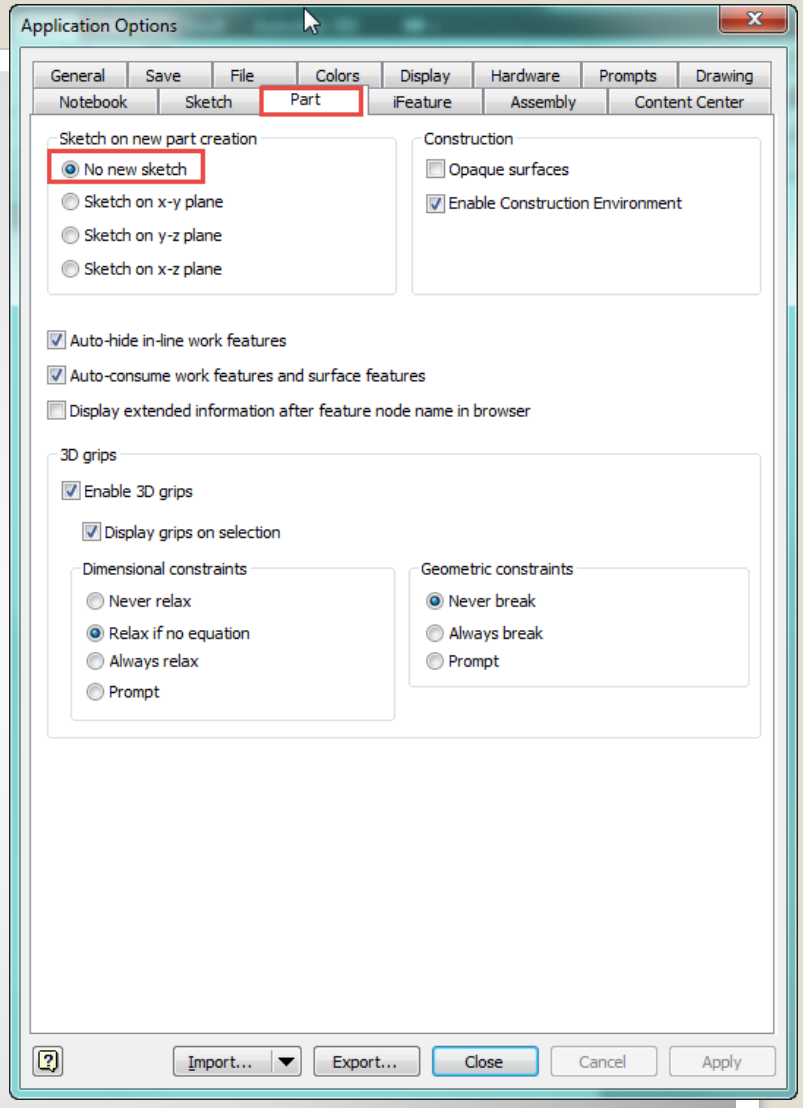
- other possibilities.
- Carefully adjust (only if needed)



c. Options - General

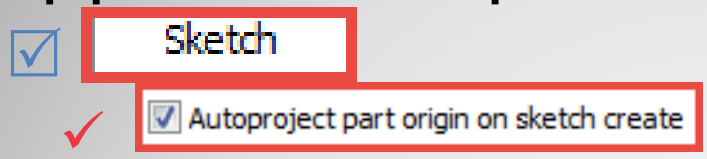
- Application Options:

Note:
- Optional setting
- You may prefer another choice
- Other possibilities

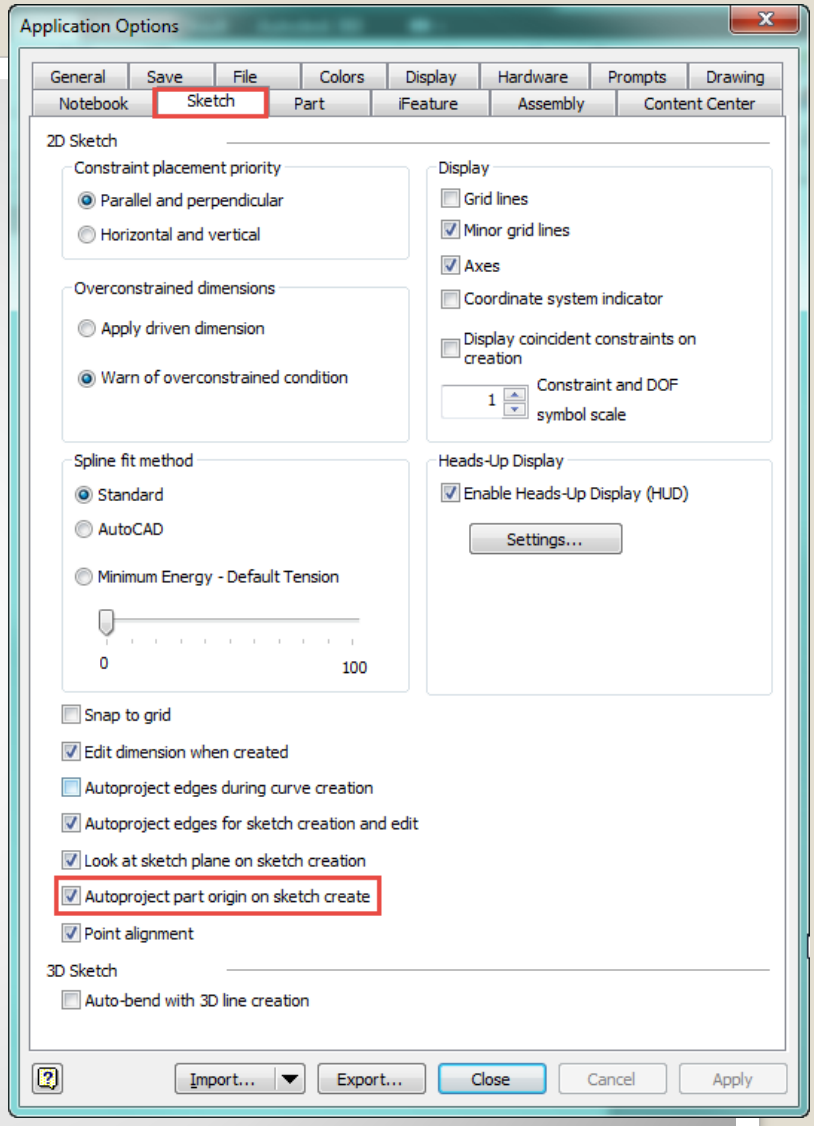


d. Options – Part

- Application Options:



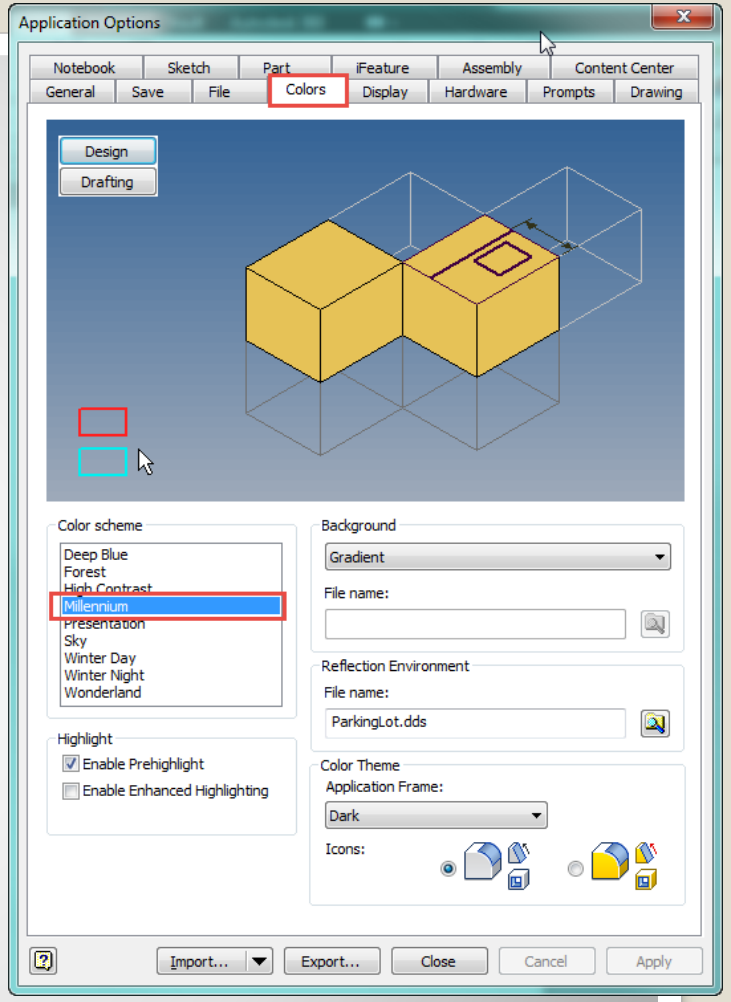
Note:
- other possibilities
- Selectively change
- Remember what you change



e. Options - Sketch

- Application Options:
  - ☑ Colors
    - ✓ Millennium

Note: Other possibilities



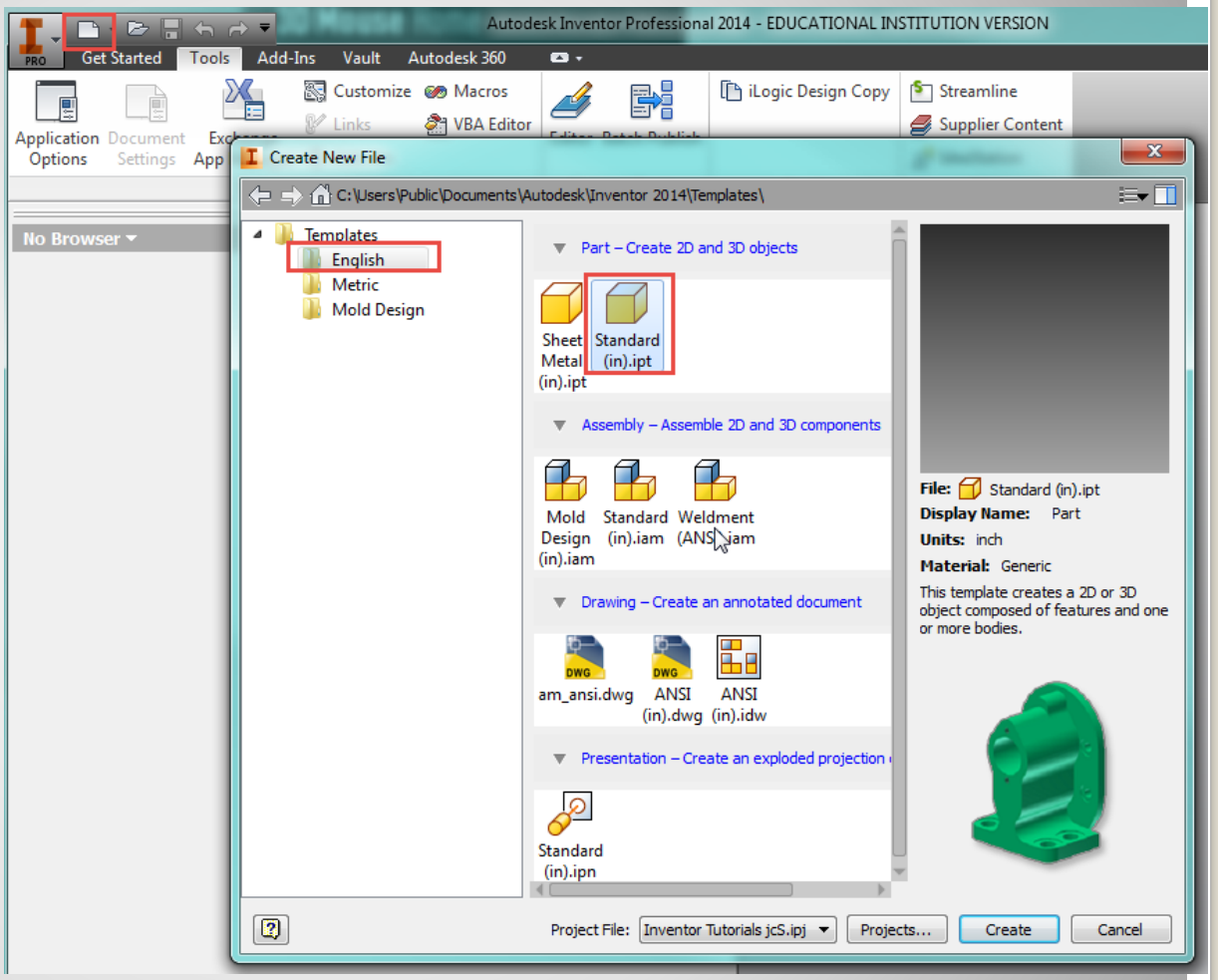
f. Options – Colors

2. NEW PART MODEL

- New 

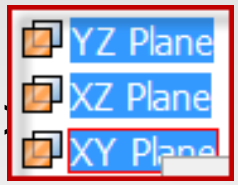
☑ English

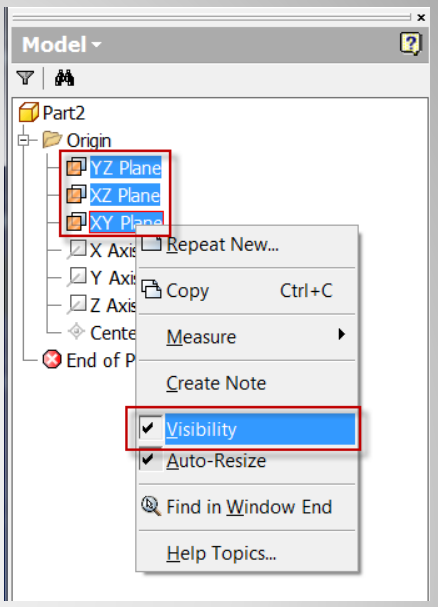
☑ Standard (in).ipt

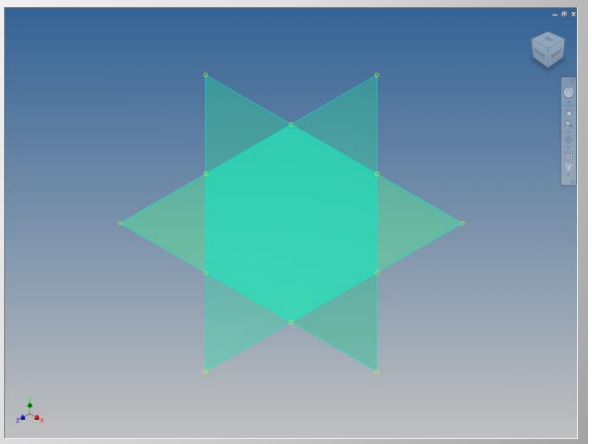


a. New Part Model

- Reference Planes
  - ☑Select: <Ctrl>+<lmb>
  - ☑Visibility
- View
  - ☑Home
  - ☑Other
    - ✓Navigation
    - ✓Pan
    - ✓Zoom
    - ✓Orbit
    - ✓View Face

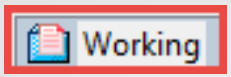




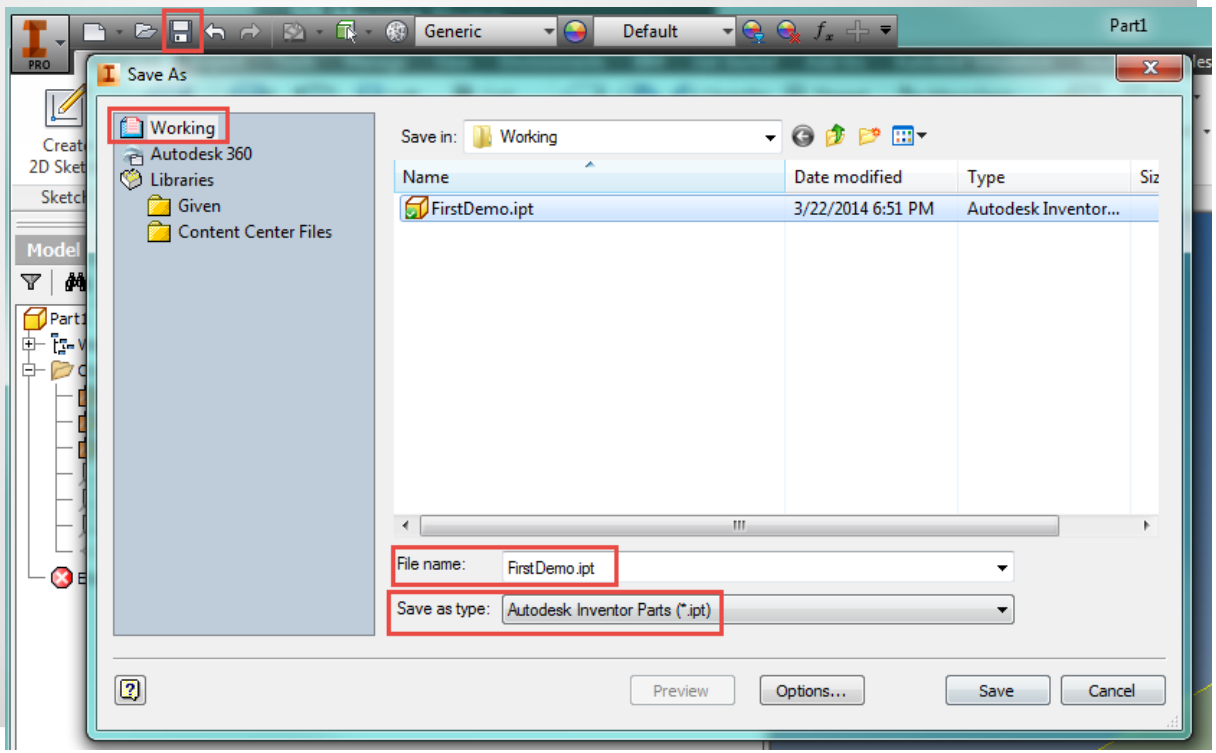


b. Adjust Display

- Save
  - ☑ Workspace



  - ☑ File name: FirstDemo.ipt
  - ☑ Save as type: Autodesk Inventor Parts (*.ipt)

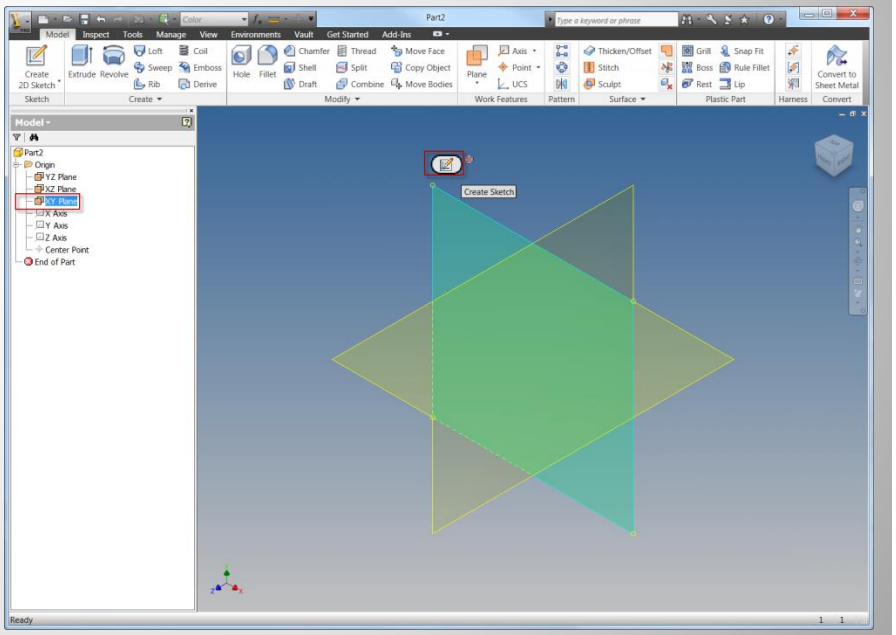


e. Save

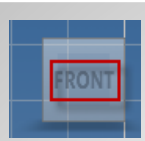
3. BASE FEATURE

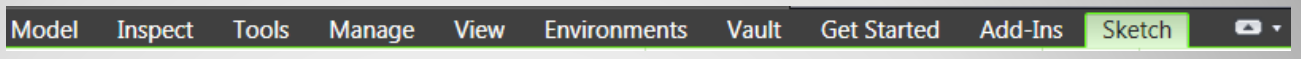
- ☑ Pick: 

- ☑ Pick: Create Sketch 

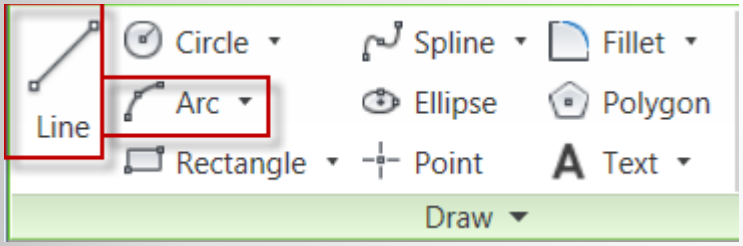


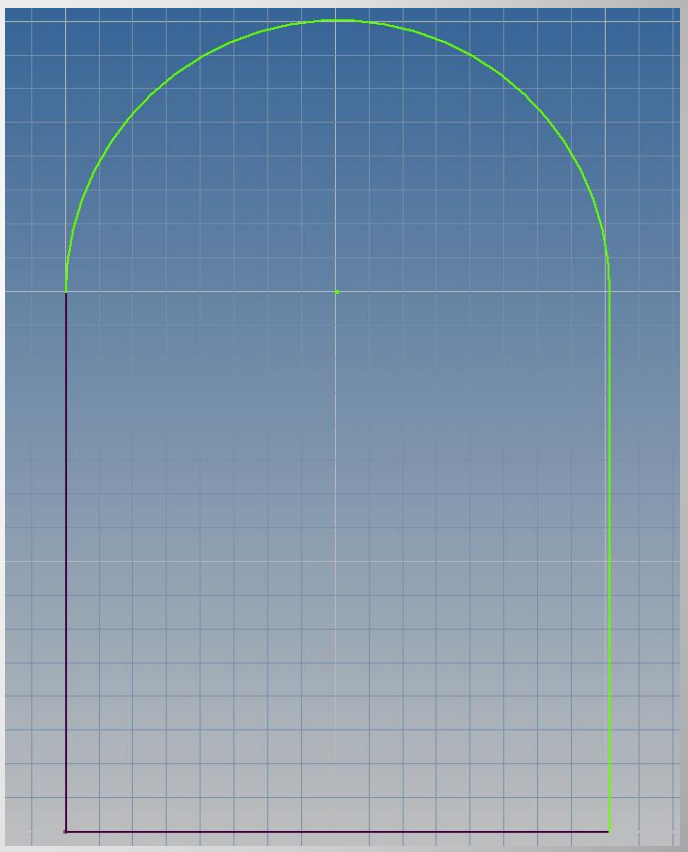
a. Sketch Plane

- Front 

- Sketch 

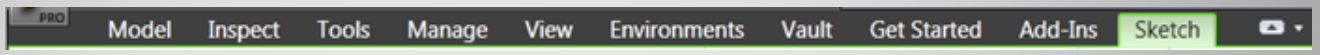
  - ☑Line
  - ☑Three Point Arc



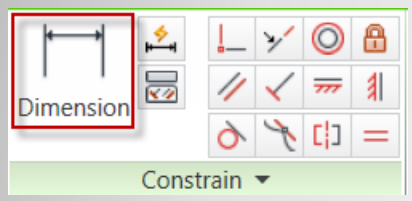


b. Sketch

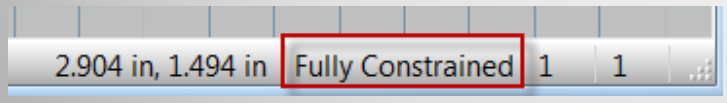
- Sketch

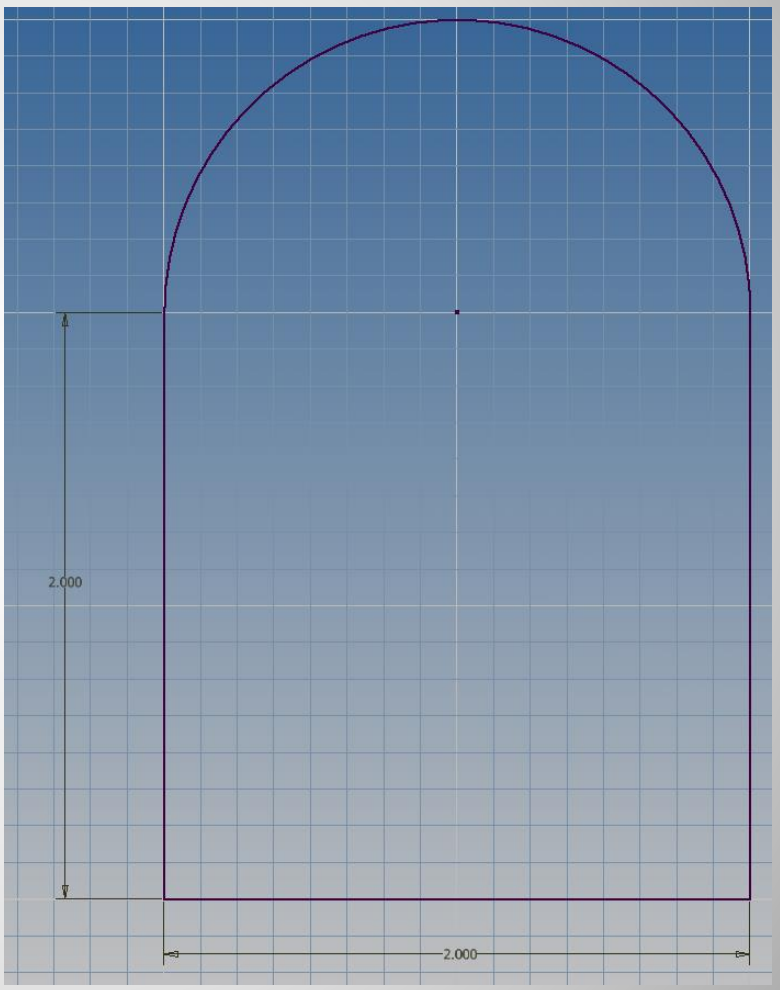


☑General Dimension



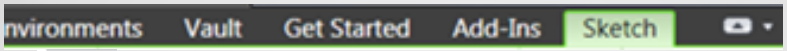
Note: Fully Constrained





c. Dimension

- <RMB>Show All Constraints
- Sketch



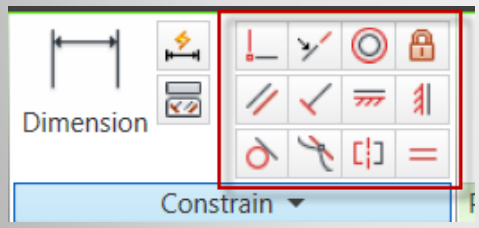
☑Tangent

☑Vertical

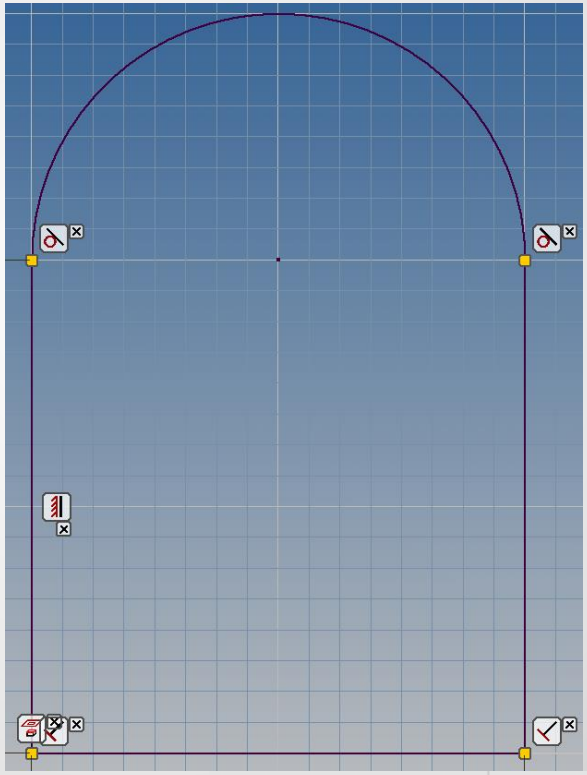
☑Parallel

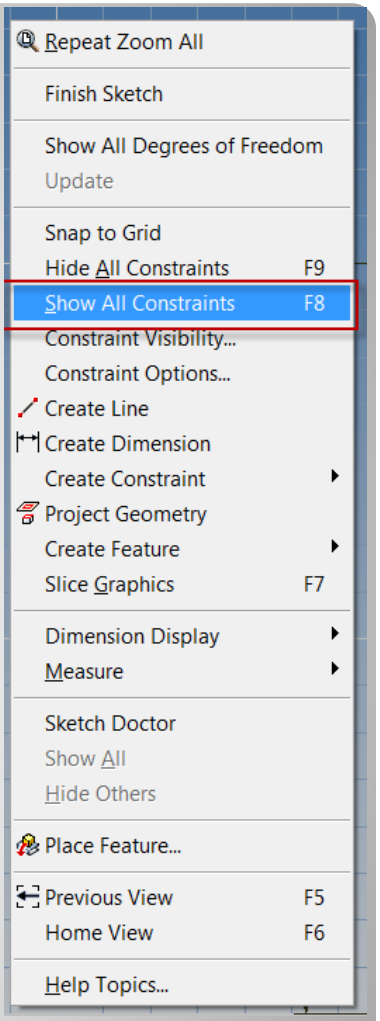
☑Perpendicular

☑Coincident

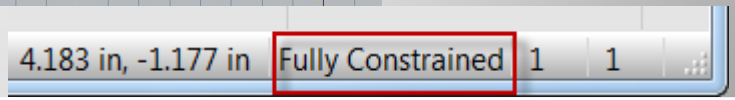


Note: Fully Constrained



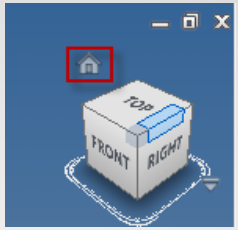


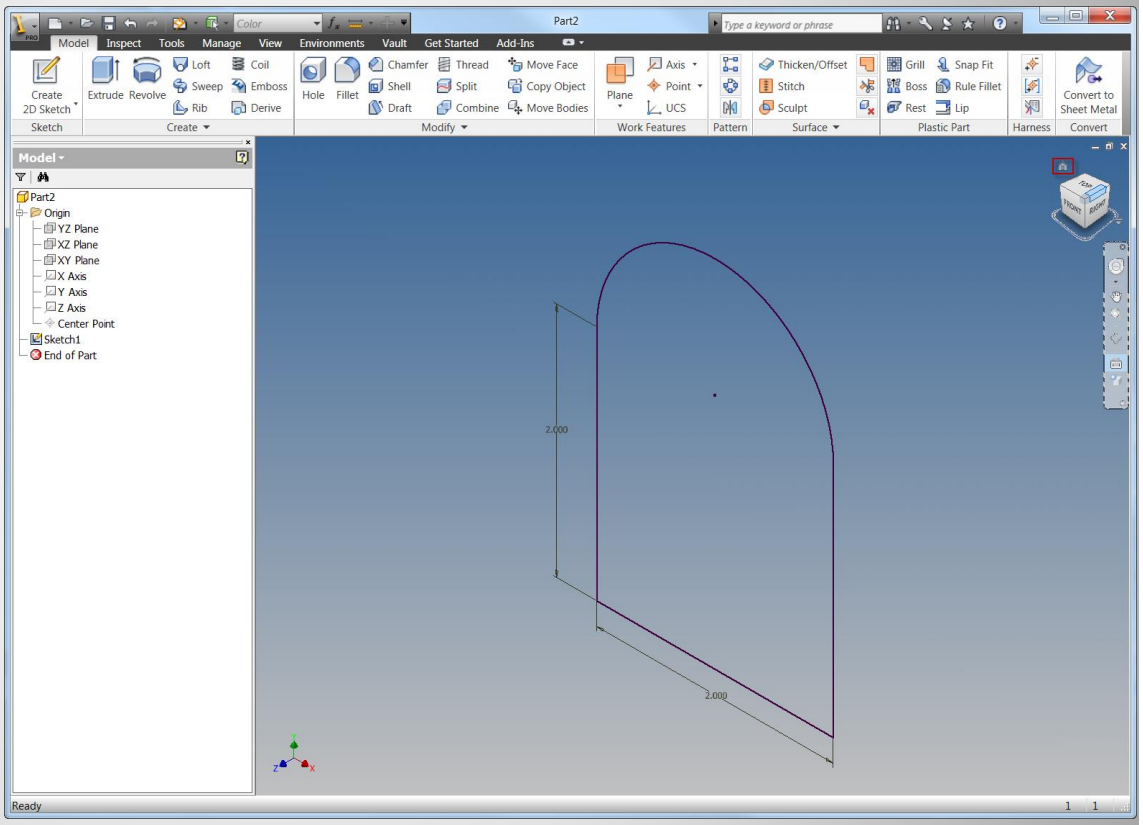
d. Constraints



- ☑ Finish Sketch 

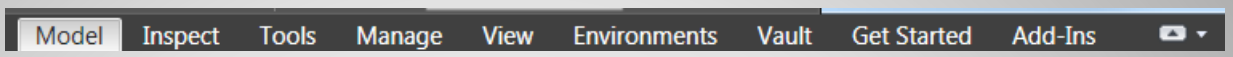
- ☑ Home View 

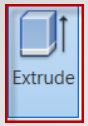


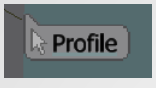


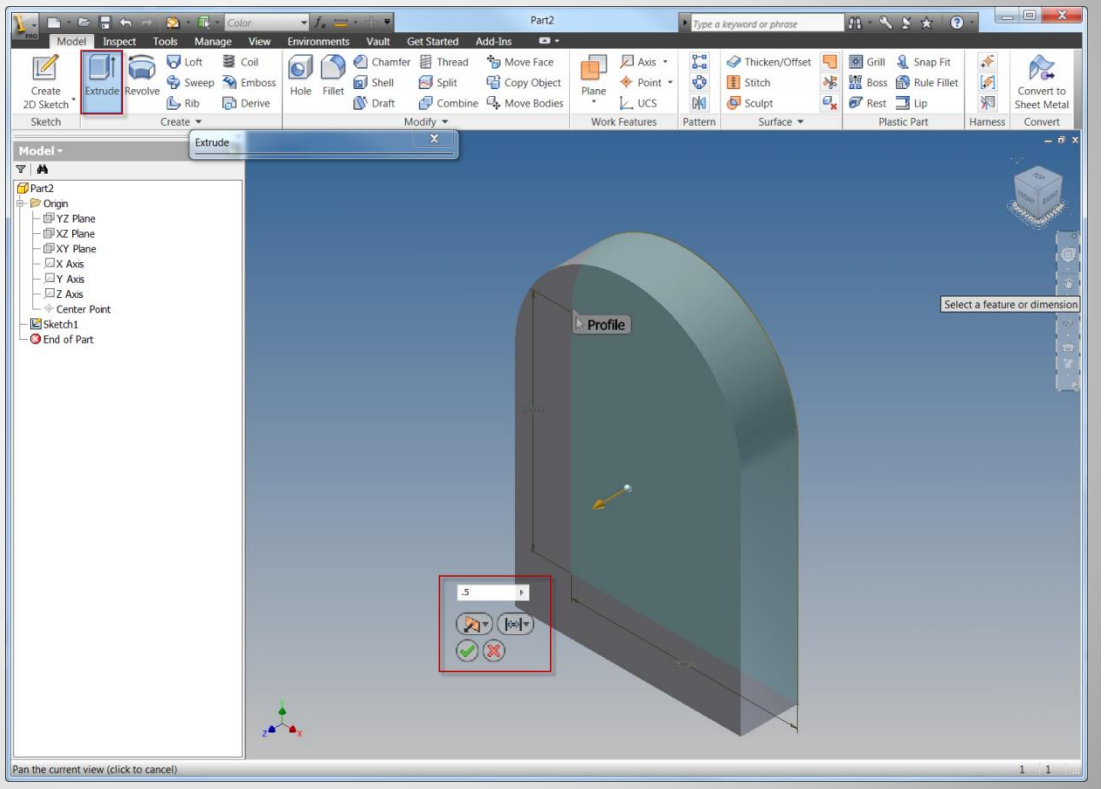
e. Finish Sketch

- Model
  - ☑Extrude
    - ✓Profile
    - ✓Join
    - ✓Direction
    - ✓Distance
    - ✓OK





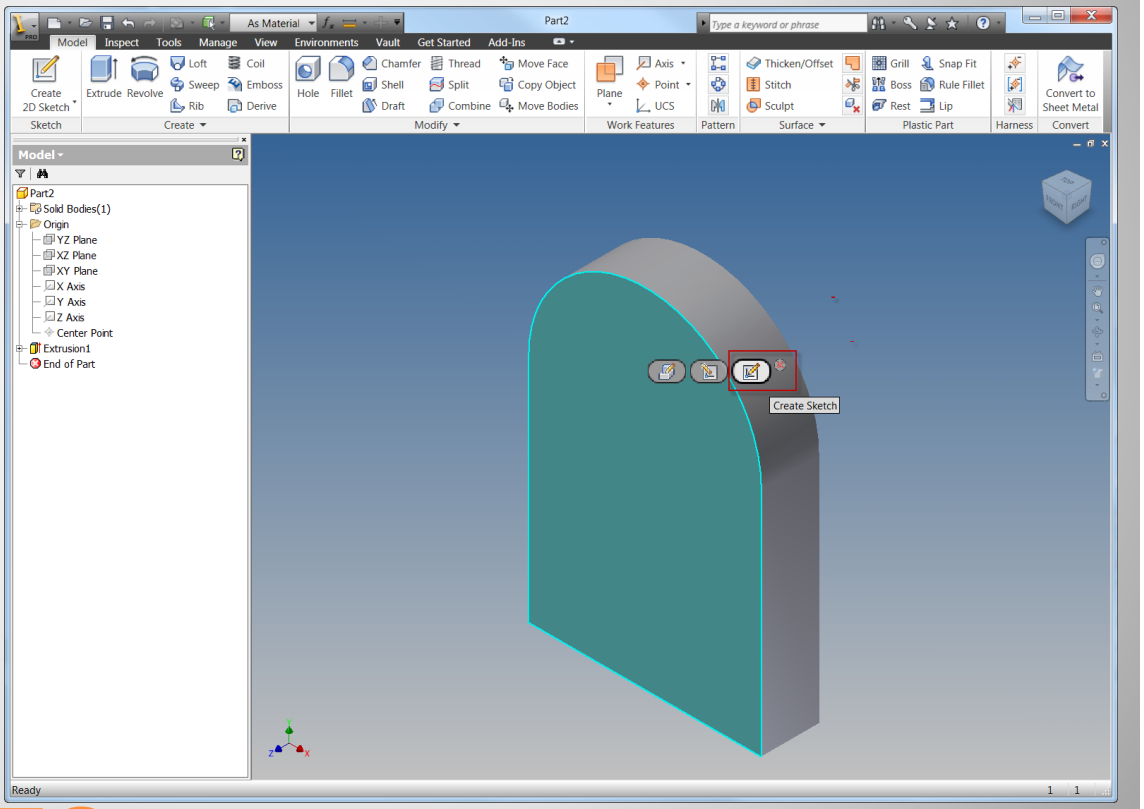




f. Extrude

4. SECOND FEATURE

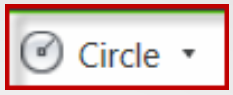
- ☑ Pick: Front Face
- ☑ Pick: Create Sketch



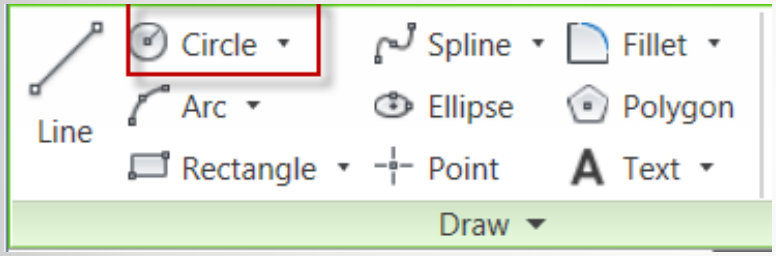
a. Initialize

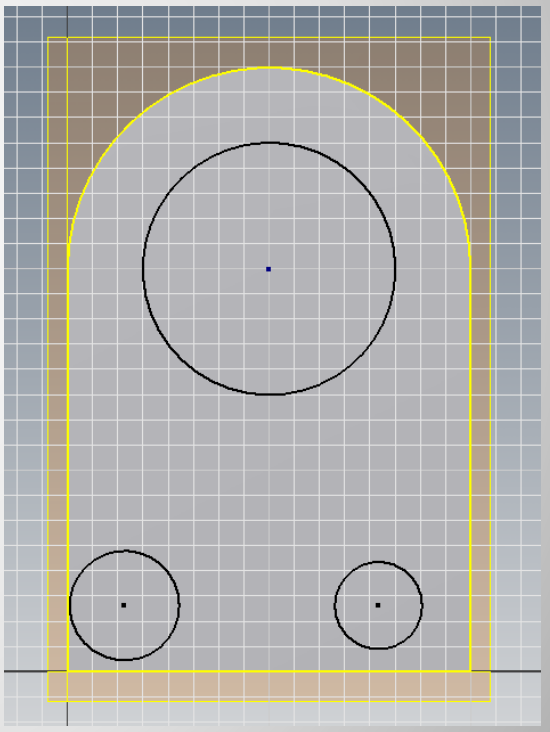
- Sketch

  - ☑Center Point Circle



    - ✓Center Point
    - ✓Diameter:

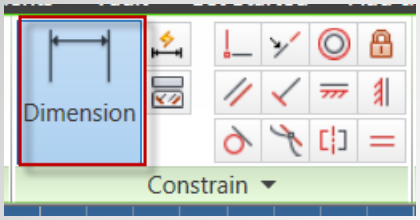


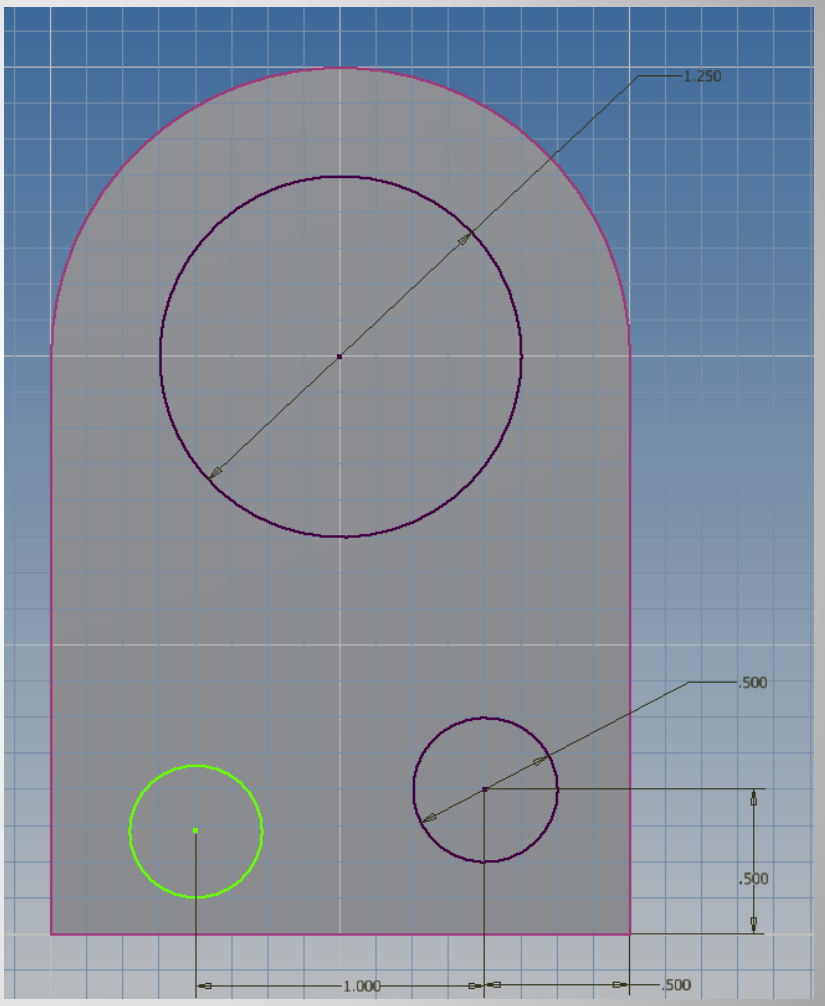


b. Sketch

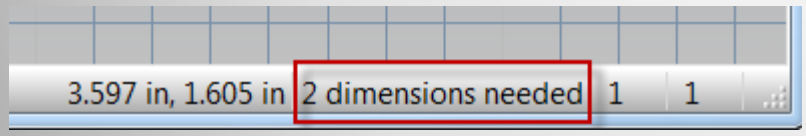
- Sketch

☑General Dimension



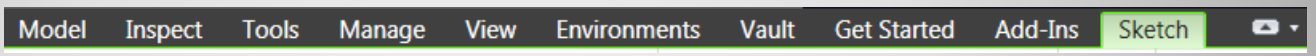


Note: 2 dimensions needed



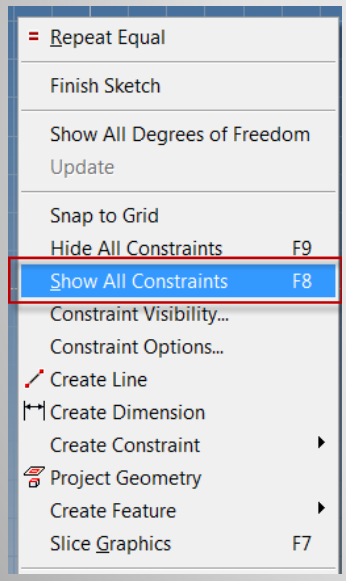
c. Dimension

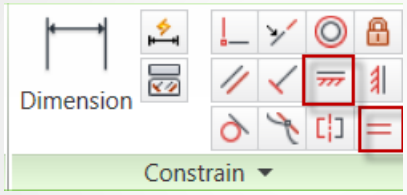
- <RMB>Show all Constraints
- Sketch

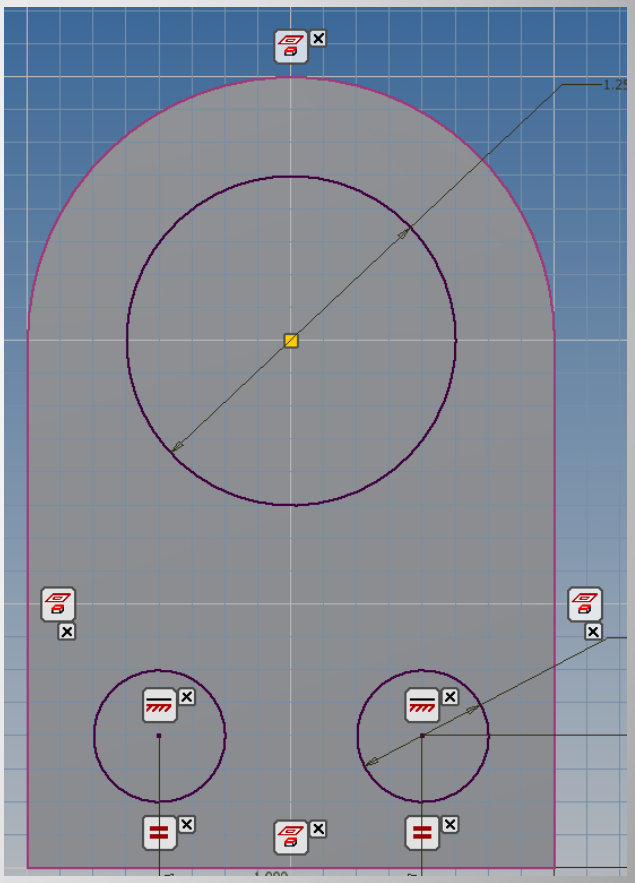


☑Equal 

☑Horizontal 

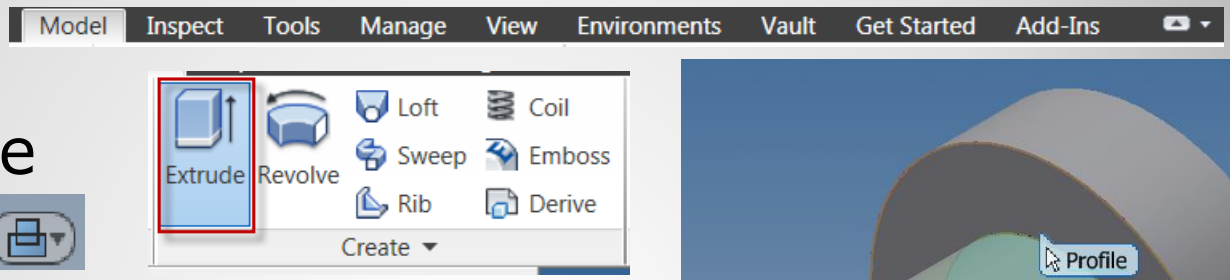


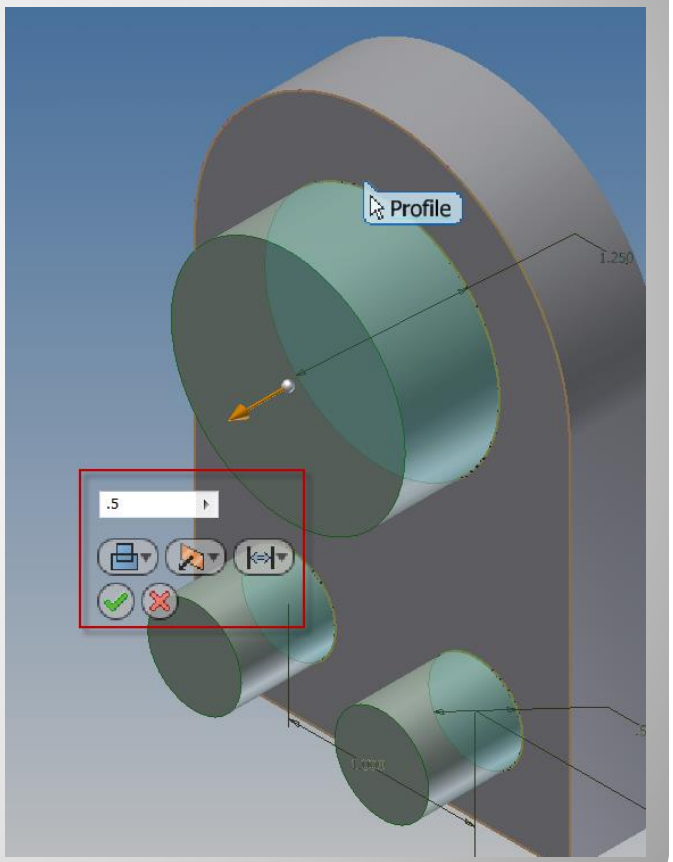




d. Constraint

- Home View
- Finish Sketch
- Model
  - ☑Extrude
    - ✓Join
    - ✓Other-previous

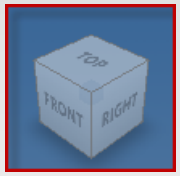


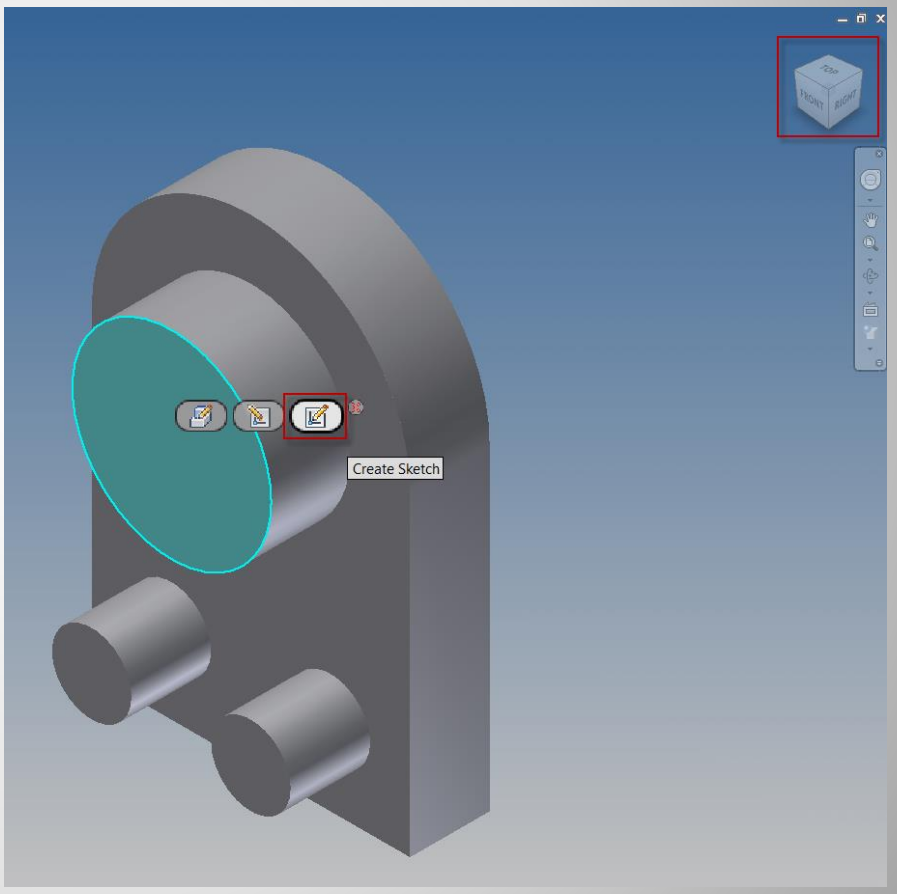


e. Extrude

5. THIRD FEATURE

- Home View
- New Sketch

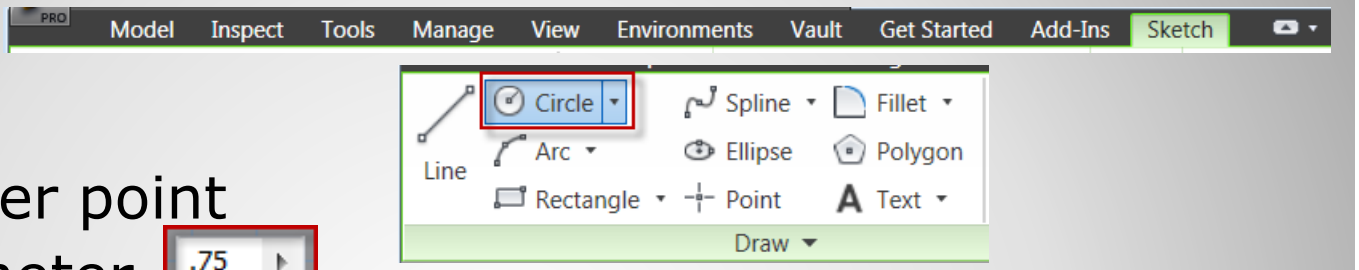




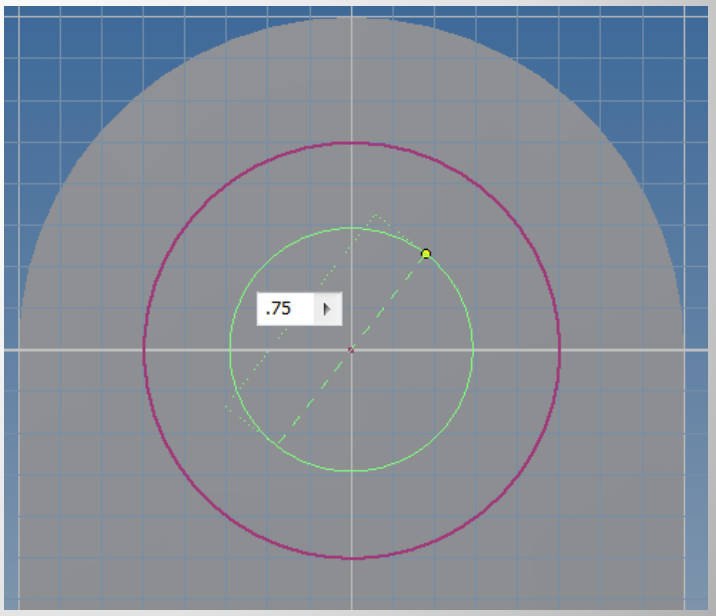
a. Initialize

- Front

- Sketch

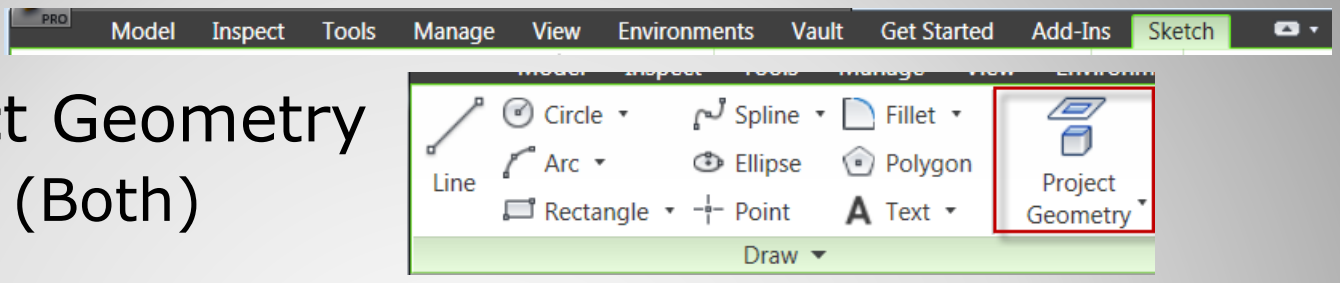


  - ☑Circle
    - ✓Center point
    - ✓Diameter

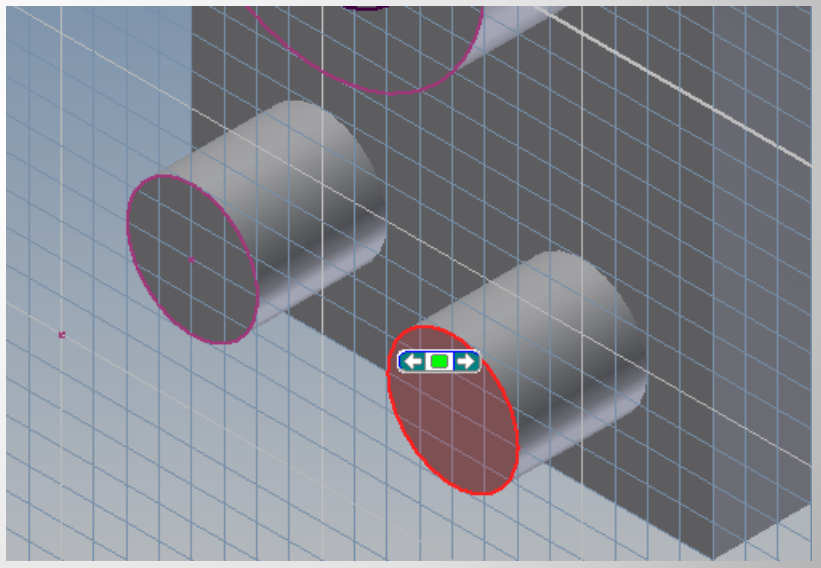


b. Sketch

- Sketch
  - ☑Project Geometry
    - ✓Edge (Both)

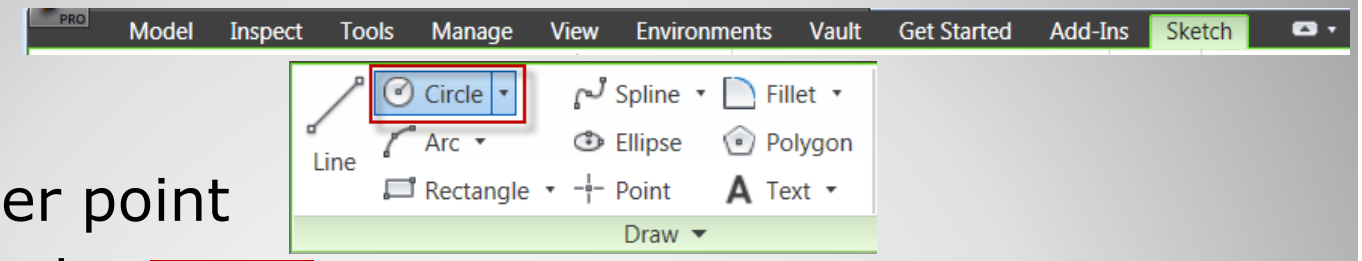


<u>Important:</u> Project Geometry allows the edges to be used in the sketch.
This will be used to make new features concentric to the edges.

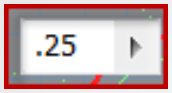


c. Sketch (Cont)

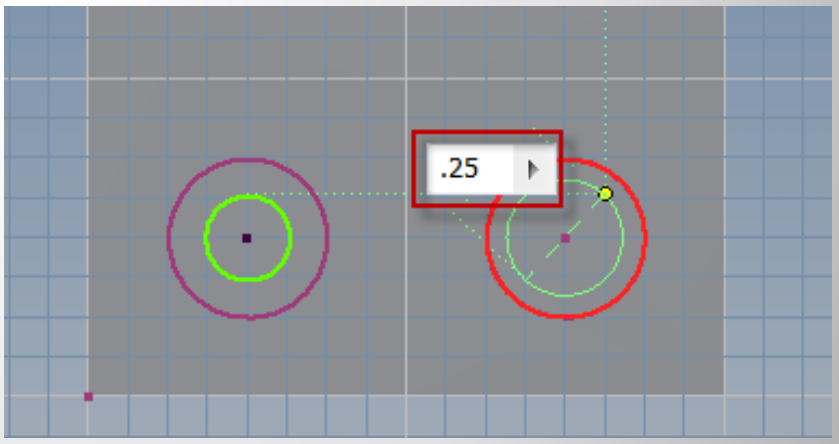
- Sketch



  - ☑Circle
    - ✓Center point
    - ✓Diameter

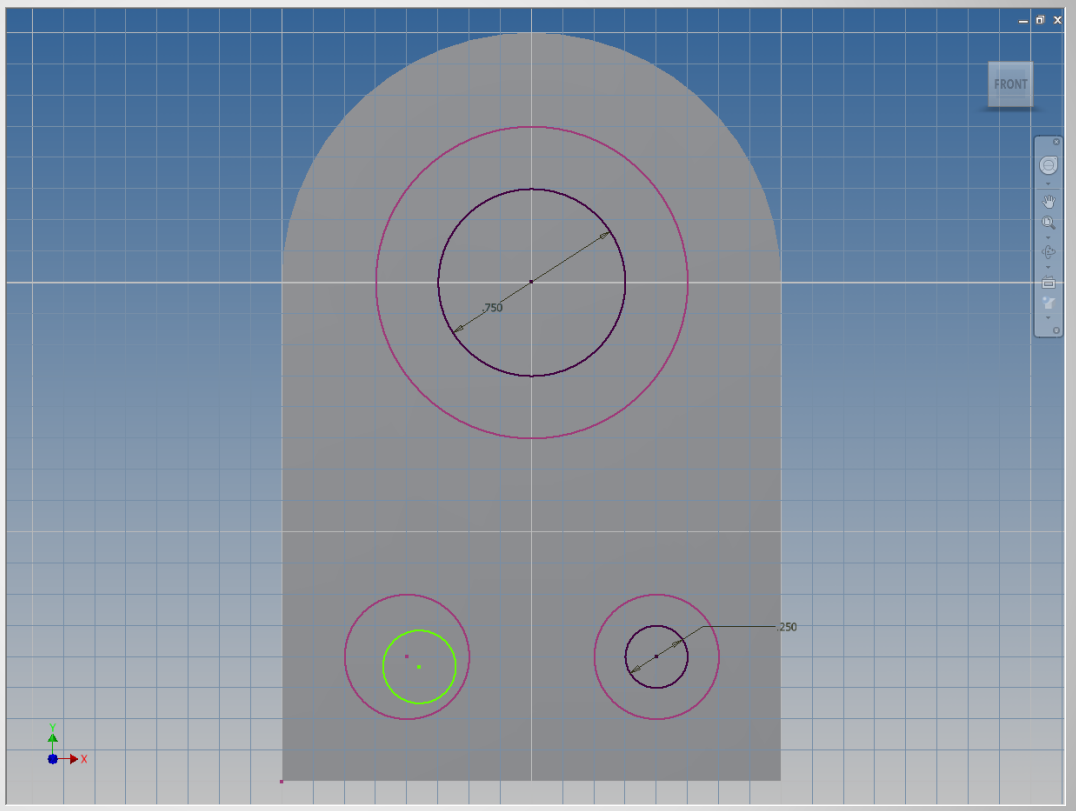


Important: Project Geometry allows the edges to be used in the sketch.
This will be used to make new features concentric to the edges.



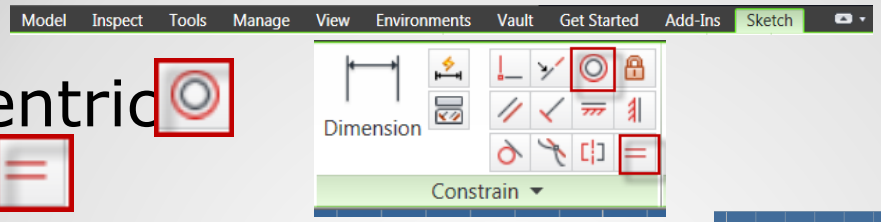
d. Sketch (Cont)

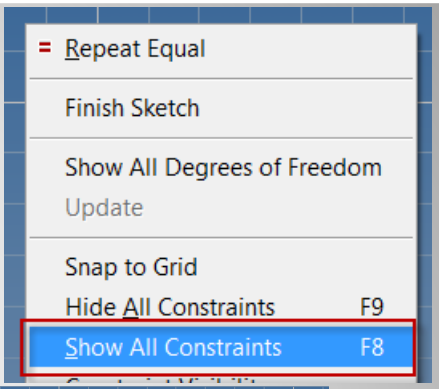
- No additional dimensions required

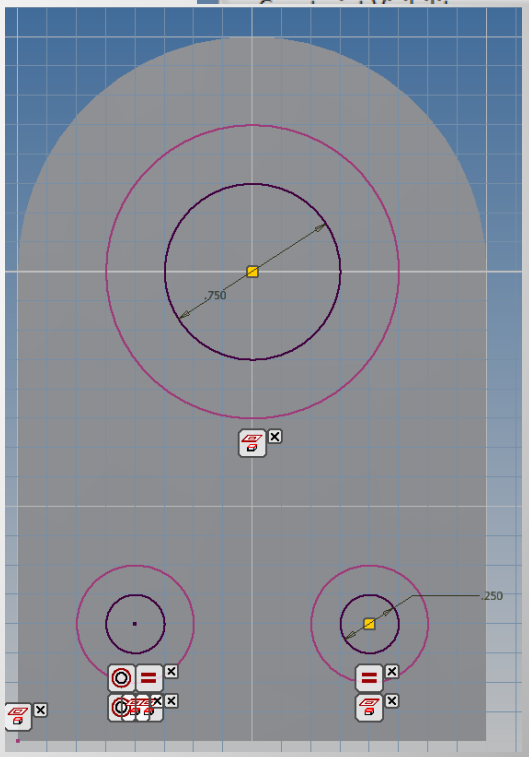


e. Dimension

- <RMB> Show All Constraints
- Sketch
  - ☑Concentric
  - ☑Equal



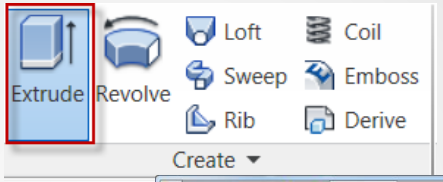




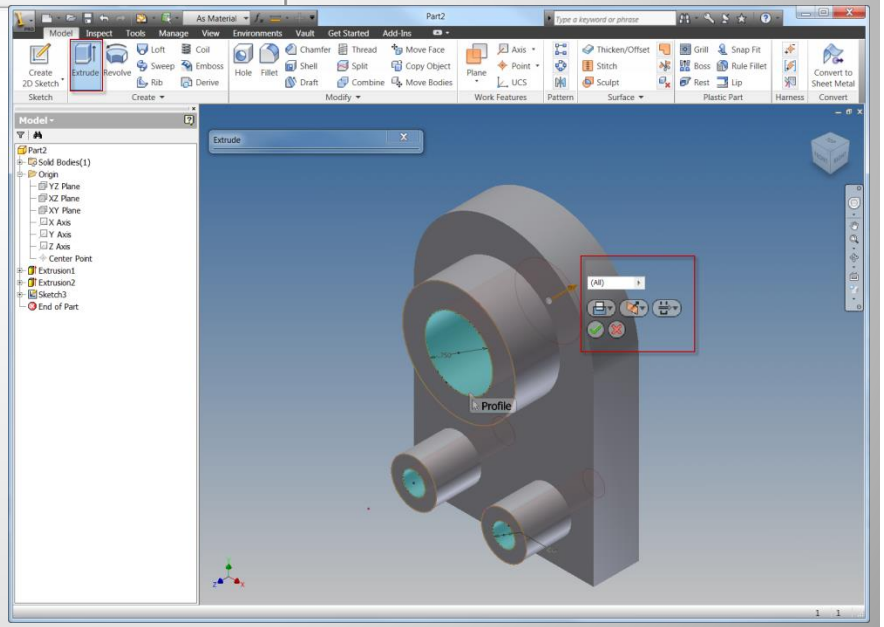
f. Constrain

- Home View 
- Finish Sketch 

- Model 

  - ☑Extrude 

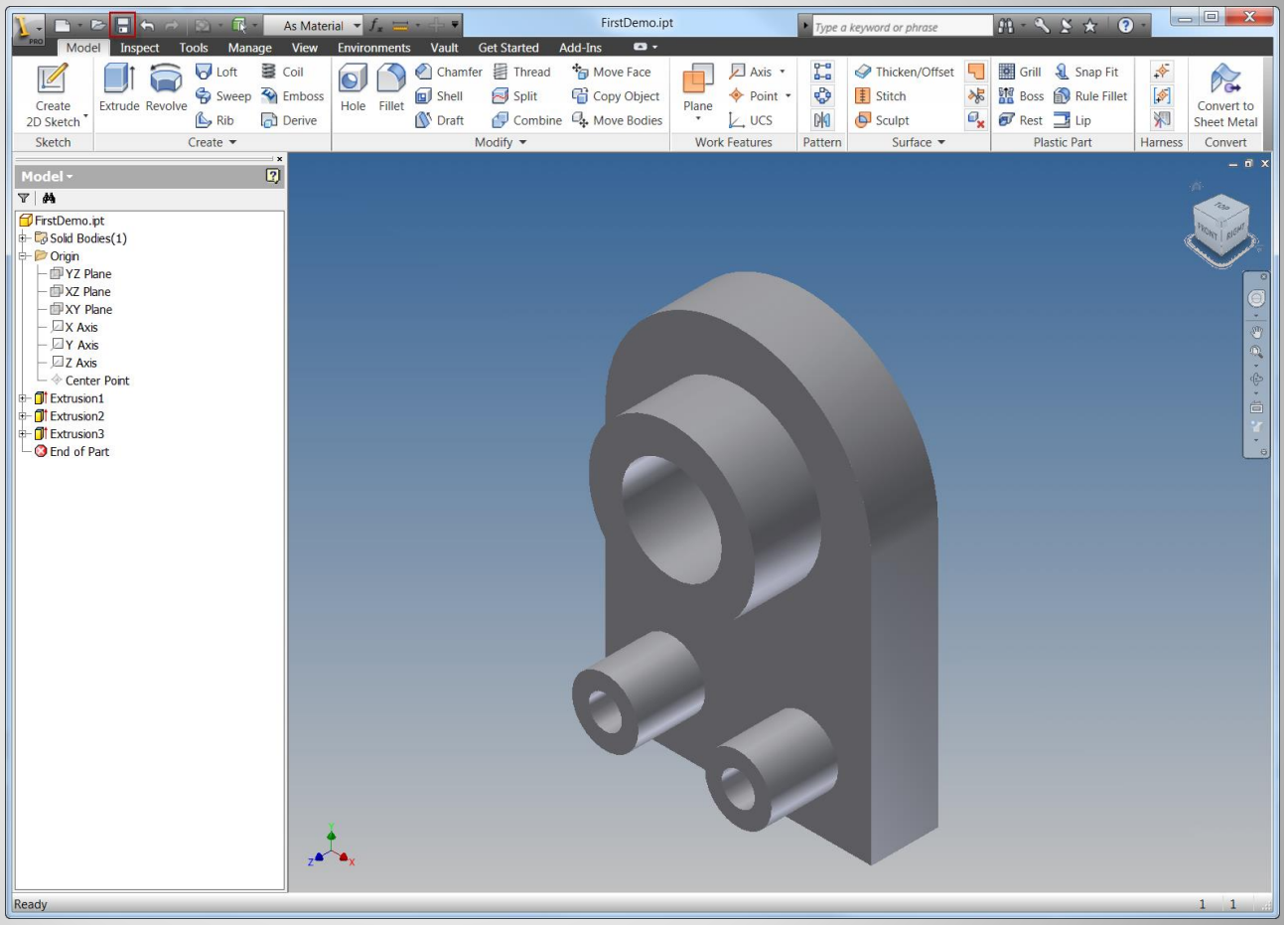
    - ✓Cut 
    - ✓Through All 



g. Extrude

5. FINISH THE PART

- Save 



a. Save the part