

# **Autodesk Inventor**

## **In Engineering Design & Drafting**

**By Edward Locke**

# Engineering Design Drafting Essentials

## ***Working Drawings:***

- Orthographic Projection Views (multi-view, auxiliary view, details and sections)
- Drawing Set (Part, Assembly, Explosion)

## ***Presentation Drawings:***

- Axonometric (Isometric, Dimetric and Trimetric)
- Oblique (general with  $\frac{3}{4}$ -Z, cabinet with  $\frac{1}{2}$ -Z, cavalier)
- Perspectives (1-point, 2-point and 3 point)

## ***CAD Programs:***

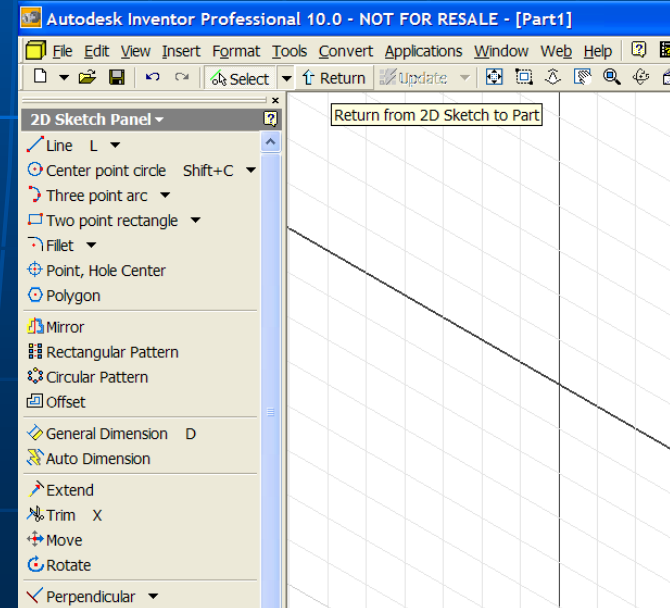
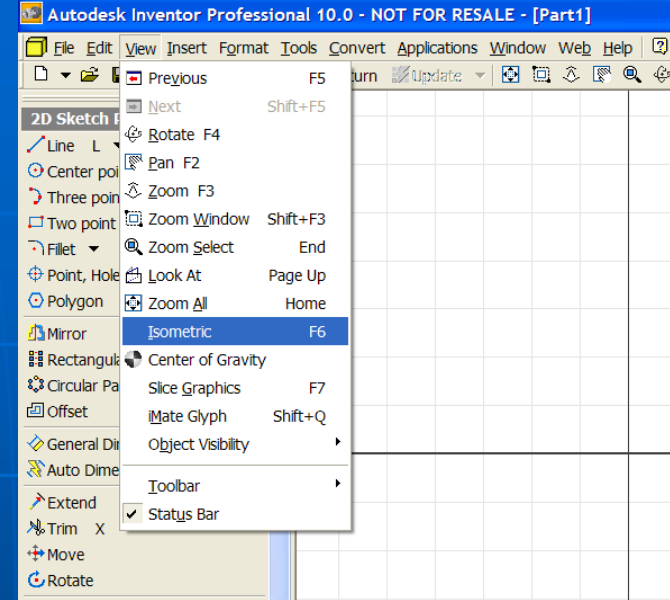
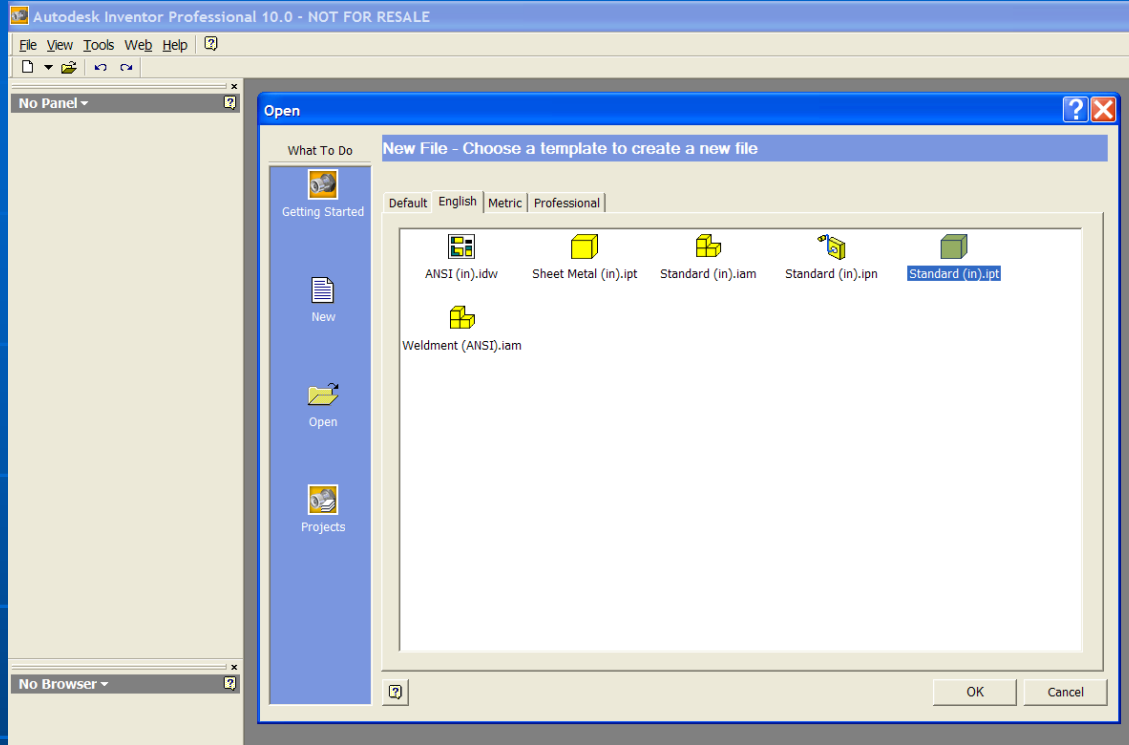
- Autodesk family (AutoCAD & AutoCAD Mechanical, Mechanical Desktop, and Inventor)
- Parametric 3D (Mechanical Desktop, Inventor, SolidWorks, SolidEdge, and CATIA)

# Parametric 3D Modeling:

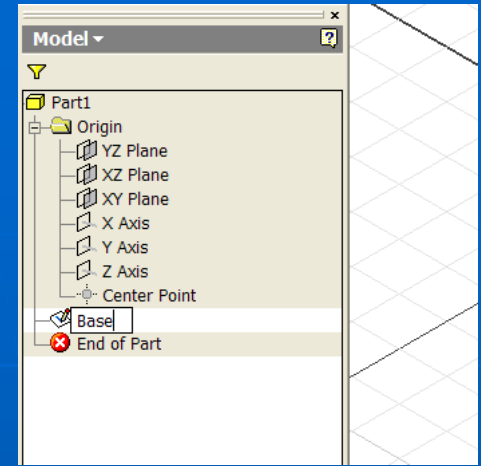
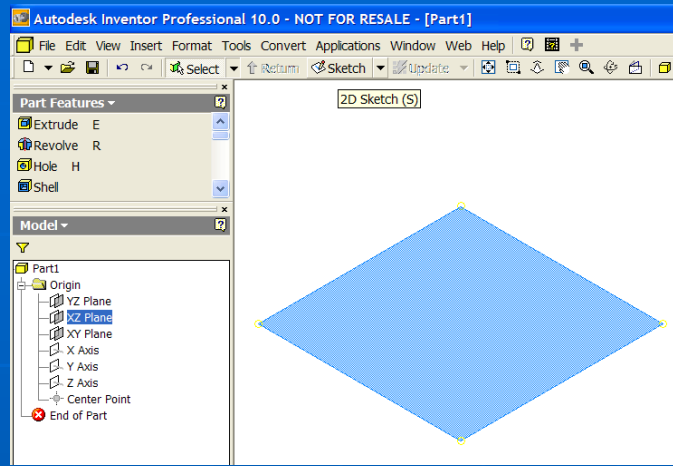
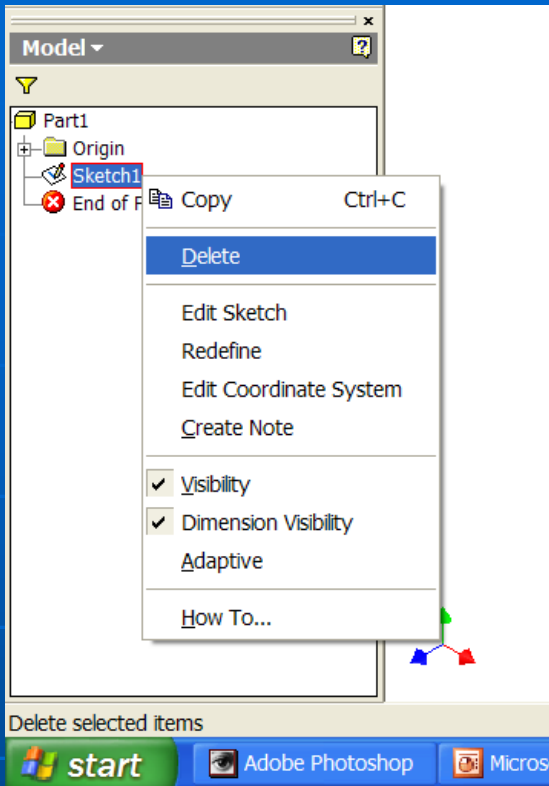
## Basic Steps

- ***Part File:*** Creates parts based on sketch with constraints to “sketched feature” and “placed feature”. Easy modification with parameter change, in design history recorder (the Model panel)
- ***Assembly File:*** Imports parts and applies geometric constraints, conducts FEA testing and/or creates presentation animation movies
- ***2D Working and Presentation Drawings:*** Automatically create and modifies BOM (bill of materials) based on changes made to part files

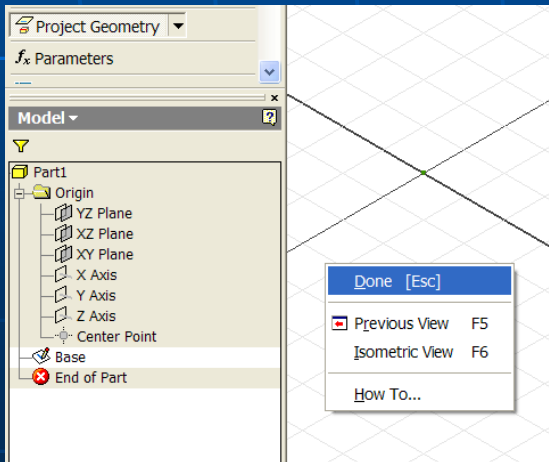
# Inventor 3D Modeling

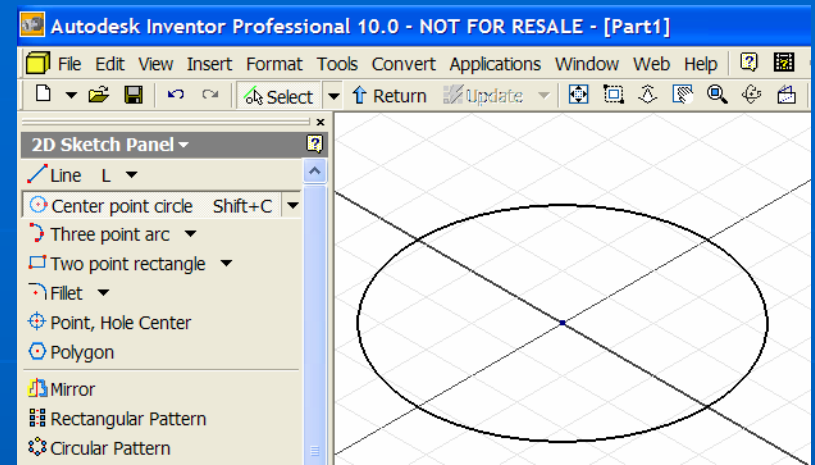
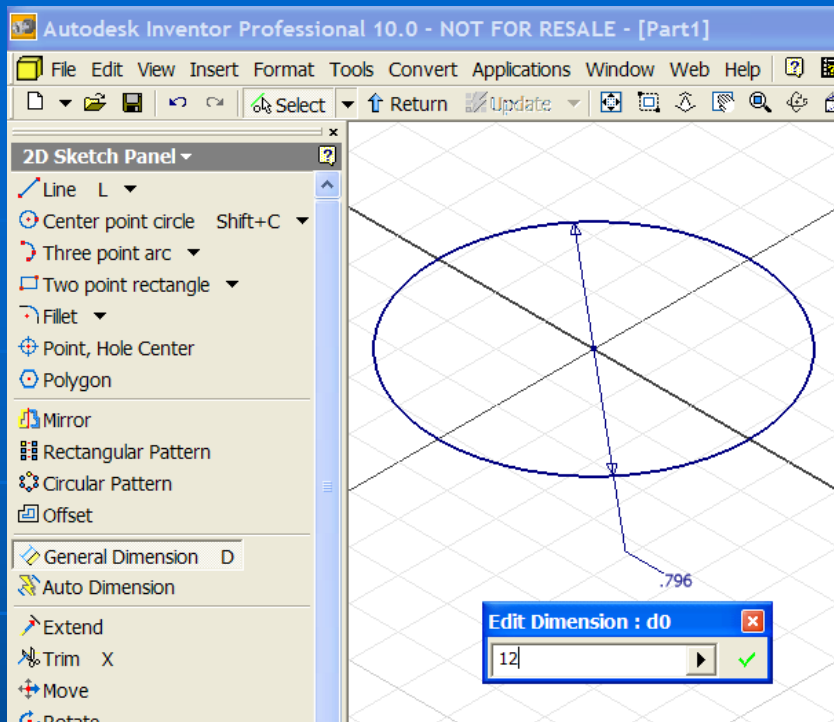


- Start an Inventor Standard (in).ipt file;
- Check the Sketch plane with View→Isometric;
- Click the Return button to dismiss the useless Sketch on the wrong plane.

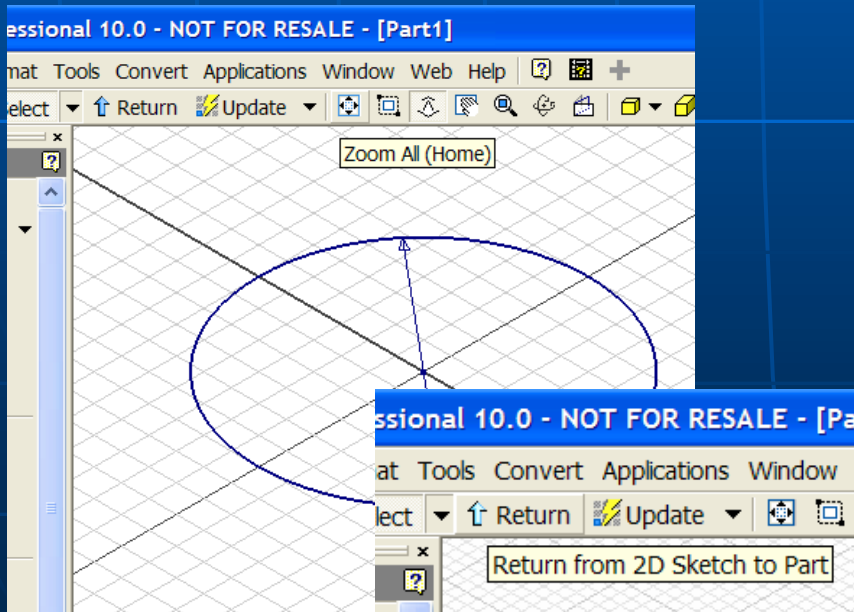


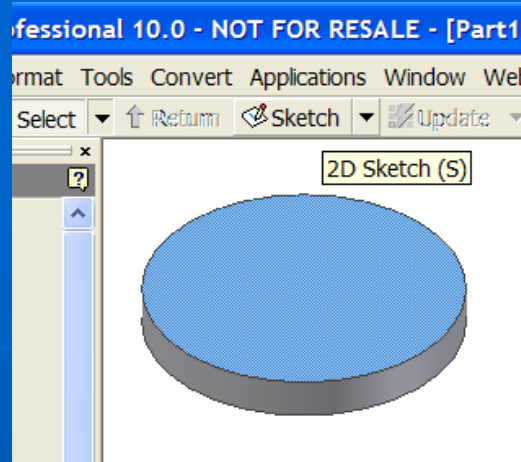
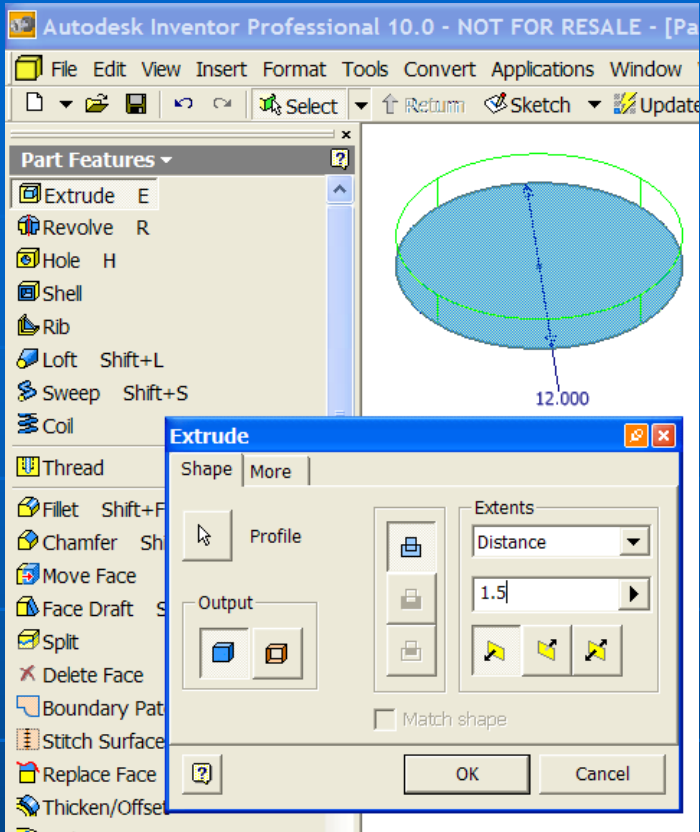
- Right-click the Sketch feature in the Model panel to delete the Sketch;
- Open the Origin folder, select the XZ Plane and click the Sketch button to start a new Sketch;
- Type Base over the highlighted Sketch1 text to rename the Sketch feature;
- Select the Project Geometry tool and the Center Point to project it onto the Base Sketch for the center of the circular profile;
- Right-click for the shortcut menu and select the Done option to end the Project Geometry tool.



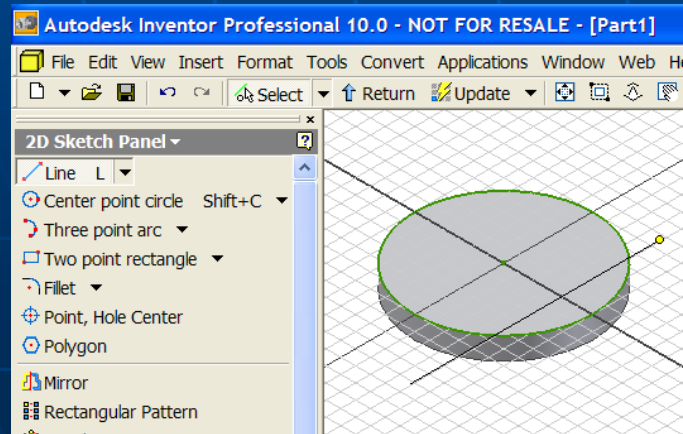


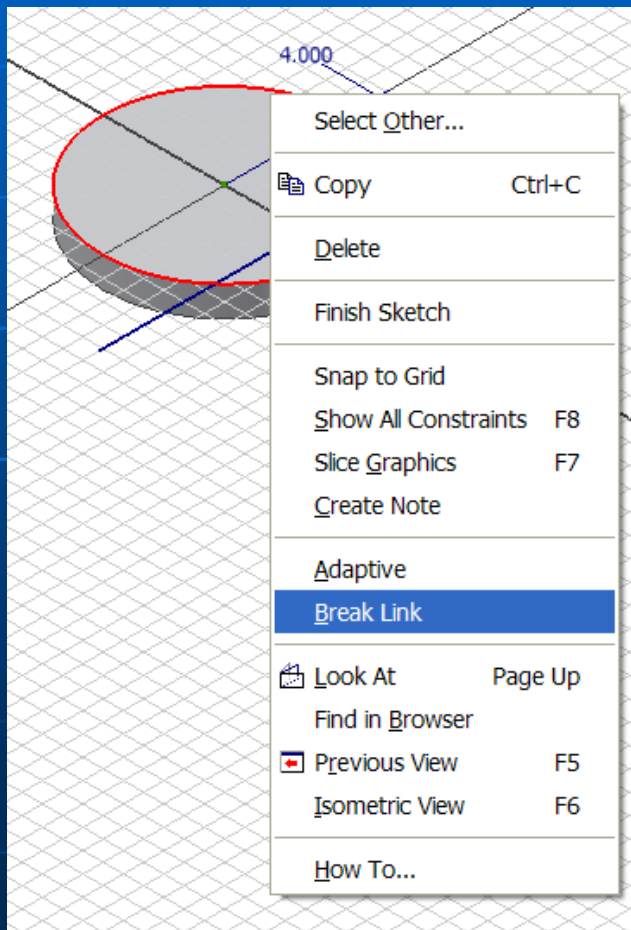
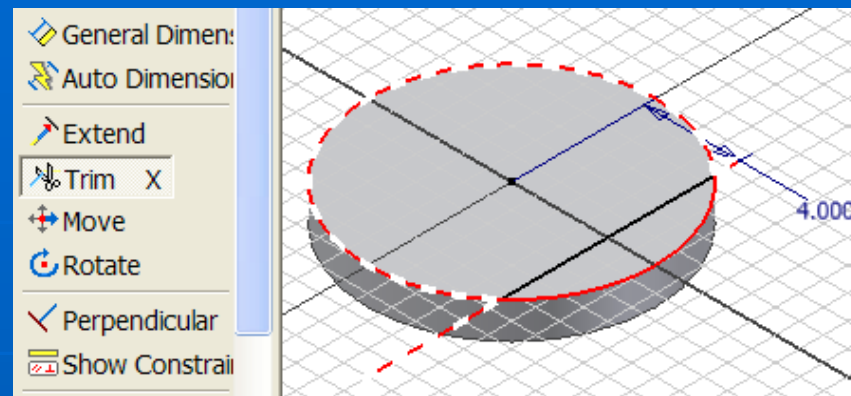
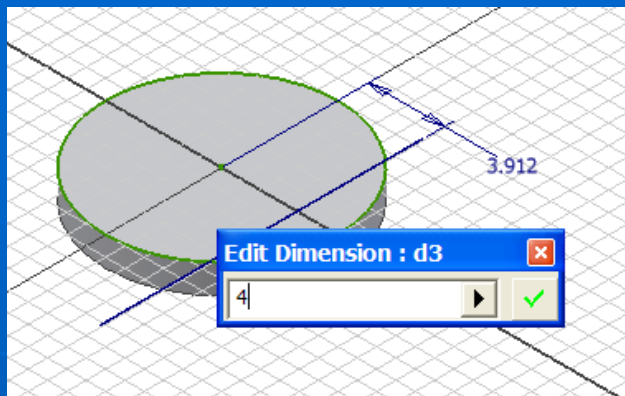
- Select the Center Point Circle tool, click at the projected Center Point (it turns green), drag out and click at any point to create a circle;
- Select the General Dimension tool, click the circle and drag out to get the default dimension, double-click the dimension to open the text field, type 12 [in] over the highlighted default text and click the green check mark to apply a 12 [in] diameter dimension;
- Click the Zoom All and then Return buttons to end the Sketch.





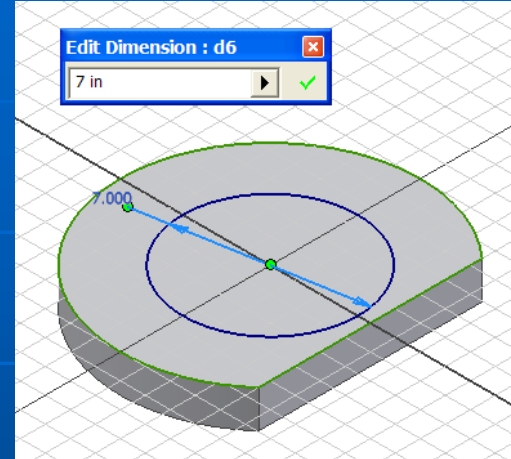
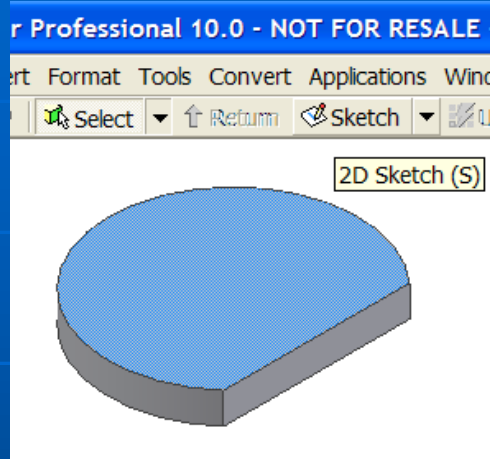
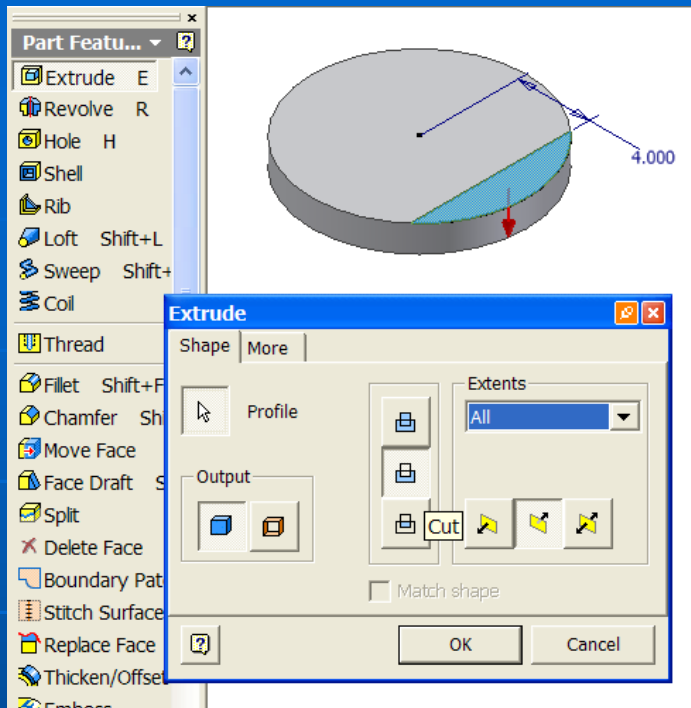
- Select the Extrude tool, accept the Distance option and type 1.5 [in] for Distance (height) in the Extents section; in the Model panel, change the name of the Extrude1 to Base Plate;
- Select the top surface and click the Sketch button to start a new Sketch, and change its name to Cut;
- Select the Line tool, click a point beneath the right portion of the circle, drag out upward and click when the vertical snap appears to create a vertical line as shown;



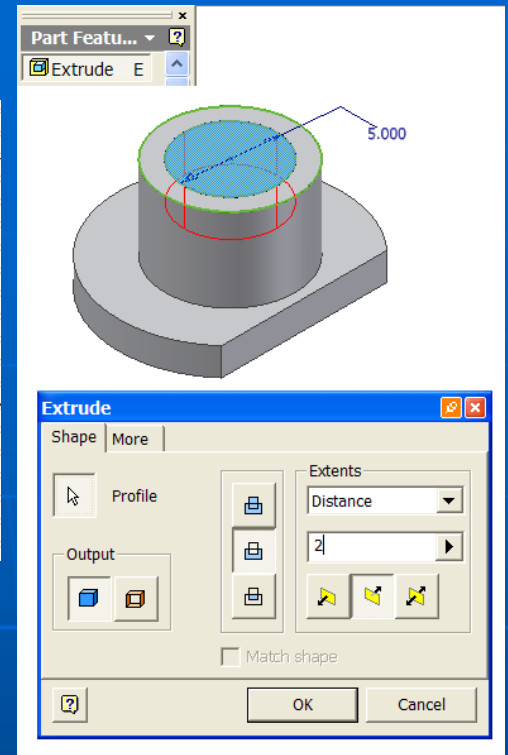
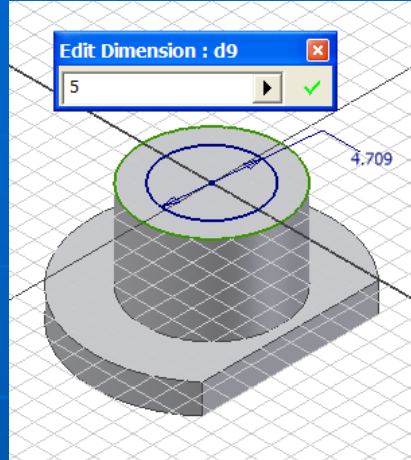
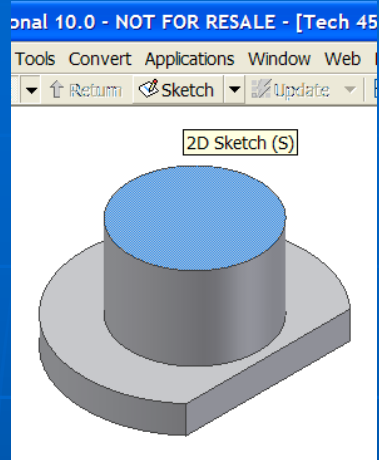
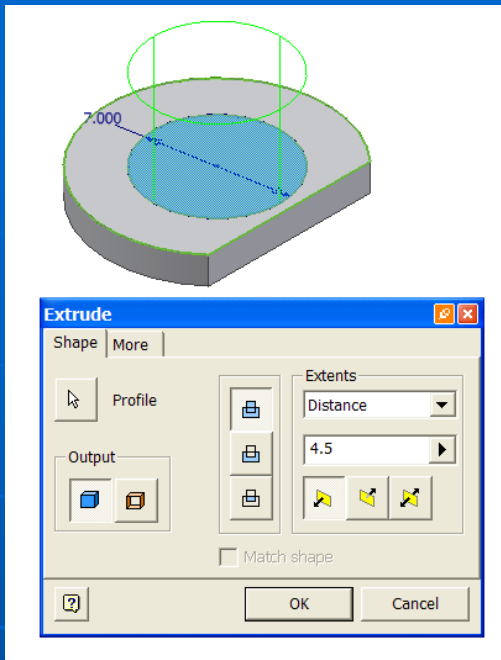


- Select the General Dimension tool to apply a 4 [in] linear dimension between the projected Center Point and the vertical line;
- Select the green projected outline of circle, right-click for the shortcut menu and select the Break Link option, to change it to editable line;
- Select the Trim tool and click the excessive segment of the circle and the vertical line as shown to trim them off;
- Click the Return button to end the Cut Sketch.

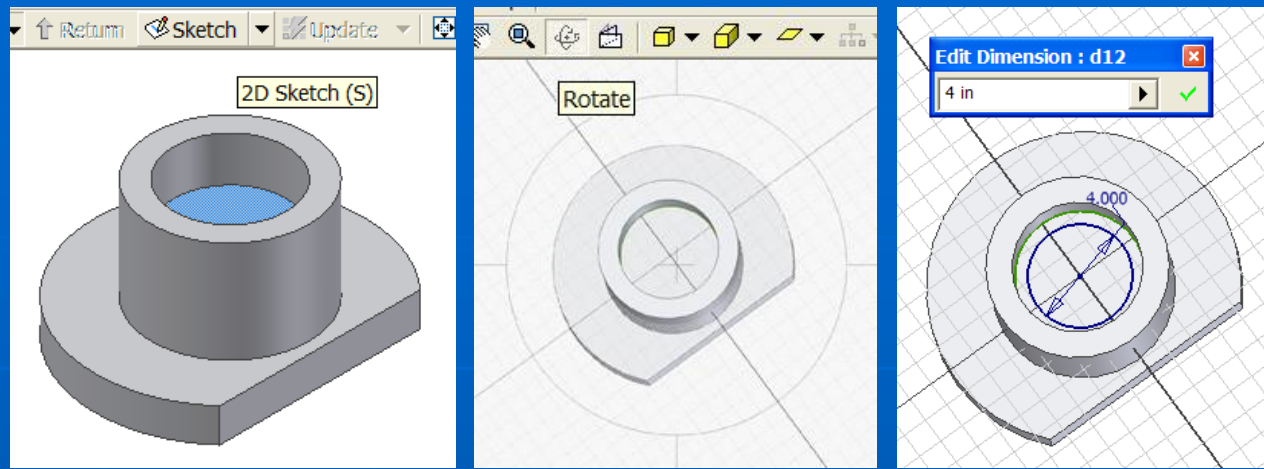




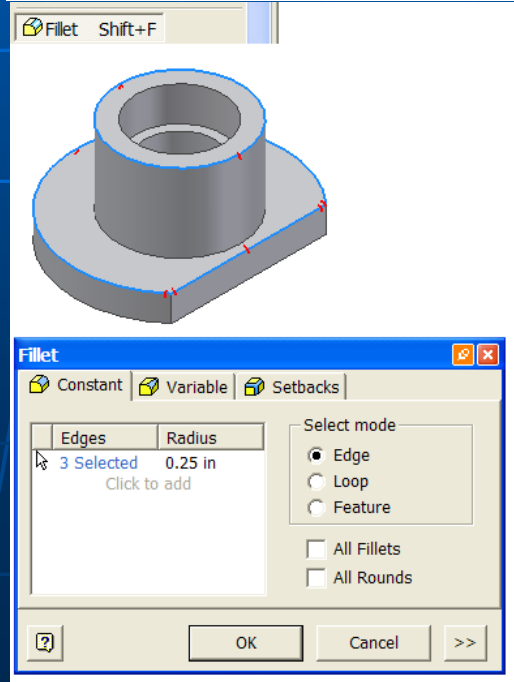
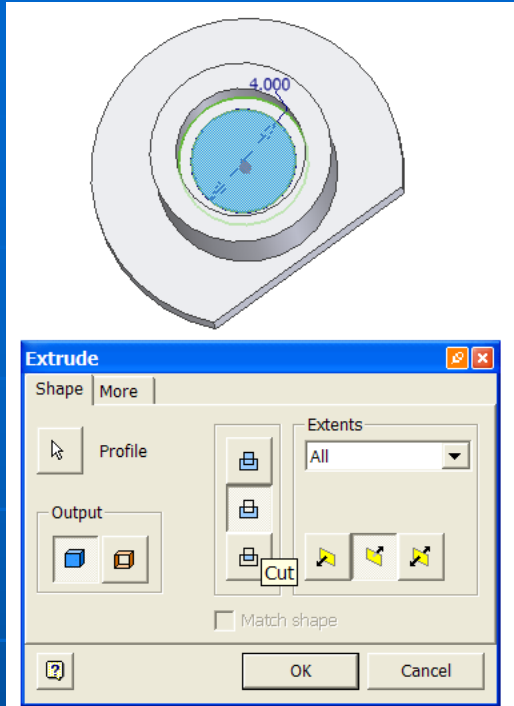
- Select the Extrude tool, the Cut and All options, and the downward Direction, and click the OK button to cut the straight edge out of the Base Plate; and rename the Extrude2 feature as Straight Cut in the Model panel;
- Select the top surface to start a new Sketch, and change the name to Bose; use the Center Point Circle and General Dimension tools to create a concentric circle; click the Return button to end the Sketch.



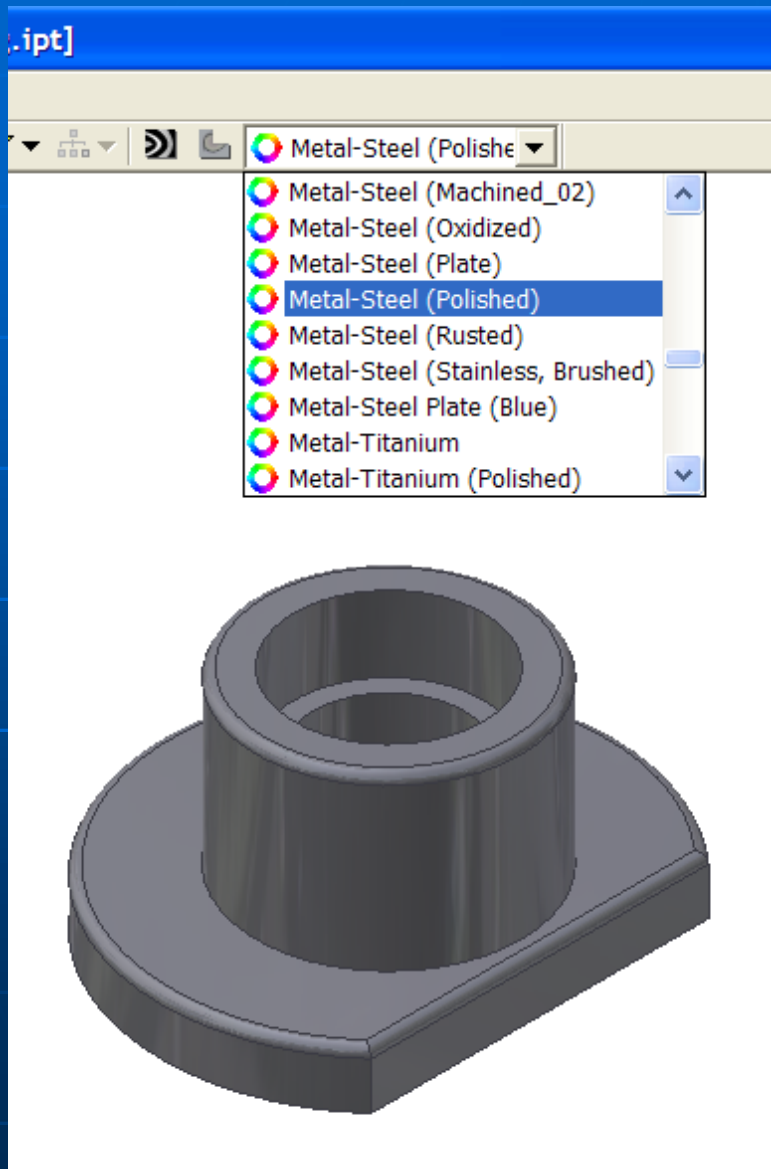
- Select the Extrude tool, select the circle as Profile, the Distance option, and the upward Direction, type 4.5 [in] for Distance, and click the OK button to create the boss; and rename the Extrude3 feature as Boss Extrusion in the Model panel;
- Select the top surface of the Boss to start a new Sketch and rename it Counterbore; use the Center Point Circle and General Dimension tools to create a concentric 5 [in] diameter circle; use the Extrude tool to cut a 2 [in] deep hole and rename the Extrude4 feature as Counterbore Cut.



- Select the top surface of the Counterbore to start a new Sketch and rename it Hole; use the Rotate tool to rotate the model to a convenient view; use the Center Point Circle and General Dimension tools to create a concentric 4 [in] diameter circle; use the Extrude tool with the All and downward (or Midplane) Direction options to cut a through hole, and rename the Extrude5 feature as Hole Cut;
- Select the Fillet tool, type .25 in for radius, select the ↖ button on the left end of the text field in the window, and select the related edges of the model to apply fillets.



# Inventor 3D Rendering



- Select a material type from the pull-down menu to apply a rendering effect.

**End of  
Autodesk Inventor 3D  
Modeling  
Presentation**

**Thank You For  
Your Attention**