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Autodesk® Inventor™ Part Modeling:
The First Step, Part I

Autodesk® Inventor™ Part Modeling: The First Step, Part I

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MA111-5 Lofts, tangents, and bores -- no need to fear Autodesk Inventor Part modeling. We'll show you everything you need to know to be the hero back in the office. This class will demonstrate most of the sketch tools and part features in Autodesk Inventor and illustrate logical techniques and guiding principles to help you create accurate and healthy parametric parts that will behave in your assemblies. Proper sketch and part modeling is the foundation for Inventor assemblies and capturing the proper design intent.

About the Speakers:

Anthony is a project manager and knowledge manager at British Petroleum. He is a former design engineer, CAD manager, software developer, ATC instructor, and CAE industry analyst. He is considered to be an agent of change and a proponent of technology, though not enamored of it (still no Blackberry). An instructor at AU for the past 14 years, Anthony will be teaching his classes with his colleague, Tom Short. Anthony's classes are designed to show users how to maximize their efficiency, and he strives to make his classes fun as well as informative. Anthony.Dudek@bp.com

Tom Short is a mechanical engineer registered as a professional in Michigan. He has taught and consulted on AutoCAD, Autodesk Inventor, and Autodesk Mechanical Desktop for many years in the U.S. and other countries. At Munro & Associates, Inc., Tom is a consultant on Lean Design, and has worked with a variety of companies helping them improve their designs for simplicity and ease of assembly. He is also an author with books on Inventor and Mechanical Desktop. Tom has presented at every Autodesk University since its inception. Notietom@cs.com



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Objectives

- Take a good look at the powerful tools available to assist you in designing incredible parts in Autodesk Inventor.
- Gain more Inventor power from these techniques to make you famous!
- Acquire a better understanding of parametric design methodology inside the latest release of Autodesk Inventor.

Outline

1:30 – 1:45	15 min.	Introductions – Class Objectives
1:45 – 2:15	30 min.	Sketching and Control
2:15 – 2:50	50 min.	Effective Part Modeling
2:50 – 3:00	10 min.	Summary - Question & Answer

Sketches, Geometric Constraints, Dimensional Constraints, etc.

Try to use geometric constraints wherever possible. They are tough little guys doing ten times work for their body size. Not very noticeable, doing their work behind the scenes, but they bring an unruly profile into line (no pun intended).

Sketch your profile to roughly the final size. Drawing a rectangle 12 X 6 when you really want to end up with a 2 X 1 final size doesn't make sense and you're asking for trouble. The rectangle example is an oversimplification, but you get the idea. Later on, as your skills improve you can use Shared Sketches and what I call "Controlling" Sketches.

Use "smart" equations for your dimensional values, not just dumb numeric values. Equations help to tie dimensions together so that if one changes perhaps they all do! The ideal is to have all your part's features tied to one or two key dimensions. Change one of those key dimensions and the part significantly changes. Most part designs evolve around a few key dimensions – the bore ID of a cylinder, the mounting holes on a bracket, etc.

Any sketch profile must survive the acid test first before it can be turned into a 3D feature. Any dimension on your profile should be able stand a reasonable dimension value change without the entire sketch profile distorting out of shape. A dimension change should produce the change you expected, if not then the sketch profile is not considered "logically" constrained; even though Inventor may have reported it as "fully" constrained.

What happens if you, like the rest of us, lose track of how many dimensions/constraints are still needed to fully constrain the profile? Use the Auto-dimension tool and it will report how many more dimensions and/or constraints left to go. In Inventor 2008 there is a new running commentary in the lower-right-hand of the screen telling you how many more dimensions (constraints) are needed!

Use construction geometry! Use construction geometry! Please! This very powerful, yet under utilized, technique is often you're only chance to fully constrain a profile. Use construction geometry. Although invisible when the profile is extruded, construction geometry is behind the scenes doing its job. Construction geometry not only has its uses in sketch profiles, but also in assemblies.



The Sketch Doctor isn't always right! Many of you have probably run into the infuriating Catch-22 type situation wherein the Sketch Doctor says that a sketch is missing coincident constraints at some vertices. However, at the same time he is also reporting *that those same vertices have redundant coincident constraints!*

Inventor Features Tutorial

We're going to do a tutorial to demonstrate some of Inventor's Part Modeling capabilities. No dataset parts are required – this will start from a new Part.

Start a new Part. There should be a Sketch already created and opened for edit.

On the Sketch Project the X axis and the Y axis from the Origin folder. This is an industry-accepted practice, although some only Project the Center Point.

Draw a rectangle and dimension it so that it is centered using the projected axis lines. Make the dimensions parametrically defined so as to center the rectangle on the Origin. Example: $D0/2$

F6 to change the display to Isometric

Add .50 fillets to the sketch at each corner.

Hit E on the keyboard – Extrude. Specify an extrusion height of .50"

Expand the Origin folder in the Browser and in turn left-click on each of the Origin Workplanes. Notice how they bisect the Base feature evenly in all three dimensions.

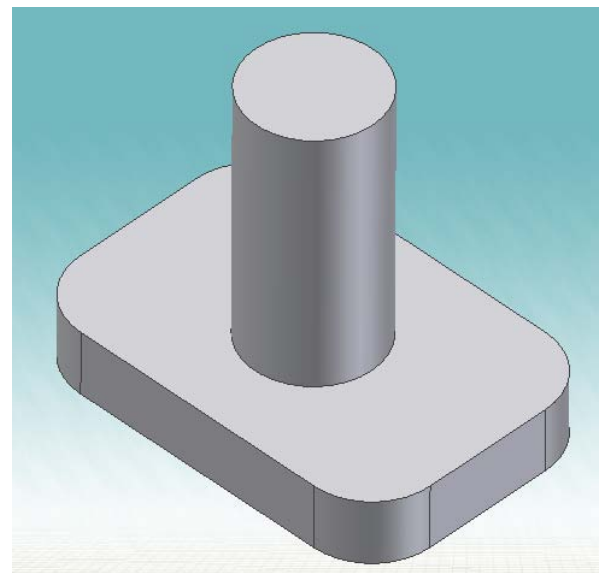
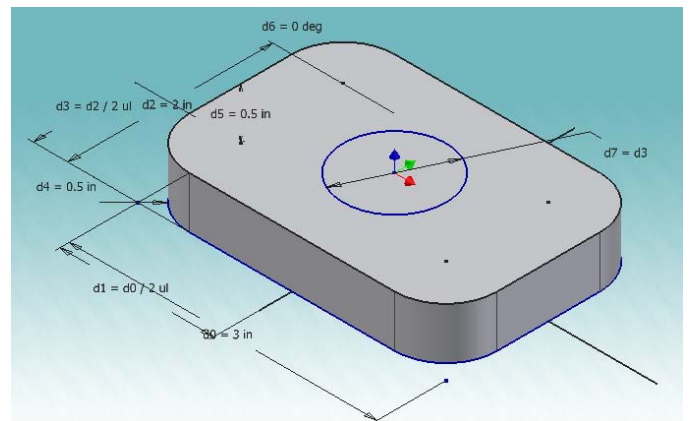
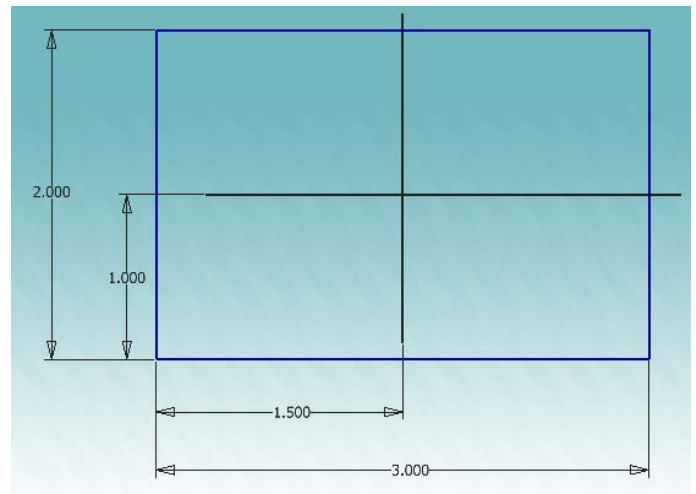
Start a secondary sketch on top of the Base feature.

Project the Center Point from the Origin folder.

Draw a Circle centered on the projected Center Point.

Dimension its diameter - tie it parametrically to the vertical half-dimension using Show Dimensions. It is not necessary to locate it dimensionally. Do you understand why?

Type E on the keyboard for Extrude. There is the possibility of two Sketches and so Inventor needs you to specify which - the area inside the circle or outside of it.



At this point it may be beneficial to the student to understand the three possible Boolean operations that are available. They weren't available before with the first Sketch because there was no solid geometry to interact with. They are Join, Cut & Intersect – graphically shown down the center of the dialog box. We will be adding material in the shape of a cylindrical Boss feature so we will use Join. Cut would be used if you were to extrude the circle down into the Base to produce a void or a hole. Intersect would create a solid based on the common volume of the two features – the Base and the extruded cylinder. Try to imagine what that would look like.

To continue, choose Join and specify a height of 2.00" for the cylindrical Boss.

Create another Sketch on top of the cylindrical Boss feature merely to create a Hole Center Point. At times this is not necessary – Inventor let's you drill holes at any point in a Sketch. We don't have that situation here though.

Using the Hole tool drill a tapped Hole using 3/4-12 UN for the Pitch, Major for the Diameter, 2B for the Class of fit.

The Hole tool requires that you position "Point, Hole Center" points on a sketch plane on the part or a work plane and then use the "Hole" tool on the Feature menu to specify the size and type. All the holes in one sequence have the same sizes and characteristics. The advantages of using holes over extruded circles are:

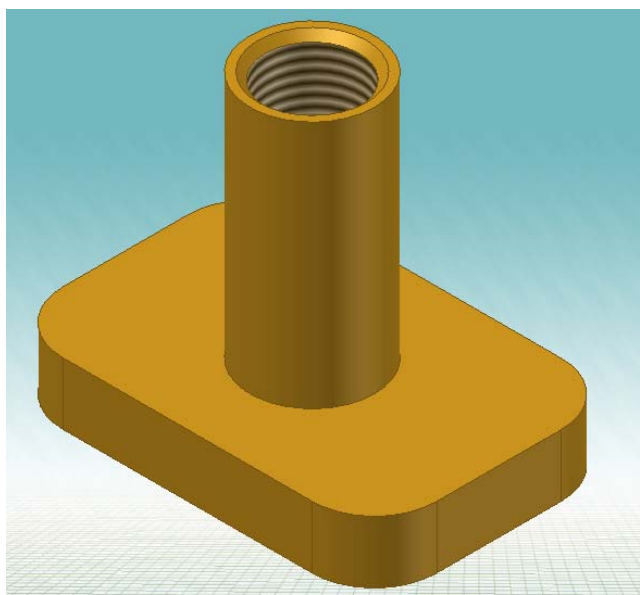
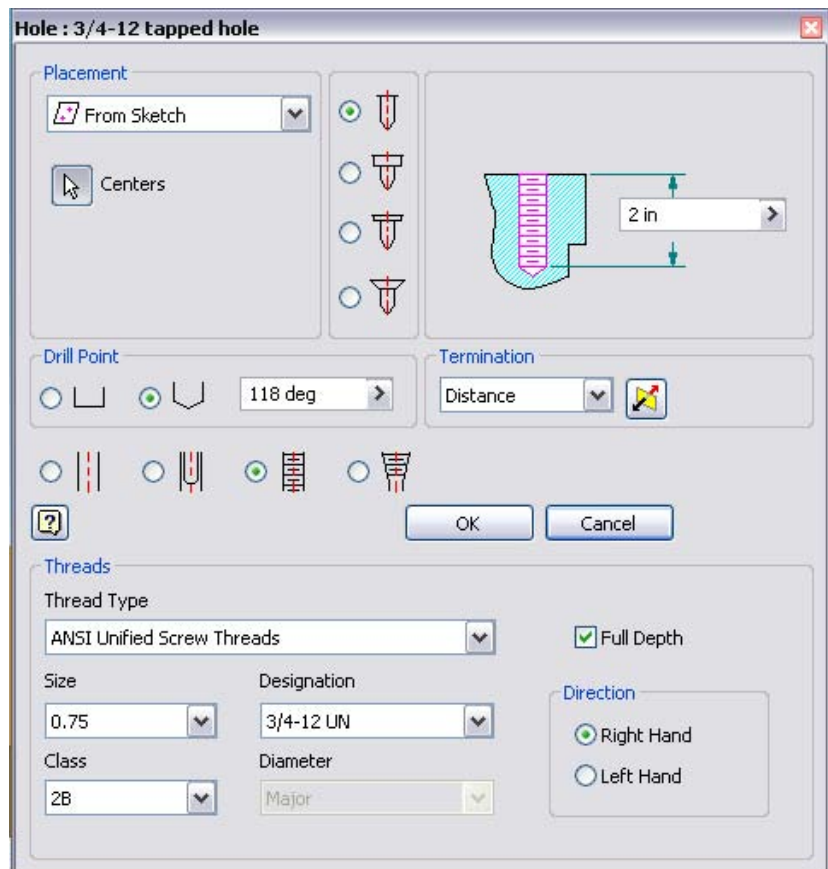
The holes can be drilled, counterbored or countersunk.

Thread sizes and specifications can be easily applied.

Threads will be displayed in the shaded image.

Threaded holes will display correctly in the part drawings.

"Hole Notes" can be automatically applied to the part drawings.



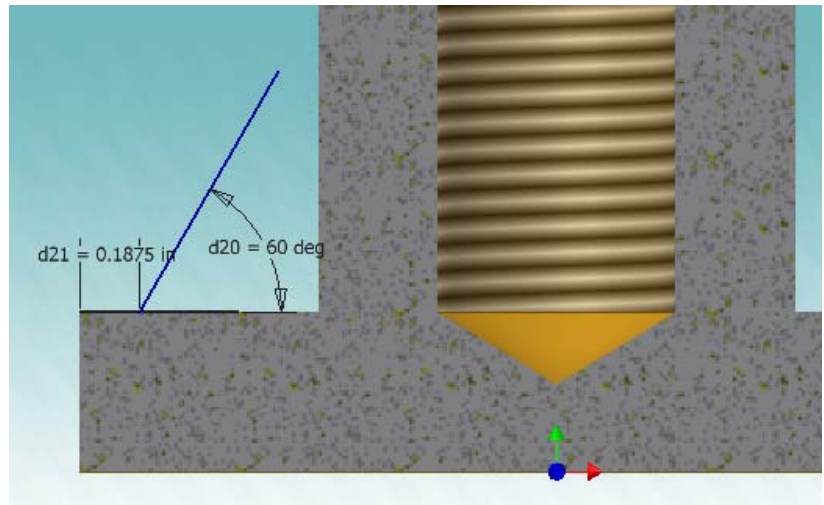
“Hole Tables” can be created in the part drawings.

There is a Thread tool also. This is meant for internal / external threads on extruded cylinders that were not created with the Hole tool.

Using the Chamfer tool put a .06” Chamfer on the opening edge of the tapped Hole.

Using the Color Style droplist on the right-hand side of the Standard toolbar change to Metal Brass or Beige Light.

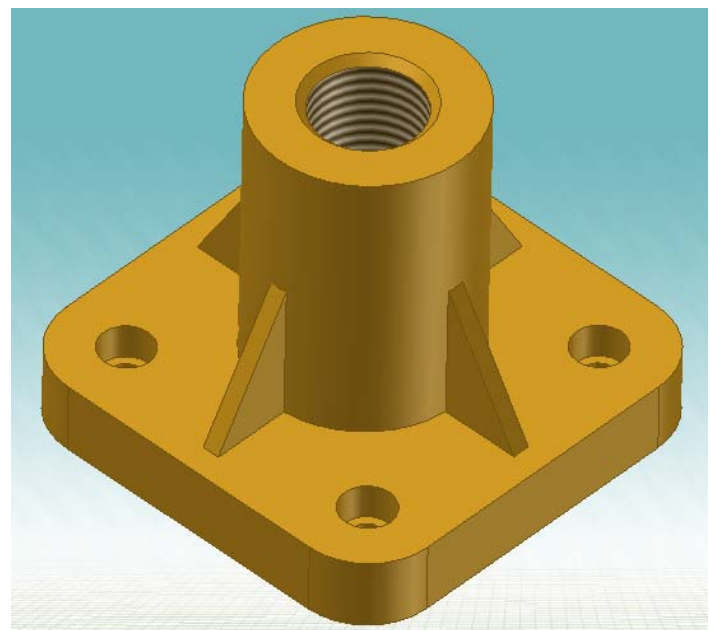
Select the YZ Plane of the Origin folder in the Browser and key ‘S’ on the keyboard. The F7 function key will slice the part visually with the Sketch Plane. Successive use of the F7 key simply toggles the effect on and off. F4 Rotate the screen to look at the other side of the Part and use F7 again. Notice that the visual slice is viewpoint dependent. Select the Sketch in the Browser and hit Page Up on the keyboard to get a plan display of the Sketch. Project the top edge of the Base (left side). Draw a line from touching the projected geometry and up and to the right. Dimension it from the edge (.01875”) and at a 60 deg. angle. Not necessary to fully constrain it. (refer to Figure)



Use the Rib tool to create a Rib feature. Use the midplane option and make it (.125”) thick. Use the Direction button to specify that the Rib should extrude down towards the body of the Part. A visual cue will help you. It should not be necessary to Extend the Rib feature so that it meets the solid geometry of the Part – Inventor will assume that.

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Select the Rib feature in the Browser by left-clicking on it. Choose the Circular Pattern tool from the Tool Panel and specify the cylindrical Boss feature as the axis for the Pattern. Choose a count of 4 for the Ribs and an angle of 360 to fill with the Pattern. Expand the << button and examine the Methods. The Creation Method dictates whether the members of the Pattern will all be alike or will adjust to conform to the Part’s perhaps changing geometry. The Positioning Method determines whether the angle value (360 in our example) is used to Fit all the members in it (yes in our example) or whether the value will be used to specify angular spacing between members.





Locate the Base feature in the Browser and Edit the sketch so that the Base feature is 3" square. Notice the changes.

Create a sketch on the top face of the Boss. Notice that the edges and the hole centers are automatically projected. This is a setting and is controlled by the Tools -> Application Options -> Sketch dialog box settings. Use the Hole tool to create four counterbored holes at the hole center points. Take note that this creates (4) holes, but only one Feature in the Browser – maybe this is good, maybe not. It all depends on your design intent.

Take a moment and examine what's been done. Now's a good time to rename the Features in the Browser.

Now it's time for the coup de grace. Usually the last operation is the application of Fillets. Refer to the figure as you do this. You need only use the Fillet tool once and create one Fillet Feature in the Browser.

The best way to do this is to use the Loop selection to pick the bottom of the cylindrical Boss and the top face's edge of the Base, then use the Feature selection to pick the Rib Feature as a whole and then the Pattern Feature. Choose OK and you should have it done. Oila!

We are now going to Engrave some text on our part as if we were stamping a part number on it. On the left-front side of the Base feature left-click on the face. Press 'S' on the keyboard and you will be in Edit Sketch mode.

Draw a construction line from the midpoint (green dot) of the left vertical line to the midpoint of

the right vertical line. Draw another construction line from the midpoint of the top horizontal line to the midpoint of the bottom horizontal line. These will be used to center the Engraving text in the center of the face. Pick the Create Text tool from the 2D Sketch Panel and pick a point anywhere in the face. In the dialog box type in "PART NO. AU-PBY24" and choose horizontal and vertical justification from the buttons along the top of the dialog box. Pick OK to exit the dialog box. You'll notice a frame around the text. You'll dimension to this frame to control the size of the text string. Use a value of 1.50 for the length and .125 for the text height. The location of the text string will be geometrically constrained by using Coincident constraints between the horizontal and vertical midpoints of the text frame and the





horizontal and vertical construction lines created earlier. Select the Auto Dimension tool to ensure that you've eliminated all of the constraints needed. Pick Return to exit out of Edit Sketch mode. Select the Emboss tool from the Tool Panel. The same tool is used for Engraving (sunken letters) and Embossing (raised letters). The Profile will need to be specified carefully. You do not want to select the large face – only the text letters themselves. Refer to the Figure using a value of .05 for the depth of the letters and the other selections necessary – Engrave and direction.

The Emboss Feature should be in the Browser – right-click on it and choose Properties from the context menu. Change the Feature Color Style to Red as shown in the Figure below.

In Closing

I hope that everyone agrees that effective Part Modeling is something that can be learned and applied to make your part designs smarter and more efficient. It has been shown that Autodesk Inventor is packed with the tools and features that are required to produce today's complex parts in today's complex world. I hope that you now have the knowledge you need to go back to the office and design it better, faster and cheaper.

We sincerely thank you for your time and attention,

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