



AUTODESK UNIVERSITY 2006

Realize Your Ideas

Nov 28 - Dec 1 The Venetian Hotel Las Vegas, USA



Autodesk

Becoming an Autodesk Inventor® Professor in 90 Minutes

J.D. Mather – Pennsylvania College of Technology

MA13-3 Looking for that one tip that will justify your time and expense of attending AU. For the Inventor user this class is where you will find it. A no-nonsense rapid fire delivery of productivity tips and problem solving strategies gained from teaching hundreds of students how to use Inventor. If your co-workers don't call you the Inventor Professor after this class you should consider another major.

About the Speaker:

J.D. Mather is an assistant professor of CAD and Product Design at Pennsylvania College of Technology in Williamsport, Pennsylvania. He has taught AutoCAD courses for 12 years and has been an Autodesk Inventor Certified Expert since release 7. His advanced Inventor Surface Tutorials are studied by users around the world.

http://home.pct.edu/~jmather/content/DSG322/inventor_surface_tutorials.htm

He was the winner of the Autodesk Inventor 11 Shape Description contest.

<http://home.pct.edu/~jmather/stapler%20assembly%20-%20jd%20mather.dwf>

Previously he worked in industry for 15 years.

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More AU 2006 class information at www.autodesk.com/auonline

<http://au.autodesk.com>

Unabridged document <http://home.pct.edu/~jmather/AU2006/MA13-3%20Mather.pdf>

Dataset <http://home.pct.edu/~jmather/AU2006/MA13-3%20Dataset.zip>

INTRODUCTION

The Obvious and the Not so Obvious

This is an intermediate class, but everyone comes from different backgrounds so I will resist the temptation to make certain assumptions of prior knowledge. I apologize for the stuff you already know. I expect everyone here will learn at least one new feature. Research shows that I should present 7 ± 2 new items. If you didn't know how to touch type (or have basic experience with Inventor) I might have to go much slower as you would be struggling just to find the keys. However if you know the keyboard (Inventor) I can go faster. I assume in an intermediate class you have some familiarity with the Inventor interface so I am going to go rather rapidly. Don't worry about memorizing everything – it is all available through the “collaboratory”.

There is a Take-home Assignment for this Class

Place checkmarks beside anything new to you. Your assignment is to email your checklist to me and I will post a graphic of the most common selections. So everyone must pay attention.

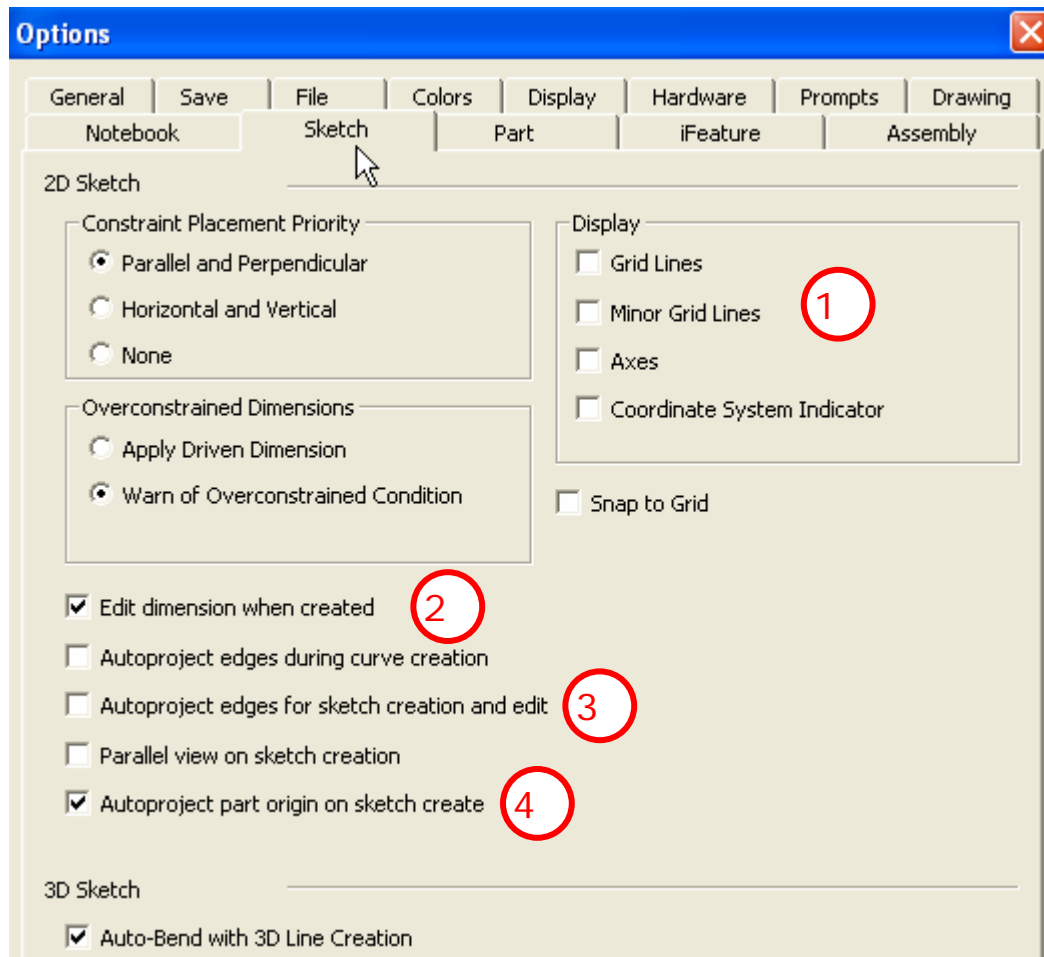
Inventor Shortcuts for AutoCAD Users

Is taking shortcuts is faster? I could be wrong, but I doubt it. (Shaq can get away with that and nobody argues). Many people ask me why they have to worry about constraints. They got along just fine in AutoCAD without them. For those of you coming from AutoCAD what if I said, “Why do I have to worry about osnaps, you can't see it on the print anyway?” Would you want to edit my drawings? How many of you had to take a machine tool class in college? Remember seating a part on parallels in a vise with a dead-drop hammer on the Bridgeport mill? Working without constraints is roughly the equivalent of taking that part out of the vise to measure and then trying to place it back in the vise to take another cut. How much control do you have? Constraints give you absolute, predictable control.

I have taught hundreds of students how to use AutoCAD, MDT and Inventor over the past 12 years. The students with the best quality, and quantity, of work were always the most disciplined. I have not seen any evidence that working without constraints is faster than working with constraints. I could be wrong, but I doubt it.

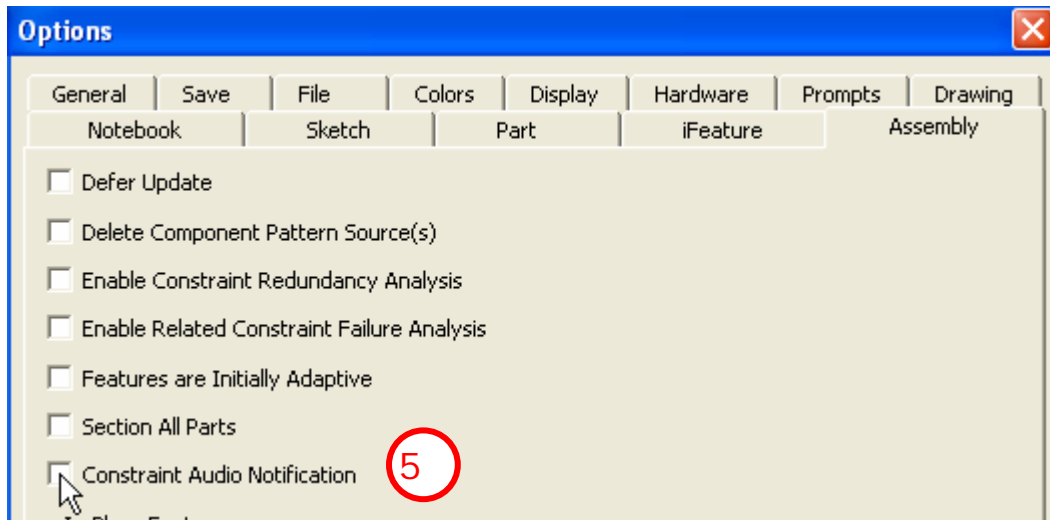
Setting up the Environment

Before we get into the class proper, we need to do some housekeeping so that we are all using the same “keyboard” (user interface).

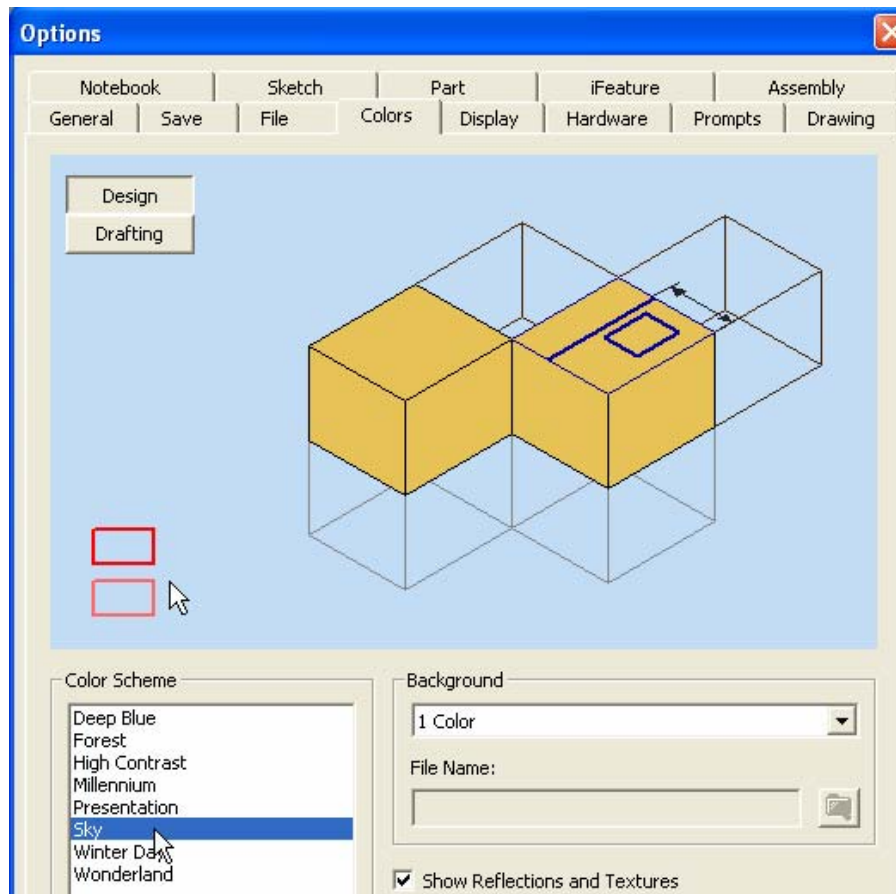


Sketching is the foundation of Inventor. I am pretty particular about how I set up the sketch environment

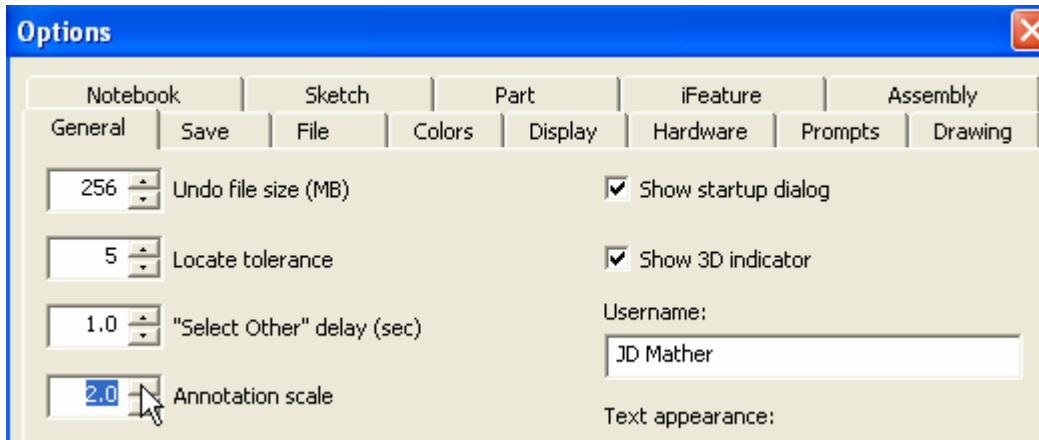
- 1. I turn off the **Grid Lines**.
I like the clean screen graphics window. In Inventor we use a sketching technique rather than the direct distance entry technique of AutoCAD. The rough equivalent workflow in Inventor is achieved if you
- 2. Checkmark the **Edit dimension when created** box.
- 3. Uncheck the **Autoproject edges for sketch creation and edit**. (I'll explain why later.
A new option in Inventor 11 is the Autoproject part origin on sketch create. I like to think Autodesk included this feature just for me after several years of persistent requests. (Thanks go to Neil Munro for the precursor add-in.)
- 4. Checkmark the **Autoproject part origin on sketch create**.



- 5. On the Assembly tab uncheck **Constraint Audio Notification** or change the Program Files\Autodesk\Inventor 11\Bin\Connect.wav file to something really cool.



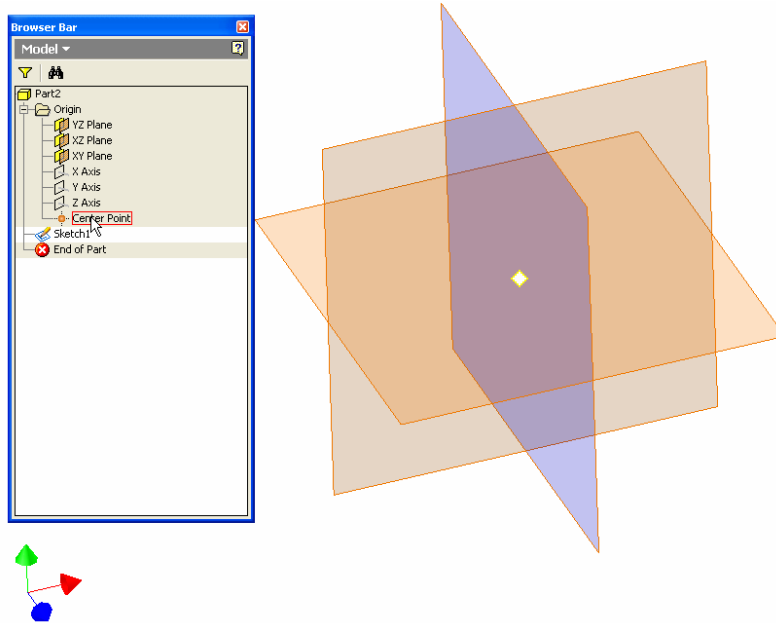
- 6. For projection in a training classroom the **Sky Color Scheme** seems to give the best contrast.



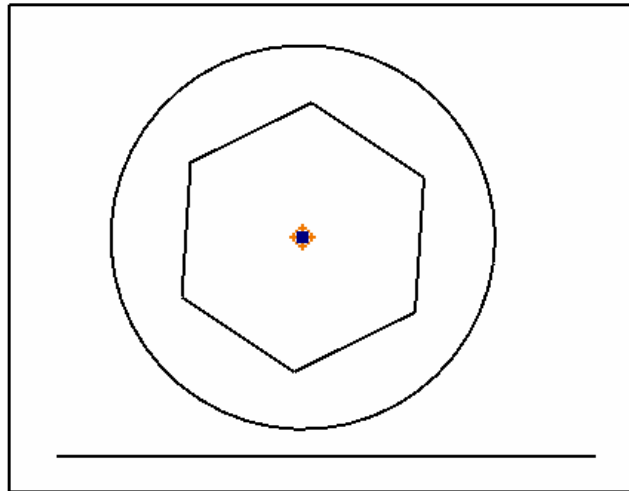
- 7. If you are projecting in a classroom or if your eyesight is poor you might want to increase the **Annotation scale** on the General tab. This will make sketch dimensions easier to see from a distance.

Sketching is the Foundation

In this class I will use the using the Base Orphan Reference Node (**BORN**) Technique.



- 8. Expand the Origin folder and RMB turn on the visibility of the **Center Point**. Use the origin geometry to your advantage. It has no parents and cannot be deleted. It will always be there for you. Use it to anchor your work. Use it for symmetry. Use it for consistency. You don't have to have it visible to use it. Just Use it!



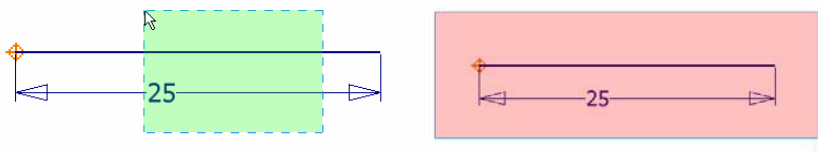
- 9. Create your first line, circle, rectangle or polygon and dimension it immediately. This will give you a sense of scale for the rest of your sketch. This step is particularly important for AutoCAD users new to Inventor who are used to using direct distance entry. As we start adding constraints your sketch could get very difficult to control if it isn't to a relative scale. We can avoid perhaps the biggest frustration for AutoCAD users' right here.



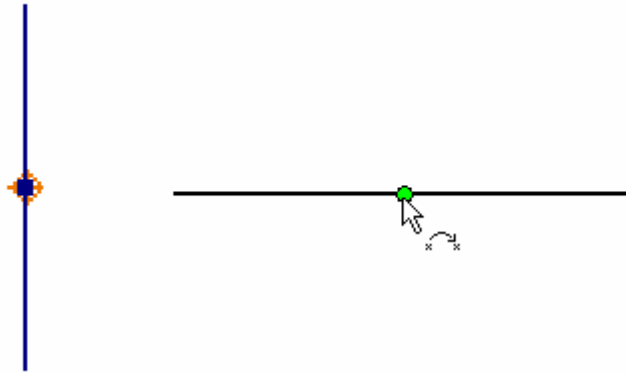
- 10. If your origin is projected you should get a green coincident “hard snap” to the origin centerpoint. If it is not projected it might appear to snap, but this is really a “soft snap” that does not result in a constraint.



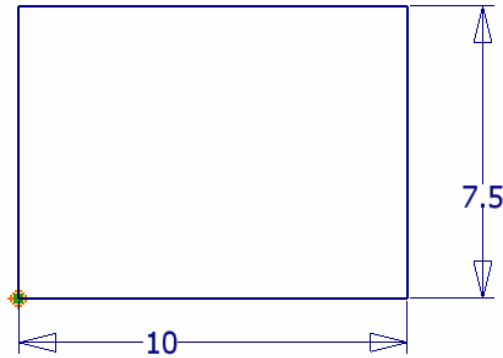
- 11. After adding a dimension the sketch line changes color indicating a fully constrained sketch (the actual color change depends on your background color scheme – for printing this document I used the white presentation background, but I would normally use the sky background in a classroom). An implied coincident constraint was added when we snapped to the projected origin center point and an implied horizontal constraint was added when we clicked the endpoint for the line tracking in a horizontal direction.



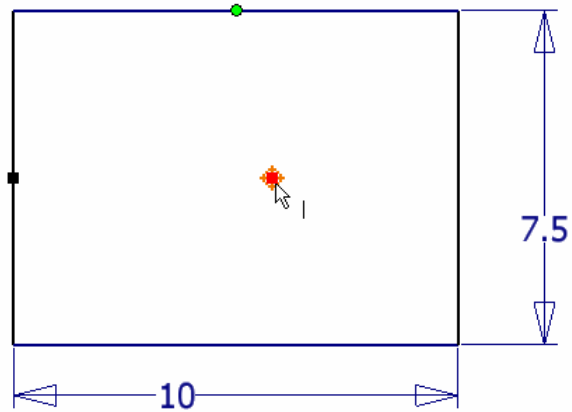
- 12. If you need to select the sketch or part of the sketch you can use a crossing window or window selection.



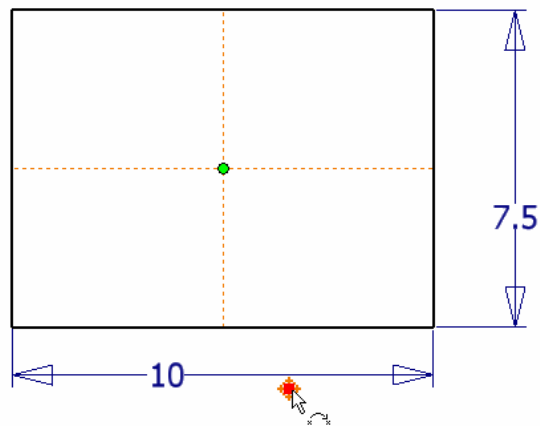
- 13. I often use a vertical or horizontal line of symmetry. I create the line close to the origin and then add a coincident constraint between the midpoint of the line and the projected origin.



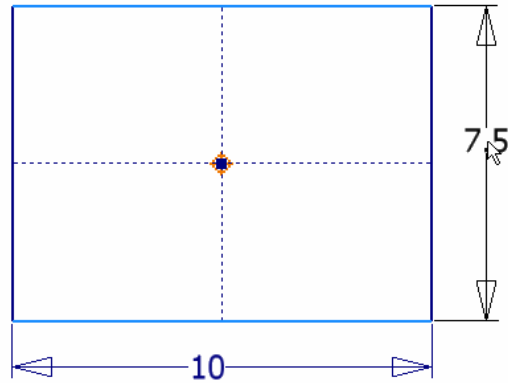
- 14. A rectangular shape is a common starting point for a model. One option is to simply start one corner of the rectangle at the projected origin. (From here on I will simply refer to the origin – it is assumed that you will ALWAYS project it.)
- 15. Coming from an AutoCAD background I used to always use this method as I frequently create Rapid Prototype models from stl files. In AutoCAD if your model isn't in the positive xyz octant you get errors trying to export the stl file. In Inventor this is not a problem. Inventor will place the geometry in the positive octant when saving as a stl file. *Tip: The Inventor 11 DWF Extension will allow you to save an assembly (iam) directly to a stl file.* The DWF extension is available free to subscription customers.
<http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id=7430881>



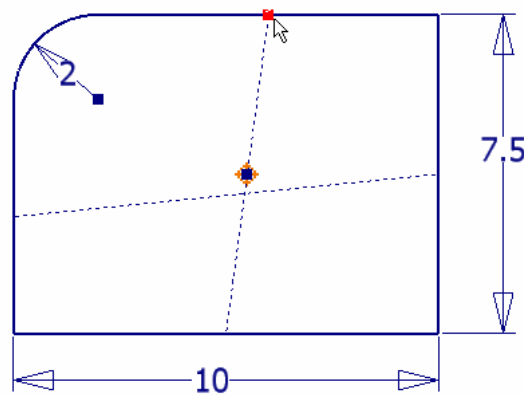
- 16. Putting the corner of a rectangle at the origin does not, however, make the best use of the origin geometry for **symmetry**. Some modelers apply horizontal and vertical constraints between the midpoints and the origin.



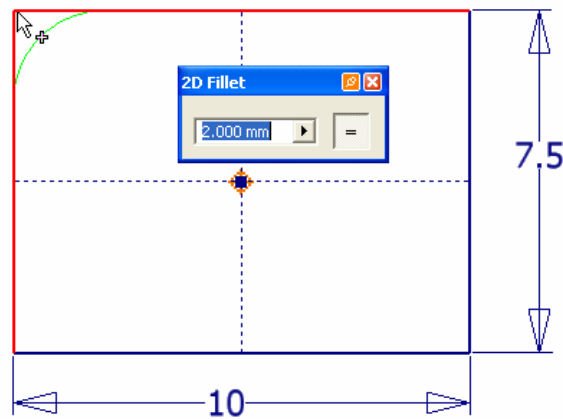
- 17. You could use bisecting **construction lines** as a **visual indication** that can't be missed later on if the sketch needs editing. (Remember back on the drawing board when you were instructed to not erase construction lines?)



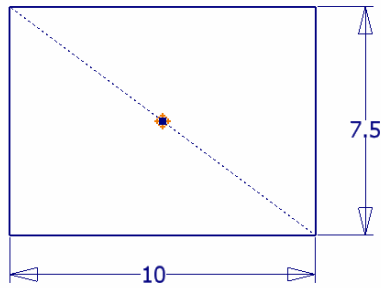
- 18. Notice that I prefer to dimension from side to side rather than single lines as that method picks the endpoints which could be easily lost in editing.



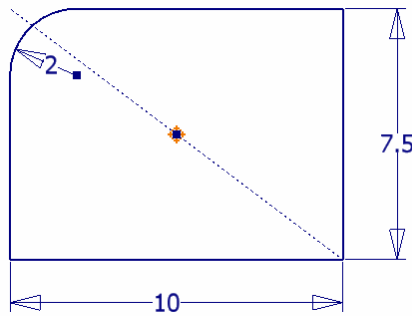
- 19. The problem with using midpoint coincident constraints is that they can shift.



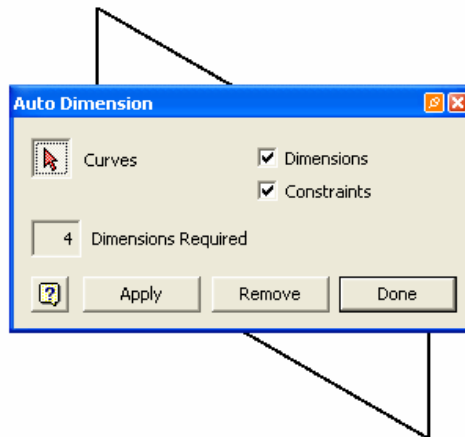
- 20. Tip: Did you notice when I created the fillet I selected the corner instead of two edges. Either way works for fillets and chamfers when there are only two edges at the intersection.



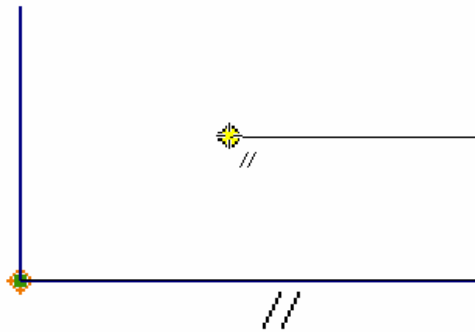
- 21. My favorite method of starting a basically rectangular shape is to use a diagonal construction line constrained at the midpoint.



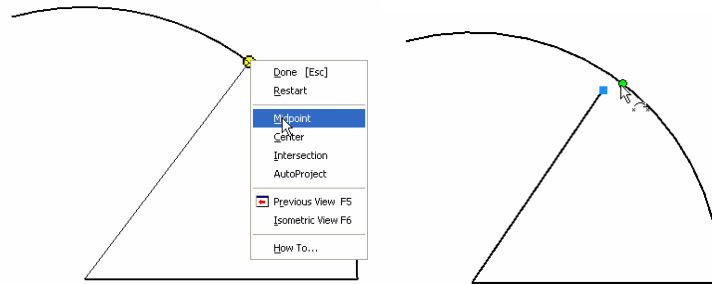
- 22. Anytime you are not sure how a sketch will react, click and drag corners. **Very Important Tip: This is the easiest way to figure out the remaining constraints or dimensions that are needed to fully constrain a sketch.** If you are training others make sure demonstrate this.



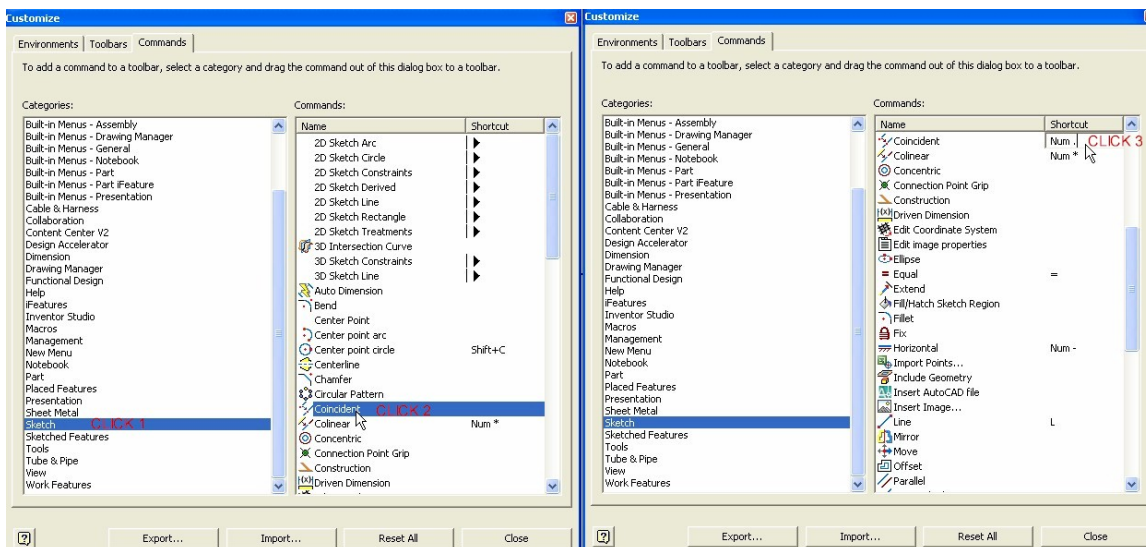
- 23. Many books and trainers emphasize using the Auto Dimension command for help in figuring out the remaining constraints or dimensions that are needed to fully constrain a sketch. However, I don't think this helps in understanding geometry.



- 24. Sometimes implied constraints can be a nuisance. What if I don't want this line constrained parallel, horizontal or perpendicular. You can override automatic placement of constraints by holding the Ctrl key.

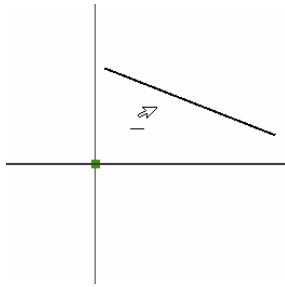


- 25. Here is an interesting tip. I can't create a line to the midpoint of an arc even if I RMB and choose the Midpoint option, but I can add a coincident constraint to the midpoint.

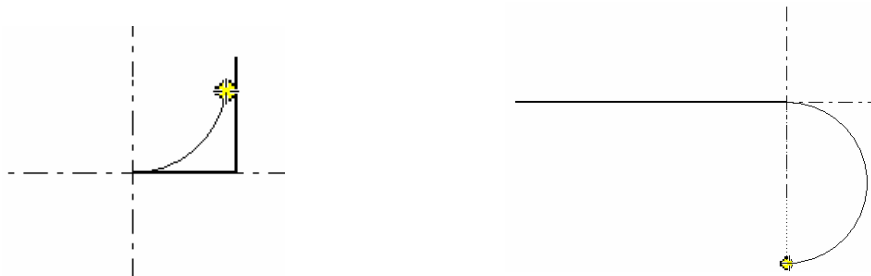


- 26. Tip: Under Tools>Customize>Commands you can set up your numerical keypad for quick access to sketch constraints. C:\Program Files\Autodesk\Inventor 11\Samples\Customized UI\Sample Shortcut Scheme\shortcuts.xls

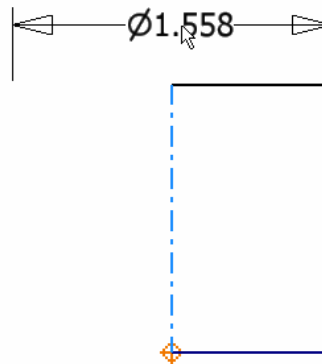
There are many pre-defined shortcuts, for example X for **Trim** and once in the trim command if you hold Shift you can **Extend**.



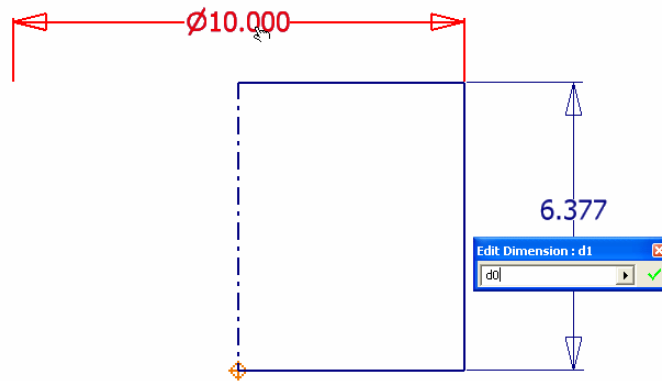
- 27. If I had left on the horizontal and vertical grid axis notice that the Horizontal grid axis is thicker.



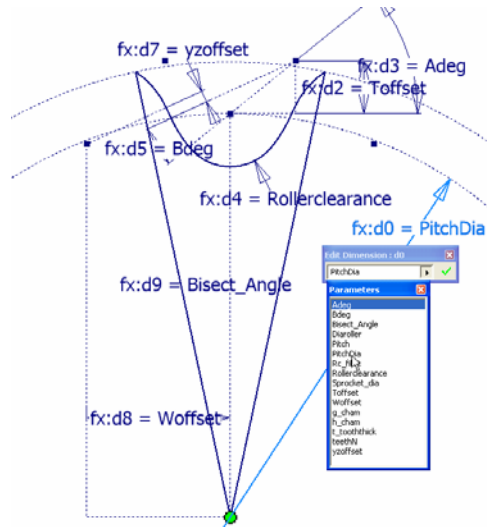
- 28. You can use the shortcut **L** to create a **Line** and an arc from a line endpoint by click dragging from the endpoint while in the line command.



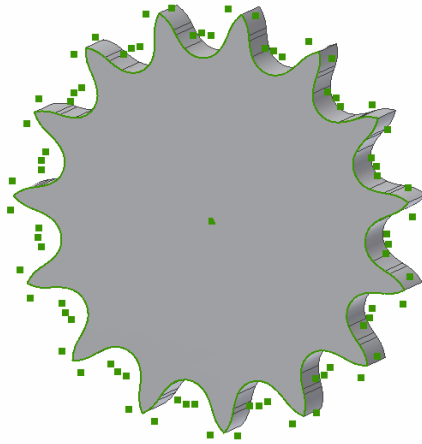
- 29. Use the shortcut **D** to start the **Dimension** command. If you dimension to a centerline type you will get a diameter dimension.



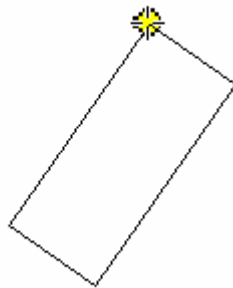
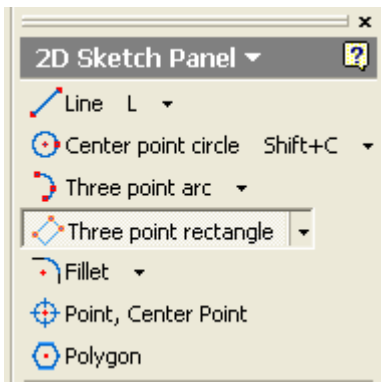
- 30. You can make one dimension dependant on another dimension by simply clicking on the source dimension while editing the dependant dimension.



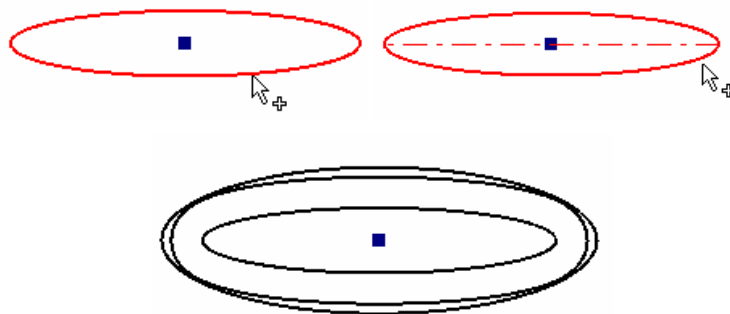
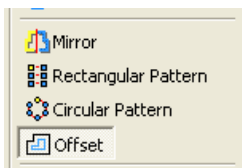
- 31. Use **List Parameters** and set dimensions to predefined variables. You can use units (in, mm) PI and functions in formulas. Your formula is not lost after accepting.



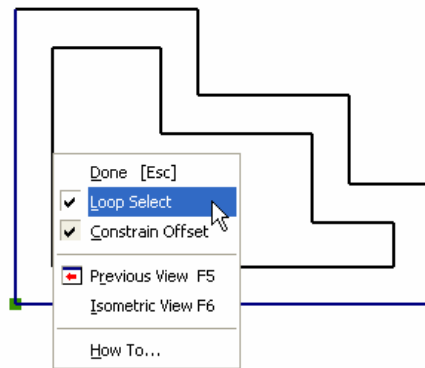
- 32. If you leave Autoproject for sketch creation turned on you can get unnecessarily complex sketches.



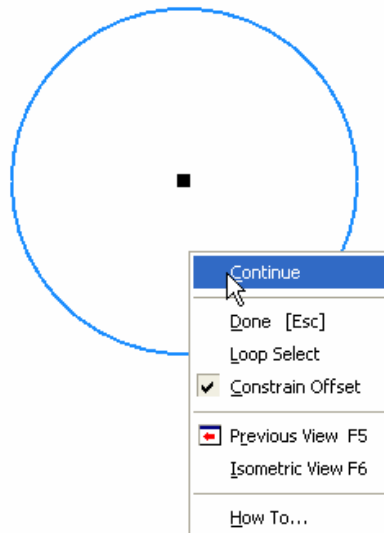
- 33. Don't forget the **3-point rectangle tool**. It can come in very handy. The lines are created with perpendicular constraints. If there are other autoconstraints that you don't want, simply hold the **Ctrl** key while sketching.



- 34. When offsetting an ellipse you will get different results depending on whether one of the axis is highlighted or not.



- 35. RMB to get additional options when Offsetting.



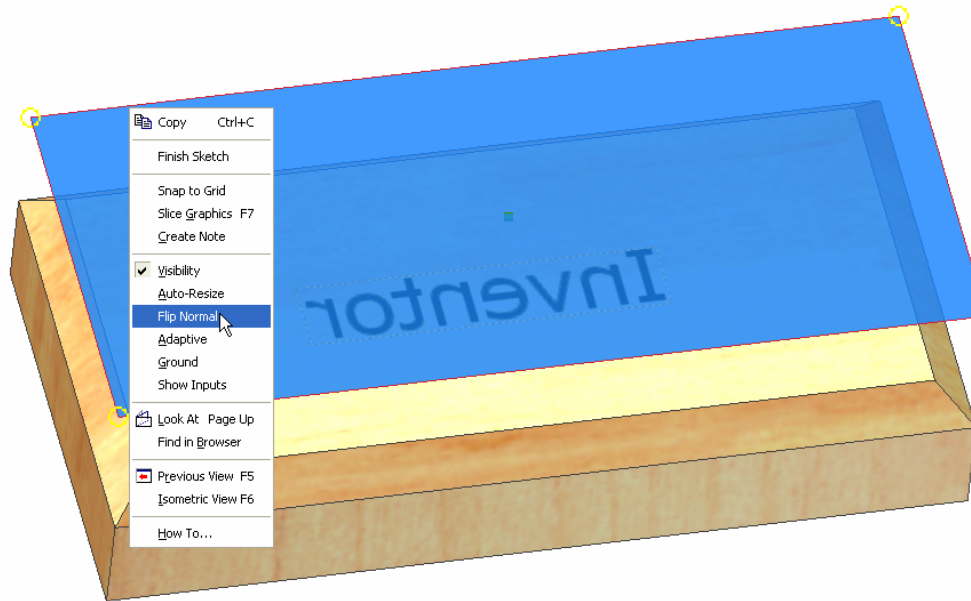
- 36. If you have Loop Select turned off you will have to RMB Continue to offset a circle or ellipse. **Watch out for this one.**



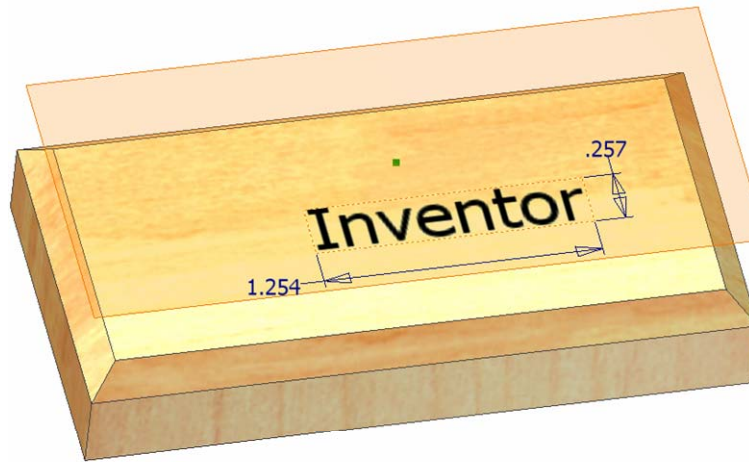
- 37. Autoproject edges can cause difficulty in selecting text sketches, but you can use the Select Other tool to get the text.



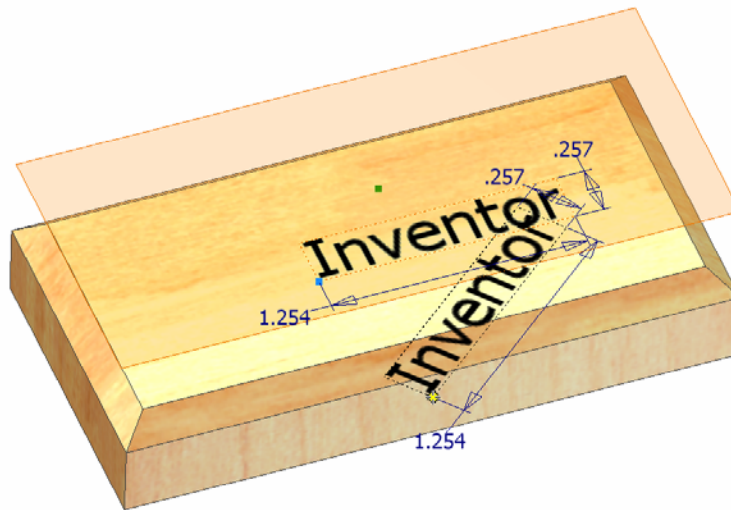
- 38. Often you will want to create text sketch on an offset workplane. A workplane has a front side and a back side. The backside is a darker color. Notice that the text is placed right-reading on the front side.



- 39. You can Flip Normal (the vector perpendicular to the front side of the workplane) by RMB on the plane.



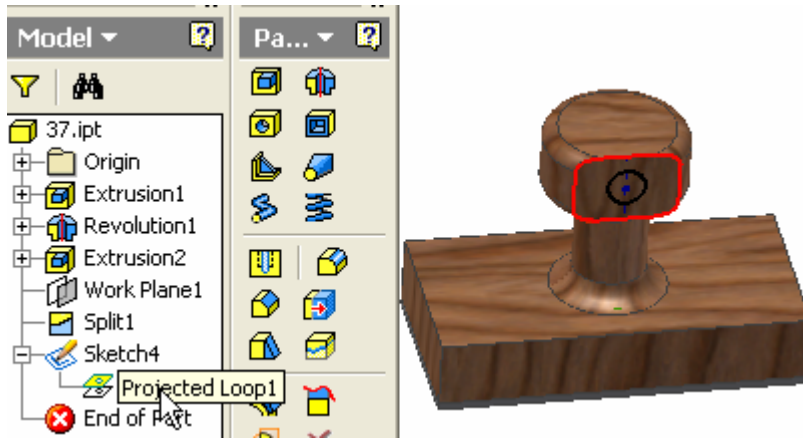
- 40. Whenever you place sketch text you should immediately dimension the text box. (The actual dimension values don't matter).



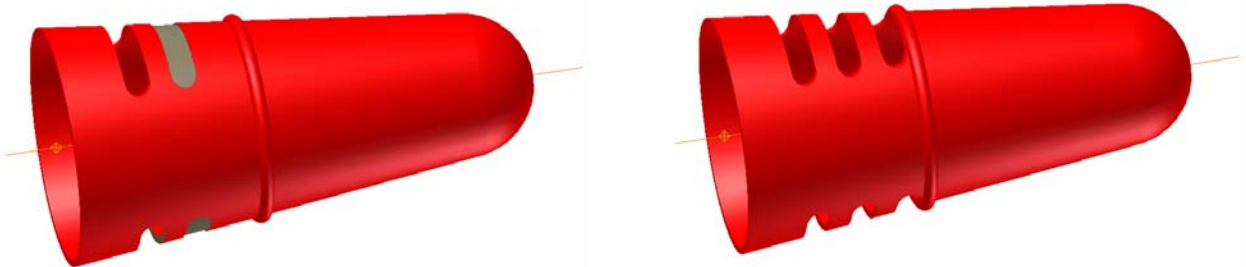
- 41. You can then grab and drag the lower left corner to roughly reorient and then constrain just like any other sketch geometry.

Creating Robust Model Features

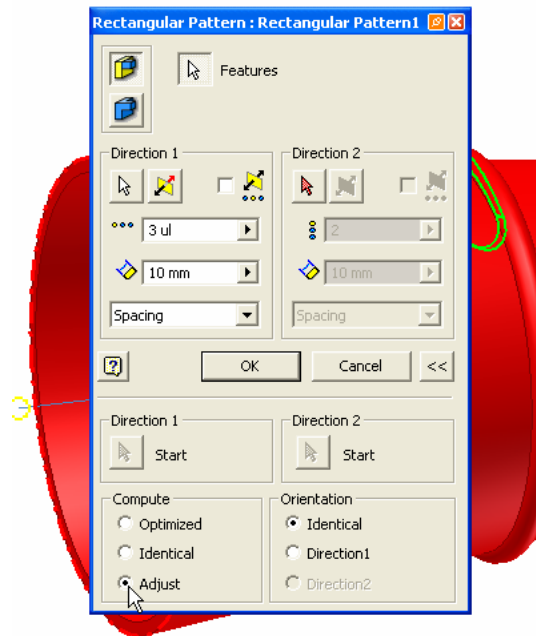
In my experience almost every design undergoes changes. It is faster make the effort to set up a robust model than it is to edit a poorly done model.



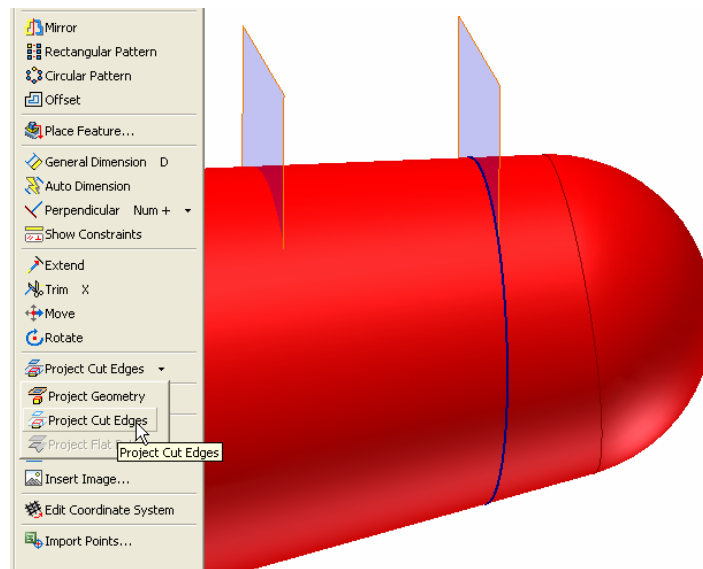
- 44. A **Projected Loop** is created by selecting a face with the Project Geometry tool rather than selecting individual edges. I would recommend not using projected loops. For example, you can't copy a feature created with a Projected Loop. If you are a trainer this is the kind of thing that will drive you crazy in a class of 25 people where some have projected the edge and some projected the loop.



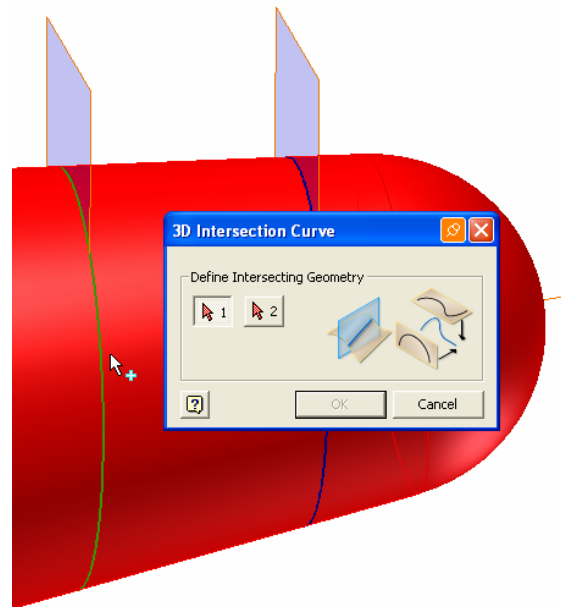
- 45. When patterning cut features you may get unexpected results where an expected material cut is not made.



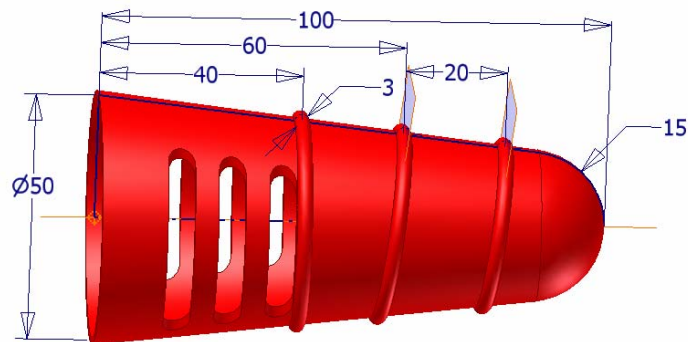
- 46. The trick to getting all of the slots in this model is to expand the Pattern Feature dialog box and **Adjust** to model.



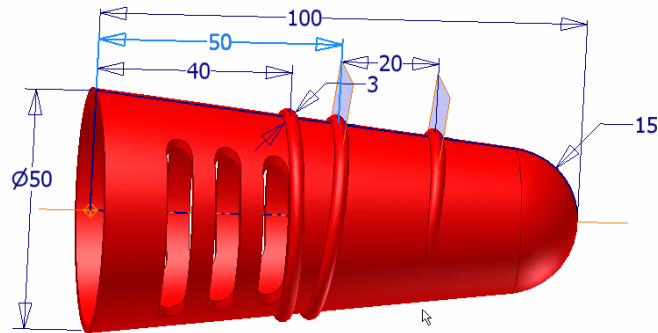
- 47. You can create a 2D sketch by **Project Cut Edges** where an inclined workplane intersects a conic feature.



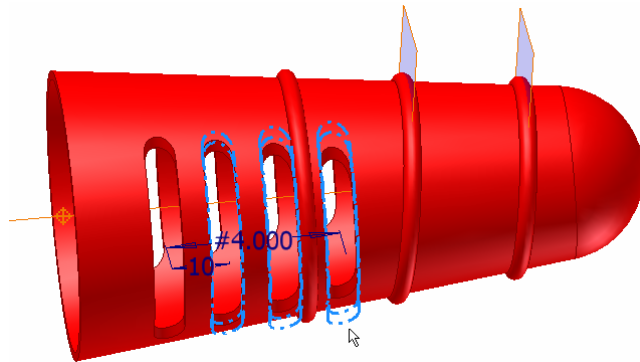
- 48. Or create a similar 3D sketch using the 3D intersection between an inclined workplane and a conic feature.



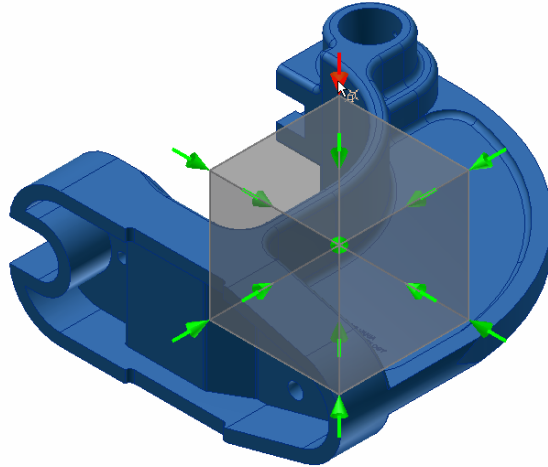
- 49. Now I **Sweep** two circles along the 2D and 3D intersection paths. To change the dimensions on the original sketch I simply Double Click on the Revolution1 feature in the browser. (I do not need to turn on the visibility of the sketch or edit the sketch.)



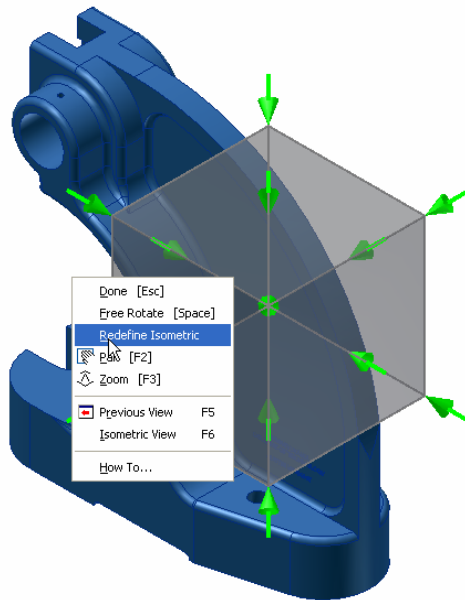
- 50. I have changed the 60 dimension to 50 and **Updated** the part. Notice that one of the revolved features updates as we might expect and the other one does not. (*When changing dimensions effecting work geometry you must do an Update, and I believe sometimes a Rebuild All to see the effect of the change.*) The lesson is DO NOT use Projected Cut Edges in a 2D sketch if there is a way to get the desired geometry with a 3D sketch.



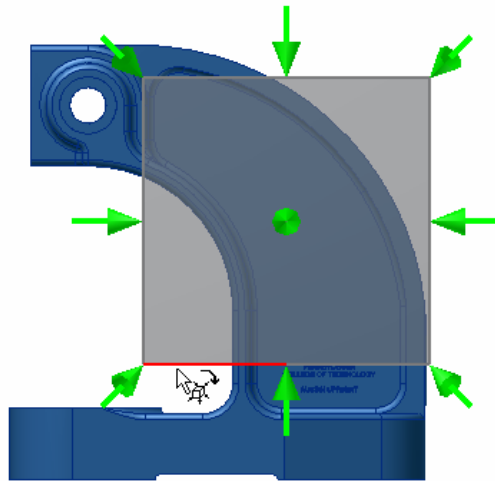
- 51. Remember – you can change feature parameters simply by double clicking on the feature in the browser. Here I have changed the # of pattern instances.



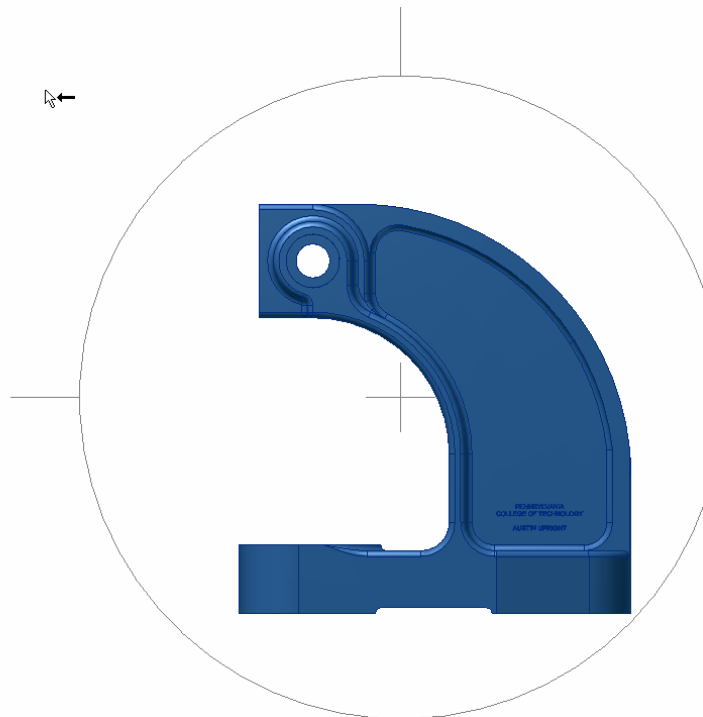
- 52. Sometimes when you open a part it is not oriented at the best viewing angle. Start the Rotate view command and hit the Spacebar and a **common view glass box** appears. Select the arrow for the view you want.



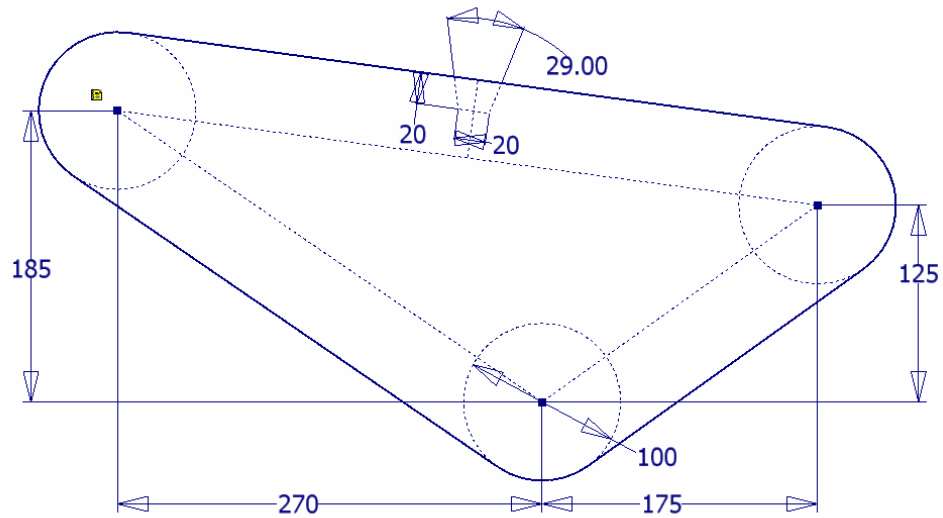
- 53. While the Common Views glass box is visible RMB and select **Redefine Isometric**.



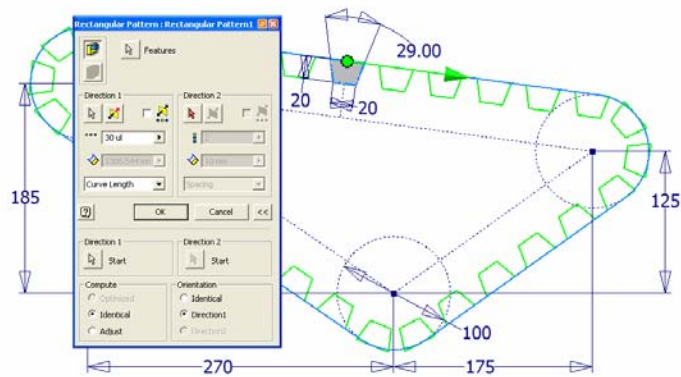
- 54. You can rotate the view of the model 90° at a time by clicking on one of the **edges** of the glass box. The icon next to the cursor indicates which way it will rotate.



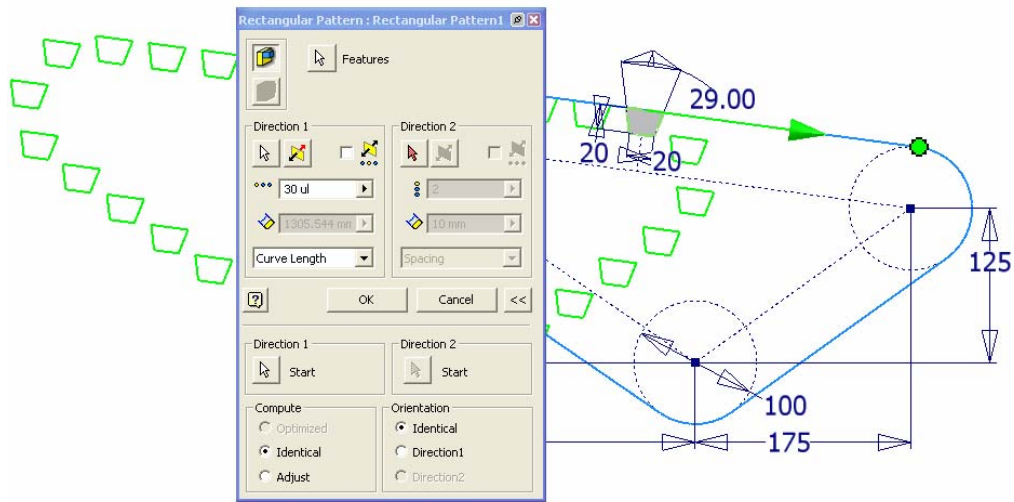
- 55. You can get back to free rotate by hitting the Spacebar again. You can center the Rotate view with a long click. Leave Rotate command by moving the cursor out of the arcball as shown and clicking in the graphics window or by hitting Esc.



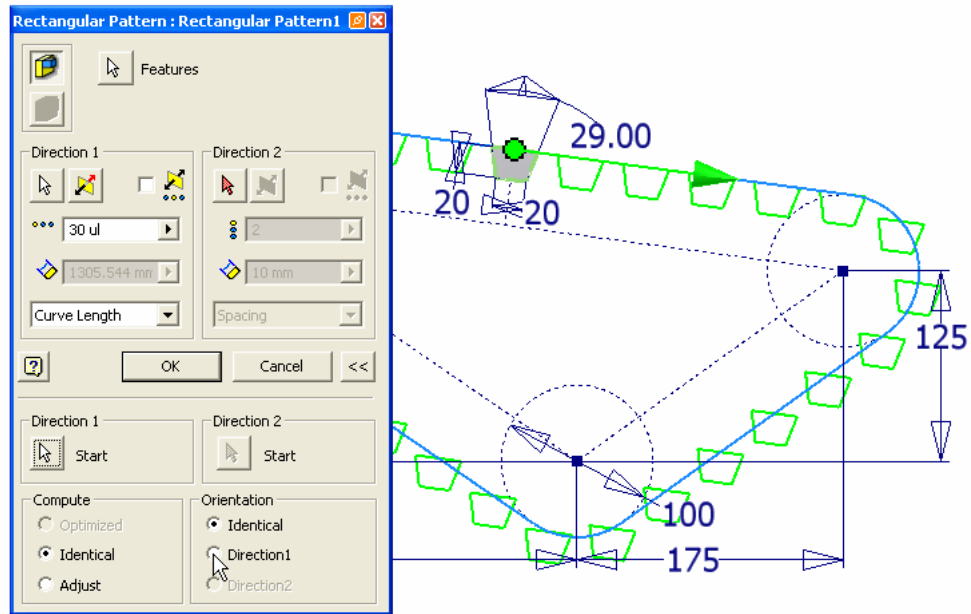
- 56. The **Rectangular Pattern** tool can be used for more than just rectangular patterns.



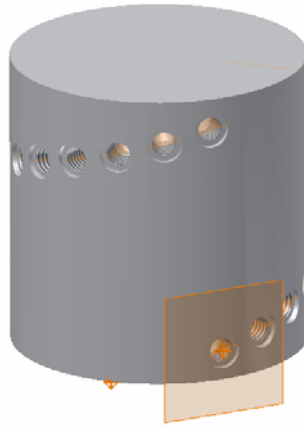
- 57. A non-rectangular pattern can be created by selecting a pattern curve.



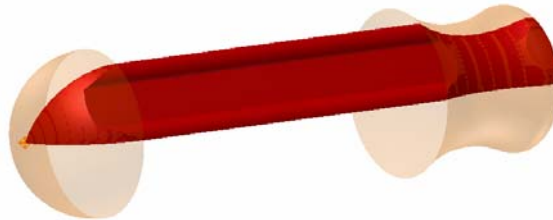
- 58. The most common problem experienced by those who know this feature is somehow possible, but they can't get it to work right is that their pattern is shifted and they forget that they can choose a new start point.



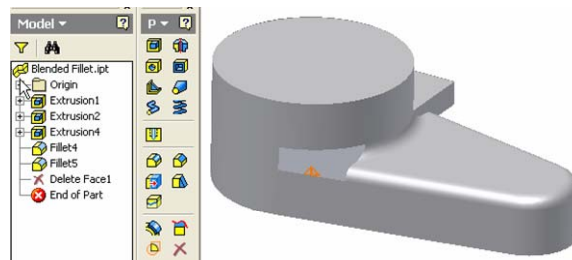
- 59. The second most common problem is that they forget how to reorient the pattern if needed. (Perhaps because the option situated under the Direction 2 area of the dialog box. Direction 2 is of course for a rectangular pattern.) The Rectangular Pattern now has a midplane option.



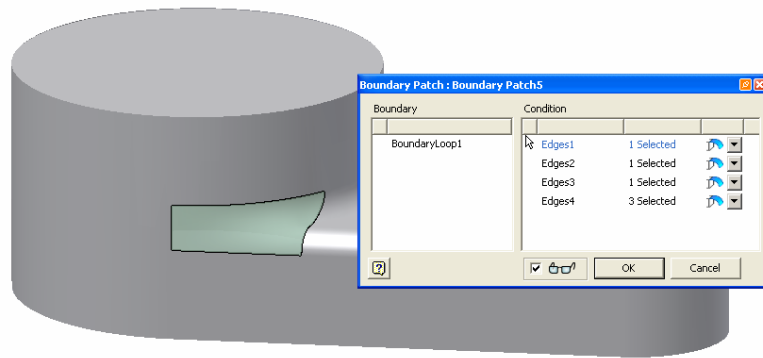
- 60. The **Coil** command in Inventor 11 now allows creation of a surface. The edge of the coil surface can be used for a pattern curve.



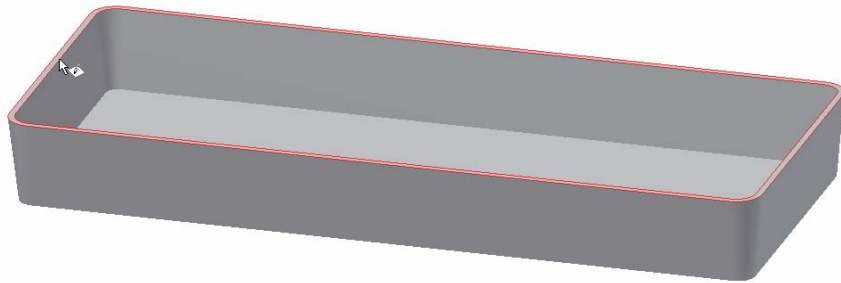
- 61. Many people associate **surface modeling** with complex shapes, but they can also be used in the creation of fairly simple geometry.



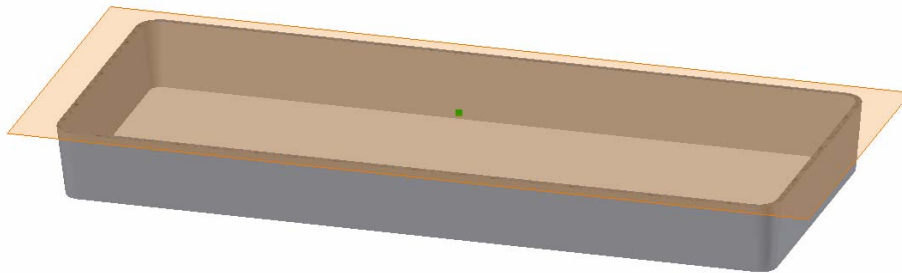
- 62. If you start working with surfaces be sure you understand when you have a surface and when you have a solid. *(A surface feature will be translucent or a surface created from a solid by the Delete Face tool will show at the top of the browser as a surface icon. If you use the Delete Face tool solid options will be disabled until you stitch your surfaces back to a solid.)*



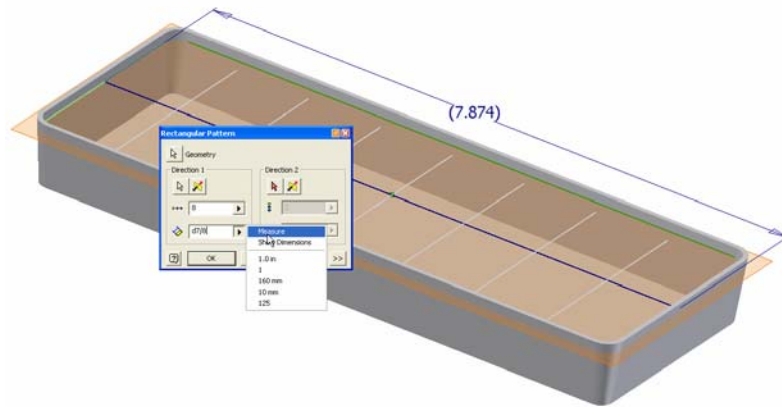
- 63. A hole in a surface model can be patched with the **Boundary Patch** tool.



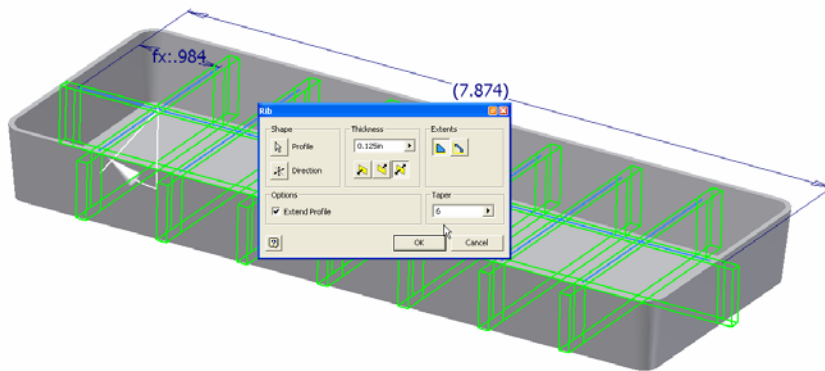
- 64. You can create an **offset Workplane** and a new sketch at the same time by click and dragging from another planar face.



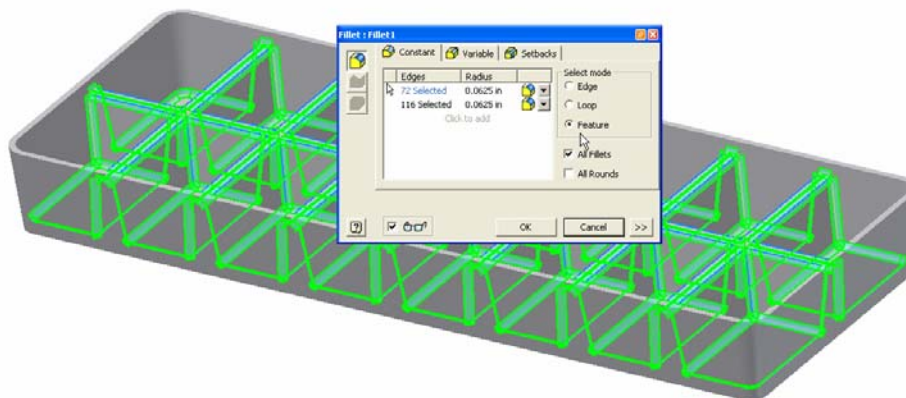
- 65. Slice into a part the current sketch plane by hitting F7 or RMB select Slice Graphics.



- 66. Using Reference dimensions is preferred over Measure as the reference dimension will update if the part changes.



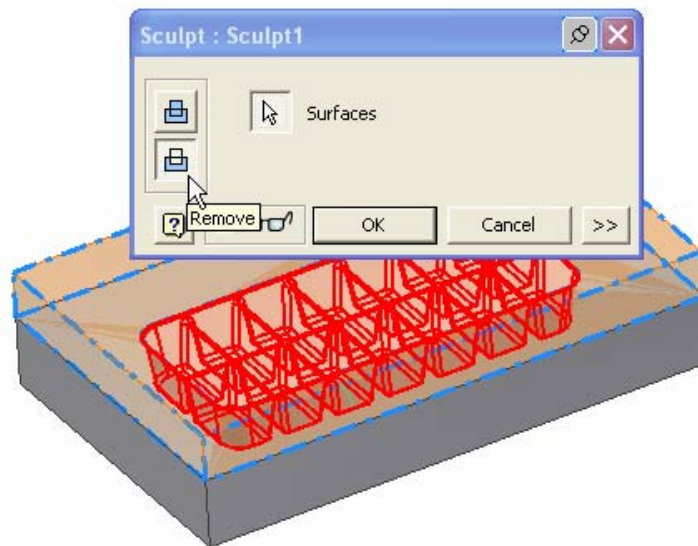
- 67. The **Rib Feature** tool now has a **Taper** option.



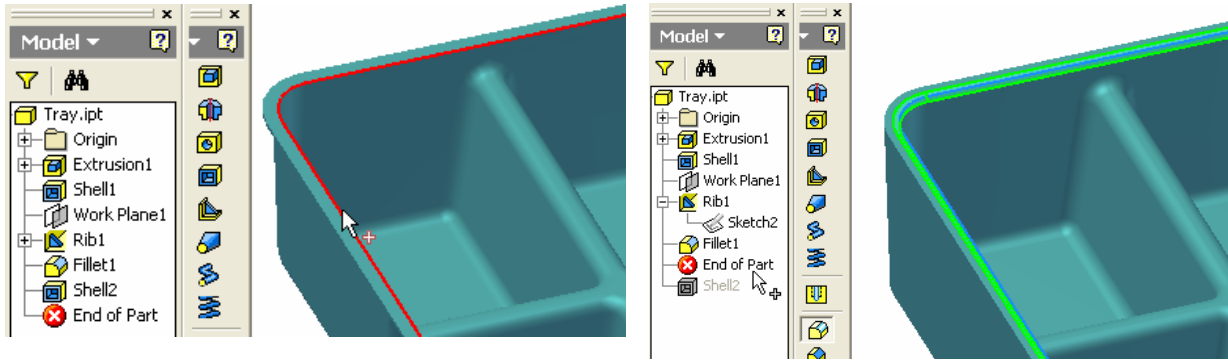
- 68. Use the **Feature Select Mode** to add fillets to the entire feature.



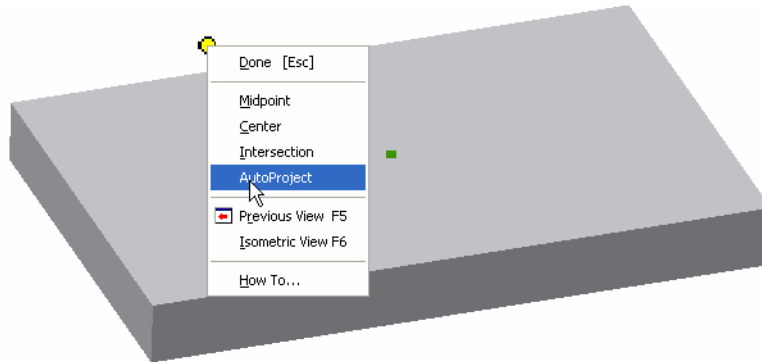
- 69. Use **Derived Components** to create tooling dependencies. If the part changes the tool changes. Use the **Delete Faces** with the **Lump or Void** option to delete an internal void.



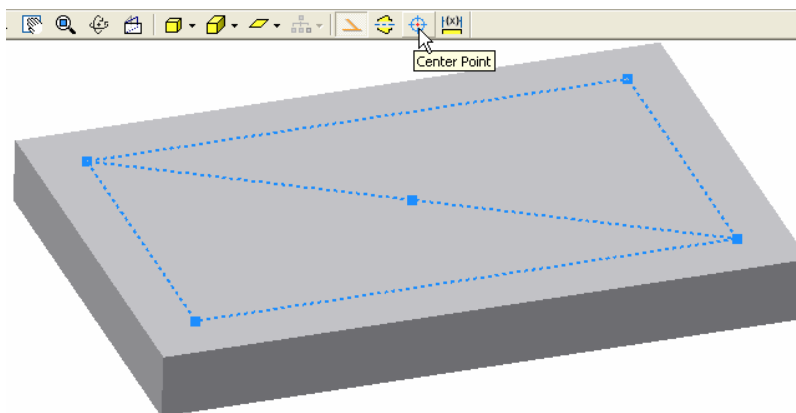
- 70. By using the **Sculpt** tool in combination with derived parts you can quickly create tooling.



- 71. Did you forget a feature? You can insert features in the history by rolling up the End of Part marker. *(Tips: Depending on the feature you can sometimes drag the feature to a different point in history. To get a small file size for email attachments you can drag the EOP marker all the way to the top of the browser).*

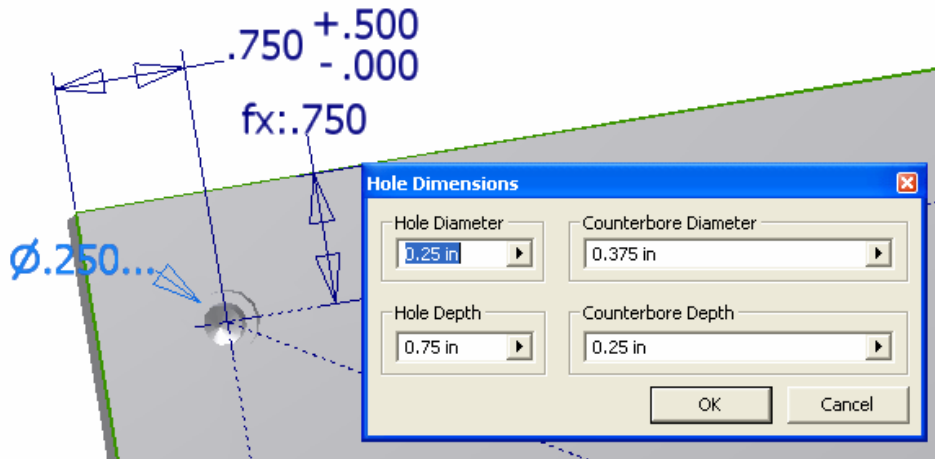


- 72. Although I prefer to turn off AutoProject Edges on sketch creation in the Application Options you can AutoProject as needed by RMB while sketching and “scrub” the edge you want to project.

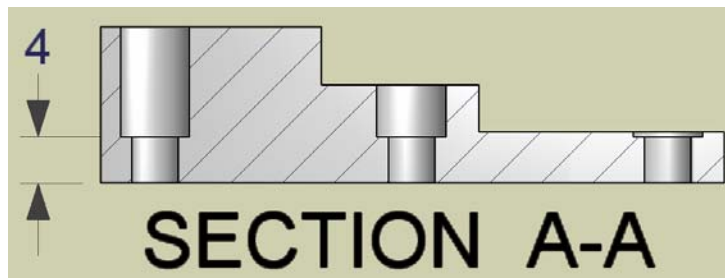


- 73. Here is a trick to get a pattern of hole centers. Create a rectangular sketch and select it. Then click on the **Center Point** tool.

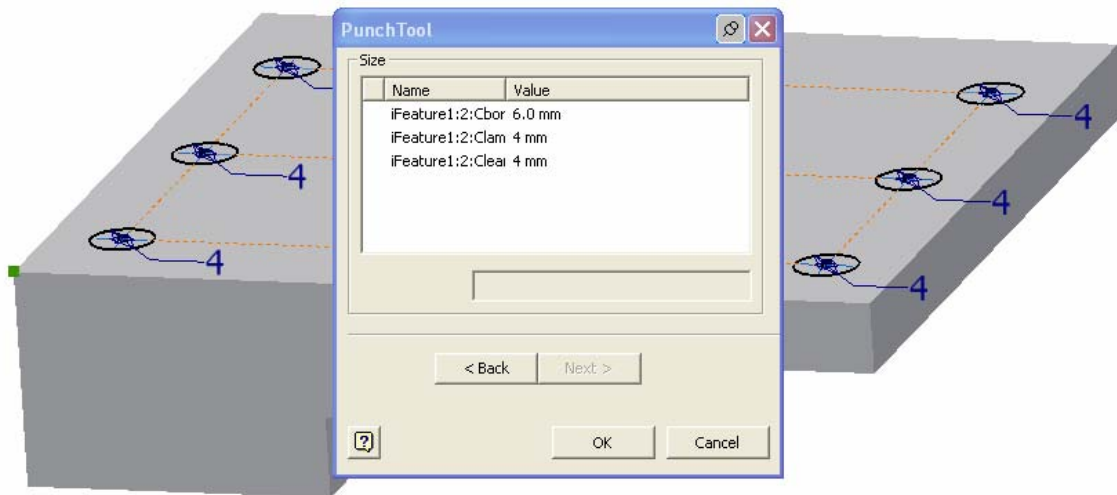
- 74. Need to change the dimensions of a hole? You could edit the feature or double click on the feature and you then get access to all dimensions including the sketch dimensions.



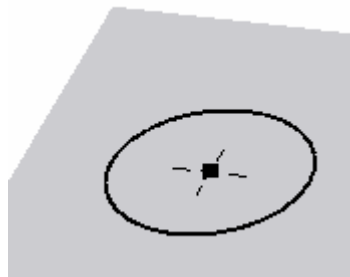
- 75. You can **evaluate** dimensions at Median, Max, Min and Nominal conditions. This can be really useful when using the model to create CNC code where a deviation tolerance is specified. You don't need to calculate the Median. (I exaggerated the tolerance and used nice easy numbers for visual purposes in this example.)



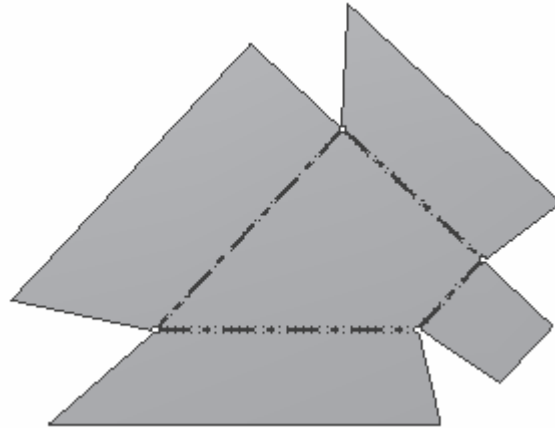
- 76. On your drawings you specify counterbore depths as shown to account for standard length fasteners and/or cast surfaces. With the Hole tool you would have to calculate the depths. Create iFeatures for standardization. In this case we will create a special iFeature called a **Punch** in the Sheetmetal Environment.



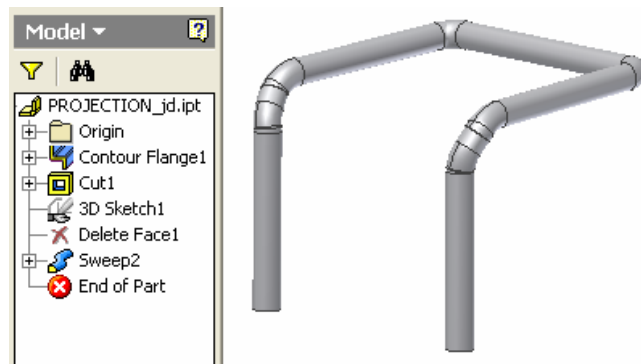
- 77. You can go back and forth between the Sheetmetal Environment and the Part Features Environment as needed.



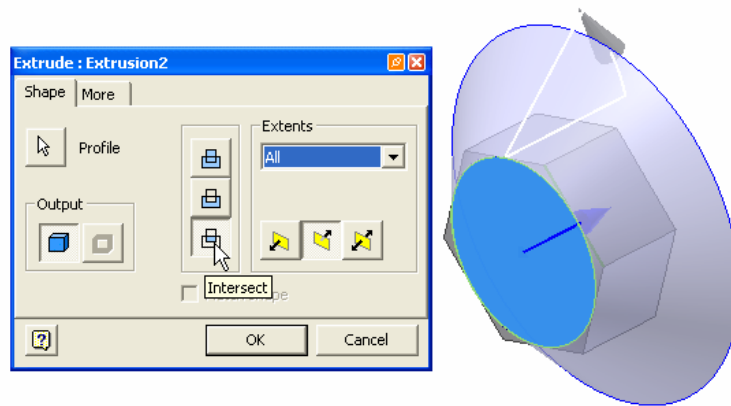
- 78. When creating an iFeature or Punch tools you must have a Point/Hole Center and **start your sketch on a part face** rather than a Workplane. I follow this rule whether it is going to be a Punch tool or standard iFeature. Create your iFeatures so that all subsequent features are dependents. *Creating iFeatures are one time when it is acceptable to have the sketch under constrained.*



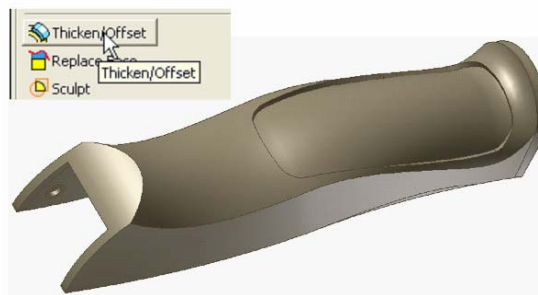
- 79. Sheet metal flat patterns can be reoriented for x and y direction before exporting dxf or other format by RMB on the sheet metal flat pattern icon at the top of the browser.



- 80. Just as you can go back and forth between the Modeling and Sheet Metal environments and use Part Features to create sheet metal parts, you can also use Sheet Metal Features to create standard parts.



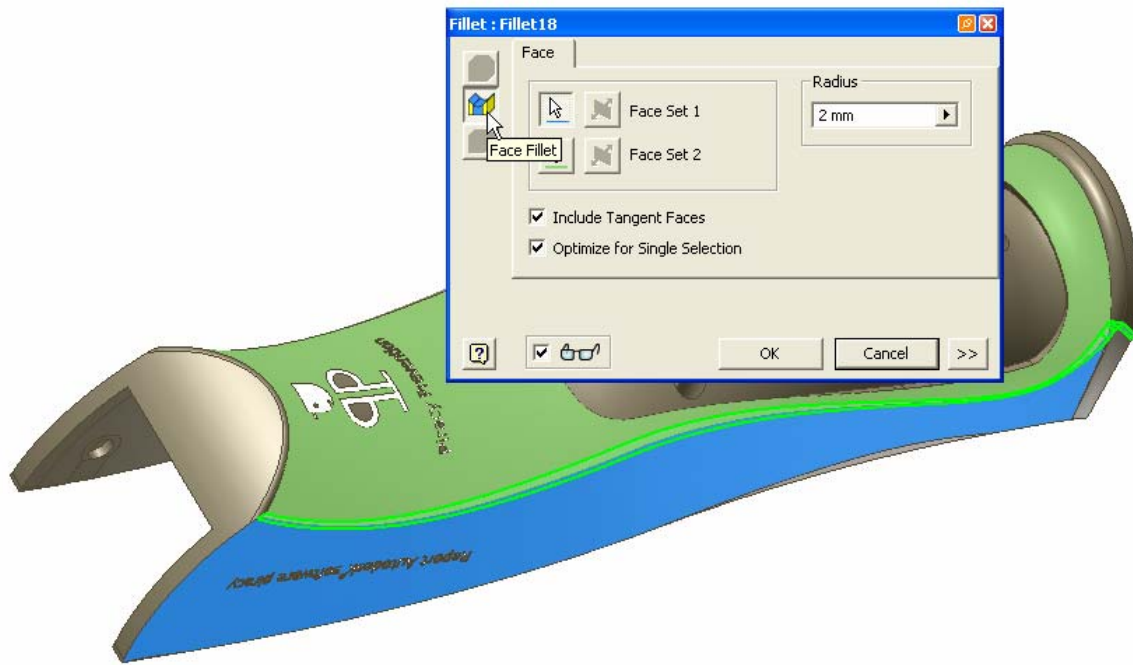
- 81. Perhaps one of the most underutilized feature options is the **Intersect** option available with many part feature tools. (*Polygons must have a vertical or horizontal constraint to be fully constrained. Creating the tangent circle is an excellent use of the RMB Autoproject option.*)



- 82. Check out the **Thicken/Offset** with the Cut option too.



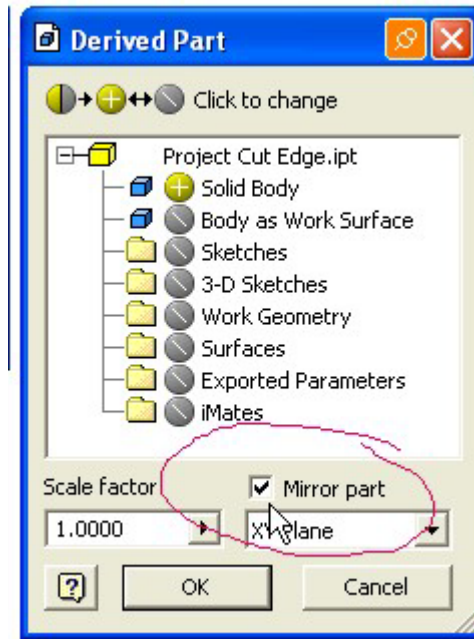
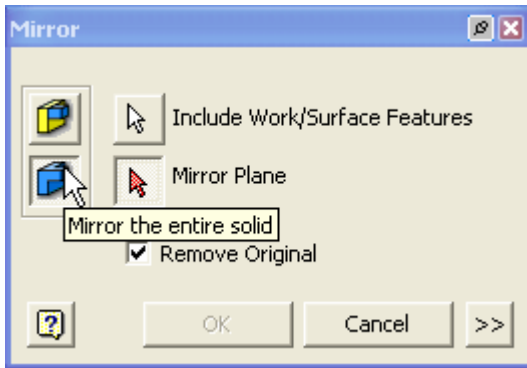
- 83. Change the Thicken option to **Intersection** and we have our mating part.



- 84. And be sure to check out the new **Face Fillet** option.



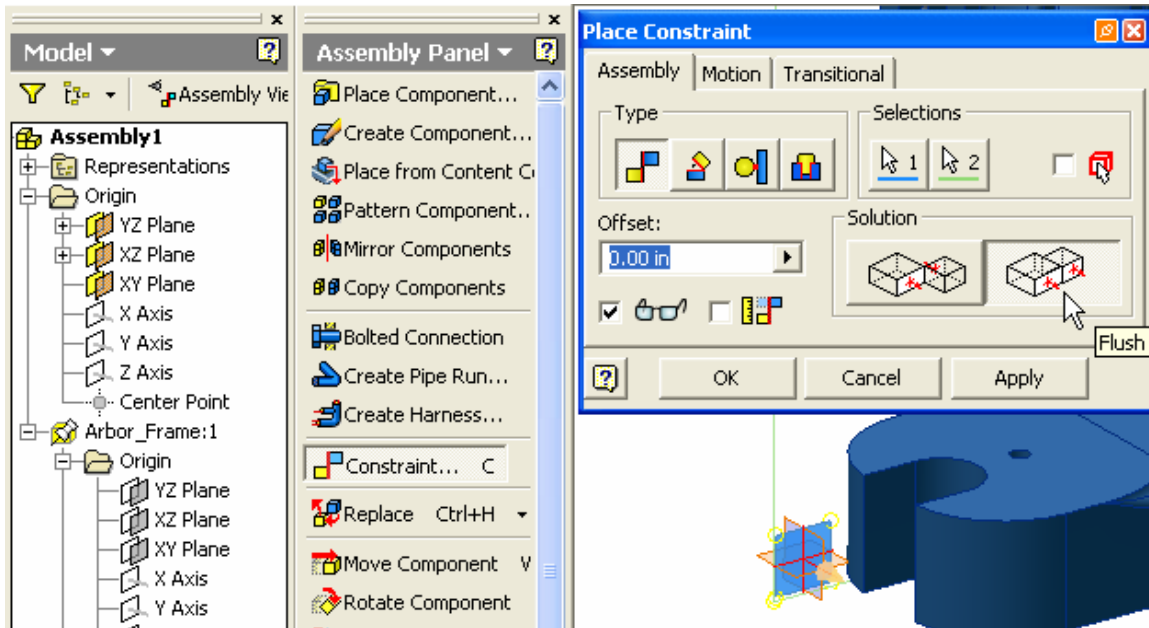
- 85. Combine the Rectangle Pattern, Thicken with intersection options and some of the other tricks we have learned and you can create some fairly complex geometry. We might have to delve into Loft, Sculpt Trim/Extend Surface and some of the other advanced tools, but that is another class.



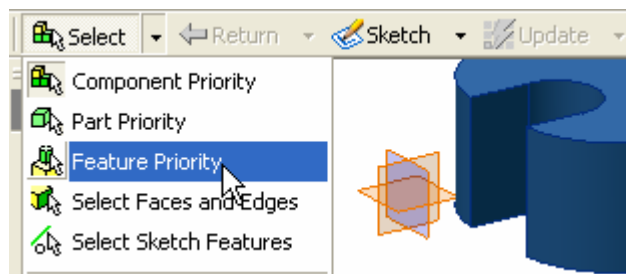
- 86. AutoCAD users always struggle at first in mirroring a part in Inventor. There are a couple of ways to mirror a part. The first method is within the part modeling environment select **Mirror** the entire solid and Remove Original. The second (and generally preferred) method is to Derive the Mirrored part.

Assemblies are a Snap

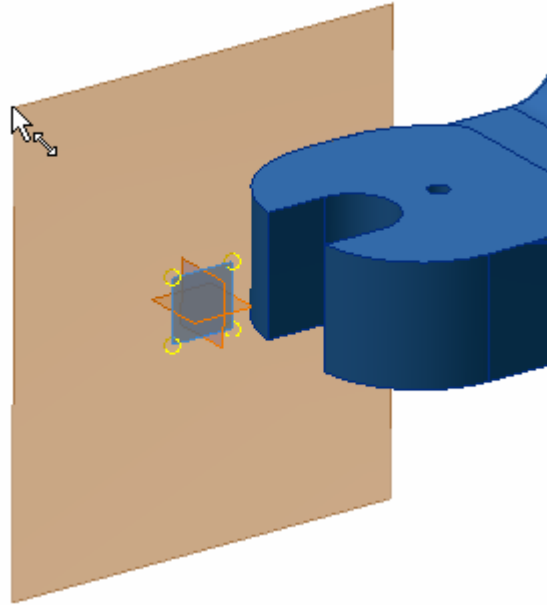
When creating assemblies try to mimic the procedures you would use with physical parts.



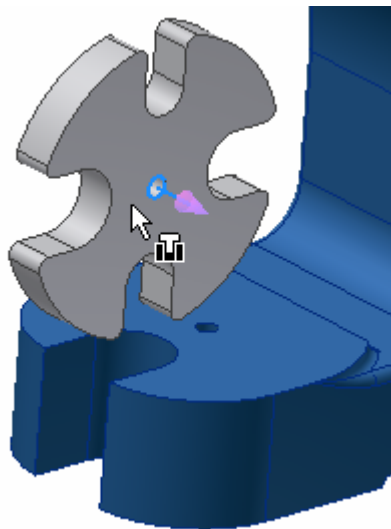
- 87. Apply **Flush** mate constraints between your assembly origin planes and a logical grounded part within assemblies and sub assemblies.



- 88. To resize assembly workplanes change the selection filter to **Feature Priority**.



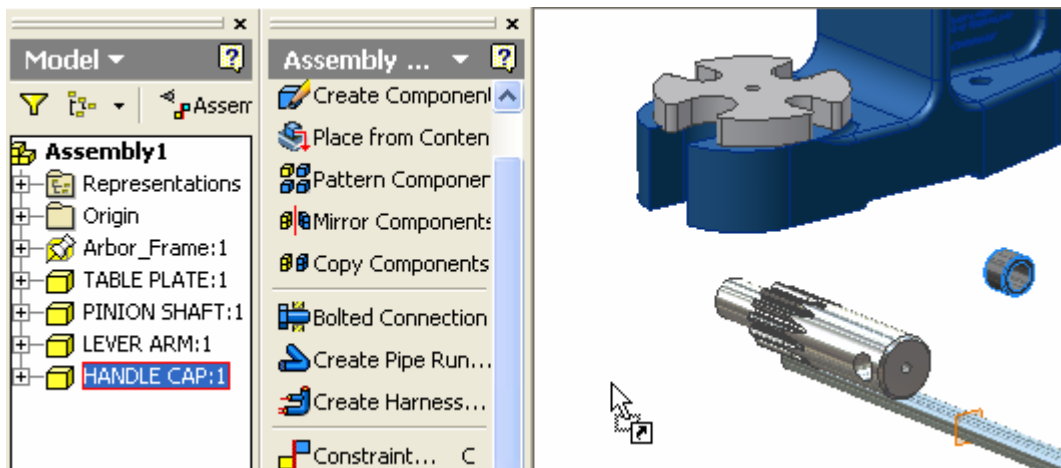
- 89. To resize drag the edge with the two-headed pointer. To pan move the cursor around till you see a four-headed arrow.



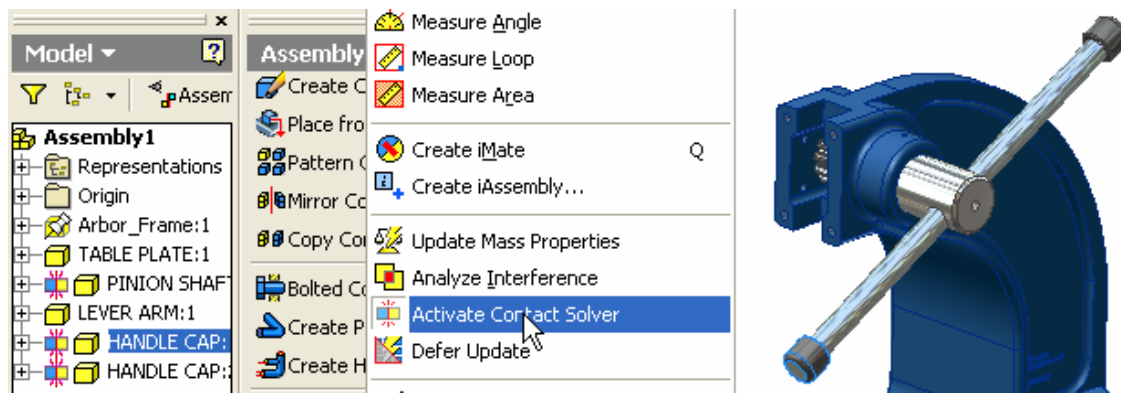
- 90. You can apply auto assembly constraints by Alt select-drag method.



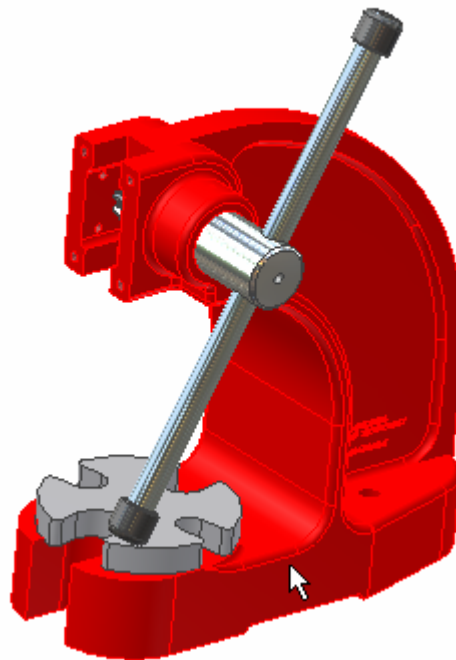
- 91. If you open Windows Explorer and set the Thumbnails preview you can drag and drop parts into your assembly.



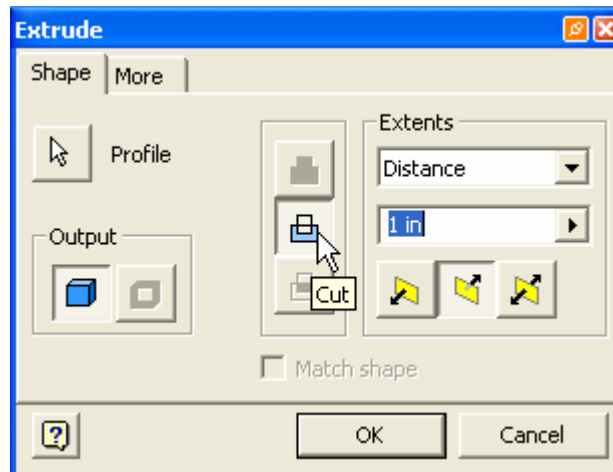
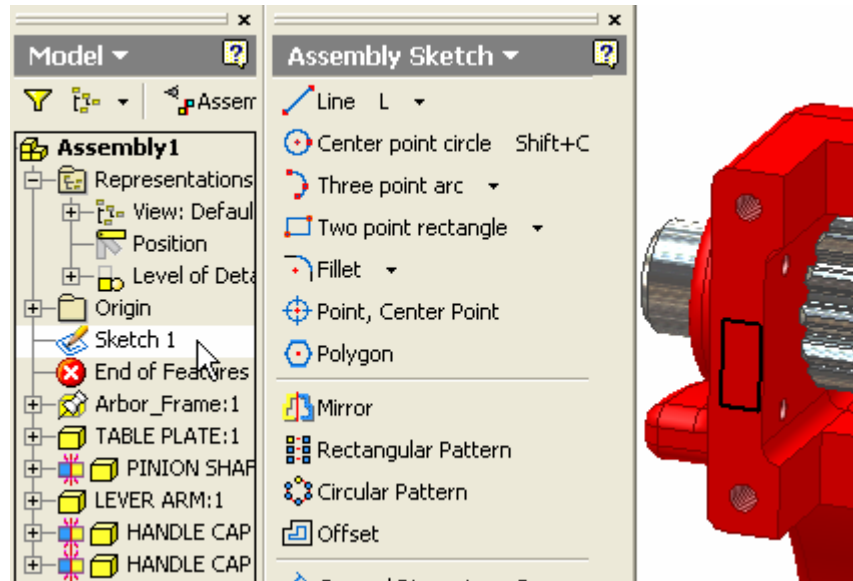
- 92. Additional instances of components can be drag copied from the browser tree to the graphics window.



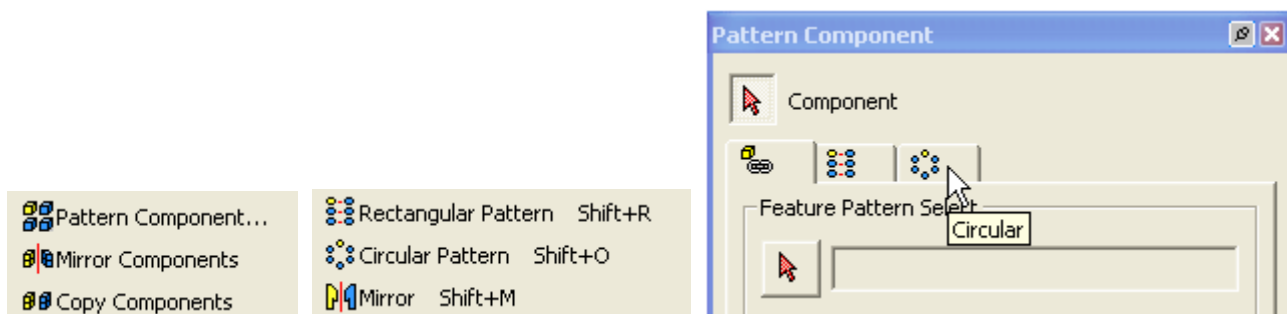
- 93. Add only those parts you need to the **Contact Sets** and Activate the **Contact Solver**. Once you have verified the contact turn off the solver.



- 94. Assembly colors do not transfer to the part file. You can create different **Design View Representations** for example, for different paint schemes.

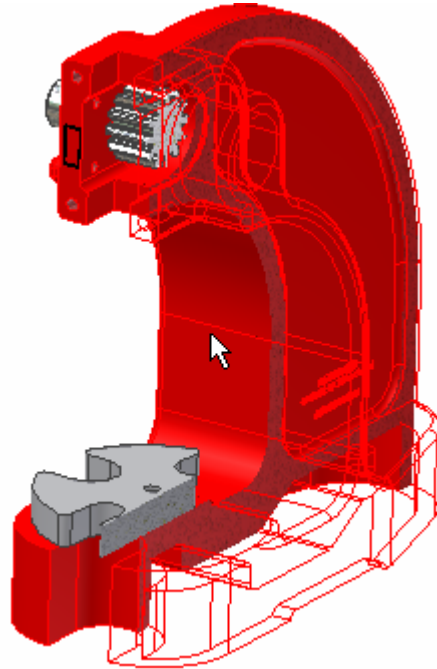


- 95. Make sure you know whether you are in an assembly level sketch or a part level sketch. Assembly features remove material only (with the exception of **Move Face**) and do not transfer to the part level.



- 96. Make sure you know whether you are selecting a Component operation or a Feature operation. Beginners often complain that they can't pattern or mirror components in an assembly when they

have mistakenly selected the pattern or mirror features tool. They especially make this mistake when trying to do a circular part pattern because the Circular **Pattern Component** icon is hidden.

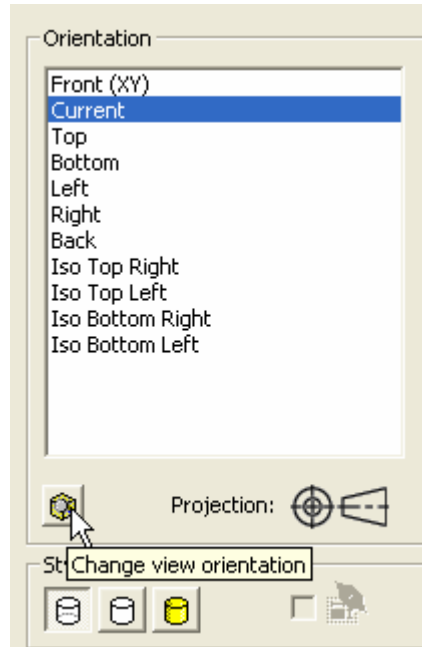


- 97. To see into an assembly un-ground the frame part and drag our assembly through a **Section** plane. Because we added flush constraints to the origin workplanes it is easy to return to the original position.

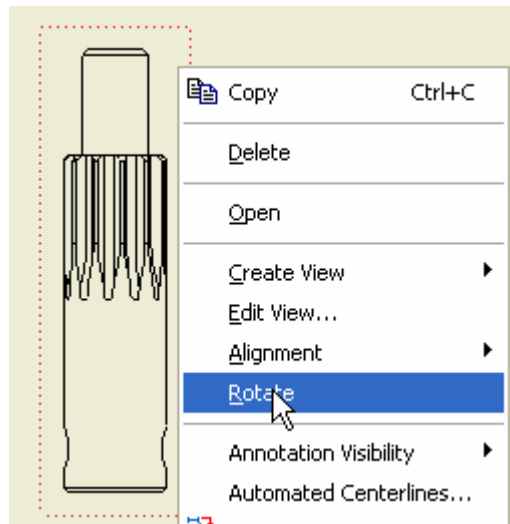
I consider Adaptive Parts or Cross-part projections to be an advanced topic. Users should be able to create rock-solid parts and assemblies before tackling adaptive features. There are entire classes at AU devoted to the topic. Also look for information on “skeletal” or “master sketch” modeling techniques.

2D is Gravy

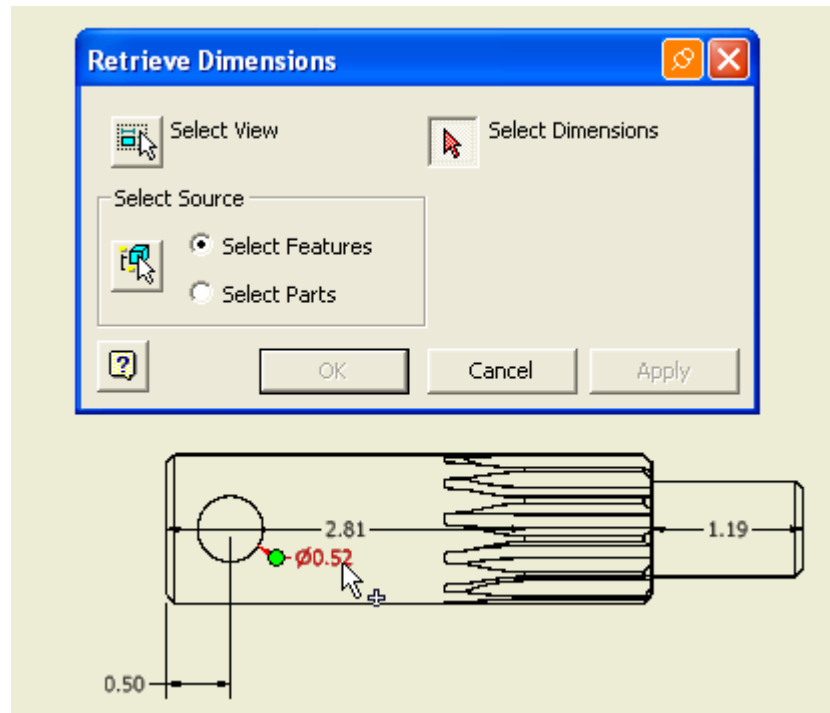
Traditional detailed 2D orthographic projections are almost an afterthought when created from 3D models.



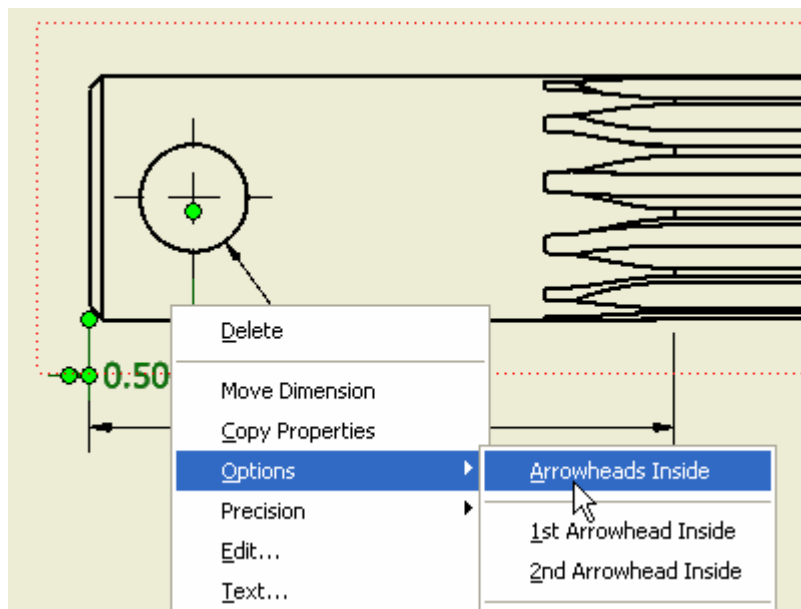
- 98. When you place a **Base View** the Orientation name, Top, Front, etc. is all relative. I don't usually worry too much about the names and instead go by the visual. Frequently I use the "Current" view and change the view orientation to suit.



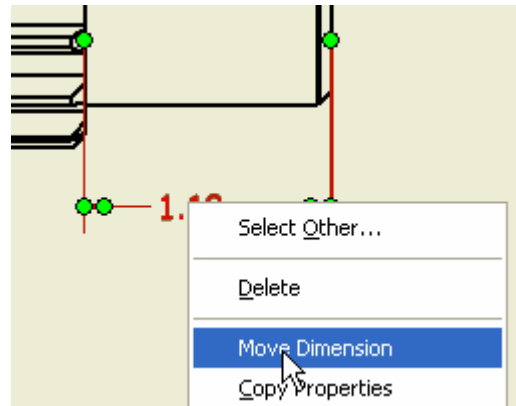
- 99. If the base view comes in at an orientation that doesn't fit well aesthetically it is easy to **Rotate** the initial Base View.



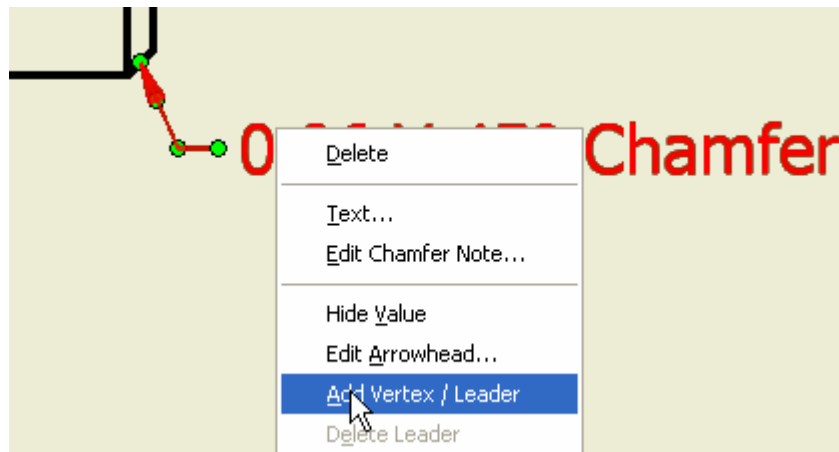
- 100. The model has already been dimensioned during creation; this work can be utilized by using **Retrieve Model Dimensions**. Depending on how Inventor was installed these dimensions in the idw can also be used to edit the ipt.



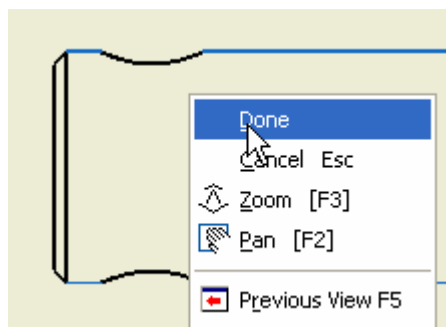
- 101. Once retrieved or placed the dimensions can be edited for aesthetic considerations such as **Arrowheads** inside/outside.



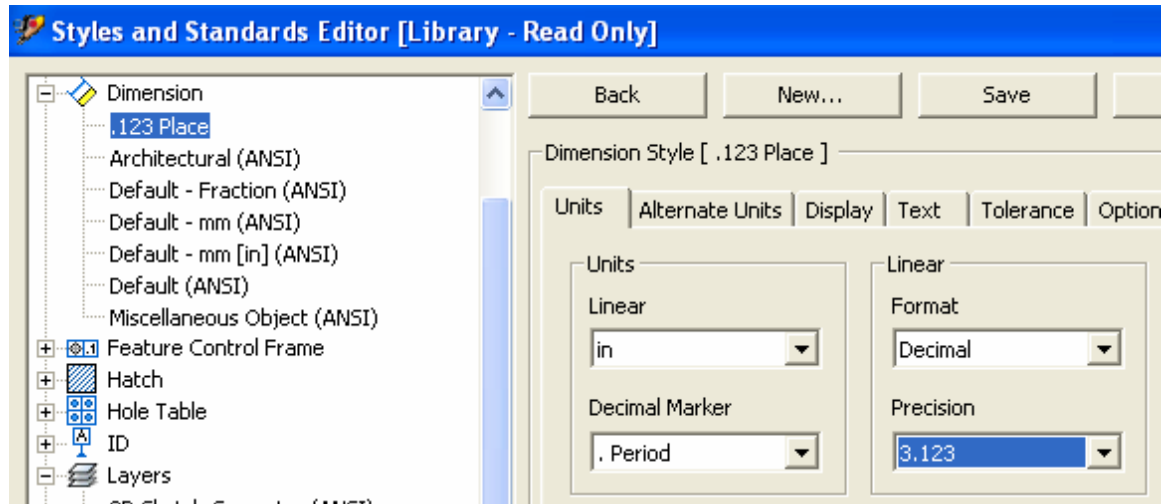
- 102. You can also RMB **Move Dimension** from one view to another.



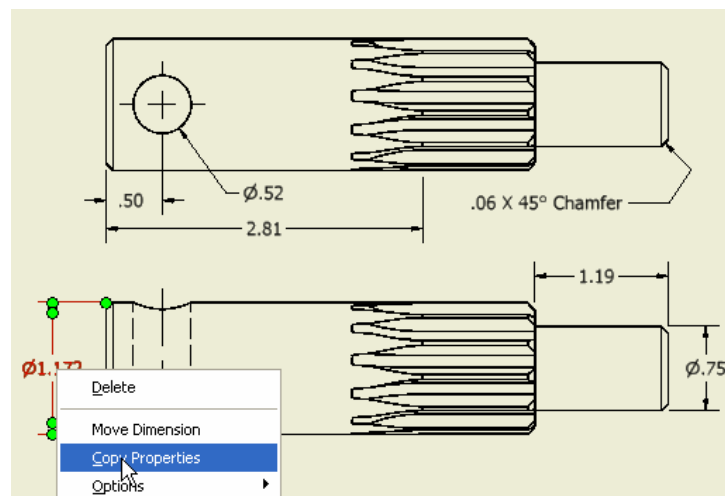
- 103. **Chamfer notes** can now be dimensioned from Chamfer Features and vertexes added to the leader as needed.



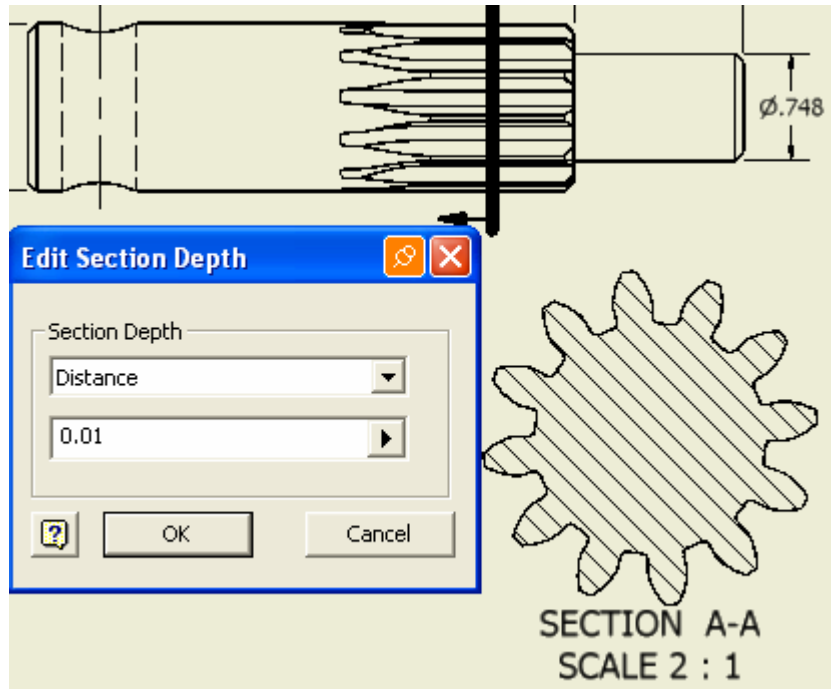
- 104. When projecting sketch geometry in an idw you must RMB Done. This is a different behavior than in the ipt environment.



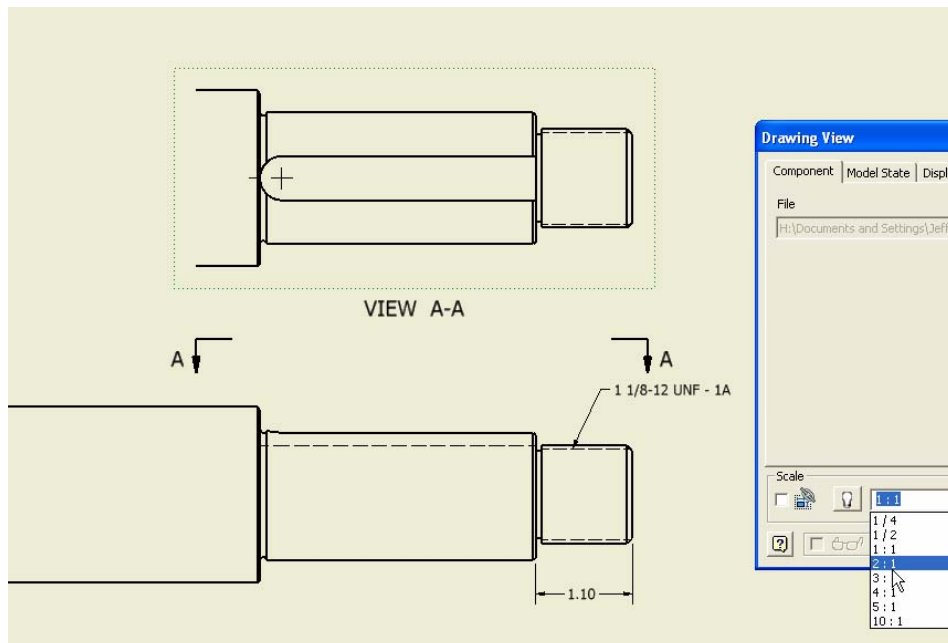
- 105. Set up standard dimension **Styles** for your company. There are entire AU classes devoted to setting up Styles and Standards.



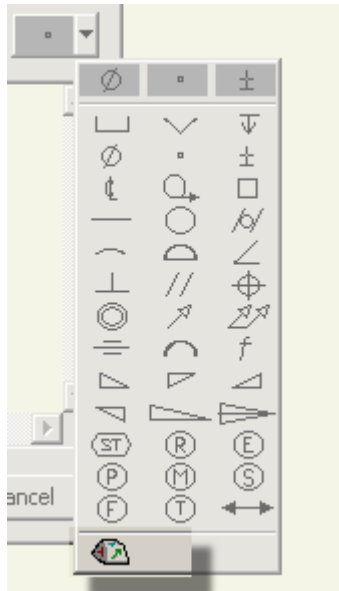
- 106. I prefer to go through a drawing setting one dimension of each style needed and then use the **Copy Properties** to change any other dimensions as needed.



- 107. The **Section Depth** can be edited and the section view placed anywhere on the drawing sheet by holding down the Ctrl key during placement. Views that are not aligned can have horizontal or vertical alignment added by RMB clicking on the view.



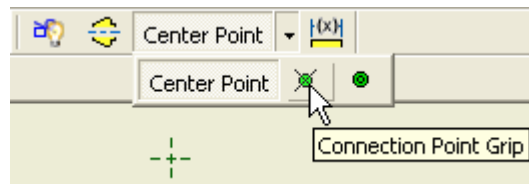
- 108. In addition to using the **Detail** view tool, details can also be made using the standard Section tool with the cutting plane off of the part.



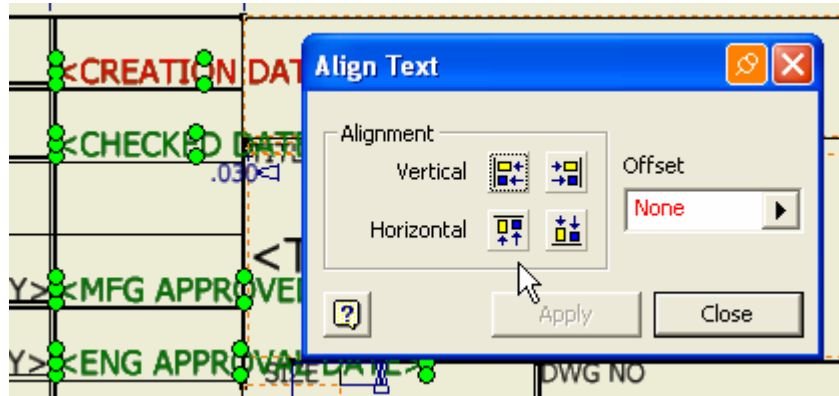
- 109. The Windows **Character Map** can be launched from the Inventor symbols.



- 110. To speed up the view creation bitmaps are used by default for shaded views. For maximum image resolution on-screen set **Offline Only** for shaded images.



- 111. Many users are not aware of the **Connection Point** and **Insertion Point** tools in creating **Sketched Symbols**.



- 112. They might not be aware of the feature to **Aligning Text** in idw views. The creation of a company standard title block and sketched symbols is a topic worthy of a class by itself.

Collaboratory Learning

The word “Collaboratory” is synthesized from the works collaboration and laboratory. The internet has become the site of many specific collaboratories. Contact me at jmather@pct.edu with your tips, suggestions and comments. And be sure to visit the Autodesk Inventor discussion forum at <http://discussions.autodesk.com>

The Student Engineering & Design Community is a collaboratory set up especially for students and faculty with free Autodesk software downloads and help forums.

- For those of you in education –
<http://engineersrule.org>
For more information check out **MS21-2: Education Industry Main Stage** 8AM tomorrow.
- For those of you in manufacturing –
For more information check out **MS21-3: Manufacturing Solutions Industry Main Stage** 8AM tomorrow.