SkillsUSA University

60 AUTODESK INVENTOR TIPS IN 60 MINUTES

J.D. Mather – Pennsylvania College of Technology

For educators and students in mechanical design.
A rapid paced delivery of the top tips and tricks gathered from years of experience teaching Autodesk Inventor software. Looking for that one class that will justify your travel to SkillsUSA University - if you teach mechanical design/drafting using Autodesk Inventor - this is it. After this class, when you return to your home school you will be referred to as the Inventor Guru.
1 p.m., Big Joe Turner A.

Key Learning

> Basic Interface Settings
> BORN Technique
> Robust Sketch Techniques
> Neutral File Translation
> Basic Assembly Tips
> Drawing Tips

About the Speaker:
J.D. Mather is an Assistant Professor of CAD and Product Design at Pennsylvania College of Technology, Penn State. He previously worked in industry for 15 years, including positions as journeyman machinist, research and development technician, and industrial engineering technician. Dr. Mather has been an Autodesk Inventor Certified Expert since release 7. He is also a Certified SolidWorks Professional. He is experienced with AutoCAD and Pro/E as well. Dr. Mather placed second for Autodesk Inventor category at the Autodesk University 2006, Avatech Altogether Smarter Challenge, and placed first in the same category at AU 2007. He has presented AutoCAD and Inventor classes at Autodesk University 2006-2009.
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Reference Materials:
http://home.pct.edu/~jmather/SkillsUSA University.pdf
http://home.pct.edu/~jmather/content/DSG322/inventor_surface_tutorials.htm
http://www.autodesk.com/discussiongroup-inventor
http://www.vdssolutions.co.uk/
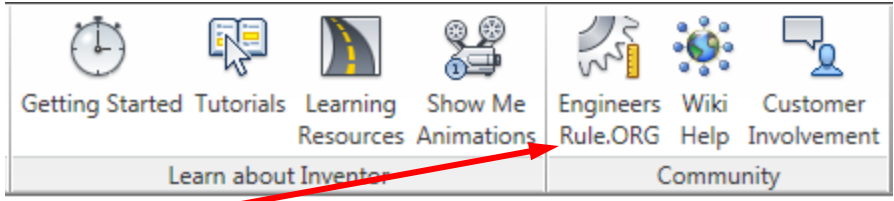
Tip 1. You should be aware that you and your students can get a free learning license of many Autodesk products including Inventor Professional from the Autodesk Student Community.

Before we get started I want to define a common interface.

Click on the Tools tab and then Application Options.
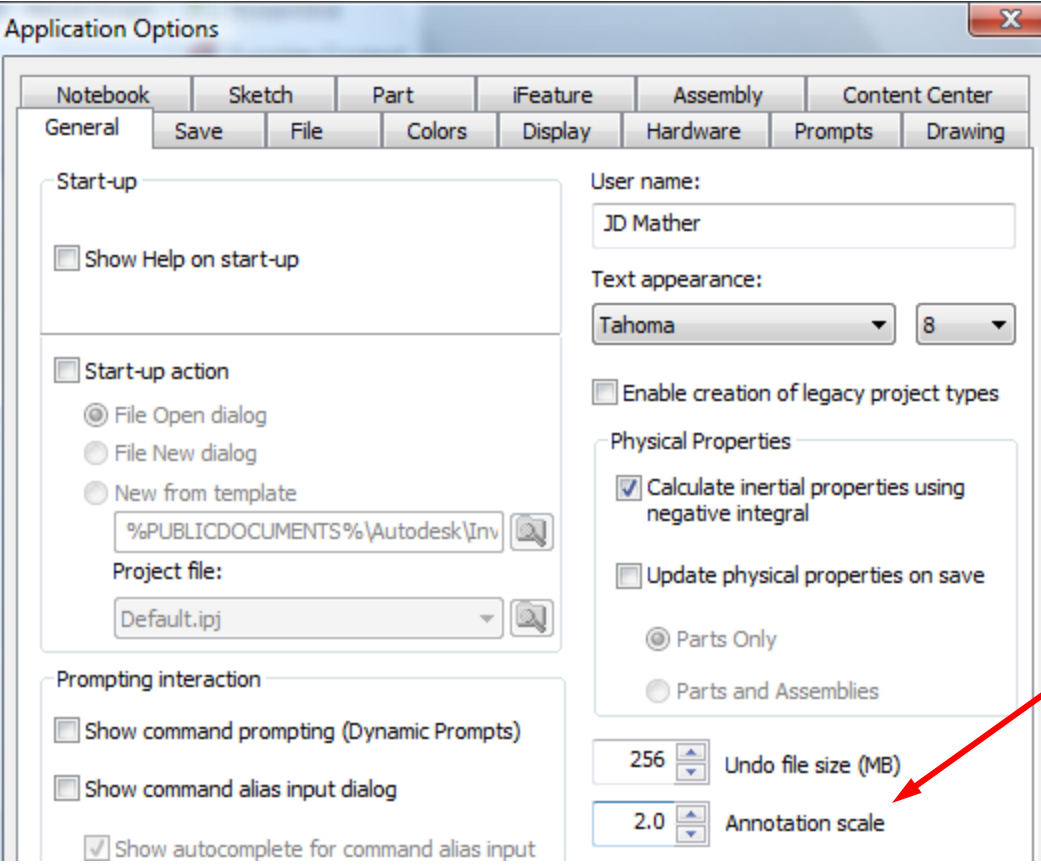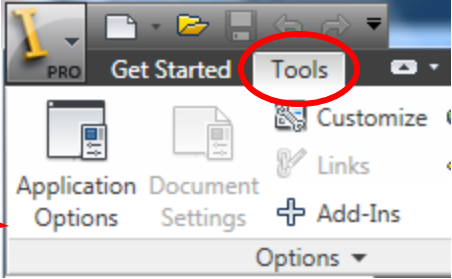
Tip 2.

If you normally present on a projection screen you might want to increase the Annotation scale so that everyone can see your sketch dimensions.
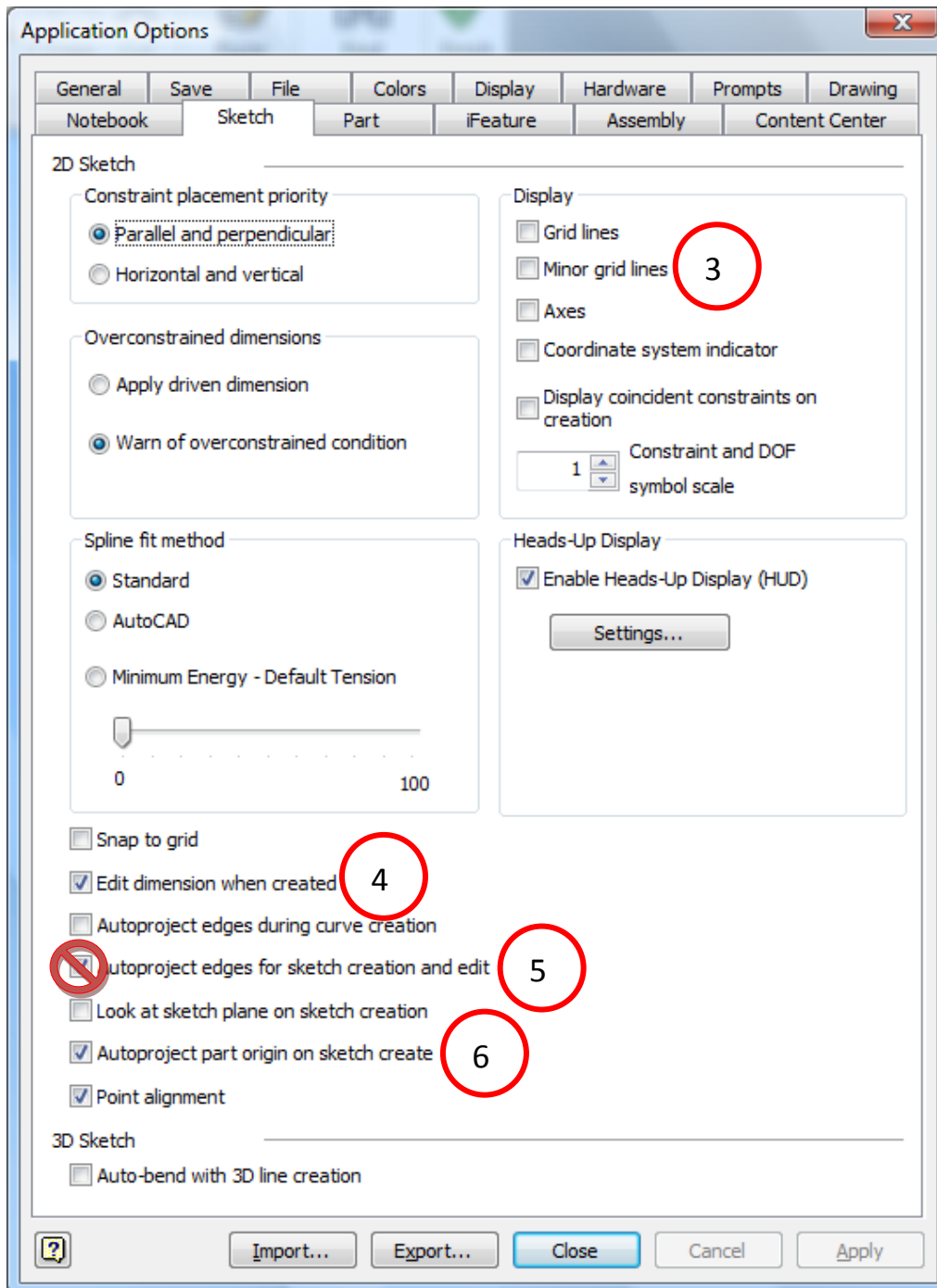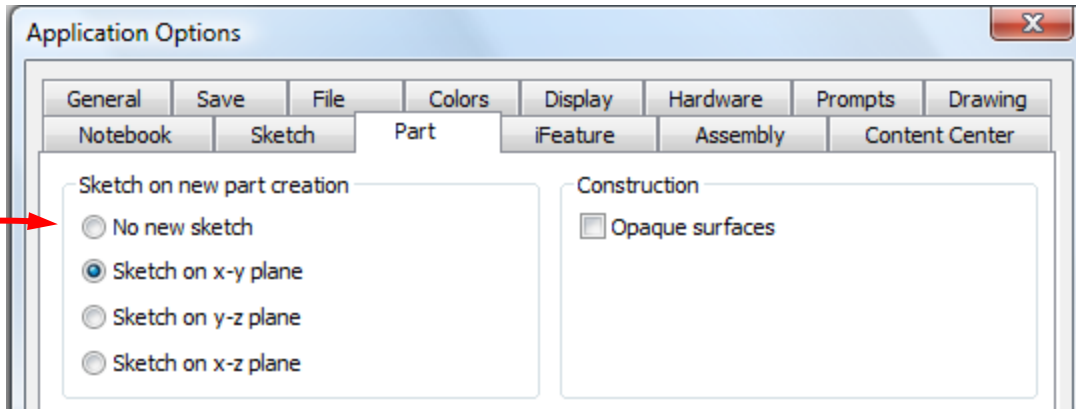
**Tip 3.** On the sketch tab turn off Display – Grid items.

Tip 4. Turn on Edit dimension when created.

Tip 5. Turn off Autoproject edges for sketch creation and edit (<mark>Note: I will leave on for now to demonstrate later why you should turn it off</mark>).

Tip 6. Make sure Autoproject part origin on sketch create is always turned on.
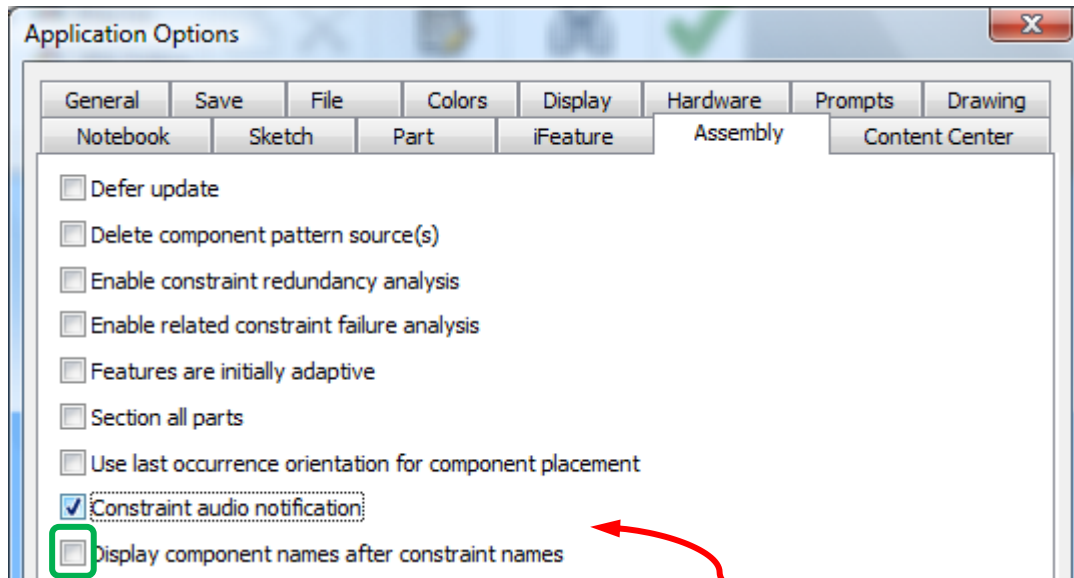
Tip 7. On the Part tab you might want to set No new sketch – when you start a new part you will select the plane on which to create the first sketch rather than Inventor automatically starting the first sketch.
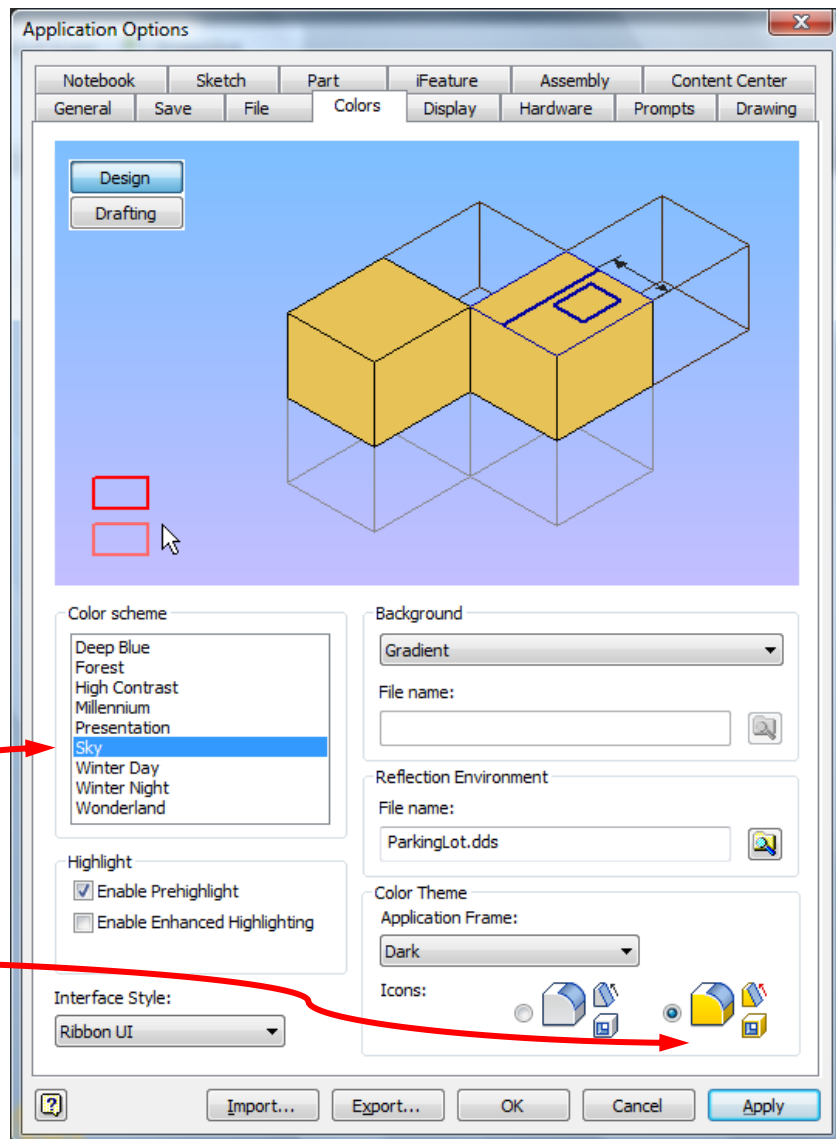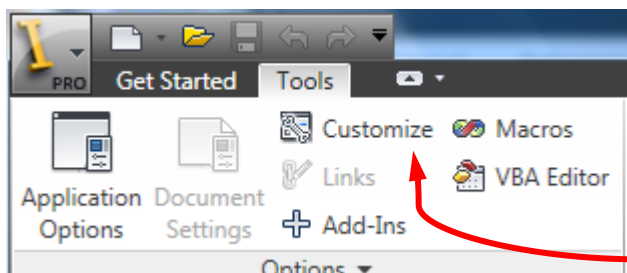
Tip 8

On the Colors tab I prefer to use the Sky color scheme with Amber icons for high-contrast projection.

Tip 9 I would uncheck Constraint audio notification and checkmark Display component names after constraint names on the Assembly tab. (Set opposite of the default settings I show here.)

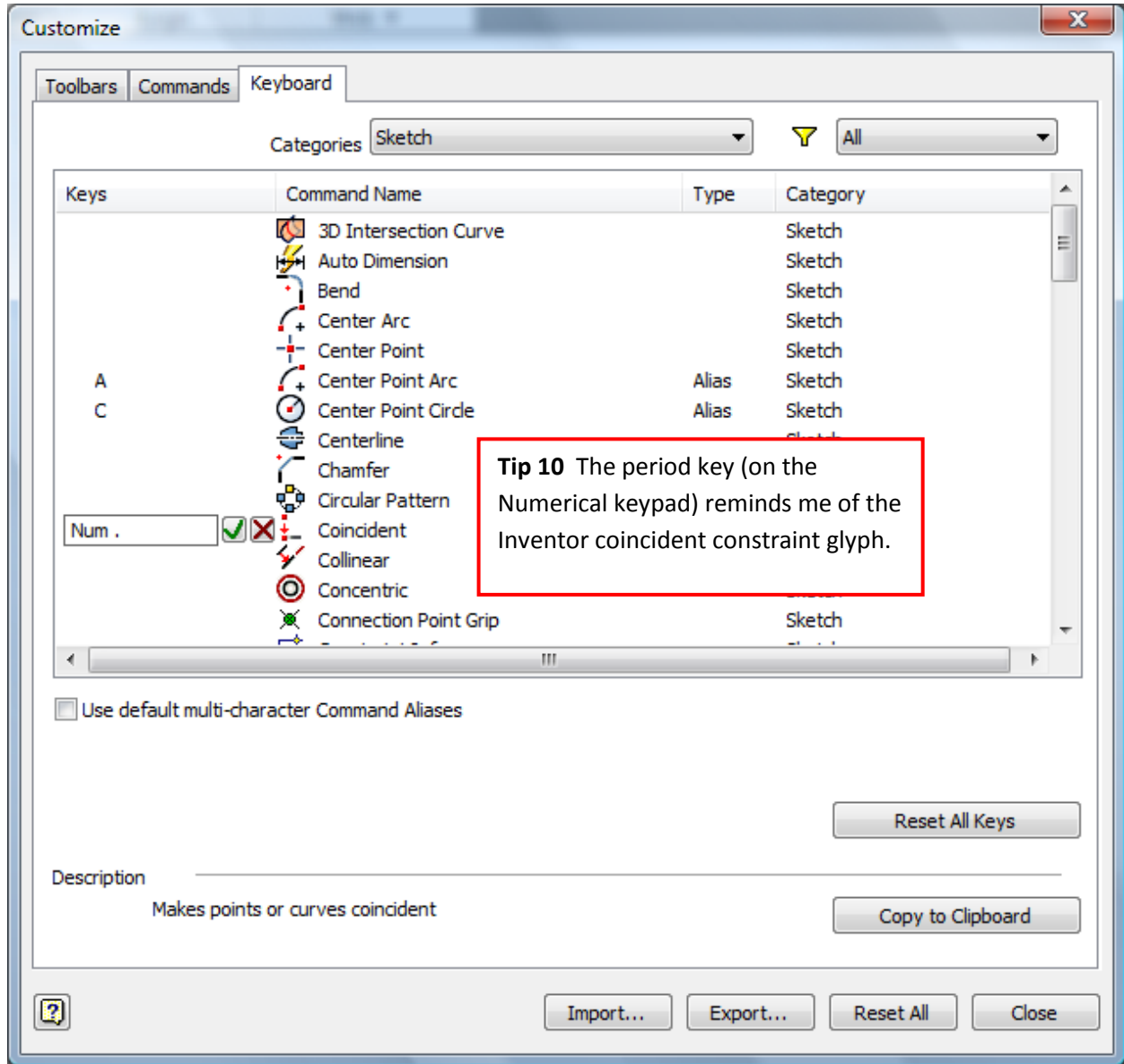Next let's customize some of our keystrokes.

One of the most powerful features of Inventor is the use of geometric constraints like tangent, collinear, horizontal, vertical and perpendicular.

We use these geometry constraints so frequently that it makes sense to assign "hotkeys" on the keyboard to quickly access these constraints.

If you use the Inventor sketcher properly most of your geometry constraints will be applied automatically, but there will still be a frequent need to manually place constraints.

I have not seen any evidence that working without constraints is faster than working with constraints. I could be wrong, but I doubt it.

Select the Keyboard tab and then the Categories – Sketch. Click to the left of the command for which you want to define a hotkey.

Tip 10 The period key (on the Numerical keypad) reminds me of the Inventor coincident constraint glyph.

Tip 11 Then minus key is a horizontal line.

Tip 12 The plus key is two perpendicular lines.

Tip 13 The asterisk reminds me of a circle.

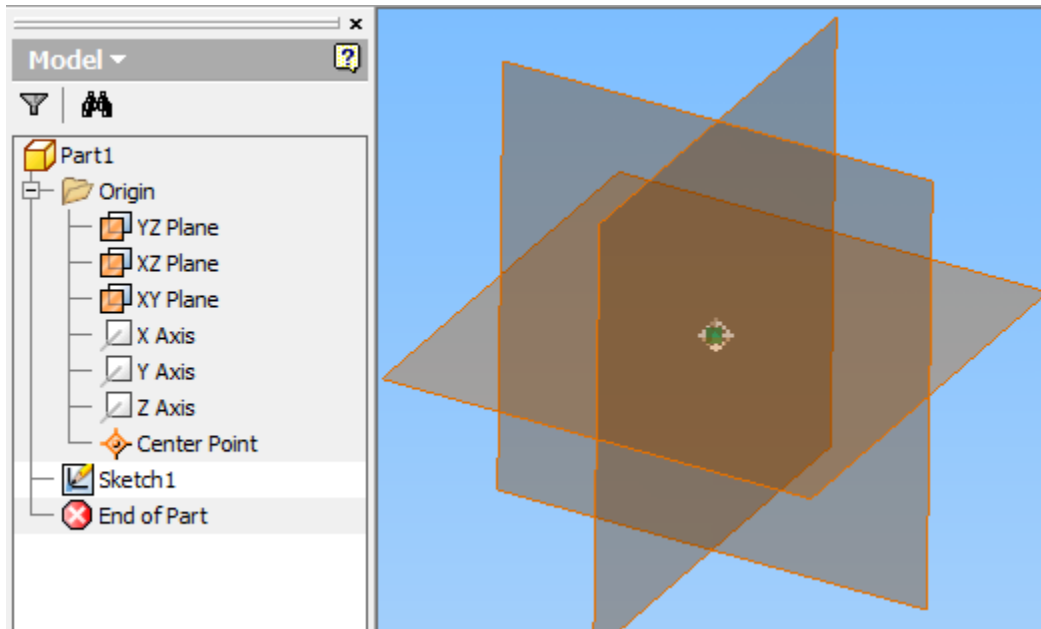**Tip 14** The forward slash key is almost vertical.

All of these settings can be Exported for Import on another machine.
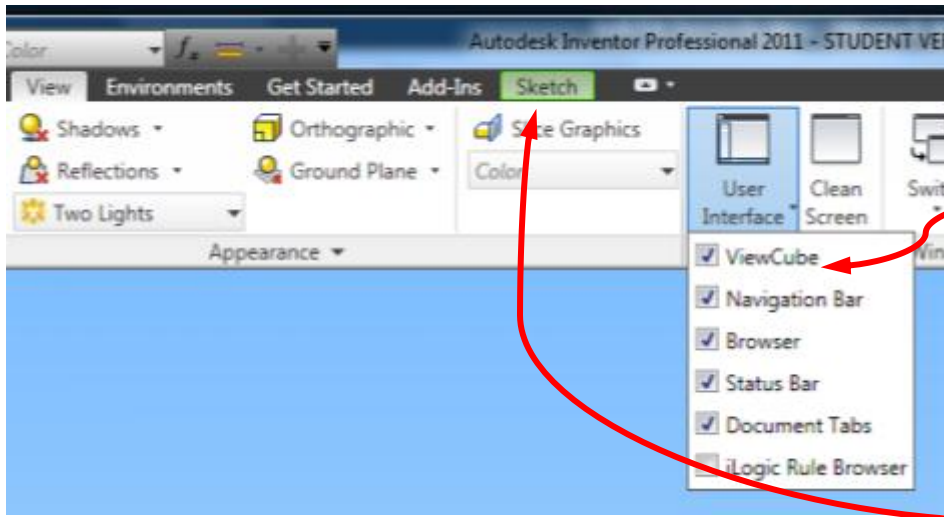
Tip 15 I see a lot of reports of people losing their Browser panel.

You can get it back from the View tab – User Interface.

Notice that any green tab is your active environment.

Tip 16 In this class we are going to use the **B**ase **O**rphan **R**eference **N**ode **T**echnique (Born Technique).

In this class 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!
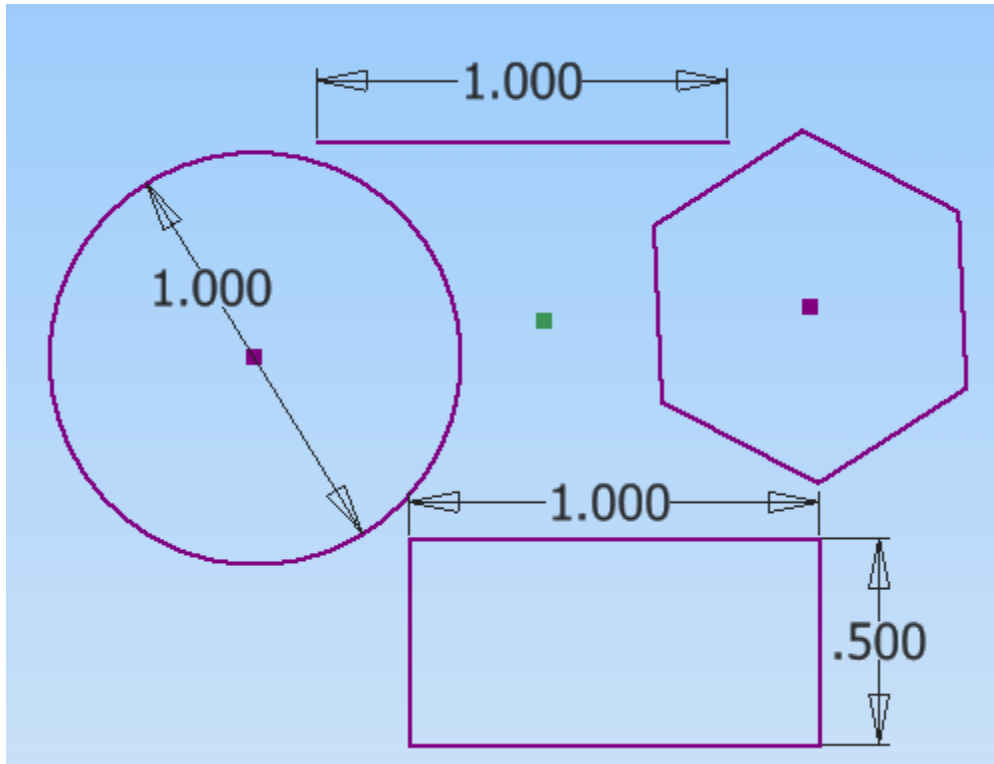
**Tip 17** Sketch one line, or a circle, a rectangle or a polygon and dimension it ==immediately==.

This will give you a sense of scale for the rest of your sketch. In general – keep your sketches simple – I recommend no more than 7-10 entities in most sketches.

Tip 18 Following the BORN Technique create your sketches ==as much as practical== on origin workplanes rather than on part faces.
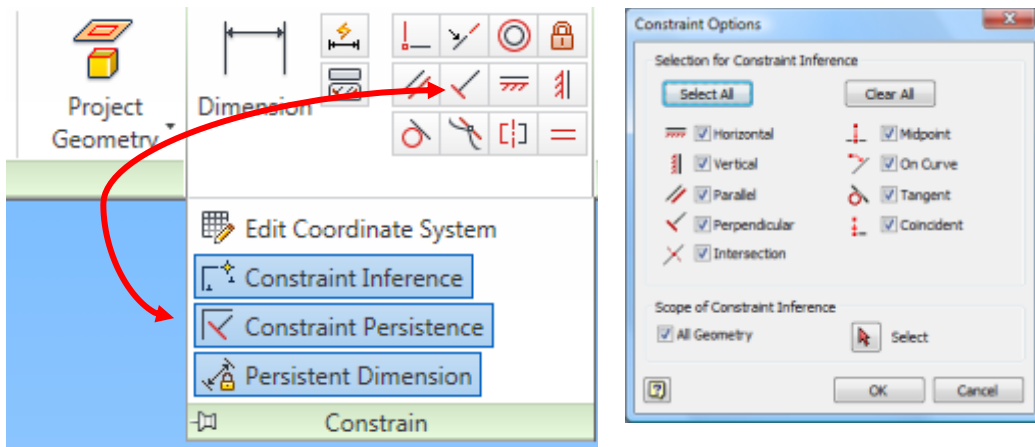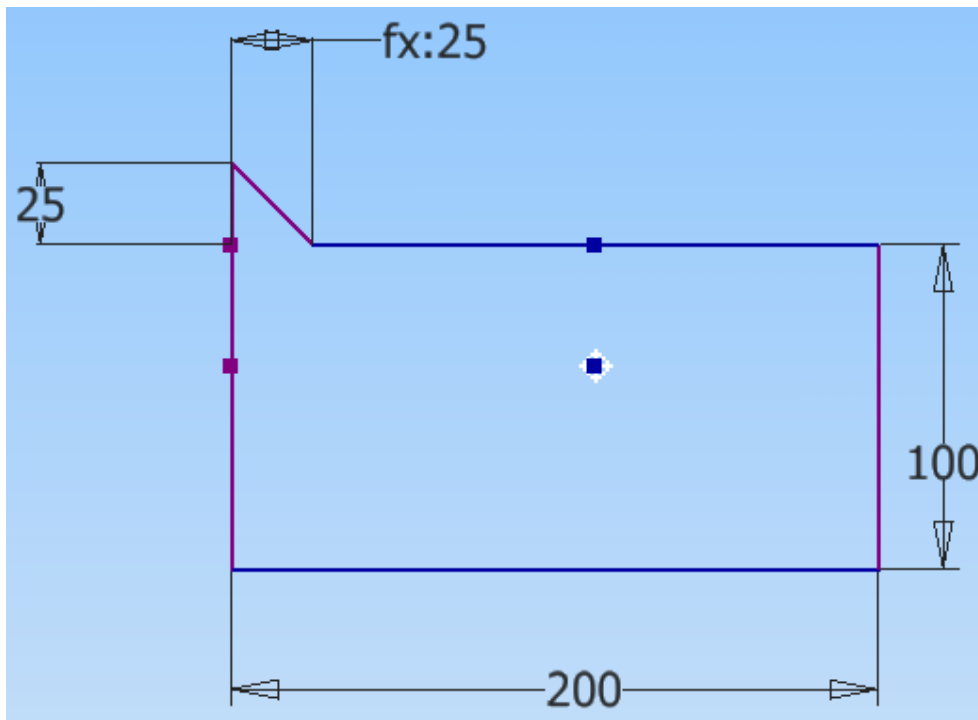

Tip 19 Watch out for the Constraint Persistence icon (particularly before Inventor 2011) as it looks very much like the Perpendicular constraint.
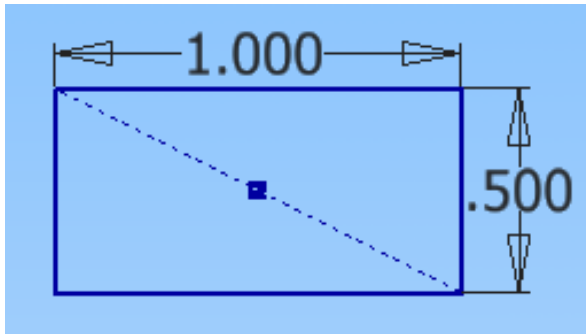
Beginners hit this and don't realize they have turned off auto-constraints.
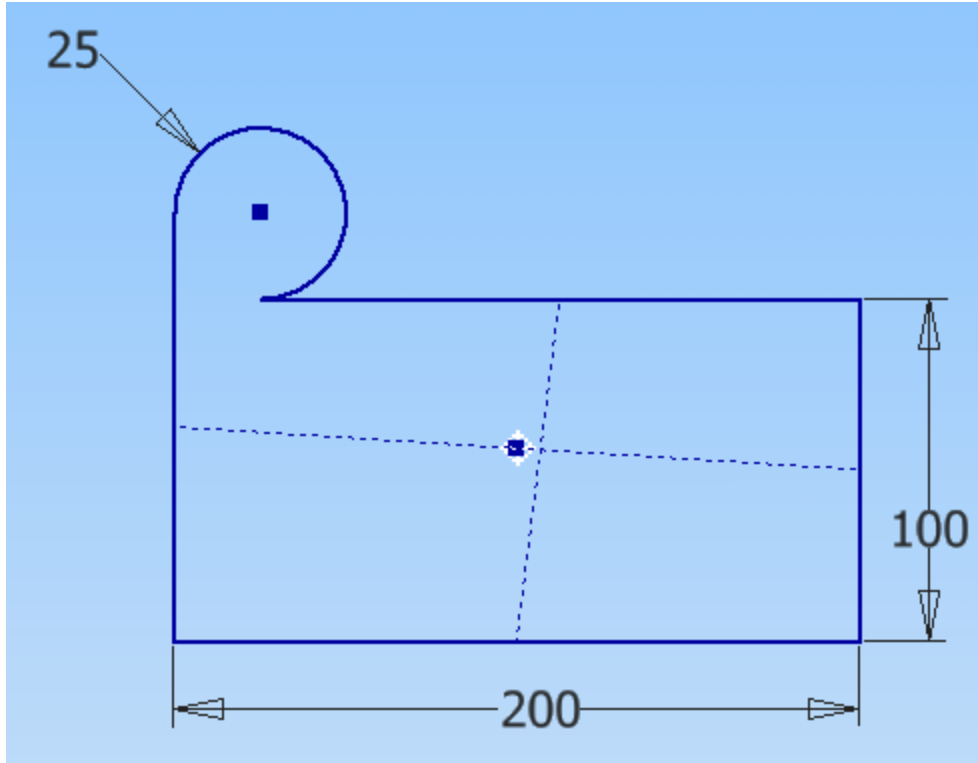
Tip 20 Using a horizontal and vertical line to midpoints can lead to unexpected results.

Tip 21 Using a horizontal and vertical constraints to midpoints can lead to unexpected results.

Tip 22 The most robust technique I have found is to use a diagonal construction line to constrain rectangles with symmetry about the origin.

Notice that a sketch changes color when it is fully constrained/dimensioned.

Make generous use of construction lines.

Tip 23 You can click the right mouse button and turn on sketch Degrees of Freedom as a visual guide to what dimensions/constraints are needed in a sketch.

I recommend simply dragging endpoints.
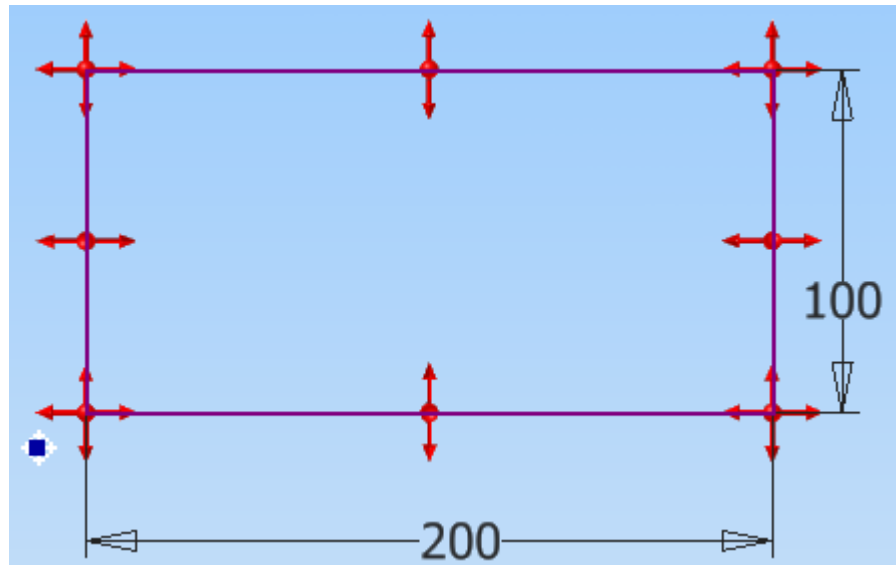
Inventor will automatically place many constraints.
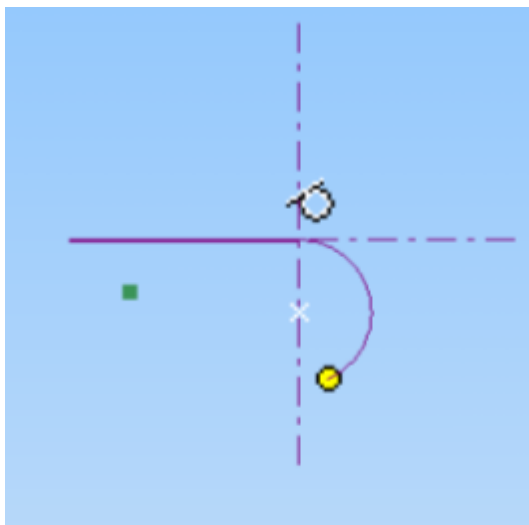
Tip 24 You can go directly into a arc while in the line command by clicking and dragging off the endpoint of the line.

Tip 25 If you click on a circle and hold the mouse button down and drag your second endpoint to a second circle Inventor will automatically place tangent constraints at both endpoints.
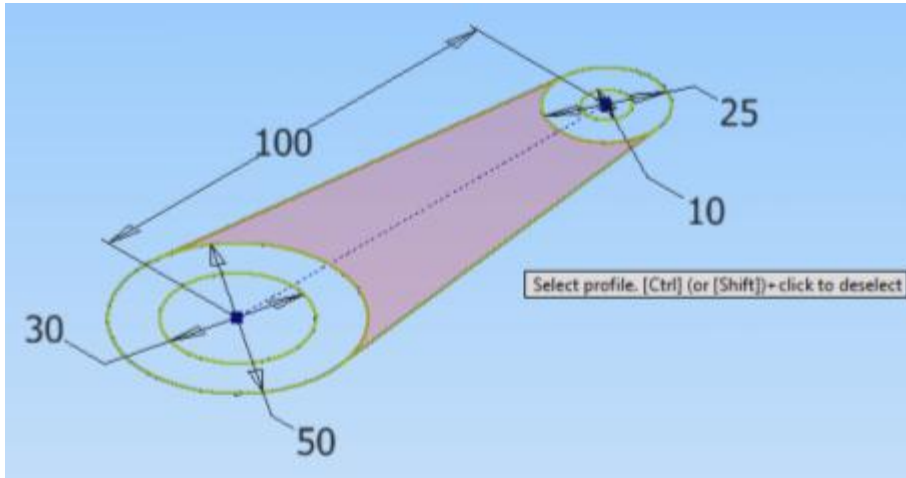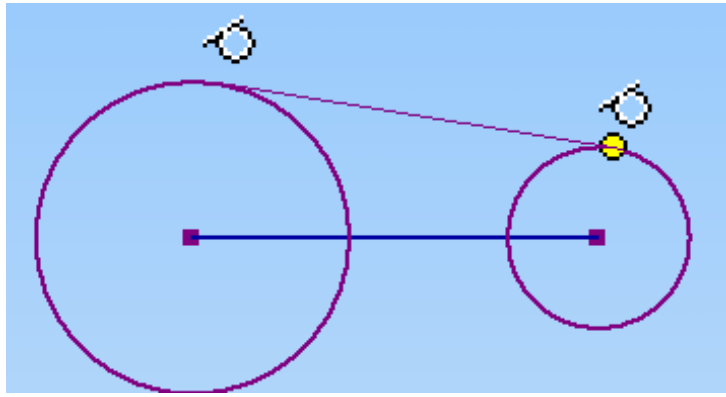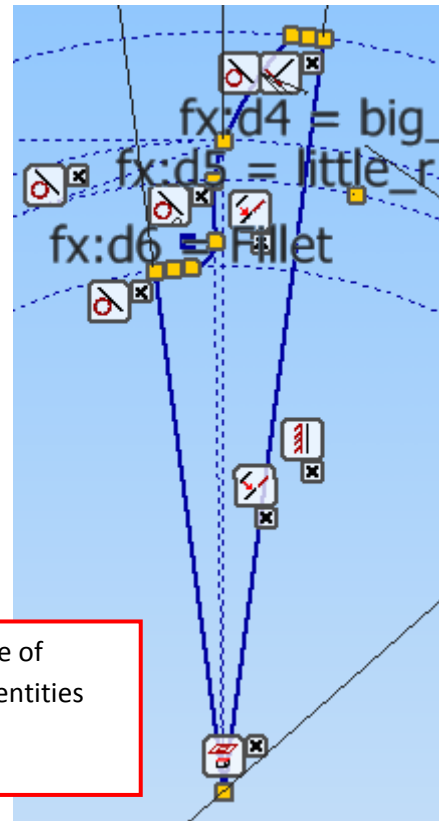
Tip 26 Avoid the use of Mirror of sketch entities if possible.
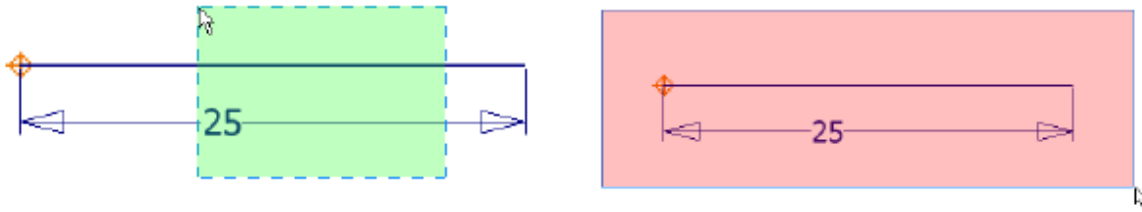
Use Symmetry constraints instead.

8 = Tooth_Bisect_Ang

fx:d4 = big_
fx:d5 = little_r
fx:d6 = Fillet
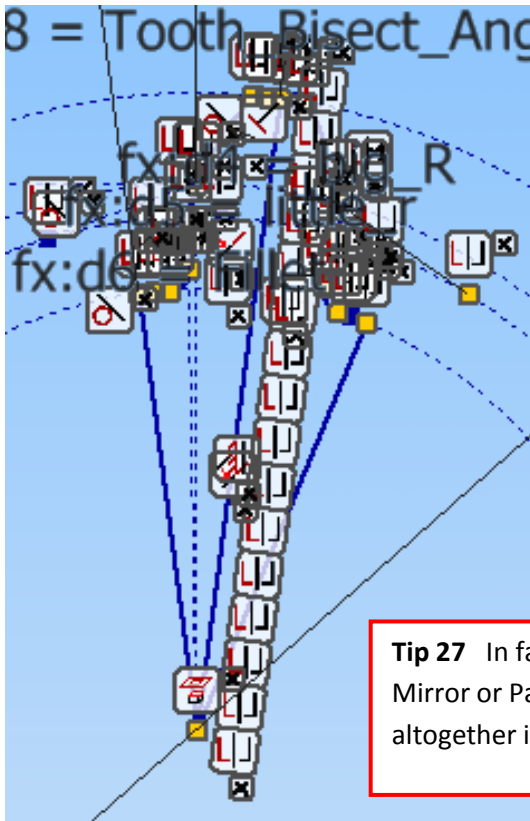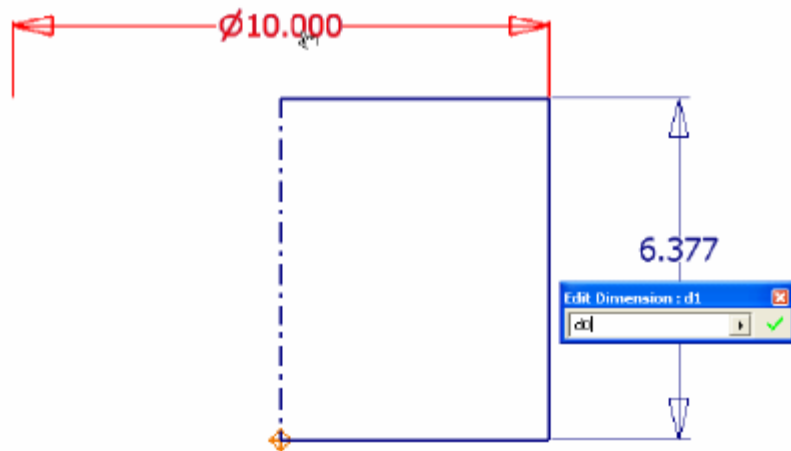
From this….

…to this.

Tip 27 In fact. Avoid the use of Mirror or Pattern of sketch entities altogether if possible.

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Tip 28 You can use window selection or crossing window selection just like AutoCAD.

**Tip 29** Dimension to a centerline (not endpoint) to get a diametral dimension.

Tip 30 Notice the hand glyph – do you know what it signifies?


Tip 31 You can enter units at any time and mix and match appropriate units as well as use function operators. If the equation in the dialog box is red then it is not valid – if black then it is valid.
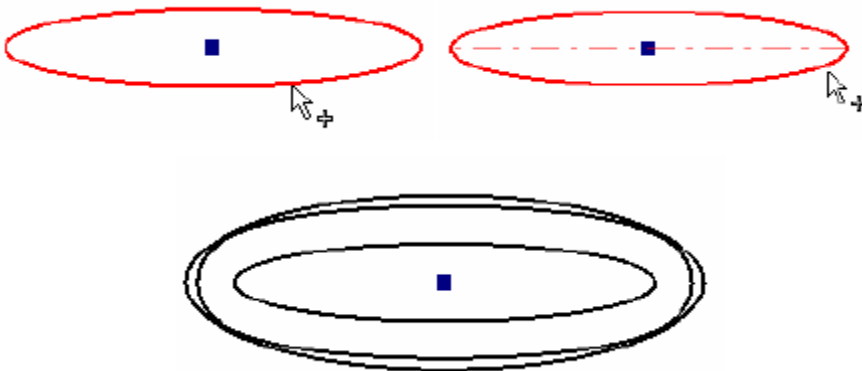

Tip 32 Offsetting an ellipse results in different solutions depending on where you select the ellipse. One result is a mathematic ellipse, the other is a constant offset distance.
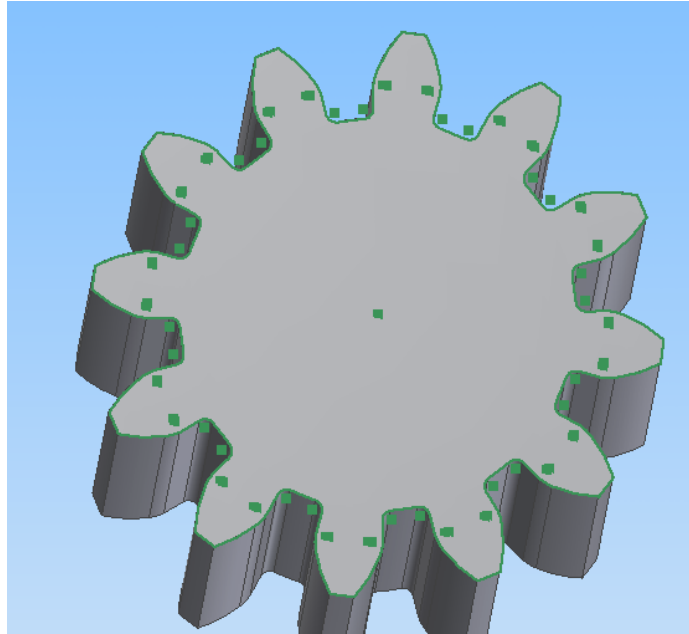
Tip 33 Remember earlier I suggested turning off the default Autoproject edges on sketch create. Notice how complex this sketch is with autoproject turned on.
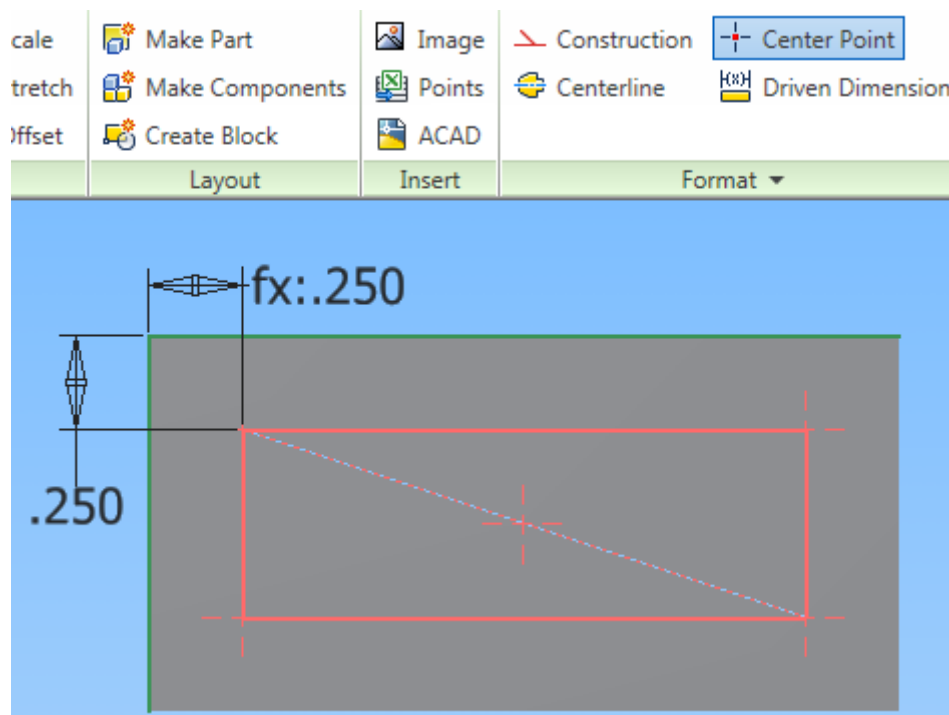

Tip 34 Copy sketches and features to save time and effort. Toggle endpoints to centerpoints for hole placement. You can Ctrl unselect hole centerpoint locations.
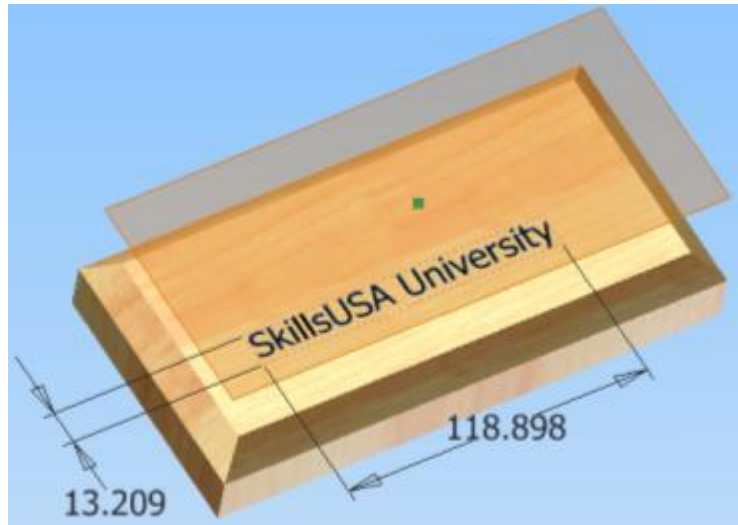
Tip 35 You can click and drag off of a planar face while in the Workplane command or simply start a new sketch and click and drag off of the planar face. (It is useful to know every method of creating a workplane.)

Tip 36 When placing text sketch, dimension the text box immediately – it doesn't matter what the dimensions are.
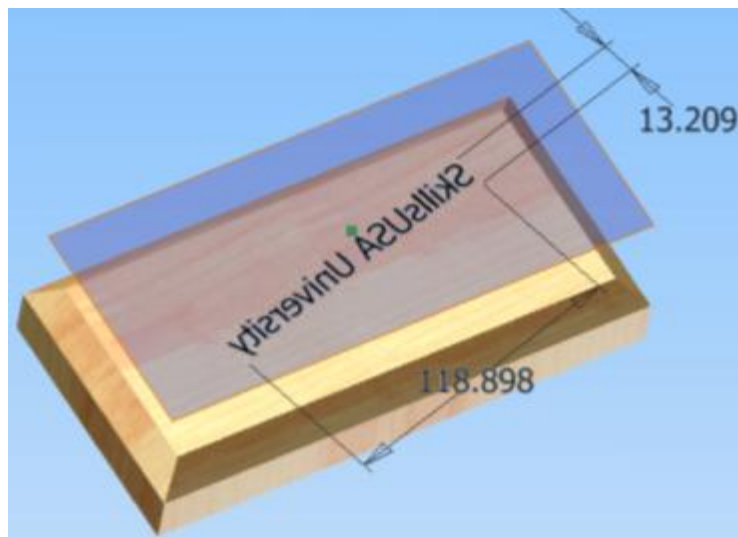

Tip 37 A workplane has a front side and a backside. Right Mouse button click on the workplane and select Flip Normal to flip a user created workplane.

Tip 38 A dimensioned text box can be dragged to any angle (predictable results from lower left corner).

14
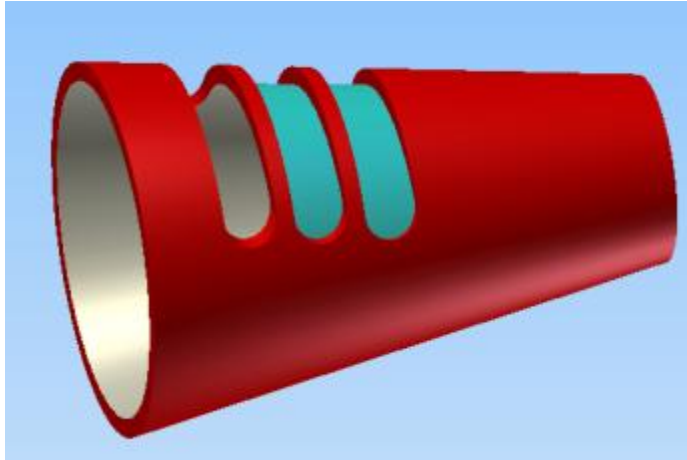
Tip 39 The Rectangular feature pattern command should probably be called Curve Driven Pattern.

But sometimes you can get unexpected results. In this case the design intent was for the patterned slots to cut entirely through the solid just like the first slot.
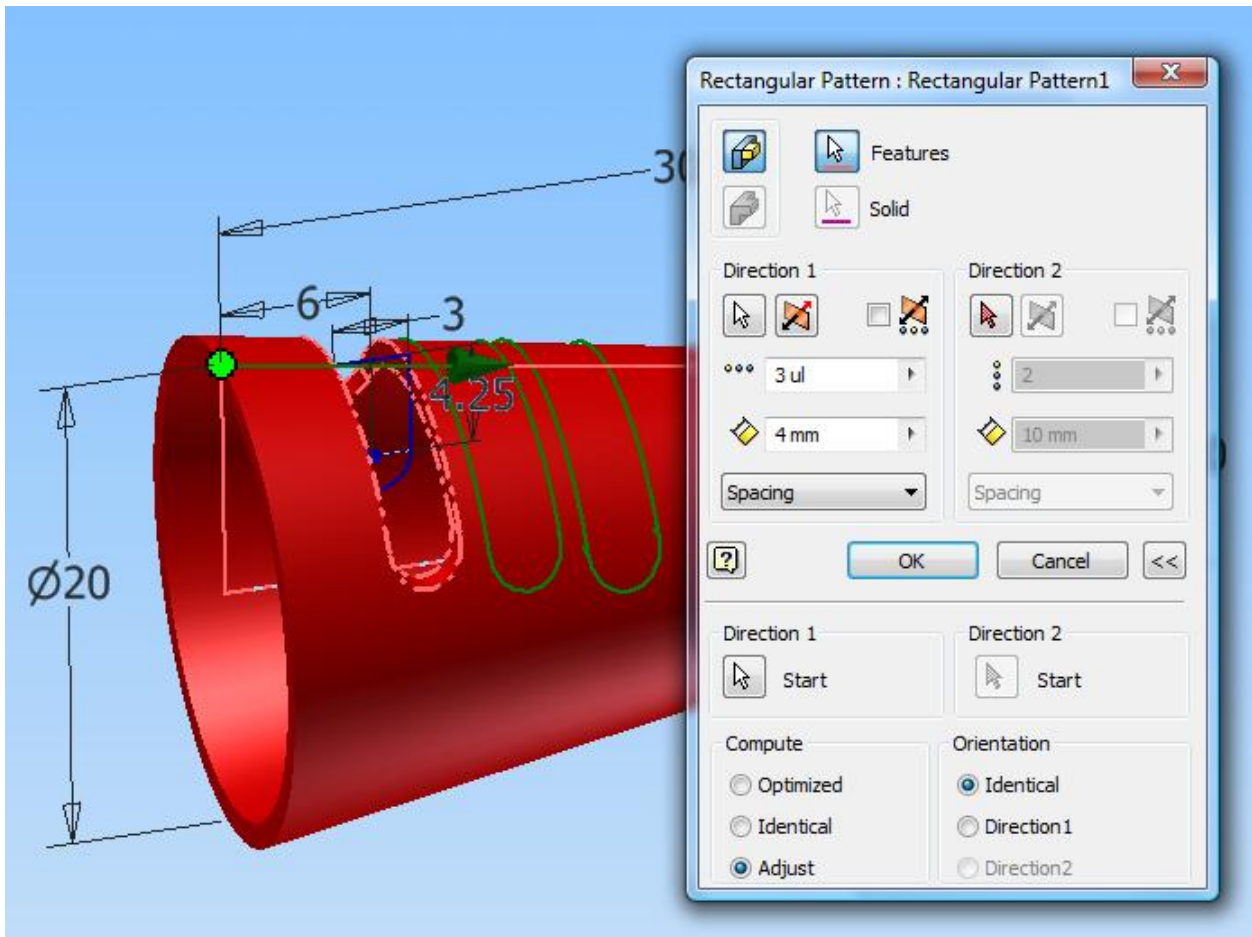
Tip 40 The design intent can be achieved by expanding the dialog box and selecting Adjust under the Compute heading.
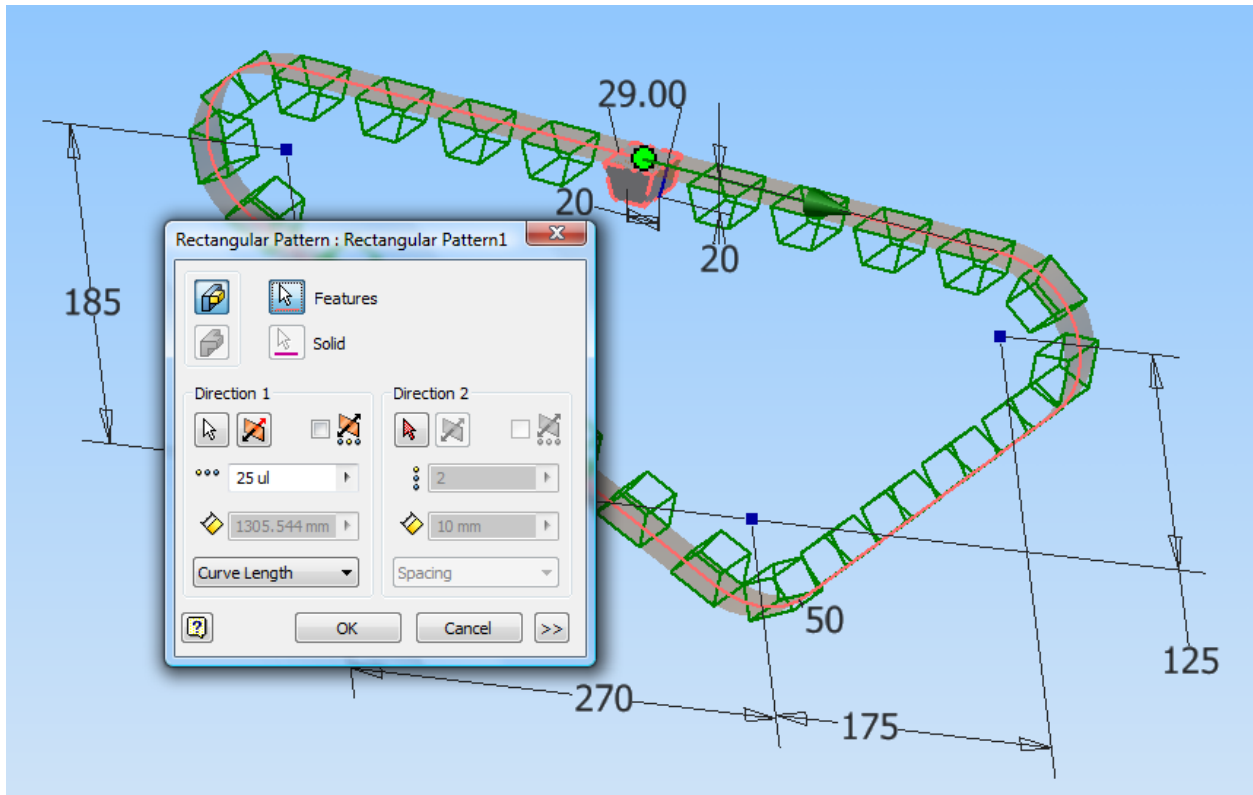
**Tip 41** For a more complex curve driven pattern expand the dialog box and move the Start Point to an appropriate location and set Orientation to Direction 1.

The design intent is for a pattern of tapped holes going around the cylinder and up the face of the cylinder.

Looks like a Circular pattern and a Rectangular pattern will be needed.

Tip 42 But the design intent can be accomplished with one "Rectangular Pattern.
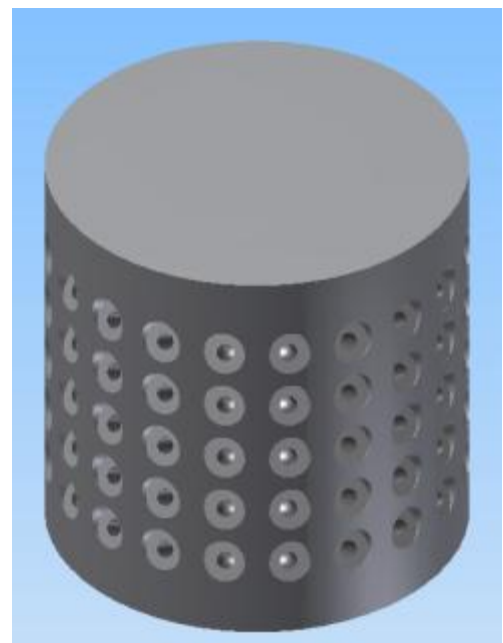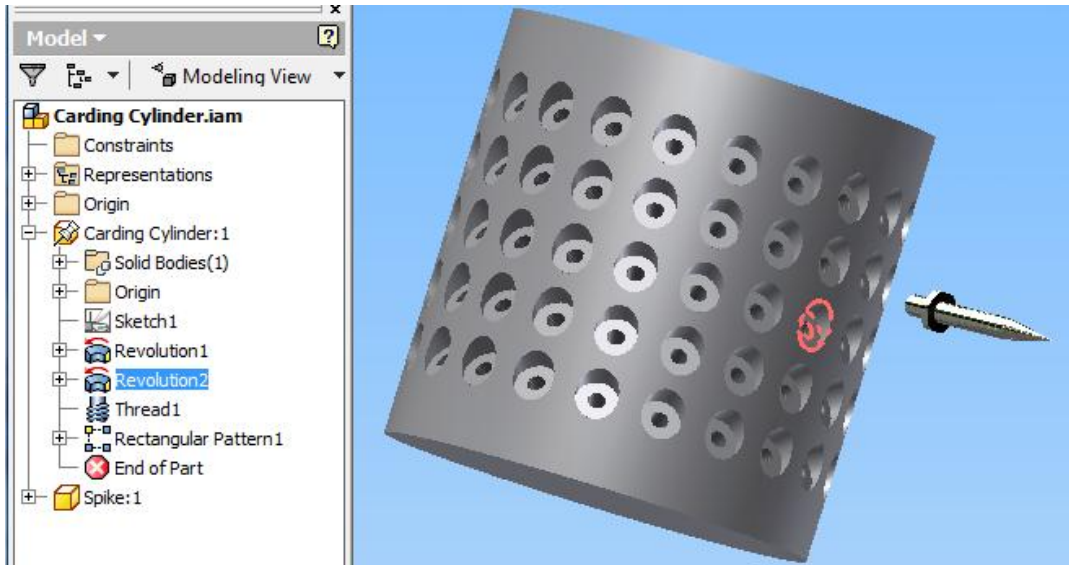
Tip 43 The first part brought in to an assembly is automatically grounded at the origin, but I recommend also constraining the first component to the origin by adding 3 Mate-Flush constraints between the XY, XZ and YZ planes.

Tip 44 You can use Place Component or simply drag and drop from Explorer to Assembly.

Tip 45 In an assembly set your browser to Modeling View to gain access to part feature tree.
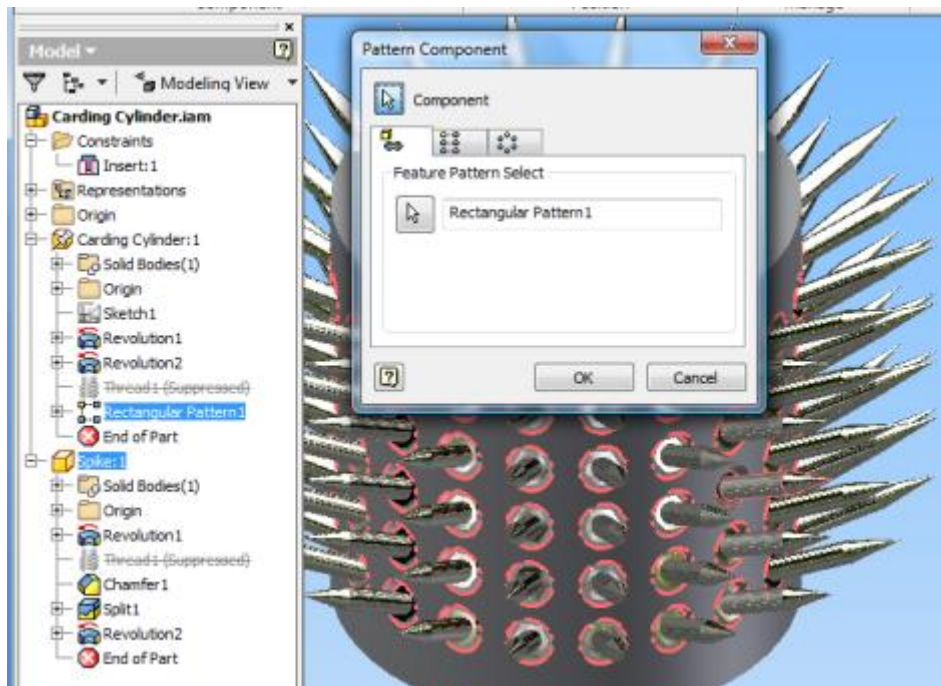
Tip 46 Feature Patterns can be used to generate Component Patterns.
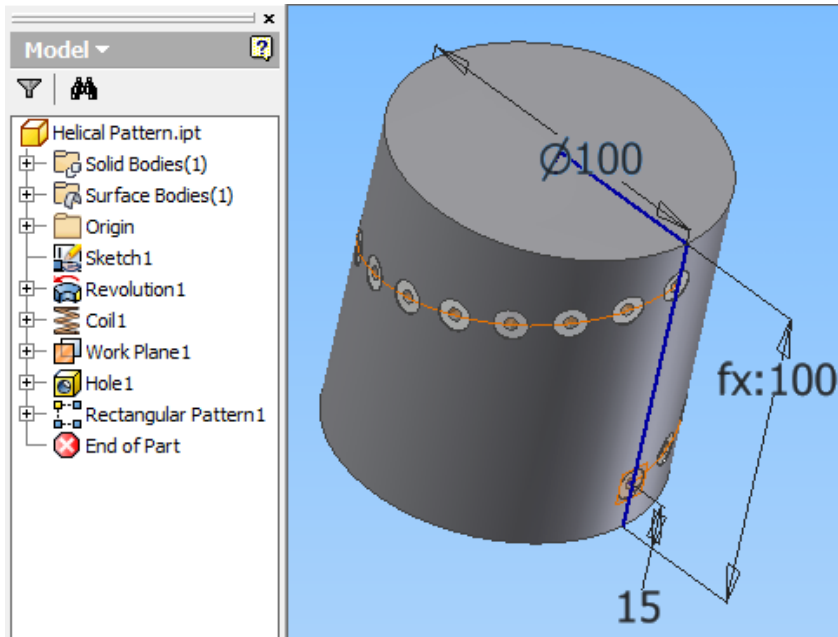
Tip 47 You can use the edge of surfaces for a curve driven pattern.

Tip 48 I usually don't worry too much about Top, Front or Right Side – it is all relative. You can change how your part is viewed on opening a file.
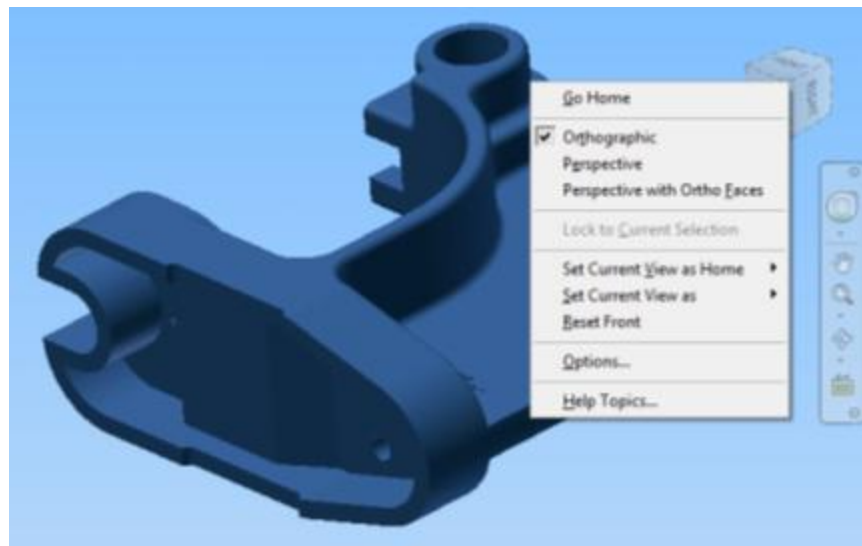
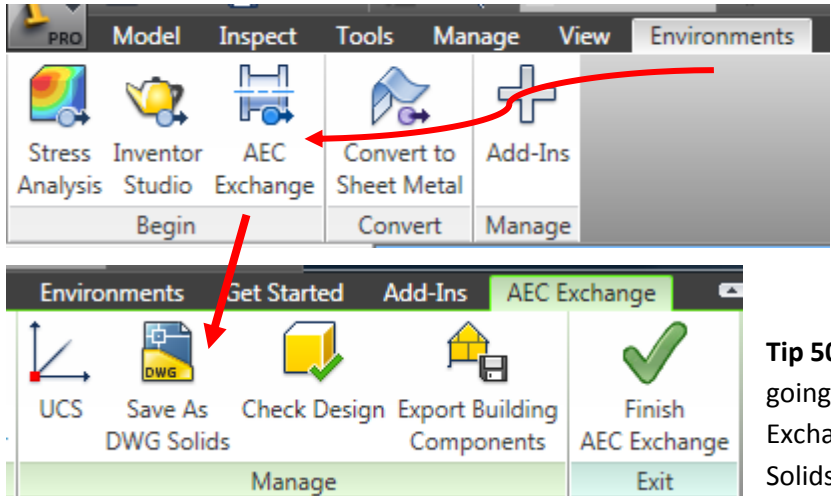**Tip 49** To add this icon to the Navigation Bar click on the arrow in the circle. You can then easily change view styles without going to the View ribbon tab.

Tip 50 You can Export DWG Solids by going to the Environments tab>AEC Exchange and select Save As DWG Solids.

Tip 51 When reusing AutoCAD data in Inventor you can opt to Open in idw environment or Import into ipt environment.
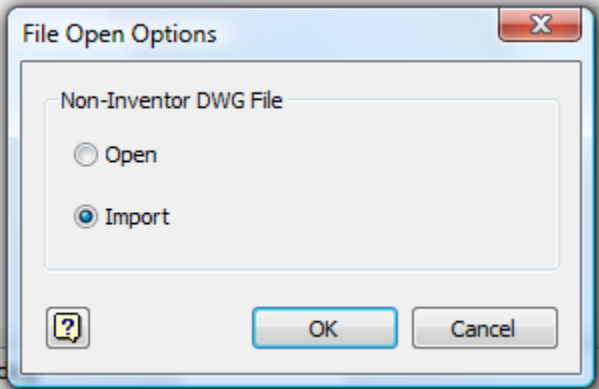
Tip 52 There are many other proprietary and neutral format file types that can be imported as well. Sometimes imported data is not of the highest quality. For information on repairing neutral format dat in Inventor see http://au.autodesk.com/?nd=class&session_id=3056 .

You might also review the other AU On-line Classes available on Inventor.
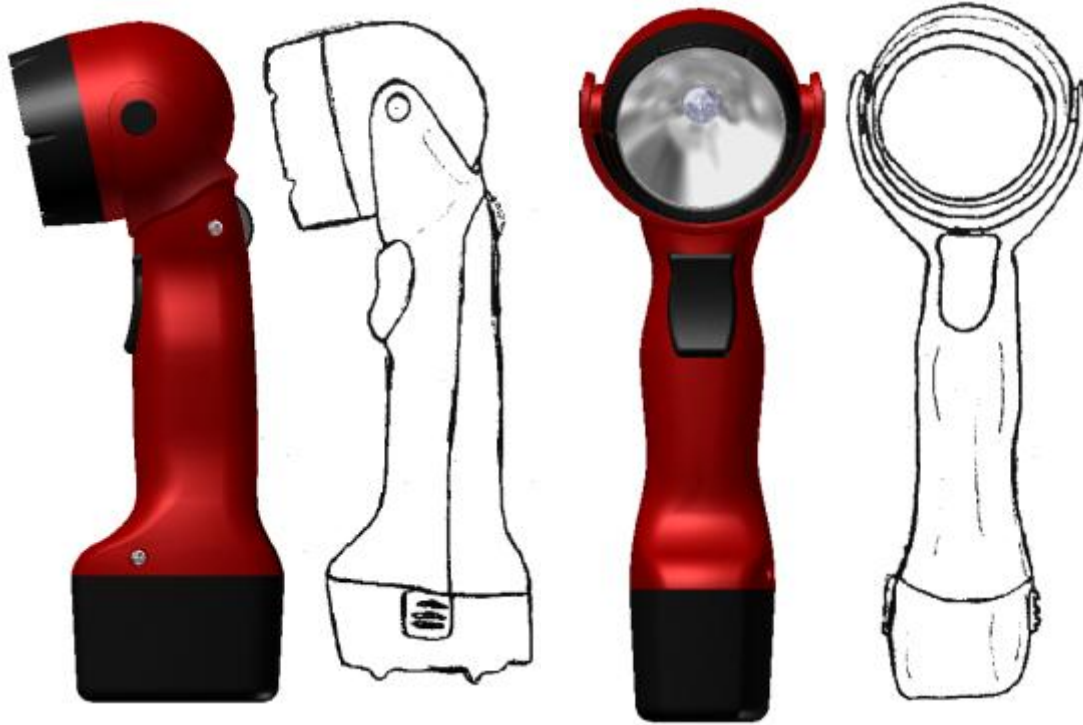
Tip 53 You can use scanned hand-drawn sketches or photographs as reference for creating models. Just don't take the imported image too literally (especially if, like me, you aren't an artist – add your design flare in Inventor).
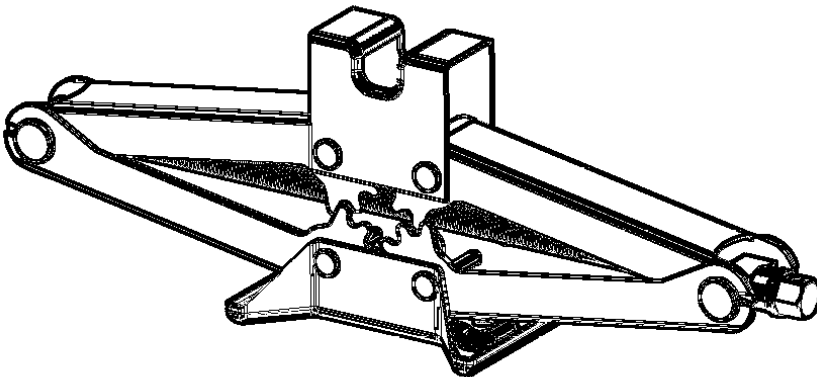
You might want to test out the kinematics[1] of a mechanism before taking time to fully model the components.

1. The branch of mechanics that deals with pure motion, without reference to the masses or forces involved in the motion.

Create schematic block representation of your mechanism.
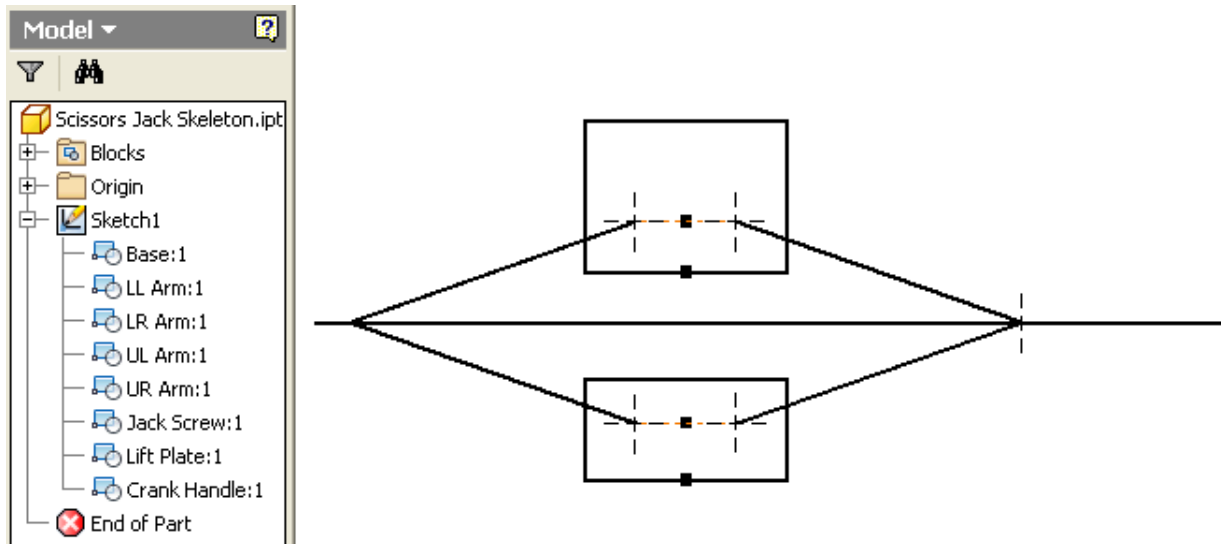
At this point we can test the 2D motion but not the 3D motion of the crank handle. Eventually we will want to create 3D components for each part of the mechanism, so let's go ahead and push out the individual parts now and create an assembly.
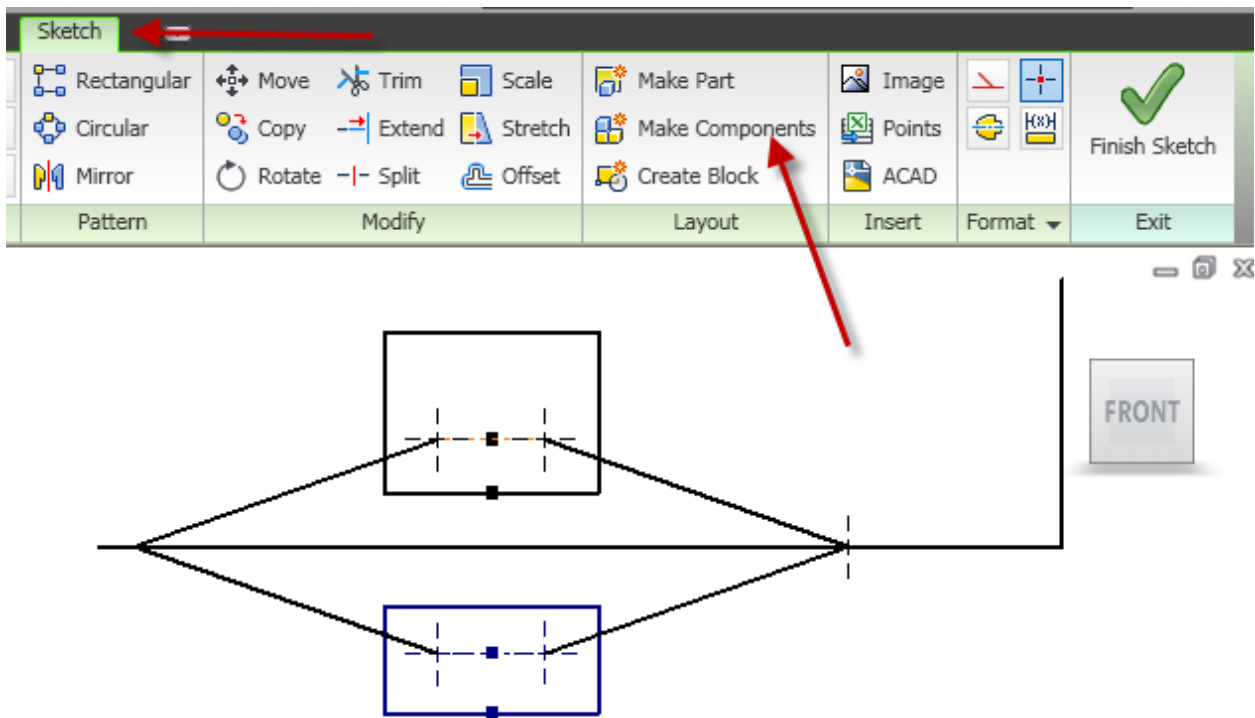
You must be in Sketch mode – click on the Make Components icon.

Make Components : Selection

Remove from selection

☑ Insert components in target assembly

- Base
- LL Arm
- LR Arm
- UL Arm
- UR Arm
- Jack Screw
- Lift Plate
- Crank Handle

Target assembly name
Scissors Jack.iam

Template
Standard.iam

Target assembly location
C:\Documents and Settings\jmather\Desktop\SkillsUniversity\3D Kinematic

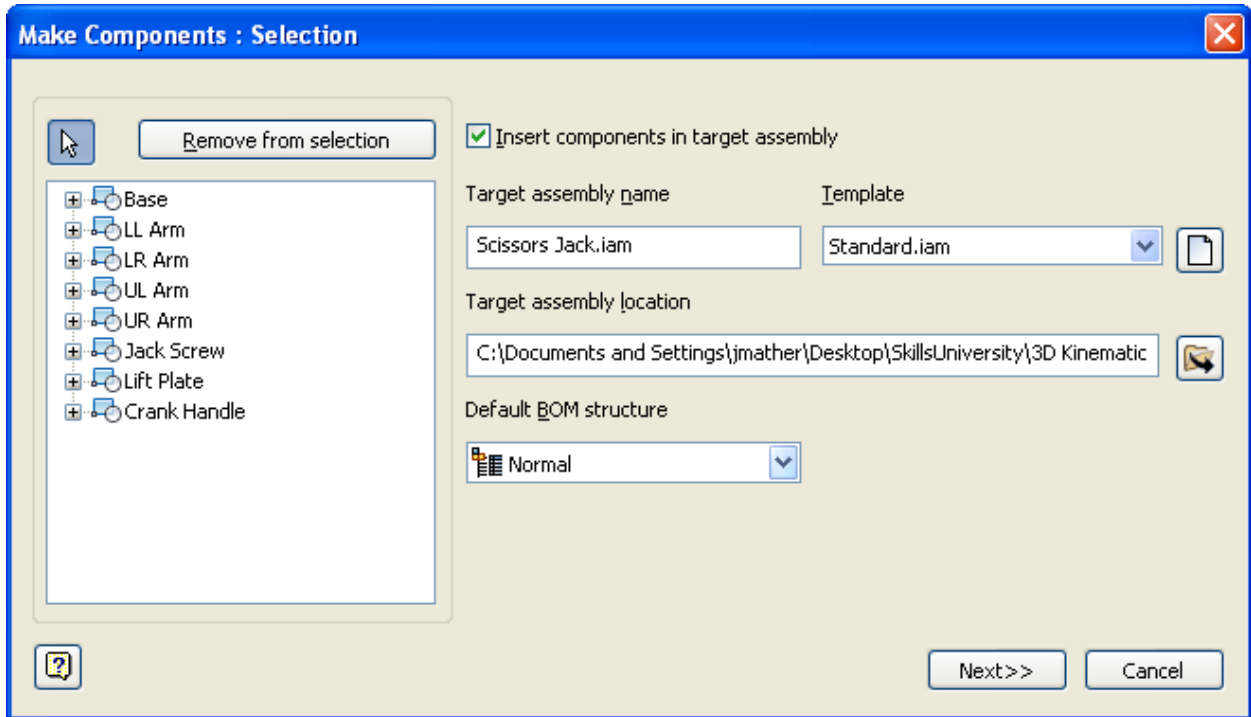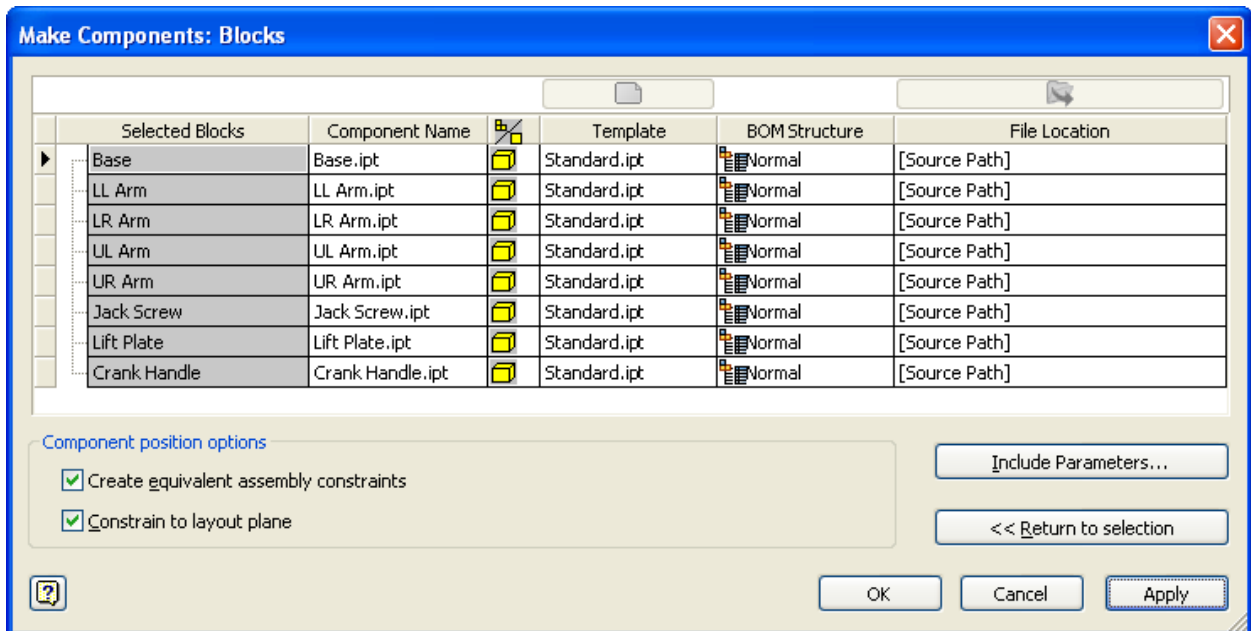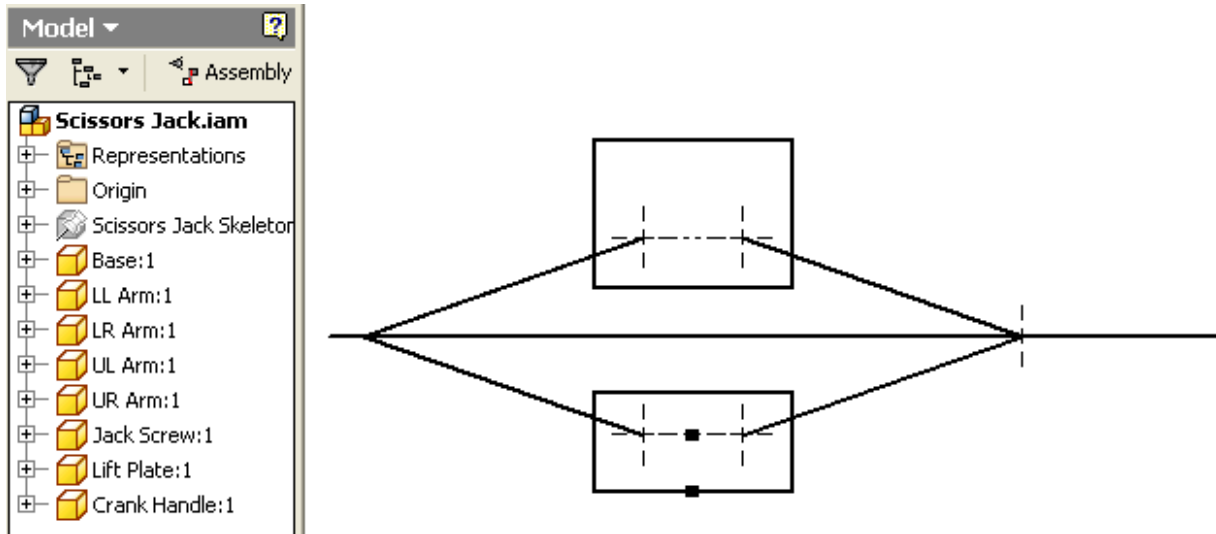Default BOM structure
Normal

Next>>   Cancel

Select all of the blocks from the browser to create components.

Make Components: Blocks

Selected Blocks	Component Name		Template	BOM Structure	File Location
Base	Base.ipt		Standard.ipt	Normal	[Source Path]
LL Arm	LL Arm.ipt		Standard.ipt	Normal	[Source Path]
LR Arm	LR Arm.ipt		Standard.ipt	Normal	[Source Path]
UL Arm	UL Arm.ipt		Standard.ipt	Normal	[Source Path]
UR Arm	UR Arm.ipt		Standard.ipt	Normal	[Source Path]
Jack Screw	Jack Screw.ipt		Standard.ipt	Normal	[Source Path]
Lift Plate	Lift Plate.ipt		Standard.ipt	Normal	[Source Path]
Crank Handle	Crank Handle.ipt		Standard.ipt	Normal	[Source Path]

Component position options
☑ Create equivalent assembly constraints
☑ Constrain to layout plane

Include Parameters...

<< Return to selection

OK   Cancel   Apply

Make sure the Component position options are selected for constraining.

We now have an assembly of components.

If we try to drag a component to test the motion we see that we have an unconstrained parallelogram. In the real world assembly the gear teeth on the arms will keep them parallel, but as we haven't modeled the parts yet we will have to add an extra assembly constraint.
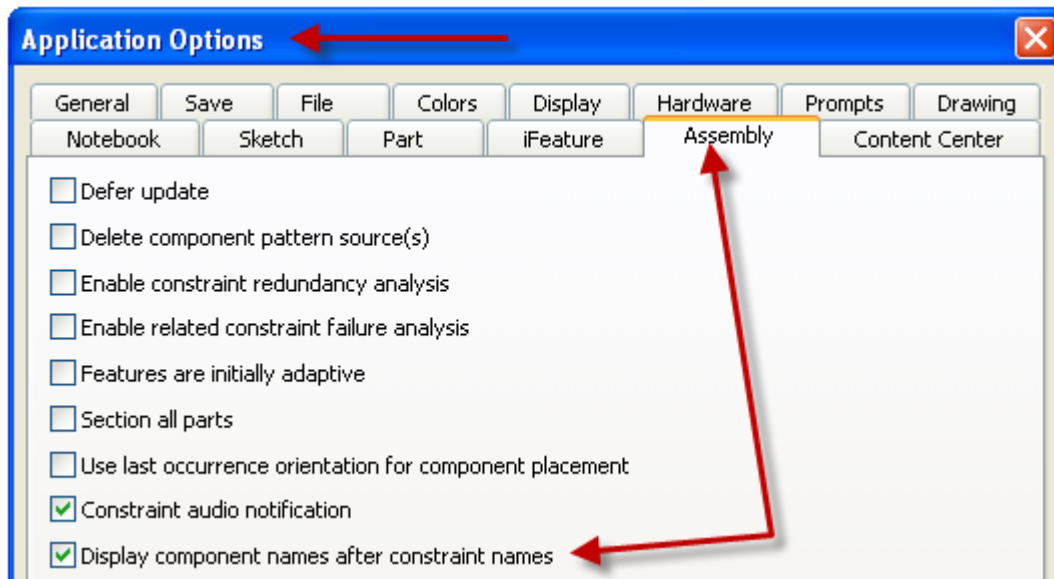
Before we continue it will be helpful to go to Tools>Application Options>Assembly tab and turn on Display component names after constraint names.
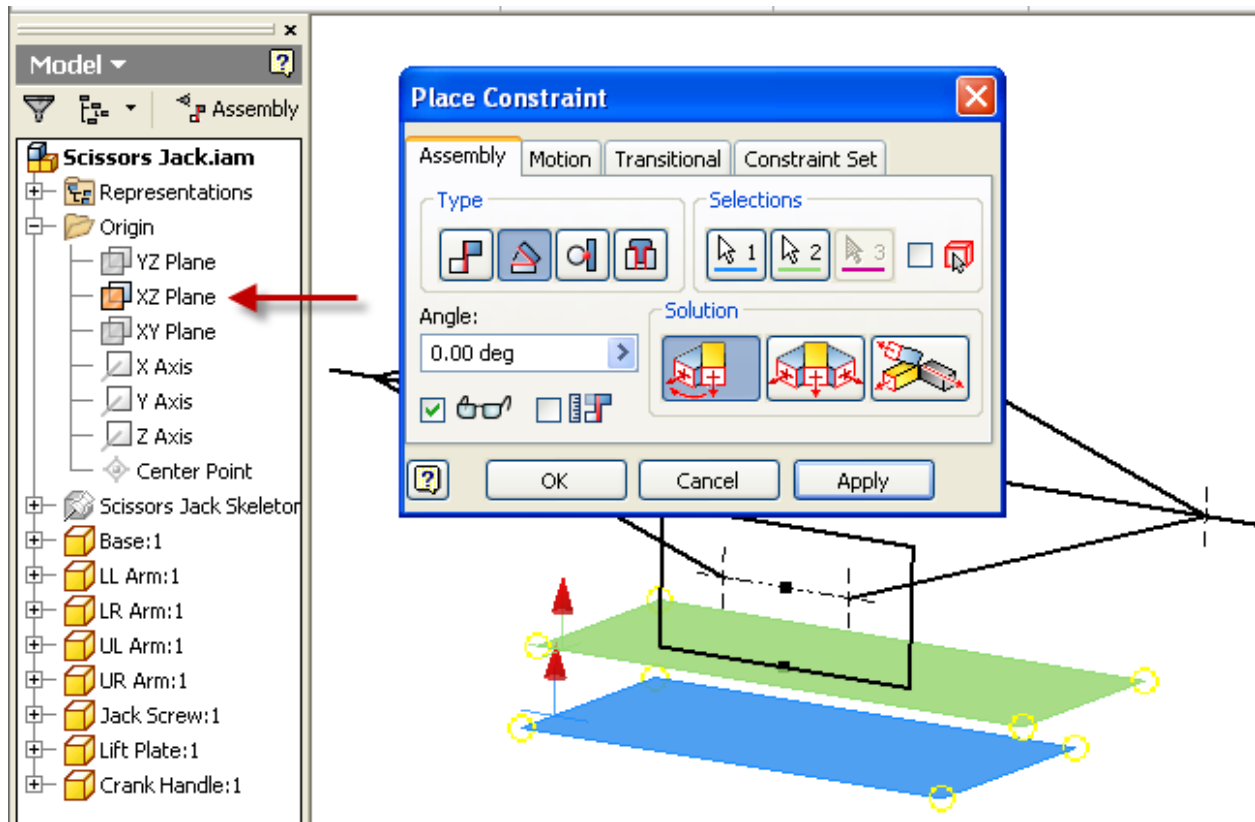
To force the Lift Plate to remain horizontal as the mechanism moves we place an angle zero mate assembly constraint between the XZ Plane of the assembly and the XZ Plane of the Lift Plate.
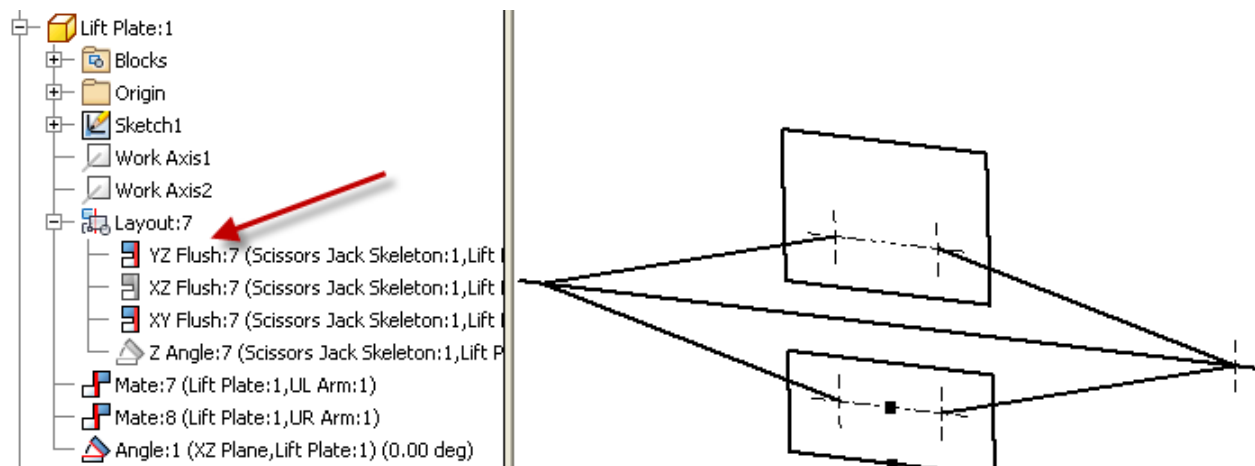

Since we haven't modeled the teeth on the arms yet we need to un-suppress the YZ Flush constraint that Inventor automatically applied to the Lift Plate.

Now we need to go through the rest of the assembly constraints that were automatically placed by Inventor and set the appropriate condition for each. A strong understanding of assembly constraints is

required to complete the rest of the problem. A disciplined step-by-step approach is required or it is easy to become confused. I expand each part one-by-one and analyze each constraint and the effect on the assembly. The Base component is fully constrained – so we don't need to change any of its constraints.
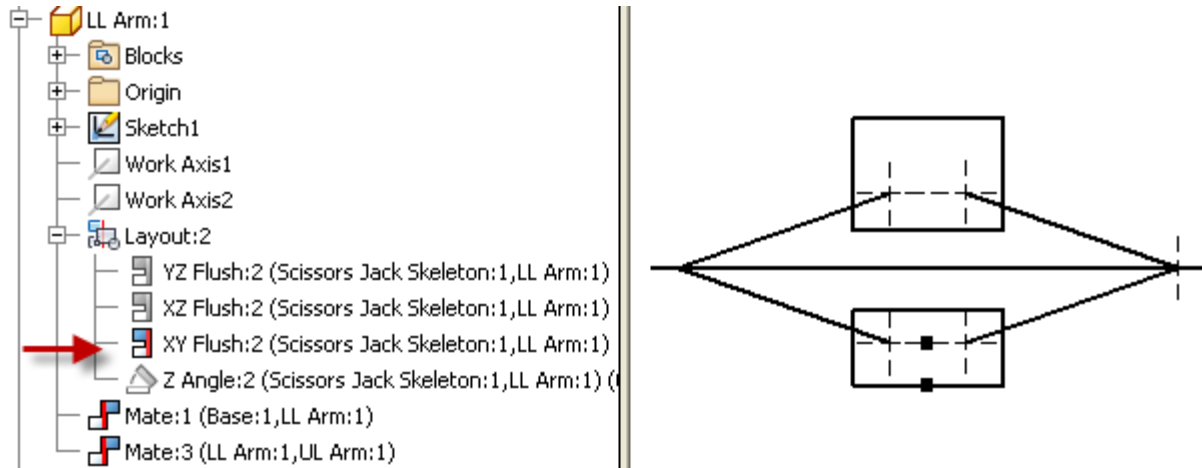

The Lower Left Arm (LL Arm) motion is constrained to the XY Plane – it needs to be free to move up and down on the XY Plane. It is constrained at one end to the Base and on the other end to the UL arm. No changes need to be made. The constraints on all of the arms are correct.
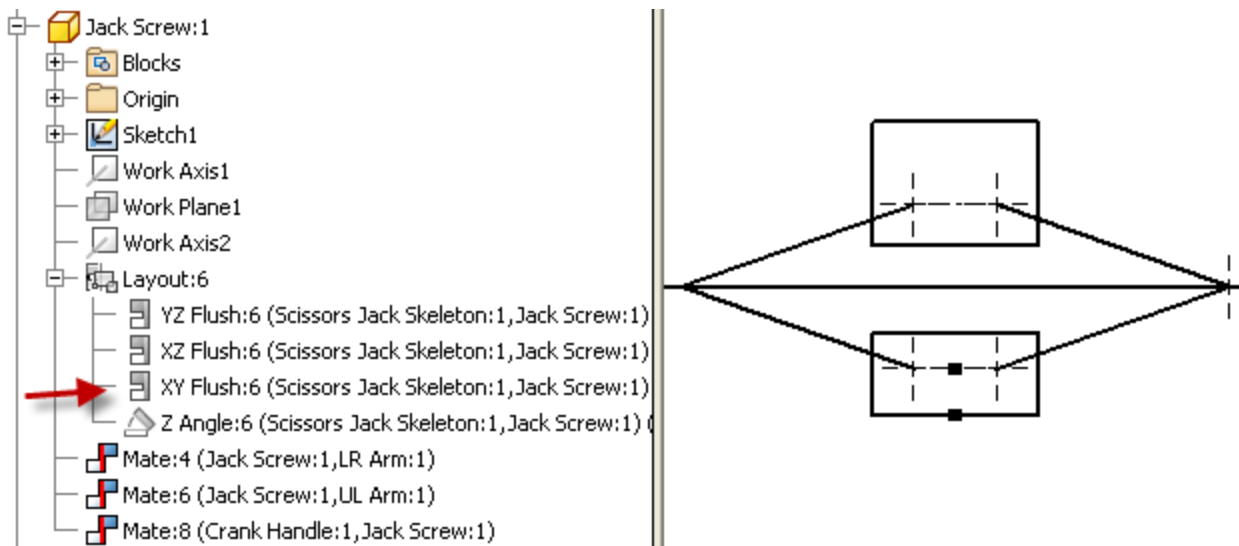

The Jack Screw need to be free to rotate, therefore we need to Suppress the XY Flush constraint.

Tip 54 Be careful – as you change between 2D and 3D Kinematics the constraint number and position in the list <mark>changes</mark>. You must change all three remaining constraints on the Jack Screw to 3D Kinematic. Check the Crank Handle – suppress the XY Flush constraint and verify that the Mate between the Crank Handle and the Jack Screw is set to 3D Kinematic. The actual identification numbers of the constraints were created automatically by Inventor and might be different for you. Pay attention to the part names.
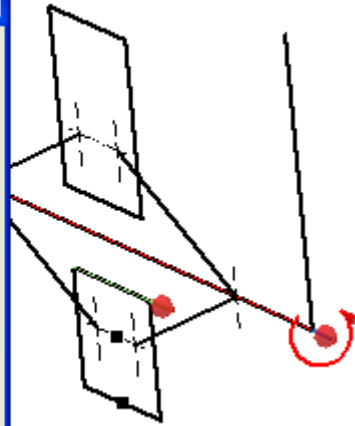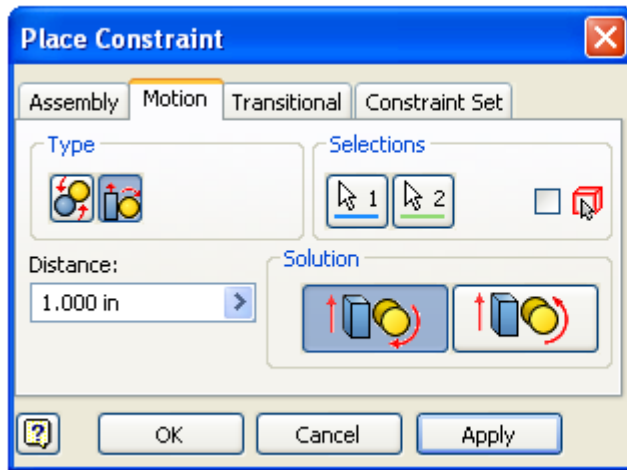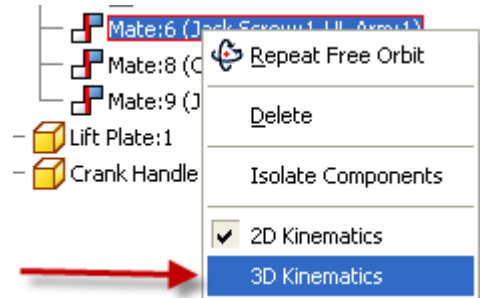
Testing our mechanism we see that the 2D motion is correct – we need to add one final constraint to get the 3D Motion. For now We will leave the Distance (ratio) on 1 as we can change that at any time. The easiest way to check the 2D and 3D kinematic motion is to click and drag the Lift Plate.
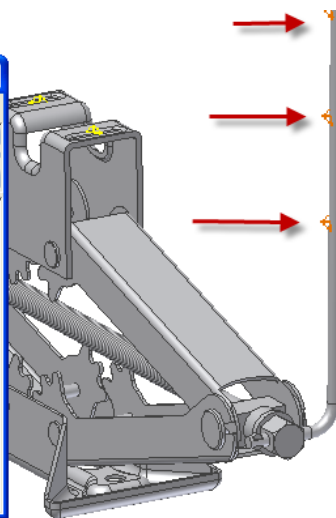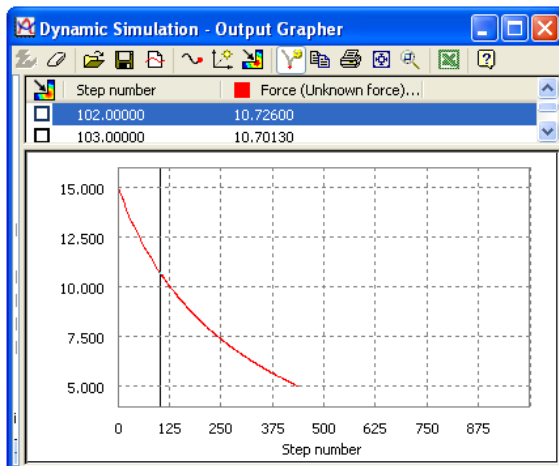
If you have Inventor Professional with Dynamic Simulation you can go beyond simple kinematics analysis and to a full Dynamic Simulation of the mechanism when you have the parts modeled. In this example the experiment was to see how the required input force would change depending on hand placement on the Crank Handle and position of the Lift Plate.
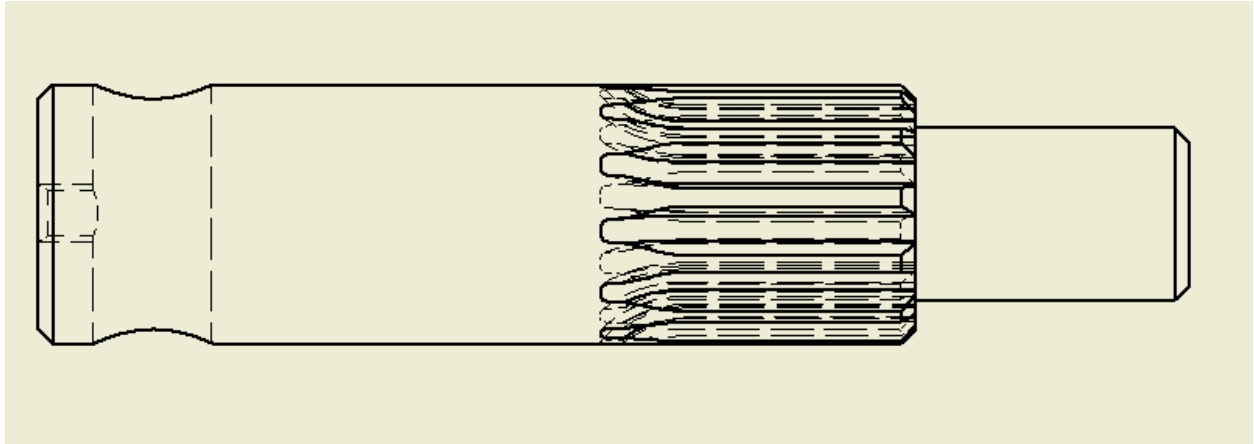
When creating 2D drawing views from 3D models the software creates the views quite literally – including details that might make the view a bit obscure. We can edit the view for clarity.
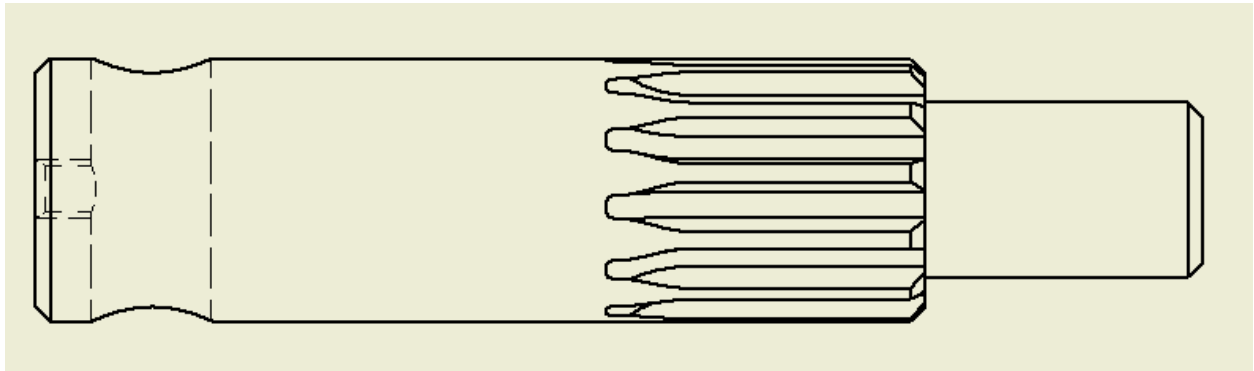

Tip 55 The job of editing can be simplified by judicious use of layers. I temporarily turn off the Visible line layer and select the lines I wish to hide. You can hide lines by right clicking and selecting Hide after selecting the entities to hide. Then turn the Visible line layer back on.
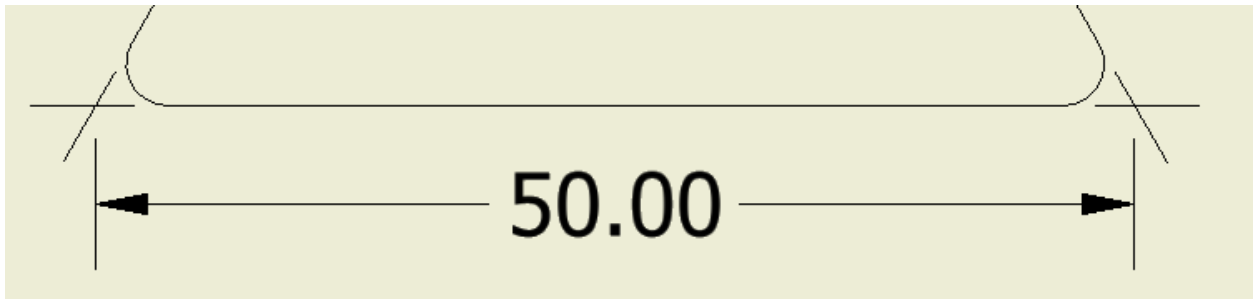
Tip 56 One of the most common questions that come up is, "How do I dimension to an apparent intersection?" It is actually quite easy, but takes a bit of practice at first. Pick the first line and then Right Mouse Button and select Intersection and click the intersecting line. In this example there are two projected intersections – so pick the third line and Right Mouse Button select the intersecting line again after opting for intersection.
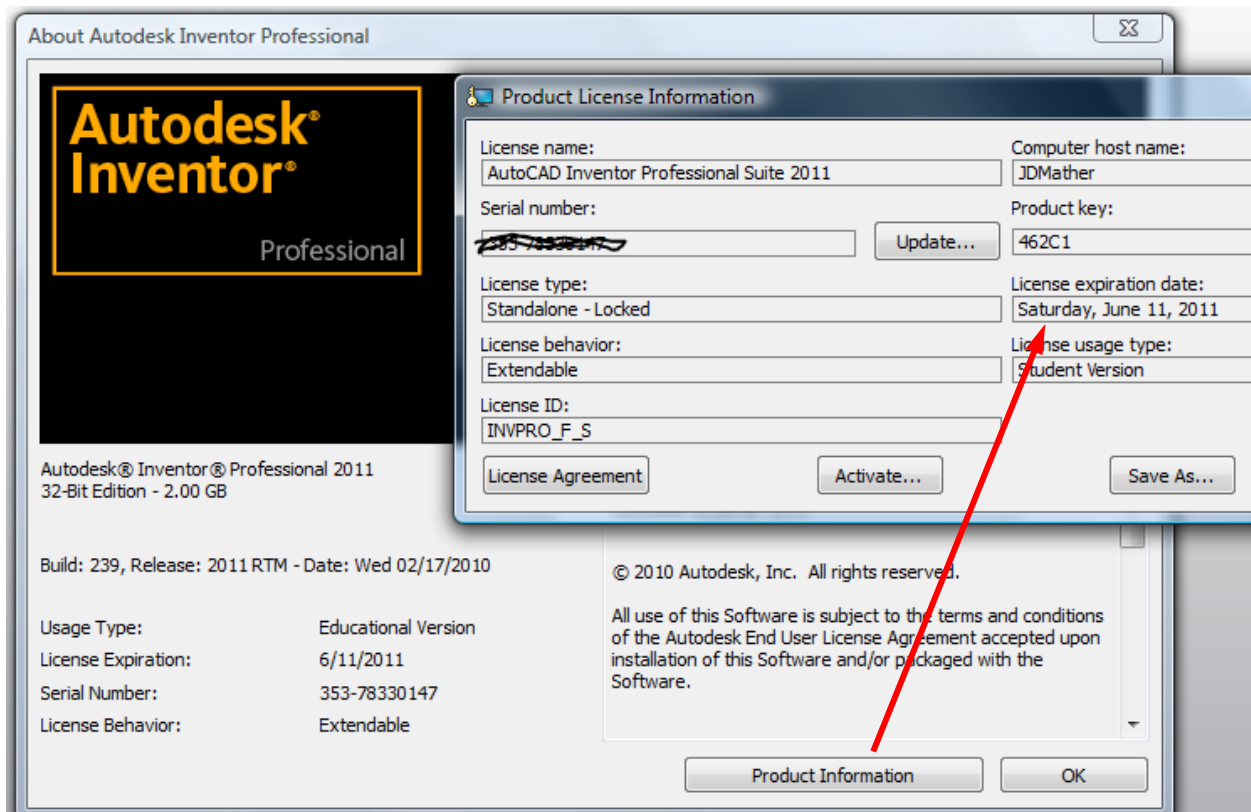
Tip 57 When you are annotating a drawing you can gain Windows Character Map in any command that uses text.
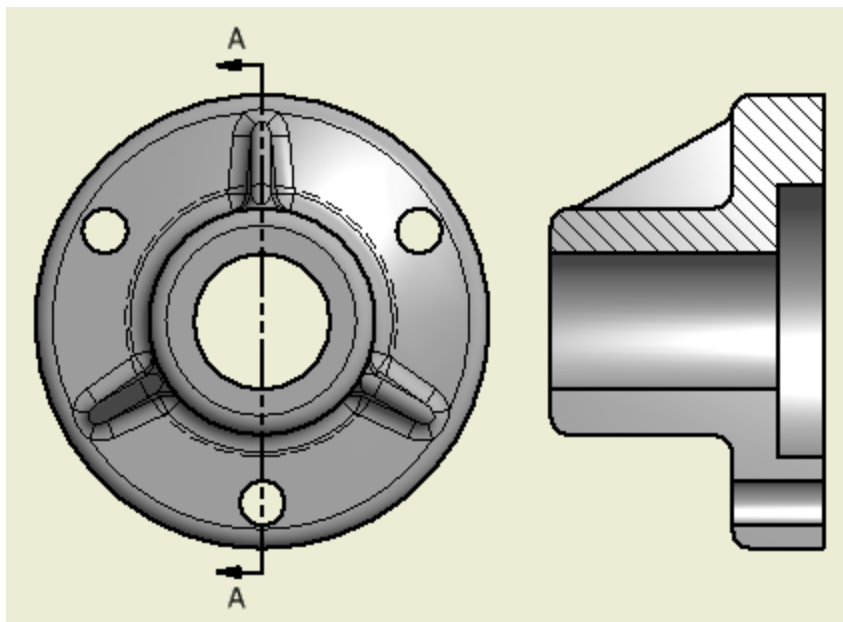
Tip 58 What is the purpose of creating drawings?

To communicate design intent.

We can do what is needed to convey design intent while also incorporating standards. An example is the traditional practice of showing ribs in section.

Tip 59 If your students are running the student version on their own computers they can verify the license expiration date by going to Help>About Inventor>Product Information.

Tip 60 <mark>Do Not</mark> double click on Big I - as this closes Inventor (or any Windows Ribbon program). Sometimes you might click on this icon and it is a bit slow to expand, you might then click again and the program suddenly closes. When students do this they might get the impression that the program just crashed, but this is normal Windows Ribbon behavior.

Thanks, and enjoy SkillsUSA University.

Send questions to jmather@pct.edu