Team 3176 White Paper

Title: Intro to Autodesk Inventor	Author(s): Angelo Botha, Jacob Engel, Vincent Phan
Subteam/Function: Design Training	Subject: CAD
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Abstract

This paper is designed with the intention of helping team members who are training to be on the design team gain a basic understanding of how to use Autodesk Inventor, excluding drawings. It is oriented towards those teaching someone who is new to using this program, or a person who is attempting to teach it to themself with the aid of this document.

Definitions

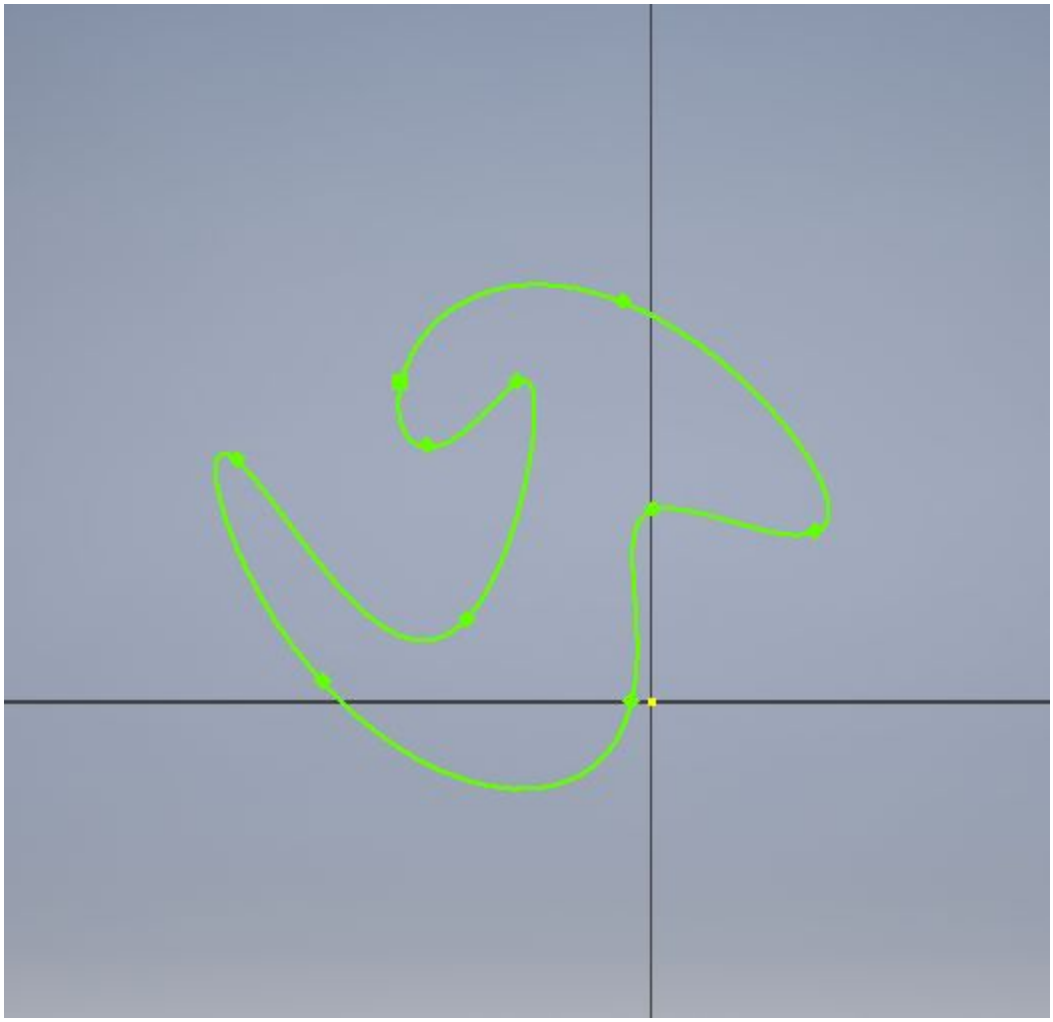
CAD - Computer Aided Design (such as Autodesk Inventor) used for creating 3D models of parts or assemblies

Sketches: Jacob Engel

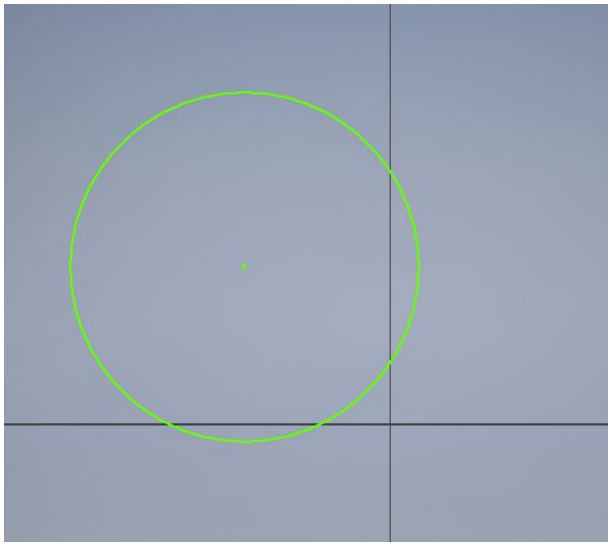
Before any parts are created, a 2D sketch should be made in order to have some base to work with. Once you have selected the plane on which you are to begin sketching on, there are many tools that you should be comfortable with.

-The Line tool does exactly what it sounds like it does, you select/create 2 points and it creates a line between them. The drop down from this tool includes spline, equation curve, and bridge curve.

-The Spline tool creates a spline curve (a curve using piecewise functions) through multiple points selected the same way as a line but with as many points as you like. It is not recommended that this tool is used when making parts for the robot as they are very irregular and hard to dimension.

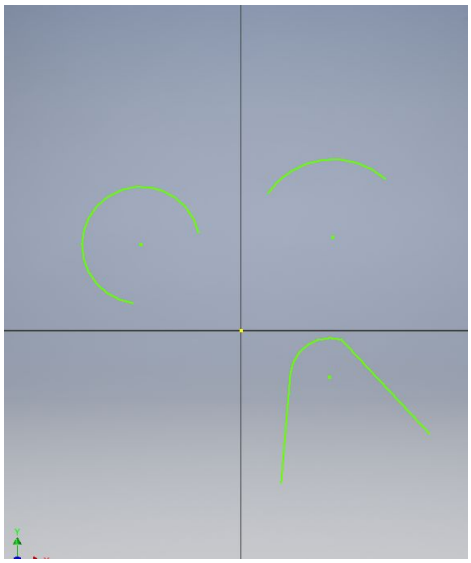


Circle- Select a point then it created a circle centered around it of any size.

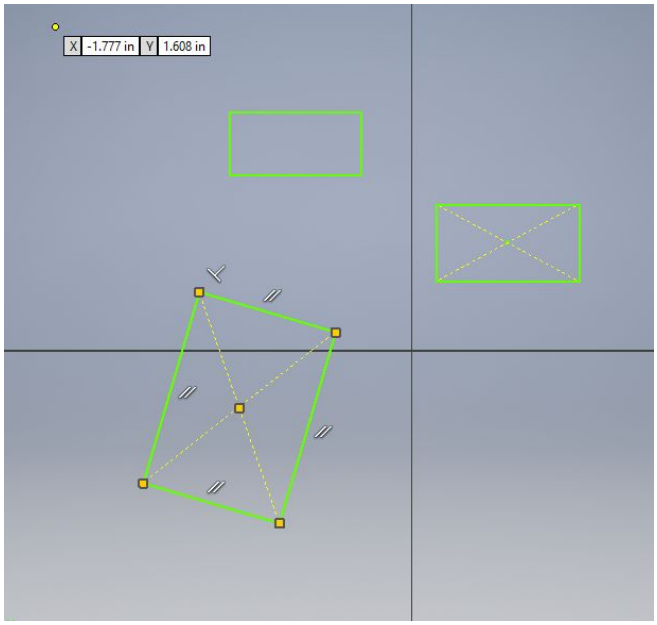


Ellipse- This tool selects a point and then you input two dimensions, for the arc to extend to on each side.

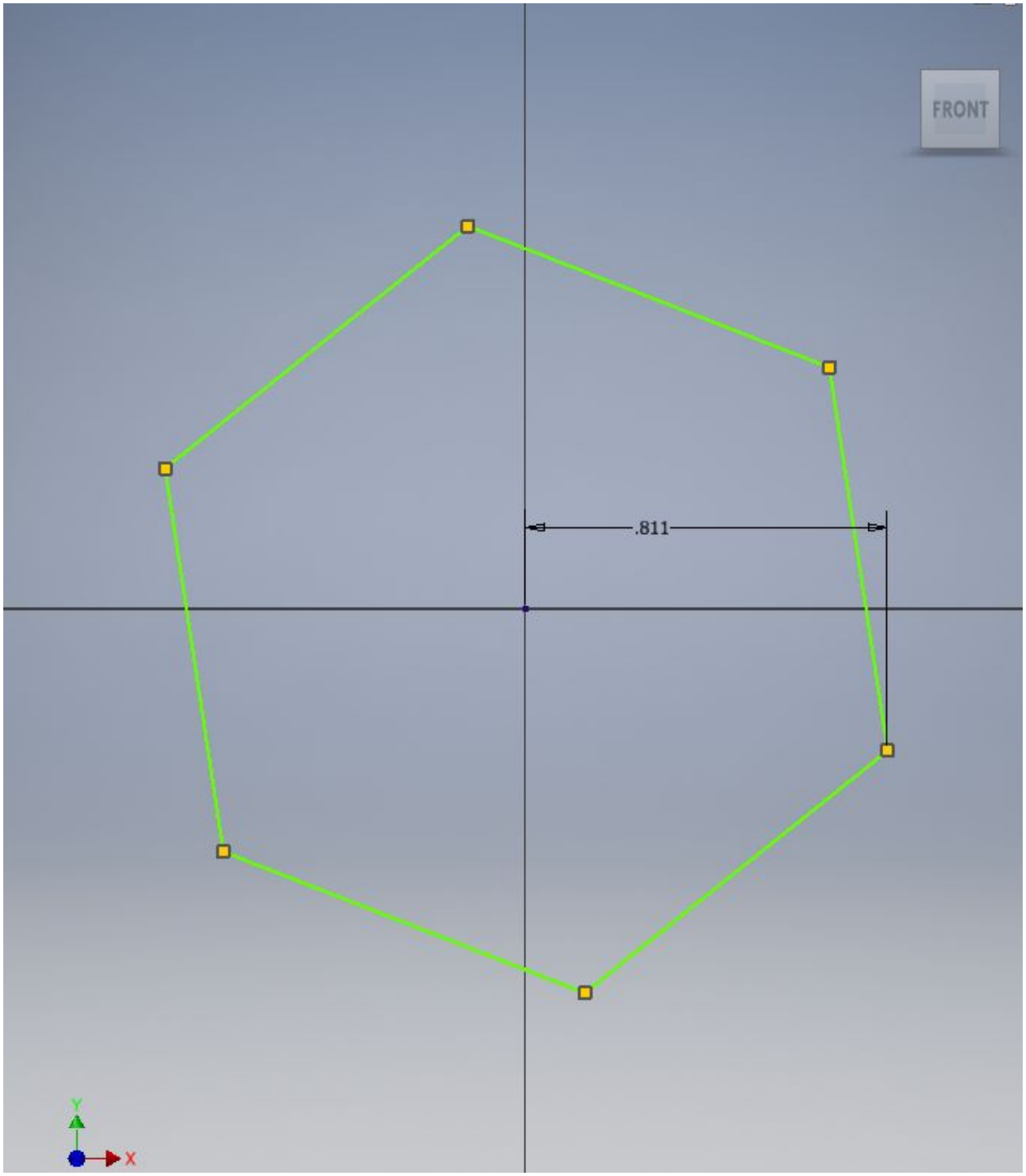
Arc- There are three methods of arc (a circle segment) creation, one is to select two points and then drag out how much of the circle you want incorporated. The second is to select a centerpoint ad drag the arc segment that you want around that at a fixed distance. The last way is to select two points at the ends of lines and an arc will be created tangent to them.



Rectangle- There are three methods for creating rectangles. The first is to use two diagonal points, spaced out however far you want each side to be. The second is to have a centerpoint that you want selected and place a corner away from that at the distance that creates your desired rectangle size. The last method of rectangle creation is to select 3 corner points so that it sits at an angle.

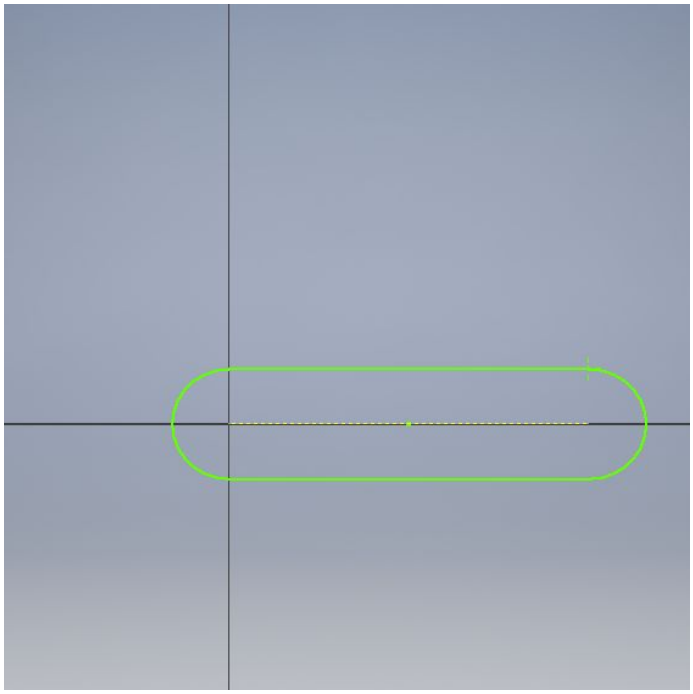


Polygon- First, click under rectangle in the 2D sketch. Select polygon at the bottom of the column. Then, choose what side you want to spawn the polygon from and select the number of shapes you want the polygon to have. Finally, click and hold to where you want the polygon to go on the 2D sketch. You can edit the size of the polygon in dimensions. Your polygon will always be regular.

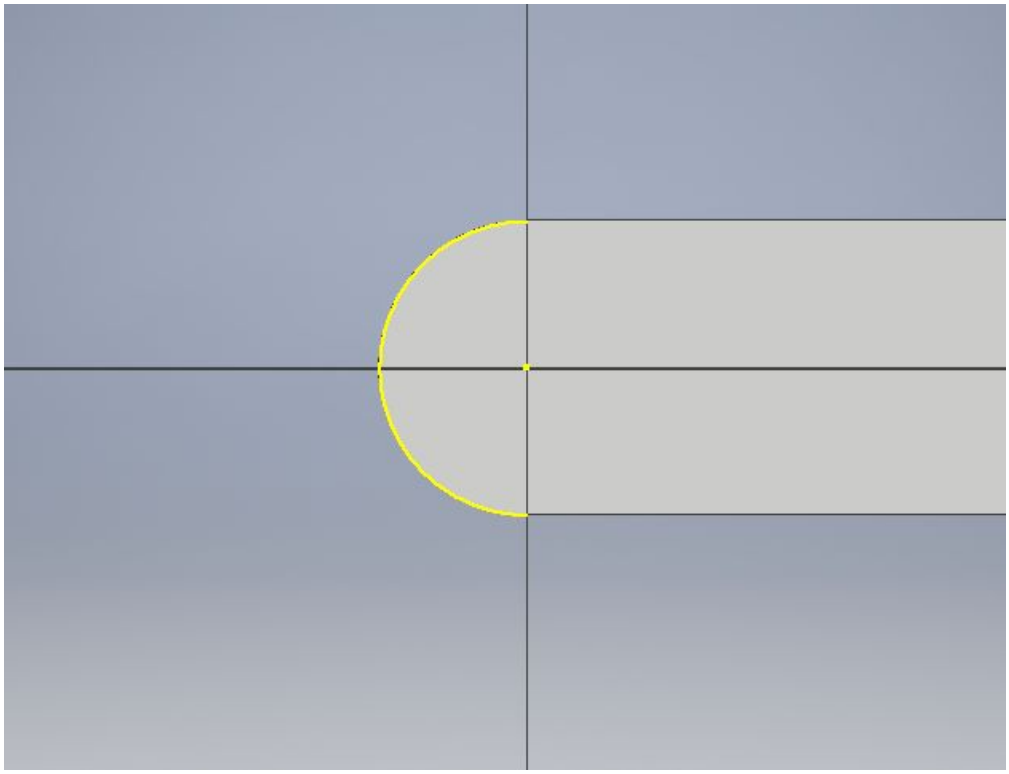


Fillet- When you select the fillet tool it creates an arc between two connected lines, of any desired radius, that is tangent to both of the lines.

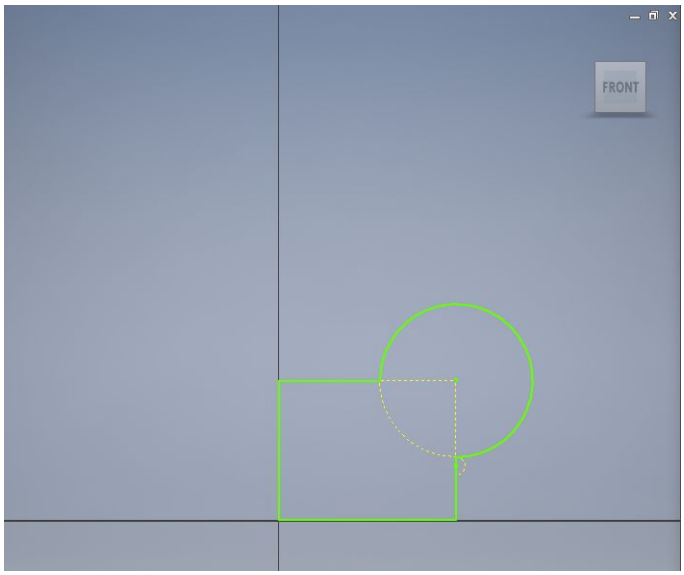
Point- If you click on a point, it creates a centermark there that will remain even if not constrained to anything.



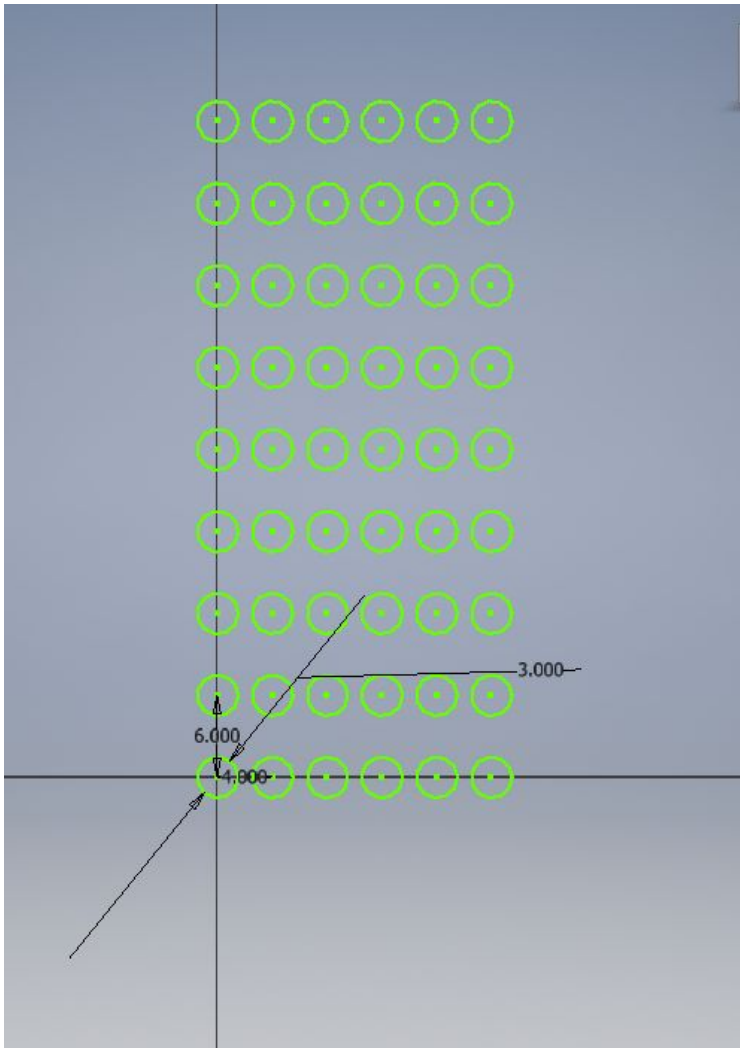
Project Geometry- This will overlay geometries from a different plane onto this one, fully constrained.



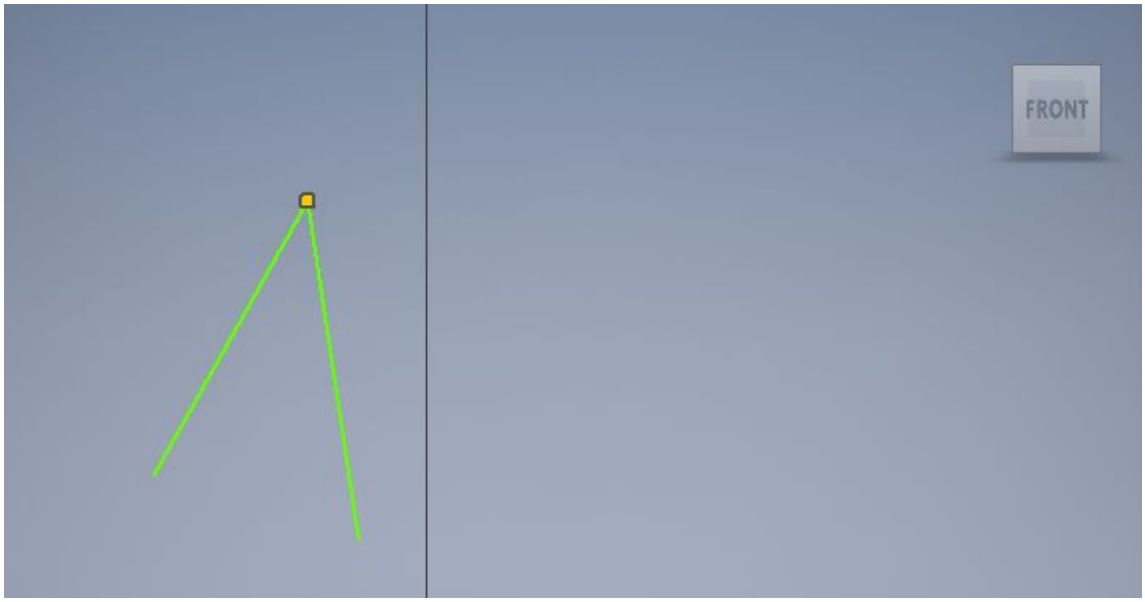
Trim- This tool will remove excess of selected intersecting lines



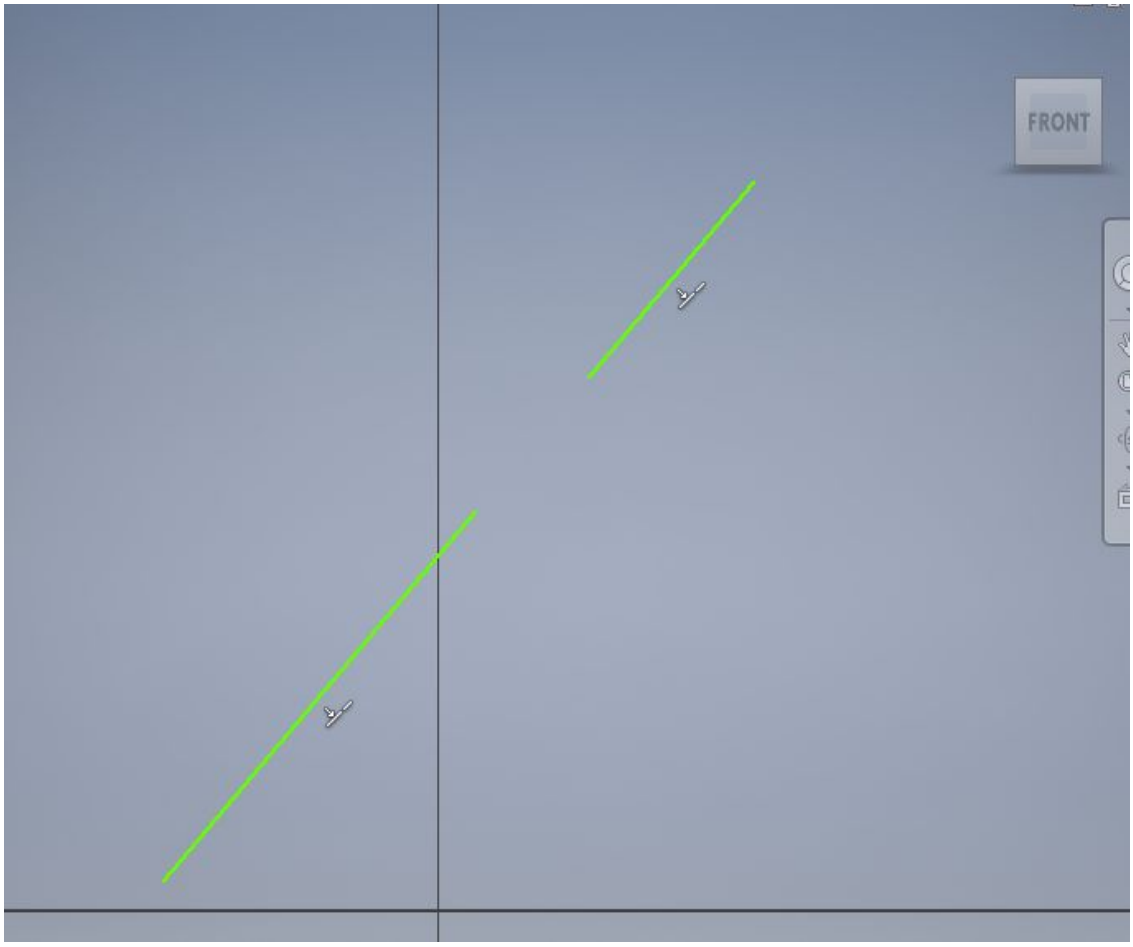
Rectangular Pattern- This tool will allow you to repeat an already created in any one or two directions with any desired number or spacing, separate for each selected direction.



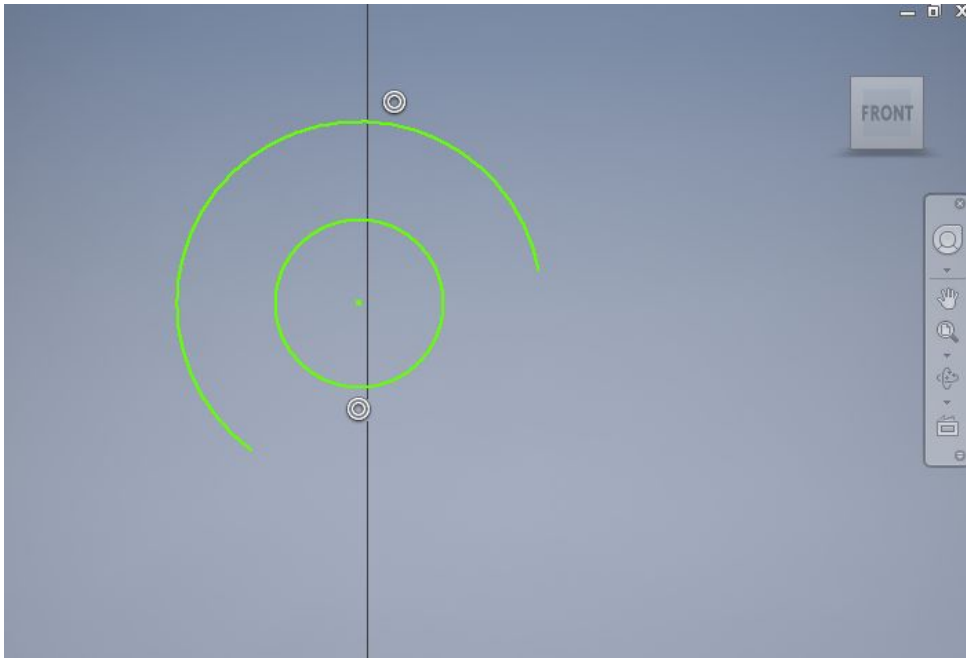
Coincident Constraint- Select 2 points on any sketched geometry and this tool will join them to each other.



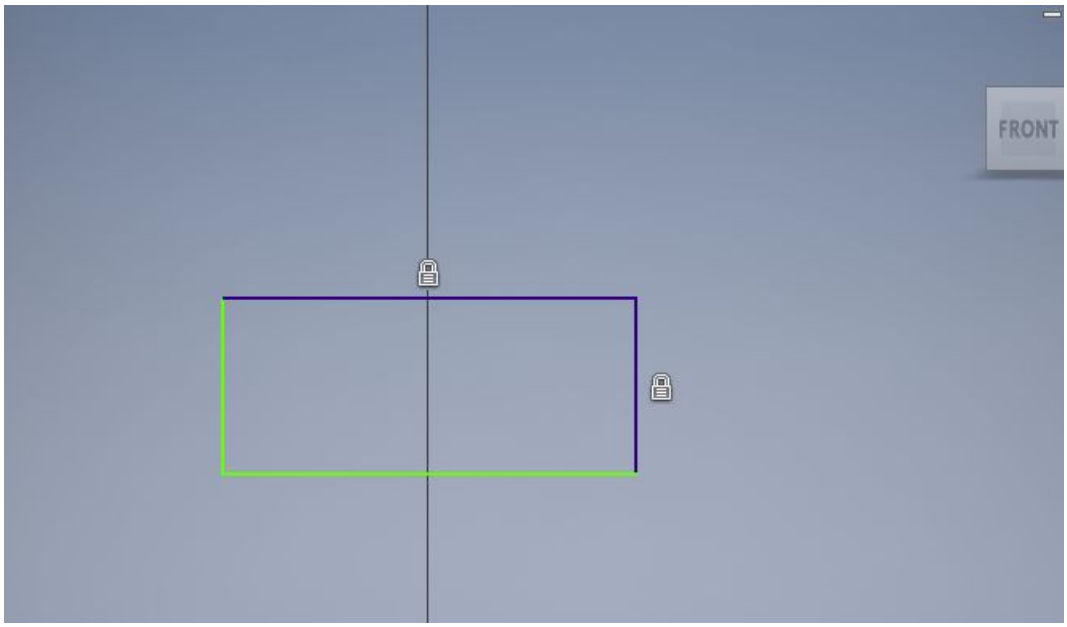
Collinear Constraint- This will constrain two lines to exist in line with each other.



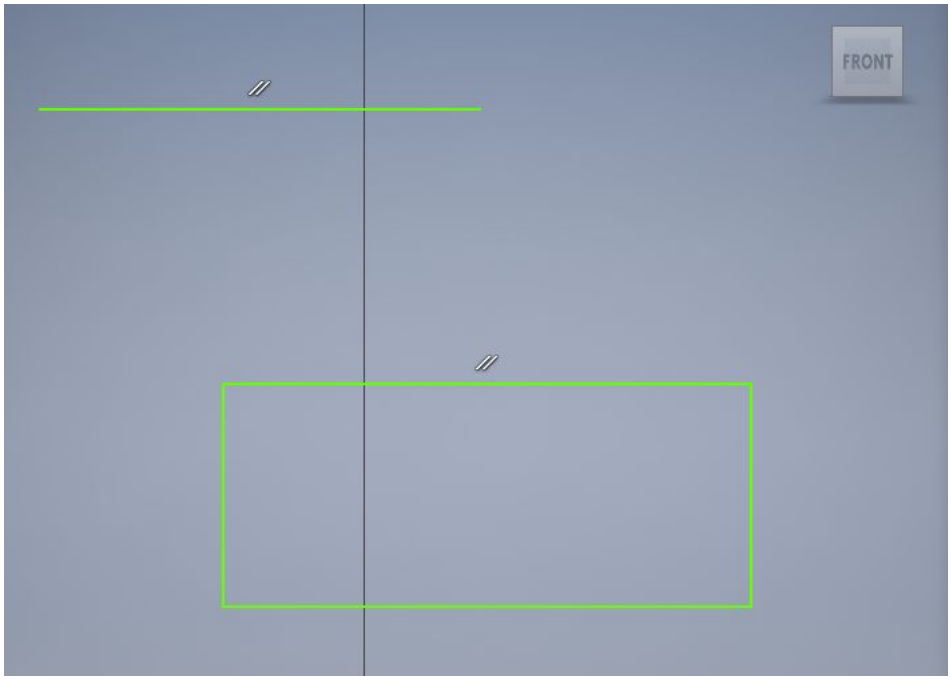
Concentric Constraint- This constraint holds two center points of circles, arcs, points, etc. to each other.



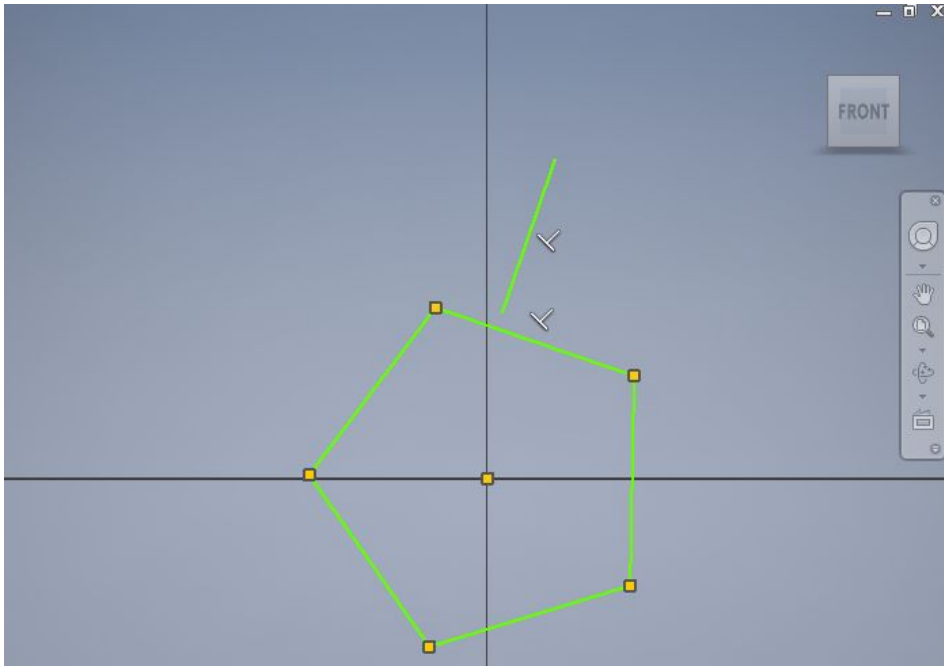
Fix Constraint- This tool holds any selected geometry in place, it will not change side or position no matter what.



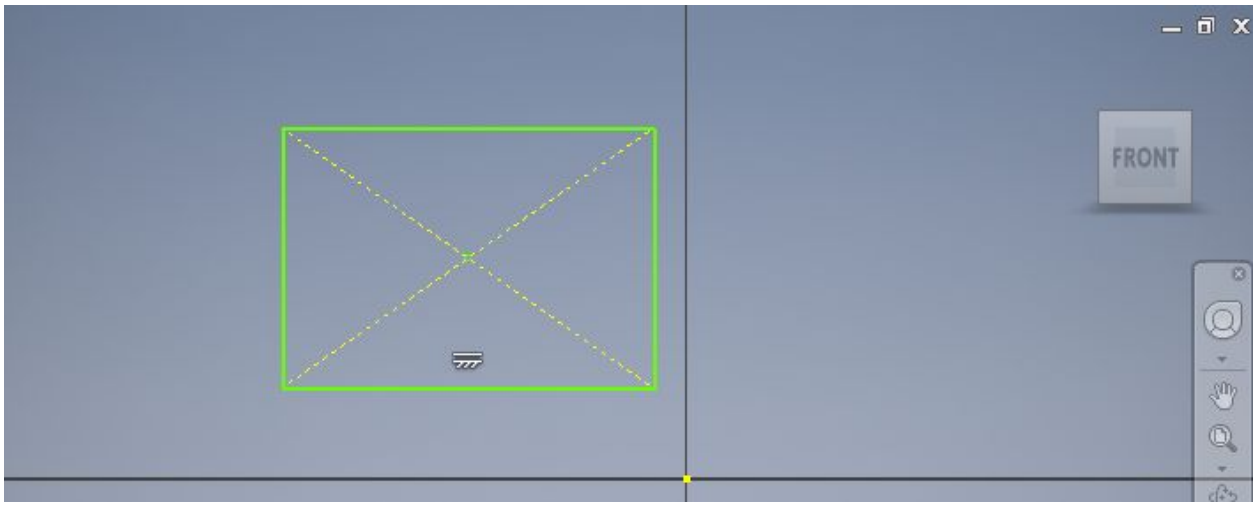
Parallel Constraint- This tool will hold any two selected lines parallel to each other.



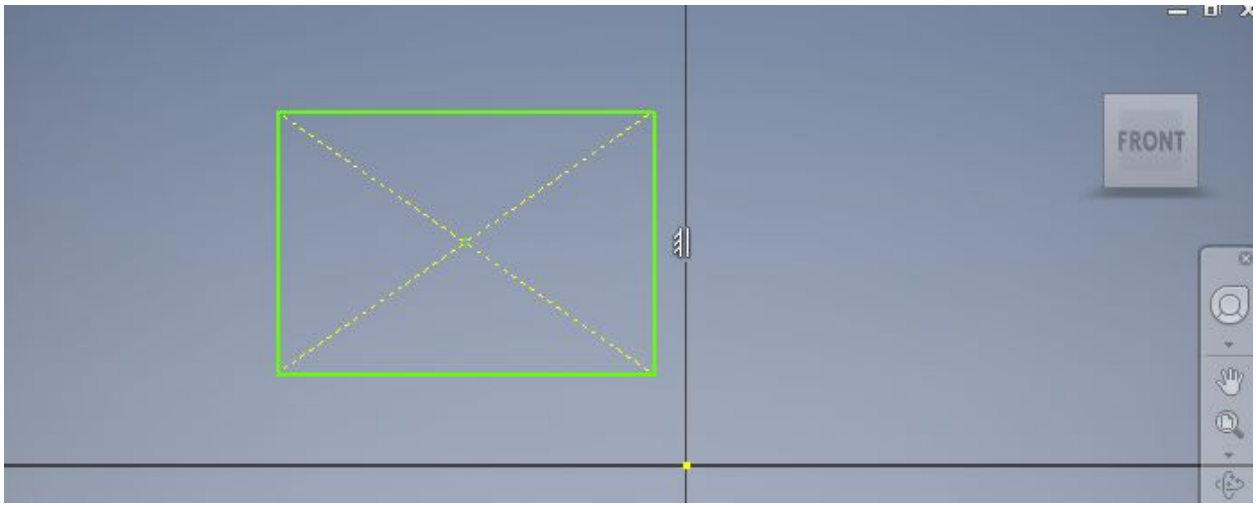
Perpendicular Constraint- This tool will hold any two selected lines parallel to each other.



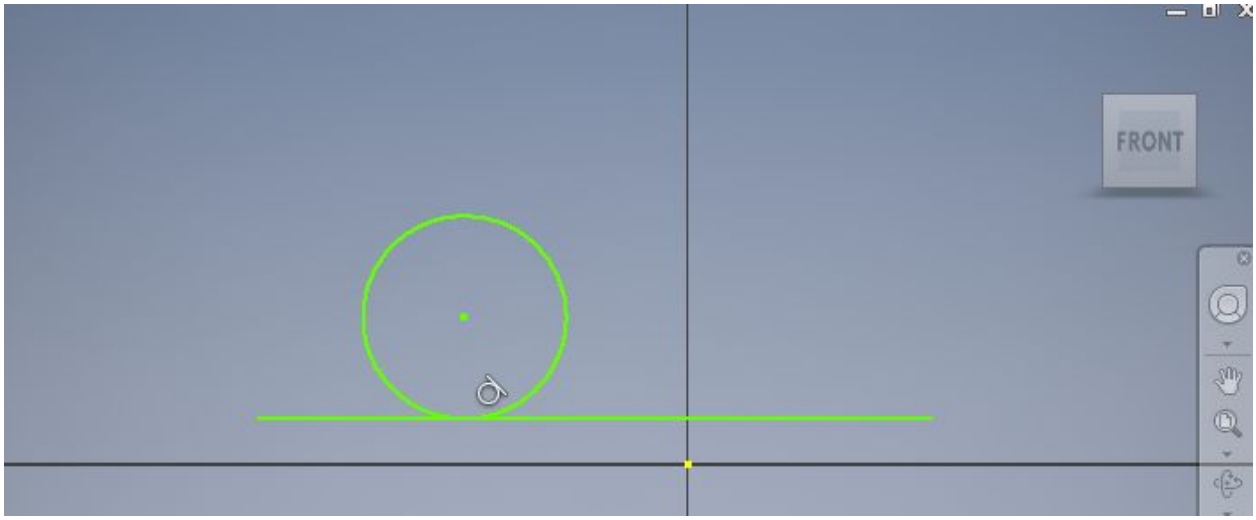
Horizontal Constraint- This constraint holds any line parallel to the horizontal axis of the plane you are working in.



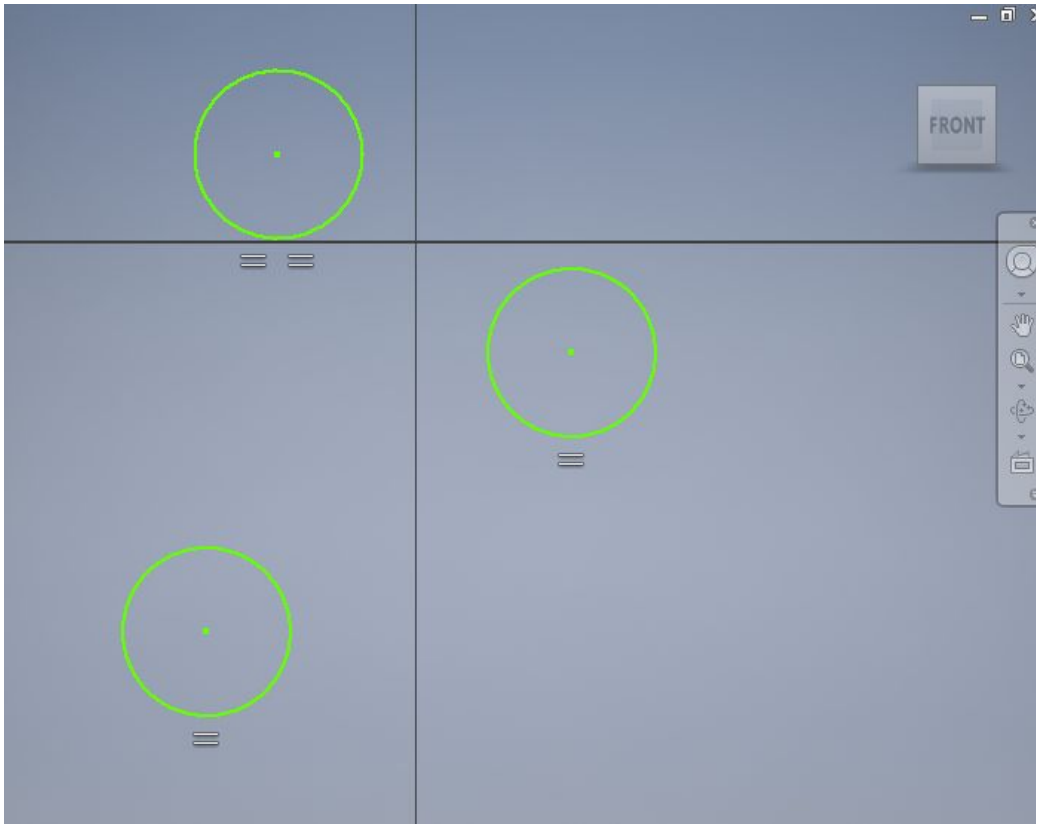
Vertical Constraint- This constraint holds any line parallel to the horizontal axis of the plane you are working in.



Tangent Constraint- This constraint lines up a curves to go into a line smoothly so that the radius of an arc intersects it perpendicularly.

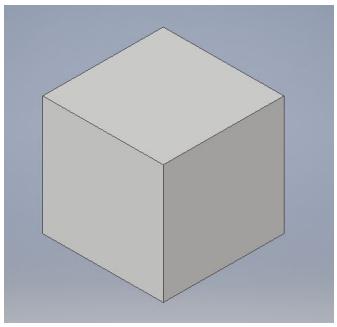


Equal Constraint- This constraint is used to mae values of any two geometries measurements equal to each other so that only one needs to be dimensioned of many.

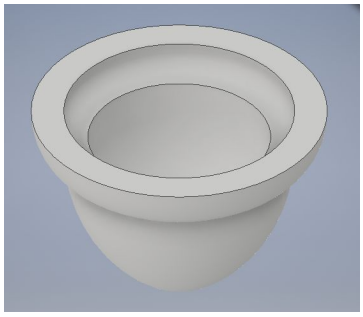


3D Model: Angelo Botha

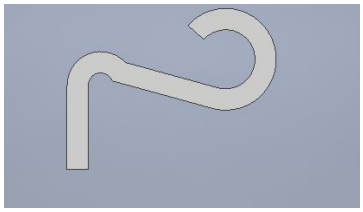
Extrude - Extrusions are used when you want to simply turn your 2D sketch into a 3D model. It extends the plane out making it 3D. To use, select the 2D sketch you want to use. On the top ribbon, underneath the 3D Model tab, select extrude. Specify the distance you want to extrude by, and click “Ok”. The sketch should now be extruded and 3D.



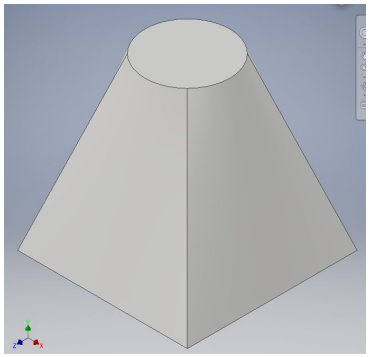
Revolve - Revolve is what is used when you want to make a spherical object/circular object. It takes an axis of your choosing, and it revolves around said axis. To use, make a 2D sketch. Find the revolve tool in the 3D Model tab. Select it, and when the window pops up, click profile and select the sketch. Next, select axis, and select the axis in which you want to revolve around. Alter the angle to your liking and click “Ok”.



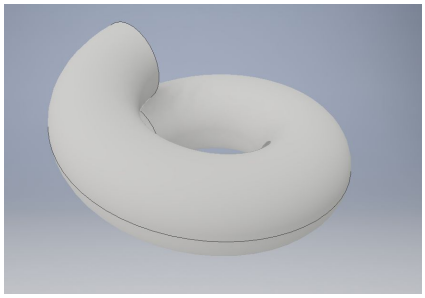
Sweep - Draw a 2D sketch of what you want to sweep. On a separate plane, sketch a line you want it to be swept across. Select the 2D sketch as your profile and the line it will follow as your path. The sketch will now be traced along the line in any shape you drew it as.



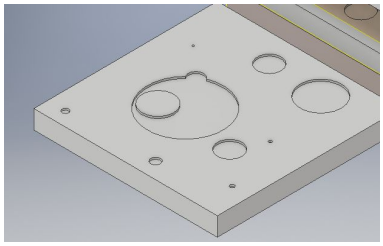
Loft - The loft tool allows for you to be able to take a shape that you have made, and sort of transform it the dimensions of a different shape on another plane. First, you want to make a sketch of the shape you want. For my example, I'm using a 10 x 10in square. You want to set up the shape with the dimensions you need, then finish sketch and don't extrude. Now you will need to offset the plane from your sketch. To do this, you need to go to the Plane drop down in the 3D Model tab. Select "Offset from Plane", and click on the edge of your shape. Enter the distance you want your eventual second shape to be from your primary shape. Next start a 2D sketch on the new plane. Project Geometry of your edges on the first shape, and create your new shape on the new plane. Select the loft tool from the 3D tab, select "Click to add" and select first shape, repeat but now select second shape. Choose "Ok" and your loft should be complete.



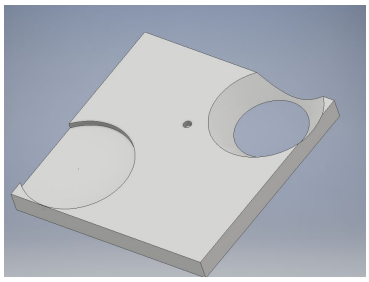
Coil - You can use the coil tool to be able to make a cylinder curve around an axis like a spring. This can be very useful in CAD, as there are many instances where you may need something to coil around. This one is very simple. You want to start a 2D sketch and create a shape of any size. Next to that shape, you need to use the line tool to create an axis next to the shape. Select the coil tool under the 3D Model tab and select your shape, then on the coil GUI, select axis and click on the line you made. You can change the pitch and revolutions, as well as other parameters in the GUI as well. Select "Ok" when you are ready and your shape should be coiled.



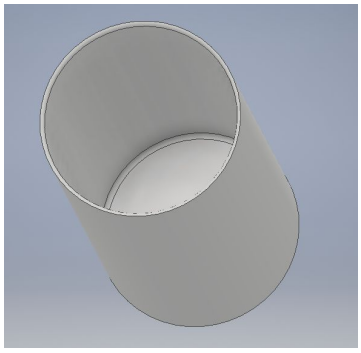
Emboss - The emboss tool is a tool that is used to make either a raised pattern on an object, or an engraved pattern into the object. This is useful in that many products have raised or engraved features in them, whether that be for measurements or something else. To use this feature, you first need to start with an object. I am starting with a 10 x 10in cube. Once you have your object, you would want to create a 2D sketch on the part of the object where you want the embossment. Draw whatever it is you need to emboss, and finish sketch. In the 3D model tab, select the emboss tool and then click on what you need embossed. You can then change the depth/height as well as choose between making it a raised feature or an engraved feature. Select "Ok".



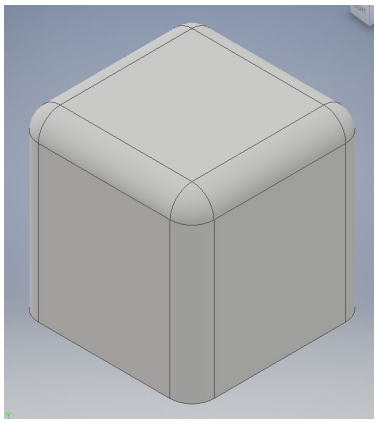
Hole - Hole is a tool that simply creates a hole in your object. You need to have your object, then you will need to create a 2D sketch on top of the face you want the hole. Use the point tool to put a point where you want your hole. Finish the sketch and then find the Hole tool in the 3D Model tab. From here you can change a number of things, from the thread to the counterbore, and all other kinds of options. Select your dimensions and when you're done select "Ok" and your hole should be there.



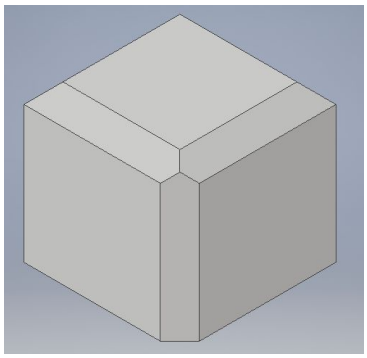
Shell - Shell is a tool used to create just that, shells. It is useful in making things like cups, or other objects that have a shell. First simply create your object. Then select the shell tool from the 3D Model tab, and click on the face you need the shell to be. You can change the thickness of the shell; how thick or thin the lip is. Press "Ok" and it should be done.



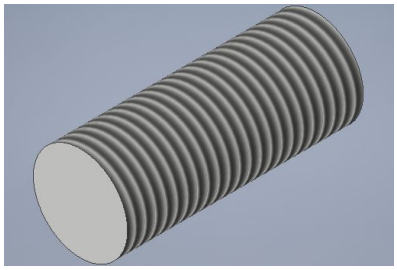
Fillet - You should use the Fillet tool when you want to add a curve to a straight edge on your extrusion. It helps to smooth out the edges on the model. To use Fillet, have your 3D Model out. Find fillet in the 3D Model tab and select it. A window will pop up, and you will need to select an edge with witch you want to smooth out. Enter a value that you are happy with for the radius and select “Ok”.



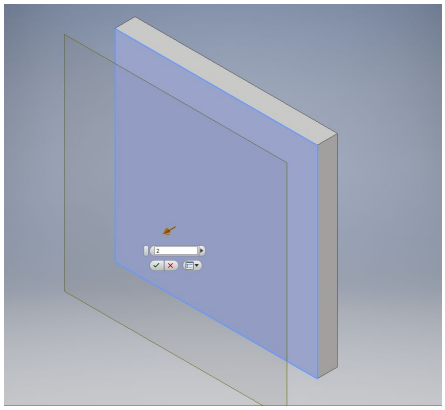
Chamfer - The Chamfer tool is a lot like the Fillet tool, in that it works with the edges of your extrusion and edits them. Fillet applies more of a rounded edge, while Chamfer just chops the edge. To use this tool, have your extrusion out, select Chamfer underneath the 3D Model tab and select the edge(s) that you wish to edit. Alter the distance to how much of a Chamfer you want, and click “Ok” when done.



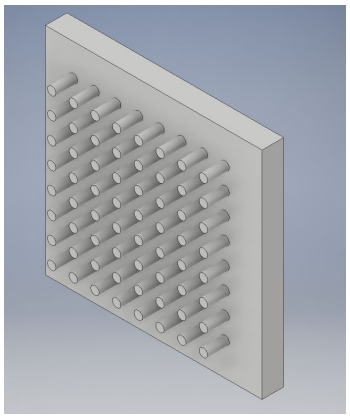
Thread - Thread is a simple tool to use when you want to make a cylinder with a thread on it. It's simple and effective. First, extrude a circle to make a cylinder. Select the thread option in the 3D Model tab, and it should place thread on your object. You can change the length of the thread/offset it. In the specification tab, you can change the thread type, size of the thread, class, and the designation. Once you change what you want to change, select "Ok".



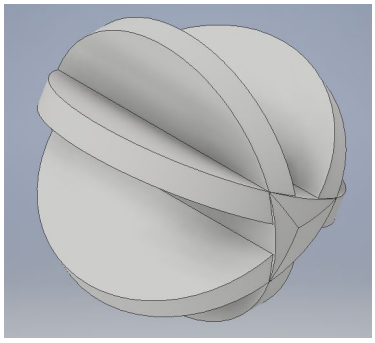
Offset from Plane - The Offset from Plane tool is a tool located in the dropdown menu of the "Plane" tool. You use this tool to be able to make another plane on your object offset from your original one. To use, select the tool, then select the face you want to plane to be offset from. Put in the distance of the offset you want and select "Ok", and you will have another plane offset from your original one.



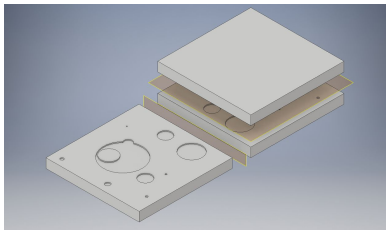
Rectangular Pattern - A rectangular pattern is very useful for many things. It can help whenever you need to repeat accurately and precisely a pattern on a rectangular object. To make one of these, you need to first of all you need a rectangular object. Once you have a 3D rectangular object, you can begin work on your rectangular pattern. First you need to make a 3D sketch of what you want to be patterned on top of the face where you want the pattern to be. Next, head over to the 3D Model tab. Click on the "Rectangular Pattern" under the pattern options. You need to click on the object that you want to be patterned. Click on "Direction 1" and click on one edge of the face you want the pattern on. This will be one direction that the pattern goes in. You can then change the number of recursions in that direction. For the second direction, click "Direction 2" click on a seperate edge of the face. You can also change the number of recursions you want in that direction. Click ok and your pattern should process.



Circular Pattern - A circular pattern is similar to a rectangular pattern, and it makes a pattern around an axis in which all objects are of equal distance to said objects. To do this, you need to have your object. Once you have your object, find the circular pattern option. Click on the feature that you want, (your object) and then click on the axis option. On the left side, select the axis you want the pattern to be on. Select the amount of placements you want, and hit “Ok”.

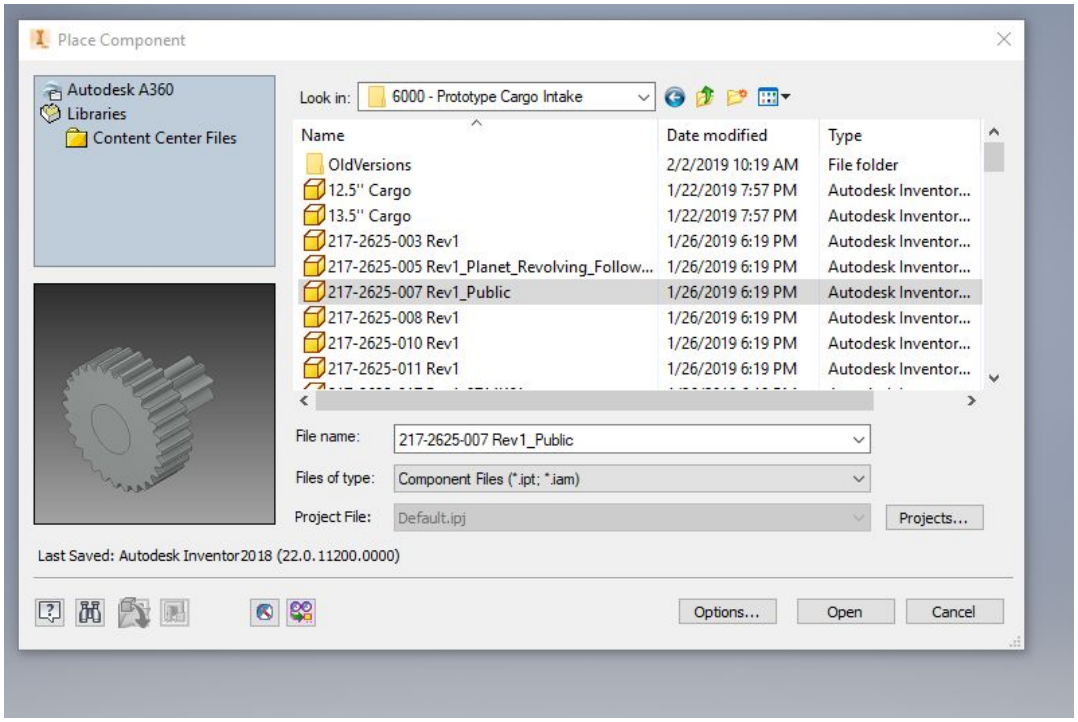


Mirror - Mirror is a useful tool when you have a project that you need something to be mirrored on one side to the other. It is useful in making builds symmetrical and and even. To start, you need to have what you are going to mirror. Once you have that, you need to use the Offset from Plane(tutorial above) to create a plane which is the plane where the object will be mirrored. Click on the mirror tool in the 3D Model tab. Select “Feature” and click on the object. Select “Ok” and the object should be mirrored.



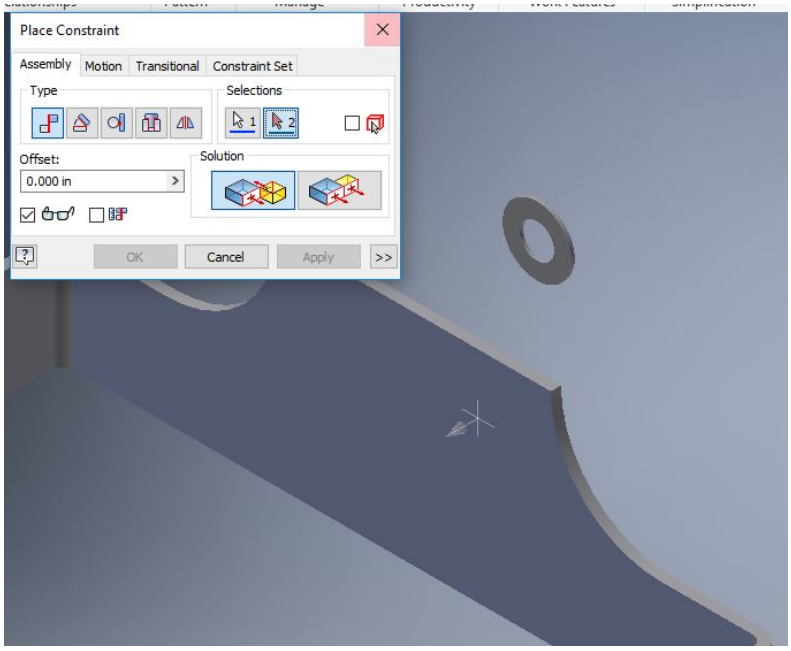
Assemblies: Vincent Phan

Place - When you click the place tool, a pop up menu opens and allows you to insert parts or even another assembly into the assembly. Whenever you click on a part there's also a preview screen on the bottom left of the screen to confirm the part before opening it. To place the part, simply click on the screen and if it is the first part of the assembly, the view will change to fit the object. Press escape to unselect the place tool or press "P" to pull up the pop up screen again.

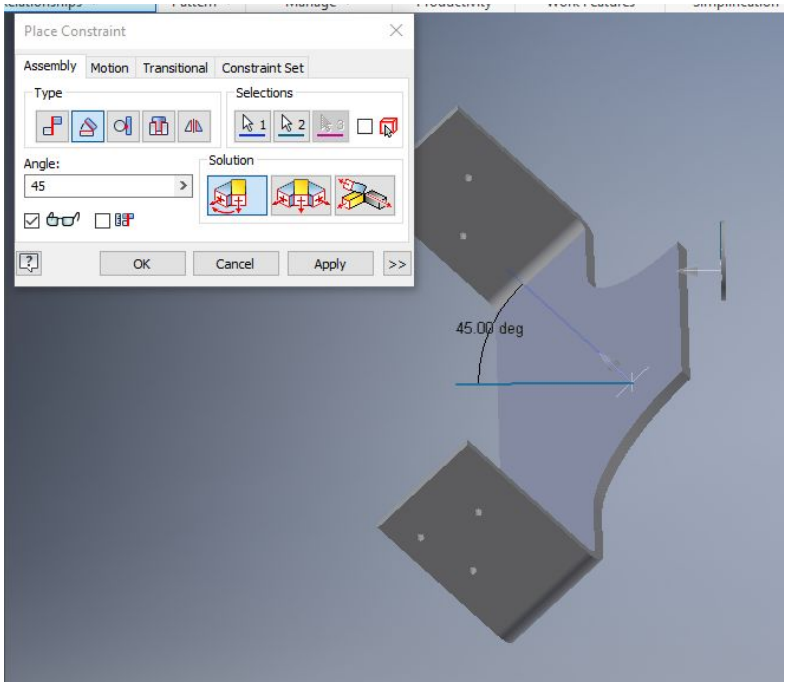


Constrain - The constraint tool as 5 sub sections under the assembly tab that can be used to attach certain parts in different ways. The motion, transitional, and constraint set tabs are not often used to constrain things together.

-**Mate** is the most commonly used constrain tool which connects planes, lines, and/or centerpoints of curved faces. Click on the 2 planes or lines that need to be constrained and adjust the offset if needed. The two options for solutions goes as follows: “Mate” puts the two selections facing each other and connected, “Flush”puts the two selections next to each other and facing the same direction. Mate can also constrain the centerline of a curved face to other planes and lines.

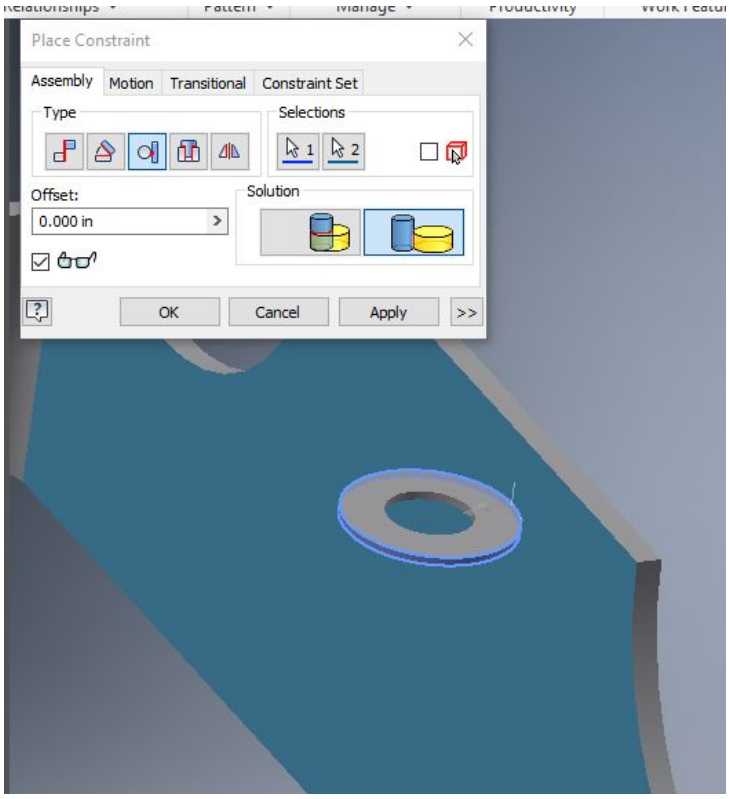


-**Angle** constraints aren’t often used but can constrain 2-3 planes or lines at an angle to each other. Select the 2 or 3 selections and then specify the angle between them, adjusting the solution options to what is needed.

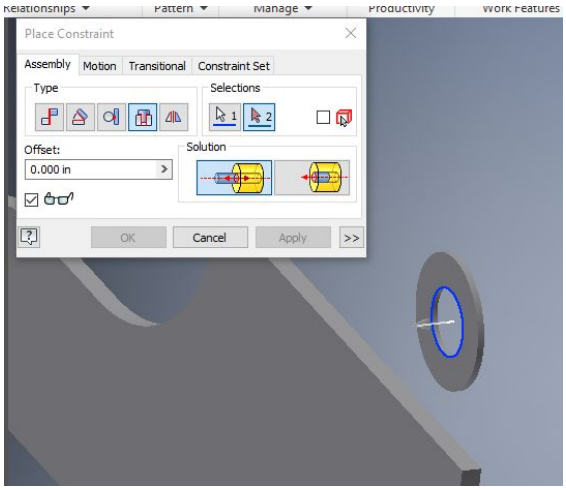


-**Tangent** constraints connect a rounded face to either another rounded face or a plane.

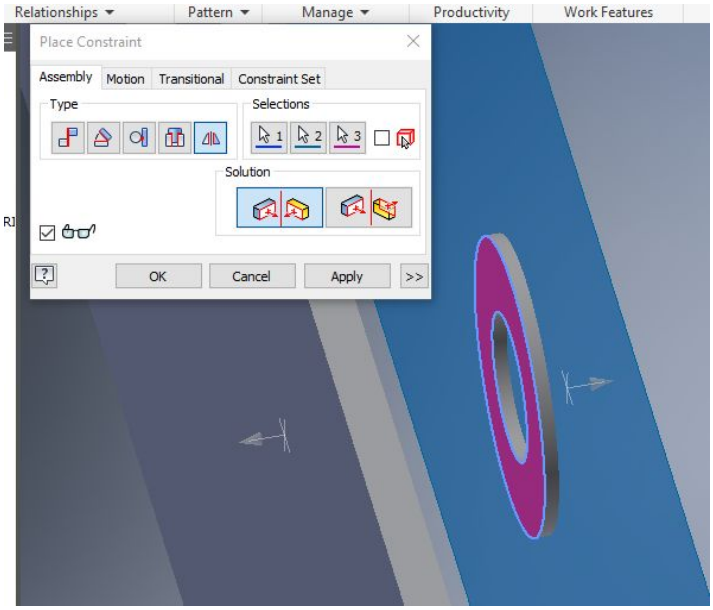
The functions operate similarly to the mate constraint but instead connect the faces to each other.



-**Insert** constraints mainly constrain 2 holes or cylinders to each other. Select 2 (usually) circular selections and make sure to check which direction the arrow is facing to make sure the correct areas are being constrained.

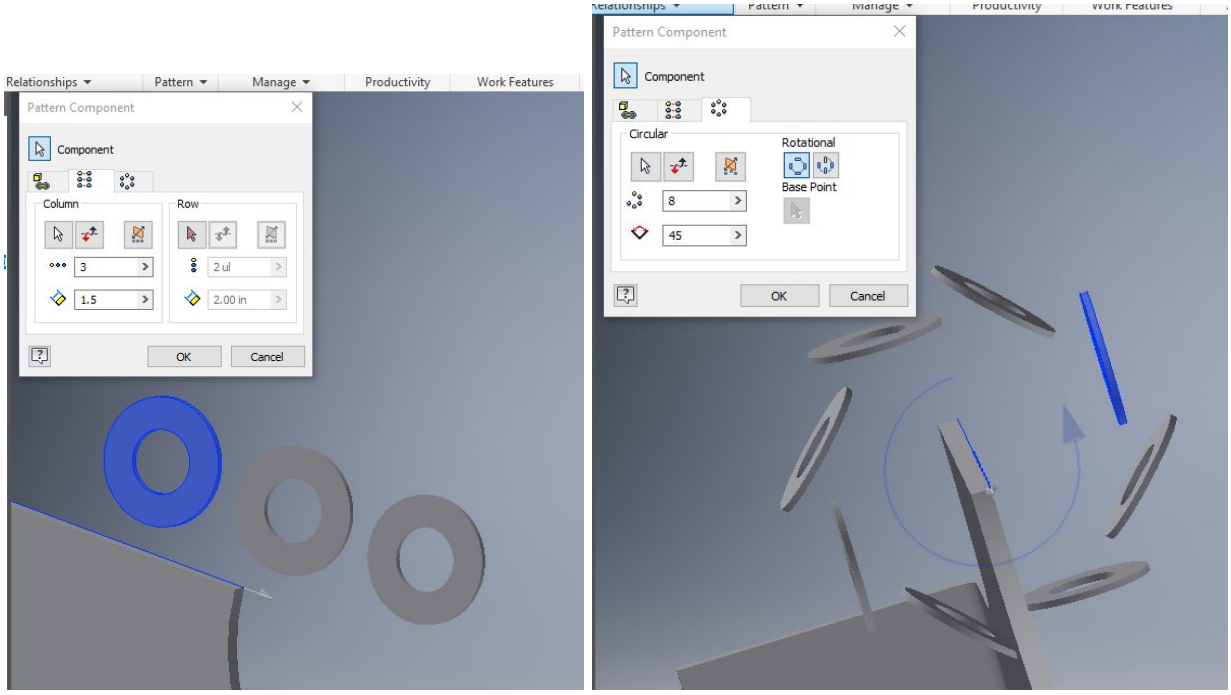
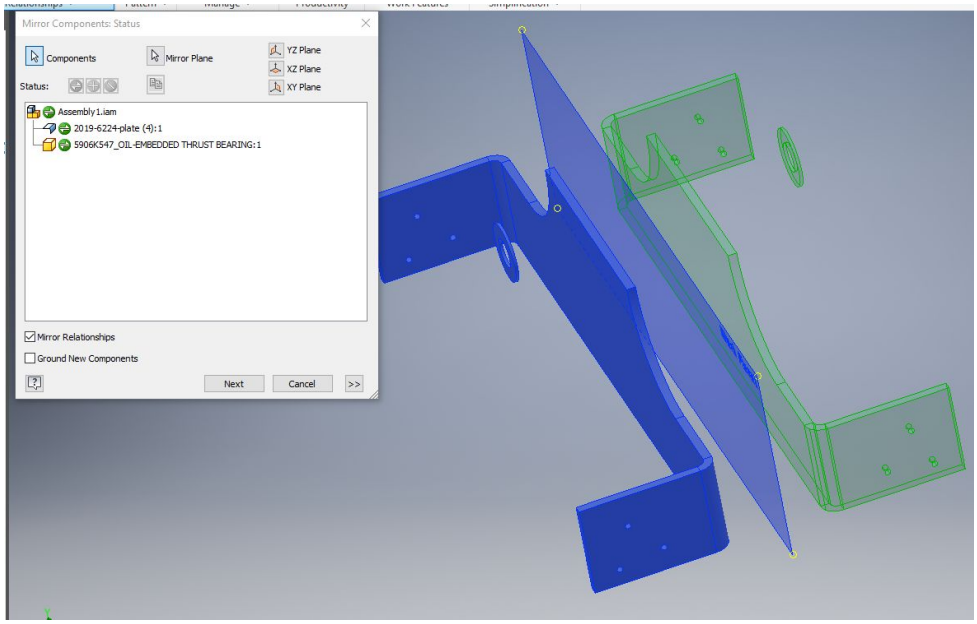


-**Symmetry** constraints can be used to center a part or assembly between to other parts. The first two selections are the 2 planes that the third selection will be centered in between.



Pattern - Pattern constraints work very similarly to patterns in sketches and 3D models. You select the type of pattern, rectangular or circular, and then you select a component and the

direction the pattern goes. The space next to the dots is the number of repetitions and the space below that is the distance or angle between the repetitions.



Mirror - Mirror works similarly to mirrors in sketches and 3D models. You select the component(s) as well as the mirror plane and then there will be a mirrored part across the plane.