Autodesk Inventor Tutorials

by Sean Dotson
www.sdotson.com
sean@sdotson.com

Introduction to 3D Sketches

Latest Revision: 11/11/02

Revised for R4, 5, 5.3, and 6

This tutorial was originally written for R5 but many of the same principles apply to R4, 5.3 and R6. Differences in versions will be noted where applicable. (Note that all screen shots are done in R5. R6 dialogue boxes may slightly differ)

Another popular request in the Inventor NG is how to construct 3D sweeps. This is useful in tubing, piping and wiring applications. If you are going to get very detailed in piping and wire harness design there are add on packages to Inventor that will help simplify this process. But if you just need to route a few hoses or wires, 3D sweeps can help you here. Furthermore you can makes these sweep adaptive so that they will change with your base geometry.

There are two ways to make 3D sketches. Adaptive and (with the introduction of grounded workpoints in R6) non-Adaptive. We will focus on the adaptive approach. At the end of this tutorial we will discuss the new, grounded workpoint feature in R6.

We start our example with a simple assembly. While 3D sweep can be constructed in a standalone part environment, to have adaptability you need to work in the assembly environment. Our assembly consists of two parts. One bracket with a hose attachment on each end (the two extruded circles) and a plate in the middle with a hole cut through. The goal of this tutorial is to show the user how to connect the two hose attachment together with a 3D sweep that passes through the hole, no matter to where the hole (or hose attachments) are moved. (see Figure 1)

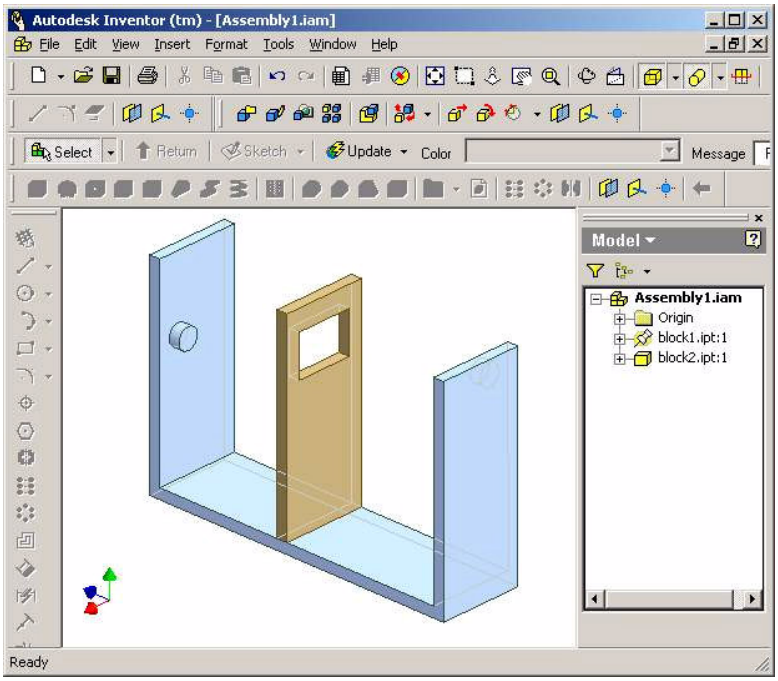

Figure 1 – Assembly

At this point we want to create the part that will be our tubing. Select the "Create Component" button from the assembly menu. Name the file, select it's location and select the template file you want to use. (see Figure 2). Be sure the "Constrain sketch

plane to selected plane or face" checkbox is checked. This assures that the part will be constrained to the first face chosen. Hit OK when done. (Note: when you exit this dialogue the parts will likely appear opaque. In this example the opacity was turned off so that the images would appear more clearly)

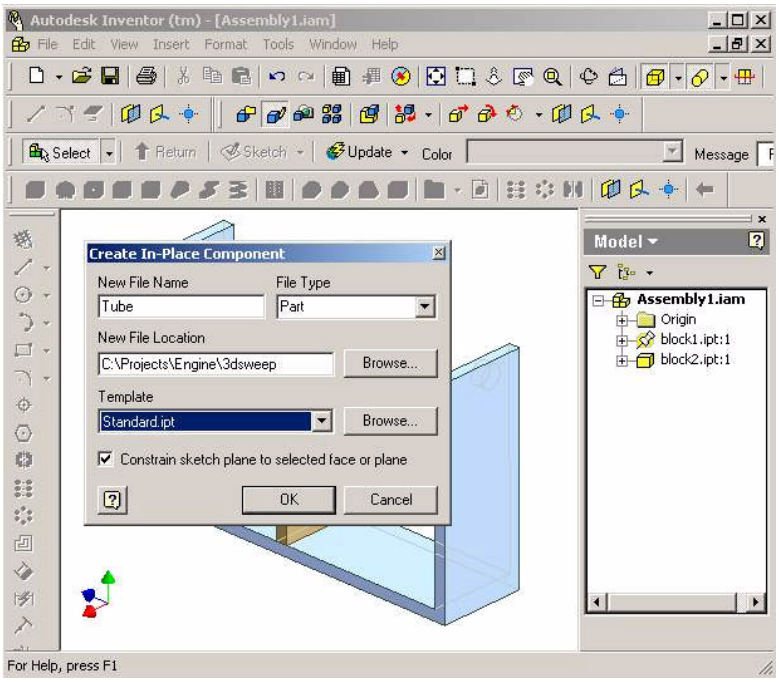

Figure 2 – Create Component Dialogue

You must now choose the face to being the sketch. Choose one of the circular extrusions. This will begin a new 2D sketch. In our example the diameter of the hose is the same as the diameter of the connections so we simply project the geometry of the circle. If the diameters are not the same, simply sketch whatever geometry you wish on this face to be used as a sweep profile. (see Figure 3)

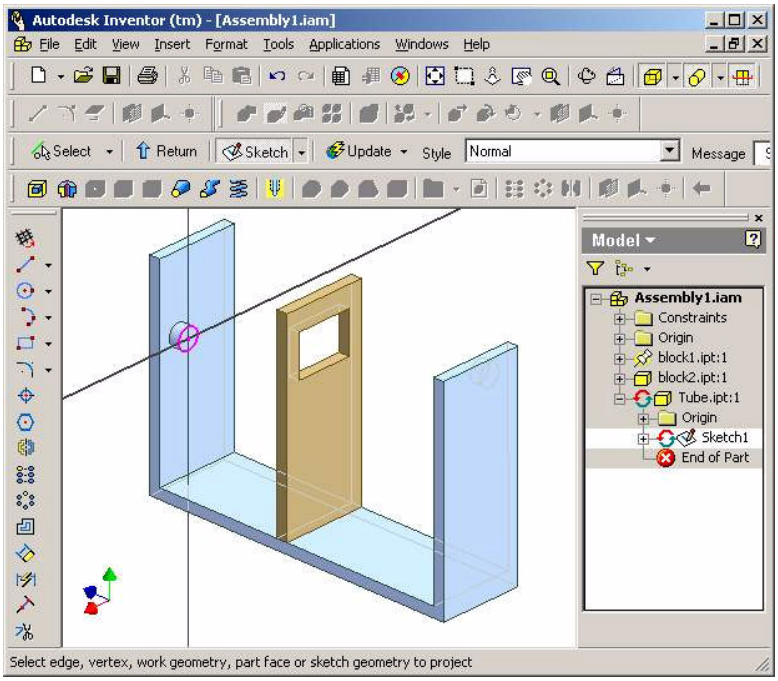

Figure 3 - 2D Sketch on Hose Connection Face

Exit the 2D sketch. We are now ready to begin the 3D sketch. On the Sketch button pull down select 3D sketch.

3D sketches are nothing more than a line in space with 3 degrees of freedom (ignoring any rotational degrees of freedom). They can be controlled by dimensions but these dimensions then "hard-code" the shape of the sweep. This can be useful in some cases (e.g. when you have a bent tube of which you know the geometry and have to design around) but the more useful and robust technique is to use work geometry to control the sweep.

A 3D sweep line can be constructed by following a series of work points. These work points must be defined in space by a plane and an axis, two intersecting axes or three intersecting planes. By constructing a series of workplanes and axes based off existing geometry the user can then place workpoints onto these planes and axes to form a "connect-the-dots" framework for the sketch.

In our example we begin with an axis through the first hose connection. We place a workpoint on the face of the connection and through the workpoint. We then place a work plane a distance off the face of the connection and place another workpoint on this plane and the hose connection axis. This provides us a small "straight" section before leading into the connection. Failure to do this can result in the face of the hose not being planar with the hose connection face when base geometry is changed.

Additional workplane and axes are placed off the hole in the middle of the plate to constrain the work points to flow through the middle of the cutout. Again we provide

two planes set off both sides of the middle plate to make sure the hose is straight as it goes through the middle plate.

We continue this logic until we reach the other hose connection where we place another offset worklane and point (for the short straight section) and a workpoint on the face of the connection and an axis running through the middle. (see Figure 4). (Note that the background color was changed to provide contrast of the workpoints)

Now choose the line tool on the 3D sketch toolbar. Creating the sweep path is as simple as connecting the workpoints. Click on the first workpoint, then the next, then the next etc. until complete. Inventor will automatically provide radii in the intersections of non planar sections of line. These can be changed (as shown in the example) to whatever is required. Note that the first radius is a controlling dimension in that all of the following radii are parametrically referenced to this dimension. Changing the first dimension will change them all. If the second or third dimension requires editing, change that particular dimensions. (see Figure 5)

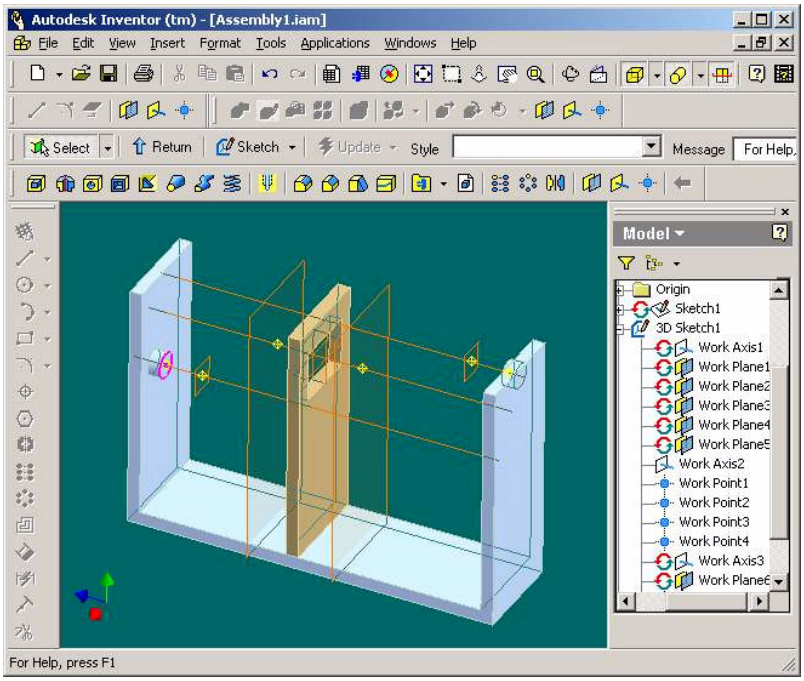

Figure 4 – Workpoints, Planes and Axes

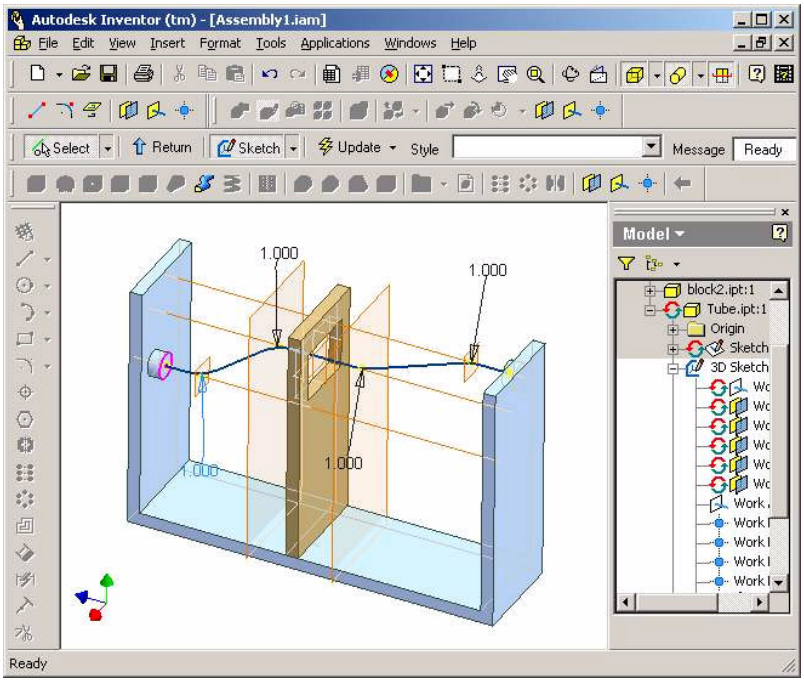

Figure 5 – 3D Sketch completed

The next step in constructing the tube is the actual sweep. While still in the edit part mode choose the sweep command. In this command you will be asked to choose the profile to be swept (the 2D sketch) and the path to follow (the 3D sketch). (see Figure 6)

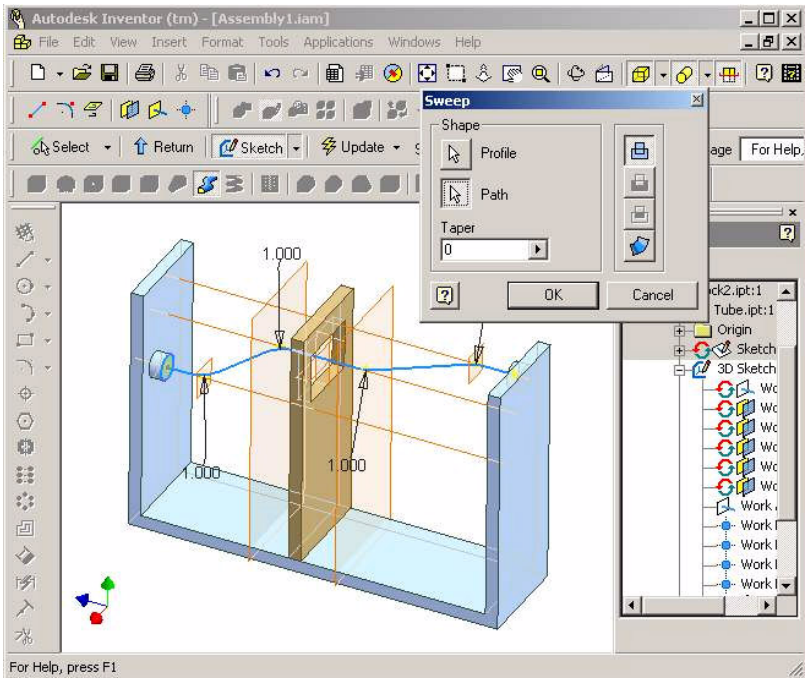

Figure 6 - 3D Sweep Dialogue

Once OK is chosen the sweep will be generated. The tube connects the two hose connections together while passing through the center of the cutout. (see Figure 7)

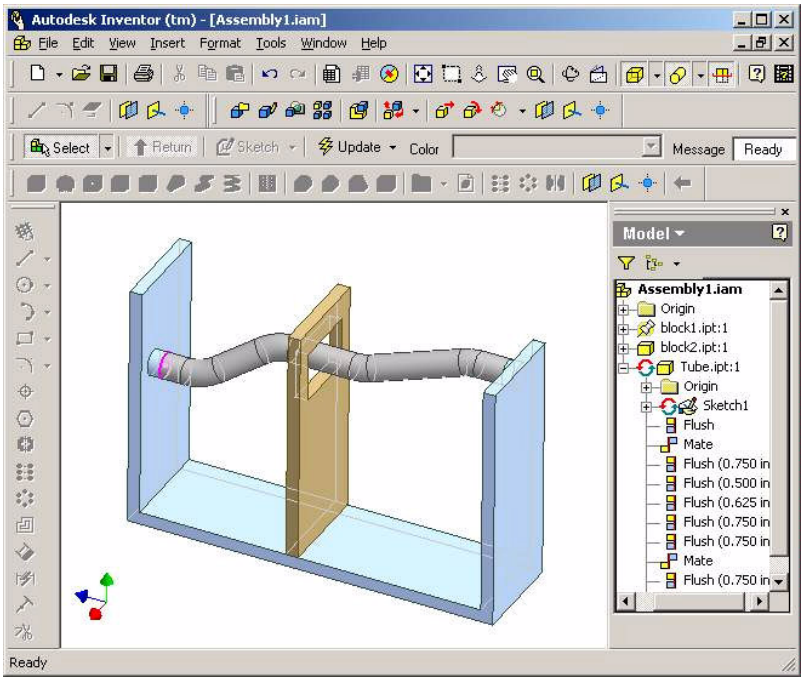

Figure 7 - Completed Sweep

To test the adaptivity of the tube, open the middle block part model and edit the position of the cutout. Save then return to the assembly and update. The hose should have followed the position of the cutout. (see Figure 8) Now open the blue channel and move the positions of the hose connections. Save the file and update the assembly. Again the hose should follow the position of the hose connections. (see Figure 9)

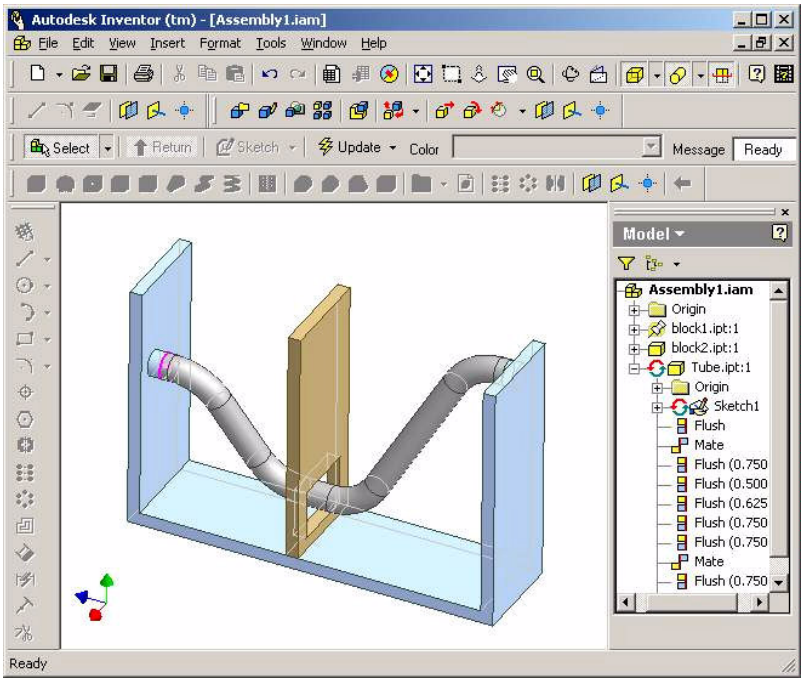

Figure 8 - Moving the Cutout Changes Hose Geometry

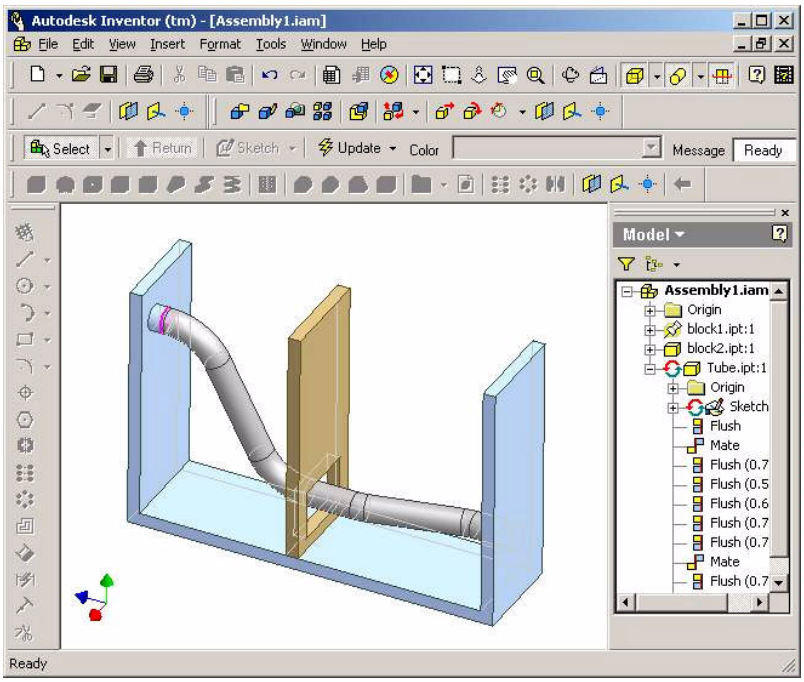

Figure 9 - Moving the Hose Connections Changes Hose Geometry

It is usually a good idea to only change one part at a time and the update the assembly. Changing multiple parts at once can cause some instability in the 3D sweep.

Another limitation of 3D sweeps is the fact that some forethought must be put into how the planes and points are constructed. A change to geometry that will force a radius to go beyond zero or infinity will results in errors in the tubing. This generally happen when

the radii and the path are too small for the profile to be swept without the profile "folding in" on it's self. Changing the radii generally clears up the error.

Another good resource for 3D sketch techniques is at Drew's Inventor Tips & Tricks page. If can be found at **http://www.mymcad.com**. Click on Inventor at the top of the page then Inventor Tips and Tricks. The tip is #1012. A direct link to the page is **http://www.mymcad.com/iv_tipstricks_answer1012.html** but I would suggest looking at the main page as there is a wealth of information there.

With the introduction of grounded workpoints in R6 you can make 3D sweeps much quicker but with the loss of adaptivity. Grounded workpoints are points that are specified using X,Y and Z distances from other points or features. Since you do not have the overhead of adaptive workplanes and axes the creation of these sweeps is much faster but with this speed you loose the adaptability of the sweep. This is useful for creating reference geometry or parts that you know cannot change in shape (e.g. and existing steam line in a plant).

To place a grounded workpoint enter a new 3D Sketch and choose the grounded workpoint icon (point with a pushpin)

Choose a point on a part or a feature (or an intersection of an axis and a pane, or any other combination of choices that define a point in space. You will be presented with an offset dialogue box. The first point should have a delta of 0,0,0 (as this will be the beginning of your path). Before clicking OK, go to the More tab and check the Repeat checkbox (this will allow you to remain in the command). Go back to the first tab and click OK. (see Figure 10). Now using the triad you can drag the next point to the proper position (similar to presentations). You can also enter the offsets by the X, Y and Z boxes in the dialogue. You can continue to place points until the path is complete.

Once your points are placed the process to create the sweep is the same as the methods described above. Simply connect the points with the line tool and sweep the profile along the path.

You can edit the position of the points by right clicking on each point and selecting **3D Move/Rotate.** Furthermore you can have a mixture of grounded and non-grounded workpoints in your sweep allowing you to have a semi-adaptive sweep.

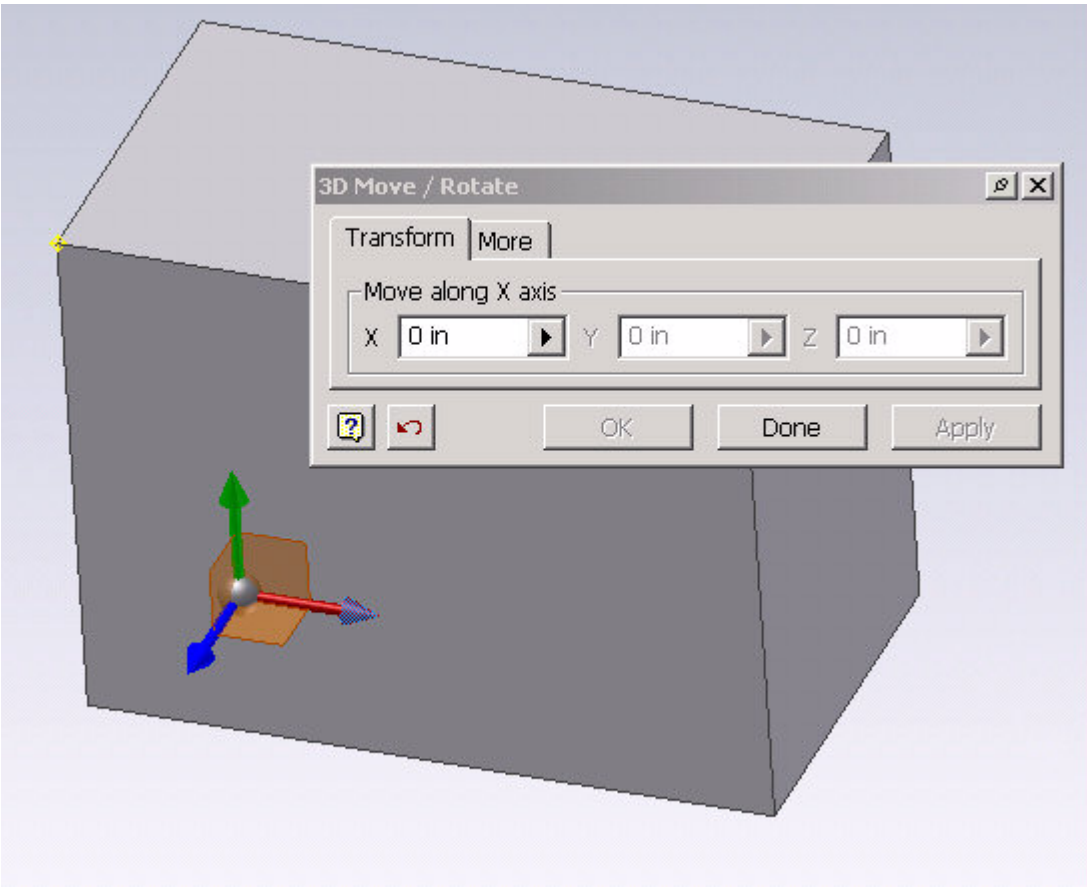

Figure 10 - Grounded 3D Workpoint

While this tutorial is a basic lesson in 3D sweeps the author hopes that you will take the opportunity to experiment with these techniques. If you find a better or faster way to do any of these steps feel free to send me some comments.