

Using Work Features in Autodesk Inventor® 2008

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Learning how to properly use work features in 3D modeling is probably the most important area to master. Unless you can properly and accurately place work features anywhere in a part or assembly environment, you simply cannot expect to be able to create complex geometry. This article should help you learn how to place work features anywhere in your design.

Work features are defined as workplanes, work axes, and work points. Learning how to properly create, place, and edit work features is tantamount to good design. Properly created, these work features will remain parametric within your model or assembly, and allow you to create complex geometry with ease.

Workplanes

A workplane is a planar feature within most modeling systems. The workplane is useful for many applications within modeling. A workplane may be used for constraining parts or assemblies together, creating a new sketch on a non-planar face, used as projected geometry in sketches, used for measurements, or many other purposes. The workplane is the primary work feature needed for creating extremely complex geometry. For reference purposes, Inventor creates three origin workplanes in every Inventor 3D file.

In Inventor 2008, workplanes have a different color on each side to the workplane. The yellow side is designated as the "Normal" direction, or the side on which the sketch will be created. To flip the normal, right mouse click on the workplane and select **Flip Normal**. Figures 1 and 2 illustrate the color change when the normal is flipped.

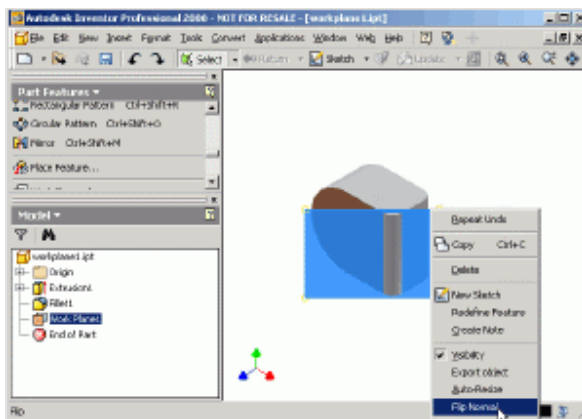


Figure 1: RMB – Flip Normal

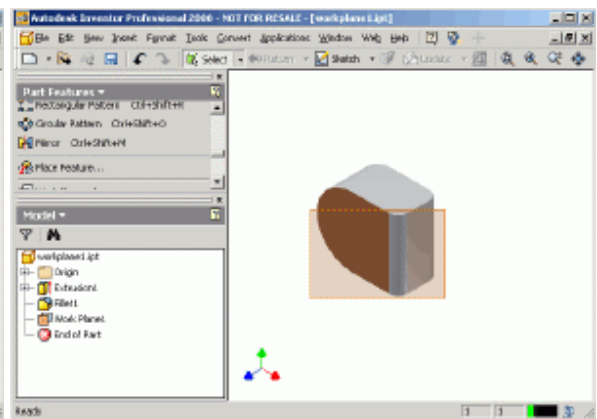


Figure 2: Normal (Yellow) side.

Let's begin by taking a look at creating a few simple parametric workplanes on various parts.

Create workplanes by vertices:

In our first example, we will utilize a simple model, and learn how to create workplanes through various points or vertices. You may create these models yourself, or download model files usable in versions 11 through 2008. **Download Models11-2008.zip**.

A vertex can be described as an intersection between two edges, a midpoint of a single edge, a control point on a spline, or a placed work point.

1. **Open** the file - **workplane1.ipt**
2. Select the **Workplane** command.
3. Select 3 vertices as shown in Figure 3. Yellow circular glyphs will appear when you are at a vertex.

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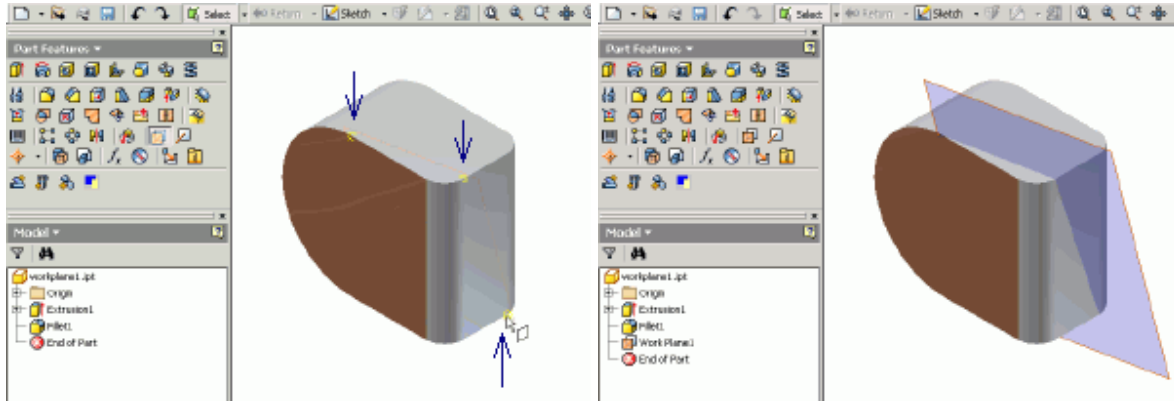


Figure 3: Select 3 vertices

Figure 4: Workplane is created.

4. You will see a preview of the workplane. When the 3rd vertex is selected, the workplane is created. (Figure 4).

5. Notice that the created workplane consists of a "blue" and a "yellow" side. The yellow side is called the normal side. If desired, you may flip the normal by selecting the surface and **Right Mouse Click- Flip Normal** command.

Undo the workplane and proceed to the next example.

Create workplane by plane and vertex

Using the same part, we will create a workplane through the middle of the part, parallel to the copper face. When a plane and a point are selected, the created plane will always be parallel to the original face or plane.

1. Select the **Workplane** command.
2. Select the copper colored face, then the midpoint of the top front edge as shown below (Figures 5, 6)
3. You will notice that the normal side (yellow) of the workplane is facing toward the original selected face.

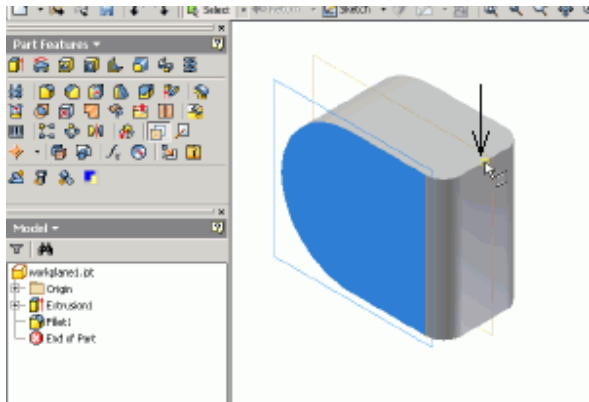


Figure 5. Select Face and vertex.

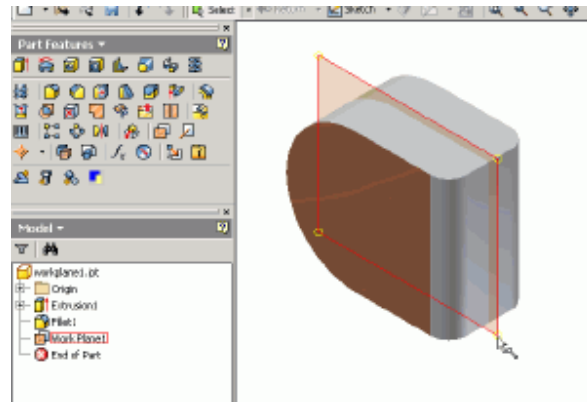


Figure 6. Workplane created (Normal Side Visible)

Undo the workplane and proceed to the next example.

Create workplane mid plane between parallel faces

Using the same part, we will create a workplane through two parallel faces. This will create a workplane identical to the method above.

1. Select the copper colored face, then select the face on the opposite side of the part. A workplane will be created midway between the two faces. If the thickness of the part is adjusted, then the workplane will move to remain midway between the two faces. Be sure to pick **Update** to update the workplane position after any change. **Undo** the workplane and proceed to the next example.

Create workplane by two edges

Using the same part, we will create a workplane through two parallel edges. The edges that we will select are the tangent edges of the forward facing vertical radius.

1. Select the two vertical tangent edges. The workplane is now created, and should be at an angle of 45° to the front face (Figures 7, 8).
2. Do not select Undo, because we will use this workplane in the next step.

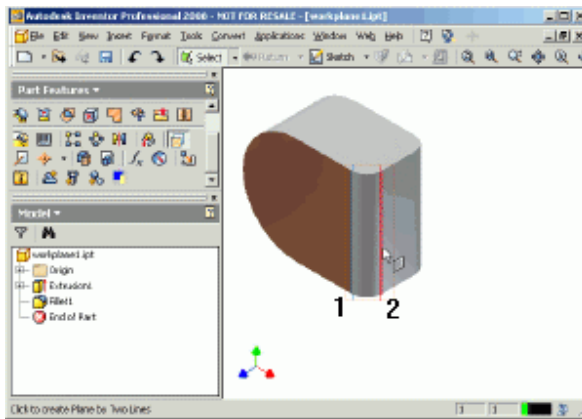


Figure 7: Select 3 vertices

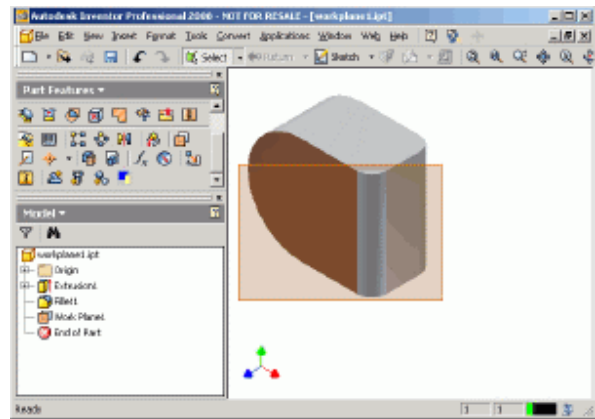


Figure 8: Workplane is created.

Create offset workplane tangent to a face

Using the workplane created in the previous step, you will now create a workplane parallel to the created plane or face, and tangent to the front face of the radius. In addition, you can select an origin plane, then any curved face including the sphere, to create a workplane tangent to the object. As always, pick **Update** to update the workplane position after any change in the model.

1. Select the previous created workplane, then pick on the front curved face of the radius. An offset tangent workplane is now created. (Figures 9, 10)

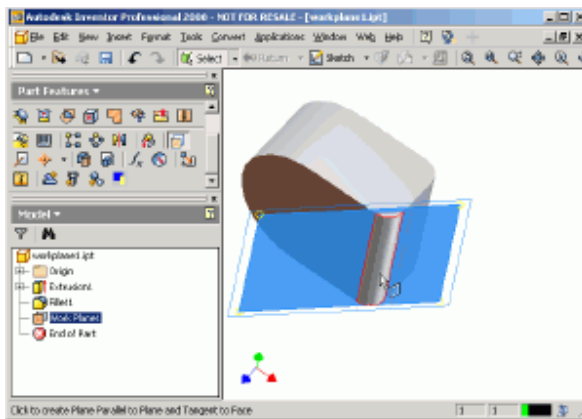


Figure 9: Select existing workplane, then face

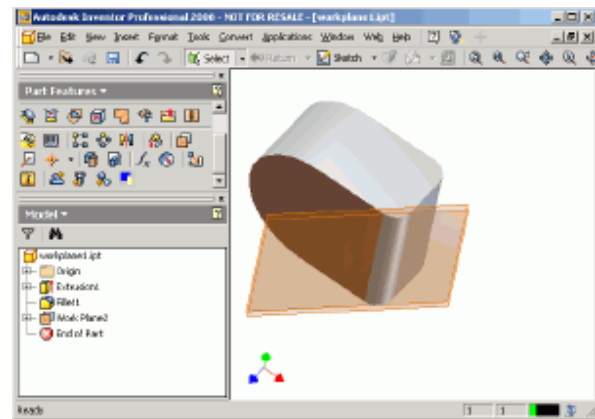


Figure 10: Offset workplane is created.

Undo the workplane and proceed to the next example.

Create angled workplane

An angled workplane may be created by selecting a line, axis or edge, then selecting a parallel workplane or face. A dimension box will appear so that you can enter the desired angle as either a plus or minus number. A preview of the workplane to be created will appear so that you can easily judge the correct positioning.

1. Using the workplane command, pick a line or an edge, then pick a planar face or existing workplane that will create a parallel plane.
2. Input the angle in the dimension box (figure 11) that appears. Figure 12 shows the finished workplane.

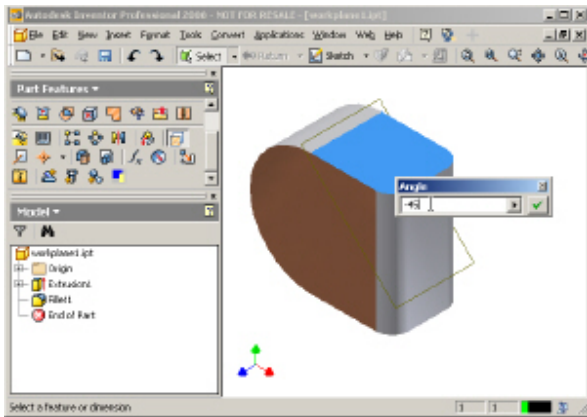


Figure 11: Select Plane, then edge - angle

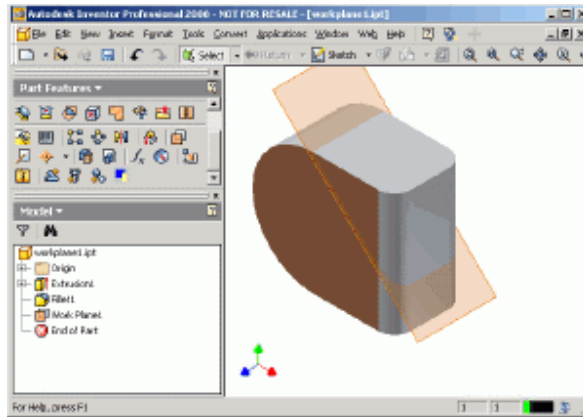


Figure 12: Angled workplane is created.

Create compound angled workplane

You may create a compound angled workplane using the angle of workplane technique shown above, creating a second workplane offset to the edge of the part, then creating a sketch on that workplane. The sketch is then used to create a third workplane at a compound angle.

1. Create an angled workplane. Then create a second angled workplane by selecting the first angled workplane and the top front edge of the part. Set the angle of this workplane to 0.
2. Use the second angled workplane to create a new sketch. Sketch a line angled from one of the origin axis (Figure 13).
3. Use the angled workplane and the sketch line from step 2 to create a third angled workplane at a compound angle (Figure 14).

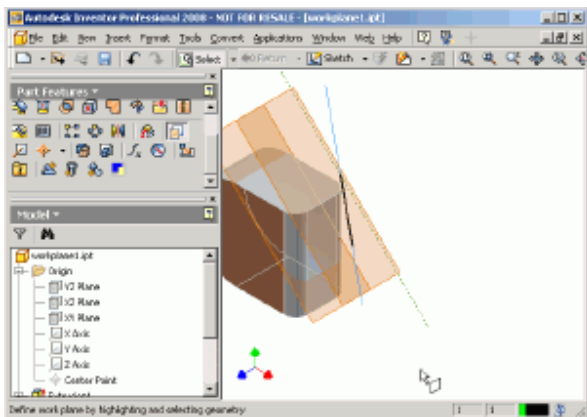


Figure 13: Create a sketched line on the workplane

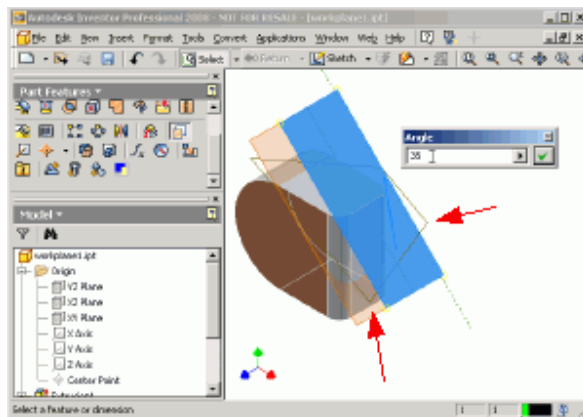


Figure 14: Use the line/workplane to create 2nd WP

Create a workplane at the end of a path

If you are planning to use the sweep or the loft command in inventor, you will need to create a sweep or loft path or paths, and then create a second 2-D sketch at the origin of the path. The easiest way to accomplish this is to create a workplane that is perpendicular to the path, and then create a new sketch on that workplane.

1. Using 2D or 3D sketch, create a path for the loft or sweep.
2. Select the **Workplane** command. Pick the endpoint of the path created in step one, where you plan to place your profile sketch. After picking the endpoint, select the path line. The workplane is created. Be sure to flip the normal of the workplane for proper sketch orientation. (Figures 15, 16)

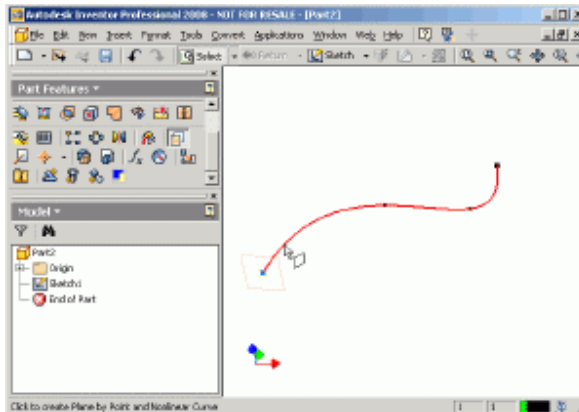


Figure 15: Select endpoint, then line/curve

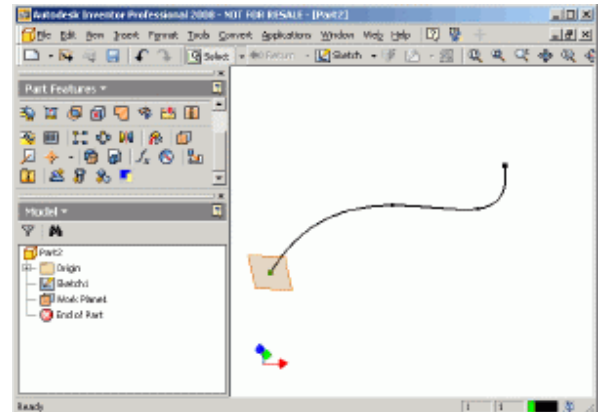
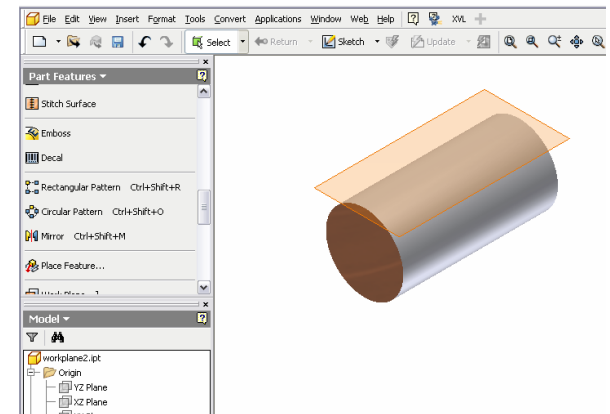
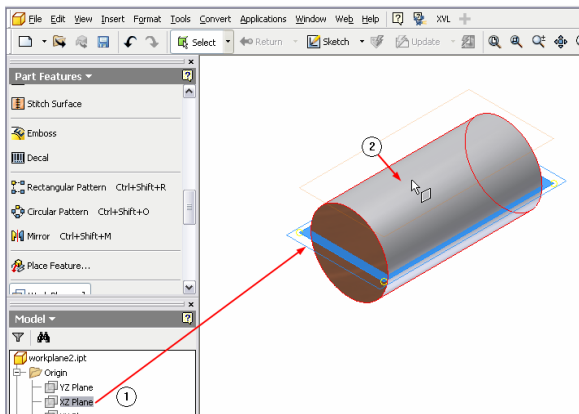


Figure 16: Workplane is created at end of path

Workplanes on cylindrical objects

Workplanes may be created on a cylindrical face by utilizing origin planes or other workplanes as a reference plane.

1. Open the file - **workplane2.ipt**
2. Select the **Workplane** command.
3. Select the reference plane in the browser, or if visible, it may be selected in the graphics window.
4. Select the side of the cylindrical face where you want the workplane to be created.

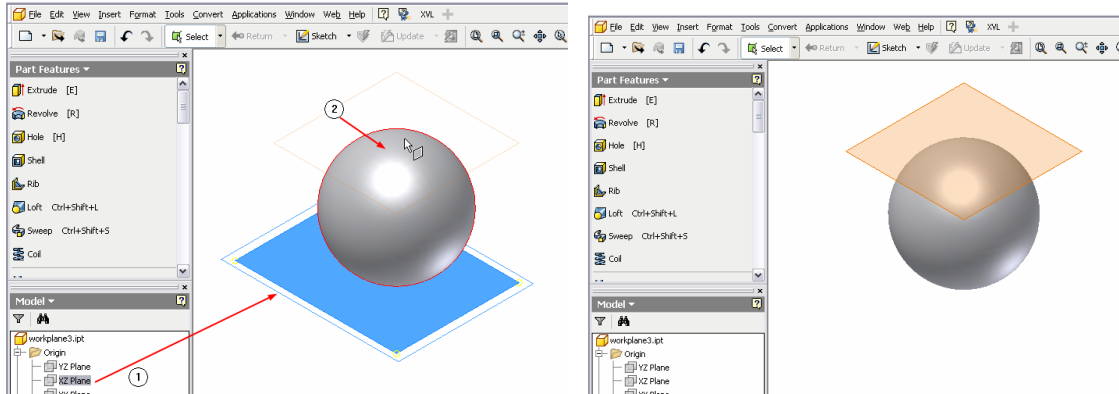


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Workplanes on spherical objects

Workplanes may be created on a spherical face by utilizing origin planes or other workplanes as a reference plane.

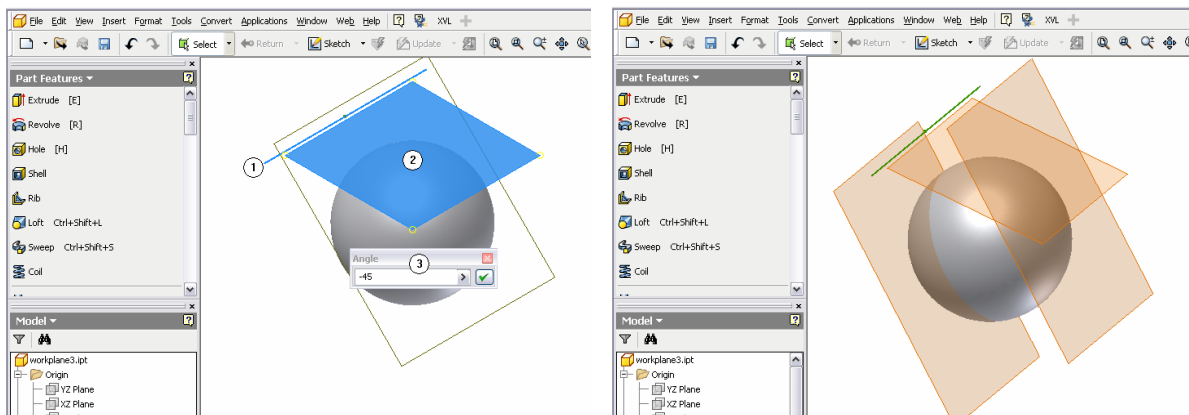
1. **Open** the file - **workplane3.ipt**
2. Select the **Workplane** command.
3. Select the reference plane in the browser, or if visible, it may be selected in the graphics window.
4. Select the side of the spherical face where you want the workplane to be created.



Angled and Compound Angle Workplanes on Spheres

Angled workplanes can be created on any object including cylindrical as vertical objects. In order to create an angled workplane you must have an existing workplane or planar face on the part and an edge or axis.

1. **Open** the file - **workplane3.ipt**
2. Select the **Workplane** command.
3. Select the reference plane in the browser, or if visible, it may be selected in the graphics window.
4. Select the side of the spherical face where you want the workplane to be created.
5. Select the newly created workplane, right mouse click - New Sketch. (Note: existing work Axis may replace Step 5 and Step 6).
6. Create a sketch line within the new sketch. Anchor the sketch line and Finish Sketch.
7. Select the **Workplane** command .
8. Select the sketched line created in step 6.
9. Select the workplane created in step 4.
10. The angle dialog box will appear. Type in the desired angle which may be preceded by a negative sign to flip the angled direction in the preview as shown below.
11. If needed, create an offset workplane from the angled workplane, tangent to the desired face.
12. Uncheck the visibility on any unneeded sketches or workplanes.



Use your imagination

This list of how to use is in no way complete. A great way to learn all of the ins and outs of creating and using workplanes is to branch out onto your own and try different approaches to solving problems. Practice on all three files included in the downloaded zip file.

Creating and Using Work Axes

Along with workplanes, work axes and work points allow considerable flexibility for creating new sketches, reference points and constraining features. Properly implemented, work features will allow you to create virtually any type of complex geometry through the use of 2D and 3D sketches.

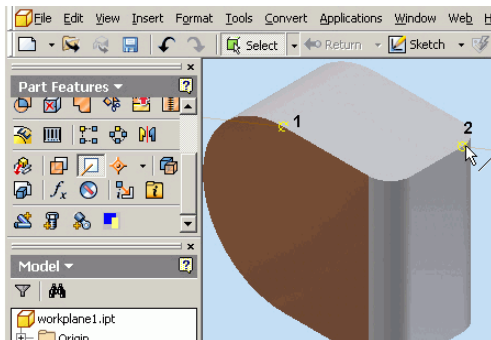
Work Axes

We normally think of a work axis as a straight object representing the centerline of a cylinder. However a work axis may be created in virtually all geometry using various selection methods. a number of excellent video tutorials are available within the Inventor 2008 help files. Be sure to view all of the videos on work axis creation, and explore all the options available for work feature creation within the Inventor help files.

Unfortunately, due to space limitations in this article we cannot cover all of the potential options that you may encounter, but instead we will strive to further wet your interest in work features. To follow along, please make sure that you have downloaded the exercise files from part one of this article. If you have a version of inventor older than version 9, then you will have to create examples similar to the ones provided. In your version of inventor you will want to open workplane1.IPT.

Axis By Point Method

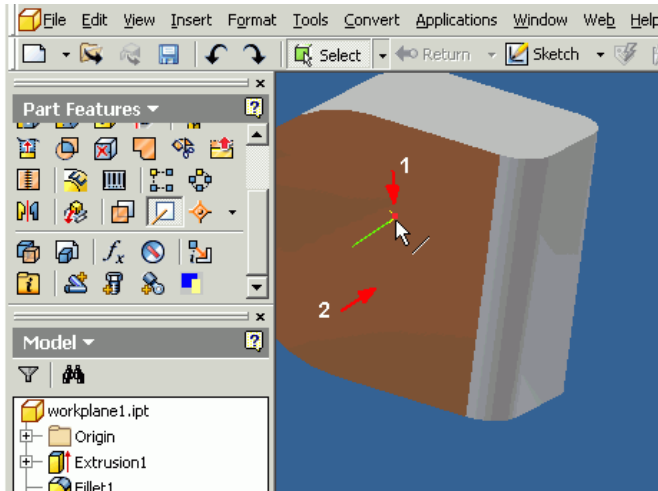
An axis may be created by selecting two vertex points on an object.



Axis By Face/Line Method

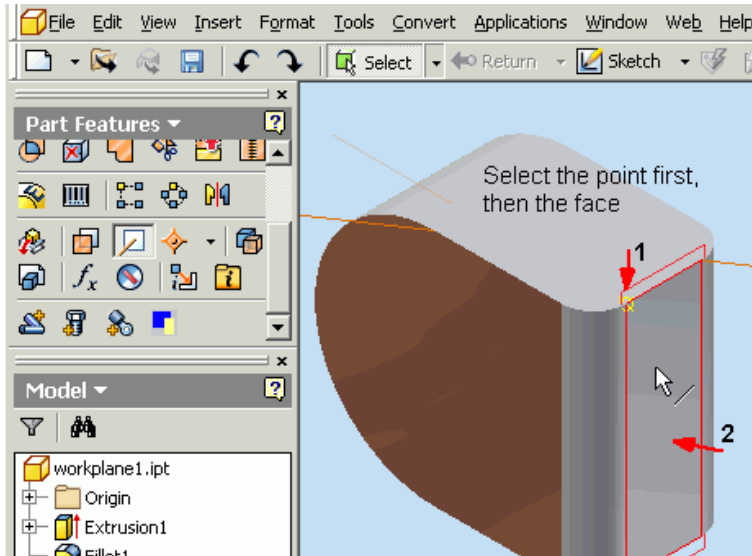
You may create a work axis by selecting the endpoint of the sketched line and a face. Alternatively, you may also select the sketch line to create the work axis.

1. Create a sketch on the brown face of the example part. Draw a diagonal short line away from the origin. Finish the sketch.
2. Create a work axis by selecting an endpoint of the sketch line, then the desired face. Selecting a face perpendicular to the sketch will produce a different work axis.
3. You may also create a work axis by selecting the sketch line only.



Axis By Face/Point Method

You may create a work axis perpendicular to a planar face, intersecting with a point, by selecting the vertex point, then the face.

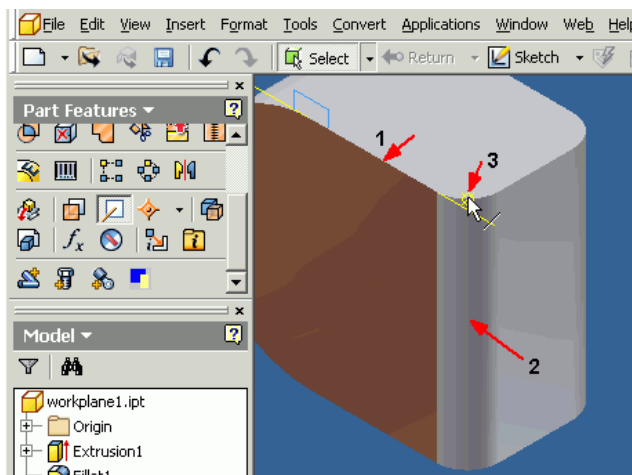


In-Line Work Axis

An in-line work axis is created by selecting edges, vertex points, and faces in conjunction with using the right click in-context menus available within the work axis command. In the illustration below, if we select the top edge (1) before we right mouse click, we will merely create a work axis on the selected edge. However, if we right mouse click and select Create Plane before we select the edge, we can then continue to select faces points and edges to create the desired work axis in the proper location.

Create a work axis by selecting options from the right click menu

1. With the work axis command active, right mouse click and select Create Plane. Pick the top front facing edge.
2. Select the front curved face as shown below.
3. Select the midpoint of the curved edge of the same face.
4. The work axis is created perpendicular to the brown face at the midpoint of the arc at the top of the face.



In the next part of this series, we will explore the work point options, then learn how to combine all three work features to provide total control over work feature creation and placement.

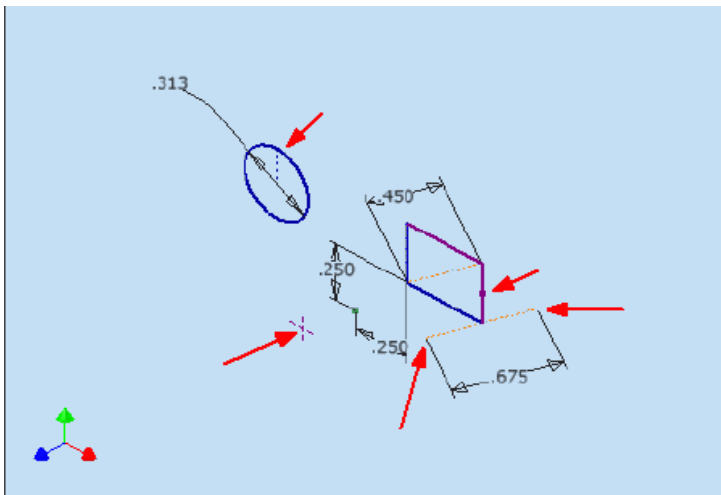
In your version of Inventor, you will want to **open Workfeatures4.IPT**.

Work points

Work points allow considerable flexibility for creating new sketches, reference points, and constraining features.

Work points by sketch method

Work points may be attached to various sketch objects, allowing the position of the work points to be determined by the size and shape of the sketch. In Figure 1 below, attach a work point to each of the five indicated locations. When the work points have been attached, edit the dimensions in the sketch to see how the points follow the sketch. When the sketch geometry has been changed, if needed, select Update from the Standard toolbar to update the points.

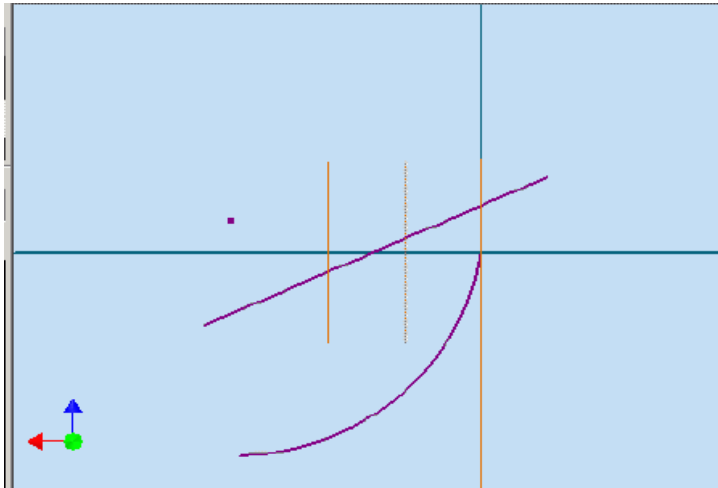


Work point by face/curve method

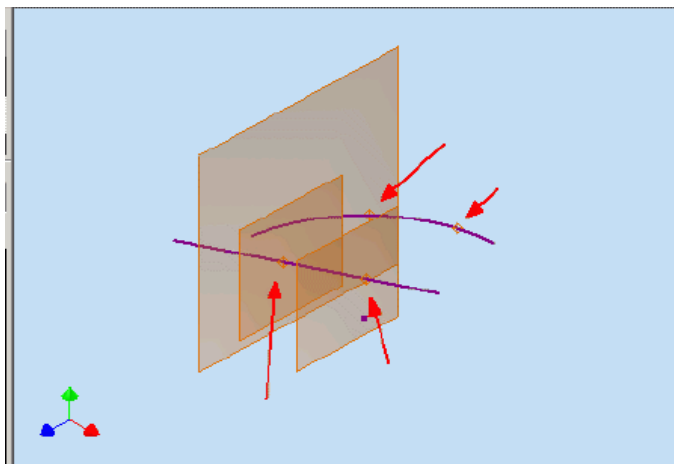
It will often be necessary to place points along a line, curve, or other geometry for the purpose of anchoring sketches, creating a 3D path, or similar workflows. Valid work point locations include part or assembly level vertices, sketch object endpoints, and other selectable objects. In many cases, the placement of work points will require the combined selection of two objects.

In this example, we will explore placing work points by one of the multiple selection options. To follow along, open the file **workfeatures5.IPT**.

1. Activate sketch2 for editing.
2. Using the Look At tool, orient sketch2 by selecting it in the browser.
3. Select the Arc tool and create an arc starting at the origin, with the second point in the lower left quadrant of the sketch, as shown in Figure 2.



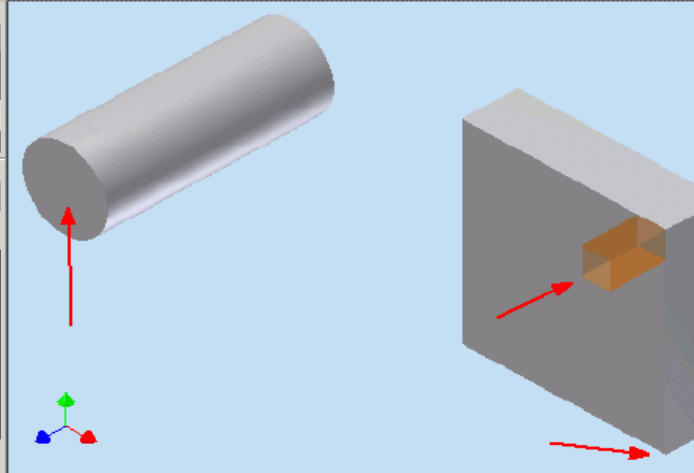
4. Finish the sketch and select Isometric View.
5. Using the Work Point tool from the Part Features panel, pick the line and then the YZ origin plane -- a work point will be created at the intersection. Repeat the Work Point command for each workplane/line or arc intersection to create a string of work points.
6. Edit workplane1 and workplane2 to change the offset distances from the origin YZ plane.
7. Pick Update to move the workplanes into the edited positions; the related work points will follow. See Figure 3.



Placing a 3D path by work points

3D paths are used for many different types of designs, including a sweep, loft rail, pipe or tube path, or other type of objects. It's next to impossible to create a 3D path by itself without some sort of reference objects or vertices.

For this example, we will use the downloaded **3Dpath1.IAM** file. This assembly contains a cylinder and a box that includes a surface feature.

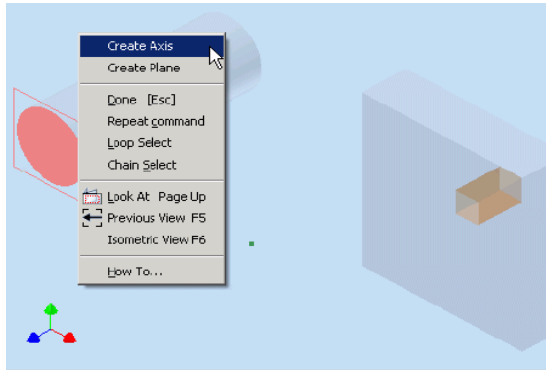


Our challenge will be to create a 3D path from the center of the circular face to the corner of the surface feature and then continue to the corner of the box as indicated. In order to create the 3D path, we will need to create a new part named Route.IPT.

1. Select the Create Component tool from the Assembly panel and create a new part called Route.IPT. Pick anywhere in the graphics window to create the part. Exit Sketch1.
2. Right-click and select New 3D Sketch from the in-context menu.
3. Select the Work Point command.
4. Activate the cylinder for editing within the assembly by either double-clicking the cylinder, or by selecting the cylinder, right mouse click and select edit. Then, select the circular face, right-click and select Create Axis (see Figure 5). The work point is now created in the center of the circular face. Pick Return at the top of your screen to return to the assembly environment.

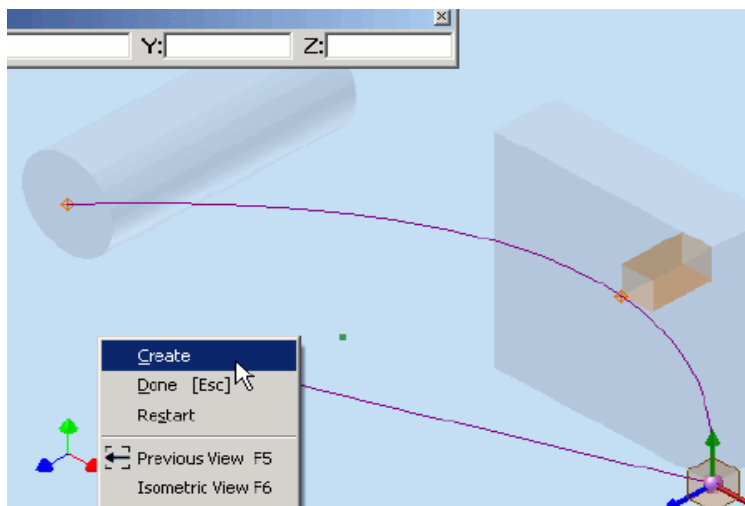
The reason that the cylinder must be edited outside of the assembly environment is because the right-click commands for work point are not accessible within the assembly environment, but only at the part level. The right-click commands available at the part level are:

- **Create Axis**
- **Create Plane**



5. Repeat the Work Point command and pick on the two vertices on the box as indicated in the illustration below.

6. Remaining in the current active 3D sketch command, select the Spline tool and connect the three work points to create a 3D path, as shown in Figure 6.



7. After creating the 3D path, return to the top level assembly, adjust the Mate2 constraint on the box, and watch how the path will change accordingly.

Get more information about Inventor's work features

There are a wealth of tutorials and examples about work features within the **Inventor 2008/2009 help** files under the topics: Create a Work Axis, Create a Work Point, and Work Plane References. If you take the time to explore these three topics, you will find valuable methods for using work features within your part and assembly files.

Review Questions (please answer)

1. What part objects may be used for work feature creation? Circle all that apply.

- Origin Work Features
 - Linear part edges
 - sketch geometry
 - part vertices
 - user created work features
 - _____
-

2. Describe the steps that you would use to create an angled hole in a flat plate.

ASSIGNMENT - WORK FEATURES

The ability to create work features on complex parts is essential to success in 3-D modeling. Proper placement of workplanes allows creation of features on any location within a 3-D part or assembly. Because of the essential nature of work features, successful completion of all exercises contained in this document is required for receiving a certificate at the completion of the course.

Your challenge for this week's assignment is to complete all of the tasks in the above tutorials, and to save the files for grading.

Creative Design With Autodesk Inventor 2009 Courseware

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Once all of the files and exercises have been completed, combine all files into a single zip file. Name this file; (my name).zip and e-mail the zip file to **djeffrey@design-excellence.com** (100MB email box).

This assignment should be completed prior to the next class. However if time constraints prevent the completion of the entire assignment, you may still submit the assignment at a later date. However, your certificate of completion will not be available until all required assignments are submitted and graded.

If you experience difficulty in creating the work features described in this paper, please do not hesitate to e-mail me, or call me direct. We can then discuss the problem, or initiate a Live Meeting session to clarify the issue.
