# Lesson 15
# Creating a Base Sweep

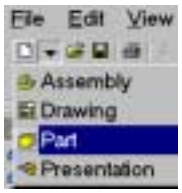
## Learning Objectives

In this lesson, we will gain mastery over the following tools:

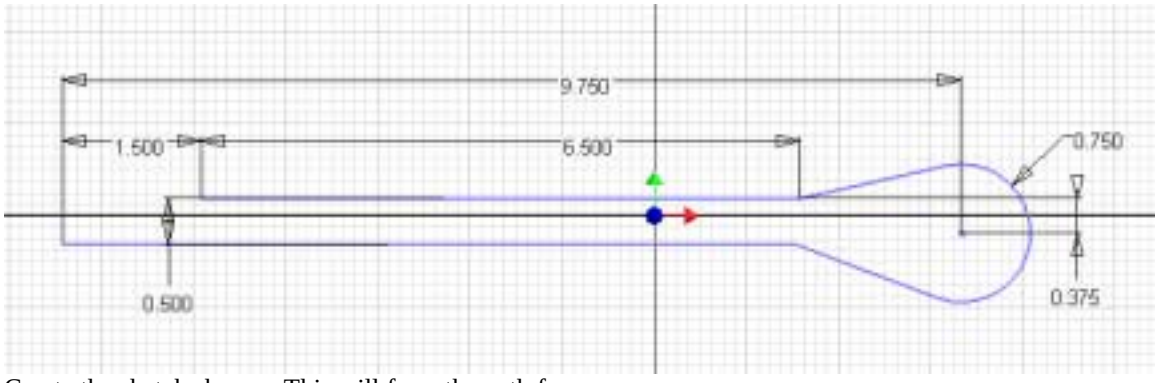
- Sweep

The features of "real world" parts can most often be modeled using *Extrude* and *Revolve*. *Sweep* and *Loft* are powerful commands because of their capability to produce complex part geometry. In this exercise, we will create a cotter pin using a Sweep operation.

Sweeps are unique in that they require two profiles. A good practice is to create the path sketch and rename it as 'path' to make it easy to locate in the browser. Then create the profile sketch and rename it 'cross_section' in the browser.




Start a new part using inches.

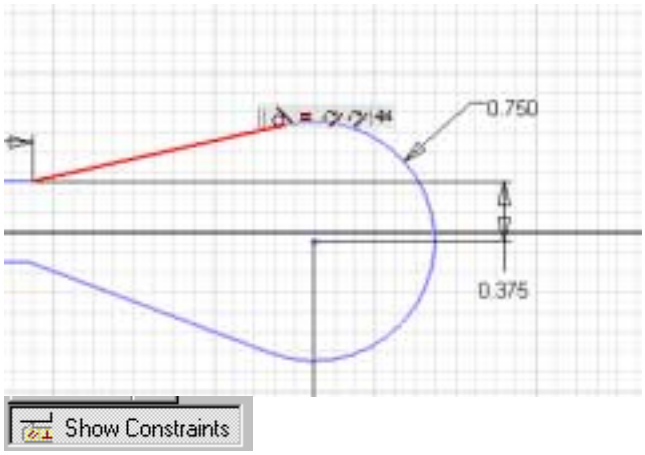



Create the sketch shown. This will form the path for our sweep.

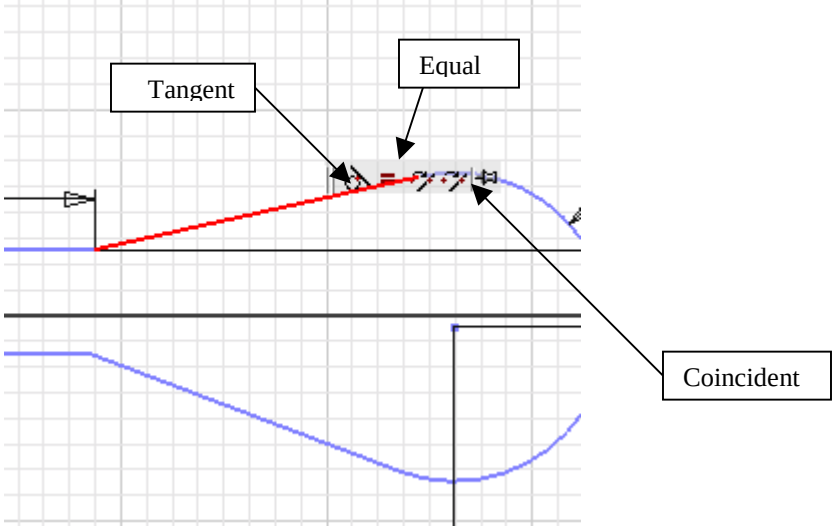
**Hints on sketch creation for the path:**

Use constraints to help create the geometry.
The slanted lines connecting the horizontal lines to the arc should be constrained equal.
Add a tangent constraint between the slanted lines and the arc.

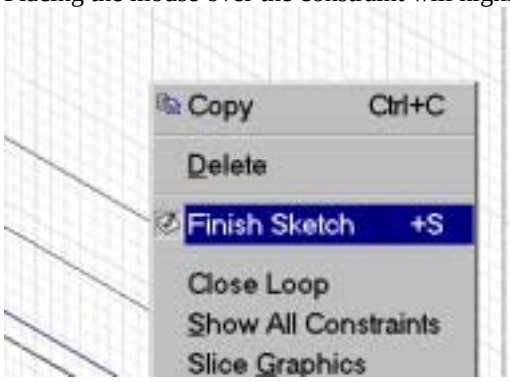
**TIP:** A path may consist of lines, splines, or arcs. It must be a 2D sketch and reside on a single work plane. It must be an open sketch and should not intersect itself.




Use Show Constraints to see the constraints applied to the geometry.




The image shows the constraints applied on the upper slanted line.
Placing the mouse over the constraint will highlight the geometrical relationship.




Right click and select ‘Finish Sketch’.

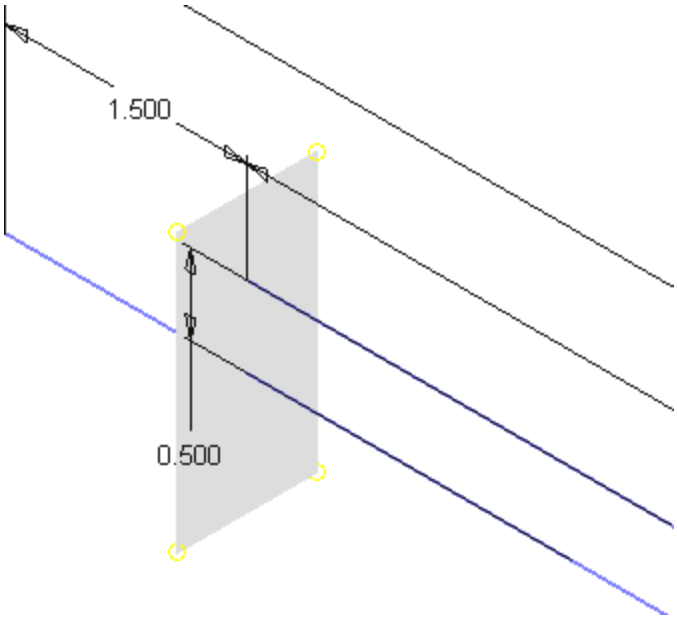

Locate the sketch in the browser and rename it ‘path’.

## Paths

Paths must be on a plane that intersects with the sweep profile plane.

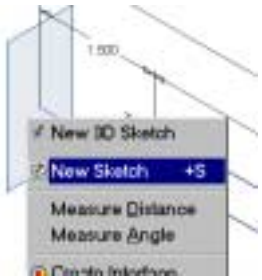
**Create Work Plane**

Select the work plane tool from the Features toolbar.

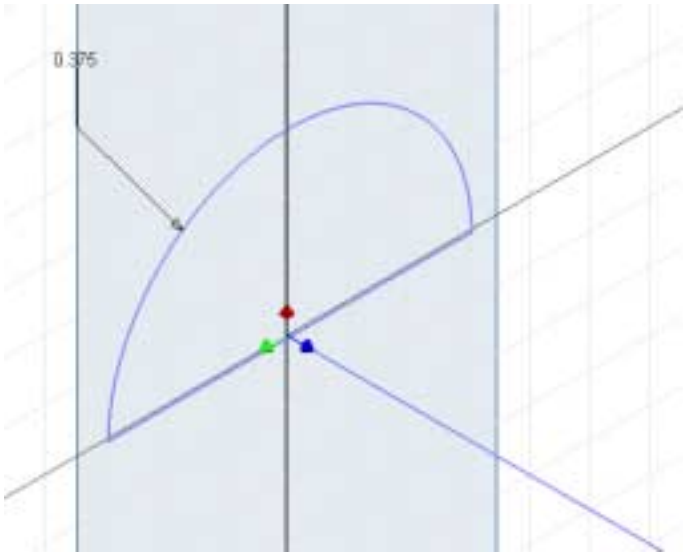




Select the endpoint of the bottom line and then the top line. A work plane will be placed normal to the line. It may take a few tries before you are able to place the work plane successfully.

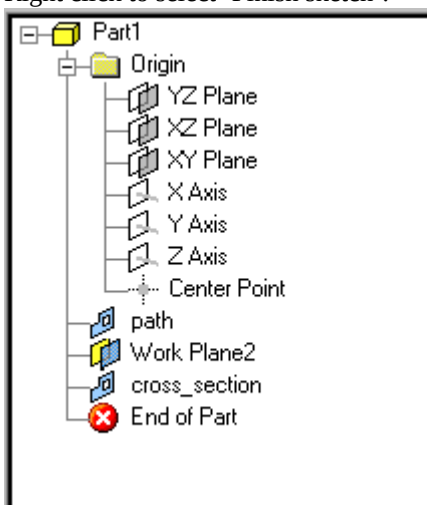



Select the plane just created in the browser or in the graphics window. Right click and select 'New Sketch'.




Create the sketch as shown. It should be oriented so the center of the hemisphere is aligned with the endpoint of the path. The radius is 0.375.

Right click to select 'Finish sketch'.



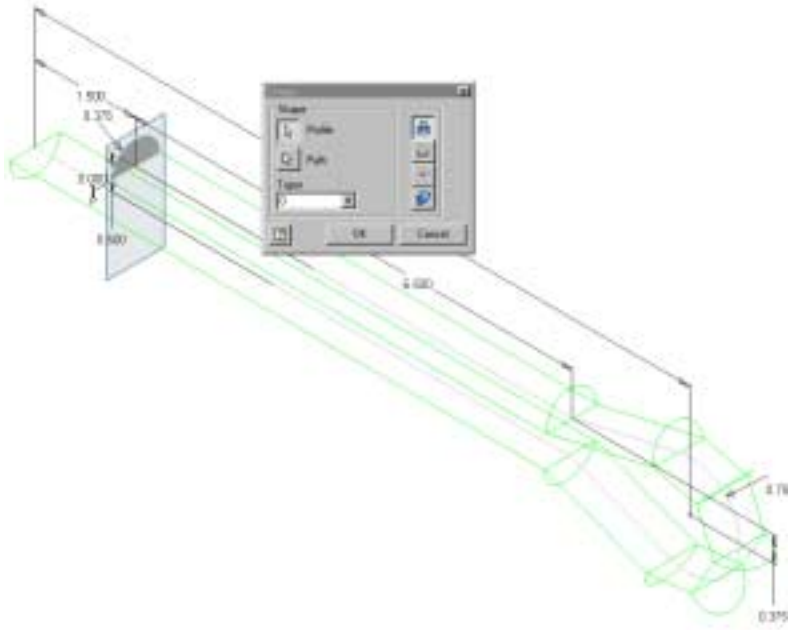

Rename the sketch 'cross_section'.

Time to create the sweep.

**TIP:** The user can specify more than one profile to sweep along a path.
Profiles may be nested but may not be intersecting.
To select more than one profile, press the Control key and select the desired profiles.
Profiles may reside on the same plane or different planes, but all profiles must be perpendicular to the path.

## Creating a Sweep

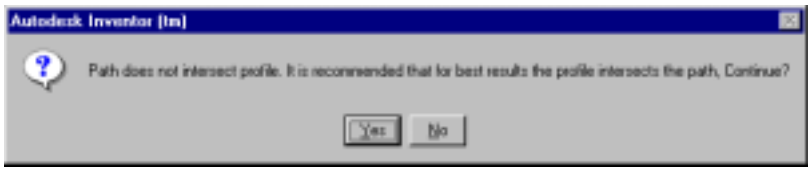
 Select the Sweep tool from the Features toolbar.



Inventor will make some assumptions on which profile is the path and which profile is to be used for the sweep. We use the preview to assist us in deciding if everything has been set up OK. We can always use the selection tools in the Sweep dialog box to make changes.

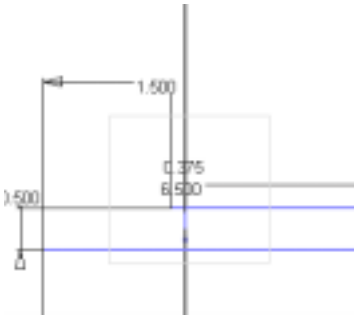
To use the selection tools, press the Profile button and then select the half-circle sketch. Then select the Path tool and select the path. You can select the cross_section and path from the browser or the graphics window. Notice that the default is the 'Join' button – the top button.
The preview in this case looks perfect, so we select 'OK'.

## If the sweep gives an intersect error

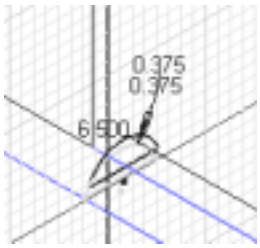



It means that your path and profile are not aligned.

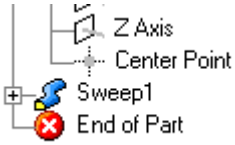

Go to the cross section sketch in the browser. Select, right click and select 'Edit Sketch'.




With your mouse, draw a window around the sketch.
The sketch will grey out. We can now move our sketch.




Hold the mouse down and drag the sketch into a new location.
Right click and select ‘Finish Sketch’.
Try performing the sweep again.




The sweep feature appears in the browser.

Use Save to save our completed cotter pin. Name the file ‘cotter-pin’.

## Review Questions

1. The Sweep tool is located on this toolbar:

    A. Feature
    B. Sketch
    C. Drawing
    D. Sheet Metal

2. To perform a sweep, the user needs to create these two sketches:

    A. Profile and Sweep
    B. Top and Bottom
    C. Handle and Bristles
    D. Profile and Path

3. If an intersection will occur:

    A. Inventor will make the sweep any way
    B. Inventor shows a dialog box warning the user and asks if the user wishes to continue
    C. Inventor will lock up and cause a fatal error
    D. The Design Doctor will automatically pop up to correct the problem

4. To add a draft angle to a sweep, the user should add a value to this option in the Sweep dialog:

    A. Draft Angle
    B. Taper
    C. Angle
    D. Face Draft

5. When creating a sweep, all of the following are true EXCEPT:

    A. There can only be one path
    B. There can only be one profile
    C. There can only be one direction
    D. There can only be one draft angle

6. The dialog box for the Sweep command changes depending on:

    A. How many profiles are defined
    B. Whether the sweep is a base feature or not
    C. What type of path is defined
    D. Whether or not a draft angle is defined

ANSWERS: 1) A; 2) D; 3) B; 4) A; 5) B; 6) B