

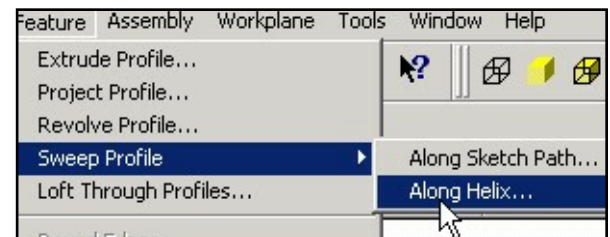
Pro/DESKTOP Tips

Pro/DESKTOP uses five primary features to generate 3D models from 2D sketches: Extrude, Project, Revolve, Loft and Sweep. One of the coolest is Sweep.

By Stephen Yaffe

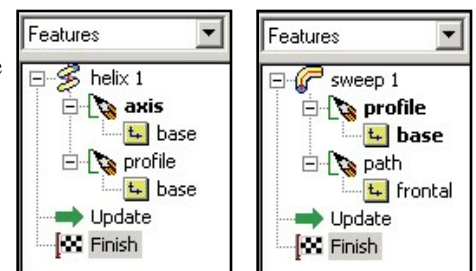
Clean Sweeps

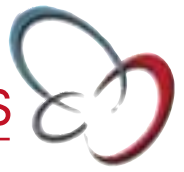
There are two ways to accomplish a **Sweep**, but three ways to access the feature, and three sets of nomenclature! If, on the one hand, you access **Sweep** from the feature icon toolbar at the top of the screen, you



will be presented with a single option to **Sweep Profile Along Profile**. That's perfectly straightforward. On the other hand, if you opt to access the **Sweep** command from the Feature menu at the top of the screen, you will be presented with what appear to be two additional options: **Sweep Profile Along Helix**, and **Sweep Along Sketch Path**. So, here's the first tip: Despite the different terminology, **Sweep Along Profile** and **Sweep Along Sketch Path** are the same thing.

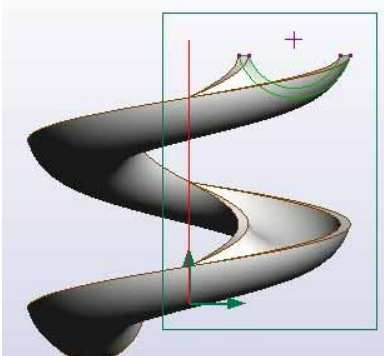
The two approaches have different sentence structures in the object browser.





Sweep Along Path = Feature > applied to two sketches > applied to two perpendicular workplanes

Sweep Along Helix = Feature > applied to two sketches > applied to one workplane

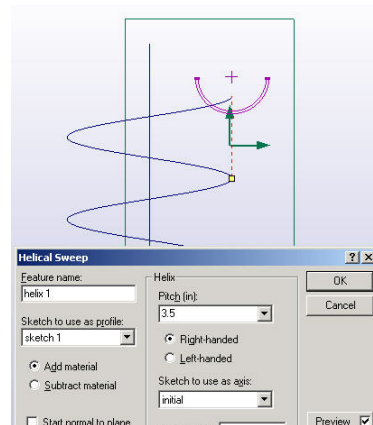
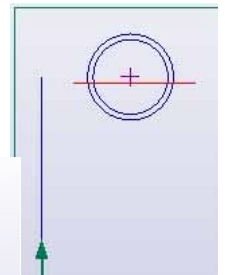


SWEEPING ALONG A HELIX

Sweeping along a helix is much like revolving a profile.

Here are the steps for making a helical sweep (like the one shown here) using an abbreviated, short cut notation system.

1. Open a new design file: **[Ctrl + N] > Enter.**
2. Drag a straight line starting at (0,0): **[S] > Drag straight line, 5 inches long.**
3. Rename this sketch: In the **Workplane browser**, click on the plus sign by the base workplane > Double click on the word Initial to open properties box > Rename sketch "Axis."
4. Create a new sketch: In the **Workplane browser**, right click on the **Base** workplane > select **New Sketch** > Name it "Profile."
5. In this new sketch, drag two concentric circles originating at (2,5) > Drag a line cutting off the top three fifths of these circles > **[D]** Delete the upper line segments > Leave the segments that connect the two arcing segments to complete a valid profile.
6. Create the sweep. Once you have completed these two sketches, from the **File** menu choose **Feature** > Select **Sweep** and then **Along Helix**. The **Helical Sweep** dialogue box will appear.
7. Select **Profile** as the sketch to use as the profile and make sure that the axis is selected (it will turn red).
8. Then, drag the handle down to set the **Pitch**. > Click on **OK** when you have a pitch that suits your purposes.

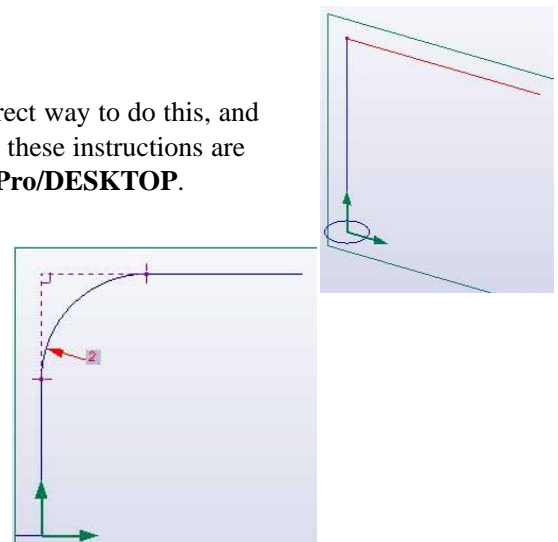


SWEEPING ALONG A PATH

Now comes **Sweep Profile Along Path** (or profile). First we will cover the correct way to do this, and then we will discuss common pitfalls and how to resolve them. Please note that these instructions are abbreviated, assuming that readers have had at least a passing experience with **Pro/DESKTOP**.

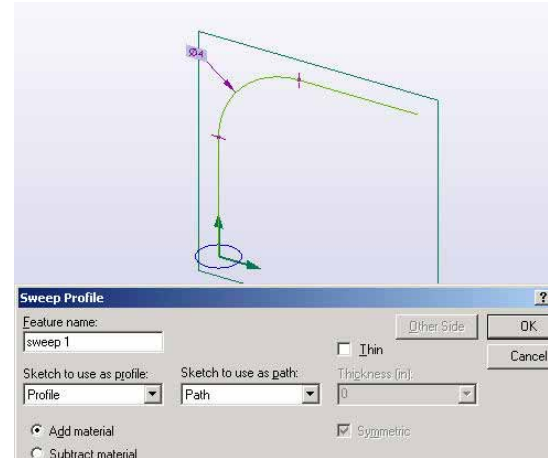
To set up, you will need two sketches in two perpendicular workplanes.

1. Open a new design **[Ctrl + N] > Enter.**
2. Rename the initial sketch "Profile:" In the **Workplane browser**, click on the plus sign by the **Base** workplane to access the sketch. > Click once on the word "Initial", then click on it a second time to rename > Change the name to "Profile."
3. **[C]** Sketch a circle: Snap to grid (0,0), diameter 1 inch.





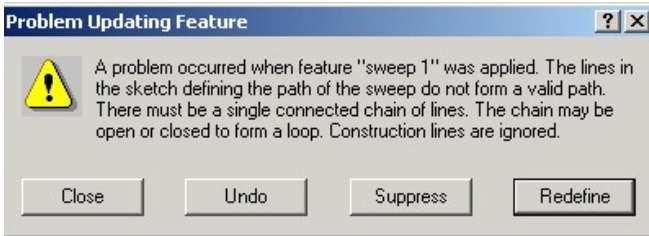
4. Select a perpendicular workplane (there are two to choose from): [W] > Click on one of the two workplanes perpendicular to the base turning it red > Right click > Select New Sketch > Name it "Path."
5. Drag a vertical straight line 5 inches long (holding the shift key constrains lines to be vertical or horizontal) > Starting at the end point of the first line, drag a horizontal straight-line, length 5 inches.
6. Apply a 2-inch radius to the corner. Click on the Sweep icon at the top of the screen > In the dialogue box that opens, make sure that each sketch is in its proper field, "Profile" in Profile, "Path" in Path > Click OK.



Now the pitfalls.

COINCIDENCE. It is important to remember that for a profile to be valid, it must form a single, connected chain of lines. Pro/DESKTOP provides visual clues to indicate that your lines are connected.

In this image you see tiny red squares at the intersection of the radius and the straight lines. These squares are called coincident points, and they indicate that the lines have been successfully connected. If you don't see them – as in the image to the right – you are likely to get this **Problem Updating Feature** error message.



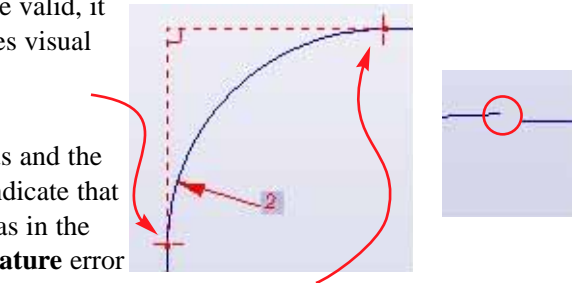
message.

Another visual clue is the dashed line right at the coincident points. This dashed line indicates that the radius is *tangent* to the straight line.

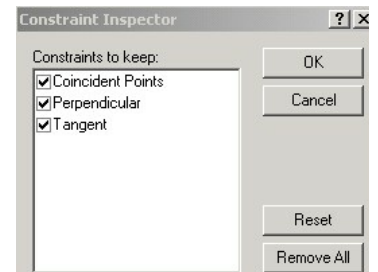
Sweep isn't working, you can look to see if these dashes and squares are present. You can also click on any two lines in the sketch (shift functions as the word "and") and from the **Constraint** menu select **Inspector**.

If your sketch doesn't have these constraints, you can apply them yourself. Just click on any two adjacent lines > From the **Constraint** menu select **Tangent**. The two lines should move together and the dashed line should appear indicating that the lines are tangent. It is a good practice to **Fix** a line in place before using constraint commands or the lines might move in unpredictable ways. With a line selected (red), go to the **Constraint** menu at the top of the screen > Select **Toggle Fixed**.

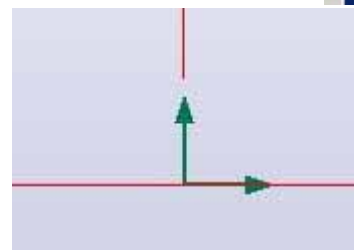
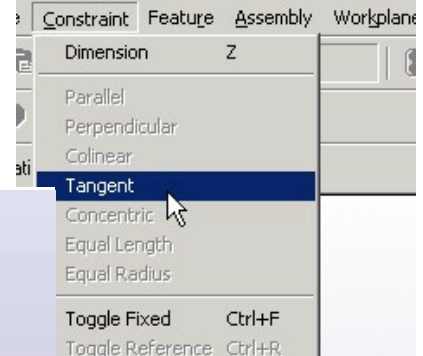
PERPENDICULARITY. What if all your lines are connected and tangent to radii and your sweep still doesn't work? Bear in mind that for a sweep to work, you need two sketches in two perpendicular workplanes. It is vital that the first line of the path meet the profile *absolutely perpendicularly*. The most effective way to ensure that the path is perpendicular to the profile is to hold the shift key while dragging the line; this ensures that a line is vertical or horizontal.



If you have made a complex sketch to use as your **Path** profile and your



Between the selected lines and others
 Within the selected lines



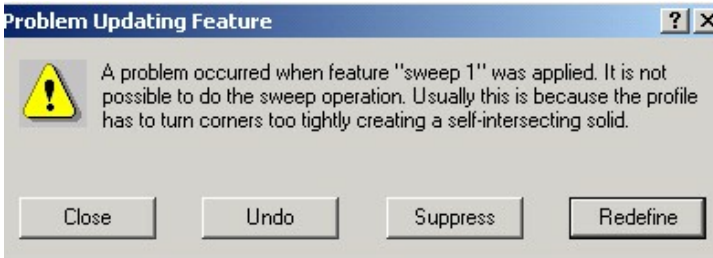


Ok, but what if the sweep still isn't working? After banging the desk you might want to check one other possibility. When we say that sketches have to meet perpendicularly, we mean the have to meet perpendicularly. The easiest way to ensure that the sketches meet is to start them both at (0,0) on the perpendicular workplanes. You can also zoom in [Shift + Z] really close to see if they are missing, like here.

If they are missing, click on the end of the path line and move it around, dragging it right to coordinate (0,0).

SELF-INTERSECTING SOLIDS. This problem occurs under the following conditions:

- > An angle is too acute
- > Your profile is too wide for you path



- > One line is much shorter than adjacent lines at a right angle to it

A FINAL TIP

If you are working on a very complex sweep project, set up the profile, set up the path sketch, and begin building the path by adding one line at a time. *Good luck and have fun!*

To learn how you can receive training in teaching with Pro/DESKTOP, go to www.ptc.com/for/education/schools/index.htm.

Stephen Yaffe

training manager for PTC's Design & Technology in Schools Program, has trained teachers all over the United States how to teach with Pro/DESKTOP 3D design software. A former small business owner, he holds a bachelor of fine arts degree from Massachusetts College of Art and a master's degree in business administration from Northeastern University.

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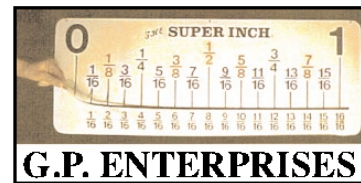


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