

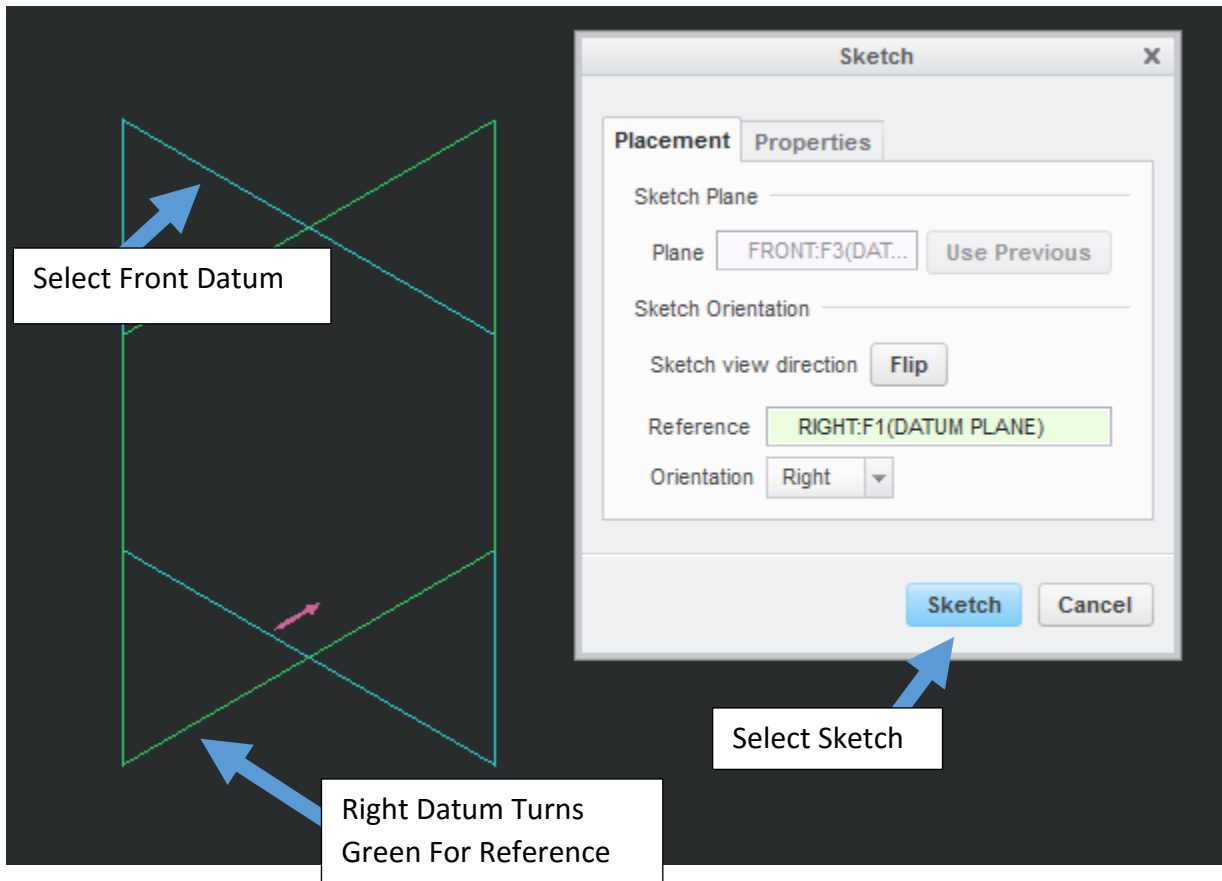
Sweep Modeling Tutorial

By: Matthew Jourden

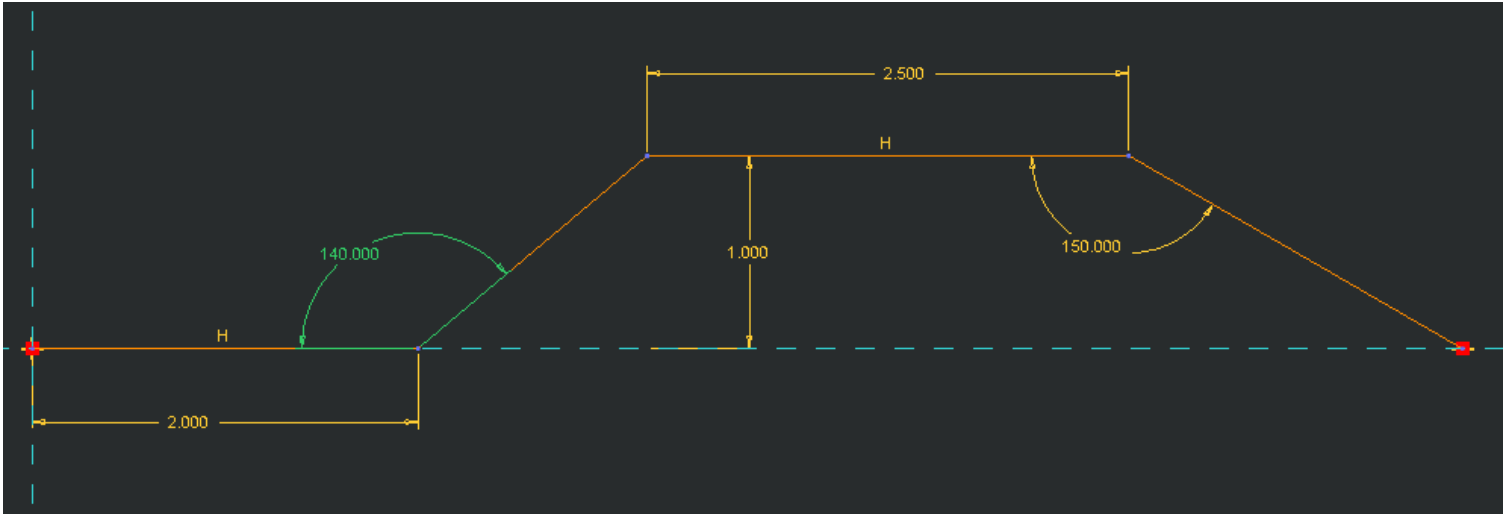
Brighton High School

Sweep: Allows the user to draw a path and a profile. The profile will follow the path. Sweep is similar to an extrude, but unlike an extrude a sweep does not have to go in a direct linear line.

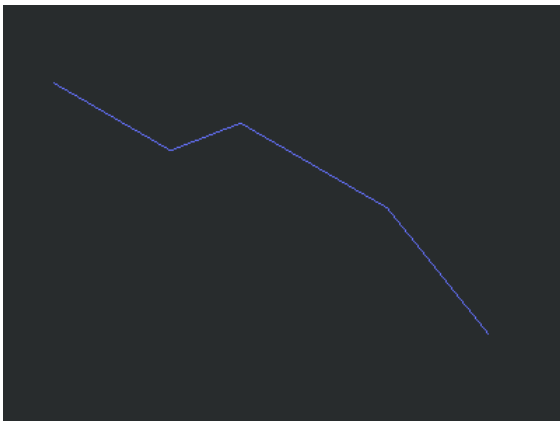
1. Open Creo > Start a new Part > Name it SweepTutorial
2. Path Creation: Select Model Tab > Sketch Icon (Sketch Icon is used to create a 2-Dimensional feature) > Select Front Datum > Select Sketch on Sketch Info box



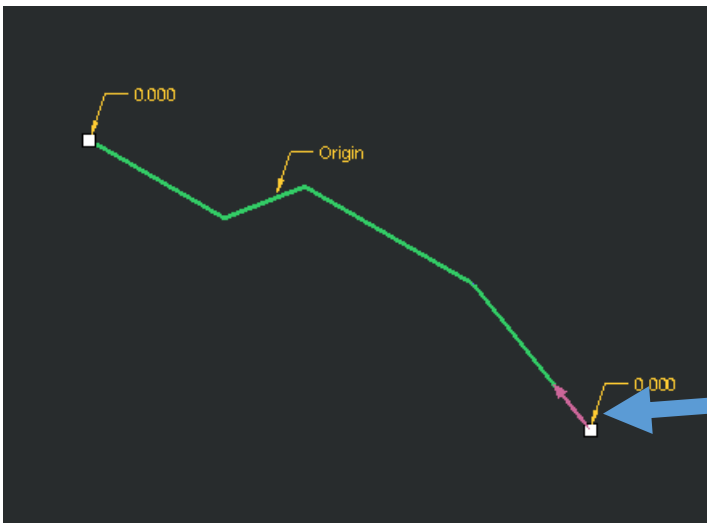
3. Sketch the Following Path on the Front Datum



- 4. Green Check to OK the Sketch
- 5. Sketch will appear as a blue line.

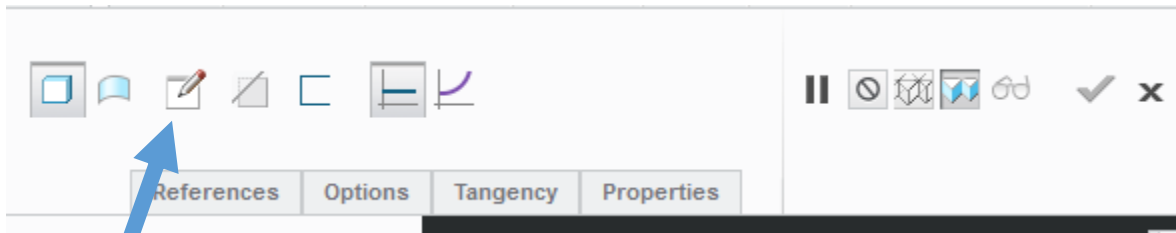


6. Profile: Select Model Tab > Select Sweep Icon > Select Path



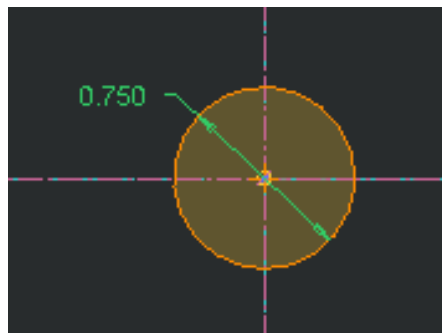
Start Point for Sweep. If the other end point is desired to start from > Left Click on the arrow and it will reposition it at the other end of the Path

- Place the arrowhead at the Origin of the Part
- Click on Sketch Icon in the Sweep Info Bar. (This will place the user into a sketch menu. NOTE the Sketch will rotate perpendicular to the Path and place the origin at the starting endpoint)



Sketch Icon

- Sketch a circle with the centerpoint of the circle at the origin (which is the starting endpoint of the Path)

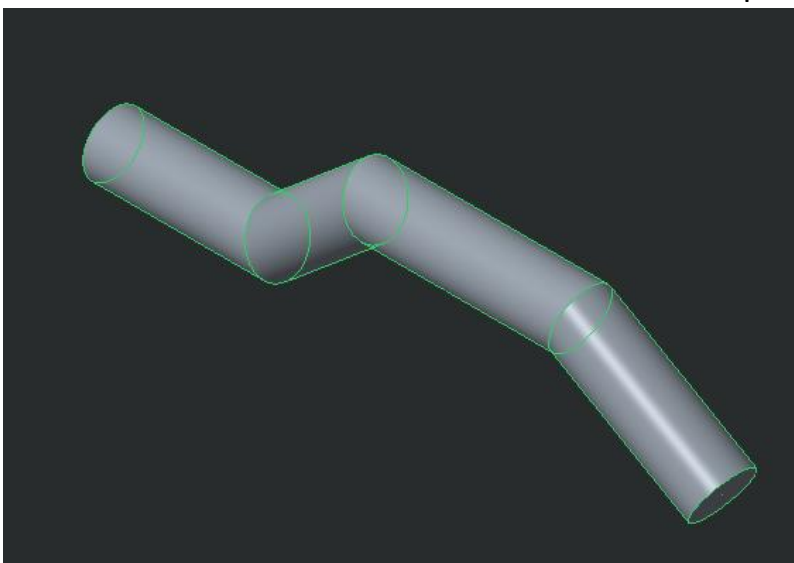


NOTE: the Profile could effect the outcome of the final product. If the Profile is too big then the profile will intersect itself on the Path; In this case 1 of 2 things will need to occur

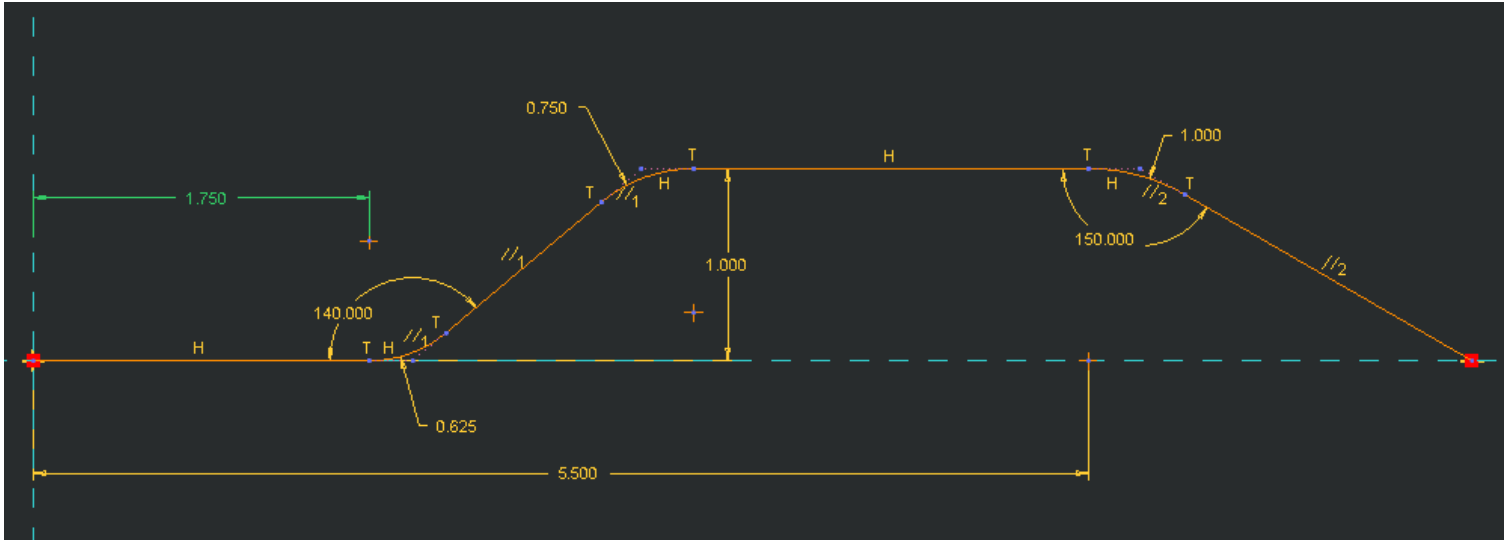
Option 1: Change the size of the profile

Option 2: Modify the Path

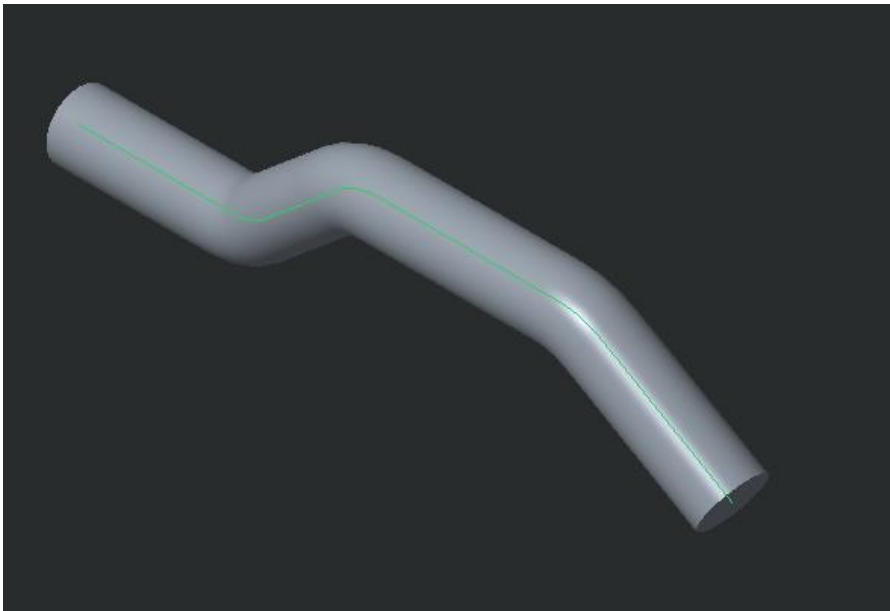
- Green Check in the Sketch > Green Check to complete the Sweep



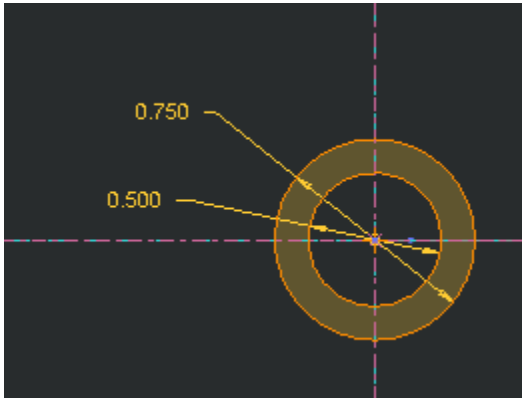
11. Modify Path: Typically the more rounded the path is the better chance the profile will be able to follow the path; this will also provide less pinch points on the joints
Right Click on Sketch1 > Edit Definition > Modify the Sketch as follows (Use the Fillet Tool in the Sketch Menu to make the arced points)



12. Green Check out of the Sketch. Notice the joints of the tube how smoothly the circle moves along the path



13. Hollowing out the Tube: Right Click on Sweep > Edit Definition > Click on Sketch Icon > Add a circle inside of the existing circle as shown > Green Check to Accept the Sketch > Green Check the Sweep to accept Sweep



Finished Part

