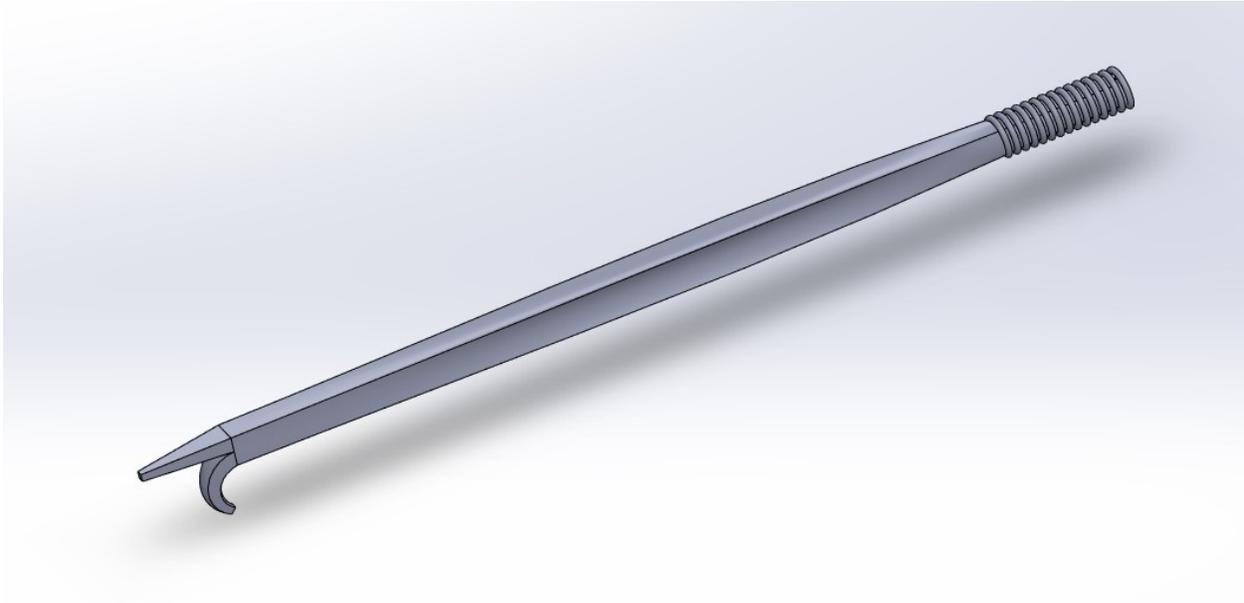


Loft and Sweep Tutorial

Topics covered: Reference Planes (Offset, Perpendicular to a curve), Simple Lofts, Loft Guide Curves, Pierce Constraint, Helix/Spiral, Sweeps.

1) Tutorial Overview

In this tutorial, you will create this basic fire poker using the loft and sweep features.

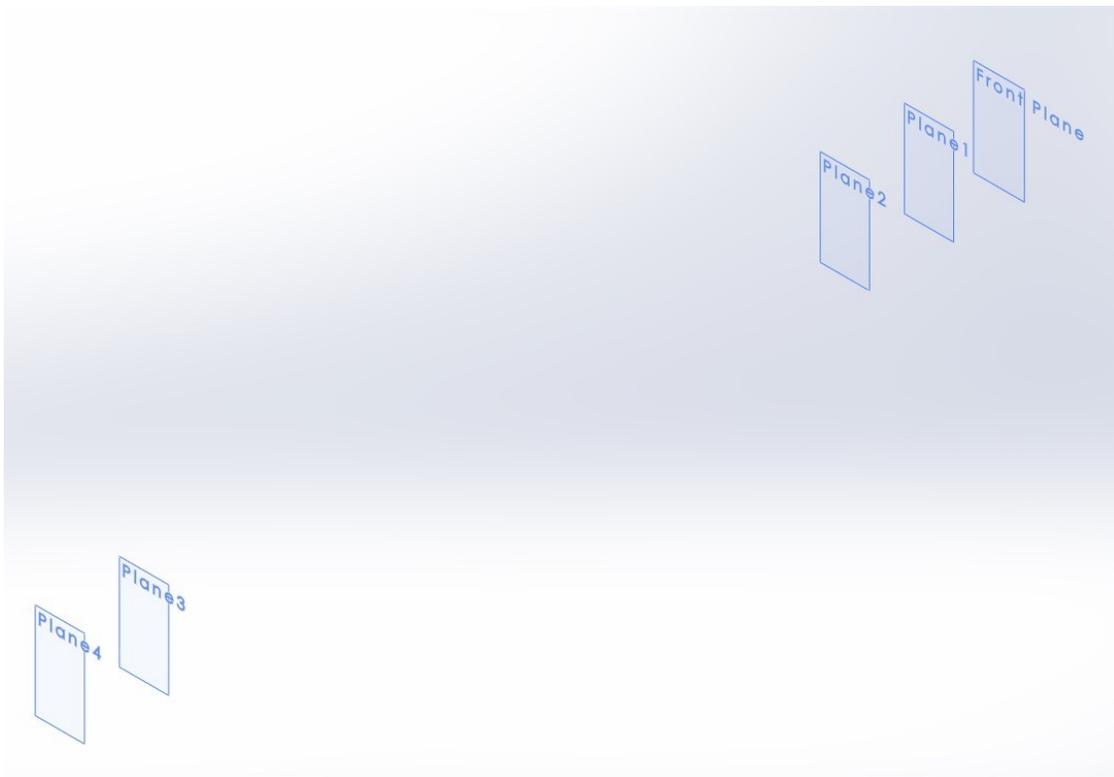


This tutorial will explain what lofts and sweeps are and lead you through the necessary steps for creating them. For the purposes of this tutorial, the positive y-direction will be considered “up”, unless stated otherwise for a specific instance.

2) Getting Started with Reference Planes

A loft is a boss or cut that is created by connecting multiple profiles, or cross-sections. To create a loft, we begin by sketching the profiles we wish to loft between on faces or on planes. If you have existing planes and faces, you can sketch on and loft between them, or you can create new planes to loft between. In this step, we will create the planes needed for the poker.

1. Open a new part by selecting the **New** icon on the Standard toolbar or by pressing **Ctrl+N** and selecting **Part**.
2. For this tutorial, it will be useful to have planes be visible, so you should verify that the **Planes** option is selected by going to **View>Hide/Show** or by clicking the **Hide/Show** dropdown. (The arrow next to the eyeball at the top of the workspace)
3. Right-click on the **Front Plane** in the design tree and select **Show**
4. With the **Front Plane** still selected, click on the **Reference Geometry** button and select the **Plane** option. In the **First Reference** box, you should see the **Front Plane** already selected. Change the offset value to 2.5 inches and click the green check to create the plane, which should be named **Plane 1**.
5. Select the new plane in the design tree and repeat the process, this time setting the offset distance to 3.0 inches. This will create **Plane 2**.
6. Again, select the new plane in the design tree and repeat the process with an offset of 25 inches. This will create **Plane 3**.
7. Finally, repeat the process one more time, setting an offset of 3.0 inches from the newest plane to get **Plane 4**.



3) Adding Sketch Profiles

Now, we will create the sketches that we wish to loft between. Note that we can loft through multiple sketches in a single feature, as long as we want a continuous solid body between them, so we will make all of the sketches for our first loft in one step.

1. Select the **Front Plane** and begin a sketch on it. Sketch a circle centered at the origin with a diameter of 1.10 inches.
2. Select **Plane 1** and begin a sketch on it. Draw another circle centered at the origin, this time with a diameter of 1.15 inches.
3. On **Plane 2**, sketch a square centered at the origin with a side length of 1.0 inch.
4. On **Plane 3**, sketch the same square again.
5. On **Plane 4**, sketch a square with a side length of 0.25 inches, with the midpoint of the bottom line coincident with the origin.

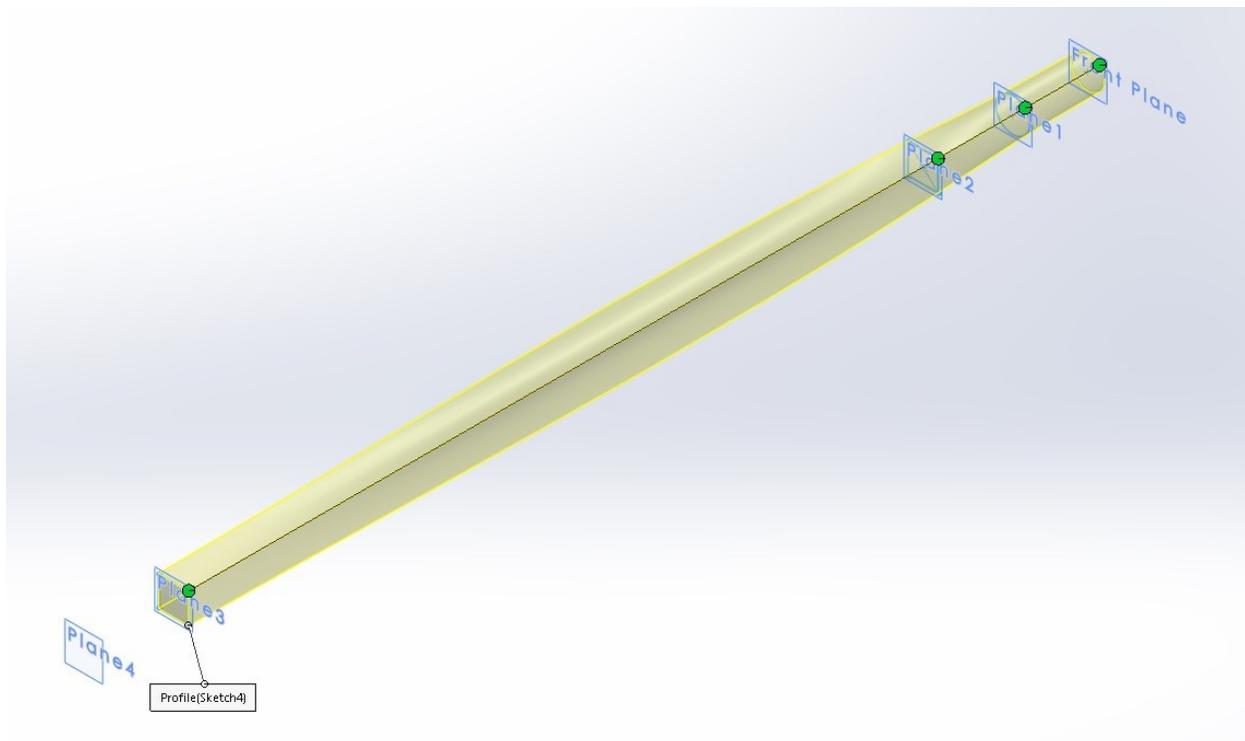


4) Lofting Through Multiple Sketch Profiles

In this step, we will make our first loft to create the shaft of the poker, as well as the tang that we will use later to make a handle.

1. First, select the **Lofted Boss/Base** tool.
2. In the workspace, click on each profile in succession, starting with the sketch on the **Front Plane**
3. You should see a yellow preview showing the loft as it will be. If needed, move the green dots to correct any twists in the loft, then click the green check to complete the loft.

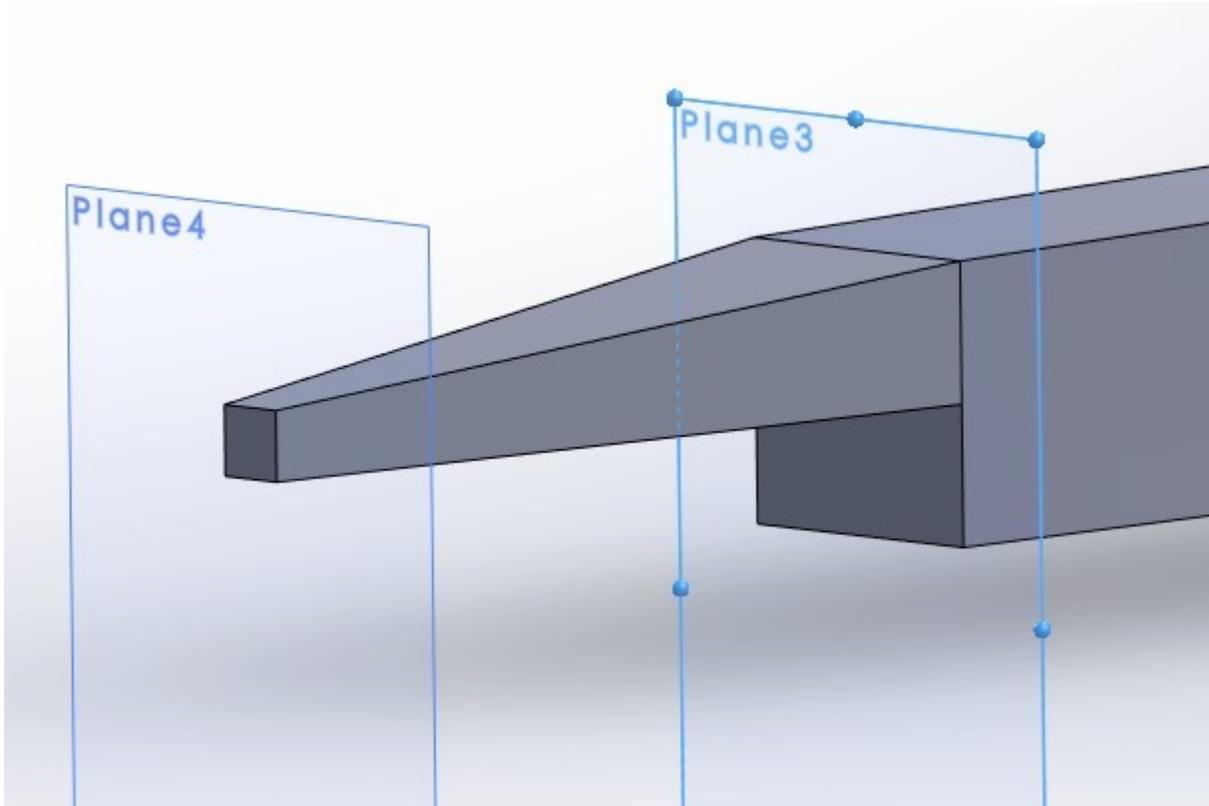
You will notice that the loft has probably created some strange geometry, particularly at the corners of the shaft. The reason for this will be explained later, and we will demonstrate the method for dealing with it.



5) Lofting the Point

Next, we will loft the straight point of the poker.

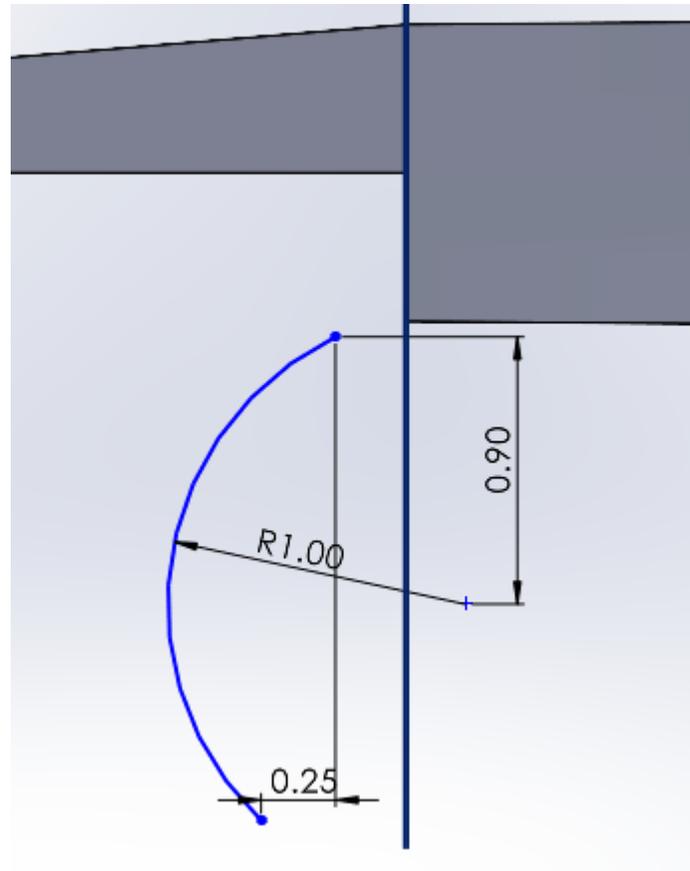
1. To begin, sketch a rectangle on the end face of the loft, making it the full width of the face and half of the height, with the bottom line coincident with the origin.
2. On **Plane 4**, sketch a square with a side length of 0.25 inches. Place the midpoint of the bottom line coincident with the origin.
3. Now, create a second loft between the two new sketches.



6) Lofting the Hook with a Guide Curve

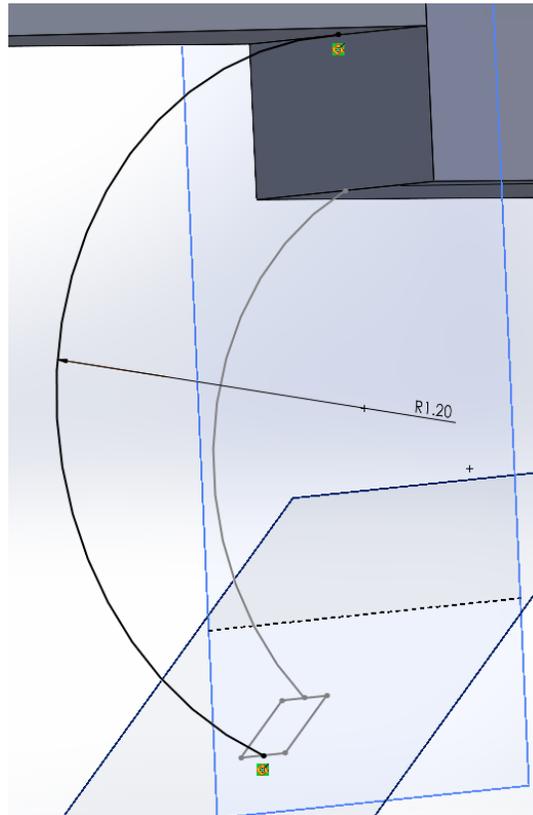
Remember the strange geometry created by the first loft? This is a result of Solidworks attempting to create the “best” path for the loft. Here, we will use guide curves to get a loft to follow the specific path we want it to take; in this case, that will be the hook of the poker.

1. A guide curve is simply a path that the edge of the loft will follow. To make the first guide curve, sketch a **Three-point Arc** on the **Right Plane** near the end of the poker. Apply the three dimensions shown below.

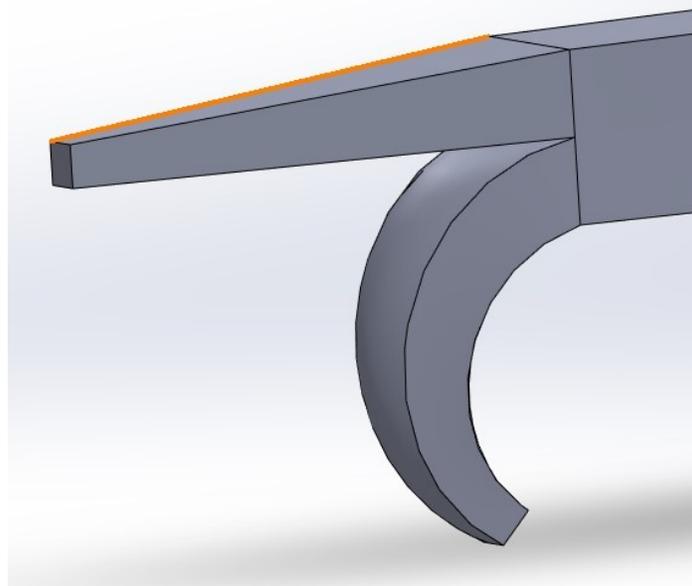


2. To fully define the arc and ensure that it will work smoothly with the loft, we need to use the **Pierce** relation. To do this, select the endpoint of the arc closest to the poker and the bottom edge of the end face of the poker. This should allow you to select the **Pierce** constraint.
3. Now we need to sketch the end profile of our hook loft. To do this, we need to make a new plane at the end of our guide curve. To do this, select the **Plane** option under **Reference Geometry** and use the arc as your first reference, making the plane perpendicular to it. To fully define the plane this way, you need another reference, so select the endpoint of the arc to be coincident.
4. On this new plane, sketch a 0.25 inch square. To fully define it, you will need to select the arc and the midpoint of the edge of the square that is closest to the poker and add another **Pierce** constraint.

5. Begin another sketch on the **Right Plane** and draw another arc. Give it a radius of 1.20 inches and add **Pierce** constraints at either end, opposite of the first arc.



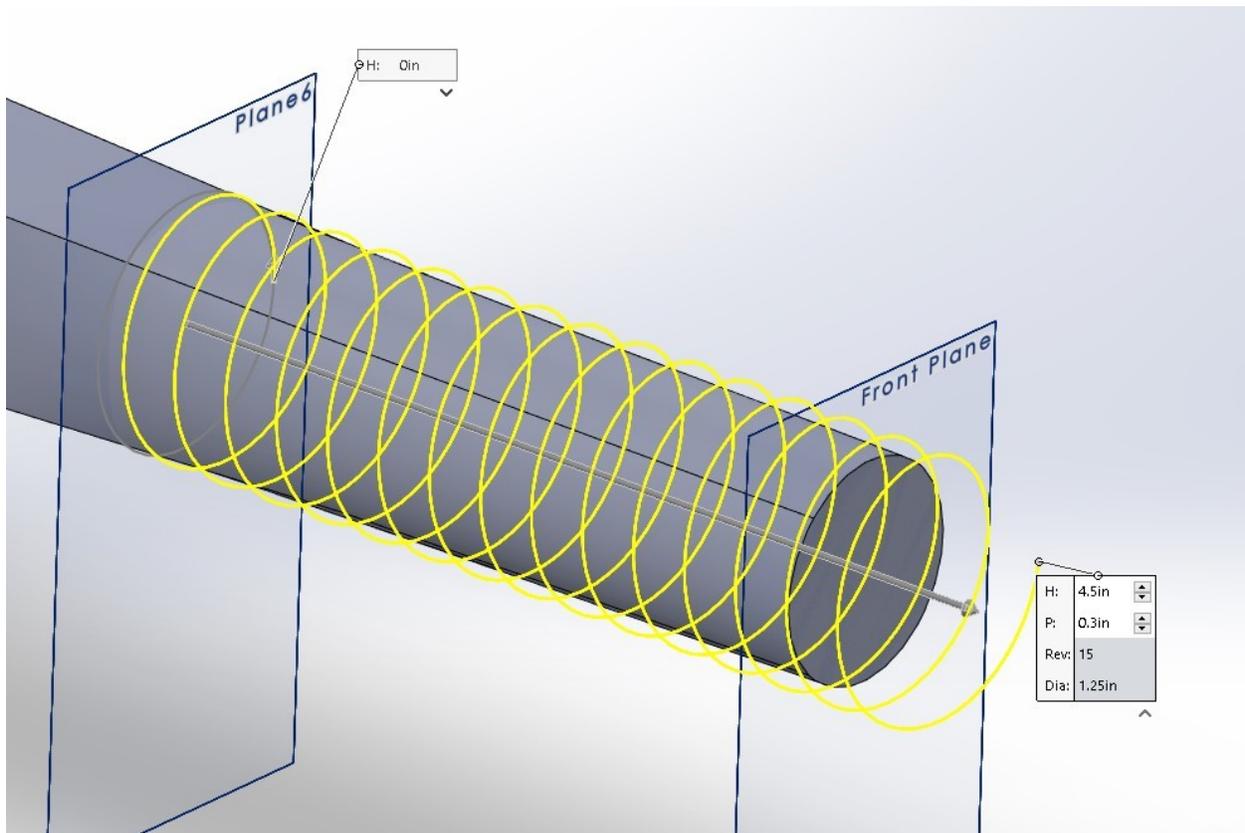
6. Finally, create a loft from the remaining face of the poker shaft to the end profile. At first, this will likely not give you a valid path, so click on the **Guide Curves** box and select the two arcs. This should give you a valid path, and you can click the green check to complete the loft.



7) Creating a Swept Helical Grip

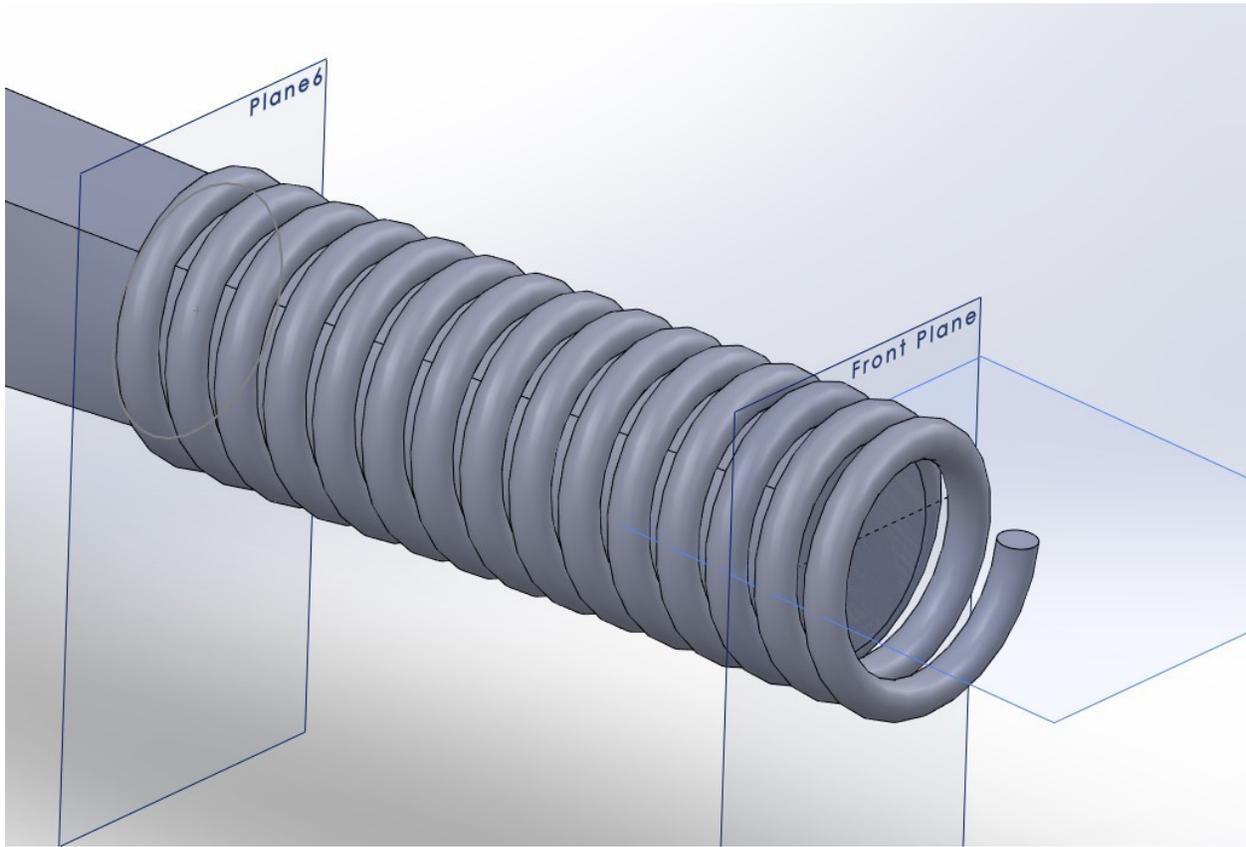
Now, we will move on from lofts to create a sweep. A sweep creates a 3D object by moving a profile along a path. We will use this in conjunction with the helix tool to create a coiled wire grip for the poker. Before beginning this step, you may find it useful to go to the design tree and hide the planes you used for the lofts, to keep your view from becoming too cluttered.

1. First, select the **Front Plane** and create a new plane offset 4.0 inches from it, inside the poker.
2. On this new plane, sketch a circle centered at the origin with a diameter of 1.25 inches.
3. Now, select the **Helix and Spiral** tool, either from the **Curves** dropdown next to the **Reference Geometry** dropdown, or by using the search bar to find it. Once you have selected the tool, choose the circle you just sketched. In the **Property Manager** for the helix tool, make sure your helix is defined by **Height and Pitch** using the dropdown menu, and set a height of 4.5 inches with a pitch of 0.30 inches. Click the green check to create the helix that will be the path for the sweep.



4. Now that we have a path, we need a profile, and to sketch the profile, we need a plane. Create a new plane using the helix as your first reference, then use the endpoint of the helix as a second reference to fully define the plane.
5. On the new plane, sketch a 0.20 inch diameter circle, and use the **Pierce** constraint to fully define it with the center of the circle intersecting the helix.

6. Finally, select the **Swept Boss/Base** tool, choose the circle as the profile and the helix as the path, and click the green check to complete the helix.



8) Finished!

Congratulations, you've completed the tutorial! Hopefully, you are now familiar with the loft and sweep features of Solidworks. If you want your poker to look more attractive, or function differently, you can play adding guide curves to the earlier lofts, changing the location of planes, moving the green dots in lofts, using fillets to smooth out the finished part, or applying a different material. **Note:** if you want to use more than one material in a single part file, you can uncheck the "Merge result" option from within most **BoSS** (i.e., additive) features.

