

# Creating a Square Spring Feature in SolidWorks

By Rishi Kumar Patel

1. Start a new part file and select Insert › Curve › Helix / Spiral to open the Helix / Spiral PropertyManager. Select the Top Plane as the plane to sketch the helix.
2. With the center at the origin, draw a circle of 50 diameter and exit the sketching environment.
3. Enter 50 in the Pitch edit box, 8 in the Revolutions edit box, and 0 in the Start Angle edit box of the Parameter rollout. Change the current view to the isometric view; the preview of the helix is displayed, as shown in Figure 1.
4. Choose the Ok button from the Helix / Spiral PropertyManager to accept the parameters of the helical curve.
5. Now, choose the Sketch button from the Command Manager; you are prompted to select the sketching plane. Select the Right plane from the Feature Manager Design Tree.
6. The view is now oriented normal to the screen. Sketch the profile, as shown in the Figure 2. Exit the sketcher after sketching the profile.

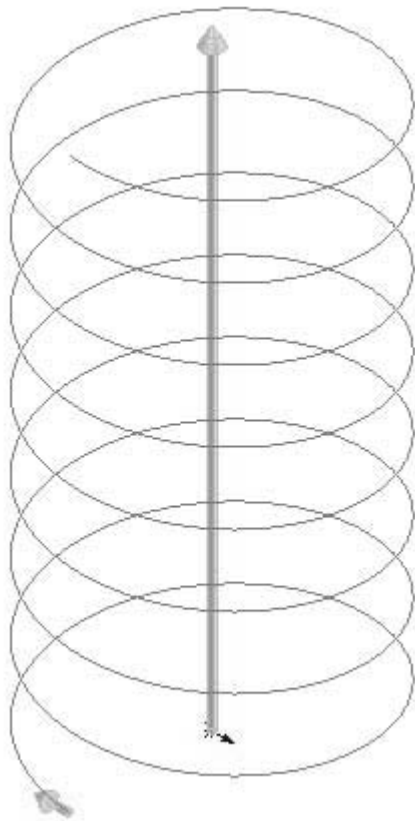


Figure 1 - Drawing a Helical curve

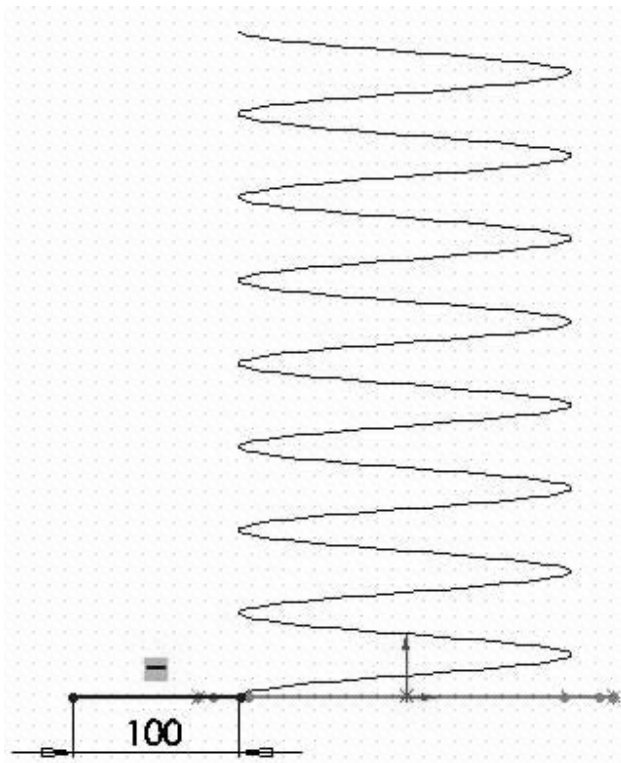


Figure 2 - Profile to sweep

6. Select Insert › Surface › Sweep to open the Surface – Sweep PropertyManager. You are prompted to select the sweep profile. Select the Line as the profile.
7. Next, you are prompted to select the sweep path. Select the Helix as the path.
8. The preview of the swept surface is displayed, as shown in the Figure 3. Choose the Ok button to confirm the creation of the swept surface.
9. Next, choose the Sketch button from the Command Manager; you are prompted to select the sketching plane. Select the Top plane from the Feature Manager Design Tree.

10. Sketch the profile, as shown in the Figure 4, and then exit the sketching environment.

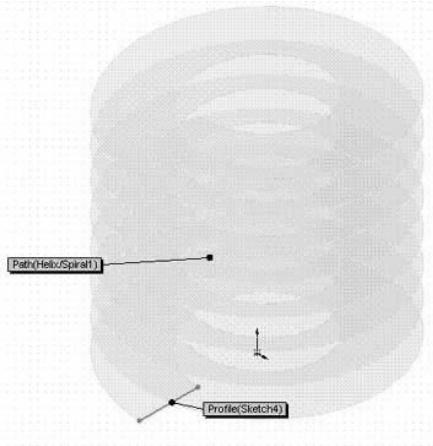


Figure 3 - Preview of the swept surface

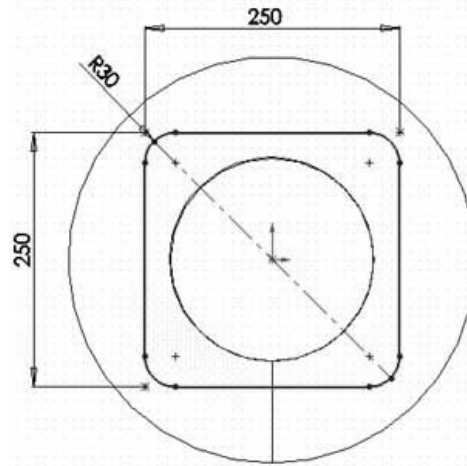


Figure 4 - Dimensions of the profile to be extruded

11. Select Insert › Surface › extrude to open the Surface – Extrude PropertyManager; the filleted rectangle is automatically selected as the profile.

12. The preview of the extruded surface is displayed. Select the drag handle to specify the depth of extrusion. Make sure that the extruded surface exceeds the limits of the swept surface on both the directions, as shown in the Figure 5.

13. Choose the Ok button to confirm the creation of the extruded surface.

14. Right click on the Helix / Spiral feature in the Feature Manager Design Tree and choose the Hide option from the shortcut menu to hide the helical curve.

15. Select Insert › Curve › Split Line to invoke the Split Line PropertyManager.

16. Select the Intersection radio button from the Type of Split rollout. Select the swept surface as the tool entity.

17. Click on the Faces/Bodies to Split area of the Selections rollout. Now, press and hold the CTRL key down and select all faces of the extruded surface. Alternatively, you can also select the Surface-Extrude feature from the Feature Manager Design Tree. Make sure the faces of the extruded surface are selected, as shown in the Figure 6.

18. Choose the Ok button to confirm the selection. Split lines are created at the intersection of the surfaces.

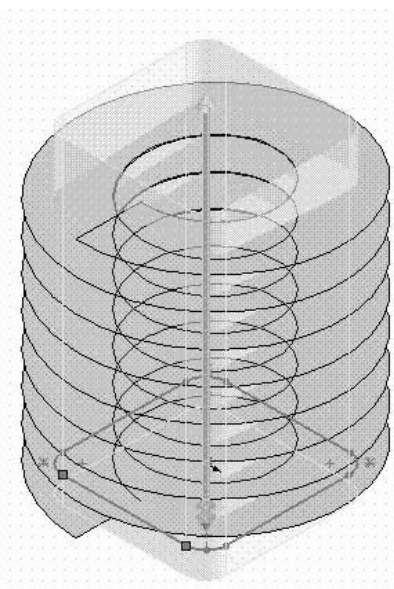


Figure 5 - Preview of the Extruded Surface

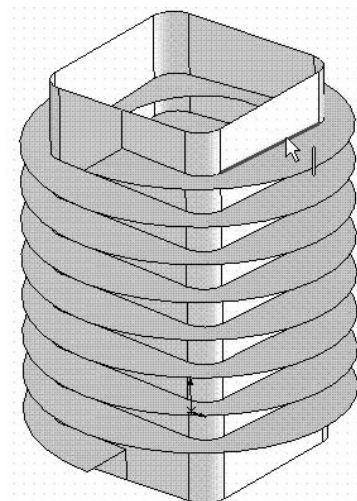


Figure 6 - Selections for creating Split Lines

20. Move the cursor near the split line; you will notice that the split lines are highlighted separately. This is because split lines are created as separate entities, as shown in the Figure 7.

21. Next, you need to change the split line into a composite curve. To do so, select Insert > Curve > Composite from the menu bar. The Composite Curve Property Manager is invoked. One by one, select all the split lines from the drawing area to make them composite, refer to Figure 8.

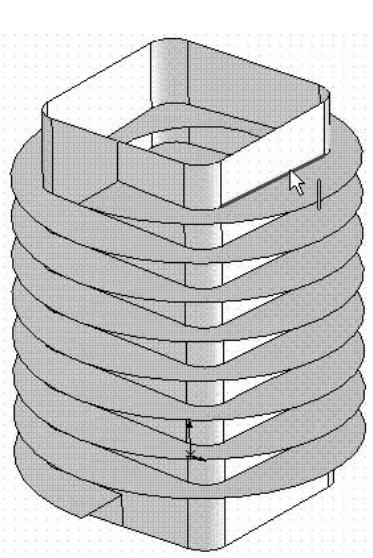


Figure 7 - Preview of the Split Line

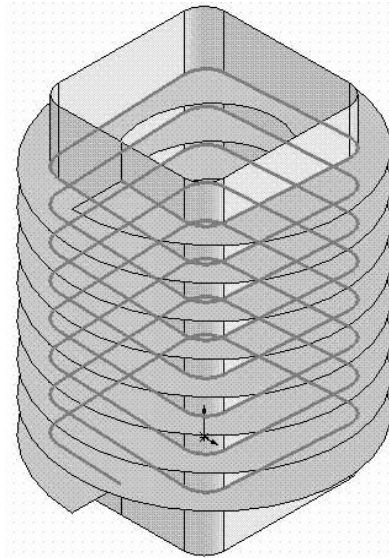


Figure 8 - Selections for creating Composite Curve

23. Choose the Ok button to confirm the selection; the composite curve is created.

24. Hide all the other features, except the composite curve. The resultant composite curve is as shown in Figure 9.

25. Select, Insert > Reference Geometry > Plane to invoke the Plane Property Manager.

26. Select the composite curve. The plane will be created normal to this curve.

27. Specify the lower endpoint of the composite curve as the location at which the plane will be normal to it; the preview of the reference plane is shown in Figure 10.

28. Choose the Ok button from the Plane PropertyManager to confirm the plane creation.

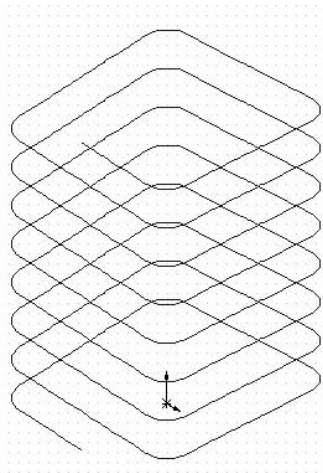


Figure 9 - The Composite Curve

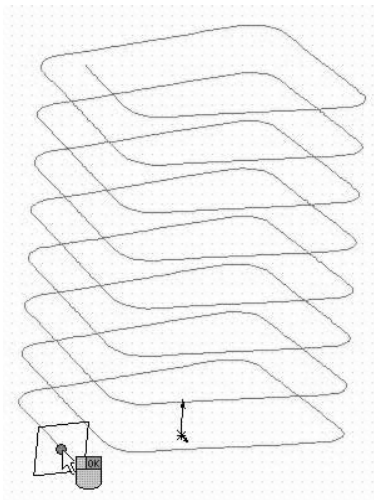


Figure 10 - Preview of the Reference Plane

29. Next, choose the Sketch button from the Command Manager; you are prompted to select the sketching plane. Select Plane1 as the reference plane from the Feature Manager Design Tree as the sketching plane.

30. Sketch the profile, as shown in the Figure 11. Exit the sketcher after sketching the profile.

31. Select Insert › Boss/Base › Sweep to open the Sweep PropertyManager. You are now prompted to select the sweep profile. Select the circle as the profile, if it is not automatically selected.

32. Select the composite curve as the sweep path.

33. The preview of the swept feature is displayed, as shown in the Figure 12. Choose the Ok button to confirm the creation of the swept feature.

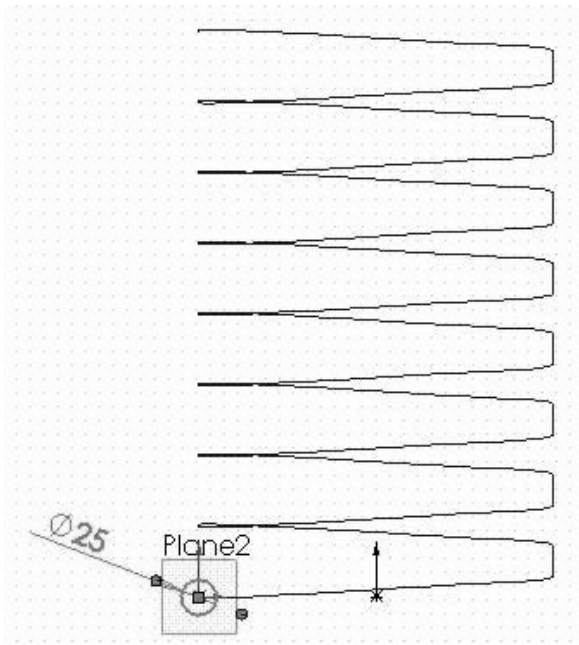


Figure 11 - Profile to sweep

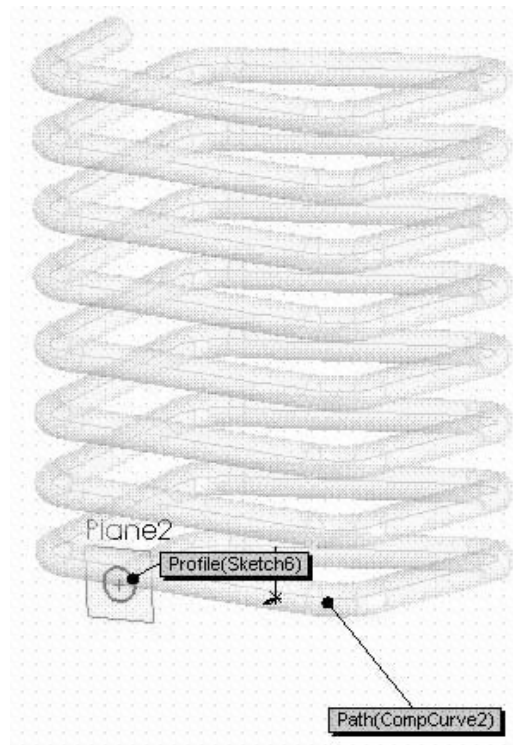


Figure 12 - Preview of the swept feature

34. The square spring feature is shown in Figure 13

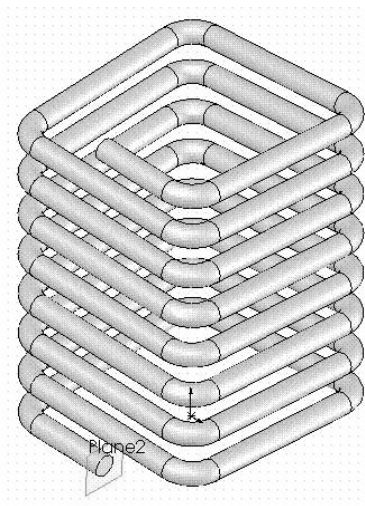


Figure 13 - Square spring feature