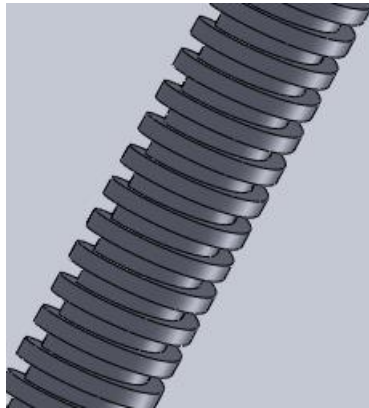


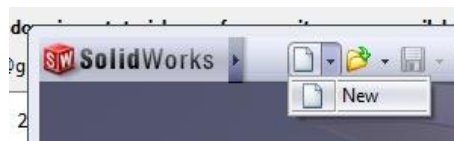
SWEPT CUT FEATURE IN SOLIDWORKS



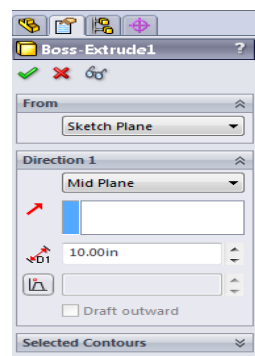
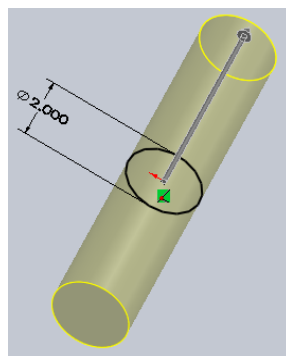
This is one of the basic features used in part modeling in solidworks. You will find its use in modeling threads in nuts and bolts. This feature is similar to swept boss/base tool. To use this tool you have to create a profile and path to sweep along. This tool removes materials unlike the Swept boss/base feature which adds material to the object / part.

Let's Begin to Design

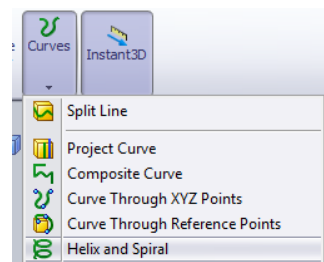
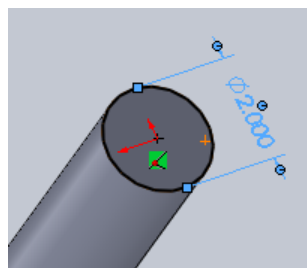
Select a New part file



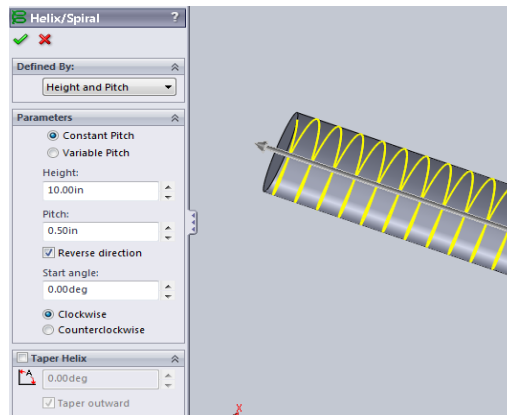
Extrude a cylinder of 10inch in length and 2 inch in diameter



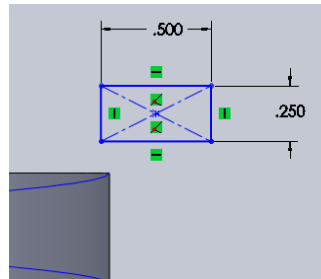
Now draw a helix wrapping the cylinder. To do this first go to Curves -> Helix and select the back face of the cylinder to sketch and sketch a circle of 2 inch in diameter and exit the sketch by click exit sketch button



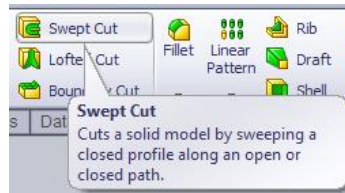
In the helix options select height and Pitch then enter 10in in height and .5in in pitch. Set start angle as zero and click OK.



Select the top plane and sketch a centered rectangle of .5in width and .25 length as shown.



Exit the sketch by clicking exit sketch button . Go to features tab, then click on swept cut button



Select the profile as newly drawn sketch and the path to sweep to be the helix as shown and click ok.

The final part will be like this

