

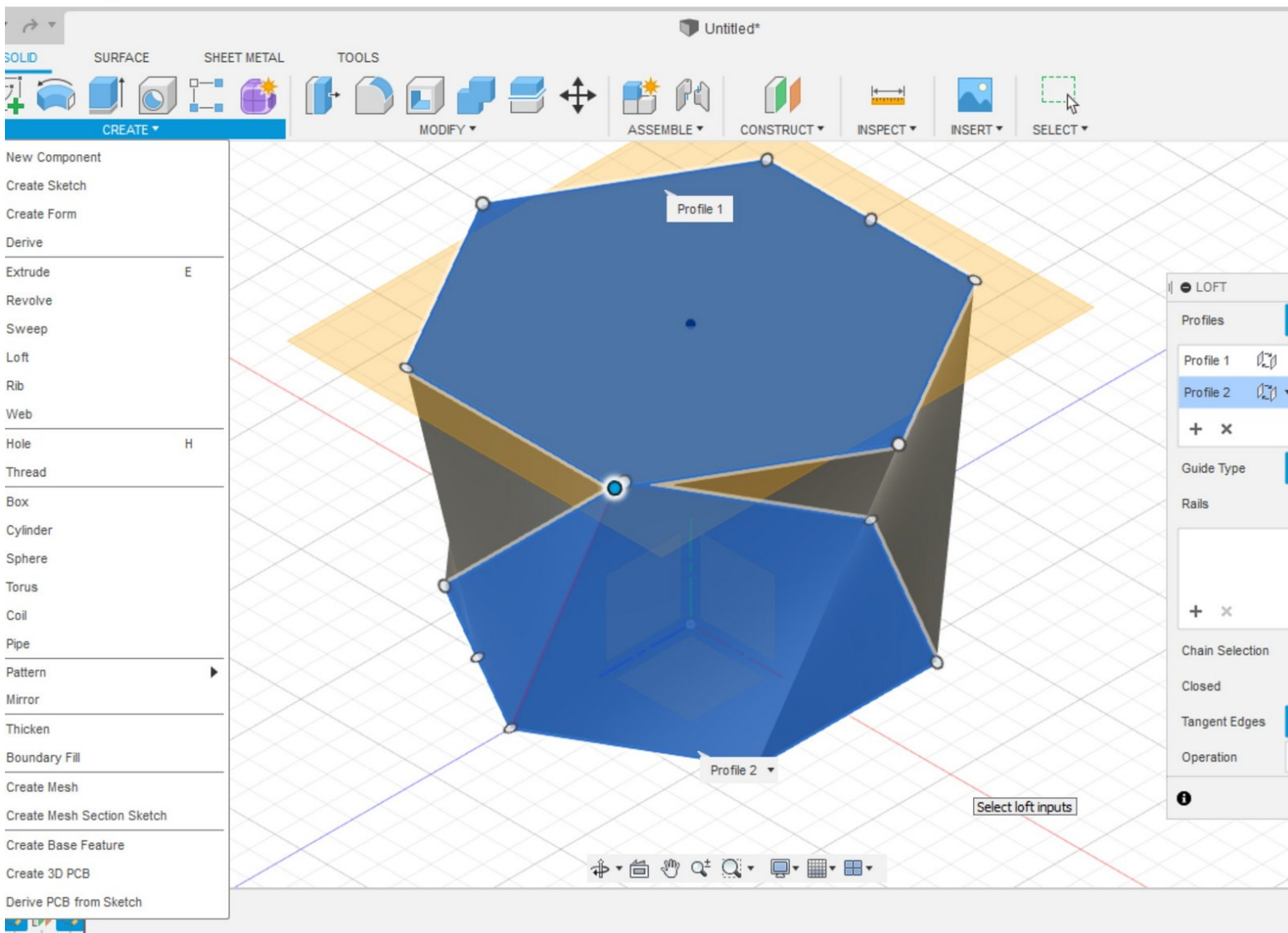


Intermediate Solid Features in Fusion 360

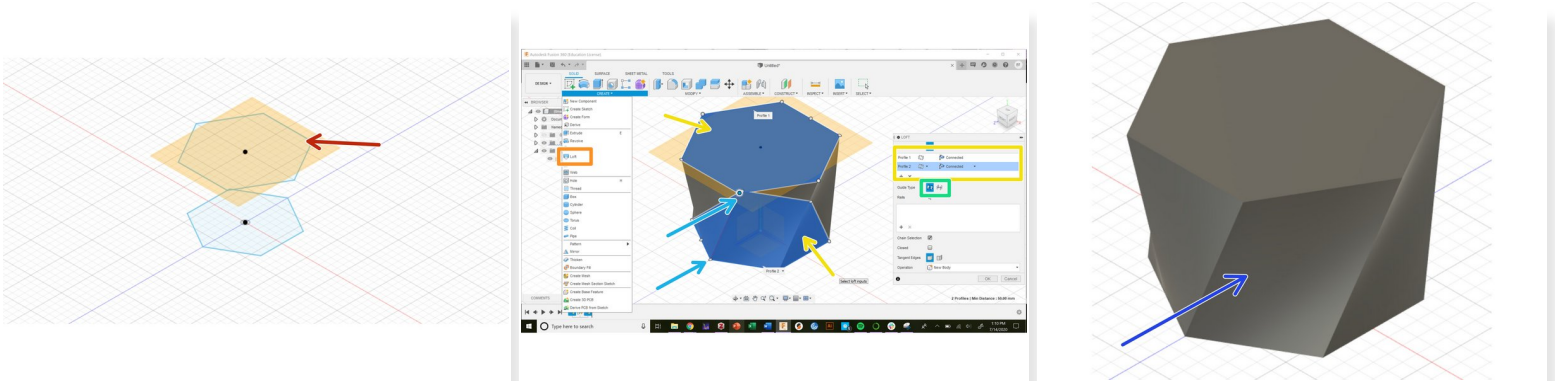
This guide covers a few of the more advanced solid features in Fusion 360

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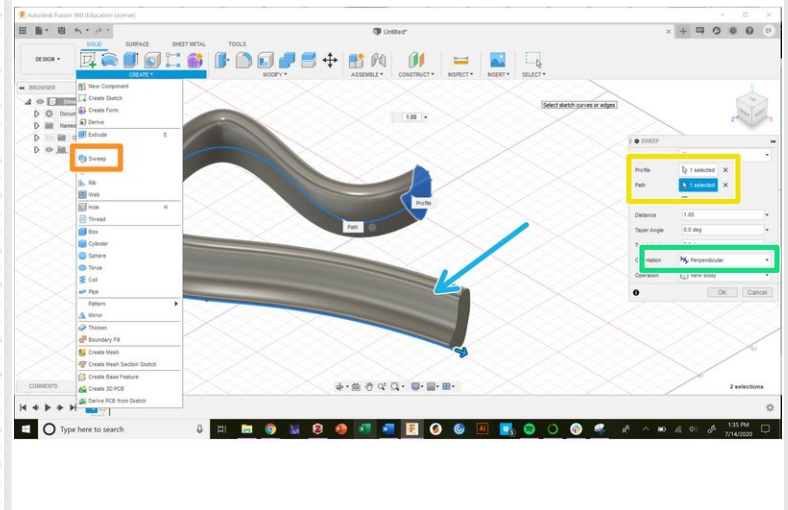
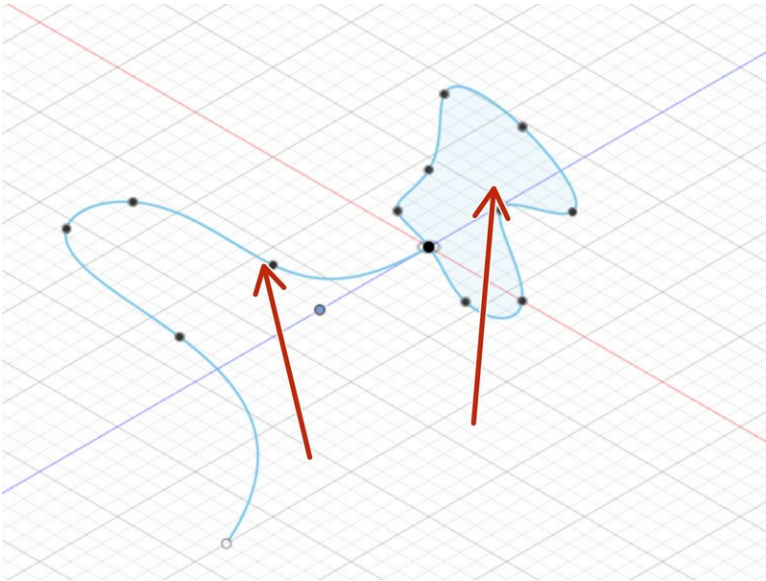


Step 1 — Lofts



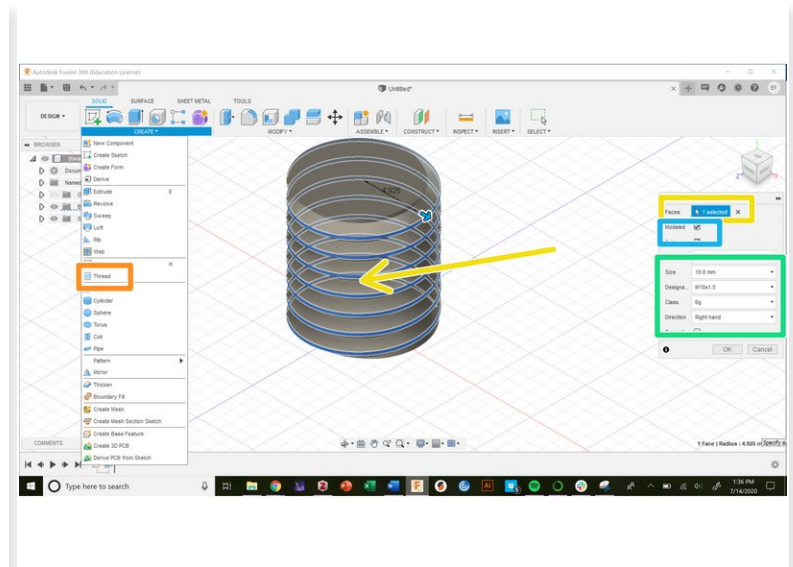
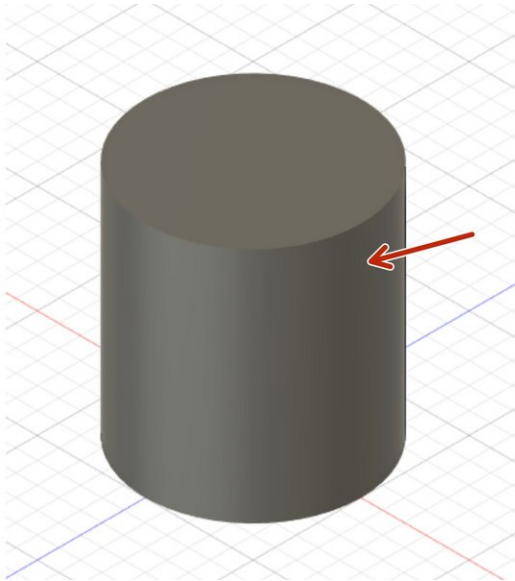
- i** To perform a loft, we need sketch profiles on different planes
- Before making this loft, we prepared a sketch on a construction plane
 - The loft tool is under the drop-down menu in the "Create" section of the tool bar
 - First we select the two profiles to loft together
 - Then we select the guide type. In this example, we selected guide rails to constrain the volume created between the two profiles.
 - Selecting "guide rails" allows us to choose the points connecting the rails and move them.
 - This loft produces the solid indicated with the dark blue arrow.

Step 2 — Sweeps



- i To create a sweep, we need a profile and a path
 - A sweep creates a pipe along a path with a cross section given by a sketch profile
 - For this example, we created an arbitrary shaped profile and path
 - The sweep tool is in the drop down "Create" menu
 - Select the profile to sweep and the path to follow
 - The orientation determines whether the profile lies perpendicular to the path or parallel to itself. This example shows a perpendicular sweep
 - The sweep tool creates the shape indicated by the blue arrow

Step 3 — Threading



i Creating threading in Fusion 360 requires a solid body compatible with threads, i.e. cylindrical bodies or holes.

- This example starts with a cylinder made by extruding a circle 10 mm in diameter
- The drop down "Create" menu houses the threading tool
- Select the surface on which to create threads
- Denote the threading specifications
- Choosing to model the threads actually cuts them into the body. Leaving the "Modeled" box unchecked creates cosmetic threads.