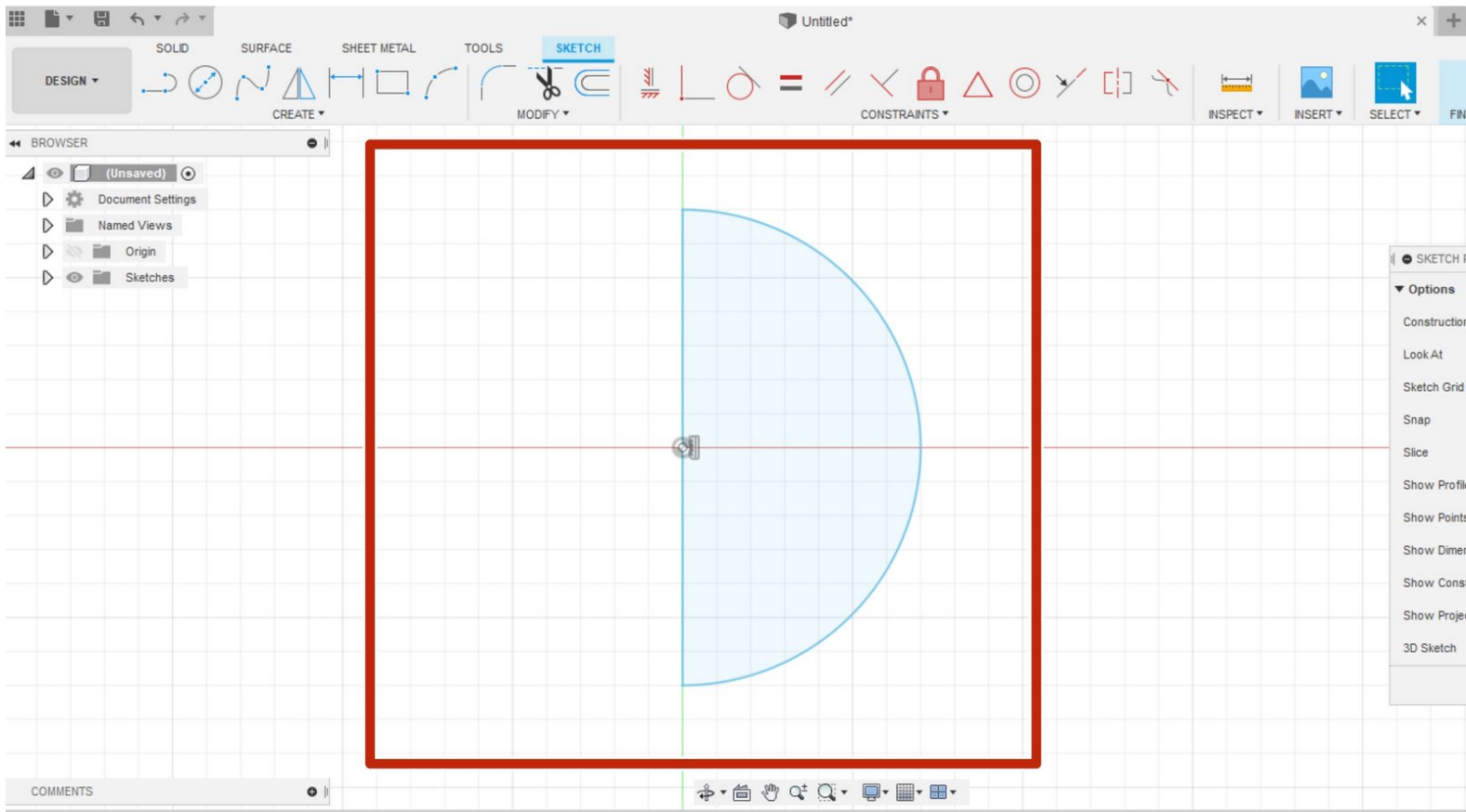




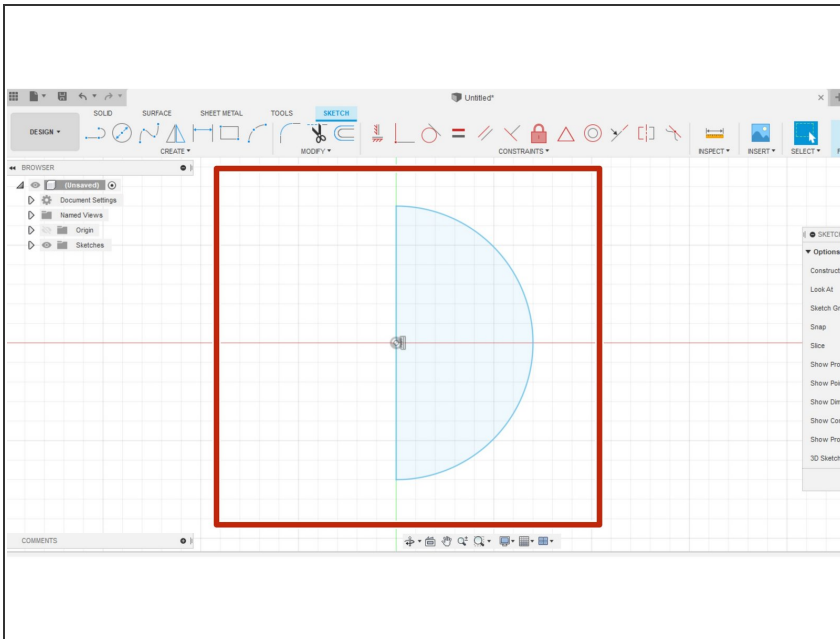
Introduction to Solid Features in Autodesk Fusion

This guide gives an overview of some of the most common tools for creating 3D solid bodies in Fusion 360

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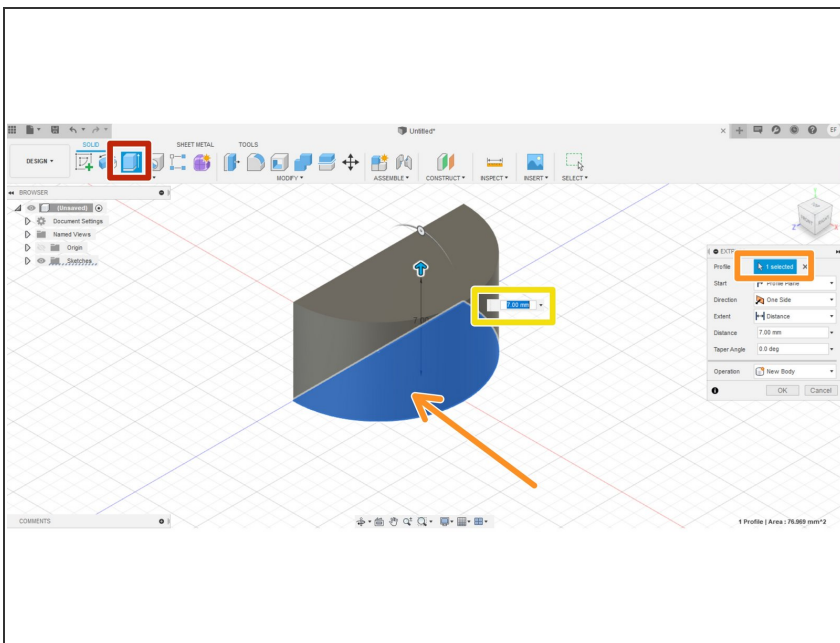


Step 1 — Introduction



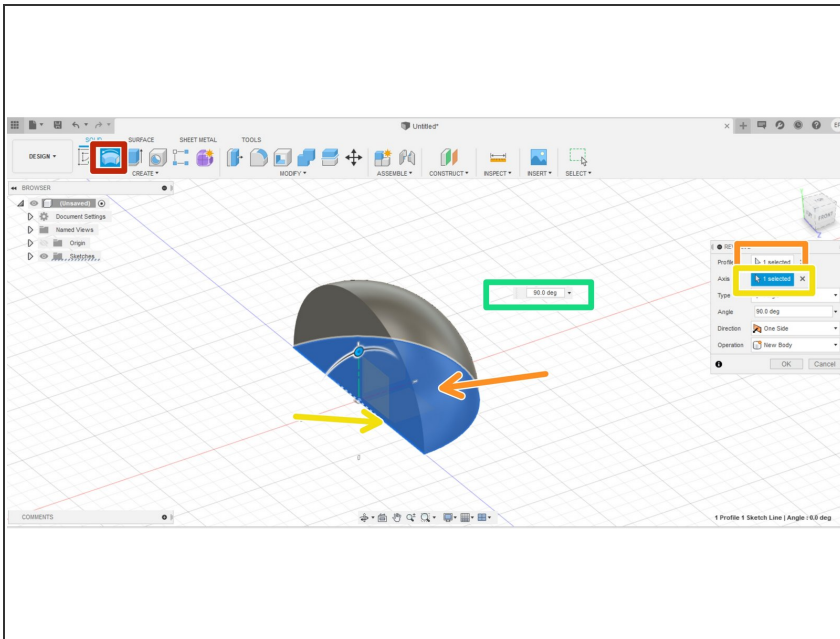
- In general, CAD software uses the following process to create 3D solids: **create a sketch then manipulate that sketch into a solid body**
- This guide provides an overview of some of the more common tools for turning sketches into solid bodies
- The figure in the red box is an example of a sketch. To learn more about creating sketches, refer to [Introduction to Sketches in Fusion 360](#)

Step 2 — Extrude



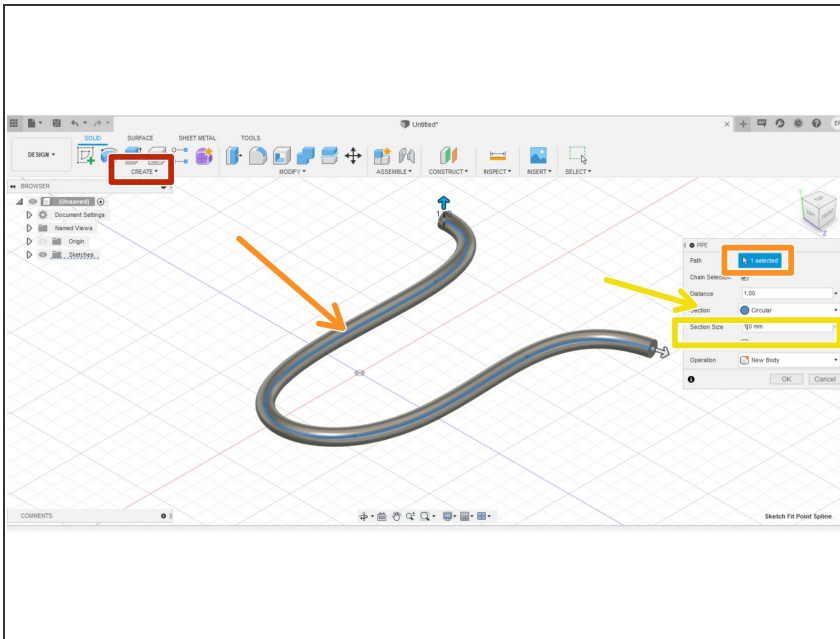
- An extrusion essentially pulls a 2D profile into a 3D body where all cross sections in the direction of the extrusion match the original profile.
- The extrusion tool icon is found in the "Create" section of the Toolbar
- Creating an extrusion requires a closed profile.
- Specify the height of the extrusion. Extrusions with negative heights can cut through other solid bodies.

Step 3 — Revolve



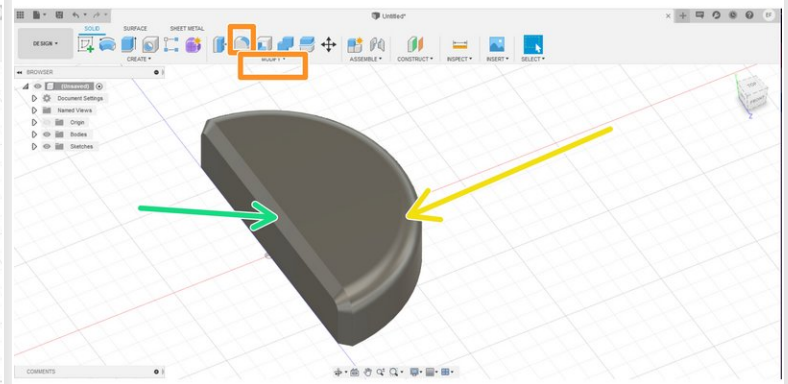
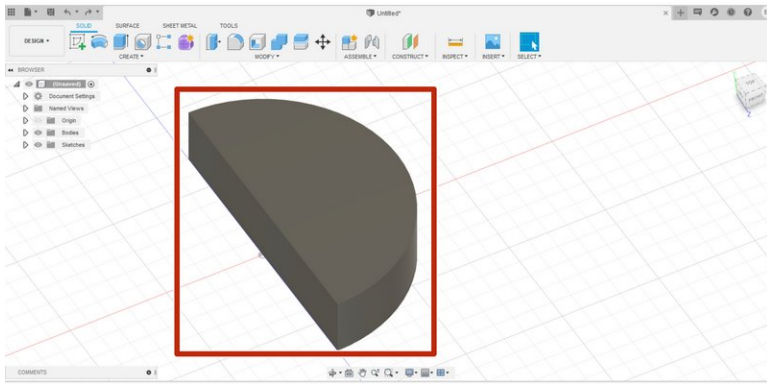
- The revolve tool creates a 3D body by rotating a closed profile about an axis.
- The icon for the revolve tool is found in the "Create" section of the tool bar
- The revolve tool requires a closed profile, so first we select the closed section of the sketch.
- In this example, we selected the line in the semicircle as the axis about which to revolve.
- After selecting the profile and axis, we finish constraining the revolve by specifying the degrees of the rotation.

Step 4 — Pipe and Sweep



- The pipe tool creates a solid by sweeping a basic shape through a specified path. The pipe tool is just a special type of a sweep; in a sweep tool rather than using a basic shape, the user creates a profile in a sketch.
- Both the pipe and sweep tool are found in the drop down "Create" menu.
- Using the pipe tool requires an open path from a sketch curve
- The path tool also allows for the selection of shape used and specification of the shape's relevant dimensions.

Step 5 — Modifying a Solid Body



- Fusion 360 provides several tools for modifying a solid body. Fillet and chamfer comprise two of the most commonly used tools for modifying solid bodies. They either smooth an edge into a curve or cut it into a slope respectively.
- The figure indicated with a red box consists of an extruded semicircle.
- The fillet tool is located in the Toolbar under the "Modify" section, while the chamfer tool is under the "Modify" drop down menu
- The fillet tool, applied to the outer curved edge indicated with the yellow arrow, smoothed the edge into a curve.
- The chamfer tool, applied to the top and side edges on the flat face indicated with the green arrow, cut the edge into a slope

Step 6 — Concluding Thoughts



- This guide provides an explanation of some of the most commonly used tools for making solid bodies in Fusion 360.
- This guide is by no means exhaustive, but it will give you a decent basis on which to start making your first designs
- We recommend you practice these tools by trying to recreate objects around you in Fusion 360.
- If you want some more practice, please head over to the [Make a Planter in Fusion 360](#) guide for your first Fusion 360 project.