



# 3D Modeling and Design Glossary - Advanced

**Body:** a modeling tool. You can use separate bodies to add or remove geometry to achieve the final shape of your design. For example, you may create a coffee mug by making a body for the cup and another body for the handle, and then join them together. Bodies can be compared to shapes in Tinkercad - for example: where shapes can be grouped together into parts, bodies can be combined into components. Fusion 360 supports three types of bodies: solid/surface bodies, sculpt bodies, and mesh bodies.

**Combine:** to unite two bodies or components into a single component. Combining can be compared to grouping in Tinkercad.

**Components:** "containers" for other modeling entities, which can include sketches, construction geometry, bodies, and even other components. Components represent real-world parts; something that is manufactured and that may be assembled to one another. If you already have a plan for what you are making, it is best to begin your design with a component and then construct the bodies within it.

**Cross section:** to cut an object off at right angles to an axis. In Fusion 360, you might do this to analyze your design.

**Freeform:** See "Sculpt."

**Joint:** used to assemble components and create mechanical relationships between components, including defining movement. Learn more about different types of joints here.

**Loft:** Transitioning from one shape to a different shape over a specified distance. For example: from a rectangle to a circle or like the hull of a ship.

**Mesh:** is a collection of vertices, edges, and faces that can describe the shape of a 3D object. A vertex is a single point. An edge is a straight line segment connecting two vertices. A face is a flat surface enclosed by edges. While Fusion 360 creates solid/surface bodies, it supports the modeling of mesh bodies also.

**Parametric modeling:** a tool that lets you describe what you want to make numerically and allows designers to change an entire object by changing only one of its attributes through assigning constraints - or parameters. For example, a parameter could describe an endcap on a pipe. When you create the model, the endcap is automatically placed at the end of the pipe, no matter what changes you make to the pipe's length.

**Path:** a line that appears when you sketch in Fusion 360. A path is made up of a series of points called "anchor points" and line segments between these points. The anchor points on the either end of a path have "control handles" and these can be used to control the direction of the curved path.

**Profile:** a 2D sketch that can be extruded to make a 3D object.



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**Render:** to create a final image of a model that shows all of the surface properties that have been applied to an object. This process involves adding all colors, shading, and other elements, such as the physical appearance of materials, that add realism.

**Revolve:** to sweep a 2D object around an axis, resulting in a three-dimensional shape.

**Sculpt:** a modeling approach that creates organically shaped models using direct manipulation. You can start with one of five basic Freeform shapes to do this: Box, Quadball, Cylinder, Torus, and Sphere. In the Sculpt Workspace, you can rapidly explore forms by simply pressing and pulling on subdivided surfaces.

**Shell:** to remove material from a part interior, creating a hollow cavity with walls of a specified thickness.

**Simulate:** to examine and test a design through a computer-aided imitation of how it might function in the real world. These quick iterations or tests are meant to provide useful visual feedback in order to better understand and improve a design before it is actually fabricated.

**Slice:** to divide a solid object into two or more separate 3D objects.

**Sweep:** to extend a profile along a curvy line into a 3D object. It requires two sketches - one for the profile and one for the path.