Solids from sketches

The following commands in the Design workspace, in the Solid,>, Create panel, let you create a solid body from a closed sketch profile, open sketch curve, or planar face in Fusion 360.

You can use the following commands to create a solid body from a sketch:
- Extrude
- Revolve
- Sweep
- Loft
- Rib
- Web
- Emboss

Note: Some commands also appear in the Generative Design, Simulation, and Manufacture workspaces. The organization of commands differs in each workspace.
You can specify the type of operation you need for your design.

| Operation	Description
Join	Combines the new shape with an existing body.
Cut	Cuts an area out of an existing body.
Intersect	Creates a body at the intersection of an existing body and the new body.
New Body	Creates a new body in the active component.
New Component	Creates a new body in a new component.

Extrude

The Extrude command adds depth to sketch profiles or faces.
You select an extrusion type, select sketch profiles or faces to extrude, then specify the distance and taper angle. For the Thin Extrude type, specify the wall thickness.
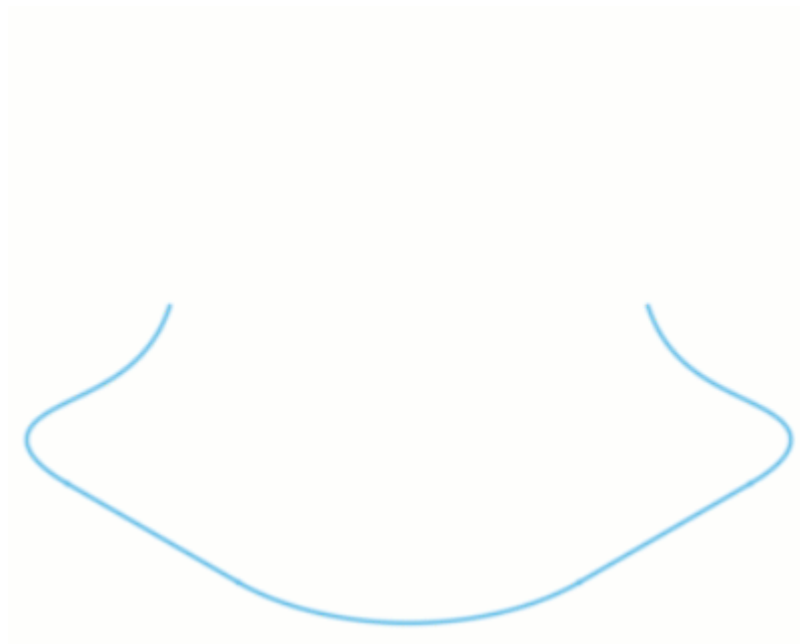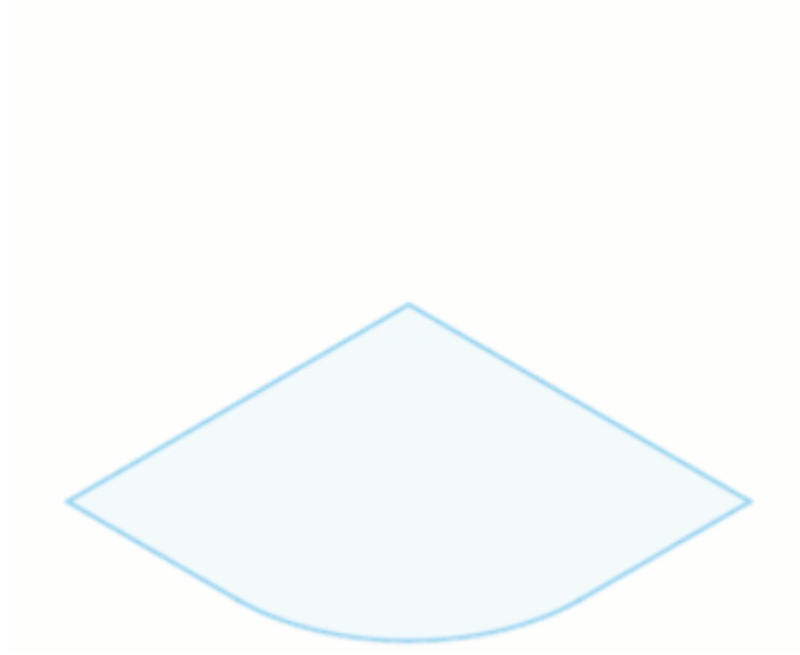
Revolve

The Revolve command revolves a sketch profile or planar face around an axis.
You select a profile or planar face, then select the axis to revolve around.
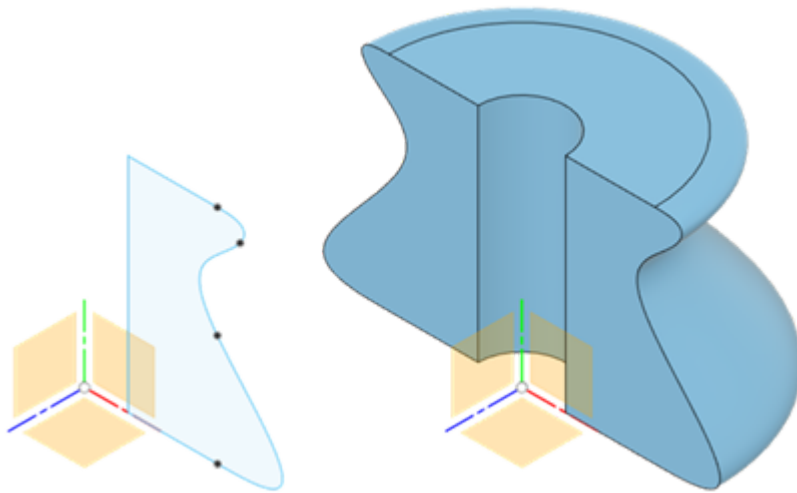
Sweep

The Sweep command sweeps a sketch profile or planar face along a path.
You select a profile, then select a path to sweep along. Optionally, select a guide rail or guide surface to control the scale and orientation fo the swept body.
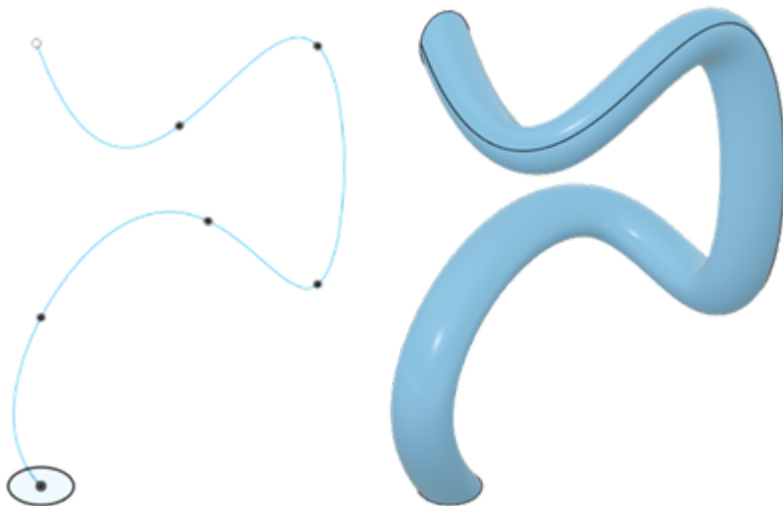
Loft

The Loft command creates a smooth shape that transitions between two or more sketch profiles of planar faces.
You select a series of profiles or planar faces to loft between. Optionally, select rails or a centerline to guide the shape.
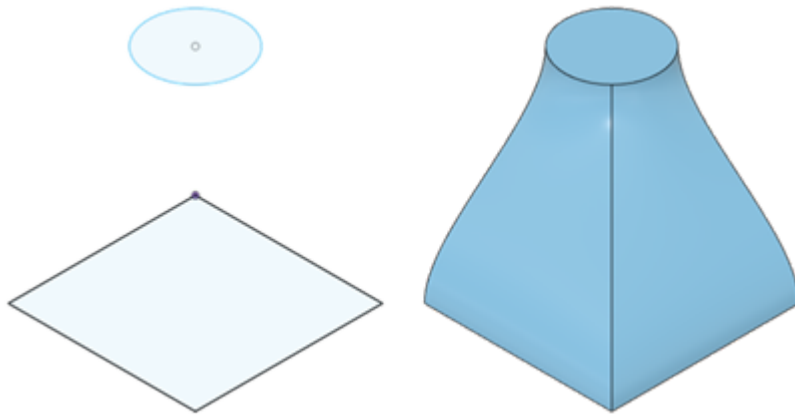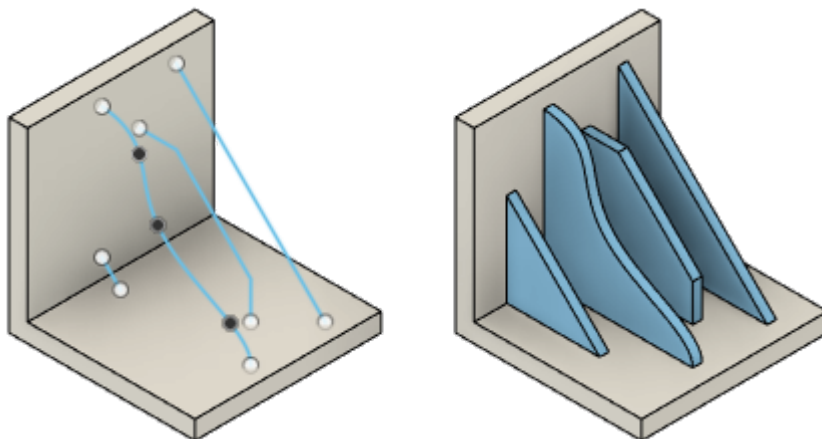
Rib

The Rib command creates a thin feature from an open sketch profile, extruded to the nearest faces on a solid body.
The rib feature is extruded in a direction parallel to the sketch plane.
You select an open sketch profile, then specify the thickness.
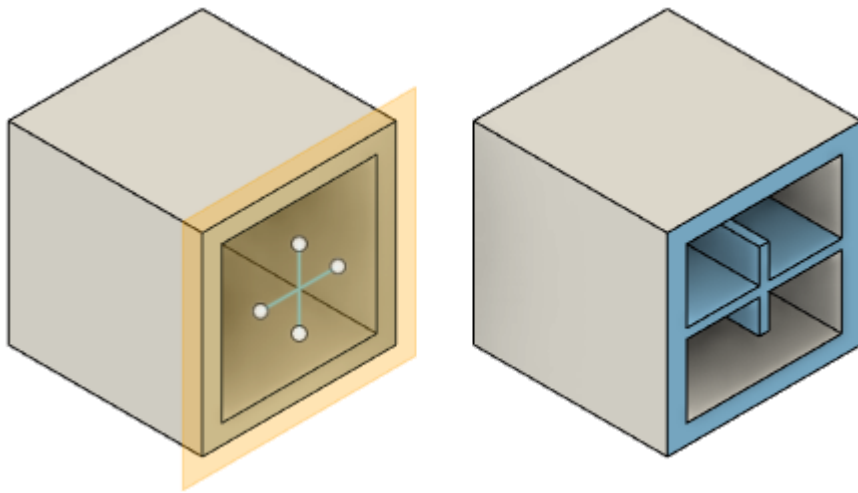
Web

The Web command creates a thin feature from an open sketch profile, extruded to the nearest faces on a solid body.
The web feature is extruded in a direction perpendicular to the sketch plane.
You select an open sketch profile, then specify the thickness.

Emboss

The Emboss command raises or recesses a sketch profile relative to faces on a solid body by a specified depth and direction.
You select sketch profiles, select faces on a solid body to emboss, then adjust depth, effect, and alignment.

Solid primitives

The following commands in the Design workspace, in the Solid >, Create panel, let you create a solid body from a primitive shape in Fusion 360.

You can use the following commands to create a solid body from a primitive shape:

- Box
- Cylinder
- Sphere
- Torus
- Coil
- Pipe
-

Note: Some commands also appear in the Generative Design, Simulation, and Manufacture workspaces. The organization of commands differs in each workspace.

You can specify the type of operation you need for your design.

| Operation	Description
Join	Combines the new body with an existing body.
Cut	Cuts an area out of an existing body.
Intersect	Creates a body at the intersection of an existing body and the new body.
New Body	Creates a new body in the active component.
New Component	Creates a new body in a new component.

Box

The Box command creates a solid body in the shape of a primitive box.
You select a plane or planar face, place the first corner, specify the length and width, then specify the height of the box.
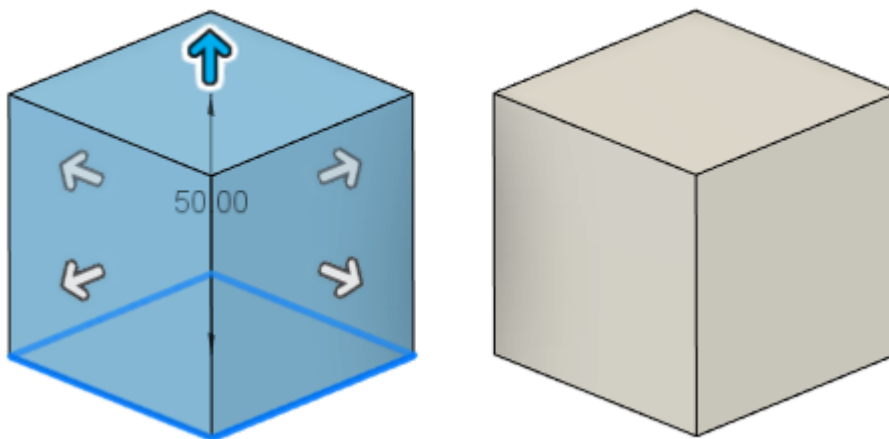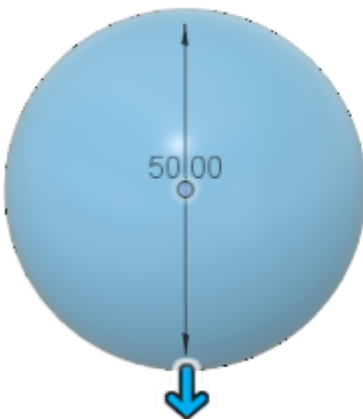
Cylinder

The Cylinder command creates a solid body in the shape of a primitive cylinder.
You select a plane or planar face, place the center point, specify the diameter, then specify the height of the cylinder
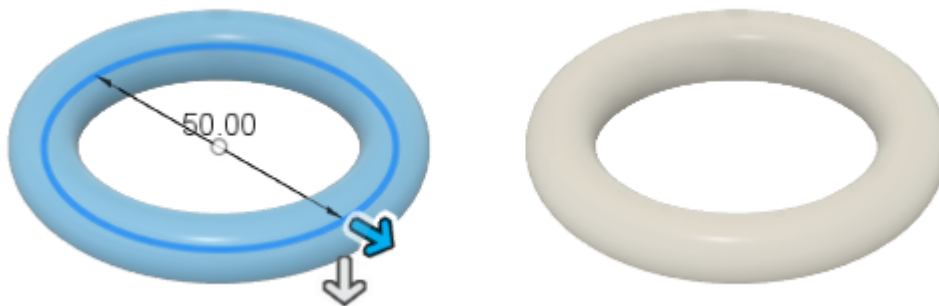
.
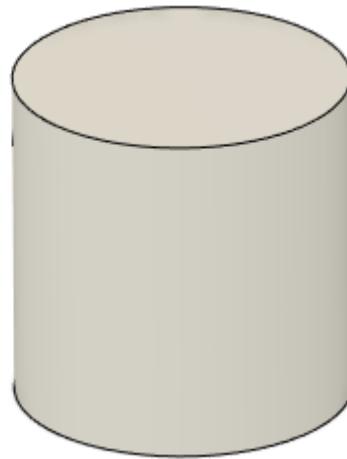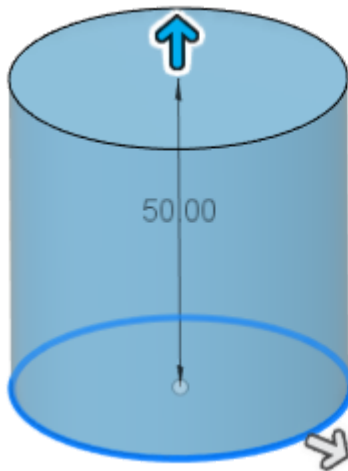
Sphere

The Sphere command creates a solid body in the shape of a primitive sphere.
You select a plane or planar face, place the center point, then specify the diameter of the sphere.

Torus

The Torus command creates a solid body in the shape of a primitive torus.

You select a plane or planar face, place the center point, specify the inner diameter, then specify the torus diameter. You can also select the position of the torus relative to the inner diameter.


Coil

The Coil command creates a solid body in the shape of a primitive coil.
You select a plane or planar face, place the center point, specify the diameter, then adjust the coil settings. You can adjust the coil type, rotation, diameter, number of revolutions, height, angle, and section shape.
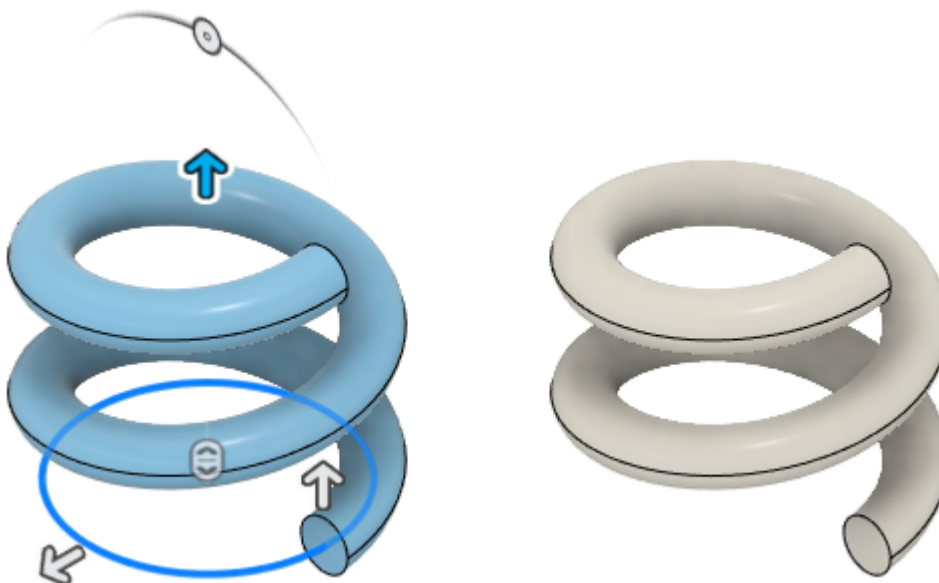
Pipe

The Pipe command creates a solid body in the shape of a primitive pipe that follows a path. You select a path for the pipe to follow, then specify the distance, section shape, and section size. You can also choose to follow the pipe.
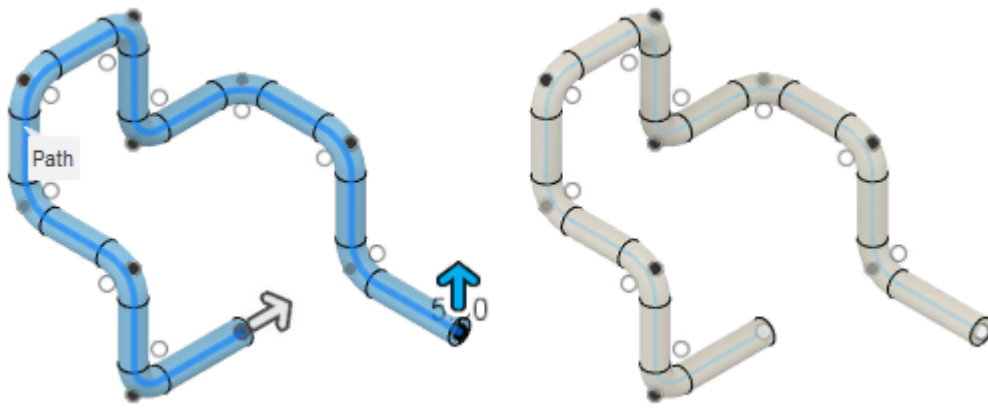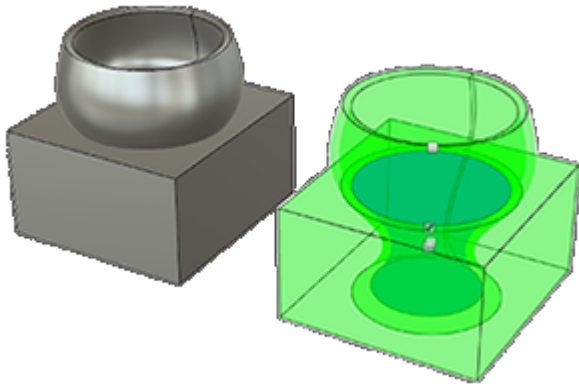
Solids from surfaces

You can create solid bodies from surface bodies by adding thickness to surfaces or by using cells to generate solid bodies in Fusion 360.

The Thicken tool offsets one or more faces, surfaces, or quilts, and adds sides to create a solid. All surfaces are offset at an equal distance from the originals.

Boundary Fill creates, joins, or removes volumes using bounding volumes formed by tool selections. Select solids, surfaces, or workplaces as tools to form volumes (or cells). These cells can be used to cut existing solids or to join existing solids or create a new solid.

Pages in this section

Solids from meshes

The following commands in the Design > Solid > Create panel let you convert a solid body to a mesh body or convert a mesh body to a solid body in Fusion 360.

- Mesh to BRep
- BRep to Mesh
-

Holes and threads

The tools in the Design > Solid > Create panel let you create a simple, counterbore, or countersink drilled hole and add threads to cylindrical solid geometry in Fusion 360.

You can use the following commands to create holes and threads in a solid body:
- Hole
- Thread

Note: Some commands also appear in the Generative Design, Simulation, and Manufacture workspaces. The organization of commands differs in each workspace.
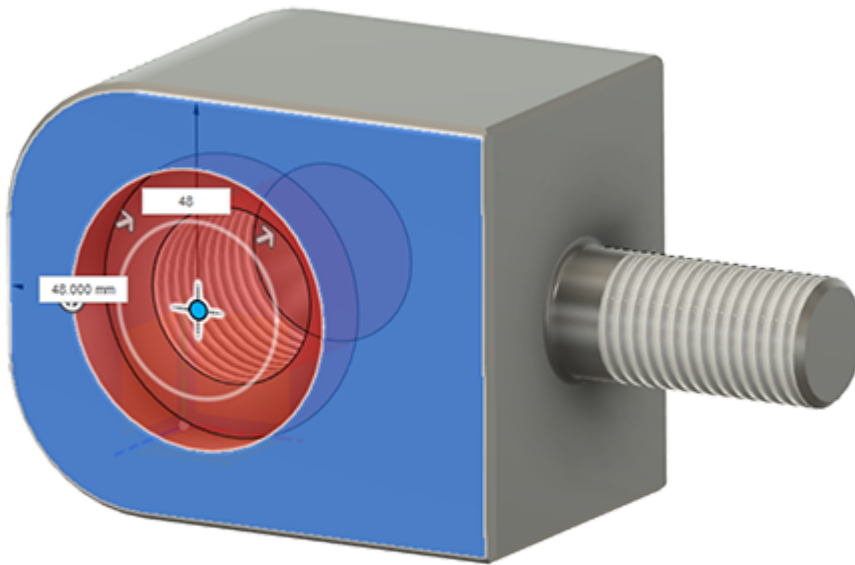
You can specify:

- The placement and extents to locate a hole
- The hole type, tap type, drill point, depth, diameter, and angle settings to customize a hole
- The length, offset, type, size, designation, class, and size of a thread
- Whether threads are cosmetic or accurately modeled

Hole

The Hole command defines a simple, counterbore, or countersink drilled a hole in a solid body.

Thread

The Thread command adds internal or external threads to cylindrical solid geometry.

Mirrors and patterns

The Mirror and Pattern commands let you create an organized set of identical faces, features, bodies, or components in Fusion 360.
You can use the following commands to mirror or pattern objects in your design:

- Mirror ⚠
- Rectangular Pattern ⬚
- Circular Pattern ⬚
- The pattern on Path ⬚

Mirror

The Mirror command creates a copy of selected faces, bodies, features, or components, mirrored across a plane.
The result is an identical copy mirrored on the opposite side of the plane. If the mirrored body intersects with the original body, you can choose to join them into a single body.
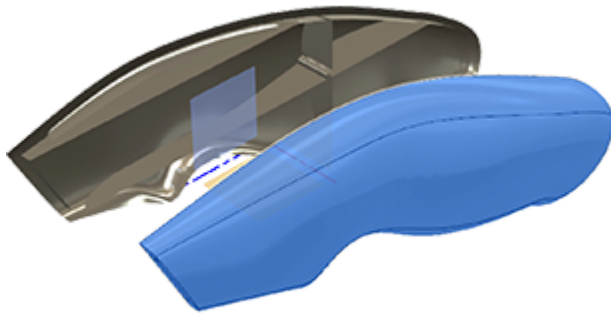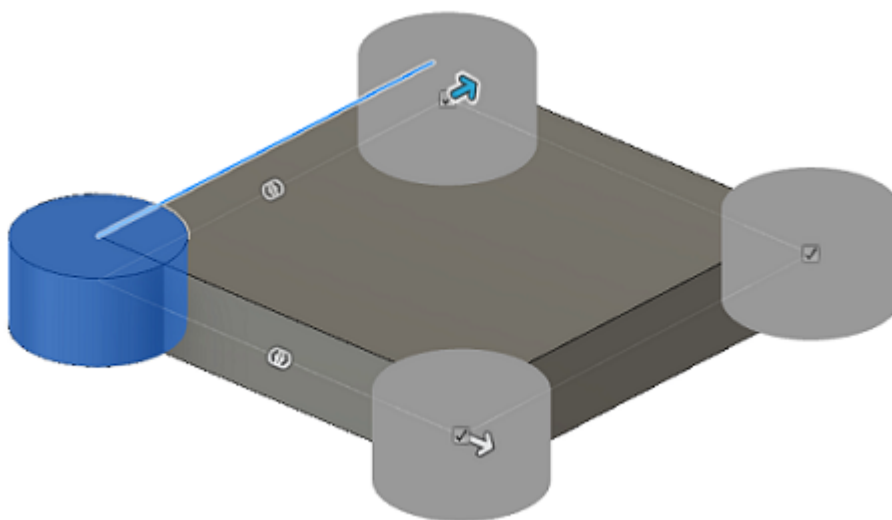
Patterns

You can specify:
- The direction, axis, or symmetry the pattern follows
- The distance and direction type
- The number of copies created in each direction

You can also suppress individual copies within the pattern.

Rectangular Pattern

The Rectangular Pattern command creates copies in one or two directions that you select.
The result is a series of identical copies arranged evenly in rows and columns.
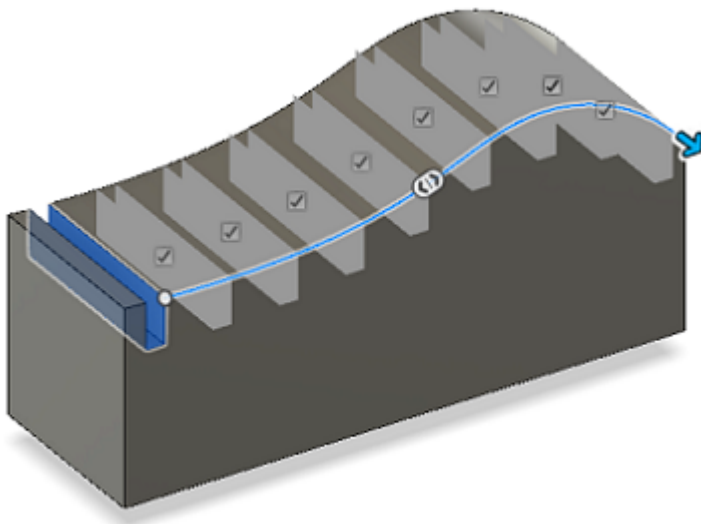
Circular Pattern

The Circular Pattern command creates copies around an axis that you select.
The result is a series of identical copies arranged evenly around a common axis.
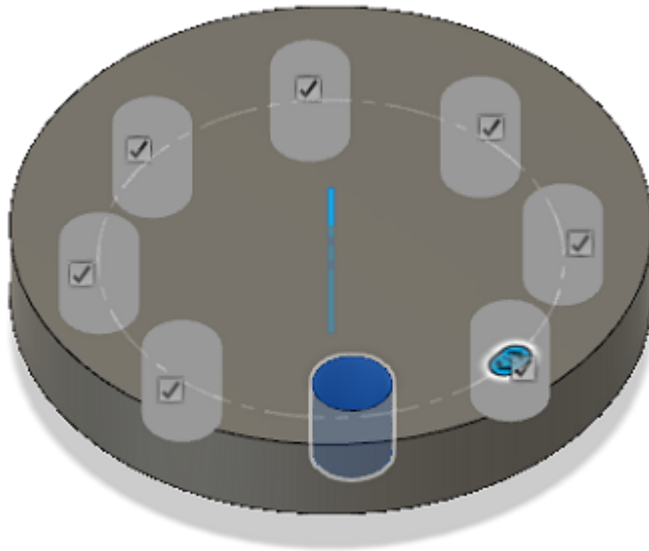
Pattern on Path

The Pattern on Path command creates copies that follow a continuous path that you select.
The result is a series of identical copies arranged evenly along a path.

Create surfaces to cover openings

The internal Fluid Volume command requires surfaces that cover all openings in a model. The surfaces can be:

- Boundary patches that exactly match the opening.
- A work plane or surface that extends past the opening.
- A developed surface that conforms to a curved shape.
- For a planar surface, create an Offset Work Plane. If the work plane is inside the model, anything past it is not included. To remove sections of the model that are not required for simulation, create an internal bounding surface.
- For an opening that has a connected set of edges, use Patch. Create a surface that matches the opening. A Boundary Patch can also blend across a curved shape.
- For more complicated openings, use surfacing commands, such as Loft, Sweep, and Revolve.