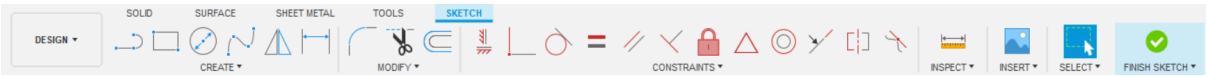
Design Sketch

- Sketch contextual tab

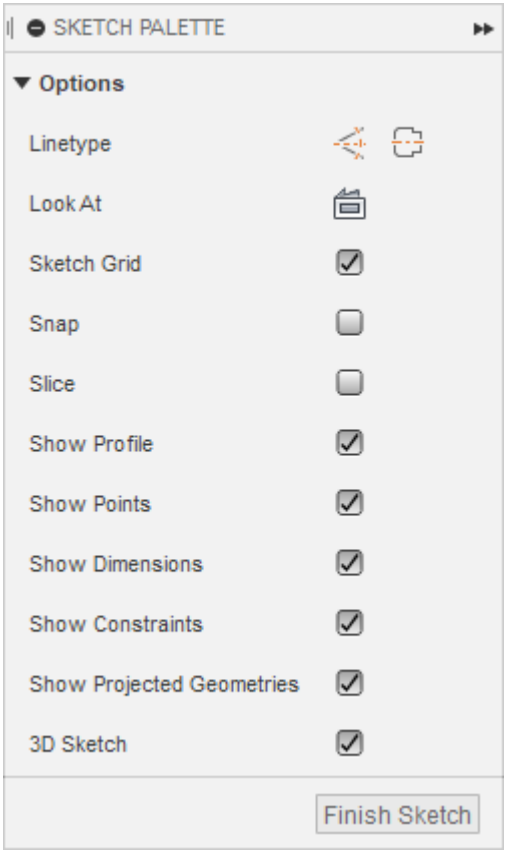
When you create a new sketch or edit an existing sketch, the Sketch contextual tab displays alongside the other toolbar tabs on the toolbar.

The Sketch contextual tab contains tools that let you create, modify, and constrain 2D and 3D sketches that drive the 3D geometry of a design.



- Sketch Palette dialog

When you create a new sketch or edit an existing sketch, the Sketch Palette dialog displays in the canvas.



You can control the following options in the Sketch Palette dialog:

- Feature Options: Displays options specific to the active sketch command.
- Linetype: Converts selected sketch geometry to a different line type.
    - Construction: Creates construction geometry that you can reference without contributing to the boundaries of sketch profiles.
    - Centerline: Creates centerline geometry. Centerlines contribute to the boundaries of sketch profiles.
- Look At: Rotates the camera to look directly at the active sketch plane.
- Sketch Grid: Shows or hides the sketch grid in the canvas.
- Snap: Enables or disables the ability to snap to the sketch grid.
- Slice: Temporarily cuts through bodies where they intersect with the active sketch plane.
- Show Profile: Shows or hides blue shading for closed sketch profiles.
- Show Points: Shows or hides sketch points.
- Show Dimensions: Shows or hides sketch dimensions.

- Show Constraints: Shows or hides sketch constraints.
- Show Projected Geometries: Shows or hides projected geometry in the active sketch.
- 3D Sketch: Enables or disables the ability to sketch in 3D.

Start a sketch on a plane or face

- Choosing a sketch plane
- Using construction planes to create sketches at unique angles
- Using the slice tool to hide existing geometry
- Creating a new sketch on an existing sketch face

Change sketch geometry to construction geometry

- In the Sketch Palette dialog, next to Linetype, click the Construction option. Any sketch geometry you create is now created as construction geometry.
- Or select existing sketch geometry, next to Linetype, click the Construction option.
- Or select existing sketch geometry, right-click, then click Normal/Construction from the Marking Menu.

Change sketch geometry to centerline geometry

- In the Sketch Palette dialog, next to Linetype, click the Centerline option. Any sketch geometry you create is now created as centerline geometry.
- Or select existing sketch geometry, next to Linetype, click the Centerline option.
- Or select existing sketch geometry, right-click, then click Normal/Centerline from the Marking Menu.

Finish a sketch
Learn how to finish an active sketch and return to the Design workspace in Fusion 360.
Do one of the following to finish the active sketch: - In the Design workspace, Sketch contextual tab, click Finish Sketch . - Right-click to display the Marking Menu, then select Sketch > Finish Sketch.

Edit a sketch

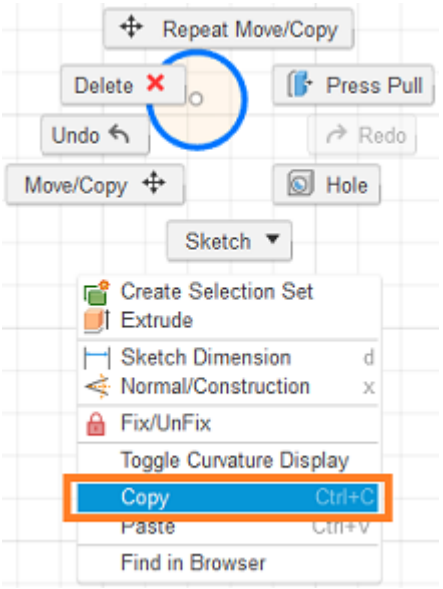
Create a sketch on a selected plane or face.

The Create Sketch icon appears in the following Design workspace toolbar tabs:

- Solid
- Surface
- Form
- Sheet Metal
- Base Feature Solid
- Base Feature Surface

Copy sketch elements to a different sketch

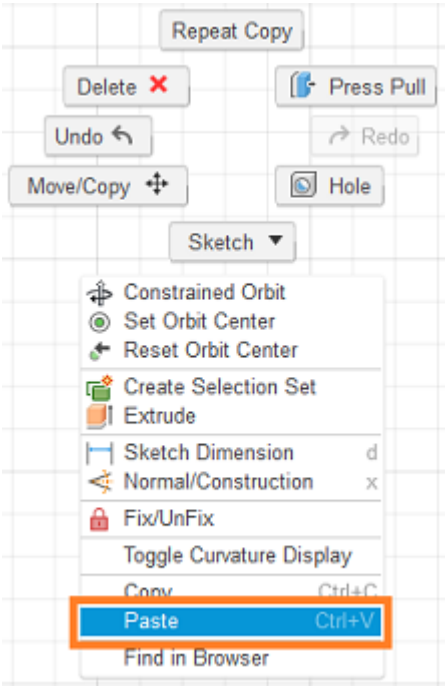
1. With the sketch active, select the elements you want to copy.
   Note: You can hold down the Shift key and click individual elements or window select all the elements.
2. Right-click the selected elements and select Copy. Or use the standard Win/Mac shortcuts, Ctrl+C (Windows), or Command+C (MacOS).



3. Click Finish Sketch.
4. Create a new sketch or locate the sketch you want to paste the copied objects into and make it the active sketch.
   Note: You can use either the browser or the timeline to make a sketch active.

Right-click the desired sketch node and select Edit Sketch. Or double-click the sketch node in the timeline.

5. With the sketch active, right-click an empty area of the canvas and select Paste. Or use the standard Win/Mac shortcuts, Ctrl+V (Windows), or Command+V (macOS).



6. Optionally: Move the copied objects.

Redefine a sketch plane

Demonstrates how a sketch's plane can be redefined if it’s removed as a result of making modifications.

Key concepts and best practices include:

- Why you would need to redefine a sketch plane.
- How to redefine a sketch plane.

Sketch commands supported by 3D Sketch

You can use the following commands to create a 3D Sketch:

- Line
- Arc
- Spline
- Rectangle

- Circle
- Ellipse
- Point
- Text
- Conic Curve

Sketch constraints supported by 3D Sketch

You can use the following constraints to create a 3D Sketch:

- Horizontal/Vertical
- Coincident
- Tangent
- Equal
- Parallel
- Perpendicular
- Fix/Unfix
- Midpoint
- Concentric
- Colinear
- Curvature

Lines in sketches

The Line tool in the Sketch > Create panel lets you create a connected series of lines and arcs as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following command to create lines in an active sketch:

- Line 

Rectangles in sketches

The rectangle tools in the Sketch > Create panel let you create different types of rectangles as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create rectangles in an active sketch:

- 2-Point Rectangle 
- 3-Point Rectangle 
- Center Rectangle 

Circles in sketches

The circle tools in the Sketch > Create panel let you create different types of circles as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create circles in an active sketch:

- Center Diameter Circle 
- 2-Point Circle 
- 3-Point Circle 
- 2-Tangent Circle 
- 3-Tangent Circle 

Arcs in sketches

The arc tools in the Sketch > Create panel let you create different types of arcs as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create arcs in an active sketch:

- 3-Point Arc 
- Center Point Arc 
- Tangent Arc 

Polygons in sketches

The polygon tools in the Sketch > Create panel let you create different types of polygons as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create polygons in an active sketch:

- Circumscribed Polygon
- Inscribed Polygon
- Edge Polygon

Ellipses in sketches

The Ellipse tool in the Sketch > Create panel lets you create ellipses as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following command to create ellipses in an active sketch:

- Ellipse

Slots in sketches

The slot tools in the Sketch > Create panel let you create different types of slots as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create slots in an active sketch:

- Center to Center Slot
- Overall Slot
- Center Point Slot
- 3 Point Arc Slot
- Center Point Arc Slot

Splines in sketches

The spline tools in the Sketch > Create panel let you create different types of splines as sketch geometry or construction geometry in an active sketch in Fusion 360.

A spline is a smooth, freeform curve that passes through or near a set of points that influence the shape of the curve. There are two kinds of splines in Fusion 360.
You can create open or closed splines.
You can use the following commands to create splines in an active sketch:

- Fit Point Spline 
- Control Point Spline 

Conic curves in sketches

The Conic Curve tool in the Sketch > Create panel lets you create conic curves as sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following command to create conic curves in an active sketch:

- Conic Curve 

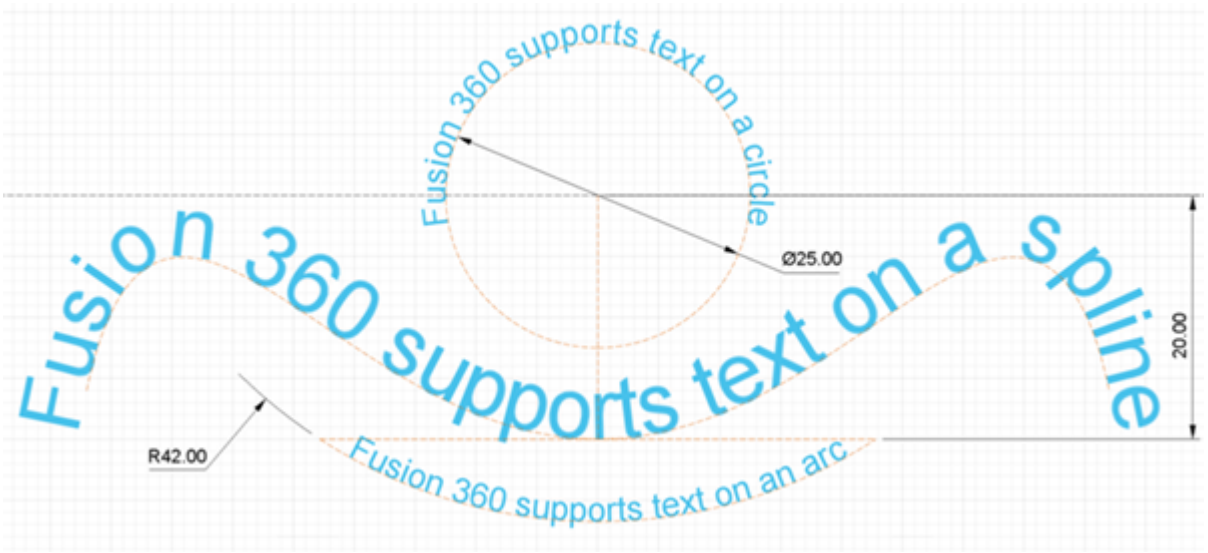
Points in sketches

The Point tool in the Sketch > Create panel lets you create individual points as construction geometry in an active sketch in Fusion 360.
You can use the following command to create points in an active sketch:

- Point 

Text in sketches

The Text tool in the Sketch > Create panel lets you create different types of text as sketch geometry in an active sketch in Fusion 360.



You can use the following command to create text in an active sketch:

- Text A

Note: Before you can create sketch geometry, you must use the Create Sketch command to create a new sketch or right-click an existing sketch and select Edit Sketch to enter the Sketch contextual environment.

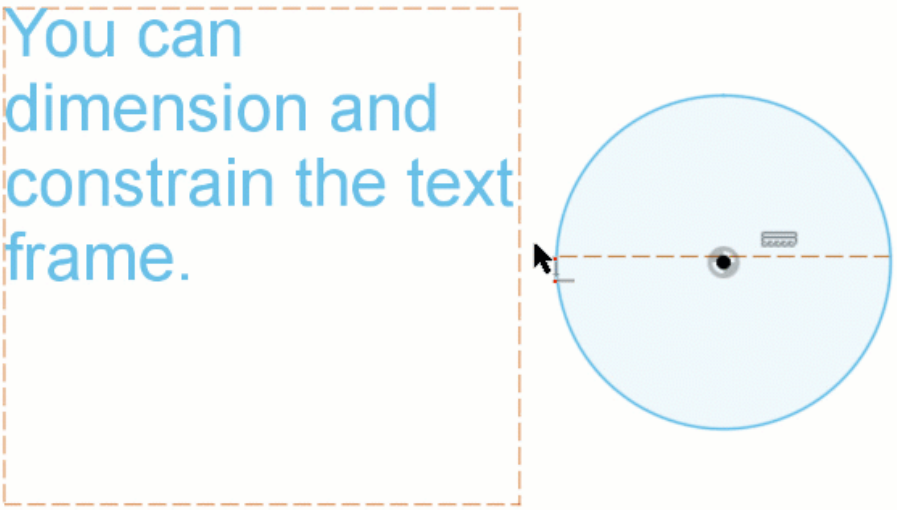
Text

The Text command creates text inside a rectangular frame or along a selected path in the active sketch, which you can use as a profile to create 3D geometry.

You click to place the text frame or select the path for text to follow, then enter and format your text.

After you place the text frame, use the rotation manipulator handle to rotate it. Click any snap point on the text frame to change the point around which you rotate the text frame.

When Type is set to Text A, you can dimension and constrain the text frame to control its size and shape in relation to other sketch geometry.



When Type is set to Text on Path and you modify the path later, the text will adjust along with the path. If the path is a circle, after you create the text, you can use the Move/Copy command to rotate it around the center point of the circle to align it more precisely.

Fit curves to mesh sections

The Fit Curves to Mesh Section command in the Sketch > Create panel lets you fit sketch curves to a section line created from a mesh body by the Create Mesh Section Sketch command, in order to reverse-engineer a mesh in Fusion 360.

- Fit Curves to Mesh Section

Mirrors and patterns in sketches

The mirror and pattern tools in the Sketch > Create panel let you create identical patterns of sketch geometry or construction geometry in an active sketch in Fusion 360.
You can use the following commands to create patterns in an active sketch:

- Mirror
- Circular Pattern
- Rectangular Pattern

Project and include geometry in sketches

The project and include tools in the Sketch > Create panel let you project and include geometry from outside a sketch into an active sketch in Fusion 360.
You can use the following commands to project or include geometry into an active sketch:

- Project
- Intersect
- Include 3D Geometry
- Project To Surface
- Intersection Curve

Dimensions in sketches

The Sketch Dimension tool in the Sketch > Create panel lets you add dimensions to sketch geometry and construction geometry in an active sketch in Fusion 360.
You can use the following command to create dimensions in an active sketch:

- Sketch Dimension

Sketch modification tools

The tools in the Sketch > Modify panel let you modify sketch geometry in Fusion 360.
You can use the following commands to modify sketch geometry:

- Fillet
- Equal Distance Chamfer
- Distance And Angle Chamfer
- Two Distance Chamfer

- Trim
- Extend
- Break
- Sketch Scale
- Offset
- Move/Copy
- Change Parameters

Fillet

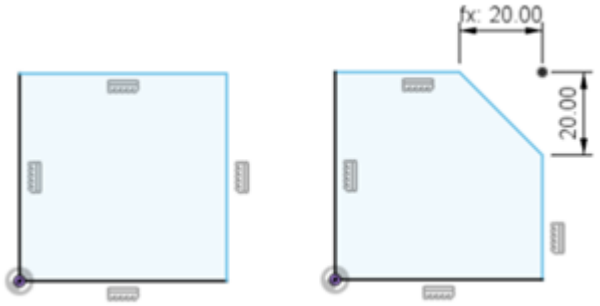
The Fillet command places an arc of a specified radius at the intersection of 2 lines or arcs.
You can create fillets between:

- 2 intersecting lines
- A point at 2 intersecting lines
- 2 parallel lines
- A-line and an arc that intersects
- 2 circular arcs that intersect

You select a vertex, 2 lines, or 2 arcs, then specify the radius.

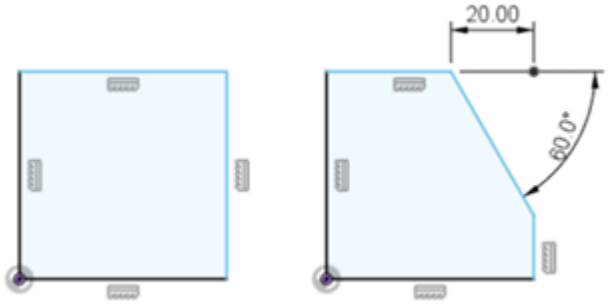
Equal Distance Chamfer

The Equal Distance Chamfer command creates a chamfer at the vertex, intersection, or extended intersection of two lines in a sketch, based on an equal distance to each side.
You select a vertex or two lines, then specify the distance value for the chamfer.



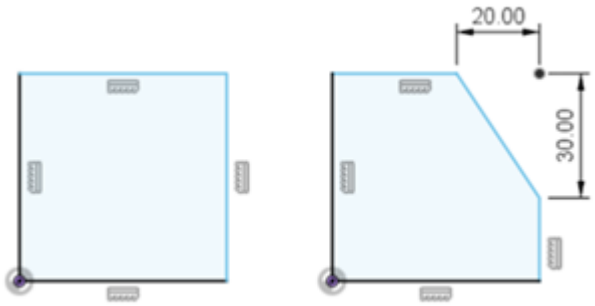
Distance And Angle Chamfer

The Distance And Angle Chamfer command creates a chamfer at the vertex, intersection, or extended intersection of two lines in a sketch, based on a single distance and an angle.
You select a vertex or two lines, then specify the distance and angle values for the chamfer.



Two Distance Chamfer

The Two Distance Chamfer command creates a chamfer at the vertex, intersection, or extended intersection of two lines in a sketch, based on a different distance to each side.
You select a vertex or two lines, then specify the two distance values for the chamfer.



Trim

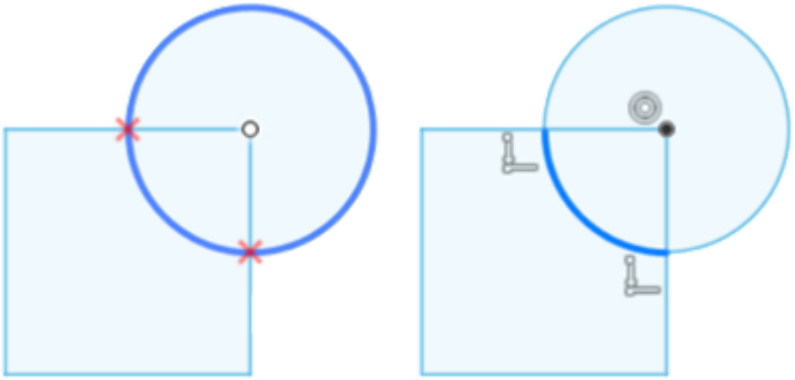
The Trim command trims sketch geometry to the nearest extended intersection with other geometry in the active sketch.
You pause the cursor over the sketch geometry to preview the segment to be trimmed, then click the segment to trim it.

Extend

The Extend command extends sketch geometry to the nearest extended intersection with other geometry in the active sketch.
You pause the cursor over the sketch geometry to preview the segment to be extended, then click the segment to extend it.

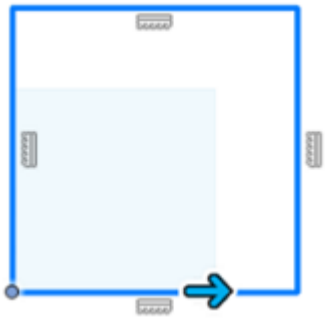
Break

The Break command breaks sketch geometry into two or more segments where it intersects other sketch geometry.
You pause the cursor over the sketch geometry to preview where it will break, then click to break it.



Sketch Scale

The Sketch Scale command scales sketch geometry.
You click to select sketch geometry, then specify the scale factor.



Offset

The Offset command copies sketch geometry a specified distance from the original geometry.
You select the sketch geometry to offset, then specify the offset distance or drag the distance manipulator handle in the canvas.
The size and position of the new geometry are associated with the original geometry. To remove this association, select the offset glyph in the canvas, right-click, then select Delete.

- You can offset any sketch geometry on the active sketch plane, including individual lines, curves, chains of connected geometry, and profiles. You can only create one selection to offset at a time.
- When you offset sketch curves, an offset constraint glyph is displayed on the original geometry and the offset geometry.

Move/Copy

The Move/Copy command moves selected sketch geometry. You can move sketch geometry off the sketch plane to create a 3D sketch.
You select the sketch geometry to move, then specify the new location.

Change Parameters

The Change Parameters command lets you create equations and relationships to control the size of objects in your Fusion 360 design.
You can create and name parameters as you add dimensions to sketches, or as you add and modify bodies.