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






Sketch tools










Sketches are typically two-dimensional entities (lines, circles, arcs, points or splines), created on a plane or existing flat face of a part. You can draw sketch elements or extract edges from the faces of existing models. It is a good practice to lock the position of these elements with dimensions or constraints. Once a sketch is finished, you can use it to create the 3D geometry.

Sketch tools

Tool	Purpose
Create panel	
 Line (L)	Create a series of lines. While using the Line tool, you can transition to the Tangent Arc tool by holding down the mouse button, and back to the Line tool, by releasing the mouse button.
Rectangle	<p>As a skeleton for building more refined sketches.</p> <ul style="list-style-type: none"> 2-Point Rectangle - creates a rectangle defined by two clicks in opposite corners of the rectangle. The sides of this rectangle are horizontal and vertical. 3-Point Rectangle - creates rectangles that are at an angle. The first two points define the edge of the rectangle. The third point defines the width. All of the constraints added to this rectangle are parallel instead of vertical and horizontal, since this rectangle can be drawn at any angle. Center Rectangle - draws a rectangle, with horizontal and vertical sides, based on a center point and a corner. This rectangle includes diagonal construction lines, to ensure that it is centered on the point used to create it.
Circle	<p>Create curved elements, and to aid in the creation of other entities that are drawn in a circular direction.</p> <ul style="list-style-type: none"> Center Diameter Circle (C) - defines the circle location based on its center point. The first click places the center point and the second sets the diameter. 2-Point Circle - creates a circle defined by two points that you position, and which set the diameter. 3-Point Circle - positions the circle at three key points in a sketch. The first two points force the circle to pass through those points, and the third sets the diameter. 2-Tangent Circle - creates a circle tangent to two lines that you select. As you move the cursor to change the diameter of the circle, the circle stays tangent to the lines even if it's not touching them. Click to place the third point for the

Tool	Purpose
	<p>circle.</p> <p> 3-Tangent Circle - creates a circle defined by three lines that you select.</p>
Arc	<p>Create detailed sketches of profiles that include curvature between two points or entities and general curved geometry.</p> <p> 3-Point Arc - connects two points with a curve. Place the two endpoints, move the mouse to change the radius of the arc and preview it, and click a third time to place the arc.</p> <p> Center Point Arc - used to create concentric features using a common center point, for example, a cutout. Click to set the center point, click a second time to define the radius and the start of the arc, and click a third time to complete the arc. Repeat using the same center point, but a different radius, to create concentric arcs.</p> <p> Tangent Arc - connects two sketch entities with a curve that is tangent to the first entity selected. The tangent arc command automatically adds a tangent constraint to the first point, but also adds one to the second point, if the two points are spaced appropriately.</p>
Polygon	<p>Build sketches of multi-faceted geometry.</p> <p> Circumscribed Polygon - uses the center point you select, the number of sides you specify, and the radius and orientation you define, to create a polygon. The radius defines the length from the center of the polygon to an <i>edge</i>.</p> <p> Inscribed Polygon - uses the center point you select, the number of sides you specify, and the radius and orientation you define, to create a polygon. The radius defines the length from the center of the polygon to a <i>corner</i>.</p> <p> Edge Polygon - creates a polygon based on the edge length and orientation you define, and the number of sides you specify.</p>
 Ellipse	Aid in the creation of non-circular, curved entities.
Slot	<p>Create linear slots in 3D geometry.</p> <p> Center to Center Slot - creates a slot based on the center point of each arc, the distance between the two arc centers, and the arc radius or slot width.</p> <p> Overall Slot - uses the orientation, length, and width you provide to create a linear slot.</p> <p> Center Point Slot - creates a slot based on the center point of the slot, the center point of the arcs, and the arc radius or slot width.</p>
 Fit Point Spline	Create a free form curve that passes through a series of points.
 Conic Curve	Create a curve that is defined by two end points and a rho value. Rho determines if the curve is elliptical, parabolic, or hyperbolic.
 Point	Position hole features, construction planes, axes, and to create sketch entities.

Tool	Purpose
 Mirror	Copy and flip sketch entities across a line of symmetry, with a relationship between the mirror and the original entity. To mirror bodies, use the Mirror command in the Create panel.
 Circular Pattern	Create a specified number of copies of sketch entities around a center point.
 Rectangular Pattern	Create copies of sketch entities with a specified spacing and number of instances.
Project/Include	<p>Create sketch geometry from selected objects.</p> <ul style="list-style-type: none">  Project (P) - creates copies of existing model geometry and sketch entities in the active sketch.  Intersect - creates sketch curves from the intersection of the selected objects and the plane of the active sketch.  Include 3D Geometry - projects faces, edges, and points into the active sketch, as 3D sketch geometry.  Project to Surface - creates a 3D sketch by projecting 2D sketch objects to BRep faces.  Intersection Curve - intersects 2D sketch geometry with faces or other 2D sketch geometry, to create a 3D sketch.
 Sketch Dimension (D)	Control the size and position of sketch curves.
Modify panel	
 Fillet	Round corners and edges.
 Trim (T)	<p>Remove unwanted curved and linear line segments. If a sketch has elements that need to be extended, it is best to create the extensions first and trim away the unwanted segments after.</p> <p>Note: Trimming sketch entities is not critical to create 3D features. They can be created from any closed-loop profile; however trimming all the extra lines will make the sketch more robust as well as more clearly defined.</p>
 Extend	Extend sketch elements, both curved and linear, so that they connect with other lines without having to re-sketch entire sections.
 Break	Break curve entities into two or more sections.
 Sketch Scale	Enlarge or reduce a selected sketch geometry based on a specified scale factor.
 Offset (O)	Create new sketch geometry from existing sketches or edges.
 Move/Copy (M)	Move or copy a face, body, sketch curve, component, or sketch geometry.
Constraints panel	
 Coincident	Constrains the position of two points or a point and a line or curve together.
 Collinear	Constrains two or more objects so that they share a common line.

Tool	Purpose
 Concentric	Constrains two or more arcs, circles, or ellipses to the same center point.
 MidPoint	Constrains a point or object to the midpoint of another object.
 Fix/UnFix	Locks the size and location of a point or object.
 Parallel	Constrains two lines so that they extend in the same direction and never intersect.
 Perpendicular	Constrains two objects so that they lie perpendicular (at a 90-degree angle) to each other.
 Horizontal/Vertical	Constrains a single line, or two points, to lie on either the horizontal or vertical axis, whichever is closer to the current alignment.
 Tangent	Constrains a curve and another object so that they touch at a single point but never cross each other.
 Equal	Constrains similar objects so that their sizes are identical. When the size of one object changes, the others adjust, too.
 Symmetry	Constrains two or more objects so that they are symmetrical (identical to each other in relation to a common axis).