

# Sketch Tools

*Note: all tools can be accessed by **left clicking** on them and employed by **left clicking** in the design space.*



***Double clicking** on many tool end points will end that feature. To exit the tool press **esc** or **left click** on the tool again.*

*In SolidWorks 2009 hitting **esc** will cancel the current action, and hitting **enter** will accept any changes made while sketching.*



## Sketch:

Opens a sketch on a selected plane or surface. To open, **left click** on the icon and pick the desired plane or surface. To exit a sketch **left click** again on the icon or hit the **exit sketch**

icon  or hit **esc**. To cancel any progress on the sketch press the **cancel icon**  in the upper right corner of the design space.

Additional tools such as 3D sketch can be found under the drop down arrow. 3D sketch performs the same except that it allows the user to sketch in the X, Y, and Z directions, whereas the sketch tool only allows for two dimensions.



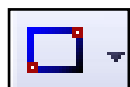
## Smart dimension:

Creates a dimension for a selected sketch entity.



## Line:

Sketch a horizontal, vertical, or angled line. **Double click** to end the line. Also start a line by **holding the left mouse button** and end the line by **releasing the button**. The centerline tool can be accessed through the drop down arrow and performs the same way.



## Corner Rectangle:

Sketch a rectangle or square. Each rectangle tool shows the user inputs in the tool by denoting the mouse “clicking” points as small red circles.

Additional tools such as parallelogram, center rectangle, 3 point corner rectangle, and 3 point center rectangle can be found under the drop down arrow. These function the same as the corner rectangle tool.




## Polygon:

Sketches a polygon. Allows the user to input the number of sides on the polygon. Inserts pre-defined reference geometry to minimize sketch dimensions.



## Perimeter Circle:

Sketches a circle by using three points specified by the user. Can define the circle by two

points instead of three by checking the circle icon  under circle type in the property manager.

Additional tools such as circle can be found under the drop down arrow. The key difference is that the circle is defined by a center point and the radius.

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Tangent Arc:



An incomplete circle defined by its tangency to the sketch entity. Specify the tangent arc by **left clicking** a sketch endpoint and then **left click** to specify the endpoint of the arc.

Additional tools such as center point arc and 3 point arc can be found under the drop down arrow. They function similarly to the tangent arc by asking the user to input the defining arc points.

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Sketch Fillet:

A tangent arc inbetween two sketch entities. Specify the radius by **left clicking** in the radius text box in the property manager

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Spline:

A two or three dimensional curve defined by a set of control points. Points within a spline can be added or deleted by **left clicking** on the spline point and saying delete, or add spline point.

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Ellipse:

Sketches a complete ellipse. **Left click** in the design space to place the center of the ellipse. Next, define the major and minor axis by **left clicking** and then dragging the cursor to the desired position.

The partial ellipse and parabola tool can be found under the drop down arrow. They function similarly to the ellipse tool.

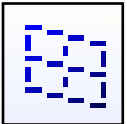
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Point:

Inserts a point or vertex into a sketch entity. This cannot be deleted if it is part of sketch endpoint or center point.

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Plane:

Inserts a plane in a 3D sketch. Must be in the 3D sketching tool to use it. Useful as construction geometry to help define the sketch entity. Define the plane through either the sketch or the models reference planes. The property manager will appear on the left hand side to aid in defining the plane.

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Text:

Sketches text on faces, curves, and sketch entities. Need to have a guided edge, curve, or sketch entity for the text to follow. Make sure that the curves box is highlighted in blue and select the path in the design space for the text to follow. Then **left click** in the text box located in the property manager on the left hand side of the screen.

To change the font's appearance: **left click** the check mark next to "Use document font" and alter the font, style, and its size. The property manager also gives the user the ability to bold, italicize, rotate, and mirror the text. Additional options include changing the width of the letters and the spacing.

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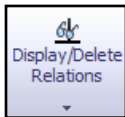
Trim Entities:

Trim sketch entities to the desired shape. To trim a section **left click** on the desired sketch entity to cut away. Several trim options include power trim, corner, trim away inside, trim away outside, and trim to closest.

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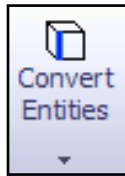
### Display/Delete Relations:



In order to define a sketch the user can relate sketch entities to each other. These relations both defined by the user and solidworks can be viewed by selecting the display/delete relations tool.

The add relations tool can be found under the drop down arrow. This enables the user to pull up the add relation property manager and manually add relations between sketch entities.

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### Convert Entities:

Project a sketch entity, face, edge, curve, or contour onto the open sketch plane. To convert entities **left click** on the desired sketch, face, edge, contour, or curve and then **left click** on convert entities. A new sketch will be created on the plane fully defined using the geometry of the selected entity.

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### Offset Entities:

Copy a sketch entity to a specified distance from the original entity. Must have a sketch entity already drawn and selected to use this tool. Select the offset distance in the property manager by **left clicking** in the offset distance text box (  ) and specifying the value.

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### Sketch Picture:

Insert an image file into the background of a sketch. This allows the user to 2D sketches based off of the image. Use the sketch picture property manager to alter the inserted image.

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### Mirror Entities:

Create a reflected copy of a desired entity about a centerline or axis. Make sure the entities to mirror box is highlighted in the property manager. Select the desired entities to mirror, or replicate by **left clicking** on each. Select the mirror about box in the property manager by **left clicking** in the text box. Then **left click** the centerline/axis. A preview will appear in the design space. Press  (or enter) to accept the mirror or  (or esc) to cancel in the property manager, and exit the mirror entities tool.

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### Linear Sketch Pattern:

Pattern a sketch entity either horizontally, vertically, or angled. Under the property manager the user can specify the distance between the original sketch entity and the patterned copy, the number of instances, and an angle the pattern will appear. Sketch entities can be skipped by selecting the drop down arrow in **instances to skip**, selecting the text box, and then **left clicking** the patterned entities to skip.

The circular pattern tool can be found under the drop down arrow. It performs similarly to the linear sketch pattern except that it requires the user to specify a point to revolve about and a radius.

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