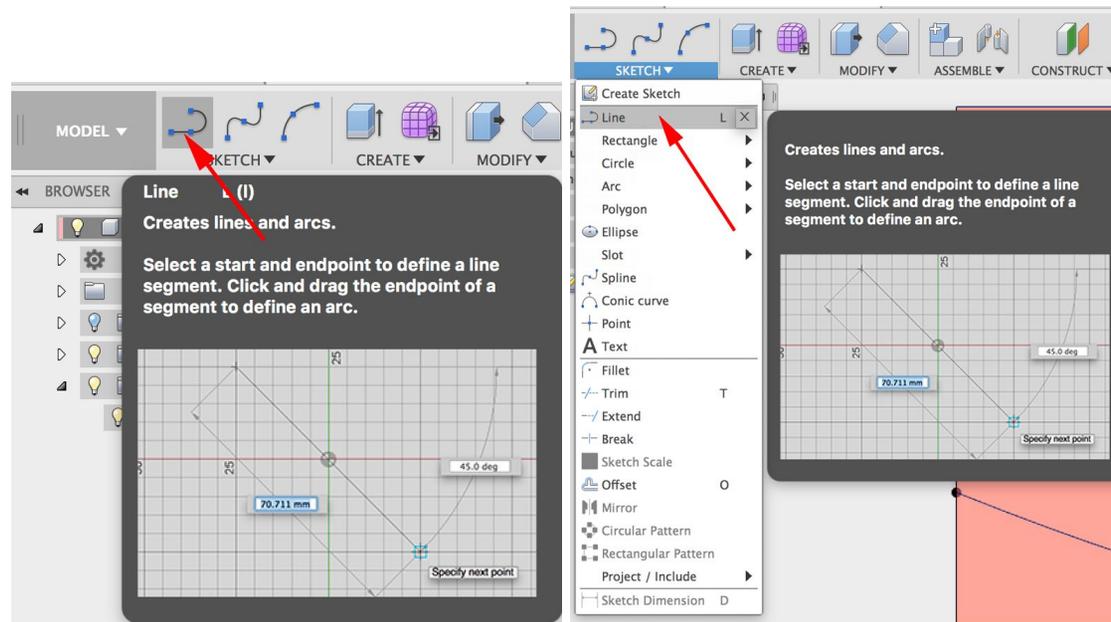


Fusion 360 CAD Help

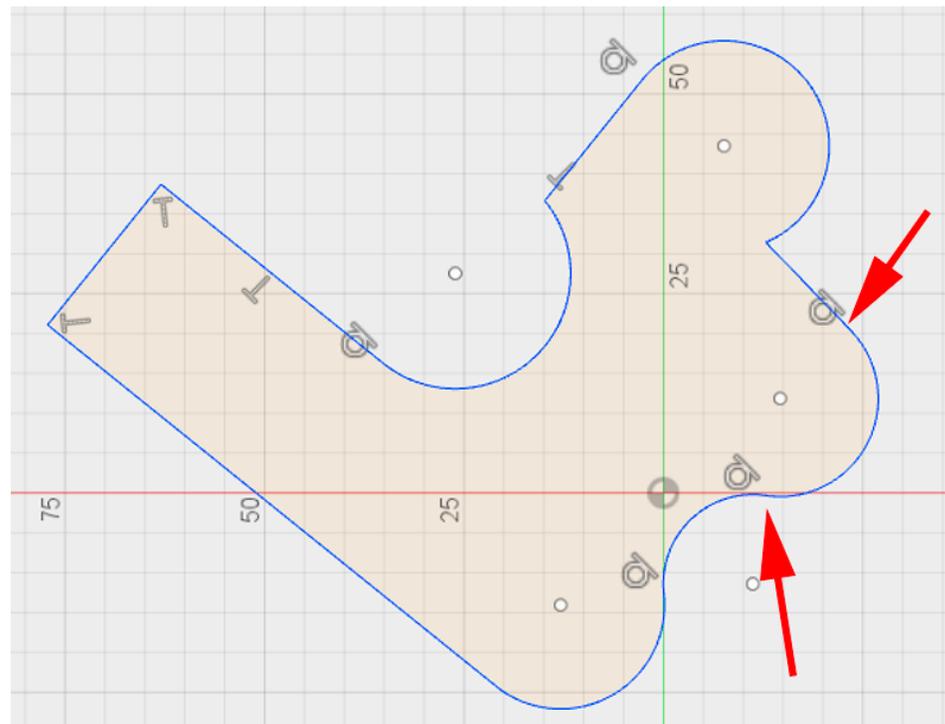
Lines

Lines can be drawn between any two points on the sketch plane, creating any length or angle needed. The line tool can be accessed in a number of ways. It is in the toolbar at the top, in the Sketch menu which, and can be activated by pressing the **L** key. It can also be activated in the right-click menu, under **Sketch** or in the Marking Menu at the bottom. This corresponds to the right-click gesture going straight down.



With the line tool active, draw a line by clicking two endpoints. Clicking a second time on screen completes the line. The line tool is still active, and you can continue drawing lines. Fusion 360 keeps the line tool active so that you can create a chain of lines connected together. When you want to end the chain, click on the check mark shown nearby the line, or press the Escape key.

To create straight segments, just click on two endpoints. If you click on one endpoint and keep the mouse depressed while you move to another position and then release, you will get a curve:

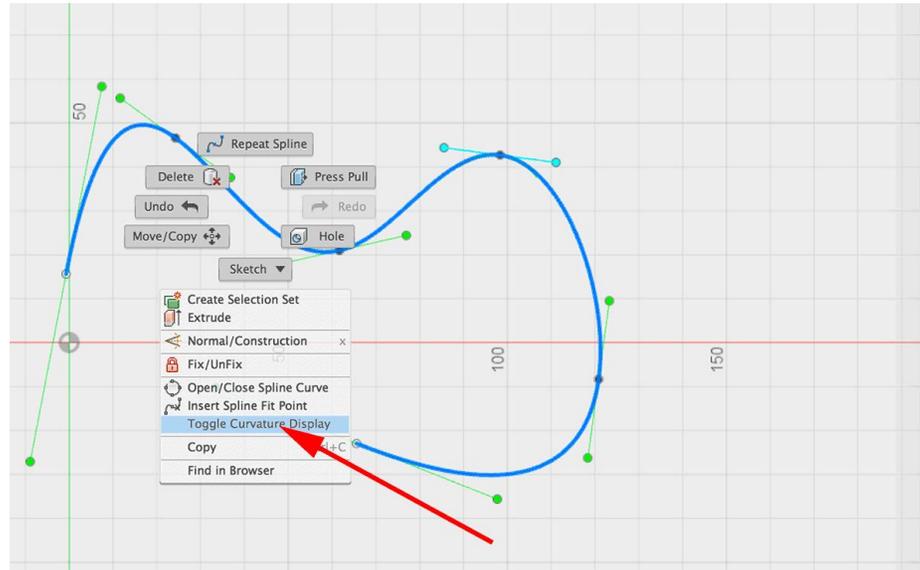


Spline

The **Spline** command is found in the sketch menu and is used to create organic curves between points. To create a spline, enable the command then specify key points along the curve. As you place each point notice the solid green line that shows the current spline between the specified point. To stop the spline, click on the green checkmark, then press ESC to deactivate the spline commands. You can change the shape of a spline by clicking and dragging any of the spline points.

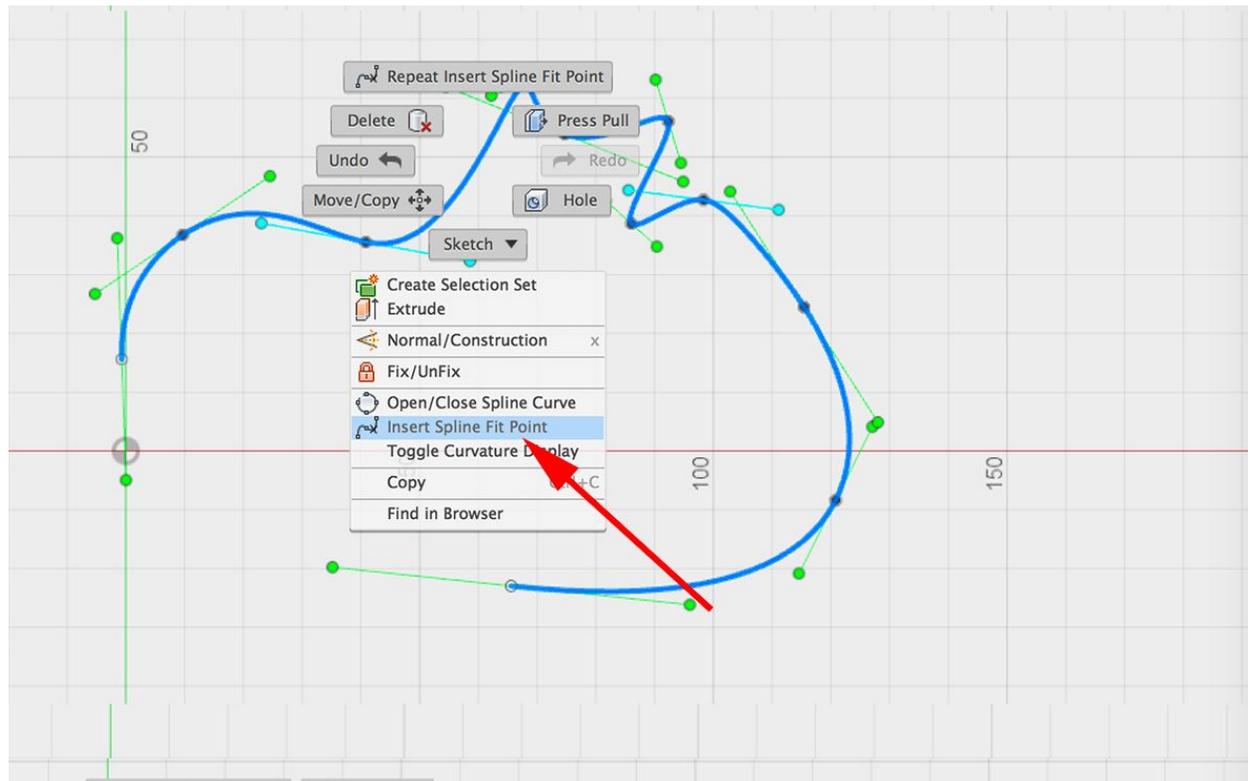
If you click on the spline itself to select it, notice the green tangent handles that appear at each spline point. These handles can be used to adjust the curvature of the spline at that point, also by clicking and dragging. You can also extend the length of the handle to widen the curve around that point. The changes you make affect the spline as a whole, not just this small section. However the spline point doesn't move, allowing for better control while manipulating the spline.

To help understand the spline's curvature, display the spline's curvature comb, which is a graphical representation of the curvature at each point overlaid directly on the spline. To turn it on I'll select the spline then right click, then select **Toggle Curvature Display** to turn the curvature comb on and off.

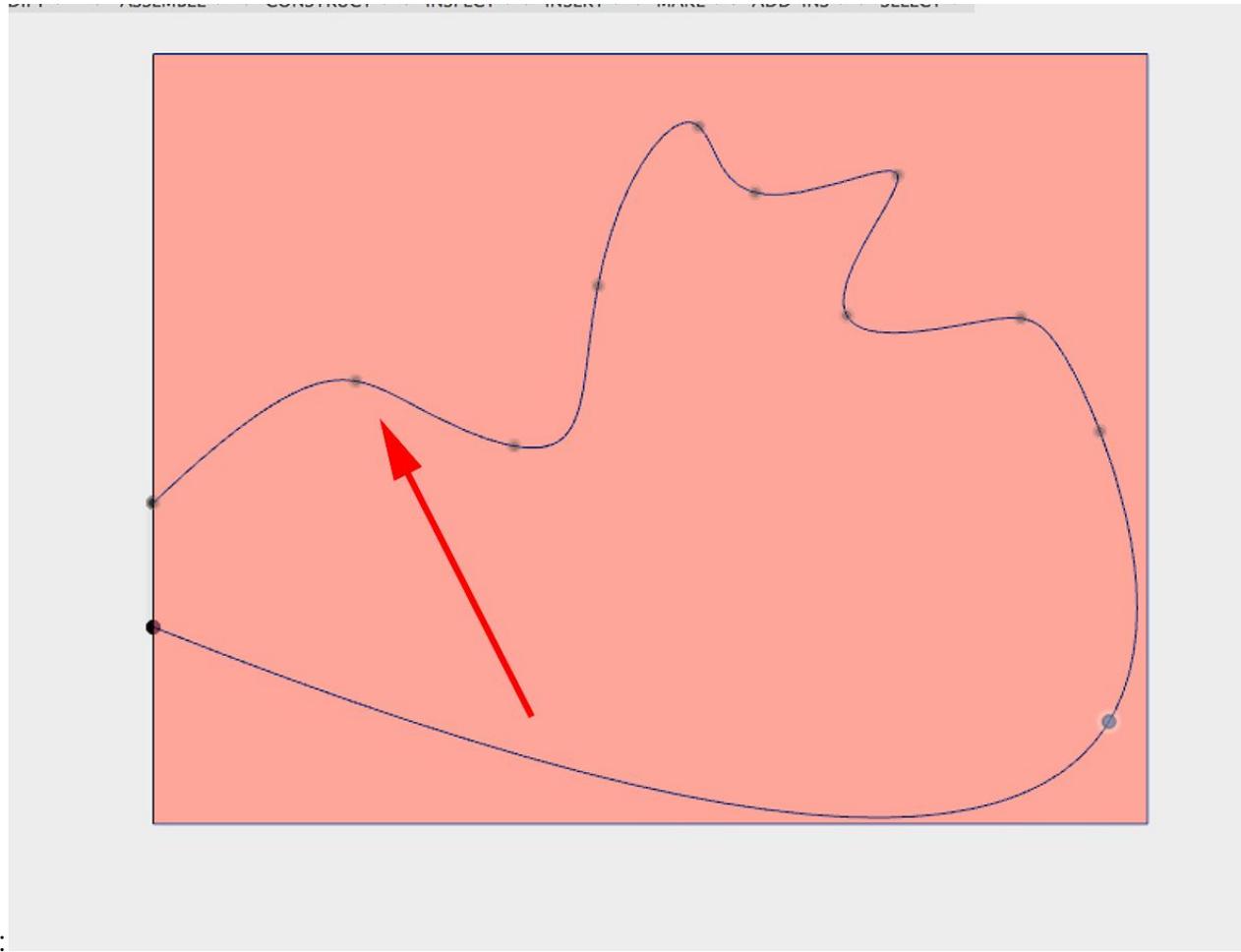


The longer lines on the curvature comb indicate larger rate of curvature, which also means there is a small radius at this point. If needed the comb density and scale can be adjusted in the dialog box.

You can add more points to your spline by selecting the spline then right clicking. Then choose **Insert Spline Fit Point**. Click on the spline to add the point, then press ESC. Now you can edit the spline.



You can also continue to edit the spline without editing the sketch. Just click and drag the points



Keyboard Sticker Sheet



**AUTODESK[®]
FUSION 360[™]**

⌘+B | Ctrl+B
Compute All

Alt+1  ⌘+S 
Box Display Scripts

Alt+3  ⌘+J 
Smooth Display As-built Joint

S Toolbox		J  Joint		
Q  Press Pull	E  Extrude	F  Fillet	H  Hole	M  Move
V  Visibility	SHIFT+N  Component Color Cycling	I  Measure	P  Project	L  Line
X  Normal / Construction	DH  Dimension	C  Center Dia. Circle	R  2-Point Rectangle	T  Trim
1  Window Selection	2  Freeform Selection	3  Paint Selection	A  Appearance	O  Offset