Introduction to Sketches in Fusion 360

This guide introduces the concept of sketching in Fusion 360

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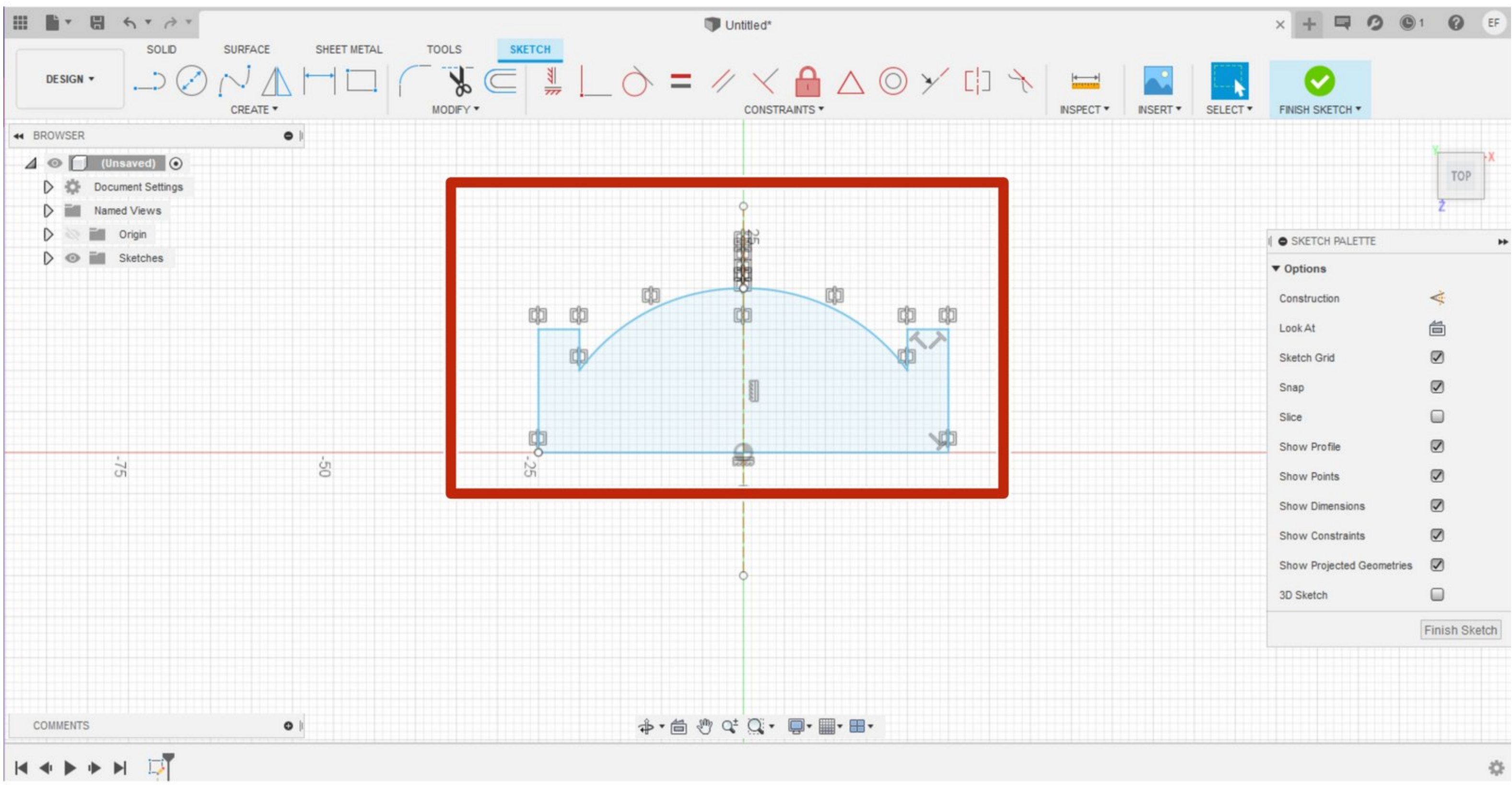
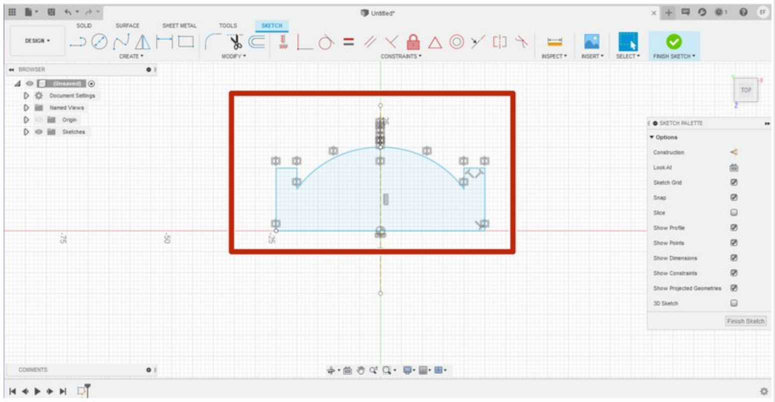
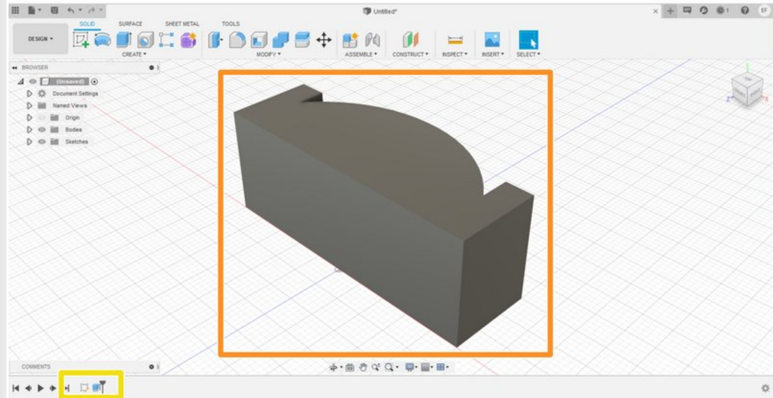


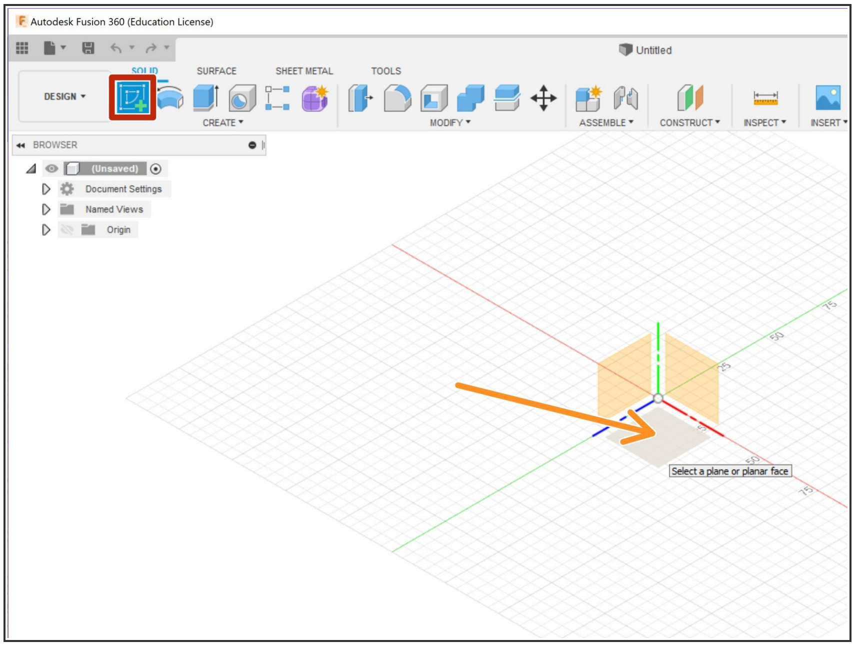
Step 1 — Making 3D Designs in CAD





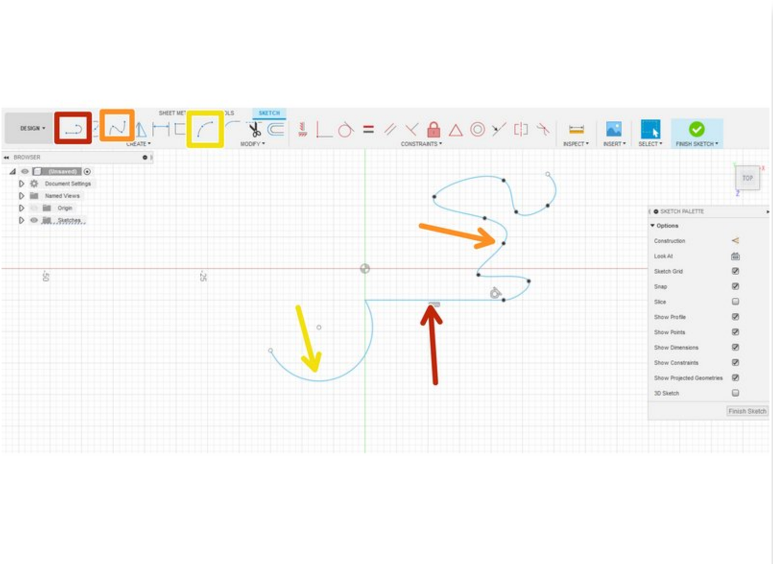
- The general process of creating a 3D design in most CAD software: **first create a sketch, then manipulate that sketch into a 3D solid**
- The figure in the red box is a sketch. It's a collection of lines, curves, and closed shapes. To create a solid, the sketch must form a closed profile.
- We can make the 3D shape in the orange box by applying the *extrusion* tool to the sketch.
- The design timeline in the bottom left of the window shows which tools we used. As indicated there, this shape was made with a single sketch and extrusion.
- This is just one example of the general process used to create solid bodies in CAD.
- This guide gives an overview of some of the tools and techniques available when making a sketch. By no means is this list exhaustive. If you want to learn more advanced sketch tools, feel free to check out the module series for intermediate Fusion 360 users.
- The guide on [Introduction to Solid Features in Autodesk Fusion](#) discusses some of the tools and techniques for turning a sketch into a solid body

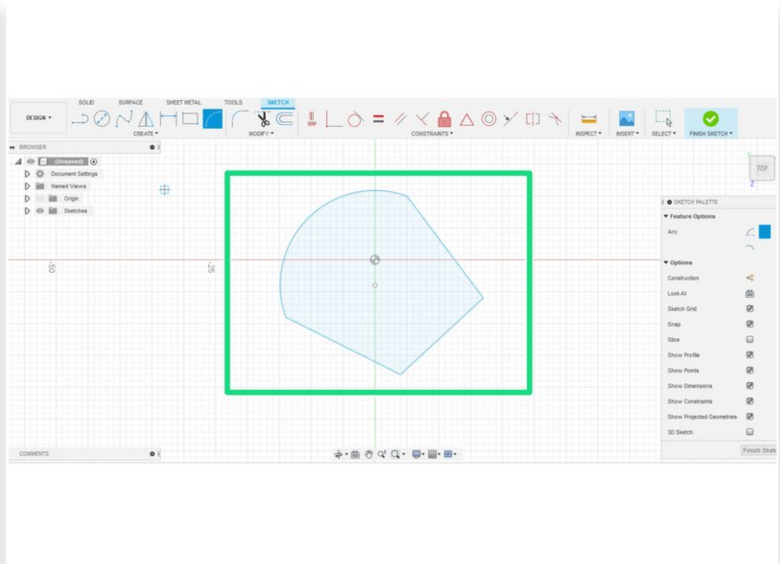
Step 2 — Opening a Sketch



- To start a new sketch, click on the "New Sketch" icon
- Sketches require a reference plane. Click on the plane on which you wish to create the sketch

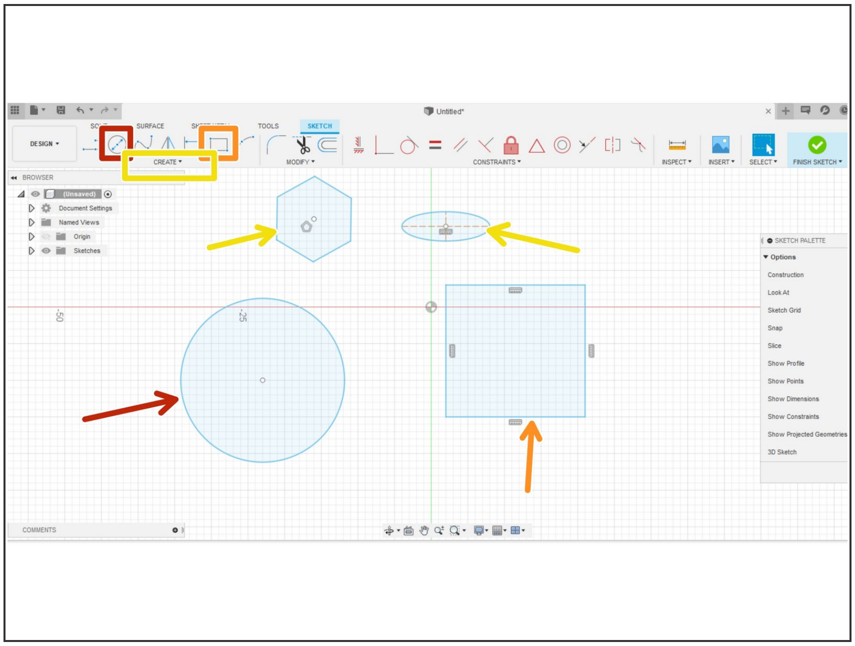
Step 3 — Lines and Curves





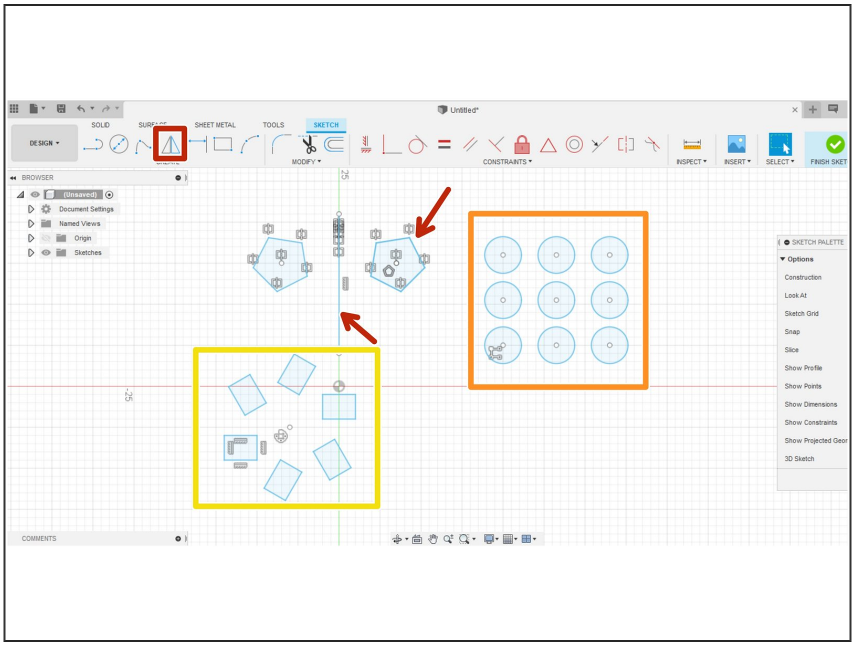
- This figure shows a few of the most commonly used tools to create lines and curves in a sketch.
- The line tool, shockingly, creates a line
- The Spline tool creates a curve that interpolates several points.
- The arc tool creates an arc defined by either three points, or two points and a radius of curvature.
- Connecting any line or curve can create a closed profile, denoted with a blue-shaded interior. This profile consists of three lines and an arc.
- We encourage you to take a pause here and try out these tools in Fusion 360 before continuing

Step 4 — Shapes



- In addition to lines and curves, Fusion 360 has tools to make shapes in sketch mode. Note that all of the shapes are automatically closed profiles
- Typically when making a shape, the user has the option of specifying important dimensions.
- Multiple tools exist to make each shape by specifying different dimensions. For example, this circle was made by placing a center point and specifying the diameter.
- The two point rectangle shown here is made by placing the points on opposite corners.
- The polygon tool enables the creation of a regular polygon with a number of sides specified by the user either circumscribed or inscribed into a circle. This tool is located in the drop down menu.
- The ellipse tool enables the user to make an ellipse by specifying the center point, major and minor axes. this tool is also typically located in the drop down menu.

Step 5 — Patterns



- Pattern tools repeat the same sketch features (line, curve, or shape).
- The mirror tool reflects a shape over a line. This command can mirror both closed profiles and lines or curves.
- The rectangular tool makes a pattern of a sketch feature in a rectangular lattice. By default, the two axes of the rectangle align with axes of the sketch plane; however, the user can specify the axes as a line.
- The circular pattern rotates a sketch feature around a point

Step 6 — Concluding Thoughts

- This guide gives an introduction to a few of the most important and commonly used tool for making sketches in Fusion 360.
- We encourage you to take some time after reading this guide to play with each of the tools at home. Try to recreate the cross section of objects a Fusion sketches.
- To learn some of the basics for turning a sketch into a 3D model, refer to Introduction to Solid Features in Autodesk Fusion.