3D CAD Basics for Guitar Design

**Description of Activity**

* This activity is a very simple introduction to 3D CAD concepts and leads in to creating a guitar shaped sketch for use with STEM Guitar Project CNC no-profile model.
* Purpose of this activity is to serve as a quick-start guide for designing a Guitar Shaped Object using AutoDesk Fusion 360.
* This activity is intended for grades 9-14.

**Learning Objectives:**

**(List measureable objectives)**

1. Students will create a sketch.
2. Students will create a 3D extrusion.
3. Students will

**Standards:**

|  |  |  |  |
| --- | --- | --- | --- |
| **Student Performance Objective(s):** | | | |
| **HS-ETS1-2** | Design a solution to a complex real-world problem by breaking it down into smaller, more manageable problems that can be solved through engineering. | | |
| Science and Engineering Practices | | Disciplinary Core Ideas | Crosscutting Concepts |
| **Constructing Explanations and Designing Solutions**   * Design a solution to a complex real-world problem, based on scientific knowledge, student-generated sources of evidence, prioritized criteria, and tradeoff considerations. (HS-ETS1-2) | | **ETS1.C: Optimizing the Design Solution**  ▪ Criteria may need to be broken down into simpler ones that can be approached systematically, and decisions about the priority of certain criteria over others (trade-offs) may be needed. (HS- ETS1-2) | **Systems and System Models**  ▪ Models (e.g., physical, mathematical, computer models) can be used to simulate systems and interactions— including energy, matter, and information flows— within and between systems at different scales. |

Common Core State Standards for Mathematics:

**MP.4** Model with mathematics.

List The Common Core Math, Next Generation Science Standard and/or SME Competency Gaps. For example: <NEED STANDARDS>

**Materials Required:**

* Computer station with AutoDesk Fusion 360 installed.

**References:**

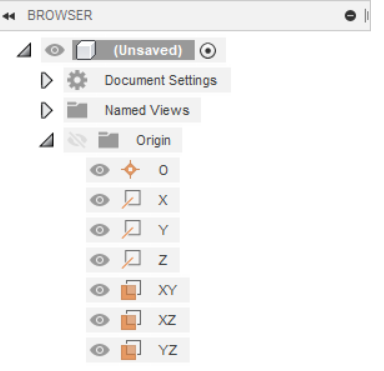
* Add any references or sites for further exploration here.

**Activity:**

All 3D CAD programs function similarly regardless of vendor. Concepts and terms are largely the same from one platform to the next. Learning the first may be the most difficult. Migrating to another platform after mastering the first involves a much shorter, shallower learning curve.

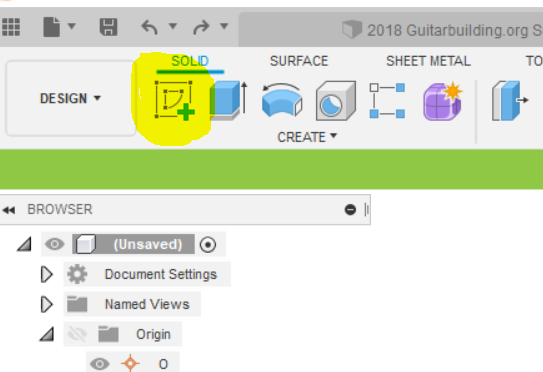
Planes and Axes

You may define as many workplanes (geometric plane) and axes as you wish, but 3D CAD programs start you off with three of each. There are X,Y, and Z axes. There are XY,XZ, and YZ planes. Depending on the computer system, the orientation of the axes and planes varies. To keep it all straight, you will rely heavily on the browser. The browser is typically in the left side of the screen and shows a list of workplanes, sketches, and features.

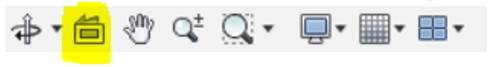


Making a shape – 2D sketch

To begin in 3D CAD, choose “New Sketch”, “Create Sketch” or something similar.

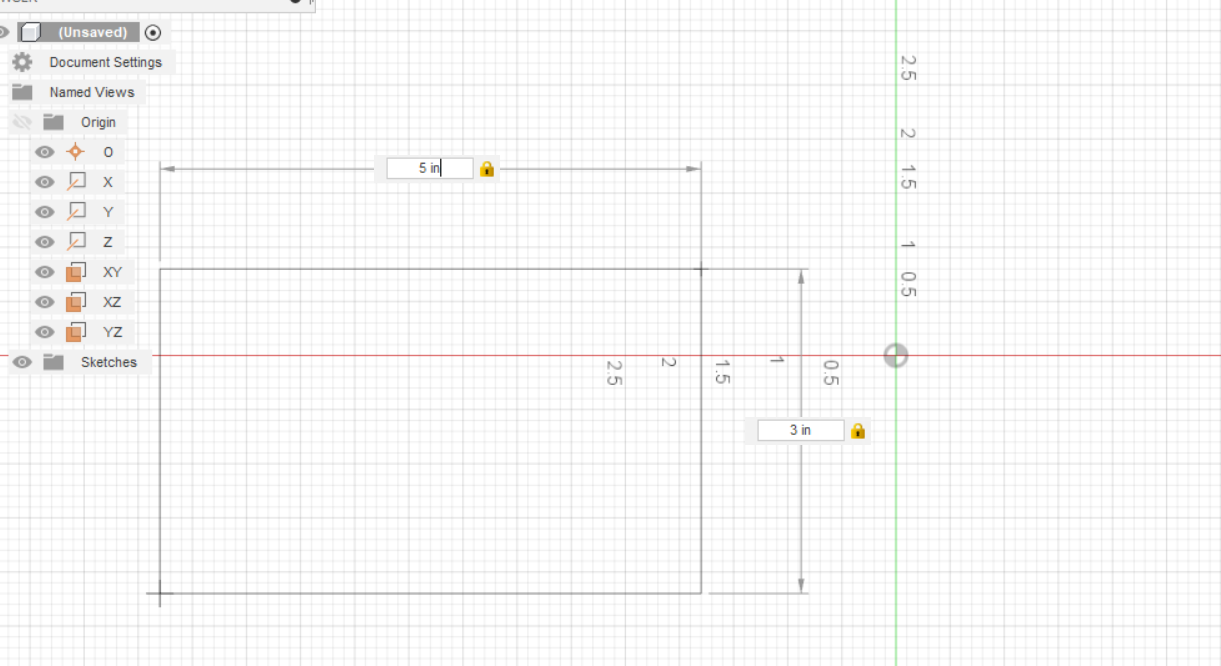
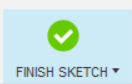


The button may be in the top command ribbon, or you might right click to find it. You may need to choose a workplane first, or be prompted to select a workplane after clicking this command. There is a Look At button at the bottom of the screen you can use to look at your sketch 2D. This makes it easier to draw lines, curves, circles, ellipses, rectangles. You also will be able to establish sizes and distances using dimensions.



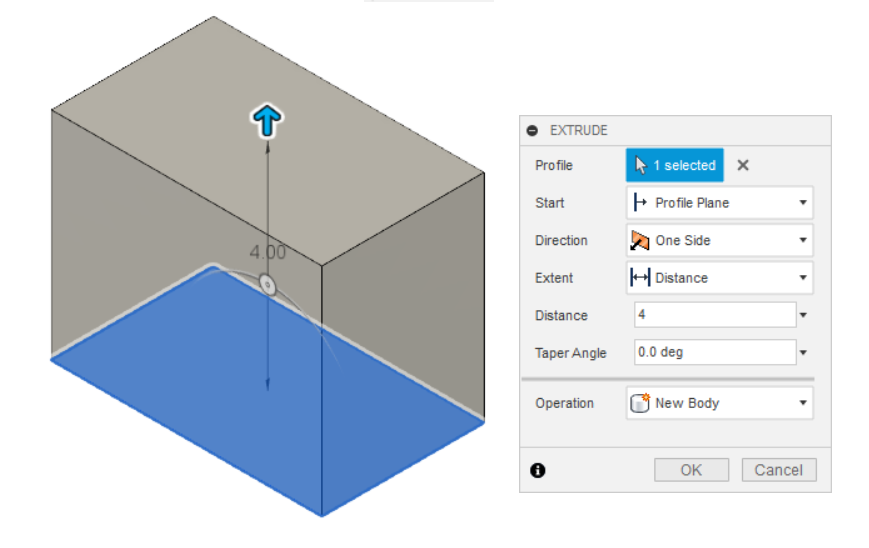
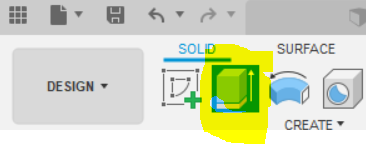
\* *These dimensions become some of the parameters that can be edited to modify shapes without needing to redraw from scratch. This feature of editing parameters classifies 3D CAD programs as parametric modeling. Parametric = parameter.*

Draw a rectangle. Look for a checkmark or a button to close or finish a sketch.

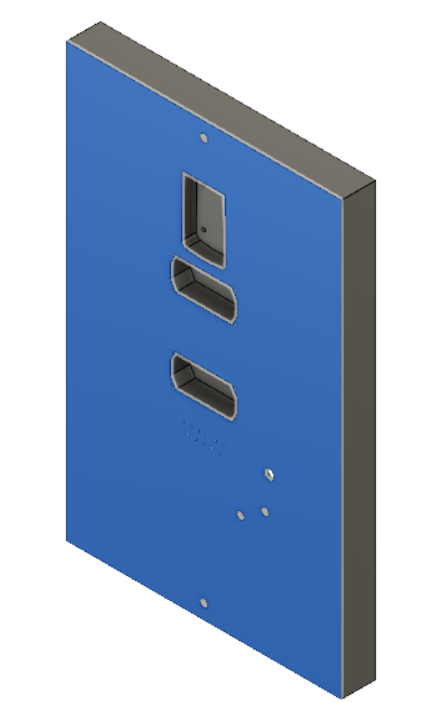
Making a shape – 3D feature

With the sketch closed – and it is important that the sketch be closed – find the extrude command. Enter a distance. Most programs typically provide a preview feature in the extrude command so you can verify the 3D feature that will be created. It is possible to add or subtract material (sometimes called cut). So if you start with the Sinclair STEM Guitar Body model, you could draw your own pickup pockets in say, and extrude them subtracting material below the top surface to create a cavity for the hardware.

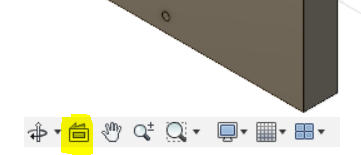


Make a guitar shape

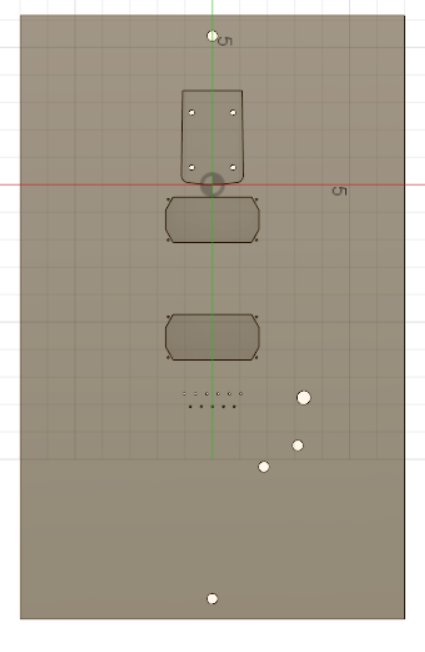
Click the front face of the Sinclair STEM Guitar no profile body blank. It will turn blue.



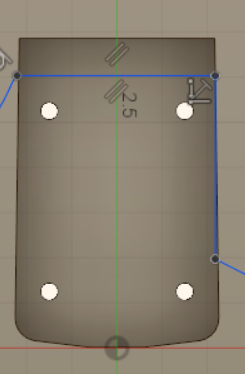
Click the Create Sketch button. Then use the Look At button at the bottom of the screen to view the guitar top straight on 2D.



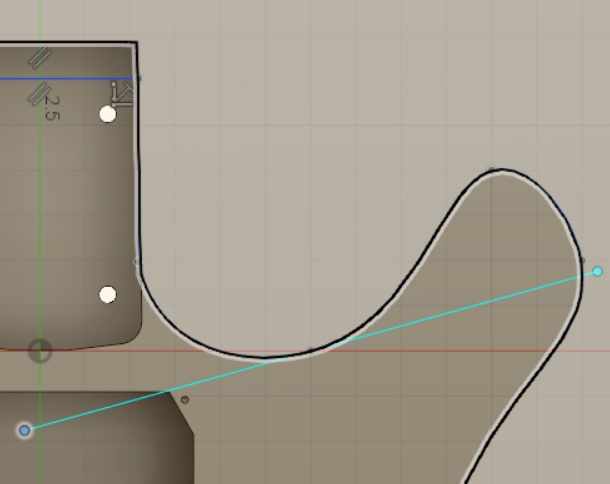
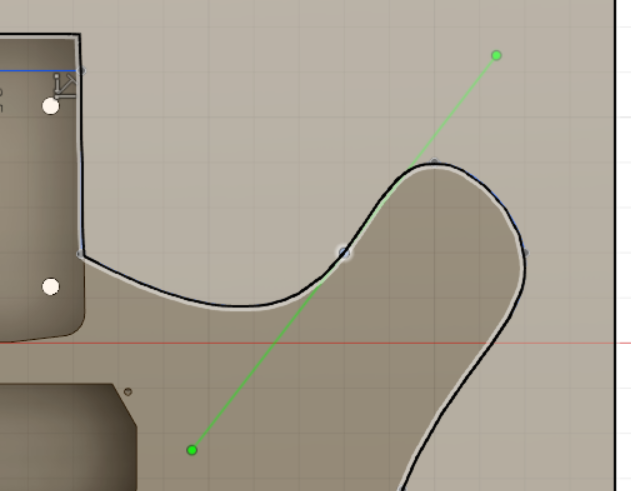
You should be looking at it straight on, and see some graph paper grid too. Now you are ready to draw a guitar shape.



You can use straight lines, curves, splines, arcs… whatever you would like to create the shape. It is essential that the endpoints of separate 2D drawn pieces connect to form a closed polygon. To begin, draw a horizontal line across the top of the neck pocket. Draw this down a bit from the top edge of the neck pocket. The extra on the neck pocket is to make sure the neck clears radiused part of the router bit’s path and the neck fits. Draw a vertical one down the treble side. Even if you plan to have a bass side cutaway, that side usually does not require a vertical line on that side of the pocket.

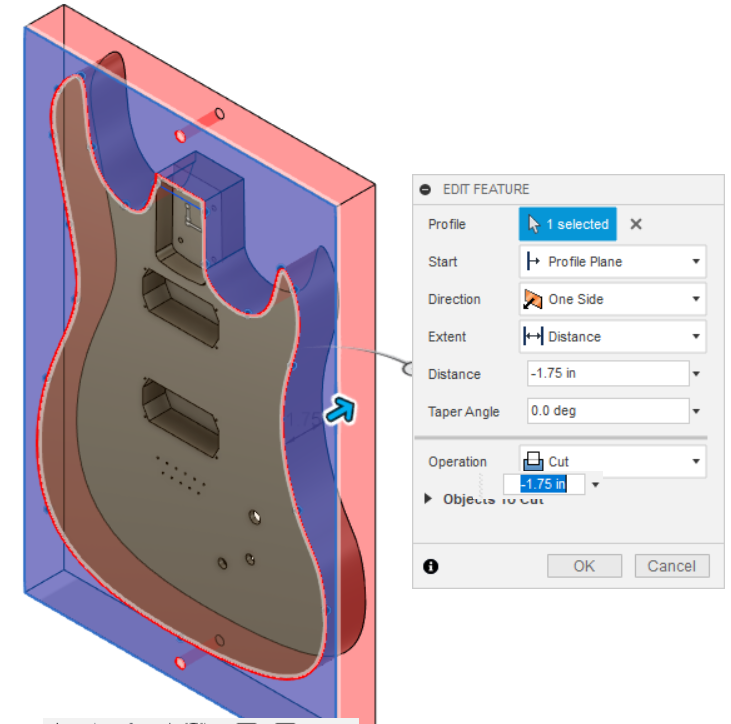
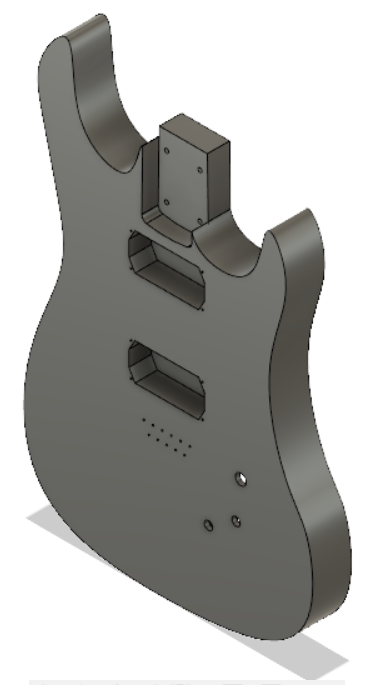


Use the spline tool to draw around the perimeter of your desired guitar shape. Once you complete the spline, you can edit it by clicking to drag a vertex, or clicking to drag and resize the handles that affect how the curve flows through the point.

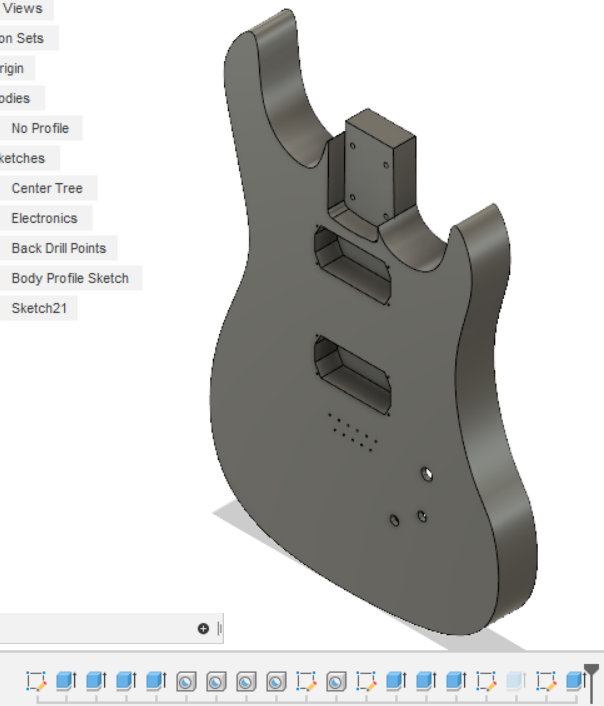


Before and After editing the point inside the treble cut-away.

After you have edited your shape, finish the sketch. Enter the Extrude command and choose the scrap portion between the edge of the block and your sketch. Enter -1.75 for the distance so that the scrap will be removed and leave only your guitar shape.

Extrude feature menu, and after clicking Ok.



While most programs list the 3D features in the browser with the 2D sketches, AutoDesk Fusion 360 lists 3D features as icons across the bottom of the screen. The user hovers over each icon to see the feature’s name. For documentation purposes, it’s a good idea to use a descriptive name for a feature or sketch so that in the future one can make sense of the model creation rather than be stymied by “Sketch21” or “Extrusion8.”

**Quiz:**

Quiz Questions

1. \_\_\_\_\_ this function creates a 3D solid feature, or removes material
2. \_\_\_\_\_ this list on the left shows 2D sketches and workplanes
3. \_\_\_\_\_ the CAD term for an adjustable curve
4. \_\_\_\_\_ to see sketch as 2D
5. \_\_\_\_\_ point on curve
6. \_\_\_\_\_ how you adjust how curve flows to the next point
7. \_\_\_\_\_

Word list (more words than questions, that’s on purpose.)

1. Point
2. Spline
3. Browser
4. Extrude
5. Handle
6. Look At

* Include at least 10 quiz questions with answer key. (Questions must be Multiple Choice, and/or Matching).

**Reviewing Faculty Cohort Members:**

* Include at least two names and schools of reviewing faculty cohort members (refer to email list for faculty cohort member email addresses).