

CP467409

## **Fusion 360 Case study: Designing a musical effect with ECAD-MCAD Workflows**

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### **Learning Objectives**

- Discover the clear benefits of an integrated ECAD-MCAD workflow.
- Learn how to solve electronic design challenges by thinking in terms of combining simple circuit blocks to achieve an end goal.
- Learn how to design your product enclosures based on your electronic PCBs.
- Learn how to solve the typical challenges associated with integrated product design.

### **Description**

In this instructional demo, we'll show how Fusion 360 Electronics software can be maximized with Fusion 360 strong modeling tools to produce a distortion-effect pedal end to end. We'll discuss the design of the effect in detail, as well as how having an integrated workflow between electronics and mechanical tooling simplifies and expedites the design process. Throughout the demo, we will be highlighting the specific features in Fusion 360 that enabled us to go from idea to completed prototype.

## Speaker(s)



### **Jorge Garcia**

I'm a Community Manager for Fusion 360 Electronics/EAGLE, here at Autodesk. I have been working with EAGLE for 10 years. I earned my Bachelor's Degree in Electrical Engineering from Florida International University in 2008. I love electronics and building projects focusing on power and control applications. I have an affinity for embedded systems and programming.



### **Paul Sohi**

Paul Sohi is a British born, Iranian architect and industrial designer based in Oakland. Paul focusses on emerging technologies in the additive manufacturing field and their applications across multiple industries

## Key References for the Fusion Drive

- <https://www.electromash.com/tube-screamer-analysis> -This page provides an in-depth circuit analysis of the original Tubescreamer. It goes into the detailed calculations of components values and they do a great job visualizing the frequency responses of the circuit.
- <http://www.hollis.co.uk/john/circuits.html> - John has a lot of interesting circuits but I got the idea for being able to switch between hard and soft clipping from his Omnidrive design.
- <https://www.electronics-tutorials.ws/> -This site has an excellent series of beginner tutorials in Electronics. If electronics is completely new to you this is where I recommend starting.
- <https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-OVERVIEW> – This is the documentation for Fusion Electronics.

## Fusion Electronics

Please refer to the class recording to see how the commands are used practically. In all honesty this design could be autorouted since there are no critical traces and the frequencies involved are so low. However, that is not a good habit to get into so this design was manually routed and the key concepts of setting that up are covered in the main class presentation.

### Creating the Schematic

Almost all of the components come from EAGLE standard libraries. If there is any components you wish to glean and use for your own designs you can simply copy them to your personal libraries. Make sure you have the library you want to copy into open in Fusion 360 and then right click on the component in the schematic/board and select copy to library.

**TIP: On the schematic editor, make sure to always stick to a 0.1”(2.54mm) grid. This is the default. If you deviate from this grid setting, it will become difficult to establish your connections on the schematic. The same is true for the symbol editor in the library which is beyond the scope of this handout.**

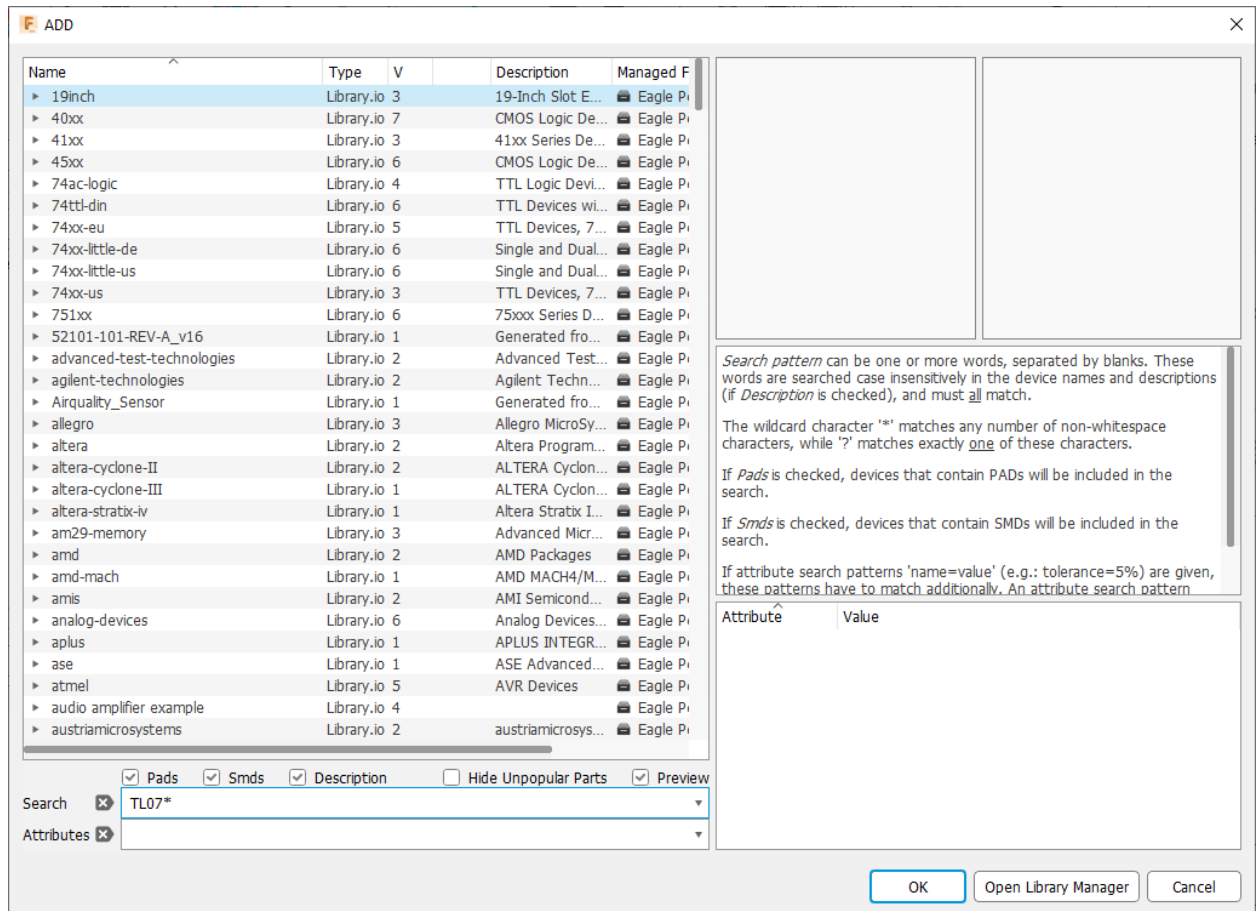
### Commands used when making the schematic

#### Net

This is one of the most important command in the schematic editor, it is used to connect your components together. The command snaps to your component pins which simplifies making the connections.

#### Add

This command allows you to choose components from the large number of Electronics/EAGLE libraries and bring them into your design.



*TIP: THE SEARCH IN THE ADD COMMAND USES THE \* AND ? CHARACTERS AS WILDCARDS. USE THESE TO GENERALIZE YOUR SEARCH*

## SPICE Simulator

The circuit originally started inside EAGLE and the simulation studies were done in that program. The process works the same in Fusion 360. The simulations used idealized models, but in a simple circuit like this one their accuracy is sufficient. See the class and dataset for the full details.

This video covers the setup in detail:

[https://www.youtube.com/watch?v=BCpYbKRfPa4&list=PL1rOC5j\\_Fyi46HXbWw8QFAT\\_EWRJ-hfpBa&index=4](https://www.youtube.com/watch?v=BCpYbKRfPa4&list=PL1rOC5j_Fyi46HXbWw8QFAT_EWRJ-hfpBa&index=4)

## Creating the PCB Layout

The layout is greatly simplified when you have a good schematic since all of the connections you defined there are automatically transferred to the layout. Some general rules of thumb for successful layout:

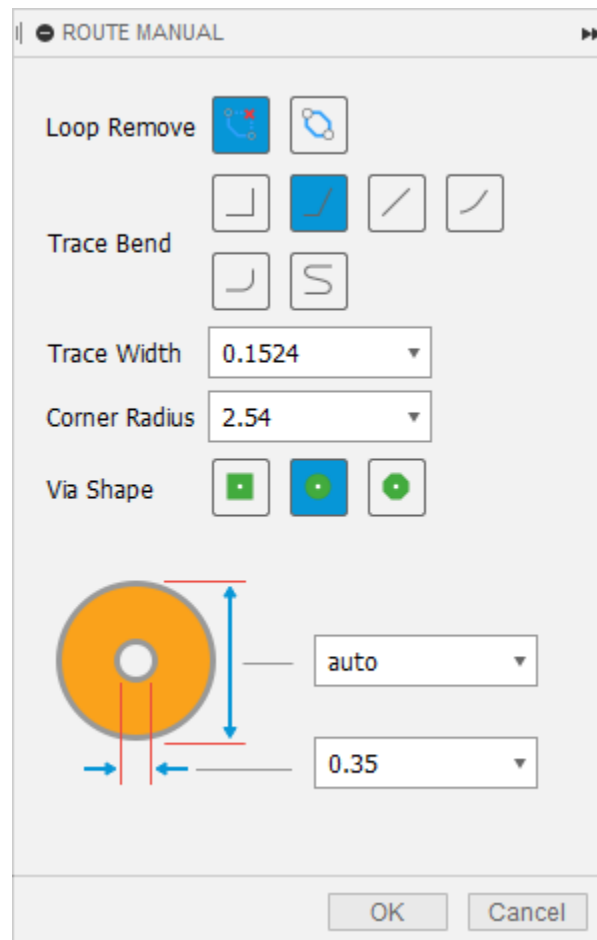
- Keep trace lengths short
- Try to route the circuit in functional groups this will allow you to make sure you can keep related components together which helps with the first point.
- Setup your design rules to match your board manufacturer early in the layout process to avoid unnecessary rework down the road.

## Commands used in the layout

### Ignore, Push, and Walkaround Violators

These environment modes are a powerful unique feature of Fusion 360 Electronics. In most ECAD tools these modes are tied to routing engine and are only available in that engine. However, in Fusion 360 Electronics these modes apply to the environment so ALL commands can leverage them. The demo in our class highlights the use of this feature.

### Route Command



This is one of the key commands of the PCB editor. With this command active, clicking on an airwire allows you to convert it to copper. The loop remove function in the image above is useful for rework or exploring alternative options since it allows you to redo traces without having to rip them up.

The route command has many options that simplify the routing task, for example if you press the backspace key you can undo the last segment you routed, continuously pressing the backspace key can take you all the way back to the start of the trace. You can also use the space bar to transition between layers and much more.

You can also benefit from several sub-commands that are especially helpful in more complex cases.

- Differential Pair – Signals are routed as differential pairs in order to improve their noise immunity along with other benefits. This mode allows you to route both members of the pair at the same time.
- Route Multiple – This mode as it's name implies allows you to route multiple traces together at the same time. This is especially helpful when routing busses.

Fusion Documentation covers this command in detail.

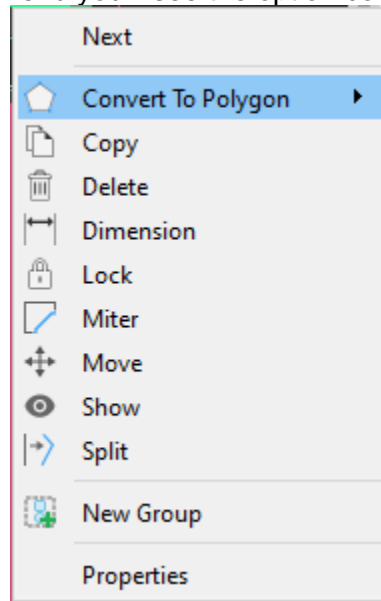
## Polygon

Polygons or copper pours are a vital part of printed circuit design. They provide many benefits including EMI shielding, low impedance return paths, heat dissipation and much more.

To create a polygon, simply click on the polygon icon and click wherever you want to place the vertices of the polygons. More often than not it's desirable to have a polygon's outline to follow the contour of the board. As long as the board outline has been correctly defined (closed contour, no gaps) you can create a polygon by right-clicking the outline.



In the context menu you'll see the option convert to polygon.



*TIP: THE CONVERT TO POLYGON OPTION LET'S YOU EITHER DUPLICATE THE OUTLINE ON ANOTHER LAYER OR REPLACE IT. FOR POLYGONS COPY IS THE BEST OPTION*

## CAM Export

Traditionally, generating manufacturing data for PCBs has been a complex task because every main feature of the board gets exported as it's own file. In an effort to simplify this process the CAM Export command analyzes your design and then selects most appropriate template to process your manufacturing data. In one click it will generate all the manufacturing files needs to manufacture your board. Besides the gerbers it generates drill files, a bill of materials, as well as a pick and place file.

## Mechanical Assembly

Please refer to the class recording to see these elements in action should any of the descriptions below be difficult to understand in application.

Mechanical assembly basics in fusion 360 are very straight forward. For working on more basic geometry, the general workflow will be

Create a sketch > create geometry from sketch > modify geometry to desired results

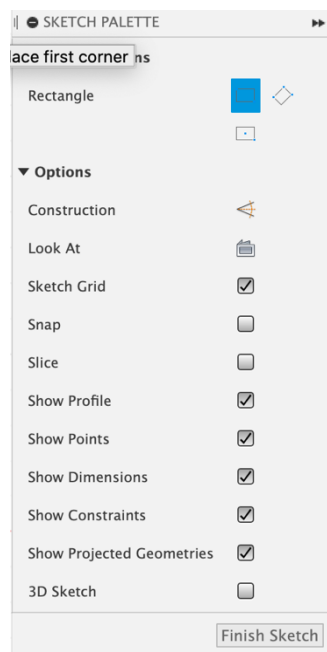
### Creating Sketches

When starting a sketch, you will be asked for what plane you wish to start using. Note that your sketch is bound to this plane unless you are using 3d sketches (which is not covered here)

The red and green lines denote your X and Y axis, where they cross, a small crosshair is present to show you the origin. We recommend you use start you first sketch of any design touching the origin in some capacity, as it will be useful later

### Tools used to make the sketch

#### Rectangle



2 point, 3 point and center!



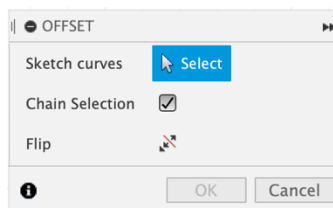
Rectangle can be used in 3 different ways, each giving you a few different options for how the rectangle is drawn, you can add the dimensions as you go along, or draw and approximation and then add the dimensions in afterward

### Sketch Dimension

Used to add dimensions to sketches in fusion 360, these can be used to directly apply a dimension to a sketch element, or used to reference one dimension to another. Rely on this tool for dimensioning over manually typing dimensions in for a faster workflow!

### Sketch offset

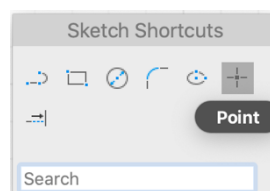
If you need to make concentric dimensions, either to create wall thicknesses or otherwise in Fusion 360, sketch offset will be useful. In our example we use this extensively for some internal relative dimensions. Offset can be used both for a chain of lines (like a rectangle) or single selection for offsetting a single line



*OFFSET SKETCH ALSO INCLUDES THE ABILITY TO FLIP THE DIRECTION*

### Point

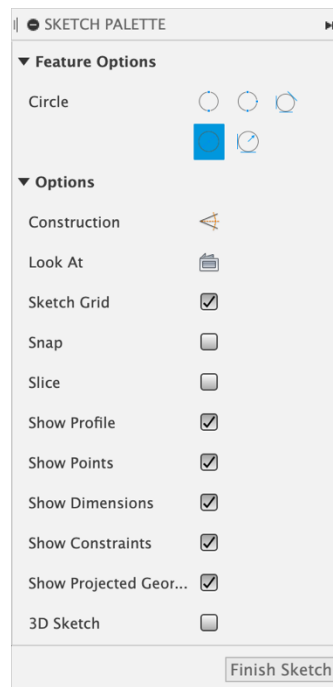
Points are really useful in the sketch environment as references for use both in 2d and 3d spaces. In our case we're using it to create references for drill holes in our 3d environment later



*A HANDY TIP IS TO PLACE THE POINT IN YOUR SHORTCUTS FOR EASIER ACCESS!*

## Circle

Circles are quite straightforward! Lots of drawing options again here, with the added benefit of being able to define by radius or diameter!

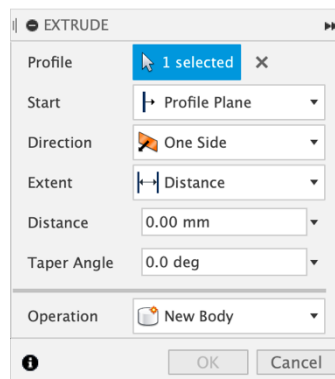


*THE CIRCLE TOOL, LIKE THE RECTANGLE TOOL INCLUDES A MULTITUDE OF WAYS TO BE USED*

## 3D Tools Used

### Extrude

One of the most powerful 3d tools, this will take 2D sketches and let you make them into 3D. Giving additional options for both symmetrical extrusions and 2 different sized extrusions from the same profile, you can use this tool to make the first elements of your geometry, and even includes handy tools like tapering to make it easier to prep castable or injection molding parts!



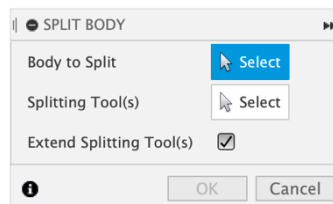
*TAPER, EXTRUDE, EXTRUDE WITH SYMMETRY, ALL IN ONE TOOL*

### Construction Plane

Working in 3D can be confusing at times. Finding reference points or planes to create the modifications you wish to can be difficult, which is where construction planes come in. These planes, axis, and point tools are used to generate references for other tools such as split both

### Split Body

Does exactly what it describes! Tell it what you want to split, and what you wish to use for splitting. In our case we used the construction plane to split the box up

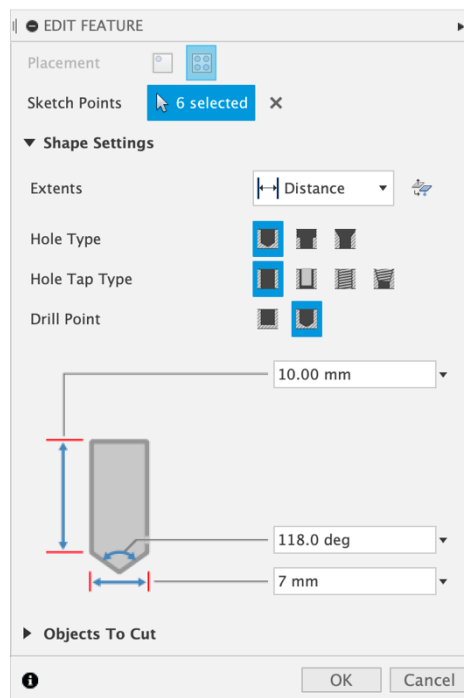


## Shell

Shell is an incredibly powerful tool that enables you to turn solid volumes into hollow ones! This can be used both to shell an object in its entirety, or to shell from a single face, creating an opening, like in a vase. Shelling can also be used on an internal volume, to take the outer limits to be the new inner wall and thicken outwards.

## Hole

The hole tool is used for creating machine holes in geometry, primarily for fasteners. The hole tool has considerations for this in mind, enabling you to create multiple holes in one function, as well as add the tapped thread in, counter sink and counter bore holes, as well as modelling a drill point for the hole, or leaving it flush!



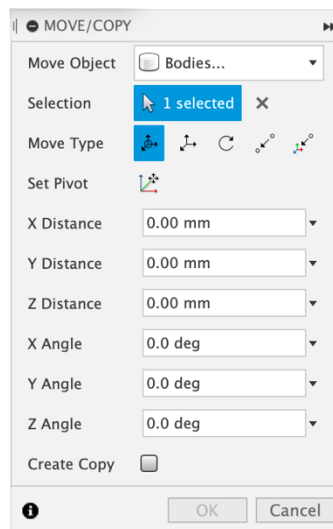
## Mirror

Mirror, like many of the tools above is very powerful. You can mirror negative space, objects, faces, and even features and functions from your timeline with it. We have used this to mirror a hole feature from one side to the other of the geometry in this exercise, making them associative, so when one is modified, the other is modified to reflect it

## Move

We used move extensively during this exercise, but worry not, it's very very simple to use. Move is designed to help you move any geometry or sketches around the space. You can use this to move and rotate, from a handful of options. The most commonly used method of moving is the free move tool, which will enable you to move your object locked to an axis, plane, or rotate about its axis.

Point to point will move object exactly as described. Pick a point on your object to move, and then pick a point anywhere else you want to move to. You will notice that when selecting a point, centers and mid points of objects will always be visible options, so long as a clear center point can be found (such as on squares or arcs)



## CONCLUSION

The goal of this document is to serve as “cheat sheet” of the most important commands that enable the sophisticated ECAD-MCAD workflows we demonstrated in the class. Fusion 360 is an exceptionally powerful tool and many documentation pages, tutorials and examples have been made about it. We hope that this class has given you a glimpse into what is possible with Fusion 360