CP500021

# Consumer Product Design Re-Imagined

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## Learning Objectives

- Learn Fusion 360 modeling techniques for plastic part design.
- Learn how to create an electronic design using Fusion 360's electronics workspace.
- Predict risks of manufacturing defects through integrated mold flow simulation.
- How to adapt to changes during the design process

## Description

Electronics can be difficult to design due to the reliance on various assembly components. Add to that the occasional need to swap between various software applications when designing specific components, such as the PCB, it can become difficult to get everything sync'd up and a more wholistic picture of the design. This class will showcase how a part designer can use Fusion 360 for everything from part design, PCB design, and even manufacturability.

## Speaker(s)

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| Kristen Kilroy<br>Product Marketing Mgr. | Kristen is a product marketing manager at Autodesk. Her focus is on simulation products, such as Fusion 360 simulation, Autodesk Moldflow™, and Autodesk CFD. |
| Edwin Robledo<br>Technical Marketing Mgr. | Ed is a Technical Marketing Manager at Autodesk. His focus is on Fusion 360 electronics, and Autodesk Eagle. |
| Garin Gardiner<br>Sr. Product Mgr. | Garin is a Product Manager at Autodesk. His primary focus is on Fusion 360. |

## Introduction

Electronics can be difficult to design due to the reliance on various assembly components. Add to that the occasional need to swap between various software applications when designing specific components, such as the PCB, it can become difficult to get everything sync'd up and a more wholistic picture of the design.

This is where we can utilize Autodesk Fusion 360. A part designer/engineer can use Fusion 360 for everything from part design, PCB design, and even manufacturability. And for times where collaboration is needed, the connectivity of designs within Fusion 360 provides quick turn-arounds for working with project stakeholders, while having confidence that we're working on the most up-to-date version of the design.

## Fusion 360 for Plastic Part Design

The Fusion 360 product team has created a series of tools focused on aiding part designers the ability to quickly add the often tedious structural and assembly components typical of injection molded parts to their designs. This toolset is intended to be added to the Product Design Extension of Fusion 360.

### Plastic Features

Designing plastic parts for their end use may require adding geometric features to the design for the purpose of improving structural strength and simplifying assembly. These features are often created manually, which can be repetitive. Automated workflows to add common plastic part features not only speeds up the design time, but it can also add consistency throughout the part design.

#### Boss Creation

A boss is a circular feature plastic molded parts often have which can be used to add support or assembly capabilities. The boss tool in Fusion 360 introduces an automated process for adding these features.

#### Web (rib) Creation

A web, often referred to as a rib, is a wall feature that is added to plastic part designs for added structural support. The web tool in Fusion 360 introduces an automated process for adding these features.

#### Adding Pre-sets

It's possible to create pre-set boss and web sizes to quickly duplicate identical features in a design. This is especially helpful in instances like that of an electronic consumer product that a PCB needs evenly supported across a part – rather than rebuilding the same dimensions over and over, creating the initial boss and saving it as a preset allows for one click to get the correctly dimensioned boss.

### Geometric Patterning

In addition to those plastic-specific design features, the patterning tool within the extension promotes flexibility to part designs through automating customized pattern features. These features are typically manually added to parts during a lengthy and tedious process, but within Fusion 360, the designer only needs to specify the surface it will live on, and then add values to presets for shape, sizing, and spacings, to customize as desired.

### Material-Dependent Sizing

Several of the tools within this extension are pre-loaded with links between the plastic material being used and the wall thickness of the feature (boss, web/rib, etc.). When changing the physical material, the wall thickness of those features automatically update according to pre-loaded recommendations.

This link to plastic material is important when looking at the end-product's quality. Injection molding plastic differs from other manufacturing methods in that melted material is added into a cavity at high speed and pressure and then cooled quickly before ejecting from the mold. This process adds stress to the polymer chemical makeup that can result in poor molded part quality if different regions of a part cool at different rates, which is why the part design needs to consider this when sizing part features.

In addition to the influence of geometric design, so modifying this initial thickness based on your original inputs will update the recommendations for sizing based on that material's predicted shrinkage rate.

More information on the concept of plastic shrinkage can be found at the following link:
Autodesk Simulation Blog: Dynamics that Affect the Shrinkage of Injection Molded Parts

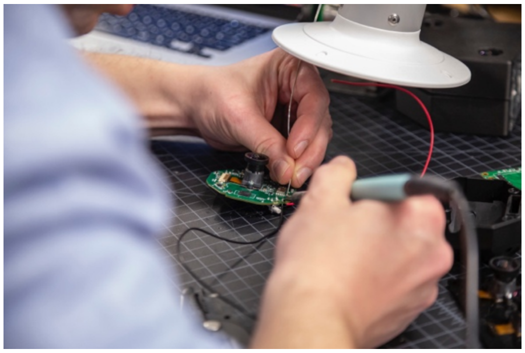


*Figure 1 Electronics*

## Fusion 360 Electronic Design

Intelligent products are currently part of our daily living. From the watch that woke you up this morning, the coffee maker that has your favorite beverage ready to be poured, or just checking your appointments on your phone or tablet.  Technology is surrounding us, and daily it's getting better and faster. Now more than ever, innovators like you need the tools that give you and your team the necessary confidence to design, collaborate and deliver in the shortest time possible.

### Electronic Workspace

The initial electronic workspace will allow us to create a new schematic or link an existing schematic. Linking a current schematic will enable us to re-purpose available designs in new products; that way, we do not have to re-invent proven technology. The same goes for the PCB.

## Electronic Document

You will access the primary electronics design space from the same design canvas used to access the mechanical workspace of your latest innovation. From the design workspace, you can launch a brand new schematic or load an existing one. Having the capacity of loading an existing schematic allow you and your team to use proven technology on future circuit board designs. Fusion 360 electronic document will monitor the annotation between the electronic schematic and circuit board. They are assuring you that the PCB will adopt changes made on the schematic. Change can include updating a component value, name, or changing connections.

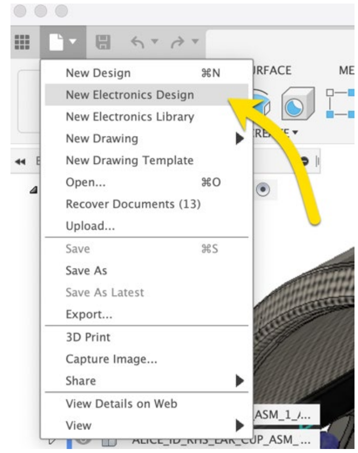



*Figure 2 Access primary electronic design space*

## Schematic Editor

The schematic editor workspace interface conveniently matches the mechanical workspace of Fusion 360. The schematic editor is surrounded by valuable panels that make it easy to navigate and find assets in your design. Components and signals will be centered and marked when selected using the design manager (Fig. 4).

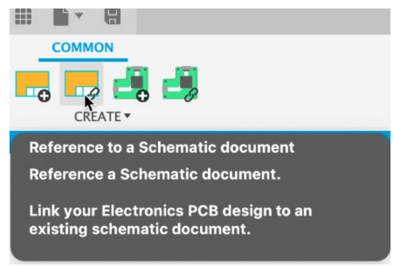



*Figure 3 Electronic Document*

Depending on the selection, you can make changes to the selected assets in the inspector panel. It is possible to filter the type of selection you want to by using the selected filter. The schematic is divided into multiple sections referred to as sheets. These sheets can be used to organize various electronic parts of your product, such as power, audio codec, Bluetooth, and others.

The schematic and PCB are linked; thus, the PCB updates in real-time when adding components to your schematic. Because of this connection, updating the design with different parts and connections doesn't affect your design time.

The building block for all electronic designs is the component libraries. Fusion 360 does have a team of librarians that are constantly adding and updating components.

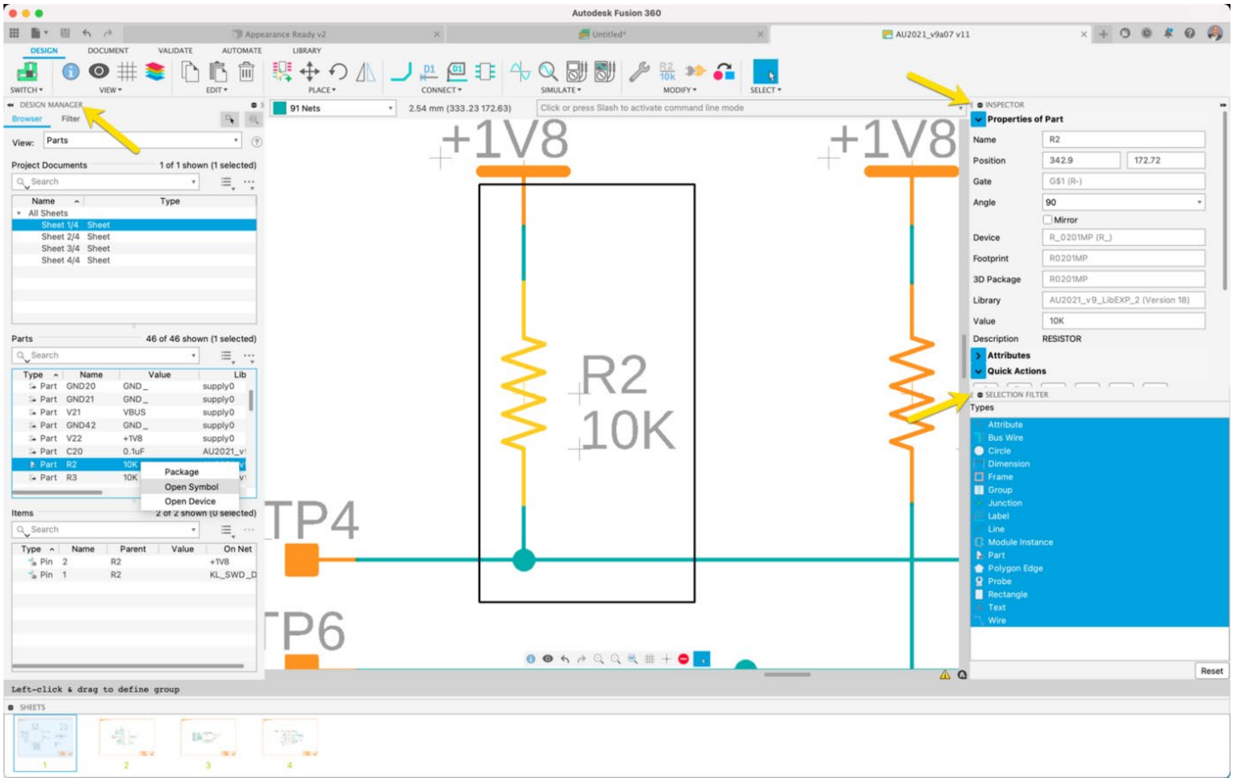



*Figure 4 Fusion 360 Schematic Workspace*

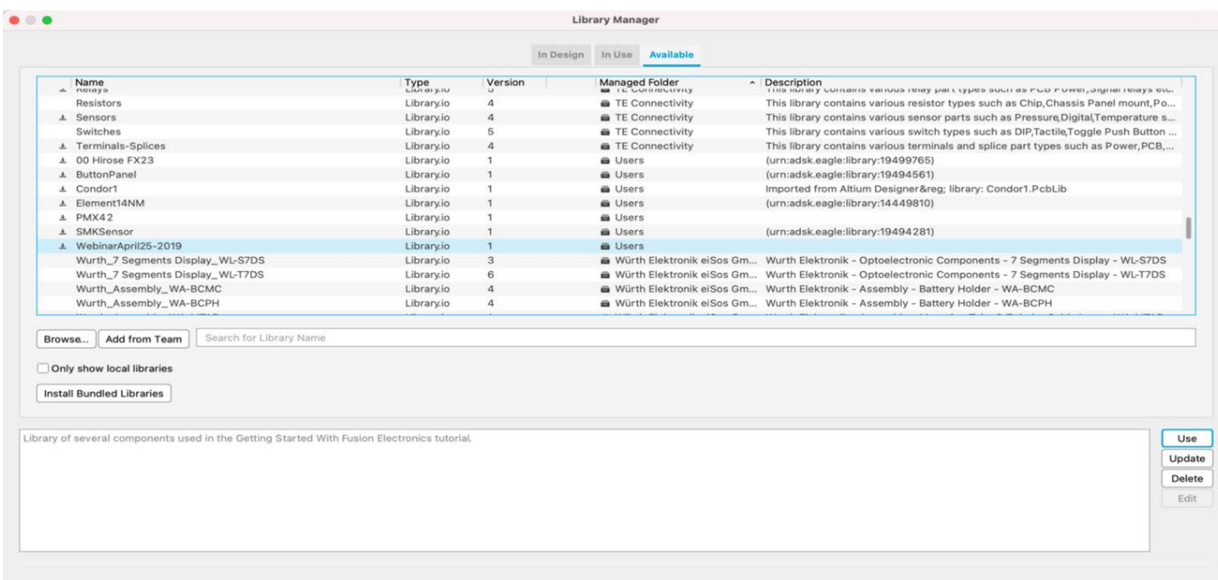



*Figure 5 Library Manager*

We have partner-up with distributors and manufacturers to make their library content is available through our repository. This content is managed by the distributor or component manufacturer, which guarantees that you will always have the latest components updates.

Use the ADD command to access our default library options. To access additional libraries made by our partners or shared by your team, select library manager from the add dialog box.

*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 set, 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.*

These libraries help optimize your design time, avoiding the need to create most manually, if not all, of the components within your design.

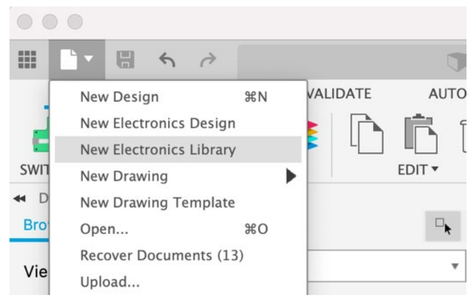



*Figure 6 Access library editor*

## Libraries

As you can see, from our vast repository of components will most probably have the part you need for your next innovation. If you don't find it, don't worry, Fusion 360 does include an easy-to-use library editor in which you will be able to build components in moments. After accessing the library editor (Fig. 6) you will notice that we continue to use the same interface as Fusion 360. Always working in a similar interface will make it very easy to navigate and operate in any available workspaces.

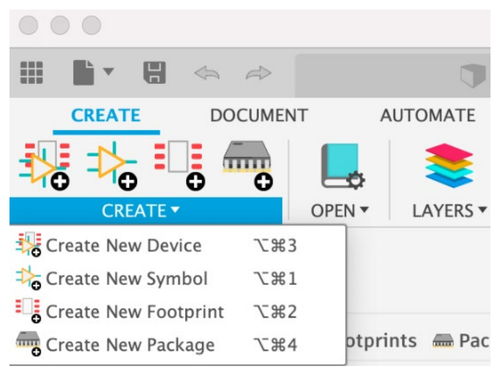



*Figure 7 Symbol creation*

The library editor is divided into four sections:

- Symbols: Logical representation of the component
- Footprint: Mechanical representation of the component
- Package: Mapping of 3D Model with Footprint
- Device: Unification of Symbol and Footprint

While creating a symbol, device, or footprint, it is possible to access the available libraries to re-use components that you might have already made us, your team, or our partners.

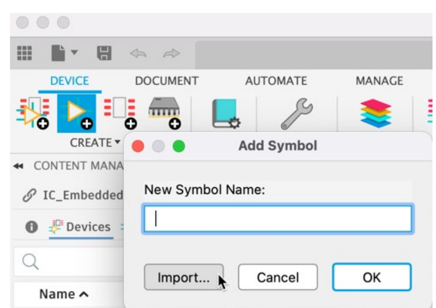



*Figure 8 Import symbol*

Mapping 3D models to footprints are done in the package editor. Many of our default components already have 3D models mapped. If you don't have the 3D model, not to worry, the package editor includes a package calculator that includes ALL IPC Components and some non-IPC components such as headers and PCB hardware.
After selecting the template based on the component specification sheet, you will enter the mechanical details for the part.

Confirming your values will create the 3D model and the footprint in the same step. All pertinent material be assigned to the component. This will help you get e-cooling simulation results as close as possible to actual life results.

The last step is to build a device (Fig. 10) with all symbol pins assigned to the footprint pads.

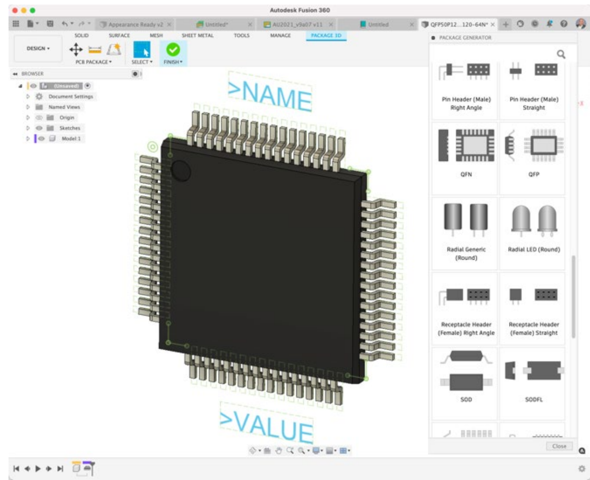



*Figure 9 IPC Compliant templates*

## Printed Circuit Board (PCB)

PCB workspace conveniently has a similar user interface as the schematic. With this familiarity, you don't have to learn a different set of commands nor their location.  The information panels are used in the same way we did in the schematic. For your convenience, the components and connections will appear next to an empty PCB default outline (Fig. 12). The PCB will have a different color than the rest of the workspace. You are making it easy to identify your layout area. The lines connecting the parts are the signals you defined in the schematic.

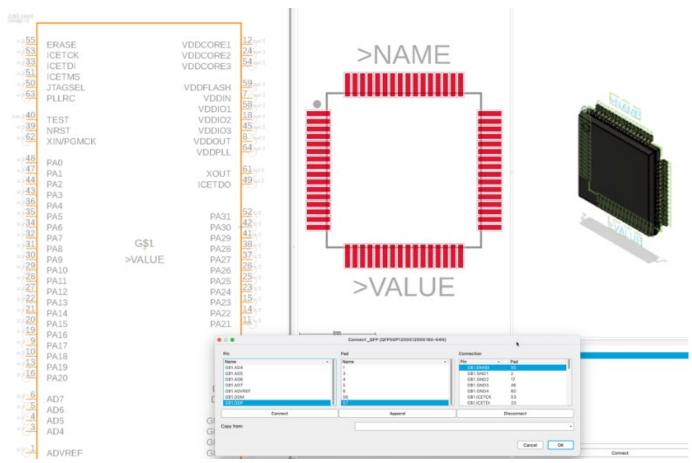



*Figure 10 Device Completed*

The next step is to move the components into the PCB area, but it is necessary to use the correct PCB outline before you do this. The PCB has been defined by the mechanical engineer in the Mechanical design Workspace. Traditionally, the Mechanical and Electronics PCB engineer would need to find a file format converted and then imported. With Fusion 360, you can select the sketch profile and adapt it to the PCB.

From the "Create" menu, select the Derive PCB from sketch. This option will allow you to make changes to the PCB sketch adopted by the PCB.

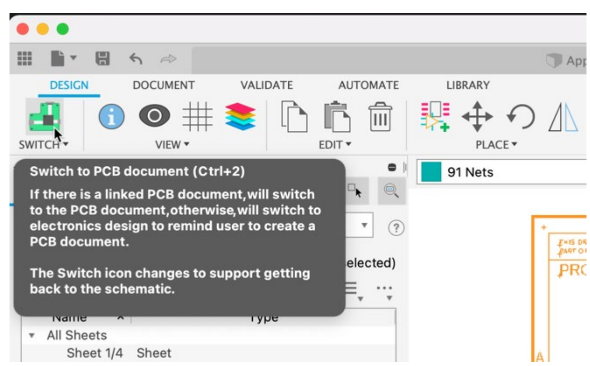



*Figure 11 Switch to Circuit Board*

After selecting the sketch profile, this will be converted to a 3DPCB. The 3DPCB will be the interface between the mechanical workspace and the PCB. With the 3DPCB generated, you can associate it with the Printed Circuit Board you are working on. In a moment, you will have the correct PCB profile shape next to all the components. This action instills a connection between the electronics workspace and the design workspace. Any changes made within the 3DPCB will now be shown on the 3D representation of the PCB shown in the design workspace.

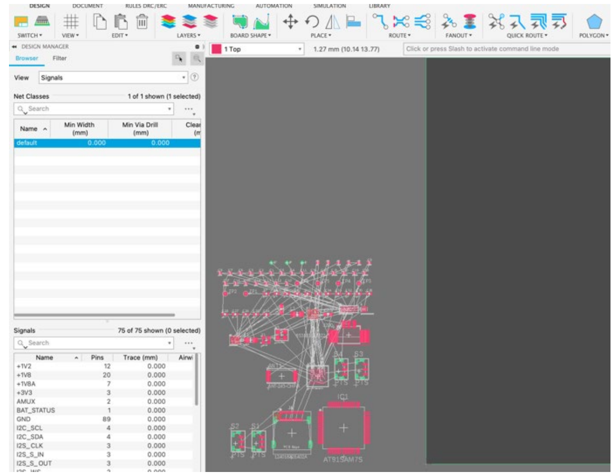


Figure 12 New PCB

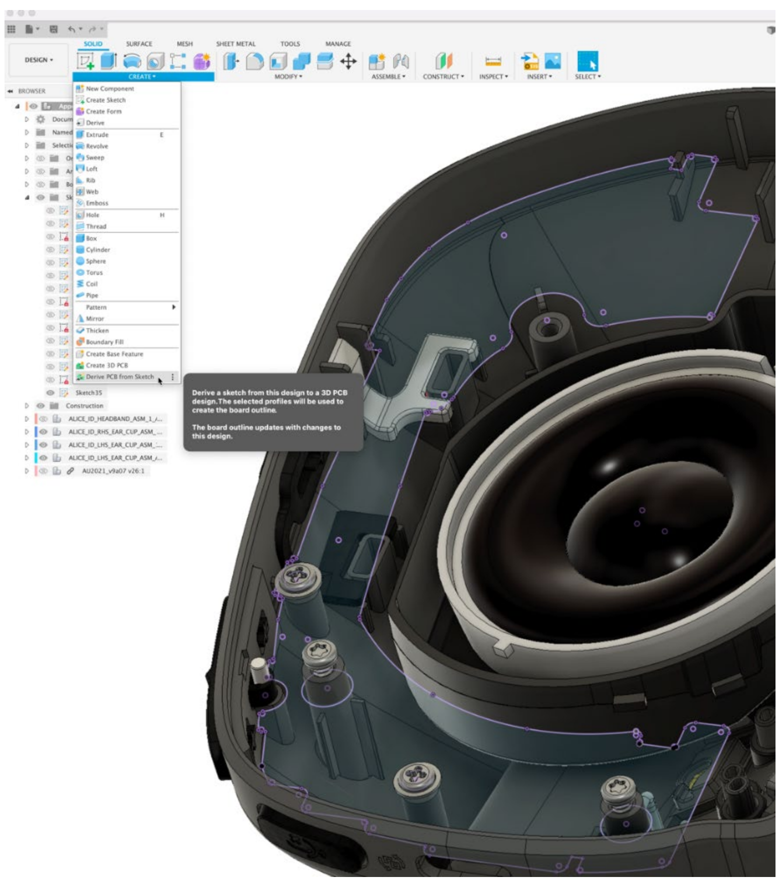



Figure 13 Capture PCB outline from design workspace

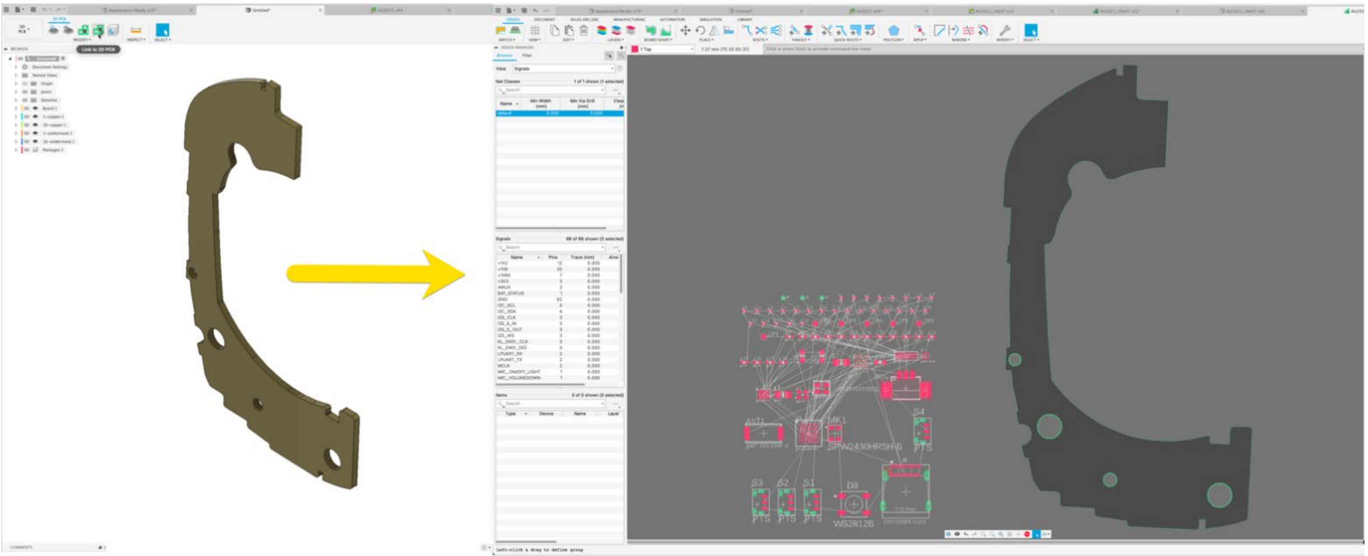


*Figure 15 Link the 3DPCB to PCB*

Being able to link the outline from the sketch profile to the PCB reduces the error-prone steps of converting files or drawing such a complicated outline in the PCB design workspace. This is a true paradigm shift regarding seamless electromechanical workflows.

Before you start adding parts to the PCB, you will need to set your manufacturing rules, usually referred to as your Design Rules Checks (DRC). These are the parameter you will need to establish to make sure your board can be manufactured.

### Design Rules

Fusion 360 has real-time alerts that warn the electronic engineer when electrical design rules may be present as the PCB is being designed. Many PCB manufactures provide Fusion 360 design rule files that can be loaded into your DRC environment.

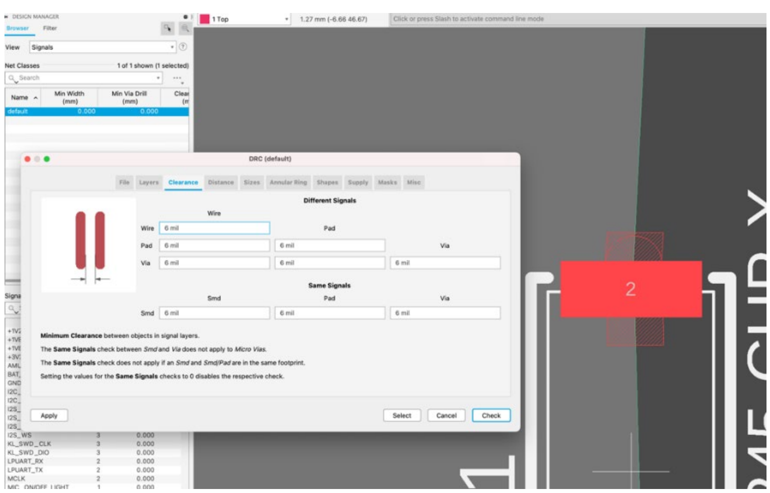


*Figure 164 DRC setup*

Visit your manufacture's site or contact them to see if they have the Design Rule file you can use. If not, they will provide the ideal DRC setting to reduce errors and design re-spins.

Fusion 360 provides a simple tabulated system to set up your design manufacturing parameter.

After placing all the components in the PCB outline, I will push for a 3D representation of the PCB. The 3DPCB will be generated after a few moments; now its time to insert the 3D model of the PCB into the enclosure. At this time, the mechanical engineer can verify any design errors such as interference or component placement.

The engineer can invoke the “Edit in Place” feature within the design workspace environment, which allows you to move electronic components in the design workspace. These changes will trigger alerts to the PCB designer by which he can adapt and make routing changes if necessary.

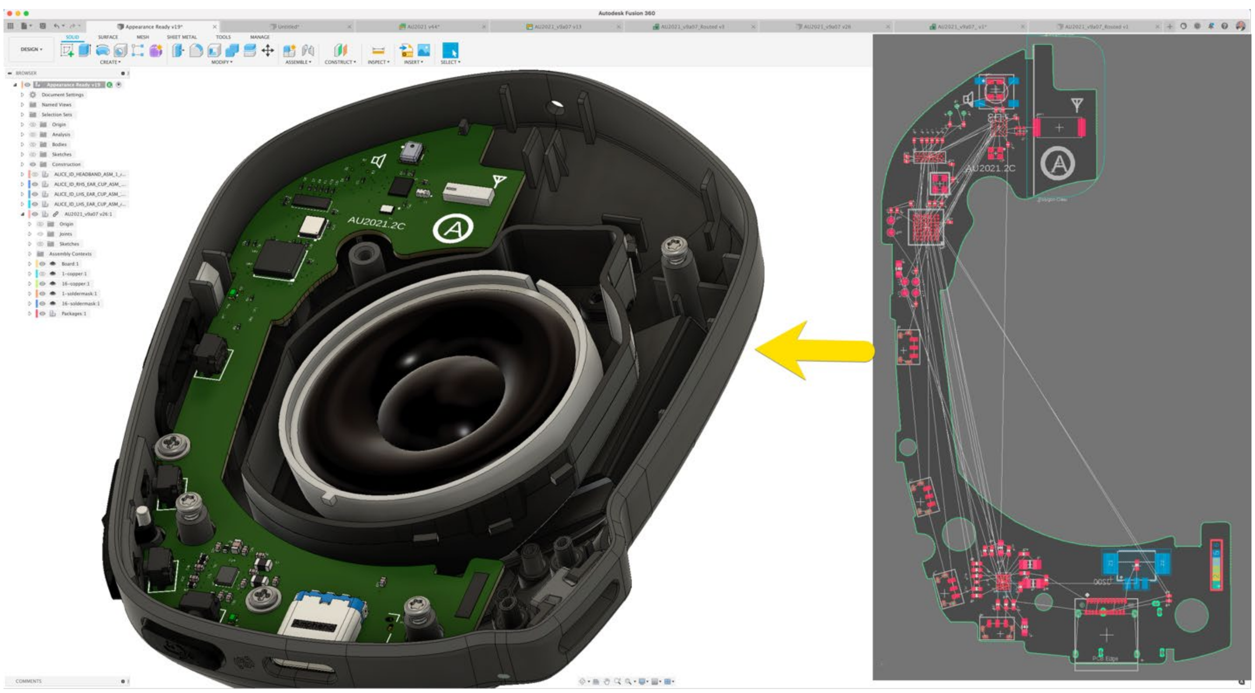



*Figure 17 Insert PCB in the enclosure before routing*

## Routing

After adopting all the changes made to the PCB in the design workspace, it is time to begin routing our signals. Fusion 360 provides a host of routing tools for complete manual or automated control. For quick path creation, the assisted manual routing tool gives the designer control by drawing the paths. The Autorouter option is a fully automatic algorithm that provides multiple results, where the designer just needs to select the option that works best for them. Using a combination of manual and automated routing lets you create the best design for your PCB, depending on the complexity of your design.

With the PCB completed routed, I will update the 3DPCB with all the adopted changes. The mechanical engineer will receive an alert about the PCB changes. After a few moments, he will now see the PCB fully routed in the enclosure.

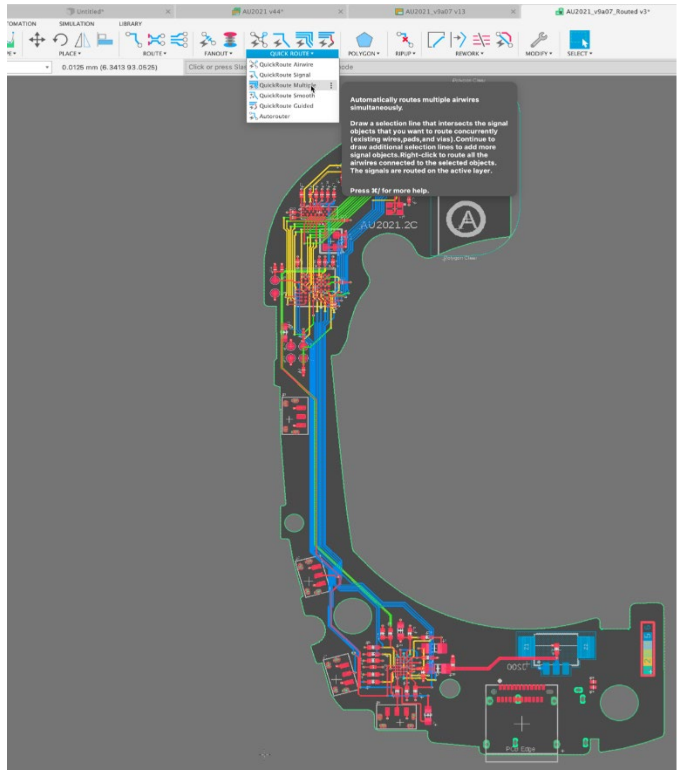
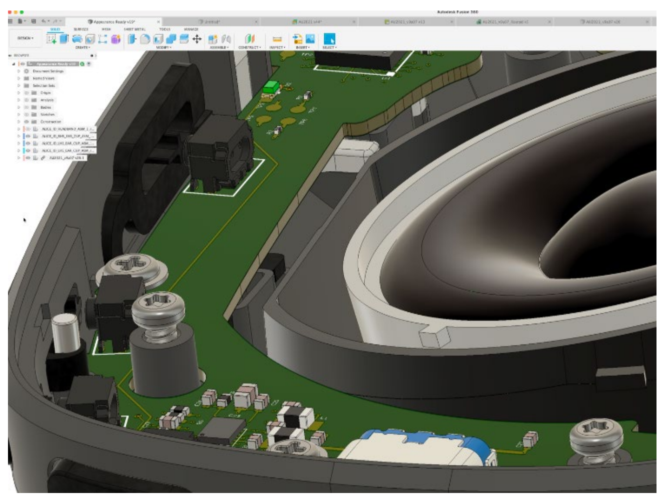


*Figure 18 Routed PCB*



## Fusion 360 Plastic Part Manufacturability

*Reference: Fusion 360 Help: Injection Molding Simulation (Concept)*
*Note: This feature is a preview. Previews provide early access to features while still in a conceptual stage. Learn more.*

The Injection Molding Simulation for Fusion 360 is a tool that enables the part designer to identify how design features will be influenced by an injection molding manufacturing process. The manufactured part quality of an injection molded plastic part is influenced by several variables.

From material to injection location to process temperatures (just naming a few) - each of these can influence if an acceptable part will be created. While working through designing a part, the part designer can use this simulation to experiment with part designs to identify problematic areas prior to the design moving forward in the design process, minimizing the need to rework injection molds.

### Injection Molding Simulation Setup

When inside of the Simulation workspace, the Injection Molding Simulation is almost immediately ready to solve. Default values of materials, injection location, and process temperatures, appear once you select a target part. This is helpful for those initial steps within part design, when some or all of those variables are still unknown. When more accurate results are desired, the Setup Summary is a list that guides you to changing these variables.

#### Target Body

*Reference: Fusion 360 Help: Target body*

In many cases, the part or product design stages include several assembly bodies or components. The Injection Molding Simulation is a part-only simulation tool, so it’s necessary to choose one body, the Target Body, to focus that study and results on. Additional assembly components can be simulated individually through creating new studies within the Simulation workspace.

#### Material

*Reference: Fusion 360 Help: Materials in a plastic injection molding study*

The material chosen for an injection molded plastic part is critical in determining the product’s end use performances, such as brittleness, flexibility, yield strength, chemical permeability, etc. It also is important to understand that different plastic materials process differently during the molding sequence. Because of this, adding the specific material type, and even the specific material grade, is critical to getting relatable simulation results to match toward the actual manufactured molding process.

Fusion 360 has a material library of over 11,000, tested material grades from sources like the material manufacturers and the Autodesk Material Testing Lab. When opening the study material menu, the first materials listed are generic material grades. This is because as stated previously, part designers may not have the specific material grade available at the time of initial simulations. The generic grades allow users to select a material with the general characteristics of that particular material family to gain insights close to the actual molding process (although it will likely still vary from the actual material used for molding that part).

Selecting any material, you can also review the specific material properties for added information on the material selected.

### Injection Location

*Reference: Fusion 360 Help: Injection locations*

The injection molding manufacturing method involves injecting a molten plastic material into a mold cavity at high speeds and pressures. The location where the material enters the cavity is another critical variable to the moldability of a part.

Multiple injection locations can be added, however this is typically for instances when it is difficult to fill the full cavity (such as larger or more complex parts). It is a best practice, however, to try to minimize the number of injection locations, as multiple can introduce weld lines in the material or other part defects.

It is also recommended to place the injection location on a surface that is not considered a “show” surface, or a region where it’s important to have minimal surface blemishes.

When the injection molding simulation is selected, Fusion 360 applies a general location for this injection point based on the centroid of the part. Although it can provide some insight into the moldability of the part, it is best practice to relocate this for improving accuracy when comparing to the end manufactured product. Selecting the injection location menu lets you select, delete, and move injection locations.

### Processing Conditions

*Reference: Fusion 360 Help: Process settings of a plastic injection molding study*

The process settings within an injection molding manufacturing method is heavily dependent on the material chosen and part geometry. After selecting a material, Fusion 360 automatically adds the recommended process temperatures, however they can be modified if looking to experiment with processability. The recommended temperatures are provided by the material manufacturers.

The injection time is also listed as a variable. This is because thermoplastic material is a non-Newtonian material, which simply means it flow differently as it is stressed. If forcing the material into the mold cavity at a quicker rate (lower injection time), it has a high likelihood to fill the cavity differently than that of a slower rate, and different injection times can affect the part quality based on how the selected material is affected by those shear stresses.

### Aesthetic Faces

*Reference: Fusion 360: Aesthetic faces*

With so many variables and potential for defects appearing on the plastic part during the molding process, Fusion 360 has an option for defining the aesthetic faces, or faces where there is a requirement to minimize the defects on that surface. This comes in handy when reviewing the simulation results.

## Simulation Results

Like other simulation capabilities in Fusion 360, the injection molding simulation uses cloud solving to allow for solving multiple studies. Cloning studies enables designers to see how part designs are influenced by different materials, gating locations, or process conditions. Once solved, the simulation results can be reviewed to identify need to adjust part designs while still able to, and rerun the simulation to validate the design changes. To help, Fusion 360 has two ways of viewing results: Guided Results and Standard Results.

### Guided Results

*Reference: Fusion 360 Help: Injection molding guided results*

Guided results for the injection molding simulation enable quick identification of problematic areas in terms of cavity Fill, Visual Defects, Warpage, and Flatness. In addition to the results, these plots also lead the design engineer to the Next Steps, describing why something is a problem and what to do to minimize or resolve it.

The Fill Confidence results show a red, yellow, green scale on the part for how easily those regions fill within the cavity with the given study settings.

The Visual Defects guided result shows the likelihood of sink marks and weld lines appearing on the molded part. Sink marks are a result of thick regions of material with thinner sections around it cooling at different rates (such as where bosses or ribs meet the part wall). Weld lines are areas where plastic material meet and can form a visible line on the surface of the part and may influence structural integrity at those areas. This is also where any Aesthetic faces defined during setup are shown and will indicate warnings if either of these defects are predicted to be on those surfaces.

The Warpage guided result shows the prediction of how internal stresses from the molding and cooling process influence deflection of the plastic part. Plastic material shrinks more than other manufacturing materials, leading to increased need to plan for this material shrinkage in the part design.

Tied to the Warpage results are the Flatness results. This option enables you to select a flat CAD face on the part to reference while looking at part warpage, ultimately showing if edges of that surface will be flat (within a tolerance) or not.

### Standard Results

*Reference: Fusion 360 Help: Injection molding results*

Additional results can be found when switching to the, “Results”, tab. These results provide a deeper insight into the simulation. The largest difference with these results when comparing with the Guided Results is that these results do not share the Next Steps recommendations.

There are several more results to dig into predicted part quality, including a summary of the process, and for further understanding of these, check out the Fusion Help.

Within the Standard Results, you'll notice you now have the ability to use comparison tools, surface probing, and section cuts, all to enable more intense analysis of the simulation predictions.

## Conclusion: Connecting these Concepts

Multiple disciplines of engineering are necessary for the development of any product. Therefore, each team will use different applications that best suit their design needs and hope they can easily collaborate. Traditional applications have a broken process since updates are not automatic and frequently require importing and exporting new versions, and occasionally CAD translating.

Fusion 360 solves this broken process by eliminating the need for file conversions, and environment changes that each project stakeholder (designer, mechanical engineer, electronics engineer, fabrication drawing, E-cooling solves, manufacturability, and much more).

The CAD design tools, electronic design tools, and injection molding simulation tools may be within different workspaces of Fusion 360, however each stakeholder has access to the most up-to-date version, and is notified as it is modified. They then have the ability to work from the same model so many moving pieces come together without delays from waiting on others. No need to import or export, and no model translating, making the design process streamlined for a quicker time to market which providing early insights to manufacturability of product designs.