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Electromechanical Product Design Reimagined

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Learning Objectives

- Learn Autodesk Fusion 360 modeling techniques for plastic part design.
- Learn how to create an electronic design using the Autodesk Fusion 360 electronics workspace.
- Learn about predicting risks of manufacturing defects through the Injection Molding Simulation feature of the Simulation Extension.
- Learn about adapting to collaborative design changes during a product design process.

Description

During Autodesk University 2021, we shared a developed workflow for how designers can bring data together through designing the part, the PCB (printed circuit board), and even how to manufacture the plastic components. With the complexity of electronic part designs, there are often various aspects that we need to consider in order for the part to both work and perform as expected. In this class, we'll demonstrate how the Autodesk Fusion 360 workflow has matured to incorporate mechanical engineering aspects to smooth this product design workflow. We'll use a sample product, the Autodesk University Digital Badge, to demo how to integrate mechanical design requirements and simulation into the product design process.

Speaker(s)

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Introduction

Over the past several years, as you look around stores and online shops, the number of consumer products available is becoming exponential. Seeing new versions of small (and large) electronics a few times a year is a regular occurrence. This is great if you are someone looking to keep up to date with the next phone… But when you're on the other side of the product – aka the designers and engineers who develop it, the strains of bringing these products to market so fast cannot be ignored.

Product development teams are challenged with faster and faster timelines, which recently has changes into agile workflows with design sprints to cut as much time out of the equation as possible. But how can they do this? Look at your phone, for example. The engineers/designers need to develop not only the basic housing of the device, but also plan for the electronics based on those designs and then develop the software it needs to run. These usually involve multiple teams, and various stakeholders who all need to know what the others are working on to move forward with the designs.

But, what happens when someone hits a roadblock? If the design engineer decides to make a mnor revision to where that power button is. Doesn't seem like it would be very disruptive to those other teams, so they may not initially discuss this with each team to save time and get the design finalized more quickly. Unfortunately for the engineer designing the PCB, if they aren't made aware of these so called, "minor changes", they might not easily identify the design update, and then when the product moves forward to the next stage, alarms raise and a delay is caused from having to rework his/her/their PCB design.

Autodesk Fusion 360 introduces a solution to this aspect of product design – a connected and collaborative approach to product design. With it already having been established as a powerful "Swiss-Army knife" of CAD software, Fusion 360 connects the product design engineers with the electrical engineers, manufacturing engineers, process enginers, and machinists. Marketing teams can also stay in the loop with pulling high-resolution rendered product imagery sooner in the design process.

Autodesk University Factory Experience

In this Autodesk University class, Ed and Kristen related the aspect of collaboration challenges that product development teams encounter with the collaboration capabilities behind Fusion 360. They used an example of a digital ID badge, which was used during the Autodesk University Factory Design Experience exhibit.

During the planning behind the scenes for the Factory Design Experience, an Autodesk team went through the agile product design process, using short sprints to approach each aspect of the product systematically and thoroughly. The following handout sections will have a strong focus on product capabilities and workflows to help those with similar product development cycle challenges identify ways to incorporate Fusion 360 into their workflow.

The Badge Model

After deciding to develop an ID badge for the Factory Experience, the team established goals to help guide them through its development. Primarily, it was to digitally display the conference attendee’s name, title, and company. Next, they wanted to add the functionality to manually connect two badges for an interactive component (ultimately deciding on a simple game). With those two goals established, they came up with the design below. Figure 1 is a refined version that needed several iterations until they landed on this, however the iterative process they encountered involved the collaboative workflow between the electronics engineer and the product development engineer.

Figure 1 Factory Experience Badge

The badge used in the Factory Experience is inspired by the Open Hardware project called the “Py-badge”, developed by a leading maker company called Adafruit. If you tinkered with electronics, I’m sure you have heard of them.

During the design development for the Factory Experience, we made many modifications to the original PCB design to get to what we have now. The processing power of the badge is the RP2040 Microcontroller. It’s 30 GPIO pins provide enough ports to control all the connector and TFT (thin film transistor) liquid crystal displays. The Raspberry PI MCU signature is its high performance, ease of use, and low cost.

The battery charger is based on the Microchip MCP73831 controller. This controller is ideal for our project since it has a low profile and is commonly used for portable projects that are ideal for our badge. Now that you have some basic understanding on the design we will be working on, let’s explore the Fusion 360 electronics schematic workspace.

The housing was developed based off of that base PCB that is being used. The screen is the focus, with the connection plug at the base of it so two badges can connect. The housing for this badge was determined to be injection molded plastic top and bottom halves, with the buttons and plug connection points being 3D printed components for easy customization.

Fusion 360 Collaboration

Electronics is a unique discipline since it requires much research before beginning to design the necessary product intelligence—a decision on which component to use and how to connect them based on the product environment. But circuit boards are not the completed product. It's one of the steps of the final process. The PCB layout needs to meet the product enclosure restriction, but there were occasions when the enclosure must meet PCB requirements. The primary collaboration between mechanical and electronic engineers is essential. The occasional need to swap between various software applications to accomplish a final product can be challenging and prone to many design errors.

Are there component collisions with the enclosure? Are the enclosure vents for aligned to assure proper operation of the component? Did the I file convert correctly? I'm I working with the correct version? This and many more factors need to be taken into consideration.

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

Fusion 360 Electronic Design

The demand for your electronics products to do much more is rising. Consumer products continue getting better and smaller, requiring designers to use the tools necessary to comply with this demand. Now more than ever, innovators like you need the tools that give you and your team the confidence to design, collaborate and deliver in the shortest time possible.

Electronic Workspace

The initial launch of Fusion 360 will land in the design workspace. Use the file pull-down menu to access the electronic workspace. From the created electronic document, it is possible to create a new schematic or link an existing schematic.

The benefit of having the option to link an existing schematic is that it will enable you 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.

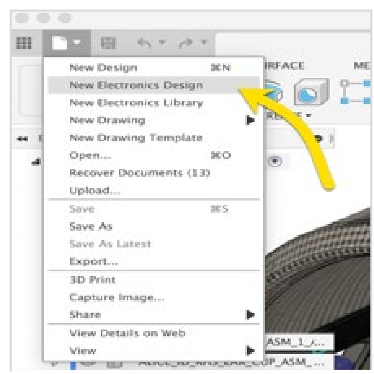

Figure 2 Access primary electronic design space

Electronic Document

Access the primary electronics design space from the same design canvas used to access the mechanical workspace. You can launch a brand-new schematic from the design workspace or load an existing one. Having the capacity to load an existing schematic allows 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 assure you that the PCB will adopt changes made to the schematic. Change can include updating a component value or name, deleting a component, or changing connections.

Figure 3 Electronic Document

Schematic Editor

Fusion 360 schematic editor workspace interface conveniently matches the mechanical workspace of Fusion 360. The workspace has valuable panels that make navigating and finding assets in your design easy. Components and signals will be centered and marked when selected using the Design Manager (Fig. 4).

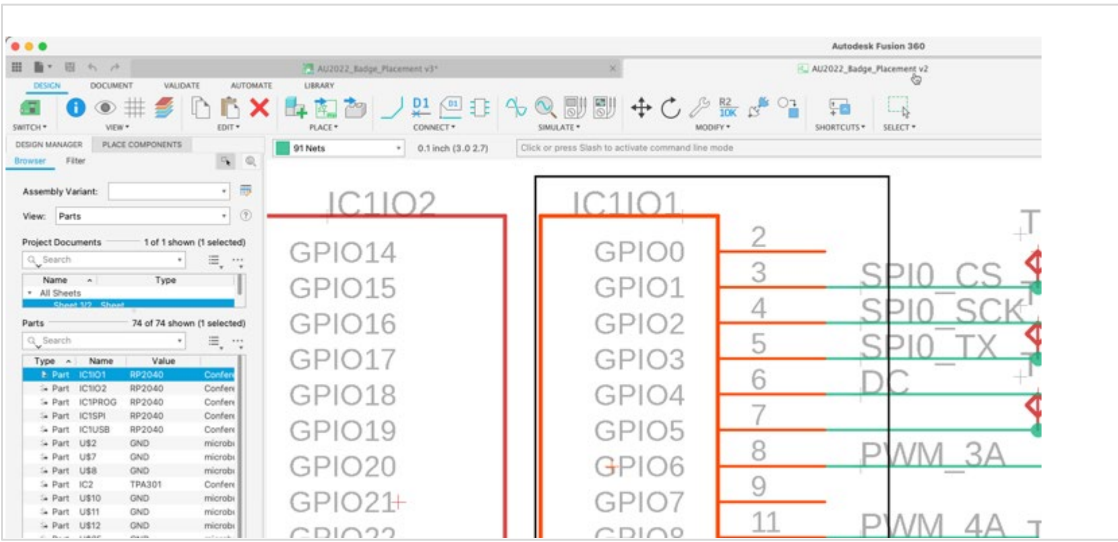

Figure 4 Schematic Editor / Design Manager

With the selection filter panel (Fig. 5), you can specify the type of assets you must select. The selection filter makes it easy for you to choose a polygon or a component in congested areas on the PCB or Schematic.

To organize your schematic into multiple areas of functionality, we strongly recommend that you either define permanent groups or use multiple schematic sheets. Example of the groups or sheets could be for power supply, 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.

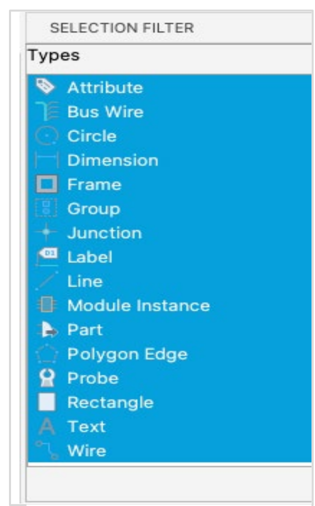

Figure 5 Selections Filter

The building block for all electronic designs is the component libraries. Fusion 360 has a team of librarians constantly adding and updating the Fusion 360 components repository. Use the Place Component panel to select the component needed for your design. Use the library manager to access additional libraries. We partner with distributors and manufacturers to make their library content available through our repository. This content is managed by the distributor or component manufacturer, guaranteeing you will always have the latest components updates.

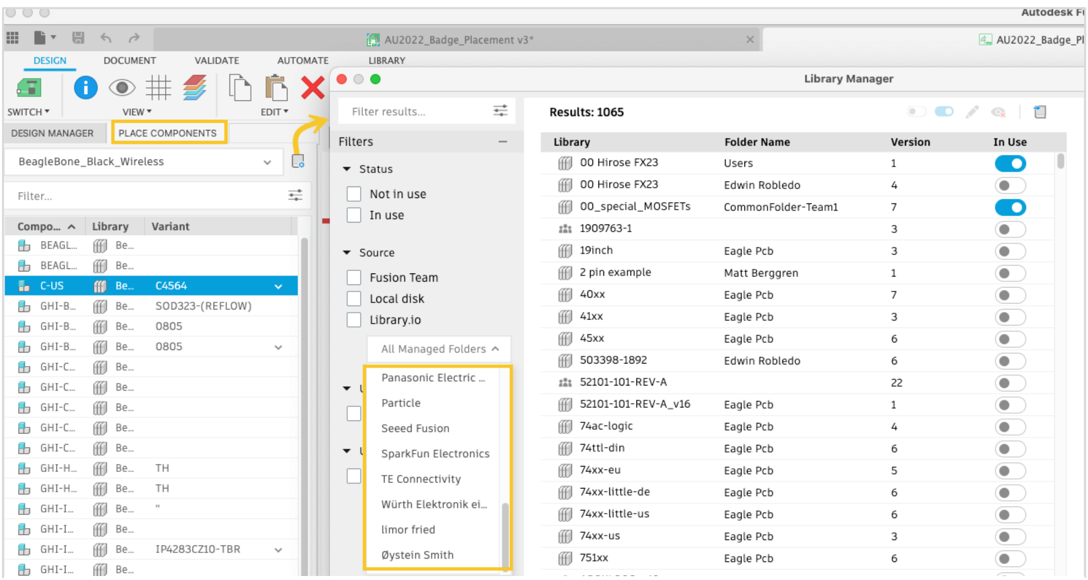

Figure 6 Place Component Panel and Library Manager

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 the required parts needed for your design. Make sure to thoroughly explore all the available library. Many manufacturers provide their assets in EAGLE library format. Fusion 360 is 100% component with EAGLE libraries and design files.

Libraries

As you can see, 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.

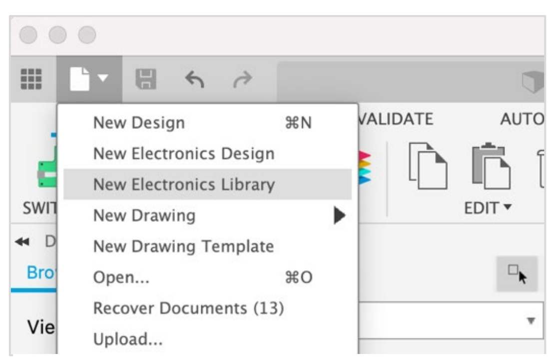

Figure 7 Access library editor

`The library editor is divided into four sections (Fig. 8):

- 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

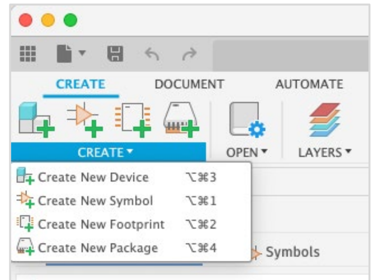

Figure 8 Library editors

While creating a symbol (Fig. 8), 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.

Mapping 3D models to footprints are accomplished in the package editor. Many of our components already have 3D models mapped. If you don't have the 3D model, not to worry, the package editor includes a package generator that includes ALL IPC Components templates and some non-IPC templates 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 (Fig. 9.)

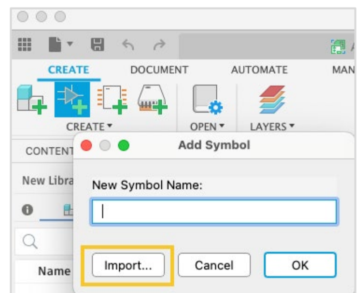

Figure 9 Import symbol

The last step is to assign the Schematic symbol pins to the footprint pads (Fig. 10). With the component completed, you can add it to your schematic by accessing it from the Component Place Panel. Begin adding components to your design by assessing the Place Component.

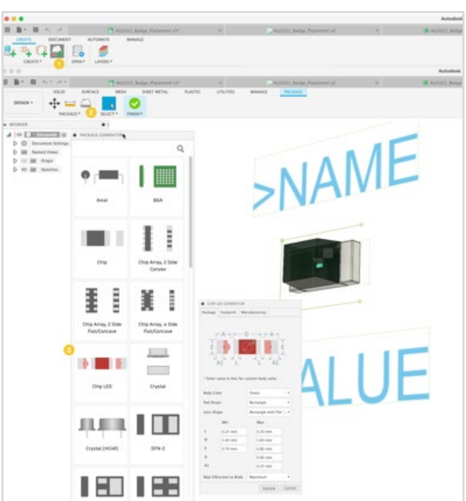

Figure 10: Realitu

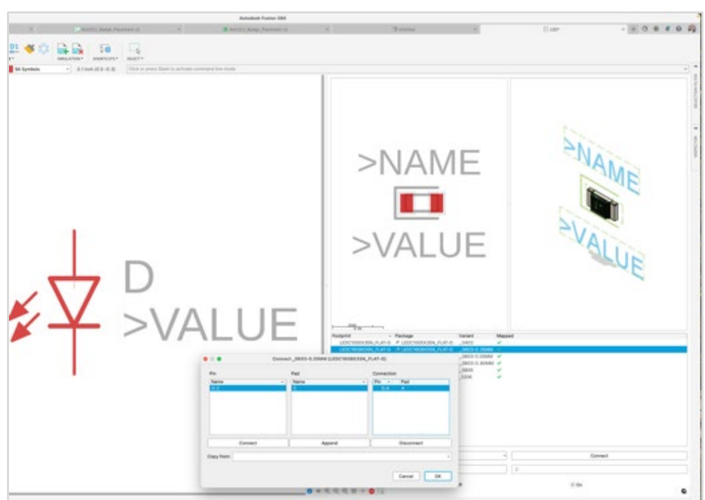

Figure 10 Device Completed

Printed Circuit Board (PCB)

From the Tabs, select the PCB workspace. Notice that is 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 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.

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, scroll to the Create PCB option. From here. You will select an Associated PCB or an independent PCB. The associated option will link the selected profile and the PCB. Therefore, changes to the enclosure outline used to reference the PCB profile will send an alert to the PCB designer with the option of adopting them. Very convenient option during the initial stages of product development.

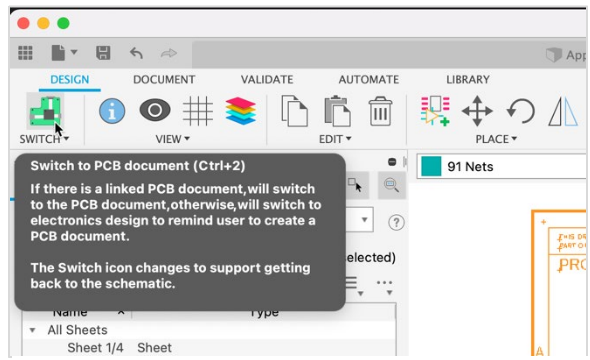

Figure 11 Switch to Circuit Board

The Create PCB dialog will allow you to select the profiles and define what will be used as the outline origin in the PCB workspace.

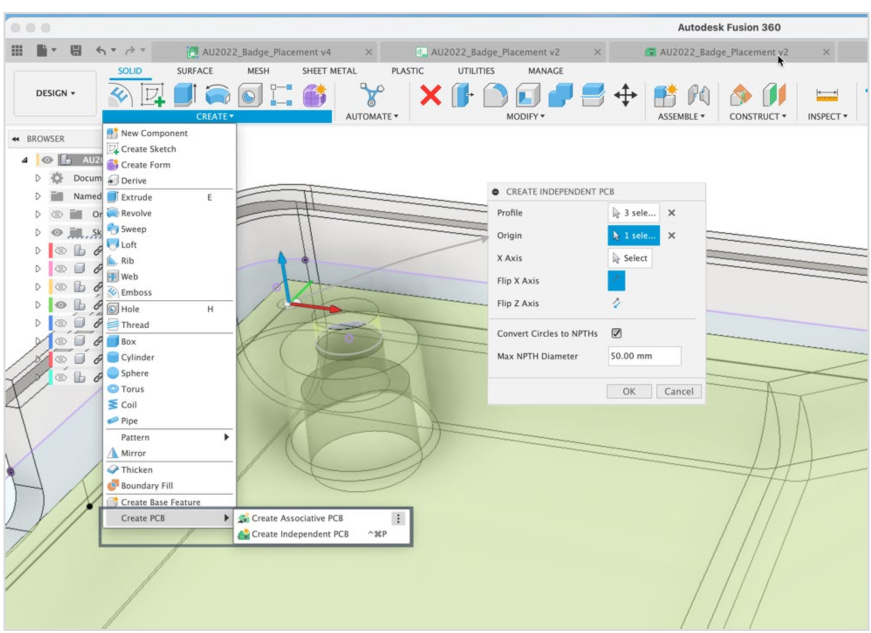

Figure 12 Link the 3DPCB to PCB

After selecting the sketch profile and select OK, notice that the outline has open the 3D PCB Workspace. This workspace will be the link between the mechanical workspace and the PCB Workspace.

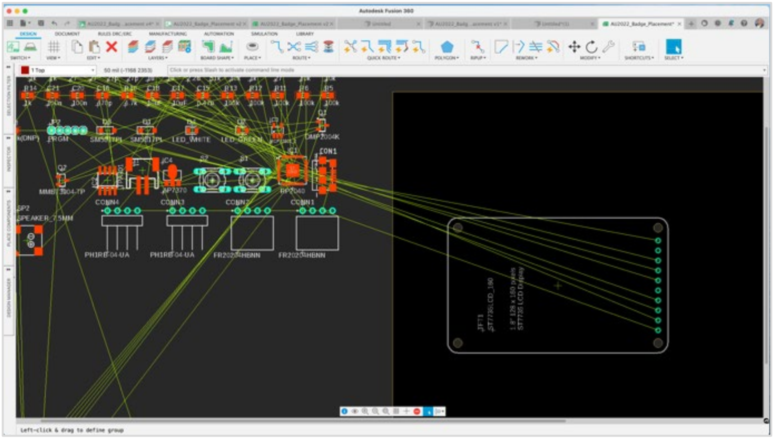

Figure 13 New PCB

With the 3DPCB generated, you can link it with the Printed Circuit Board you are working on. In a moment, you will have the correct PCB profile next to all the components. This action instills a connection between the electronics workspace and the design workspace. Changes made within the 3DPCB will reflected on the 3D model will update the PCB.

Linking the outline from the sketch profile to the PCB reduces the error-prone steps of converting files or manually 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 or adopt 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 manufacturers provide Fusion 360 design rule files that can be loaded into your DRC environment.

Visit your manufacturer'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.

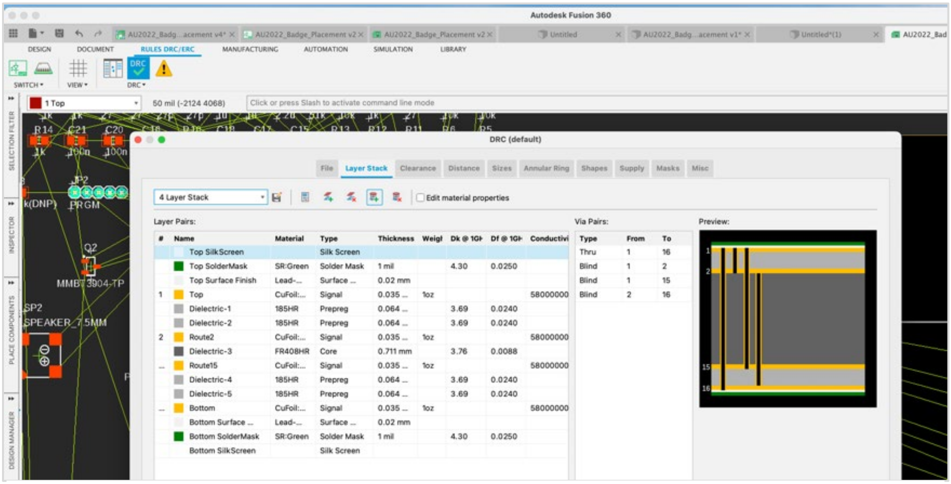

Figure 14 DRC setup

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; this is a functional 3D representation of the PCB. The vias, pads and holes are all constructed from extruded bodies with materials assigned. This is paramount since it will take it into consideration when doing an e-cooling simulation.

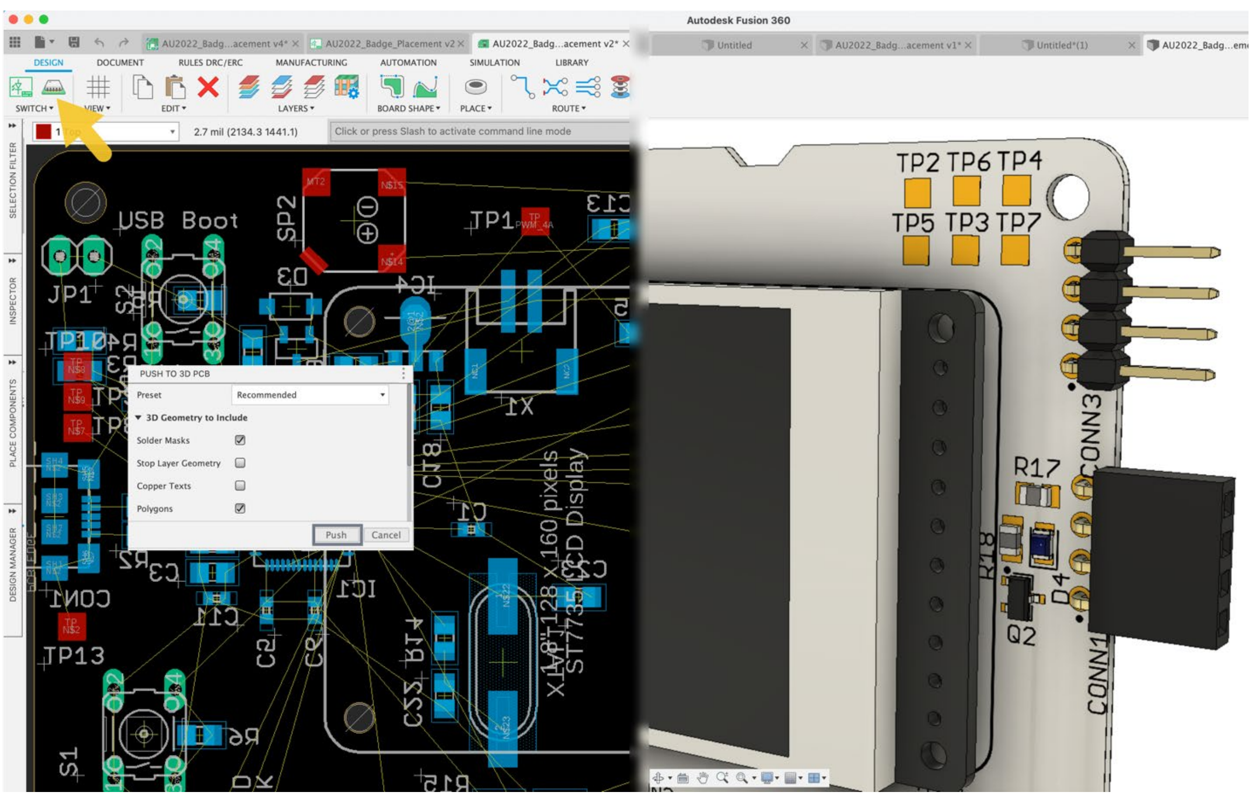

Figure 15 Push to 3D PCB

Modifications, such as moving components from location or layer, can be done in the 3DPCB workspace. These changes will be reflected on the PCB in a matter of moments. With a more realistic vision of the PCB, it makes it easier to verify component placement and adapt when necessary.

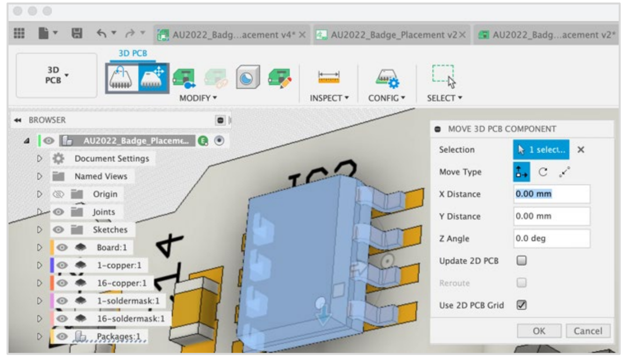

Figure 16 Move components on 3D PCB

After making all the necessary changes that benefit the PCB layout, it's a good time to inform the mechanical engineer to insert the 3D model into the enclosure. The mechanical engineer can verify if the PCB fits correctly into the enclosure and check for design conflicts, such as interference or collisions.

Edit In Place

After inserting the 3D model of the PCB into the enclosure, it is possible to check for any. Interferences. Also, make sure that your connectors or displays are correctly aligned. If they are not, use the EDIT in place option to make the changes.

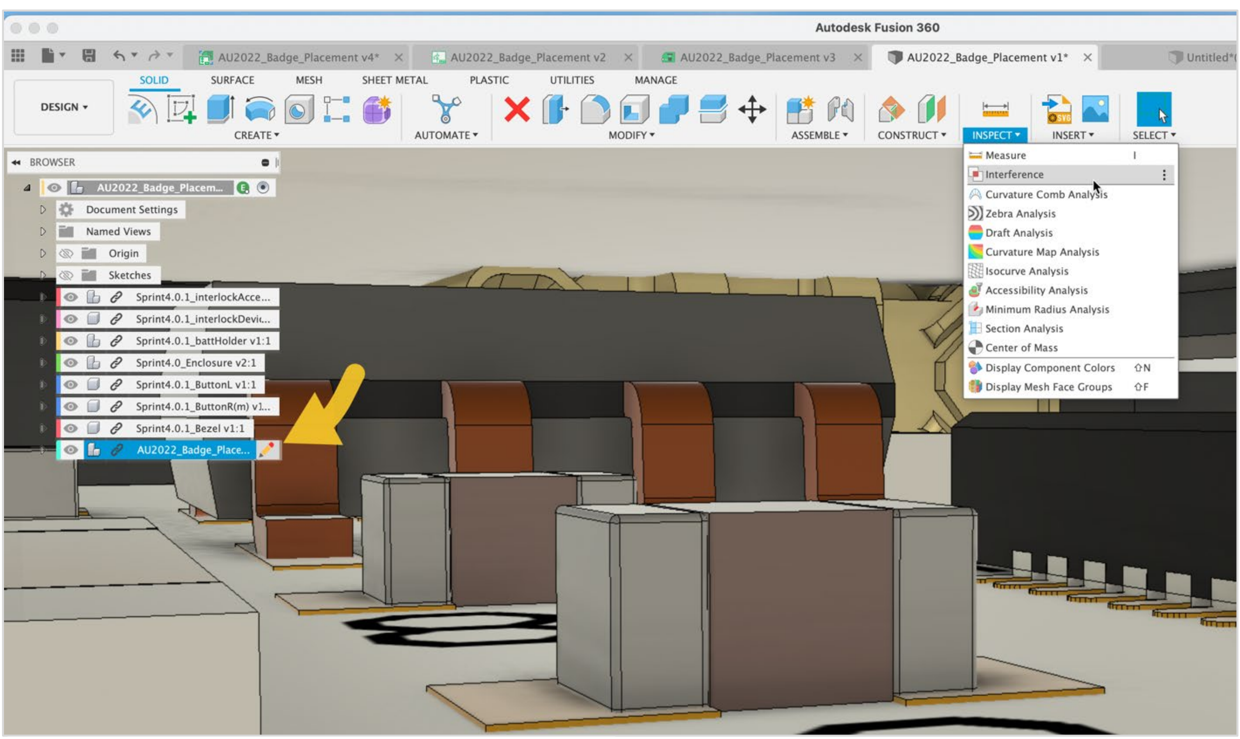

Figure 17 Use edit in place for interferences

One of the highlights of Fusion 360 is that moving parts on the PCB layout can be performed in the 2D or 3D model of your PCB. Using the Edit in Place features allows you to modify the placement or change the layer of components used on the PCB. The 2D designer will receive an alert to accept these changes. This collaboration between the electronic and mechanical engineers fully completes the eternal gap that made it so difficult to collaborate.

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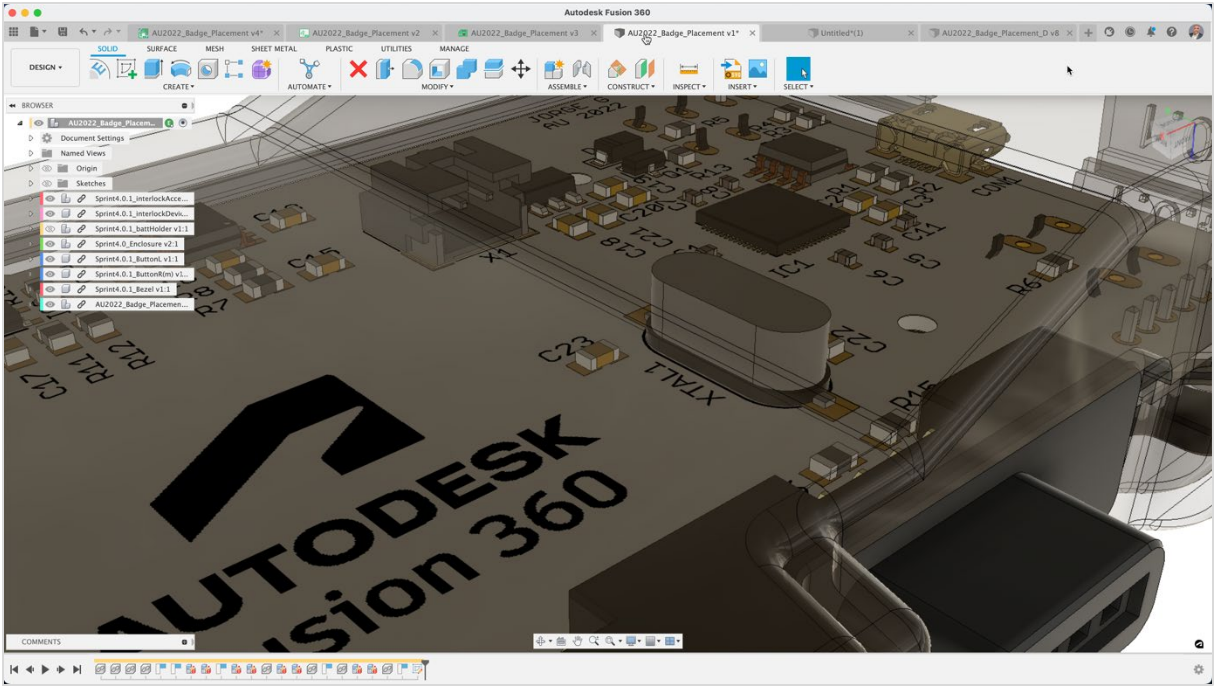


Figure 18 Insert PCB in the enclosure before routing

Routing

After adopting all the changes made to the PCB in the design workspace, it is time to begin routing our signals. Let me tell you a few key tips you should follow before you begin routing your board:

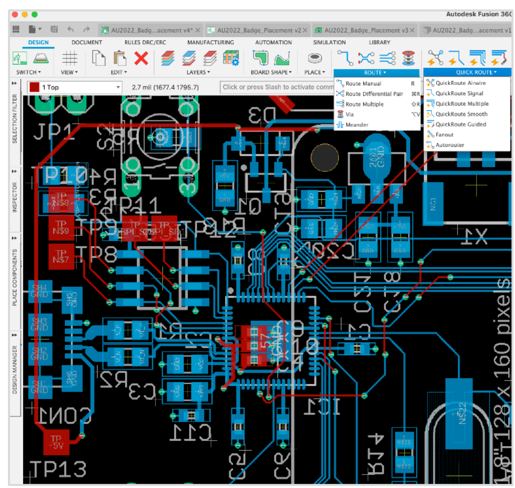
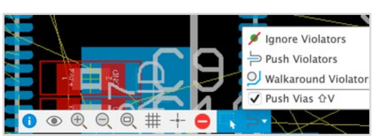


Figure 19 Routed PCB

1. Before you begin routing make sure you have updated your DRC preference with those recommended by the manufacturers. This will guarantee you get your boards faster in with fewer possibilities of errors.
2. Take advantage of our violator modes.



a. Ignore violators – this is a fully manual routing tool that will allow violations but will trigger DRC alerts.
b. Push Violators – this feature will move traces and vias to accommodate components or routed traces.
c. Walkaround Violator – is primarily used for routing traces and avoiding DRC violations.

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. 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 fully routed, let's move over to the design workspace and have the design update with the routed version. Just as the EE received an alert earlier now 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.

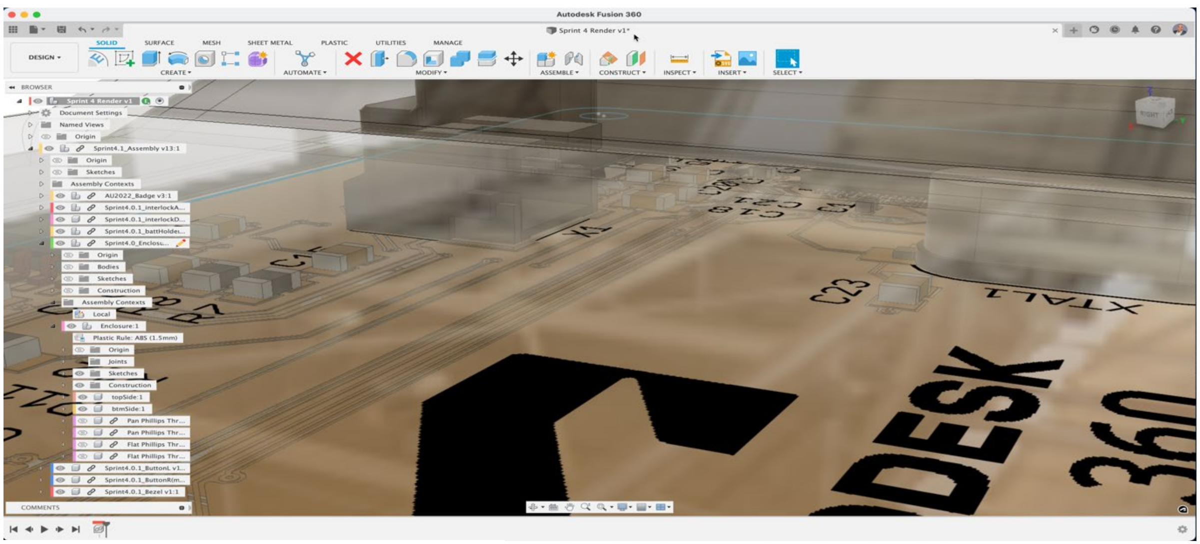


Figure 20 Routed PCB

Outputs

Ansys

If you have Ansys simulation tools installed, you can export directly to Ansys from the Simulation tab. With Ansys, you have a host of simulation capabilities which include Signal Integrity, Power Integrity, EMI analysis, and others.

ODB++

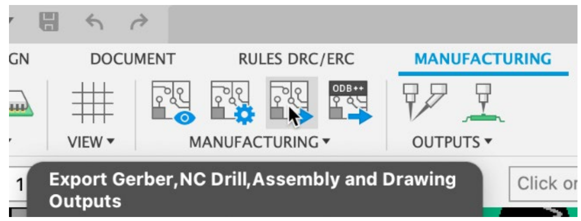
ODB ++ is a format for data exchange between CAD, CAM, and assembly in the development and manufacture of electronic equipment. ODB ++ is, compared to the Gerber format, a new data format that helps prevent many errors and data misinterpretation, especially in printed circuit board manufacturing. You can access this format export from the Manufacturing Tab.

Gerber

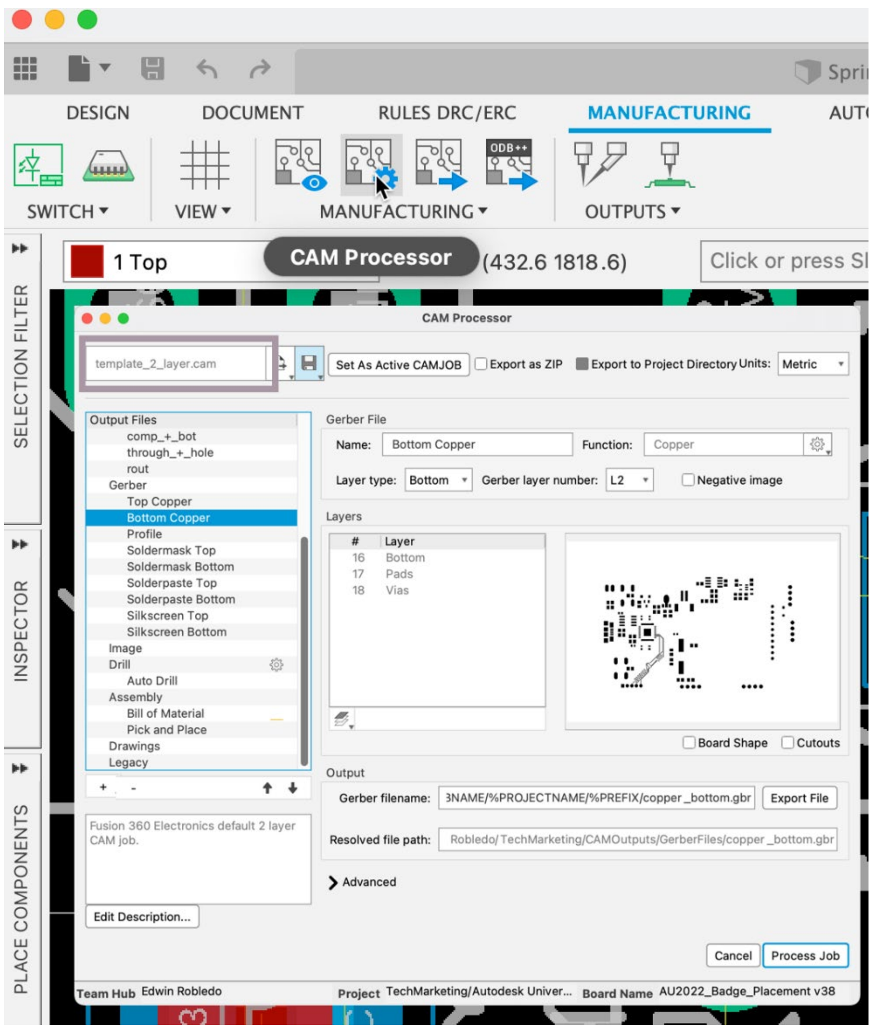
It is an open ASCII vector format for printed circuit board (PCB) designs.[1] It is the de facto standard used by PCB industry software to describe the printed circuit board images: copper layers, solder mask, legend, drill data, etc.

CAM Processor

The CAM Processor is going to be used to deploy your Gerber files. There are a few ways to access the CAM Processor:



- *One Click Solution* - This will automatically load your export template based on your DRC layer stackup. If you are working with a 4-layer board, then the CAM job template for four layers will load.
- *CAM Processor* - The Cam Processor provides the flexibility of making changes to the output. It is possible to change the layer configuration or customize the amount of layers into a section.



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Fusion 360 Plastic Part Manufacturability

Reference: Fusion 360 Help: Injection Molding Simulation (Concept)

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.

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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.

Fusion 360 Electronics Cooling

Technical Preview within Fusion 360
Reference: Fusion 360 Help: Electronics Cooling

The Fusion 360 electronics cooling simuation within the simulation workspace is a newer technogy that the Fusion 360 developers collaborated with the Autodesk CFD simulation team to generate. Although it is currently (as of this handout's creation) still a technical preview, the calculation methods are par for how Autodesk CFD calculates thermal influences of PCB heat distributions, including airflow.

This simulation tool allows users to access insights into the impacts of placement of various features and components, and the setup is as straight forward as the other simulations within Fusion 360, enabling fast adoption for designers/mechanical/electrical engineers.

Electronic Cooling Simulation Setup

In the simulation workspace, you can find the Electronic Cooling study type. This can be run on individual components (such as the PCB), or the full assembly. Using the full assembly helps to show how your current airflow through the parts will influence the thermal characteristics of the PCB components. Simply hiding the assembly pieces you aren't interested in will exclude them from the simulation.

Because the 3D PCB was generated using the electronic component libraries within Fusion 360, when we create this electronics cooling simulation study, the materials for the PCB components are all automatically added. This just leaves the assignment of the other assembly components (if applicable), which may have already been assigned materials within the design workspace.

Next, follow the prompts to setup the thermal loads on the appropriate 3D PCB components. During a workflow such as the one demonstrated during the AU presentation, Edwin was the electronics expert. Because of this, I asked him to assign those thermal loads so he can accurately represent the heat distribution. Either of us are then able to proceed with the remainder of the setup, allowing him to get to other projects while I continued to digitally prototype the part.

Cooling loads can be applied in situations where there are heat sinks or fans – in this example, we did not have any so they were left out, but more advanced PCB boards may have one or several of these to aid with relieving heat build-ups in critical areas.

The ambient temperature can be assigned to more closely match the real-world environment the product is meant to perform within.

The Critical Temperatures settings are used to help with component failures of PCB parts through manual assignment of failure temperatures. This takes the simulation one step forward to help designers identify how impactful a heat spot may be within their PCB.

Once finished, the simulation pre-check can be run to make sure all necessary components are assigned for a simulation to run. If all is OK, initiate the solve through cloud solving.

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.