

CP125430

Integrated ECAD, Modeling, and CAM in EAGLE/Fusion 360

Paul Clauss

Technical Support Specialist, Fusion 360
Autodesk, Inc.

Edwin Robledo

Product Support Specialist, EAGLE
Autodesk, Inc.

Learning Objectives

- Discover the advantages of designing with multidisciplinary workflows in Fusion 360 and EAGLE.
- Learn how to save money, effort, and time by finding and addressing design problems before beginning manufacturing processes.
- Introduce various electrical/mechanical design situations and how to approach them using the Fusion 360 and EAGLE integrated workflows.
- Understand the communication benefits of fully integrated electrical and mechanical design processes.

Description

Inspired by the integrated product-development capabilities in Fusion 360, this class will walk through the advantages of using the multidisciplinary workspaces (ECAD, modeling, and CAM) of the product innovation platform. You will learn how to save time and money while avoiding frustration by rapidly exploring iterations of a design to optimize a printed circuit board (PCB) and mechanical components for an electromechanical assembly.

To fully encompass the design process, we will use the tools in both Fusion 360 and EAGLE illustrate basic PCB workflows. We will then discuss how to use the integrated electrical/mechanical design tools to approach various design scenarios in Fusion 360. After this, we show how the CAM workspace can be updated immediately with changes to either a mechanical or electrical component. These demos will illustrate the efficiency and interdisciplinary communication benefits of integrated product development in Fusion 360 and EAGLE.

Speakers



Paul Clauss is a graduate of the Mechanical Engineering Technology program at Montana State University, where he worked to integrate mechanical and electrical designs in a pressure sensitive bicycle saddle. Currently a Technical Support Specialist for Fusion 360 and HSM, he is very interested in leveraging the multiple workspaces of Fusion 360 for integrated design, particularly from a DIY perspective. In his free time, he enjoys bicycle riding, snowboarding, and travelling.

Edwin Robledo graduated as an Electronic Engineer the Polytech University of Puerto Rico. Currently, he is a Global Support Manager for the Autodesk EAGLE support Team. His career began primarily working in the communications industry before joining the EAGLE team 25 years ago. His passion for support has lead him to publish best practices articles, Blogs and hundreds of video tutorials. His favorite are live Online Workshops or onsite boot camps demonstrations. If he is not working on electronics, he is usually mountain biking trails in South Florida.



Integrated ECAD, Modeling, and CAM in EAGLE/Fusion 360

As many mechanical products today make use of electrical PCBs, it is important for electrical and mechanical designers to be able to work together for the entirety of the design process. These design teams must be able to:

- Work together to create a design that meets both the electrical and mechanical requirements of the product,
- Be confident that they are working with the most recent version of the electrical and mechanical designs,
- Quickly respond to design changes made by other teams.

The new PCB workspace in Fusion 360 allows for PCB designs from Autodesk EAGLE to be integrated into a mechanical Fusion 360 designs. This cross-product integration strives to provide multidisciplinary design teams with the tools to meet the requirements above.

With these new cross-product tools, designers can create and maintain a link between an EAGLE board file and a mechanical design created in Fusion 360. This allows multidisciplinary teams to:

- Drive mechanical designs in Fusion 360 off of PCB requirements from EAGLE.
- Make changes to PCB components in Fusion 360.
- Pull the changes made to the PCB in Fusion back into the original EAGLE board file.
- Make further edits to the PCB in EAGLE and push the updated board back to Fusion.

The workflows made possible by the PCB workspace in Fusion 360 and the Fusion Sync button in EAGLE allow one file to be used through the entire design process – no more file conversions to maintain up-to-date board files! And because the CAM workspace in Fusion 360 will update when changes are made in the Model workspace, this advantage carries all the way through to the manufacturing team.

The shape and size of a PCB may be limited by mechanical components or, in other situations, it may drive a mechanical design. Before the Fusion 360/EAGLE integration, an update to the board would require;

1. A new 3D package of the board design to be output from EAGLE.
2. That 3D package to be uploaded to a mechanical CAD program.
3. The existing mechanical components need to be edited to meet the requirements of the new board design from EAGLE.
4. The new mechanical components would then need to be exported and used in a new file in CAM software.

And this would need to occur every time the PCB was updated throughout the design process! With the integration between Fusion 360 and EAGLE, changes to the board can now be performed in either EAGLE or Fusion 360, and the PCB component can be easily updated in both the mechanical and electrical designs. This workflow offers true ECAD/MCAD synchronization in which changes can flow back and forth, which gives multidisciplinary teams

more freedom during the design process and the ability to quickly develop design iterations with changes to both the mechanical and electrical components.

To take full advantage of the Fusion 360/EAGLE integration features, there are some tools and workflows to be aware of. This class will present the specifics of the Fusion 360 PCB workspace and the Fusion Sync tool in EAGLE, the key workflows used during ECAD/MCAD integration in Fusion 360/EAGLE, and describe some best practices for working with electrical/mechanical designs.

- Enabling the PCB integration features in Fusion 360/EAGLE
- The PCB workspace in Fusion 360 and the Fusion Sync button in EAGLE
 - The PCB workspace in Fusion 360
 - The Fusion Sync button in EAGLE
- Key Workflows for ECAD/MCAD Integration in Fusion 360/EAGLE
 - Pushing an EAGLE PCB to Fusion 360
 - Creating a PCB Component in Fusion and pulling into EAGLE
 - Creating a new version of a 3D package in an EAGLE library
- Considerations when using Fusion 360 and EAGLE integration
 - Design Scenarios
 - A standard PCB design is used in multiple different mechanical assemblies
 - A mechanical design dictates the allowable form of a PCB,
 - Both the mechanical and electrical components are in the early stages of the design process and may be changed
 - Communication tools for ECAD/MCAD integration in Fusion 360/EAGLE
 - Notes on CAM operations and PCB manufacturing
- Additional Resources

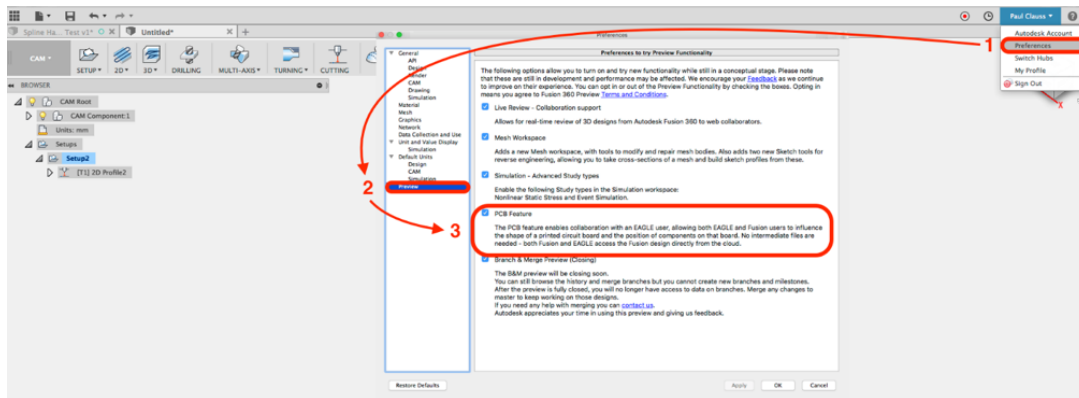
ENABLING THE PCB INTEGRATION FEATURES IN FUSION 360/EAGLE

Activating the PCB Preview in Fusion 360

The EAGLE/Fusion 360 integration tools are offered as a preview feature at this time. Previews refer to tools within Fusion 360 that are still undergoing development, but are available in Fusion 360 for you to try and provide feedback on.

You can [learn more about previews in Fusion 360 at this link](#).

To access the PCB workspace in Fusion 360, start by activating the PCB Preview flag in your Fusion 360 preferences:



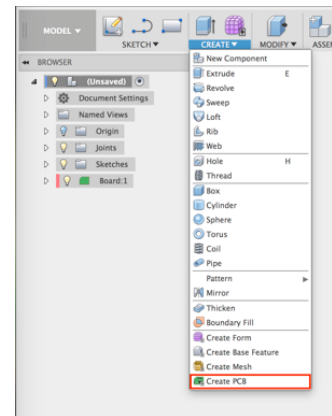
1. Click on your name in the upper right-hand corner and select "Preferences" from the drop-down menu.
2. Enter the Preview tab of the Preferences dialogue.
3. Enable the flag for the PCB Feature preview.

After enabling the PCB Feature preview in Fusion 360, we can now create PCB components through the Create > Create PCB option and access the PCB workspace in Fusion 360.

In EAGLE, you must be using version 8.3.0 or later to access the Fusion Sync button to push and pull designs to and from Fusion 360. You also must be connected to the internet.

There are two ways to create a PCB component. Depending on your workflow and the design requirements, you can choose from either of the options below;

- Create a PCB component in the PCB workspace of Fusion 360 and *pull* it into EAGLE,
- Create a PCB board file in EAGLE and use the Fusion Sync Button *push* it into Fusion 360.

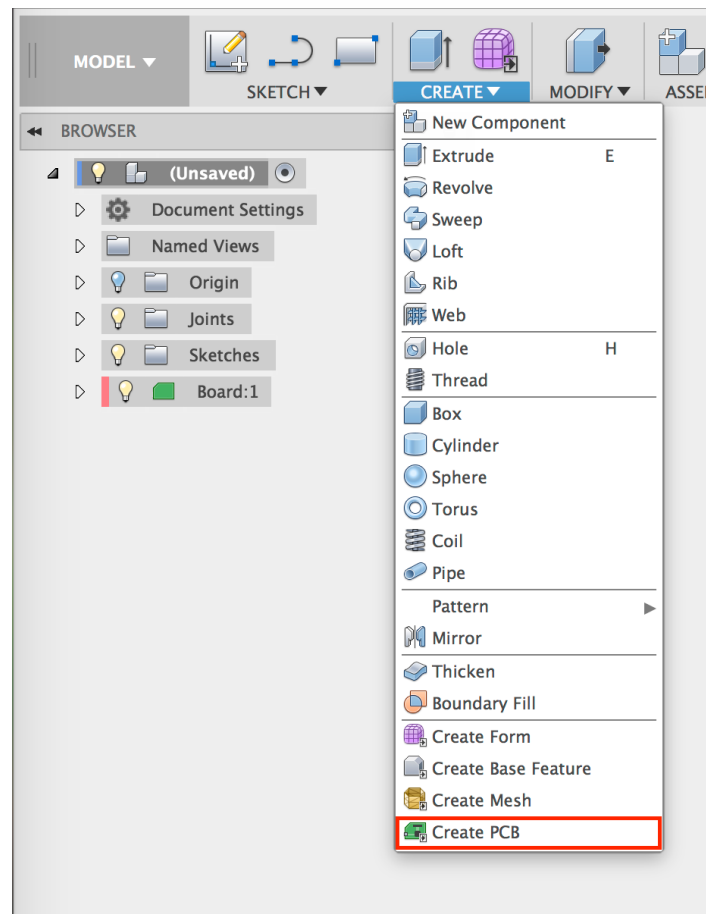


THE PCB WORKSPACE IN FUSION 360 AND THE FUSION SYNC BUTTON IN EAGLE

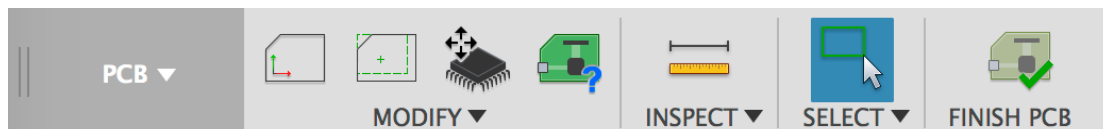
The PCB workspace in Fusion 360 and the Fusion Sync button in EAGLE are the tools that make it possible to edit the size, shape, and layout of PCBs in Fusion 360 and push/pull boards between EAGLE and Fusion. Using the PCB workspace in Fusion, mechanical designers can create board profiles, move PCB components, and check to see if the board needs to be updated in either Fusion or EAGLE with a push or pull. And with the Fusion Sync button in EAGLE, electrical designers can push and pull design iterations of the board back and forth between the two programs.

The PCB workspace in Fusion 360

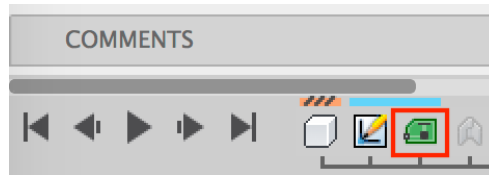
The PCB workspace is accessible once the PCB preview has been activated. To create a PCB component in a Fusion design, use the “Create PCB” command in the “Create” drop down menu of the Model workspace:



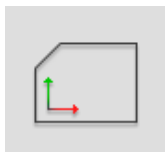
This will bring Fusion 360 into the PCB workspace. The ribbon below will be shown:



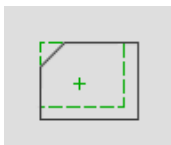
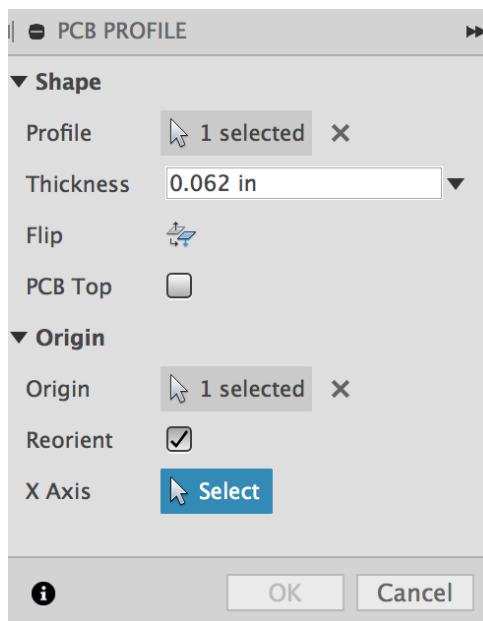
Once a PCB component has been created in a Fusion 360 design, the PCB workspace can be accessed by either double or right-clicking the PCB icon shown below and selecting “Edit.”



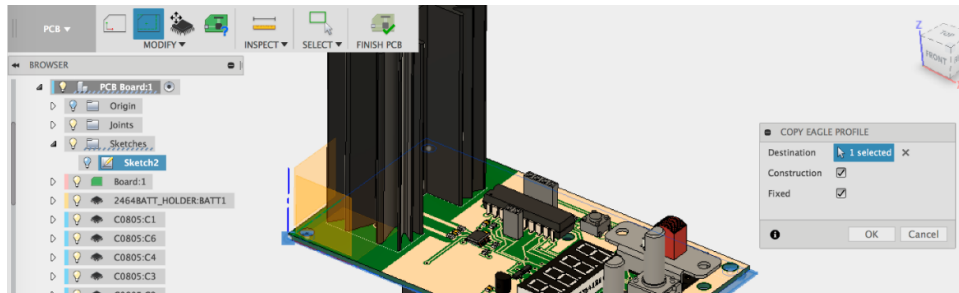
The PCB workspace includes five new tools; PCB Profile, Copy EAGLE Profile, Move PCB Components, PCB Status, and FINISH PCB. These tools are explained below:



- **PCB Profile:** This option allows you to select a closed Fusion sketch profile as a PCB outline/profile, define the top and bottom of the board, set the board origin point and axes, and create a board component in your Fusion 360 design. This creates a Fusion Board Shape within the Fusion design file. The PCB Profile dialog box is shown below.

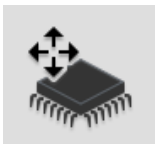
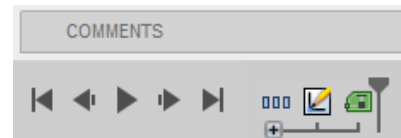


- **Copy EAGLE Profile:** This option will only be shown after a board has been pushed from EAGLE to Fusion. It allows a copy the sketched profile of the current EAGLE version of the board (the EAGLE Board Shape) to be added to a sketch in the Fusion design. We can choose which sketch the EAGLE profile will be copied into and if the sketch will consist of fixed and/or construction lines, as shown at the top of the next page.

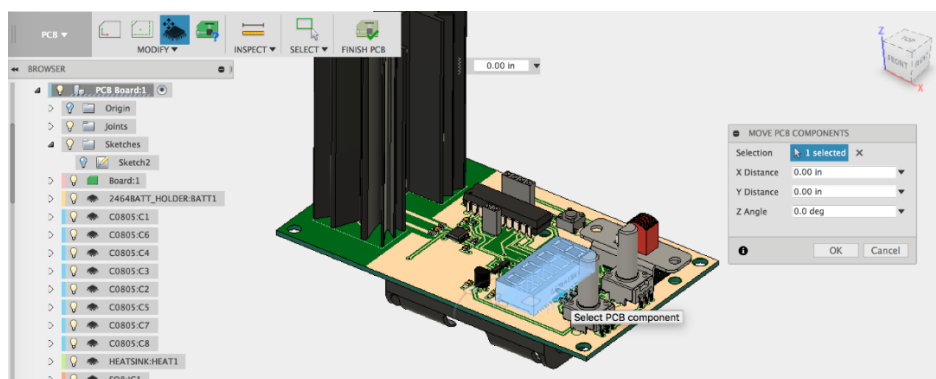


The Copy EAGLE Profile tool is useful when a PCB is pushed into a Fusion 360 design in which no Fusion board shape has been created. To define a Fusion 360 board profile, a sketch must be present before the PCB component in the design timeline, as editing the board will roll the timeline back to when the PCB component was created.

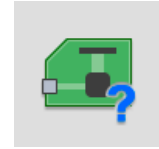
A sketch of the board profile from earlier in the timeline than the PCB component feature is required to define and edit a Fusion board shape. It can be difficult to work with because rolling back the timeline to create a sketch before the PCB component in the timeline hides the PCB – it was not present in the model at that point in the design timeline. This creates problems when trying to sketch the Fusion board profile in the correct location and on the correct plane, as we cannot see the existing EAGLE board when creating the sketch. The Copy EAGLE Profile tool allows us to copy the EAGLE sketch into an existing sketch from before the PCB component was pushed from EAGLE and use that sketch for reference when creating a Fusion board profile. This makes it easier to create a Fusion Board Shape in a design that only contains an EAGLE Board Shape. Fusion vs. EAGLE board shapes will be explained later in more detail.



- **Move PCB Components:** This tool allows you to move PCB components on a board that has been had 3D models linked to it through a library component in EAGLE after the board has been pushed back to Fusion. Simply select the circuit component that you would like to move and use either the manipulator handle or numerical entry fields to move it translationally or angularly on the plane of the board top. Circuit components moved with this tool are moved on both the EAGLE and Fusion board shapes within the Fusion design.

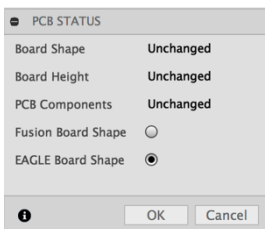


- **PCB Status:** This tool allows you to check if the Fusion PCB component has been altered *since the last push from EAGLE to Fusion 360*.



The three status lines (Board Shape, Board Height, and PCB Components) in the PCB Status tool show if the shape and height of the board, as well as the layout of circuit components, have been changed since the last push from EAGLE in the Fusion 360 design. When both systems are synced, the two board shapes (Fusion and EAGLE) included in the Fusion design will be the same.

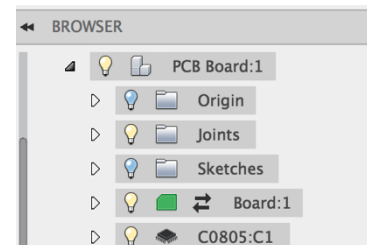
When the EAGLE and Fusion board shapes are different, there is a danger of an inconsistency between the electrical and mechanical designs. The PCB Status tool can help manage these inconsistencies.



In the PCB Status tool, we see the option to activate either the Fusion or EAGLE board shape. The Fusion board shape refers to the PCB profile as defined by the PCB Profile tool in Fusion and the EAGLE board shape refers to the footprint that was last pushed to Fusion from EAGLE.

The shape choice radio buttons (Fusion Board Shape and EAGLE Board Shape) can be used to select if the EAGLE or Fusion board shape will be shown in Fusion 360. If the currently used shape is the EAGLE shape, the board shape will always show as "Unchanged," as the shape being used will be the shape from the last push from EAGLE – the EAGLE board shape cannot be altered in Fusion. Selecting the Fusion board shape will show if the Fusion board shape has been edited since the last push from EAGLE.

We can also see if the board has been changed in Fusion since the last push from EAGLE in the Fusion 360 browser tree. The arrow icons shown to the right on the board component will be shown if the Fusion board shape has been changed since the last push from EAGLE. This can indicate that a push/pull cycle needs to be done to keep the Fusion and EAGLE board shapes in sync and can be used much like the PCB Status tool.

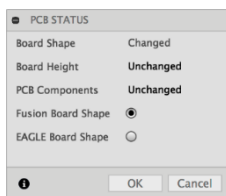
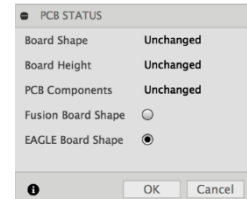


A note on this is that the board shape selected in the PCB Status tool will also be the board shape used during the next pull from Fusion to EAGLE. This is important to consider when the shape of the board is edited in Fusion after a push from EAGLE because, if the EAGLE board shape is selected, the changes to the Fusion board shape will not be pulled to EAGLE during that exchange. Because Fusion only keeps one copy of the PCB components, changes to the circuit components or PCB layout are always made on both board shapes and will always be included in a pull to EAGLE.

If the Fusion board shape is selected and shows that it has been "Changed" since the last push from EAGLE, you will have the option to pull the new Fusion board shape into EAGLE. The need for the pull will be shown in EAGLE in the Fusion Sync button after the Fusion design is saved, as well as in the PCB Status tool and in the browser tree in Fusion.

The Board Height and PCB Components fields are reset each time the board is pushed from EAGLE to Fusion 360. Any changes made to the board height or position of the PCB components since the last push from EAGLE will be shown when the Fusion Board Shape radio button is selected. Please note – changes made in the EAGLE board file (in EAGLE) after the most recent push to Fusion will not be shown in the PCB Status tool.

In the screenshot to the right, the EAGLE Board Shape is selected. This means that the EAGLE board shape will be shown in the Fusion 360 design and that, since the last time it was pushed to Fusion, no changes have been made to the Board Height or the PCB Components within Fusion. The EAGLE board shape cannot be changed in Fusion, so the EAGLE board shape will always show as “Unchanged” – it is possible that the Fusion board shape has been edited since the last push. Changes to the board file in EAGLE since the last push will not be shown here, so the electrical designer must be sure that the “Push to Fusion” button shows no changes in the Fusion Sync dialog in EAGLE. This also means that the EAGLE board shape will be used in a pull to EAGLE.



In screenshot to the left, the Fusion Board Shape is selected. This screenshot shows indicates that the Fusion board shape will be shown in the design and that the Fusion board shape has been changed since the last time the board was pushed from EAGLE. It also shows that the board height and PCB components have not been edited in Fusion since the last push. With this selection, the Fusion board shape will be used in a pull to EAGLE. This configuration could also show that holes

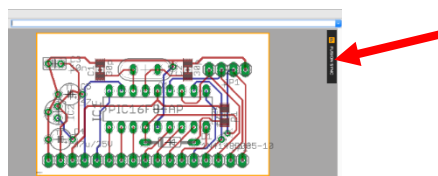
were punched in the Fusion board shape when PCB components were added in the last push from EAGLE, even though the profile was not changed. While these holes do not trigger the PCB components on the Fusion board to show as changed, they will trigger the Fusion board shape to show as changed. The PCB Components are unchanged because Fusion only sees one set of PCB components (from the last push from EAGLE), and these have not been altered since the last push from EAGLE. They have, however, required pin holes to be added to the Fusion board shape, which is shown in the PCB Status tool. When this occurs, we can perform another push/pull cycle between EAGLE and Fusion to ensure everything is synced.



- **FINISH PCB:** This command allows you to exit the PCB workspace and return to the Model workspace of Fusion 360 after creating or editing a PCB component is complete, at least for the time being.

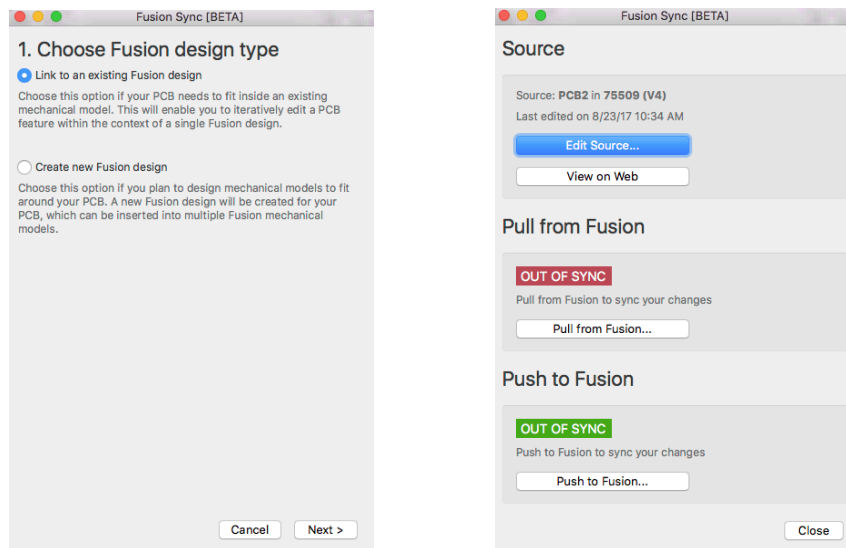
The Fusion Sync button in EAGLE

The Fusion Sync button in EAGLE allows electrical designers to pull updates to the board component from Fusion 360 and push changes made in EAGLE back to Fusion 360. This button is accessible to the right side of an EAGLE board window.



In an EAGLE board with no link to a Fusion 360 design, you will see the option to either create a link to an existing or new Fusion design in the Fusion Sync button in EAGLE. This is shown to the left below.

In a board that is already linked to a Fusion design, you will see if the board has been changed in either Fusion 360 or EAGLE since the last push to Fusion. For the “Pull from Fusion” section to show as “Out of Sync,” a Fusion design with changes to either the Fusion board shape, height, or the circuit components layout must be saved as a new version. A Fusion Sync dialog in which the board has been edited in both Fusion and EAGLE since the last push is shown to the right below.



When working in EAGLE, you will notice that the color of the Fusion Sync button changes. A RED button means that the board design you are working with in EAGLE has been altered in Fusion 360 and saved in a version of the file there, and a GREEN button means that you have made changes to the PCB in EAGLE after the last push that need to be sent to Fusion in a new push.

The “Push to Fusion” section of the Fusion Sync button is the only place that changes to the EAGLE board file will be shown after a board is pushed to Fusion from EAGLE, as the PCB Status tool is unable to pull information about EAGLE board file versioning into Fusion 360.

KEY WORKFLOWS FOR ECAD/MCAD INTEGRATION IN FUSION 360/EAGLE

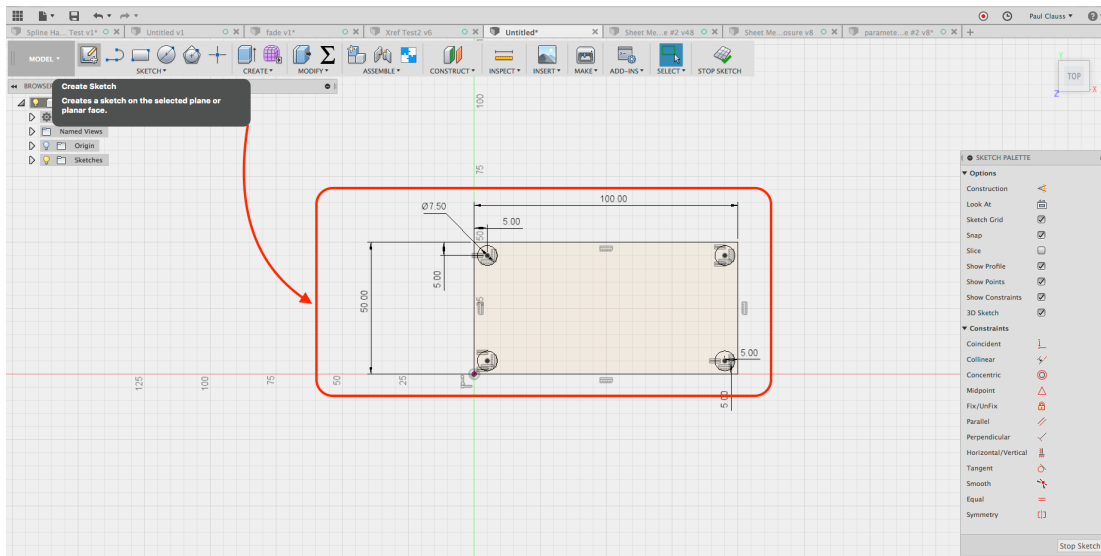
There are a few common workflows to be aware of when working with PCB components linked between an EAGLE board file and a Fusion 360 design. Step-by-step instructions for the workflows listed below will be presented in this section.

1. Creating a PCB component in Fusion 360 and pulling it into EAGLE.
2. Pushing an existing EAGLE board file to a Fusion 360 design.
3. Creating or updating a new version of a 3D package in an EAGLE library.

Creating a PCB component in Fusion 360 and pulling it into EAGLE

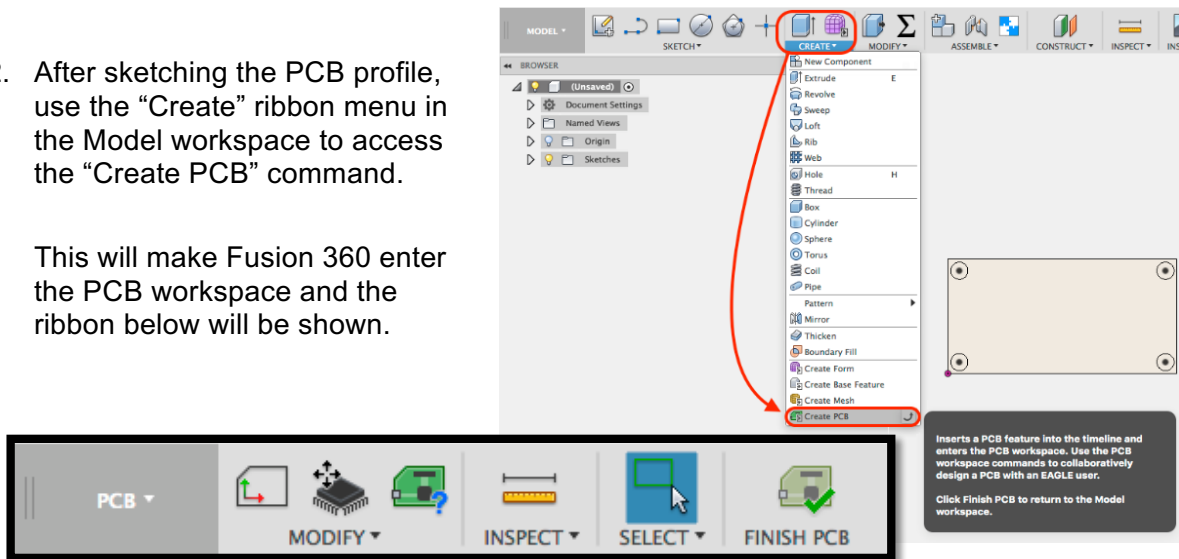
There are design situations in which the form or size of mechanical components dictate the allowable size and shape of a PCB. If a mechanical designer knows what size and shape a PCB is required to be, they can create a PCB board with the required profile in Fusion 360 that can be pulled into EAGLE by the electrical team.

1. Create a closed profile sketch in Fusion 360. This sketch can be any shape or size, can contain mounting holes or other through features, and will define the footprint and origin of the PCB component.

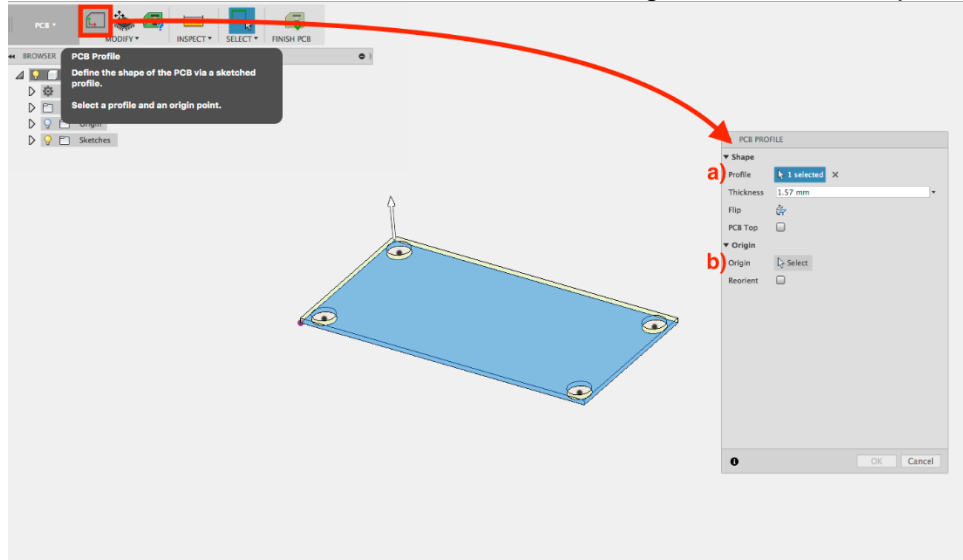


2. After sketching the PCB profile, use the “Create” ribbon menu in the Model workspace to access the “Create PCB” command.

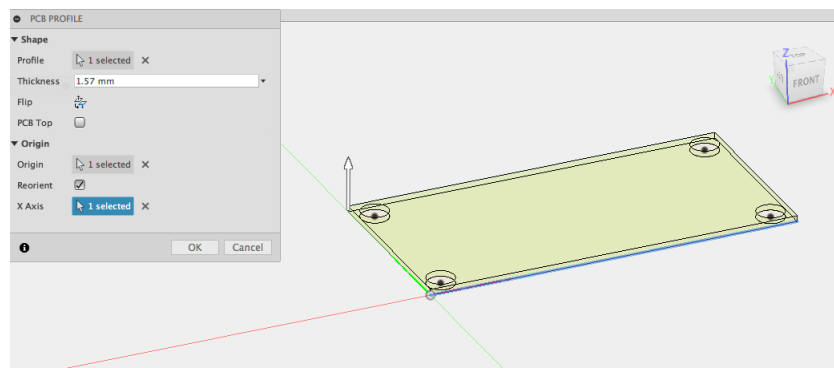
This will make Fusion 360 enter the PCB workspace and the ribbon below will be shown.



3. Use the PCB Profile tool to define the Profile and Origin of the PCB component:

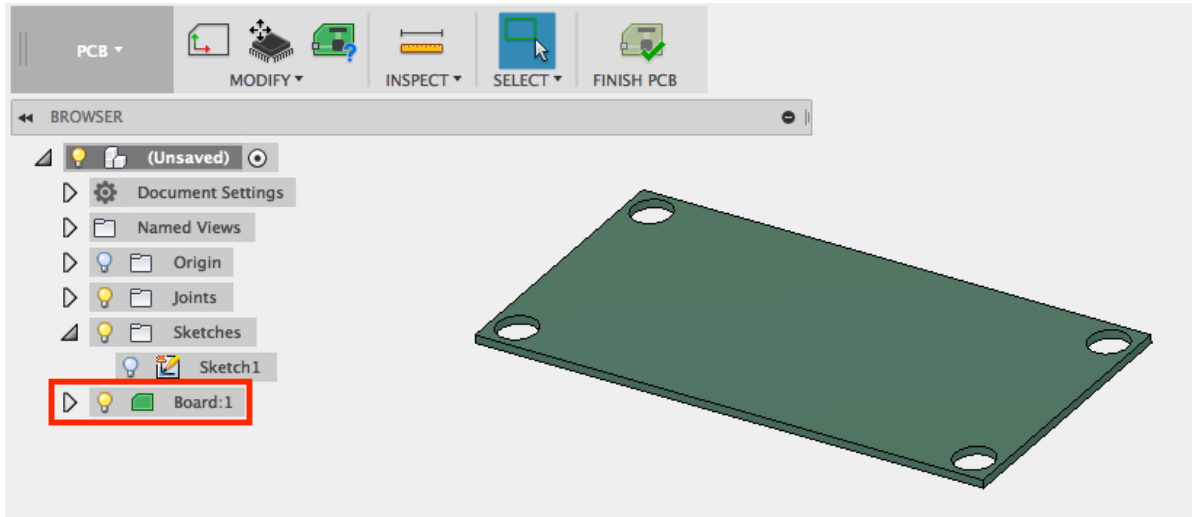


- To define the shape of the board component, select the closed sketch profile created in Step #1. You can reverse the direction of the extrusion for the board with the “Flip” option and define the sketch as the top of the PCB sketch by enabling the “PCB Top” flag.
- When defining the origin of the PCB component, you can select any point. This point will define the board origin point that will be used in both Fusion and EAGLE. After you select a point for the origin, you may reorient the X and Y axis of the board by enabling the “Reorient” flag and then selecting the desired X-axis. The axes defined here will be the axes used for the board component in Fusion and in the EAGLE board after it is pulled to EAGLE.

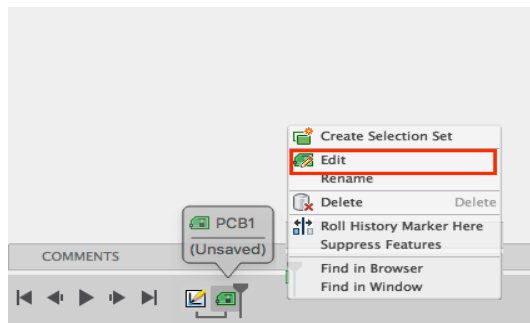
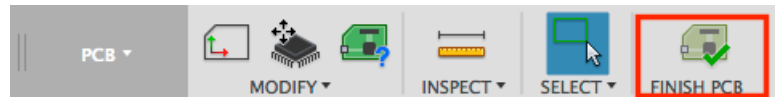


When setting up the X and Y axes of a PCB component in Fusion 360, the X-axis will be previewed in red and the Y-axis will be previewed in green as shown above.

4. Select “OK” to exit the PCB Profile dialogue box and create the PCB. The board will now be shown as a solid green body and a Board:1 component will be shown in the browser.



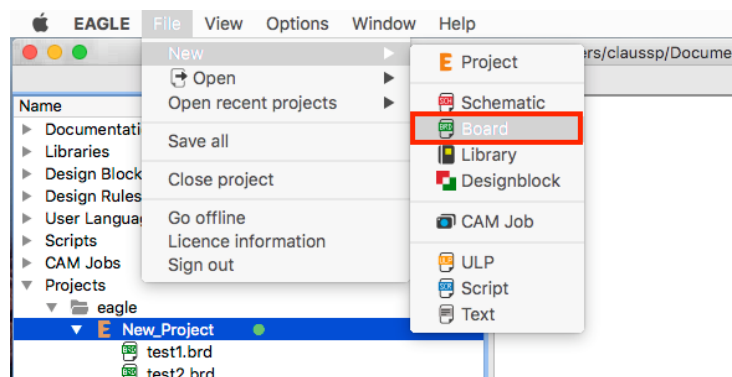
5. Use the “Finish PCB” tool in the ribbon to exit the PCB workspace and re-enter the modeling workspace.



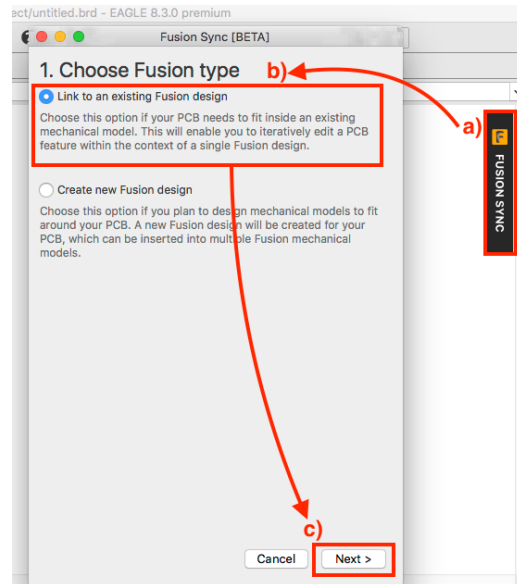
6. After the PCB is finished, you may need to make changes to it. To edit the PCB component, right-click the PCB icon in the design timeline and select “Edit.” This will allow you to re-enter the PCB workspace.

7. After saving the Fusion 360 design, the PCB component can be pulled into EAGLE.

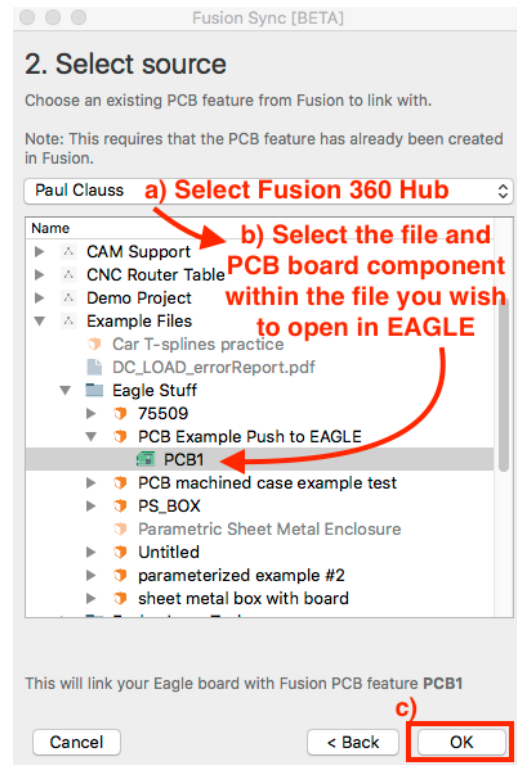
8. Open EAGLE (version 8.3.0 or higher) and sign into your Autodesk Account.
9. Ensure EAGLE is connected to the internet and you are signed into your account
10. Create a new board file through the File menu as shown to the right.



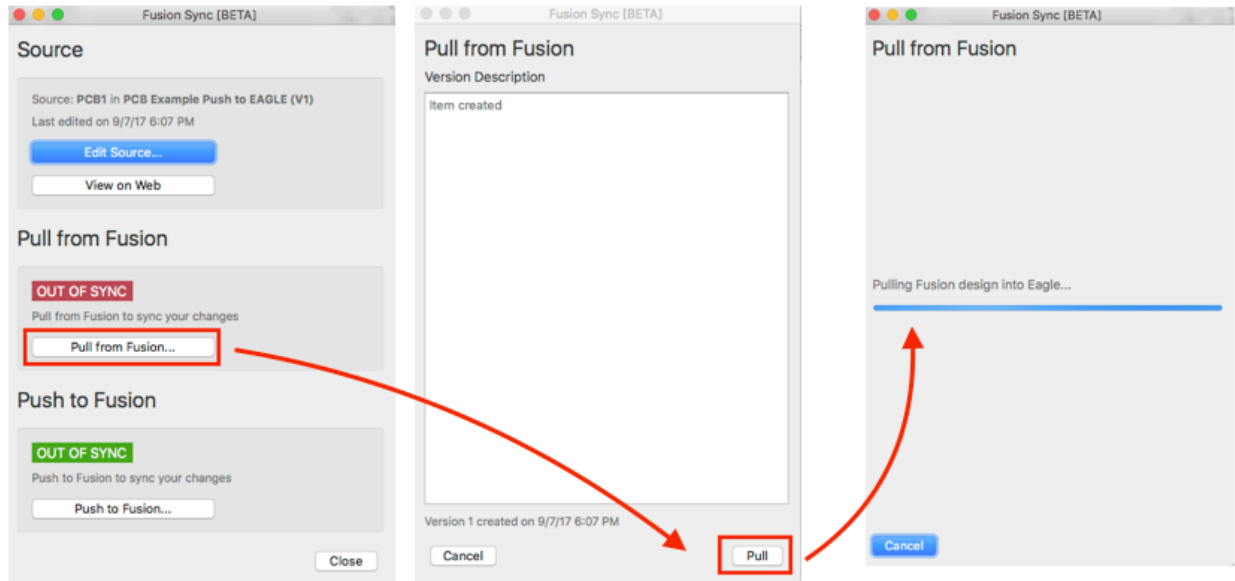
11. Click the Fusion Sync button (a) and select “Link to an Existing Fusion Design,” (b) then “Next” (c).



12. Select the source file from Fusion 360 in the Fusion Sync dialogue. Select the Fusion 360 hub first (a), and then navigate through the folders shown to the design containing the PCB component (b) that you would like to open in Eagle. Use the arrows to expand folders and files to show the PCB components in each file. After selecting the source file, hit “OK” (c).



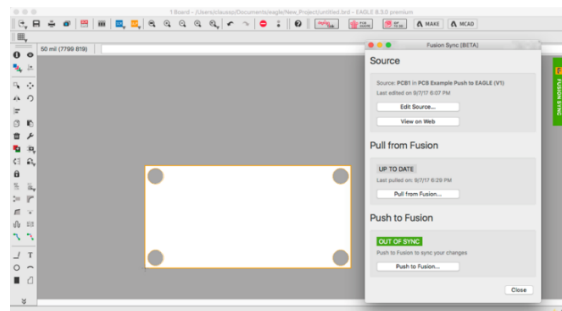
13. After clicking “OK”, the dialogue to the far left below will be shown. The “pull from Fusion” field and the “push to Fusion” field will both be shown as “Out of Sync” because the board has not been pulled into EAGLE yet. Click the “pull from Fusion” button to begin the pulling process.



14. The version description of the Fusion PCB component will be shown (as in the middle screenshot above). Select “Pull” to bring the PCB component from Fusion into EAGLE. This will show final dialogue shown above (to the far right).

15. After the pull is complete, the Source dialogue will show as “Up to Date” under the “Pull from Fusion” heading and the board profile will be shown in the EAGLE board workspace. The board can now be edited in EAGLE and pushed back into the original Fusion design!

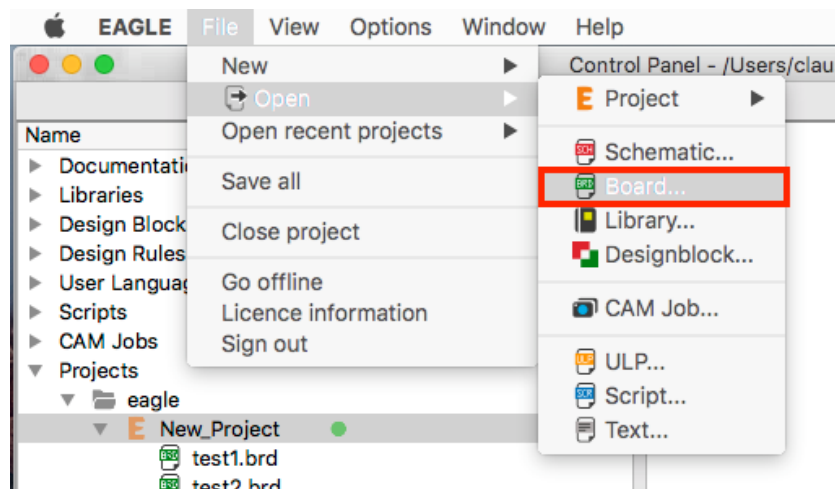
While the board can be edited in EAGLE at this point, it is recommended to push the board back to Fusion 360 before editing the board in Fusion or EAGLE. This completes the integration cycle and syncs the EAGLE and Fusion board shapes in the beginning of the design process, as well as guarantees that both a Fusion and EAGLE board shape is present in the Fusion design.



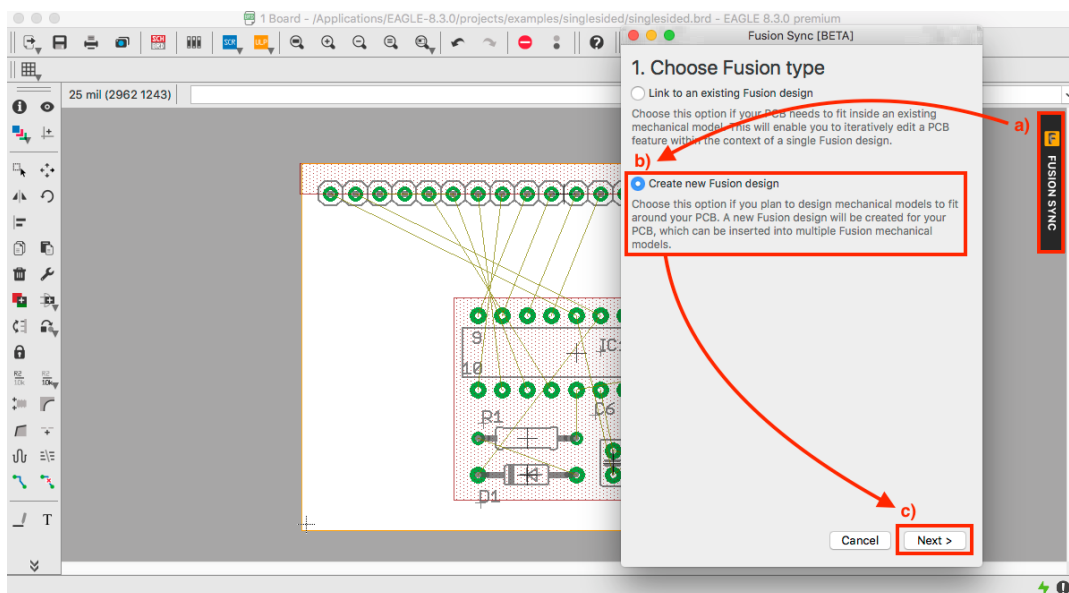
Pushing an existing EAGLE board file to a Fusion 360 design

There are also times in which a mechanical design must be built around an existing PCB. In these situations, electrical designers may want to push a board design to the mechanical team, who can then work off the existing PCB model to develop mechanical components. The process to push an existing EAGLE board into a new Fusion 360 is detailed below:

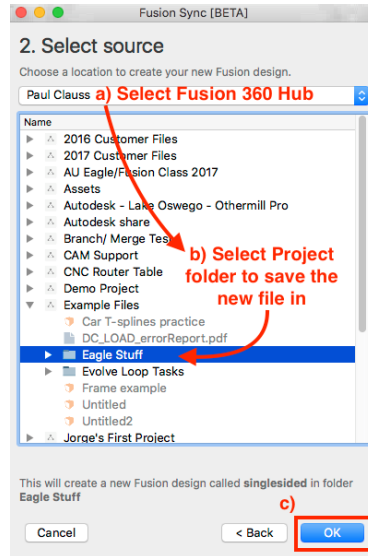
1. Open EAGLE version 8.3.0 or higher.
2. Open an existing board design in EAGLE through the File> Open menu.



3. After the board has opened, click the “Fusion Sync” button (a) and select “Create New Fusion Design” (b), then click “Next” (c).

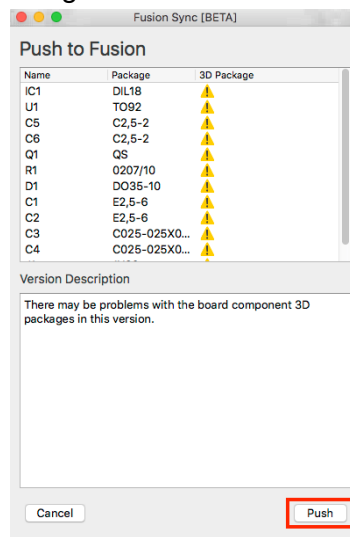


4. Select the location in your Fusion 360 Data Panel for the new Fusion 360 design that will be created when the board is pushed. Select the Fusion 360 hub you are working in (a) and the project folder you wish to create the new Fusion 360 design in (b). You can then click the “OK” button (c).

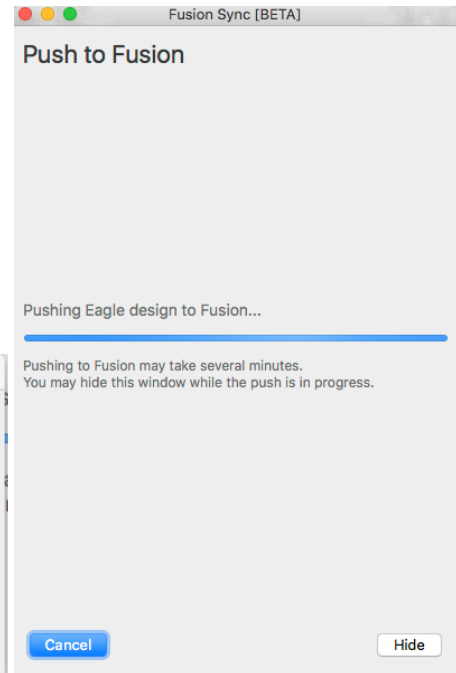
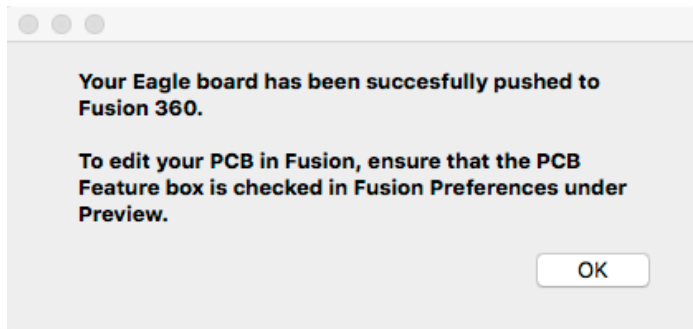


5. After clicking “OK”, you will see a list of the components on the PCB. If there are problems with the 3D model for any of the components, they will be marked with a yellow flag, as shown in the screenshot below. Components with no errors in their 3D models will display green checks instead of the yellow flags.

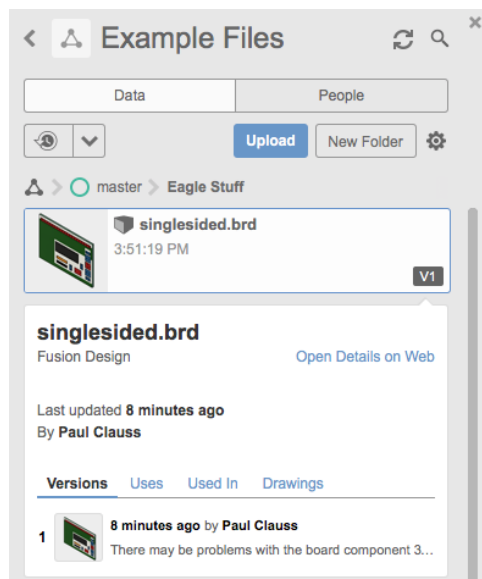
You will also be given the option to add a version description for the Fusion design in this dialogue. These notes will be shown in the version of the file shown in the Fusion 360 Data Panel. After adding any notes and checking the 3D packages, click the “push” button to begin pushing the design to Fusion.



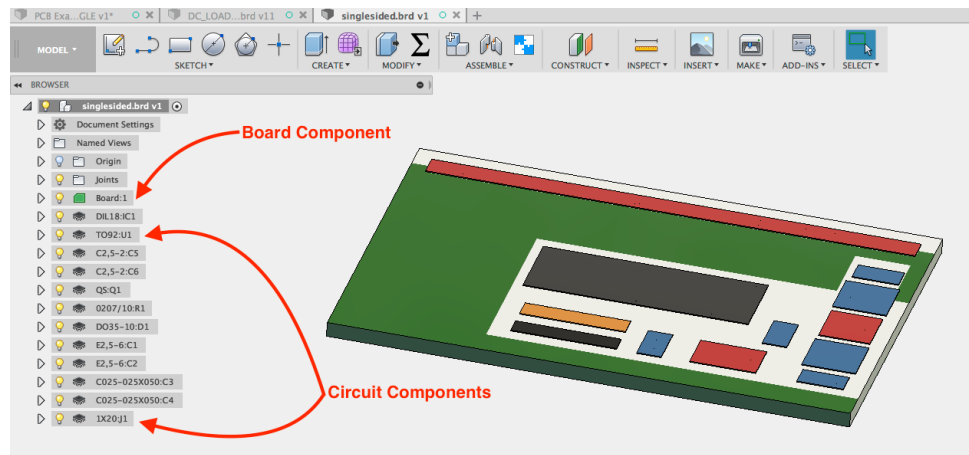
6. After clicking “push,” you will see the dialog shown to the right. You may choose to hide the window or leave it open while the board design is pushed to Fusion 360.
7. When the push to Fusion 360 is completed successfully, you will see the message shown below.



8. Open Fusion 360 and navigate to the file location the board was pushed to in step #4 in the Data Panel. The new Fusion 360 design created during the push from EAGLE will be shown and the comments made during the push will be shown next to the version description.



9. The design can now be opened in Fusion 360! When the design is opened, the board will be shown as a board component and the 3D packages placed on the board in EAGLE will be shown as circuit components.



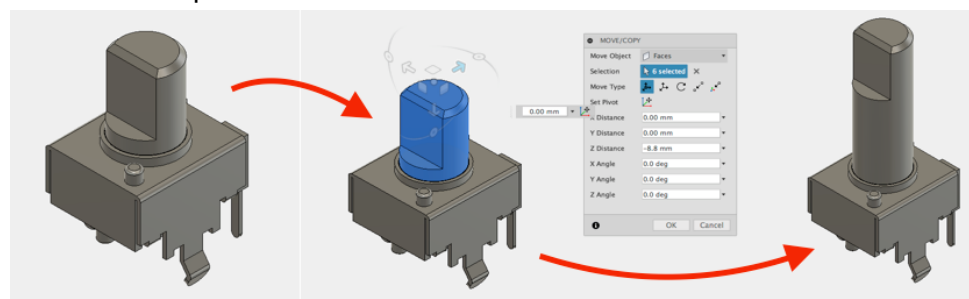
You can now begin editing the board in the Fusion 360 file and designing a mechanical enclosure around the PCB component!

Creating a new version of a 3D package in an EAGLE library

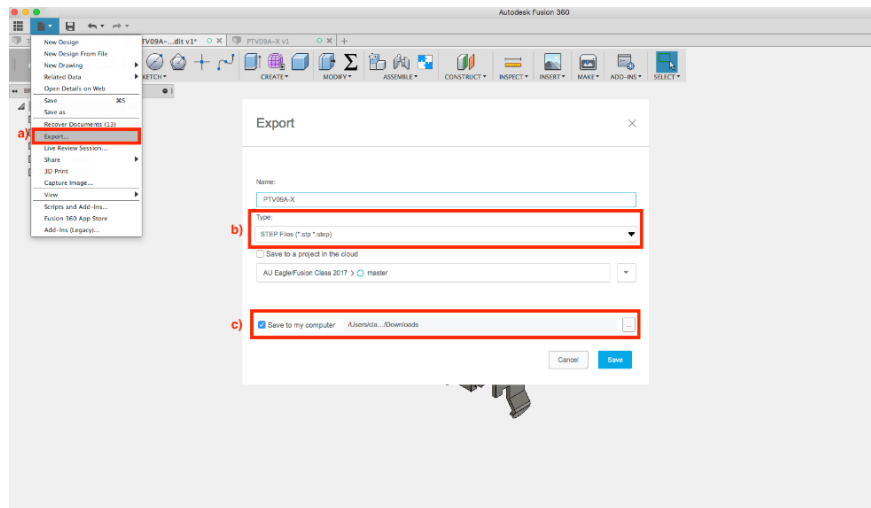
Another issue that can occur during PCB design is that it can be hard to find 3D models of electrical components to add to a design library of 3D packages in EAGLE. Sometimes a model for only one size of a component is available, and sometimes custom electrical components will need to be created. While Fusion 360 designs cannot be directly linked to 3D packages in an EAGLE library at this time, they can be exported in STEP format and uploaded as a new version of the EAGLE library component.

If an electrical designer has a 2D drawing for a circuit component but not a 3D model, they can share it with a mechanical colleague who could model the component in Fusion 360. The electrical designer might also notice that the model for a circuit component in their library is the incorrect size. In this case, the electrical designer could share the STEP file for the incorrect component with the mechanical designer, who could use the modelling tools in Fusion to edit and fix the model. An example of how to upload a new version of an EAGLE library component created in Fusion 360 to the circuits.io library manager is included below:

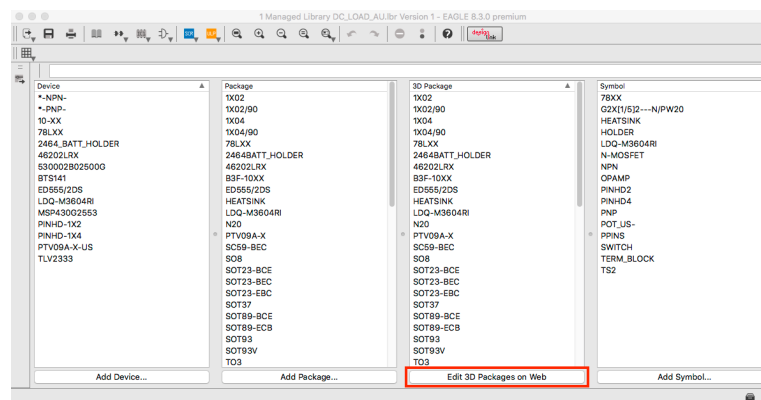
1. Create or edit an existing design for a circuit component in Fusion 360. In this example, the height of a potentiometer knob was altered to meet the specifications for a different version of a standard part with the "Move Faces" command.



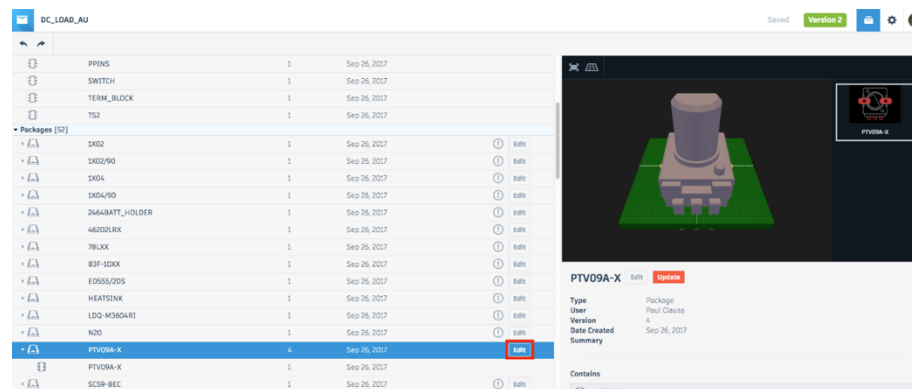
- Use the (a)File > Export command to save a (b)STEP format copy of the file (c)locally.



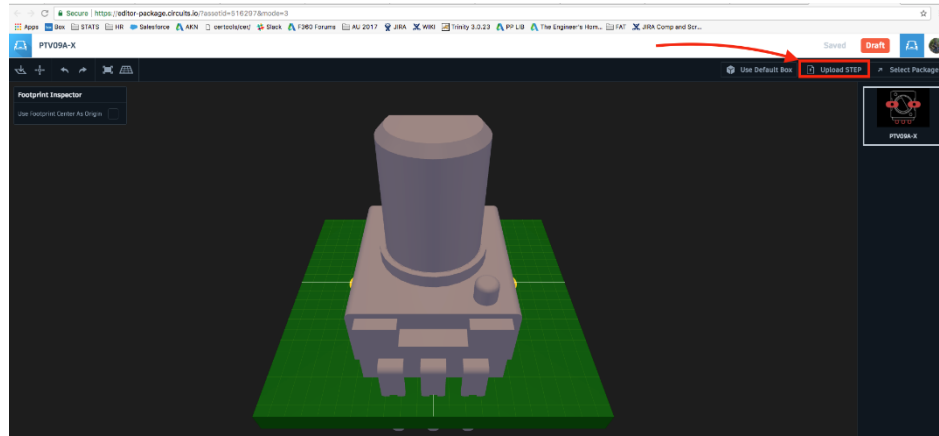
- Open EAGLE and open the library containing the part being edited. After opening the library, click the “Edit 3D Packages on Web” button.



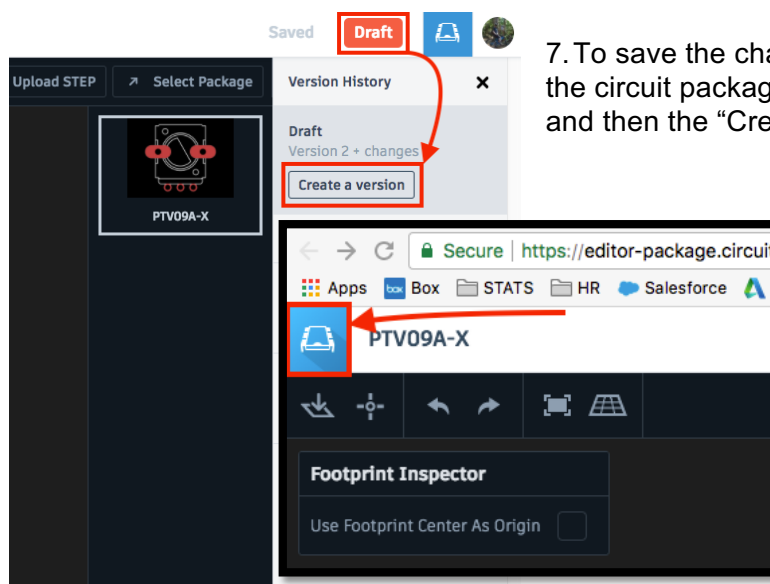
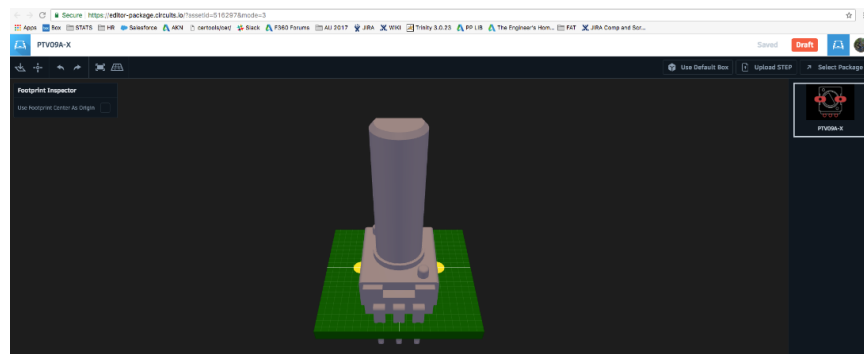
- A web browser will open and navigate to the library editor at circuits.io for the EAGLE library. Find the component that is being altered under the “Packages” header and click the “Edit” button.



5. A new tab will be opened in the web browser and the current 3D model used for the library component will be displayed. Click the “Upload STEP” button to import the file created in step #2 to the library.



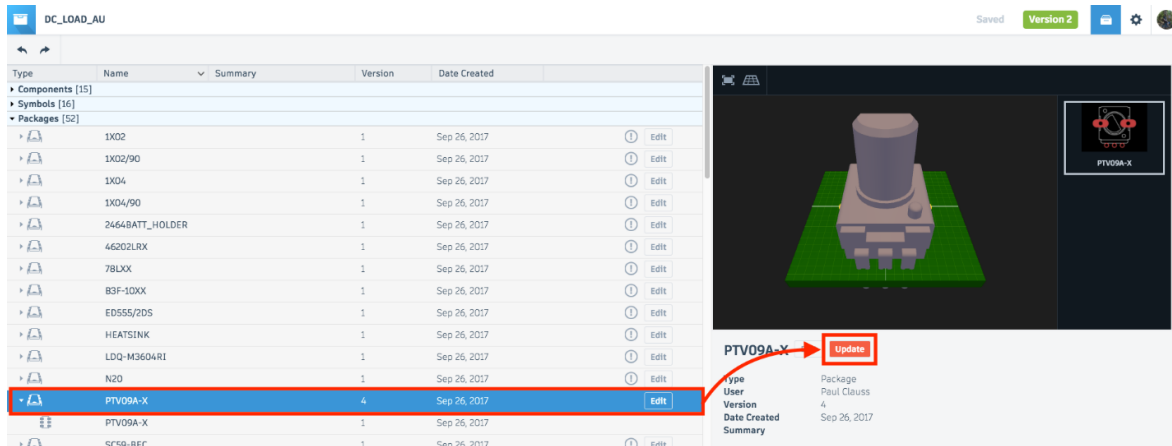
6. Select the STEP file exported from Fusion 360 in step #2 from the location it was saved to locally and select “Open.” The new STEP file will be uploaded to the library and shown in the workspace.



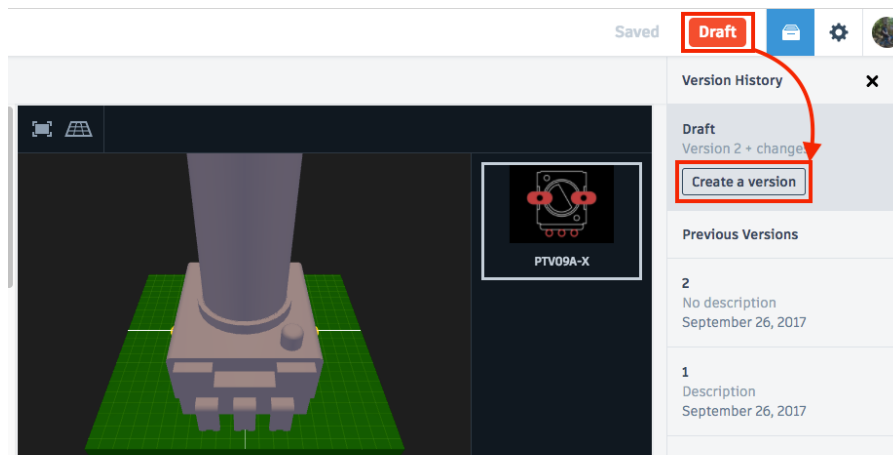
7. To save the changes as a new version of the circuit package, click the “Draft” button and then the “Create a version.”

8. Click the blue circuit component button in the upper left-hand corner to return to the library in the web browser. This will bring us to the webpage from step #4.

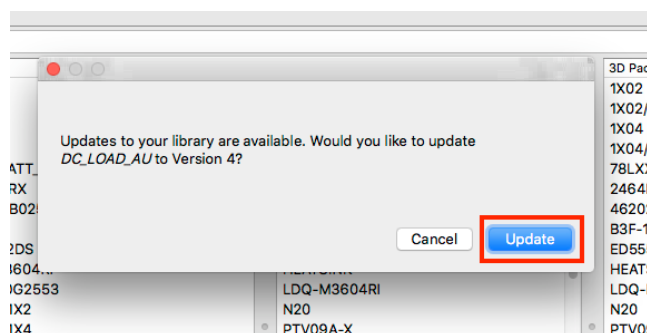
- Back at the library page, select the package that was edited and click the “Update” button. The changes you just made will be shown when the update completes.



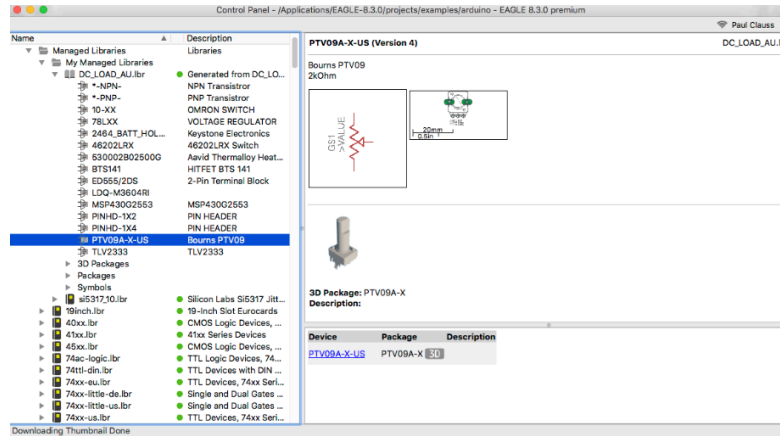
- Save the updates to the library by clicking the “Draft” button and then the “Create a version” button.



- Open the library in EAGLE. A notification that changes were made to the library will be displayed. Select “Update” to update the library.



- The changes made to the 3D package will now be shown when viewing the circuit component through its library in the EAGLE control panel and the new 3D package will be used when pushing designs using the component and library to Fusion 360.



CONSIDERATIONS WHEN USING FUSION 360/EAGLE INTEGRATION

Because the integrated tools in Fusion 360 and EAGLE allow a variety of different workflows to be used, the design requirements and process should be taken into consideration when creating a linked electrical/mechanical design.

When designing an integrated electrical/mechanical product, there are three design situations that commonly occur:

- A standard PCB design is used in multiple different mechanical assemblies,
- A mechanical design dictates the allowable form of a PCB,
- Both the mechanical and electrical components are in the early stages of the design process and may be changed.

Each of these design situations requires special considerations when creating models for the mechanical and electrical components. Descriptions of a potential workflow for each of these situations, along with some notes on communication tools and manufacturing, are included below.

Design Situation #1: A standard PCB is used in multiple different mechanical assemblies

Certain PCB designs may be used in a variety of mechanical enclosures. These PCBs are often existing designs that are rarely changed and can be thought of as standard or mass-produced parts. Examples of these PCBs could include;

- A standard microcontroller such as an Arduino board.
- A switch or load circuit used in multiple mechanical assemblies.

While creating a mechanical design for these boards, the size, shape, and layout of the board may need to be designed around in a variety of different mechanical assemblies. When working with a finalized PCB that will need to be used in multiple mechanical designs, a good workflow is to;

1. Open the finalized BRD file for the PCB in EAGLE.
2. Push the PCB from EAGLE into a new Fusion 360 design.
3. Insert the Fusion 360 design for the board into the assembly designs as an x-referenced component.



X-referenced components inserted into Fusion 360 designs are linked to their original files. They cannot be edited in the assembly designs, but changes made in their original design file can be brought into the assembly design by updating the x-referenced components with the “Get All Latest” command. While x-referenced components cannot be edited in the assembly design, their geometry can be referenced by the modeling tools in Fusion within the assembly design.

When including a finalized board design in multiple mechanical assemblies, this workflow is advantageous as it:

- Allows a single board file to be inserted into multiple designs,
- Gives electrical designers the ability to push any changes made to the standard board to a single Fusion 360 design to alert the mechanical team,
- Minimizes the number of pushes from EAGLE required to update multiple assembly designs because the changes pushed from EAGLE into the board-only file will trigger a prompt to run the “Get All Latest” command in every assembly file (the board does not need to be pushed to every assembly file from EAGLE),
- Gives mechanical designers the ability to update any changes to the board easily in every assembly file with the “Get All Latest” tool (which needs to be run in each assembly file that will be updated),
- Permits mechanical designers to reference accurate board geometry when developing their designs,
- Keeps the board design and layout in the hands of the electrical team, as mechanical designers will rarely use the original file in which the board can be edited in Fusion.

If the board needs to be customized in one particular assembly design, the link to the x-referenced board file can be broken and the PCB component in that file can be pulled into a new EAGLE or existing EAGLE design, edited, and pushed back to Fusion.

This workflow could be used to create a Fusion project folder containing a “library” of commonly used PCB designs which mechanical designers could insert into assembly files. If the boards did need to be changed, a single push to Fusion from the electrical team and then an update of all the linked assembly files could quickly make as many designs as necessary current. One

thing to note is that x-referenced designs do not maintain their link across projects at the highest level of folder hierarchy in the Data Panel, so we would need to create a project folder at the highest level and then create a PCB folder and a Mechanical designs folder within that top-level project to use this technique.

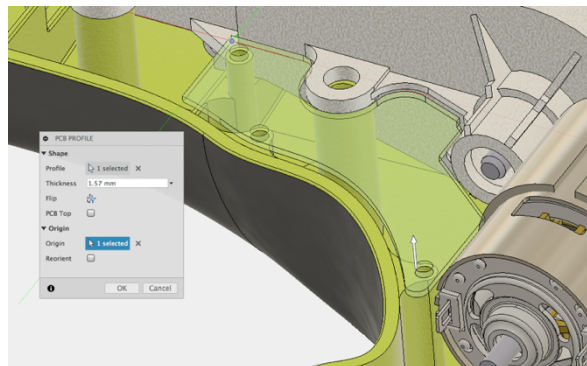
Design Situation #2: A mechanical design dictates the allowable form of a PCB

In other scenarios, the physical form of a mechanical component will dictate the shape, size, and layout of a PCB. These boards are usually manufactured specifically to meet the needs of specific mechanical designs. Examples of this design situation may include:

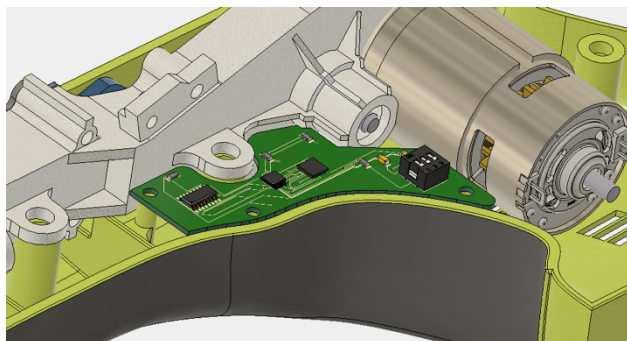
- A PCB must fit in a complex mechanical assembly such as an electric gearbox.
- A graphics card for a desktop computer must fit properly with other components in the tower.

In these situations, it is recommended to create a PCB component in Fusion 360 and pull it into EAGLE. The benefits of using this process are described below:

- The existing mechanical geometry can be used to develop the profile of the PCB. Projected geometry and visual verification can ensure the board will fit correctly and mount properly in a complex mechanical form.



- The need to bring a complicated board profile sketch into EAGLE as a DXF file is eliminated. This increases efficiency and reduces errors due to file conversions.
- The mechanical team can edit the board profile and PCB layout within the Fusion 360 design to confirm mounting positions, fit, and interference due to PCB components.



- If the mechanical team does come across any interference or fit issues, they can edit the PCB within the assembly design using the PCB workspace, after which it can be pulled into EAGLE by the electrical team.

This workflow allows multi-disciplinary design teams to meet difficult design constraints due to unique geometry when designing a PCB for a mechanical assembly.

Design Situation #3: Both the mechanical and electrical components are in the early stages of the design process and may be changed

We may find that the design of both the PCB and the mechanical assembly are still being finalized and are be subject to change. In these scenarios, designers may need to:

- Drive the electrical design off of the mechanical assembly or vise-versa.
- Allow both the electrical and mechanical teams to make changes to the PCB.

Examples of this design situation may include:

- An enclosure is being developed concurrently with the PCB.
- A standardized enclosure design may be used with various PCBs of different sizes.

In this scenario, it is also recommended to start by creating a PCB component in Fusion 360, pull the board profile into EAGLE, apply PCB components, and push the board back to Fusion. We should then use the Fusion board shape to drive the footprint of the PCB.

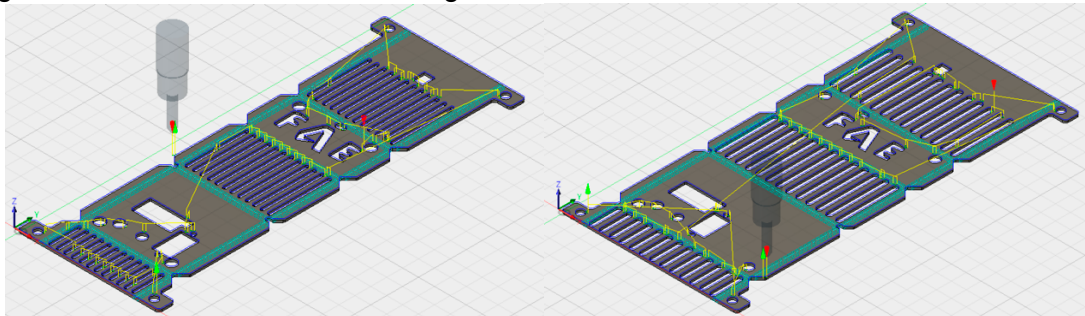
The reason we want to start with a PCB component in Fusion is due to the fact that two board profiles are created when a board design has been pushed and pulled between EAGLE and Fusion. Starting the design with a PCB shape in Fusion and then continuing to use Fusion to make changes to the size or shape of the board presents the least risk of making changes to the board that will damage the assembly design. Using the Fusion board shape to drive the design in this situation also minimizes the risk of losing references to board geometry, as the Fusion board profile is native to the mechanical design and maintained in Fusion through pulls/pushes to/from EAGLE.

Using the Fusion board shape to drive the design is also advantageous because any change made to the Fusion board shape will be shown instantaneously in the assembly design. Updating the EAGLE board shape in Fusion requires a push from EAGLE, but editing the Fusion board shape can immediately update both the electrical and mechanical designs (and any CAM programming present in the Fusion design). The most recent version of the Fusion board shape can then be pulled to EAGLE by the electrical team when changes are made.

Creating the Fusion 360 PCB component first, using it to drive the PCB and mechanical design, and pulling the Fusion board to EAGLE when changes are made will allow designers the freedoms described below:

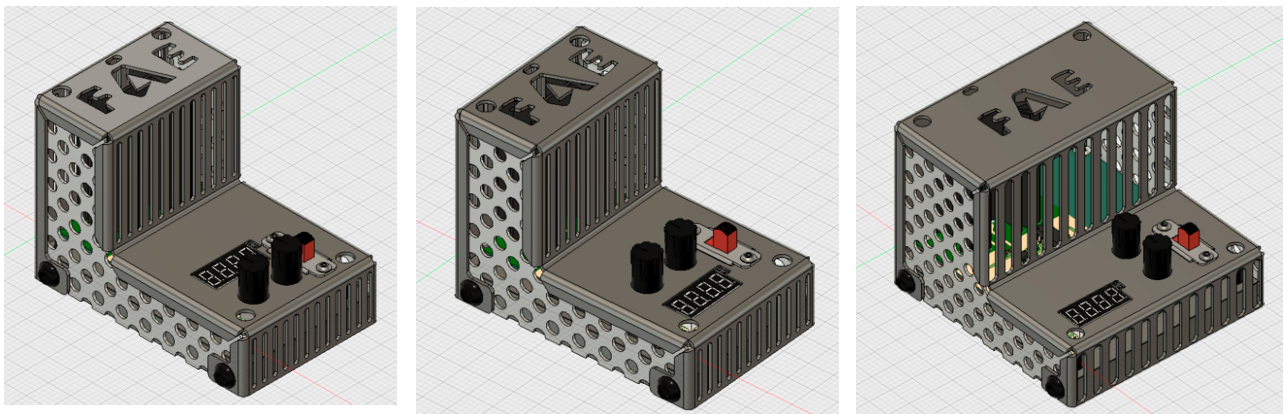
- Changes to the mechanical assembly are visualized immediately when changes to PCB are made.

- The Fusion board shape can be parameterized and used to drive the size, form, and position of the mechanical assembly or vice-versa (the mechanical assembly can be used to drive the Fusion board shape).
- As the CAM workspace requires toolpaths to be regenerated when model changes are made, manufacturing operations for mechanical components altered by a change to the PCB are easily updated. The screenshot at the top of the next page shows the before (right) and after (left) when a PCB board housed in a sheet metal enclosure was changed. The sheet metal pattern was updated and the waterjet toolpath was regenerated to meet the model changes after the PCB was made wider.



- File conversions do not need to be performed when a new version of the PCB is created by the electrical team and mechanical designers do not need to remove old versions of inserted files and insert the new version in assembly designs.

This workflow allows mechanical designers to take advantage of all the parametric modeling tools in Fusion 360 while integrating PCB designs into assemblies. Through use of model parameters and the top-down assembly modeling tools, designers can adapt mechanical assemblies to changes made to the PCB or drive the form and layout of a PCB with mechanical geometry.



The screenshots above show various configurations of a PCB and enclosure design. When changes are made to the Fusion board shape, we can easily visualize the effect on the enclosure, as these changes are shown instantly. The Move PCB Components also allows us to visualize the effect of changes to the PCB layout on the mechanical design, as seen throughout the visuals above.

When using this workflow, it is also important to remember to pay close attention to when the Fusion board shape should be pushed to EAGLE by watching for the arrow icons in the browser

and keeping an eye on the Fusion Sync Button in EAGLE. The Fusion board shape should be pulled to EAGLE anytime the EAGLE board shape does not match the current Fusion board shape.

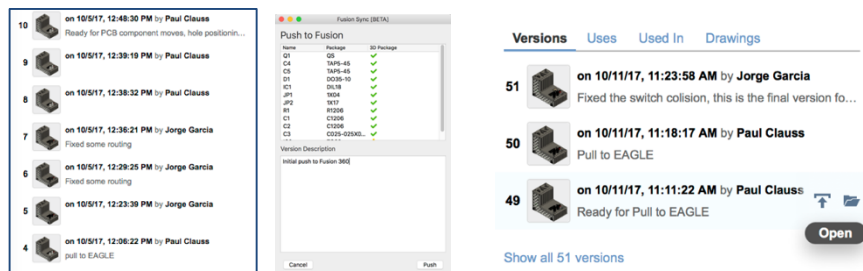
Communication tools for ECAD/MCAD integration in Fusion 360/EAGLE

While the EAGLE and Fusion 360 integration tools make it much easier to manage versions of both mechanical and electrical design files, it is still important to provide documentation for each design iteration, particularly when the PCB component is altered. Fusion 360 provides a variety of tools to include comments on both the mechanical and electrical designs. The PCB Status and Fusion Sync tools in Fusion 360 and EAGLE have already been discussed and the communication tools always available in Fusion 360 are described below:

- *Fusion 360 versioning and version comments*

Each time a Fusion 360 design is saved manually, a new version is created. Whenever a new version is created, Fusion 360 gives the option to include a comment on that specific version. Before a PCB component can be pulled from Fusion 360 to EAGLE, the board must be saved in Fusion. This gives mechanical engineers an opportunity to provide the electrical team with insight into any changes made to the PCB in Fusion 360 in the version description, which will be shown in the Fusion Sync dialogue when pulling to EAGLE.

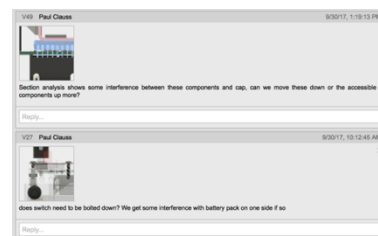
When an electrical engineer pushes a board or board update from EAGLE to Fusion 360, they are creating a new version of the Fusion design and will be given the option to provide comments on any changes made in EAGLE while pushing the board update in the “Version Description” field of the Fusion Sync dialogue.



If an EAGLE board design needs to be rolled back to a previous version and a back-up of the file was not created, the mechanical team can roll the Fusion 360 design back to a previous version (containing the previous version of the PCB design) and the electrical team can then pull the previously used board back into EAGLE from the previous Fusion 360 design version.

- *Comments within a Fusion 360 design*

Because the electrical and mechanical teams will have access to the assembly design in Fusion 360, both teams can use the “Comments” tool in Fusion 360 to provide insight on the design. Screenshots and markups can be included in these comments.



Along with these comments, the Fusion 360 and EAGLE integration requires a new version of the design to be created with each push or pull and when changes made to the board in Fusion are saved, so a new version will be created and visible in the Fusion 360 data panel even if no comments are made.

Notes on CAM operations and PCB manufacturing

One of the most important advantages of the Fusion 360/EAGLE integration is the ability to include PCB designs throughout all the workspaces in Fusion. CAM operations, simulations and renderings can now be created with both the mechanical and electrical components in a single design file. Because the CAM workspace in Fusion 360 will update with model changes, no additional CAM programming needs to be done when mechanical assembly components are altered. This allows for an extremely fast concept to physical prototype cycle as it:

- Reduces file conversions between both the mechanical/electrical designs as well as between the mechanical design/manufacturing files.
- Minimizes repeated work for manufacturing engineers due to model changes.
- Ensures all project stakeholders are aware of changes to either the mechanical or electrical design.

Because the CAM workspace of Fusion 360 cannot create and apply toolpaths to machine traces in a board design, PCB manufacturing plans still need to be created elsewhere. However, because toolpaths created in the CAM workspace of Fusion 360 will regenerate to match model changes, design alterations to the mechanical assembly no longer require CAM programming to be repeated.

- Notes on PCB manufacturing

As the CAM workspace in Fusion 360 is currently incapable of recognizing PCB traces, the manufacturing plans for PCBs still need to be created using the CAM tools in EAGLE or in a different software than Fusion 360. Some desktop milling machines like the Othermill include their own machine controller software that can import Gerber and EAGLE board files.

- Notes on working with EAGLE/Fusion 360 integration in the CAM workspace

Because the electrical, mechanical design, and manufacturing teams can now work in a single design file, manufacturing engineers may also make edits to the PCB based on manufacturing concerns and easily make the electrical team aware of these updates.

When working in the CAM workspace of Fusion 360 with a design utilizing the EAGLE/Fusion 360 integration features, there are no differences from working with a normal Fusion 360 CAM program in which mechanical parts may be altered. Some general tips for maintaining the link between the Model and CAM workspaces of Fusion 360 are included below:

- Remember to regenerate CAM toolpaths and update sheet metal patterns when the mechanical assembly or PCB component is changed.
- When working on CAM toolpaths for components in large assembly design, you may want to insert the entire assembly as an x-referenced component to separate designs. The CAM operations for separate components can be created in separate files using this method, but each CAM file maintains its link to the master assembly file and can be updated with changes made to that file.

- Follow the usual best practices of the CAM workspace in Fusion 360! If you are unfamiliar with the CAM workspace in Fusion 360, the resources linked below should help you get started:
 - [Fusion 360 CAM Learning Materials](#)
 - [Fusion 360 CAM Adoption Portal](#)
 - [Fusion 360: CAM Deep Dive](#)

Additional Resources:

- [The Fusion 360 Forums](#)
- [The EAGLE Forums](#)
- [Fusion 360 Adoption Portal](#)
- [Fusion 360 Learning Materials](#)
- [EAGLE Blog](#)
- [EAGLE/Fusion 360 YouTube Playlist](#)
- [EAGLE Support](#)
- [Fusion 360 Support](#)
- [The Autodesk Knowledge Network](#)

If you need to contact the Fusion 360 technical support team, please either post in the [Support Forum Board](#) or send an email to support.fusion360@autodesk.com. We are happy to hear from you!