

Fusion 360 Electronics Migration Guide for EAGLE Users

Table of Contents

1	<i>Introduction</i>	- 4 -
2	<i>Where Are my Fusion 360 Files Stored?</i>	- 4 -
3	<i>Organize your Electronics Projects in Fusion</i>	- 5 -
	Create Projects in Fusion	- 5 -
	Start a new Electronics Design	- 6 -
	Projects and Folders	- 7 -
	Brand-new Design with a New Schematic	- 8 -
	Bring Your EAGLE Design into Fusion	- 8 -
	Save your files in Fusion	- 10 -
	Share Projects	- 11 -
4	<i>Maximize your editing workspace in Fusion 360 Electronics</i>	- 12 -
5	<i>Work with Multiple Windows and Monitors</i>	- 14 -
6	<i>Where are my Fusion 360 settings and preferences?</i>	- 15 -
7	<i>Menus and settings in Schematic and 2D PCB</i>	- 16 -
	Schematic Editor	- 16 -
	DESIGN menu	- 17 -
	DOCUMENT menu	- 18 -
	VALIDATE menu	- 18 -
	AUTOMATE menu	- 19 -
	LIBRARY menu	- 19 -
	Quick Access toolbar	- 19 -
	2D PCB Editor	- 20 -
	DESIGN menu	- 20 -
	DOCUMENT menu	- 21 -
	RULES ERC/DRC menu	- 21 -
	MANUFACTURING menu	- 22 -
	AUTOMATION menu	- 22 -
	SIMULATION menu	- 22 -
	LIBRARY menu	- 23 -
	Command shortcuts	- 23 -
	Fusion Default Assignment	- 23 -
	Electronics-specific Command Assignment	- 25 -
8	<i>User Language Programs and Scripts in Fusion 360 Electronics</i>	- 26 -
9	<i>Use Your Eagle Libraries and Manage Libraries in Fusion 360</i>	- 27 -
	Library Manager	- 27 -
	Filtering options	- 28 -
	Why are there libraries at different locations?	- 29 -
	What are the library sources?	- 29 -
10	<i>First use of the Library Editor</i>	- 29 -
	Create New Library	- 29 -
	Edit a Managed Library Created in EAGLE	- 30 -
	Import Libraries	- 32 -
	Content Manager in the Library Editor	- 32 -

- Library Editor Overview** - 33 -
 - CREATE menu - 34 -
 - Import from Another Library - 34 -
 - MANAGE menu - 34 -
- Device Editor** - 36 -
 - DEVICE menu - 37 -
- Symbol Editor** - 38 -
- Footprint Editor** - 39 -
- Package Editor** - 40 -

1 INTRODUCTION

This document is designed to give all EAGLE users a hand to safely start with your first steps in Fusion 360 Electronics. Please follow this guide's advice for a smooth transition from EAGLE to Fusion Electronics.

The move to Fusion 360 offers new options compared to EAGLE

- improved and new routing features in the 2D PCB, like Guided Routing, Push Obstacles mode with MOVE
- new polygon algorithm
- easier and tighter 2D/3D collaboration
- loop remove and preserve angles in schematic
- ODB++ export
- improved library experience
- Place Components panel

Fusion 360 is under constant development and so is the list of new options. The mentioned items are as of May 2022.

Additional support can be accessed

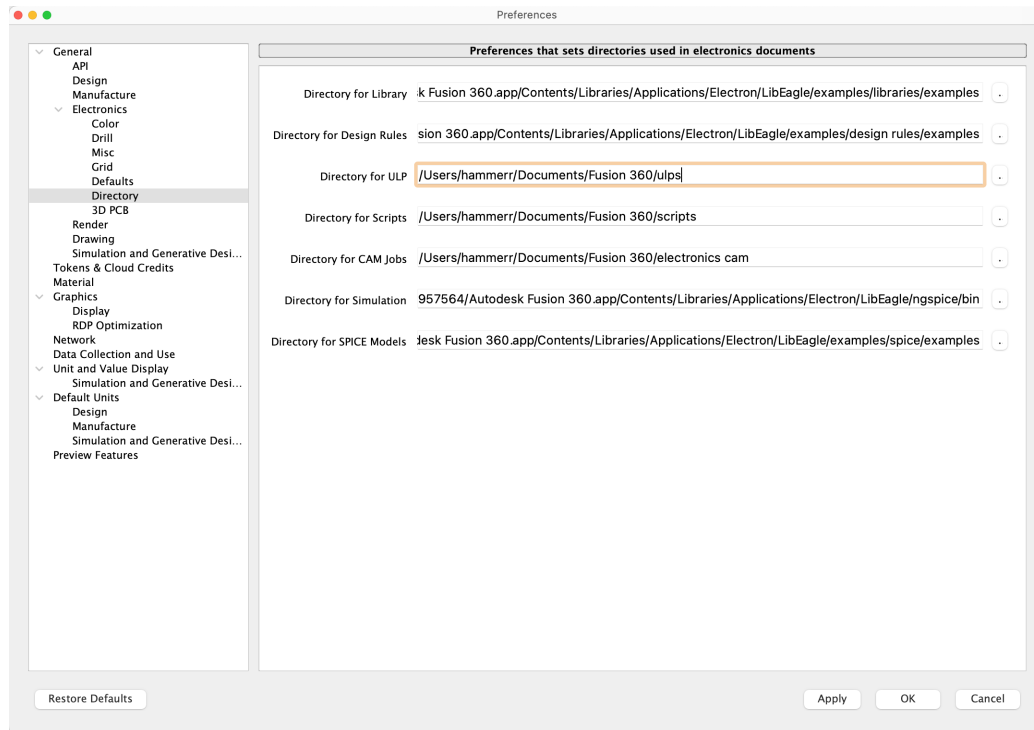
- in-app product information
- in-app self-paced learning content
- in-app learning panel
- in-app chat to connect with a support agent
- Electronics playlist in Autodesk's Fusion 360 YouTube channel
- Autodesk support channel

2 WHERE ARE MY FUSION 360 FILES STORED?

Fusion 360 is a cloud data management product. The electronics project, schematics, 2D PCBs, 3D PCBs, libraries and 3D models are stored in the Autodesk cloud. For more information, see: [Where are my Fusion 360 files?](#) and <https://www.autodesk.com/trust/privacy>.

Additional files like scripts, ULPs, CAM jobs, SPICE models are stored on your computer. The Fusion installer creates a *Fusion 360* folder with subfolders for ULPs, scripts, design rules, cam files, and spice models in your *\$HOME/Documents* folder. This is a good place for all your user files.

In Fusion's preferences *General/Electronics/Directory* menu you could adjust the paths where these files should be located.




1 Path settings for user files

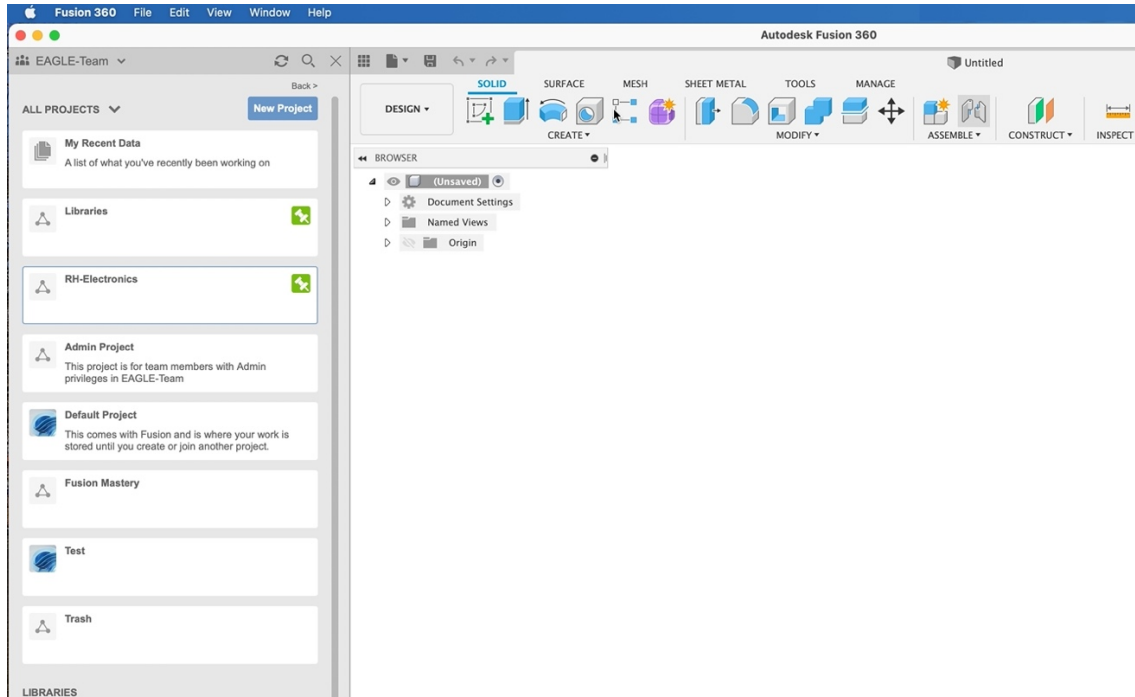
3 ORGANIZE YOUR ELECTRONICS PROJECTS IN FUSION

Before you start working on your electronics projects, think about how you organize your projects. It's good practice to create projects for your electronics designs, libraries, and trash. For collaboration you can easily share your projects.

CREATE PROJECTS IN FUSION

The following image shows the data panel with an electronics project named *RH-Electronics* and a *Libraries* project created and pinned  to the top of the list. A *Trash* folder has also been created.

Click  in the data panel to create projects for your data.



2 Projects in the data panel

Projects to be created:

Libraries Use this project for all library files you created or modified. This will be the single source of truth of all the libraries you edited. If you plan to collaborate with the mechanical world, it helps to have a subfolder in the *Libraries* project for all your 3D models.

Electronics project name Use this project for electronics files (in our example: RH-Electronics). You can also save files in subfolders of the project. There you could for example keep older versions or variations of your design.

Trash Deleting files in Fusion is not always easy. There can be several linked assets/designs in all the versions of a design and Fusion wants you to unlink all of them before you are allowed to delete. This can be tedious. As a quick workaround we use the trash project for files we don't need any more. Files moved into the trash project can be restored, if necessary.

With a double-click onto one of the project entries in the data panel you dive into project level. To create subfolders, click the  icon and name the folder.

For the Libraries project you could create a sub-folder named *3D models* in case you prefer to have libraries and 3D models in separate locations.

START A NEW ELECTRONICS DESIGN

In this section you learn how to create an Electronics Design and then start with a new schematic or reference to an already existing (EAGLE) schematic and board.

Before you create your schematic and PCB, you need to create an electronics design file. This is the place where information about the name and location of your current schematic and its related board in 2D and 3D is gathered. It also shows a preview of schematic, 2D

PCB and 3D PCB in top and bottom view, as soon as available.

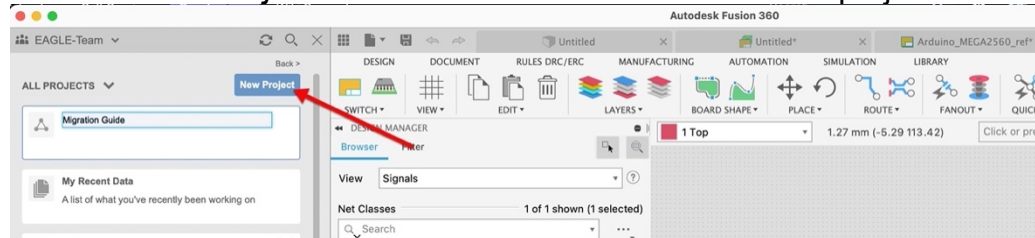
One schematic refers to one 2D PCB. In comparison to EAGLE, schematic and board can have different names and can be stored at different locations. Changes in a schematic will be annotated automatically to the 2D PCB and vice versa. This works the same way as you are used to in EAGLE.

For a flawless annotation between schematic and board, be sure to have both files opened at the same time.

PROJECTS AND FOLDERS

Before you start with the Electronics Design, think about where to save your files and how to organize your projects. It's good practice to create a *New Project* first, or a *New Folder* in an already existing project. This is done in the Fusion data panel. It is best to choose a meaningful name.

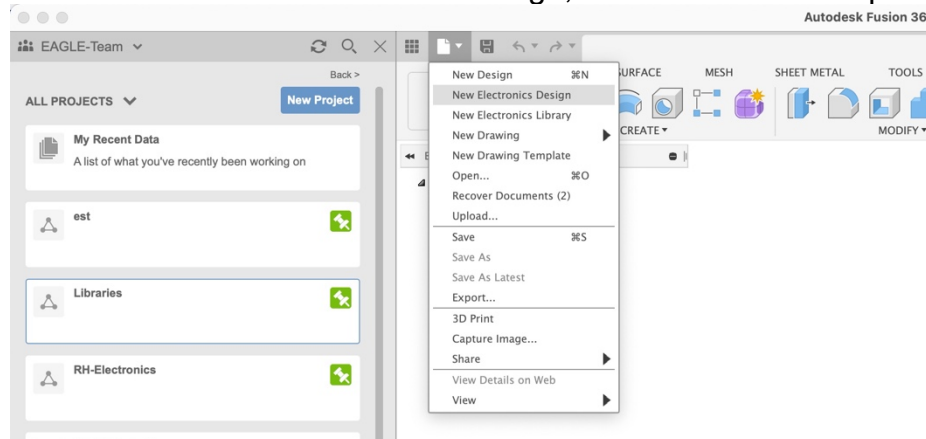
Click the *New Project* button and create a new electronics project named *Migration Guide*.



3 Create a New Project in the data panel

Type the name of the project followed by the enter key. The project has been created now. With a double-click onto the new entry, you navigate into project level.

In order to create an electronics design, select *File* and then pick *New Electronics Design*.



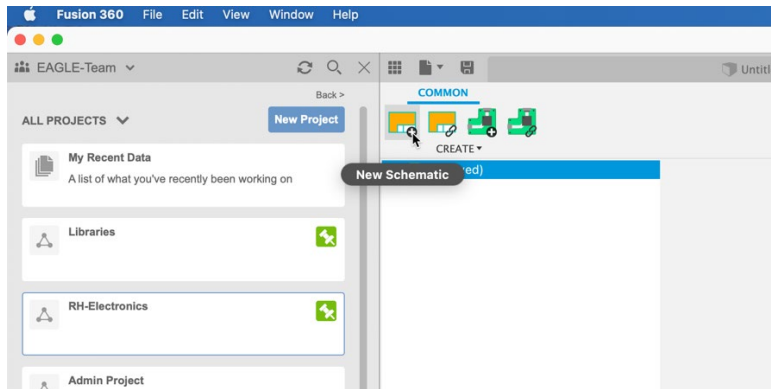
4 Starting a new Electronics Design

Now a new tab with an empty electronics design document opens. From here you start with your schematic.

An Electronics Design file links your schematic and board. This is the only way to maintain syncing between the schematic and the 2D PCB. Unlike EAGLE, the Fusion 360 schematic and the 2D PCB are not required to have the same name and do not need to be stored in the same folder.

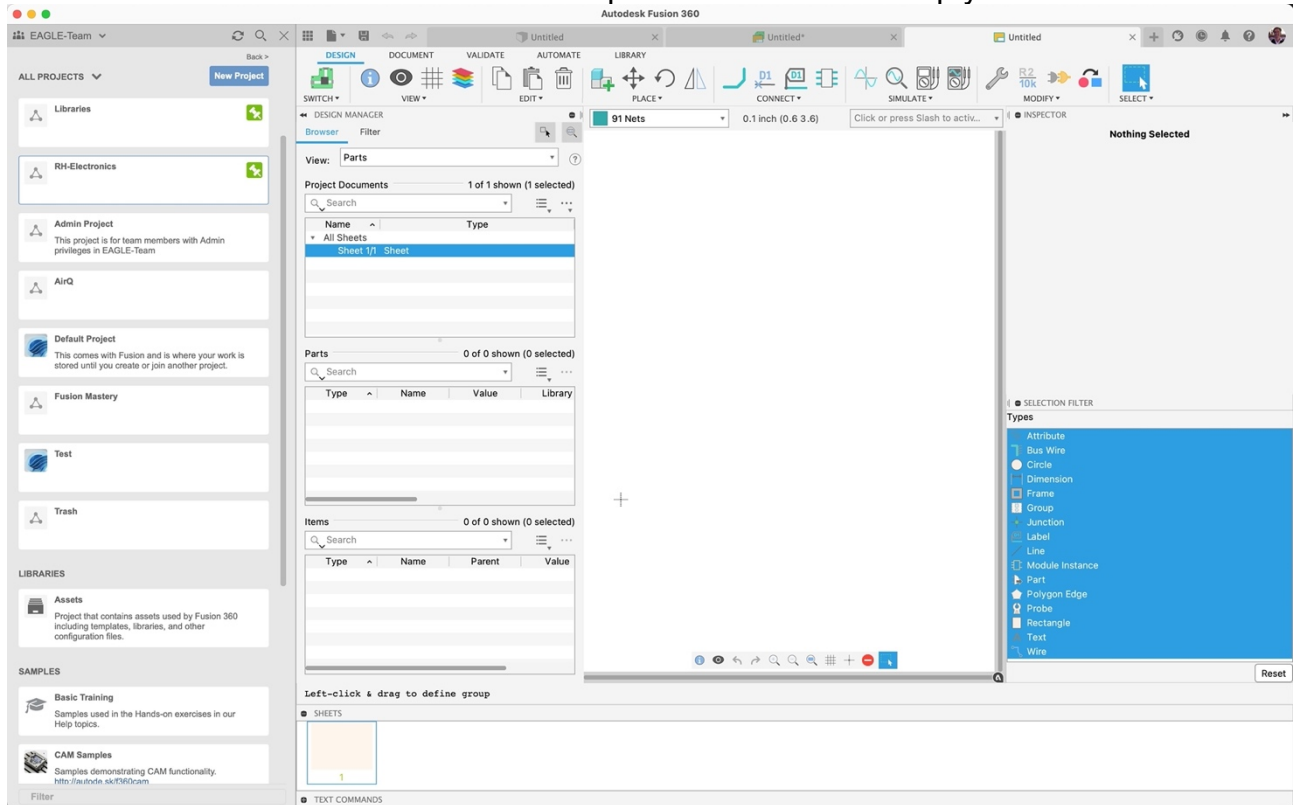
BRAND-NEW DESIGN WITH A NEW SCHEMATIC

To create a new schematic, click the *New Schematic* icon.



5 New Schematic icon in Electronics Design

Now a new tab with the schematic editor opens and shows an empty schematic.



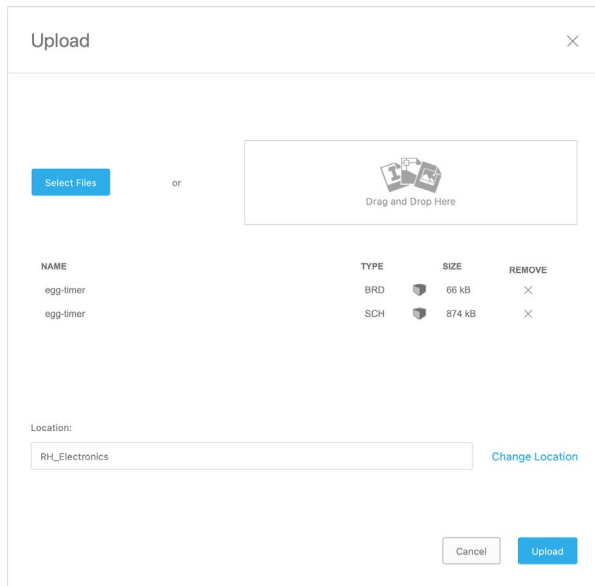
6 The schematic editor with a new schematic

Next steps:

- save your files
- start working on your schematic

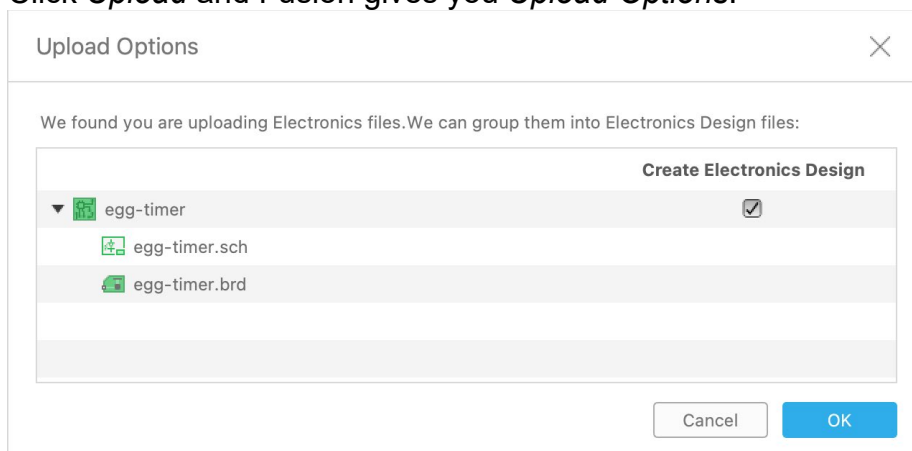
BRING YOUR EAGLE DESIGN INTO FUSION

You can easily transfer an EAGLE project to Fusion. Click the **Upload** button in the data panel. In the *Upload* dialog choose *Select files* and select the schematic and/or board files from your computer or just drag and drop your files into the dialog.



7 Upload dialog – sch and brd are selected and ready for upload

Click *Upload* and Fusion gives you *Upload Options*.

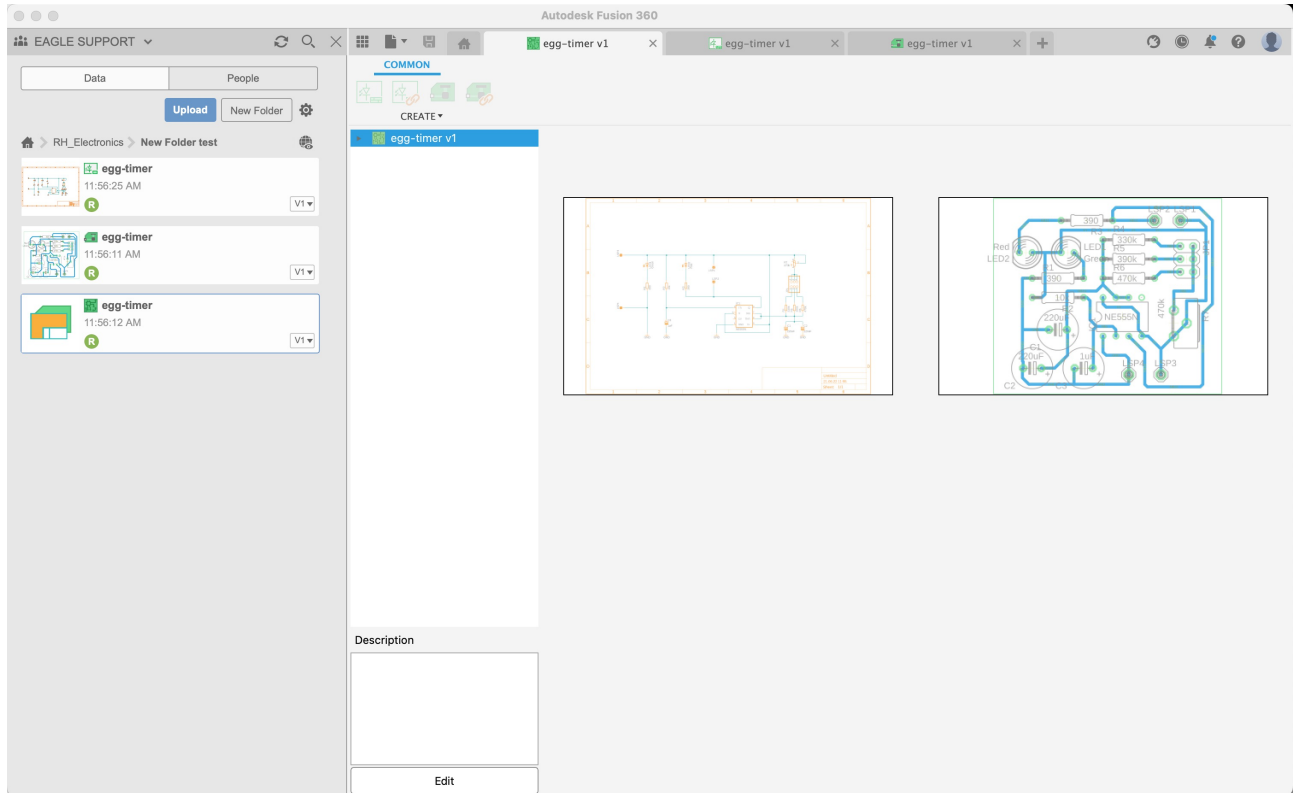


8 Upload options

If your EAGLE schematic and board files have the same name and are stored in the same folder, Fusion suggests to automatically create an electronics design file for the selected files. Click *OK* and copies of your EAGLE files will be automatically uploaded into your Fusion environment into the currently active folder. An Electronics Design file that references your schematic/board pair will be created automatically by default.

The original EAGLE files will remain untouched on your computer and won't be changed when you continue your work in Fusion.

Now double-click the Electronics Design preview in the data panel. Three tabs will be opened now. Electronics Design file, schematic and 2D PCB.

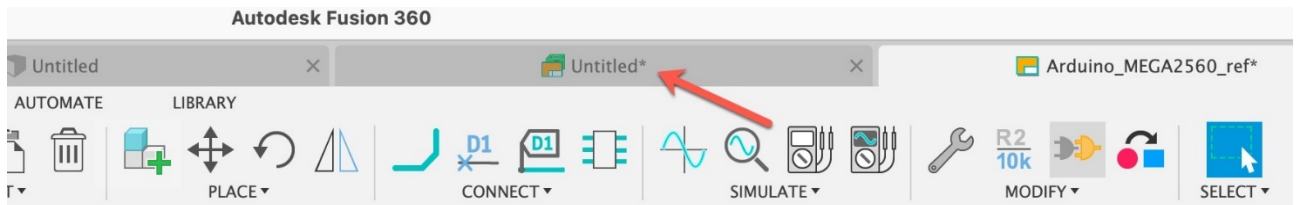


9 Example of an Electronics Design file with preview of schematic and board

SAVE YOUR FILES IN FUSION

This section shows how to save your project files.

With our EAGLE project migrating, we have three electronics related tabs open. Click the tab named *Untitled ** in the open file tabs. This is the electronics design you started with. It shows a preview of the schematic and board assigned.

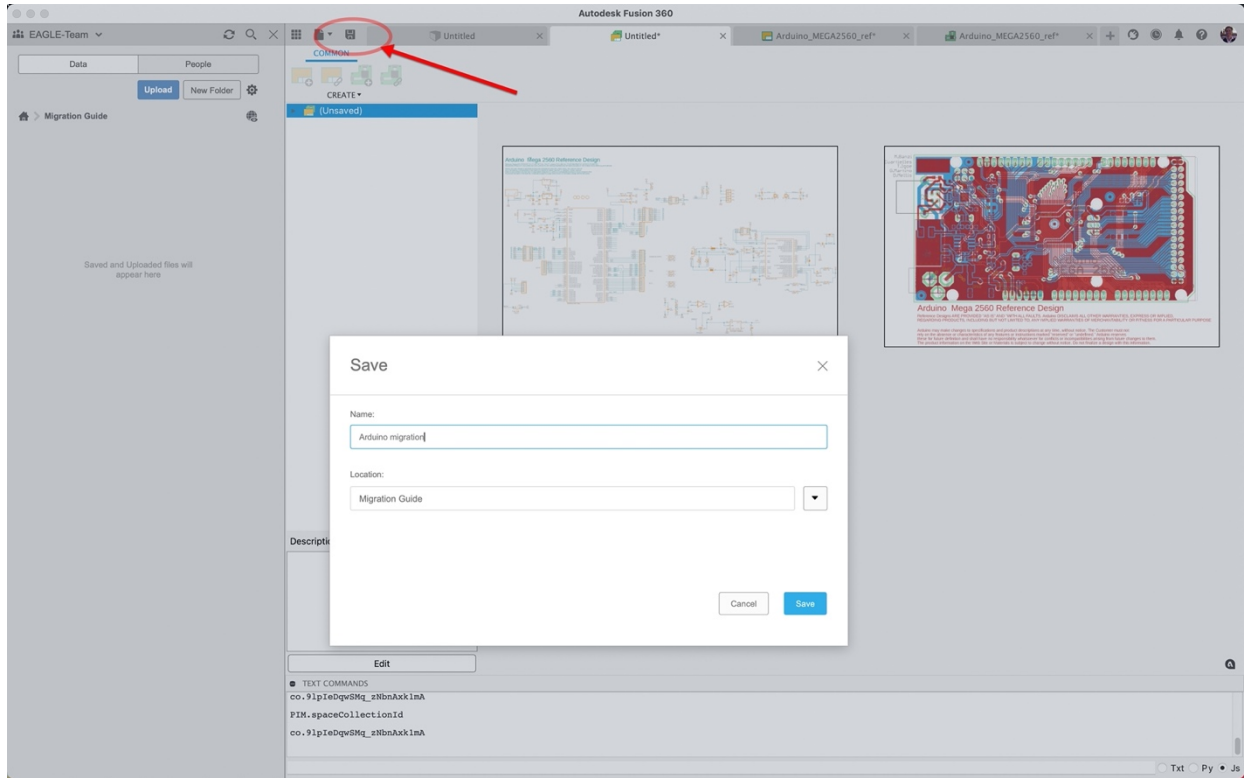


10 The Electronics Design tab, not saved and still named Untitled.

The electronics design window and all editor windows have a Save icon. In this example, schematic and 2D PCB are saved. A save in the electronics design file can also save an associated 3D PCB, if existing.

Click the Save icon in the Electronics Design window (currently named Untitled) to save all related files at the same time.*

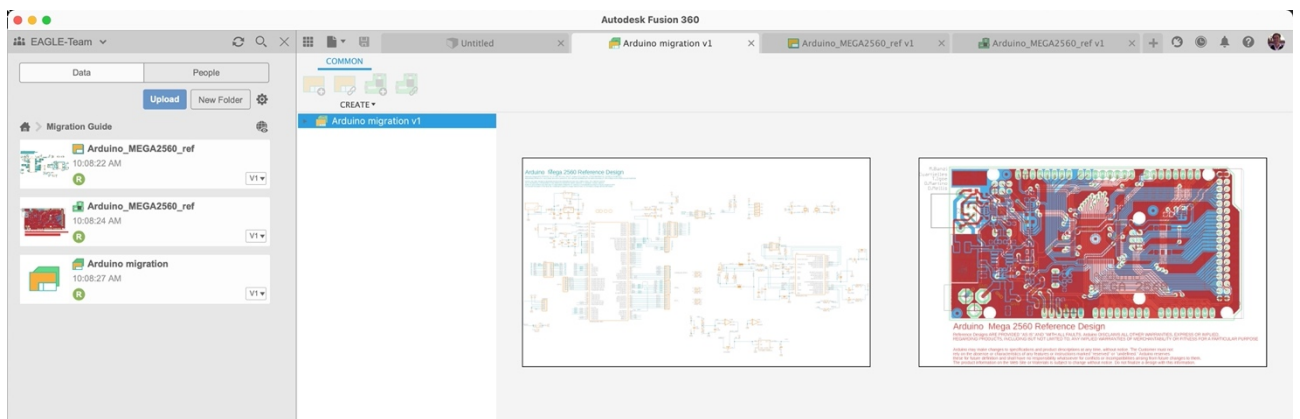
*The * in the title of the tab indicates that the current state of the electronics design is not yet saved.*



11 Save icon in the Electronics Design window

Save the electronics design file with a meaningful name. In this example, the file is saved as *Arduino migration* design. The schematic and board files are saved with their original names now.

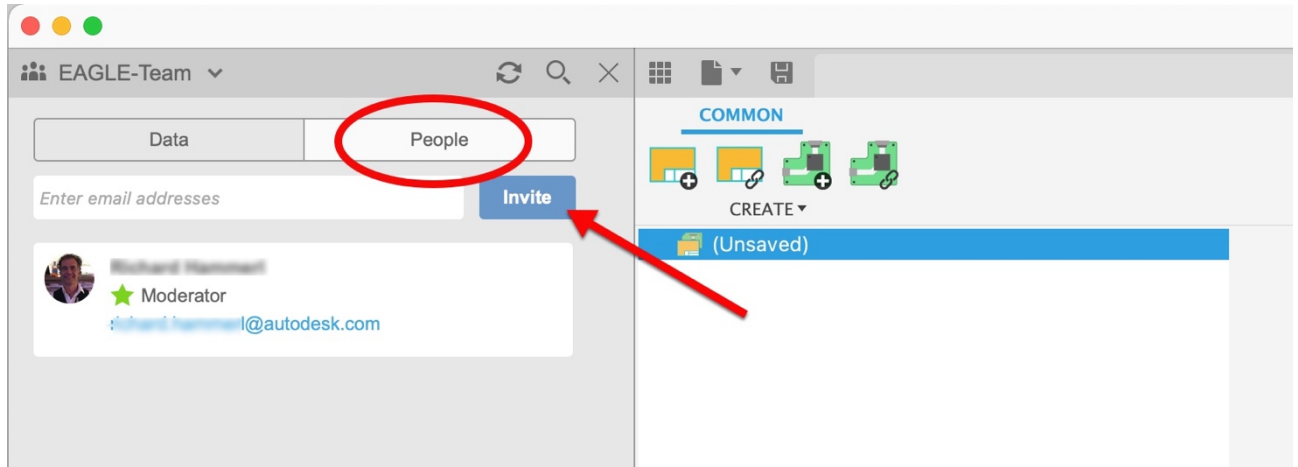
In the data panel on the left side, we now see the three files saved. From top to bottom: the schematic, the 2D PCB and the electronics design file named *Arduino migration*. These three files are stored in a project named *Migration Guide*.



12 Electronics design preview of schematic and board now migrated to Fusion

SHARE PROJECTS

You can easily share Projects with your colleagues. Activate a project folder in the data panel and then click the *People* tab at the top. Enter the email address of the person you want to give access to, then click *Invite* to send the invitation. The recipient receives an email then.



13 Data Panel - Invite People

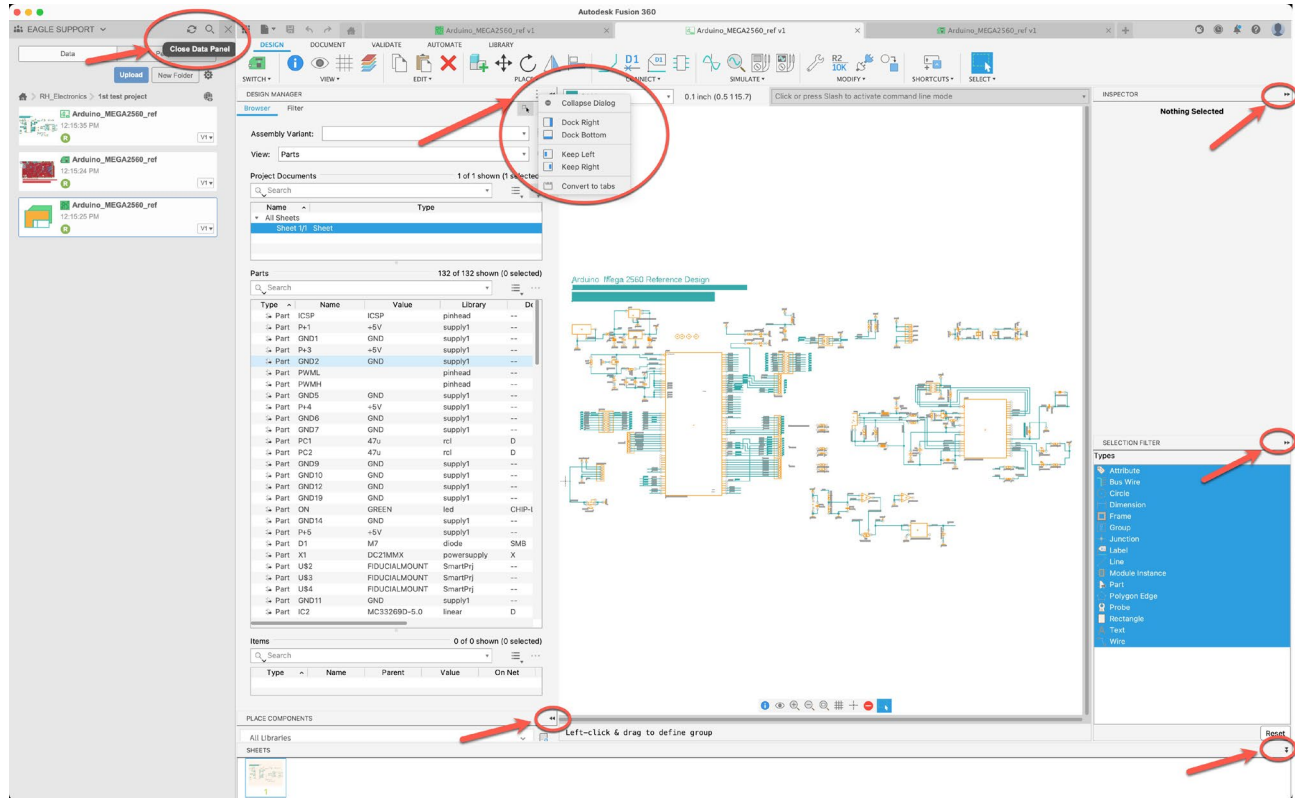
4 MAXIMIZE YOUR EDITING WORKSPACE IN FUSION 360 ELECTRONICS

In its editor windows Fusion shows several panels by default. Learn how to minimize, move, and restore them for best working experience.

The initial settings in the editor windows show all available panels, the *Data Panel*, the *Design Manager*, the *Place Components* panel, the *Inspector* and *Selection Filter* panel, and the *Sheets* preview. They consume a notable amount of space on your screen. You can collapse those panels you do not need now.

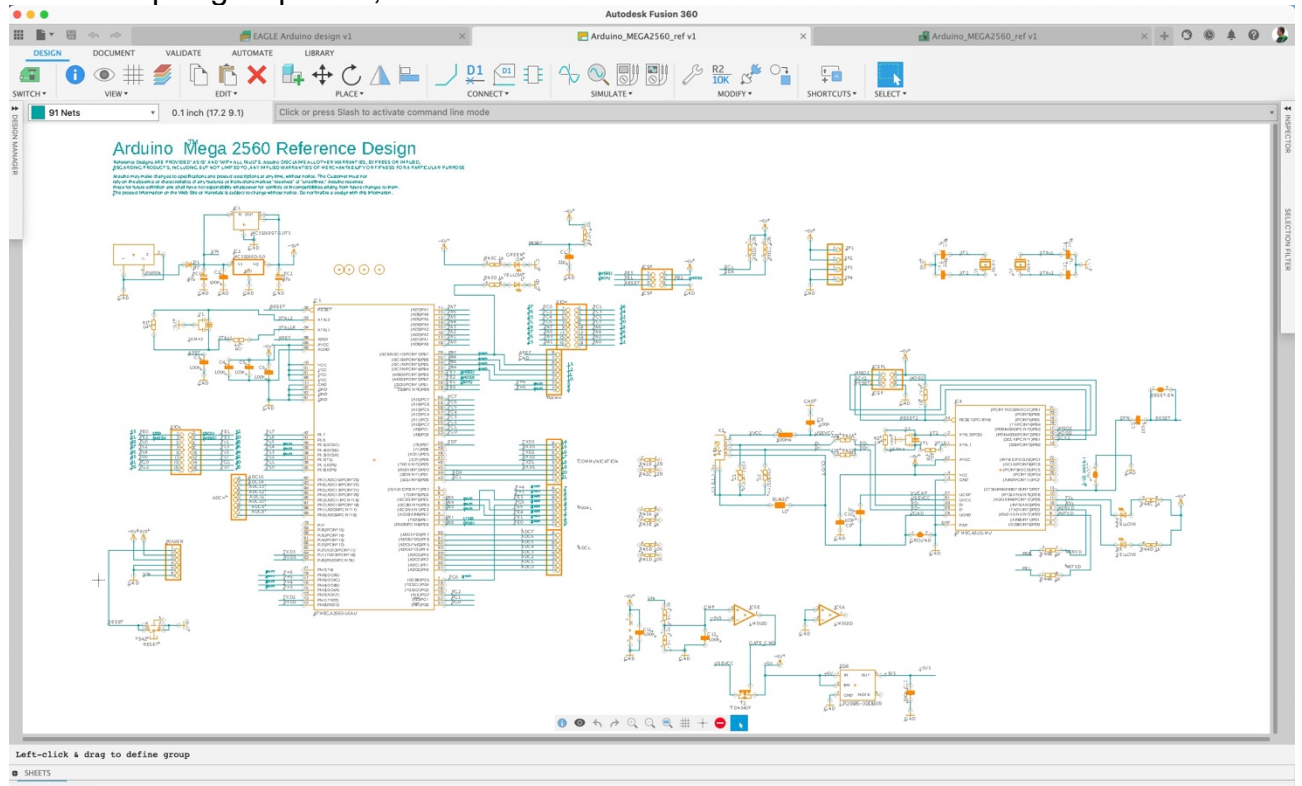
Please open the schematic editor with a double-click onto the preview in the electronics design document or with a click onto the schematic's tab. The following image shows the handles you can use to minimize or collapse each panel. Close these panels to maximize the drawing area.

With a right mouse click onto the title bar of one of the panels a context menu with several options appears. You could for example convert stacked panels to tabs.




14 Position of handles to collapse panels

After collapsing all panels, the schematic window looks like this:



15 Schematic editor with all panels collapsed

Hover with the mouse cursor over a tab of one of the collapsed panels for a preview. Click the handles again to unhide panels. Click the *Show Data Panel* icon  in the top left corner to make the *Data panel* visible.

This works the same in the 2D PCB and in the library editor!

You could also rearrange the *Design Manager*, *Inspector*, *Selection Filter*, and *Sheets* panels. Drag them out of the editor window and move it, for example, to a second screen. Grab the panel on the title bar to drag.

For more information on working with panels see:

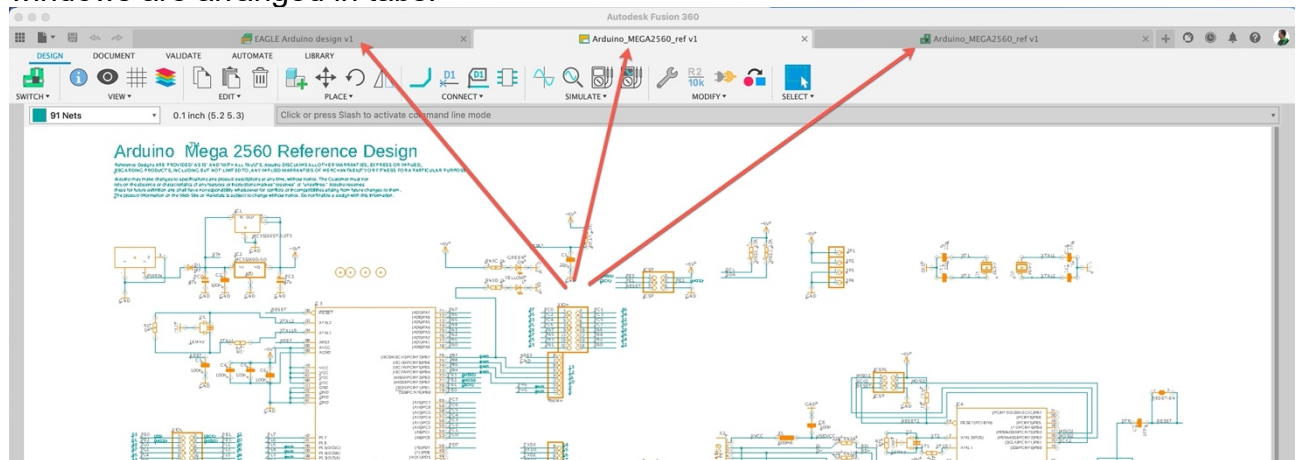
<https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-INFO-PANELS>

5 WORK WITH MULTIPLE WINDOWS AND MONITORS

Fusion supports working with multiple windows on multiple monitors. This section shows how to drag out the editor tabs from the Fusion main window.

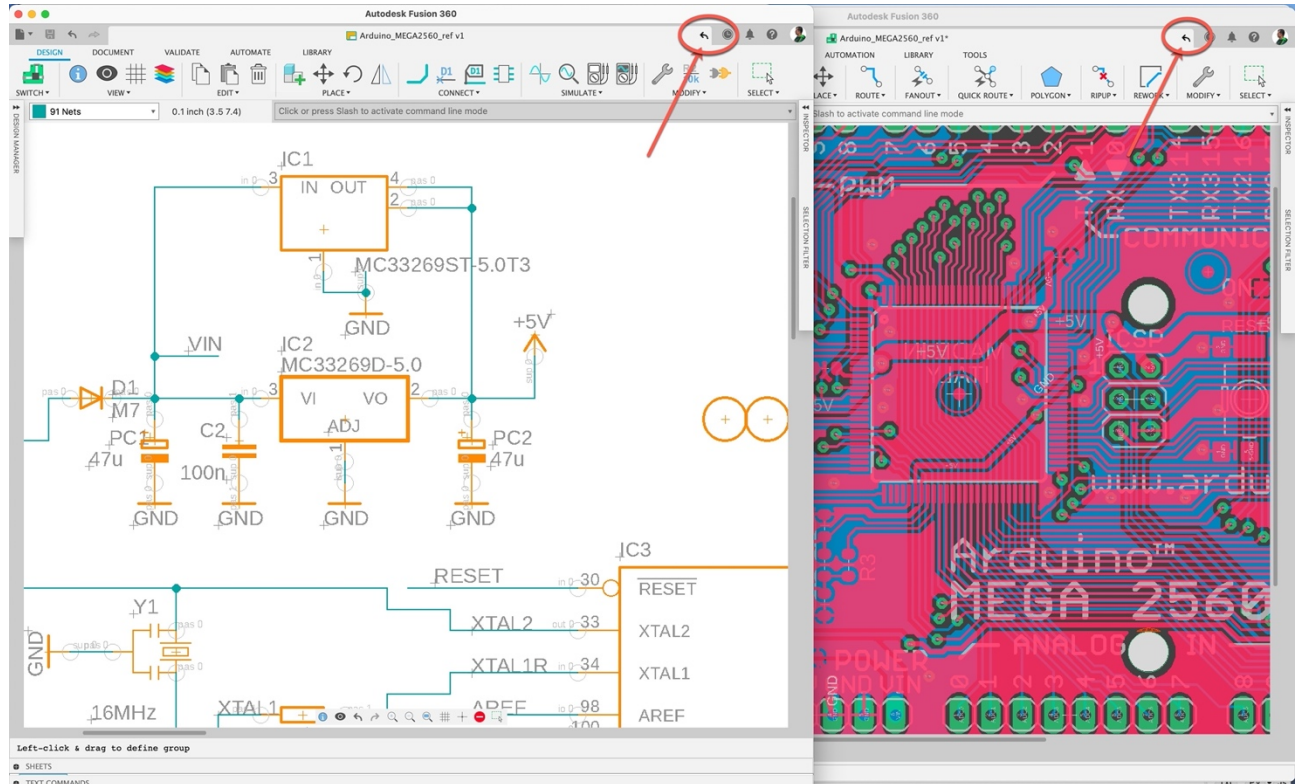
Many EAGLE users work with two monitors. One monitor displays the schematic, and the other displays the board. Highlighting an object in the schematic also highlights the object in the board.

You can use these workflows in Fusion 360 Electronics. When you start an electronics design, create a schematic, and create the 2D PCB from the schematic, the editor windows are arranged in tabs.



16 Editor windows are tabbed by default

If you have two or more monitors, you could drag, for example, the schematic tab to a second monitor and the 2D PCB to a third. To move a tab, click the tab and drag it out of the Fusion window. When you release the mouse button, the tab is a separate window. Move the new window to the desired position.



17 Schematic and board in two separate windows

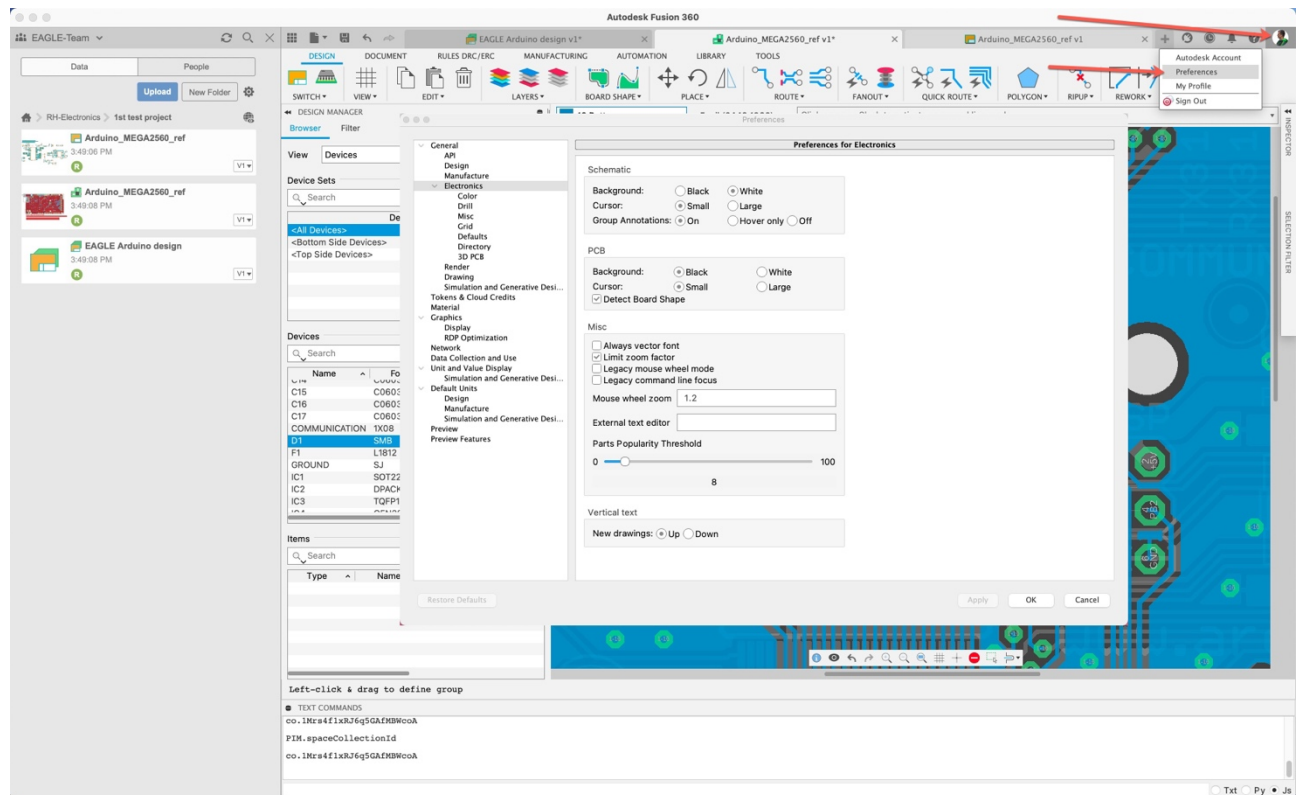
To return a window to the default tabbed view, click the back arrow in the separated window as shown circled in the previous image.

For more information, see: <https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-UNDOCK>

6 WHERE ARE MY FUSION 360 SETTINGS AND PREFERENCES?

User Interface, general settings, and default settings for editor windows known from EAGLE are all available in Fusion's Preferences menu.

The settings in EAGLE's *Options/User interface* and *Options/Set* menus and EAGLE Control Panel's *Options/Directories* menu are in one place in Fusion 360. Click your avatar in the upper right corner and choose *Preferences*. Select the *Electronics* section and change the settings to match the settings from EAGLE's *User interface*. You can also set your preferences in the *Color, Drill, Misc, Grid, Defaults, Directory* and *3D PCB* categories.



18 Preferences for Electronics in Fusion

7 MENUS AND SETTINGS IN SCHEMATIC AND 2D PCB

In this section we explore the schematic and layout editor and give an overview about the commands and tools available

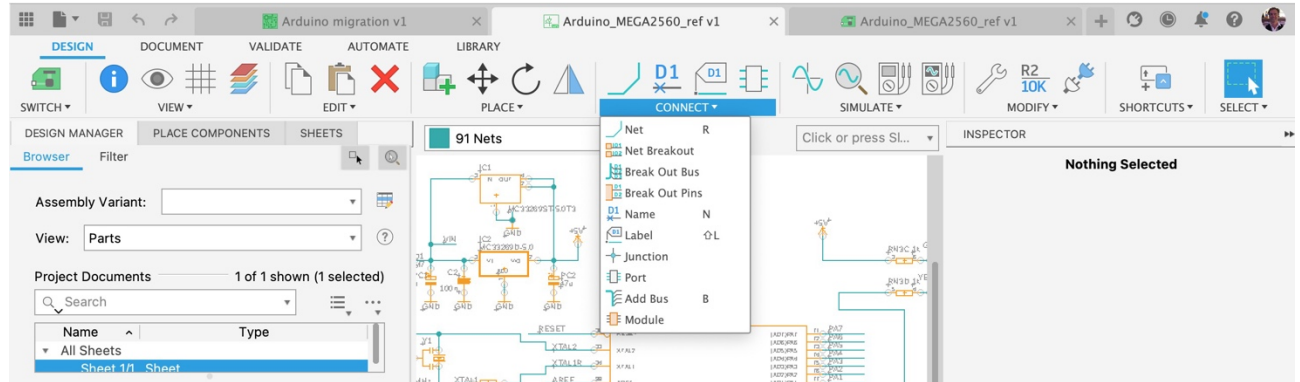
SCHEMATIC EDITOR

Fusion Electronics supports the same commands and tools you are familiar with in EAGLE plus some additional new options. The commands are available in the icon bar on top of the window.

In the following image the *DESIGN* menu shows all relevant commands in the *VIEW*, *EDIT*, *PLACE*, *CONNECT*, *SIMULATE*, and *SELECT* sections. Frequently used commands are accessed through the icons directly. Submenus contain all available commands. The *CONNECT* submenu, for example, additionally lists *Junction*, *Add Bus*, and *Module*.

On the left, selecting the *Switch* icon opens the 2D PCB editor. Next to it are the *INFO*, *SHOW*, *GRID* and *Layers* icons. These tools and the *SELECT* tool are always accessible.

DESIGN MENU



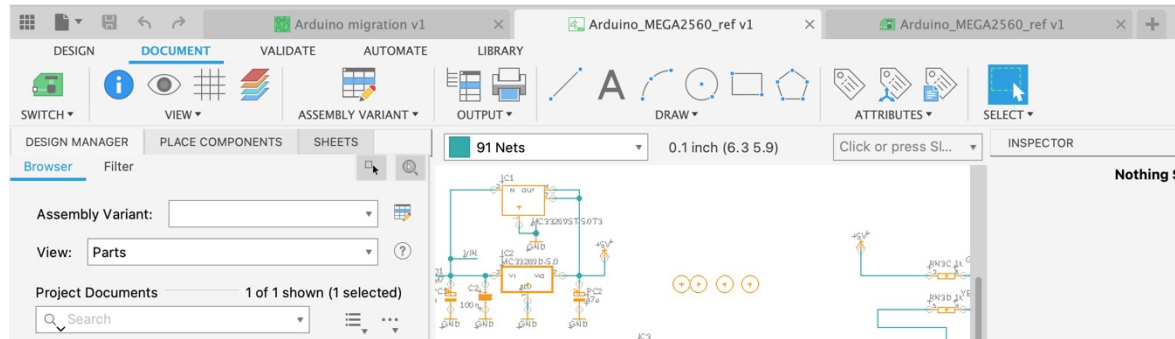
19 DESIGN menu with opened submenu CONNECT in the Schematic Editor

Tools accessible in the *DESIGN* menu:

<p>SWITCH Switch to PCB document</p> <p>VIEW Info Show Grid settings Display Zoom to Fit Redraw Zoom Window</p> <p>EDIT Copy Paste Delete Add Custom 3D Model Edit Custom 3D Model Remove Custom 3D Model</p> <p>PLACE Place Component Move Rotate Mirror Align Insert Schematic Insert Design Block</p> <p>CONNECT Net Net Breakout Break Out Bus Break Out Pins Name Label Junction Port Add Bus Module</p>	<p>SIMULATE Simulate Show Simulation Overlay Analog Source Setup Digital Source Setup Voltage Probe Phase Probe Add Spice Model Remove Spice Model</p> <p>MODIFY Change Value Split Miter Slice Invoke Replace Copy Format Pattern Pin Swap Gate Swap Optimize</p> <p>SHORTCUTS Shortcuts</p>
--	--

Additionally, explore the DOCUMENT, VALIDATE, AUTOMATE, and LIBRARY menus to become familiar with the available commands.

DOCUMENT MENU

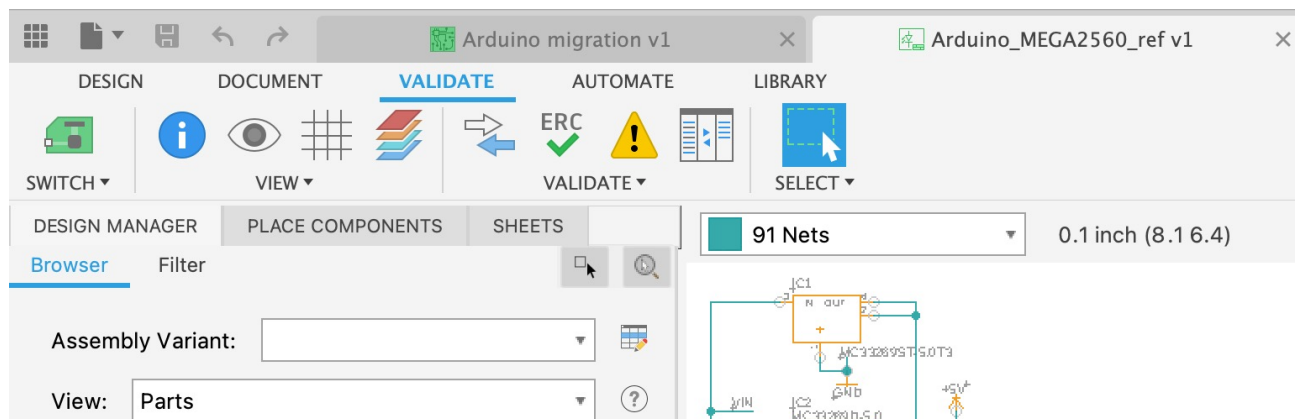


20 DOCUMENT menu with its specific sections Assembly Variant, Output, Draw.

Tools accessible:

<p>ASSEMBLY VARIANT Assembly Variant</p> <p>OUTPUT Bill of Materials Print</p>	<p>DRAW Line Text Arc Circle Rectangle Polygon Shape</p> <p>ATTRIBUTES Attribute Reposition Attributes Document Attributes</p>
---	---

VALIDATE MENU

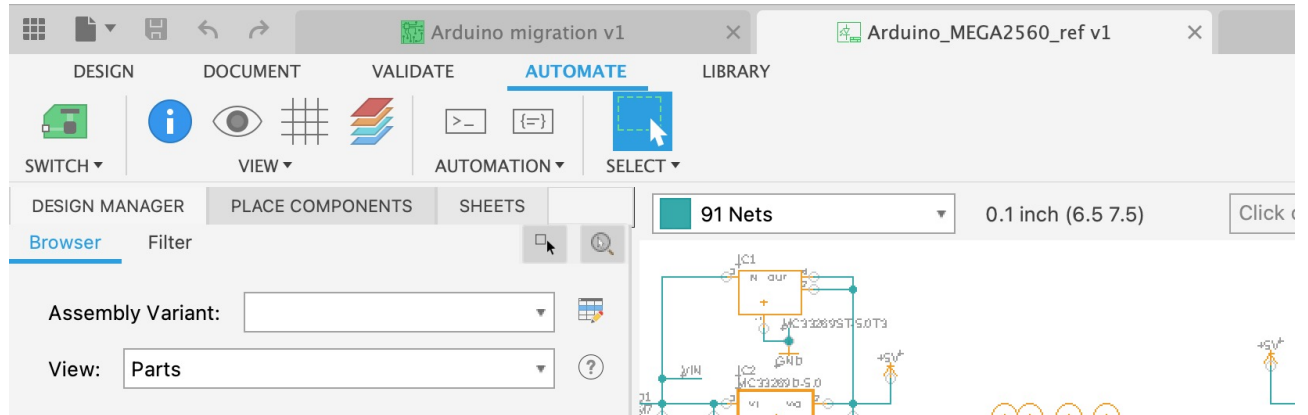


21 VALIDATE menu with its specific section VALIDATE

Tools accessible:

<p>VALIDATE Synchronize ERC Errors Class</p>	
---	--

AUTOMATE MENU

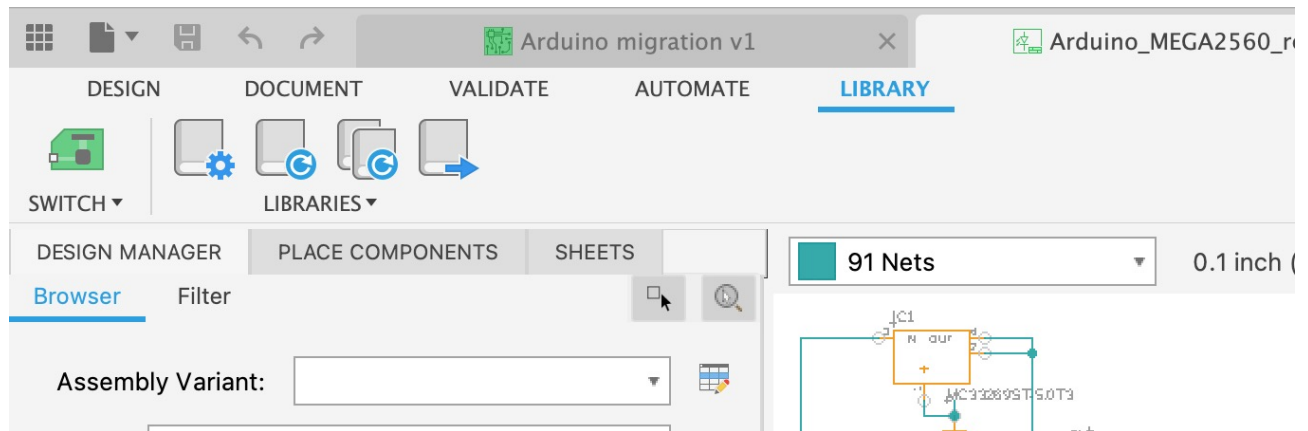


22 AUTOMATE menu with its specific section Automation

Tools accessible:

Automation Run ULP Run Script	
-------------------------------------	--

LIBRARY MENU



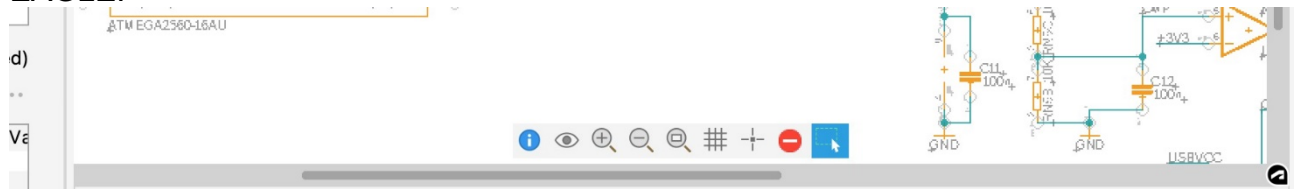
23 LIBRARY menu in the Schematic editor

Tools accessible:

Libraries Open Library Manager Update design from library Update design from all libraries used in design Export Libraries	
--	--

QUICK ACCESS TOOLBAR

At the bottom of the drawing area is the menu bar comparable to the *Action Toolbar* in EAGLE.



24 Quick access menu

The Schematic editor displays the commands *INFO*, *SHOW*, *Zoom In*, *Zoom Out*, *Zoom to Fit*, *GRID*, *MARK*, *STOP*, and *GROUP*.

For more information, see:

<https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-OVERVIEW>

<https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-CREATE-SCHEMATIC>

2D PCB EDITOR

The 2D PCB editor contains all the available commands for working in the layout.

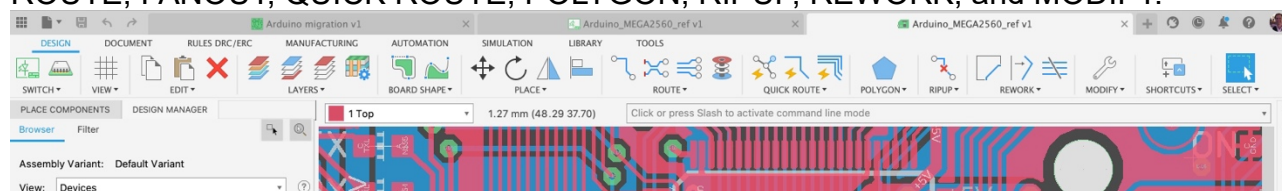
The *Switch* icons on the left allow you to *Switch to Schematic document* or to *View 3D PCB*.

Next to it is the *VIEW* section with the *Grid Settings*. Additional view options *INFO*, *SHOW*, *Zoom to Fit*, *Redraw*, and *Zoom window* are available in the submenu.

At the far right of the ribbon is the *SELECT* tool for selecting groups of objects.

DESIGN MENU

In the Design menu there are subsections for *EDIT*, *LAYERS*, *BOARD SHAPE*, *PLACE*, *ROUTE*, *FANOUT*, *QUICK ROUTE*, *POLYGON*, *RIPUP*, *REWORK*, and *MODIFY*.



25 DESIGN menu in 2D PCB

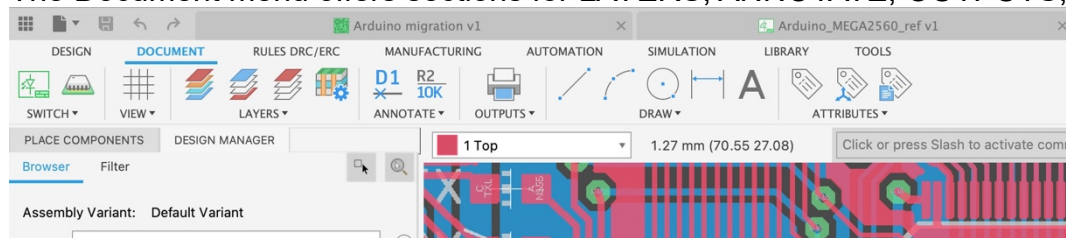
Tools accessible:

<p>EDIT</p> <ul style="list-style-type: none"> Copy Paste Delete Add Custom 3D Model Edit Custom 3D Model Remove Custom 3D Model <p>LAYERS</p> <ul style="list-style-type: none"> Display Flip Board Toggle Single Layer View Layer Stack Manager <p>BOARD SHAPE</p> <ul style="list-style-type: none"> Outline Polyline Outline Spline Outline Arc Outline Circle <p>PLACE</p> <ul style="list-style-type: none"> Place Component Signal Move Rotate Mirror Align Hole (NPTH) 	<p>QUICK ROUTE</p> <ul style="list-style-type: none"> Quick Route Airwire Quick Route Signal Quick Route Multiple Quick Route Smooth Quick Route Guided Fanout Autorouter <p>POLYGON</p> <ul style="list-style-type: none"> Polygon Pour Polygon Cutout Polygon Shape Polygon Pour from Outline Hide all Polygon Pour fills Show all Polygon Pour fills Re-fill Polygon Pours <p>RIPUP</p> <ul style="list-style-type: none"> Ripup All Ripup Connected Ripup Part to Part Ripup Signal <p>REWORK</p> <ul style="list-style-type: none"> Miter Split Slice
---	---

<p>Pattern Insert 2D PCB Insert Design Block</p> <p>ROUTE</p> <p>Route Manual Differential Pair Route Multiple Via Meander</p>	<p>Ripup and Route Device or Signal Meander Optimize</p> <p>MODIFY</p> <p>Change Name Value Replace Copy Format Lock</p> <p>Lock Lock all Lock Current Layer Unlock all</p> <p>Pin Swap</p> <p>SHORTCUTS</p> <p>Shortcuts</p>
---	---

DOCUMENT MENU

The Document menu offers sections for **LAYERS**, **ANNOTATE**, **OUTPUTS**, and **DRAW**.



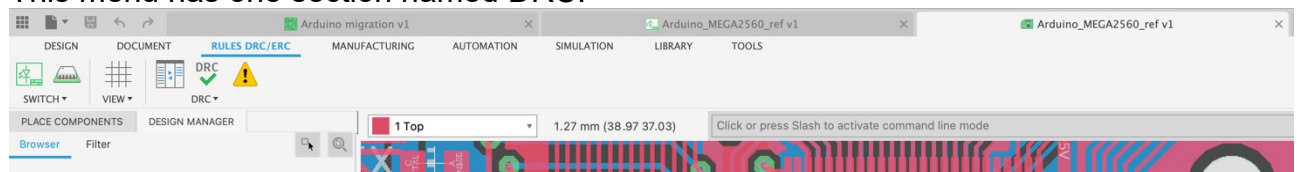
26 DOCUMENT menu in the 2D PCB

Tools accessible:

<p>LAYERS</p> <p>Display Flip Board Toggle Single Layer View Layer Stack Manager</p> <p>ANNOTATE</p> <p>Name Value</p> <p>Outputs</p> <p>Print</p>	<p>DRAW</p> <p>Line Arc Circle Text Rectangle Polygon Shape Mark Dimension</p> <p>ATTRIBUTES</p> <p>Attribute Reposition Attributes Document Attributes</p>
---	--

RULES ERC/DRC MENU

This menu has one section named **DRC**.



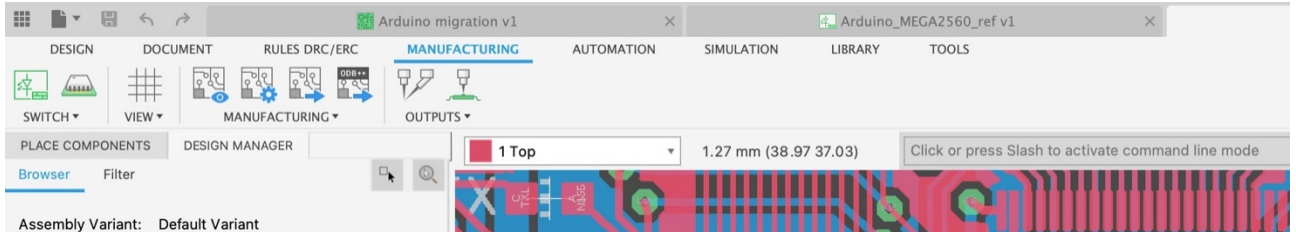
27 RULES menu in 2D PCB

Tools accessible:

DRC Class DRC Errors	
-------------------------------	--

MANUFACTURING MENU

This menu has sections for MANUFACTURING and OUTPUTS.

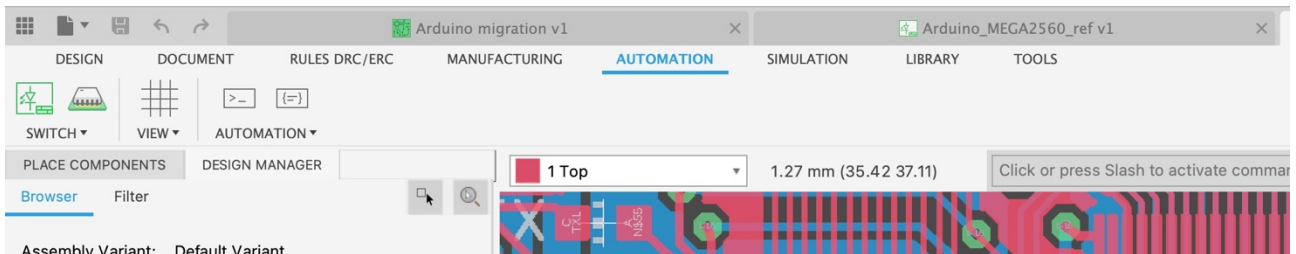


28 MANUFACTURING menu in 2D PCB

Tools accessible:

MANUFACTURING CAM Preview CAM Processor Export Gerber, NC Drill, Assembly and Drawing Outputs Export ODB++ Output	OUTPUTS IPC-D-356 Netlist PickAndPlace
--	--

AUTOMATION MENU

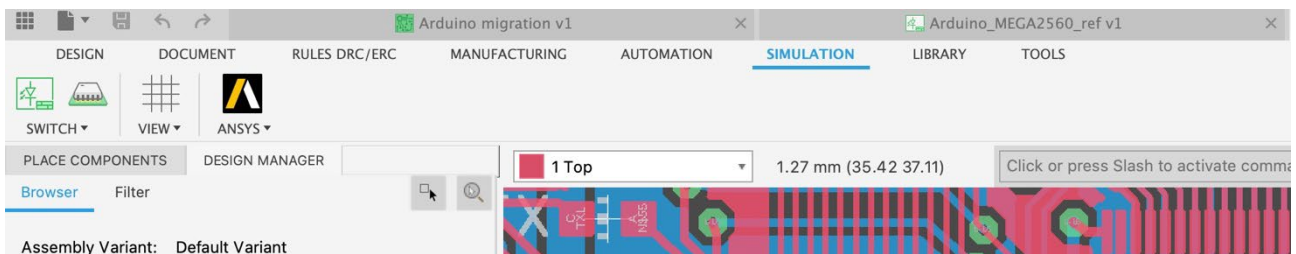


29 Automation menu in the 2D PCB editor

Tools available:

Automation Run ULP Run Script	
-------------------------------------	--

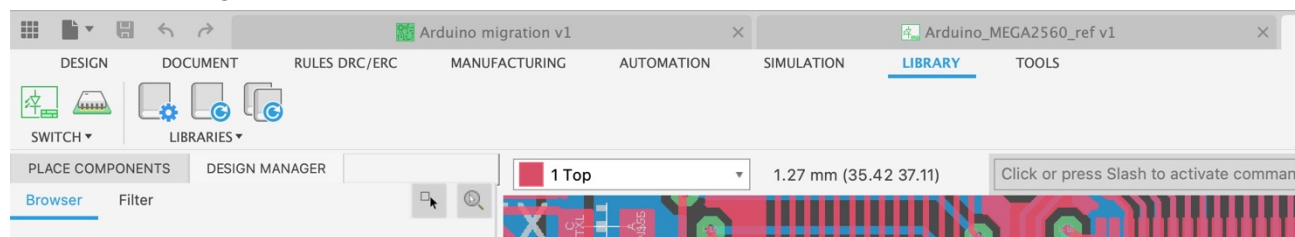
SIMULATION MENU



30 Simulation menu in the 2D PCB editor

Tools available:

Simulation Export PCB Data for Ansys	
---	--

LIBRARY MENU**31 Library menu in the 2D PCB editor**

Tools available:

Library	
Open Library Manager	
Update design from library	
Update design from all libraries used in design	

COMMAND SHORTCUTS

Fusion Electronics has an expanded shortcuts system. Besides the Fusion-like key assignments it allows you to assign Electronics-specific commands or command sequences.

FUSION DEFAULT ASSIGNMENT

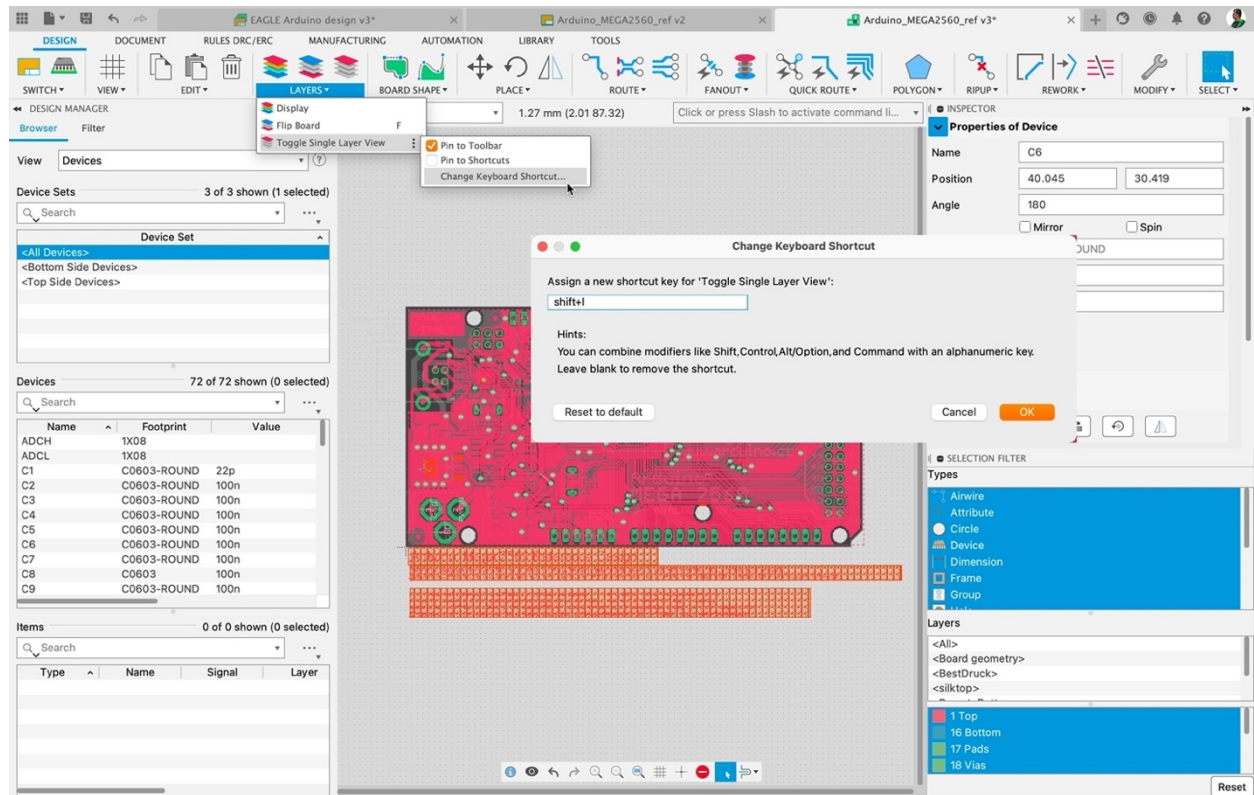
Fusion Electronics has some commands assigned to shortcuts, like A for the ADD command, or G for GRID. It is also possible to define your own key assignments. Default shortcut key assignment in Fusion Electronics:

Electronics Workspace Command	Windows Key Combination	Mac Key Combination
Activate command line	/	/
Add Hole	H	H
Add Part	A	A
Add Text	T	T
Bus	B	B
Change	C	C
Copy	Ctrl+C	Command+C
Delete	Delete	Backspace
Dimension	D	D
DRC	Ctrl+D	Command+D
ERC	Ctrl+E	Command+E
Errors	E	E
Grid	G	G
Label	Shift+L	Shift+L
Manual Route	R	R
Move	M	M
Name	N	N

Net	R	R
New Device	Alt+Ctrl+3	Option+Command+3
New Footprint	Alt+Ctrl+2	Option+Command+2
New Symbol	Alt+Ctrl+1	Option+Command+1
Package 3D Create	Alt+Ctrl+4	Option+Command+4
Pad	O	O
Pad Array	Shift+O	Shift+O
Pin	P	P
Pin Array	Shift+P	Shift+P
Redo	Ctrl+Y	Command+Shift+Z
Ripup	U	U
Ripup All Polygons	Alt+Shift+P	Option+Shift+P
Route Diff Pair	Ctrl+R	Command+R
Route Multi	Shift+R	Shift+R
Run Script	Alt+Shift+S	Option+Shift+S
Run ULP	Shift+U	Shift+U
SMD	P	P
SMD Array	Shift+P	Shift+P
Stop	ESC	ESC
Undo	Ctrl+Z	Command+Z
Value	V	V
Via	Alt+V	Option+V
Zoom to Fit	F6	F6
Switch Sch Doc	Ctrl+1	Command+1
Switch PCB Doc	Ctrl+2	Command+2
PCB 3D View	Ctrl+3	Command+3
Shortcut Dialog	S	S


Keyboard shortcuts in Fusion are user definable. To change or define a shortcut, highlight a command in one of the menu subsections and click the three dots on the right side of the menu entry. Choose *Change keyboard shortcut*. A dialog opens that allows you to create a custom keyboard shortcut.

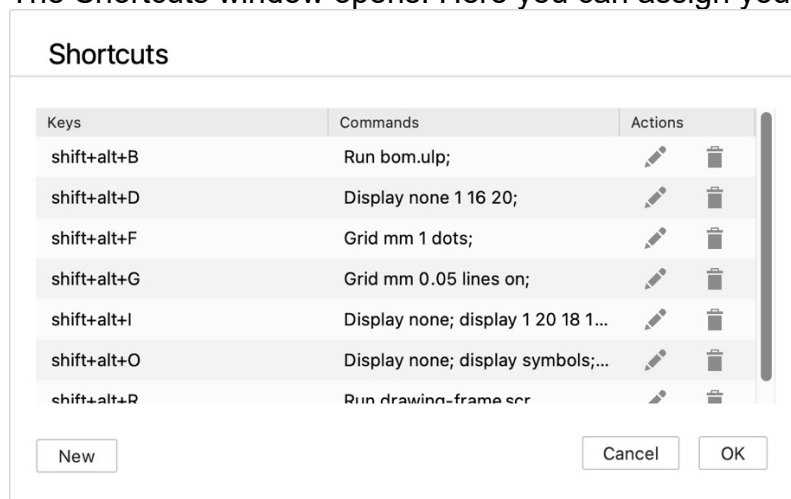
In case you tried to change a reserved shortcut, Fusion displays a message in the Change Keyboard Dialog.



32 Defining a shortcut

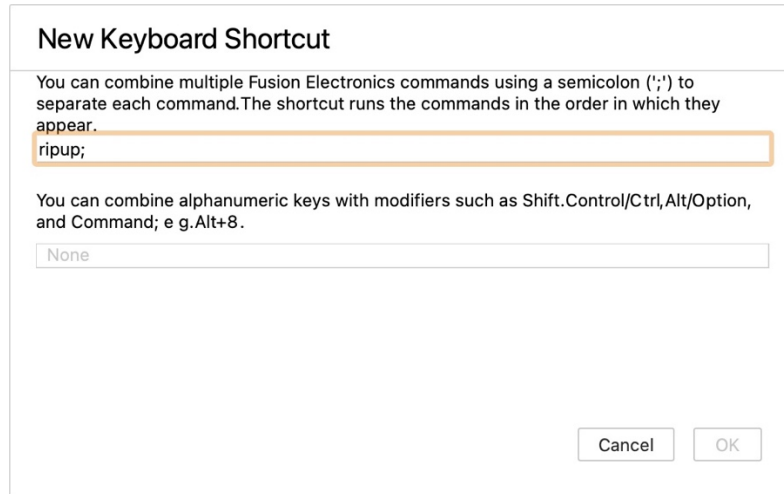
ELECTRONICS-SPECIFIC COMMAND ASSIGNMENT

For assigning electronics related commands or command sequences as it is possible in EAGLE, select the Shortcuts icon  in the Design menu in one of the editor windows. The Shortcuts window opens. Here you can assign your commands.



33 Shortcut window with some electronics-specific commands assigned

Click the *New* icon and the *New Keyboard Shortcut* window shows up. Type the command you would like to assign into the highlighted box. Now click into the box below and press the key combination you want to assign it to on your keyboard. With *OK* the assignment is settled.

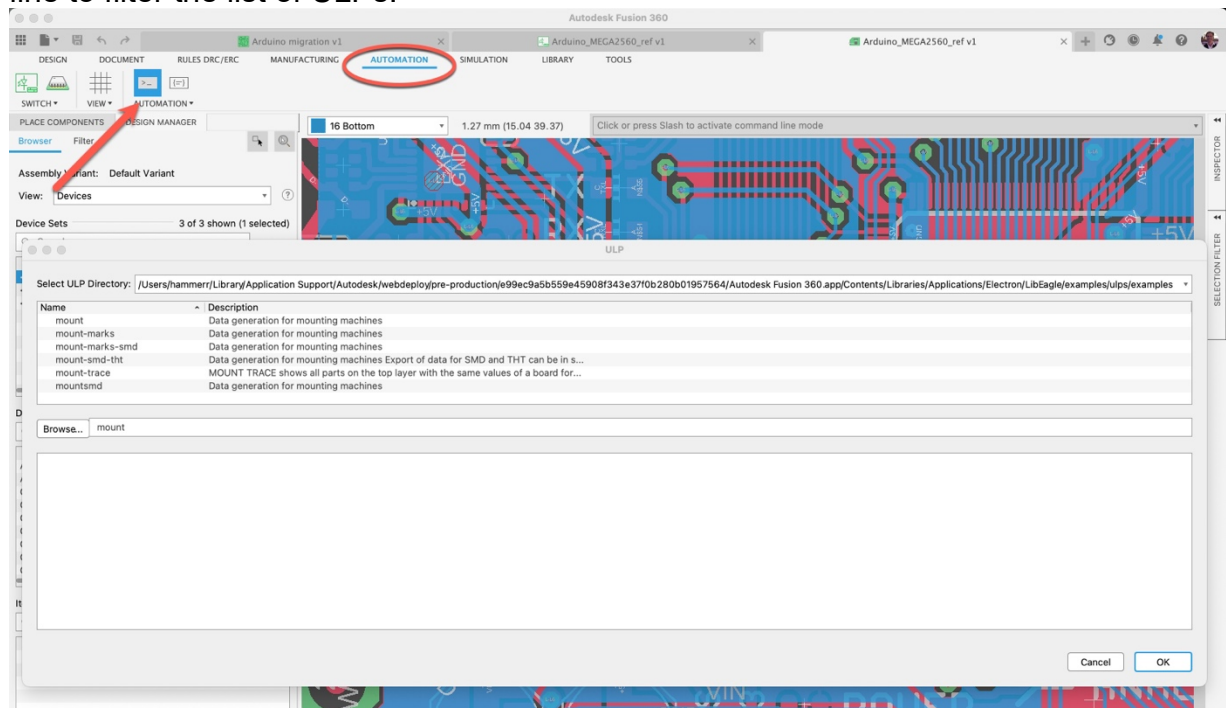


34 New Keyboard Shortcut window

8 USER LANGUAGE PROGRAMS AND SCRIPTS IN FUSION 360 ELECTRONICS

Like EAGLE, Fusion 360 Electronics supports User Language Programs (ULPs) and Scripts. Choose the *AUTOMATION* menu and click the orange *Run ULP* icon.

The ULP list opens and shows the ULPs provided with Fusion. Click Browse to search for ULPs that you've used with EAGLE before. You can type a search expression into the text line to filter the list of ULPs.



35 List of User Language Programs filtered for the search expression "mount"

You can use a similar workflow to run a script file. Click the *AUTOMATION* menu and click the blue *Run Script* icon next to the *Run ULP* icon. A list of scripts is displayed. Use *Browse...* and search filters to find other existing scripts.

The ULPs and script files provided with Fusion 360 are stored in a system directory. On a Mac computer, the path to these files is quite long and cryptic (as you can see in the image above). We recommend that you specify a path to a user folder for your own ULPs and

scripts. This can be done in the *Preferences* for Fusion, *Electronics*, *Directory*.

9 USE YOUR EAGLE LIBRARIES AND MANAGE LIBRARIES IN FUSION 360

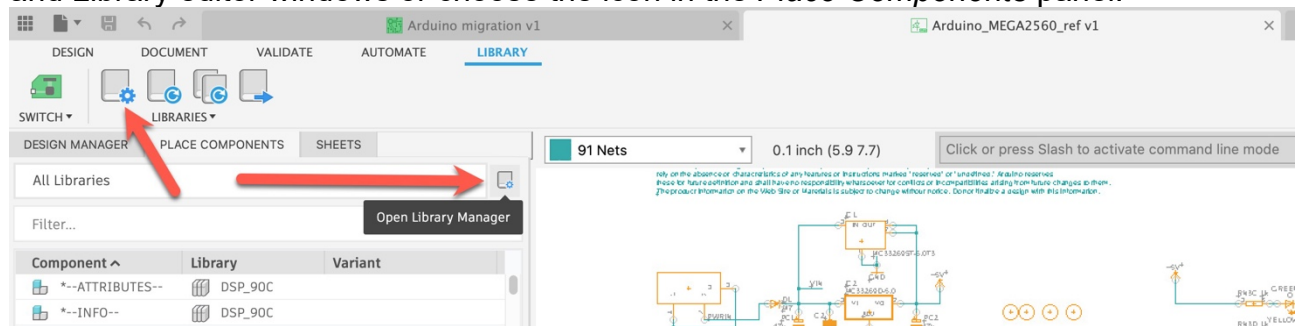
This section informs about the Library Manager, the different types of libraries available and their locations. Please read careful!

Fusion Electronics comes with several component libraries. Use the Library Manager to navigate and manage the libraries. If you are familiar with EAGLE version 8 or newer, you are aware of the Library Manager. In Fusion 360 it looks a bit different and offers convenient filtering options for navigating the libraries.

LIBRARY MANAGER

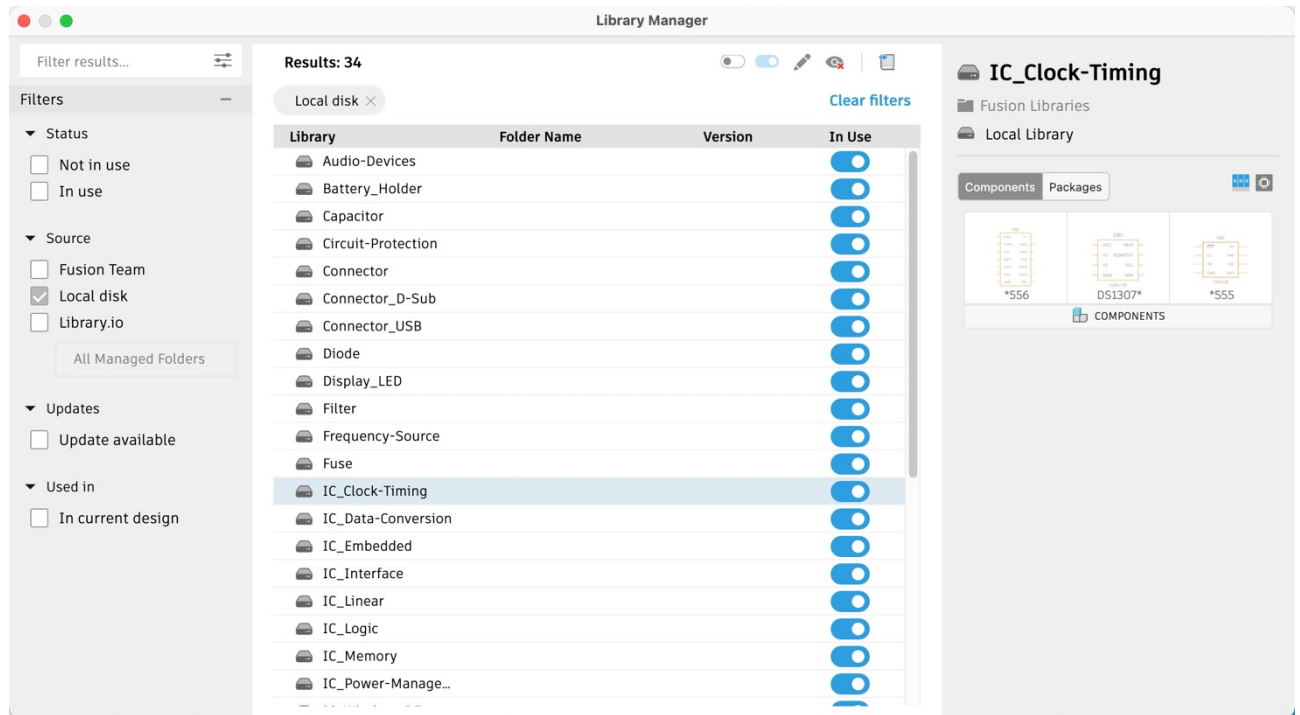
The Library Manager mainly controls the libraries available when adding components to your design.

Click the *Open Library Manager* icon in the *LIBRARY* menu in the Schematic, 2D PCB, and Library editor windows or choose the icon in the *Place Components* panel.



36 Open Library Manager

The Library Manager window allows easy navigating your libraries. *In Use* libraries will be available in the *Place Components* panel where you can take components from and place them in the schematic / 2D PCB canvas.



37 Library Manager with In Use libraries listed from your local disk

FILTERING OPTIONS

On the left side of the Library Manager window, you will find several filtering options. For searching a library, type in a search expression in the box on top. The slider icon holds options for *Name*, *Description* and *Folder name*. These filter settings determine where to search for. To refine your search, use the available filter options.

STATUS

- Not in use library is not available in the Place Component dialog
- In use library is already available in the Place Component dialog

SOURCE

- Fusion Team library stored on a Team hub, which is a cloud-based storage, accessible by invited people.
- Local disk library stored on your computer. Typically, libraries coming with Fusion by default.
- Library.io library stored on <https://library.io>. The place where Autodesk EAGLE stores all its managed libraries in managed folders.

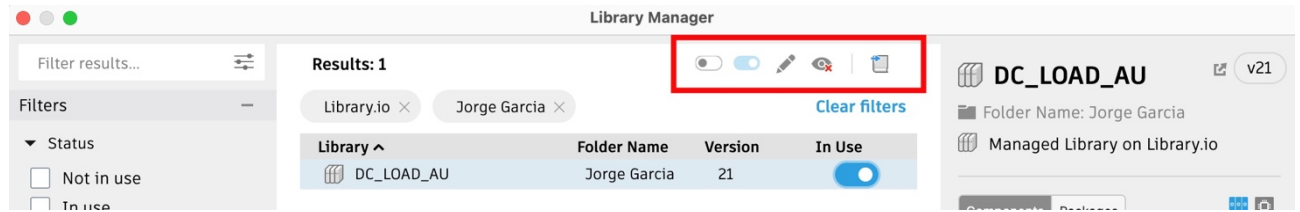
UPDATES

- Update available filters for libraries that have been updated recently, but not yet downloaded to your computer

USED IN

- In current design the libraries already used in the currently active design

Click on the *In Use* icon in the list next to the library entry to toggle the library's status. With the icons located on top of the Library Manager window (marked with a red box in the following image) you set the library status to *Not in use* or *In use*, *Edit* the selected library, *Remove it from view* and *Import libraries*.



38 Library Manager – Action icons for Not in use, In use, Edit, Remove from view and Import

WHY ARE THERE LIBRARIES AT DIFFERENT LOCATIONS?

Consider the following scenario:

An EAGLE user creates a library and stores it in a managed folder on library.io. You were provided with access to this folder for collaboration using the same libraries in EAGLE. Now you are working with Fusion Electronics which gives you access to all the libraries you previously used in EAGLE. These are listed as of type *library.io* in a managed folder. This is the folder you already had access to with EAGLE.

WHAT ARE THE LIBRARY SOURCES?

Fusion Team libraries are stored in your Fusion Team environment. Typically they are visible in the libraries project/folder(s) in Fusion's data panel.

Local disk libraries are stored on your computer. This can be self-made or downloaded libraries on your computer, typically created with EAGLE.

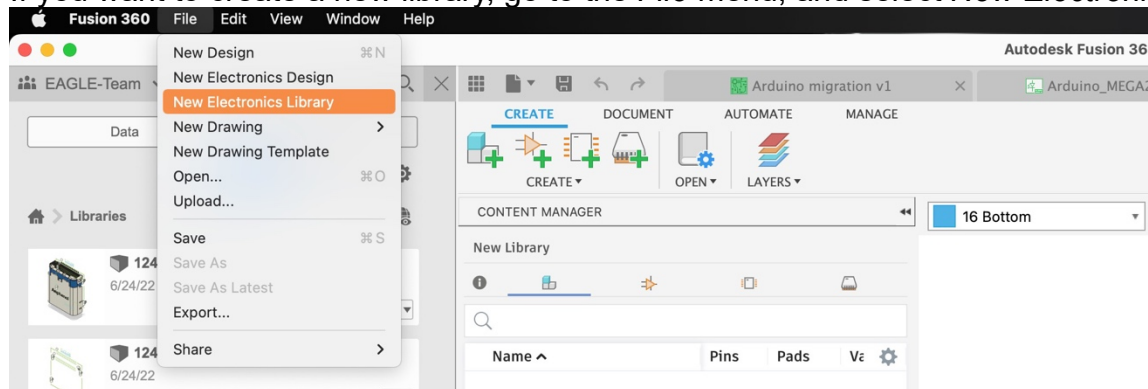
[library.io](#) libraries are stored on our Autodesk EAGLE managed libraries platform. They come with EAGLE or are made available by EAGLE users you are collaborating with. There can be different "managed folders" available.

10 FIRST USE OF THE LIBRARY EDITOR

In this section we give an overview of the library editor. How to create a new library and how to edit an already existing library.

CREATE NEW LIBRARY

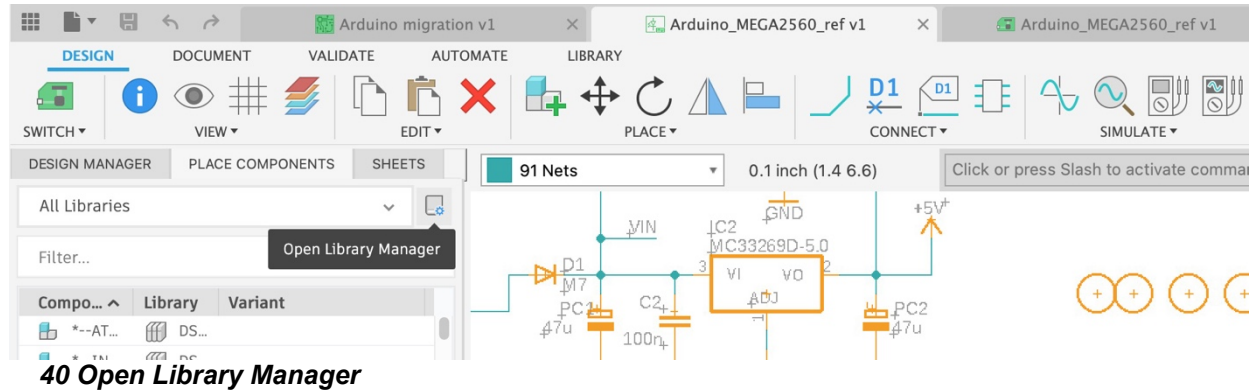
If you want to create a new library, go to the *File* menu, and select *New Electronics Library*.



39 Create New Library

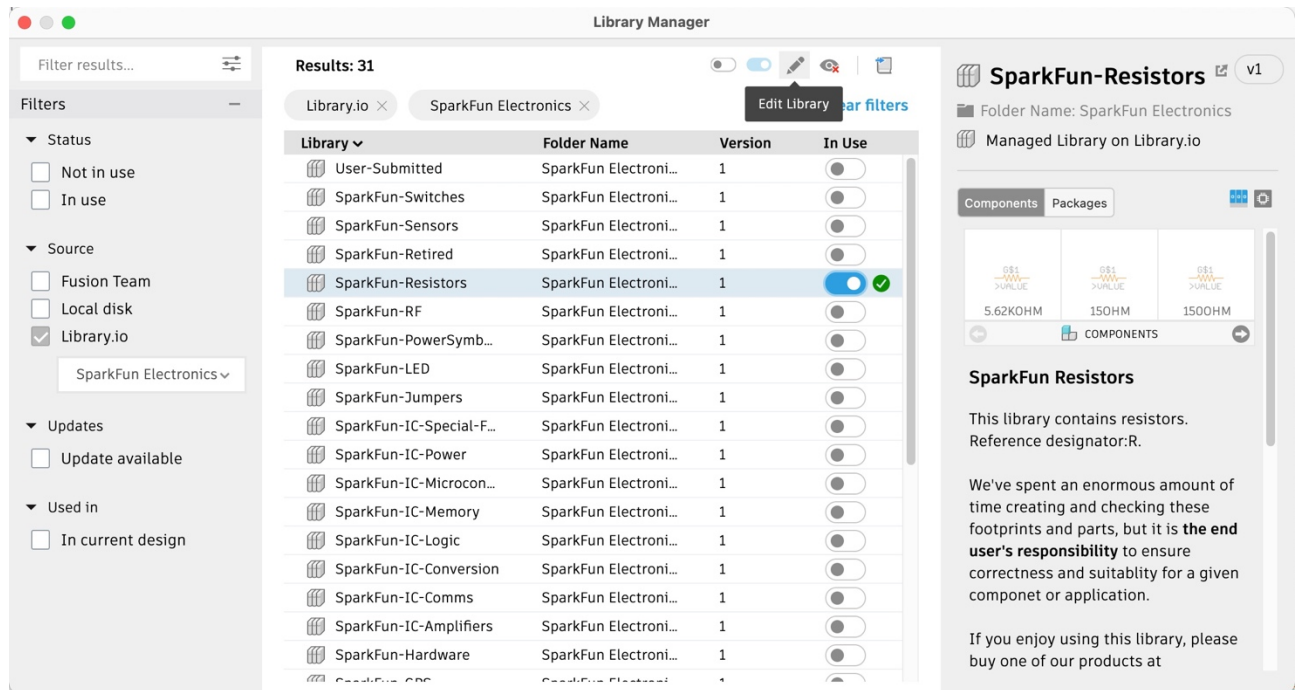
EDIT A MANAGED LIBRARY CREATED IN EAGLE

If you want to edit a library made in EAGLE, navigate to the *Library* menu in your schematic or 2D PCB editor and click the *Open Library Manager* icon.



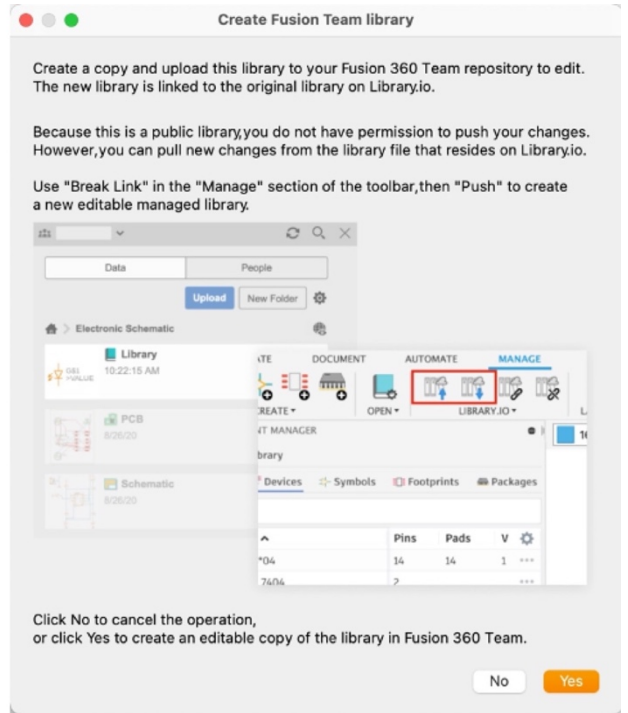
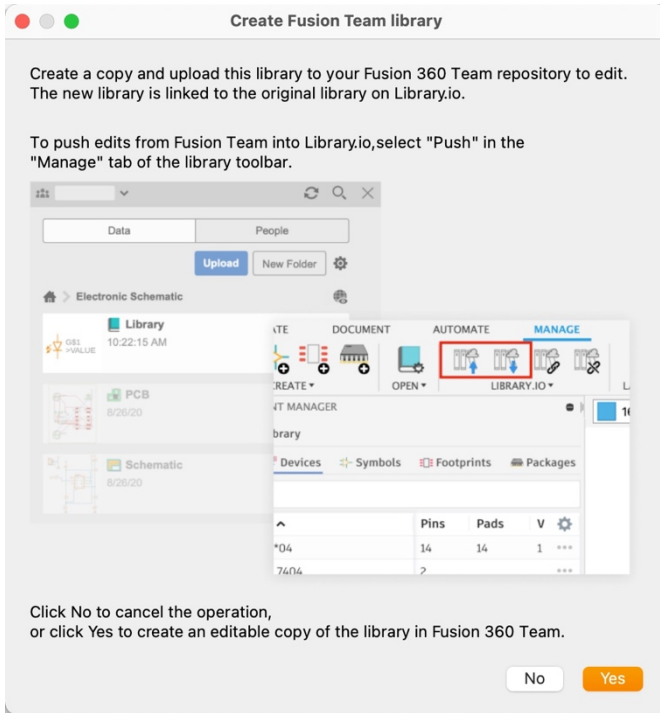
40 Open Library Manager

In the Library manager, use the filter options to locate the library. As soon as you set the status of the library to *In Use* it will be downloaded automatically. The green check mark confirms the download from *library.io*. Now the *Edit* icon is accessible.



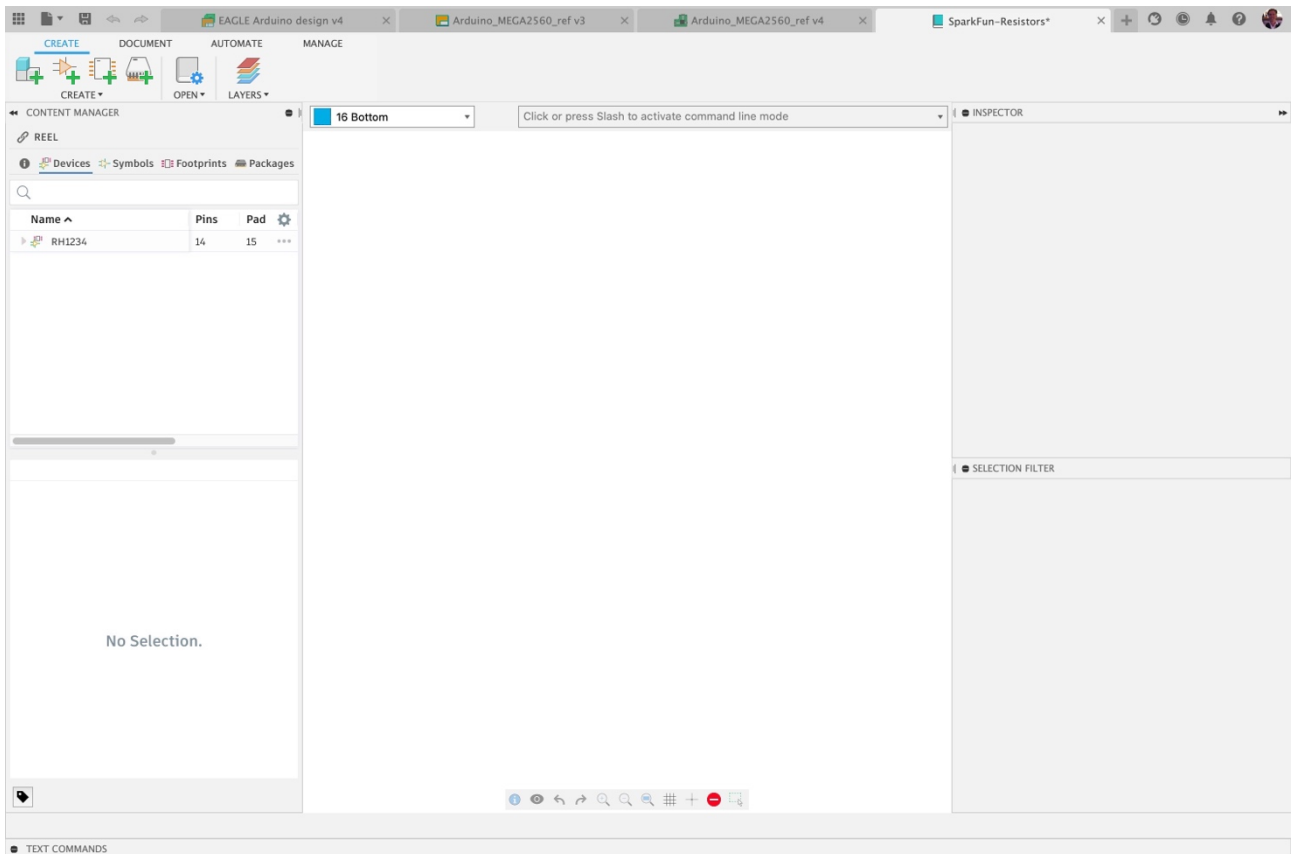
41 Library Manager – Library is in use, downloaded and ready to be edited

Click the *Edit* button and Fusion shows the *Create Fusion Team library* window. Depending on the access rights for the library, you get information how to manage your library after modifying it.



42 Create Fusion Team library info for libraries with full or restricted access

Click Yes to confirm and the library editor opens.



43 The Library editor window

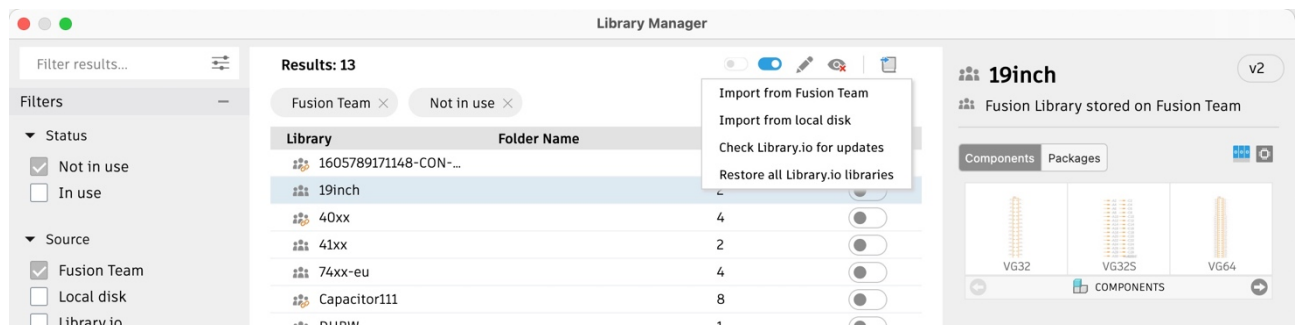
It's good practice to save immediately after opening your previous EAGLE library in Fusion 360. It is also recommended that you create a Library project and store all your library files in this project.

The original EAGLE file is untouched and unchanged! The library file was copied and saved in your Fusion environment, and it doesn't matter whether it comes from library.io or from your local/network drive.

After a managed EAGLE library becomes a Fusion Team library, and you continue to work in an existing design with components from this library, it is recommended that you update the design with the components from the Fusion Team library.

IMPORT LIBRARIES

In case your library is not listed and can't be found, you probably must import it from a Fusion Team or local disk. Therefore click the import icon and choose an option.

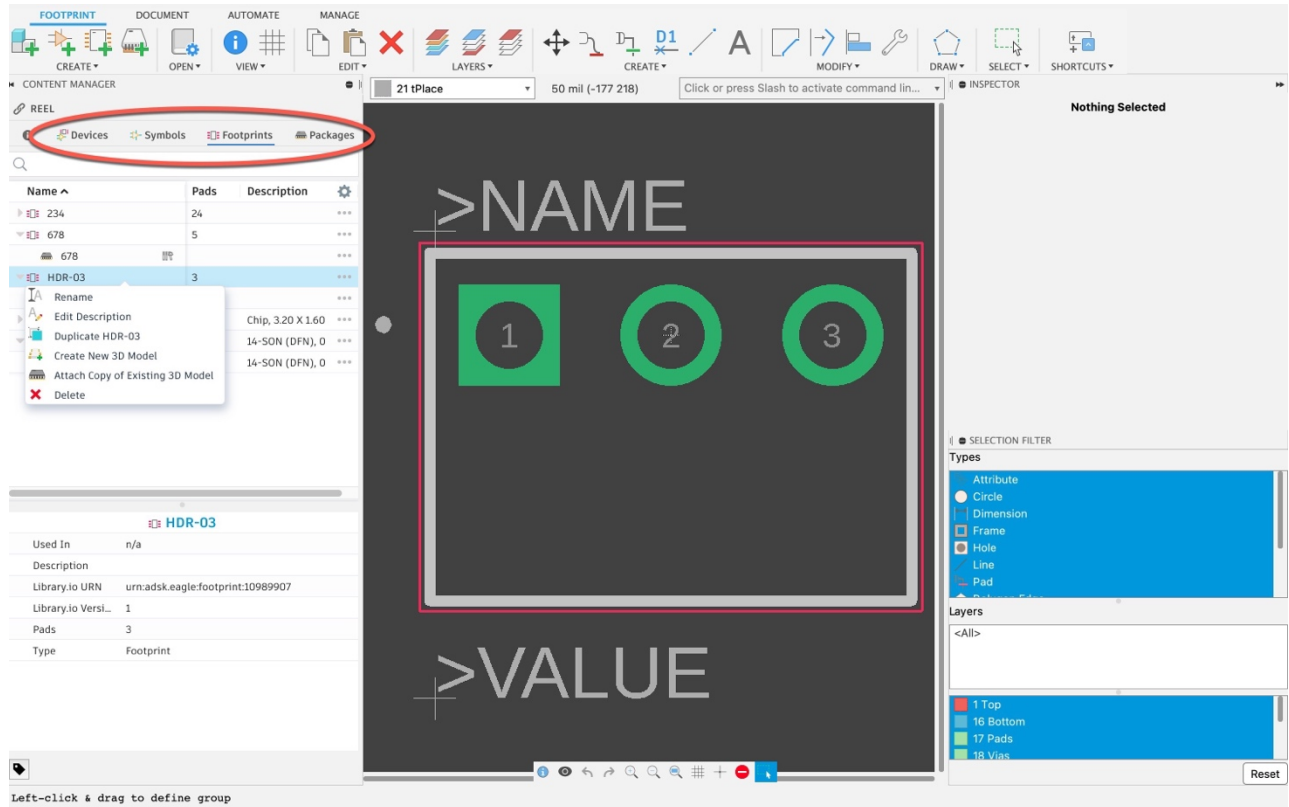


44 Library Manager – Import options

CONTENT MANAGER IN THE LIBRARY EDITOR

The Content Manager in the Library editor is comparable with the Table of contents view in EAGLE's library editor. Navigate through the Devices, Symbols, Footprints, and 3D Packages in this library file.

Click on the icons *Devices*, *Symbols*, *Footprints* or *Packages* (marked in the following image) to view the list of objects. There you can see additional information like the number of pads of the footprint, for example.



45 Library Editor - Content Manager gives access to all assets

Click the three dots of an entry in the list or right mouse-click to open the context menu. This provides access to options for the selected asset such as, *Rename*, *Edit Description*, *Duplicate*, *Create New 3D Model*, *Attach Copy of Existing 3D Model*, and *Delete*.

Double-click an entry to open the asset in editor mode. Right mouse click to open a context menu with several options. The options available depend on the selected object.

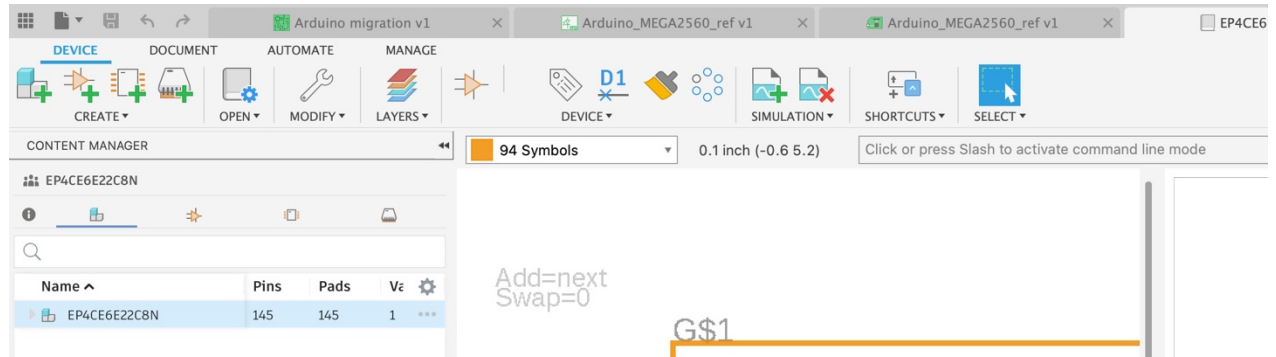
In the previous image, the footprint *HDR-03* is shown. A double-click on the entry in the list opens the footprint editor. A right mouse click on the entry in the list opens options to *Rename*, *Edit Description*, *Duplicate*, *Create New 3D Model*, *Attach Copy of Existing 3D Model*, and *Delete*.

LIBRARY EDITOR OVERVIEW

Let's take a deeper look into the library editor and explain how to create a new asset or import an already existing asset into the library.

The library editor window looks like the schematic and 2D PCB editor windows with the command menu at the top. When no editor window is open, the basic tools for libraries are displayed.

CREATE MENU

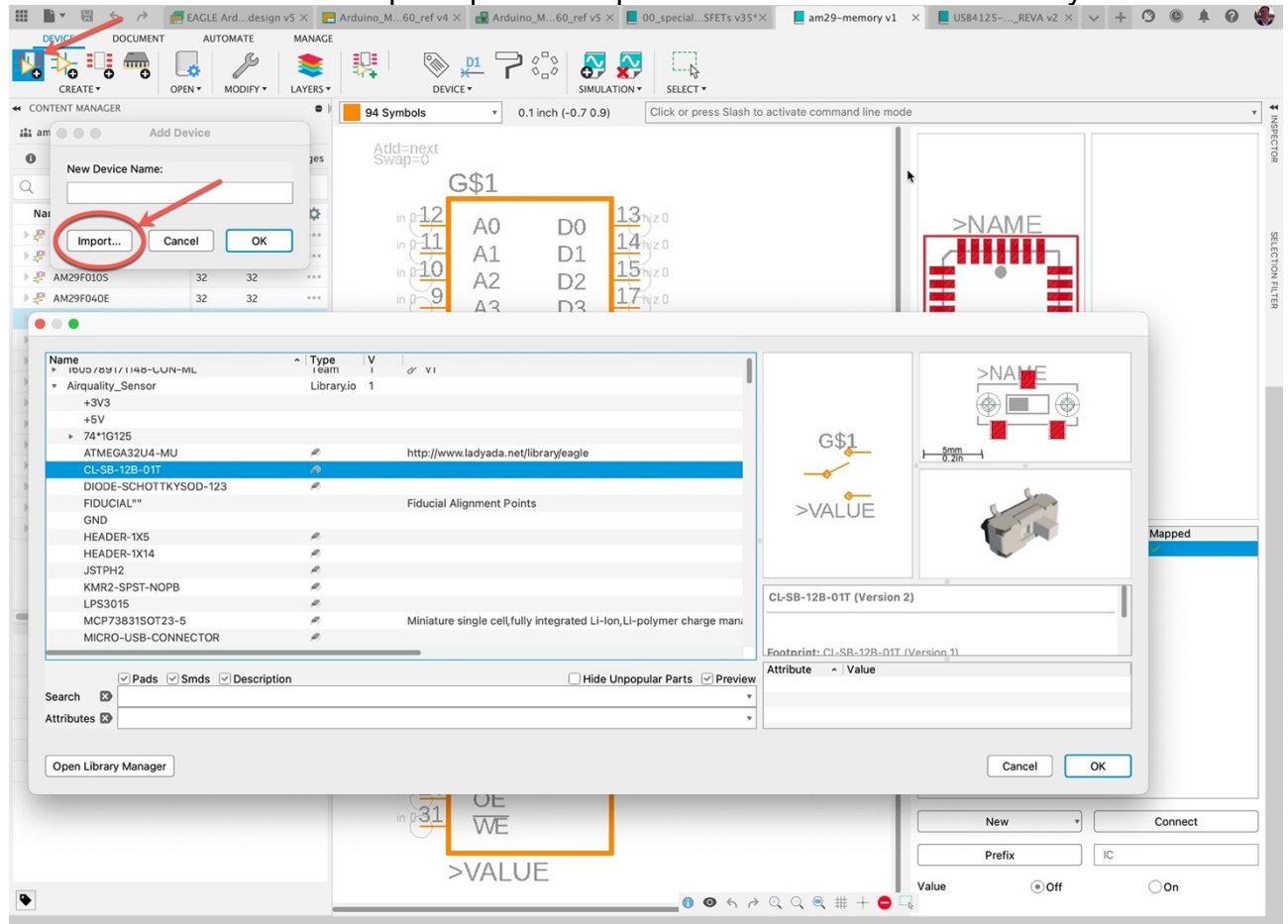


46 Library Editor - CREATE menu

From left to right in the CREATE menu you can: create a new Device, a new Symbol, a new Footprint, or a new 3D Package. You can also open the Library Manager and access Layers options.

IMPORT FROM ANOTHER LIBRARY

Use the *Create New ... Import* option to import an asset from another library.



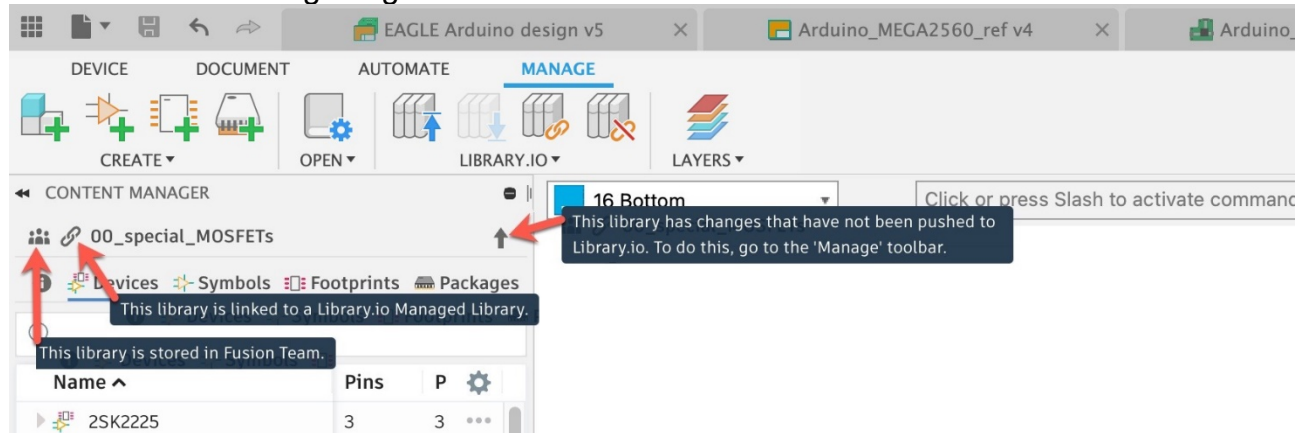
47 Create New Device and Import from Another library

MANAGE MENU

Managing a library is necessary if you want to collaborate with EAGLE users and store your libraries on library.io. This section explains how to update a library in Fusion or on library.io and how to link or unlink it.


This menu is the place where you manage your library. What does this mean? Well, let's examine the library. What information does the Content Manager panel provide? Quite a bit of information.


Examine the following image:




48 Library Editor: Manage menu


Hover over one of the icons marked to see where the library is stored and whether it is linked to a library in library.io.

 **Library stored in Fusion Team:** The library was created in Fusion or comes from library.io and has been edited and stored in your Fusion environment.

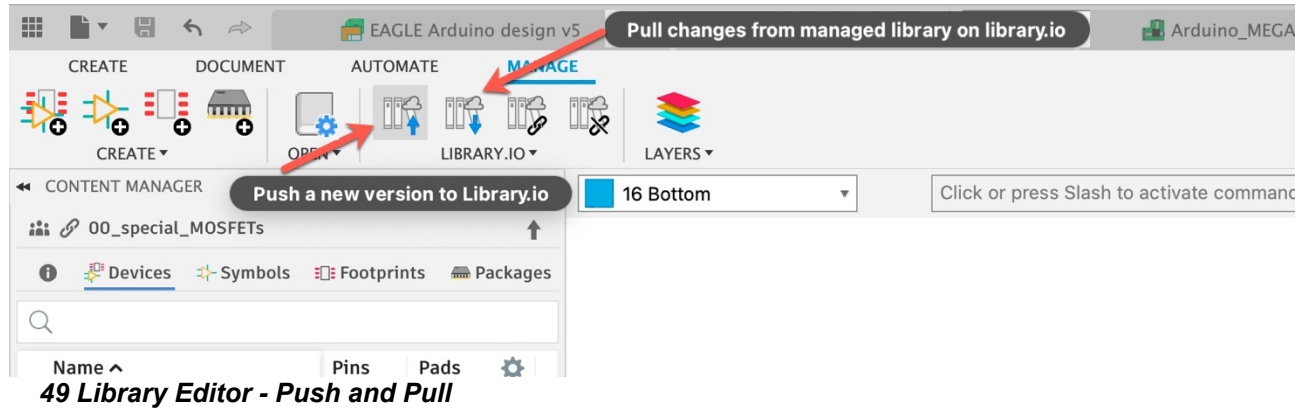
 **Library is linked to a library.io managed library:** The library is taken from library.io (which is the storage for managed EAGLE libraries) and stored in the Fusion Team environment. The link to library.io still exists. This means that changes made to this library in library.io by an EAGLE user can be pulled into your library in Fusion Team, or you can edit your library in Fusion and push the changes to library.io so an EAGLE user can consume the new version of your library. This provides the ability for Fusion and EAGLE users to collaborate.

The push and pull action are only possible if you have full access to the library.io managed folders.

 **Library has changes that have not been pushed to library.io:** You made changes to the library in Fusion and saved it in your Fusion Team environment, but you did not sync it with the linked library on library.io. An EAGLE user can't work with your current version of the library.

 **Changes from managed library on library.io available** (this option is not visible in the image above): There are changes made to the linked library on library.io which have not been pulled into your Fusion Team library yet.

The Push and Pull icons are marked in the image.



If you created your own library in Fusion and you would like to make it accessible to colleagues still using EAGLE, you can link it to library.io. Make sure that you have been granted full access to the managed library folder on library.io. Otherwise, you are not allowed to link your library.

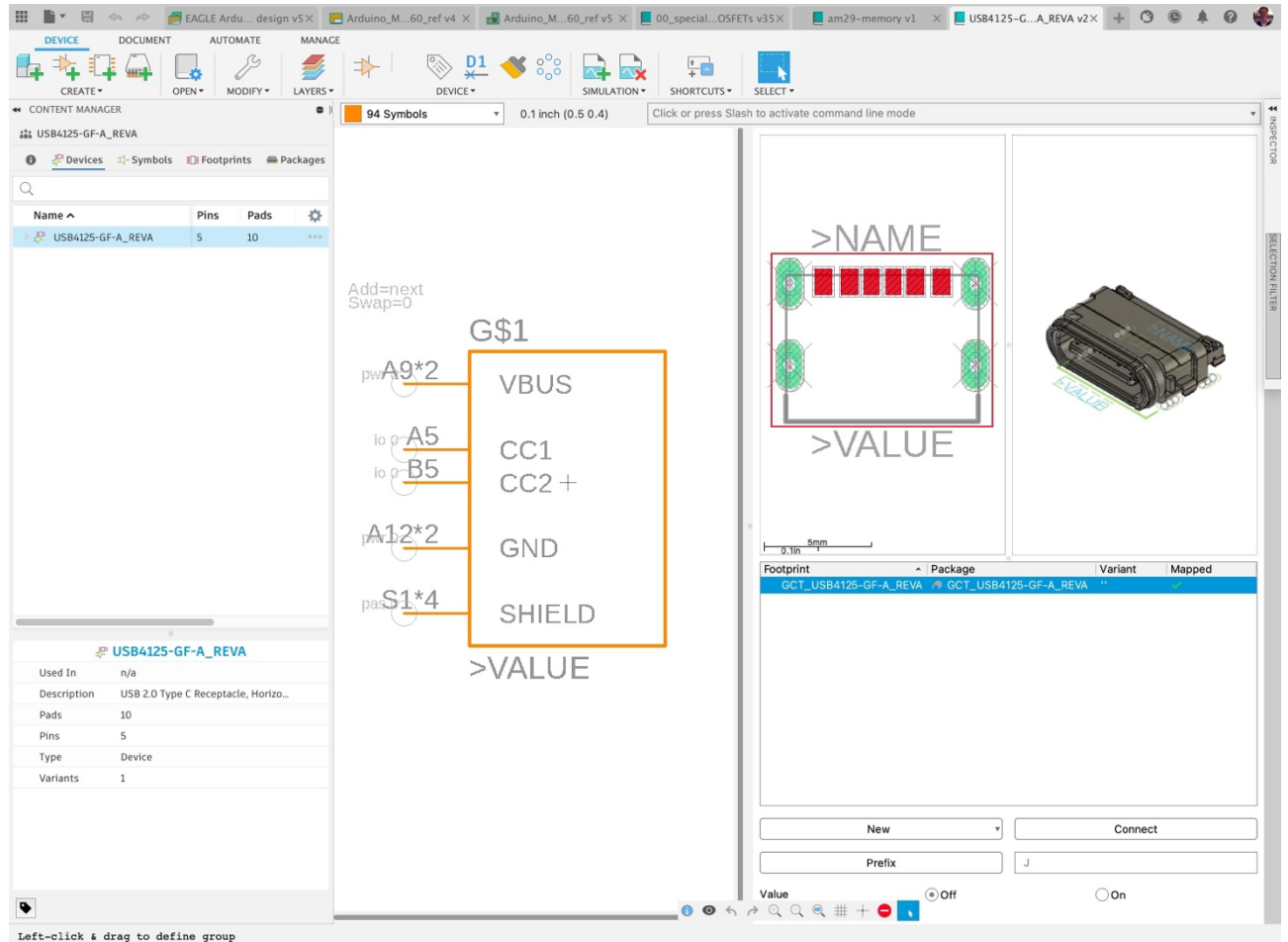
Tip: If you do not plan to collaborate with EAGLE users, you can unlink your library from library.io. Click the *unlink* icon and save your library.



DEVICE EDITOR

This section and the following sections give an overview of the Device, Symbol, and Footprint editors. Where are all the commands and tools located?

The Device editor in the library has nearly same appearance as it does in EAGLE. Click the *Create New Device* icon or double-click a *Device* entry in the Content Manager to enter device editing mode.



51 Device Editor

DEVICE MENU

The DEVICE menu contains the sections CREATE, OPEN, MODIFY, LAYERS, DEVICE, SIMULATION, and SELECT. This is where you find the relevant tools for your work in the device editor.



52 Device editor – DEVICE menu

Tools available:

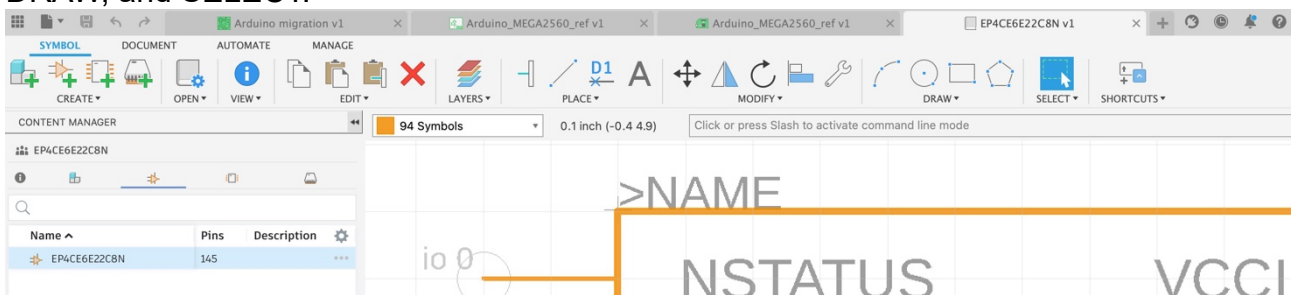
<p>CREATE</p> <ul style="list-style-type: none"> Create new Device Create new Symbol Create new Footprint Create new Package <p>OPEN</p> <ul style="list-style-type: none"> Open Library Manager <p>MODIFY</p>	<p>DEVICE</p> <ul style="list-style-type: none"> Add Attribute Name Copy Fromat Pattern <p>SIMULATION</p> <ul style="list-style-type: none"> Add Spice Model
--	---

Change LAYERS Display	Remove Spice Model SHORTCUTS Shortcuts SELECT Group Group with polygon

The DOCUMENT menu provides access to the *PRINT* command and the AUTOMATE menu provides access to *Run User Language programs* and *Run scripts*.

SYMBOL EDITOR

Click the *Create New Symbol* icon and choose a name for your new symbol. The SYMBOL menu contains the sections CREATE, OPEN, VIEW, EDIT, LAYERS, PLACE, MODIFY, DRAW, and SELECT.



53 Symbol Editor: Symbol menu

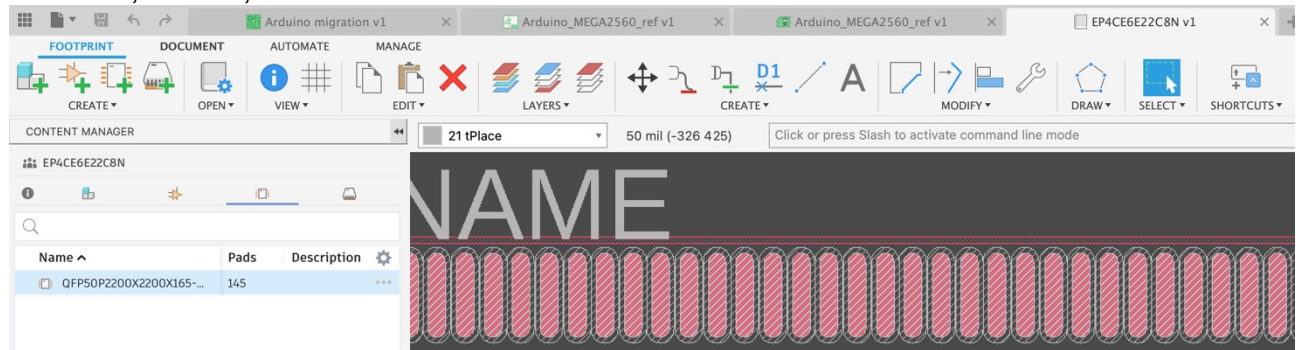
Tools available:

CREATE Create New Device Create New Symbol Create New Footprint Create New Package OPEN Open Library Manager VIEW Info EDIT Copy Paste Paste Pins Delete LAYERS Display PLACE Pin Line Name Text Pattern Pin Array	MODIFY Move Mirror Rotate Align Change Miter Split DRAW Arc Circle Rectangle Polygon Shape SELECT Group Group with polygon SHORTCUTS Shortcuts
--	---

The DOCUMENT menu provides access to the *PRINT* command and the AUTOMATE menu provides access to *Run User Language programs* and *Run scripts*.

FOOTPRINT EDITOR

Click the *Create New Footprint* icon and choose a name for the new footprint. The FOOTPRINT menu contains the sections CREATE, OPEN, VIEW, EDIT, LAYERS, PLACE, MODIFY, DRAW, and SELECT.



54 Footprint Editor - Footprint menu

Tools available:

<p>CREATE</p> <ul style="list-style-type: none"> Create New Device Create New Symbol Create New Footprint Create New Package <p>OPEN</p> <ul style="list-style-type: none"> Open Library Manager <p>VIEW</p> <ul style="list-style-type: none"> Info Grid settings Show <p>EDIT</p> <ul style="list-style-type: none"> Copy Paste Delete <p>LAYERS</p> <ul style="list-style-type: none"> Display Flip Board Toggle Single Layer View <p>PLACE</p> <ul style="list-style-type: none"> Move SMD Pad PTH Pad SMD Pad Array PTH Pad Array Name Line Text Hole (NPTH) Pattern 	<p>MODIFY</p> <ul style="list-style-type: none"> Miter Split Align Copy Format Optimize Change <p>DRAW</p> <ul style="list-style-type: none"> Arc Circle Rectangle Polygon Shape Polygon Cutout Dimension <p>SELECT</p> <ul style="list-style-type: none"> Group Group with polygon <p>SHORTCUTS</p> <ul style="list-style-type: none"> Shortcuts
---	--

Additionally, the DOCUMENT and AUTOMATE menus provide access to the *PRINT* command, to *Run User Language programs*, and *Run scripts*. The LAYERS section is also available in those menus.

PACKAGE EDITOR

The Package editor takes you into the mechanical design world. The Package editor is used for creating/editing the 3D model that we want to assign to a footprint in the library.

Click the *Create New Package* icon in the library editor. A new window opens in the Design workspace.

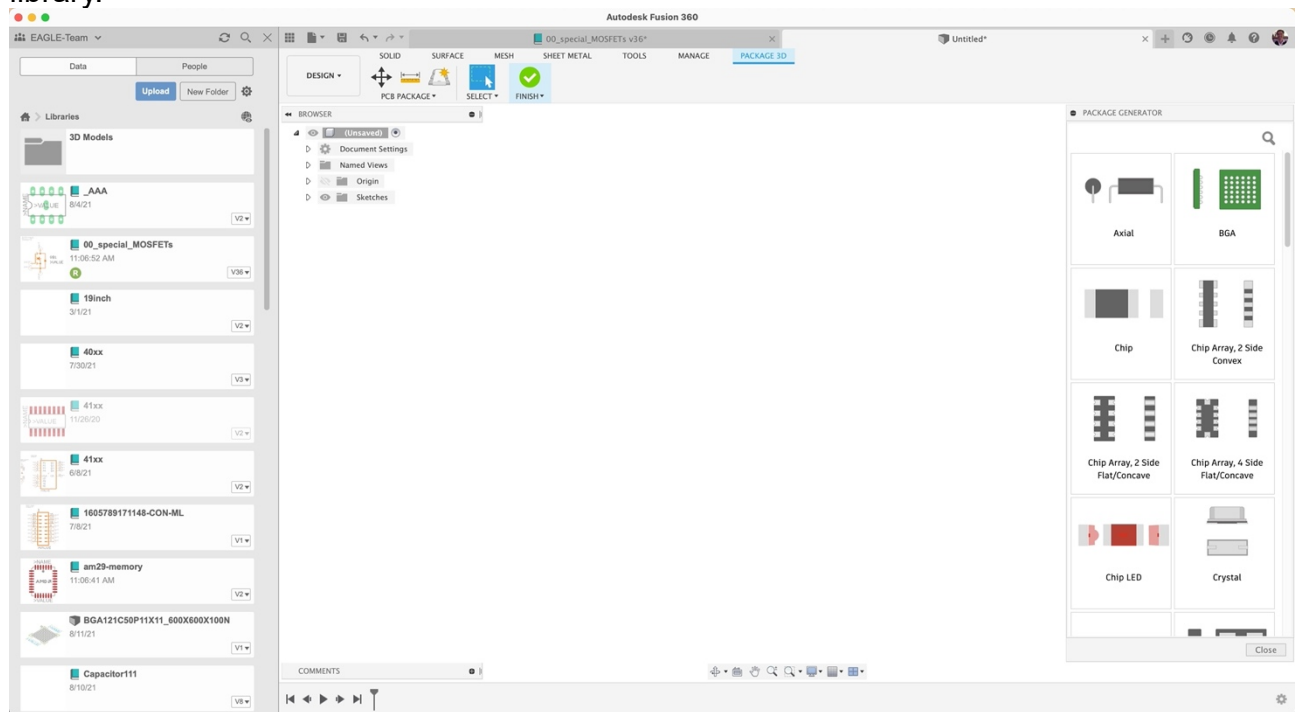
The PACKAGE 3D menu is active by default. On the right is the PACKAGE GENERATOR with templates for creating 3D models. In Fusion 360 the Package Generator offers more templates than EAGLE does.

Options to create a package:

- Use the Package Generator templates
- Insert an existing 3D model by right click on a model listed in the data panel
- Sketch/design your own model

Click the Finish icon to save the package and add it to the current library.

The 3D model is not integrated into the library file. It is a separate file which is linked to the library.



55 Package Editor with Package Generator

By default, the PACKAGE 3D menu is shown.

Tools available:

<p>PCB PACKAGE</p> <ul style="list-style-type: none"> Move/Copy Measure 3D Package Generator <p>SELECT</p> <ul style="list-style-type: none"> Select Window Selection Freeform Selection Paint Selection 	<p>FINISH</p> <ul style="list-style-type: none"> Finish
---	--

Selection Tools ... Selection Priority... Selection Filter...	
---	--

There are multiple tools in Fusion to create a model. The tools are accessed in the SOLID, SURFACE, MESH, SHEET METAL, TOOLS and MANAGE menus. Please refer to the Fusion 360 tutorials and help topics to learn how to start a mechanical design.

For further information how to work with 3D PCB in Fusion please follow this link:
<https://www.autodesk.com/products/fusion-360/blog/3d-pcb-fusion-360-tutorial/>