CP231516

Libraries: The Foundation of Electromechanical Workflow

Edwin Robledo
Community Manager
Product Support Specialist, EAGLE
Autodesk, Inc.

Jonathan Odom
Community Manager, Fusion 360
Autodesk, Inc.

Learning Objectives

- Advantages of the electromechanical workflow
- Fundamental importance of libraries in the ECAD workflow.
- Comprehensive understanding of the sections that comprise the Autodesk EAGLE Library editor.
- Introduce various methods by which Autodesk EAGLE users can assign 3D models to components footprints.

Description

One of the biggest challenges that an Electrical Engineer confronts when using an electronics design application is finding parts that are needed for his design. Manufactures are continually updating and changing their parts to adapt to today's design challenges. This means that the EE will be spending lots of their time just searching for the parts they want to use and hoping that is its in a compatible with their design applications.
A 2D render of your design is no longer sufficient in today's race to go to market. The design foundation, the libraries, needs to support the necessary functional 3D electromechanical workflow. This has been proven to save time and quickly spin iterations of the product.

This class will walk through the solutions that Autodesk EAGLE has in its library editor to solve these challenges.

AUTODESK.
UNIVERSITY

Speaker(s)

Edwin Robledo graduated as an Electronic Engineer the Polytech University of Puerto Rico. Currently, he is a Community 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.

Jonathan is a professional designer with more than 10 years of experience in digital design and fabrication. His work includes special effects, animatronics, architecture, exhibit design, product design, and interactive art. His work has been exhibited locally and internationally, published online and in print, and is enjoyed by friends, family, collectors, and clients. In his five years at Autodesk, he has published dozens of example projects, tutorials, webinars, and online classes focusing on digital fabrication with Fusion 360. He is currently a Fusion 360 specialist, content creator, and community manager at Autodesk.

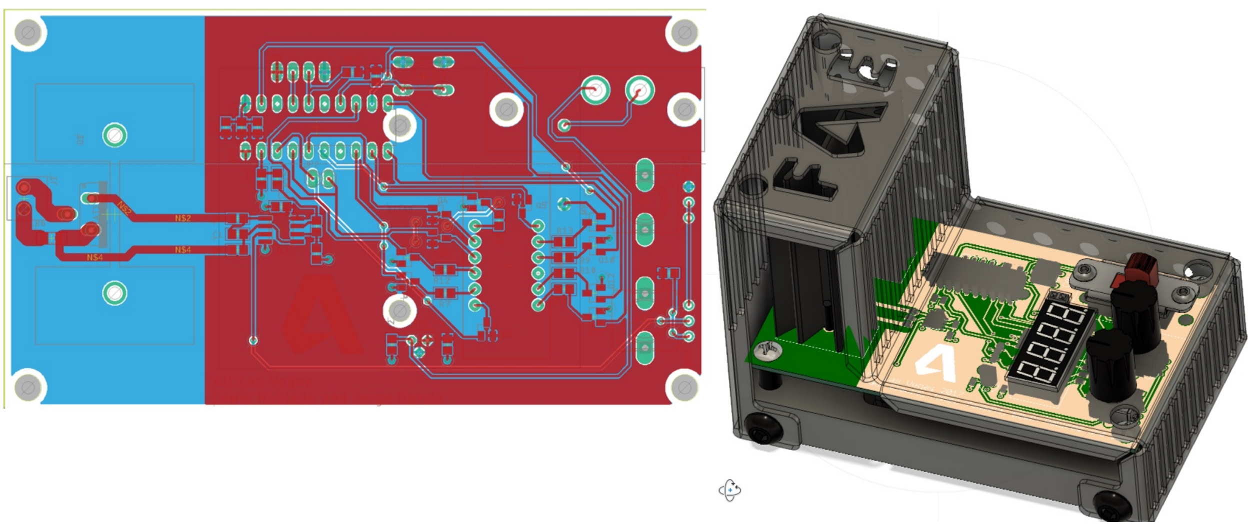
Libraries: The Foundation of Electromechanical Workflow

Advantages of the electromechanical workflow

File sharing between electronic and mechanical application is great start, but vulnerable to many errors. Design collaboration without file conversion is imperative. All intelligent products consist of multiple levels of engineering which require constant updating from the ECAD design or the enclosure restrictions. The engineering teams require to:

- Quickly make changes
- Make certain they are working with the most recent design version
- Collaboration between the electrical and mechanical engineer to ensure the product requirements are met.

Autodesk EAGLE and Fusion 360 have a system in which a bi-directional workflow is available.



Autodesk EAGLE PCB in use in Fusion 360 designed mechanical enclosure

The PCB Workspace in Fusion 360 allows PCB designs from Autodesk EAGLE to be integrated into the a mechanical fusion design. The integration goes much further than just a rendered image:

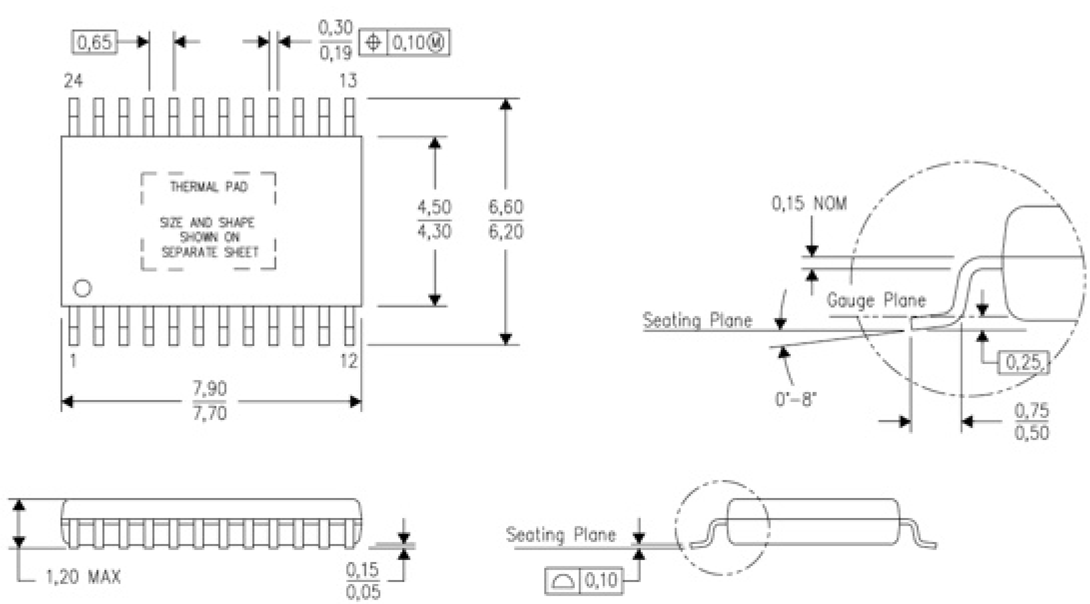
- Make changes to PCB Components in Fusion 360.
- Define PCB Outlines in Fusion 360 which are recognized and imported by Autodesk EAGLE.
- Pull changes made to the PCB outline in Fusion back to the original EAGLE board file.
- Updates to the PCB in Autodesk EAGLE are pushed to Fusion 360

Fundamental importance of libraries in the ECAD workflow

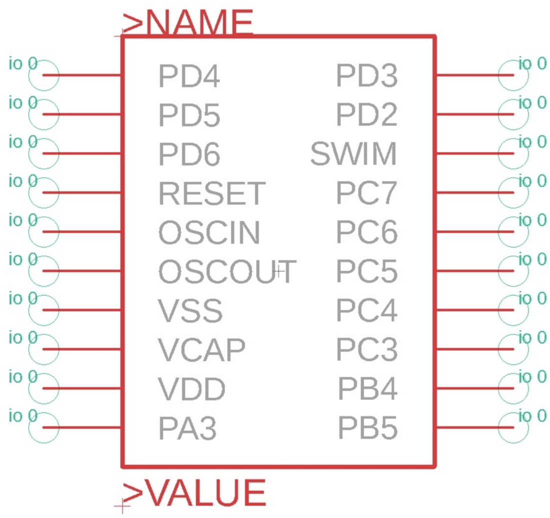
The building blocks of every electronic design is based on accessing components that will be used in the design. Libraries are the key to this and a good set of libraries are at the core of any good electronics design. They provide the foundation on which the electronics design is built and directly drive the quality and accuracy of the schematic, layout, mechanical design, and component purchasing. Libraries help provide the vehicle for sharing information between the various disciplines involved in the product design. A well designed set of component libraries will enable:

- The designer to easily find the parts he or she requires (well structured with pertinent information easily accessible).
- The designer to quickly and easily compare parts to enable them to choose the most appropriate (useful attributes).
- The designer to quickly access key information about the parts they need for their design (link to data sheets, descriptions etc).
- A schematic which is clear to read, consistent in look and easily understandable (symbols designed to a standard and with all appropriate information visible).
- A layout which is accurate, manufacturable, and consistent (PCB footprints designed to IPC standards).
- A mechanical design which factors in real component information at an early stage (accurate 3D models associated with the components in the library).
- Easy and accurate generation of the Bill of Materials for the design (well structured, additional attributes, ULP for easy and reliable BoM generation).

The components can be provided by manufacturers, distributors, 3rd party providers or created by the application. The source of the parts is always a manufacturers component datasheet. The manufacturer sheet will contain all of the functional details of the components as well as the exact dimension of the mounting pattern.



Manufacturer Data Sheet



Schematic Symbol

The mounting patterns are usually referred as ‘Footprints’, and the many manufacturers will follow the IPC standards. The IPC is a trade association which aims to standardize the assembly and production of electronic and circuit equipment. Based on these standards, calculators can be created to automate the process of creating footprints.

The majority of PCB designs will always begin with a schematic. A schematic is the logical representation of the printed circuit board. Each component will be represented with a symbol and each connecting point will be named according the function of the pin. At this stage the symbol has no mechanical reference to the footprint.

The schematic design provides the engineer:

- A detailed perspective of the electronic functionality of the design.
- Better control of the bill of material.
- Project detailed documentation.

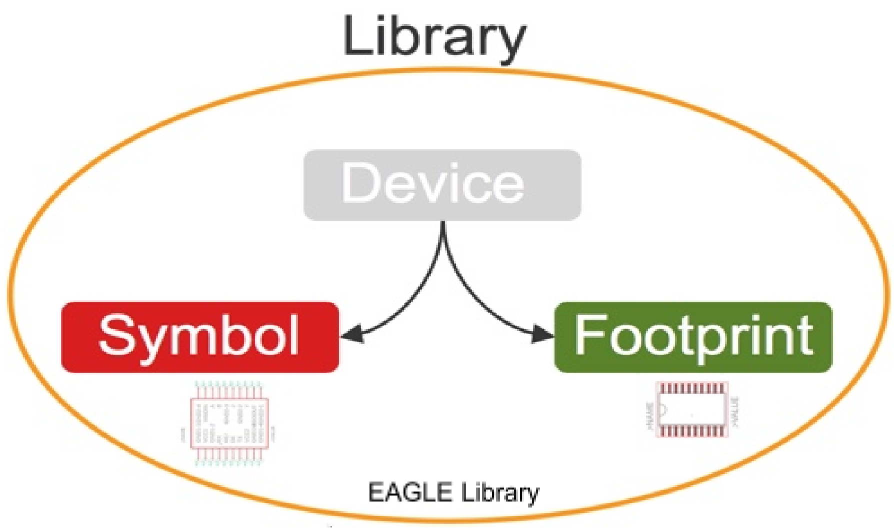
The Autodesk EAGLE and Fusion 360 integration works because the core foundation of EAGLE libraries is capable of including 3D models. Autodesk EAGLE stores the components in files called Libraries. The libraries contain from a single component to hundreds of parts. There are many ways to manage the libraries. The default installed libraries in Autodesk EAGLE are managed by function and manufacturer. Another good idea is to have a library for each design project.

Comprehensive explanation of the Autodesk EAGLE Library editor

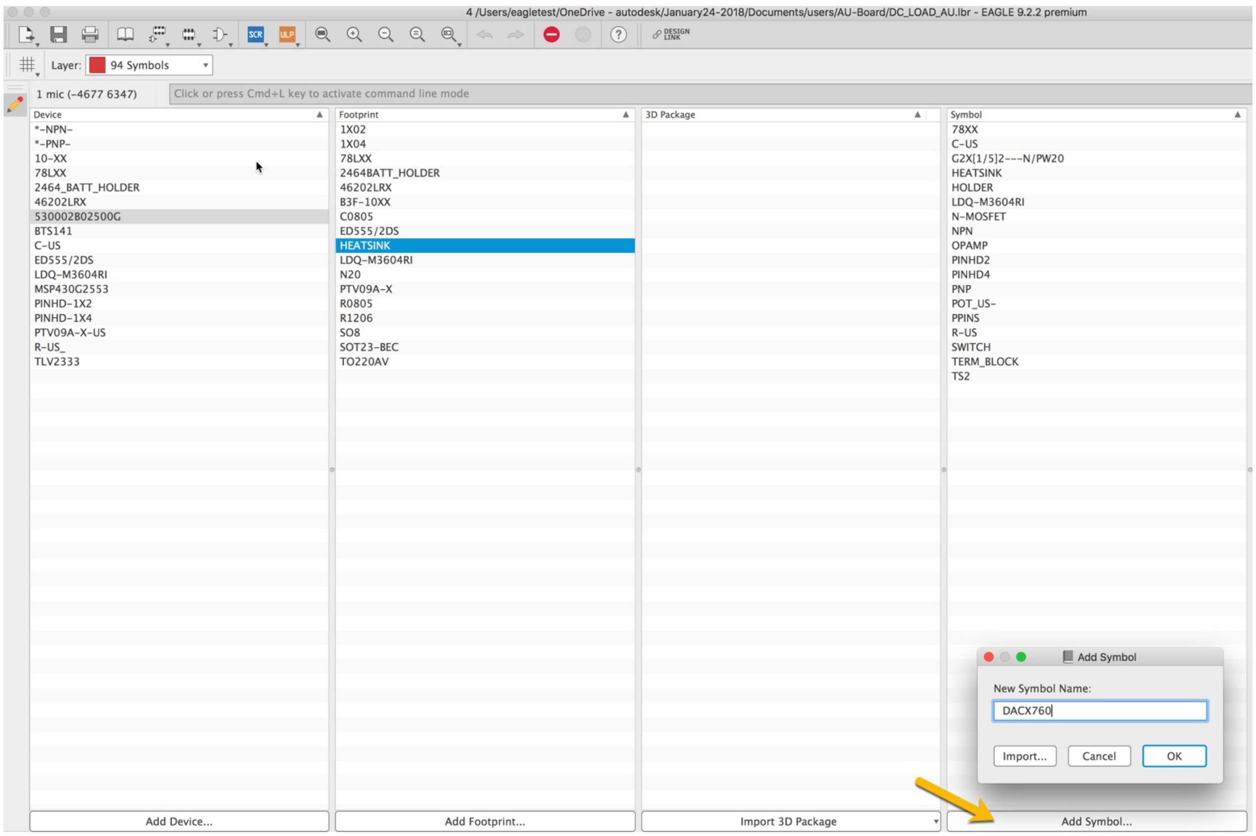
The default installation of EAGLE includes thousands of components, but there will be times that components required for a design are not available. The engineer might choose to search for the parts in different medias that are available and hope their in a compatible format with EAGLE. It is imperative to verify the dimension of the component acquired by other medias before committing to its use in your design. A badly design footprint can cause design delays, therefore wasting unnecessary funds. It is common for Electronic Engineers create their own components. Creating your own parts usually guarantees your components will be done correctly. Autodesk EAGLE includes a an elaborate library editor.

The EAGLE library editor can be referred as a platform which includes multiple editors that collaborate between each other. The main 3 stages are comprise of:

- Schematic Symbol Editor
- Footprint Editor
- Device Editor



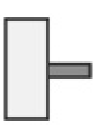
From the EAGLE control panel click on File New Library to create a brand new library or expand the library tree to edit an existing library. The EAGLE Library editor opens to the Table of content, from here it is possible to access and preview all of the assets available in a library. Click on the Add button at the bottom of the symbol column to create a new symbol.



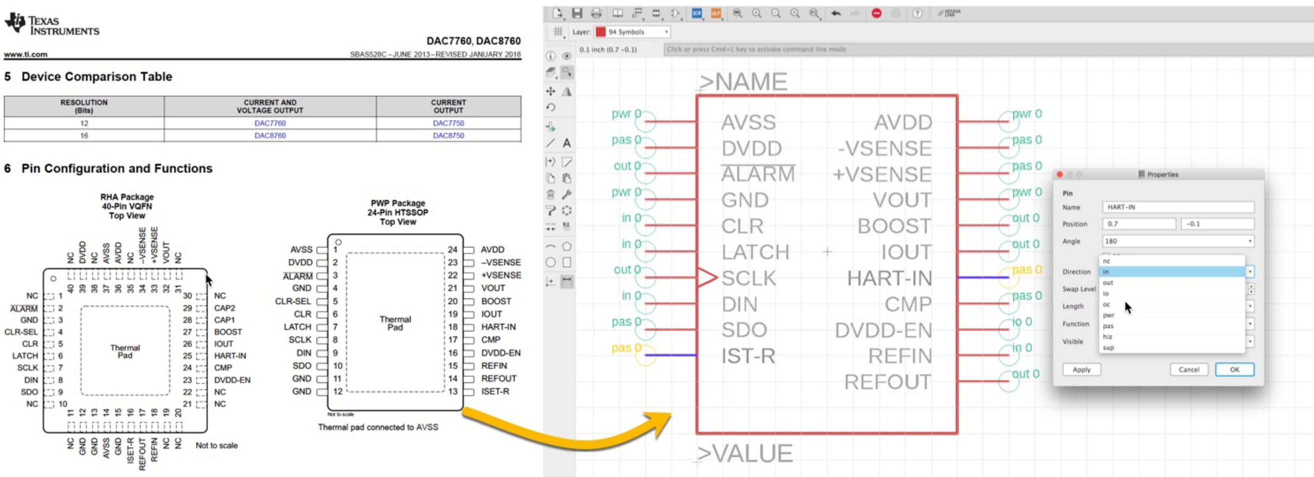
EAGLE Library Table of Content

Schematic Symbol Editor

In the device editor we will create the schematic symbol based on the manufacturer's specifications. Since this a logical representation of our footprint, the mounting details can be ignored. We are only interested on building a symbol which will consist of the required pins, pin direction and function as detailed by the manufacturer. Use the PIN command to place your pins. After placing your pins right click the pins and select properties option from the context menu. From the properties dialog name and appropriately assigned the pins function and direction.



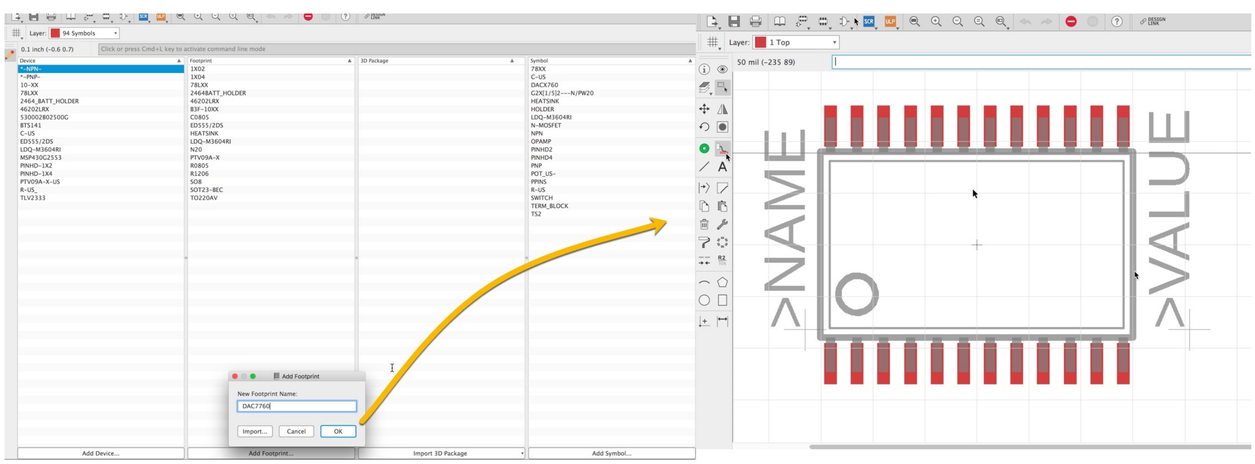
Pin command



EAGLE symbol editor

Footprint Editor

The footprint editor is considered the most important part of the electronic project since it represents physical part. The footprint that is created in the editor will need to include the exact mounting measurement details based on the manufacturer's specification. The footprints define the component landing points that the PCB will need that way components can be soldered to the board. There is a very small margin of error, a bad footprint usually means bad boards. From the EAGLE library editor table of content footprint column, click on the ADD button and name your new footprint and press OK. The loaded editor will have a similar interface as the earlier discussed symbol editor.



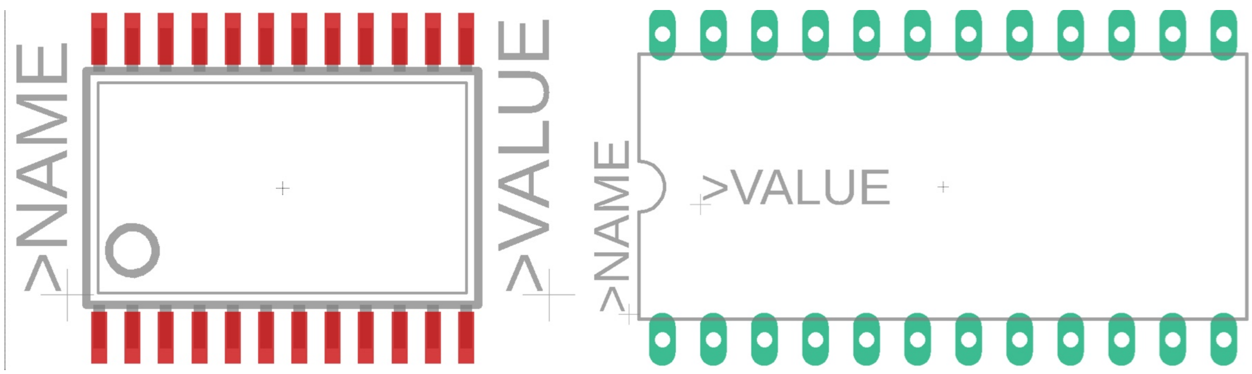
Accessing Footprint editor

There are 2 type of pads:

- Thru-hole pads
- Surface Mount pads

Thru-hole pads are plated through holes that have a conductive path from one side of the circuit board to the other.

Surface mount pads allow components to be set on the board surface, currently this is the dominant method of assembly used today.

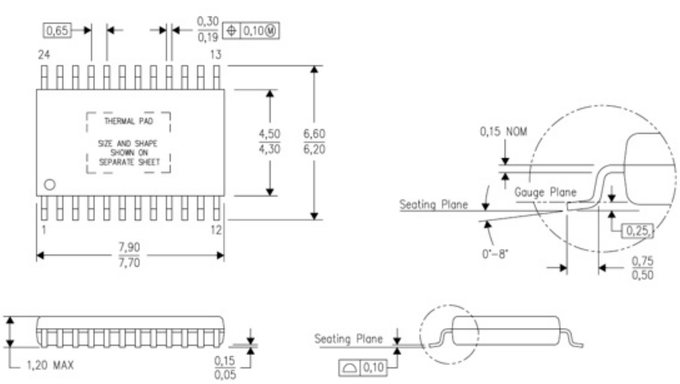


Component PCB Landing Types

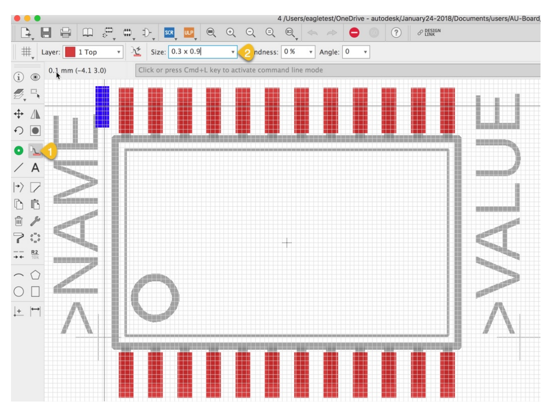
Before designing the footprint, it is a good idea to take a look at the manufacturer's specifications sheets for the exact component measurements. The initial values that should be observed are:

- Pad Spacing AKA pitch spacing
- Pad Size

Common practice is to set up the active footprint workspace grid at value ½ to ¼ of the manufacturers pads spacing. Selecting the SMD command will activate the parameter toolbar to specify the landing size.



Manufacturer's Specifications



EAGLE footprint Editor

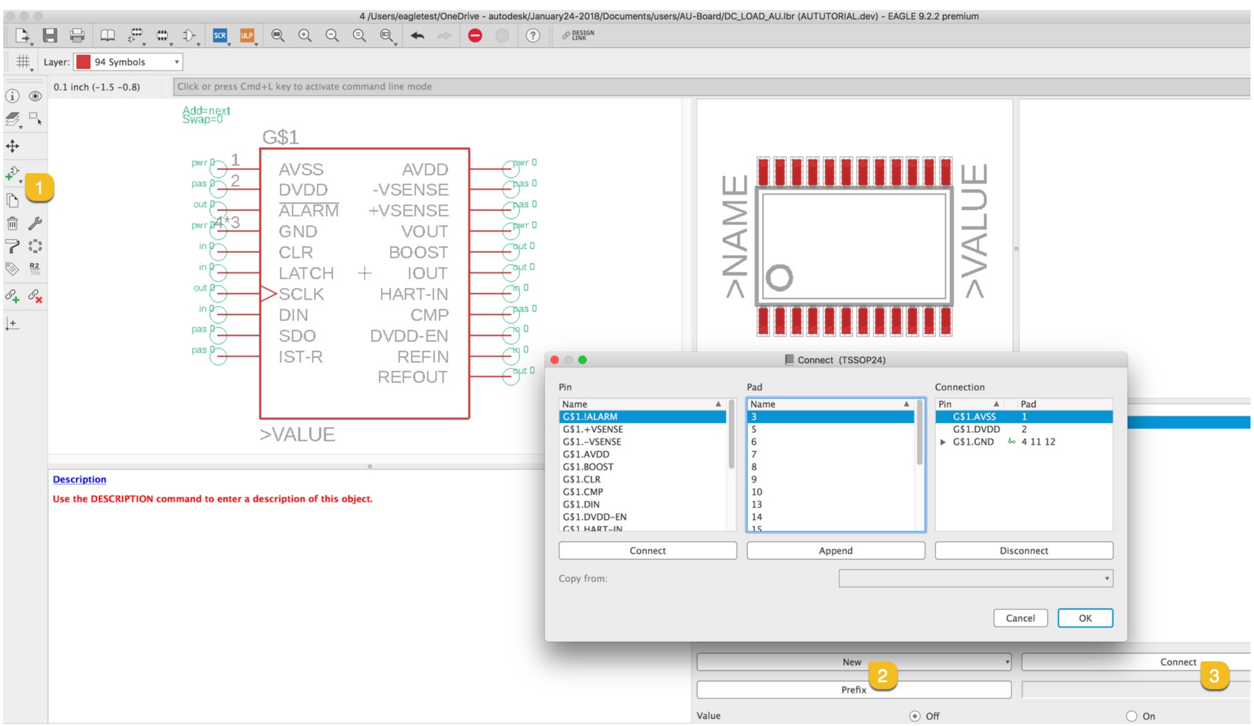
After placing the pads and drawing the component outline we can move on to the device editor.

Device Editor

Everything comes together at the in the device editor. At this stage we will associate the schematic symbol and the footprint, this is the core that provides EAGLE's a real time annotation workflow between the Schematic and the PCB editor.

From the table of content select the Add button at the bottom of the Device column, and give your device a name.

1. Click the Add icon to select the symbol that was created in the symbol editor
2. Click the New Button at the bottom right to select the Footprint that was generated during the footprint editor.
3. Use the Connect command to associate the schematic symbol pins to the corresponding pads on the footprint.



This process concludes with an accurate 2D component which can be used for our ECAD workflow, but does not provide the support for our electromechanical workflow. For the remaining of this handout, we will explain the available options to include 3D models to your footprints.

Methods available in the EAGLE library editor to assign 3D models to footprints

Traditional electromechanical workflow meant converting file format such as DXF, IDF, STEP and others between the electronic and mechanical engineers. The challenge doesn't stop here, file conversion needed to be done every time updates were done to the project. This caused many design errors and extended product release.

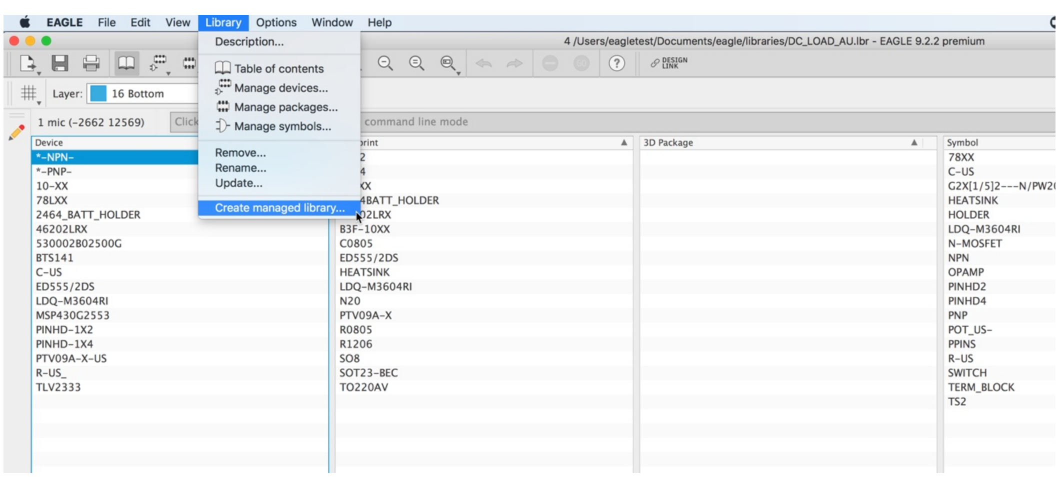
We have established that the foundation of all PCB designs is accessing component libraries required for the design. Autodesk EAGLE does provide a comprehensive library editor that extends its capability by providing the means of including 3D models for the footprints. By establishing this relation at the core of the ECAD design workflow, it facilitates the Autodesk EAGLE and Fusion 360 electromechanical workflow.

Before we begin explaining the available methods to include 3D models to the library, we need to explain the concept of Managed libraries. A Managed Library means that the local EAGLE library has an exact duplicate on the cloud (library.io). The cloud version of the library includes the 3D model. The link that exist between the EAGLE Local footprint and the cloud stored 3D model is identified as the URN (Unique Random Number). The URN identifier remains with the component when it is used in a design.

Creating a Managed Library

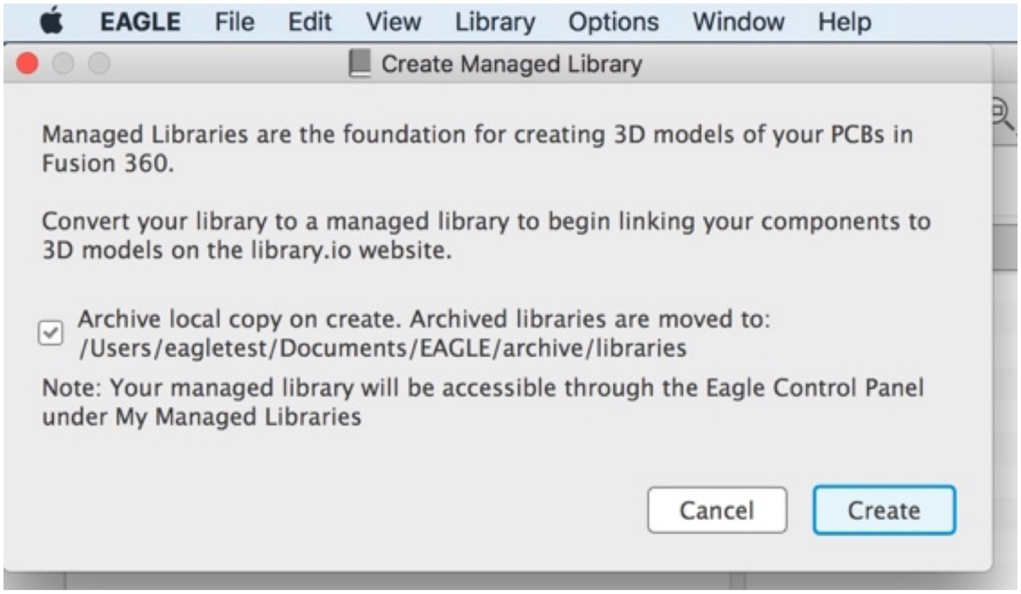
In addition to providing the 3D model mapping capabilities to the library editor, managed libraries does provide you remote access to your libraries. Updates to your managed libraries will be flagged when accessing your library from another computer. Default installed libraries are managed and constantly being updated, the managed library system makes sure that you are always using the latest library version.

From the library editor we will demonstrate how to create our first managed library. Click on the library drop menu and select the option "Create a Managed Library".



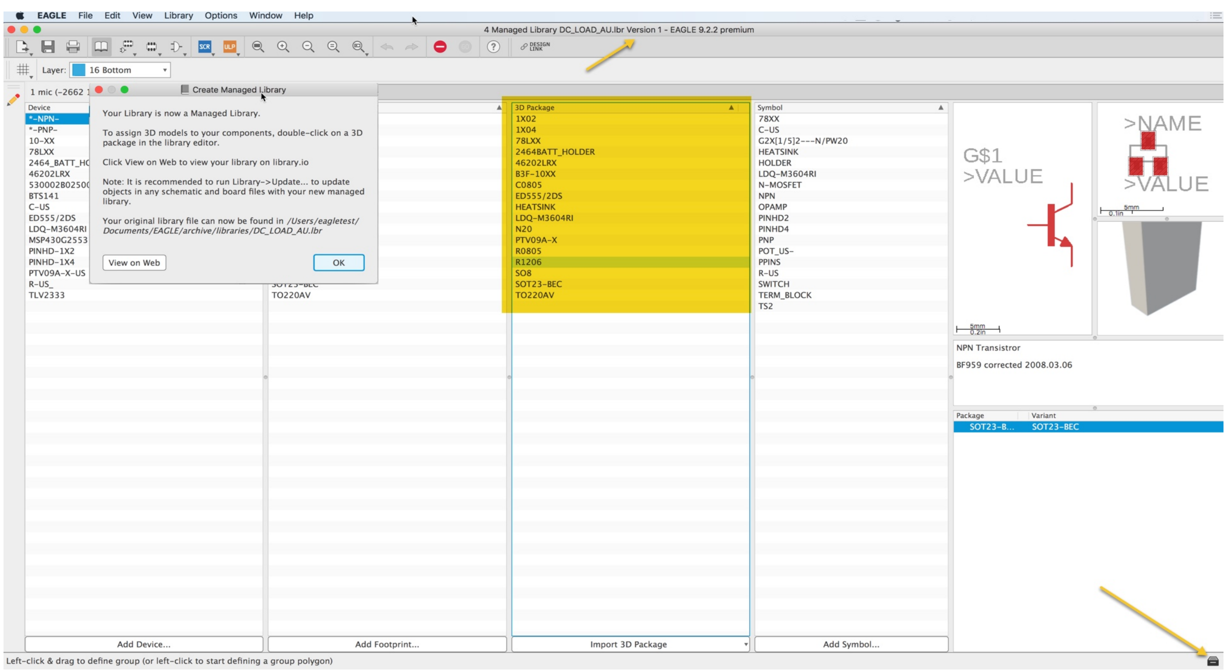
Create Managed Library

After acknowledging the link of your libraries to the Autodesk EAGLE library cloud service of library.io and the backup option, EAGLE will begin the process of generating a managed library.



Create Managed Library and Backup

Library.io is the cloud service that works in parallel with EAGLE and empowers managed libraries. The managed library will be created in a few moments, click on the **View On Web** button if you want to access your library on the library.io website. Click OK to continue.



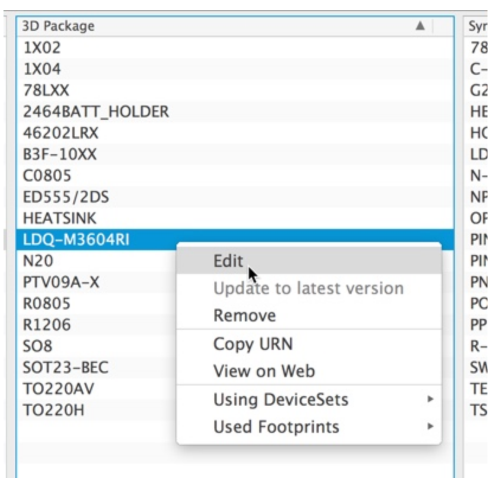
Managed library Completed

The library table of content will have some added information. Managed libraries are versioned, please notice that at the top status bar title the library name now has a version number. The 3D Package column is now populated with default packages. These are only boxes that have been preliminary added to represent the 3D package of your footprints. A thumbnail representing your the 3D model will now be added to the preview window and finally, notice that there is a file cabinet symbol at the bottom right corner of your table of content.

The file cabinet symbol will change color depending of the status of the managed library. Black means the library is up to date while blue means that local changes have not been pushed to the library.io. This can be fixed by creating a new version of the library, further along the handout we will explain more of this.

Mapping STEP Models to component footprints

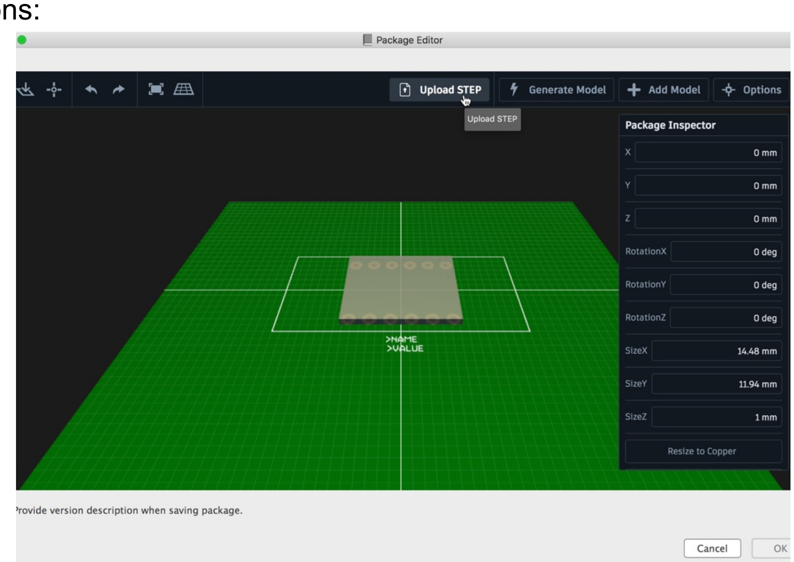
Managed libraries extend the capabilities of EAGLE by which 3D models can be associated to library footprints. STEP files is a ASCII structured file which is commonly used to share 3D models in different CAD applications. STEP models can usually be located on the manufacturer's website, Autodesk library.io and few websites that provide models. With Fusion 360 you can draw your own electronic components using the Manufacturers specification sheets. tool that can be used to create electronic components and exported in STEP format. We will expand on this later in the handout.



Edit Package

Now that the library has been managed we will start by assigning a step model to one of the available footprints. We will be using the 4 digit 7 segment LED display. Please right click the LDQ-M3604RI located in the 4D package column and select the edit option from the context menu.

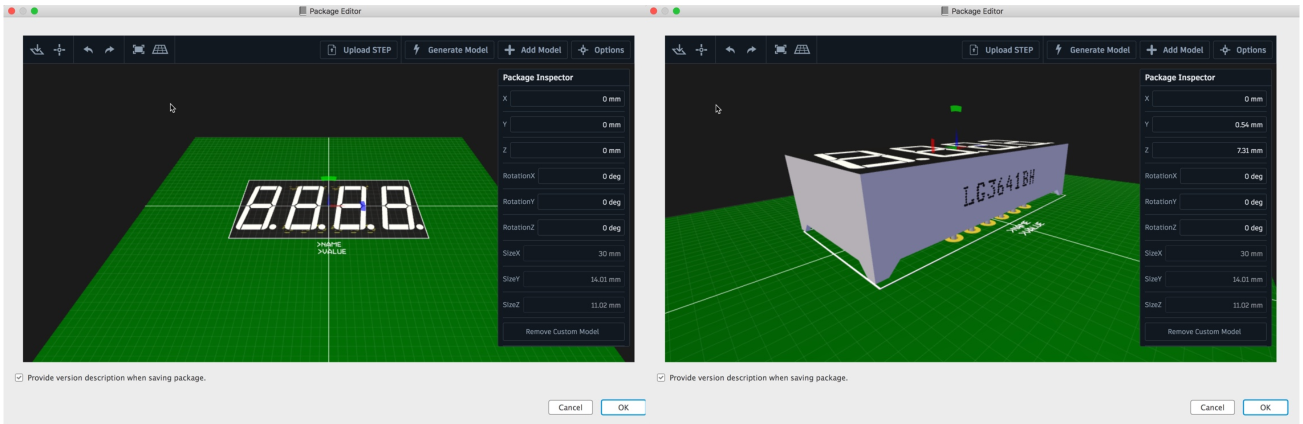
This will launch the library.io package editor which has a interface in EAGLE. At the top of the editor there 3 options:



- Upload a STEP
- Generate a Model
- Add Model

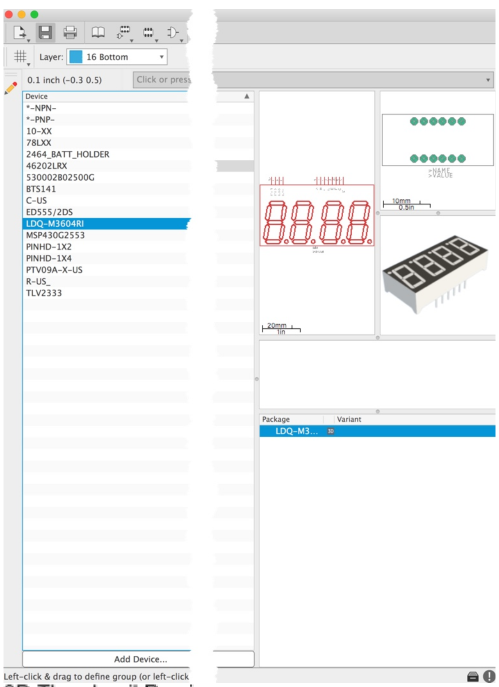
For our exercise we will use the option to upload a STEP model.

Navigate your system and select the step file you would like to use for to map the footprint.



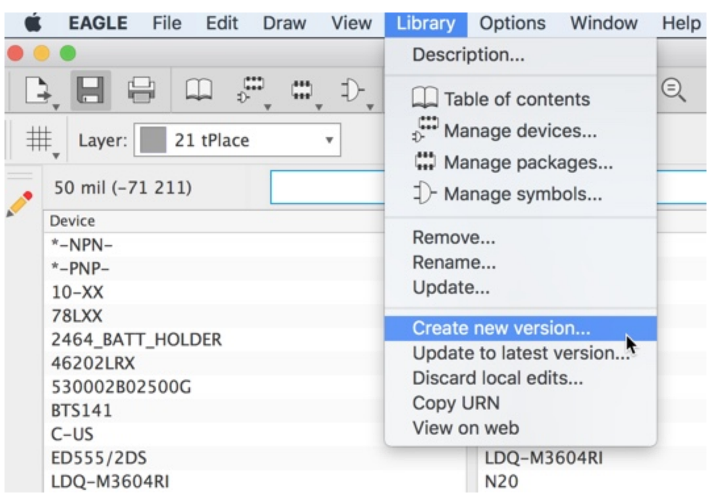
Step Model Uploaded

The imported STEP model will not always line up with the footprint. Use the package editor screen arrows to manually rotate and move to line up the STEP model. On the right side of the package editor is the Package Inspector, it has fields by which you can fine tune the coordinate values to line up the model. Press OK after lining up the STEP model, an optional comment screen will appear that way notations can be added to this package.



3D Thumbnail Preview

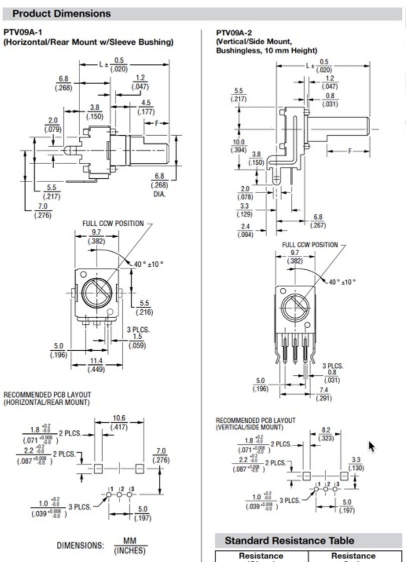
After several moments, a preview thumbnail will preview will appear for the 3D package as well as the for the device. After making changes to a managed library, it is necessary to create a new version. This implies the synchronization of the local library and the mirrored library on library.io.

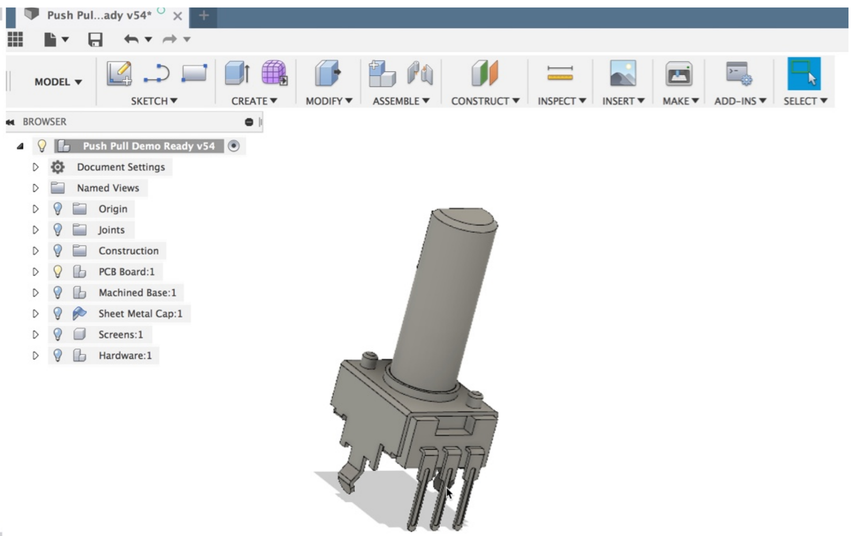


Create New Version

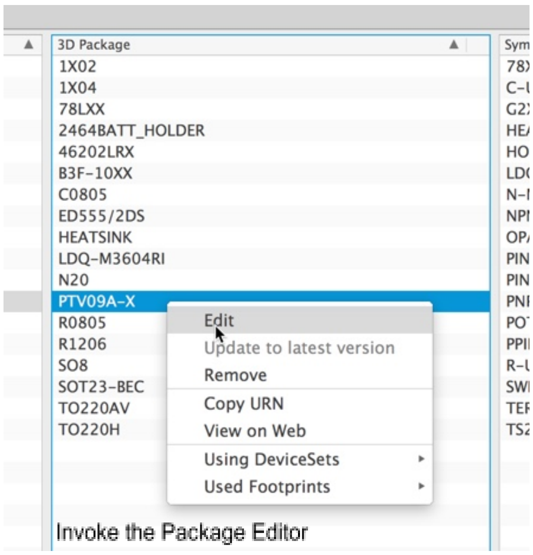
There will be instances in which a STEP model for non standard footprints are not readily available, examples of this could be heatsinks, potentiomenters, inductors just to name a few. It is possible to use the manufacturers specifications and build your own 3D step model using Fusion 360.

With the component specifications, use Fusion 360 to create Electronic Components and export them in STEP models

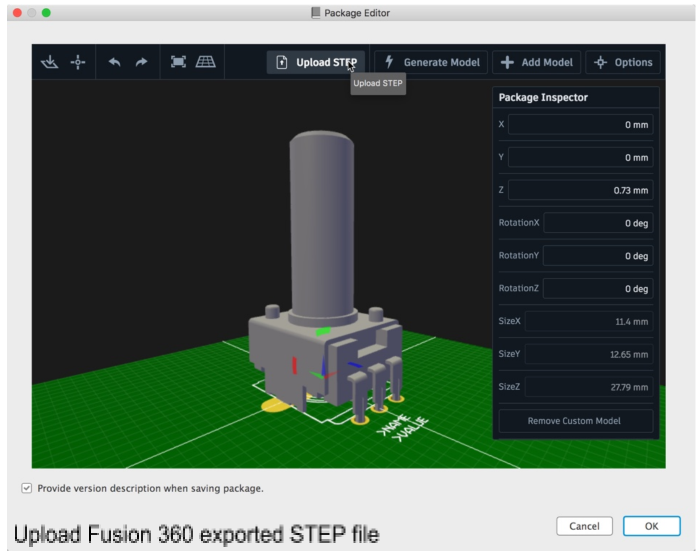




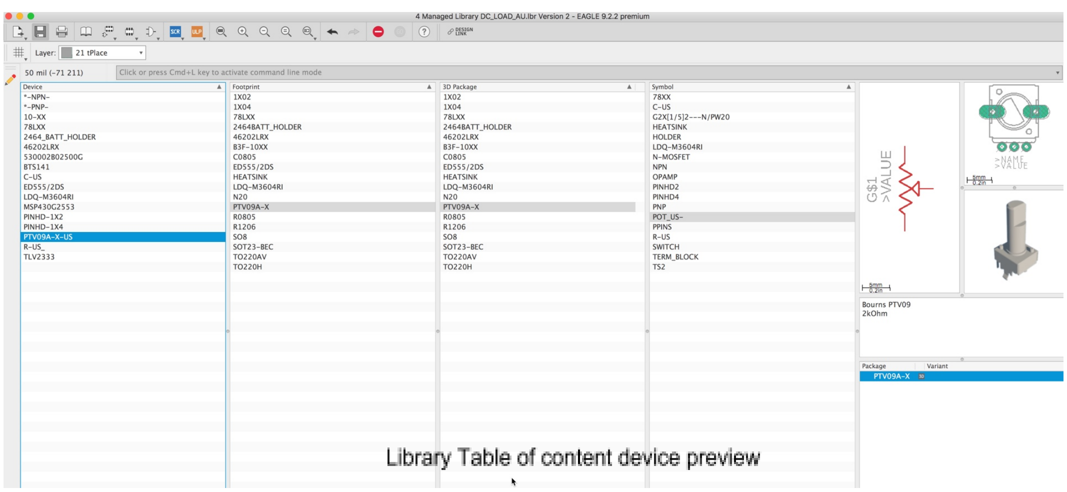
After exporting the created component from Fusion 360 in STEP format, we will proceed mapping it to our EAGLE footprint. From the EAGLE Library table of content, right click the 3D package for the PTV09A-X, from the context menu select the Edit option. In the Package editor, click the Upload STEP button and select the STEP file exported from Fusion 360 and click OK. After several moments the table of content will include the 3D Model thumbnail.



Invoke the Package Editor



Upload Fusion 360 exported STEP file

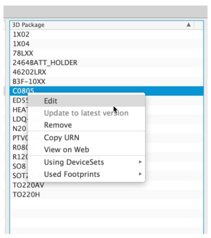


Library Table of content device preview

After saving your library it is necessary to synchronize the local library with the cloud mirror version of your library. Click on Library pull down menu and select the option, Create a new version. After this push, you will notice that the version number of this library has changed. The new version number is located on the top title bar.

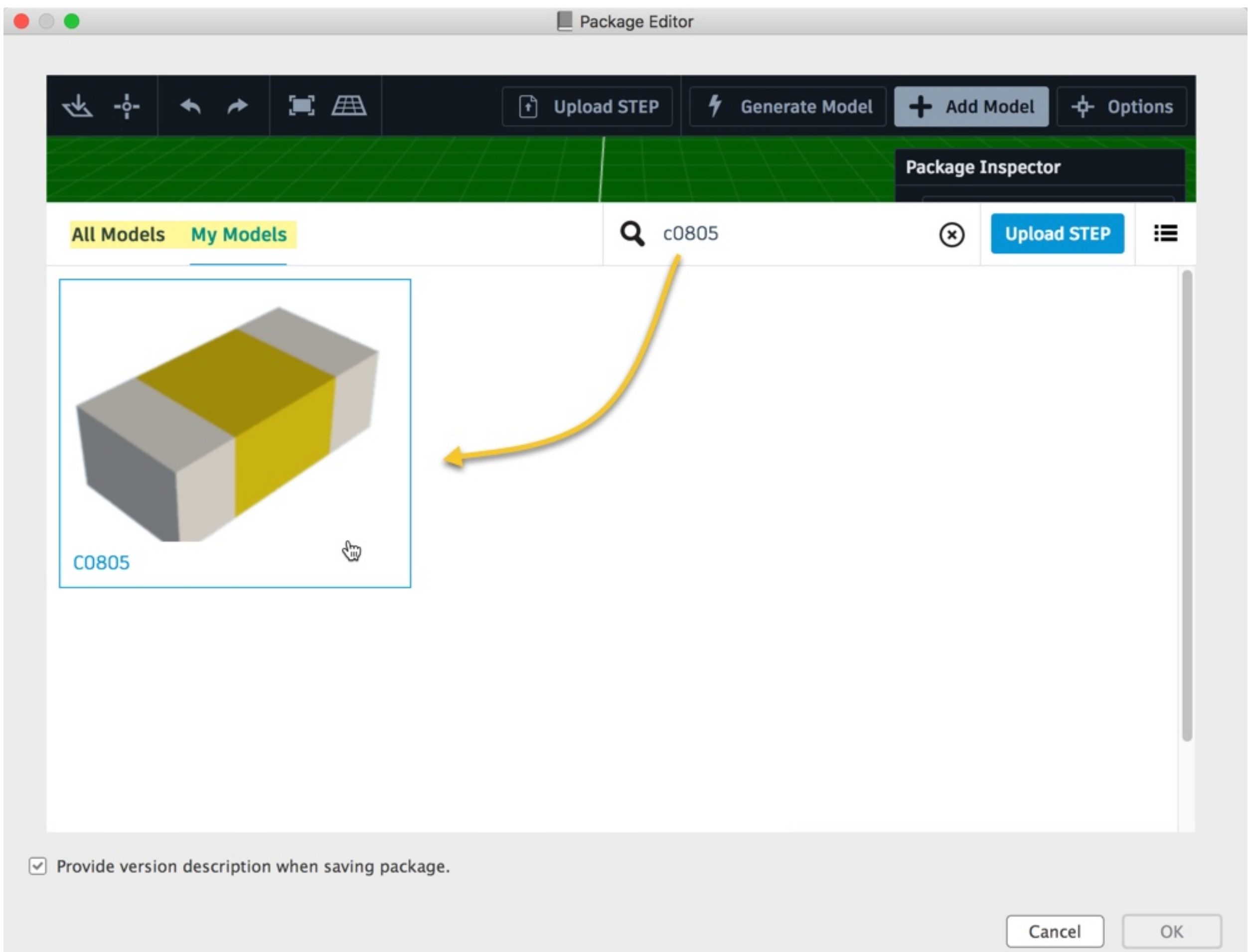
Package editor, Add model

Library.io does include a large repository of 3D models that can be used to map to your footprints. It will also make available the ones the you have uploaded from other projects. From



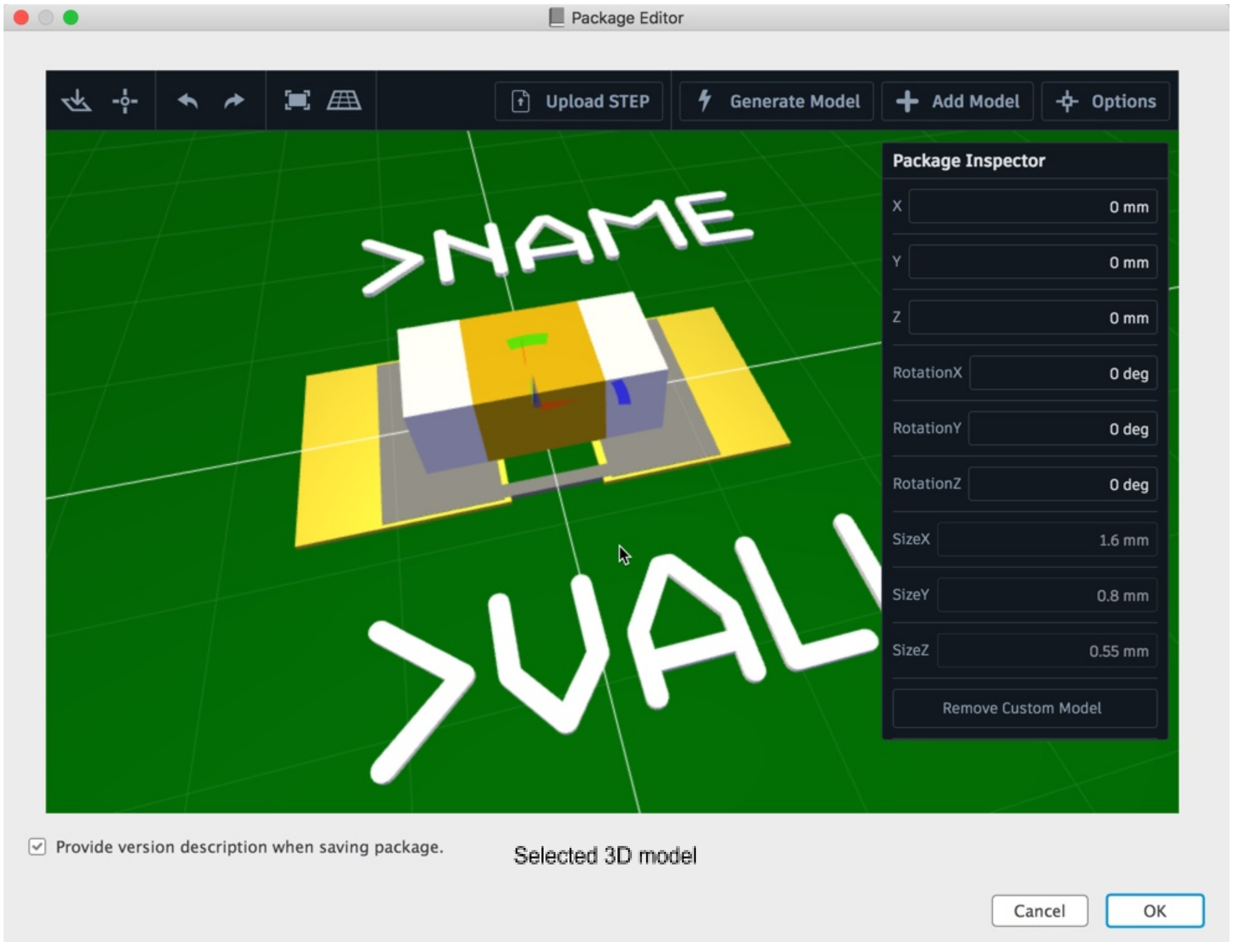
the EAGLE library editor table of content, right click C0805 package located in the 3D package column, from the context menu select Edit.

In the package editor click the button called “Add Model” at the top tool bar. You can filter between models you have upload or include the default library.io repository models. Use the search field to narrow down the component that needs to be used. For this footprint, search for the C0805 package. This is a commonly used package.



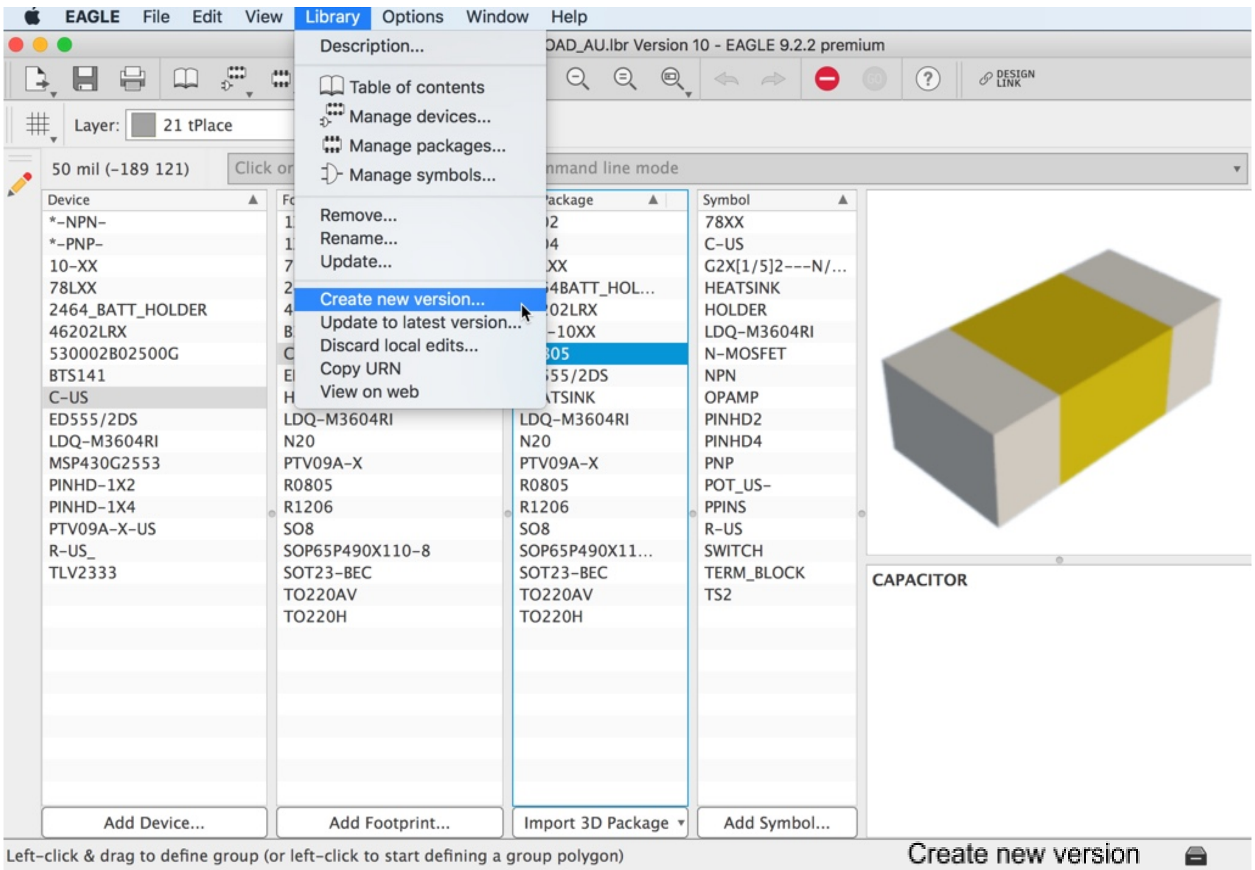
Found available 3D model

Click the model you need that matched your footprint and click OK. The model will now appear in the package editor.



Click OK in the Package Editor, the optional comment dialog box will appear. We recommend that you add some notation for example: “3D Model Added” or “STEP file from Manufacturer” then press OK.

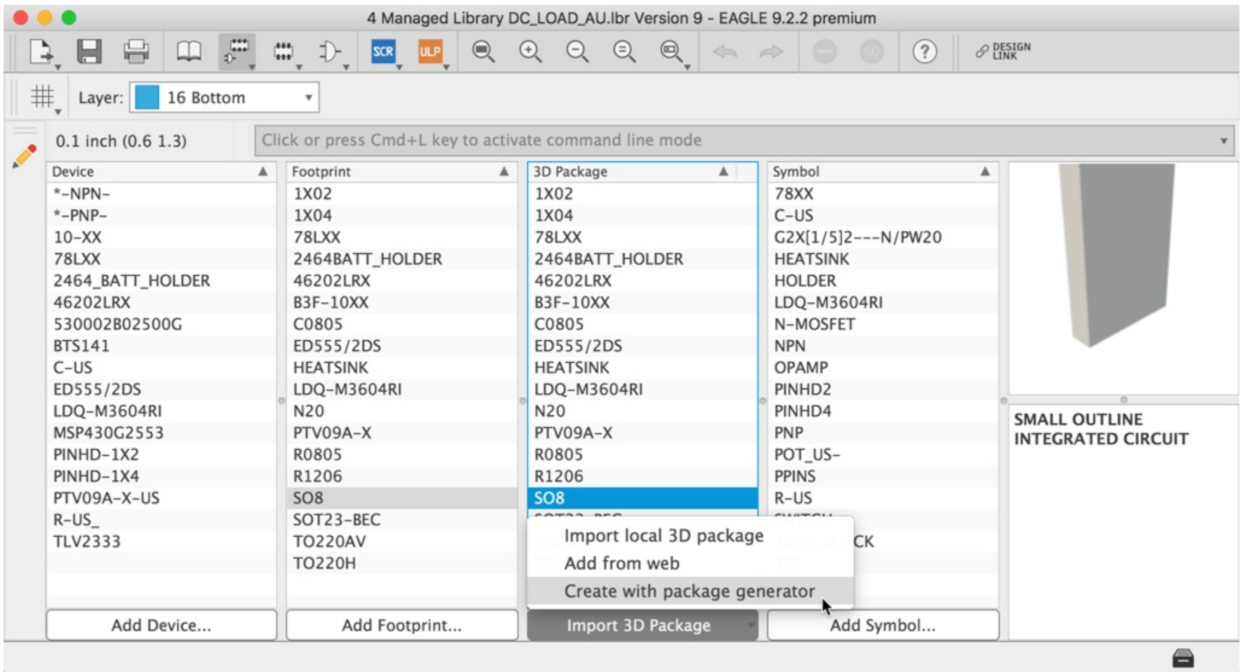
Save the library, then click on the Library Pull down menu and select “Create a New Version” to synchronize the managed library with the new edit. After several moment the 3D model thumbnail preview will appear when highlighting the C0805 3D Package.



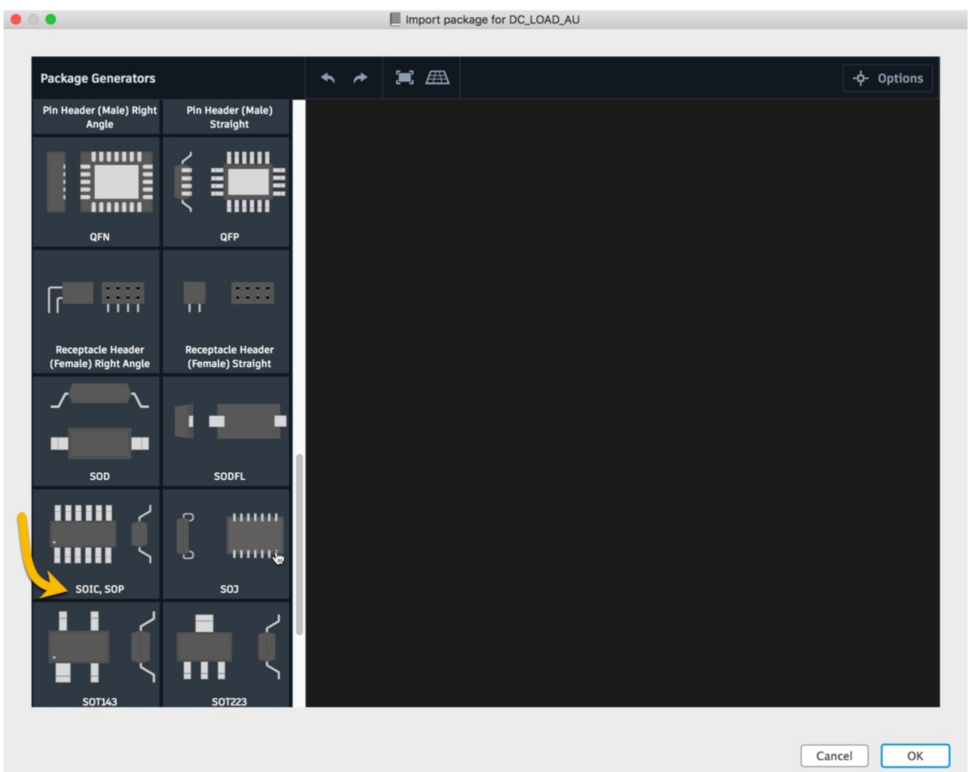
Creating Footprint and 3D model in a single step

Library.io does provide a parametric ECAD - MCAD component generator that can be accessed from the EAGLE package generator. This generator uses IPC industry standard calculators to create the footprint and 3D model. The generator is not limited to IPC based components only, there are other models available, such as pin header and axial parts. The Generator Model will provide several model templates, select the template that applies to your component.

For this example, use Texas Instrument TLV333 CMOS Operational Amplifier specification sheet. We will be using the SOIC 8 pin package footprint model. From the EAGLE Library editor Table of content, at the bottom of the 3D Package column click the **Create with Package Generator** option.

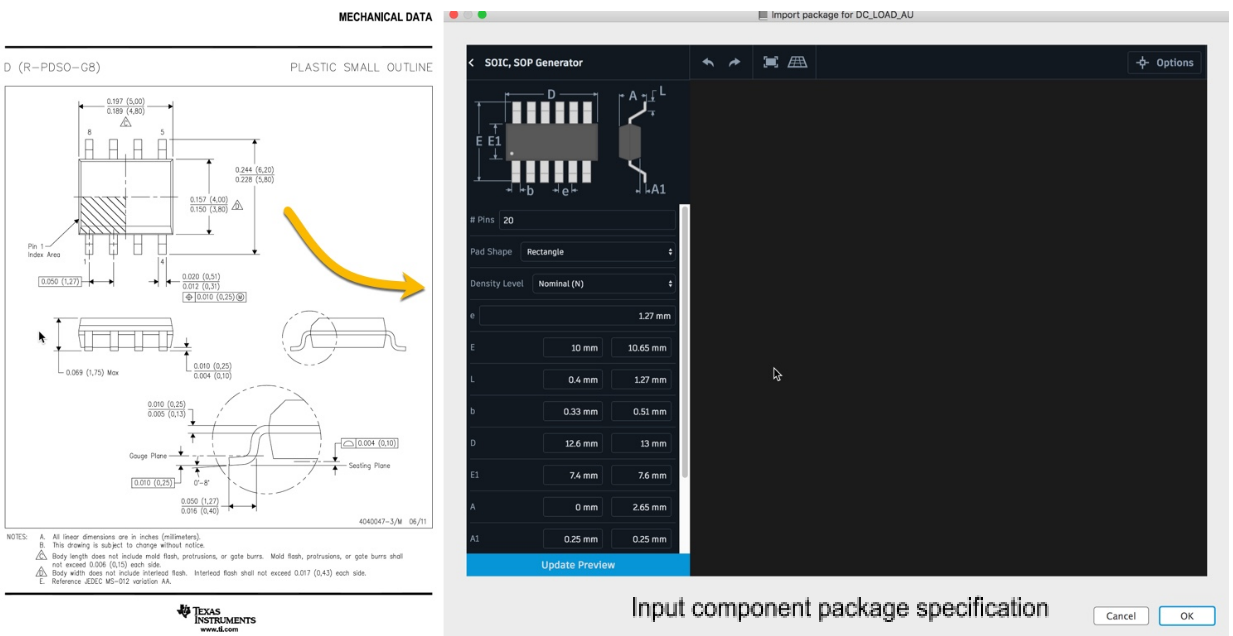


This will access the library.io package editor with the model template selection. Select the SOIC template.



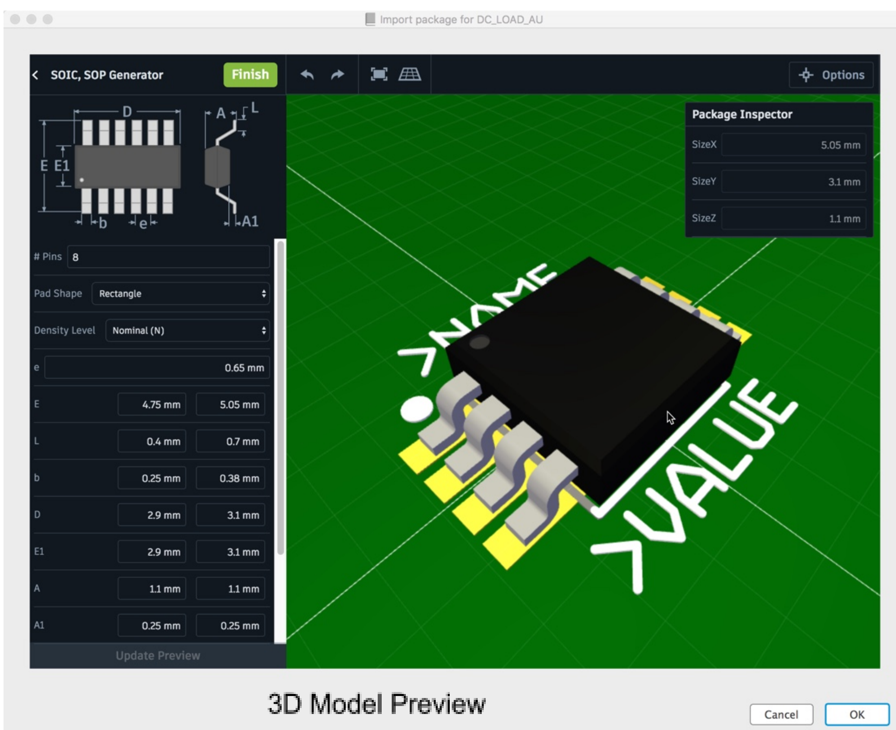
After selecting the SOIC Template, the SOIC generator will open. At this point transfer the measurements provided by the in the Texas Instrument specification sheet to the Model Generator.

Manufacturer Specification Sheet



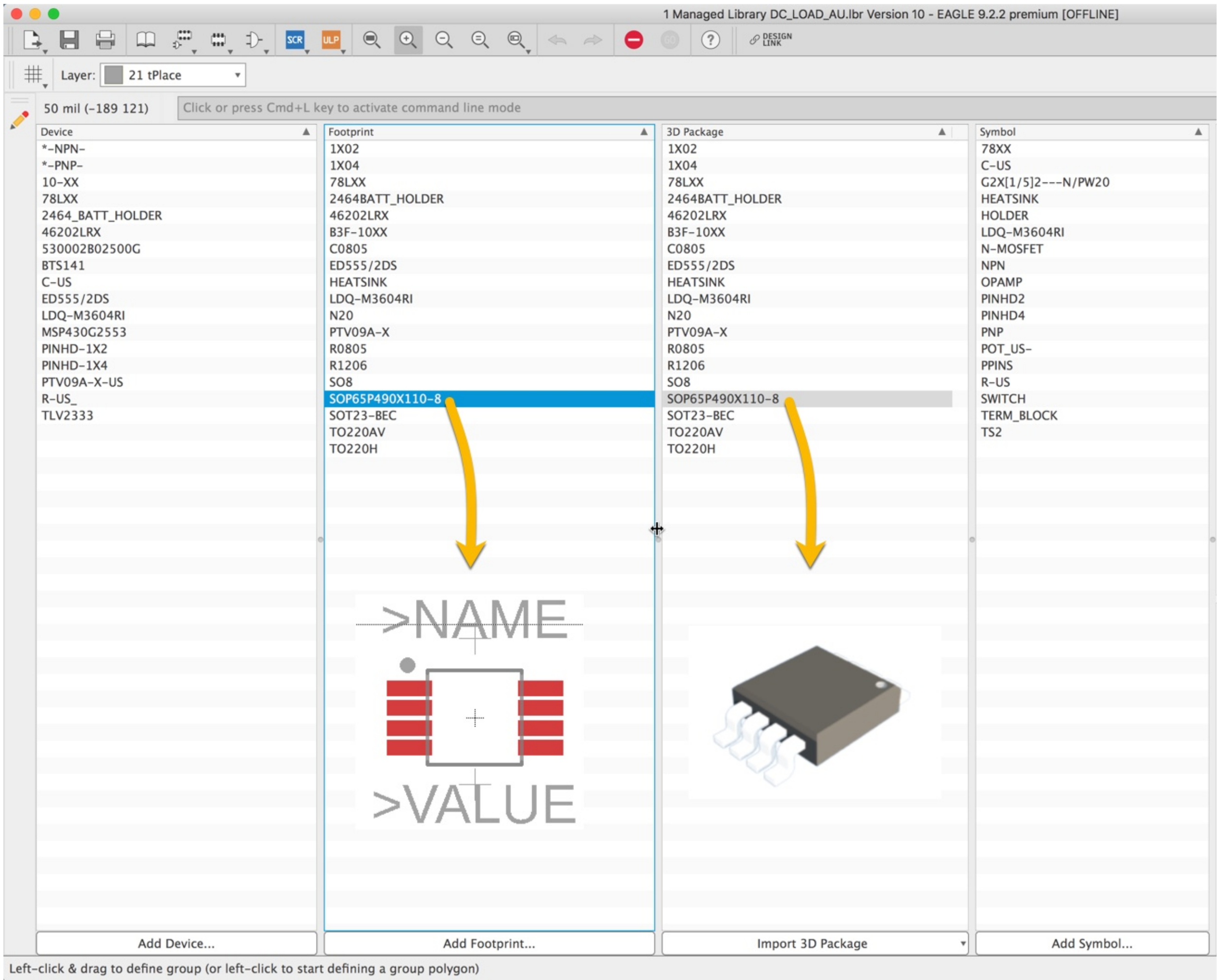
Input component package specification

Click the preview button to preview the 3D model model.

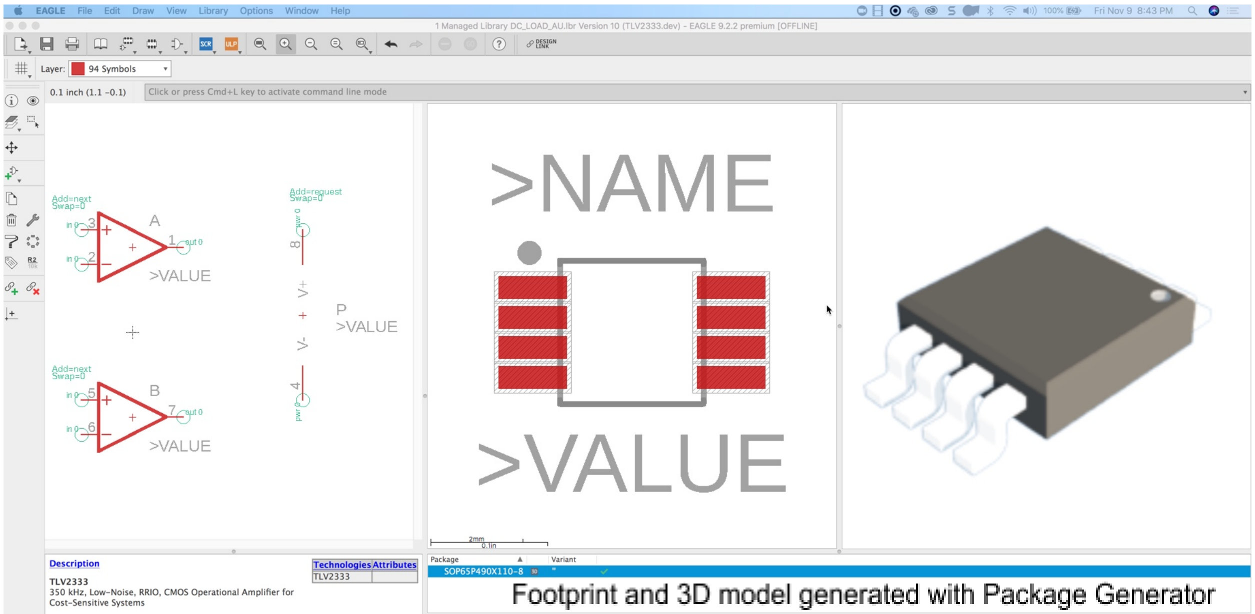


3D Model Preview

Click OK to add this model and the footprint to the managed library. From the pull down menu selections click on Library then the option “Create a new version”. This will force the managed library to synchronize.

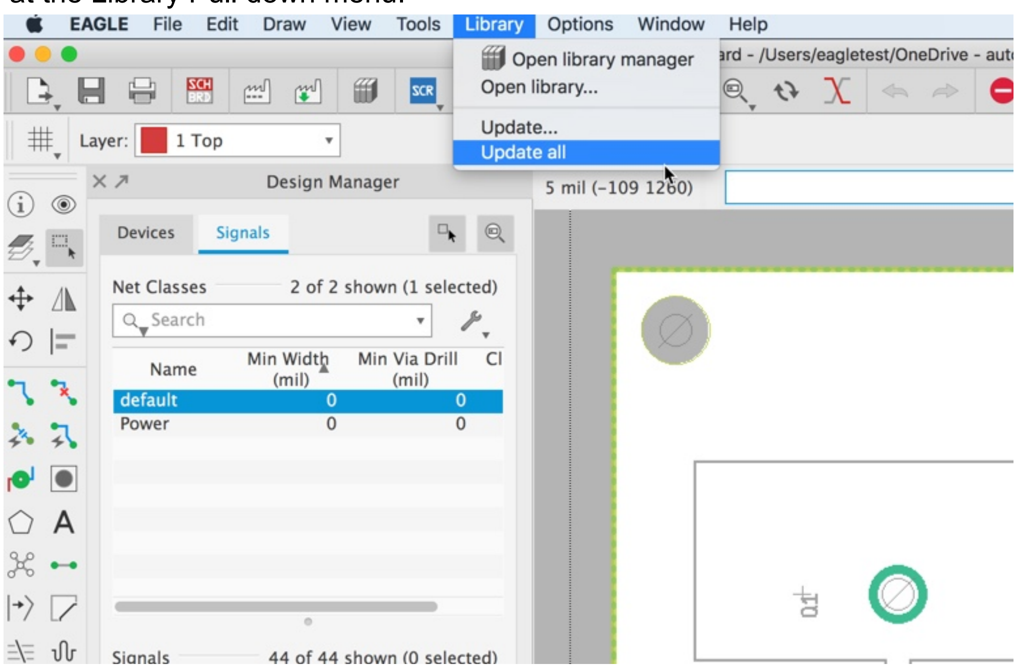


The footprint and 3D model were simultaneously generated. The ECAD engineer can proceed to use the generated footprint to create a device.



By having all the footprints associated to 3D models using the explained methods in this handout, the engineer can proceed to create a design that will use these libraries or update an existing design that was has been using this library.

To update a design, from loaded design project, click on the Update Library option that is available at the Library Pull down menu.



With the Autodesk EAGLE designs made or updated with the components that have associated 3D models, a working Electromechanical workflow with Fusion 360 is established.