[CP321596]

Creating a Pulse Oximeter: Utilizing the Cloud to Prototype in EAGLE and Fusion 360

James Youmatz
Fusion 360 Technical Support - Autodesk

Edwin Robledo
EAGLE Global Support Manager - Autodesk

Learning Objectives

- Discover the electromechanical workflow between EAGLE and Fusion 360
- Learn how to make changes to designs of schematics or 3D designs and update a PCB
- Learn how to use the cloud connectivity between EAGLE and Fusion 360 to parameterize a PCB enclosure based on changing PCB components
- Learn how to validate your results in the Fusion 360 Simulation environment

Description

Today's smarter products require multidisciplinary levels of engineering. Converging the electrical and mechanical portions of a design is a critical stage in the product development process. Design teams must have an effective bidirectional electronic CAD (ECAD) / mechanical CAD (MCAD) collaborative environment. This class will introduce the first-ever cloud-connected ECAD-to-MCAD workflow between EAGLE software and Fusion 360 software. By creating a pulse oximeter, this class will showcase the workflows of creating a printed circuit board (PCB) and assigning 3D models to the electronics in EAGLE. The PCB will be live linked into Fusion 360, where an enclosure will be built around the PCB. You'll learn strategies to ensure serial ports and LCDs are built into the enclosure. The class will demonstrate the power of cloud connectivity by making changes to the board layout, and showing how the live linking works parametrically with respect to the enclosure. We will then perform thermal validation of the board in a Fusion 360 thermal simulation.

Speakers:

James Youmatz

I've been a Technical Support Specialist at Autodesk for 4 years supporting Fusion 360 with a background in Mechanical Engineering and FEA. Been experimenting with EAGLE as well as Generative Design workflows. My current role is expanding and curating our knowledge database to better help our customers self-service any issue that may come up. Now our support team has grown internationally and I am helping to train new Fusion support folks as the Knowledge Domain Expert for Fusion.

Edwin Robledo

I am the Global Support Manager for the Autodesk EAGLE support Team. Began my career primarily working in the communications industry before joining the EAGLE team. My passion for support has lead me to publish best practices articles, Blogs and hundreds of video tutorials. I enjoy hosting live Online Workshops or onsite EAGLE boot camps. If I am not tinkering with electronics, you will find me pedaling my iron horse on the trails of South Florida.

Designing a Pulse Oximeter

Every 2 years, in Europe there is the Olympics of Engineering sponsored by WordSkills. Autodesk has been one of the sponsorsfor the past 2 terms. The most recent WordSkills event was hosted this year in Kazan, Russia. The event challenges students from different parts of the world to create something based on the given topic provided the day of the competition. There are over 1300 participants from 63 countries participating. Students must use their knowledge and know how to meet the expectation of the judges. As expected, electronics is one of the categories. 24 participants were provided the design concept that would be implemented and off they went.

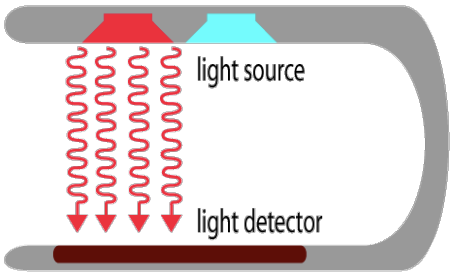
In July 2018, Autodesk was approached to produce the electronics project for the competition. This project would form the basis for the competition tasks and the reference by which the competitors would be judged. The first order of business was determining what the project would be.

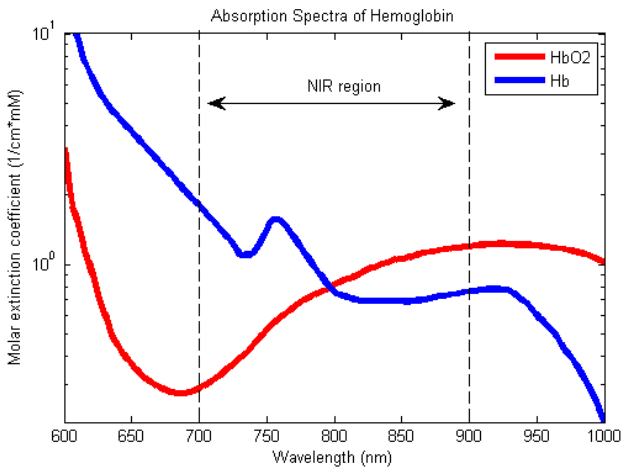
We went back and forth with several ideas including, audio projects and some robotics. In the end though we settled on making an EKG/PulseOximeter combination device. Eventually the EKG portion was scrapped due to time constraints.

How Does it work?

(For full details see AN4327 from NXP via https://www.nxp.com/docs/en/application-note/AN4327.pdf)

A pulse oximeter is technically measuring oxygen saturation in the blood. This is the ratio of oxygenated hemoglobin (HbO_2) to deoxygenated hemoglobin(Hb). This is done by shining light through a piece of tissue and then measuring the quantity of what passes through to the other side.The reason this works is because HbO_2 and Hb absorb light differently. This is why two different light sources are used. Hb absorbs more light at 660nm (Red light) where as HbO_2 absorbs more light at 940nm (Infrared Light).





By Adrian Curtin - Own work, CC BY-SA 3.0, https://commons.wikimedia.org/w/index.php?curid=20510064

By taking measurements at specific intervals and calculating the ratio of the light absorbed at 660nm to the light absorbed at 940nm we can arrive at the blood oxygen saturation value. Here's the base equation:

(AC660nm/DC660nm)/(AC940nm/DC940nm)

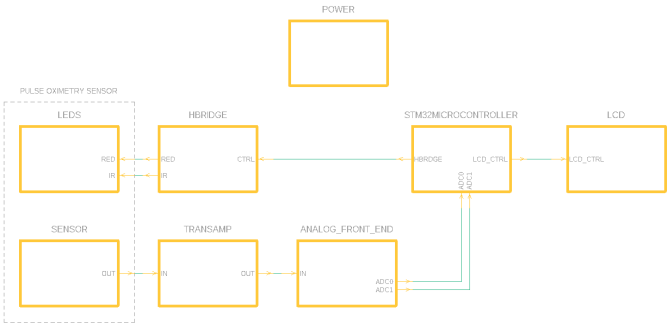
Where AC refers to the time-varying levels and DC refers to the average of the sensor levels. One simplification that can be done in hardware is to adjust the brightness of the two LEDs(light sources) so that their average level is equal. This simplifies the above equation and we arrive at the final result which is:

Sp02(blood oxygen level) = AC660nm/AC940nm

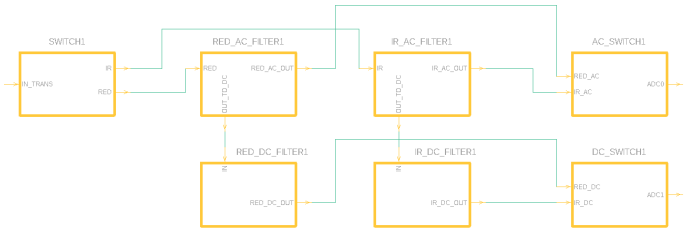
Design Overview

Our pulse oximeter design uses the STM32L052K8T6 Arm Cortex M0+ processor as the brains of our device. It coordinates the flashing of the two light sources through PWM and analyzes the signals coming from the sensor to derive a pulse oximetry measurement. It also controls the 128 x 128 pixel full color LCD screen to display sensor waveforms.

The following block diagram shows how our device works at the systems level:



As can be clearly seen the microcontroller is the key player in our design. The HBridge allows the microcontroller to control the brightness and the on times of the LEDs precisely, while supplying more current to the LEDs than the microcontroller can provide. Physically the LEDs and Sensor are part of a single assembly that was purchasd from a reputable supplier for use with our design. The transamp block converts the current signal from the sensor into a voltage signal and provides some preliminary gain and filtering before the signal is passed to the Analog Front End. The Analog Front End(AFE), warrants its own block diagram which has been included below. It's role is to further filter and process the pulse oximetry signal to remove noise and extract the signal of interest.



The AFE consists of two identical chains of filtering. One chain is for the RED LED signal and the other is for the IR LED signal. All of the switches are synchronized. The DC filters take the average of a lightly processed signal it gets from the AC filter stages. All of this processing is vital to getting a proper signal out of the sensor.

Discover the electromechanical workflow between EAGLE and Fusion 360

As many products today require both electrical and mechanical components, it is important for electrical and mechanical designers to be able to work together from the start to finish 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 as a way to save physical cost
- Be confident that they are working with the most recent version of the electrical and mechanical designs.
- Quickly respond to design changes implemented by other teams.

The PCB workspace in Fusion 360 allows for PCB board files from Autodesk EAGLE to be integrated into a mechanical Fusion 360 designs. This 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 assembly design created in Fusion 360. This allows multidisciplinary teams to:

- Drive mechanical designs in Fusion 360 off of PCB requirements from EAGLE.
- Make subsequent changes in EAGLE if necessary and push the updated board back to Fusion 360 to adjust the physical design.

The workflows made possible by the PCB workspace in Fusion 360 and the Fusion Sync button in EAGLE to allow one file to be used through the entire design process – no more endless file conversions to maintain up-to-date board files!

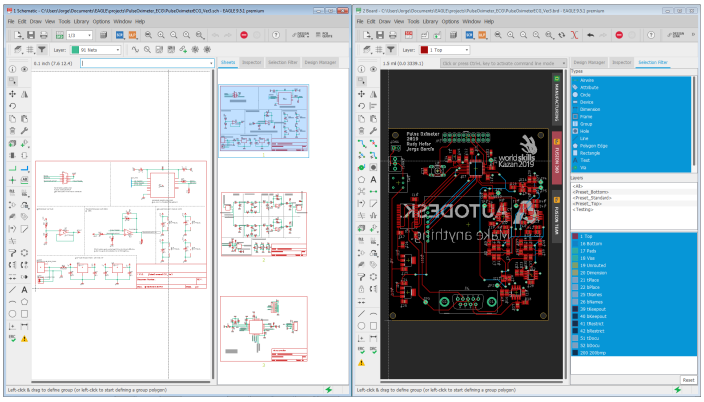
The shape and size of a PCB can often be limited by mechanical components. Before the Fusion 360/EAGLE integration, an update to the board would require;

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 to be edited to meet the requirements of the new board design from EAGLE.

This process occurs in endless iterations until the PCB and mechanical design are finalized! With the integration between Fusion 360 and EAGLE, changes to the board can now be performed in either EAGLE or Fusion 36. 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 and propels teams quickly through iterations to a final design.

Design Process – Creating the board schematic in EAGLE

Now that we have learned the details of the design, lets walk you through the process of building our design. EAGLE is a CAD application strictly used to design circuit boards and electronic schematic. The schematic is a representation - like the blueprint of a structure - of the of the board. As you notice we are working with 2 collaborating workspaces under one platform.



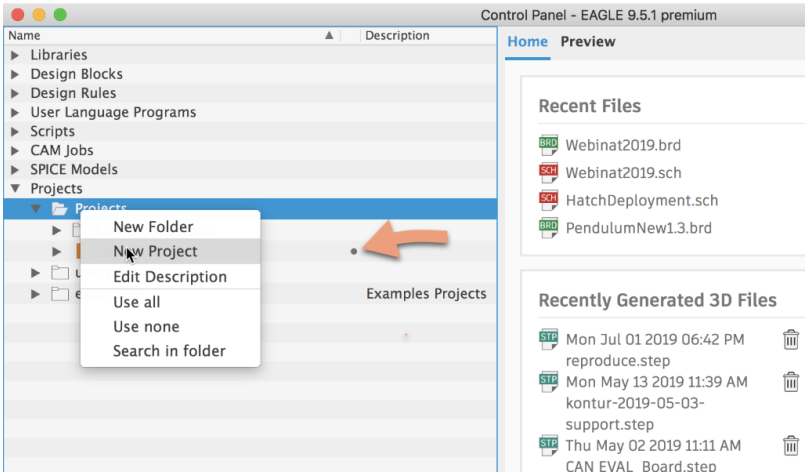
Both workspaces are continuously synchronizing in real-time. Modification done to either workspace will affect the other. It is imperative for the designer to have this capability. The concept of electronic design depends of resources available in the application component libraries. Libraries are the building blocks of any electronic design because contain the parts that will be used for the project.

Begin Designing

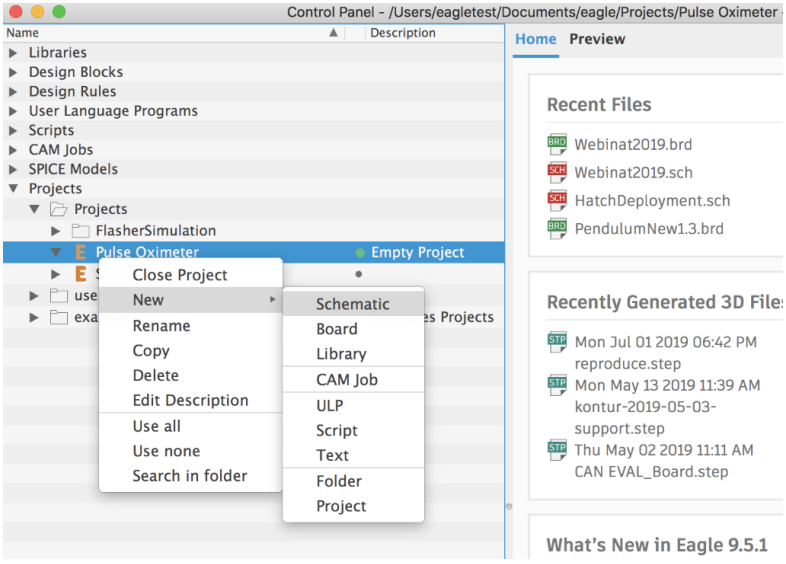
After launching Autodesk EAGLE and signing in, you will be at the Control Panel. The Control Panel appears after starting EAGLE, and serves as your control center. All the project files and libraries specific to EAGLE are managed in this powerful editor. Basic settings can also be adjusted here.

To start your brand new project, expand the projects folder in the Control Panel tree. On the project sub folder, right click and from the context menu select New Project. This newly created project folder will contain all the file pertaining to this project. The schematic file will adopt the extension, .sch, while the circuit board will have, .brd. On the right of the project's name, you will see a circle which is either gray or green.

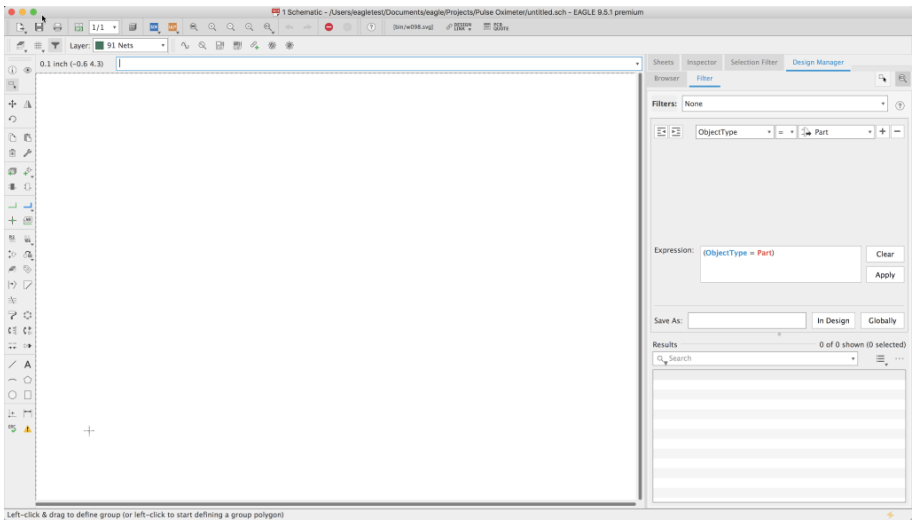
Clicking on this circle will allow you to quickly open or close a project. Once a project is open, the circle will turn green. Clicking on the green circle again or clicking onto another gray circle will close the current project and open another project after closing the current one. This process allows you to quickly switch from one project to another.



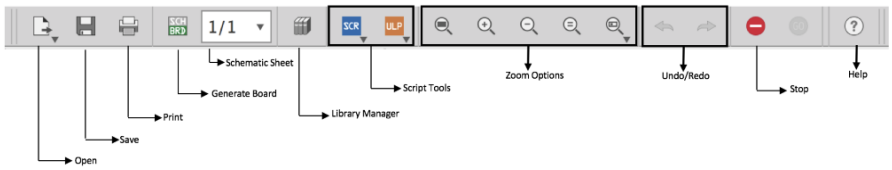
Now that we have the project folder created it is time to create our schematic. Right click the recently created project and from the context menu lets select “New Schematic”.

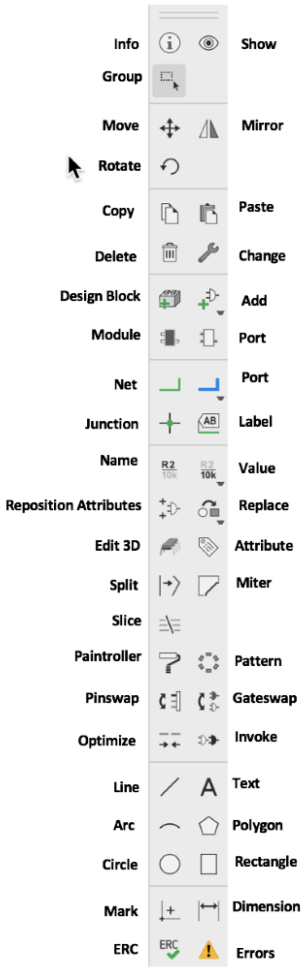


The schematic workspace will now be open. All the workspaces in EAGLE have a similar interface. The icons and command will change based on the need of that editor.

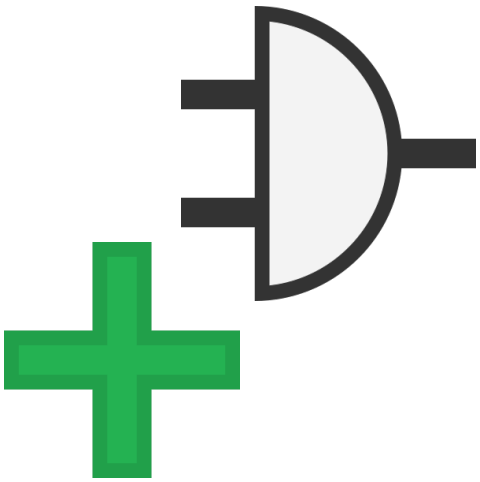


The interface does have a series of icon on the top and the right side. These can be moved based on your preference. The top tool bar is referred as the action tool bar. This contain the open, execute a script and zooming options.




Info
Show
Group
Move
Mirror
Rotate
Copy
Paste
Delete
Change
Design Block
Add
Module
Port
Net
Port
Junction
Label
Name
Value
Reposition Attributes
Replace
Edit 3D
Attribute
Split
Miter
Slice
Paintroller
Pattern
Pinswap
Gateswap
Optimize
Invoke
Line
Text
Arc
Polygon
Circle
Rectangle
Mark
Dimension
ERC
Errors

The side tool bar has the rest of the command that will be necessary to build your new design. At this stage will be using the ADD command to place parts on the schematic.



Pushing the Board Design to Fusion 360

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

In an EAGLE board with no link to a Fusion 360 design, you will see the option to create a link to the Fusion design or create a new Fusion design. This is shown to the left on the next page. In a board that is already linked to a Fusion design, you will if the board has been changed in either Fusion 360 or EAGLE since the last push to Fusion.

When working in EAGLE, you will notice that the color of the Fusion Sync button changes. A RED button means that the design you are working with in EAGLE has been altered in Fusion 360, and a GREEN button means that you have made changes to your PCB in EAGLE that need to be pushed to Fusion.

Commented [JY1]: Add Screenshots

Commented [JY2]: Is this accurate for when starting in EAGLE and pushing directly to Fusion

Parametrically designing the enclosure around a PCB

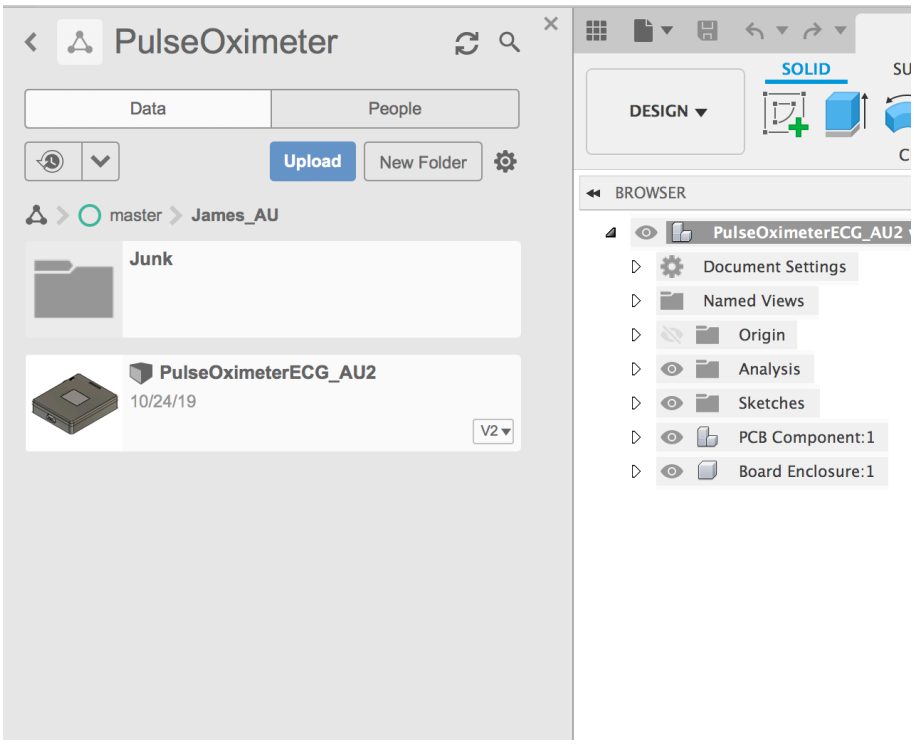
With the emergence of additive manufacturing and 3D Printing, the process for physically creating housing for PCB boards have become much easier. Now – you can very easily bring in a PCB board into any CAD program and design an enclosure to print same day.

But what makes this integration between Fusion 360 and EAGLE so helpful? It is the ability to link these two powerful softwares together. It is rare that anyone gets their design correct the first time – whether that be the electrical engineer designing the board, or the designer/mechanical engineer creating the enclosure. In the real world – this is an iterative process. Having to send new boards over with each version, and creating new enclosures everytime.

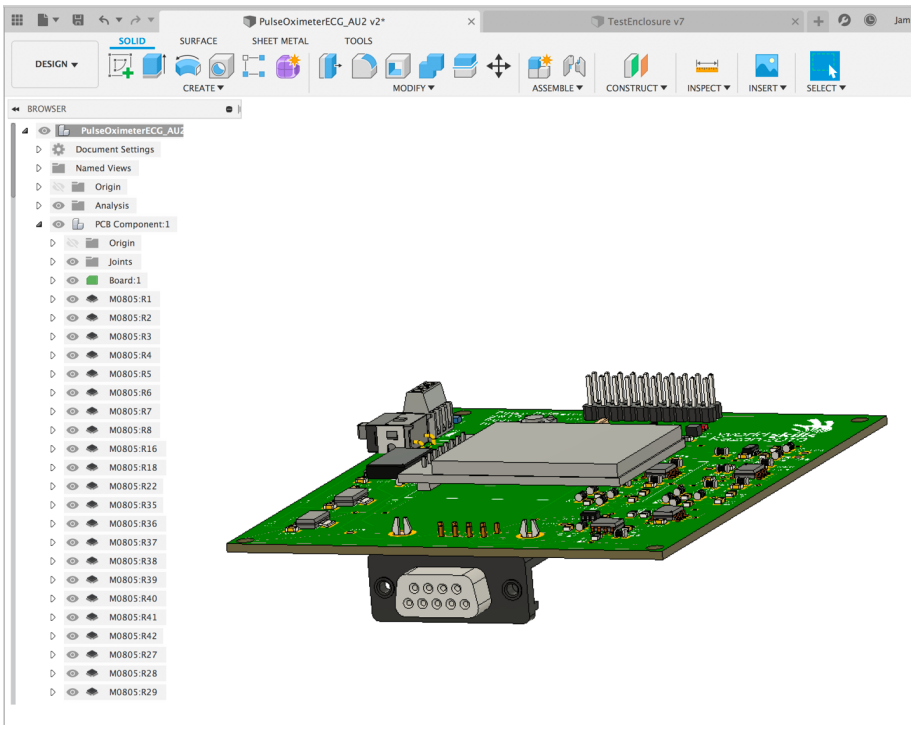
With Fusion 360 linking to EAGLE via the cloud, this is no longer necessary. Using procedures similar to what is going to be presented in this class – you can parametrically design the enclosure around the board. What does this mean? It means that if you create a slot in the enclosure for the board, and the plug location on the board moves, the enclosure slot will update with the new positioning. It means no longer having to recreate the entire enclosure when a board side change happens.

Creating the Enclosure

At this point, the board should be pushed into Fusion 360 as a native Fusion design, and linked to EAGLE. For this design, we will start by opening this file. You can, however, use this file in an assembly – but for this exercise we will be directly creating the enclosure in the same file.

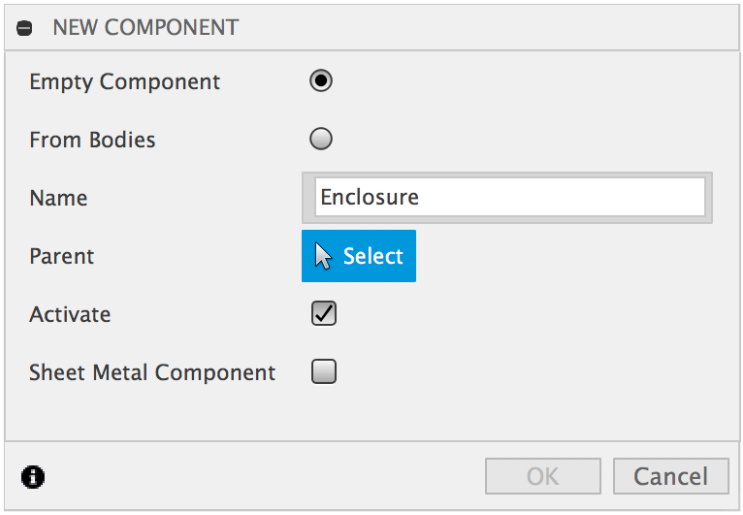


Once the file is opened, you will notice that the Board will be its own component, complete will all of the Library components built within EAGLE.



To start designing the enclosure – it is important to note Fusion 360's R.U.LE. #1, which is to create an empty component, activate it, and start building the enclosure in the empty component. This will ensure organized work – and allow for accurate file management.

To create a new blank component, in the Design workspace, under the Create menu, select New Component. In this dialog, you can name the new component, select a Parent component, denote if it is a sheetmetal component, and choose to activate the component.



For our pulse oximeter, we will to create an empty component, with no parent component. Since we will be 3D printing the enclosure – no need to tick Sheet Metal. We do want to Activate the component however, meaning we are immediately going to start designing this component after hitting OK.

After hitting OK to create our new component – you will notice the PCB is now opaque. This is done automatically when a different component is activated in order to visualize this new component easier.

There are a thousand different ways to build this enclosure – but for our purposes we need to keep a few key pointers in the back of our brain:

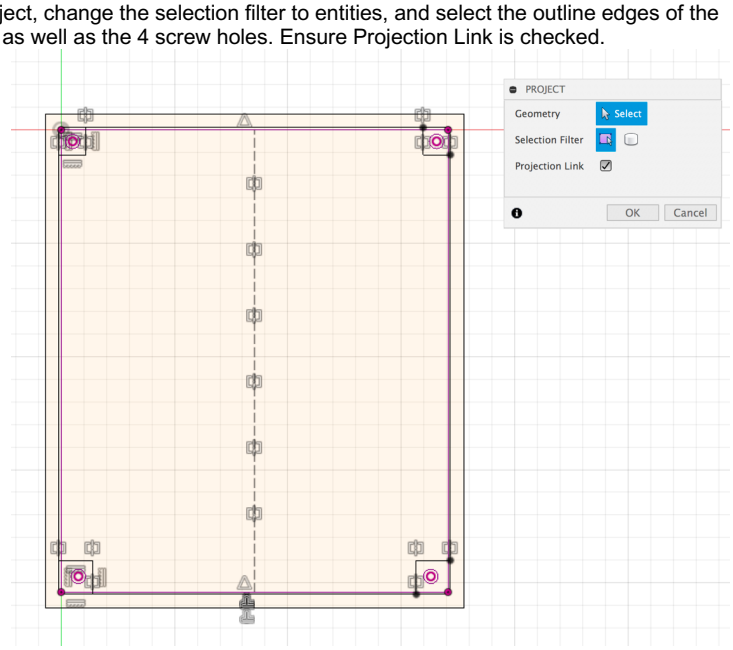
Commented [JY3]: Makee sense?

1. This is a table top tool, so it needs a flat bottom and the ability to be read and picked up easily.
2. It is going to be 3D Printed, and the PCB has to be seated and screwed to the printed board.
3. There has to be clearance for the board to fit into the enclosure so that any external board components can easily fit (ON/OFF Switch, LED Screen,
4. Has to be parametrically linked to certain board components that could move – such as screw holes, plug location, screen location, ON/OFF switch location.

Keeping those pointers in mind, I thought it would be easier to start from the underside of the board and build down. I'm doing this because I know I have to design the enclosure to be screwed in at these points. By having my starting sketch be referenced to the underside of the board – I can link the hole locations to my board and maintain a projection link. This means that

if the board is updated in EAGLE to move those hole locations, the enclosure will automatically update where the screw housing is.

To start this process – we will begin by:

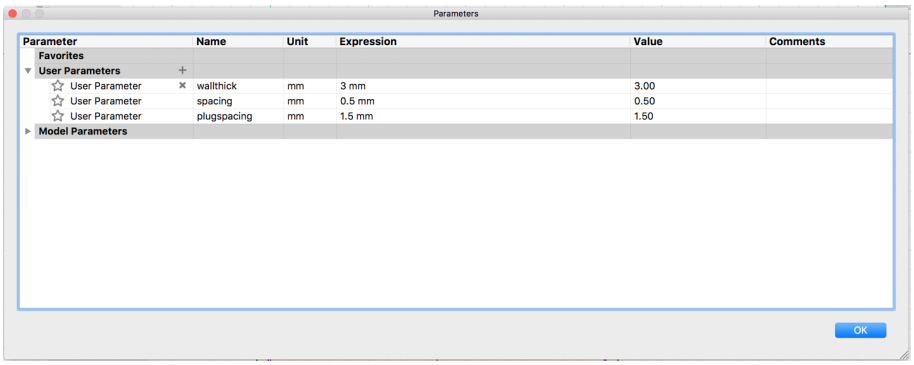
1. Left-clicking the underside of the board once to highlight the face.
2. Right-click to bring up the marking menu and select Create Sketch. Alternatively Create Sketch can be found in the Create menu
3. This will bring up the sketch menu. Under Create, we will need to select Project
    a. Project the body silhouettes, edges, work geometry, and sketch curves into the active sketch.
    b. This will allow us to maintain a link to the board outline and shapes – that way if it changes in EAGLE it will update the Fusion model that we are basing our enclosure off of.
4. In Project, change the selection filter to entities, and select the outline edges of the board as well as the 4 screw holes. Ensure Projection Link is checked.
    a. 

The basic board outline is now projected and linked onto our active sketch. It is now time to tie some parameters into the design. These user defined parameters allow us to make adjustments on a variable basis. A great example, which I will use on this enclosure is creating a parameter for both the spacing (which will be used as a tolerance value in-between the wall and the board, to adjust for printing tolerancing) and the wall thickness (which will be used to define the wall thickness of the enclosure). These are 2 parameters that may have to change later on. Defining

them as parameters now, will allow me to change these values as time goes on to adjust, and will update my model accordingly.

To set up User Parameters:

1. In the Modify menu select Change Parameters
2. Select the + Sign next to User Parameters
3. Create 2 new user parameters
    a. Name=wallthick, value=3mm
    b. Name=spacing, value=0.5mm
4. Hit Ok



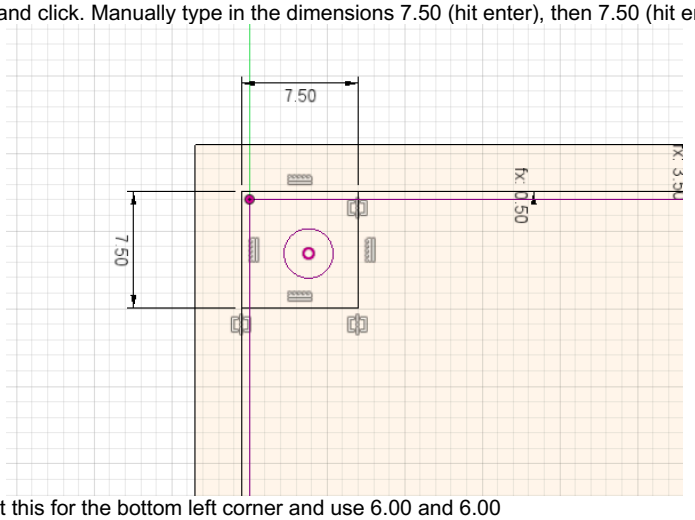
Now that the user parameters are defined, we can start to sketch out the enclosure silhouette, adjust for the 3D Printing tolerance, that we defined with the “spacing” parameter.

To do this:

1. From the Sketch->Modify Menu, select Offset
2. With the chain option ticked, select the projected edges of the board
3. For “Offset Position”, type in the parameter Spacing and hit okay
    a. Note that the dimension is represented by “fx:.50 mm” to represent the spacing variable. If the Parameter is updated in the Change Parameters table, this value will automatically adjust
4. Right-click in space, and hit “Repeat Offset”. For Offset Position enter the value of (spacing+wallthick)
    a. Currently, you can only offset the original line, not an already offset line. Because of this, we need to use an equation to represent the spacing added to the wall thickness parameter.

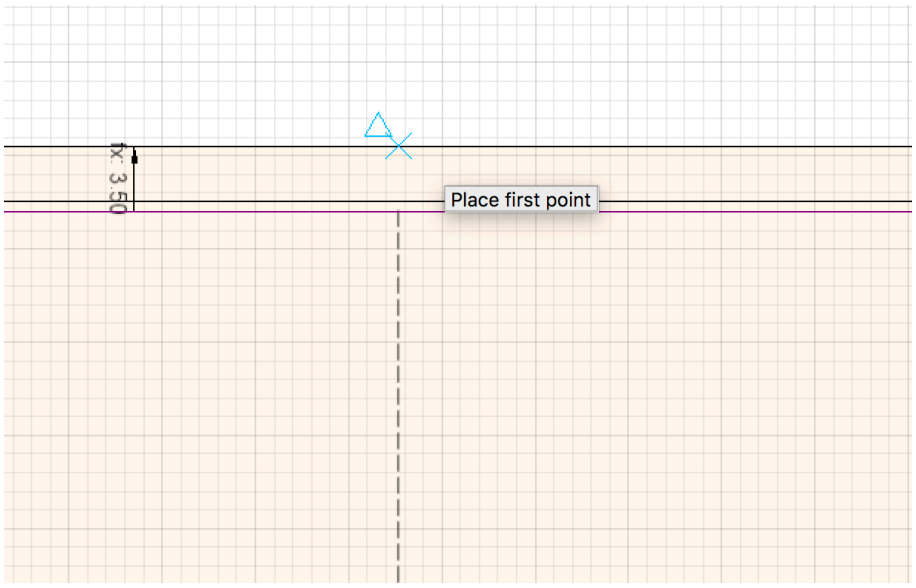
Now that the enclosure outline is defined, we need to think about how to create where the board is fastened to the enclosure via screws. Since this will be made via 3D printing, there has to be a solid structure to screw into, or else the torque of the screw will snap the enclosure. To combat this, we can simply create a rectangle in our original sketch – snapping to the enclosure edge, and fully encompassing the screw hole. This way, when it is extruded it will have a decent thickness to resist that torque.

To do this:

1. While still editing the enclosure base sketch, select 2 Point Rectangle from the Create menu
    a. Notice that the left side holes are symmetric to the right side. However the top holes are not symmetric in position to the bottom. What we will do is model the left side rectangles, and mirror to the right.
2. For the first point of the rectangle hover over the upper left edge of the projected pcb edge, and click. Manually type in the dimensions 7.50 (hit enter), then 7.50 (hit enter).
    a. 

3. Repeat this for the bottom left corner and use 6.00 and 6.00

From these two, we can mirror these to the other side. The benefit of mirror – is that if we update these two dimensions, or sizes – the other side will automatically update. First, we need to create a mirror construction line. Select Line from the Create menu while editing the sketch, and hover towards the mid point of the horizontal board edge. When you are close, hold Shift and it will automatically snap to the mid-point, denoted by an X with a triangle:



Once the line is created, we can highlight it in the Sketch Palette and toggle to Construction Line. Construction lines are used as reference edges – as opposed to normal lines which denote profiles for features.

From the Sketch Create menu, select mirror. For the mirror line, select the construction line. For the objects to be mirrored, select the two rectangles created. Hit Ok. Now all 4 screw holes have been built into the enclosure.

Updating the board in EAGLE

Validating the Results

Still working on generating results. Will post updated PDF.