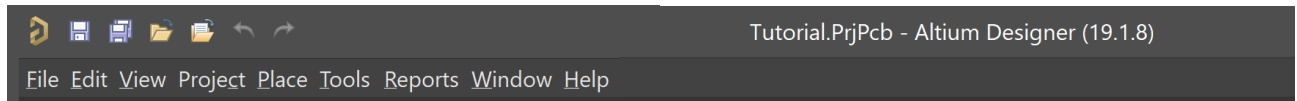

BEGINNER'S ALTIUM TUTORIAL 1 V19.1.8

JESI MIRANDA-SANTOS

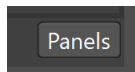
Quick Panel Overview

Top of Window:

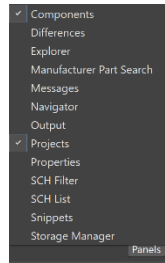
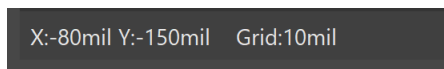
Current Project and Altium Version:



Bottom of Window (right):

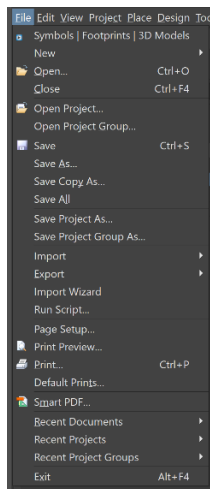


Bottom Left (grid and cursor position):

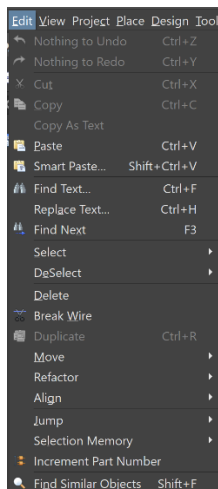


NOTE: When opening the panels components, you can move the windows to the side of the screen to pin them, when changing source documents (schematic, pcb, library, etc), you will see other panels

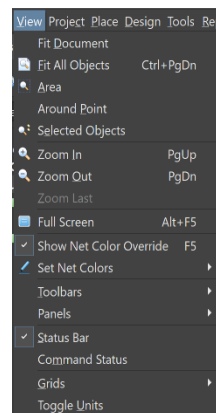
File Tab



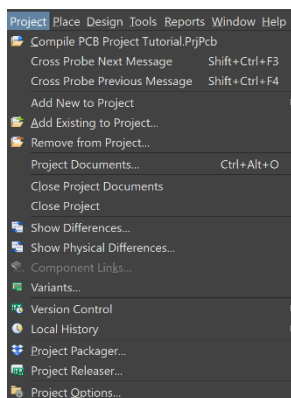
Edit Tab



View Tab

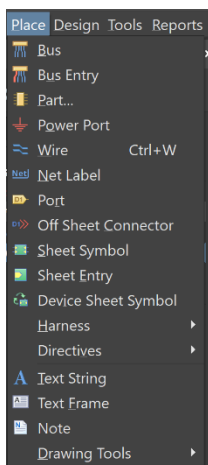


Project Tab

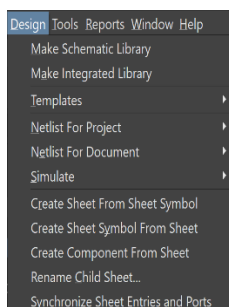


NOTE: These tabs will be different when you change from Schematic to PCB and to other folders

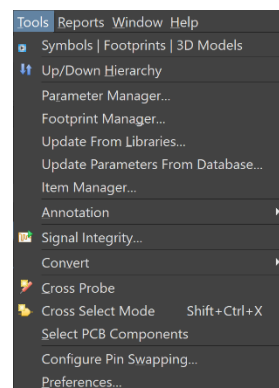
Place Tab



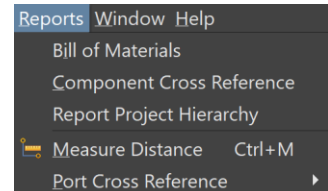
Design Tab



Tools Tab

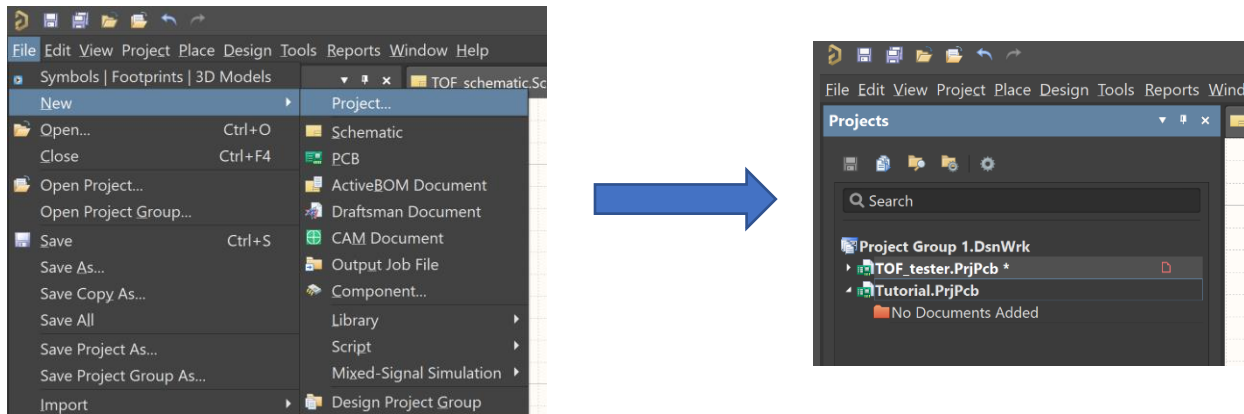


Reports Tab



Getting Started

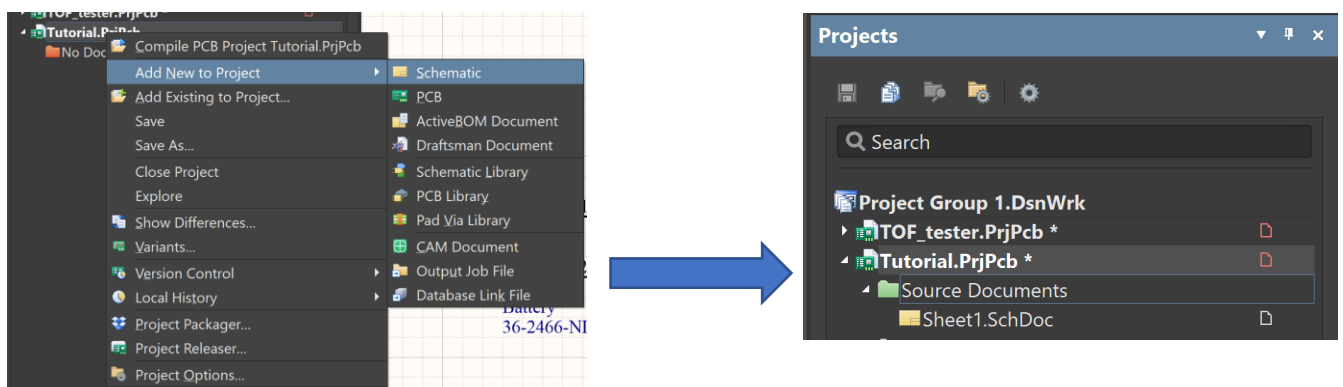
1) Create a new Project (just change the name) by doing the following:



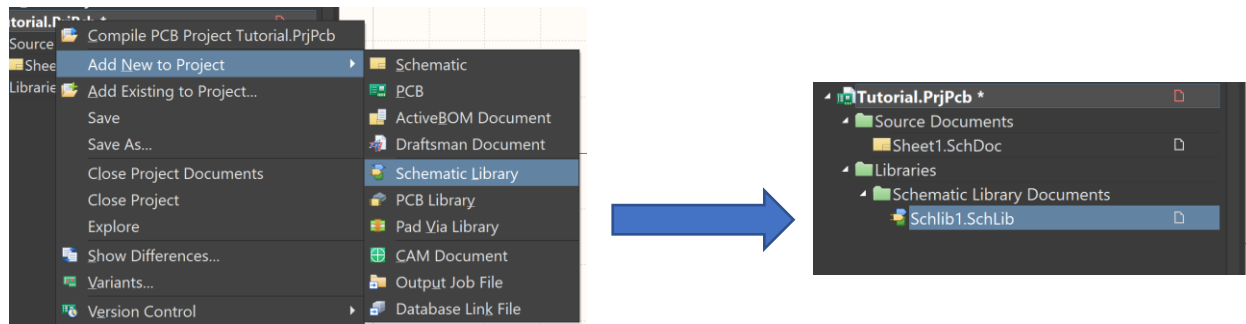
NOTE: If you have another project opened, this will be added to the “GROUP”

Select “Design Project Group” for a new “Group”.

2) Create a Schematic document



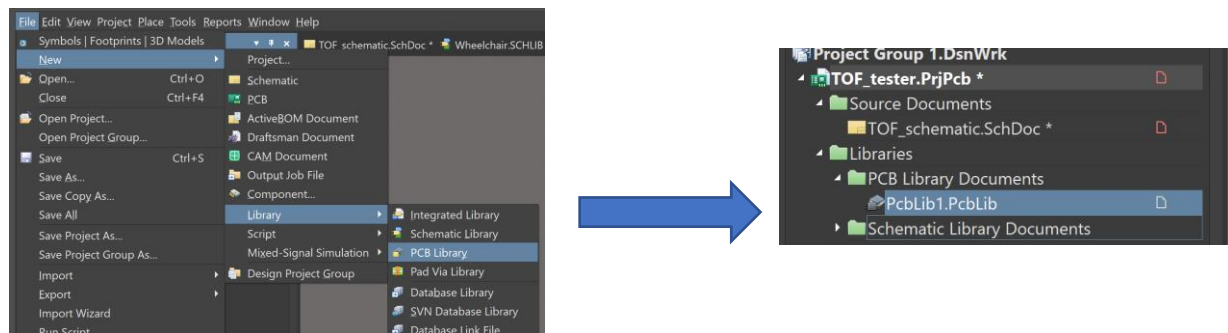
3) Create a Schematic Library



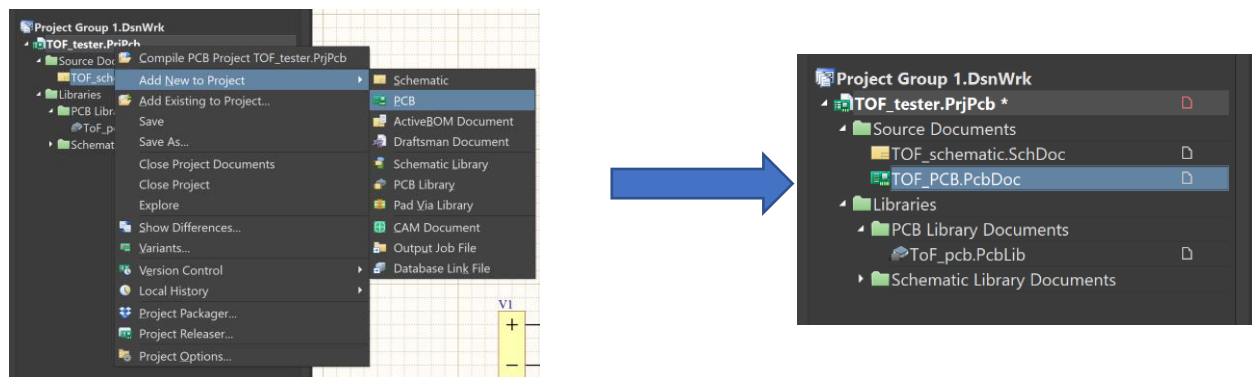
Note: You will be going back and forth on mostly all the tabs. Think of it as a change in one panel will propagate the change in the other panels and you need to **check if the change propagated properly**.

4) Create a PCB Library by repeating previous step

a. If adding from “New” tab, do the following:

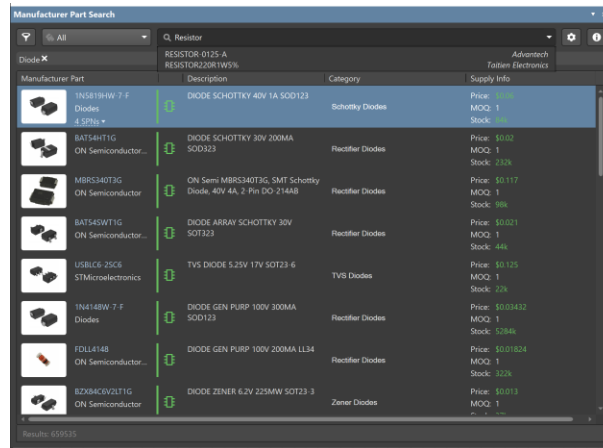


5) Create a PCB Document



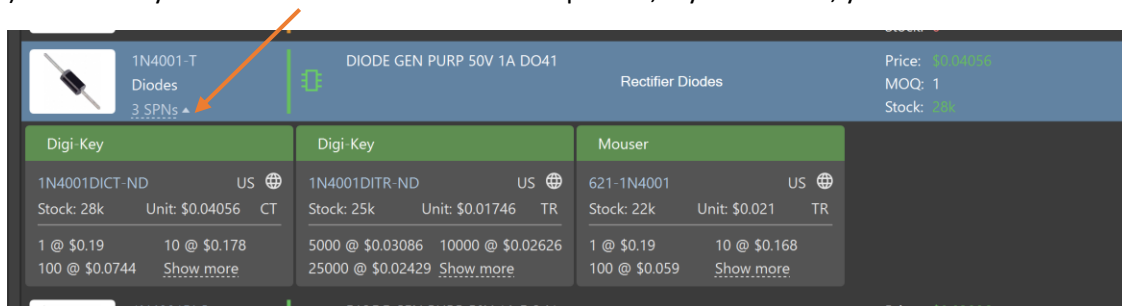
Adding Manufacture Information

- 1) For ease, click on the Schematic Library, select “Add” in the “SCH Library” tab for your part
- 2) Select “Panel” → “Manufacturer Part Search”

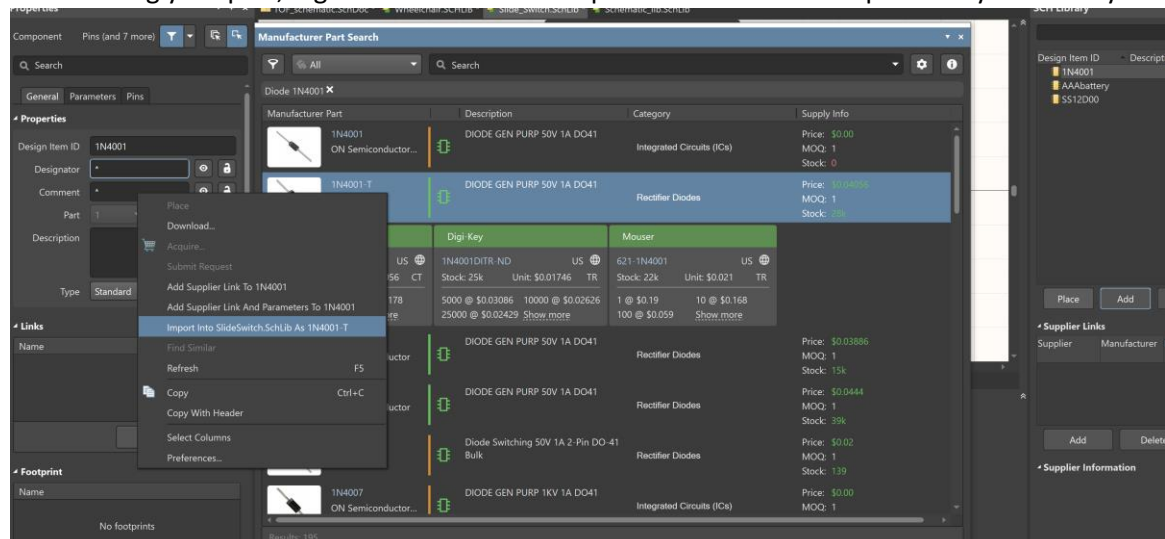


NOTE: Check the “Supply Info”, “Description” and “Manufacturer Part” for the correct component

- 3) Some may have an arrow on the side of the picture, if you select it, you find more info:



- 4) After finding your part, Right-click and select “Import Into...” to add the part into your Library



5) You should see this now:

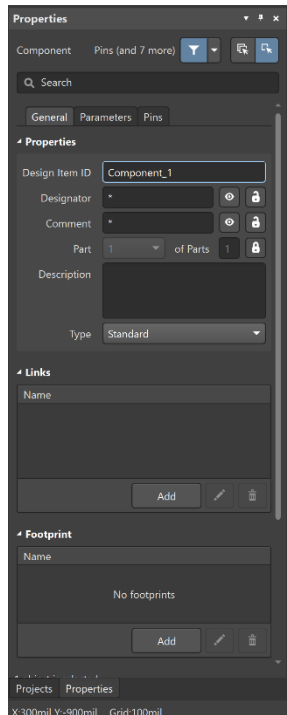


IMPORTANT: If you already have your schematic made, you can select “Add Supplier Link and Parameters ...” to update the information

NOTE: You can skip this part and move on to “Schematic Component Representation” but don’t forget to update information with Digikey info and parameters. This is very important especially if you are going to send out to the manufacturer!! **KEY NOTE:** If you can’t find it, either make it and find a supplier for the part, or just choose a different component if possible

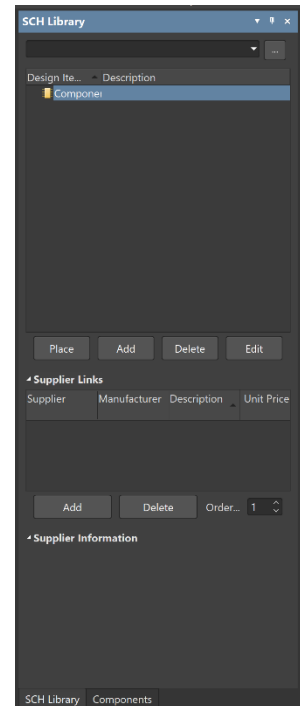
Creating a Schematic Component Symbol

- 1) When created the “Schematic Library”, you were taken to the window. On “Panel” open the “SCH Library” if it’s not open.



Double clicking on “Component” will open the “Property Panel”

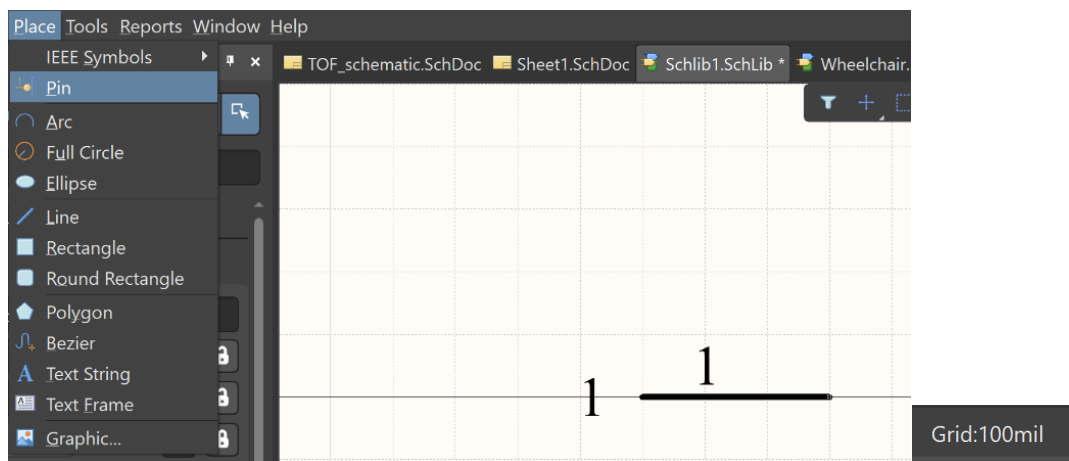
Rename the component and leave everything else untouched IF you have not added the Manufacturer info or could not find it



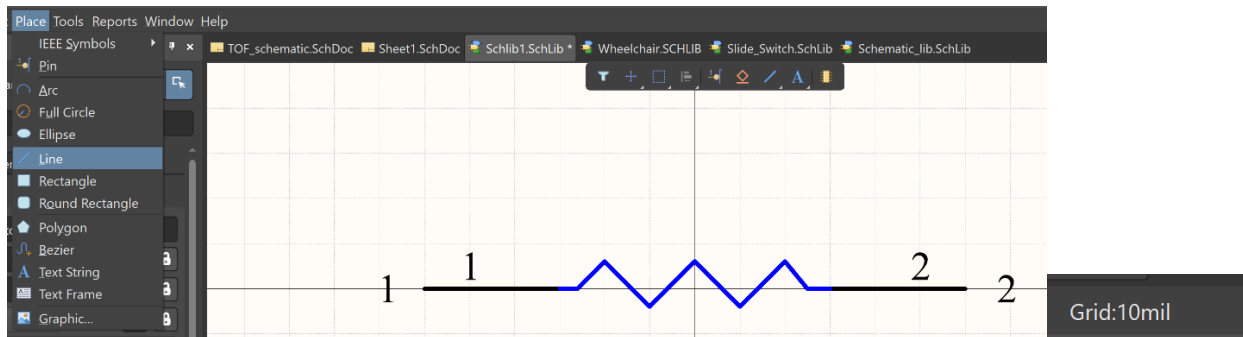
NOTE: You can add parts, this will be included in the “Library File”, so if you were to have other project and import the Library, you will be importing all the components that you created in this file

- 2) Draw component using the “Place Tab”

- Click on the schematic sheet and *press G on keyboard* to change grid until its 10 mil.
- Left click to place; Right click to cancel selection
- If needed, double click and under “Properties” change the “Name” and “Designator” to 1, then 2, etc, depending on the number of pins



- Under “Place Tab” select “Line” and draw a resistor, connecting both pins.
- Make sure to change the Color (not black) and Line Width if needed
 - Press Spacebar to change the path of the line and press G to change grid if needed

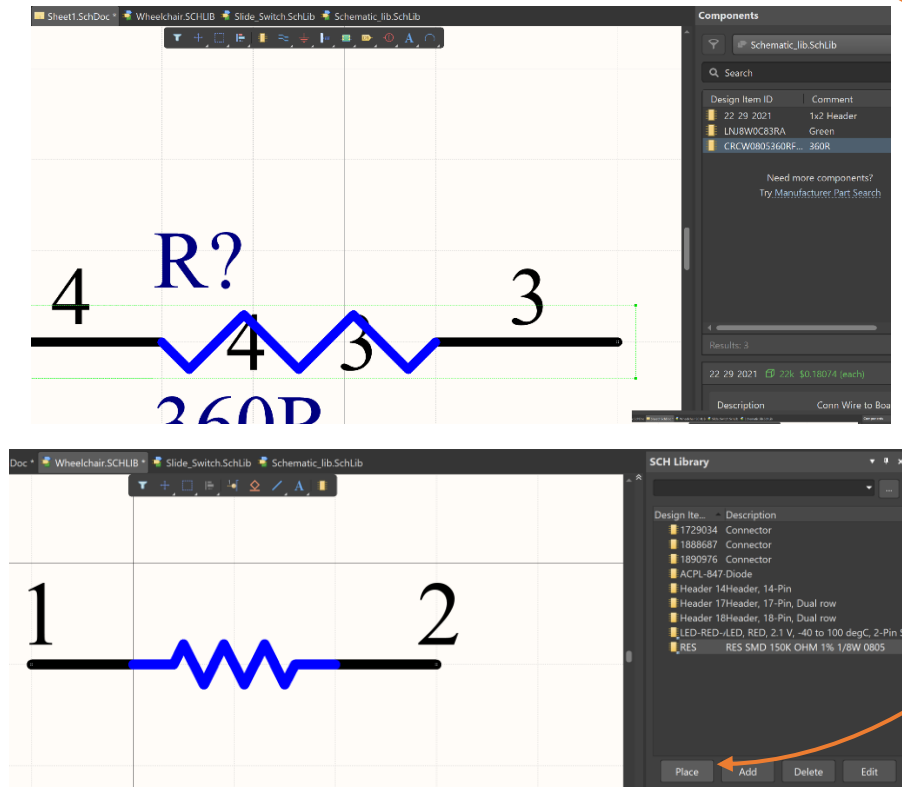


NOTE: Every time you make a change you are satisfied with Ctr+ S to save.

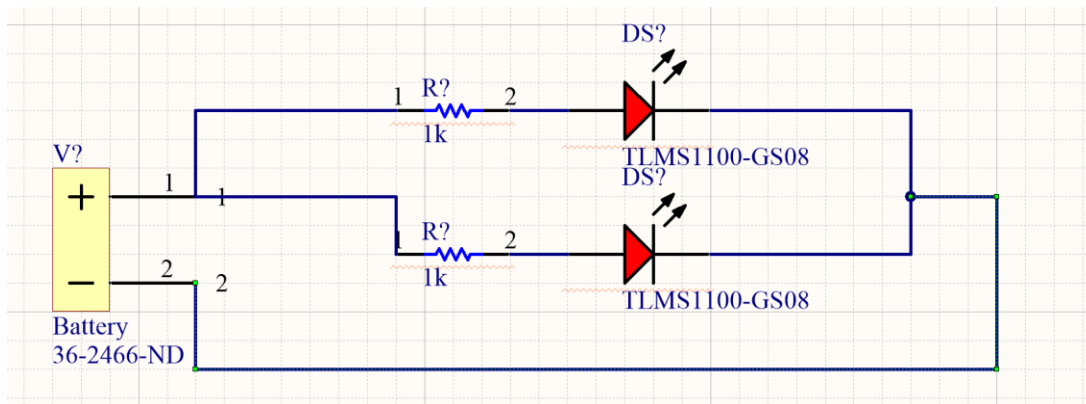
3) Save your Schematic Library.

Making your Circuit Schematic

- 1) Go to your “Schematic Document”, once you have all your schematic parts
- 2) Open “Components” from the “Panel”
 - a. You can also from your Library, open the “SCH Components” tab and select your part
 - b. Include your part in the schematic by clicking “Place”
- 3) Double-click to place part

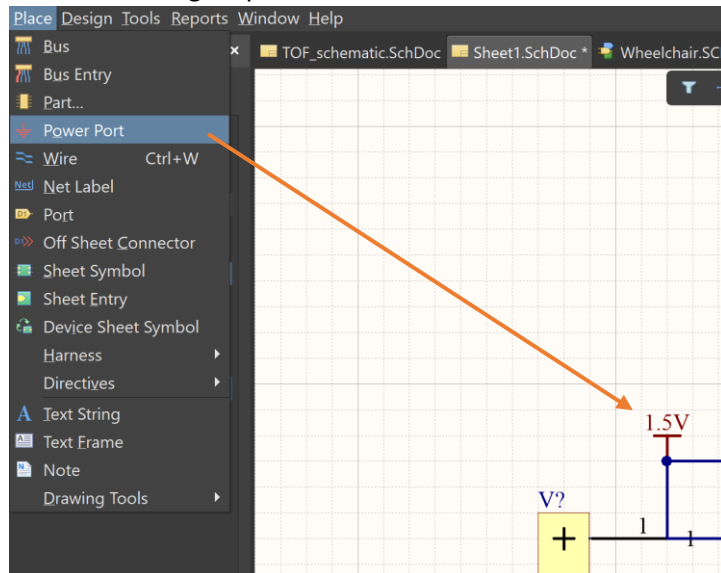


- 4) After Placing parts, select “Place” → “Wire” to connect components
 - a. Right click to cancel function
 - b. Short cut to wire is **Ctrl+W**



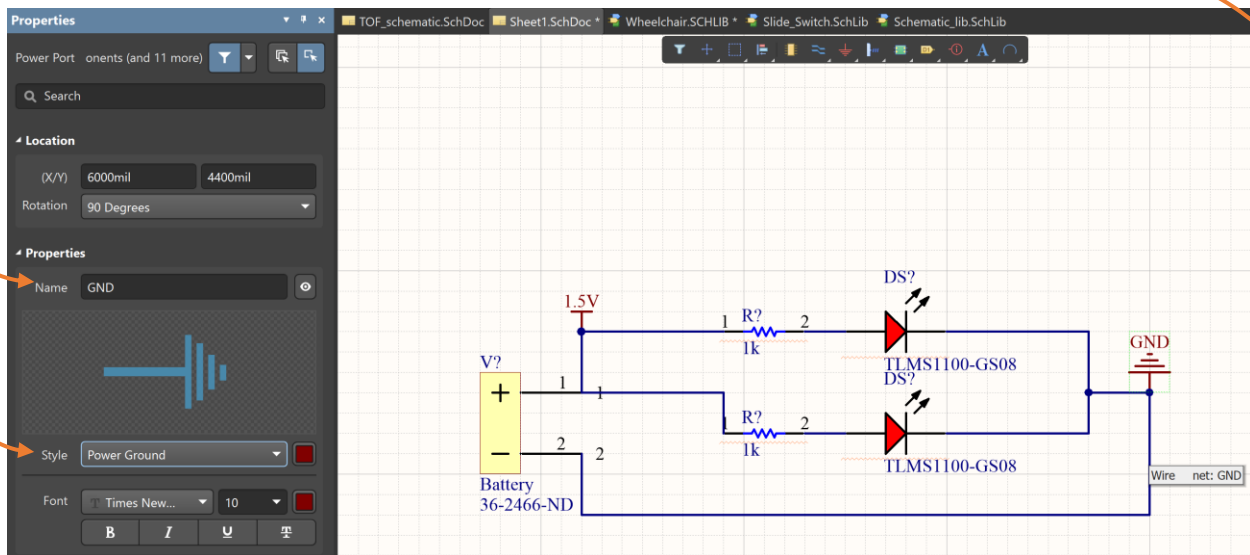
5) Connect Ground and Power input

a. Place the Voltage Input



b. Place the Ground input

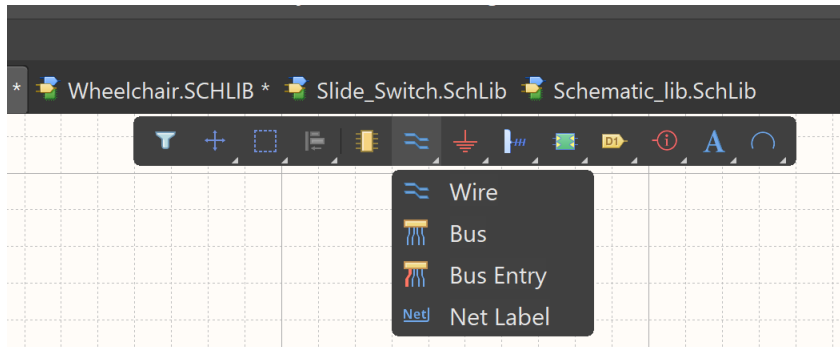
- Double-click power port and change
- Change "Name" to "GND"
- Change "Style" to "Power Ground"
- Hover over wire to make sure your wire ("nets") are connected properly



NOTE: You can start with GND then Voltage Input, you just change the "Style" to "Bar".
You can HIDE, the name, values and labels by clicking the "Eye" symbol

Adding Labels to Placed Wires (Part 1)

- 1) You can “Place” → “Net Label” or do the following to select “Net Label”:

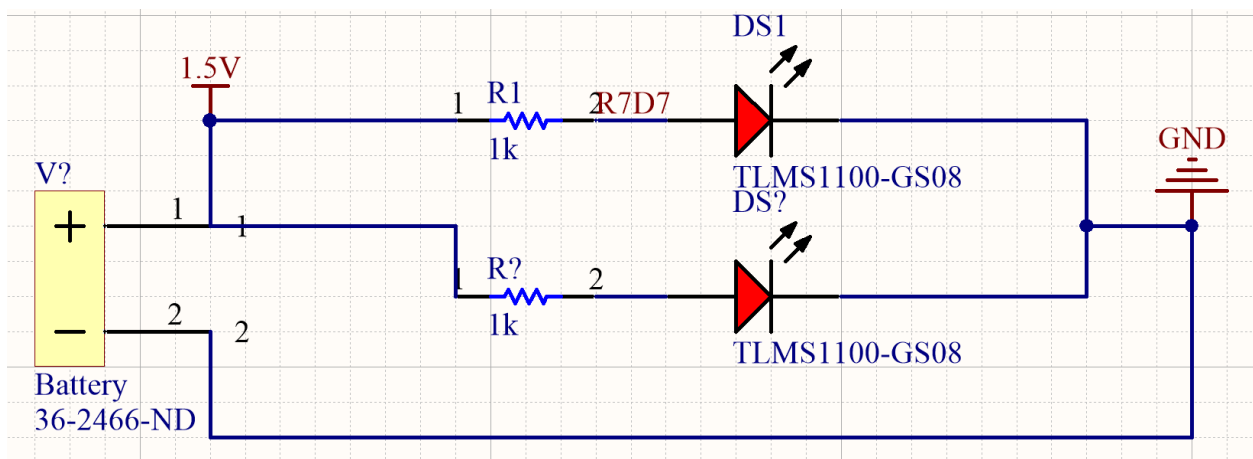


- 2) After selection, hover over a wire and click
 - a. Change the “Net Name” in the “Properties” OR double click to overwrite



NOTE: You can do this for all wires, but there is a better way. For WIRES connecting 2 components try having the name convention [Name_Comp1][#][Name_Comp2][#]

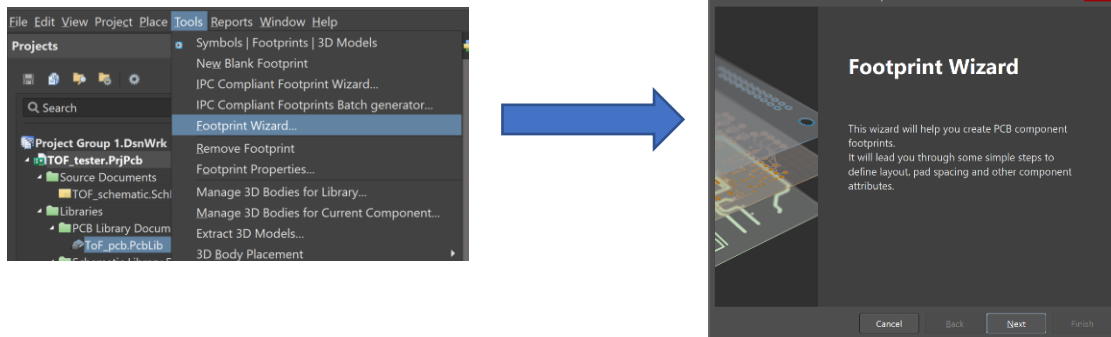
- 3) Rename your components (*if you have repeated components rename with [NAME][#] for only one component type*)
 - a. Double-click component → Change “Designator” with proper name



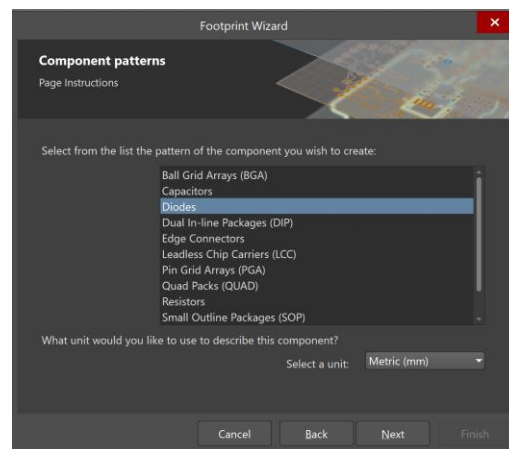
NOTE: We want to add a 3D model to every component, usually before adding more components and annotating the schematic.

Creating Footprints for Components (Part 1)

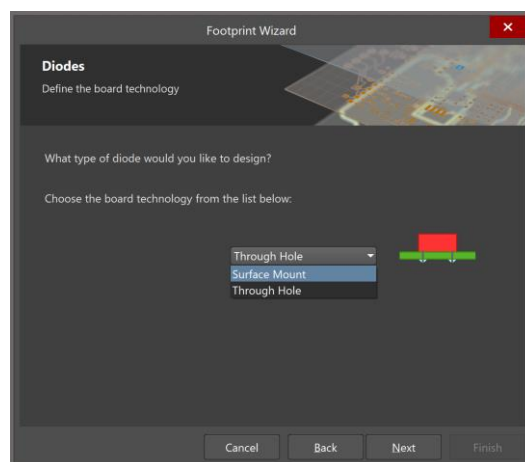
- 1) Go to the “PCB Library Document” you created in the beginning
 - a. Open Property Tab and change the “Units” to “mm”
- 2) “Tools” → “Footprint Wizard”
 - a. You will need datasheet information for your component



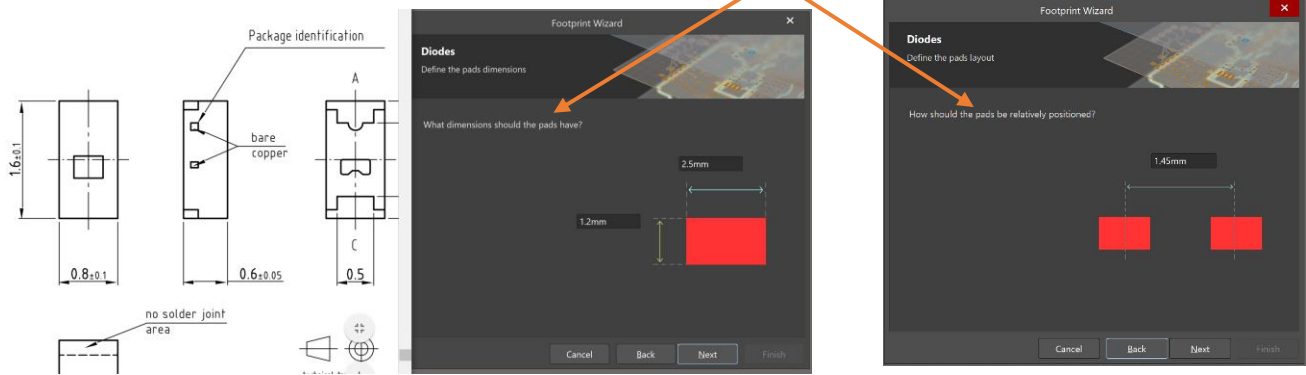
- 3) “Next” → Select type of component and change “Select a unit” to “Metric (mm)”



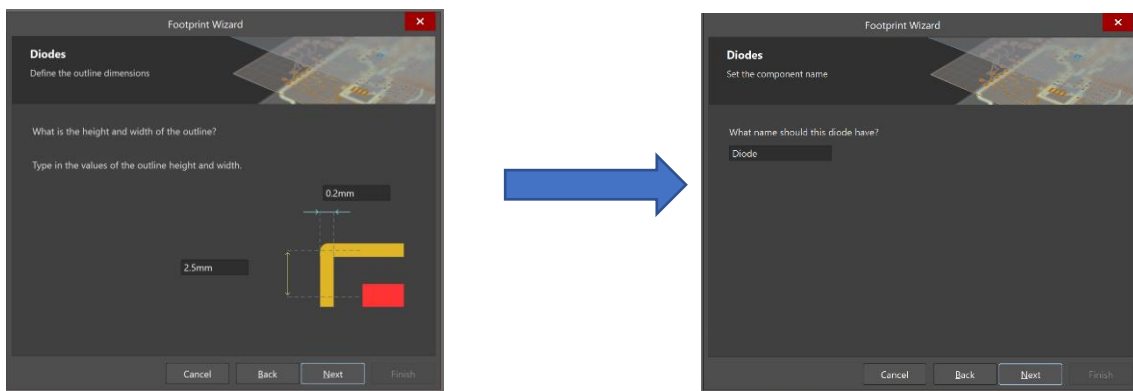
- 4) “Next” → Select the model type: Surface Mount (SMD) or Through Hole (depends on the component you decided to go with)



5) “Next” → Look at DATASHEET for dimensions of the PADs and input them

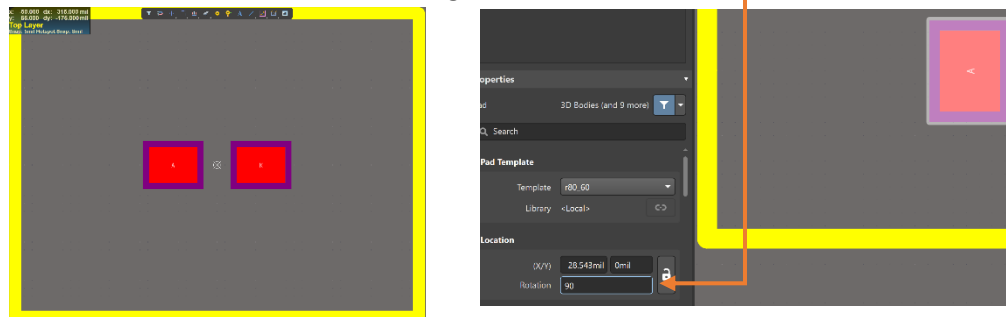


6) “Next” → Leave this part unchanged → “Next” → Change the name if needed



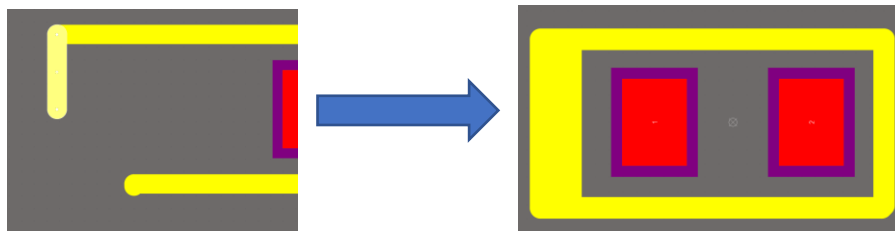
7) “Finish”

- You should see this, now we need to resize the Yellow Lines to fit well
- Rotate the pads if necessary
- Left-click and hold on Yellow Line, drag to resize



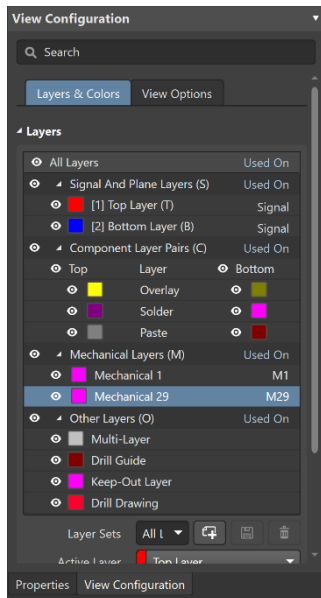
8) For the left Yellow Line, delete one of the lines, and extend the other one, then resize

- Make sure you are in the TOP OVERLAY to place a “Solid Region” to know the cathode of LED



NOTE: To view in 3D click “3” on keyboard and “2” to return to 2D

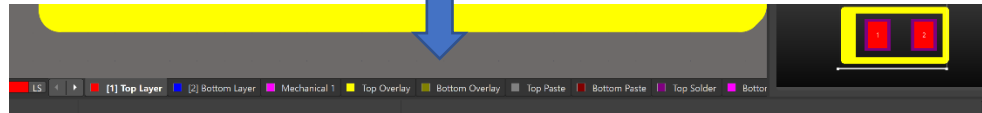
Schematic PCB Layer(s) and Purposes



While in PCB Document → “Panel” → “View Configuration” to see **all** the layers

At the bottom you can see in what layer you are currently in.

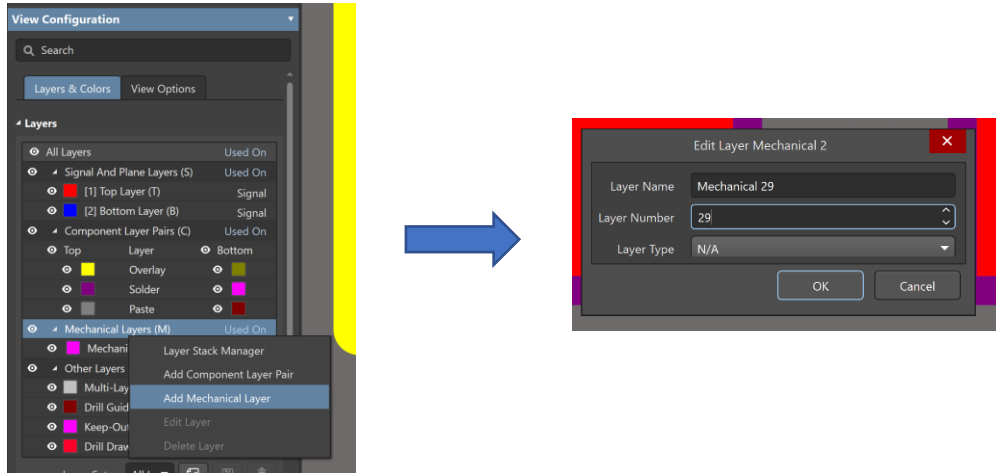
(Here we are in Top Layer)



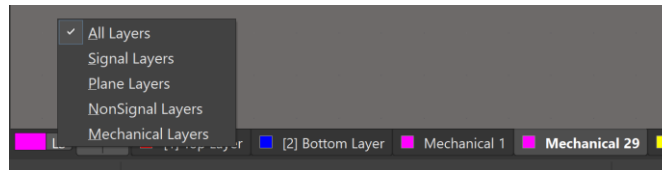
- 1) **Top Layer** – copper layer; pads where you will solder component
- 2) **Mechanical Layer 1** – This is where 3D model will be placed
- 3) **Mechanical Layer 29** – Assembly drawing layer
- 4) **Top Overlay** – White lines outlining the components
- 5) **Top Paste** – Where solder tin is placed on your pad by the machine
- 6) **Top Solder** – Mask Layer; area with no green color

Adding Assembly Drawing to PCB (usually on M29) (Part 2)

- 1) Make sure you are on the PCB Library
- 2) "Panels" → "View Configuration" to see all layers → Add Mechanical Layer 29

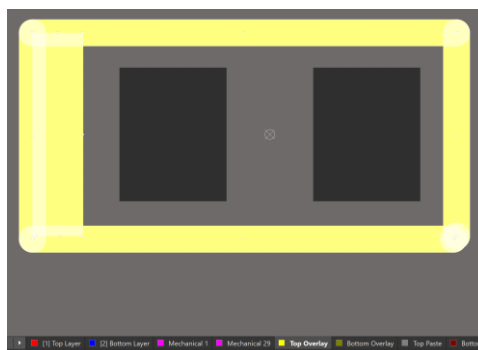


- 3) Navigate to Mechanical 29 (M29) if you are not on it to see that you have it



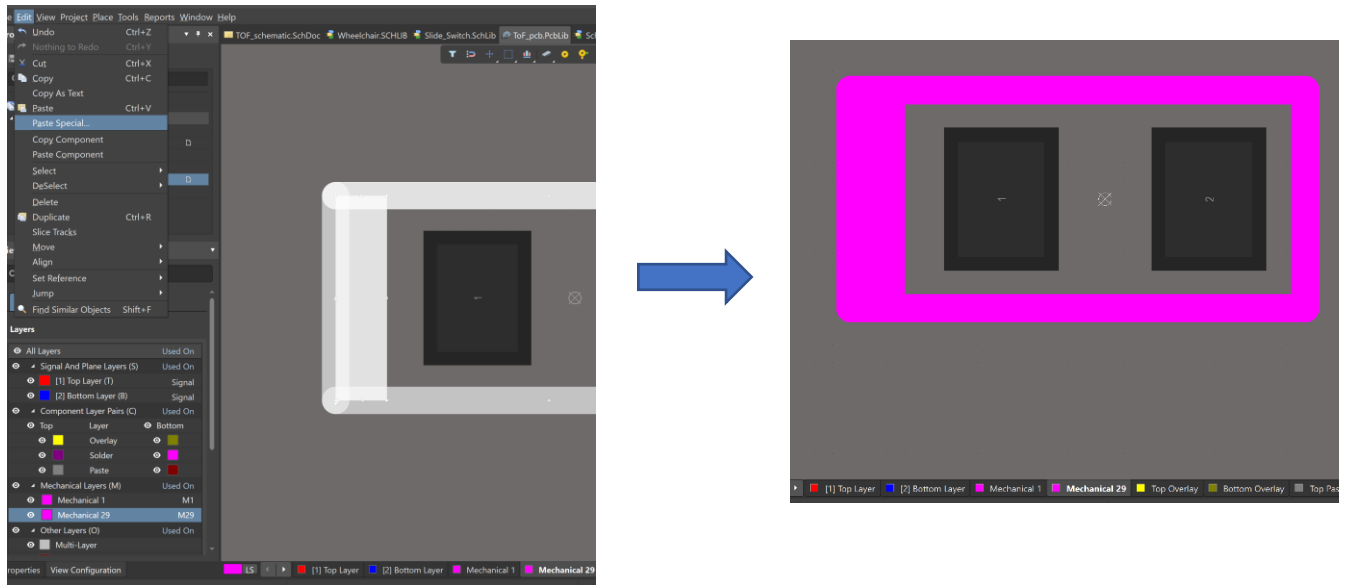
- 4) Navigate to Top Overlay and only copy the Top Overlay
 - a. Use Shift + S for SINGLE LAYER View
 - b. Select all the Yellow Line and copy, THEN click on the ORIGIN

NOTE: After copying, you need to select a reference to grab the copy, which you will see as a GREEN mark



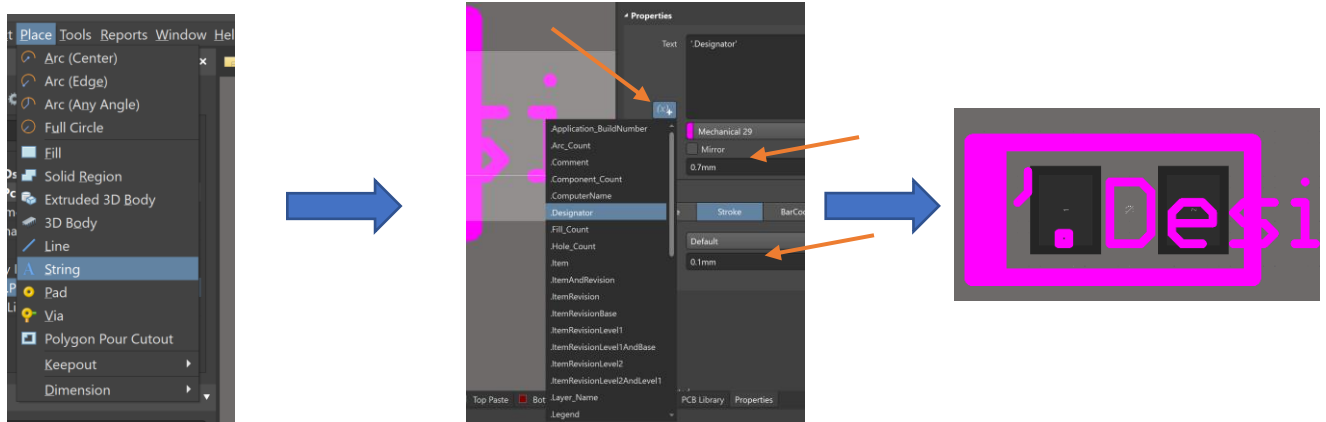
5) Once you copy, go to M29

- “Edit” → “Paste Special”
- Select “Paste on current Layer” and align with origin



6) Now, we add a Designator (its like a variable for this component)

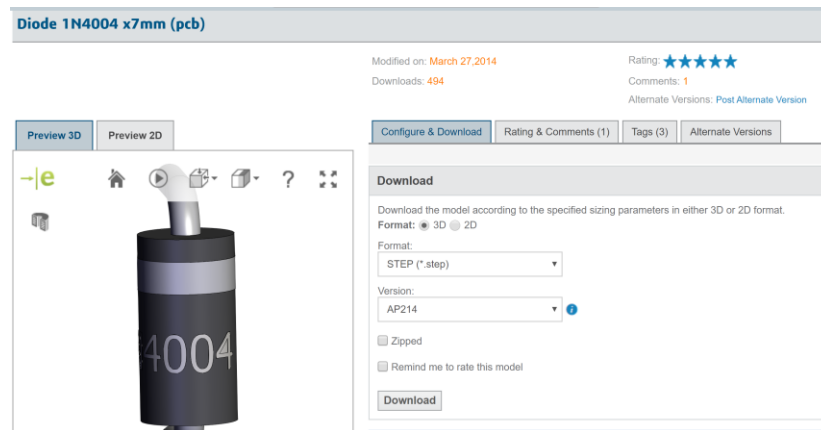
- Select “Designator” in the Properties tab
- Change following: *Text height* = 0.7mm and *Stroke Width* = .1mm
- Place the string and Shift + S, to see it all



7) Save the PCB Library

Adding .step Files for 3D rendering (Part 3)

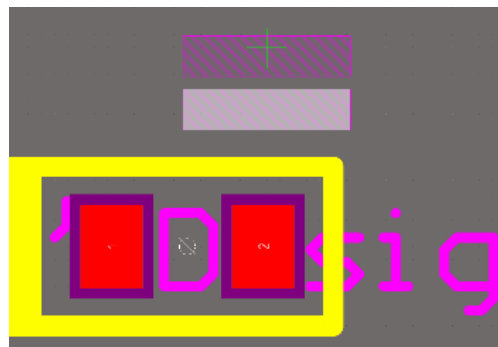
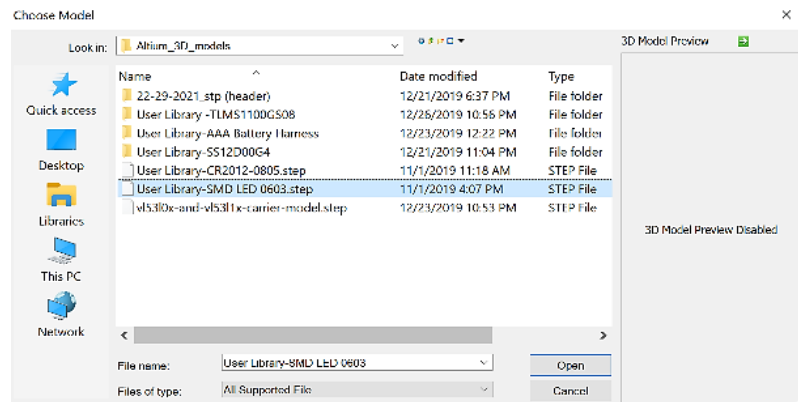
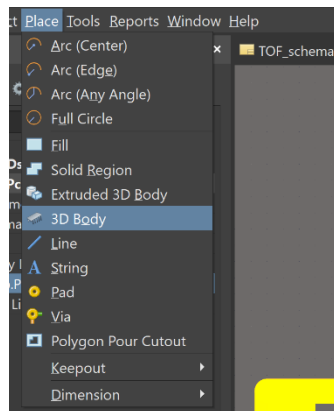
- 1) On PCB Library, you finished the assembly drawing, go to M1
- 2) Go to 3d content central and download the STEP file (search by its name)
 - a. You can use any other site, but make sure you have the right part
 - b. Make an account (its free) and make sure your download features are this:



- c. Save the file in a folder, you may reuse the .step files for other projects

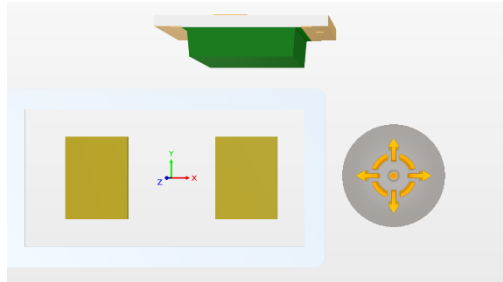
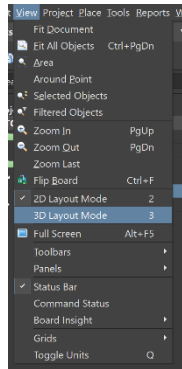
NOTE: Digikey provides EVERYTHING, but its good to only use the 3D part and create the rest

- 3) "Place" → "3D Body" → select the step file → Place file



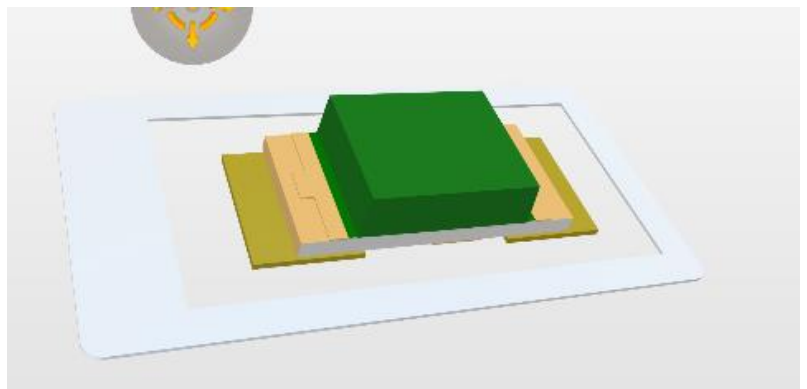
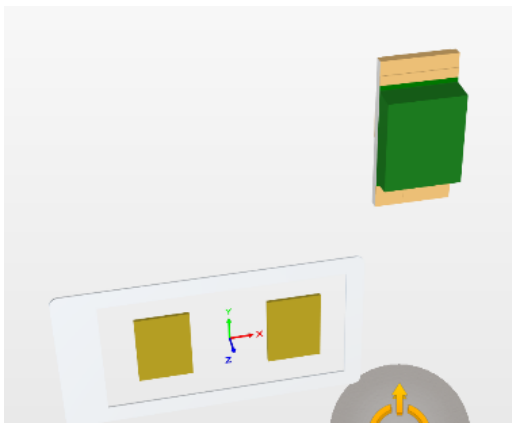
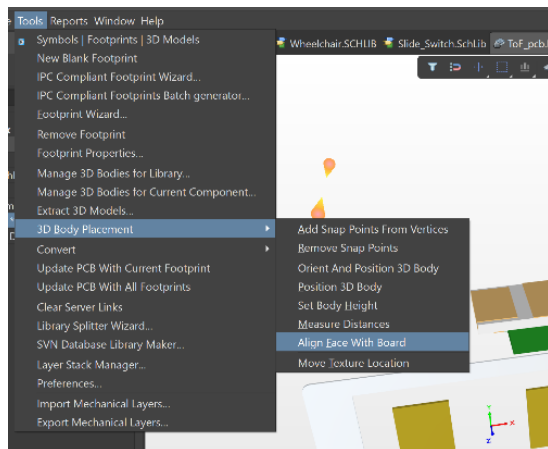
4) Render the 3D view

- You can press number “3” for 3D view and “2” for 2D view
- Now, we just move the component to the proper place



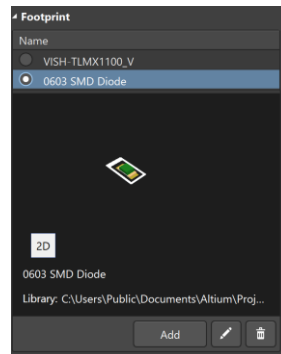
5) Align bottom face with the pads

- “Align Face with Board” → select the bottom face
- Switch to 2D to rotate and align with pads

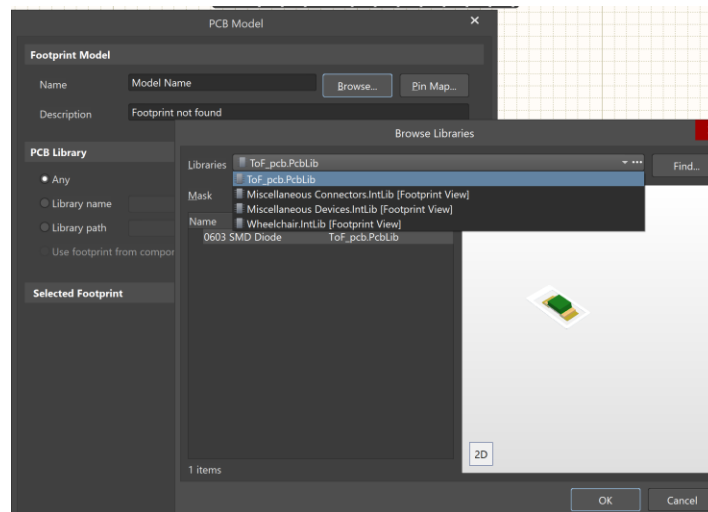


Adding 3D Model to Schematic

- 1) Go back to Circuit Schematic Document
- 2) Double-click on any component from the schematic
 - a. This will open the Properties tab
- 3) Find “Footprint”
 - a. Yours might will be empty



- 4) “Add” → “Browse” → Choose the library with your part → Select your part and select “Ok”



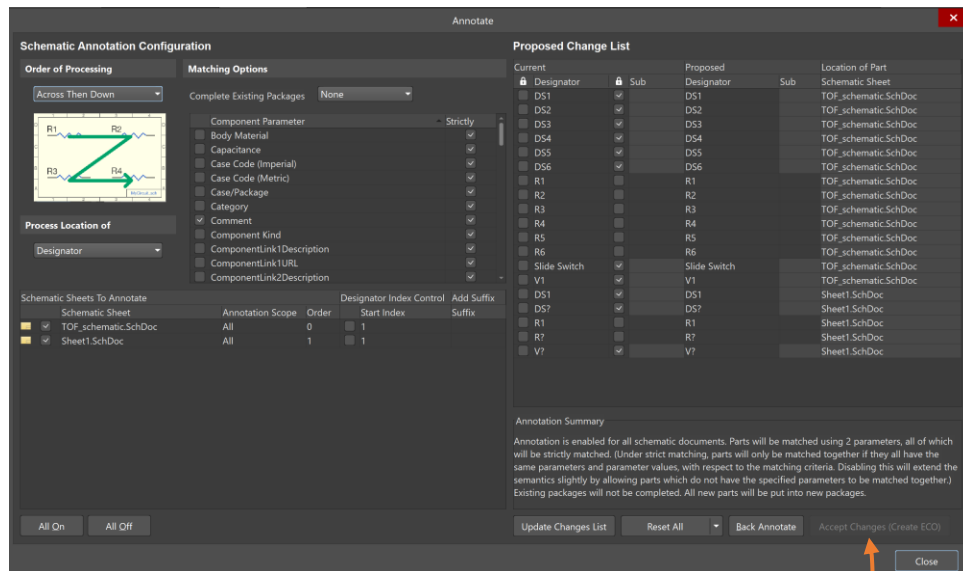
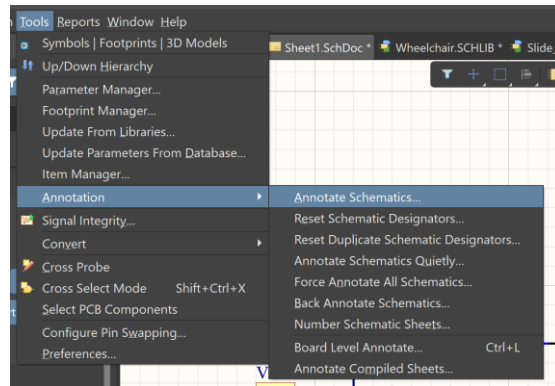
- 5) Stay on the Schematic Document if you need to annotate it.

NOTE: When you select your part, you should see the 3D model, we need to annotate the schematic so we can generate a PCB design with all of our parts connected with reference paths so we can wire them

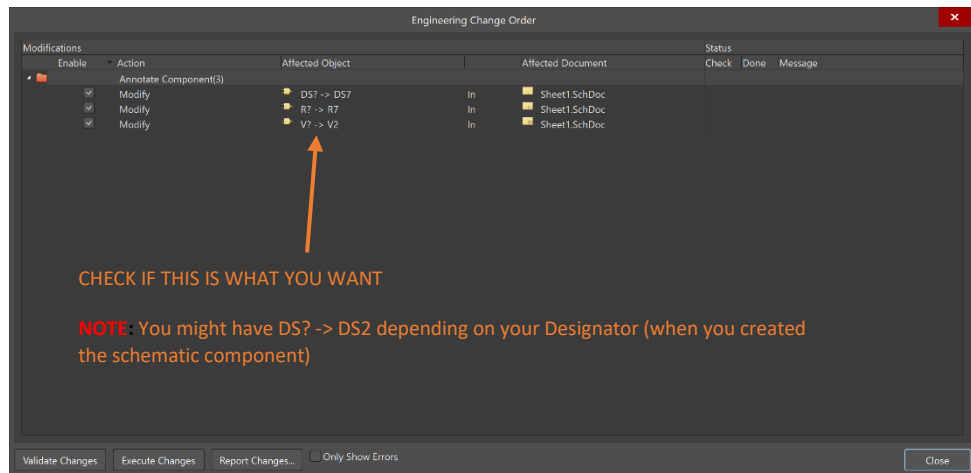
NOTE: Remember to UPDATE & SAVE all documents when you make a change!

Adding Labels to Placed Wires (Part 2)

1) Re-annotating Schematic (“the better way”)



- Select “Update Changes List”, then “ok”
- You will see “Accept Changes (Create ECO)” become available; select this for the following



- c. Select “Validate Changes” if everything is fine. You should see all green under “Check”:

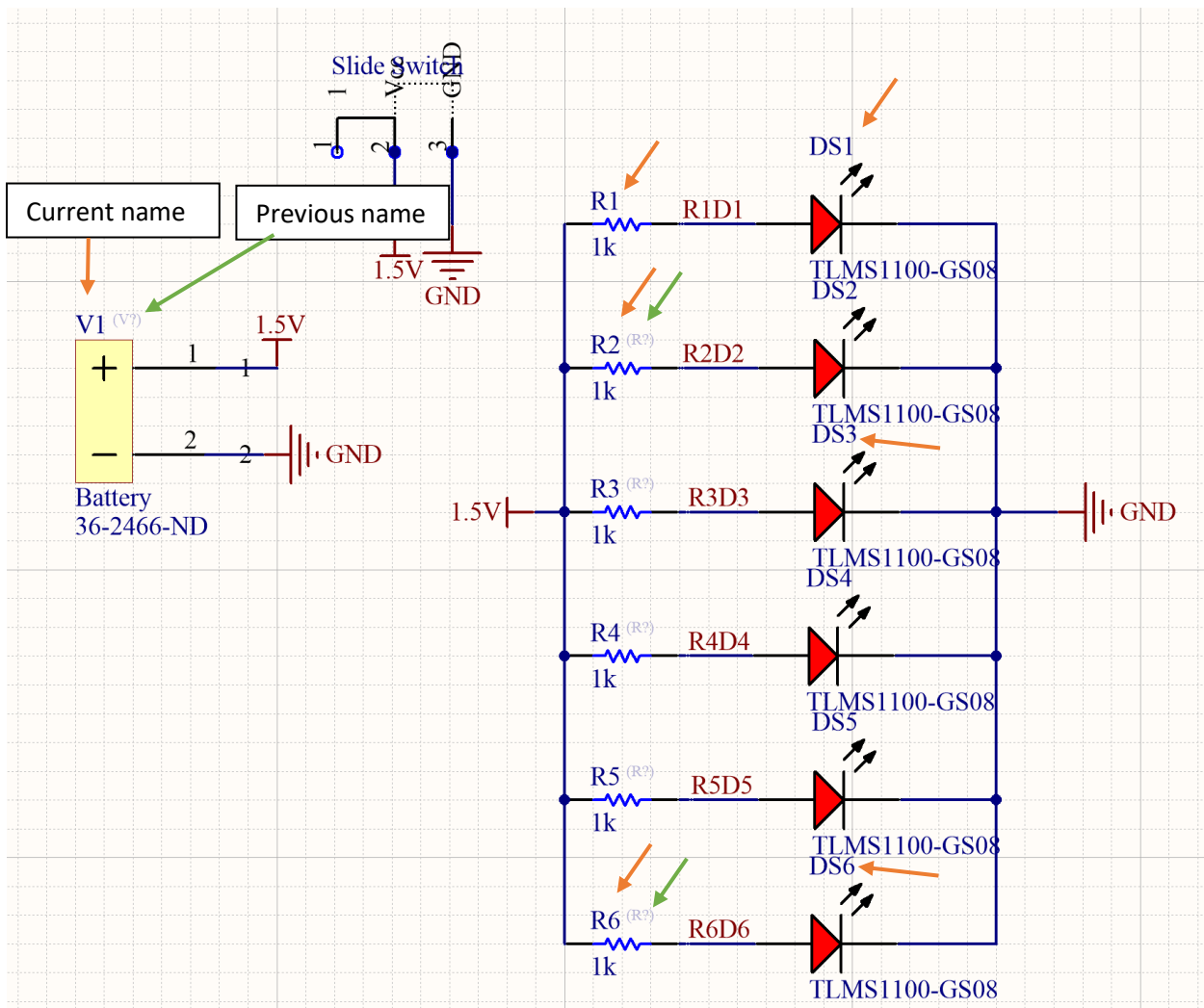
Modifications			Affected Document		Status		
Enable	Action	Affected Object			Check	Done	Message
<input checked="" type="checkbox"/>	Annotate Component(3)						
<input checked="" type="checkbox"/>	Modify	DS? -> DS7	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Modify	R? -> R7	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>		
<input checked="" type="checkbox"/>	Modify	V? -> V2	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>		

NOTE: If you get an error, check that names don't repeat or just change manually

- d. Now, select “Validate Changes” to make them official:

Modifications			Affected Document		Status		
Enable	Action	Affected Object			Check	Done	Message
<input checked="" type="checkbox"/>	Annotate Component(3)						
<input checked="" type="checkbox"/>	Modify	DS? -> DS7	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	Modify	R? -> R7	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
<input checked="" type="checkbox"/>	Modify	V? -> V2	In	Sheet1.SchDoc	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	

- e. Close all the tabs that were opened in the process. Check your schematic to see the changes (this is an example schematic with multiple parts done following the previous steps):



Short Cuts

For more info: <https://www.altium.com/documentation/altium-designer/new-in-altium-designer?version=18.1>

G – Change grid size

Spacebar – Rotate a selected component

Wire – Ctr + W

Fit all object (to screen) – Ctr + PgDn

Single Layer View – Shift + S

Ctrl + Shift + G – Set new grid location of cursor (useful when drawing on Top Overlay)

J+ L – then select “New Location” to move cursor to a specific point