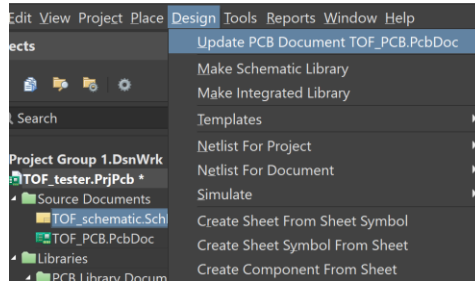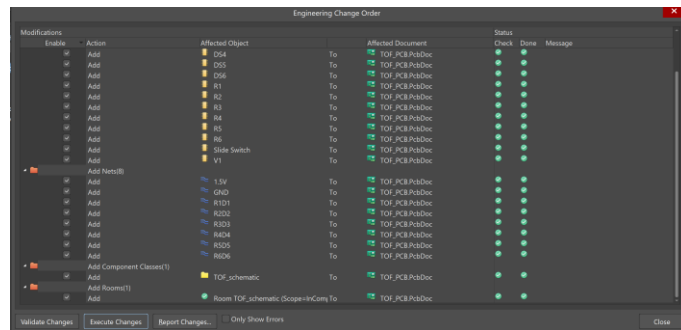# BEGINNER'S
ALTIUM TUTORIAL 2
V19.1.8

JESI MIRANDA-SANTOS
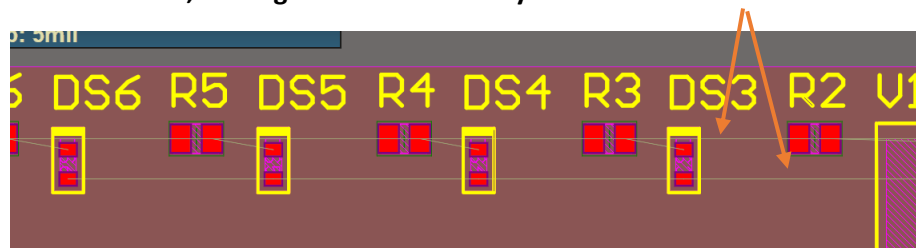
Placing Components on PCB

1) **Make sure you created the "PCB Document"**
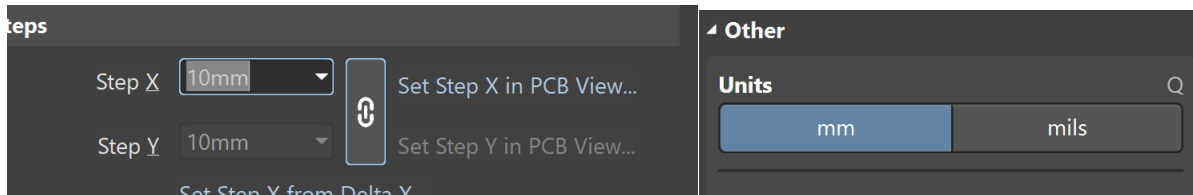2) **Go to the "Schematic Document" and UPDATE the PCB document**

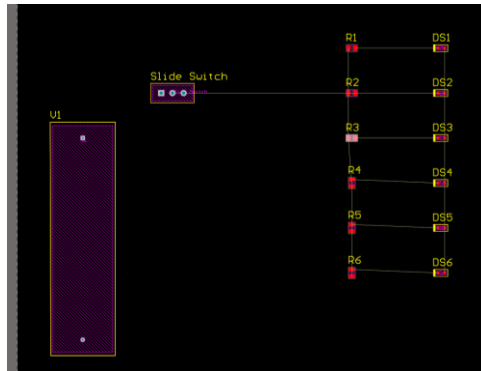3) **Validate and Execute Changes (make sure everything is good!)**

4) **On the PCB Document, Ctr+ PgDn and make sure you can see the NETS!!**

5) **In "Properties" tab, change units to "mm" and Ctr + G → set grid to 10mm**

6) **On Top Overlay, drag all the components inside the PCB**
   a. **Erase the RED room ("background") and rearrange all the components to your desire**
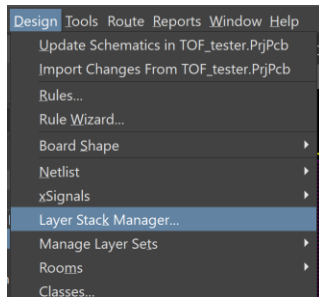

7) **Changing shape of PCB Board**
   a. You need to close your loop that you make, then right-click to exit

NOTE: You can adjust the board before adding the components if you know your dimensions. Also, you can make more shapes other than a rectangle

# Setting Basic Defined Rules

1) **Increase Dielectric Thickness to 0.6**

| #	Name	Material	Type	Thickness	Weight	Dk	Df
	Top Overlay		Overlay				
	Top Solder	Solder Resist	Solder Mask	0.01mm		3.5	
1	Top Layer		Signal	0.036mm	1oz		
	Dielectric 1	FR-4	Dielectric	1.6mm		4.8	
2	Bottom Layer		Signal	0.036mm	1oz		
	Bottom Solder	Solder Resist	Solder Mask	0.01mm		3.5	
	Bottom Overlay		Overlay				

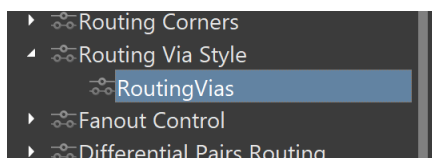2) **Defining Spacing Rules ("Clearance")**
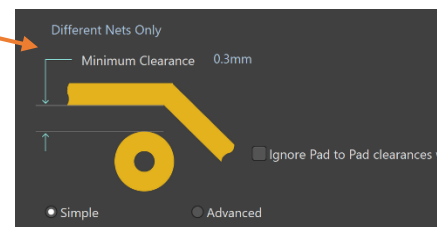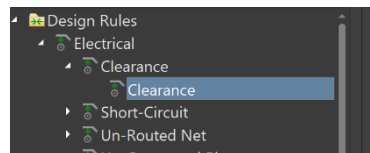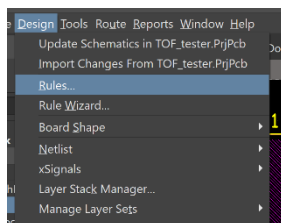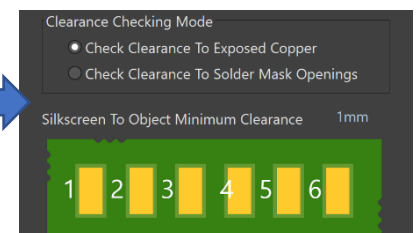
3) **Defining Routing Wires properties**
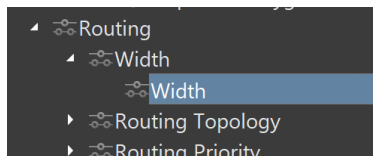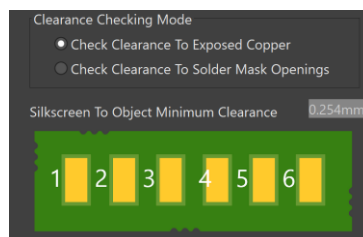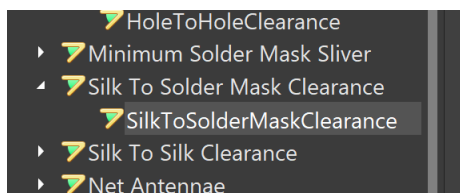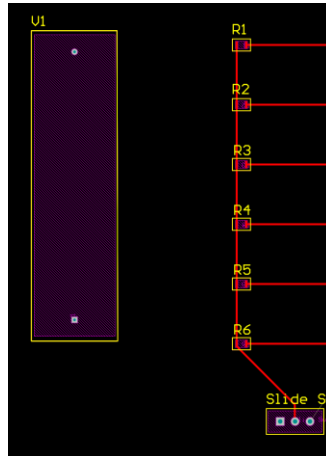
4) **Defining VIAs**
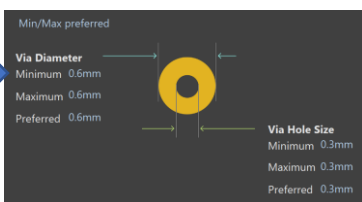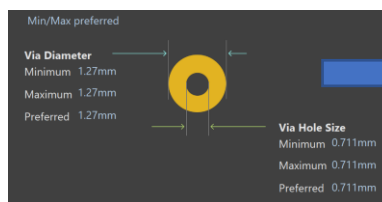
5) **Silk to Solder Mask**
   a. **You might need to adjust the PCB Library if you have Design Rule Violations**

Wiring PCB Components

1) You essentially use any layer to wire the components but try to stick to the Top Layer and Bottom Layer (easier for the use of VIAs

2) "Place" → "Line" on Top Layer

3) Add a wire from GND to somewhere, then "Place" → "VIAs" ON THE LINE YOU MADE
   a. Adding the Via on the line will automatically add the NET
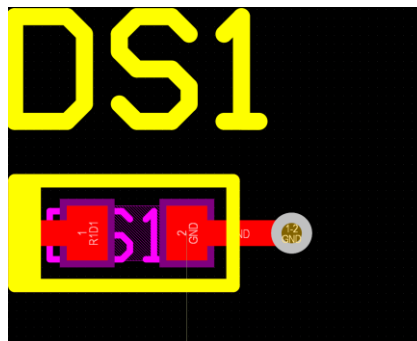
4) Ctr + F → Select "Bottom Layer" and place a "Wire(s)" for GND
   a. Connect the VIA to connect the TOP Layer and Bottom Layer

5) "Place"→ "Fill"
   a. This allows for a solid region to be dedicate for a specific Net (this case: GND and 1.5V)
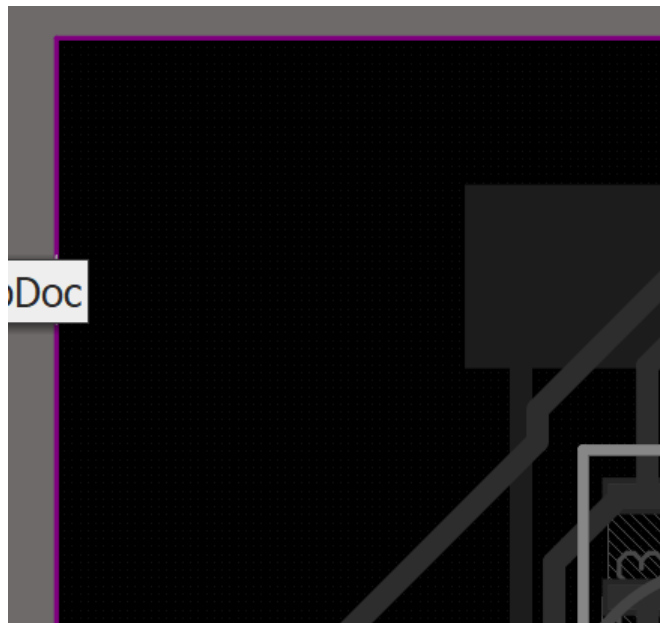   b. When you are done press "3" to see for "final product"

Jesi Miranda-Santos4
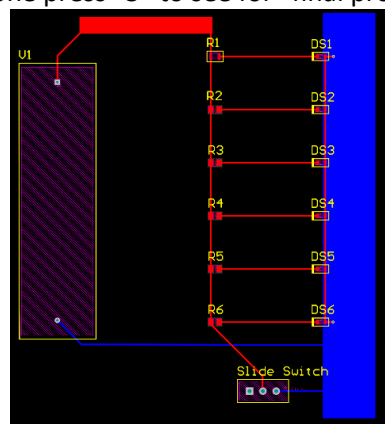
Adding Board Outline

1) **Add a Mechanical Layer (M2) and rename to "Board Outline"**

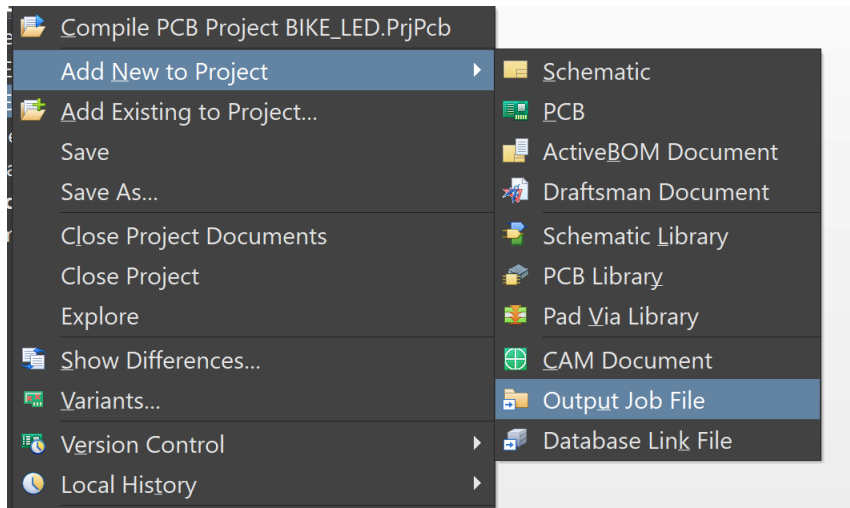2) **Make sure to be on "Board Outline" Layer**
3) **"Place"➔"Line"**
   a. Change width to whatever is desired.
   b. Place Line on the board outline to trace it out
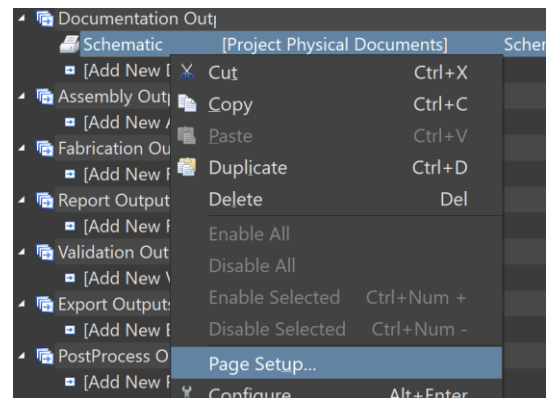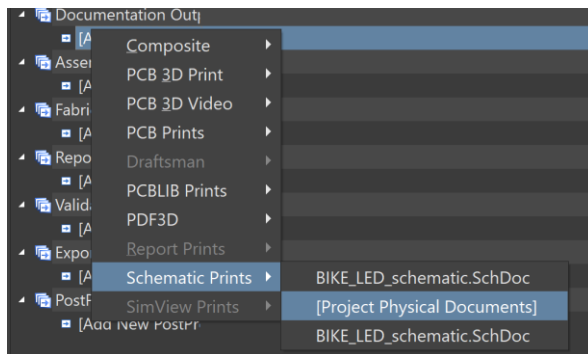
Generating Manufacturing Output

Creating PDF of Schematic
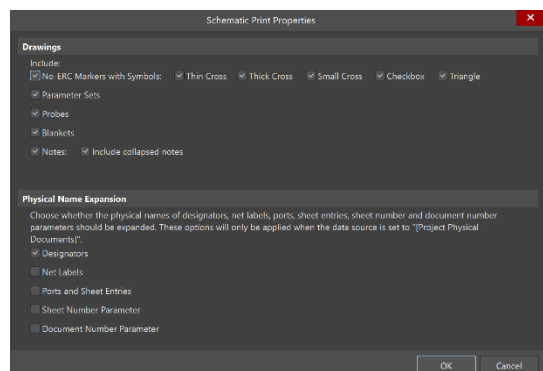
1) **Create a Output Job File**

2) **Create a PDF of the Schematic and change the Name**
   a. Left-click →Page Setup → change "Color Set" to "Color"
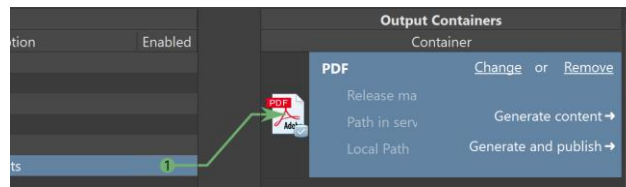

3) **Right-click and select "Configure"**
   a. A window as such will pop-out; select "OK" (no need to change anything)
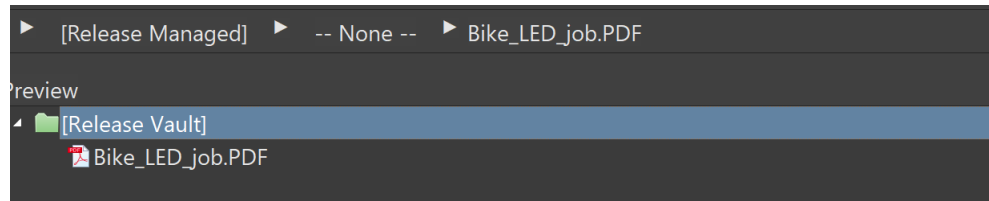
4) Now select PDF and then click on the circle under the tab "Enable" to print as a PDF
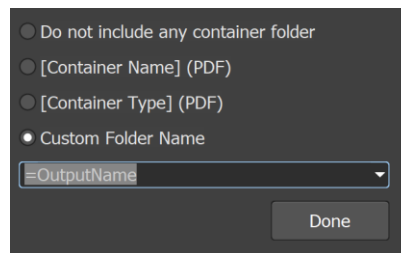

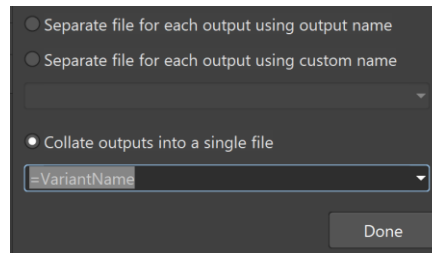**5) Select "Change" and you will click on "—None—"**

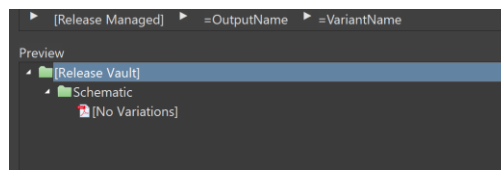a. We do this to have all the output files in the same directory
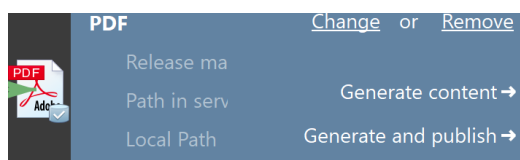

b. Do the following:

c. Now, select the name of the file (next to the one you previously selected) and do this:
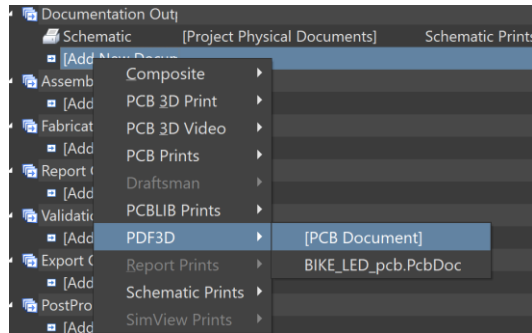

d. You should now see this

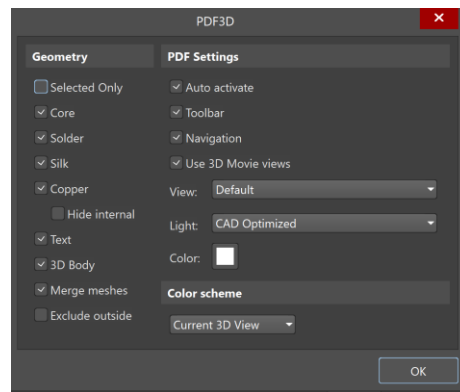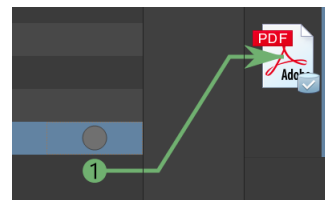**6)** Select "Generate content" to view the PDF schematic
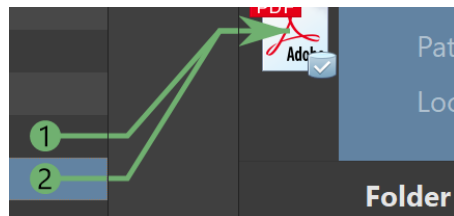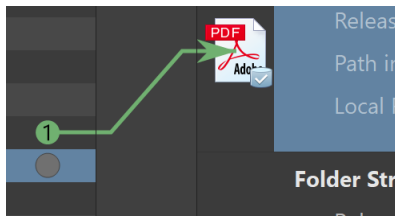
Creating 3D Model Output

1) **Under "Document Output" do the following:**

2) **Repeat step 2 and 3**
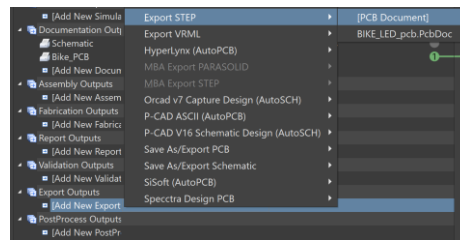3) **Click "Ok" (no need to change anything)**

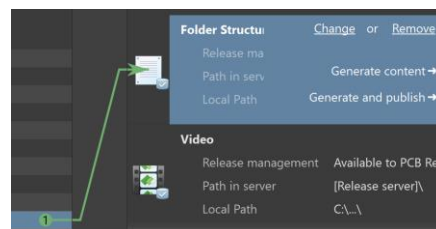4) **Now, click the gray then click on "1" to de-select**

Exporting 3D Model
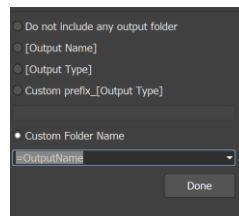
1) **Do the following:**

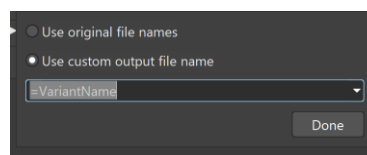
2) **"Configure" and then click "Ok"**
3) **Change the following output to match this:**

4) **Select "Change" → select "[Output Type]" and do the following:**
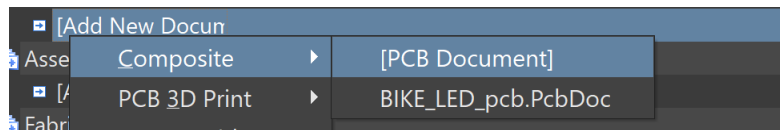

5) **Select "[Output Name]" and do the following:**
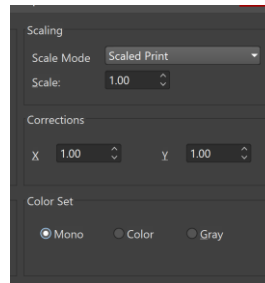

6) **Click "Ok" and select "Generate content"**
7) **To View the file, you will need to upload the file to AutoDesk Viewer**

Jesi Miranda-Santos9

Creating Layers Output

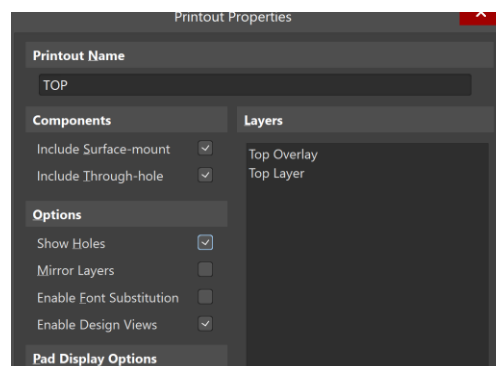1) **Under "Documentation Outputs" create a "Composite" document**

2) **"Page Setup" → change to match this →Select "Ok" → "Configure"**

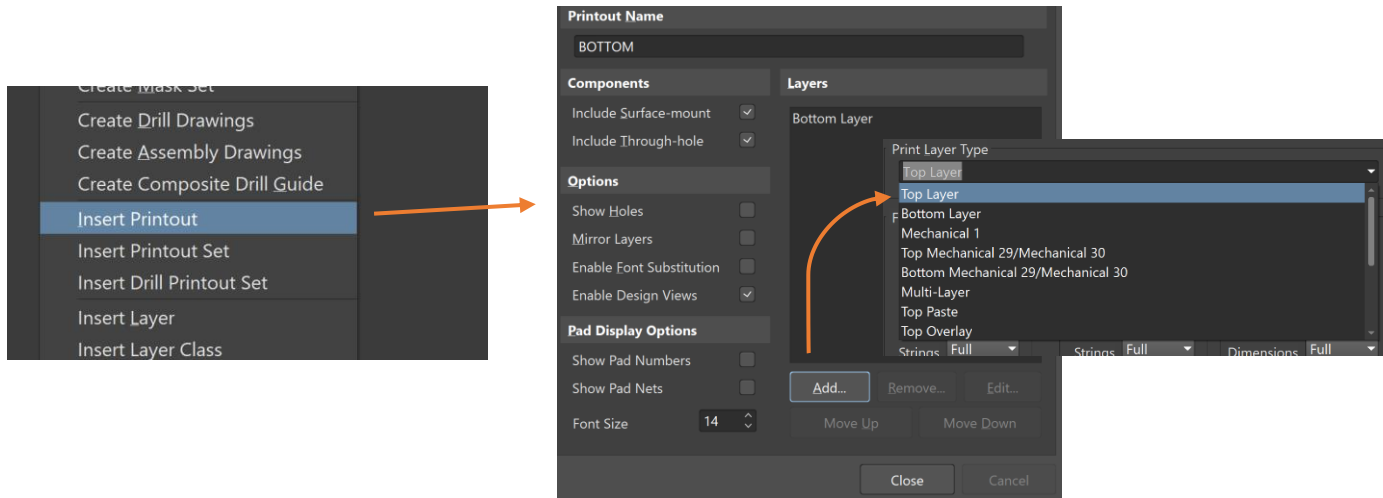3) **Double-click on "Multilayer Composite Print" to open the "Printout Properties Window"**
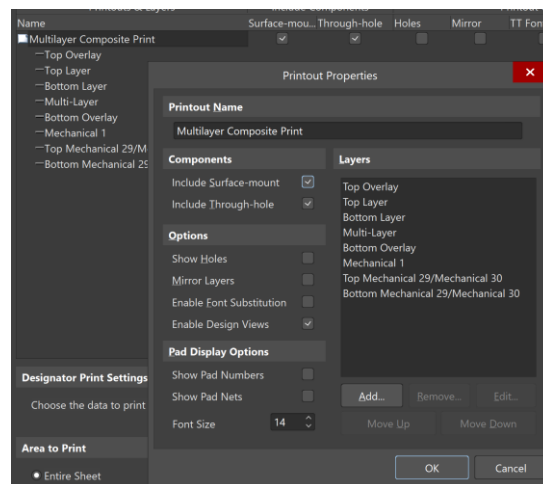
4) **Under "Printout Name", rename to "TOP"**
   a. Under Layers, only have selected "Top Overlay" and "Board Outline"
   b. Under Options, select "Show Holes" if have any through holes

5) **Create a new Printout for the BOTTOM layer:**
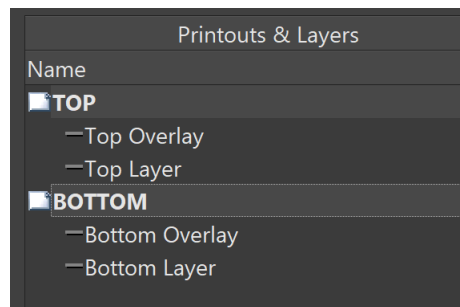   a. Click "Add", and select "Bottom Layer" and "Board Outline"
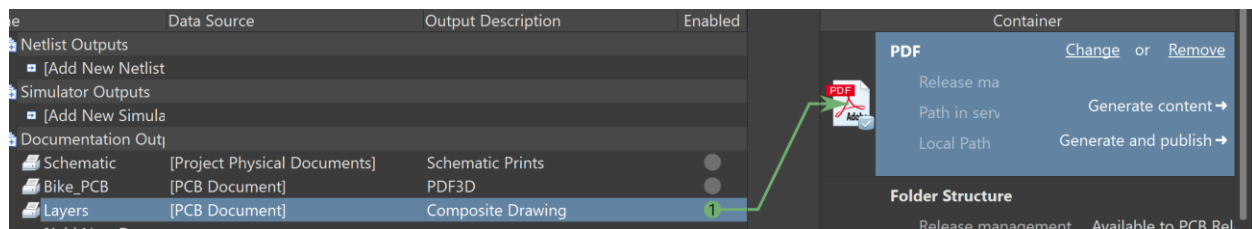

6) **Select "Show Holes" and "Mirror Layers" (because it's the bottom layer)**
   a. Click "Ok" and "Close"
7) **Right-click on "BOTTOM" and select "Move Down" to get the following:**
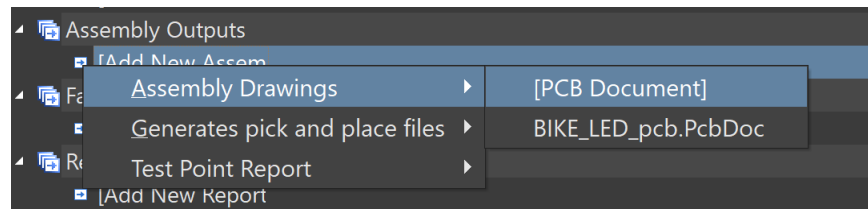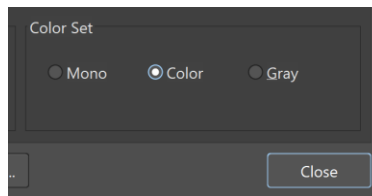

8) **Generate this as a PDF**

Creating Assembly Output
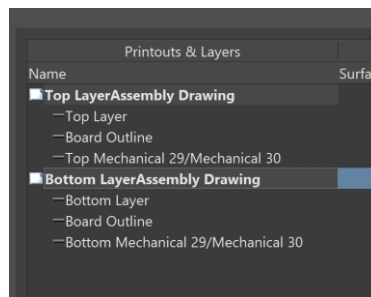
1) **Do the following:**

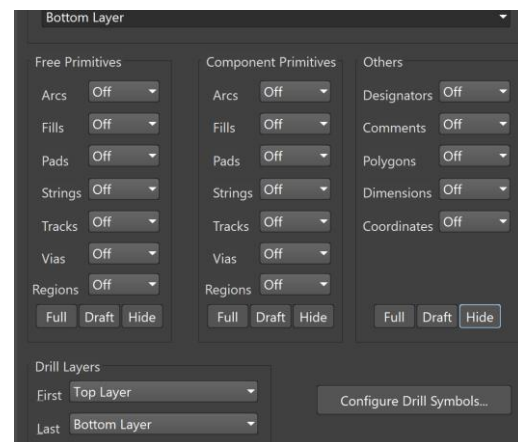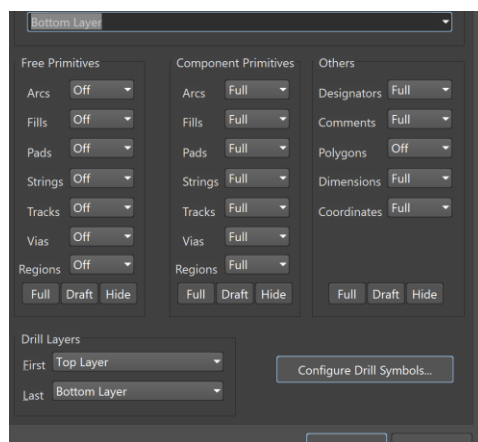2) **"Page Setup"→change "Color Set" to color**

3) **"Configure"**
4) **For both layers, just keep the "[Top/Bottom] Layer", "Board Outline", and "Mechanical #"**
   a. For Bottom layer, don't forget to "Mirror"

5) **Double-click "Bottom Layer" and select "Hide"**
   a. Repeat for "Top Layer"
   b. This is done if you have a Variant PCB

6) **Generate PDF**

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

Ctr + 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

Ctr + F – Flips Board to place stuff from the Back

L – While holding component with left click, this will put the component on the back of the board