

Altium Introduction

Henry Lau (2021)

V1.0



M2 ROBOCON
STUDIO

Topics

Creating a new project

Adding and editing schematic file

Schematic file's property and common commands

Adding and editing PCB file

PCB file's property and common commands

Change log / bug fixes

Version	Description
V1.0	Initial

IMPORTANT LINKS

Embedded Ninja Altium Guides

<https://blog.mbedded.ninja/electronics/general/altium/altium-tricks-and-standards/>

Sometimes I will just point to this website instead of explaining it here

<https://www.altium.com/documentation/altium-designer>

https://www.youtube.com/watch?v=PH69SMrmBag&list=PL3aaAq2OJU5H_Jj72DObh5kNh6Nr4xNS0

Official altium documentation and tutorial

Altium

Altium

Download and installation

How to check profile and license

How to create project, schematic, PCB, schematic library and PCB library

Learn the importance of unit

Altium Installation

Install either happpppppppy or student edition

Install Altium (happppppy version)

https://drive.google.com/drive/u/2/folders/1ufxKkO22k_HXneU2uyKhtjU6RSZp32Lt

Download the zip from here ad follow the tutorial pdf to install Altium

It is 10 times easier than installing solidworks

Install Altium (Student version)

<https://www.altium.com/solutions/academic-programs/student-licenses>

Apply for student license

<https://www.altium.com/products/downloads>

Download Altium from official website, follow the instructions on official website.

Download Libraries

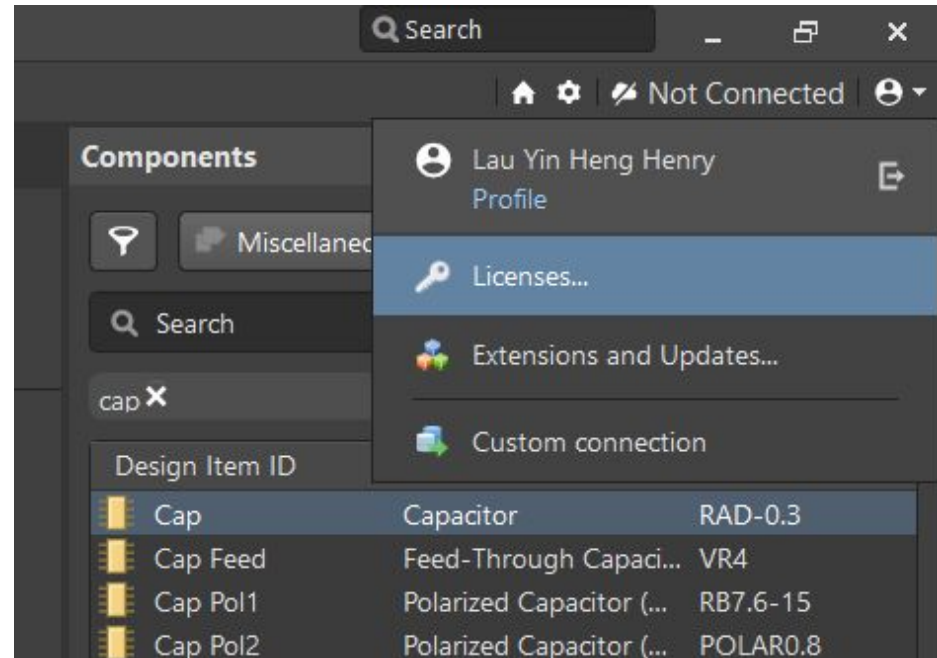
Download the necessary libraries from this link

<https://drive.google.com/drive/u/2/folders/1DtgfGTG4yzhiiBbwYUXcPySys9V6JWob>

Checking license

Press the human icon, and u could login to your account (if you have one) and check if you have legit license

Only those using student version will need to login in order to get student license

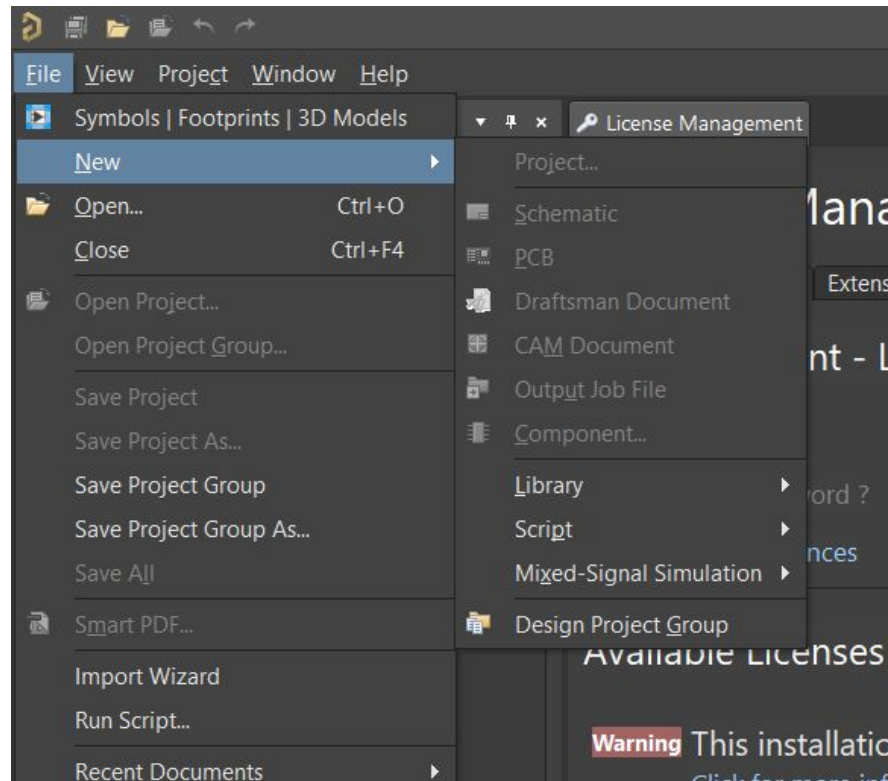


License

You won't be able to create any new files without a valid license.

If you install normal edition, you could request license with your university's student email

If you install happy version, follow the guides for that specific happy version to use standalone files

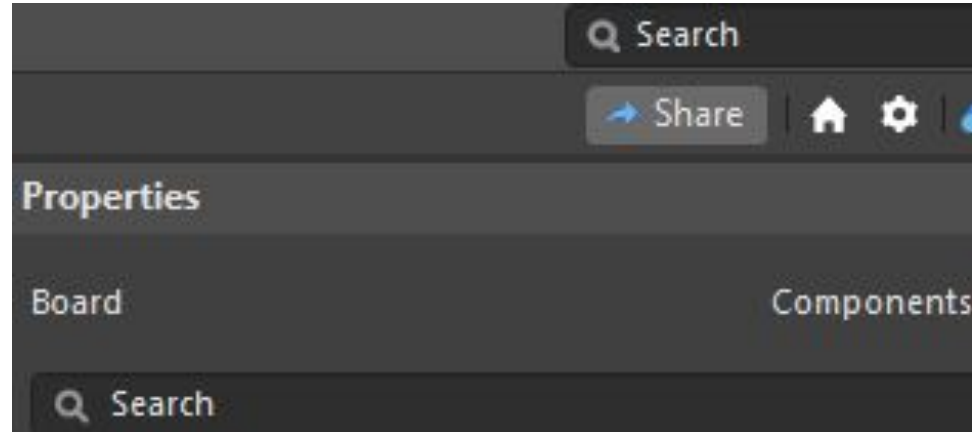


Weird license problem for happy version

For the happy version, if more than 2 people are using altium under the same Wi-Fi network (subnet). The license could clash and you may need to select another license that is not used by other people.

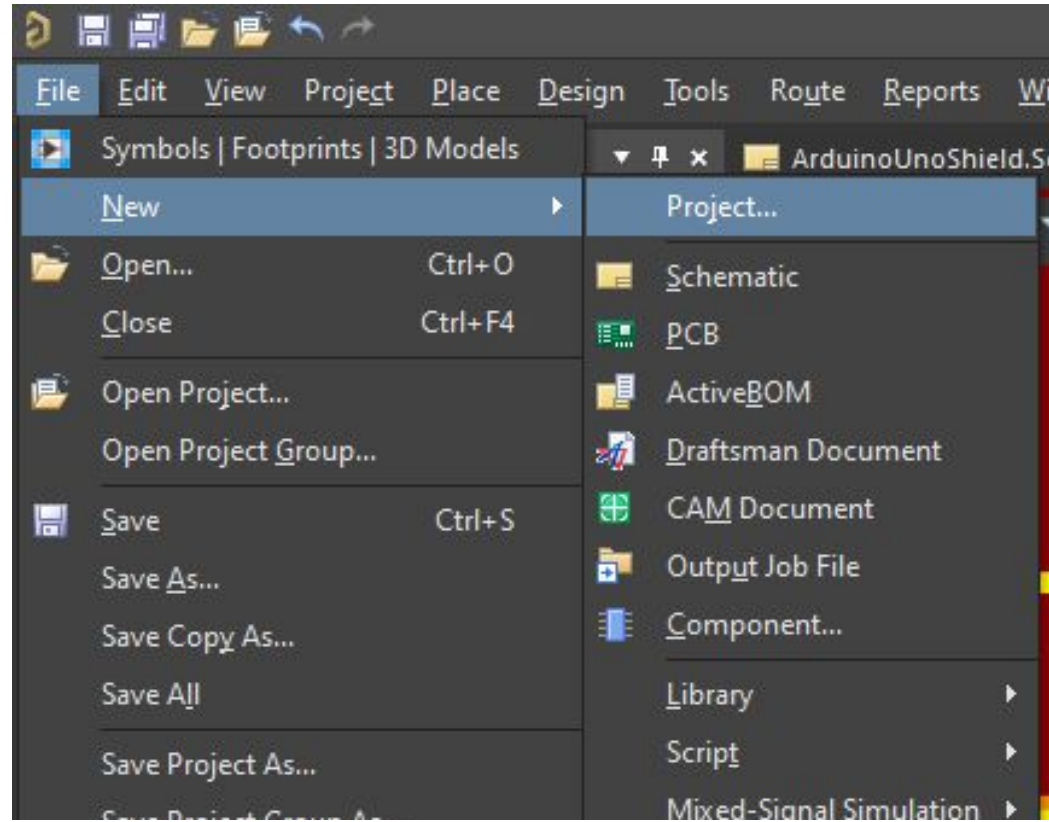
Searching Commands

Commands and menus could be searched from the top right corner



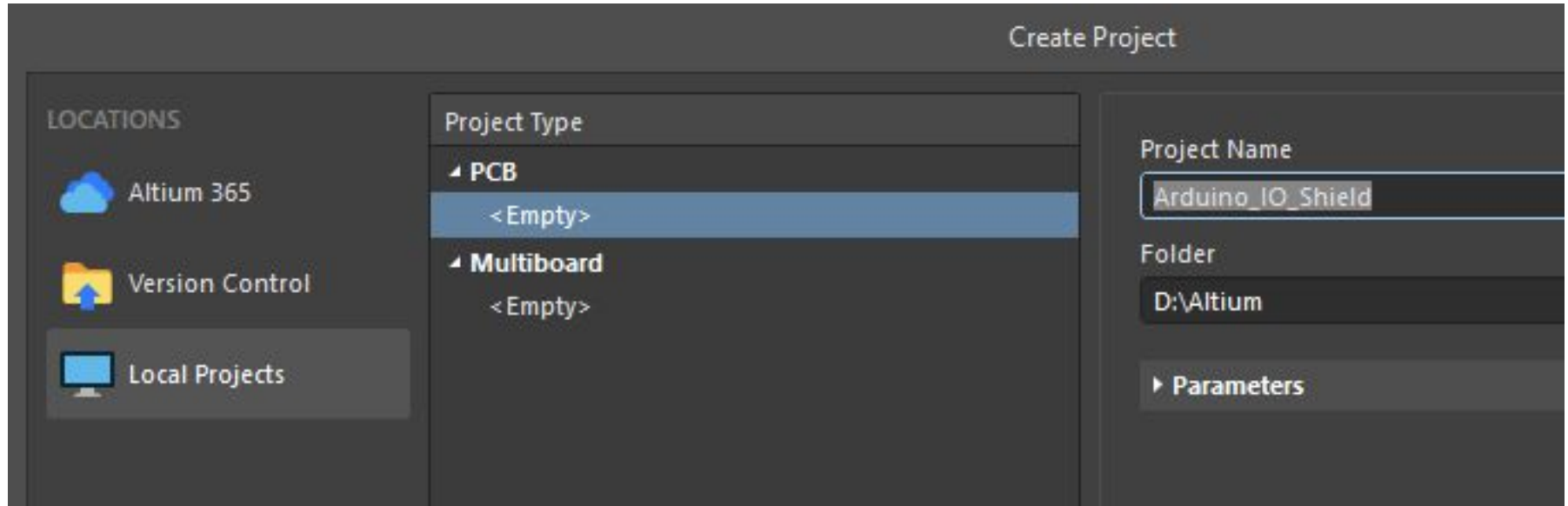
Projects

New project workflow



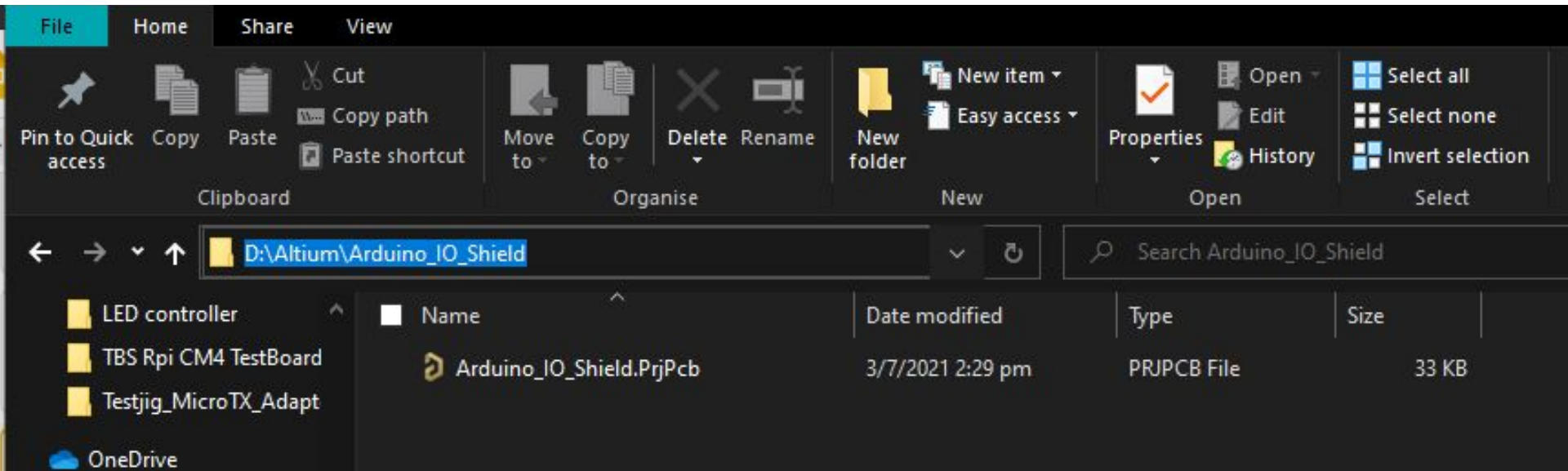
New project workflow

Give the project a proper name. Altium will auto create a folder with the same project name



New project workflow

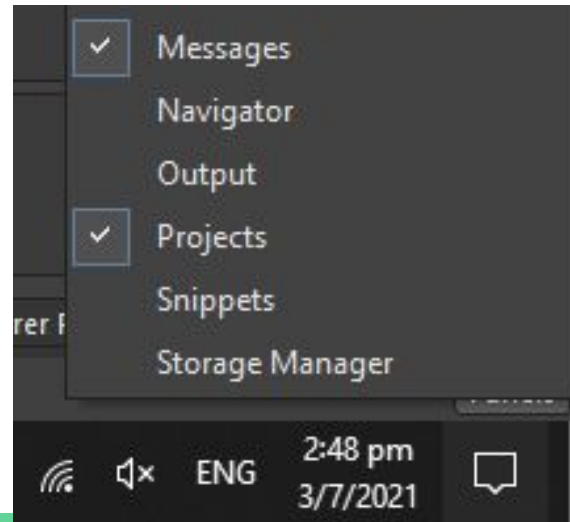
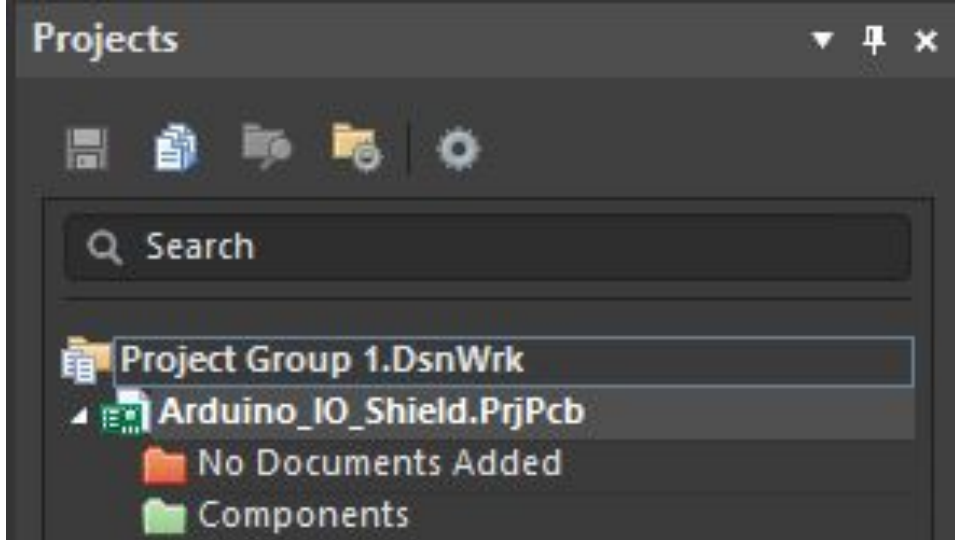
You could see the project created in this repository with the same name



New project workflow

You should see the project in the “project” panel

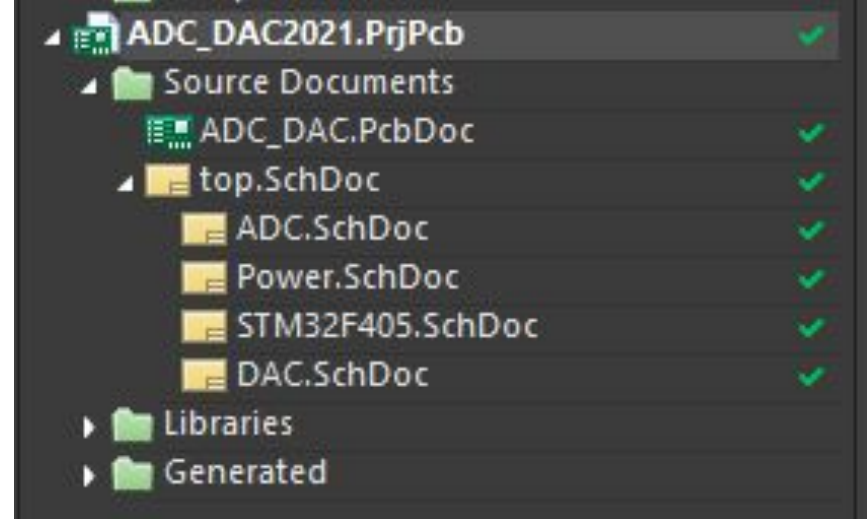
If you cannot find the project panel, you should press the “Panels” at the bottom right corner and open “Projects”



New project workflow

For each project, new files could be created, existing files could be included into the project.

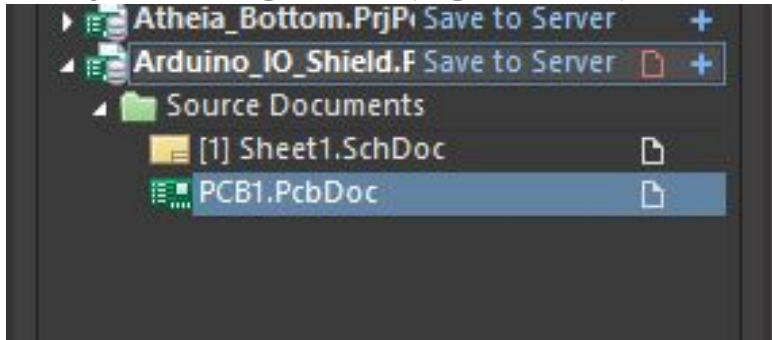
For each project you could at most have one PCB file, but you could have multiple schematic files.



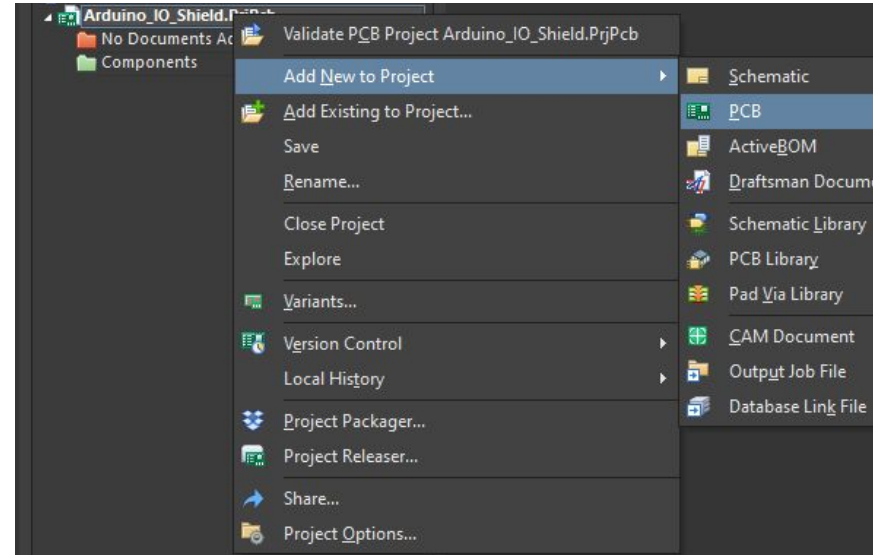
New project workflow (New Files)

You could either add existing schematic / PCB files, or create new files

First add one schematic and one pcb file into the project and save those files with (Ctrl + S) or by clicking RMB (right click) on the files



You should see 2 files added under the project

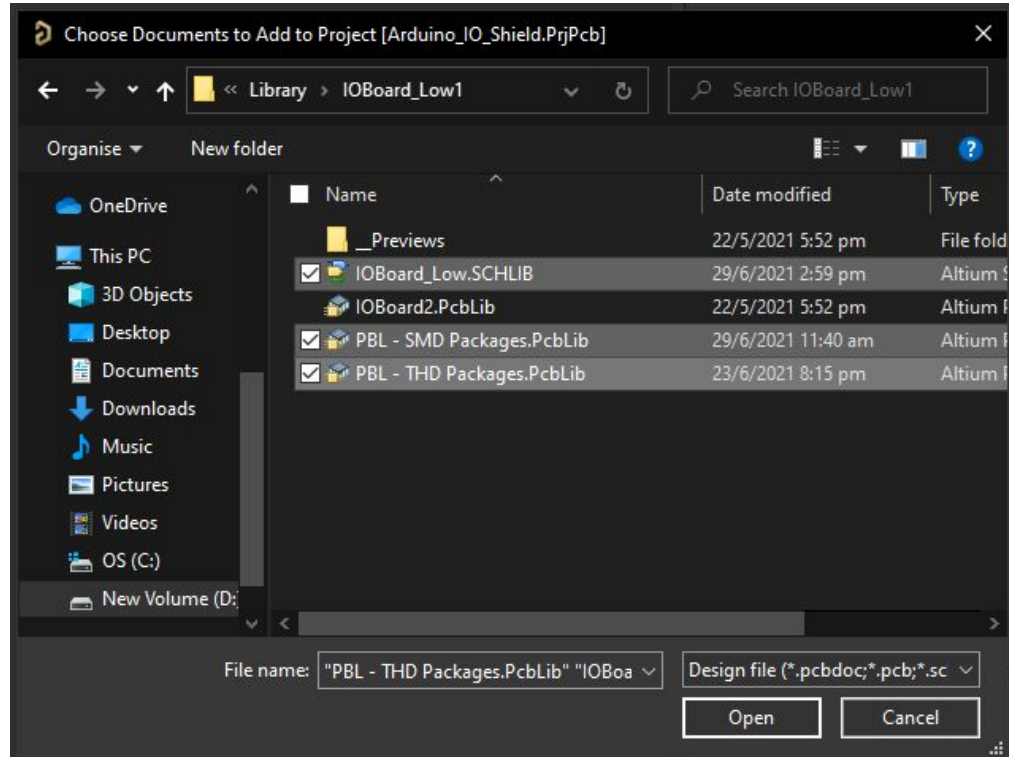


New project workflow (Add Existing)

If you pressed “Add Existing to Project”, you will see this window.

Traverse through the file system to the folder containing the libraries

Click on the files to add existing schematic libraries (SCHLIB) and PCB libraries (PCBLIB) into the project.



New project workflow (Add Library)

In the add existing tab, schematic and PCB libraries could be added into the project. Libraries are .PCBLIB and .SCHLIB file type

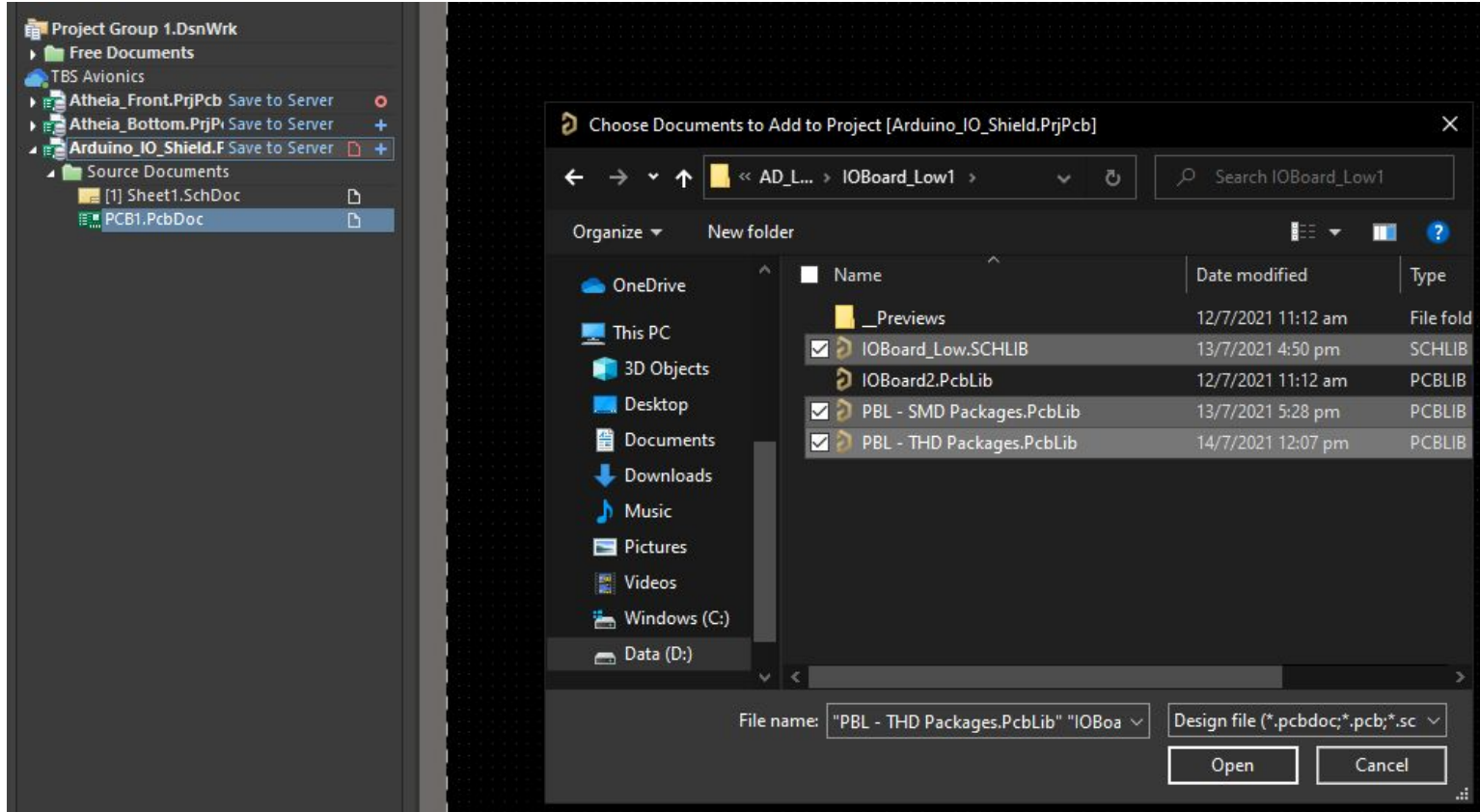
You have to download the libraries that we are going to use in the lab from here:

[Link](#)

(Not used in this lab) My personal library is here

https://github.com/onedino/AD_Library

New project workflow (Add Library into Project)



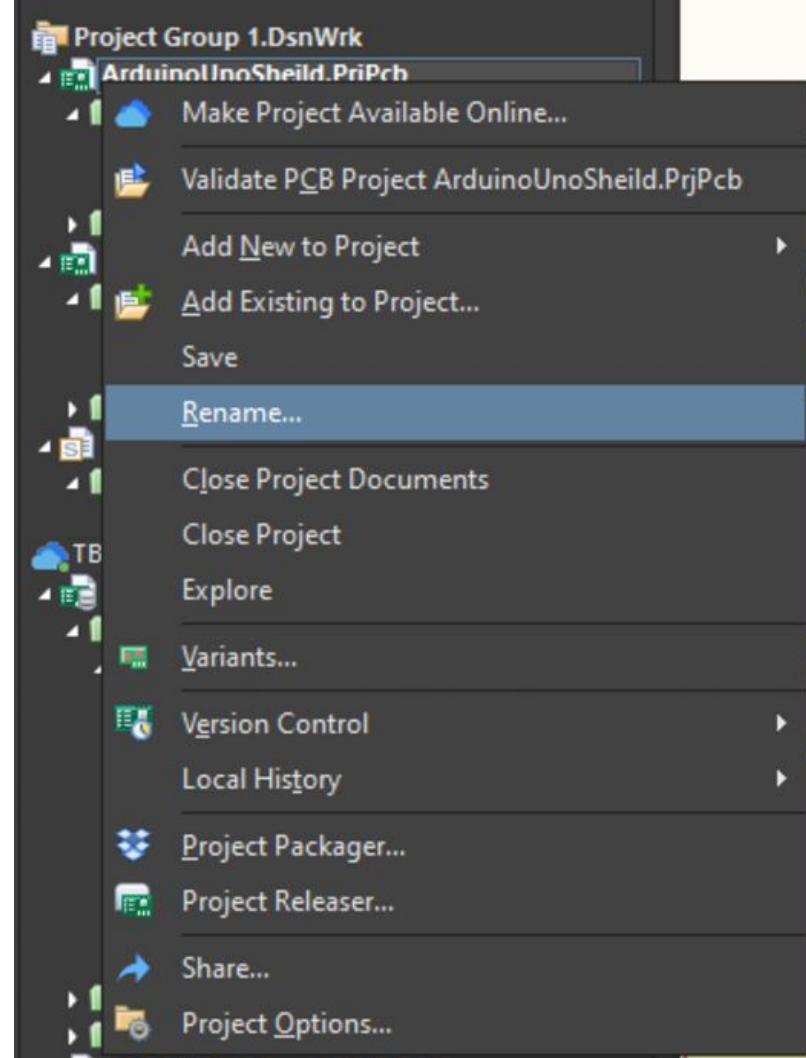
Renaming files

JUST FOR REFERENCE in case you need to rename some files.

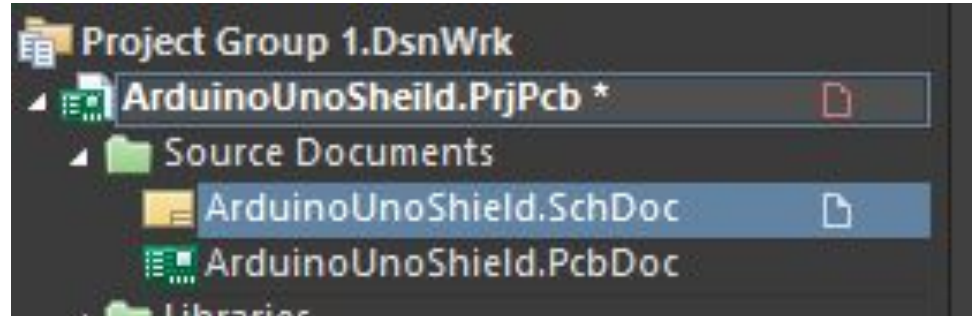
Click RMB on the file that you would like to rename and navigate to the Rename command

Files could be closed, saved, committed, pushed, reverted etc.

Explore will open the file explorer where the file is located



Unsaved files



After adding the libraries there should be a star next to the roject

The star on the right upper corner indicates there are changes made to that file, and is not saved after modification. Right click the project files to save the project file.

The project file contains the structure of the entire project, it requires saving after adding or removing files from the project

Remember to save you project!

Schematic

Traversing the Schematic

Zoom In/Out:

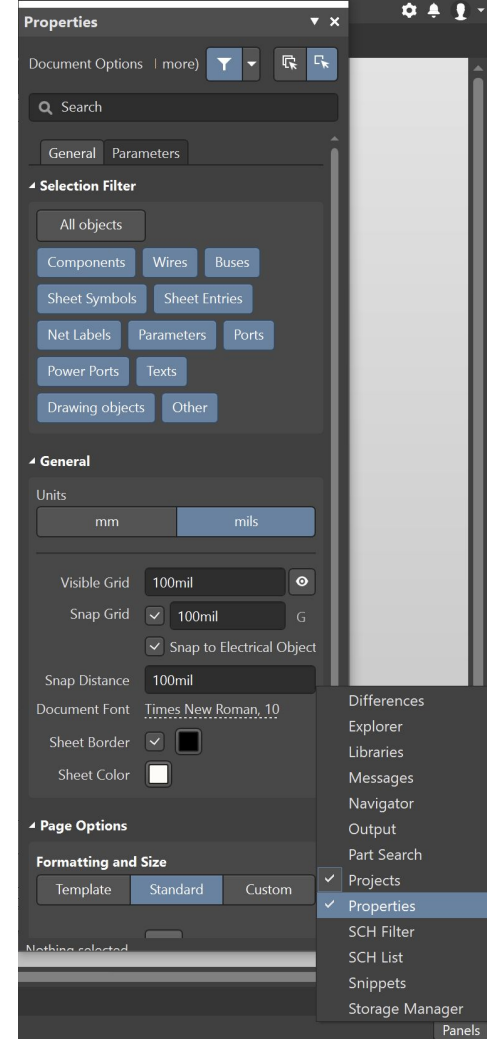
- (Ctrl + mouse wheel)
- Pushing the mouse forward or backward when holding middle mouse button

Pan (move the paper)

- Holding RMB while moving the mouse (hold rmb and drag)
- Scroll mouse wheel to move up and down
- (Shift + scroll mouse wheel) to move left and right

Properties in Panels

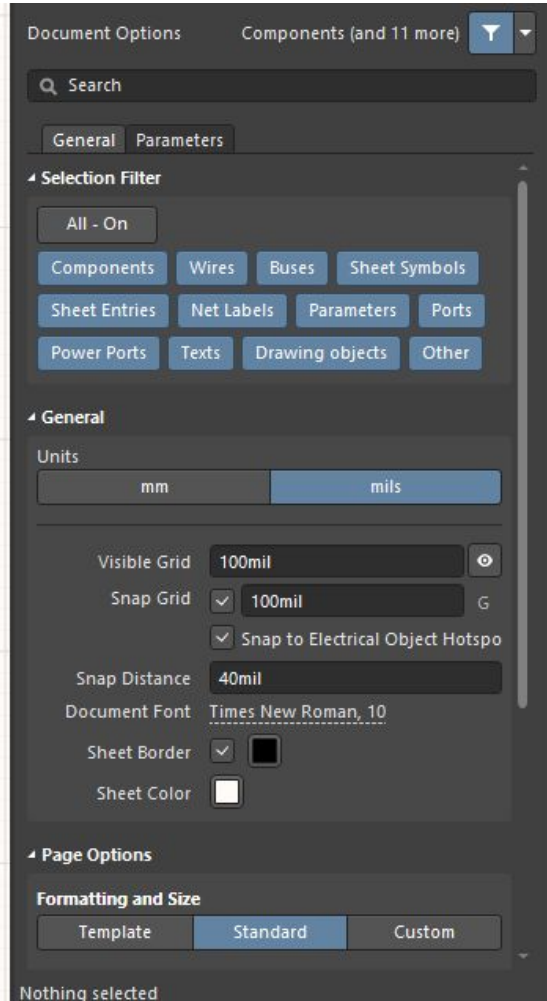
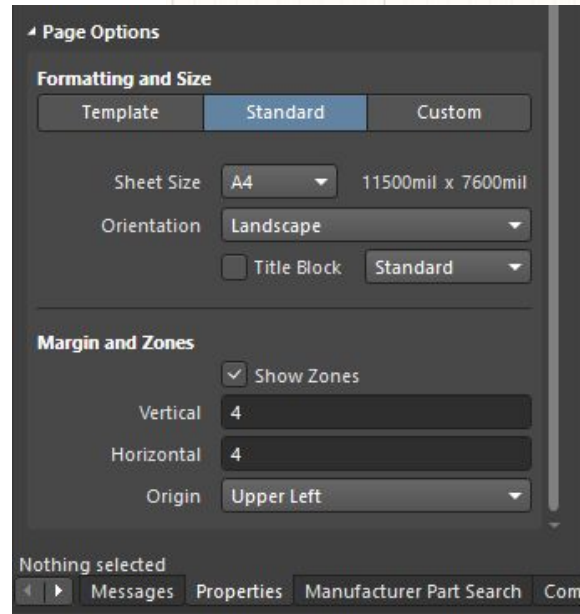
You can open up “Properties” by clicking on its button in the list of Panels



Schematic

These are the properties of a schematic file:

- Visible grid
- Snap grid
- Snap distance
- Page options
- Sheet size, orientation



Visible grid

MUST set the visible grid to **100mil**

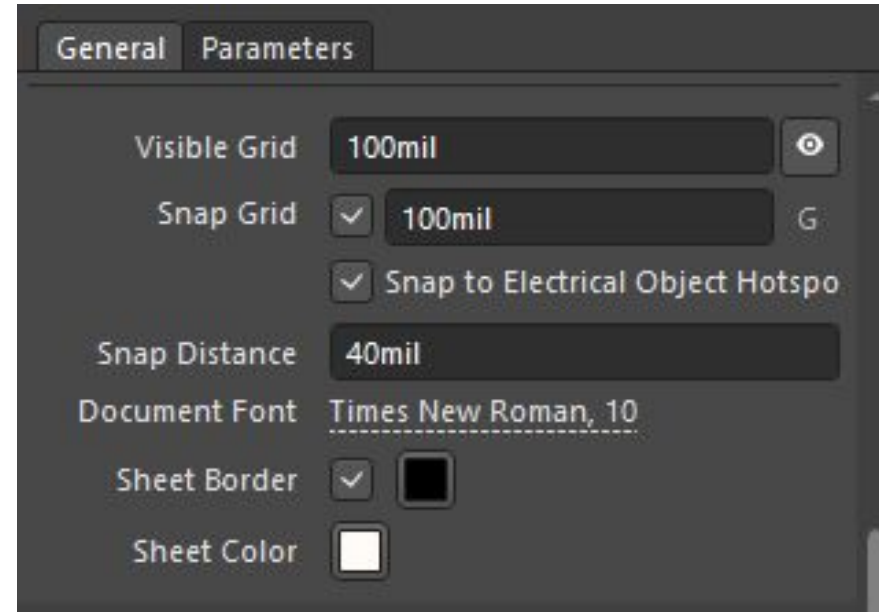
REMEMBER the unit is MIL

Mil is most commonly used in schematic, using mm in grid will lead to more components being **off grid**

Unit conversion:

1 inch = 1000 mil = 25.4mm

0.254mm = 10 mil



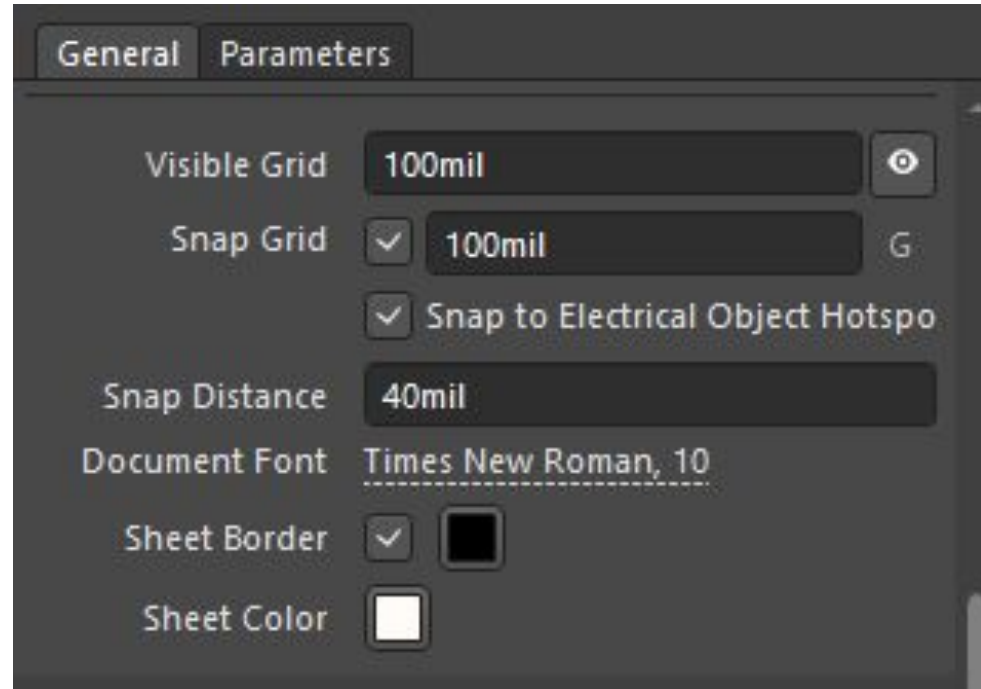
Snap grid

Snap grid is usually same as visible grid

By pressing “g” on your keyboard, the “Snap Grid” will usually change between 10, 50 and 100 mil, **keep it at 100 mil for 99% of the time**

Snap distance is the distance that the object will snap onto the grid

Usually you would not need to change this from the property tab

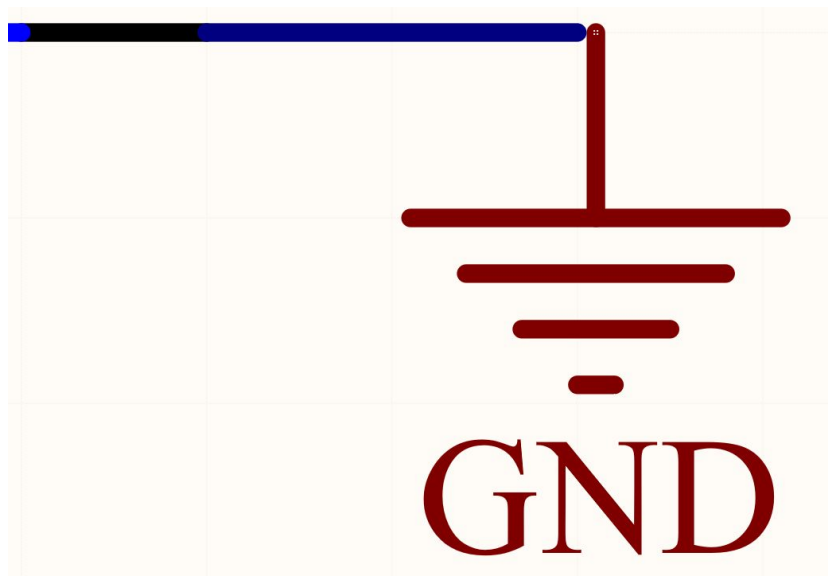


Issue of OFF GRID

When you see objects that are not on grid, which is caused by snap grid being different to visible grid

Objects should be on grid, having them off grid will often lead to **fake connection issues** (shown in the picture with Ground Power Port)

Altium will give Warning about off grid objects when compiling the schematic, but it will not be covered in this slide

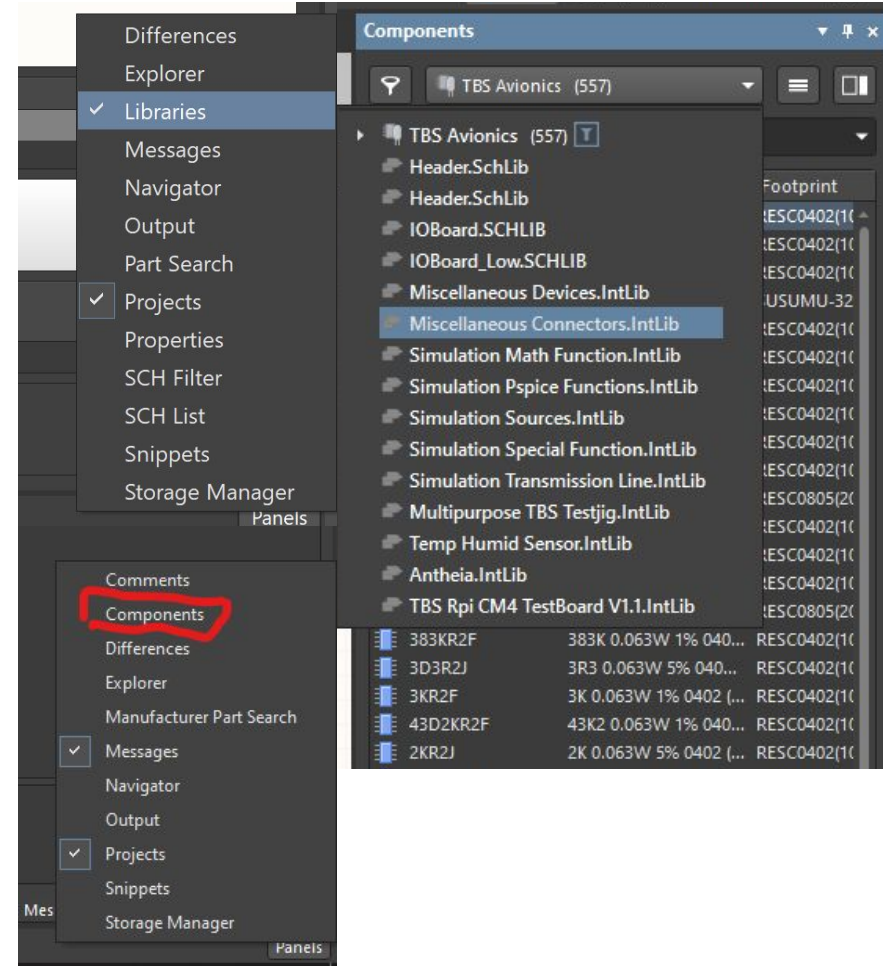


Components

One of the panel is
“Components” (For version
20 or newer)/ “Libraries”(for
happy version)

Included libraries could be
selected from here

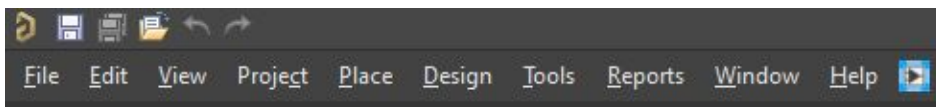
And the components
belonging to the library
would be displayed



Schematic Symbols

How to place Net Label, Power Port, Wires

Stuff needed to connect components together inside schematic files



Placing schematic symbol

Press (P) show the menu. The place menu is also available at the top toolbar

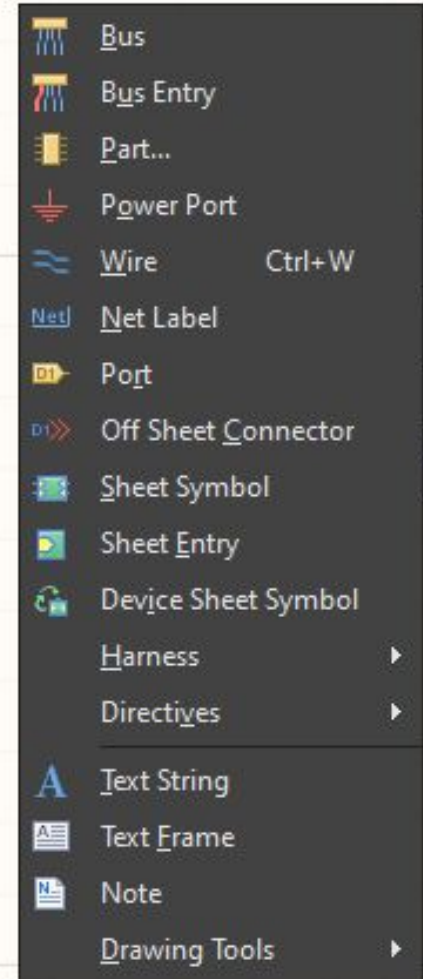
The most common commands that you could use are Wire, Power Port, Net Label, Text String for now

One letter in each command has underline, which shows that is a shortcut key to use that command

Command Short form format used throughout the lab:

(P, W) => press "P" first, then press "W"

(CTRL + W) => press both at the same time



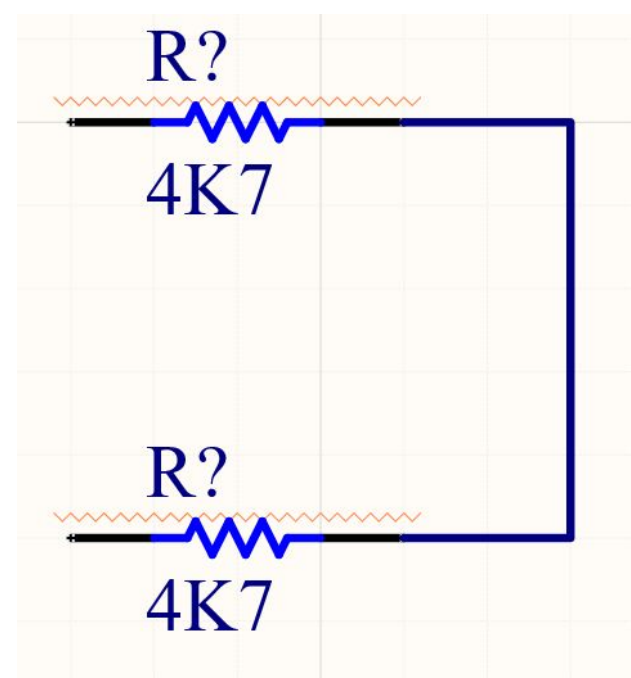
Wire (P, W) or (Ctrl + W)

Wires are the blue line in the picture

Wires are used to connect pins of different components

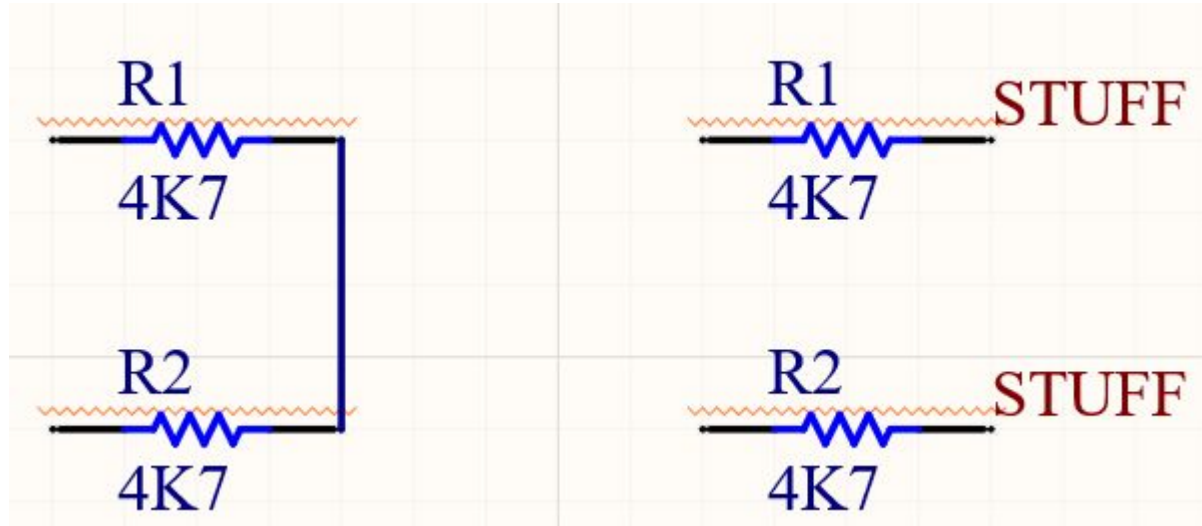
For example the wire is used to connect two different resistors

After you called the command, you will see the cursor become a crosshair, click LMB (left click) to place the vertex of the wire onto the schematic. Press Esc or click RMB to escape the command. This applies to other commands



Net Label (P ,N)

2 pins having the same net label on them is electrically connected. So in the following picture, both components are actually connected in the same way despite using different methods



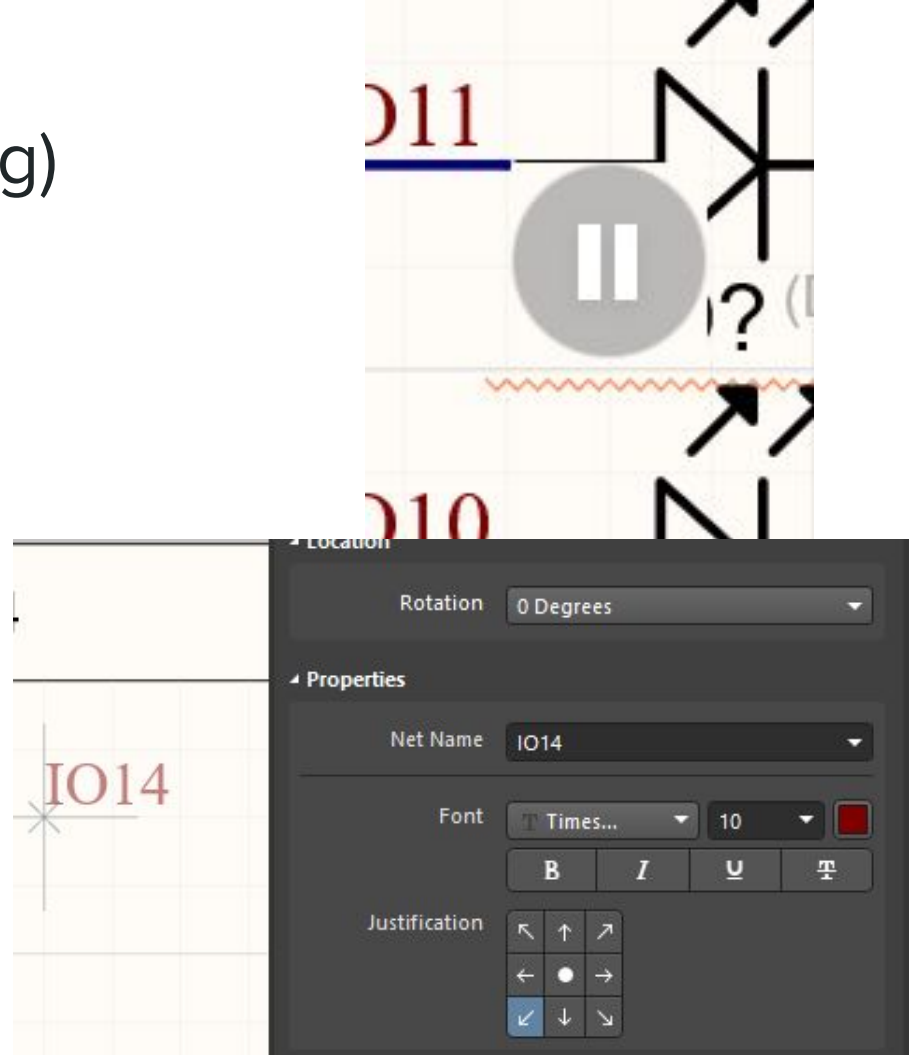
Net Label (Edit before placing)

It could be rotated by (Space Bar)

Before placing the Net Label, the properties could be edit by pressing (Tab)

A pause logo will appear in the middle of the schematic

The Net Name, font size and font type could be changed



Net Label (Increment)

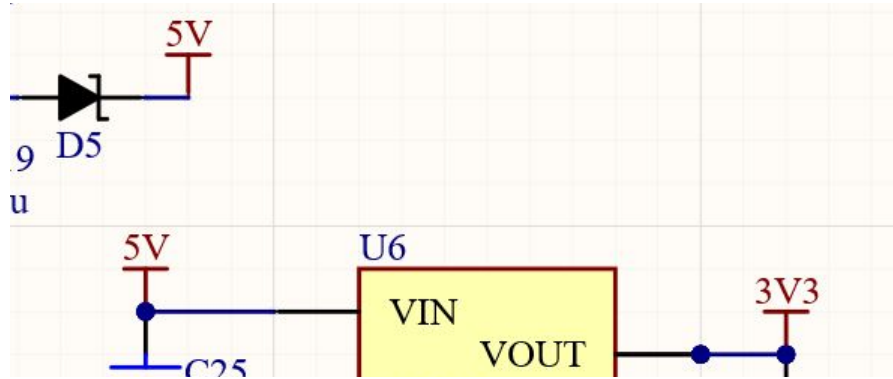
When placing Net Label with Net name ending with number, the number will increment by 1 after a NetLabel has been placed



Power Port (P, O)

Power port is very similar to Net Label, but it is reserved for nets that are power nets

Example usage of power port for connecting 5V and 3.3V shown in the picture below

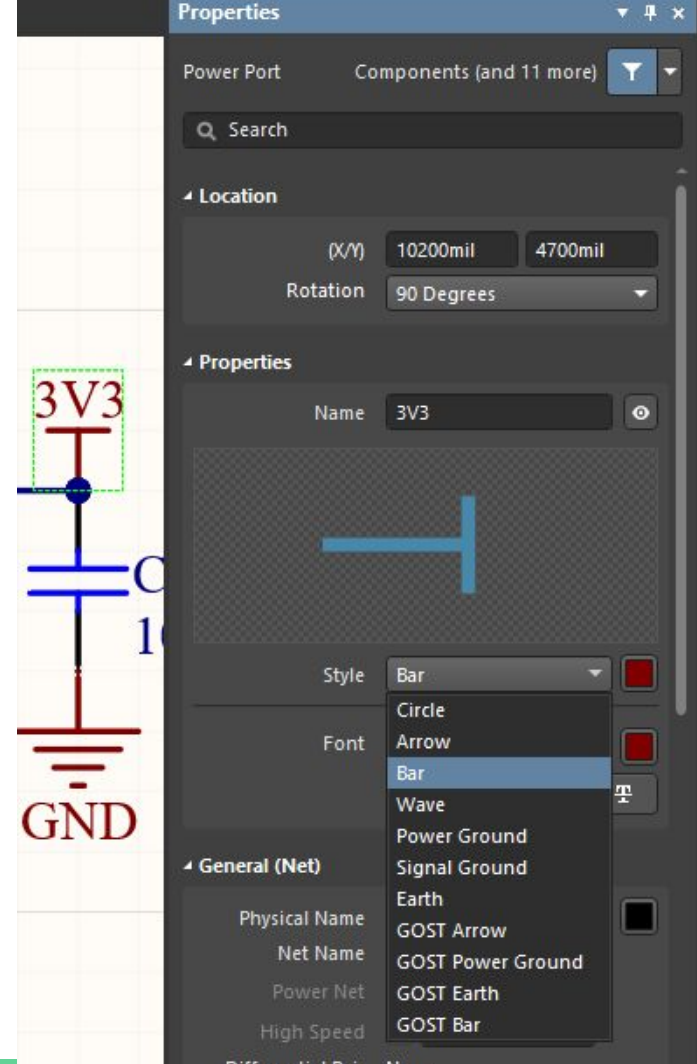


Power Port (P, O) (Properties)

Types of power ports

- Bar, Arrow for power rail
- Earth, Power Ground, Signal Ground for ground

If your circuit have multiple ground, you should use different symbols to represent them



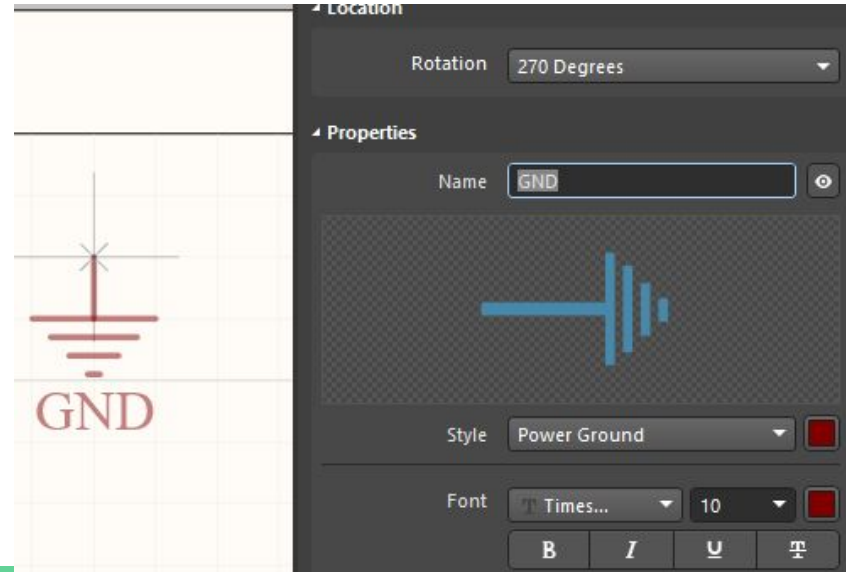
Power Port (P, O) (Edit before placing)

Similar to Net Label

It could be rotated by (Space Bar)

After pressing the command, the Net Name could be edit by clicking (Tab)

A pause logo will appear in the middle of the schematic

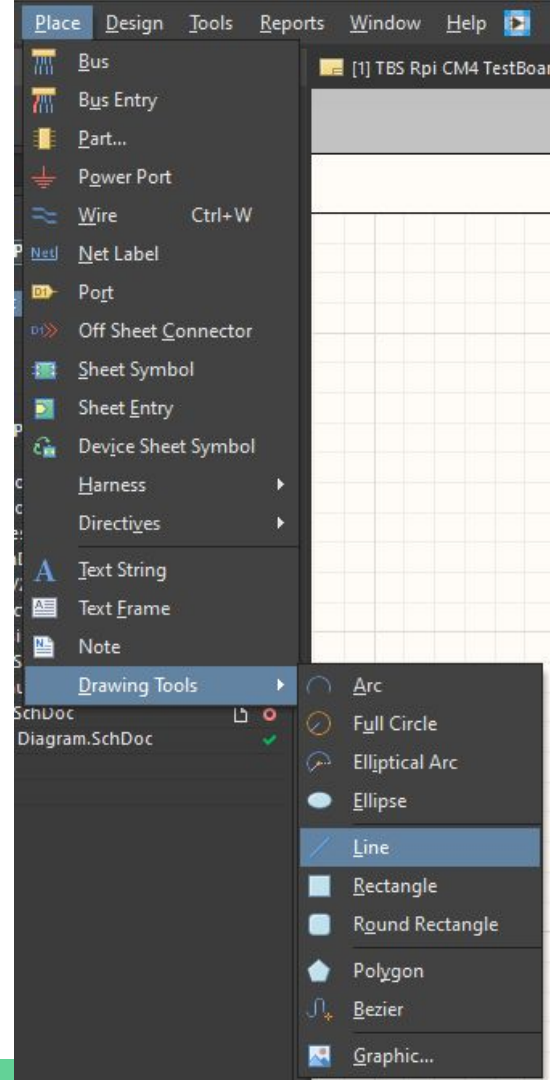


Drawing Tools

Drawing Line, Rectangle, Text String onto the schematic

Drawing Tools

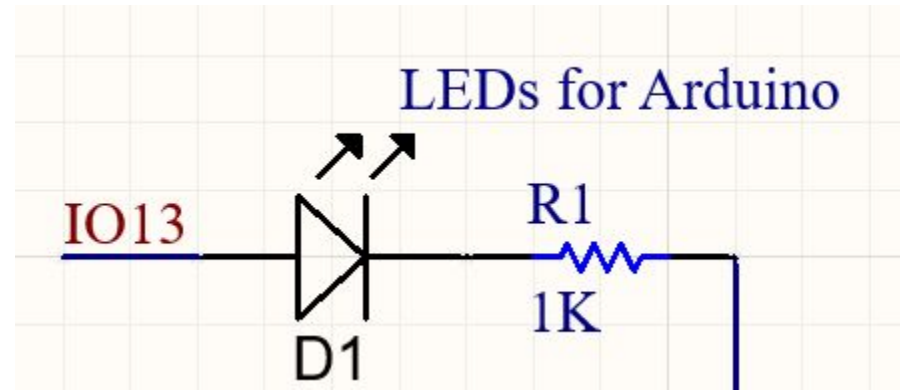
Drawing tools are used to draw different shapes onto the schematic, they does not connect objects electrically



Place Text String (P, T)

Placing text string on different part of the schematic to act as comment for the schematic

Its font, size, bold, color etc could be edited in its properties tab

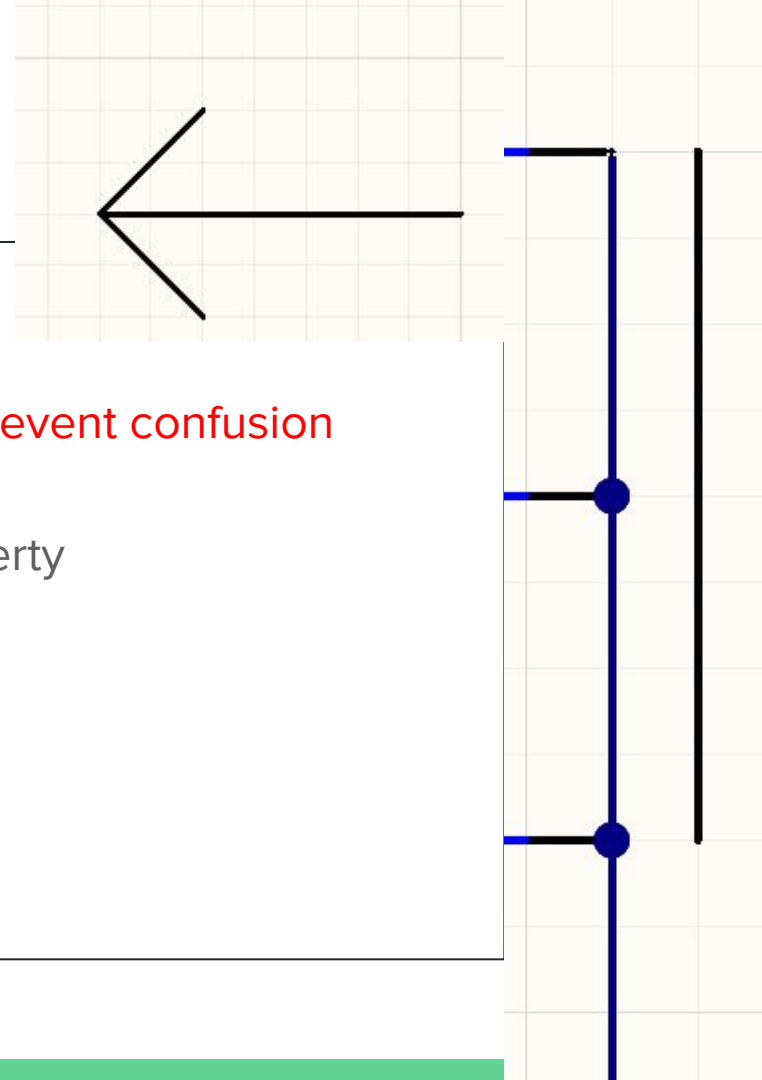


Place Line (P, D, L)

Line is usually used as marker

Do not use the same color as wire (deep blue) to prevent confusion

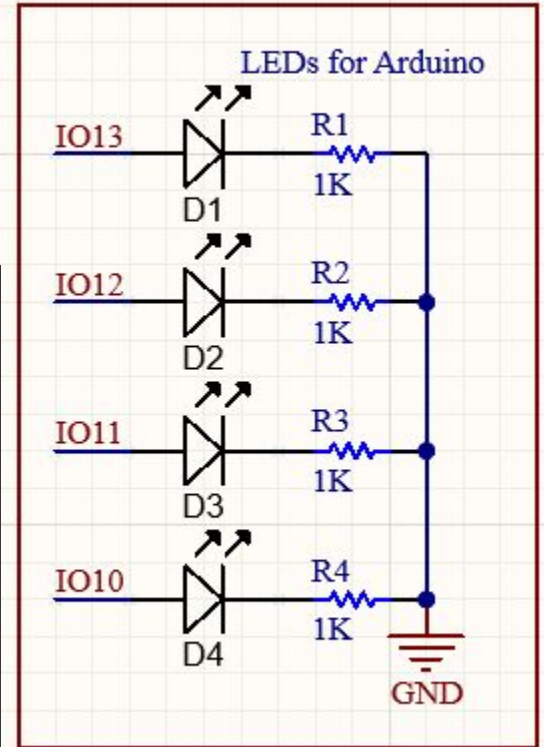
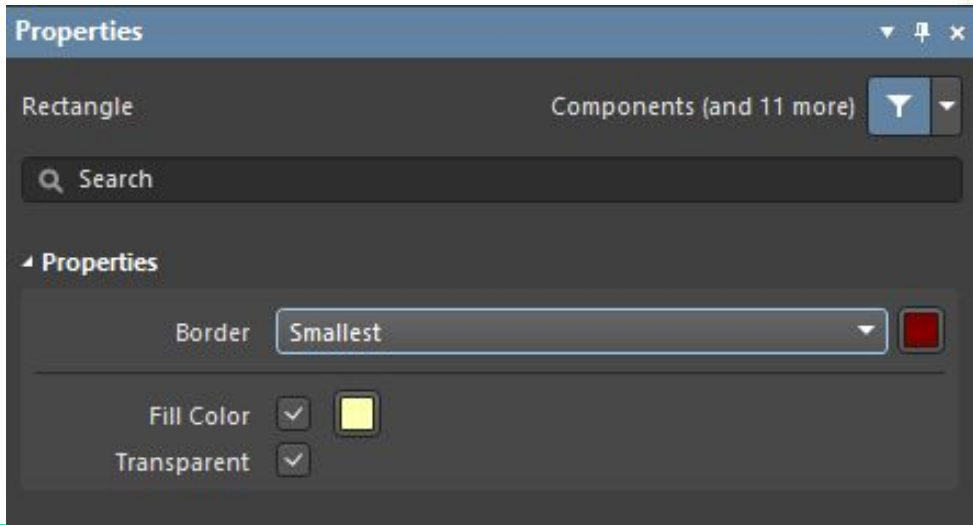
The thickness, color etc could be edited in its property



Place Rectangle (P, D, R)

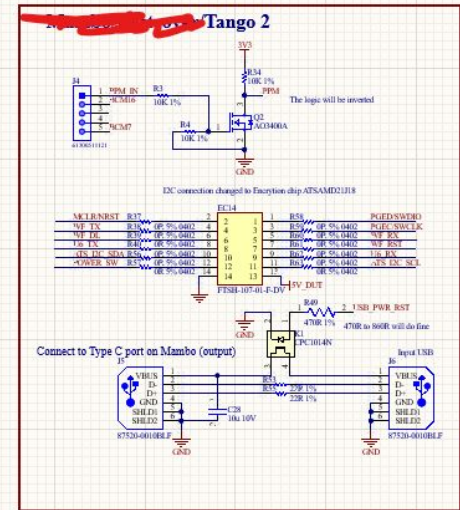
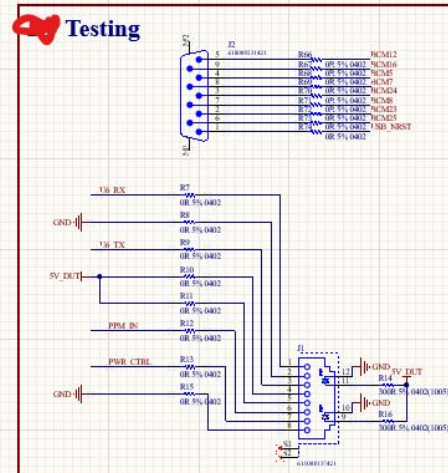
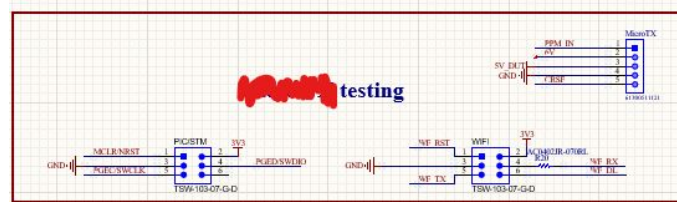
Rectangle could be used to circle the objects that the text string is referencing

In its property tab, you could change its Border, Its fill color and its transparency



Drawing Tools

Use them wisely to increase the readability of the schematic



Edit the Schematic

Add components and connecting with schematic symbols

Select, Drag, Delete, Copy/Paste objects

Editing Schematic

Now, we will try to add a LED to the Arduino shield

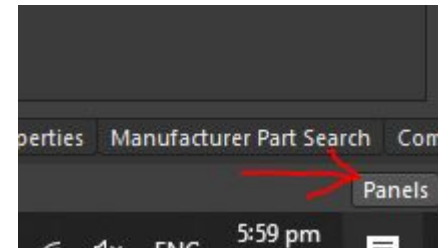
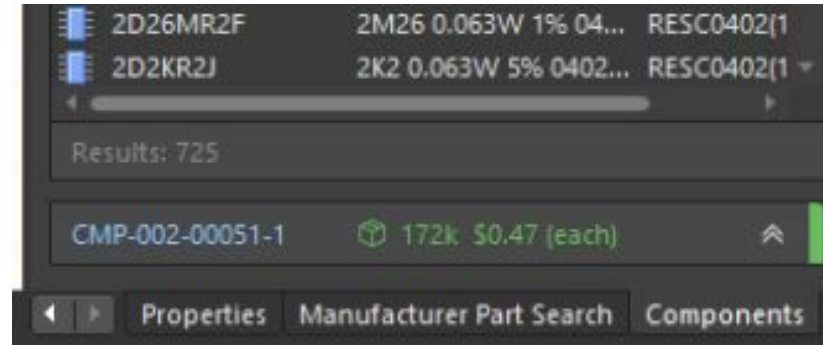
First find the required components from the altium library

- Grab a LED and resistor from the library
- Connect the LED to a GPIO port and connect the current limiting resistor

Adding Parts

Find the Component tab on the bottom right corner of Altium

If it is not spotted in the tabs, open it through Panels



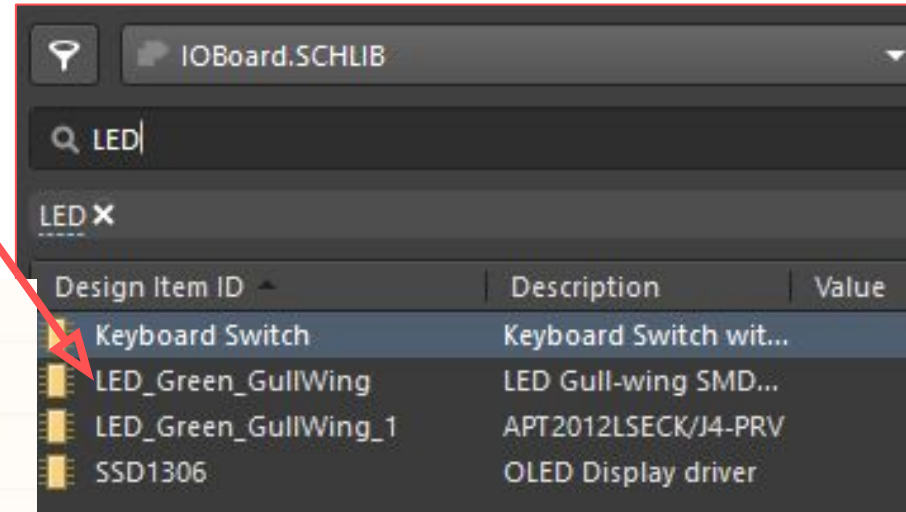
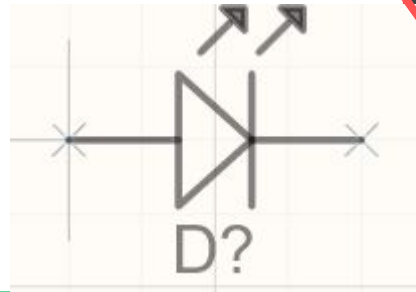
Adding LED

Select "IOBoard.SCHLIB" in the drop down menu

Search "LED" in the search bar

Double click LMB on LED_Geen_GullWing

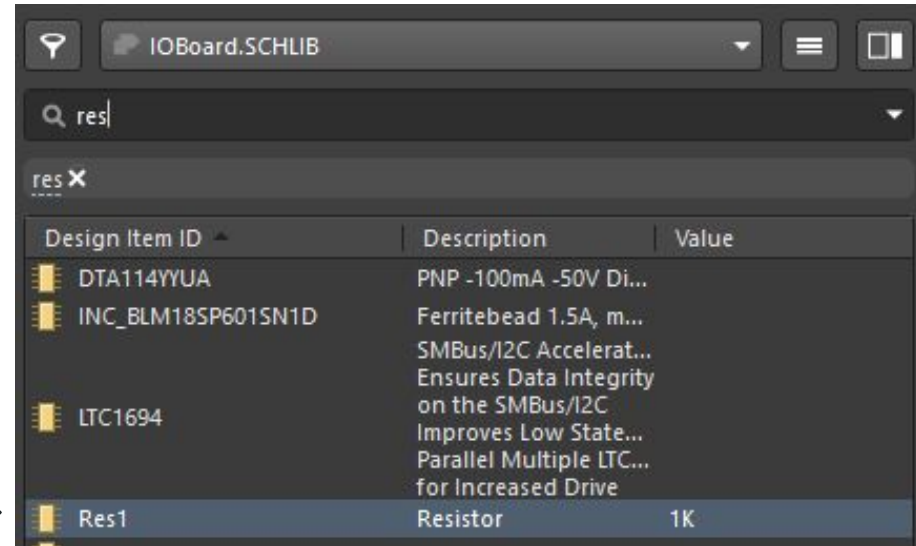
A new component should appeared



Adding Resistor

Repeat the process but search “res” in the search field

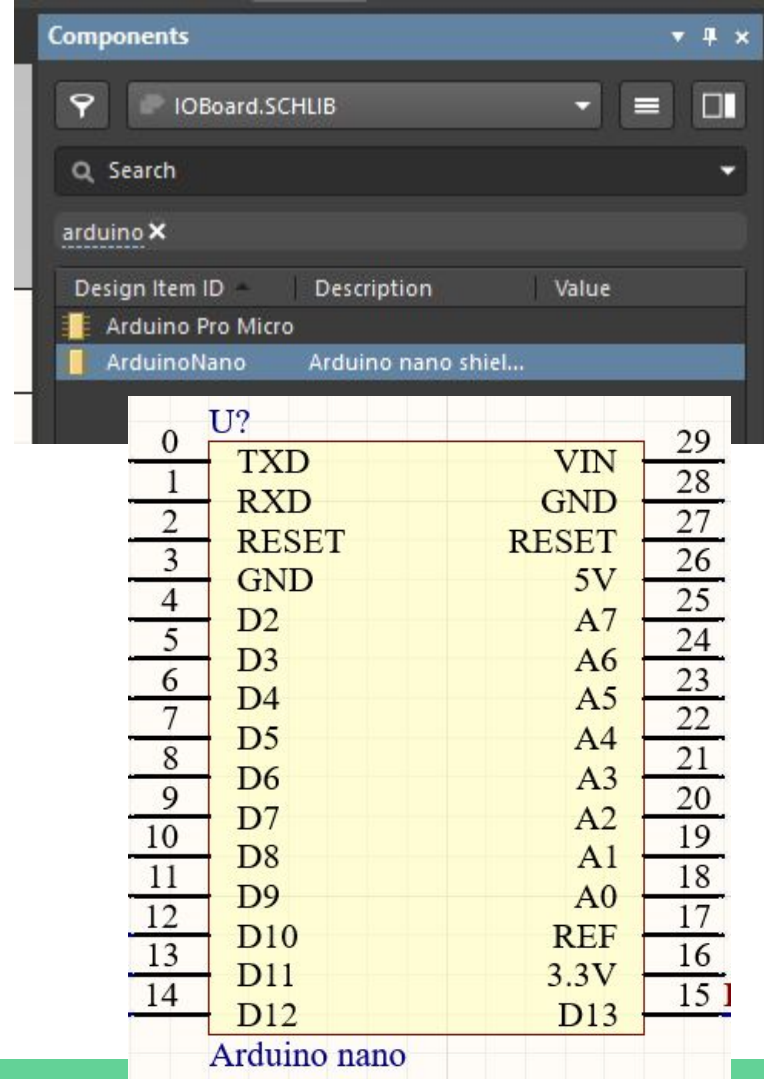
Place “Res1” onto the schematic



Adding Arduino Nano

Search “Arduino”

Select and place the Arduino nano



The screenshot shows the Altium Designer Components panel with the search term 'arduino' entered. The results list includes 'Arduino Pro Micro' and 'ArduinoNano'. The 'ArduinoNano' component is selected, showing its description 'Arduino nano shiel...'. Below the panel, a pinout table for the Arduino Nano is displayed.

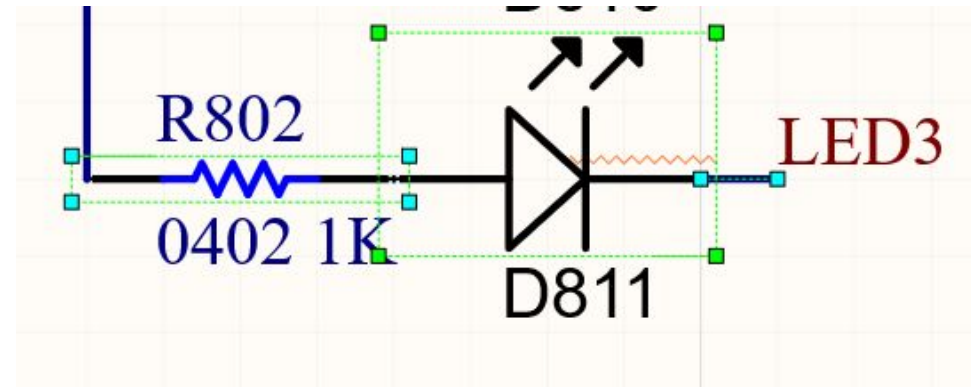
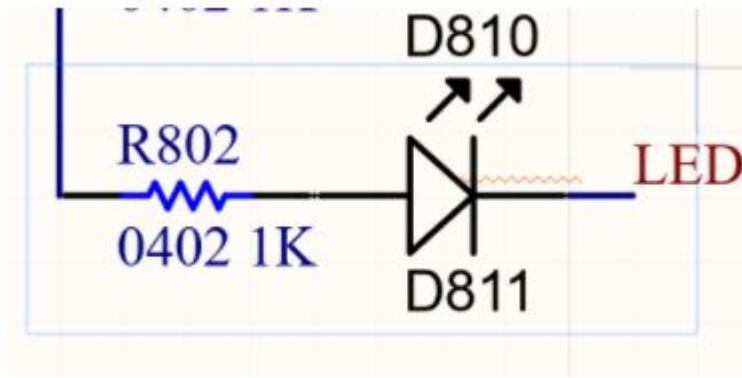
Pin	Signal	Signal	Pin
0	TXD	VIN	29
1	RXD	GND	28
2	RESET	RESET	27
3	GND	5V	26
4	D2	A7	25
5	D3	A6	24
6	D4	A5	23
7	D5	A4	22
8	D6	A3	21
9	D7	A2	20
10	D8	A1	19
11	D9	A0	18
12	D10	REF	17
13	D11	3.3V	16
14	D12	D13	15

Arduino nano

Selecting Components (Dragging from left to right)

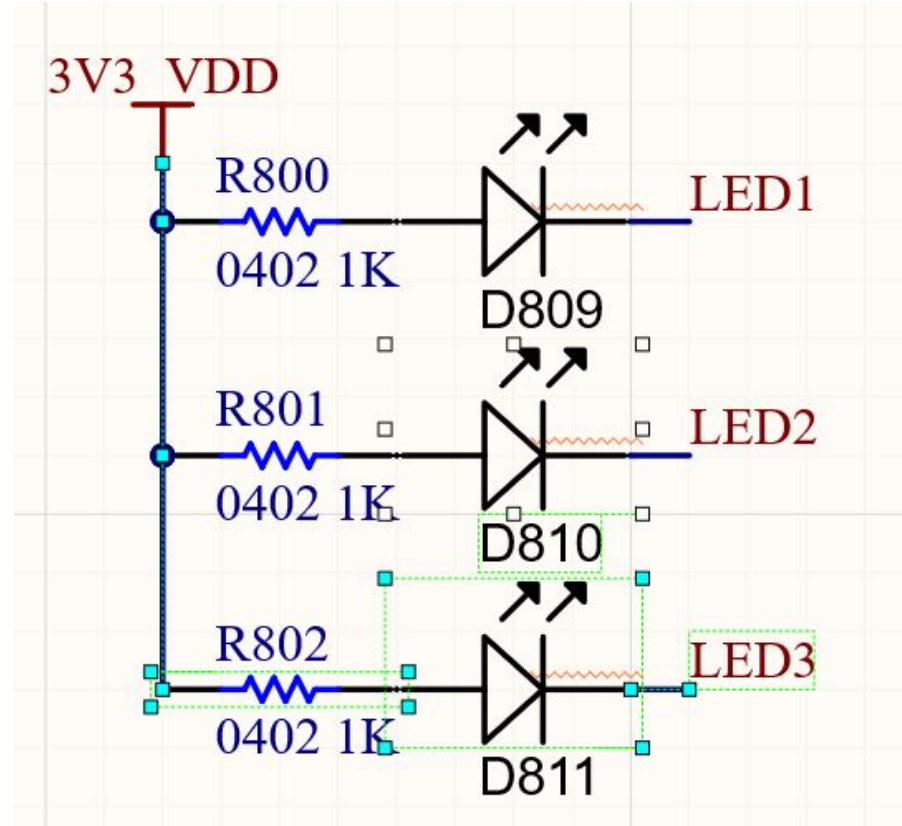
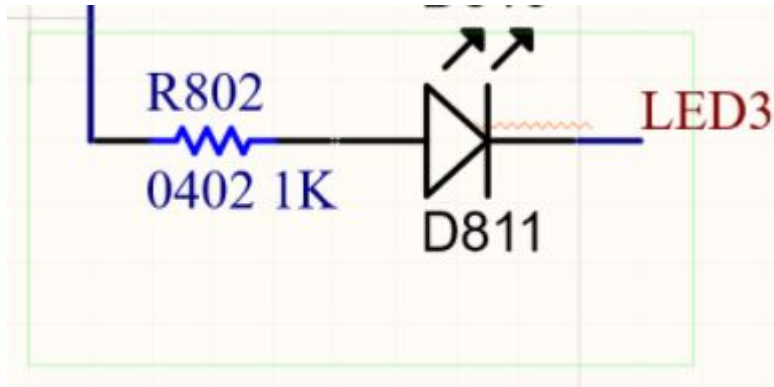
The color of the circled area will be blue

It will only select objects that are fully enclosed in the area



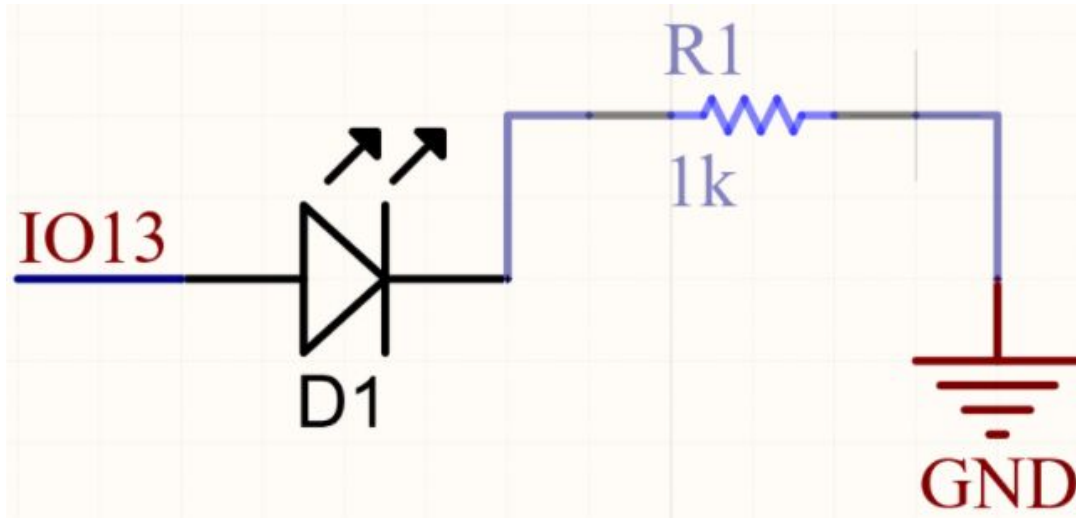
Selecting Components (Drag from right to left)

The color of the circled area will be green
It will only select objects that are in touch with the area



Dragging Components

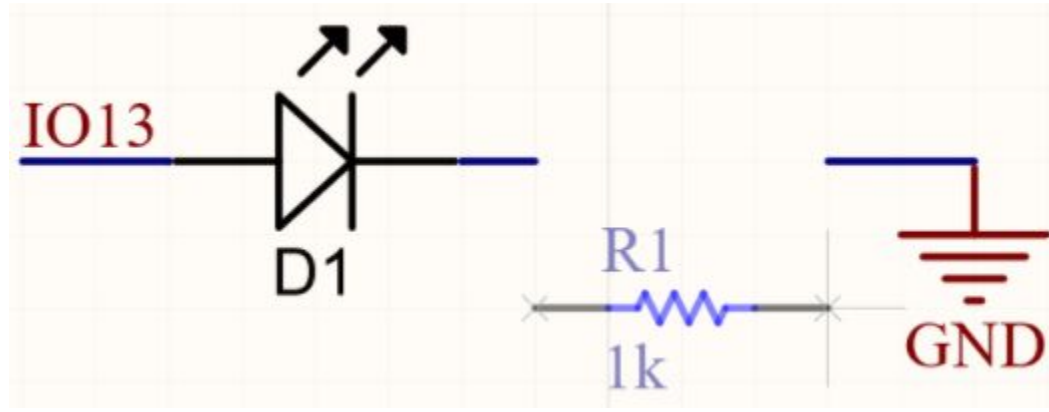
After selecting the objects, you could move them by holding LMB and dragging to the desired position. The object and the wire connecting to it will be transparent.



Dragging Components (Disconnect from wire)

To break the connection and drag the component away, use (Ctrl + LMB) to drag them out.

The component being dragged should not be connected to the wires or other components



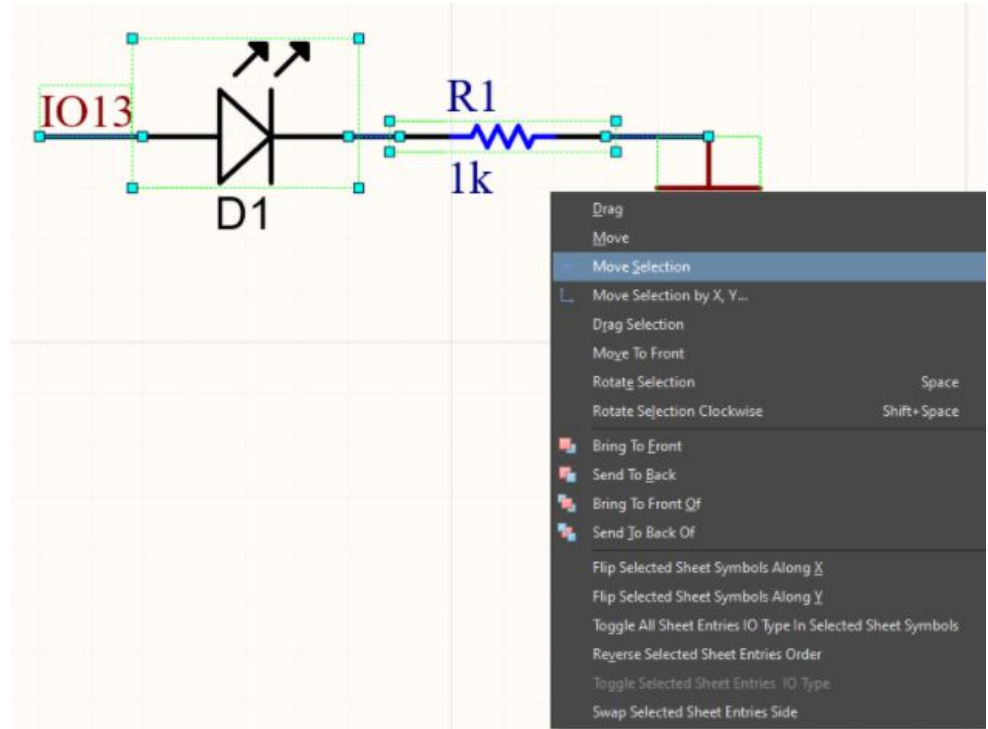
Dragging Components (Snap Grid)

The components will snap onto the grid according to the snap grid dimension that you have setted in the property of the schematic sheet, you could change the snap grid by pressing 'g'.

Dragging Components (with move command)

After selecting the objects, they could be moved with either Move Selection (m , s) or with Drag Selection (m, r)

Personally i prefer using Move Selection command, and move them around with arrow keys when i need higher precision.



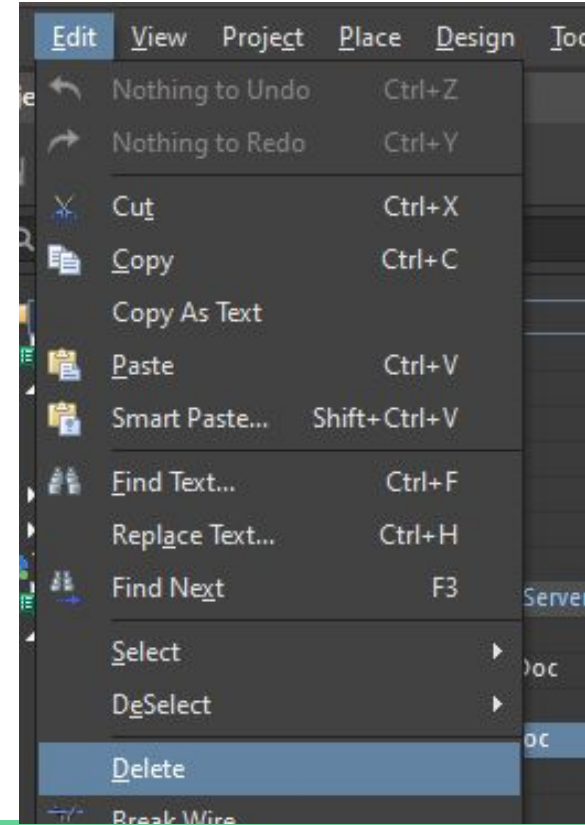
Delete, Ctrl + z and Ctrl + y

After selecting the objects that you want to delete, click (Del) on keyboard to remove the Objects

You could also delete it by using Edit -> Delete (E, D) command

You could undo your action with (Ctrl + z)

You could redo your action with (Ctrl + y)



Delete, Ctrl + z and Ctrl + y

Sometimes when working with larger and multiple files, the undo and redo function will not work because of system resource running out. The only way to regain this function is restarting the software (If you are using VM, it may cause the VM to freeze if not enough resource is allocated to it)

Having a computer with better specs could lessen the chance of this bug happening

TLDR Get better PC lol



Editing Schematic (LED example)

Components could be rotated using space bar

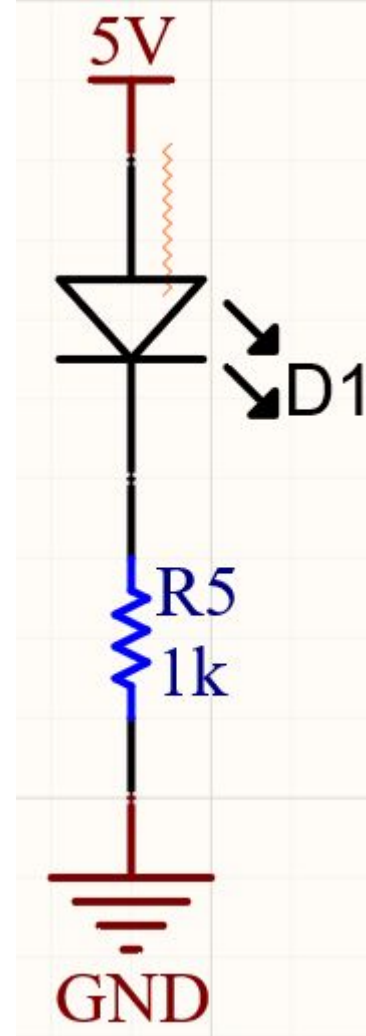
- Each click will turn it by 90 degrees

Connect the pins using “Wire”

Refer to the figure, first add a Power Port with the name “GND” and style “Power Ground”, and another one with the name “5V” and style “Bar” and connect them to the diode and ground respectively

Use Del, redo, undo to correct your mistakes

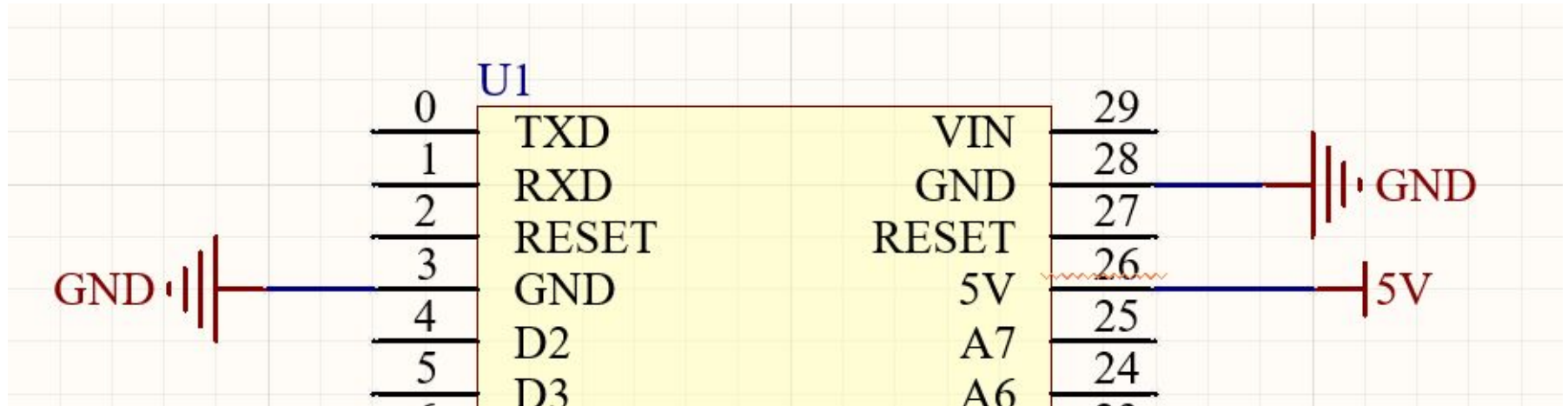
The result should be similar to the picture on the right



Adding Power Ports to Arduino

Add Power Port with the name “GND” and style “Power Ground” to the pins with the naming GND

Add Power Port with the name “5V” and style “Bar” to the pins with the name 5V

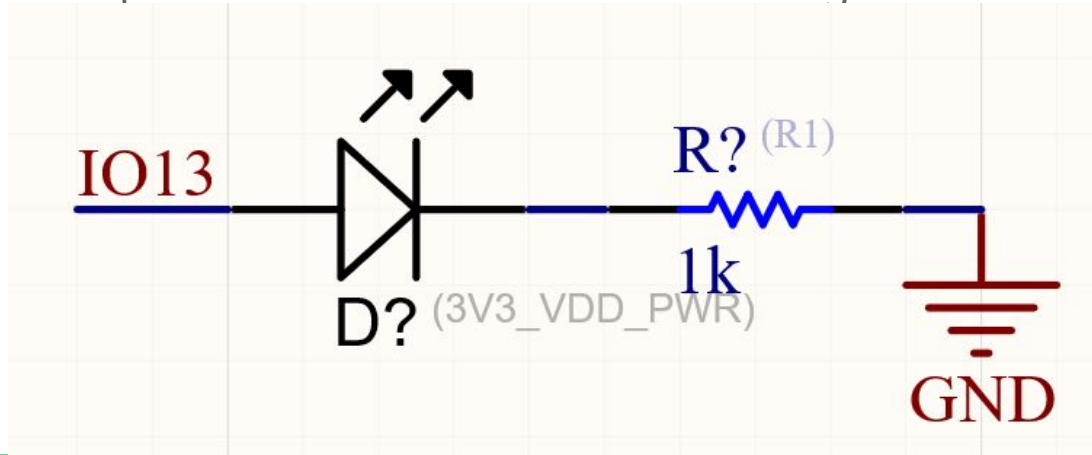


Adding LED to GPIO13 of Nano

Replace the 5V power port with a NetLabel with name IO13

After placing the components, you should see all the designators have a question mark at the end


Different kind of components has different default designator as well



Don't forgot to add the Net Label on the Arduino as well.

Add the NetLabel with the name "IO13" onto D13 on Arduino

With the same Net Label, GPIO13 of Arduino is now connected to the LED in the schematic

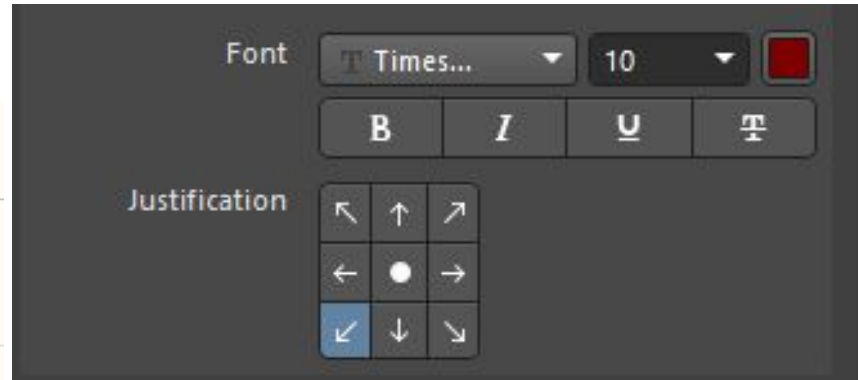


VIN	29
GND	28
RESET	27
5V	26
A7	25
A6	24
A5	23
A4	22
A3	21
A2	20
A1	19
A0	18
REF	17
3.3V	16
D13	15
	<u>IO13</u>

Font Size and Justification of Net Label

Make sure the justification is bottom left and the font size is 10 for Net Labels, since those are most common found in schematic

The bottom left picture shows net label with bottom left justification (look at the grey cross), while the middle picture shows justification at top right



Standard Reference Designator (Extra Info)

D for diode, R for resistor, C for capacitor, L for inductor etc.

Refer to the link below

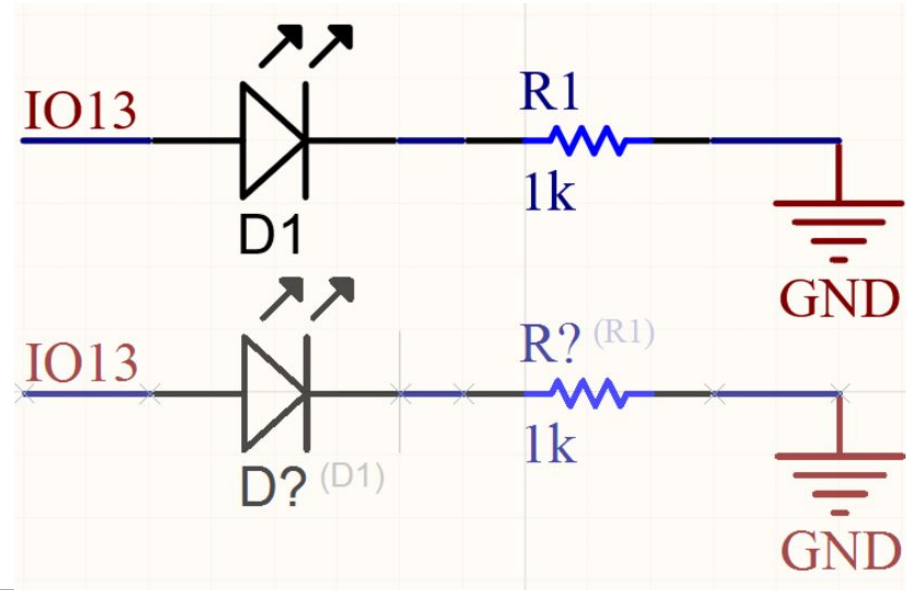
<https://dexpcb.com/manual/standard-reference-designators.htm>

Copy pasting (Ctrl + x) (Ctrl + c) (Ctrl + v)

Select the objects that you want to copy,
and use Ctrl + C and Ctrl + V to paste it.

Ctrl + X for cut and paste

NetLabels and power port will remain same,
while component designator will be reset to
?

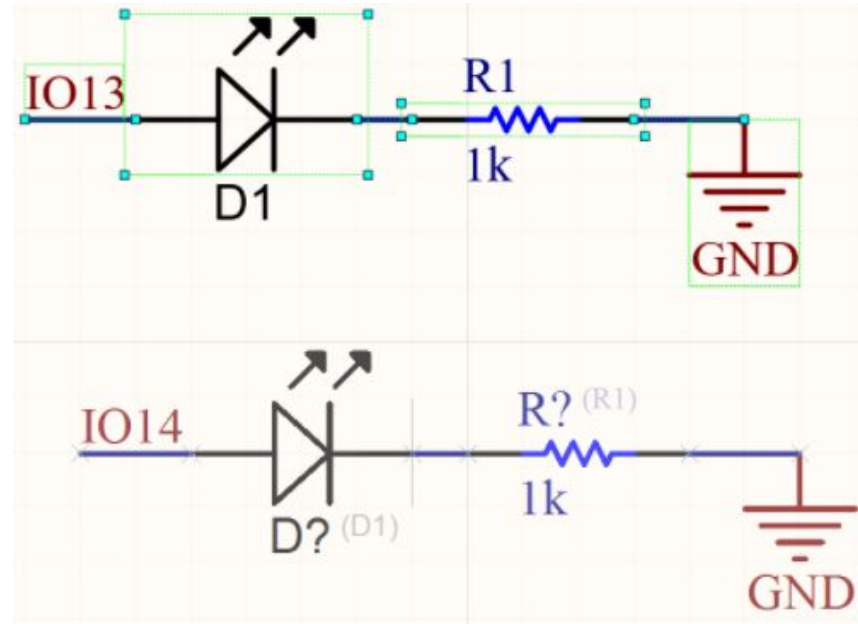


Copy pasting (Shift + LMB)

Selected objects could be also duplicated when you use hold left shift key and try to drag it

Designator will ?, but the Netlabel will be incremented by 1 if it ends with a number

IO13 being incremented to IO14



Smart Paste (Ctrl + C then Ctrl + Shift + V)

To keep the old designator, smart paste is required when copying the components.

Select the components that needs to be copied and press (Ctrl + C)

Then press (Ctrl +Shift + V) to get the smart paste manual

See next page >>

Smart Paste

Press “OK” at the
bottom right corner

Smart Paste

Choose the objects to paste

Schematic Object Type	Count
<input checked="" type="checkbox"/> Wires	3
<input checked="" type="checkbox"/> Net Labels	1
<input checked="" type="checkbox"/> Parts	2
<input checked="" type="checkbox"/> Power Objects	1

Windows Clipboard Contents	Count
<input type="checkbox"/> Pictures	1

Choose Paste Action

Paste As

Themselves

Net Labels

Ports

Cross Sheet Connectors

Sheet Entries

Harness Entries

Ports and Wires

Net Labels and Wires

Ports, Wires and NetLabels

Labels

Text Frames

Notes

No Options Available

Paste Array

☐ Enable Paste Array

Columns

Count1

Spacing0mil

Rows

Count8

Spacing0mil

Text Increment

DirectionNone

Primary1

Secondary1

☐ Remove Leading Zeroes

Summary

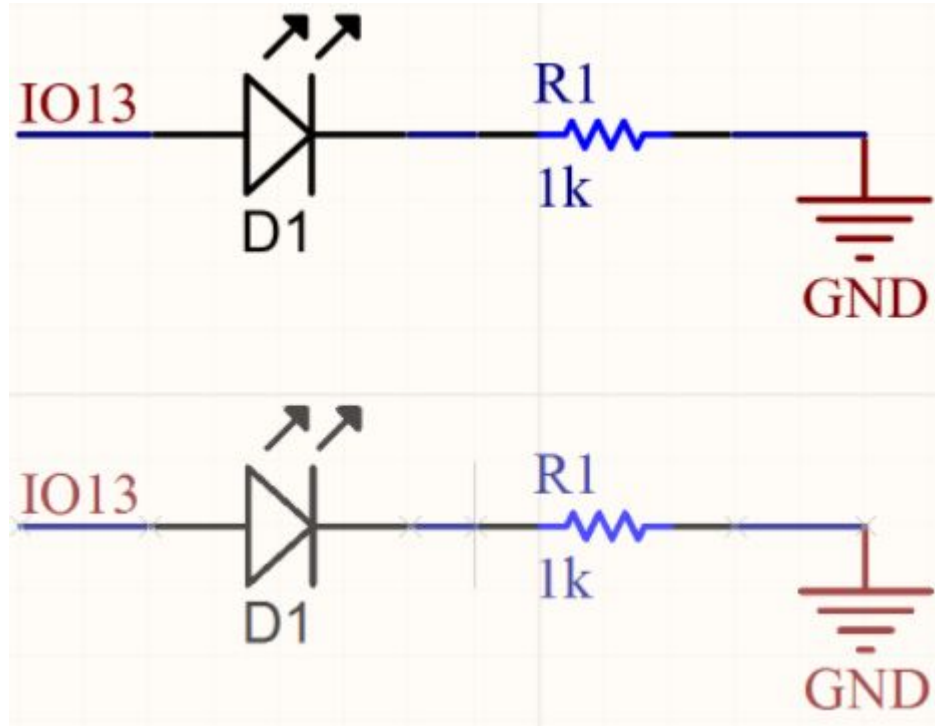
Paste 1 Netlabel and 1 Power Object as they are.

OK

Cancel

Smart Paste

Duplicated components with the same designator instead with “?”



Smart Paste

Smart paste offers a lot of other functions, but you will need to check for those functions yourself on the road.

- Copy NetLabels and convert them to Text Frame
- Copy NetLabels and convert them to Ports
- etc

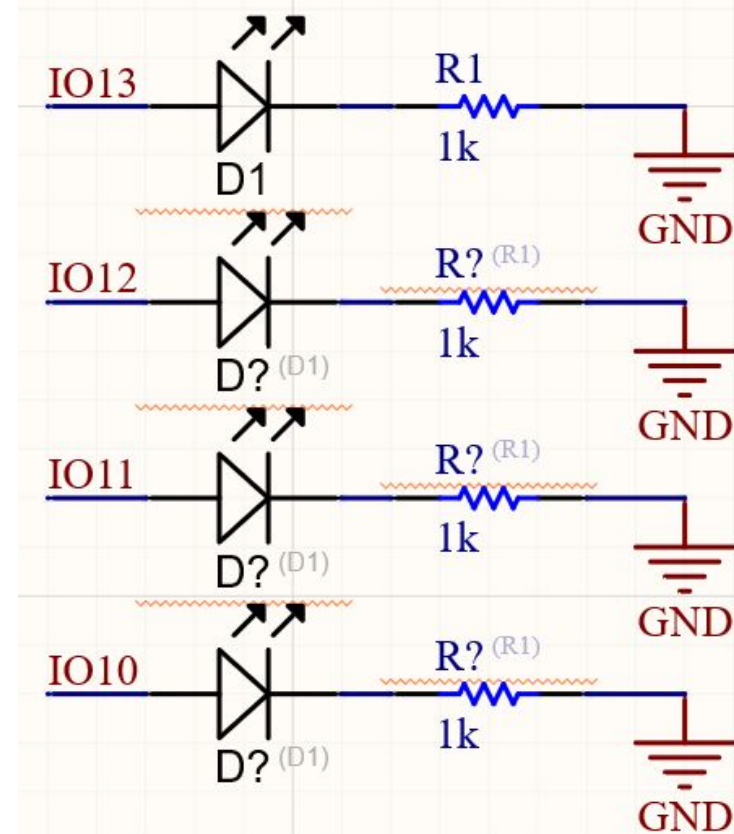
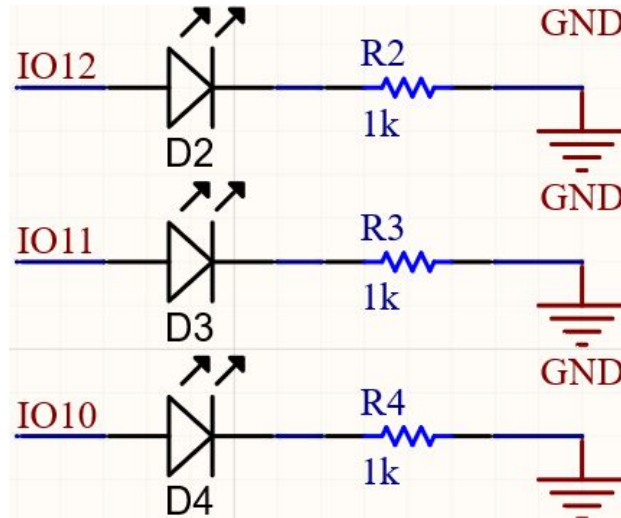
Edit the Schematic

Annotate the designators in the schematic

Adding more LED

After copy and pasting several times, there will be new components with ? in their designator.

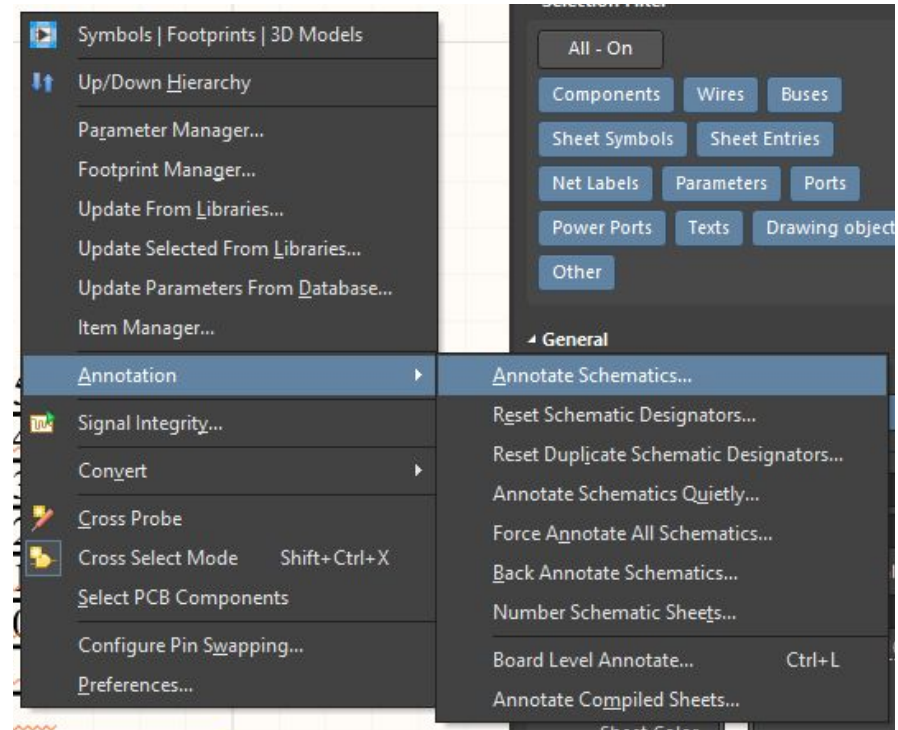
Use the Annotate tool to assign unique designator



Annotate Schematic

Annotate the schematic to assign unique designator to the components

Tool -> Annotation -> Annotate schematic (t, a, a)



Annotate Schematic

Schematic Annotation Configuration

Order of Processing
Across Then Down

Matching Options
Complete Existing Packages: None

Process Location of
Designator

Replace Sub-Parts
Off

Component Parameter (Strictly)

- ☐ Color
- ☒ Comment
- ☐ Component Kind
- ☐ Description
- ☐ Footprint
- ☐ Ibis Model
- ☐ Library Name
- ☒ Library Reference
- ☐ Manufacturer
- ☐ Manufacturer 1
- ☐ Manufacturer Part Number 1
- ☐ Material
- ☐ Package / Case

Schematic Sheets To Annotate

Schematic Sheet	Annotation Scope	Order	Start Index	Add Suffix
<input checked="" type="checkbox"/> ArduinoUnoShield.SchDoc	All	0	<input type="checkbox"/> 1	

Proposed Change List

Current Designator	Sub	Proposed Designator	Sub	Location of Part
<input checked="" type="checkbox"/> AD	<input checked="" type="checkbox"/>	AD		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> D?	<input type="checkbox"/>	D?		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> ICSP	<input checked="" type="checkbox"/>	ICSP		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> IOH	<input type="checkbox"/>	IOH		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> IOL	<input checked="" type="checkbox"/>	IOL		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> POWER	<input checked="" type="checkbox"/>	POWER		ArduinoUnoShield.SchDoc
<input checked="" type="checkbox"/> R?	<input type="checkbox"/>	R?		ArduinoUnoShield.SchDoc

Annotation Summary

Annotation is enabled for all schematic documents. Parts will be matched using 2 parameters, all of which will be strictly matched. (Under strict matching, parts will only be matched together if they all have the same parameters and parameter values, with respect to the matching criteria. Disabling this will extend the semantics slightly by allowing parts which do not have the specified parameters to be matched together.) Existing packages will not be completed. All new parts will be put into new packages.

All OnAll Off










Update Changes ListReset AllBack AnnotateAccept Changes (Create ECO)

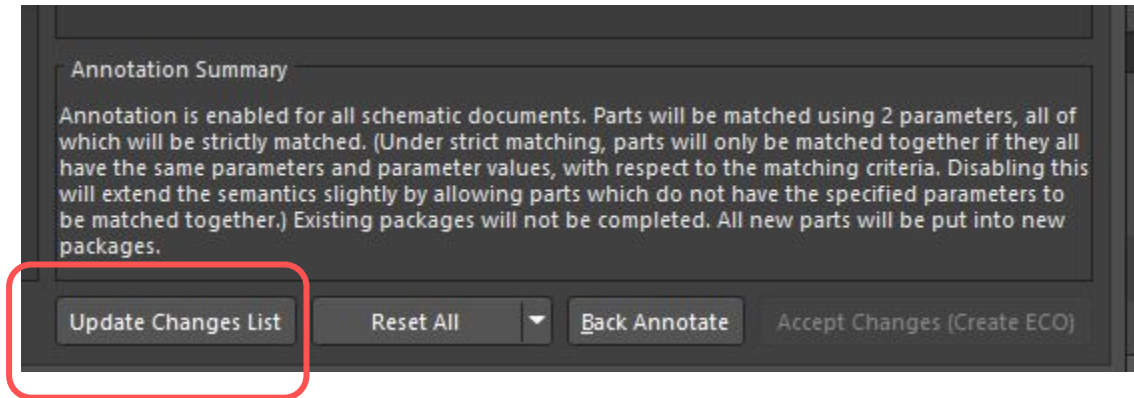
Close

Annotate Schematic

All the components could be seen on the top right corner

Press “Update Changes List” on the bottom right corner

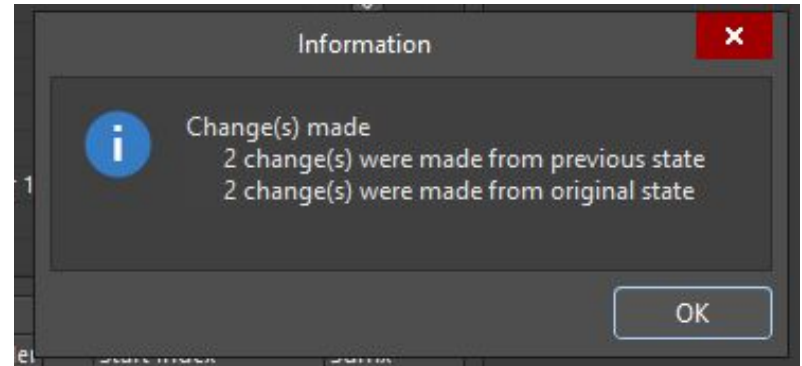
Current			Proposed		Location of Part
	Designator	 Sub	Designator	Sub	Schematic Sheet
	AD	<input checked="" type="checkbox"/>	AD		ArduinoUnoShield.SchDoc
	D?	<input type="checkbox"/>	D?		ArduinoUnoShield.SchDoc
	ICSP	<input checked="" type="checkbox"/>	ICSP		ArduinoUnoShield.SchDoc
	IOH	<input type="checkbox"/>	IOH		ArduinoUnoShield.SchDoc
	IOL	<input checked="" type="checkbox"/>	IOL		ArduinoUnoShield.SchDoc
	POWER	<input checked="" type="checkbox"/>	POWER		ArduinoUnoShield.SchDoc
	R?	<input type="checkbox"/>	R?		ArduinoUnoShield.SchDoc



Annotate Schematic

System will pop a notification with the total number of changes of designators

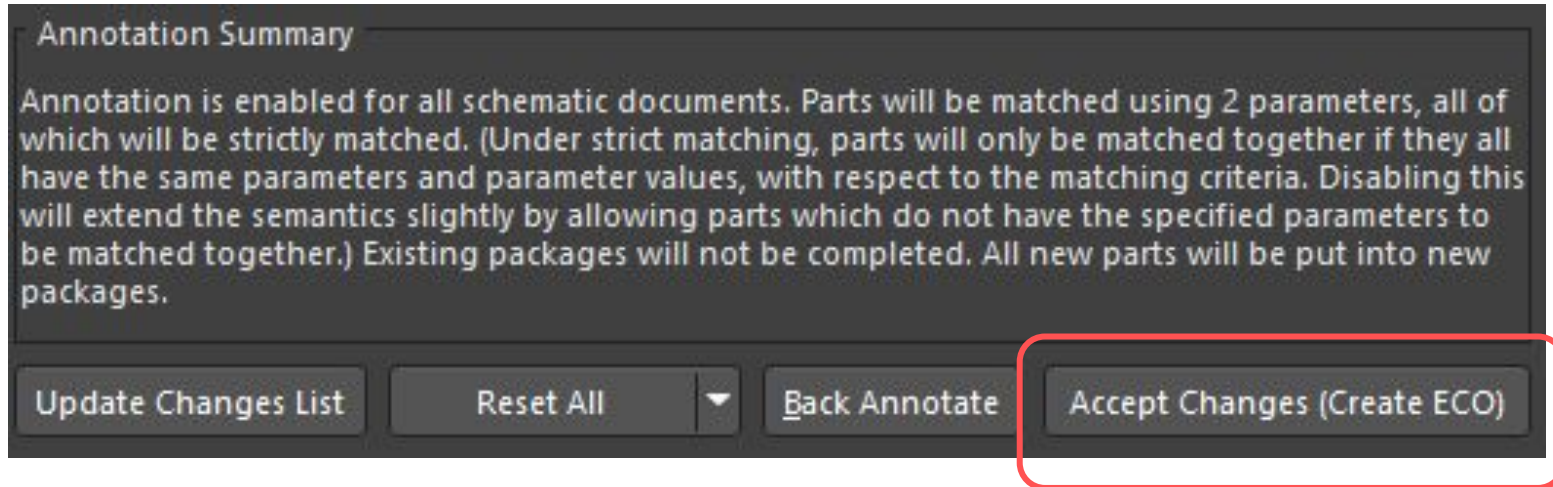
In the table at the top right corner, the proposed changes could be preview



Current				Proposed	
	Designator		Sub	Designator	
<input type="checkbox"/>	AD	<input checked="" type="checkbox"/>		AD	
<input type="checkbox"/>	D?	<input type="checkbox"/>		D1	
<input type="checkbox"/>	ICSP	<input checked="" type="checkbox"/>		ICSP	
<input type="checkbox"/>	IOH	<input type="checkbox"/>		IOH	
<input type="checkbox"/>	IOL	<input checked="" type="checkbox"/>		IOL	
<input type="checkbox"/>	POWER	<input checked="" type="checkbox"/>		POWER	
<input type="checkbox"/>	R?	<input type="checkbox"/>		R1	

Annotate Schematic

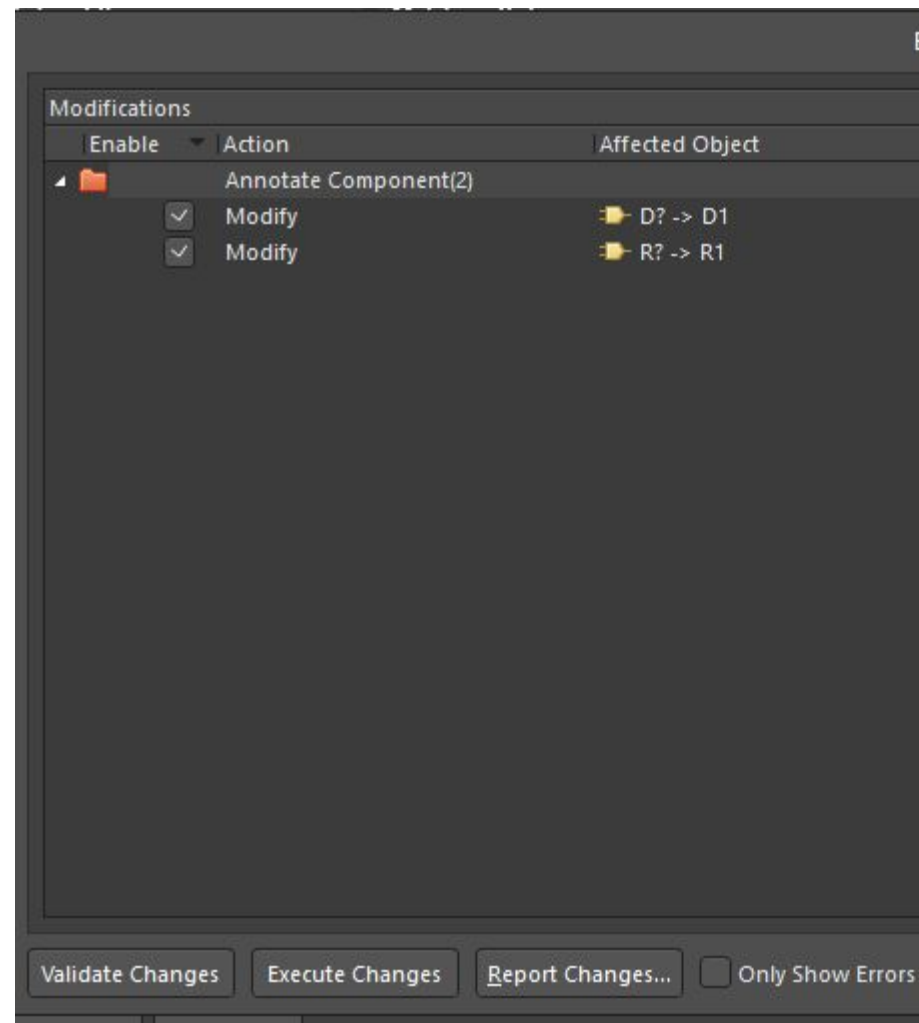
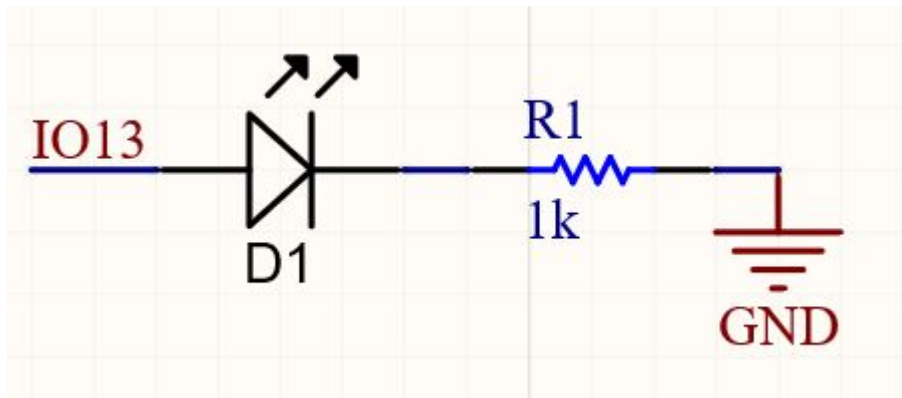
Click the “Accept Changes (Create ECO)” button to apply the changes to the schematic



Annotate Schematic

The changes could be previewed again before clicking the “Execute Changes” button

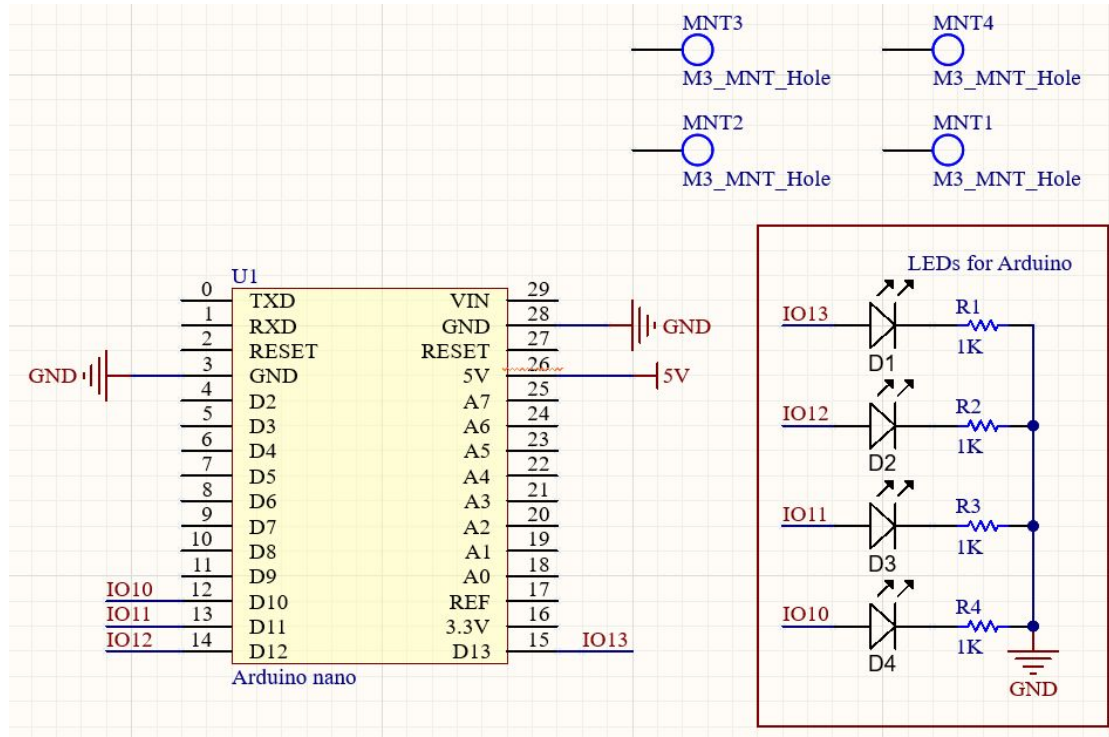
The components’ designators should be updated



Edit the Schematic

Copy/paste more LEDs and Annotate them

With the components placed and connected, we could move on to import those changes to PCB



Move to part (II) for PCB
