Altium Guide

UMSATS

Author:
Aleksa Svitlica

Last Updated: March 18, 2018

Contents

List of Figures

Chapter 1

Getting Started

First two important notes:

1. I am using Altium Designer 17.1
2. I am writing this as I learn Altium so expect mistakes. If something seems wrong it probably is.

1.1 Creating a new PCB project

- File->New->Project...
- Select PCB project from the project types options on the left side.
- For project templates simply choose <Default> because we will be importing a STEP file later.

If Altium is giving you errors when you try to create a new project that probably means you haven't selected a valid license. To fix this you can select View->Home to open up the Home page. Then click on the Admin tab, make sure you are logged in and select a license from below.

1.2 Create a PCB file

Click File->New->PCB. This should result in a screen like below:

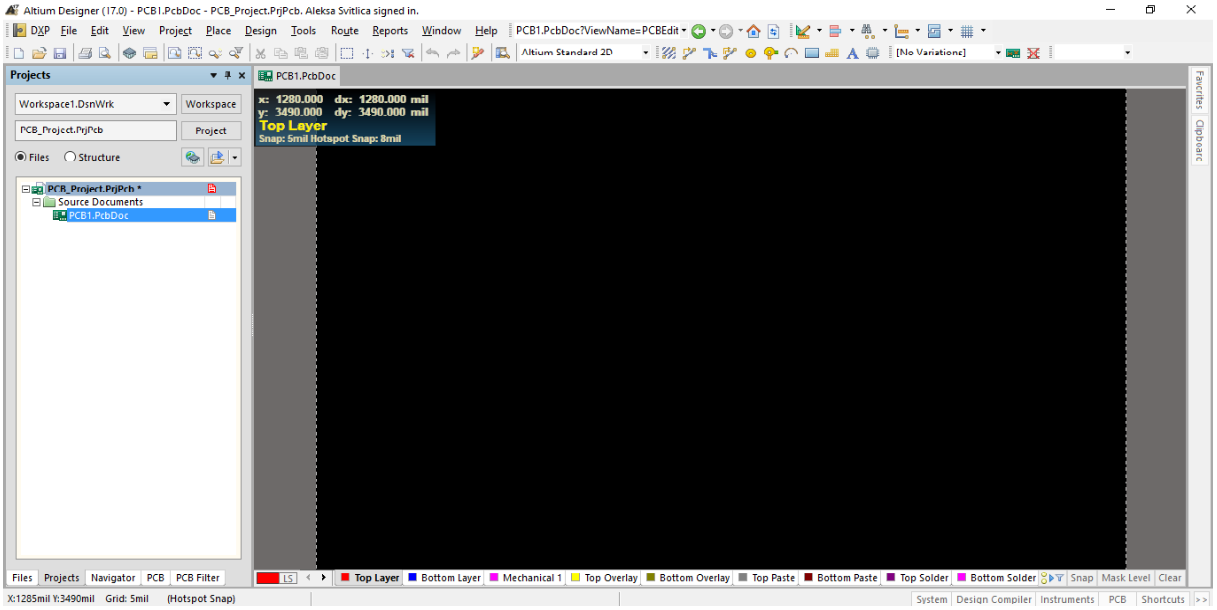
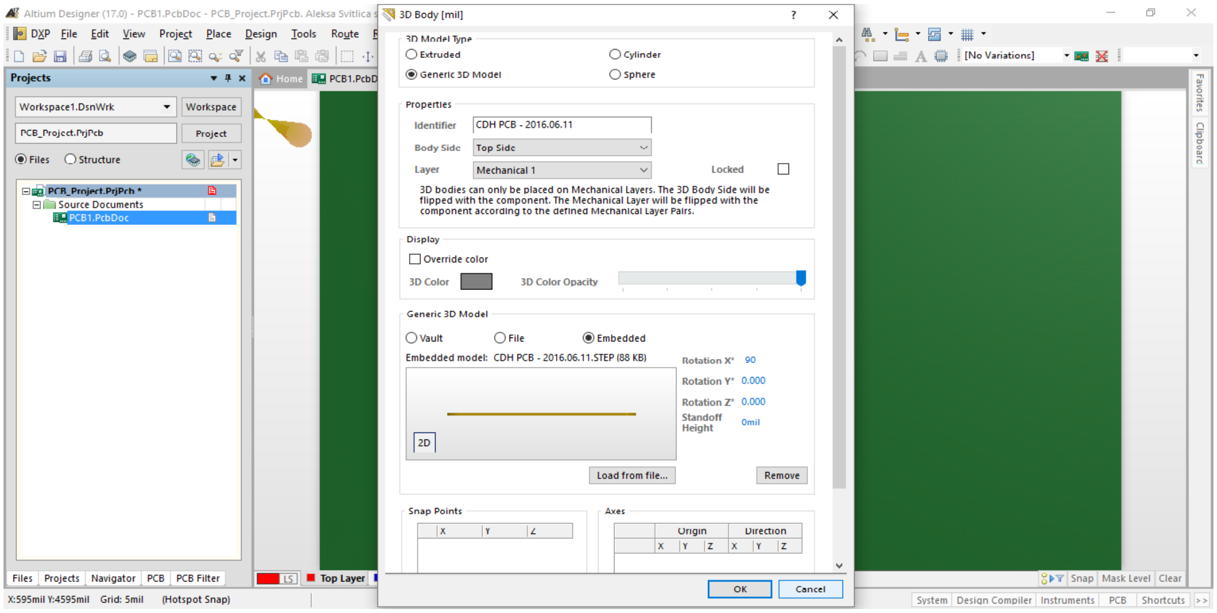

Altium Designer (17.0) - PCB1.PcbDoc - PCB_Project.PrjPcb. Aleksa Svitlica signed in.
DXP File Edit View Project Place Design Tools Route Reports Window Help
PCB1.PcbDoc?ViewName=PCBEdit
Altium Standard 2D
[No Variations]
Projects
Workspace1.DsnWrk
Workspace
PCB_Project.PrjPcb
Project
Files
Structure
PCB_Project.PrjPcb *
Source Documents
PCB1.PcbDoc
x: 1280.000 dx: 1280.000 mil
y: 3490.000 dy: 3490.000 mil
Top Layer
Snap: 5mil Hotspot Snap: 8mil
Favorites
Clipboard
Files Projects Navigator PCB PCB Filter
LS
Top Layer
Bottom Layer
Mechanical 1
Top Overlay
Bottom Overlay
Top Paste
Bottom Paste
Top Solder
Bottom Solder
Snap Mask Level Clear
X:1285mil Y:3490mil Grid: 5mil (Hotspot Snap)
System Design Compiler Instruments PCB Shortcuts >>

Figure 1.1: New PCB file.

The next step is import our step file and use that to create the physical PCB shape. To do this first click 3 to switch to 3D mode, you can switch between 2D and 3D by pressing 2 or 3. Then click Place->3D Body to bring up a new menu. In this menu there are a couple things we need to change:

- 3D Model Type: Select Generic 3D model
- Further down the window there will now be an option to select a file to import for the generic 3D model. Select embedded and then choose the correct STEP file provided to you by the mechanical team.
- After loading your STEP file there should be a model displayed in the window, if it looks like a horizontal line don't worry, we just need to rotate it. In rotation x simple put 90 or -90.


Altium Designer (17.0) - PCB1.PcbDoc - PCB_Project.PrjPcb. Aleksa Svitlica
DXP File Edit View Project Place Design Tools Route
Projects
Workspace1.DsnWrk
Workspace
PCB_Project.PrjPcb
Project
Files
Structure
PCB_Project.PrjPcb *
Source Documents
PCB1.PcbDoc
Files Projects Navigator PCB PCB Filter
Home
Top Layer
[No Variations]
Snap Mask Level Clear
System Design Compiler Instruments PCB Shortcuts
Favorites Clipboard
X:595mil Y:4595mil Grid: 5mil (Hotspot Snap)

3D Body [mil]

3D Model Type
○ Extruded ○ Cylinder
● Generic 3D Model ○ Sphere

Properties
Identifier: CDH PCB - 2016.06.11
Body Side: Top Side
Layer: Mechanical 1
Locked □

3D bodies can only be placed on Mechanical Layers. The 3D Body Side will be flipped with the component. The Mechanical Layer will be flipped with the component according to the defined Mechanical Layer Pairs.

Display
□ Override color
3D Color
3D Color Opacity

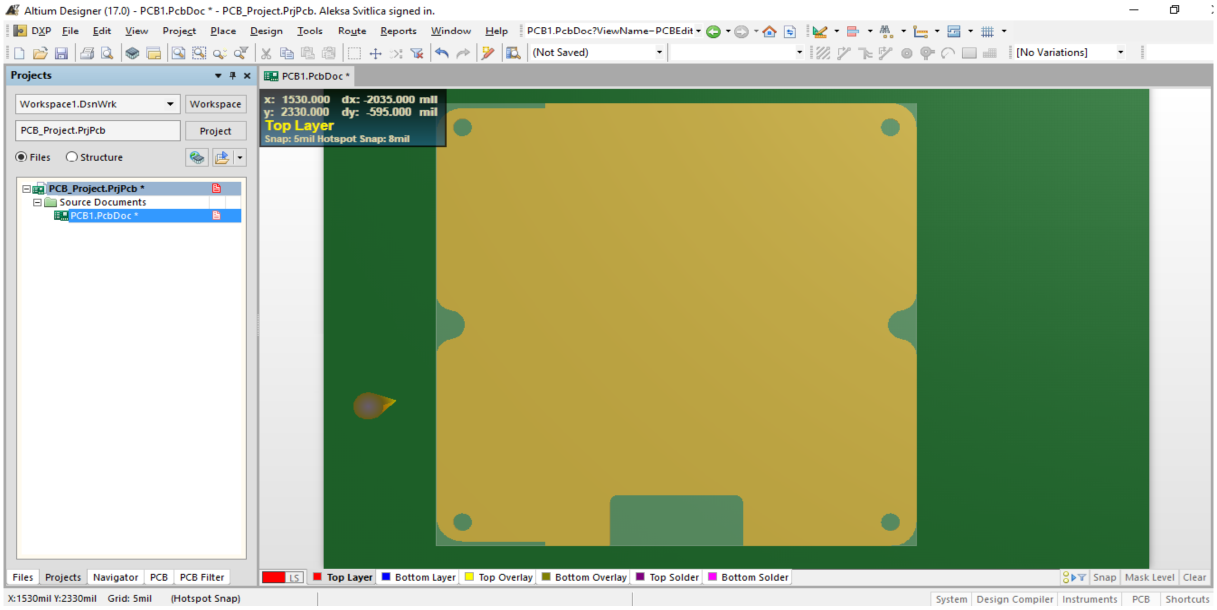
Generic 3D Model
○ Vault ○ File ● Embedded
Embedded model: CDH PCB - 2016.06.11.STEP (88 KB)
Rotation X° 90
Rotation Y° 0.000
Rotation Z° 0.000
Standoff Height 0mil
2D
Load from file... Remove

Snap Points: X Y Z
Axes: Origin (X Y Z) Direction (X Y Z)

OK Cancel

Figure 1.2: Importing STEP file as 3D body.

Click OK and left click again to place your 3D body down, where you place it doesn't matter for now.

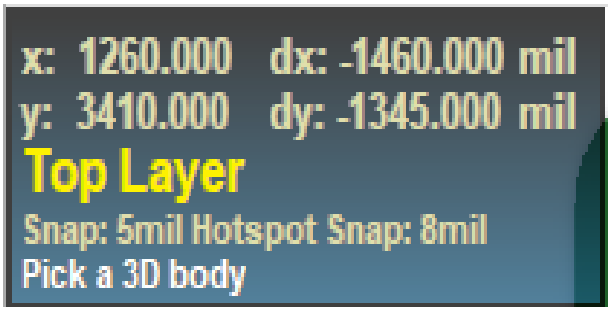

Altium Designer (17.0) - PCB1.PcbDoc * - PCB_Project.PrjPcb. Aleksa Svitlica signed in.
DXP File Edit View Project Place Design Tools Route Reports Window Help
PCB1.PcbDoc?ViewName=PCBEdit
(Not Saved)
[No Variations]
Projects
Workspace1.DsnWrk
Workspace
PCB_Project.PrjPcb
Project
Files
Structure
PCB_Project.PrjPcb *
Source Documents
PCB1.PcbDoc *
Files Projects Navigator PCB PCB Filter
Top Layer Bottom Layer Top Overlay Bottom Overlay Top Solder Bottom Solder
Snap Mask Level Clear
System Design Compiler Instruments PCB Shortcuts

x: 1530.000 dx: -2035.000 mil
y: 2330.000 dy: -595.000 mil
Top Layer
Snap: 5mil Hotspot Snap: 8mil

X:1530mil Y:2330mil Grid: 5mil (Hotspot Snap)

Figure 1.3: Placing 3D body.

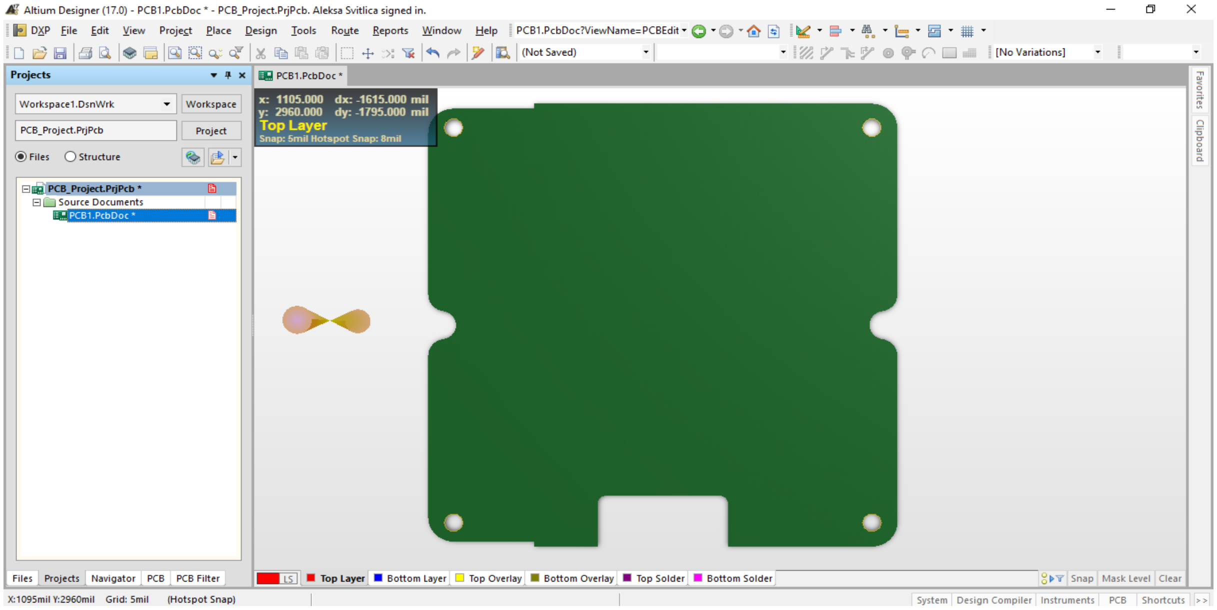
Now to make your PCB match this 3D body go to Design->Board Shape->Define From 3D Body. If this is option is unclickable for you make sure you are in 3D mode and try again. Your cursor should look like a + symbol and that window which says Top Layer in bold yellow words should also say Pick a 3D Body.



```
x:  1260.000   dx: -1460.000 mil
y:  3410.000   dy: -1345.000 mil
Top Layer
Snap: 5mil Hotspot Snap: 8mil
Pick a 3D body
```

Figure 1.4: Pick 3D body.

Click on the 3D body you just placed, the words in that window should now say select face, click again on the 3D body. It will open up a couple windows, just click cancel or closed until you see the following screen.



Altium Designer (17.0) - PCB1.PcbDoc * - PCB_Project.PrjPcb. Aleksa Svitlica signed in.

```
x:  1105.000   dx: -1615.000 mil
y:  2960.000   dy: -1795.000 mil
Top Layer
Snap: 5mil Hotspot Snap: 8mil
```

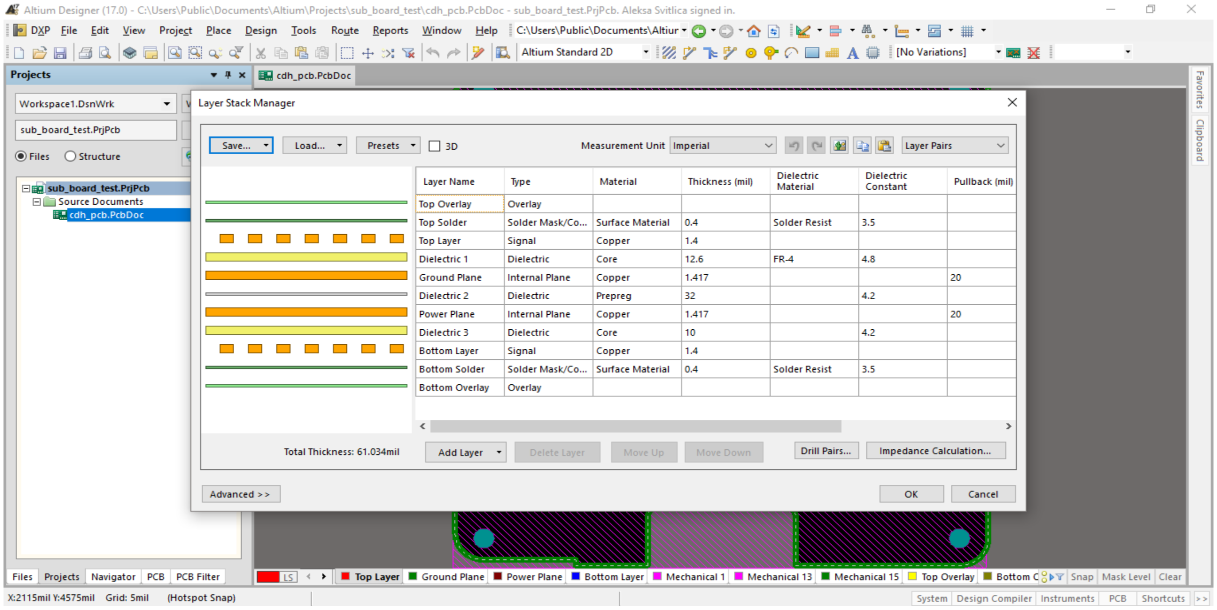
X:1095mil Y:2960mil   Grid: 5mil   (Hotspot Snap)

Figure 1.5: Finished placing 3D body.

The PCB shape is now defined, we are ready to define layers.

1.3 Layer Stack Manager

To access the layer stack manager, select Design->Layer Stack Manager. Here you can define each layer of your PCB, it will default to a 2 layer setup. Even though you see a bunch of layers listed, only the copper layers actually count when we say 2-layer board. I can't offer any suggestions about how many layers you should use, what thickness or what material since this is a topic I know almost nothing about. For reference I've included a picture of the 4-layer subsystems board that Mike Lambeta designed.



Altium Designer (17.0) - C:\Users\Public\Documents\Altium\Projects\sub_board_test\cdh_pcb.PcbDoc - sub_board_test.PrjPcb. Aleksa Svitlica signed in.

Layer Stack Manager

Measurement Unit: Imperial

Layer Name	Type	Material	Thickness (mil)	Dielectric Material	Dielectric Constant	Pullback (mil)
Top Overlay	Overlay					
Top Solder	Solder Mask/Co...	Surface Material	0.4	Solder Resist	3.5	
Top Layer	Signal	Copper	1.4			
Dielectric 1	Dielectric	Core	12.6	FR-4	4.8	
Ground Plane	Internal Plane	Copper	1.417			20
Dielectric 2	Dielectric	Prepreg	32		4.2	
Power Plane	Internal Plane	Copper	1.417			20
Dielectric 3	Dielectric	Core	10		4.2	
Bottom Layer	Signal	Copper	1.4			
Bottom Solder	Solder Mask/Co...	Surface Material	0.4	Solder Resist	3.5	
Bottom Overlay	Overlay					

Total Thickness: 61.034mil

X:2115mil Y:4575mil Grid: 5mil (Hotspot Snap)

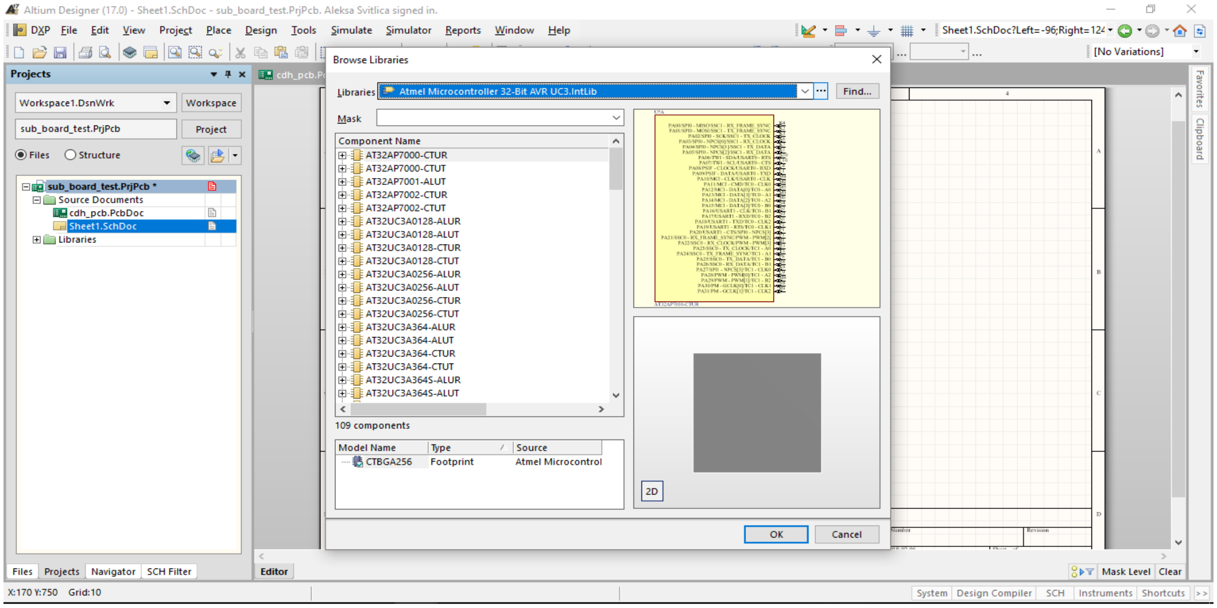
Figure 1.6: Layer stack manager.

Chapter 2

Parts Libraries

2.1 Importing Libraries

I would recommend downloading the libraries Altium provides on their website. It is much easier to work with preexisting models since making components is a hassle. You can store the libraries anywhere on your computer and import the ones you need in your project. Click on Place->Part... and then select Choose. This will bring up a menu like in figure 2.1, click on the three dots to the left of find to bring up the library import menu. After importing, the libraries will be selectable in the menu seen below and individual parts can be selected.



Altium Designer (17.0) - Sheet1.SchDoc - sub_board_test.PrjPcb. Aleksa Svitlica signed in.

Browse Libraries

Libraries: Atmel Microcontroller 32-Bit AVR UC3.IntLib

Mask:

Component Name
- AT32AP7000-CTUR
- AT32AP7000-CTUT
- AT32AP7001-ALUT
- AT32AP7002-CTUR
- AT32AP7002-CTUT
- AT32UC3A0128-ALUR
- AT32UC3A0128-ALUT
- AT32UC3A0128-CTUR
- AT32UC3A0128-CTUT
- AT32UC3A0256-ALUR
- AT32UC3A0256-ALUT
- AT32UC3A0256-CTUR
- AT32UC3A0256-CTUT
- AT32UC3A364-ALUR
- AT32UC3A364-ALUT
- AT32UC3A364-CTUR
- AT32UC3A364-CTUT
- AT32UC3A364S-ALUR
- AT32UC3A364S-ALUT

109 components

Model Name	Type	Source
CTBGA256	Footprint	Atmel Microcontrol

Workspace1.DsnWrk

sub_board_test.PrjPcb

● Files ○ Structure

- sub_board_test.PrjPcb *
  - Source Documents
    - cdh_pcb.PcbDoc
    - Sheet1.SchDoc
  - Libraries

Sheet1.SchDoc?Left=-96;Right=124

[No Variations]

X:170 Y:750 Grid:10

Figure 2.1: Browsing libraries.

Note: The link to the Altium libraries is not exhaustive, it is simply a collection of common libraries used and may not have the components you need. For most of the components I used I Googled ”Altium component-name-here” which lead to a designcontent.live.altium.com website. In the libraries found this way you can scroll through and make sure your component is included in the library.

2.2 Custom Components [TODO]

All the information I have on this topic came from this video.

To add a new library to your project, right click on the PrjPCB file in the projects tab on the left. Navigating through the menu like in the picture below you will see an option to add a new Schematic Library and a PCB Library. Essentially the PCB Library defines the footprint of the component, the physical dimensions, pad placement and orientation references on the PCB. The Schematic library is where you define the pins, input/output and anything you would need when using it in a schematic. As the naming suggests, the PCB Library defines what we see when importing that component in the PCB file; the Schematic Library defines what we see when importing the component into a schematic.

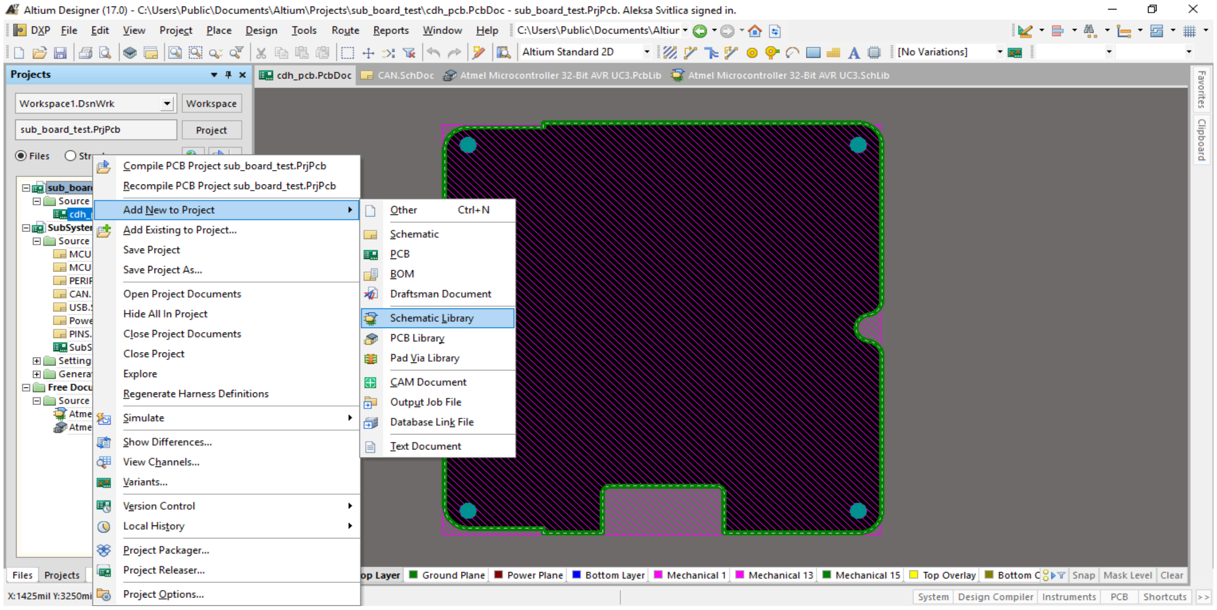


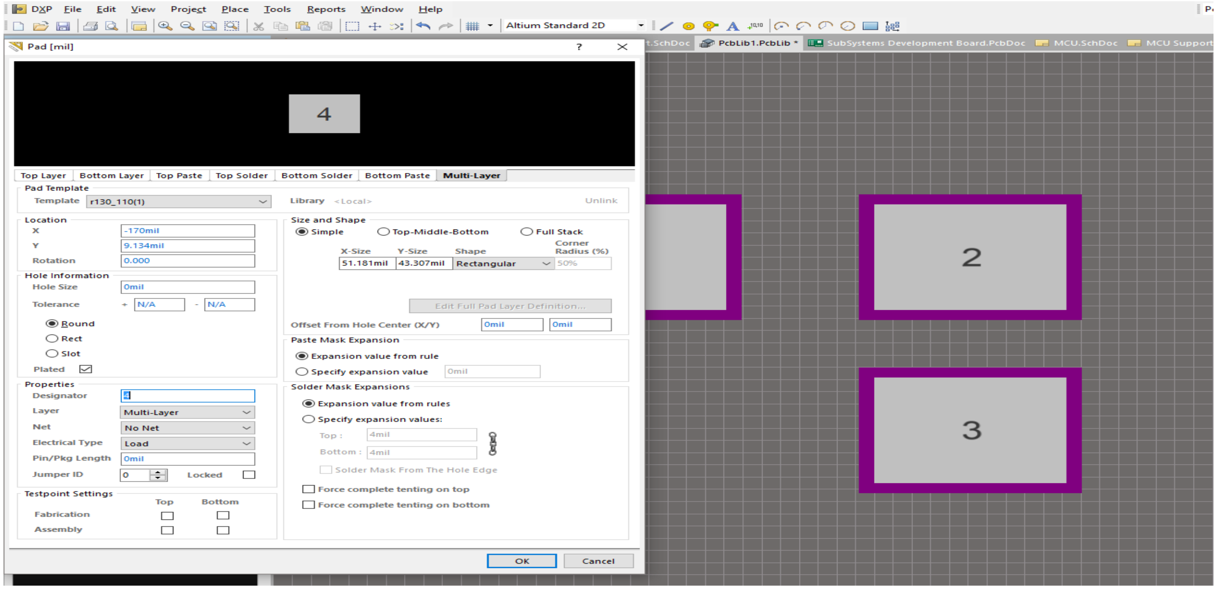
Figure 2.2: Menu for adding new schematic library.

2.2.1 PCB Libraries [TODO]

All the information needed to create a PCB footprint is located in the component datasheet. The bare minimum you need to define is the pads or holes needed to mount the component.

2.2.1.1 Creating Holes/Pads

Pins and pads should be the first thing created in a PCB footprint. They are created using the same tool, Place-¿Pad. It will default to a hole but this can be changed by modifying the properties (figure 2.3). The name should match the numbers in the datasheet to avoid confusion. To create a pad for surface mount components select Top Layer in the Layer property. Doing so will automatically change the hole size to 0 and likely change the pad color to red. An important thing to keep track of is the dimensions, datasheets will alternate between milimeters (mm) and thousandth of an inch (mil); Altium defaults to mils.



Pad Template

Template r130_110(1)

Library <Local> Unlink

Location

X -170mil

Y 9.134mil

Rotation 0.000

Hole Information

Hole Size 0mil

Tolerance + N/A - N/A

● Round

○ Rect

○ Slot

Plated ☑

Properties

Designator 4

Layer Multi-Layer

Net No Net

Electrical Type Load

Pin/Pkg Length 0mil

Jumper ID 0 Locked □

Testpoint Settings

	Top	Bottom
Fabrication	□	□
Assembly	□	□

Size and Shape

● Simple ○ Top-Middle-Bottom ○ Full Stack

X-Size	Y-Size	Shape	Corner Radius (%)
51.181mil	43.307mil	Rectangular	50%

Offset From Hole Center (X/Y) 0mil 0mil

Paste Mask Expansion

● Expansion value from rule

○ Specify expansion value 0mil

Solder Mask Expansions

● Expansion value from rules

○ Specify expansion values:

Top : 4mil

Bottom : 4mil

□ Solder Mask From The Hole Edge

□ Force complete tenting on top

□ Force complete tenting on bottom

Figure 2.3: Pad properties.

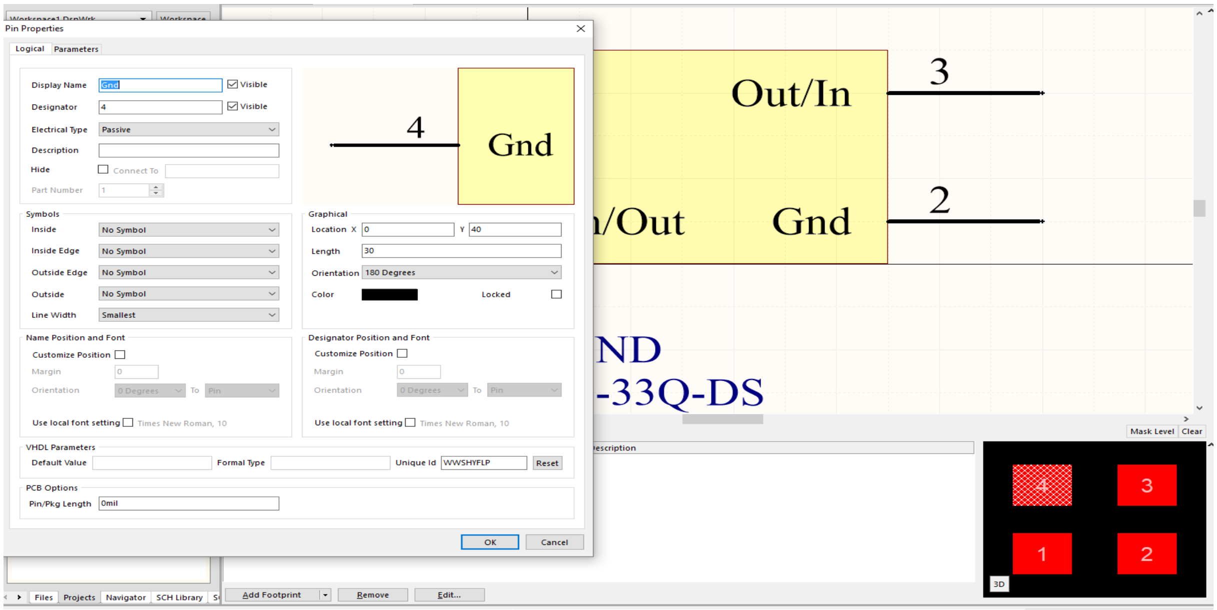
Notes: To ensure correct positioning of pads relative to each other I used the location property of the pads. If creating holes, it is recommended to use the max hole diameter (in the datasheet) plus 0.25mm to ensure easy installation.

2.2.1.2 Defining the Outline[TODO]

2.2.1.3 3D Body[TODO]

2.2.2 Schematic Libraries

The core requirement in schematic libraries is the pins. These pins need to match up with the holes or pads created in the component footprint (PCB library). Creating a pin can be done through the Place menu or the toolbar. The space bar rotates the pin. After placing a pin you will need to set the properties in the menu shown in figure 2.4. Display name can be anything but ideally describes what the pin is for. Designator must be a number which matches a hole or pad in the footprint. To add the footprint click the Add Footprint button on the bottom of the screen and select the footprint you made previously.



Pin Properties

Display Name	Gnd ☑ Visible
Designator	4 ☑ Visible
Electrical Type	Passive
Description	
Hide	□ Connect To
Part Number	1

Symbols

Inside	No Symbol
Inside Edge	No Symbol
Outside Edge	No Symbol
Outside	No Symbol
Line Width	Smallest

Graphical

Location X	0
Y	40
Length	30
Orientation	180 Degrees
Color	■
Locked	□

Name Position and Font

Customize Position □
Margin 0
Orientation 0 Degrees To Pin
Use local font setting □ Times New Roman, 10

Designator Position and Font

Customize Position □
Margin 0
Orientation 0 Degrees To Pin
Use local font setting □ Times New Roman, 10

VHDL Parameters

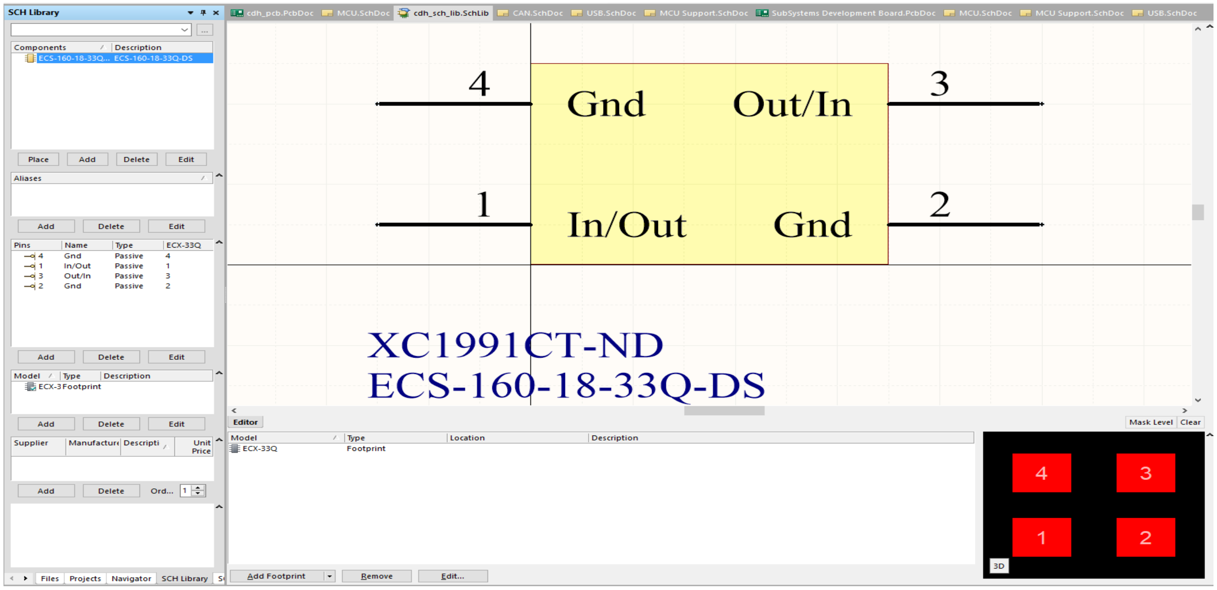
Default Value Formal Type Unique Id WWSHYFLP

PCB Options

Pin/Pkg Length 0mil

Figure 2.4: Modifying pins.

If possible it is best to place one a symbol to represent your component. If you don't have the symbol and it isn't provided by Altium, a common alternative is to place a rectangle between your pins like in figure 2.5.



Components	Description
ECS-160-18-33Q...	ECS-160-18-33Q-DS

Pins	Name	Type	ECX-33Q
4	Gnd	Passive	4
1	In/Out	Passive	1
3	Out/In	Passive	3
2	Gnd	Passive	2

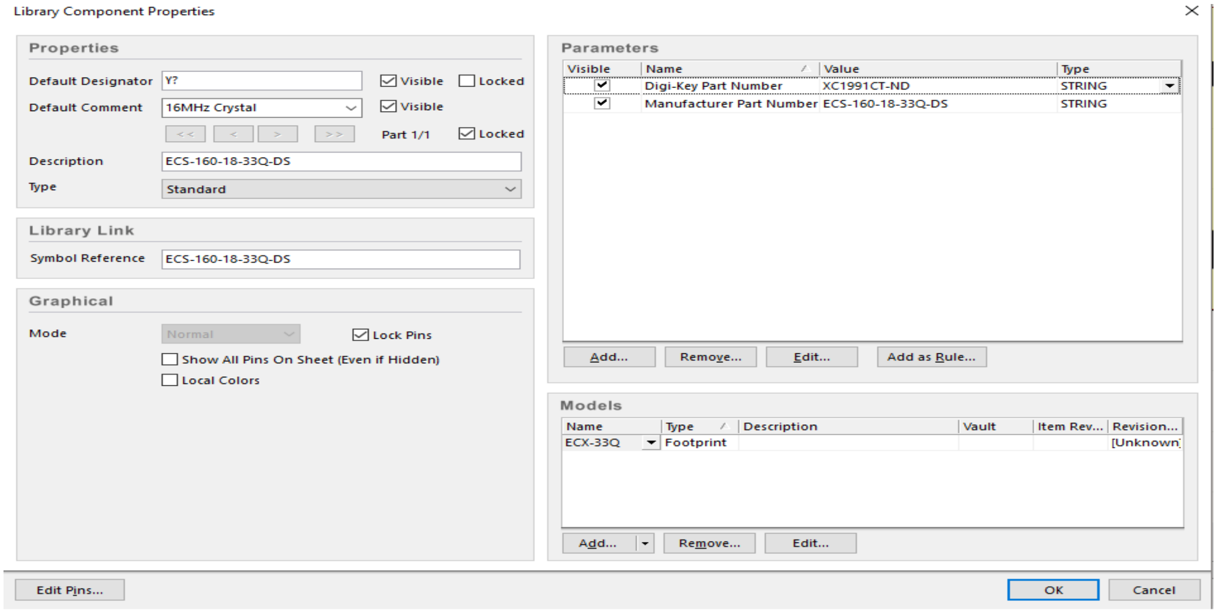
Model	Type	Description
ECX-3 Footprint		

Model	Type	Location	Description
ECX-33Q	Footprint		

Figure 2.5: Sample schematic library.

When finished making the component it is time to add additional properties and useful information. Click on the SCH Library tab on the bottom left to open up the component menu seen in figure 2.5. Click on the component you just created and select edit. This will bring up a menu like in figure 2.6. The designator field should have some standard letters followed by a question mark (R for resistor, C for capactior, U for microcontroller etc.). The comment and description fields are up for interpretation. I personally put a colloquial description in the comment and then copy the description from Digikey for the description.

The parameters on the right side of the menu are also freeform in their use. In general you want to put information which will make it easier for other people to get information about this component. I opted to include the Digikey and manufacturer part numbers.



Library Component Properties

Properties

Default Designator: Y? ☑ Visible □ Locked

Default Comment: 16MHz Crystal ☑ Visible

Part 1/1 ☑ Locked

Description: ECS-160-18-33Q-DS

Type: Standard

Library Link

Symbol Reference: ECS-160-18-33Q-DS

Graphical

Mode: Normal ☑ Lock Pins

□ Show All Pins On Sheet (Even if Hidden)

□ Local Colors

Parameters

Visible	Name	Value	Type
☑	Digi-Key Part Number	XC1991CT-ND	STRING
☑	Manufacturer Part Number	ECS-160-18-33Q-DS	STRING

Models

Name	Type	Description	Vault	Item Rev...	Revision...
ECX-33Q	Footprint				[Unknown]

Figure 2.6: Library component properties.

Chapter 3

Making Schematics

Before making any schematics, there should be some planning in regards to how components will be grouped. Unless you are designing a simple PCB with very few components, it will be useful to create multiple schematics. For example, in the subsystems development board there are seven schematics: MCU, MCU Support, Peripherals, CAN, USB, Power and Pins.

3.1 Placing Components

Click on Place->Part... and select Choose. This will bring you to the menu in figure 2.1, there you can select libraries and components from those libraries.

3.1.1 Placing the MCU

I chose to start with the MCU because everything will branch off this component. I selected the Atmel Microcontroller AVR UC3.IntLib library I downloaded earlier and searched by the component name in the mask field, AVRUC3C2512C and then select the A2UT version of this component. After clicking Ok this will take you back to the schematic file where you can left click to place the component. The MCU is a special case in component placement because most of them are divided up into smaller parts to place on the schematic. When you left click, the next component queued for placement will look slightly different, keep clicking and placing on the schematic until you notice the components are starting to repeat. Alternatively, you can check with the identifier on the top left of the component, probably U?A, and keep placing until you notice that the identifier has reached U?A again. To stop placing components press the esc key. My placement can be seen in figure 3.1. While the entire MCU is currently in the same schematic, I will be moving parts of it to separate schematics where it makes sense.

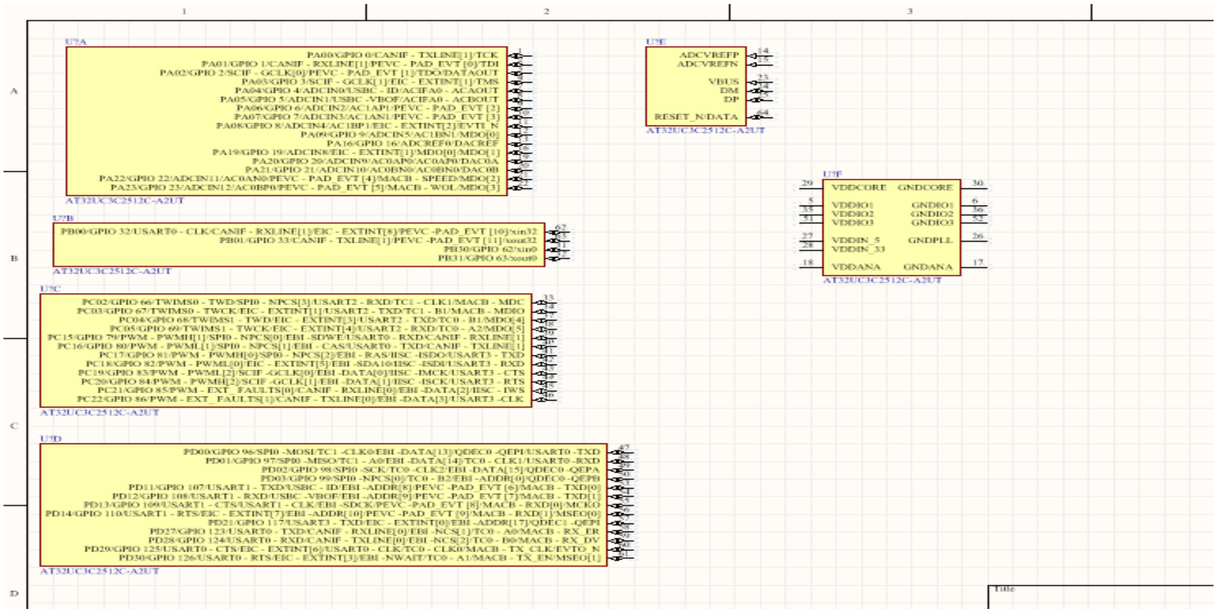


Figure 3.1: Placing the MCU

Most other components are much simpler to place because they won't be divided up into smaller sections.

3.1.2 Placing Other Components

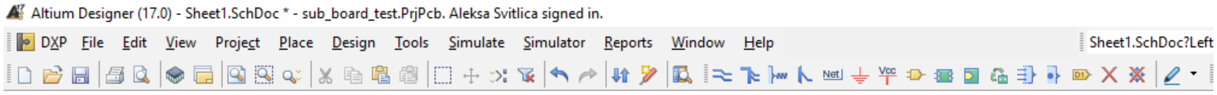


Figure 3.2: Schematic toolbar

3.1.2.1 Ground and Voltage

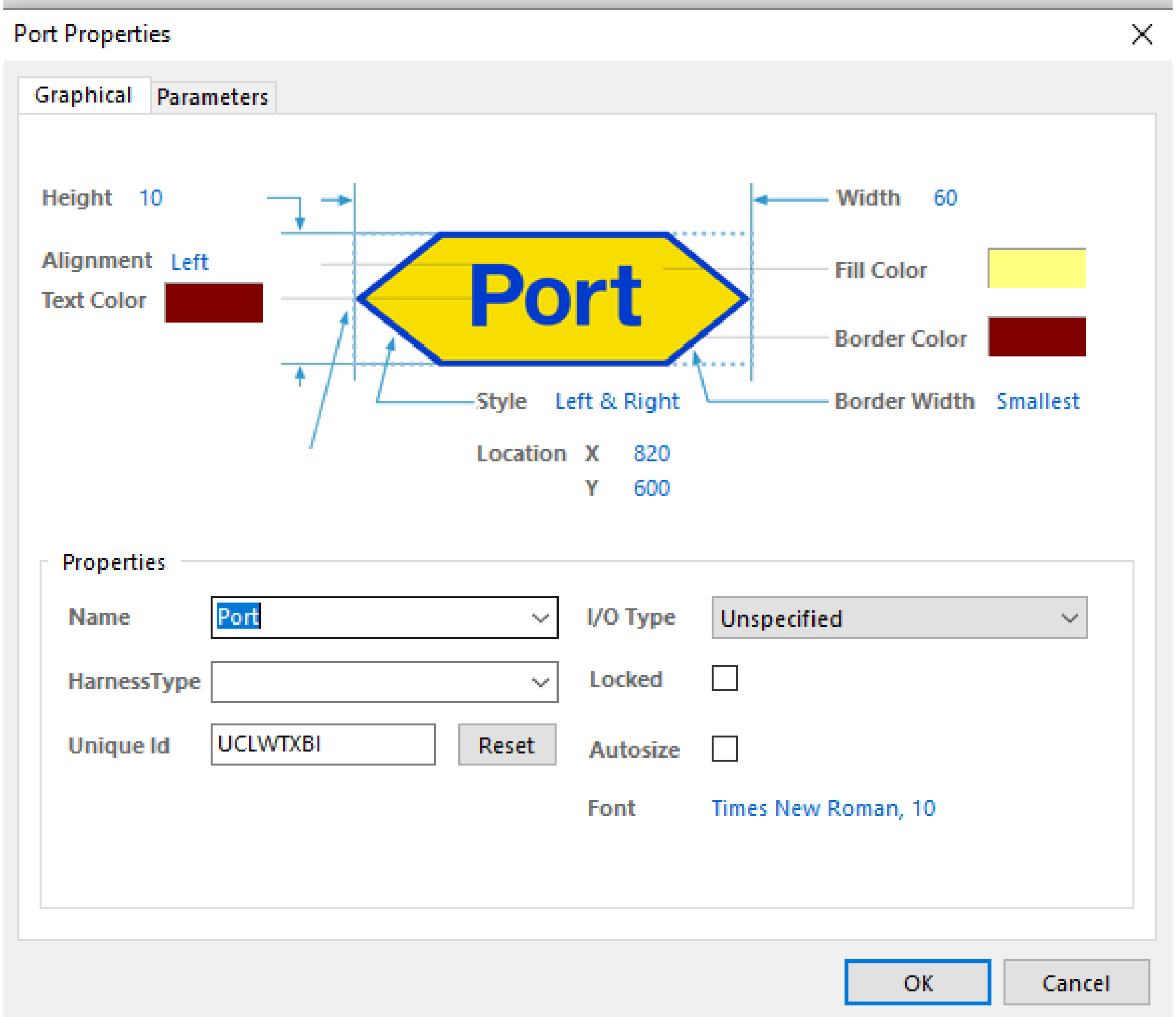
Figure 3.2 shows the toolbar available when placing components on a schematic. Ground and voltage are both options on the toolbar, identified by their standard symbols. When placing a voltage source, it will default to Vcc, to change this right click and select properties. The net property is the label shown on the schematic.

3.1.2.2 Resistors, Capacitors and Inductors

In general, the resistors, capacitors and inductors you need can be taken from the standard parts library Miscelaneous Device.IntLib. The only reason to create custom components here would be to use a footprint that isn't included in the standard parts library.

3.1.3 Adding Ports

Ports are a tool than can be used to define I/O as input, output or bidirectional. These are used on the MCU pins to allow cleaner schematics. Rather than wiring the MCU directly to the rest of the schematic, we instead add ports to each MCU pin. Each port needs to have a name and an I/O type, these are defined after placing the port by right right clicking at selecting properties. For the MCU all ports should be bi-directional because we are going to place a matching port elsewhere to be used in the schematic.



Height 10

Alignment Left

Text Color 🟥

Width 60

Fill Color 🟨

Border Color 🟥

Style Left & Right

Border Width Smallest

Location X 820
Y 600

Name	Port	I/O Type	Unspecified
HarnessType		Locked	□
Unique Id	UCLWTXBI	Autosize	□
		Font	Times New Roman, 10

Figure 3.3: Port properties

Chapter 4

Routing