

Altium Designer Guide

Beginning & Intermediate Version



Glenn Mercier

mercierg@unlv.nevada.edu

Research Engineer, TBE-B3 | I

University of Las Vegas, Nevada



Part I: Beginning Guide

- This guide is a beginner's guide to PCB design using Altium Designer and is geared towards the following individuals:
 - Possess ample theoretical electronics knowledge
 - Has little or no PCB design experience
 - Has little or no Altium Designer experience



Beginning Guide

- Before proceeding to the actual software tutorial, It is important to understand when it is absolutely necessary to use a PCB rather than design with a breadboard
- Most students have a reluctance to learning new software and spending money for something they could possibly build on a breadboard.



Breadboard vs. PCB

- For many electronic designs, one has a choice to build a circuit on either a breadboard or on a printed circuit board.
- Most students have a certain comfort zone working with breadboards, but there comes a time when the complexity of the project or the physical requirements requires electronic design through CAD (computer aided design).



Breadboard Advantages

- Very quick to prototype using standard components
- Can easily make changes to schematic or the functional working of the circuit
- Easy to connect to electronic equipment such as function generator, oscilloscope, power supplies, etc.



Breadboard Disadvantages

- Unprofessional appearance
- Difficult to troubleshoot due to human error and poor connections
- Works very poorly for high speed design
- Difficult to modify complex SMD components for prototyping
- Excessive capacitance
- Difficult to replicate



Printed Circuit Board Advantages

- Professional appearance
- Repeatable and controllable stray inductance and capacitance
- Can handle most power requirements
- Can make very compact
- Excellent high speed capabilities
- Easy to assemble and replicate



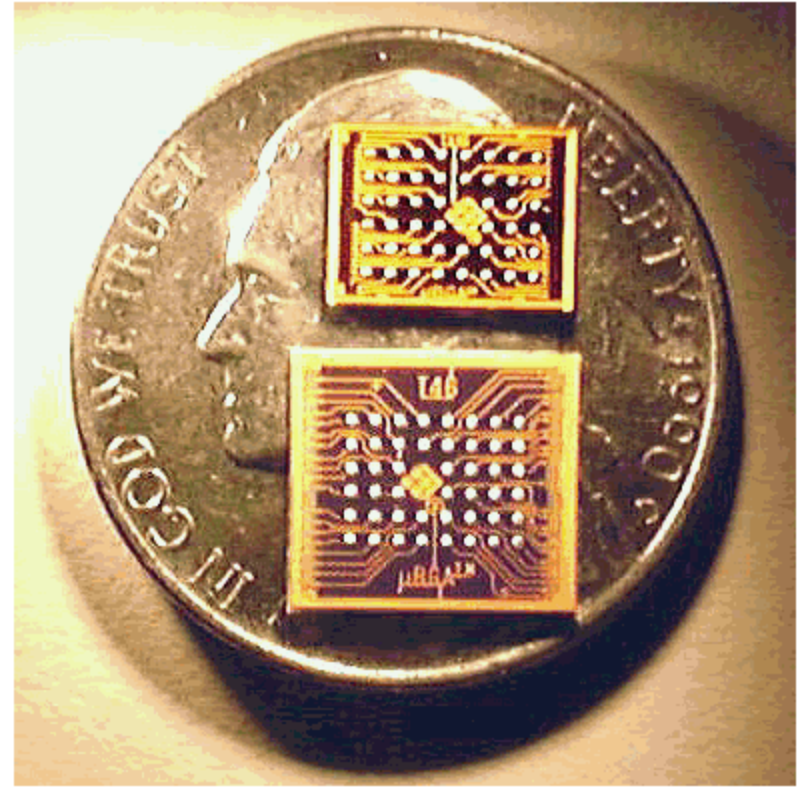
Printed Circuit Board Disadvantages

- Increased design time
- Drastically increased schematic-to-finished product timeframe
- Development Cost
- Difficult to modify once board is fabricated
- Must learn at least a basic form of CAD software
- Quality of PCB can be affected by limited knowledge of software package
- Capabilities vary greatly with different software packages.
- High end CAD software is very expensive

Required PCB Design

- Small Packages, such as a BGA, MLF, QFP are practically impossible to prototype on a breadboard due to their small size and **MUST** be designed on a PCB.

From Computer Desktop Encyclopedia
Reproduced with permission.
© 1999 Tessera Inc.



http://content.answers.com/main/content/img/CDE/_MCROBGA.GIF



Required PCB Design

- Just dealing with the sheer number of pins on modern practical designs required a PCB.
- It is common to see single components containing up to 1,500 pins
- High speed design cannot be performed using traditional prototyping methods



Altium Designer

- This is based off the current version of Altium Designer, 6.9.0.12759
- The Altium Designer Suite contains many unified features such as:
 - FPGA schematic design
 - VHDL/Verilog compilers
 - C/ASM compilers
 - Scripting
 - Simulation
 - 2D Field Solver
 - Simulation Engine
 - Library Management
 - Database and advanced query language
 - CAM display management
 - Schematic Capture
 - PCB Layout



System Requirements

Recommended system for optimal performance

- Windows XP (Professional or Home)
- 3 GHz Pentium 4 processor or equivalent
- 1 GByte RAM
- 2 GByte hard disk space (Install + User Files)
- Dual monitors with 1280x1024 screen resolution
- 32-bit color, 64 MB graphics card*
- Parallel port (if connecting to a NanoBoard-NB1)

Minimum system requirements for acceptable performance

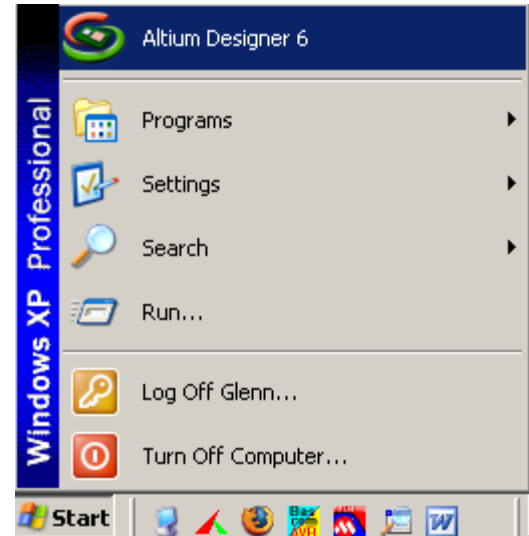
- Windows 2000 Professional SP2
- 1.8 GHz processor
- 1 GByte RAM
- 2 GByte hard disk space (Install + User Files)
- Main monitor 1280x1024 screen resolution
Strongly recommended: second monitor with minimum 1024x768 screen resolution
- 32-bit color, 32 MB graphics card*
- Parallel port (if connecting to a NanoBoard-NB1)

Getting Started

- Start the software by clicking

START
PROGRAMS
ALTIUM DESIGNER
Altium Designer 6

- Create a start bar shortcut if you plan on using the software a lot. (shown on right)



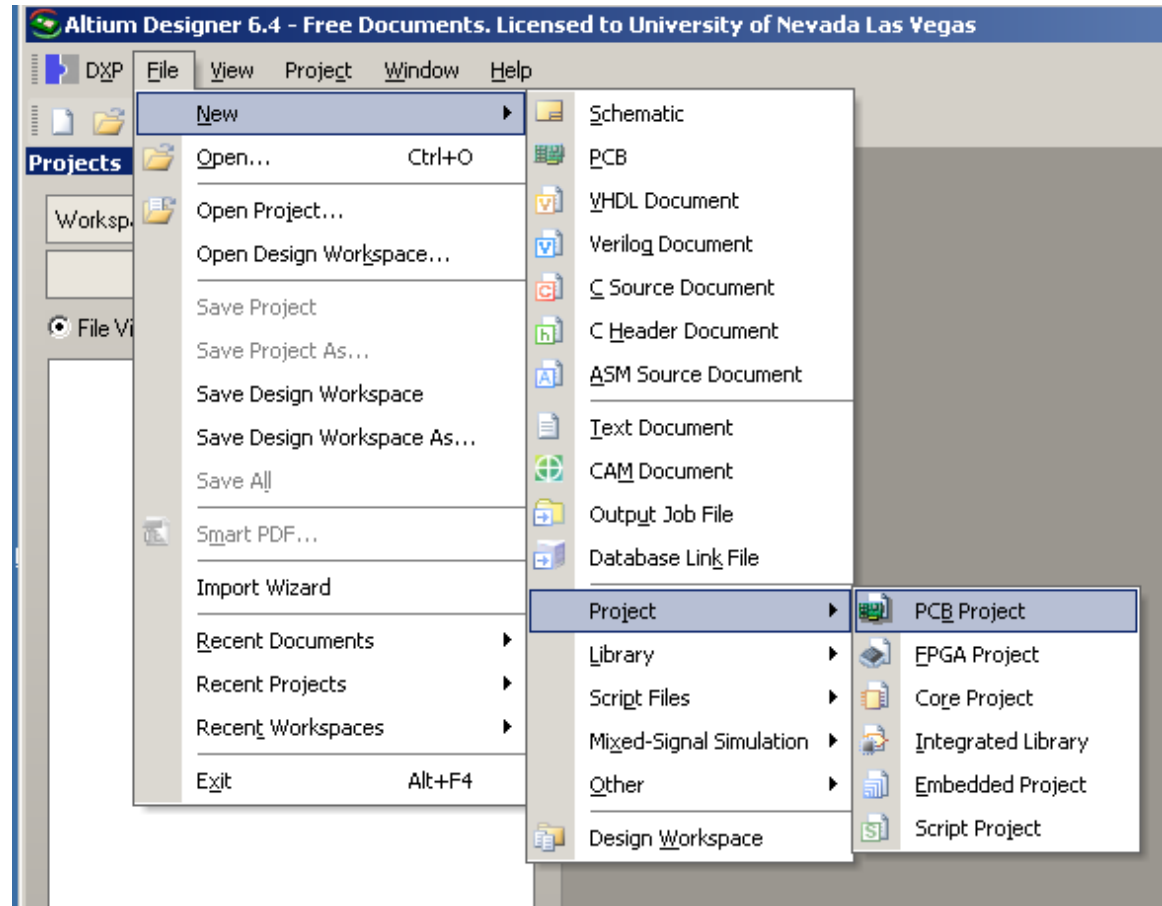


Project File

- Project files are like a container for your project. There are many different things you will need for even a basic project, such as:
 - PCB Footprint libraries
 - Schematic libraries
 - Schematic documents
 - PCB documents
 - Scripts
 - CAM files
 - Settings
 - Generated reports
 - Generated files
 - Version Control features
 - Etc.

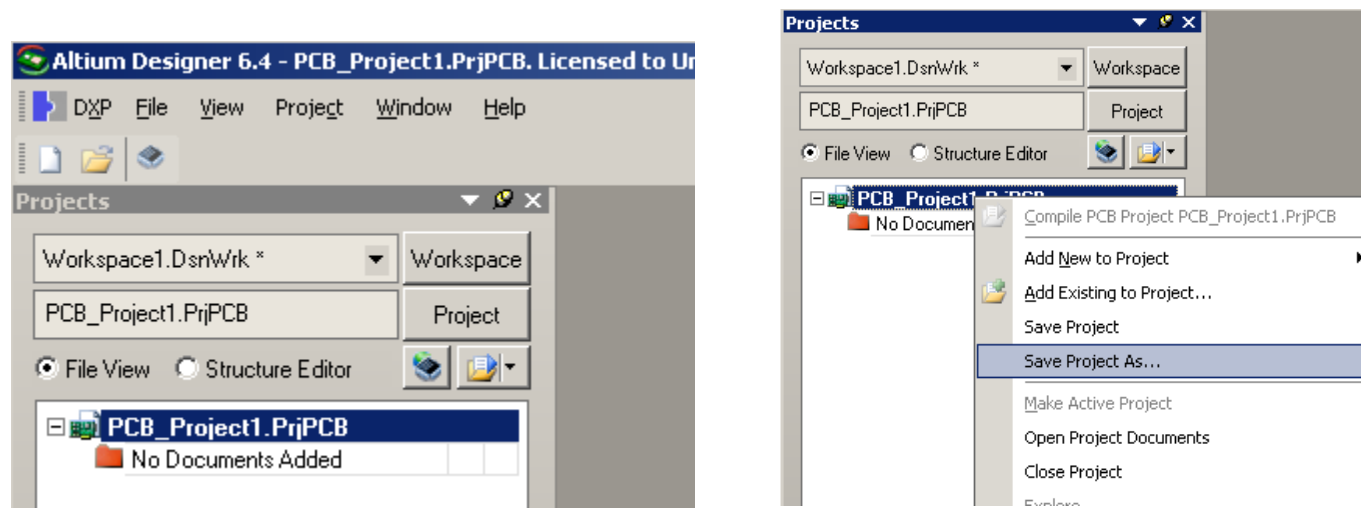
Creating a Project

- When the software starts, follow the menu sequence in the image to create a new 'PCB Project'
- Many people skip this step and just create a schematic file. This a huge mistake! Absolutely everything in Altium Designer is based on project files



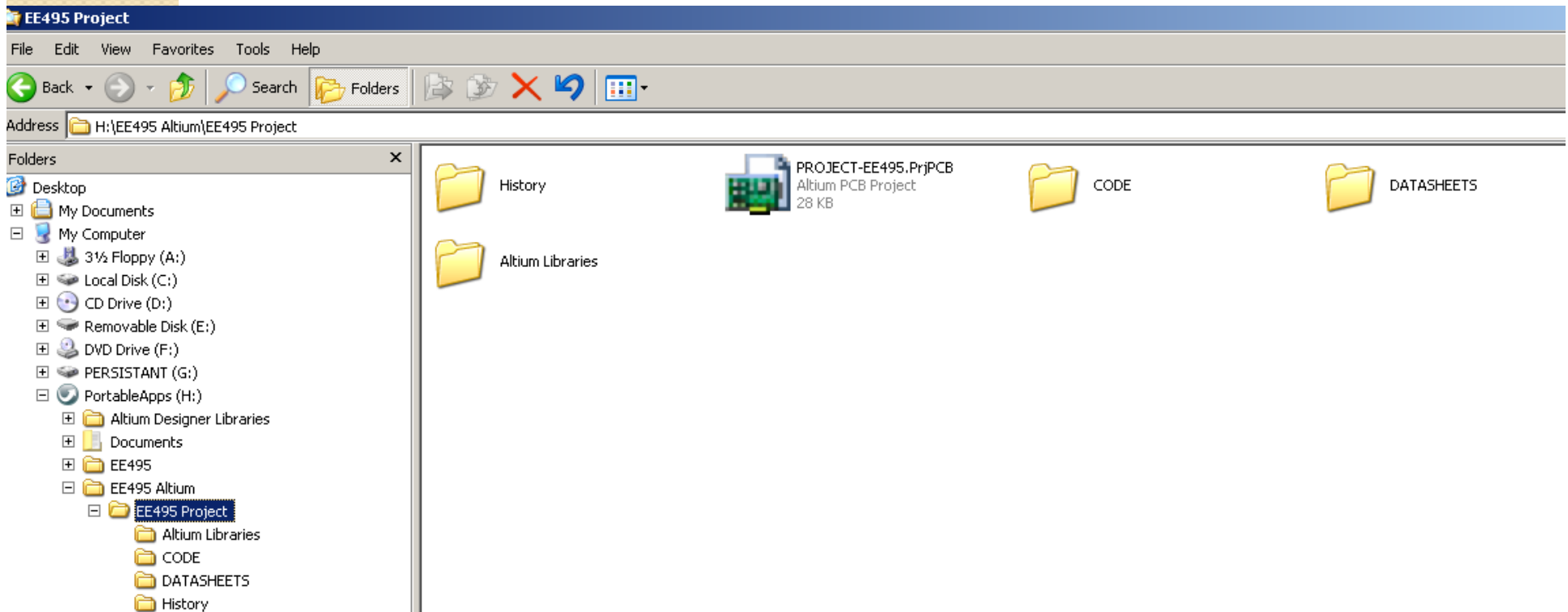
Project Name

- We need to save our project and rename the project to something more meaningful.
- Right click on PCB_Project1.PrjPCB and save the project as 'PROJECT- EE495' to a new folder (preferably on a flash drive) called 'EE495 Project' inside a root folder called 'EE495 Altium'
- EX: F:\EE495 Altium\EE495 Project\PROJECT-EE495.PrjPCB
- Inside this folder, we will place all relevant project data



Organization

- It is useful to place all project information inside this folder.
- Inside this root folder, create subfolders called 'CODE' and 'DATASHEETS'
- Organization and centralization is also important for backing up all your data properly, version control, and copying project data to a CD for all finished projects (which you should do for future reference)

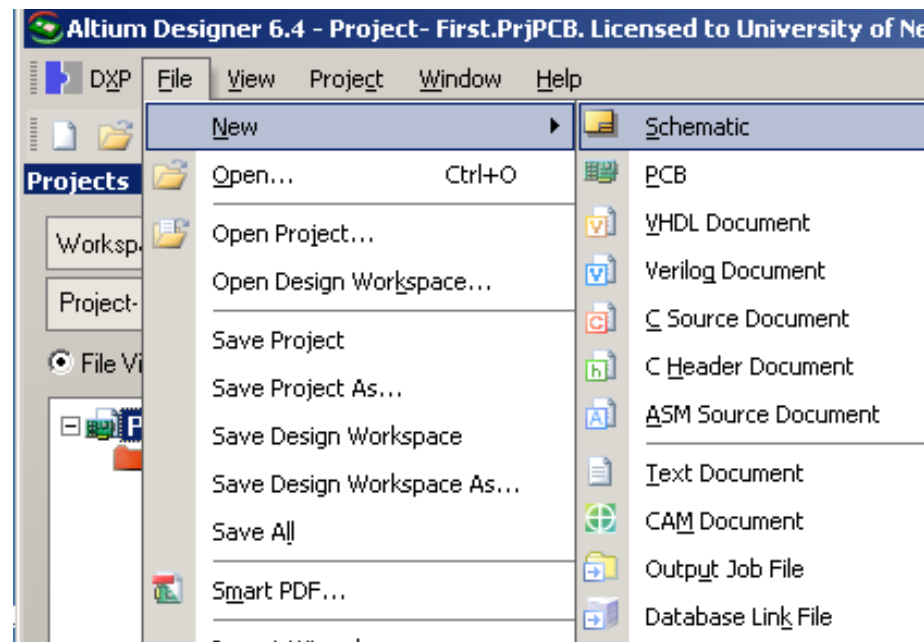


(Group) Quiz Project #1

- For the first quiz project, we are going to create a schematic that will contain:
 - Atmega8 - Atmel Microcontroller , 16 MHz, QFP
 - USB-B Connector
 - FT232 USB- UART interface chip , TSSOP28
 - 2x5 Programming Input Header (0.1" Pitch)
 - SMD (Surface Mount Device) LEDs , 0805 Size
- This project contains all the necessary parts to create a PCB that can communicate directly with a computer through a USB (Male B- Male A) Cable.
 - You are encouraged to actually build this PCB as an inexpensive, functional development board.
 - **This project is to be finished by next Wednesday (5/28) with a possible in-class portion**

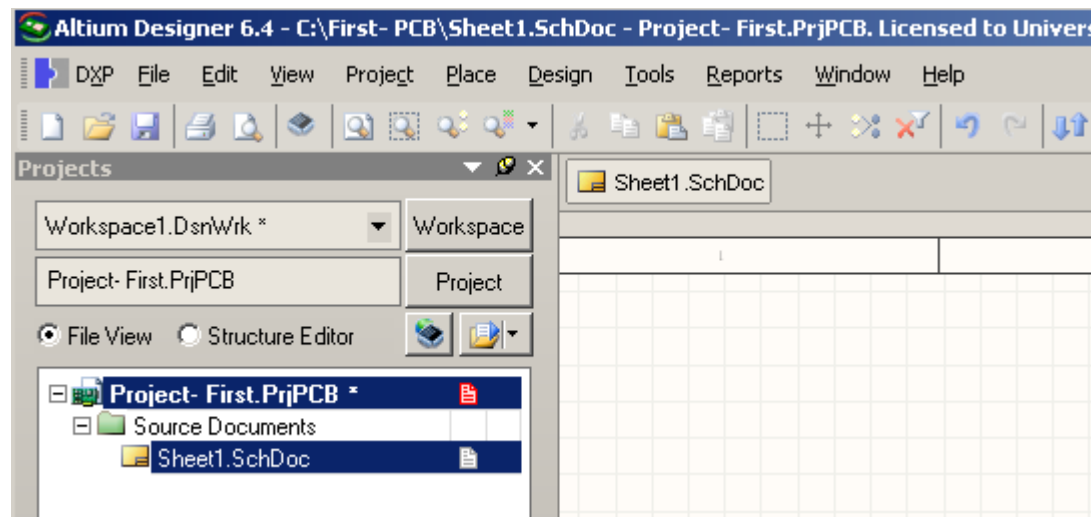
Adding a Schematic Drawing

- Follow the menu sequence shown to create a new schematic. This will automatically be added to the project tree we created.



Schematic Entry

- The 'sheet1' schematic document is now added to the project tree as shown below.
- Rename this file to something with a more meaningful name such as 'Main'. There are many times when you will have multiple schematic entries and it helps to have a descriptive name
- Any open project files will appear in the menu area above where the blank schematic page appears.





Libraries

- Similar to a real laboratory, to build a design from a schematic you must first add real parts to the project.
- For this, we must add what are called ‘libraries’ to our project. Libraries contain vital information about the parts we are going to add.
- Once libraries are added to the project, any component inside that library can be freely inserted into your design.



Types of Libraries

- There are three main types of libraries for PCB design:
 - Schematic Library- Contains schematic drawings of parts, when we look at a schematic, we are looking at a collection of schematic parts
 - PCB Footprint Library- This contains the physical dimensions of a real component. This information is readily available in datasheets.
 - Integrated Library- These are the most useful libraries because they combine a physical part (footprint) with a schematic drawing.



Libraries

- Libraries are located in the folder:
 - C:\Program Files\Altium Designer 6\Library
- Updated libraries are available online at
 - <http://www.altium.com/Community/support/Libraries/Designer6libraries/>
- Altium has since changed some of their library files, I have included a ZIP file on the course website with library files we'll be using.
- Download and unzip the file into the 'library' folder in your project directory

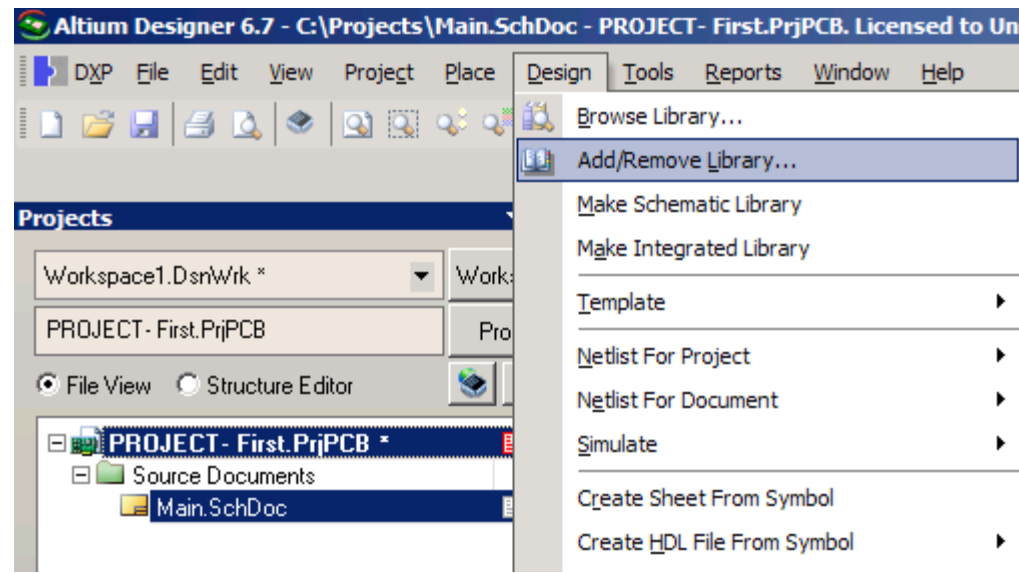


Libraries

- Copy the following Libraries to a folder called 'Libraries' in your Custom Altium folder:
- Integrated Libraries (*.IntLib)
 - Atmel Microcontroller 8-bit AVR
 - Miscellaneous Connectors
 - Miscellaneous Devices
 - NSC Operational Amplifier
 - ON Semi Operational Amplifier
- Schematic Libraries (*.SchLib)
 - <none>
- Footprint Libraries (*.PcbLib)
 - Chip Capacitor – 2 Contacts
 - Chip Diode – 2 Contacts
 - Chip Inductor – 2 Contacts
 - Chip Resistor – 2 Contacts
 - Crystal Oscillator
 - Miscellaneous Connector PCB
 - Miscellaneous Devices PCB
 - Miscellaneous Devices
 - Miscellaneous
- NOTE: When you create your own schematic and footprint libraries, you should place them in the same library directory in your project folder

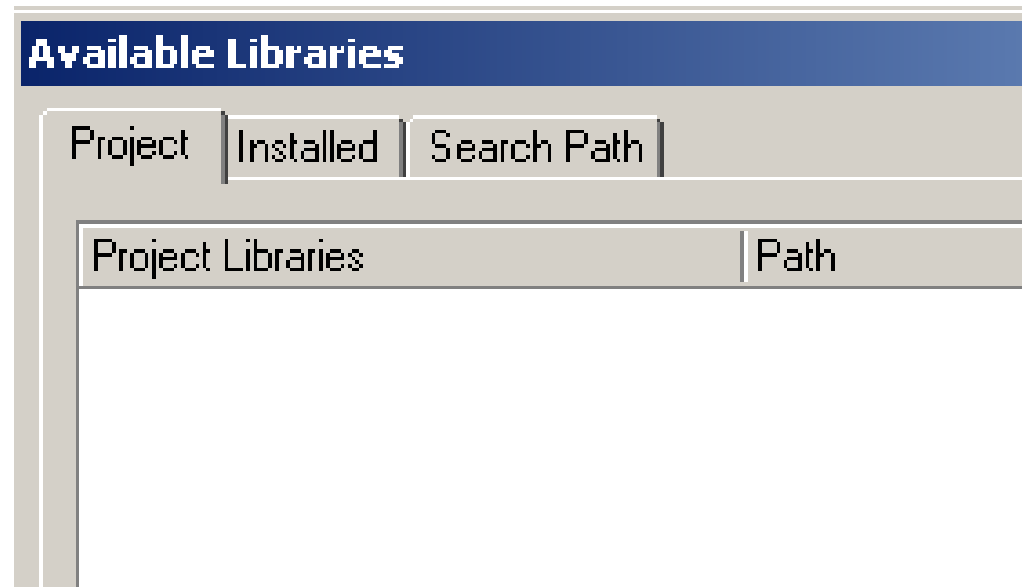
Adding Libraries To A Project

- We have organized all the libraries in a folder, now we need to add these to the project
- Follow the following prompt to add libraries to the project



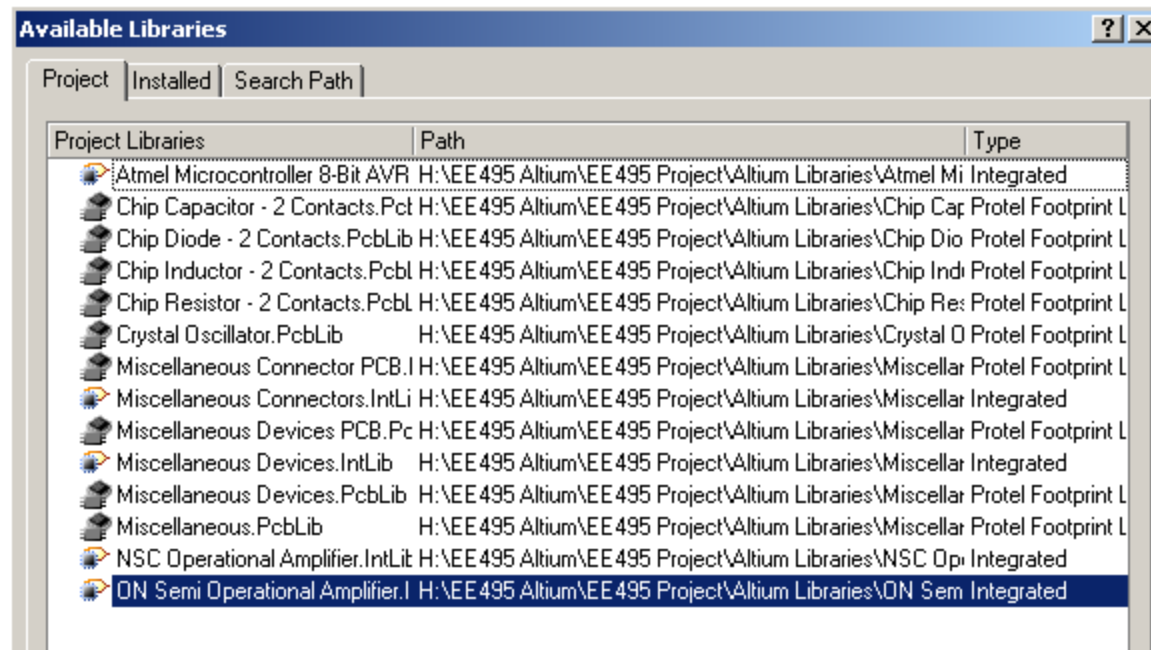
Adding Libraries

- There are two methods of adding libraries
 - Project Libraries- Libraries available only for the scope of this project
 - Installed libraries – Libraries available to ALL projects by default. These do not show up in the project tree.
 - The Search Path modifies the search location when searching for parts.



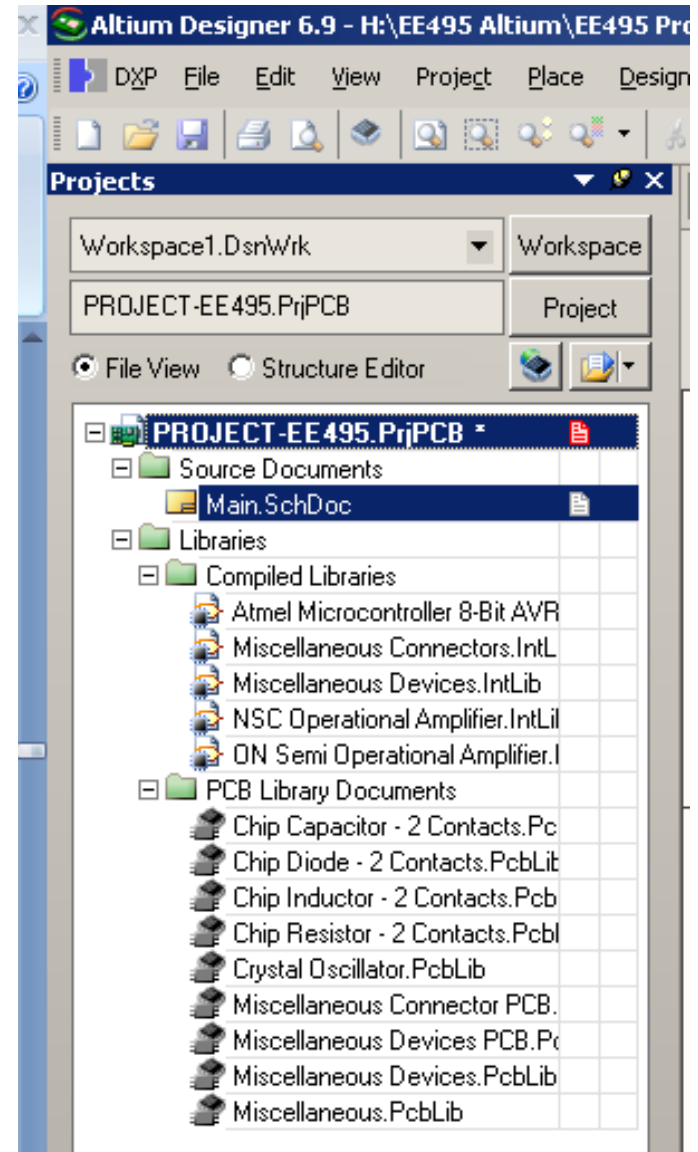
Adding Libraries

- We want to make these libraries project-specific only instead of global availability with the 'Installed' tab
- Make sure the 'Project' tab is selected, and add all libraries from your project library folder



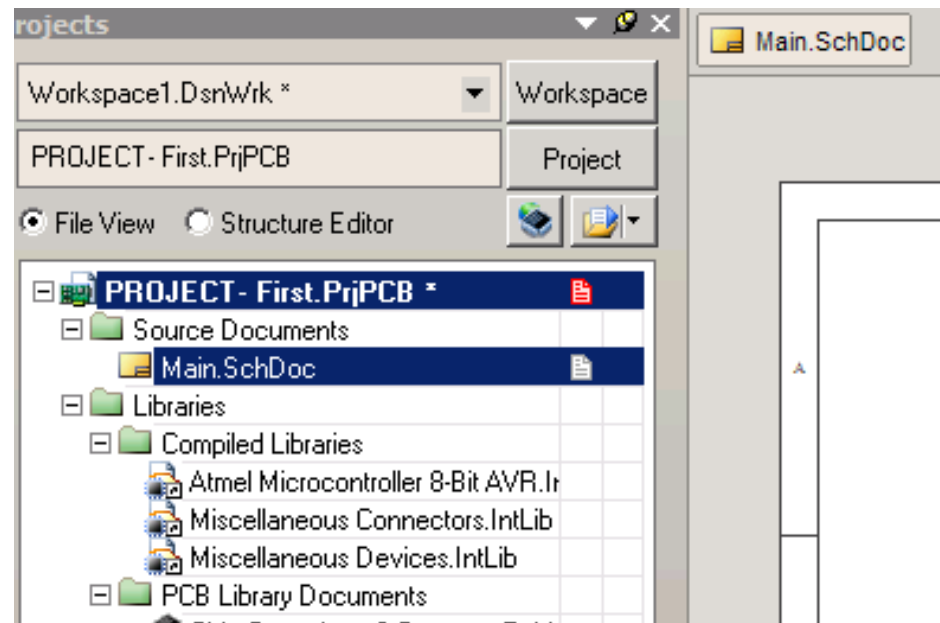
Project Hierarchy

- By expanding the '+' next to libraries, you should see a list of all the project libraries. Make sure you have all these libraries installed before continuing.
- Notice how the project tree separates source documents and libraries automatically



Open, Saved, Unsaved Files

- Open files are indicated with a 'white paper'
 - Example: Main.SchDoc
- Open modified and unsaved files are indicated with a 'red paper'
 - Example: The Project file itself has been modified.
- Closed files have nothing in the column
 - Example: The Atmel Microcontroller library
- All open files are accessible by clicking on them in the project tree, or in the top bar in a windows- taskbar type window



Project Documentation

- We can't create a schematic without knowing more details about the parts we'll be using.
- Obtain the FTDI FT232R UART IC Datasheet
- Rename the file something less cryptic such as FT232 USB.pdf



The screenshot shows a Mozilla Firefox browser window displaying the FTDI Data Sheets website. The browser's address bar shows the URL <http://www.ftdichip.com/Documents/DataSheets.htm>. The website header features the FTDI Chip logo and the text "Future Technology Devices International Ltd" with subtext "USB Device Solutions ASIC Design Product Design". A navigation menu on the left lists: Home, Products, Drivers, Documents, Vinculum Documents, Application Notes, Articles, Data Sheets, Materials Content, Installation Guides, Programming Guides, Schematics, and Solder Profiles. The main content area has a "Data Sheets" section with a play button icon. Below this, it states: "The table below lists the data sheets for FTDI devices. To view a data sheet, click 'Save Target As...'. Data sheets for Vinculum products will redirect you to the Vinculum website." It also notes: "All of the data sheets are in PDF format and require [Adobe Acrobat Reader](#) to be installed." There are three links: [ICs](#), [Vinculum Modules](#), and [FT232R Modules](#). A section titled "IC Data Sheets" contains a table with two entries: [Vinculum Embedded USB Host Controller IC Data Sheet](#) and [FT232R USB UART IC Data Sheet](#).

Project Documentation

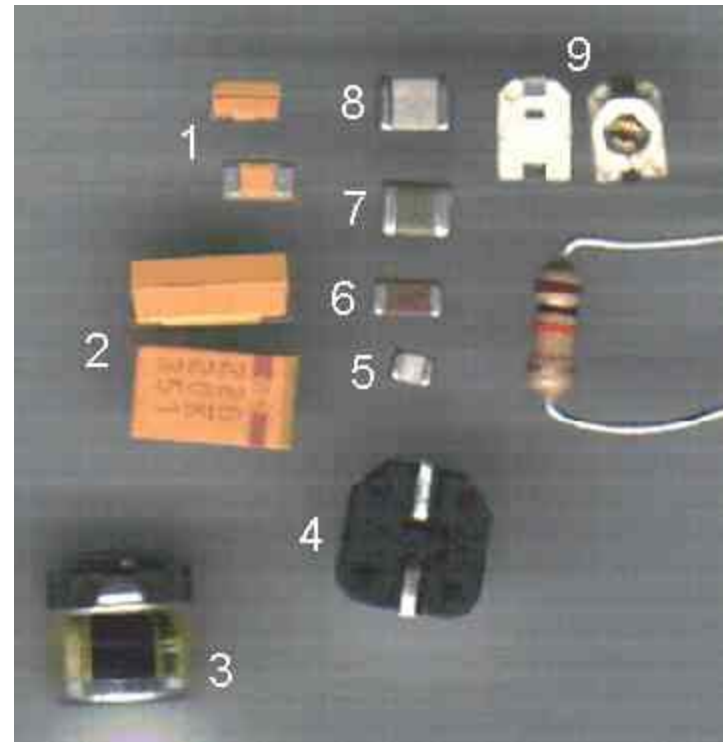
- Specifications are often given or required for components, You should check and double check each meets the project specifications.
- Atmega8 Microcontroller, QFP Package, 16 MHz, ROHS Compliant
Mouser P/N: 556-ATMEGA8-16AU
Save as 'Atmega8 QFP Atmel Microcontroller.pdf'
- USB-B (Female/Socket) , ROHS Compliant, Right Angle Mount
Mouser P/N: 806-KUSBX-BS1N-B30
Save datasheet as 'USB-B Connector'
- SMD LEDs (0805), ROHS Compliant
Mouser P/N: 638-172BHC-ANIP23T
- 2x5 Box Header (Prog. Connector), Gold Plated, ROHS Compliant, Pitch=0.1"
Mouser P/N: 649-66506-066LF
Save datasheet as '2x5 Box Header.pdf'

Chip Sizing Information

- 'Chip' components are devices that come in small rectangles
- The 0805 specification for the LED are actually dimensions of the chip component itself (Size Code).
- Split up the numbers to 08 05, These are the dimensions in inches
- The Length of the chip diode is 0.08" = 80 mils
- The Width of the chip diode is 0.05" = 50 mils
- Therefore, the larger the numbers, the larger the chip device:
- **Larger** chip sizes can handle more **power**
- **Smaller** chip size can handle **higher frequencies** (ideally)

Chip Sizing Information

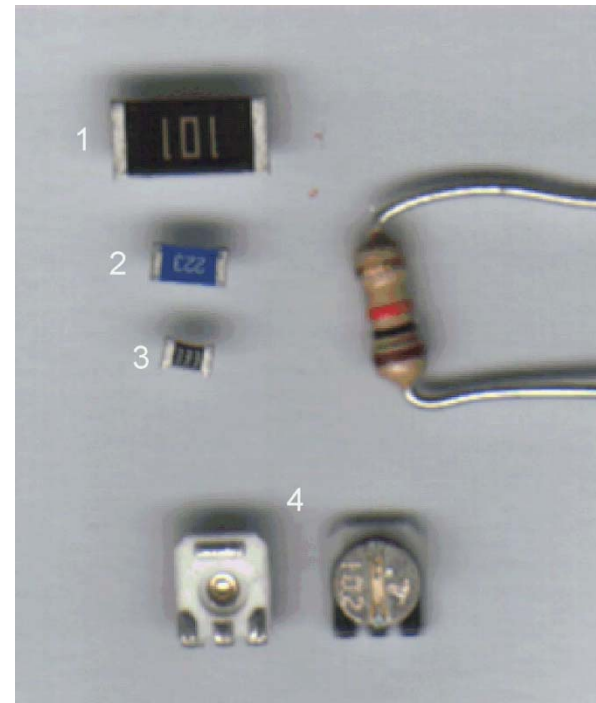
- 1) Tantalum Capacitor (A-Case)
- 2) Tantalum Capacitor (D-Case)
- 3 & 4) Electrolytic Capacitor
- 5) 0805 Ceramic Capacitor
- 6) 1206 Ceramic Capacitor
- 7) 1210 Ceramic (Same length, but wider)
- $\frac{1}{4}$ Watt through-hole resistor
given as size reference
(Same as in our lab)



<http://www.geocities.com/vk3em/smtguide/pics/caps.jpg>

Chip Sizing Information

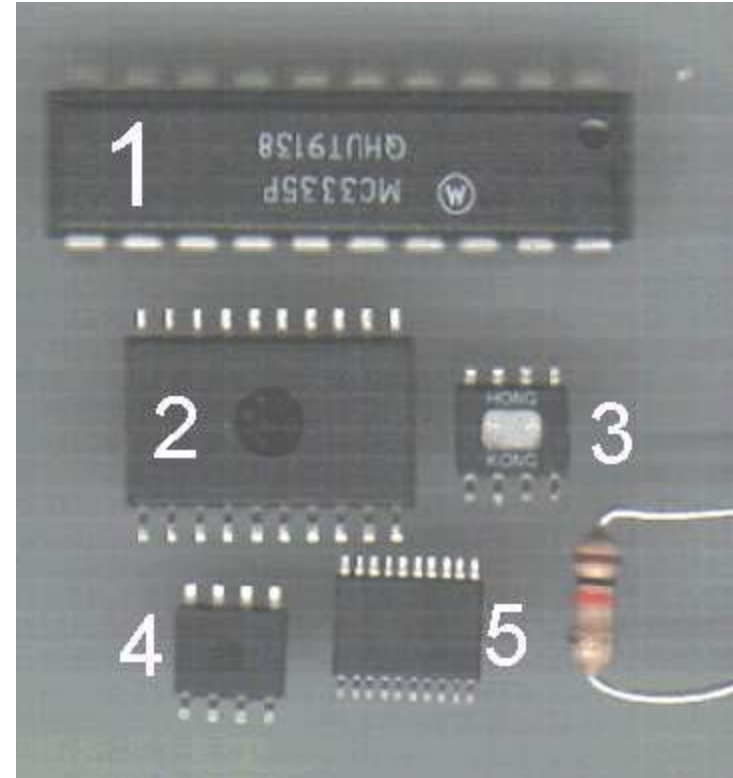
- 1) 1218 (1 Watt)
 - 2) 1206 (1/4 Watt)
 - 3) 0805 (1/8 Watt)
 - 4) Potentiometer (Bottom & Top)
-
- 1/4 Watt through-hole resistor
given as size reference
(Same as in our lab)



<http://www.geocities.com/vk3em/smtguide/pics/resistor.jpg>

Chip Sizing Information

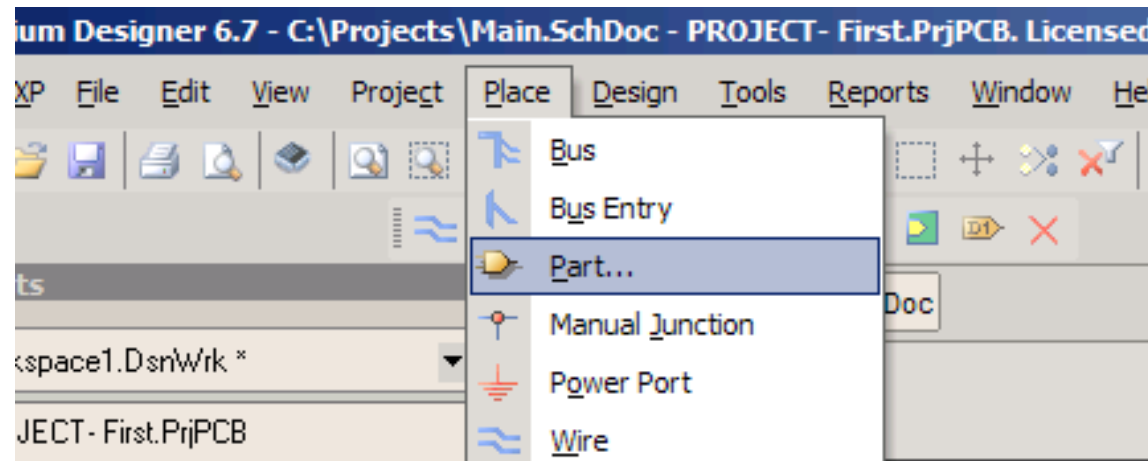
- 1) 20 Pin DIP Package
 - 2) SOIC20, Wide
 - 3) SOIC8, W/Power Tab
 - 4) SOIC8, Standard
 - 5) SSOP20
- 1/4 Watt through-hole resistor
given as size reference
(Same as in our lab)



<http://www.geocities.com/vk3em/smtguide/pics/ic.jpg>

Placing Parts

- We have now added all the necessary libraries for the project, now we need to draw the schematic.
- Follow the menu sequence below to place a part



Placing Parts

- After clicking 'place part' from the previous screen, you will have an option to place from standard libraries or database libraries. Keep this on standard libraries
- Click the button to the right of the 'History' button to bring up a list of available libraries.

Place Part

Placement Type

From Standard Libraries

From Database Libraries

Part Details

Physical Component: MHDR2X7 History... ...

Logical Symbol: MHDR2X7

Designator: P?

Comment: MHDR2X7

Footprint: MHDR2X7

Part ID: 1

Lib Path: Miscellaneous Connectors.IntLib

OK Cancel

Creating a Schematic

- At the top of the screen, there is a dropdown box which lists available schematic and integrated libraries (but not PCB footprint libraries). For the highlighted library, in this case Miscellaneous Devices, there will be a list of the parts inside that library in the bottom left.

The screenshot displays a PCB design software interface. On the left, a project tree shows the following structure:

- PROJECT-EE495.PrjPCB
 - Source Documents
 - Main.SchDoc
 - Libraries
 - Compiled Libraries
 - Atmel Microcontroller 8-Bit AVR
 - Miscellaneous Connectors.IntLib
 - Miscellaneous Devices.IntLib
 - NSC Operational Amplifier.IntLib
 - ON Semi Operational Amplifier.I
 - PCB Library Documents
 - Chip Capacitor - 2 Contacts.Pc
 - Chip Diode - 2 Contacts.PcbLib
 - Chip Inductor - 2 Contacts.Pcb
 - Chip Resistor - 2 Contacts.PcbL
 - Crystal Oscillator.PcbLib
 - Miscellaneous Connector PCB.
 - Miscellaneous Devices PCB.Pc
 - Miscellaneous Devices.PcbLib
 - Miscellaneous.PcbLib

The 'Browse Libraries' dialog is open, showing a list of libraries. The 'Miscellaneous Devices.IntLib' is selected. Below the library list, a table shows the components available in that library:

Component	Source	Type
PNP1	Miscellaneous Devices.IntLib	PI
PNP2	Miscellaneous Devices.IntLib	PI
PNP3	Miscellaneous Devices.IntLib	PI
PUT	Miscellaneous Devices.IntLib	Pr
QNPN	Miscellaneous Devices.IntLib	N
Relay	Miscellaneous Devices.IntLib	SI
Relay-DPDT	Miscellaneous Devices.IntLib	D
Relay-DPST	Miscellaneous Devices.IntLib	D
Relay-SPDT	Miscellaneous Devices.IntLib	Si
Relay-SPST	Miscellaneous Devices.IntLib	Si
Res Bridge	Miscellaneous Devices.IntLib	R
Res Semi	Miscellaneous Devices.IntLib	Si
Res Tap	Miscellaneous Devices.IntLib	T.
Res Thermal	Miscellaneous Devices.IntLib	TI
Res Varistor	Miscellaneous Devices.IntLib	V.
Res1	Miscellaneous Devices.IntLib	R
Res2	Miscellaneous Devices.IntLib	R

The schematic diagram on the right shows a resistor component labeled 'Res1' with a value of '1K'. The component is highlighted with a yellow box. Below the schematic, a table shows the component's properties:

Model Name	Source	Type
AXIAL-0.3	Miscellaneous Devic	Footprint
Res		Signal Integrity
RESISTOR		Simulation

Creating a Schematic

- Select the Atmel Library and navigate down to Atmega8. Notice there are many options for the same chip. This is because although the device is the same, there are many different 'packages' that this chip comes in.
- **NOTE:** The Atmega8 costs about \$3.50

Component Name	Library	Description	Footprint
AT90S8535-8PC	Atmel Microcontroller	8-Bit AVR Microcontroller	40P6
AT90S8535-8PI	Atmel Microcontroller	8-Bit AVR Microcontroller	40P6
ATmega8-16AC	Atmel Microcontroller	8-Bit AVR Microcontroller	32A_N
ATmega8-16AI	Atmel Microcontroller	8-Bit AVR Microcontroller	32A_N
ATmega8-16MC	Atmel Microcontroller	8-Bit AVR Microcontroller	32M1-A_N
ATmega8-16MI	Atmel Microcontroller	8-Bit AVR Microcontroller	32M1-A_N
ATmega8-16PC	Atmel Microcontroller	8-Bit AVR Microcontroller	28P3
ATmega8-16PI	Atmel Microcontroller	8-Bit AVR Microcontroller	28P3
ATmega8L-8AC	Atmel Microcontroller	8-Bit AVR Microcontroller	32A_N
ATmega8L-8AI	Atmel Microcontroller	8-Bit AVR Microcontroller	32A_N
ATmega8L-8MC	Atmel Microcontroller	8-Bit AVR Microcontroller	32M1-A_N
ATmega8L-8MI	Atmel Microcontroller	8-Bit AVR Microcontroller	32M1-A_N
ATmega8L-8PC	Atmel Microcontroller	8-Bit AVR Microcontroller	28P3
ATmega8L-8PI	Atmel Microcontroller	8-Bit AVR Microcontroller	28P3
ATmega16-16AC	Atmel Microcontroller	8-Bit AVR Microcontroller	44A_N
ATmega16-16AI	Atmel Microcontroller	8-Bit AVR Microcontroller	44A_N
ATmega16-16MC	Atmel Microcontroller	8-Bit AVR Microcontroller	44M1 N

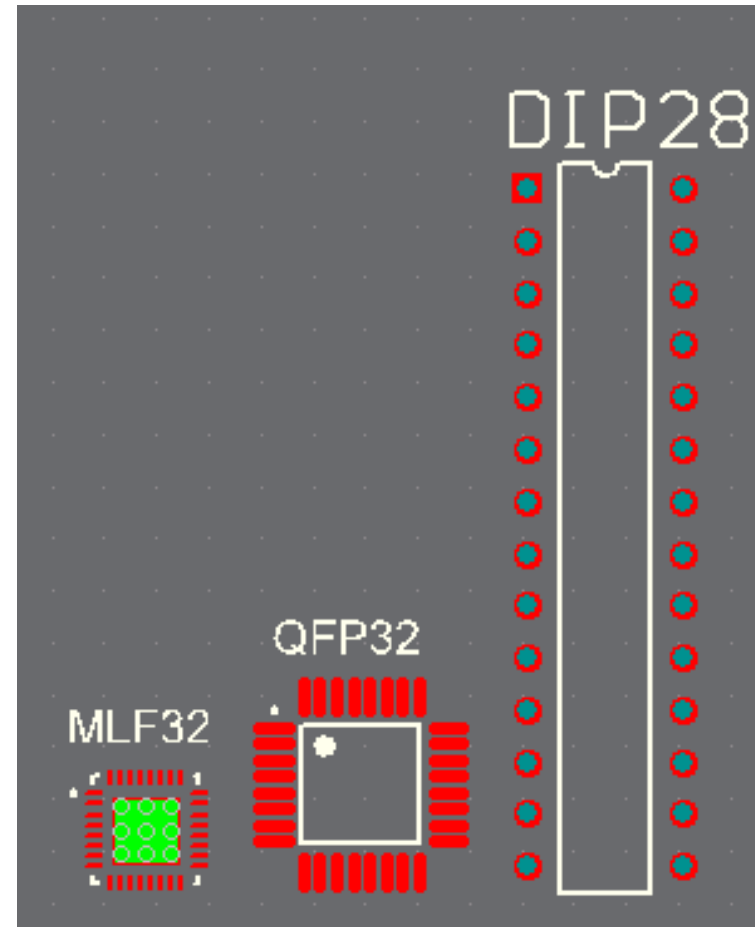
Creating a Schematic

- Add the ATmega8-16AC
- Looking at the datasheet for the ATmega8, we can see exactly what the -16AI stands for.
 - Supports up to a 16 Mhz clock
 - Commercial grade fabrication (Temperature ranges from 0C to 70C)

Speed (MHz)	Power Supply	Ordering Code	Package ⁽¹⁾	Operation Range
8	2.7 - 5.5	ATmega8L-8AC	32A	Commercial (0°C to 70°C)
		ATmega8L-8PC	28P3	
		ATmega8L-8MC	32M1-A	
		ATmega8L-8AI	32A	Industrial (-40°C to 85°C)
		ATmega8L-8AU ⁽²⁾	32A	
		ATmega8L-8PI	28P3	
		ATmega8L-8PU ⁽²⁾	28P3	
ATmega8L-8MI	32M1-A			
ATmega8L-8MU ⁽²⁾	32M1-A			
16	4.5 - 5.5	ATmega8-16AC	32A	Commercial (0°C to 70°C)
		ATmega8-16PC	28P3	
		ATmega8-16MC	32M1-A	
		ATmega8-16AI	32A	Industrial (-40°C to 85°C)
		ATmega8-16AU ⁽²⁾	32A	
		ATmega8-16PI	28P3	
		ATmega8-16PU ⁽²⁾	28P3	
ATmega8-16MI	32M1-A			
ATmega8-16MU ⁽²⁾	32M1-A			

Creating a Schematic

- Here are the three common packages for the ATmega8 microcontroller.
- These are scaled relative to each other with the close to the same functionality with each package.
- NOTE: Why does the QFP package have 4 more pins than the DIP package?





Creating a Schematic

- We were given **constraints** with the assignment of this project. Sometimes constraints are based on:
 - Economics, Size, Power handling, ability to assemble the part, etc..
- For the microcontroller, we were told it must function with a clock rate of 16 MHz, and must have a QFP Package
- It is crucial to take note of all constraints **BEFORE** entering the schematic and/or pcb layout. It is a **HUGE** waste of time to design around a part that doesn't meet the specifications.

Creating a Schematic

- Looking again at the ordering information available in the datasheet, we know that:
 - The Chip must be able to work with a 16 MHz clock, we can eliminate all 8MHz rated microcontrollers
 - The DIP package is 28 pins, we can eliminate any package that has a '28' in the package information
 - The 32MI refer to the MLF packaging, we can eliminate all those items
 - This leaves us with a decision to choose between commercial rated or industrial rated temperature ranges, Either the ending in -16AC, -16AI, -16AU are acceptable

Speed (MHz)	Power Supply	Ordering Code	Package ⁽¹⁾	Operation Range
8	5.5	ATmega8L-8MC	32A	Commercial (0°C to 70°C)
		ATmega8L-8P3	32A	
		ATmega8L-8PU	32MI	Industrial (-40°C to 85°C)
16	4.5 - 5.5	ATmega8-16AC	32A	Commercial (0°C to 70°C)
		ATmega8-16AI	32A	
		ATmega8-16AU ⁽²⁾	32A	
		ATmega8-16AC	32MI	Industrial (-40°C to 85°C)
		ATmega8-16AI	32MI	
		ATmega8-16AU ⁽²⁾	32MI	

Creating a Schematic

- Usually the industrial rated components are more costly than the commercial rated components, so we will choose the Atmega8-16AC
- QUESTION: A temperature rating doesn't affect the footprint or simulation of the device, is there a reason we can't/shouldn't just choose any of the three possible options ???

ATmega8-16AC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32A_N
ATmega8-16A	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32A_N
ATmega8-16MC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32M1-A_N
ATmega8-16MI	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32M1-A_N
ATmega8-16PC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 28P3
ATmega8-16PI	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 28P3
ATmega8L-8AC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32A_N
ATmega8L-8AI	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32A_N
ATmega8L-8MC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32M1-A_N
ATmega8L-8MI	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 32M1-A_N
ATmega8L-8PC	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 28P3
ATmega8L-8PI	Atmel Microcontroller 8-Bit AVR 8-Bit AVR Microcontroller with { 28P3

Creating a Schematic

- Notice to the right, there is the **schematic drawing** and associated **pinout** of the device, beneath that is a drawing of the PCB Footprint (this should match what you expect from the datasheet)
- The bottom shows all footprints associated with the part
 - Why do you think there are three different footprints for a QFP32 ?

The screenshot displays a CAD software interface with three main sections:

- Component Library:** A table listing various Atmel AVR microcontrollers. The selected component is ATmega8-16AC.
- Pinout Diagram:** A schematic drawing of the ATmega8-16AC showing its 32 pins and their functions.
- PCB Footprint:** A 3D rendering of the ATmega8-16AC PCB footprint, showing a square package with 32 pins.

Component Name	Library	Description	Footprint
AT90S8535-8AC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 44A_N	
AT90S8535-8AI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 44A_N	
AT90S8535-8JC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 44J_N	
AT90S8535-8JI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 44J_N	
AT90S8535-8PC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 40P6	
AT90S8535-8PI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 40P6	
ATmega8-16AC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32A_N	
ATmega8-16AI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32A_N	
ATmega8-16MC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32M1-A_N	
ATmega8-16MI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32M1-A_N	
ATmega8-16PC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 28P3	
ATmega8-16PI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 28P3	
ATmega8L-8AC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32A_N	
ATmega8L-8AI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32A_N	
ATmega8L-8MC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32M1-A_N	
ATmega8L-8MI	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 32M1-A_N	
ATmega8L-8PC	Atmel Microcontroller 8-Bit AVR	8-Bit AVR Microcontroller with { 28P3	

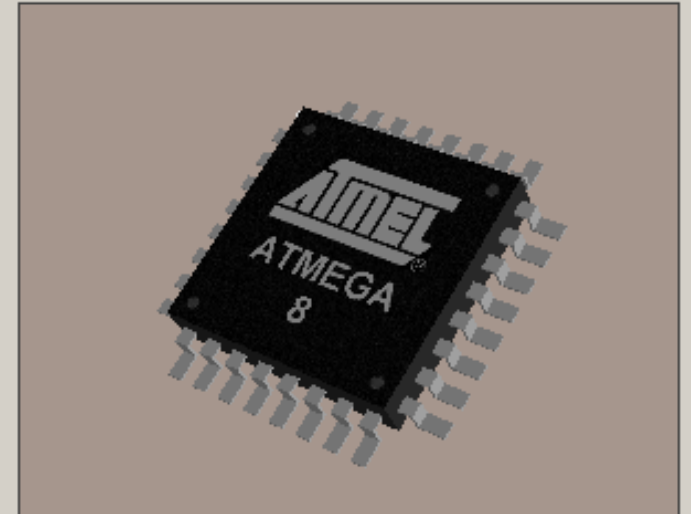
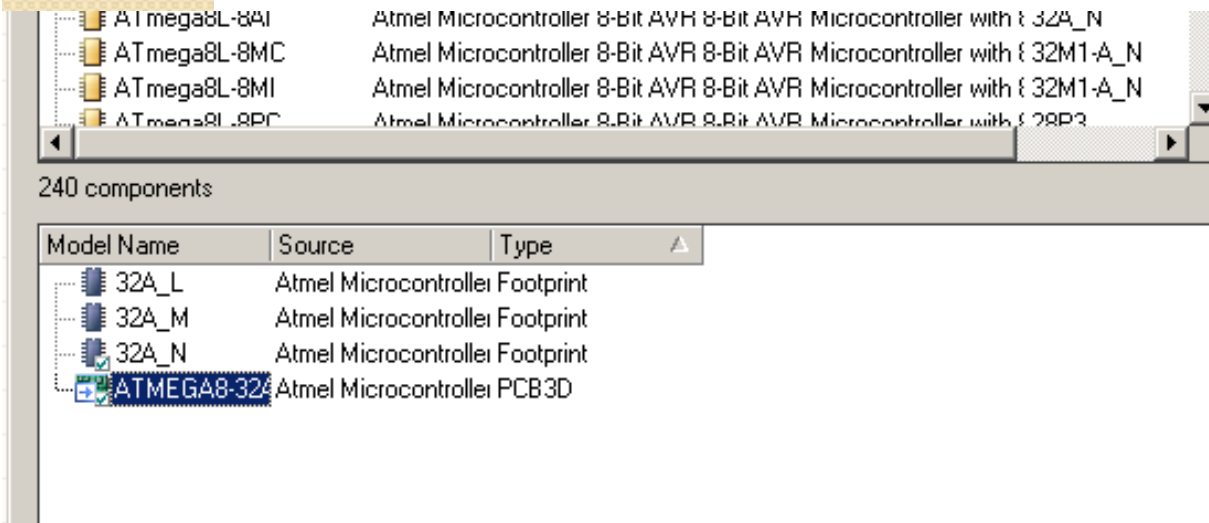
Model Name	Source	Type
32A_L	Atmel Microcontroller Footprint	
32A_M	Atmel Microcontroller Footprint	
32A_N	Atmel Microcontroller Footprint	
ATMEGA8-32	Atmel Microcontroller PCB3D	

Pinout Diagram (ATmega8-16AC):

Pin	Function	Pin	Function
1	PB0 (ICP)	21	PC0 (ADC0)
2	PB1 (CC1A)	22	PC1 (ADC1)
3	PB2 (FRC/IB)	23	PC2 (ADC2)
4	PB3 (MCP/OC2)	24	PC3 (ADC3)
5	PB4 (MIAD)	25	PC4 (ADC4/ADL)
6	PB5 (XK)	26	PC5 (ADC5/ADL)
7	PB6 (XTAL/TOX1)	27	ADC4
8	PB7 (XTAL/TOX2)	28	ADC7
9	PD0 (LXD)	29	PC4 (AREF)
10	PD1 (LXD)	30	VCC
11	PD2 (DNT0)	31	VCC
12	PD3 (DNT1)	32	VCC
13	PD4 (XCLT0)	33	AVCC
14	PD5 (IL)	34	AREF
15	PD6 (AD0)	35	GND
16	PD7 (AD1)	36	GND

Creating a Schematic

- Many vendor supplied integrated libraries contain much more than just a schematic drawing and PCB footprint
- Shown below is a rendering of a 3D Image of the chip. Altium has an option to view the PCB in 3D, and this image will represent the Atmega8 in the 3D rendering
- Also included are SPICE files and/or IBIS files (more on this later)



Creating a Schematic

- Place the Atmega8 into the schematic drawing
 - You can always move the location later
 - What function does pin 20 perform ?

Altium Designer 6.9 - H:\EE495 Altium\EE495 Project\Main.SchDoc * - PROJECT-EE495.PrjPCB, Licensed to University of Nevada Las Vegas

Workspace1.Dsn\Wk | Workspace
PROJECT-EE495.PrjPCB | Project
File View | Structure Editor

PROJECT-EE495.PrjPCB *
Source Documents
Main.Sch.Doc
Libraries
Compiled Libraries
Atmel Microcontroller 8-Bit AVR
Miscellaneous Connectors.IntLib
Miscellaneous Devices.IntLib
NSC Operational Amplifier.IntLib
ON Semi Operational Amplifier.I
PCB Library Documents
Chip Capacitor - 2 Contacts.Pc
Chip Diode - 2 Contacts.PcbLib
Chip Inductor - 2 Contacts.Pcb
Chip Resistor - 2 Contacts.PcbLib
Crystal Oscillator.PcbLib
Miscellaneous Connector PCB
Miscellaneous Devices PCB.Pc
Miscellaneous Devices.PcbLib
Miscellaneous.PcbLib

Pin	Function	Pin	Function
1	AVCC	16	PC0 (ADC0)
2	AVCC	17	PC1 (ADC1)
3	AVCC	18	PC2 (ADC2)
4	AVCC	19	PC3 (ADC3)
5	AVCC	20	PC4 (ADC4)
6	AVCC	21	PC5 (ADC5)
7	AVCC	22	PC6 (ADC6)
8	AVCC	23	PC7 (ADC7)
9	AVCC	24	PC8 (ADC8)
10	AVCC	25	PC9 (ADC9)
11	AVCC	26	PC10 (ADC10)
12	AVCC	27	PC11 (ADC11)
13	AVCC	28	PC12 (ADC12)
14	AVCC	29	PC13 (ADC13)
15	AVCC	30	PC14 (ADC14)
16	AVCC	31	PC15 (ADC15)
17	AVCC	32	PC16 (ADC16)
18	AVCC	33	PC17 (ADC17)
19	AVCC	34	PC18 (ADC18)
20	AVCC	35	PC19 (ADC19)
21	AVCC	36	PC20 (ADC20)

Atmega8-16M

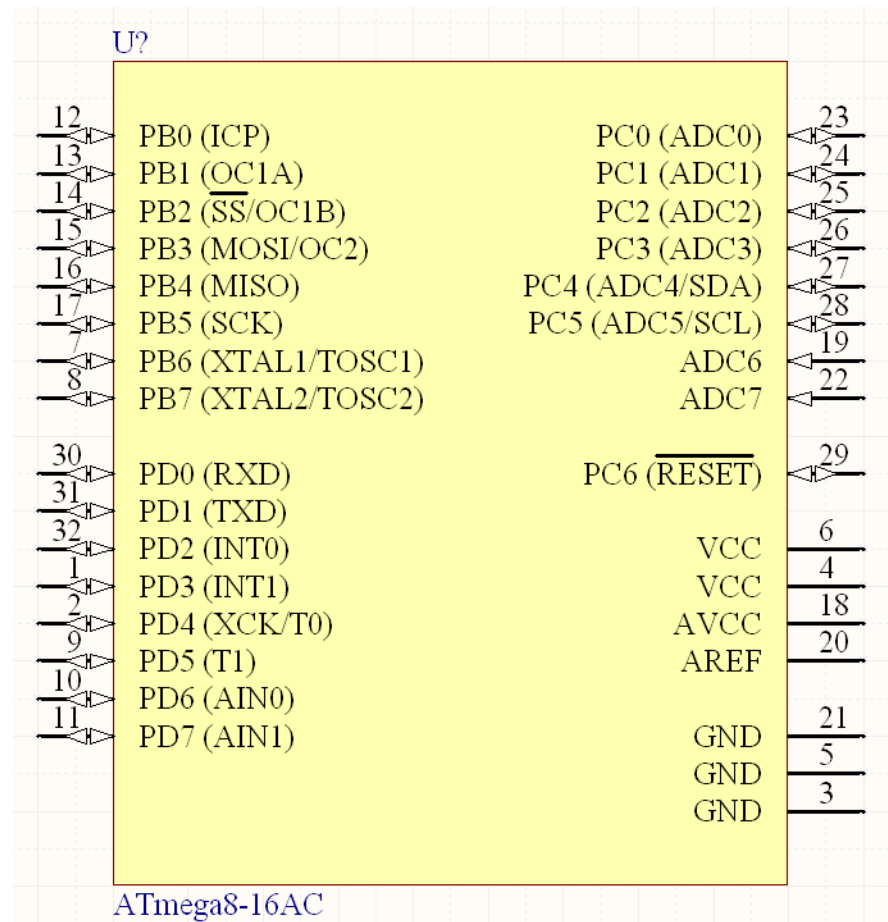
Pinout table for Atmega8-16M:

Pin	Function	Pin	Function
1	AVCC	16	PC0 (ADC0)
2	AVCC	17	PC1 (ADC1)
3	AVCC	18	PC2 (ADC2)
4	AVCC	19	PC3 (ADC3)
5	AVCC	20	PC4 (ADC4)
6	AVCC	21	PC5 (ADC5)
7	AVCC	22	PC6 (ADC6)
8	AVCC	23	PC7 (ADC7)
9	AVCC	24	PC8 (ADC8)
10	AVCC	25	PC9 (ADC9)
11	AVCC	26	PC10 (ADC10)
12	AVCC	27	PC11 (ADC11)
13	AVCC	28	PC12 (ADC12)
14	AVCC	29	PC13 (ADC13)
15	AVCC	30	PC14 (ADC14)
16	AVCC	31	PC15 (ADC15)
17	AVCC	32	PC16 (ADC16)
18	AVCC	33	PC17 (ADC17)
19	AVCC	34	PC18 (ADC18)
20	AVCC	35	PC19 (ADC19)
21	AVCC	36	PC20 (ADC20)

Files | Projects | Navigator | SCH Filter | Editor | Mask Level | Clear
X:500 Y:160 Grid:10 | System | Design Compiler | SCH | Help | Instruments

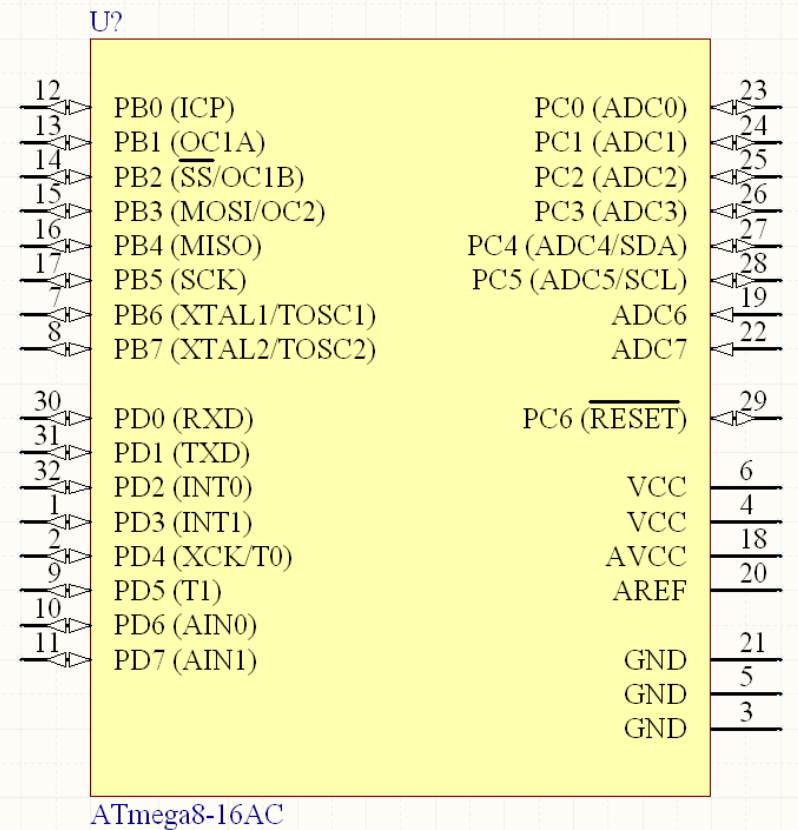
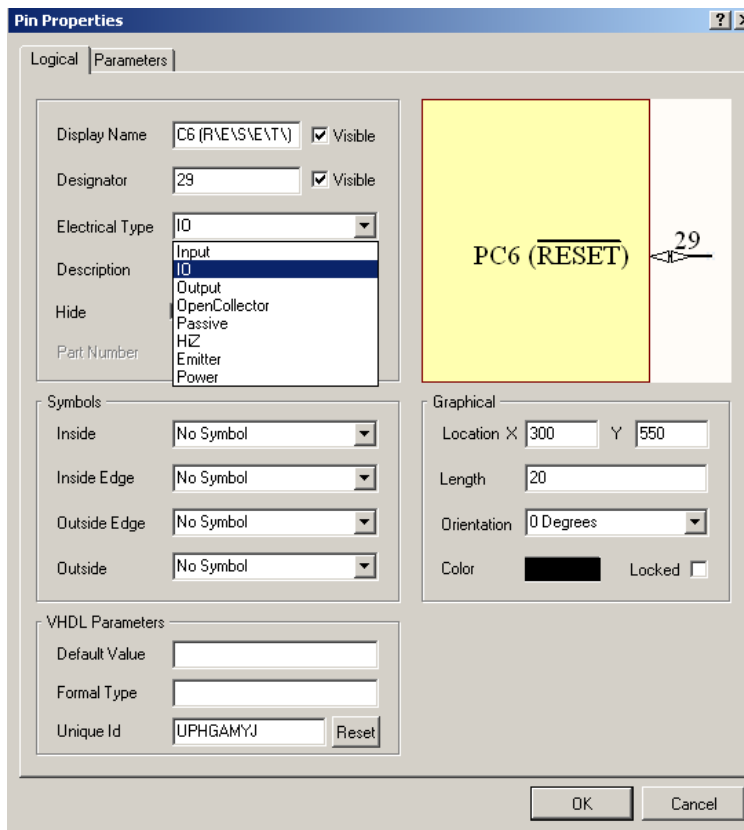
Zooming In and Out

- You can't see the function of pin 20 because the image is zoomed out.
To Zoom In/Out
 - Wheel Mouse and Control Key
 - Page Up/Page Down Key
- To Pan Left/Right/Up/Down
 - Wheel Mouse and Shift Key
 - Left/Right/Up/Down Arrow (Small Pan)
 - Shift Key and Left/Right/Up/Down Arrow (Larger Pan)



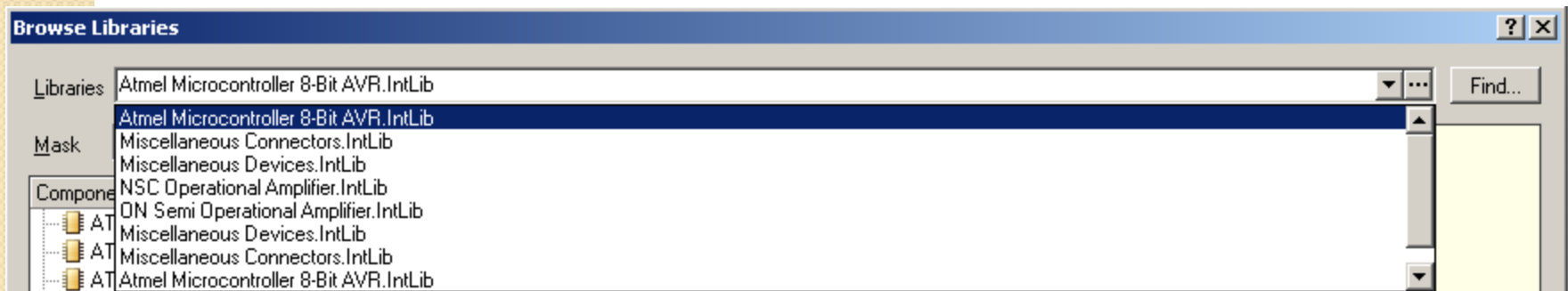
Pin Functions

- Pins can be setup for the following functions:
 - Input, Output, I/O
 - Open Collector, Passive, HiZ, Emitter,
 - Power
- Depending on how the pin is set, will determine how the DRC (Design Rule Check) Interprets the Pin and what connections are allowed



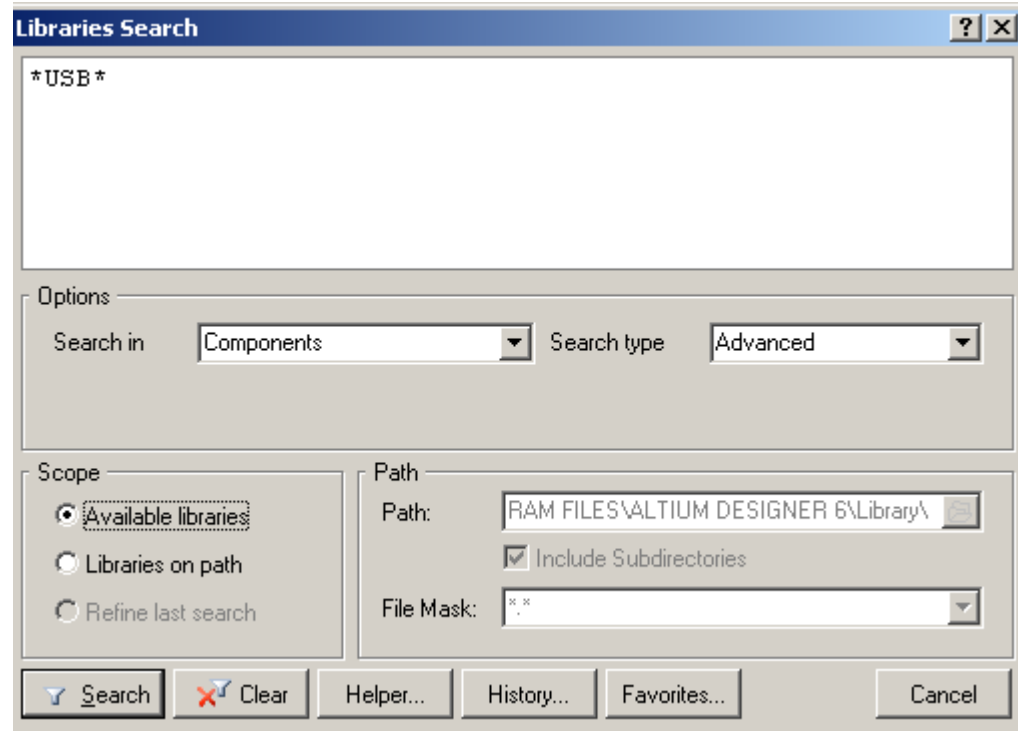
Add the USB-B Connector

- It seemed obvious which library contained the Atmel Atmega8, but now we need to add a USB-B connector.
- An easier method than searching each component and each library is to search the libraries.
- Click the 'Find' button to the right of the library selection
(It doesn't matter which library is selected/active)



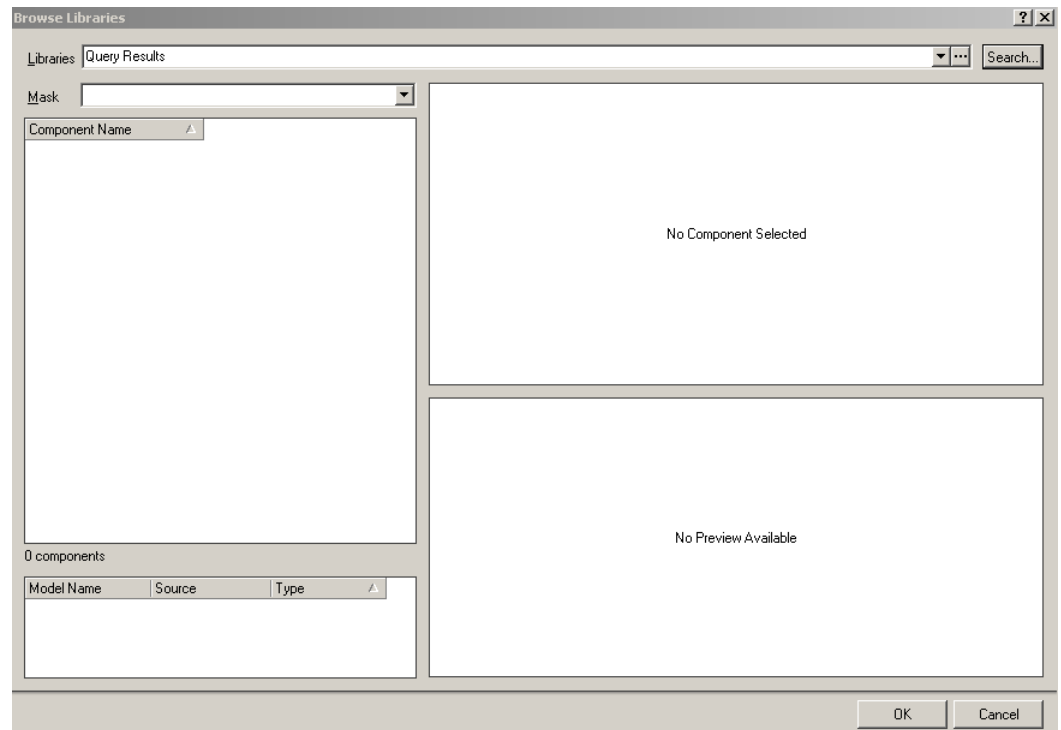
Add the USB-B Connector

- Type in the keyword you are looking for
- We are looking for a USB-B connector, For now let's just search for 'USB' and narrow down the search after that.
- NOTE: Using wildcards * before and after allow for words such as xxx**USB**xxx to be recognized.



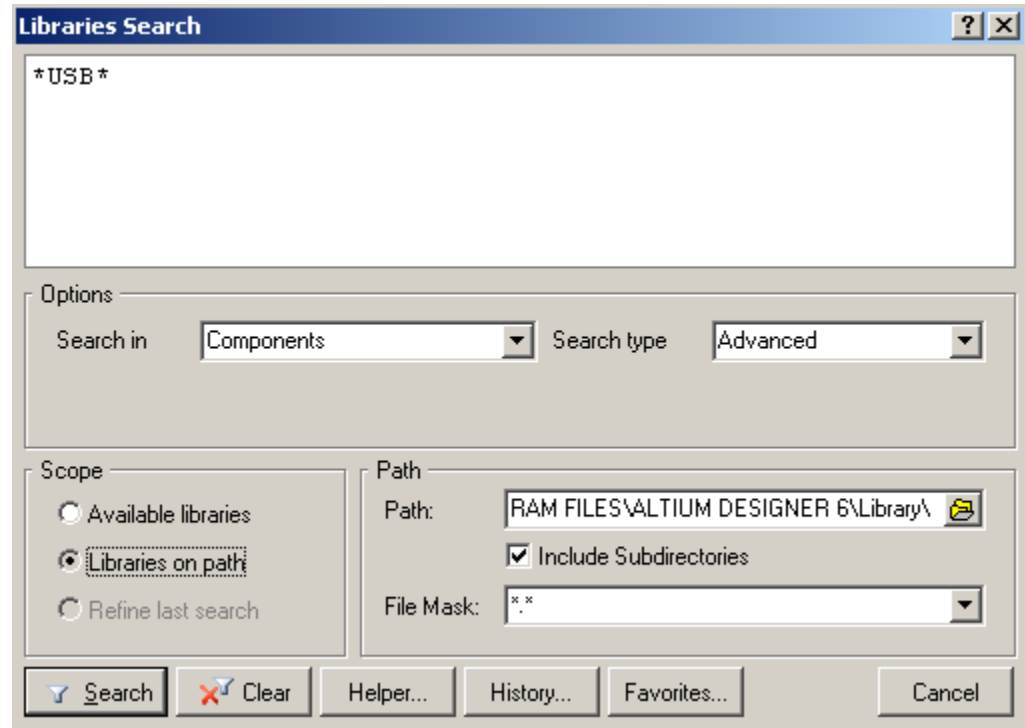
Add the USB-B Connector

- The search can take a while depending on how many libraries you are searching through.
- We are searching through only the libraries we added to the project, so it should happen very quickly.
- The result of this search came back negative, That means there are no USB items in any of our project libraries.
- Click the 'Search' tab in the top right corner



Add the USB-B Connector

- Change the scope of the search from available libraries (available to the project) to ALL libraries in the path.
- The path will automatically be entered for the default library location in Altium
- * make sure 'include subdirectories' is ticked.
- Click the search button



Add the USB-B Connector

- USB is a very common term, so very quickly the list fills up
- Expanding the box to read the description helps with identifying what each part is (in case names such as '1-353576-1' mean nothing to you)
- We are looking for a SINGLE Through Hole, Right Angle, USB-B Receptacle socket, which is located at the 8th row down

The screenshot shows a 'Browse Libraries' dialog box with a table of components. The 8th row is highlighted, showing the component '1-353576-1' from the library 'AMP Serial Bus USB.IntLib'. The description is 'USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 1-147'. To the right of the table is a schematic diagram of the component, labeled 'J?' and '1-353576-1'. The schematic shows a yellow box with four pins on the left labeled 1, 2, 3, and 4, and four pins on the right labeled VBUS, D-, D+, and GND. Below the schematic is a footprint diagram showing a red rectangular component with pins 5 and 6.

Component Name	Library	Description	Footp
1-353576-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35357	
1-353576-2	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35357	
1-353583-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35358	
1-353583-4	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35358	
1-440128-0	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, A Type, Receptacle, 4x4x 44012	
1-440128-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, A Type, Receptacle, 4x4x 44012	
1-440128-2	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, A Type, Receptacle, 4x4x 44012	
1-1470156-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 1-147	
1-1470156-2	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 1-147	
1-1470168-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, A Type, Receptacle, 4x4 f 14701	
4-1437106-7	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 14371	
4-1437106-8	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 14371	
353576-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35357	
353583-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35358	
353583-4	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, SMT, A Type, Receptacle, 4 Position 35358	
440068-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, Reverse Type, Receptacle 44006	
440119-1	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 44011	
440119-2	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 44011	
440119-3	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 44011	
440119-4	AMP Serial Bus USB.IntLib	USB 1.1, Right Angle, Thru-Hole, B Type, Receptacle, 4 Po 44011	

251 components QuickLogic pASIC 2.IntLib

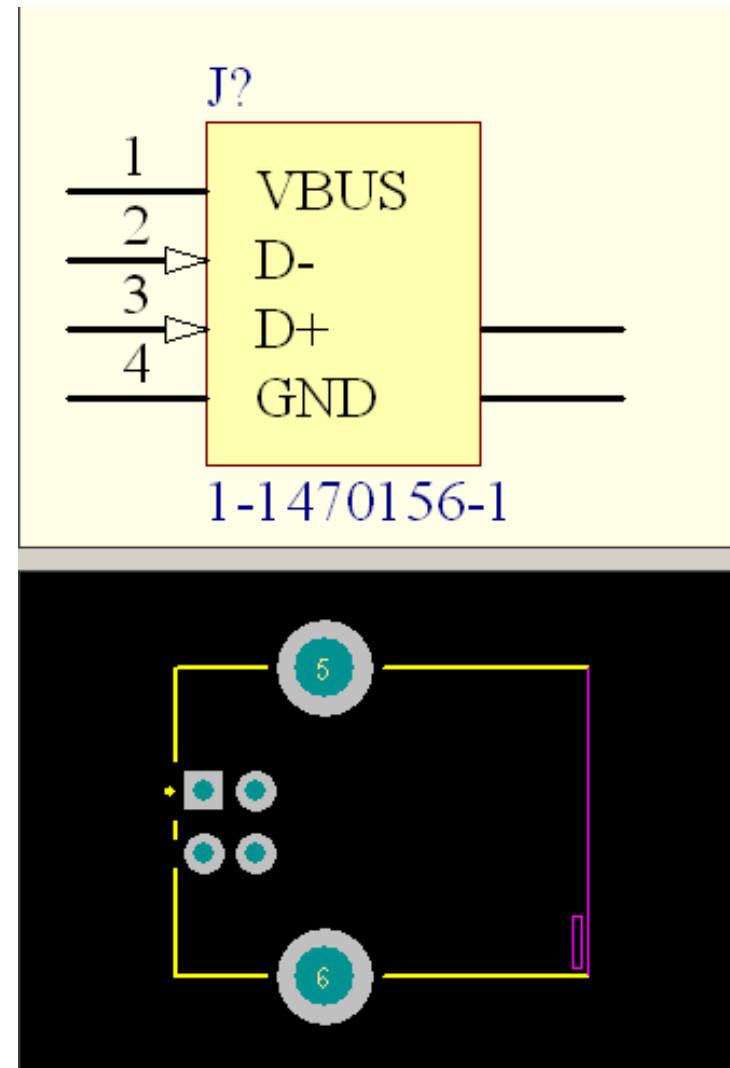
Model Name	Source	Type
353576	AMP Serial Bus USB Footprint	
USB1.1		Signal Integrity

Add the USB-B Connector

- Select the 1-1470156-1 and click OK
- You will get a message stating that the library is not available to the project, but Altium is willing to install it for you, click yes



- Click 'OK' again to add the part



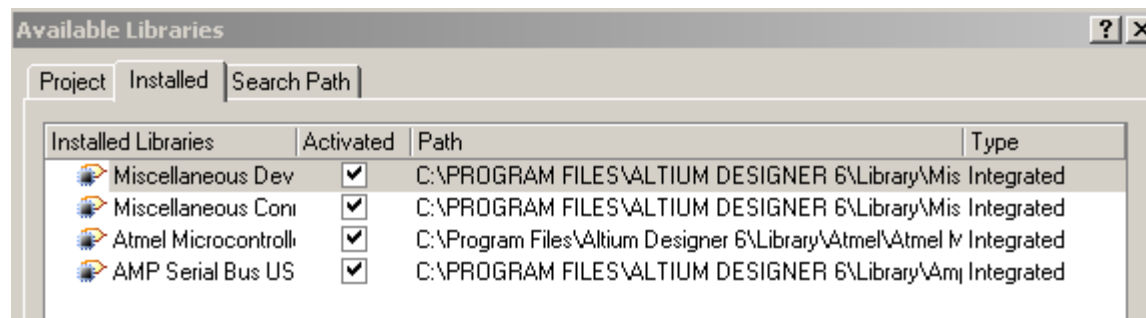
Add the USB-B Connector

- The library AMP Serial Bus is added automatically to the 'Installed' libraries, NOT the project libraries
- This action DOES NOT MOVE ANY LIBRARIES PHYSICAL LOCATION
- For ease of future design, it is a good idea to move libraries added by this method to your ever-growing list of libraries
- You can do this by copying the AMP Serial Bus USB from the path given in the 'available libraries' below

C:\Program Files\Altium Designer 6\Library\Amp

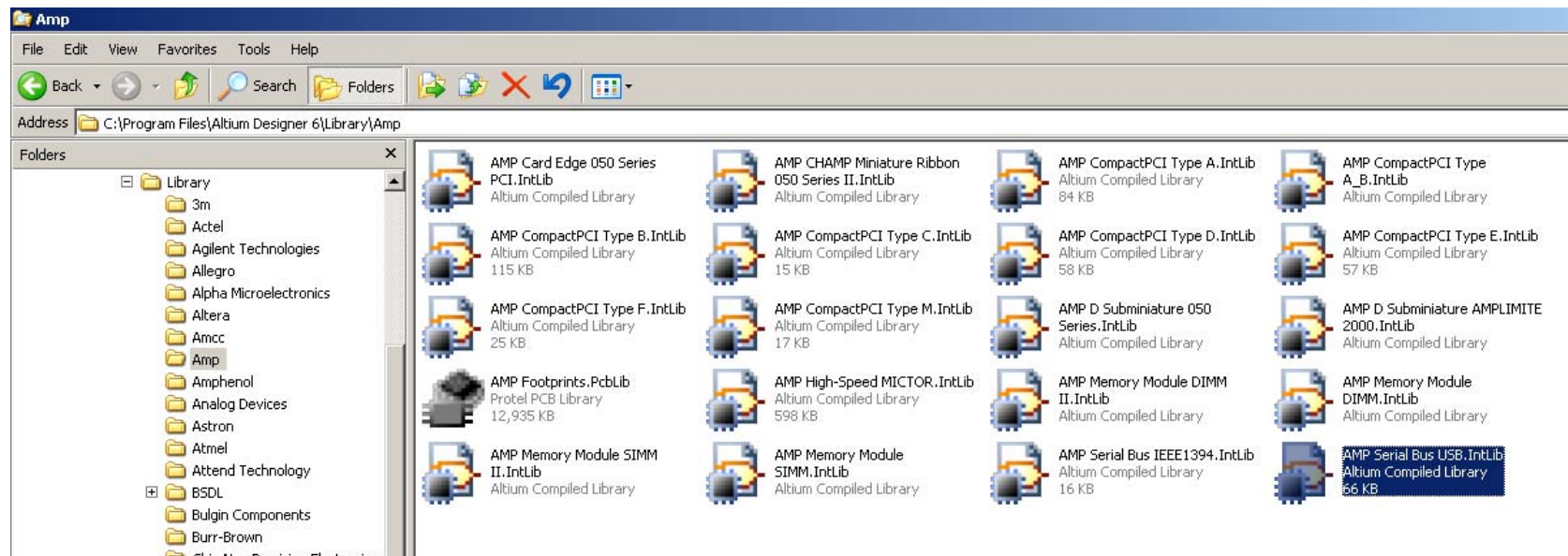
to your custom library in your flash drive

X:\EE495 Altium\EE495 Project\Altium Libraries



Add the USB-B Connector

- AMP makes several different libraries, so be sure to copy the correct one!



Rotating Parts

- The easiest way to rotate parts is to select the item and hold the left mouse button down, and tap the space button, this will rotate 90 degrees each tap.
- Another method is to double click the component to open up the component properties window, and in the bottom left corner you can set the orientation
- NOTE: the 'mirrored' tab will flip the component about its axis

Component Properties

Properties

Designator: J? Visible Locked

Comment: 1-1470156-1 Visible

Description: ht Angle, Thru-Hole, B Type, Receptacle, 4 Position, White

Unique Id: TLWEKRLR

Type: Standard

Library Link

Design Item ID: 1-1470156-1

Library Name: AMP Serial Bus USB.IntLib

Table Name:

Sub-Design Links

Sub-Project: None

Configuration: None

Graphical

Location X: 470 Y: 470

Orientation: 180 Degrees Locked

Mode: Normal Lock Pins Mirrored

Show All Pins On Sheet (Even if Hidden)

Local Colors

Parameters for J? - 1-1470156-1

Visible	Name	Value	Type
<input type="checkbox"/>	ComponentLink2URL	http://ecommas.tycoelectronics.com/c	STRING
<input type="checkbox"/>	ContactMaterial	Phosphor Bronze	STRING
<input type="checkbox"/>	ContactPlating	Gold over Nickel	STRING
<input type="checkbox"/>	DatasheetDocument	02-Jun-2001	STRING
<input type="checkbox"/>	DrillSize	Corresponds to nominal value.	STRING
<input type="checkbox"/>	HousingMaterial	Glass-Filled Thermoplastic	STRING
<input type="checkbox"/>	PackageDocument	02-Jun-2001	STRING
<input type="checkbox"/>	PackageReference	1-1470156	STRING
<input type="checkbox"/>	PanelGround	Without	STRING
<input type="checkbox"/>	PCB_Layout	Complies with manufacturer's recomme	STRING
<input type="checkbox"/>	Published	15-Jul-2004	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING
<input type="checkbox"/>	TerminationLength	3.00mm [0.118in.]	STRING

Models for J? - 1-1470156-1

Name	Type	Description
USB1.1	Signal Integrity	
1-1470156	Footprint	Connector; USB1.1, Type B, Thru-Hole; 4 Position; Right

Component Properties

- In the top left corner of the component properties, you can set comments, descriptions, designator values and library link information
- **NOTE:** Unique ID is a unique value which links the schematic part with the PCB document.
- In the bottom left corner, you can set graphical properties such as orientation, mirrored, locked (stuck in schematic)
- The 'Edit Pins' button will allow you to reassign pins, pin functions, and pin numbers. This should only be used if you are absolutely positive you know what you are doing!!

<u>Properties</u>	
Designator	J? <input checked="" type="checkbox"/> Visible <input type="checkbox"/> Locked
Comment	1-1470156-1 <input checked="" type="checkbox"/> Visible
	<< < > >> Part 1/1 <input type="checkbox"/> Locked
Description	ht Angle, Thru-Hole, B Type, Receptacle, 4 Position, White
Unique Id	TLWEKRLR <input type="button" value="Reset"/>
Type	Standard
<u>Library Link</u>	
Design Item ID	1-1470156-1 <input type="button" value="Choose"/>
<input checked="" type="checkbox"/> Library Name	AMP Serial Bus USB.IntLib
<input type="checkbox"/> Table Name	
	<input type="button" value="Validate"/>
<u>Sub-Design Links</u>	
Sub-Project	None
Configuration	None
<u>Graphical</u>	
Location X	470 Y 470
Orientation	180 Degrees <input type="checkbox"/> Locked
Mode	Normal <input checked="" type="checkbox"/> Lock Pins <input type="checkbox"/> Mirrored
	<input type="checkbox"/> Show All Pins On Sheet (Even if Hidden)
	<input type="checkbox"/> Local Colors
<input type="button" value="Edit Pjns..."/>	

Component Properties

- In the top right corner of component properties of a standard component will contain information such as publishing date, termination length, URL's, drill sizes, etc.
- NOTE: You can add your own custom fields (such as ordering information, ordering part numbers, etc.)
- NOTE: You can also add a custom PCB rule to the component while you are still in schematic capture mode
- The bottom right corner contains model information for the component, such as footprint model information and signal integrity models.

Visible	Name	Value	Type
<input type="checkbox"/>	ComponentLink2URL	http://ecommas.tycoelectronics.com/c	STRING
<input type="checkbox"/>	ContactMaterial	Phosphor Bronze	STRING
<input type="checkbox"/>	ContactPlating	Gold over Nickel	STRING
<input type="checkbox"/>	DatasheetDocument	02-Jun-2001	STRING
<input type="checkbox"/>	DrillSize	Corresponds to nominal value.	STRING
<input type="checkbox"/>	HousingMaterial	Glass-Filled Thermoplastic	STRING
<input type="checkbox"/>	PackageDocument	02-Jun-2001	STRING
<input type="checkbox"/>	PackageReference	1-1470156	STRING
<input type="checkbox"/>	PanelGround	Without	STRING
<input type="checkbox"/>	PCB_Layout	Complies with manufacturer's recomme	STRING
<input type="checkbox"/>	Published	15-Jul-2004	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING
<input type="checkbox"/>	TerminationLength	3.00mm [0.118in.]	STRING

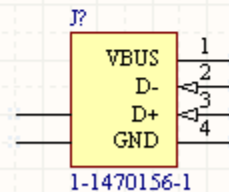
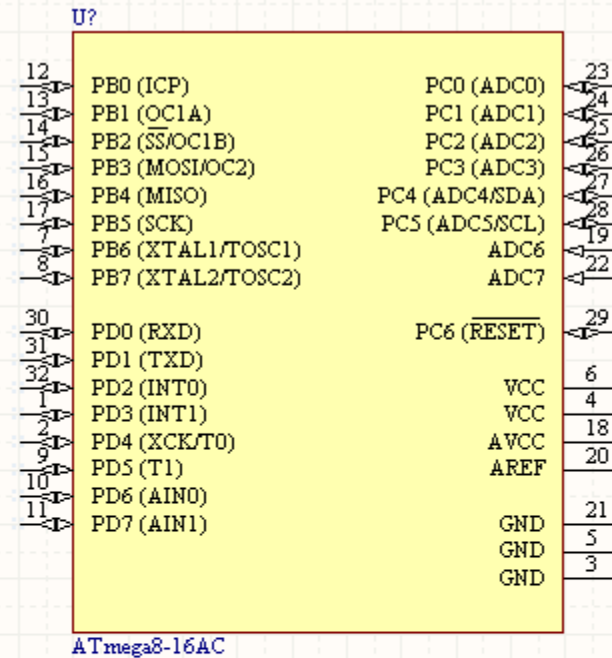
Buttons: Add... Remove... Edit... Add as Rule...

Name	Type	Description
USB1.1	Signal Integrity	
1-1470156	Footprint	Connector; USB1.1, Type B, Thru-Hole; 4 Position; Right

Buttons: Add... Remove... Edit...

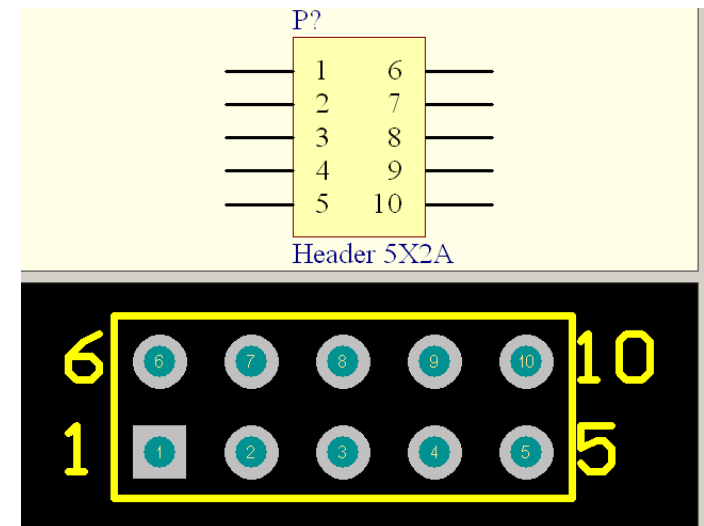
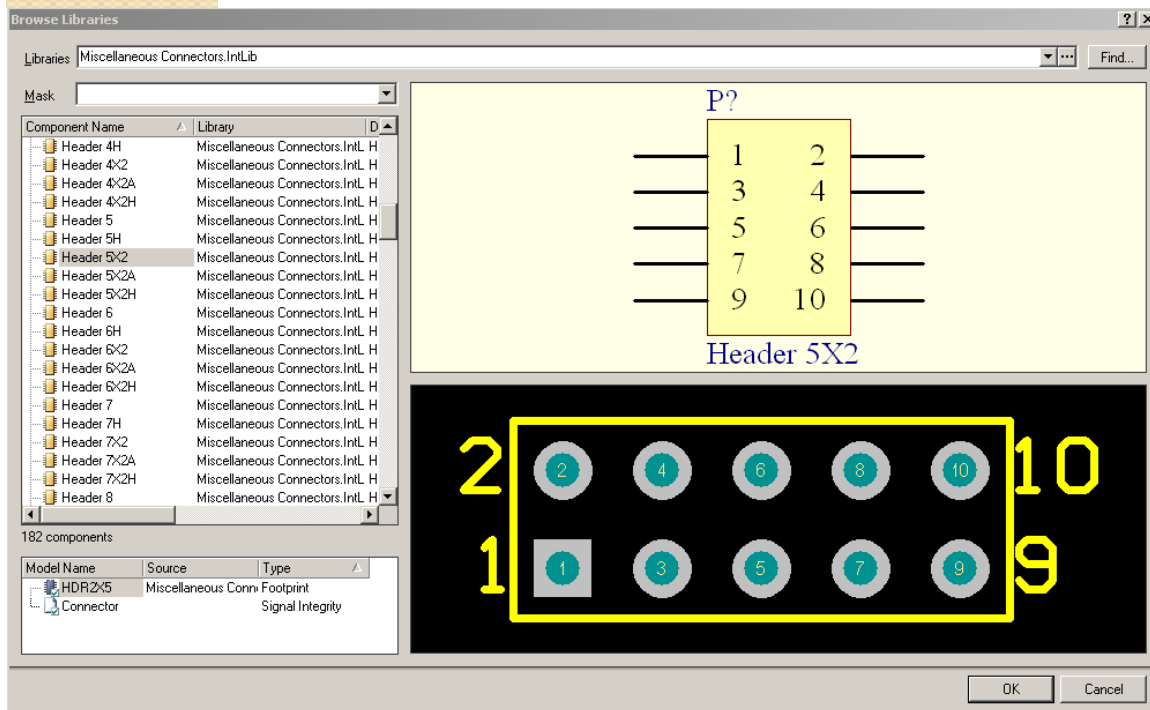
Component Properties

- Mirror the USB-B connector and orient the parts so they match the diagram to the right.



Programming Header

- Add a part from the Miscellaneous Connectors library called 'Header 5x2'
- Make sure to use caution when selecting parts. If you accidentally add 'Header 5x2a' they make look the same, but look at the numbering system of the pins. Always make sure the numbering method is the same as you expect it to be.



FTDI Chip- FT232 (TSSOP28)

- A quick search for the FT232 chip we will be using gives the following result
- Although there are three matches for the FT232, We are actually looking for the FT232RL as we will see on the next slide

The screenshot shows a software interface titled "Browse Libraries" with a search bar containing "Query Results". A table lists three components:

Component Name	Library	Desc
FT232BL	FTDI Communication UART.Inl USB	
FT232BM	FTDI Communication UART.Inl USB	
FT232BQ	FTDI Communication UART.Inl USB	

Below the table, it says "3 components". At the bottom, there is a table of footprint models:

Model Name	Source	Type
LQFP-32_L	FTDI Communicator Footprint	
LQFP-32_M	FTDI Communicator Footprint	
LQFP-32_N	FTDI Communicator Footprint	

To the right of the table is a pinout diagram for the FT232RL chip, showing a 28-pin package with pins labeled: 3VDDI, AVCC, VCC, VCC, VCC, VCCIO, I2C1, I2C2, I2C3, I2C4, I2C5, I2C6, I2C7, I2C8, I2C9, I2C10, I2C11, I2C12, I2C13, I2C14, I2C15, I2C16, I2C17, I2C18, I2C19, I2C20, I2C21, I2C22, I2C23, I2C24, I2C25, I2C26, I2C27, I2C28, I2C29, I2C30, I2C31, I2C32, I2C33, I2C34, I2C35, I2C36, I2C37, I2C38, I2C39, I2C40, I2C41, I2C42, I2C43, I2C44, I2C45, I2C46, I2C47, I2C48, I2C49, I2C50, I2C51, I2C52, I2C53, I2C54, I2C55, I2C56, I2C57, I2C58, I2C59, I2C60, I2C61, I2C62, I2C63, I2C64, I2C65, I2C66, I2C67, I2C68, I2C69, I2C70, I2C71, I2C72, I2C73, I2C74, I2C75, I2C76, I2C77, I2C78, I2C79, I2C80, I2C81, I2C82, I2C83, I2C84, I2C85, I2C86, I2C87, I2C88, I2C89, I2C90, I2C91, I2C92, I2C93, I2C94, I2C95, I2C96, I2C97, I2C98, I2C99, I2C100.

Below the pinout diagram is a 3D rendering of the FT232RL chip, showing a square package with pins on all four sides.

FTDI Chip- FT232 (TSSOP28)

- Like all aspects of PCB layout, you must pay attention to every detail on everything.
- The datasheet states that the FT232RL is a 28-pin SSOP package.
- Since this footprint is not available, we must create the part in a custom library

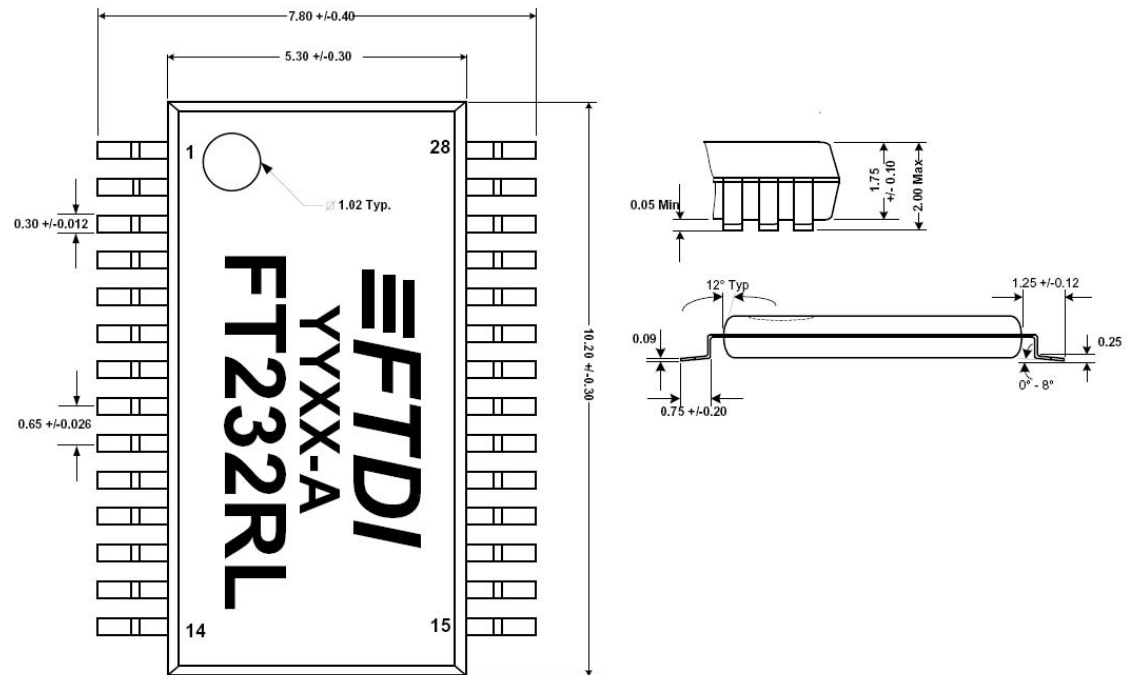
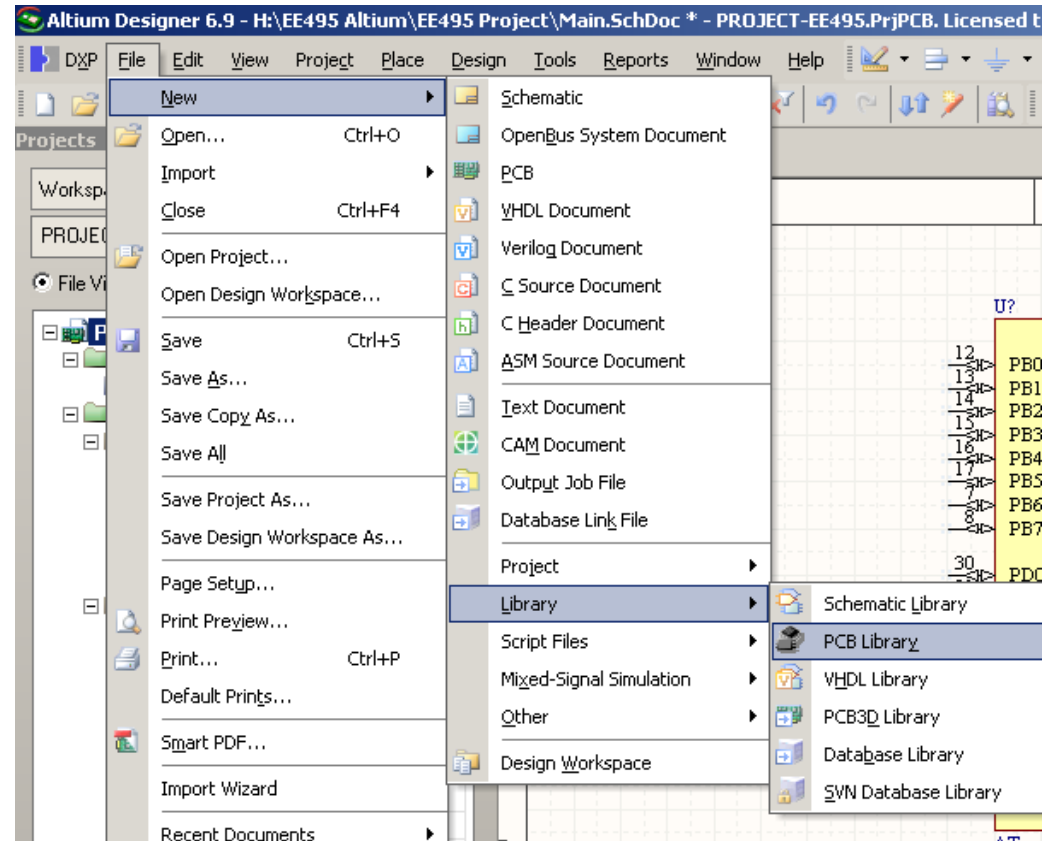


Figure 6 - SSOP-28 Package Dimensions

The FT232RL is supplied in a RoHS compliant 28 pin SSOP package. The package is lead (Pb) free and uses a 'green' compound. The package is fully compliant with European Union directive 2002/95/EC.

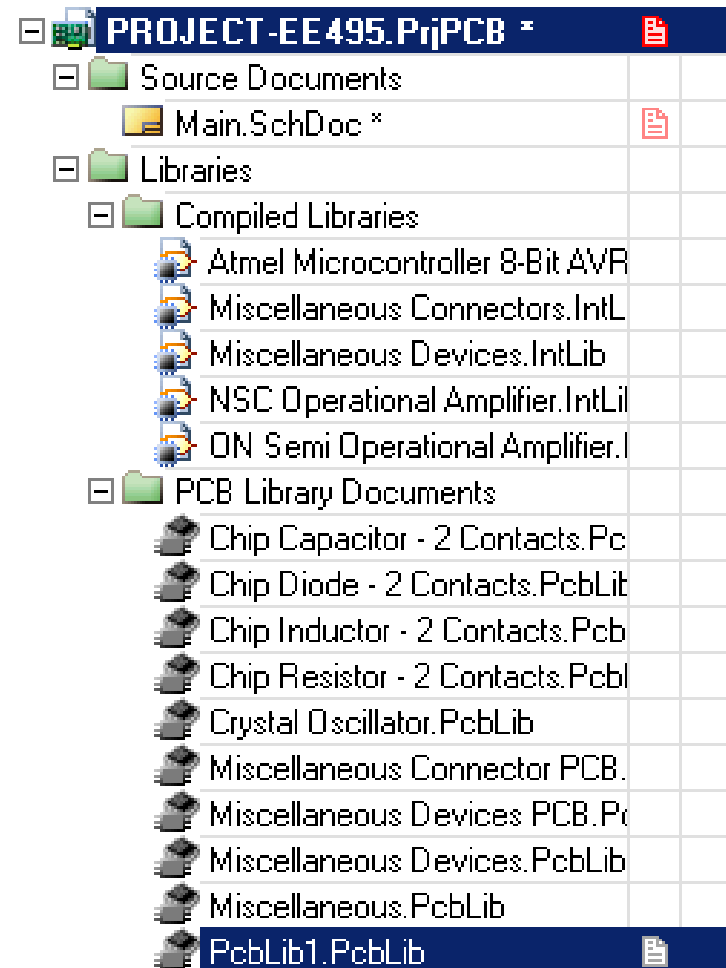
FTDI Chip- FT232 (TSSOP28)

- Follow the menu sequence shown to the right to create a new PCB library.
- This library will house ALL of your components



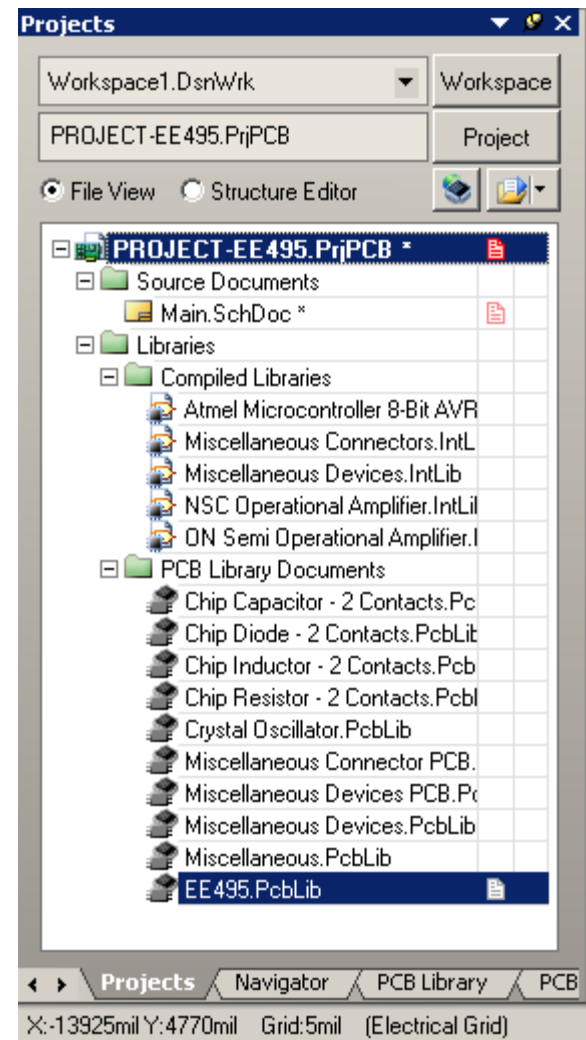
FTDI Chip- FT232 (TSSOP28)

- The new library is automatically added to the project.
- Right click and click 'save as' and rename the library to a more appropriate name: EE495 and save it in the directory with the rest of your libraries



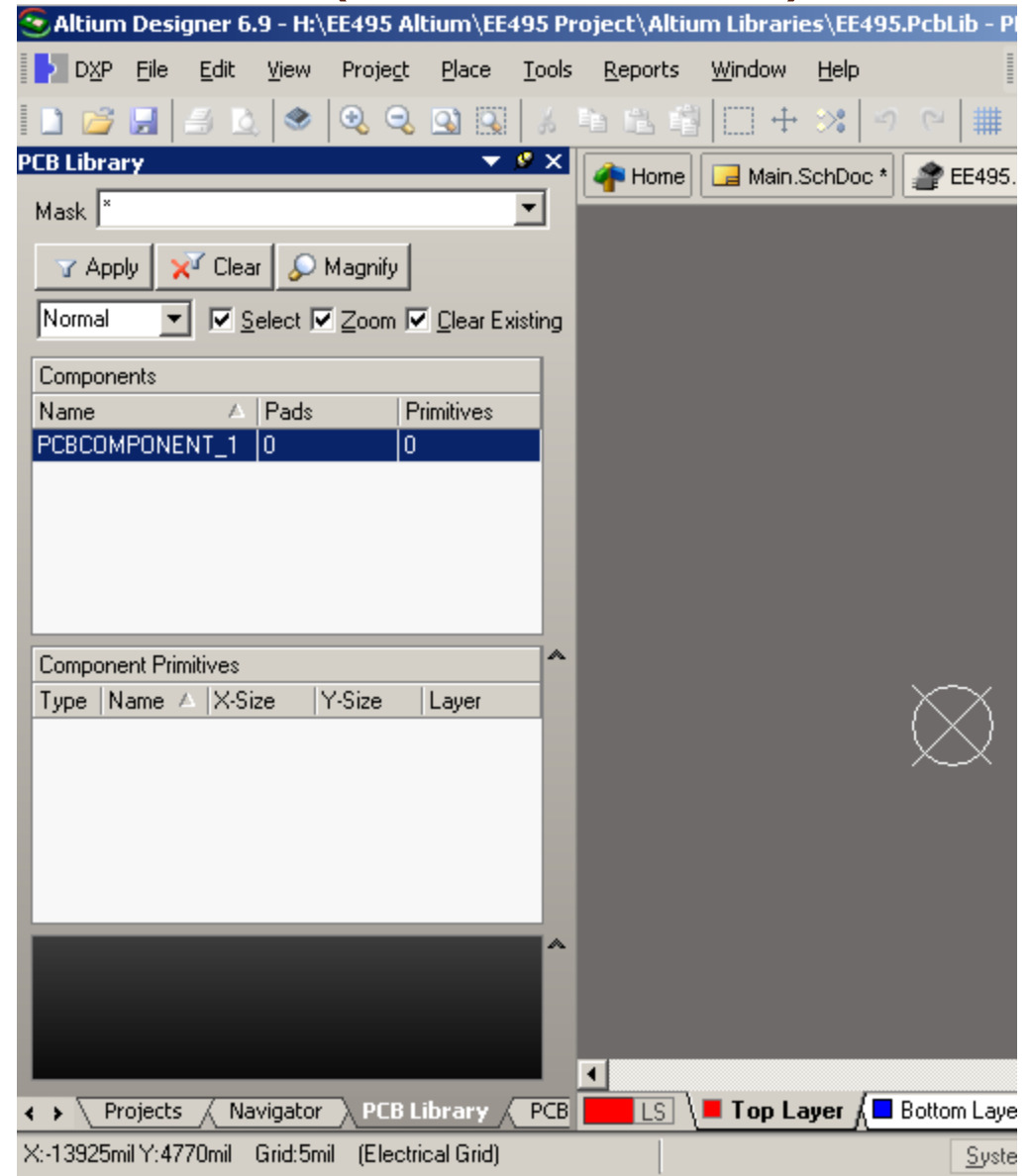
FTDI Chip- FT232 (TSSOP28)

- Notice the small tabs in the bottom left corner.
- Double click the new PCB library you created to open it, and click on the one titled 'PCB Library' to work inside that library.



FTDI Chip- FT232 (TSSOP28)

- With the PCB Library tab selected, we can see that there is only one component inside our library called PCBCOMPONENT_1



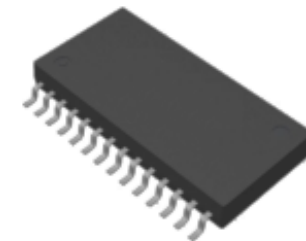
FTDI Chip- FT232 (TSSOP28)

- The first step is selecting what type of part it is. They don't specifically have SSOP, but SSOP is just a modified version of the SOP so select that
- There is a diagram of what the chip looks like to the right
- **NOTE:** check the note on the bottom of the wizard: All wizard measurements must be entered as metric (mm) units

Component Types

Name	Description	Included Packages
BGA	Ball Grid Array	BGA, CGA
BQFP	Bumpered Quad Flat Pack	BQFP
CFP	Ceramic Dual Flat Pack - Trimmed and formed Gullwing Leads	CFP
CHIP	Chip Components, 2-Pins	Capacitor, Inductor, Resistor
CQFP	Ceramic Quad Flat Pack - Trimmed and formed Gullwing Leads	CQFP
DPAK	Transistor Outline	DPAK
LCC	Leadless Chip Carrier	LCC
MELF	MELF Components, 2-Pins	Diode, Resistor
MOLDED	Molded Components, 2-Pins	Capacitor, Inductor, Diode
PLCC	Plastic Leaded Chip Carrier, Square - J Leads	PLCC
PQFP	Plastic Quad Flat Pack	PQFP, PQFP Exposed Pad
QFN	Quad Flat Pack No-Lead	QFN, LLP
QFN-2ROW	Quad Flat Pack No-Lead, 2 Rows, Square	Double Row QFN
SOIC	Small Outline Integrated Package, 1.27mm Pitch - Gullwing Leads	SOIC, SOIC Exposed Pad
SOJ	Small Outline Package - J Leads	SOJ
SOP	Small Outline Package - Gullwing Leads	SOP, SOP Exposed Pad
SOT143/343	Small Outline Transistor	SOT143, SOT343
SOT223	Small Outline Transistor	SOT223

The selected component is SOP.
This will allow you to generate SOP, SOP Exposed Pad packages.



NOTE: All wizard measurement dimensions are required to be entered as metric (mm) units.

FTDI Chip- FT232 (TSSOP28)

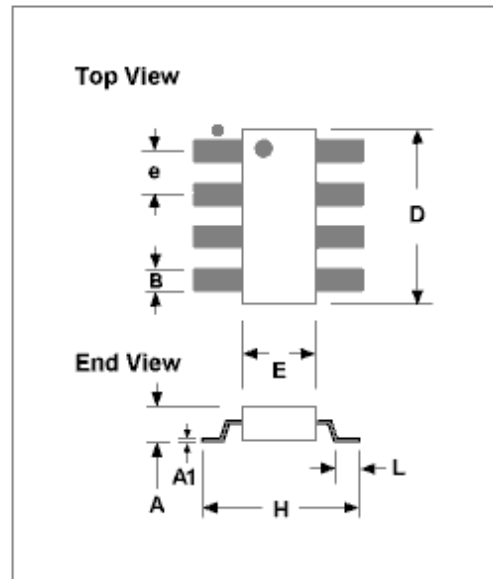
- At first this might seem a bit daunting and intimidating, but when you pull up the datasheet to find the values, you will see that the graphics and naming convention matches what we must entered

Overall Dimensions

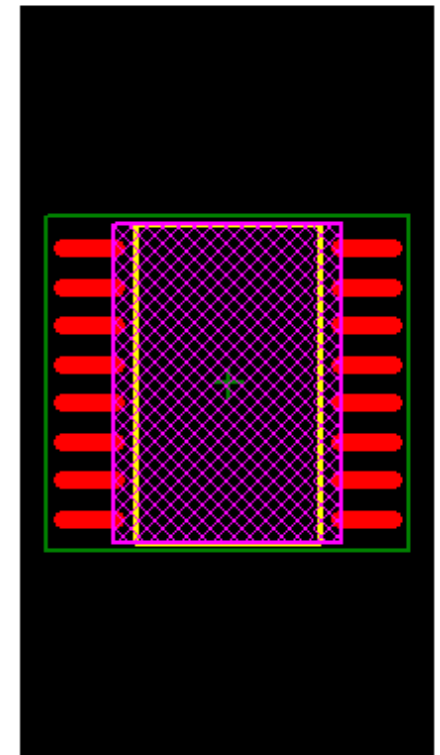
Width Range (H)	Minimum	9.97mm
	Maximum	10.63mm
Maximum Height (A)		2.65mm
Minimum standoff height (A1)		0.1mm
Maximum body Width (E)		7.5mm
Maximum body length (D)		10.5mm

Pin Information

Number of pins		16
Lead Width Range (B)	Minimum	0.31mm
	Maximum	0.51mm
Lead Length Range (L)	Minimum	0.4mm
	Maximum	1.27mm
Pitch (e)		1.27mm



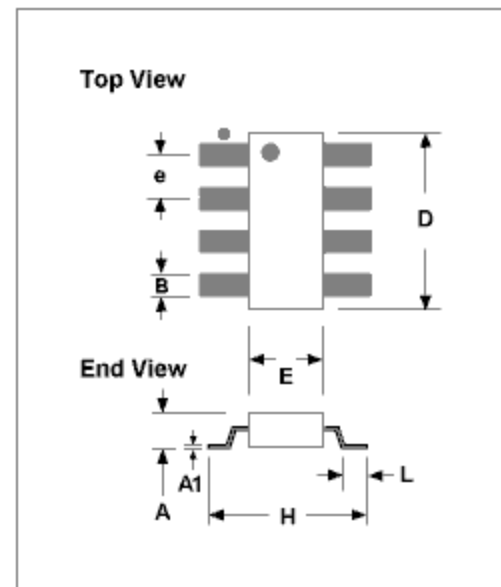
Preview



FTDI Chip- FT232 (TSSOP28)

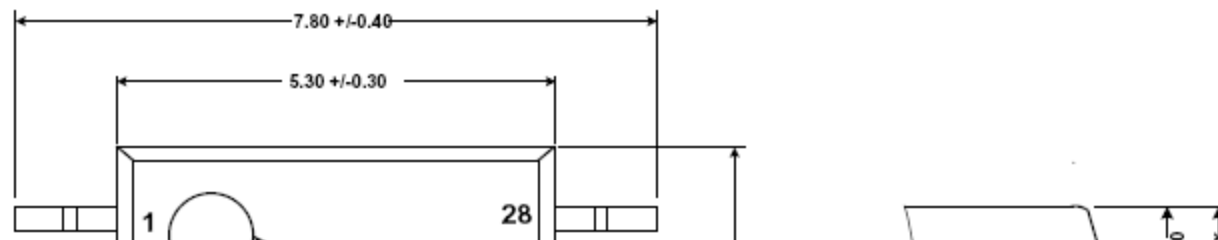
- The first entry is for width range. The datasheet gives a width range of $7.80 \pm 0.40\text{mm}$ (Min width = 7.4mm, Max = 8.2mm)

Width Range (H)	Minimum	9.97mm
	Maximum	10.63mm
Maximum Height (A)		2.65mm
Minimum standoff height (A1)		0.1mm
Maximum body Width (E)		7.5mm
Maximum body length (D)		10.5mm
Pin Information		
Number of pins		16
Lead Width Range (B)	Minimum	0.31mm
	Maximum	0.51mm



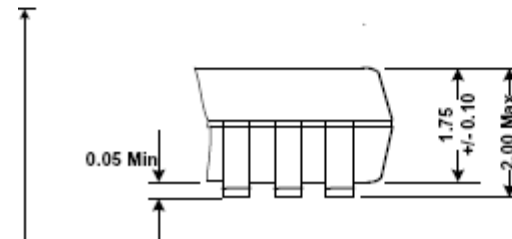
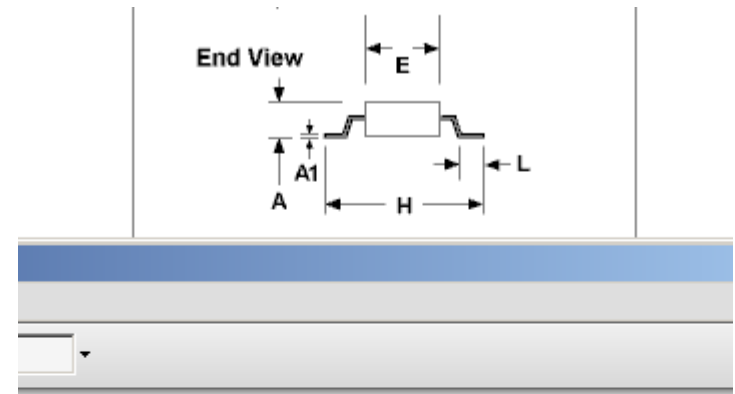
32 USB.pdf (SECURED) - Adobe Reader

Edit View Document Tools Window Help



FTDI Chip- FT232 (TSSOP28)

- The max height of the chip (A) is 2.00mm
- Enter this in for maximum height
- Question: Why would the height of the chip matter ??





FTDI Chip- FT232 (TSSOP28)

- Fill out the rest of the values, you should get the numbers below
- NOTE: Watch as you enter values, the graphic on the right will dynamically change

SOP Package Dimensions

Enter the required package values.

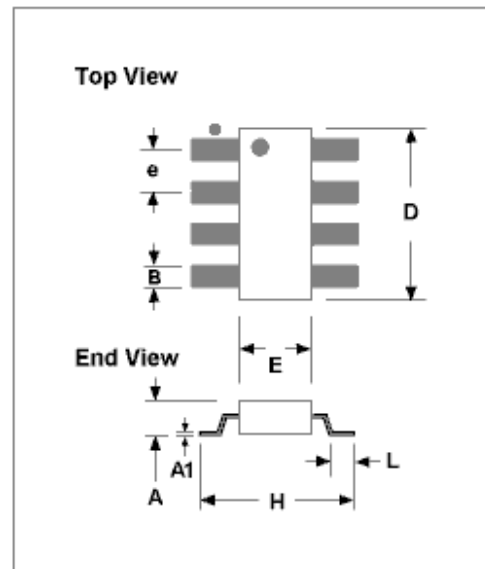


Overall Dimensions

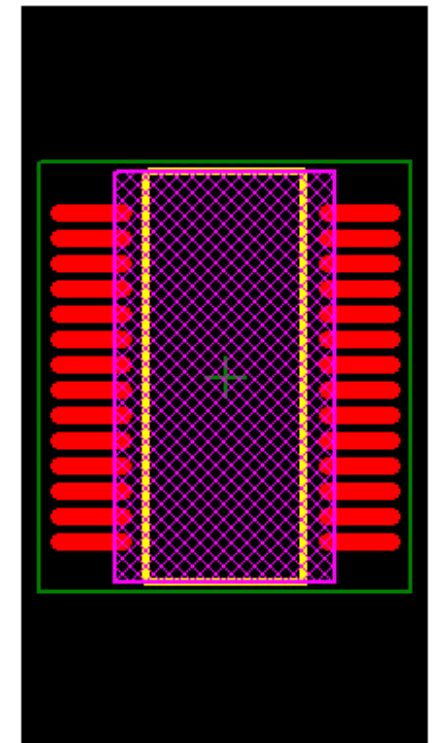
Width Range (H)	Minimum	<input type="text" value="7.4mm"/>
	Maximum	<input type="text" value="8.2mm"/>
Maximum Height (A)		<input type="text" value="2.00mm"/>
Minimum standoff height (A1)		<input type="text" value="0.05mm"/>
Maximum body Width (E)		<input type="text" value="5.6mm"/>
Maximum body length (D)		<input type="text" value="10.5mm"/>

Pin Information

Number of pins		<input type="text" value="28"/>
Lead Width Range (B)	Minimum	<input type="text" value="0.288mm"/>
	Maximum	<input type="text" value="0.312mm"/>
Lead Length Range (L)	Minimum	<input type="text" value="0.55mm"/>
	Maximum	<input type="text" value="0.95mm"/>
Pitch (e)		<input type="text" value="0.65mm"/>



Preview



FTDI Chip- FT232 (TSSOP28)

- Accept the automatically entered values for the next few slides until you get to the slide below
- Uncheck the 'use suggested values' and change the name to FT232RL
- Click 'Finish' and the part will be added to your library

SOP Footprint Description

The footprint values can now be inferred from the package dimensions.
You can review and modify them here.

Use suggested values

Name

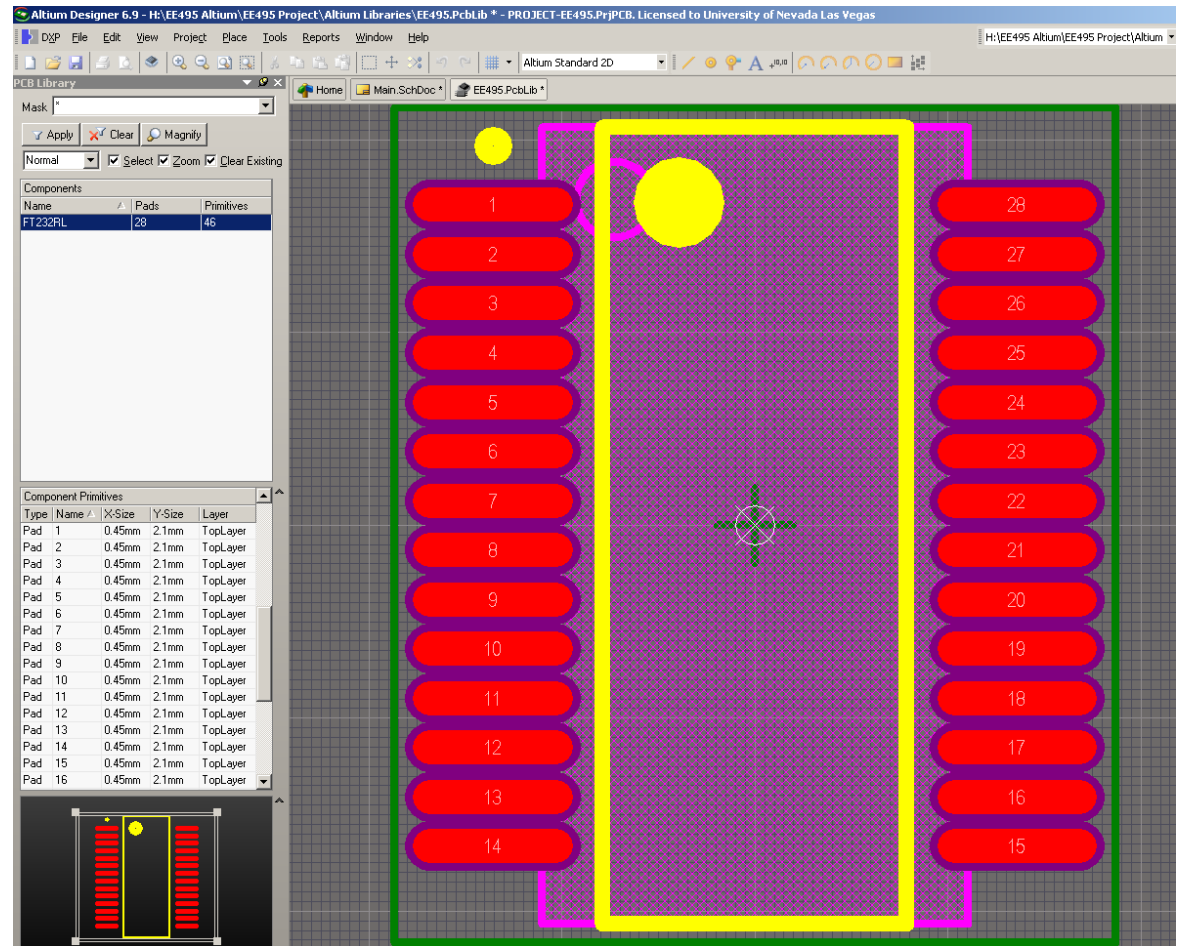
SOP65P780X200-28N

Description

SOP, 28-Leads, Body 10.5x5.6mm (max), Pitch 0.65mm, IPC Medium Density

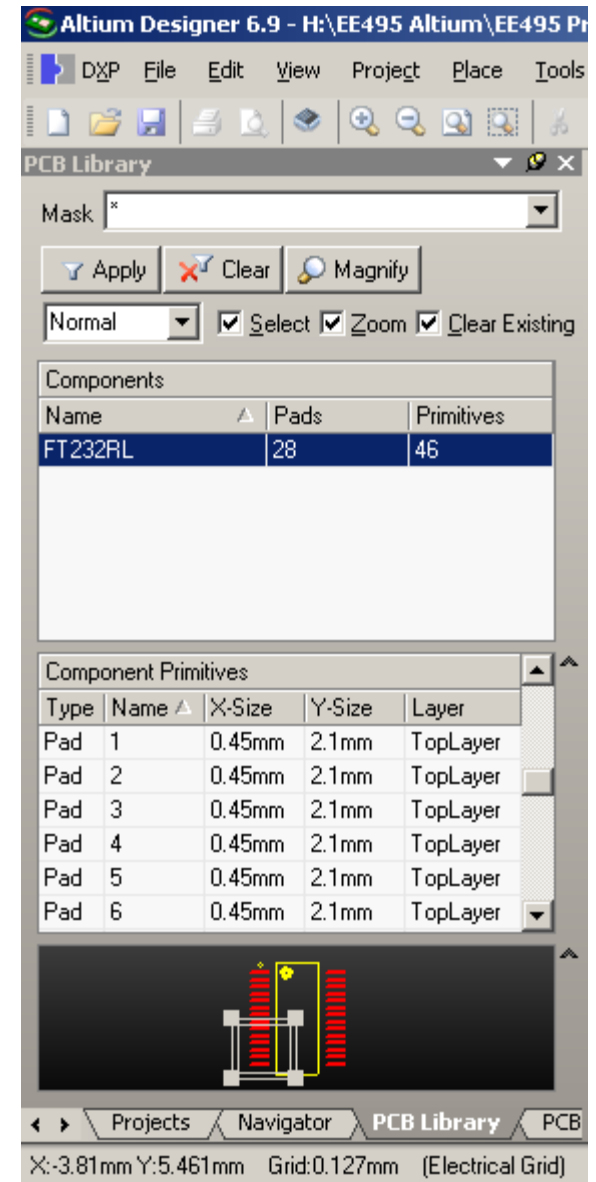
FTDI Chip- FT232 (TSSOP28)

- Your library will automatically open with your new part included.
- Notice how there are two yellow dots, those are 'pin 1 indicators' so the board is assembled properly
- You can now be confident that the FT232RL chip will fit on this footprint



FTDI Chip- FT232 (TSSOP28)

- Click the 'Projects' tab on the bottom left to return back to the project file hierarchy
- Since a PCB footprint is useless as far as a schematic is concerned, we must create a schematic drawing and link the drawing with the footprint we created



FTDI Chip- FT232 (TSSOP28)

- Keep in mind that we cannot keep the schematic drawing of the FT232 and change only the footprint because sometimes pins are added, removed, or are different.
- Shown to the right is the pinout of the device obtained from the datasheet

4. Device Pin Out and Signal Descriptions

4.1 28-LD SSOP Package

Page /

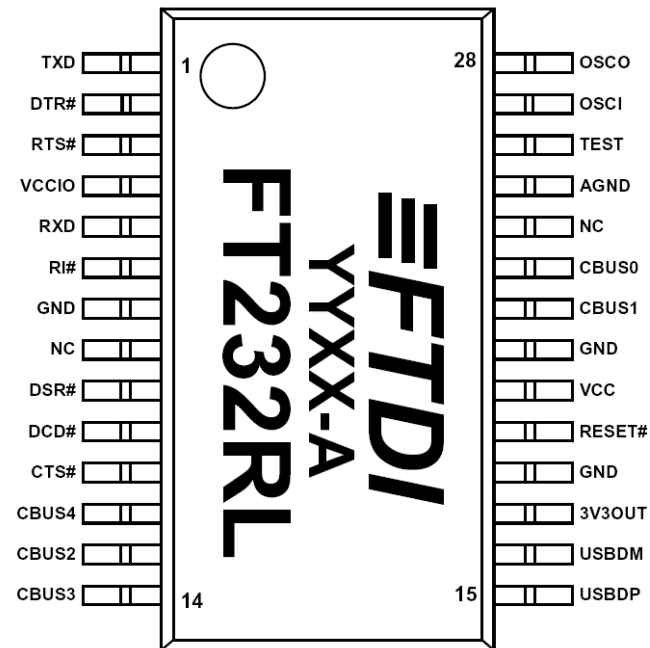
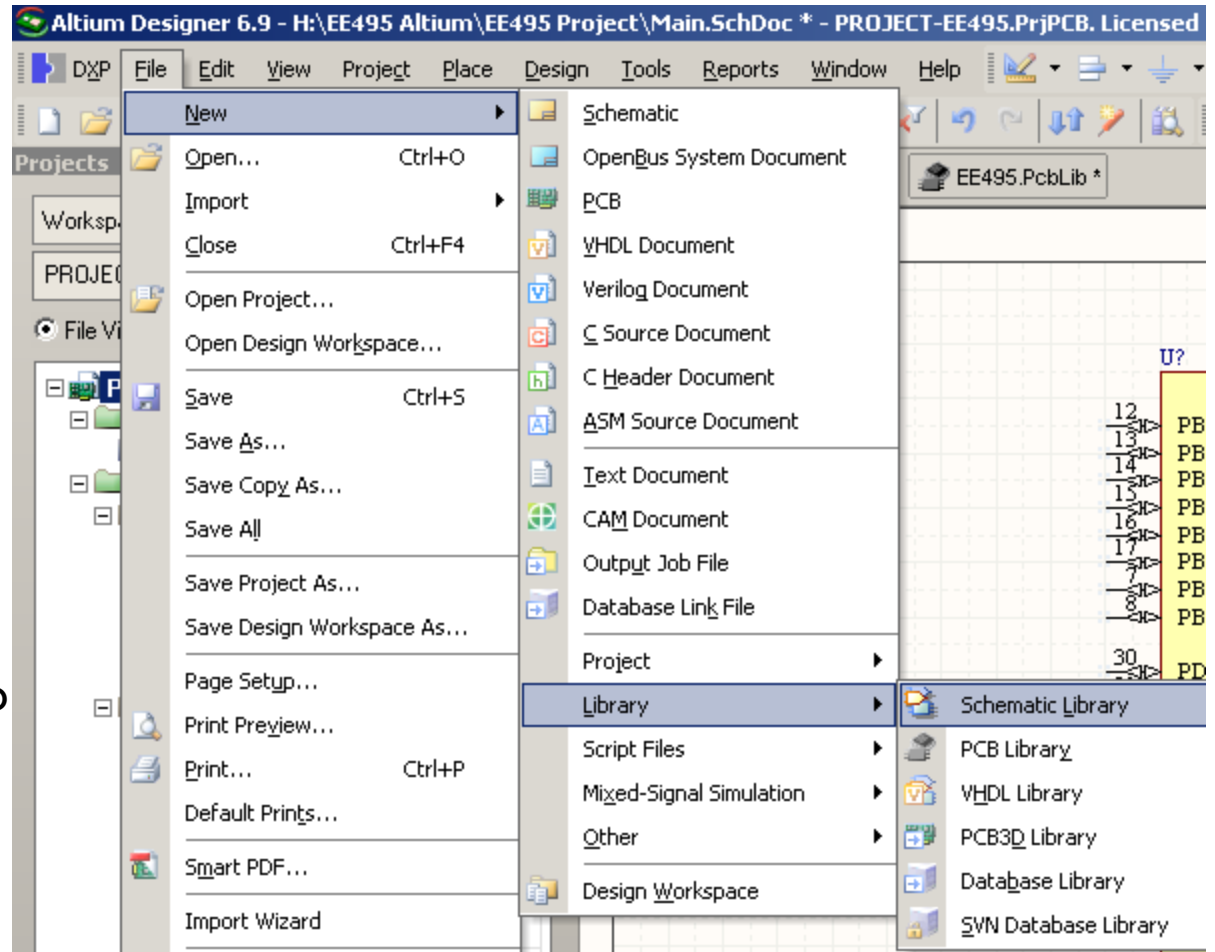


Figure 2 - 28 Pin SSOP Package Pin Out

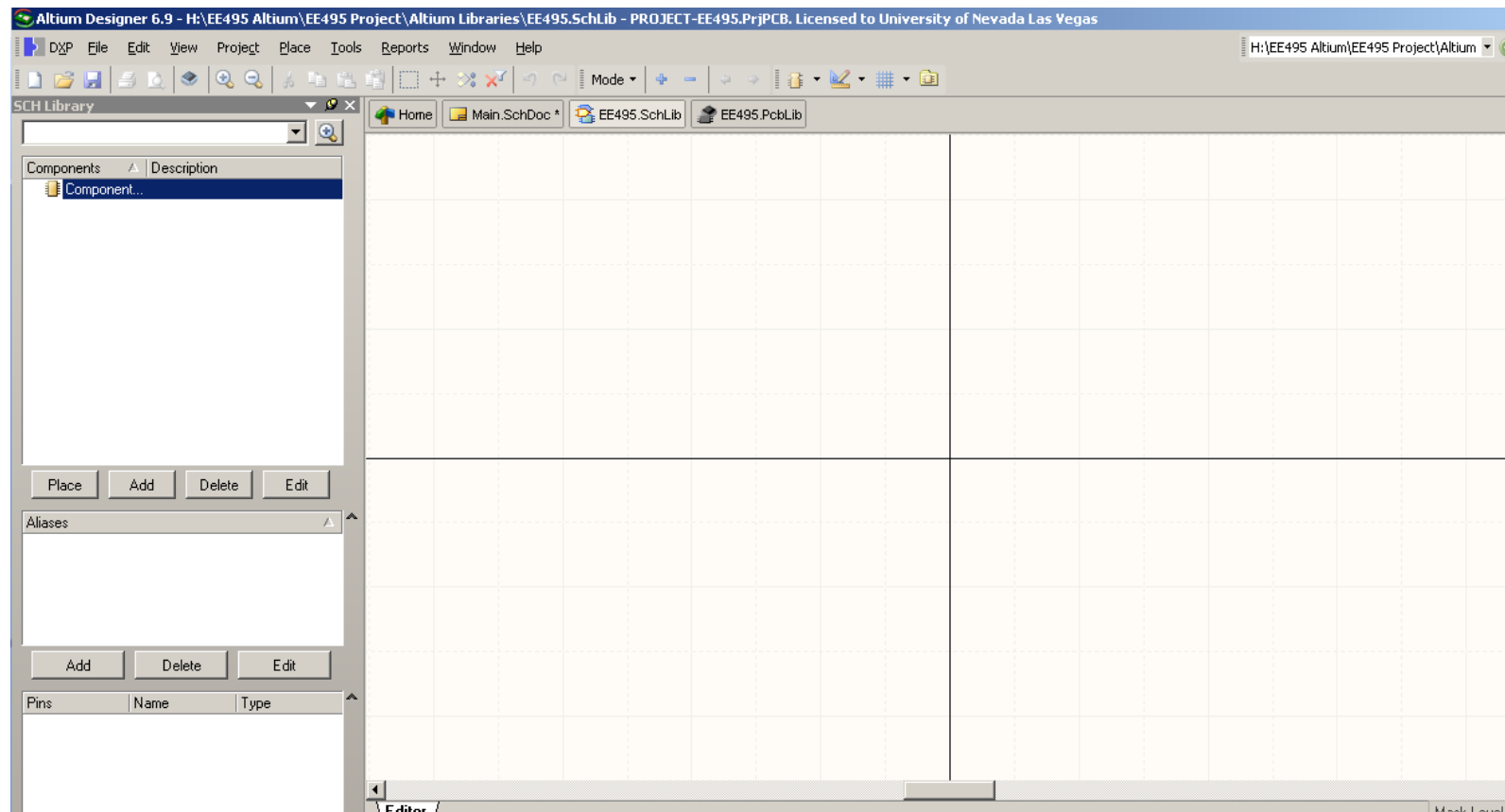
Creating a Schematic Part

- Create a Schematic library like we created a PCB footprint library
- Rename the library to EE495, and save in the same directory as the rest of your libraries.
- Double click the schematic library to open the library, and then view the library by clicking the 'SCH Library' tab in the bottom left of the screen



Creating a Schematic Part

- The large portion of the screen is for creating your schematic drawing, the top left part contains all the parts in the library, the towards the bottom left is the pins and associated function/name



Creating a Schematic Part

- Double click the component and enter the values as shown below

Library Component Properties

Properties

Default Designator: U? Visible Locked

Comment: FT232RL Visible

Part 1/1 Locked

Description: FT232RL USB <> UART

Type: Standard

Library Link

Symbol Reference: FT232RL

Graphical

Mode: Normal Lock Pins

Show All Pins On Sheet (Even if Hidden)

Local Colors

Parameters for Component_1

Visible	Name	Value	Type
---------	------	-------	------

Add... Remove... Edit... Add as Rule...

Models for Component_1

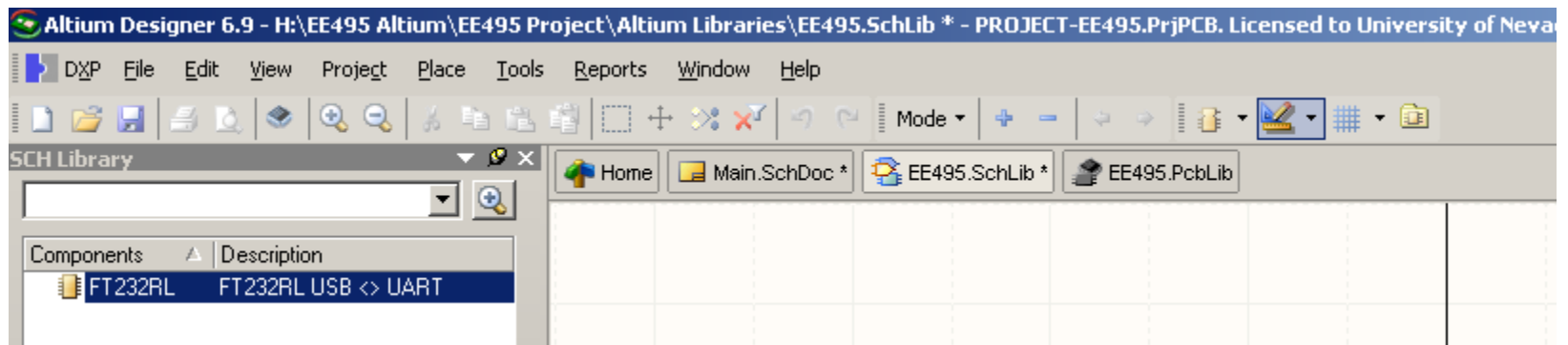
Name	Type	Description
------	------	-------------

Add... Remove... Edit...

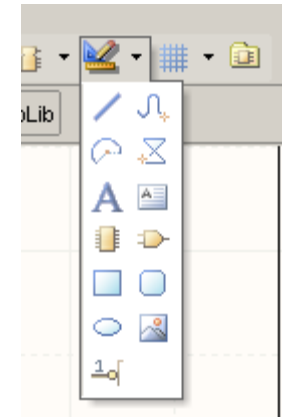
Edit Pins... OK Cancel

Creating a Schematic Part

- Your component should have a descriptive component name and description such as the one below

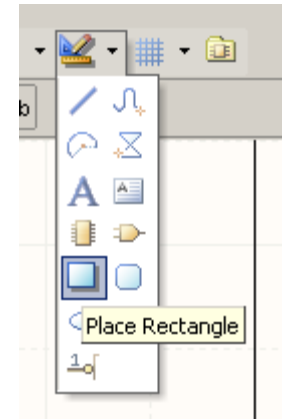
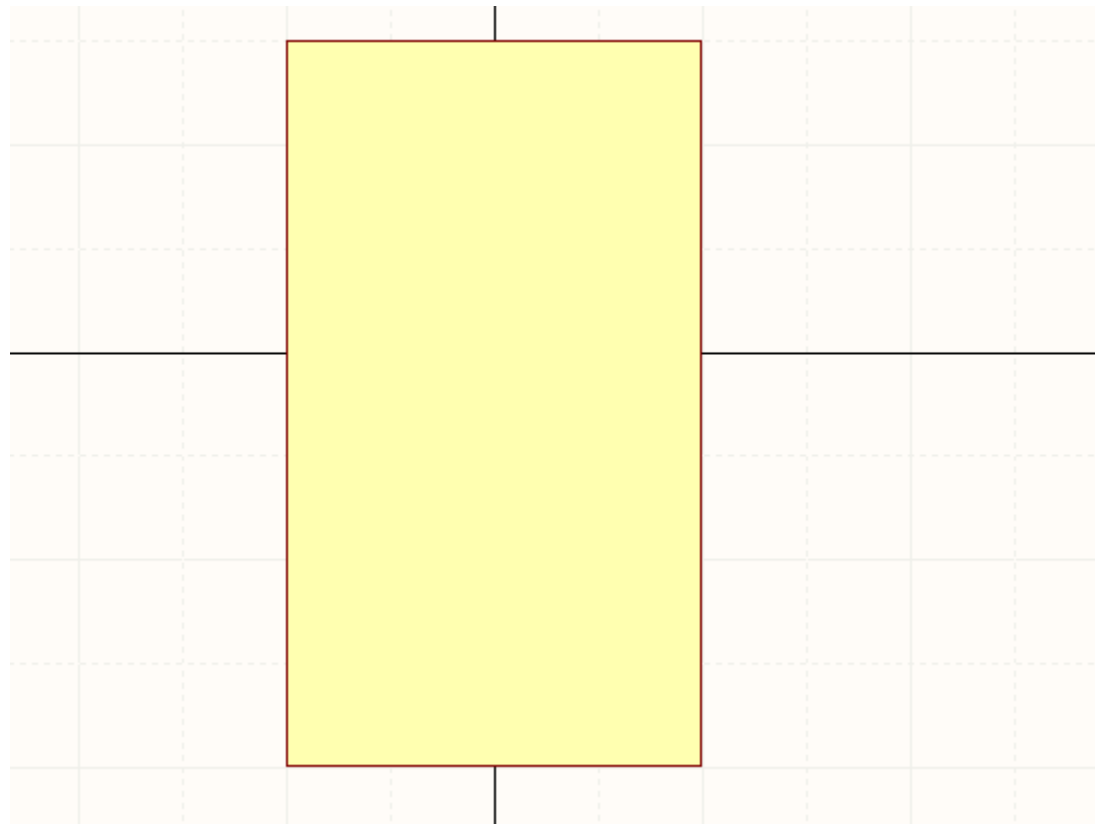


- Click the ruler tab to see the dropdown list as shown in the right. This is the main icon we'll be using in creating the schematic drawing



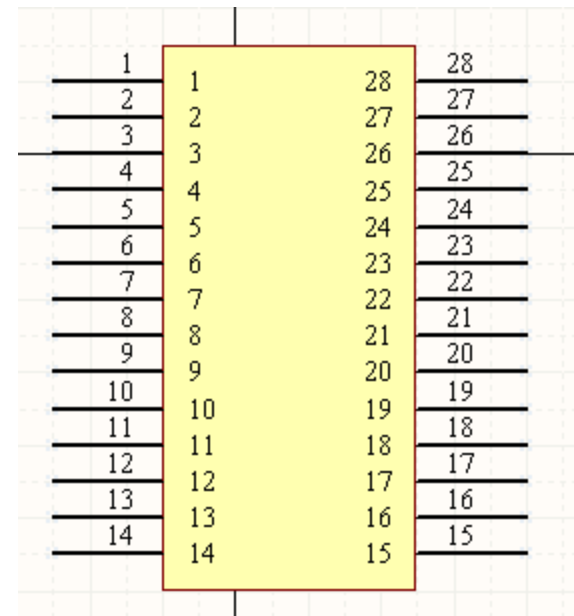
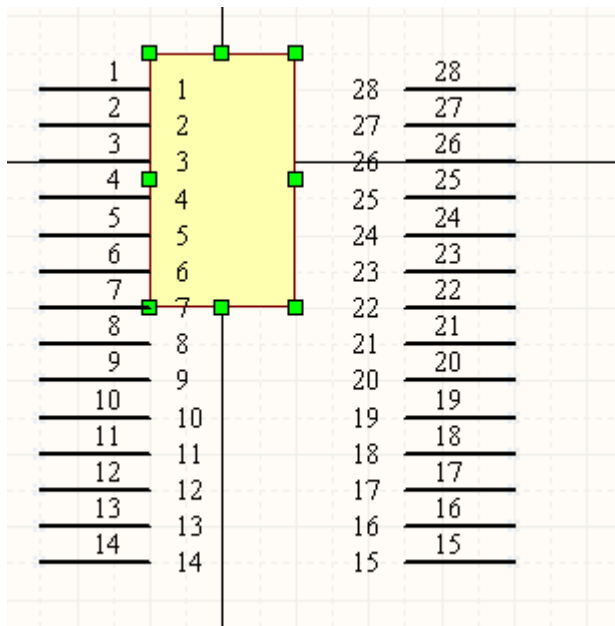
Creating a Schematic Part

- Place a rectangle and draw it so it looks like below
(Size doesn't matter)



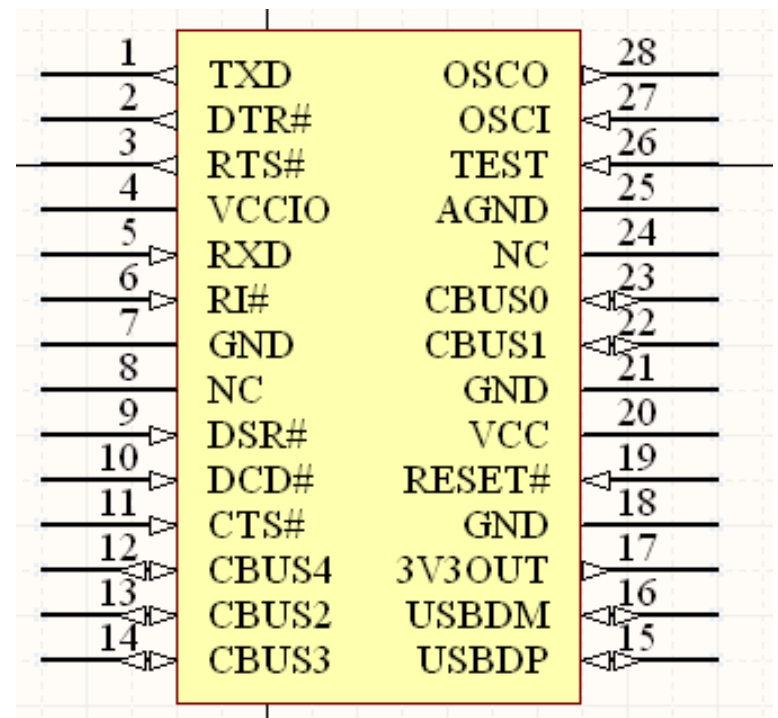
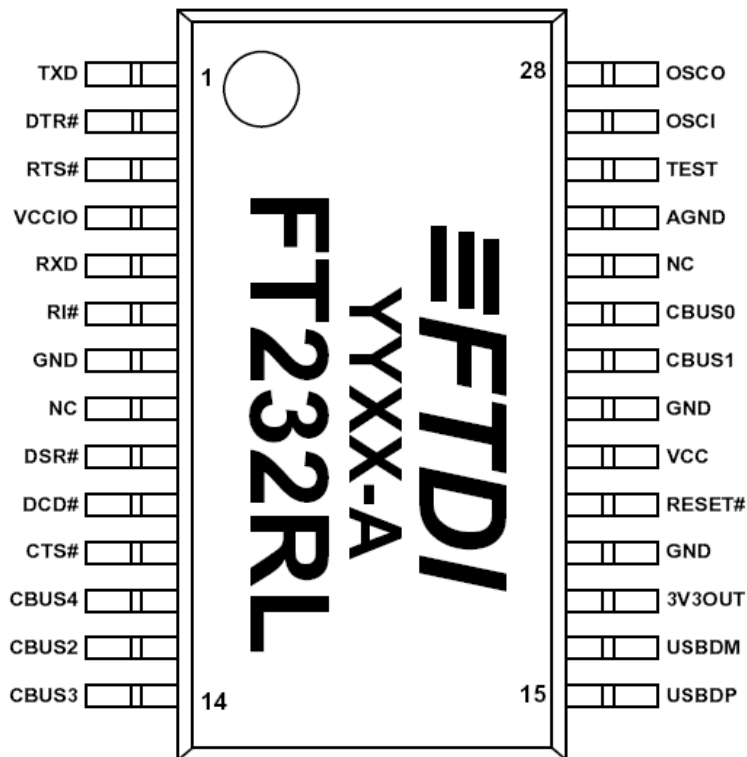
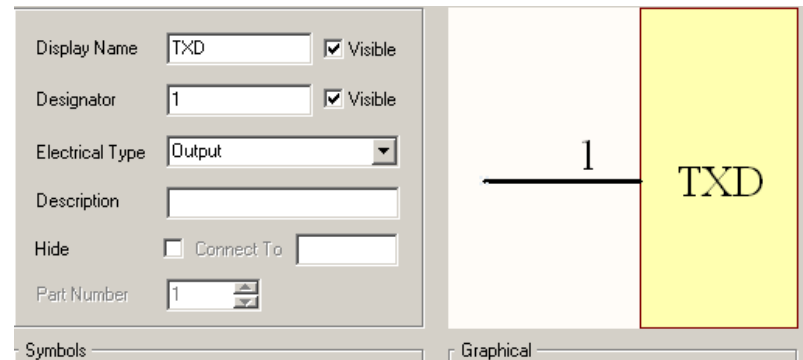
Creating a Schematic Part

- Click the 'place pin' and place 28 pins around the rectangle like below
- Click and drag the rectangle to the proper size



Creating a Schematic Part

- Double click Pin 1 and change display name to TXD,
- Change electrical type to OUTPUT



Creating a Schematic Part

- Sometimes the datasheet will explicitly give the function of the pin, this is an added bonus and not usually given
- You must usually define the pin type from reading the description of what the pin does
- Defining pin types is not a necessity, and things may work fine without defining them properly, but you should usually add the pin function to allow the DRC to catch your errors.

Table 1 - SSOP Package Pin Out Description

Pin No.	Name	Type	Description
USB Interface Group			
15	USBDP	I/O	USB Data Signal
16	USBDM	I/O	USB Data Signal
Power and Ground Group			
4	VCCIO	PWR	+1.8V to +5.25V powered design level. This pin should be noted means that in bi
7, 18, 21	GND	PWR	Device ground s
17	3V3OUT	Output	3.3V output from The prime purpo 1.5kΩ pull up re This pin can als
20	VCC	PWR	3.3V to 5.25V si
25	AGND	PWR	Device analog g
Miscellaneous Signal Group			
8, 24	NC	NC	No internal conn
19	RESET#	Input	Can be used by to VCCIO.
26	TEST	Input	Puts the device
27	OSCI	Input	Input to 12MHz
28	OSCO	Output	Output from 12M tor is used. *

Creating a Schematic Part

- There are usually typical applications schematics which will help you properly use the device.
- You should always read each and every pin description to see if you need to add functionality, but it is always a good starting point to use the typical application sheet

7.1 Bus Powered Configuration

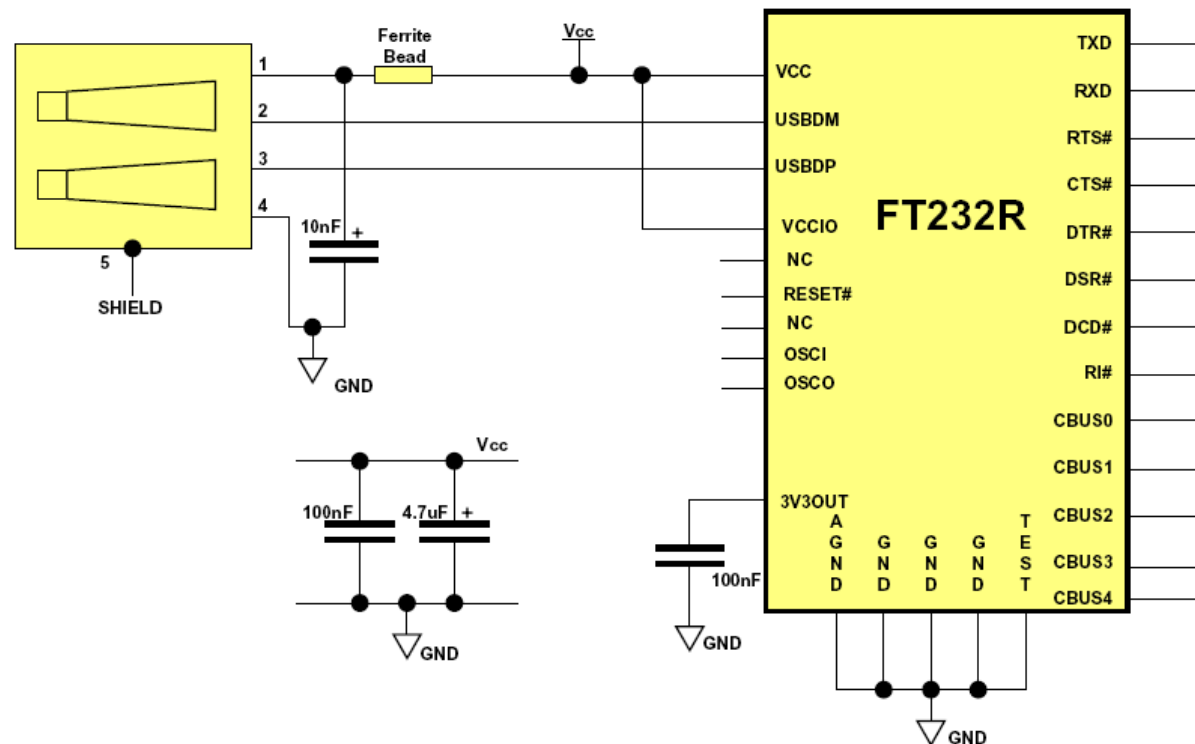


Figure 11 - Bus Powered Configuration

Creating a Schematic Part

- Usually these typical application notes have corresponding notes associated with it that you should consider.
- The following rules for this device are a combination of rules for the USB standard, and for the FT232
- These notes should be regarded as bible, written by people who understand and have tested their product to death.

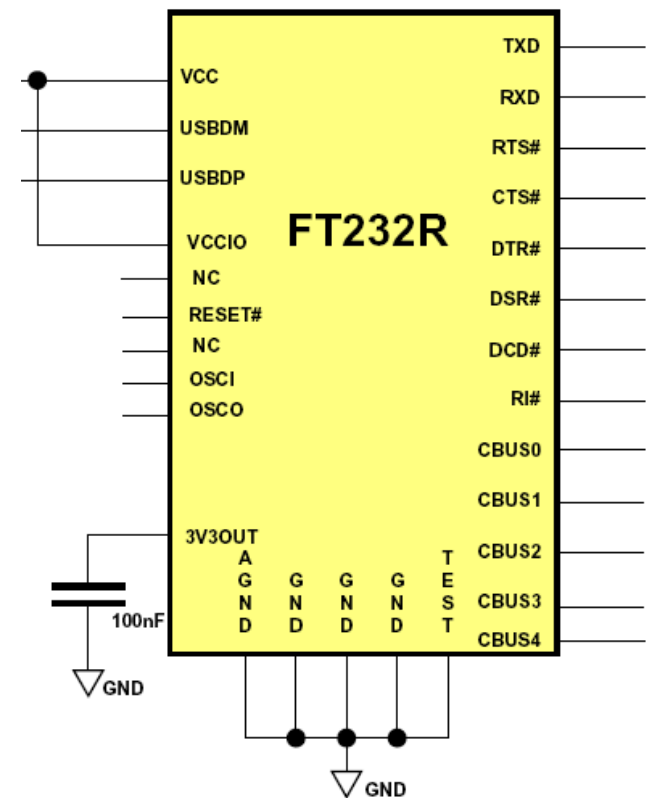
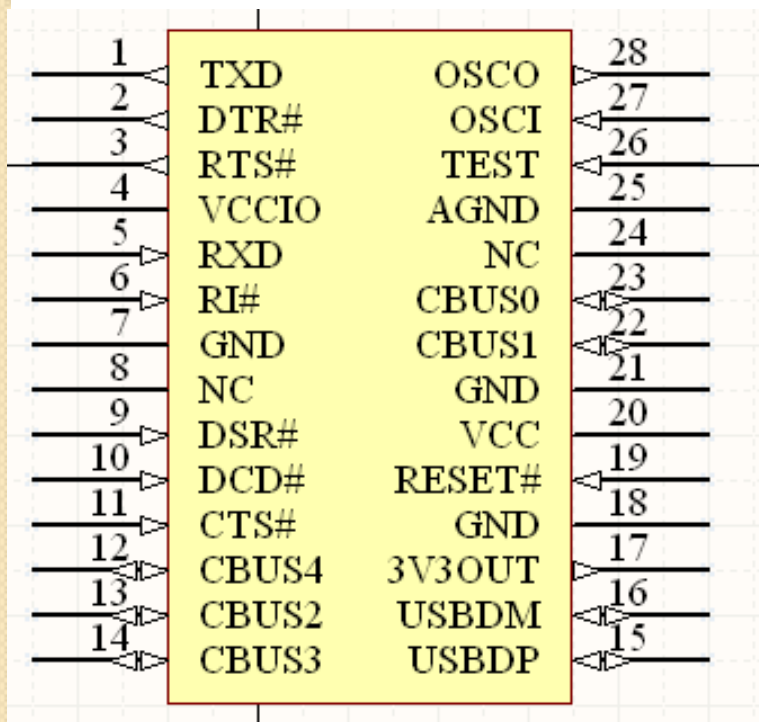
Figure 11 illustrates the FT232R in a typical USB bus powered design configuration. A USB Bus Powered device gets its power from the USB bus. Basic rules for USB Bus power devices are as follows –

- i) On plug-in to USB, the device must draw no more than 100mA.
- ii) On USB Suspend the device must draw no more than 500 μ A.
- iii) A Bus Powered High Power USB Device (one that draws more than 100mA) should use one of the CBUS pins configured as PWREN# and use it to keep the current below 100mA on plug-in and 500 μ A on USB suspend.
- iv) A device that consumes more than 100mA can not be plugged into a USB Bus Powered Hub.
- v) No device can draw more than 500mA from the USB Bus.

The power descriptor in the internal EEPROM should be programmed to match the current draw of the device. A Ferrite Bead is connected in series with USB power to prevent noise from the device and associated circuitry (EMI) being radiated down the USB cable to the Host. The value of the Ferrite Bead depends on the total current required by the circuit – a suitable range of Ferrite Beads is available from Steward (www.steward.com) for example Steward Part # MI0805K400R-00.

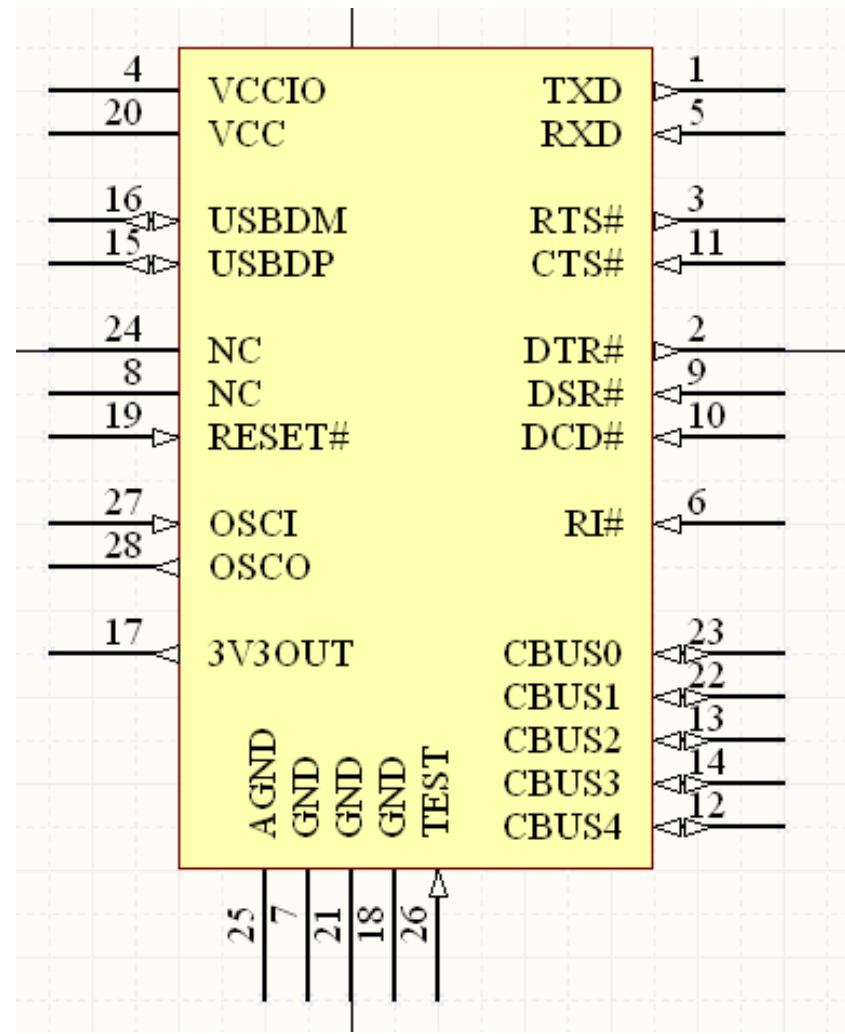
Creating a Schematic Part

- Now that we have verified that the pins all match the ones in the datasheet (with the correct pin numbers!) we can move pins around and make it look more like the application schematic given
- This usually allows for a cleaner looking schematic drawing and much easier to reference the datasheet.



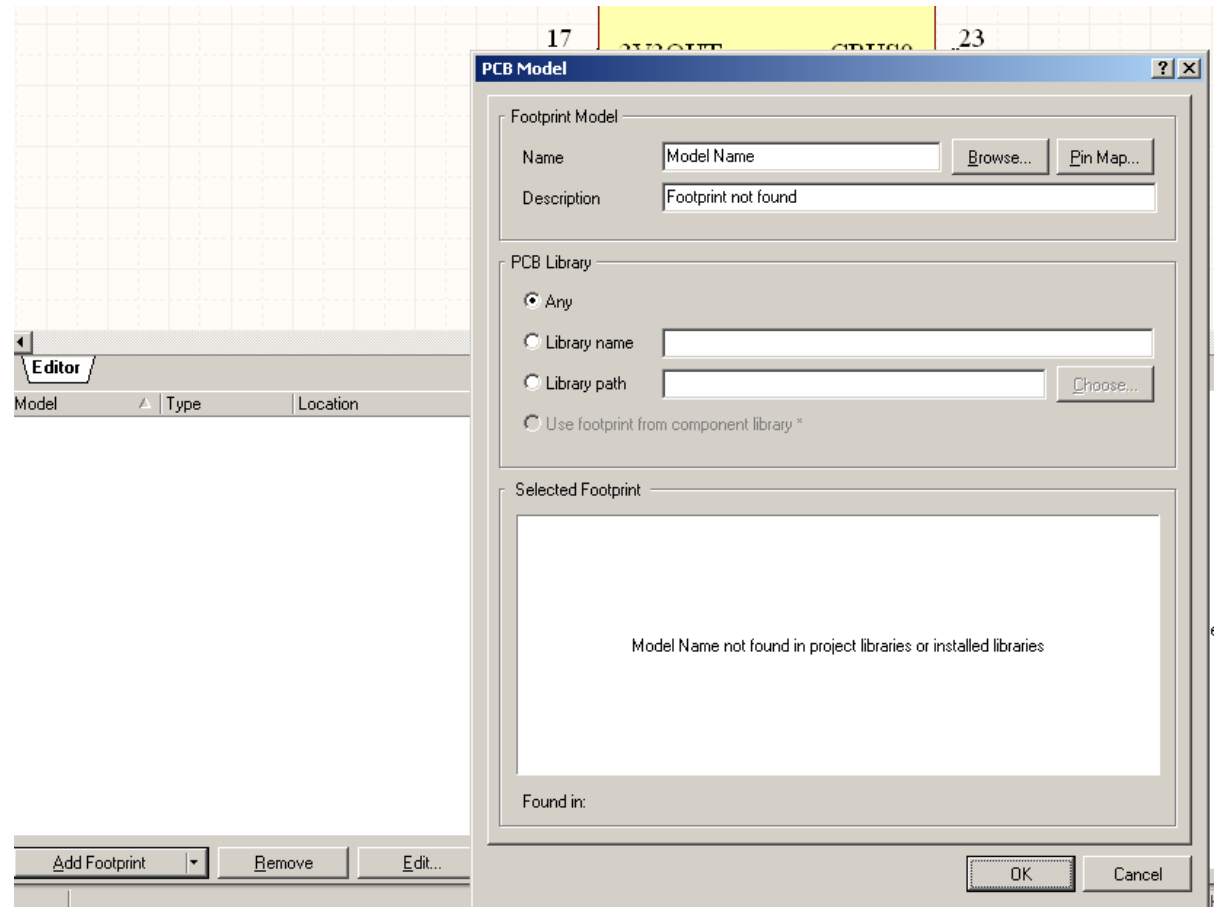
Creating a Schematic Part

- Now that we have verified that the pins all match the ones in the datasheet (with the correct pin numbers!) we can move pins around and make it look more like the application schematic given
- This usually allows for a cleaner looking schematic drawing and much easier to reference the datasheet.
- NOTE: You can also hide pins from the schematic document. Double click on the pin and check 'hide'



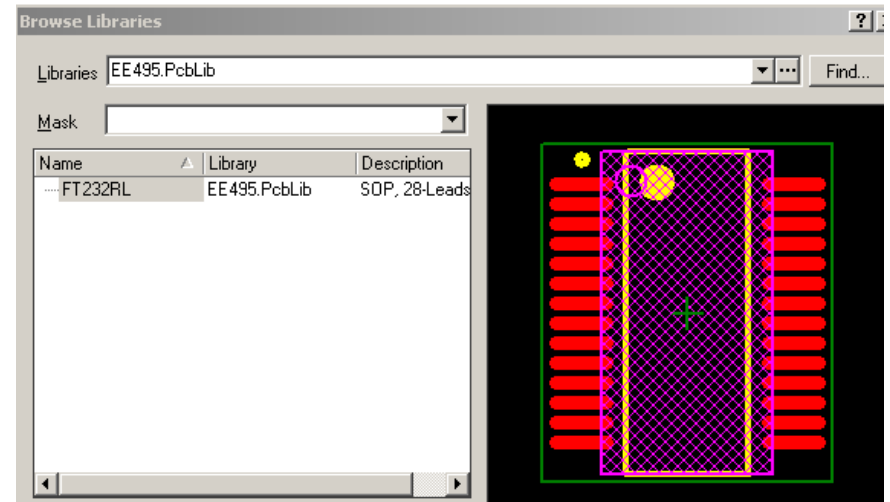
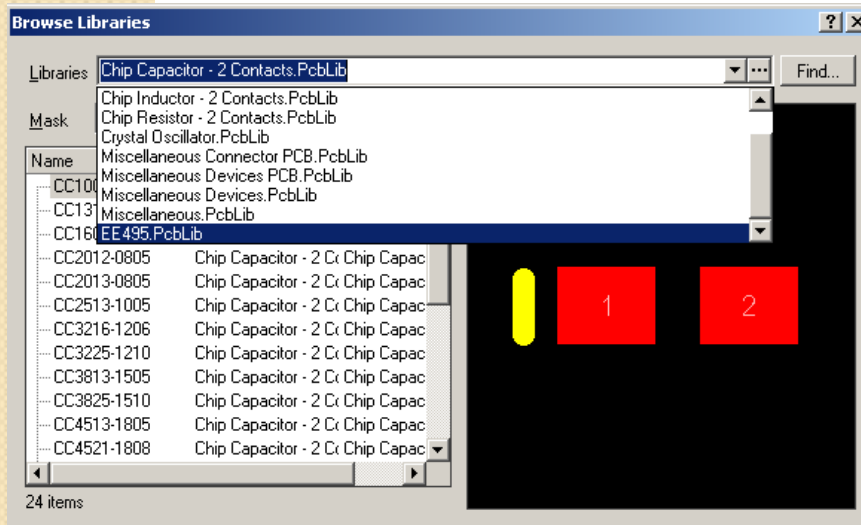
Creating a Schematic Part

- Now we need to associate the PCB Footprint we created with the schematic drawing
- Click the 'add footprint' at the bottom of the page
- Click 'browse' in the 'footprint model' section



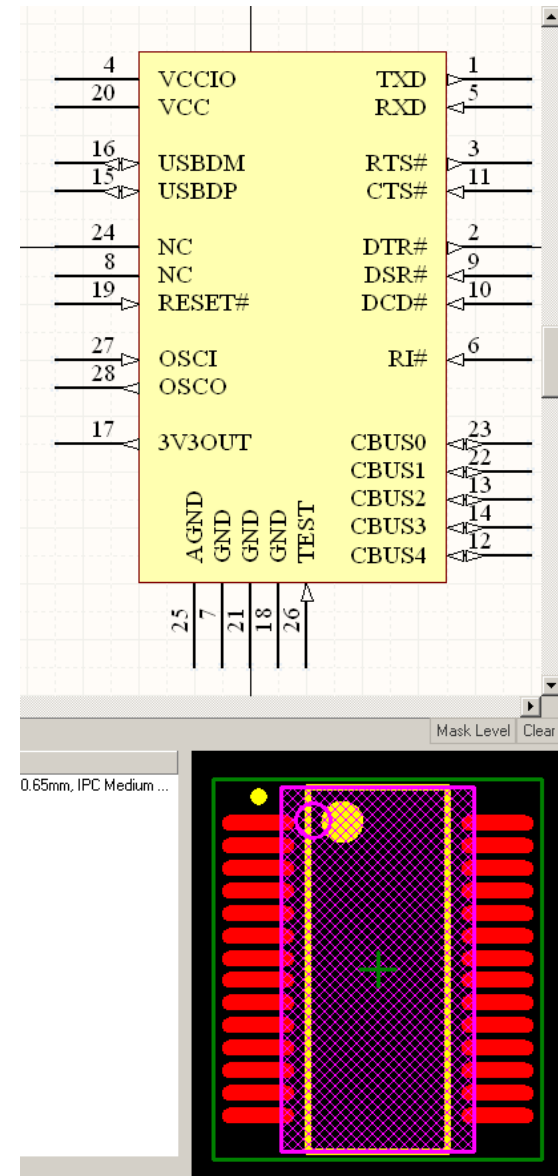
Creating a Schematic Part

- Select your EE495 Library from the dropdown list
- There is only one part inside your custom EE495 library, so the choice is easy. Select your part and click 'OK'



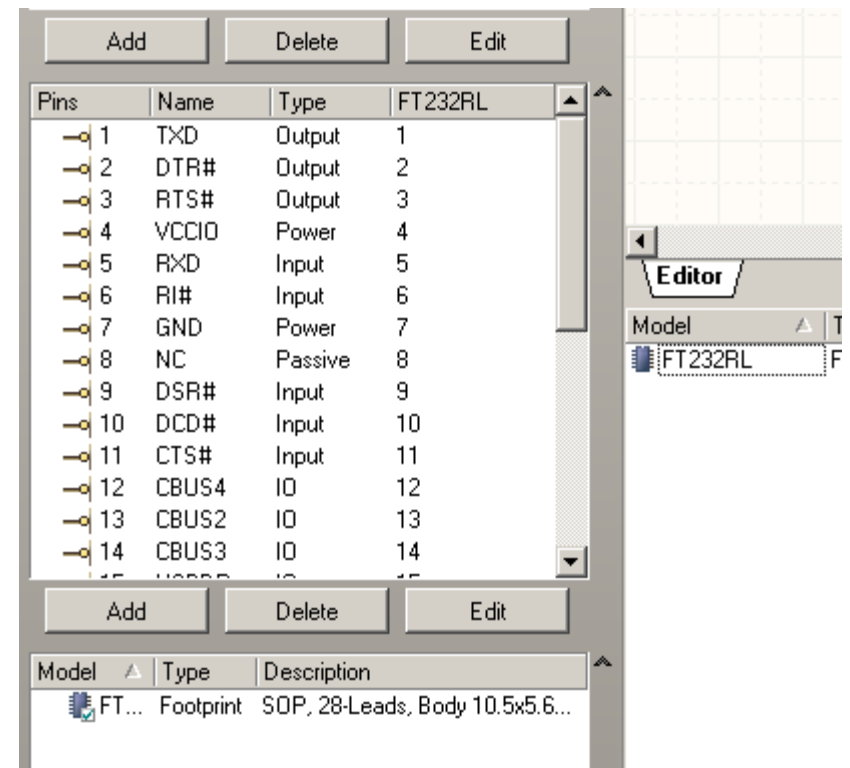
Creating a Schematic Part

- In your schematic library, you should now see the footprint you created along with the schematic drawing you made
- Finally the part is complete!!
- NOTE: This gets much faster as you do it more, but you can see the value of the large list of integrated libraries included with Altium



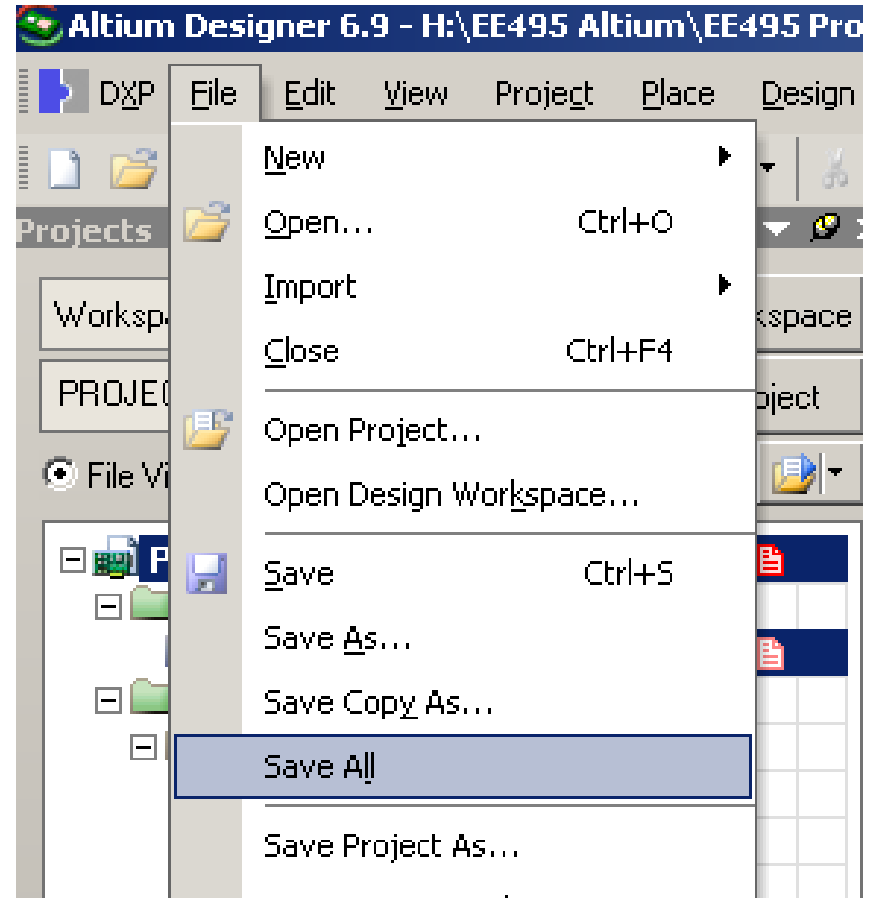
Creating a Schematic Part

- One last check is to look at the left side of the screen. The first three columns in this case all refer to data you entered when creating the schematic document
- The last column (FT232RL) is the name that you gave the PCB Footprint
- Check and compare that pin 1 on the schematic is linked to pin 1 on the PCB footprint
- Repeat for the rest. If something doesn't match, something went wrong!!



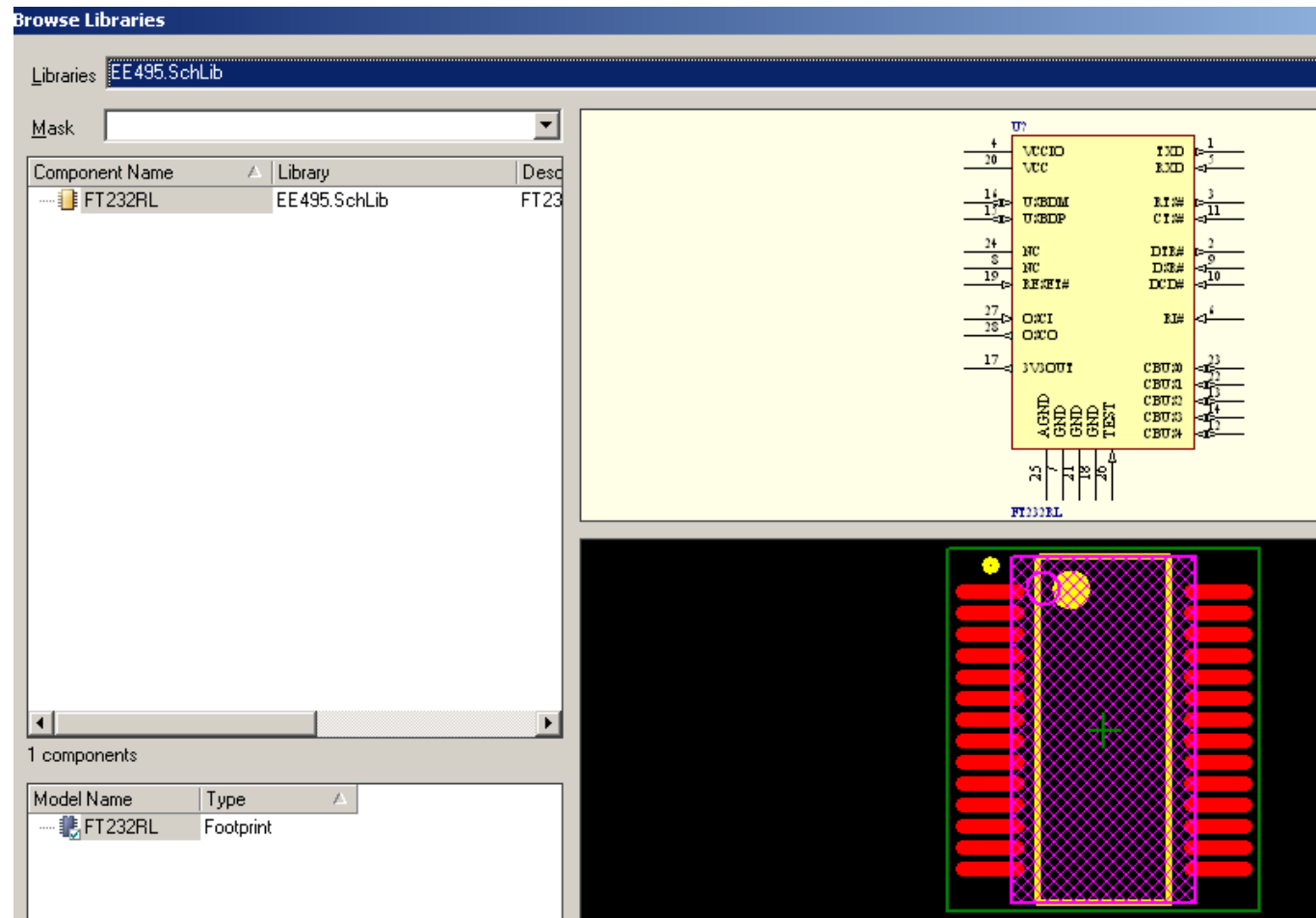
Save the Project

- Now is a good time to save the project and all the changes you made.
- Follow the menu sequence to the right to save the project



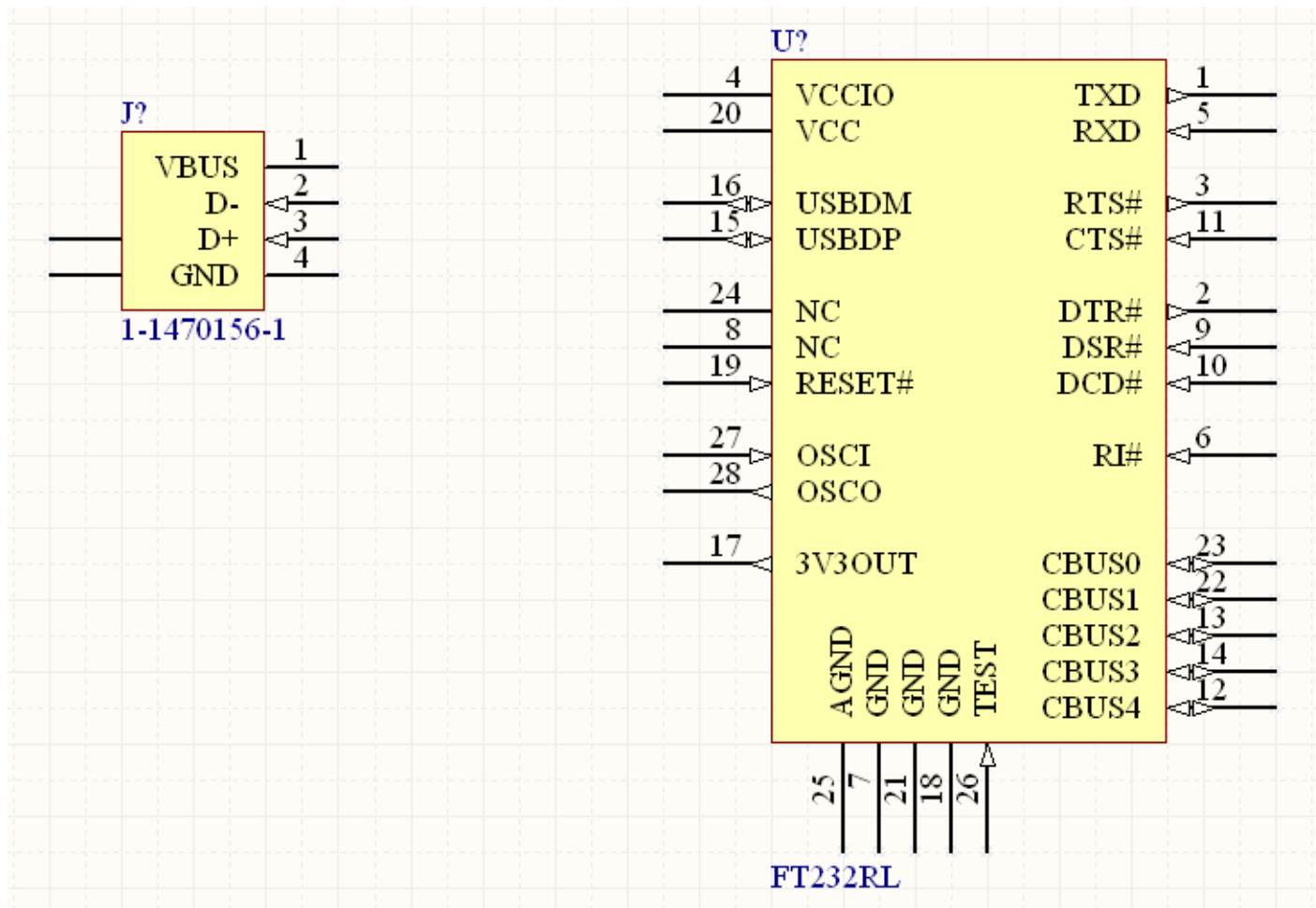
Insert The FTDI Chip

- Select your EE495 library, and place the only component in your library (so far). You should see the schematic drawing and also the PCB footprint. If not something is wrong

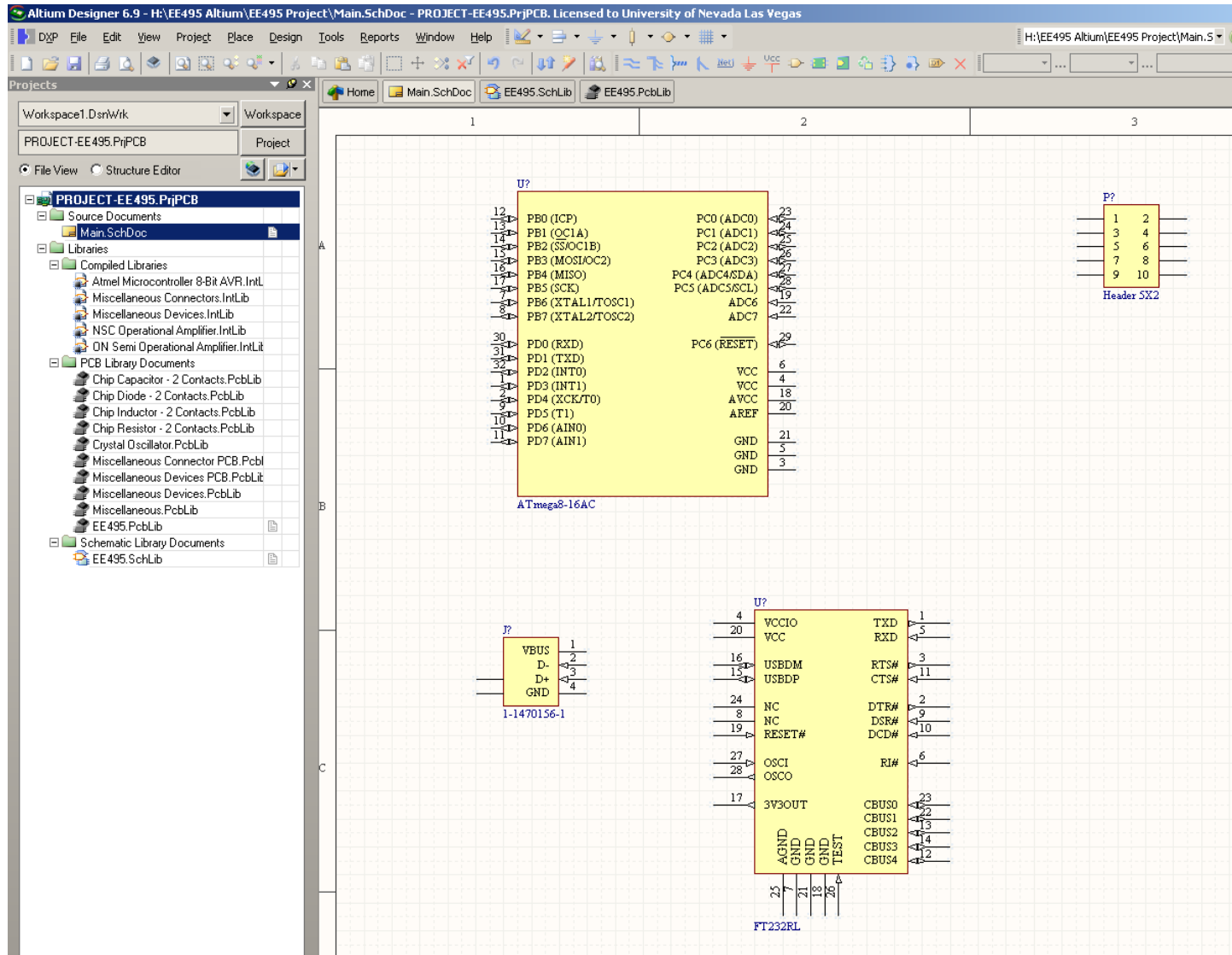


Insert The FTDI Chip

- Notice how the rotated and mirrored USB-B connector matches up perfectly with the FT232 chip. (D- goes to USBDM, D+ goes to USBDP) This did not happen by chance



Current Schematic

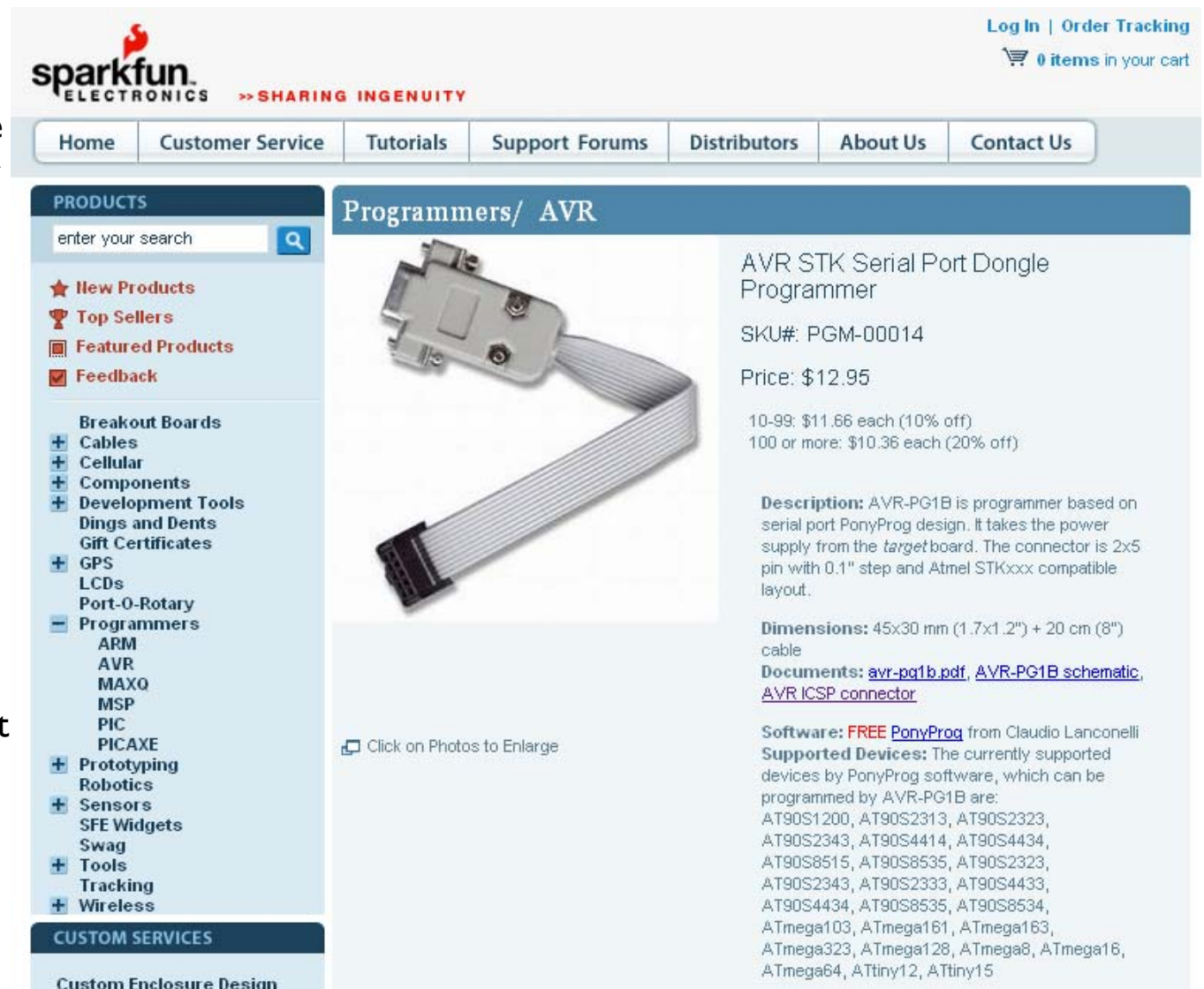


Programming Interface

- We need a device that will send our compiled HEX code to the microcontroller.

- This type of device is called a programmer

- Sparkfun sells many decent programmers that are cheap (\$13)



The screenshot shows the Sparkfun Electronics website interface. At the top, the logo "sparkfun ELECTRONICS" is displayed with the tagline ">> SHARING INGENUITY". Navigation links include "Home", "Customer Service", "Tutorials", "Support Forums", "Distributors", "About Us", and "Contact Us". A search bar is present in the "PRODUCTS" section. The main content area is titled "Programmers/ AVR" and features a product listing for the "AVR STK Serial Port Dongle Programmer". The product image shows a small metal device with a ribbon cable. The product details include the SKU# PGM-00014, a price of \$12.95, and a description stating it is based on a serial port PonyProg design. A list of supported AVR and ATmega devices is provided at the bottom of the product page.

Log In | Order Tracking
0 items in your cart

Home Customer Service Tutorials Support Forums Distributors About Us Contact Us


PRODUCTS
enter your search

- ★ New Products
- 🏆 Top Sellers
- 📄 Featured Products
- 👍 Feedback

- Breakout Boards
- + Cables
- + Cellular
- + Components
- + Development Tools
- Dings and Dents
- Gift Certificates
- + GPS
- LCDs
- Port-O-Rotary
- Programmers
 - ARM
 - AVR
 - MAXQ
 - MSP
 - PIC
 - PICAXE
- + Prototyping
- Robotics
- + Sensors
- SFE Widgets
- Swag
- + Tools
- Tracking
- + Wireless

CUSTOM SERVICES
Custom Enclosure Design

Programmers/ AVR



AVR STK Serial Port Dongle Programmer

SKU#: PGM-00014

Price: \$12.95

10-99: \$11.66 each (10% off)
100 or more: \$10.36 each (20% off)

Description: AVR-PG1B is programmer based on serial port PonyProg design. It takes the power supply from the target board. The connector is 2x5 pin with 0.1" step and Atmel STKxxx compatible layout.

Dimensions: 45x30 mm (1.7x1.2") + 20 cm (8") cable

Documents: [avr-pg1b.pdf](#), [AVR-PG1B schematic](#), [AVR ICSP connector](#)

Software: **FREE PonyProg** from Claudio Lanconelli

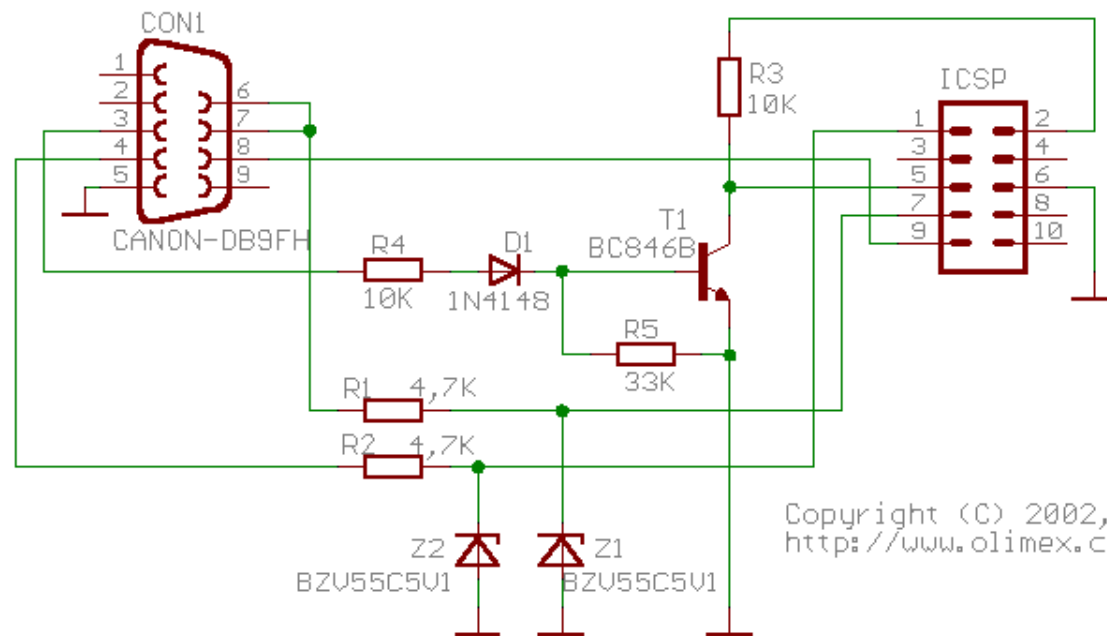
Supported Devices: The currently supported devices by PonyProg software, which can be programmed by AVR-PG1B are:
AT90S1200, AT90S2313, AT90S2323,
AT90S2343, AT90S4414, AT90S4434,
AT90S8515, AT90S8535, AT90S2323,
AT90S2343, AT90S2333, AT90S4433,
AT90S4434, AT90S8535, AT90S8534,
ATmega103, ATmega161, ATmega163,
ATmega323, ATmega128, ATmega8, ATmega16,
ATmega64, ATtiny12, ATtiny15

Click on Photos to Enlarge

Programming Interface

- If you want, you can see the schematic of the programmer, and integrate it onto your development board

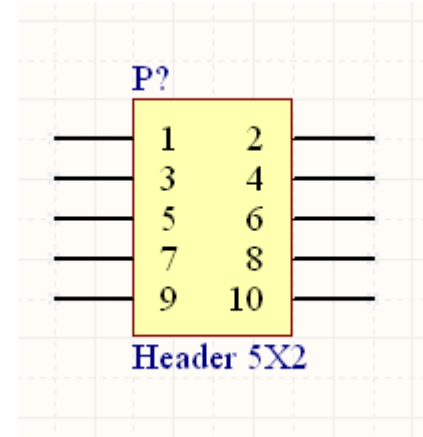
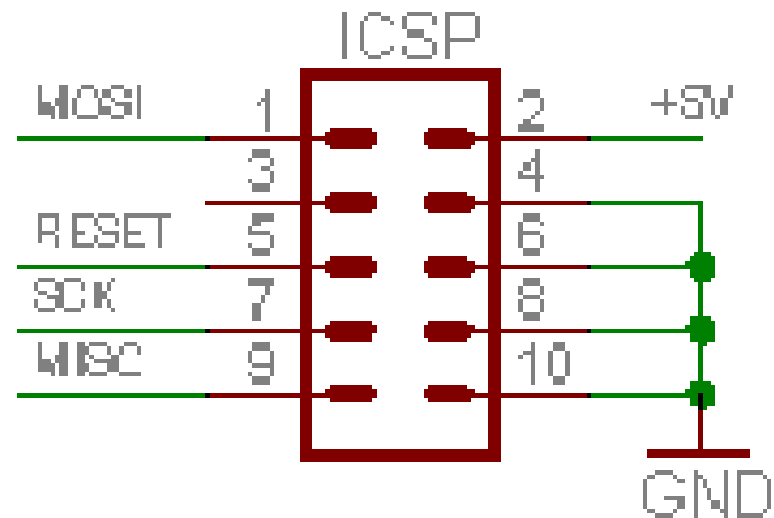
- As you can see, this is a simple device which would only add a few components to your project



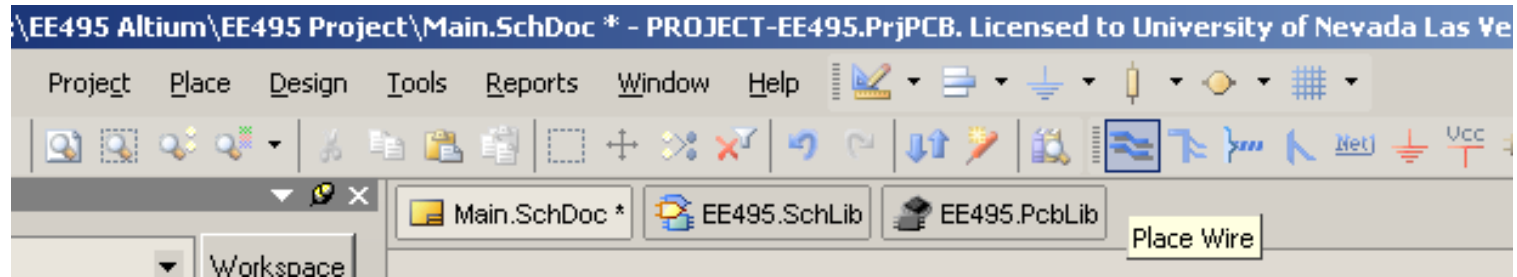
Copyright (C) 2002, OLIMEX Ltd
<http://www.olimex.com/dev>

Programming Interface

- The specification for the 2x5 header, spaced 0.1" apart perfectly matches the connection for the programmer.
- Although you may not know what these pins are, we will assign them to the programmer header

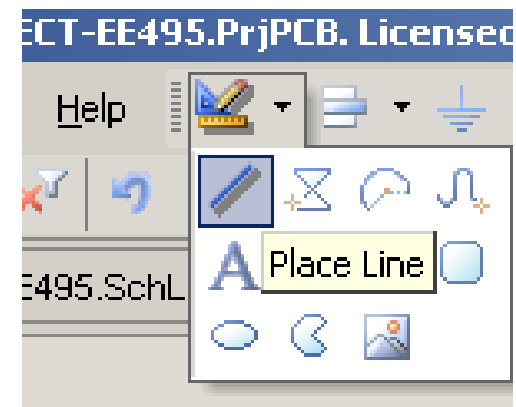


Placing a Connective Wire



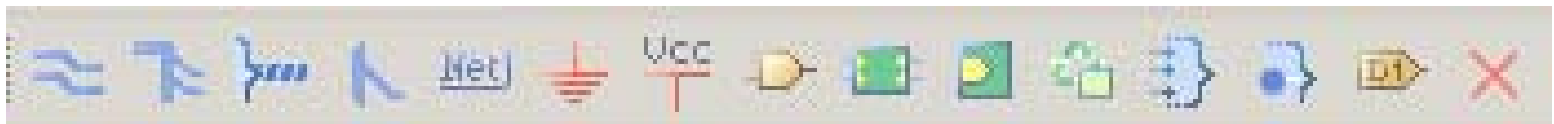
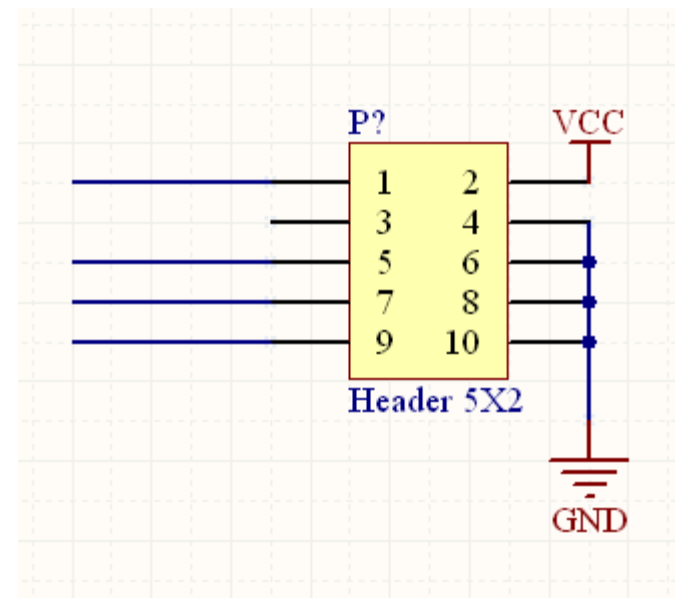
- Find the icon that looks like the one above, and click it to place a wire, it will be blue in appearance

WARNING!!: Make sure not to use the 'place line' from the drawing box. This will draw a line that LOOKS like a connective wire, but a connection won't be made



Placing a Connective Wire

- Draw wires on the 2x5 header as shown on the right
- Click the GND and VCC icons from the wiring menu (shown below) and make it look like the diagram to the right



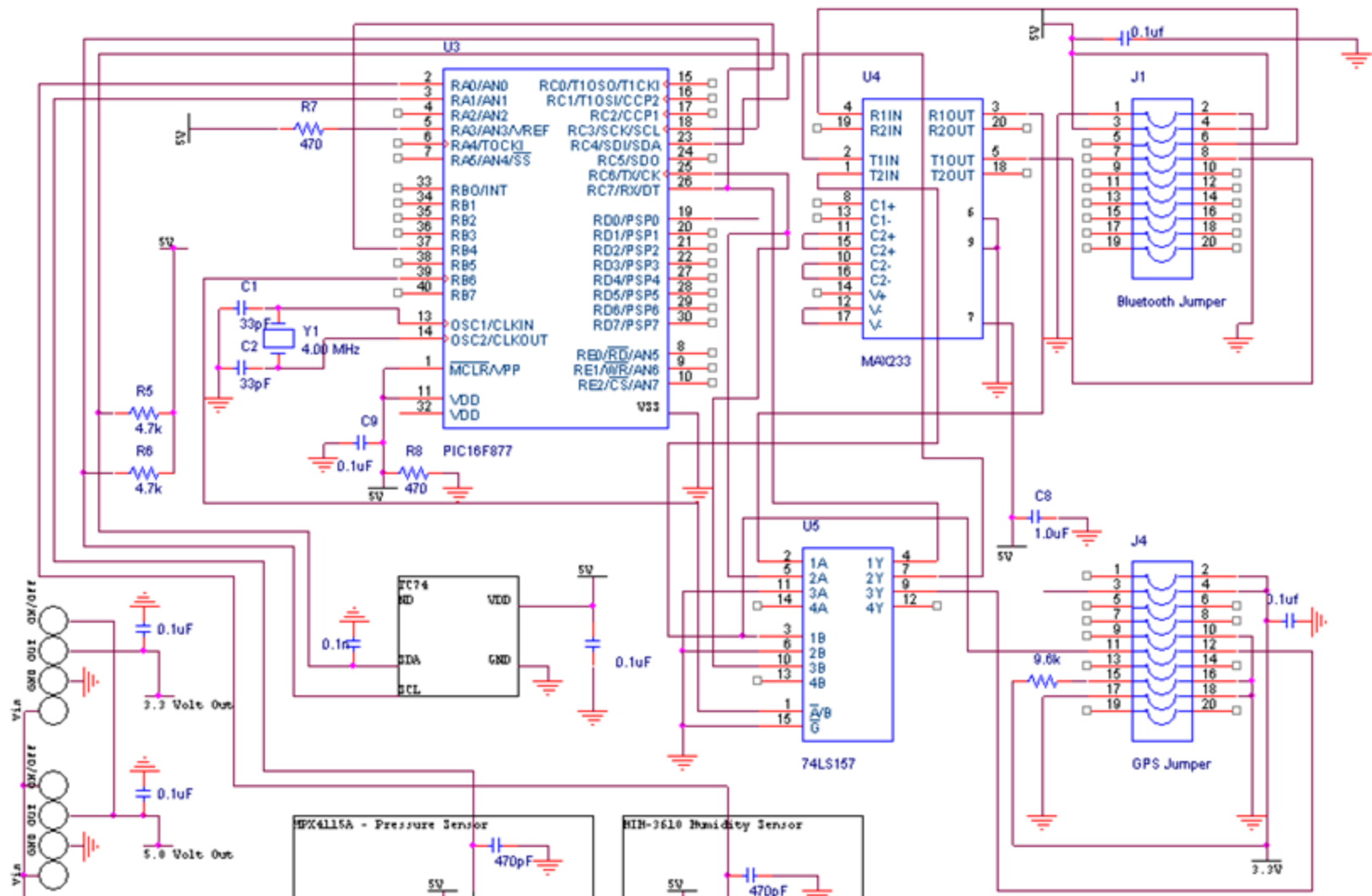


Net Labels

- A net label is a method of electrically connecting nets of the same name
- This allows a method to make a connection without requiring placing a wire
- So what's wrong with running wires ?? It is far more difficult and more confusing to run wires, especially when routing a large bus.

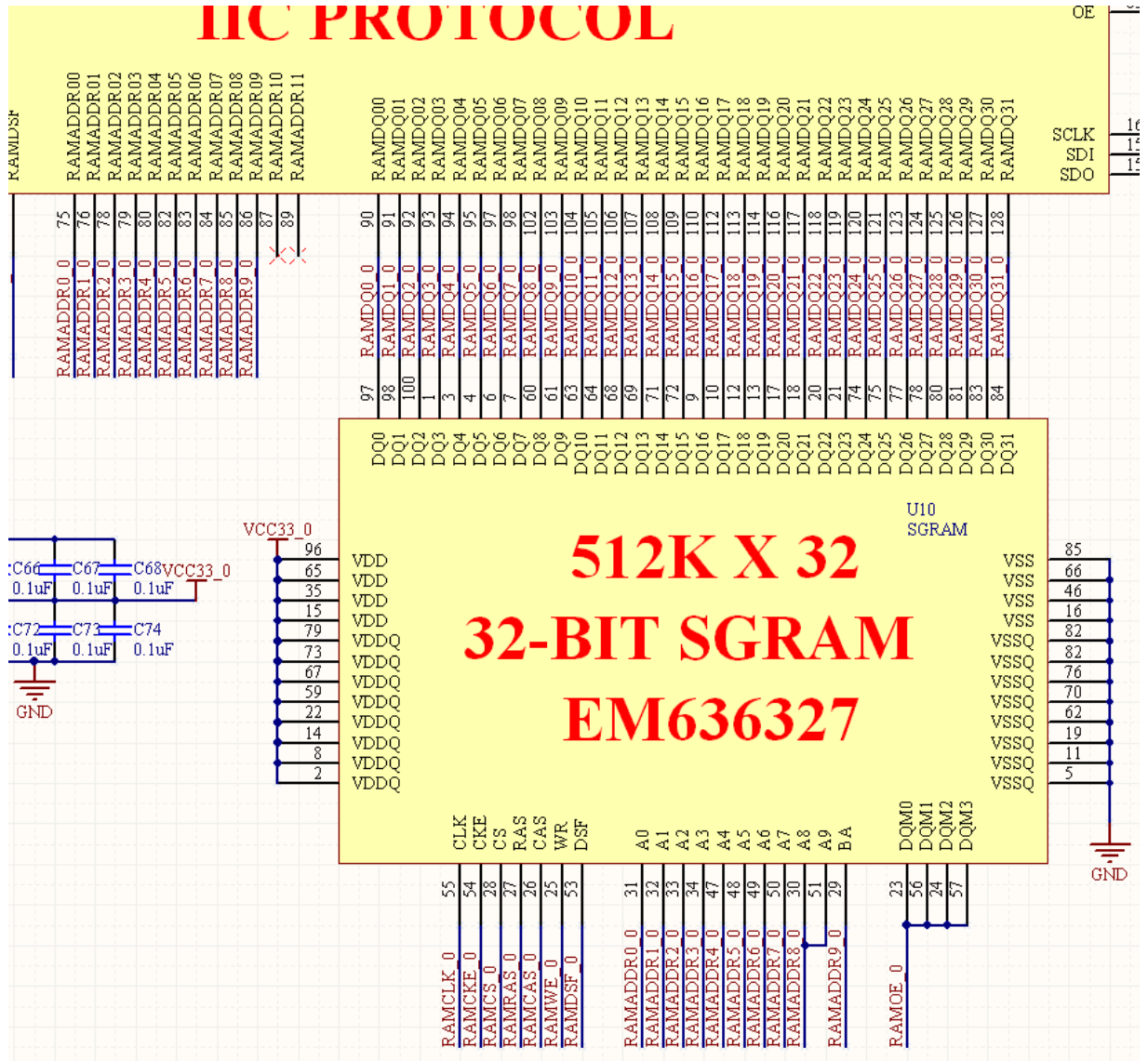
Net Labels

- Here is a schematic which is very confusing and would benefit from using net labels



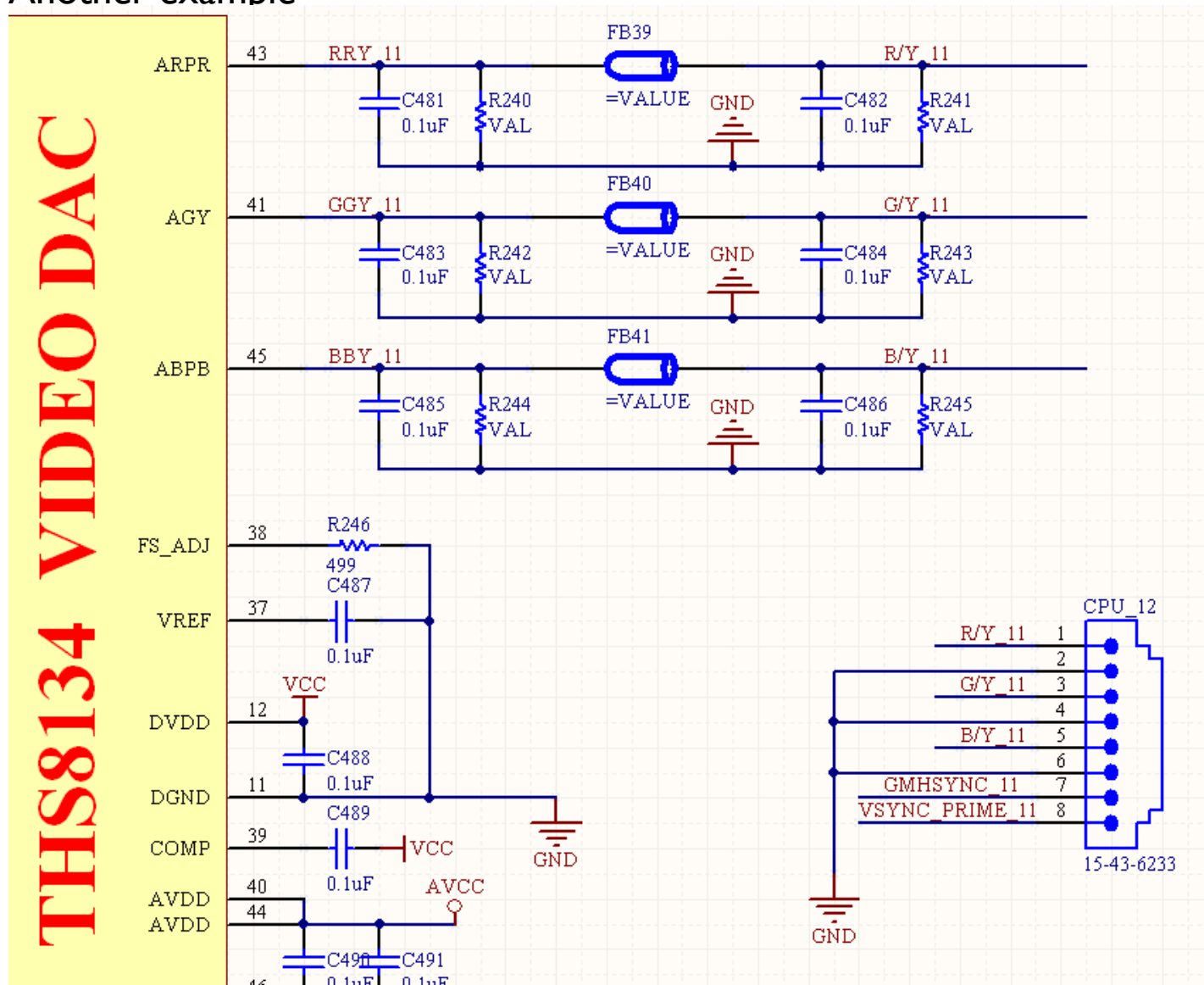
Net Labels

- This is a design that uses direct wires for short connections, and net labels for long connections.
- Notice how the schematic looks more clean and professional.
- NOTE: Even if direct wire connections are used, it's still a good idea to give the net a name for PCB reasons



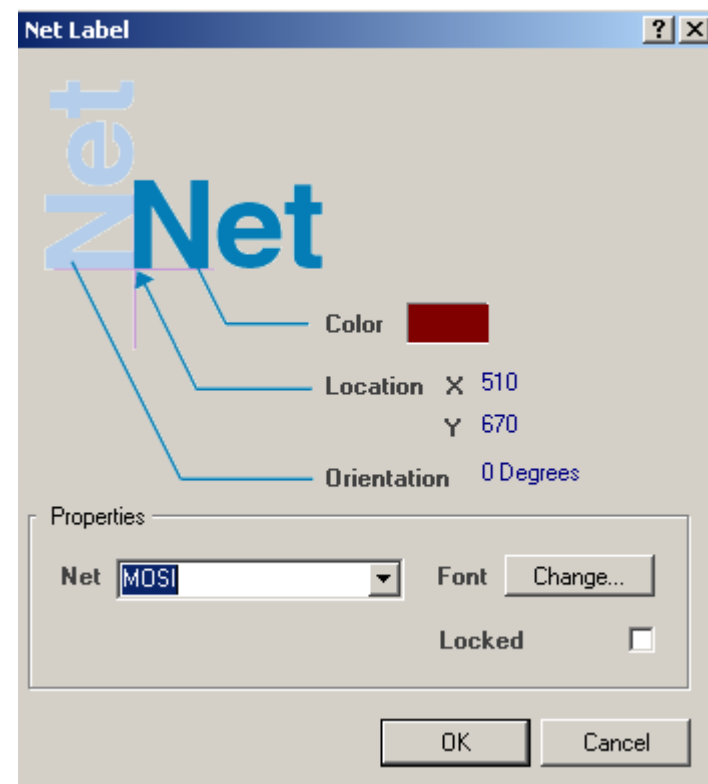
Net Labels

- Another example



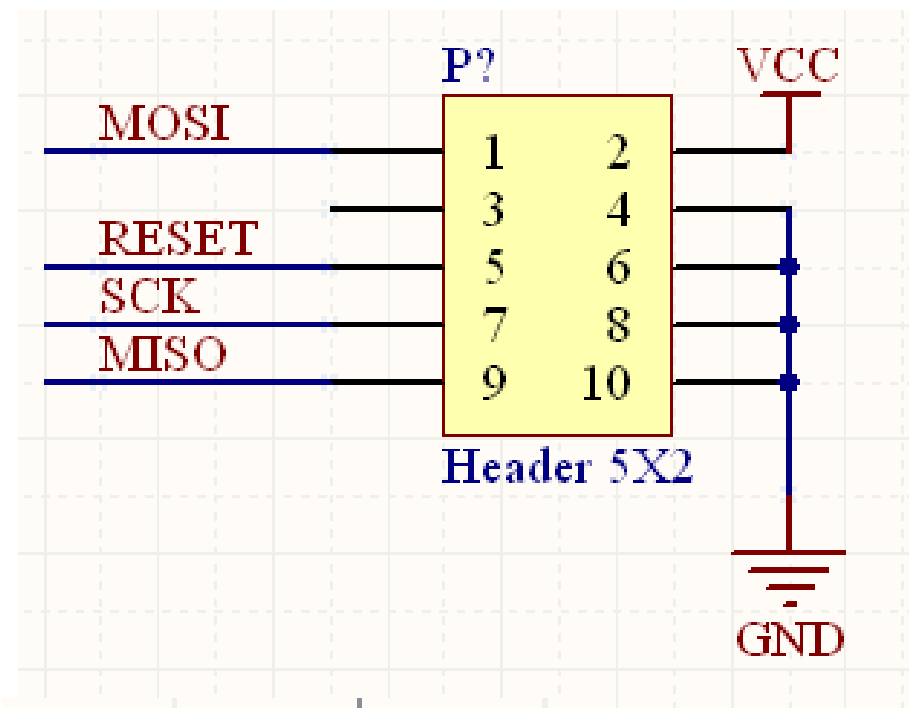
Using Net Labels

- Click on the net label icon and place the net label on each of the wires.
- Double click the net label to rename the net. You can type in a new net, or select one from the drop down list.

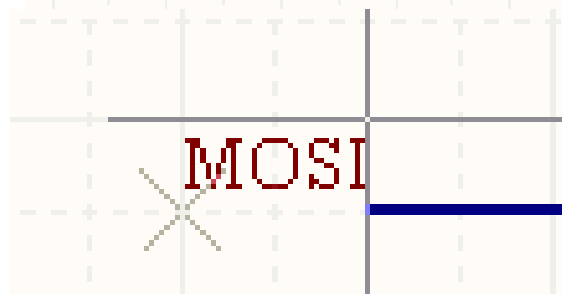


Using Net Labels

- Rename the nets to match the diagram on the right
- **WARNING:**When placing netlabels, make sure there is a red X at the connection point. If it is a gray X, there is no electrical connection



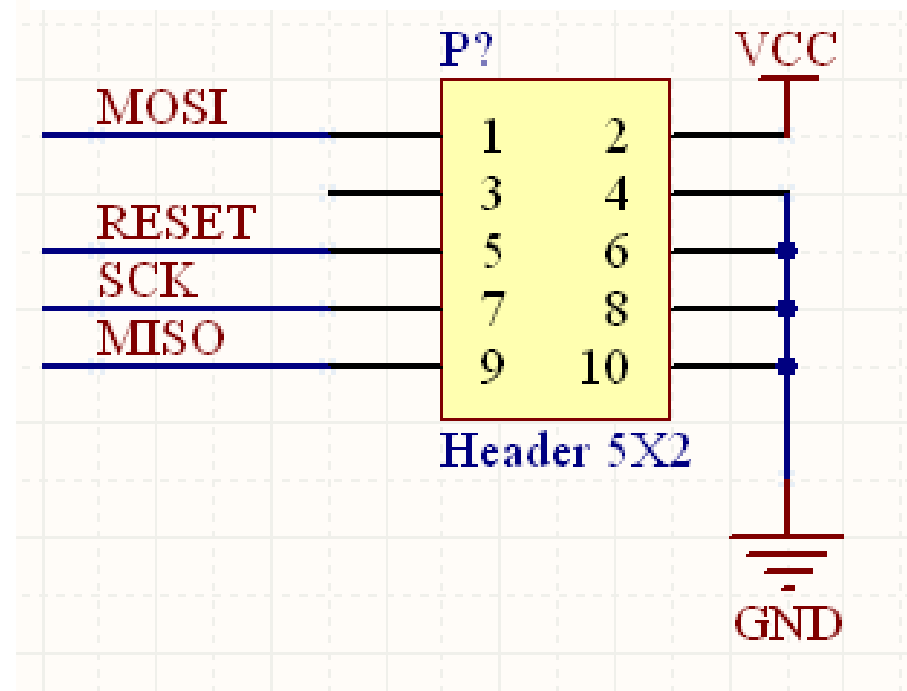
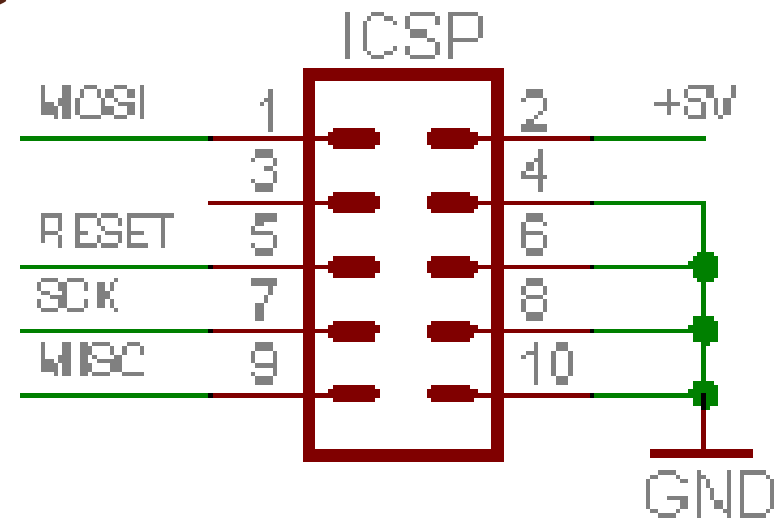
Good



Bad

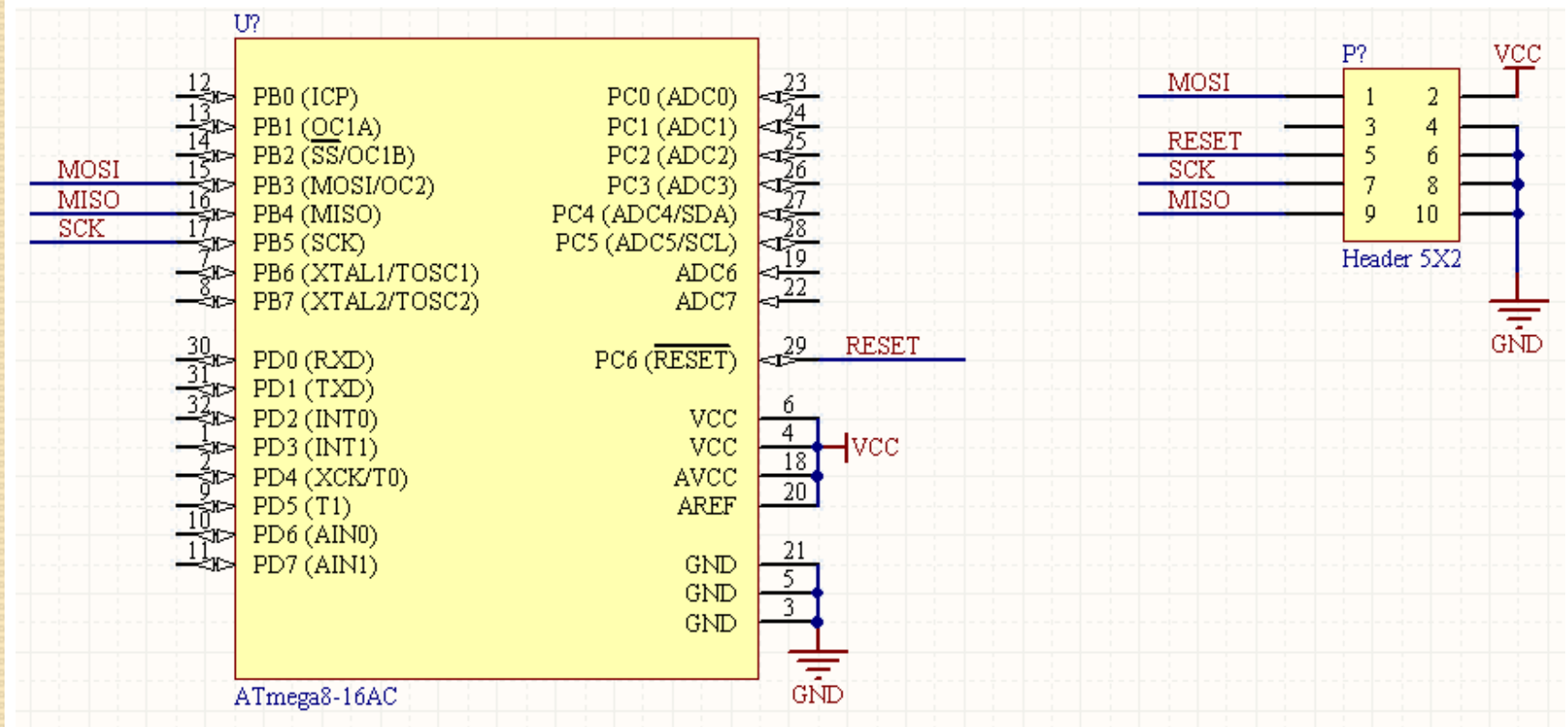
Using Net Labels

- Our programming connector now exactly matches the pinout of the programmer we selected to buy



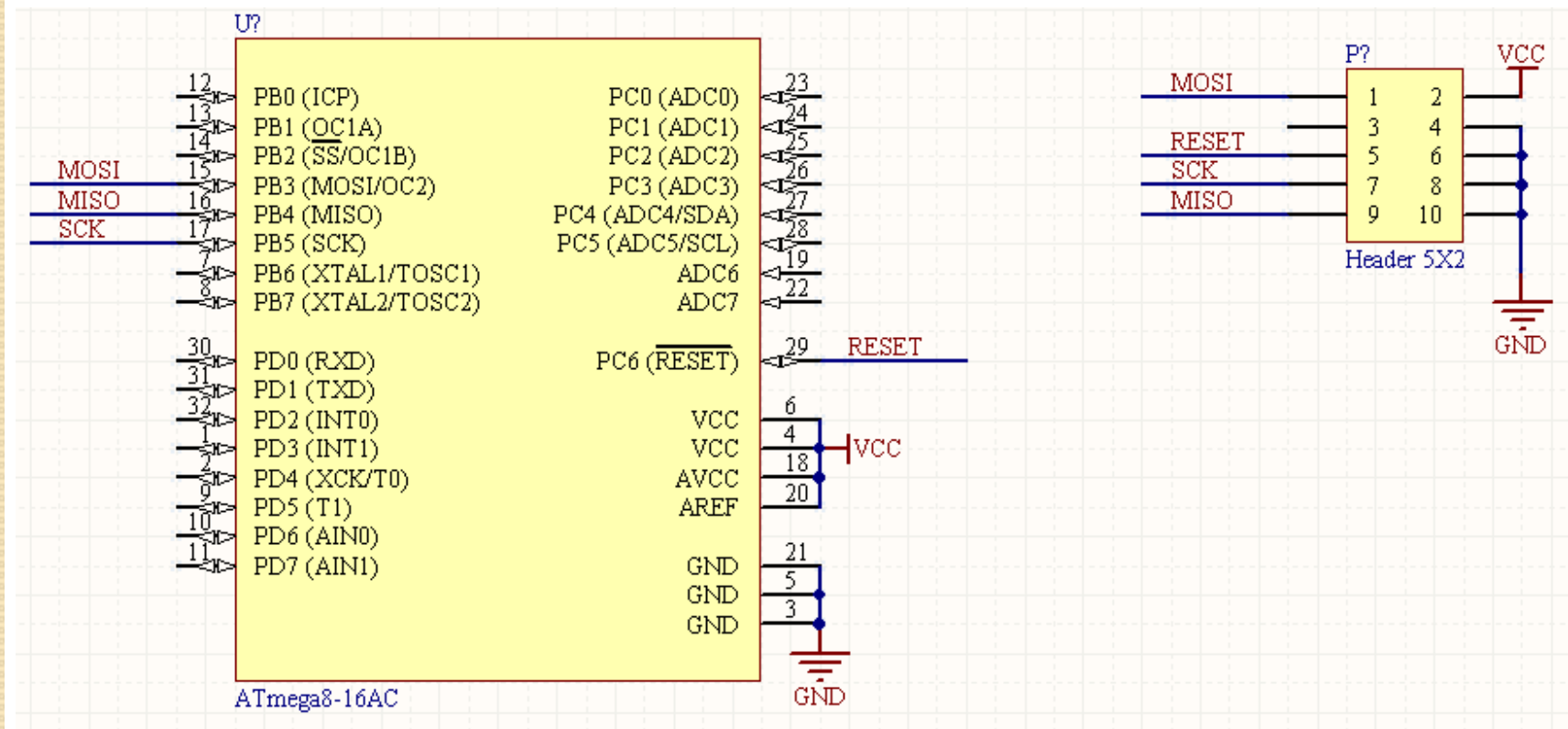
Using Net Labels

- Make the same net labels on the microcontroller as you did on the programming connector
- Connect the Power and Ground nets as shown below.
- The programmer is now electrically connected to the microcontroller

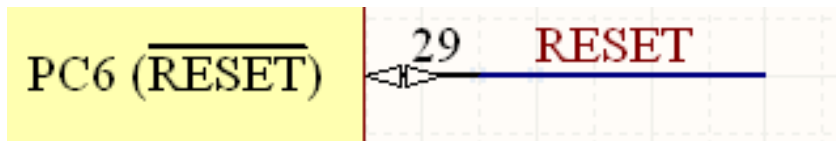


Using Net Labels

- Make the same net labels on the microcontroller as you did on the programming connector
- Connect the Power and Ground nets as shown below.
- The programmer is now electrically connected to the microcontroller



Reset Pin



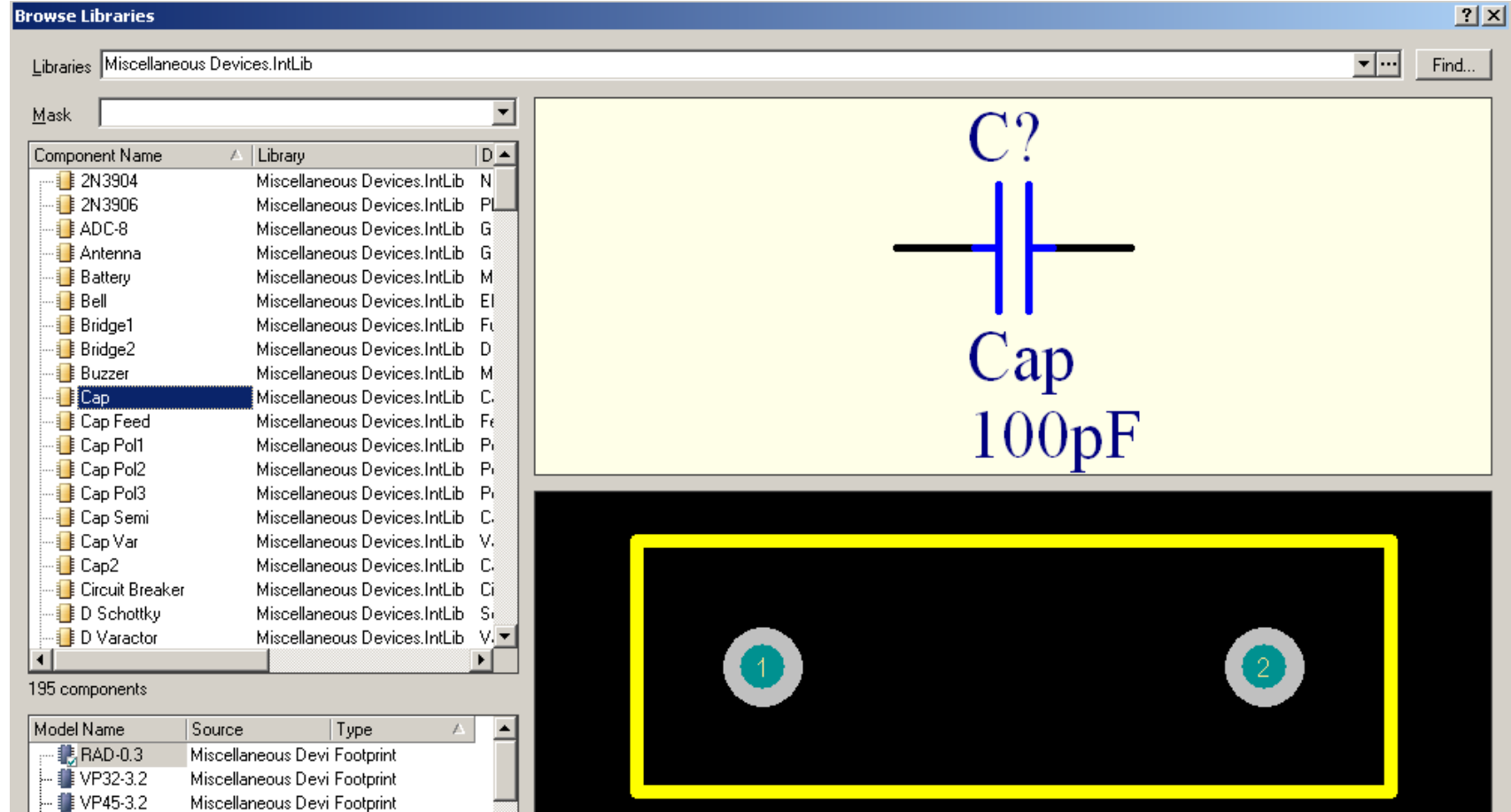
- The reset pin on the microcontroller is an **active low** pin, indicated by the bar above the name
- This means the device functions normally as long as the reset line is held high, and will go into reset mode when brought low (like rebooting a computer)
- Looking at the datasheet, we can see characteristics on each pin and how it works.

$\overline{\text{RESET}}$

Reset input. A low level on this pin for longer than the minimum pulse length will generate a reset, even if the clock is not running. The minimum pulse length is given in [Table 15 on page 38](#). Shorter pulses are not guaranteed to generate a reset.

Passive Components

- Now we need to add capacitors, resistors, and the LEDs.
- Place a part, select the Miscellaneous Devices library, and scroll down to CAP
- Place this part (Don't worry that the footprint isn't SMD 0805)



Passive Components

- Change the comment to =Value, and uncheck the visible
- Change the value to 0.1uF

The screenshot shows the 'Component Properties' dialog box. The 'Properties' section includes:

- Designator: C?
- Comment: =Value (Visible checked)
- Description: Capacitor
- Unique Id: JWPLVEHX
- Type: Standard

The 'Parameters for C? - Cap' table is shown on the right:

Visible	Name	Value	Type
<input type="checkbox"/>	LatestRevisionDate	17-Jul-2002	STRING
<input type="checkbox"/>	LatestRevisionNote	Re-released for DXP Platform.	STRING
<input type="checkbox"/>	PackageReference	RAD-0.3	STRING
<input type="checkbox"/>	Published	8-Jun-2000	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING
<input checked="" type="checkbox"/>	Value	0.1uF	STRING

Passive Components

- In the footprint section, click the 'add' button to add our 0805 footprint to this capacitor

Models for C? - Cap		
Name	Type	Description
CAP	Simulation	Capacitor
Cap	Signal Integrity	
RAD-0.3	Footprint	Capacitor; 2 Leads

Buttons: Add... Remove... Edit...

Add New Model [?] [X]

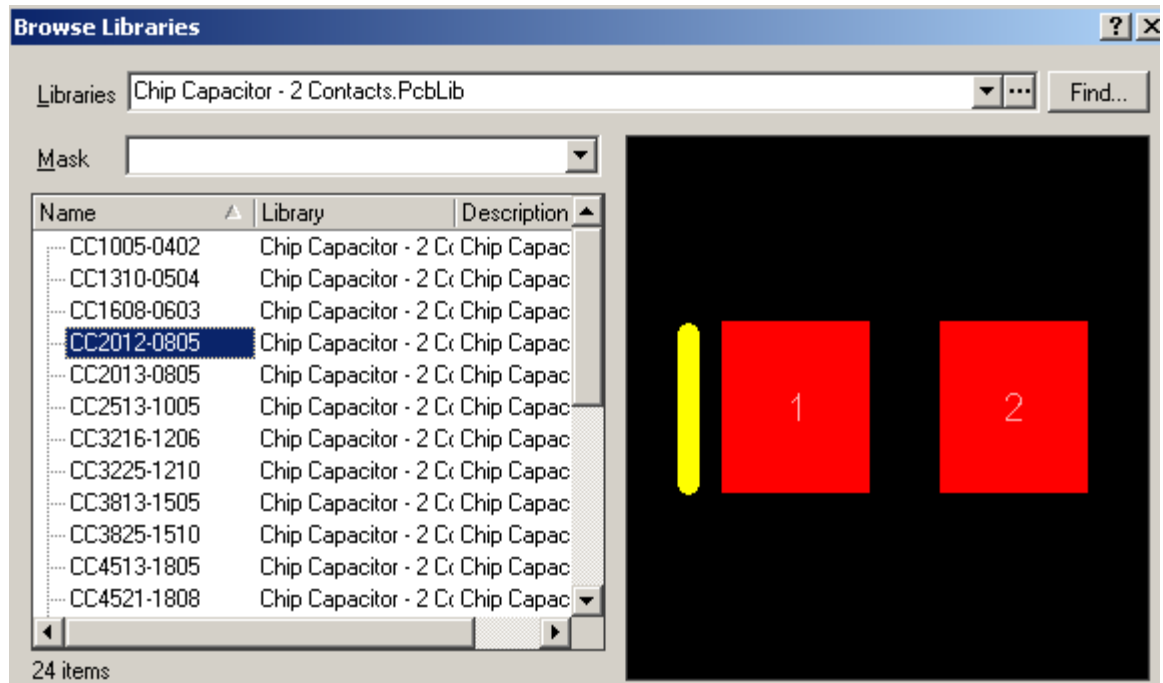
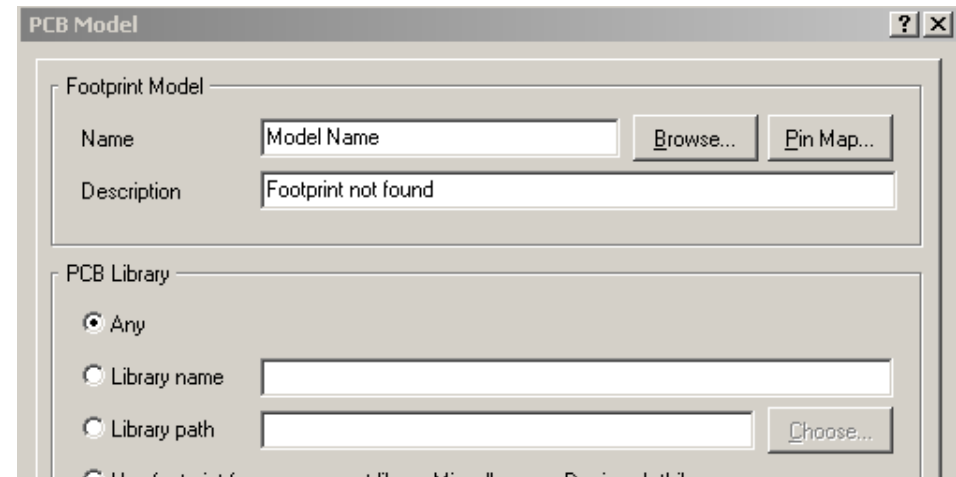
Model Type

Footprint

OK Cancel

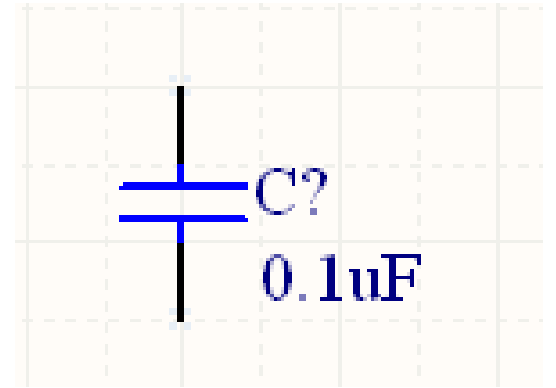
Passive Components

- Click 'browse' and make sure 'any' PCB library is selected
- Select the 'Chip Capacitor' library, and choose CC2012-0805
- This is code for:
CC= Chip Cap
2012- metric
0805 - empirical



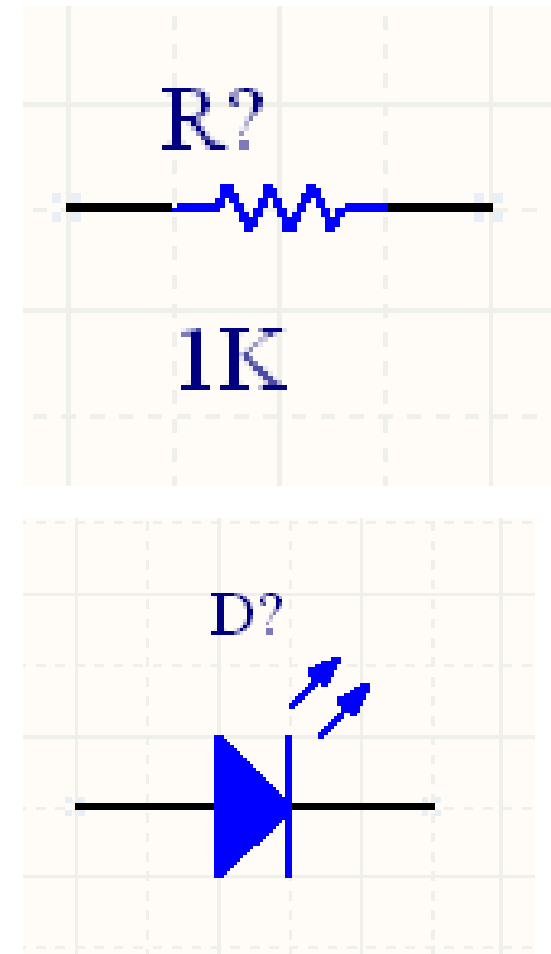
Passive Components

- Place the capacitor down. It should look something like this:
- This is now our template for a capacitor, just copy & paste whenever you need capacitors and they will all have the same footprint and settings we set for this capacitor
- Do not worry about the 'C?' , we will deal with labeling the parts at the end. For now it is a waste of time.



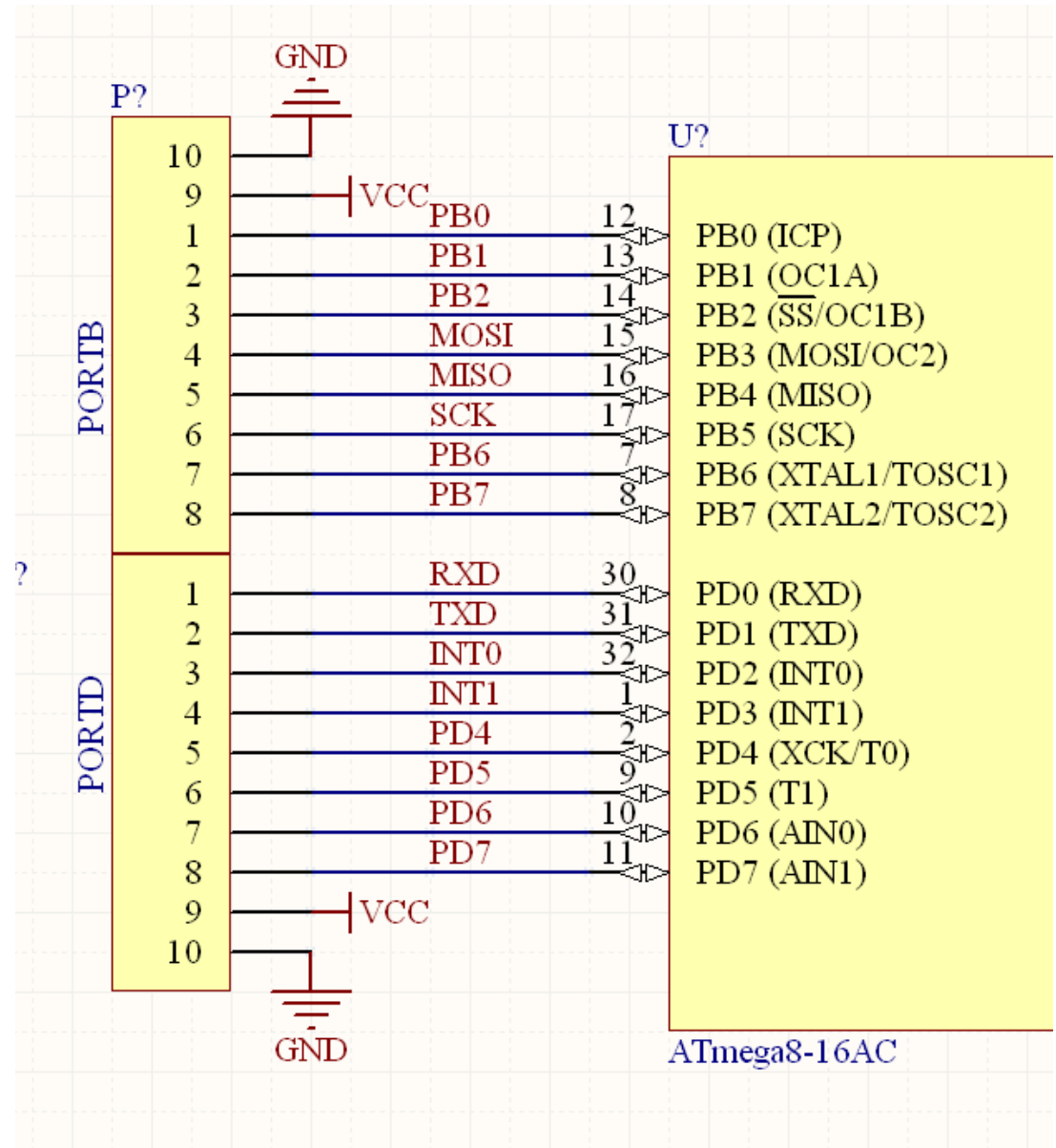
Passive Components

- Repeat the same process with resistors and LEDs (all in same miscellaneous devices library)
- Modify the settings and change the footprints
 - Diodes are from the Chip Diode library
 - Resistors are from the Chip Resistor library



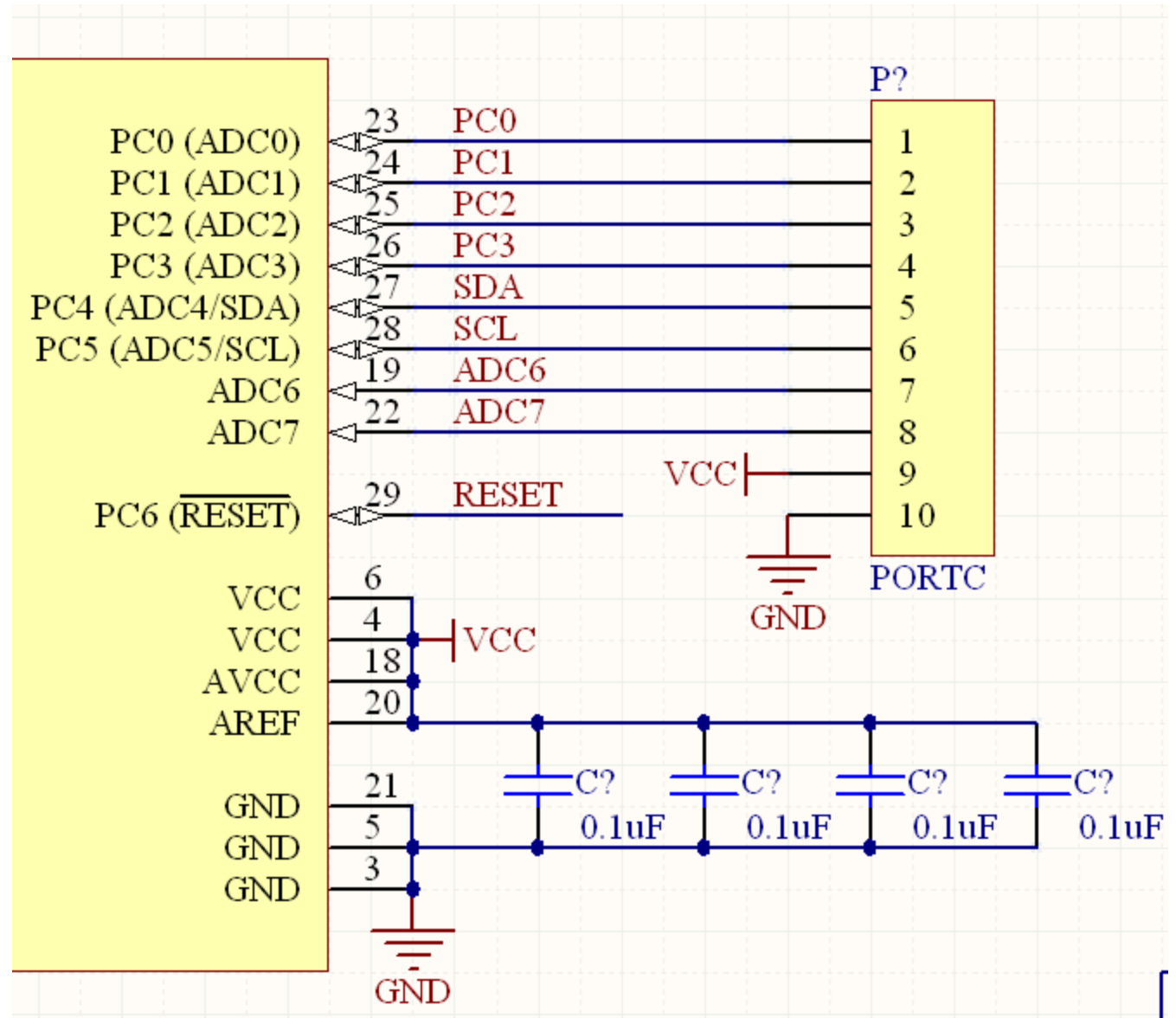
Complete the Schematic

- Copy and paste the parts as needed to finish this schematic
 - Highlight, CTRL-C (copy), CTRL-V (paste)
- The parts to the left are in the miscellaneous connectors library and are called HDRIX10
- NOTE: I moved the pin locations on the top connector, this does not change functionality at all, it just allows for a cleaner schematic. To do this you must first unlock the pins. Double click the part and uncheck the 'lock pins' to move the pins around



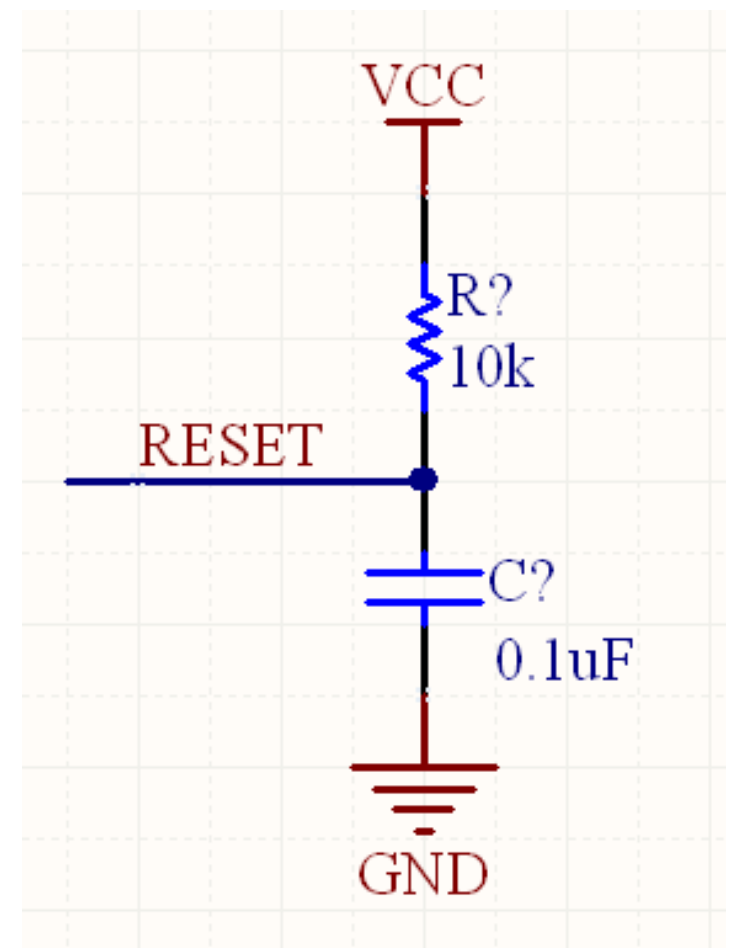
Complete the Schematic

- It is common practice to add a 0.1uF decoupling capacitor to EVERY power pin per IC



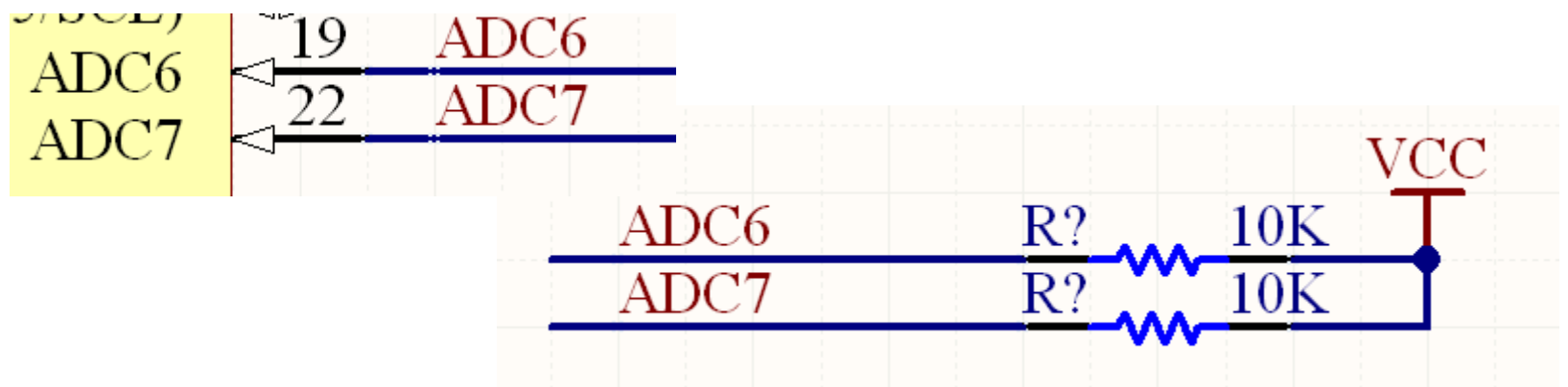
Complete the Schematic

- The reset line must be held high with external circuitry.
- Using a 10k **pull-up resistor** we can keep this out of reset
- NOTE: Try adding a switch from the reset line to ground. This is a hardware reset switch



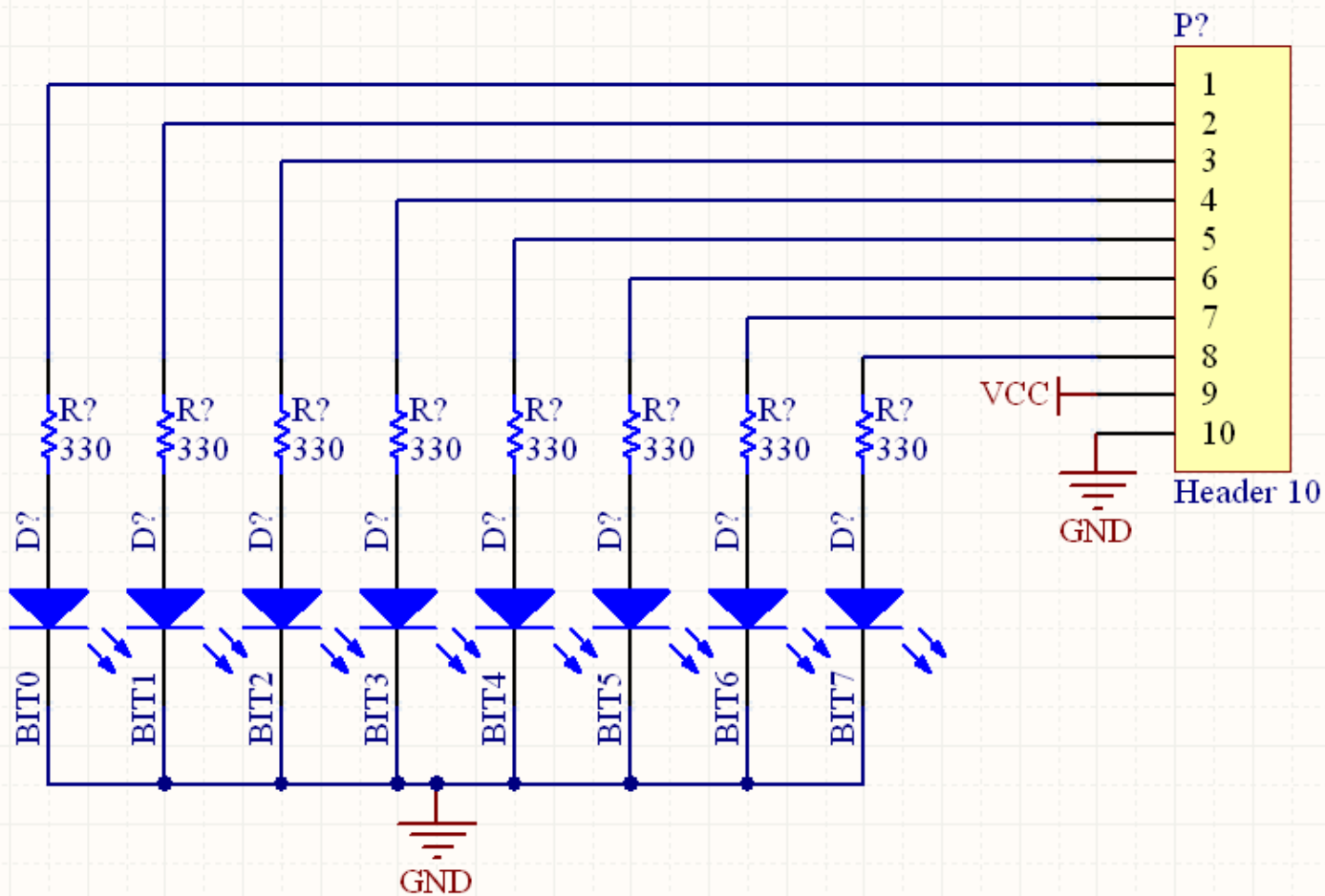
Complete the Schematic

- Pins 19 and 22 are INPUT pins on the microcontroller. These pins are both 10-bit analog to digital converters. It is necessary to pull these pins high with pull-up resistors if they are going to be unused (or possibly unused)



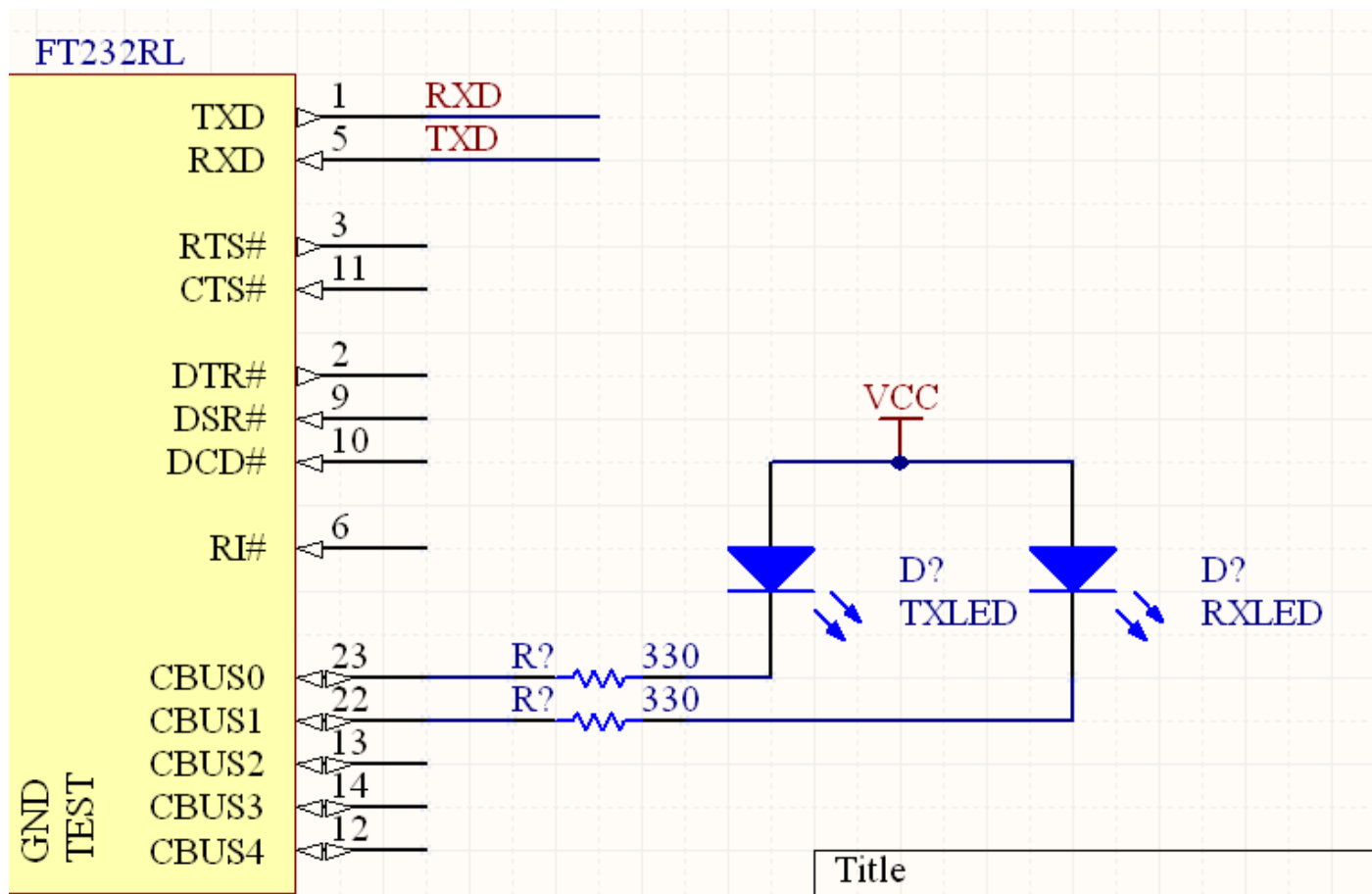
Complete the Schematic

- Don't forget we cannot apply 5V across the LED's. We must use current limiting resistors
- To ensure maximum functionality of the development board, rather than hardwiring the LEDs to a specific port, we will make it so a user can connect any of the three 8-bit ports



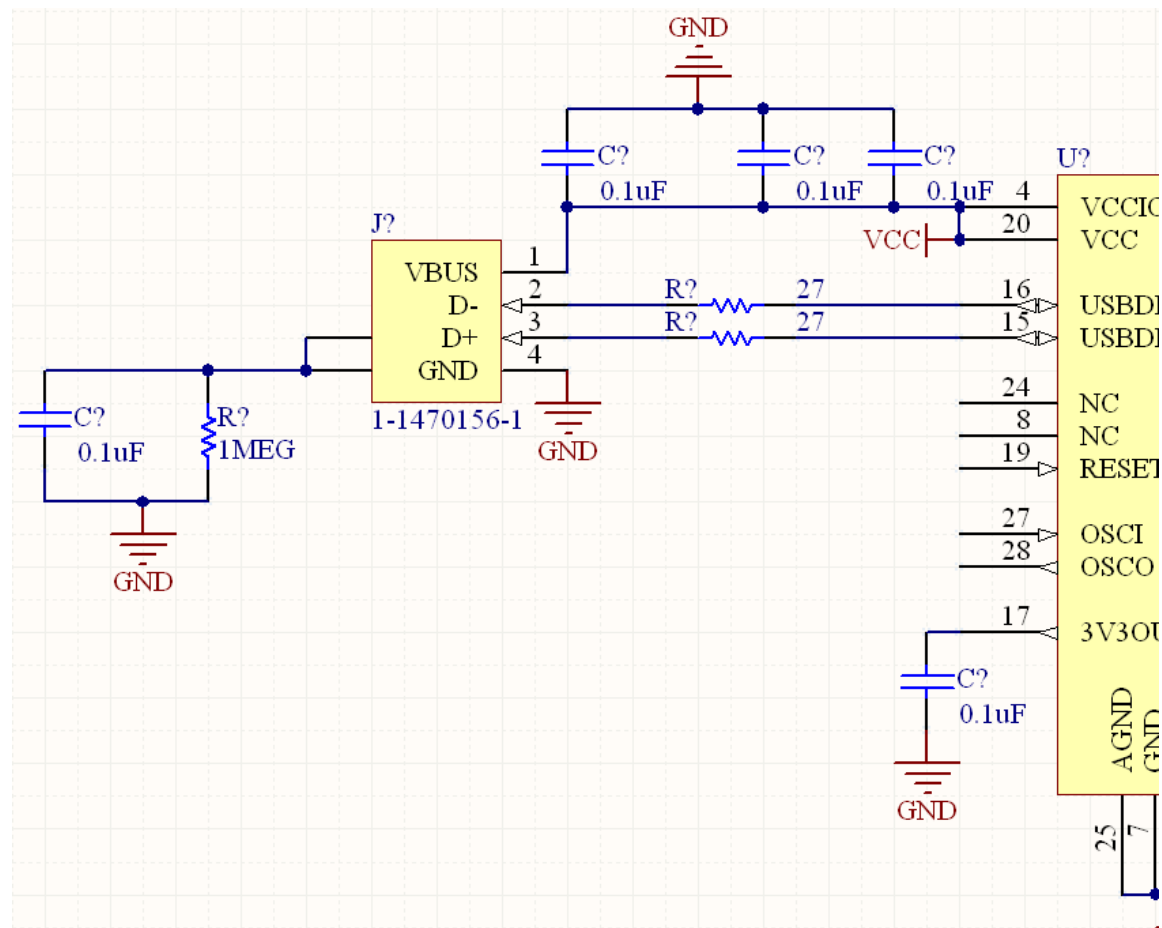
Complete the Schematic

- Notice how RXD on the microcontroller connects to TXD on the FT232 chip, This is because two transmitting signals cannot appear on the same line. One pin is an OUTPUT and one is an input.
- Because these pins are correctly defined with both parts, The DRC would catch that if we made the mistake of connecting TXD to TXD
- The LED's here are optional and light when data is being sent or received



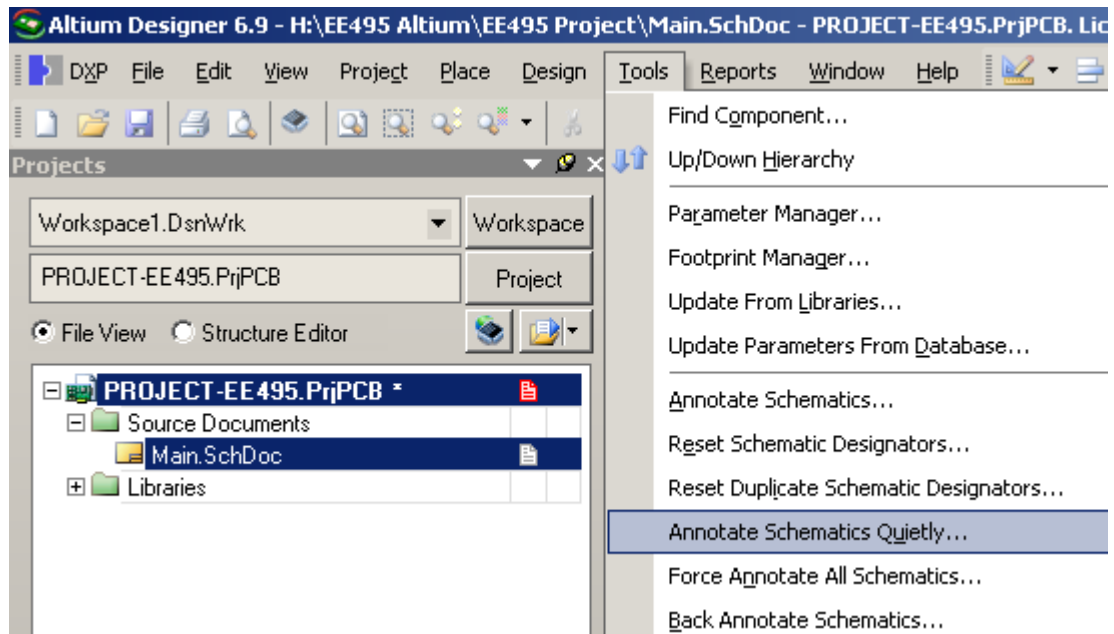
Complete the Schematic

- I have altered the typical application circuit for this in two ways:
 - I have added terminating resistors on the differential pair
 - I have not included the ferrite bead , you should include this . You can include a 0805 footprint as a placeholder for the ferrite bead which comes in a chip



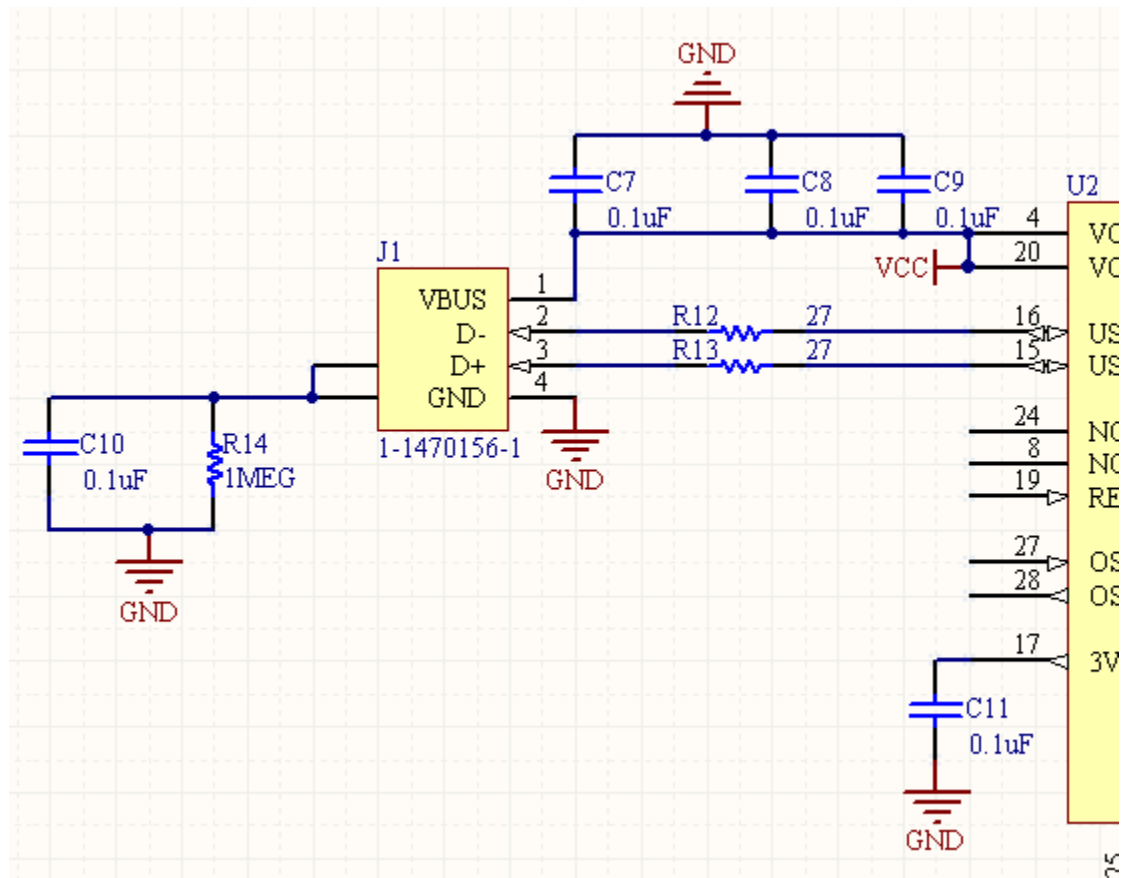
Complete the Schematic

- Now that all the parts are added, we must label all the components
- Follow the menu sequence to the right to label all the parts automatically
- Click 'Yes' when asking to update the changes

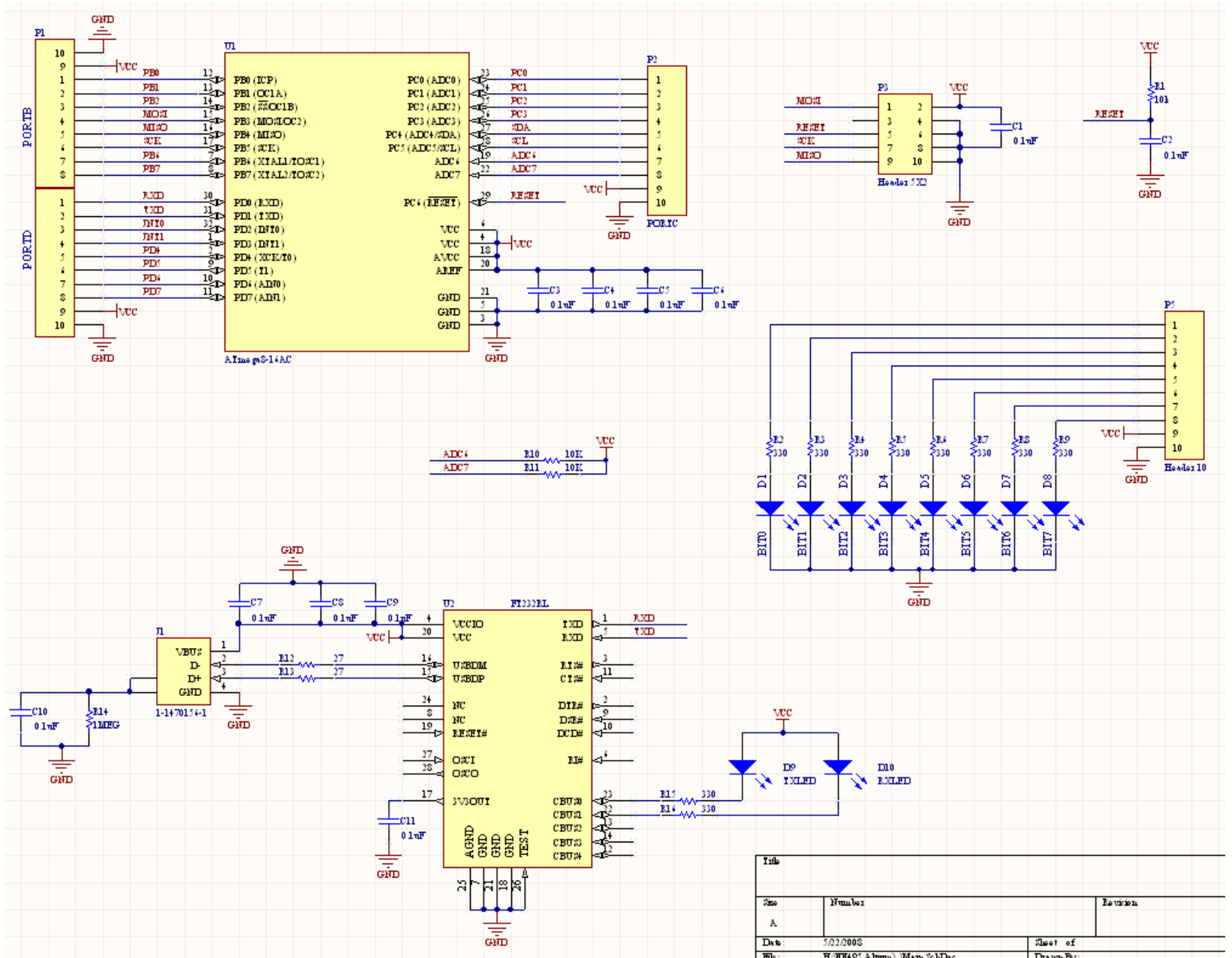


Complete the Schematic

- All the parts are now properly numbered
- NOTE: This step is necessary before working with simulations or PCB Layout

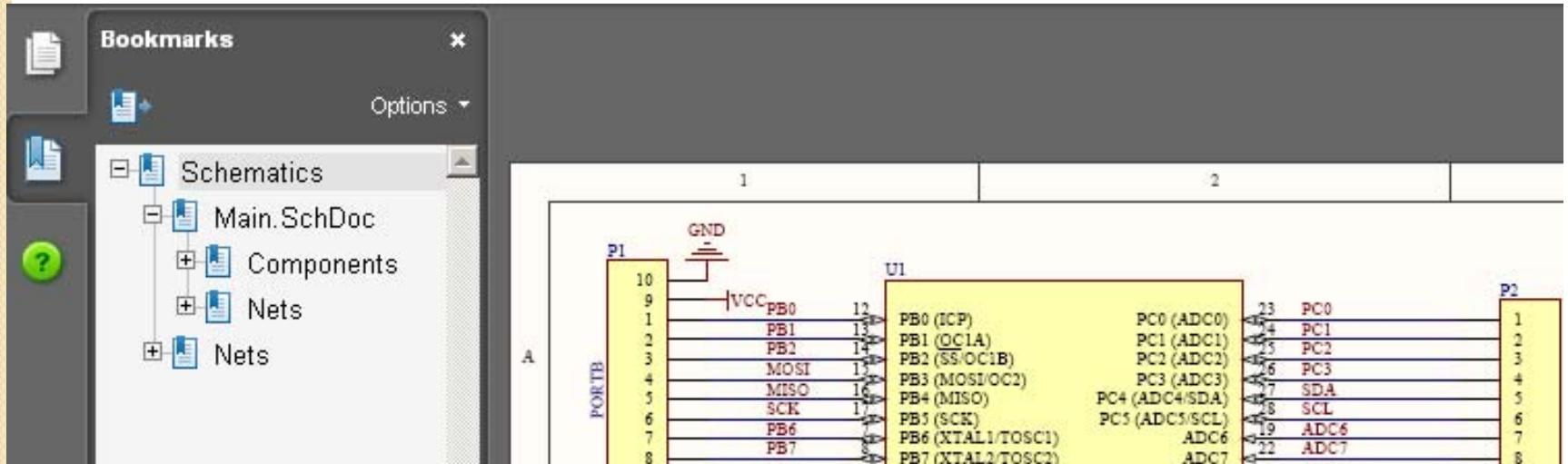


Finished Schematic



Smart PDF

- I have included a Smart PDF of the schematic for this development board in the downloads section of the website.
- In this PDF file you can select and zoom in to see parts, nets, etc ...



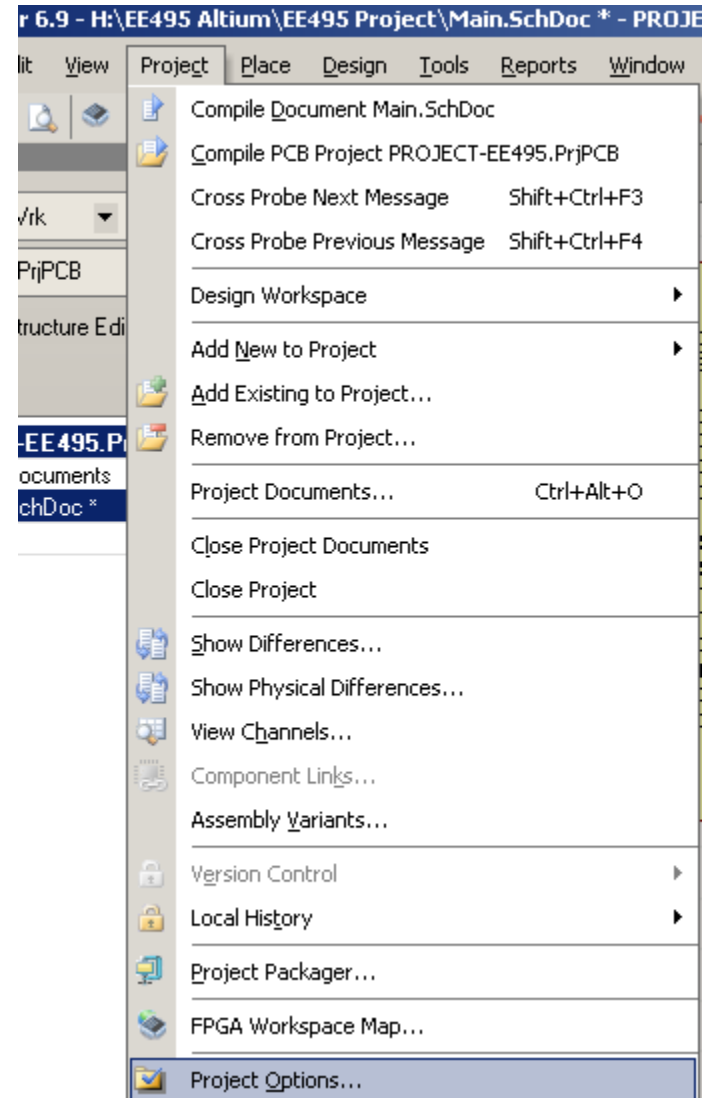


Compiling the Project

- Compiling the project is fancy terminology for error checking.
- Error checking, although not required, is a must before converting the schematic drawing into a PCB document.
- We can't error check without setting up rules, so first we must setup rules

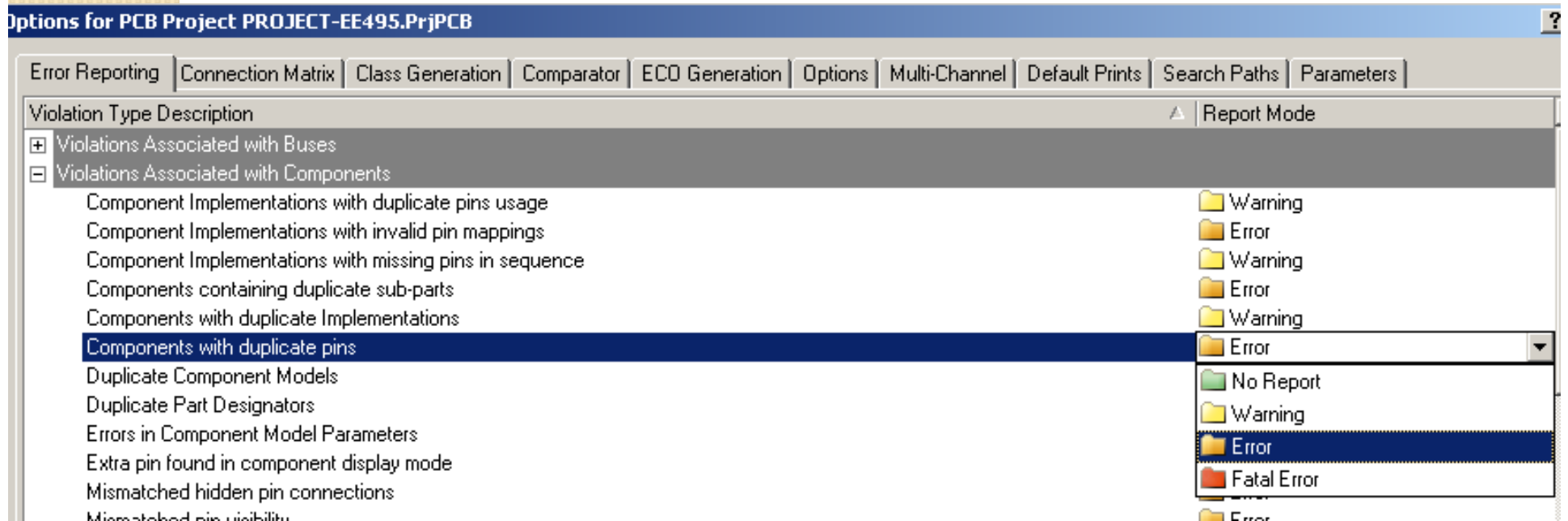
Compiling the Project

- Click
 - Project > Project Options



Compiling the Project

- Under the Error Reporting tab, you can set options for which level of report you will get.
 - Some errors are serious
 - Some are not a big deal



Compiling the Project

- A setting of 'no report' is the same as the software ignoring it
- A warning will allow an action, but won't be happy about it
- Set the 'Error' level or above to see immediate feedback
- Fatal error will not allow compilation of the project

Floating net labels

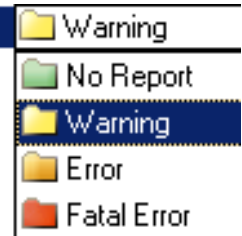
Floating power objects

Global Power-Object scope changes

Harness Object on a Bus

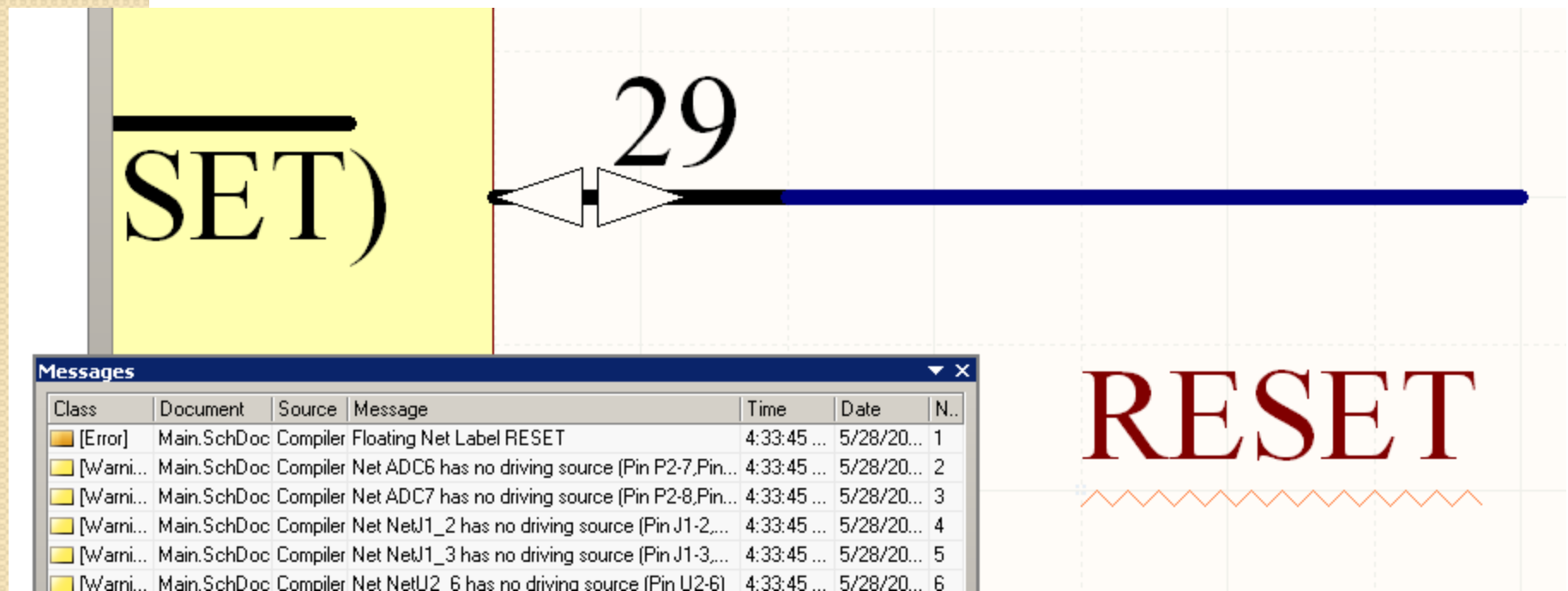
Harness Object on a Wire

Missing Negative Net in Differential Pair



Compiling the Project

- If we set 'floating net labels' to error, when we compile the document, a red squiggly line will appear beneath the floating label and it will appear in the compilation report



The image shows a screenshot of a PCB design tool. On the left, a yellow rectangular area contains the text "SET)" with a horizontal line above it. To the right of this area, a pencil icon with the number "29" is positioned over the text. A blue horizontal line extends from the pencil tip to the right. Below the yellow area, a red squiggly line is visible under the text "RESET". In the bottom left corner, a "Messages" window is open, displaying a list of error messages:

Class	Document	Source	Message	Time	Date	N..
[Error]	Main.SchDoc	Compiler	Floating Net Label RESET	4:33:45 ...	5/28/20...	1
[Warni...]	Main.SchDoc	Compiler	Net ADC6 has no driving source (Pin P2-7,Pin...	4:33:45 ...	5/28/20...	2
[Warni...]	Main.SchDoc	Compiler	Net ADC7 has no driving source (Pin P2-8,Pin...	4:33:45 ...	5/28/20...	3
[Warni...]	Main.SchDoc	Compiler	Net NetJ1_2 has no driving source (Pin J1-2,...	4:33:45 ...	5/28/20...	4
[Warni...]	Main.SchDoc	Compiler	Net NetJ1_3 has no driving source (Pin J1-3,...	4:33:45 ...	5/28/20...	5
[Warni...]	Main.SchDoc	Compiler	Net NetU2_6 has no driving source (Pin U2-6)	4:33:45 ...	5/28/20...	6

Compiling the Project

- If we set 'floating net labels' to fatal error, when we compile the document, a red squiggly line will appear beneath the floating label and it will appear in the compilation report.

SET)

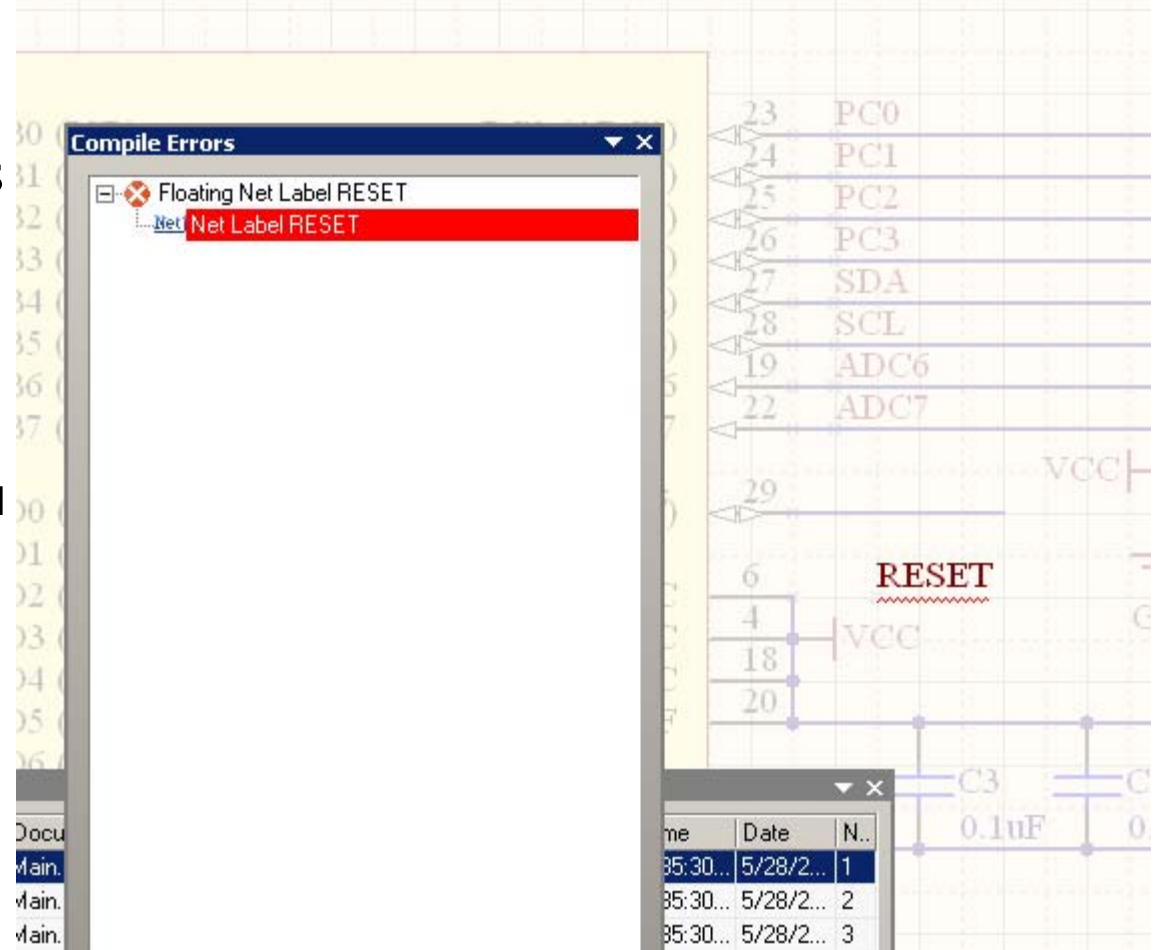
29

RESET

Class	Document	Source	Message	Time	Date	N..
[Fatal Error]	Main.Sch...	Comp...	Floating Net Label RESET	4:35:30...	5/28/2...	1
[Warning]	Main.Sch...	Comp...	Net ADC6 has no driving source (Pin P2-7...	4:35:30...	5/28/2...	2
[Warning]	Main.Sch...	Comp...	Net ADC7 has no driving source (Pin P2-8...	4:35:30...	5/28/2...	3
[Warning]	Main.Sch...	Comp...	Net NetJ1_2 has no driving source (Pin J...	4:35:30...	5/28/2...	4
[Warning]	Main.Sch...	Comp...	Net NetJ1_3 has no driving source (Pin J...	4:35:30...	5/28/2...	5

Compiling the Project

- If you double-click the error in the messages box, it will show you compile errors
- From there you can select the offending error and it will dim the schematic except for the error





Compiling the Project

- I recommend reading through and setting whatever rules you think will be appropriate.
- Keep in mind it is better to have a compilation report with a lot of information that you don't care about than to have an important error slip past
- If there are no errors, you will not get a popup screen when the compilation is complete



PROJECT UPDATE

- DUE WED. 06/04/08 !!
- Single page report with the following
 - What have you done for your project- (Be Specific!!) I expect this to be mostly schematic design and research at this point
 - What do you have left to complete
 - Who is responsible for each part of the design
 - Schematic Entry
 - PCB Layout
 - Implementing Technology (Ex: Mike will be in charge of interfacing the transmitter with a microcontroller)
 - Exam I will be based on this report.
 - Do not think that if you have done no work yet that you are not responsible for the project or that your test will be easy. The more work you have done by the 9th, the easier the test should be.
 - Every group member is responsible if someone is trying to get out of doing work. Do not cover for anyone else. If I find out that the work was not done by an individual, I will hold the entire group responsible
 - If there is a problem with someone not working, you have until the 7th to split from the group and work on the same project by yourself if you let me know.

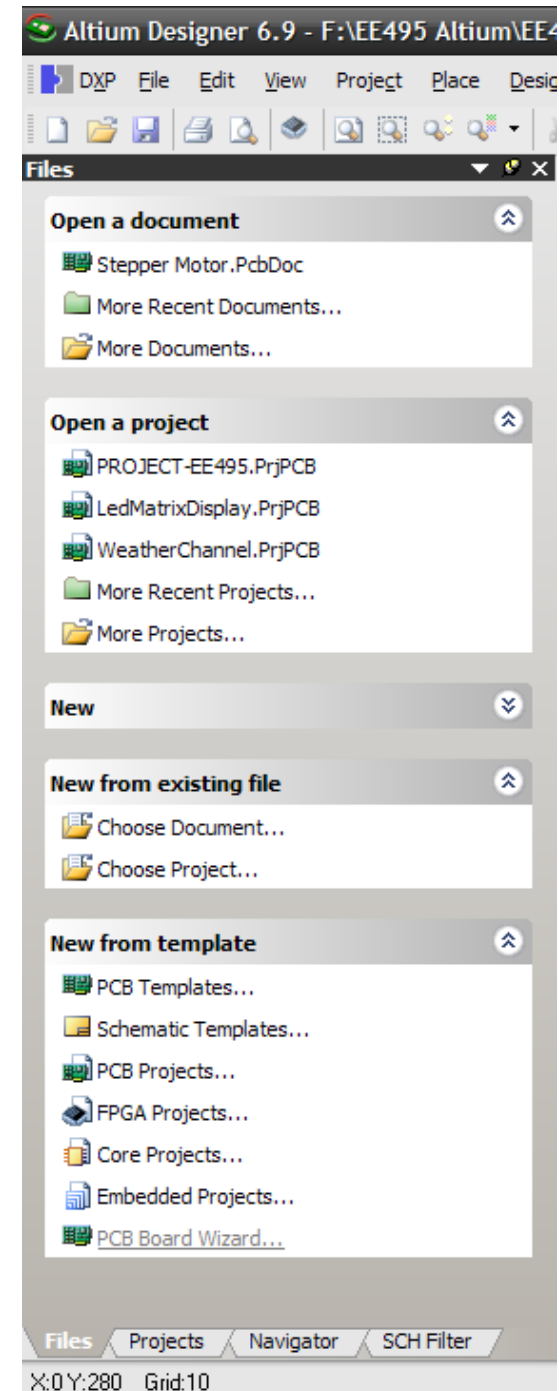


Creating the PCB

- One of the reasons why it is important to define the PCB footprints while entering in the schematic is to be able to provide a good estimate of how much board area you will need.
- Generally speaking the **following things make production more expensive**
 - Larger area
 - More layers
 - Smaller holes
 - Closer tolerances
 - Closer traces and spacing
 - Thinner or Thicker PCB
- Our goal is to keep cost as low as possible while still being able to transfer a schematic to a functioning PCB.
- There are no special characteristics of this design that require any exotics, and we can use a 2 layer PCB for low cost

Creating the PCB

- From the project tab in the bottom left corner, click the 'Files' tab and click the upward facing arrows until you can see
- NEW FROM TEMPLATE
- Select PCB Board Wizard



Creating the PCB

- You will get a very friendly wizard screen, click next to continue



Altium Designer New Board Wizard

This wizard will help you create and set up a new printed circuit board.

It will take you through some simple steps to define the board layout, manufacturing parameters and layer information.

Creating the PCB

- Choose the board units as Imperial for this since that is what most of us are most familiar with
- NOTE: You are not stuck with the choice of units that you choose here. The units specified are **ONLY** used for the PCB wizard

Choose Board Units

Choose the type of measurement units for the board being created.

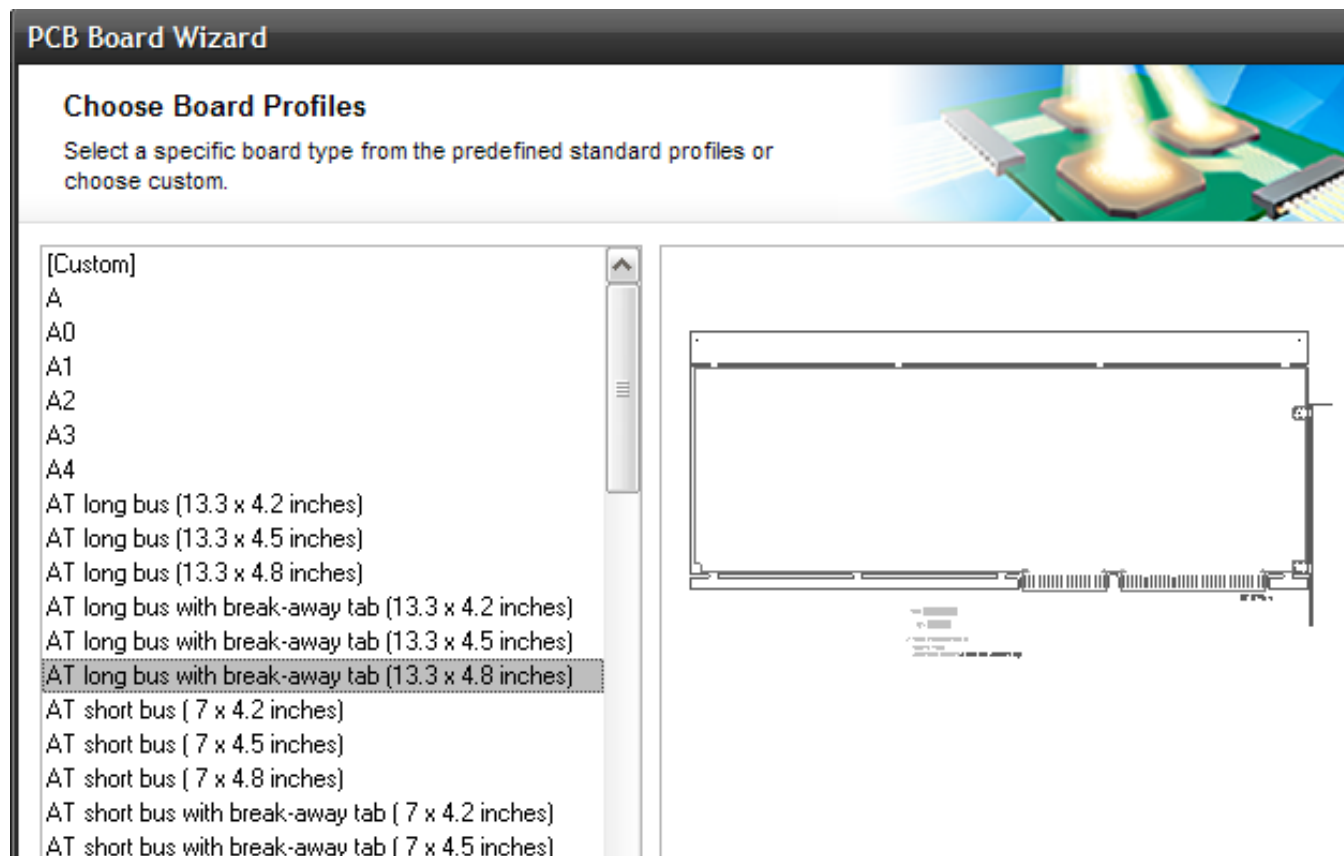
If you use mils, click Imperial. If you use millimetres, click Metric.

Imperial

Metric

Creating the PCB

- There are many predefined templates for exotic designs or standard PCB computer interface forms, such as AT Bus, PCI bus, Eurocard, and many others.
- Choose CUSTOM (at the top)



Creating the PCB

- Leave the default settings (Rectangular, 5000 mils x 4000 mils)
- The dimensions of the board will be on a generic, information layer (Mechanical layer) which is used to give information to the board house.

PCB Board Wizard

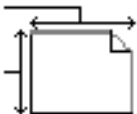
Choose Board Details
Choose Board Details

Outline Shape:

Rectangular
 Circular
 Custom

Board Size:

Width
Height



Dimension Layer

Boundary Track Width

Dimension Line Width

Keep Out Distance From Board Edge

Title Block and Scale
 Legend String
 Dimension Lines

Corner Cutoff
 Inner CutOff

Creating the PCB

- We want to have a two layer board. This is two signal layers (top of board, bottom of board) and zero internal plane layers, make this change
- Do you remember what a power plane is ? How does it differ from a signal layer? (HINT: you can have signal layers internal or external, and you can have power planes internal and external)

Choose Board Layers

Set the number of signal layers and power planes suitable for your design.

Signal Layers:

Power Planes:

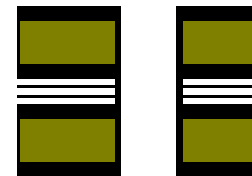
 

Creating the PCB

- Make sure the hole type is set on thruhole vias only
- Do you remember what a via is ??

- Thruhole Vias only
- Blind and Buried Vias only

- Thruhole Vias only
- Blind and Buried Vias only



Creating the PCB

- Our board contains mostly surface mount components, so check this option
- Choose that both sides of the board can contain components.
- NOTE: These settings really don't matter. They exist for the autorouter which we will not be using

The board has mostly:

- Surface-mount components.
- Through-hole components.

Do you put components on both sides of the board?

- Yes
- No



Creating the PCB

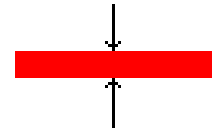
- Change track size (10mils) and trace clearance (8 mils)
- What is the metric length of 8 mils ????
- Change the minimum via hole size to 15 mils, and the min. via width to 30 mils
- NOTE: ALWAYS be sure your settings can be fabricated at a reasonable price before setting these values

Choose Default Track and Via sizes

Choose the minimum track size, via size and the copper to copper clearances to use on the new board.

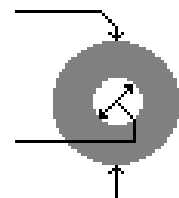
Minimum Track Size

8 mil



Minimum Via Width

30 mil

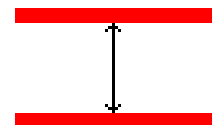


Minimum Via HoleSize

15 mil

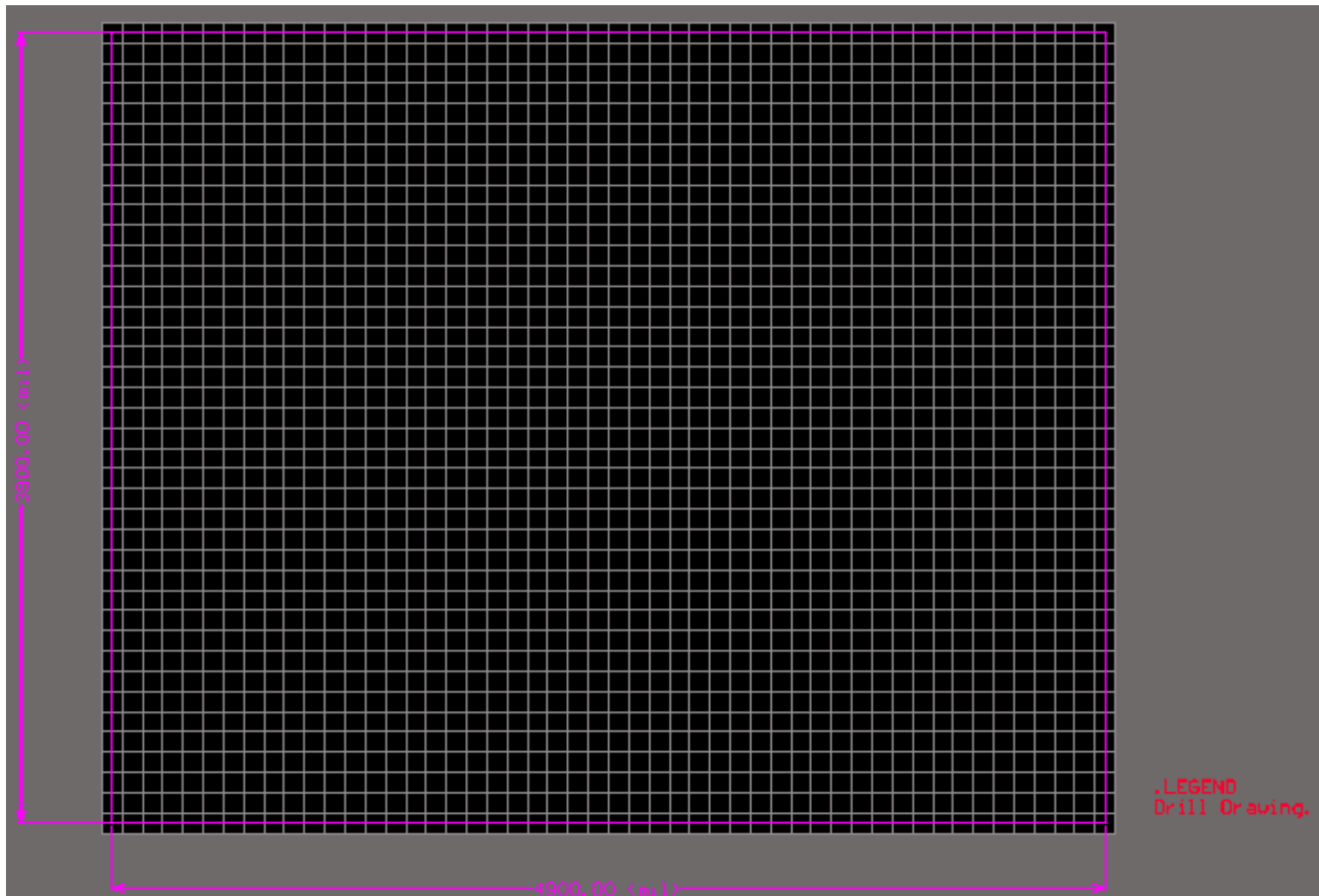
Minimum Clearance

8 mil



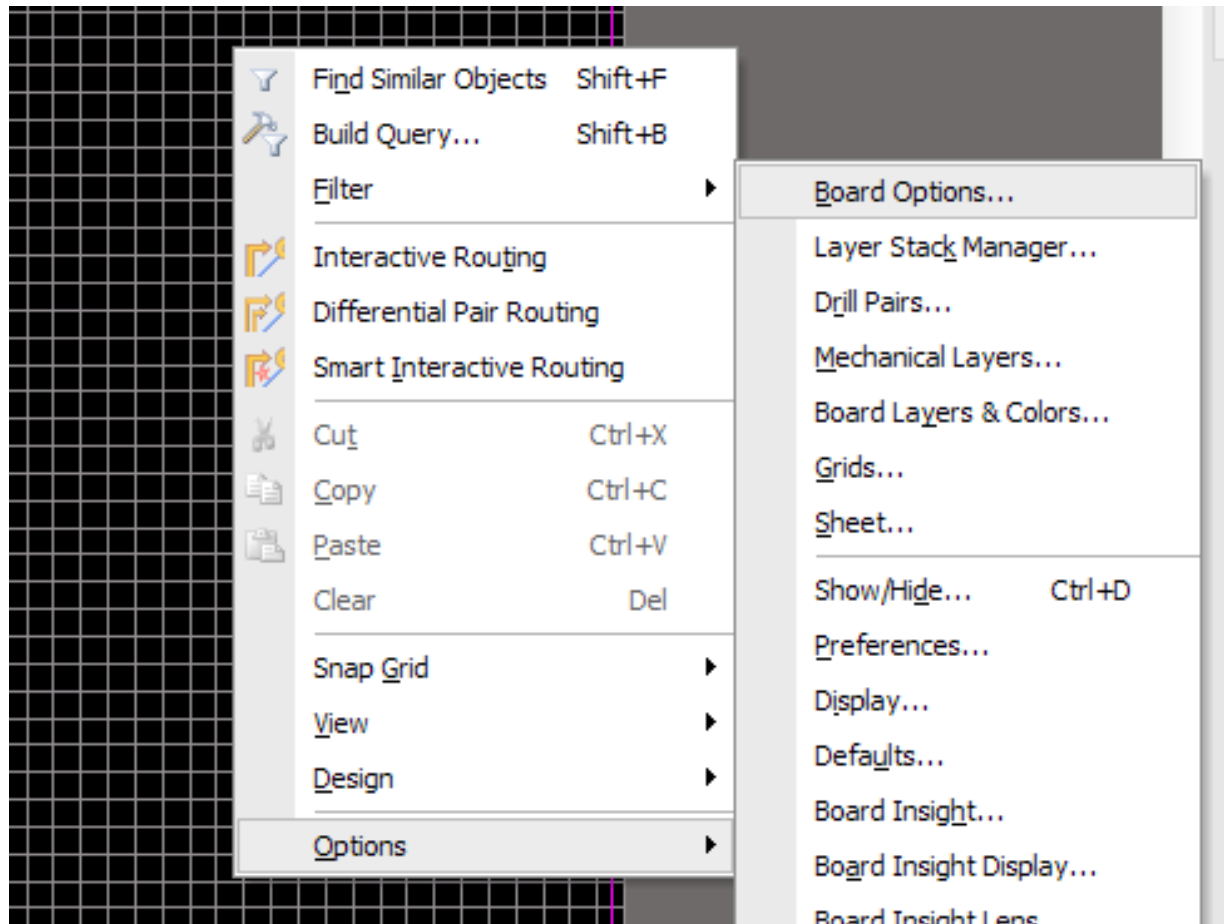
Creating the PCB

- The wizard is now complete and your PCB is created based on the values you have given.



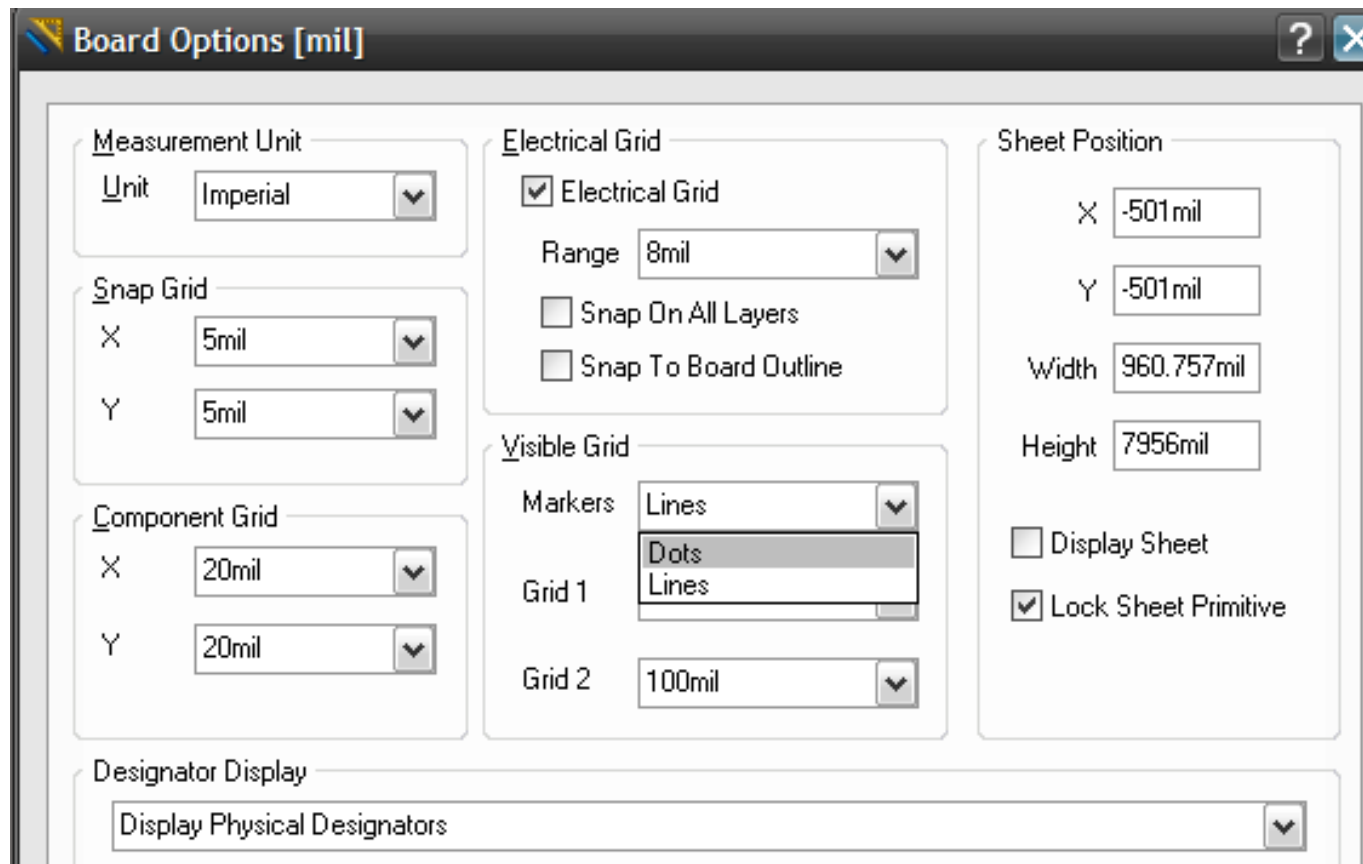
Creating the PCB

- Right click on the PCB >
- Click 'Options'
- Click 'Board Options'



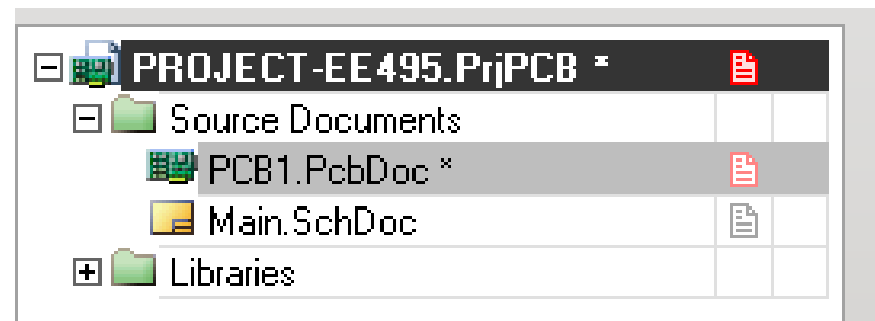
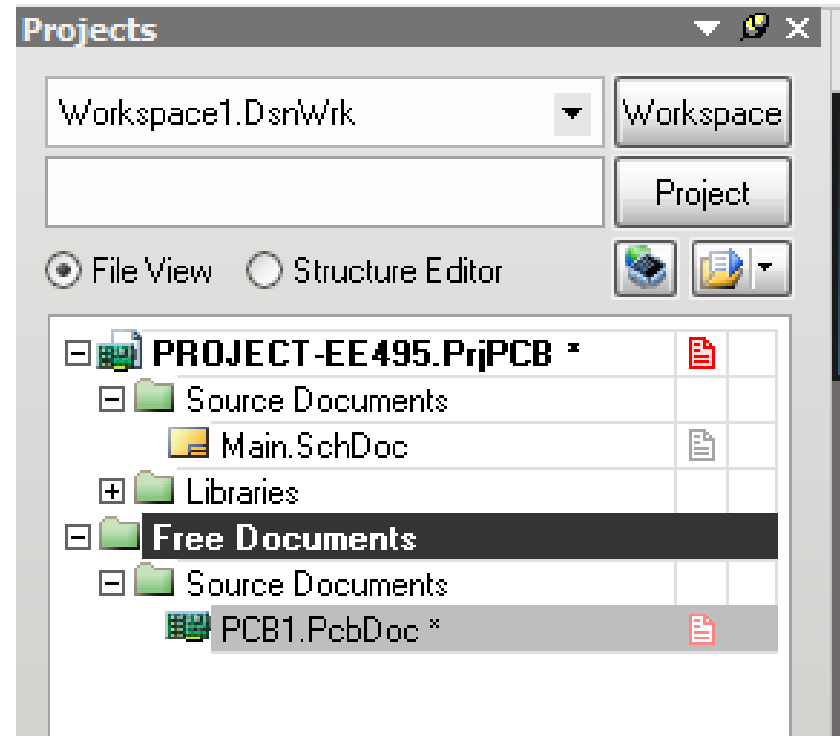
Creating the PCB

- Uncheck 'Display Sheet' (this has a use, but for now we want to reduce clutter)
- Change the markers from lines to dots and click OK.
- NOTE: The snap grid and component grid are very important, we will get into this later though.



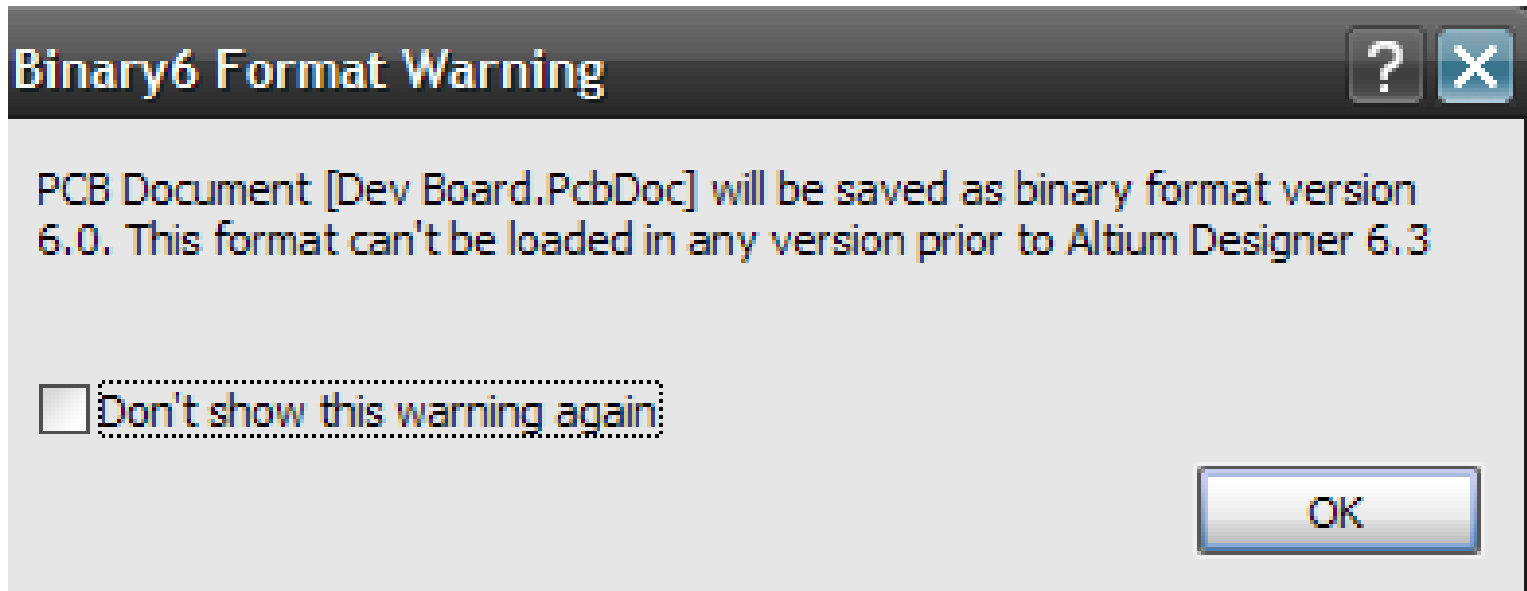
Creating the PCB

- The PCB has been added to the list of documents, but has NOT been added to the project (Notice it is listed as a free document)
- Click and drag the PCB to your project
- Right click on the PCB1.PcbDoc file and save the file in your project directory.
- Save the file as 'Dev Board'



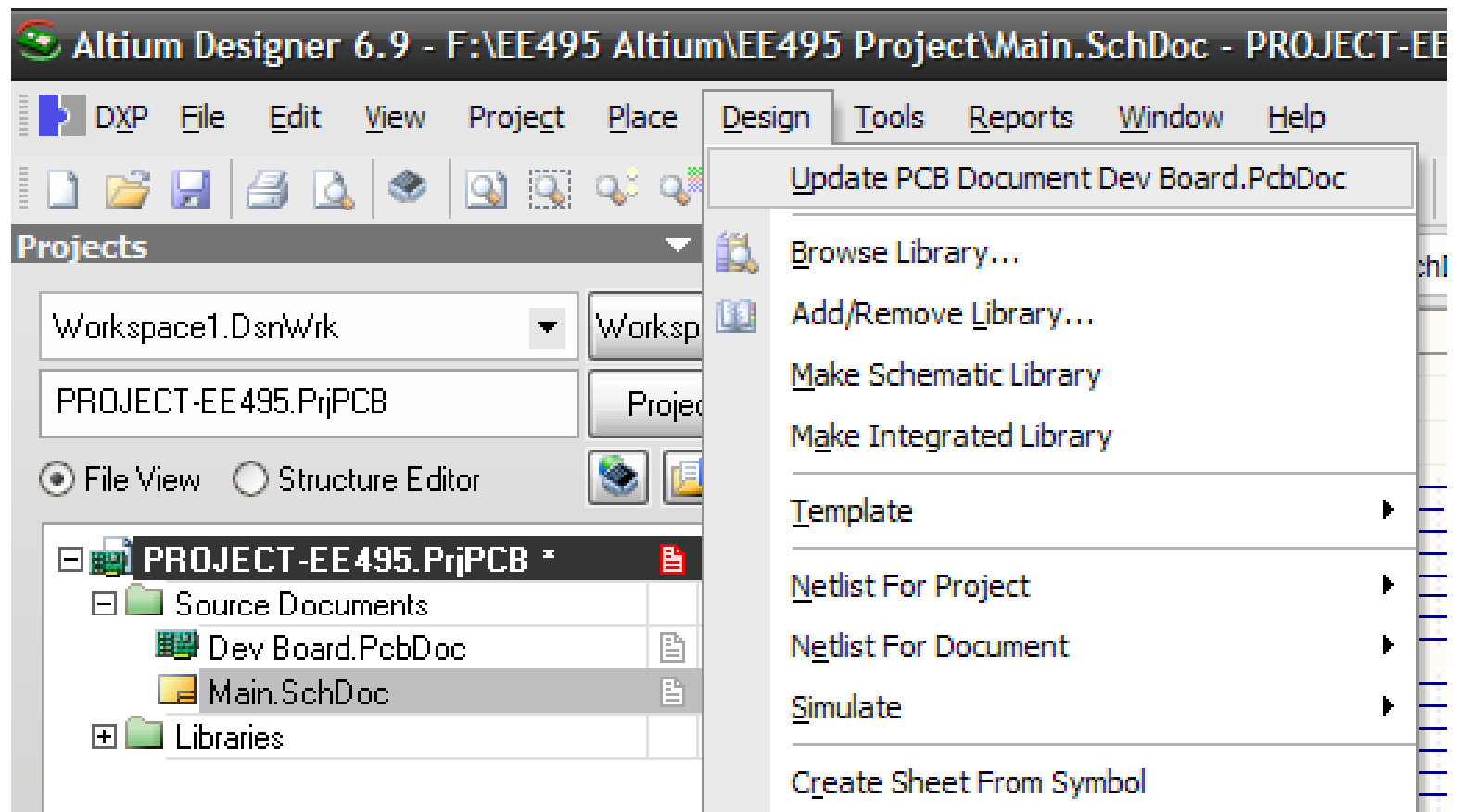
Creating the PCB

- If you get the following warning, just click OK. It is telling you there may be compatibility issues if you try to open the PCB document in an older version of Altium.



Creating the PCB

- If you select your schematic again, click the Design button up top and then click on 'UPDATE PCB DOCUMENT DEV BOARD.PcbDoc'
- This will begin the process to turn your schematic into a PCB



Creating the PCB

- This will open the **ECO (Engineering Change Order)**
- This is asking which changes to make.
- Validate changes will check for errors in the process
- Execute will actually perform the change

Engineering Change Order

Modifications				Status	
Enable	Action	Affected Object	Affected Document	Check	Done
Add Components(45)					
<input checked="" type="checkbox"/>	Add	C1	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C2	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C3	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C4	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C5	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C6	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C7	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C8	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C9	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C10	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	C11	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D1	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D2	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D3	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D4	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D5	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D6	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D7	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D8	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D9	To Dev Board.PcbDoc		
<input checked="" type="checkbox"/>	Add	D10	To Dev Board.PcbDoc		

Validate Changes Execute Changes Report Changes... Only Show Errors

Creating the PCB

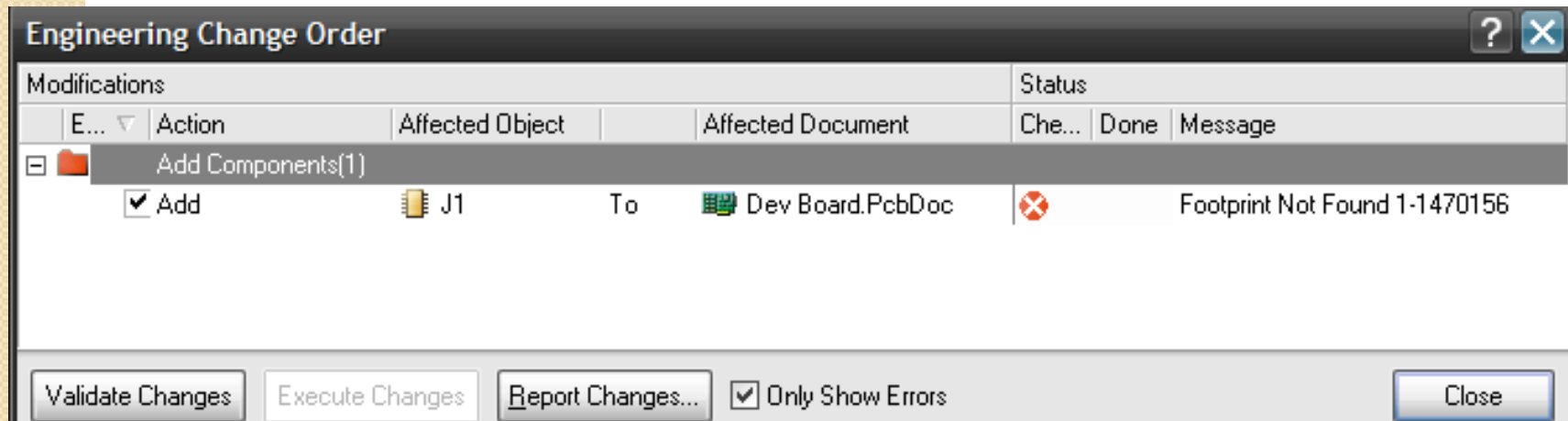
- Click 'Validate' at the bottom. This may take a while depending on the complexity of the project.
- Notice the green check boxes to the right. This means that each step with the green check mark can be performed successfully.
- I have deliberately removed the library for the USB connector to show what will happen if there is an error for a step (red x) Adding the library will fix this error

Engineering Change Order

Modifications						Status		
Enable	Action	Affected Object		Affected Document	Check	Done	Message	
<input checked="" type="checkbox"/>	Add	C6	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	C7	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	C8	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	C9	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	C10	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	C11	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D1	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D2	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D3	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D4	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D5	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D6	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D7	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D8	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D9	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	D10	To	Dev Board.PcbDoc	✓			
<input checked="" type="checkbox"/>	Add	J1	To	Dev Board.PcbDoc	✗		Footprint Not Found 1-1470156	
<input checked="" type="checkbox"/>	Add	P1	To	Dev Board.PcbDoc	✓			
<input type="checkbox"/>	...	--	-	-	-			

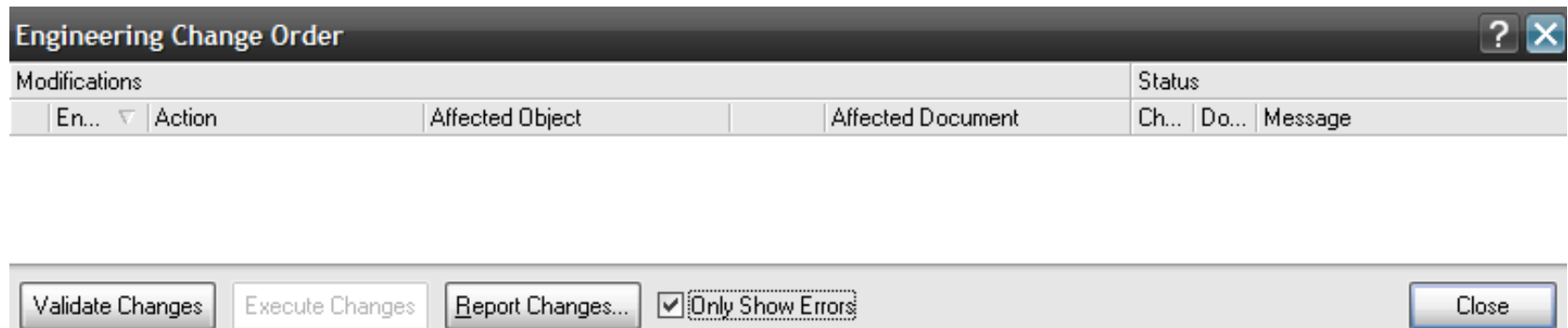
Creating the PCB

- Because there are usually many different steps to creating the ECO, it is usually helpful to check the box that says 'Only show errors' at the bottom.
- You are in good shape when no errors occur and this list is empty

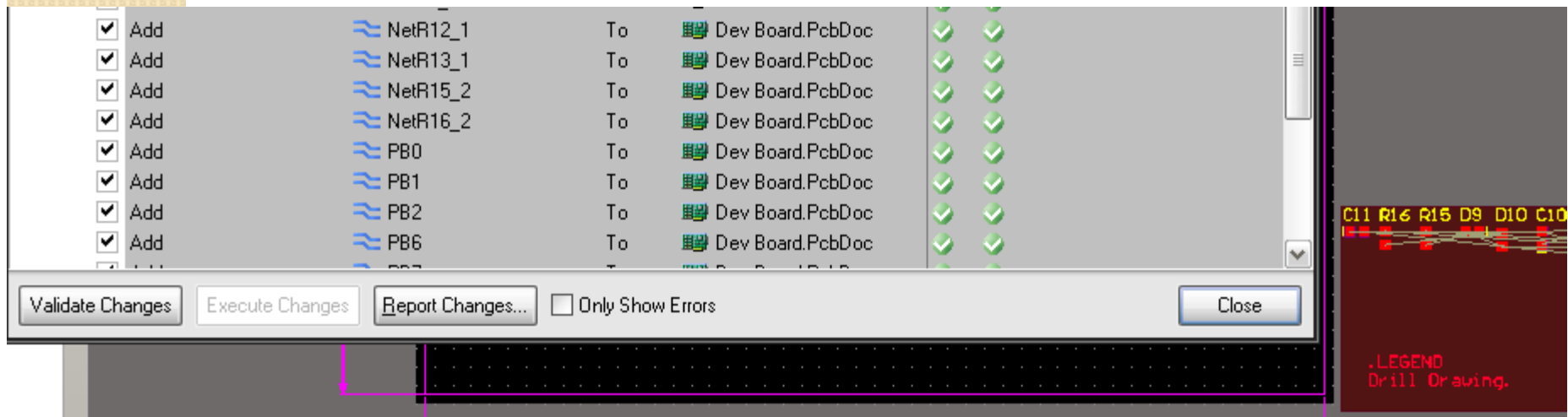


Creating the PCB

- When no errors appear during the validation step, you can click the 'execute changes' to actually import the schematic to the PCB



- You will see a second row of green check boxes when you execute the changes. Again, check for errors by checking the 'Only show errors'



Creating the PCB

- NOTE: Every time you perform an ECO, the results are saved in a text file in your project directory

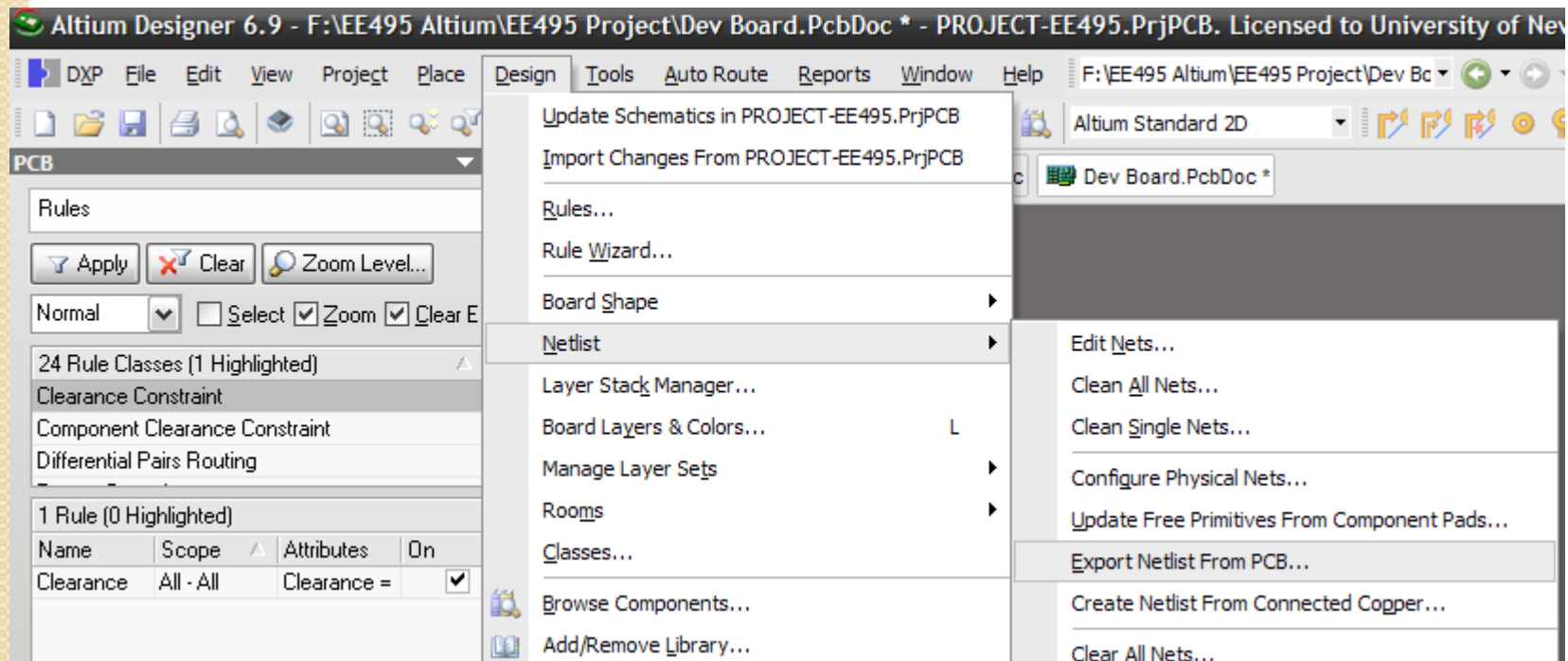
Dev Board PCB ECO 6-1-2008 2-35-19 PM.LOG - Notepad

File Edit Format View Help

```
Added Component : Designator=C1(CC2012-0805)
Added Component : Designator=C2(CC2012-0805)
Added Component : Designator=C3(CC2012-0805)
Added Component : Designator=C4(CC2012-0805)
Added Component : Designator=C5(CC2012-0805)
Added Component : Designator=C6(CC2012-0805)
Added Component : Designator=C7(CC2012-0805)
Added Component : Designator=C8(CC2012-0805)
Added Component : Designator=C9(CC2012-0805)
Added Component : Designator=C10(CC2012-0805)
Added Component : Designator=C11(CC2012-0805)
Added Component : Designator=D1(CD2012-0805)
Added Component : Designator=D2(CD2012-0805)
Added Component : Designator=D3(CD2012-0805)
Added Component : Designator=D4(CD2012-0805)
Added Component : Designator=D5(CD2012-0805)
Added Component : Designator=D6(CD2012-0805)
Added Component : Designator=D7(CD2012-0805)
```

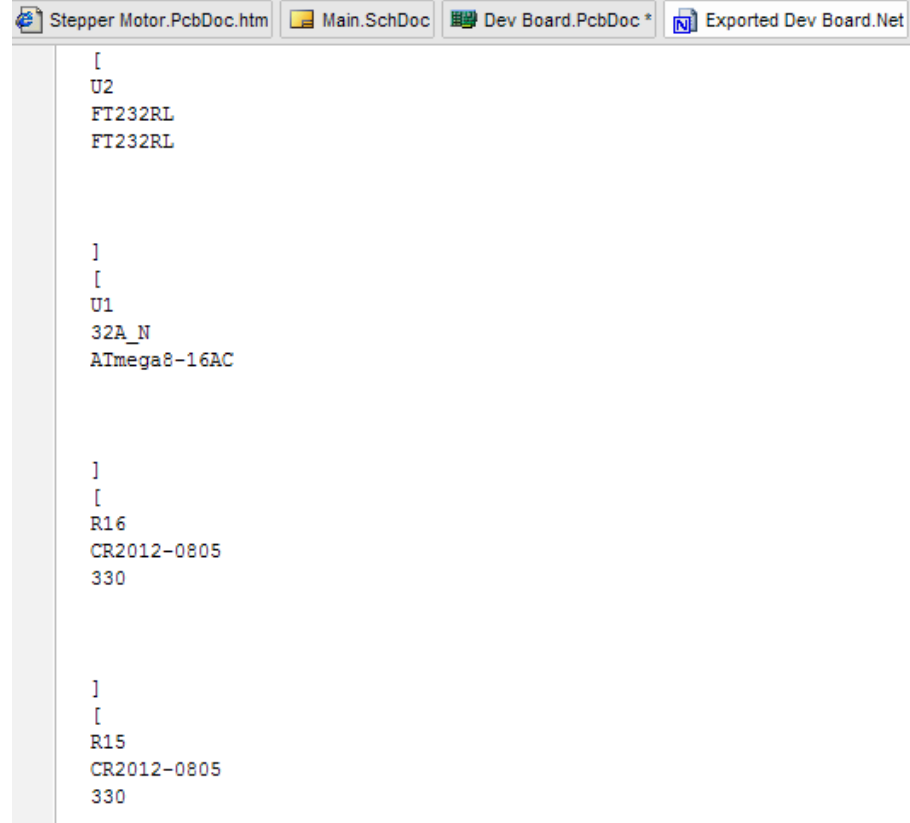
Creating the PCB

- Export the NETLIST following the menu sequence below (You must be on the PCB document, not the schematic document)
- This will export the Netlist to your project directory



Creating the PCB

- The Netlist contains information such as parts, footprint, and value (as shown below)



The screenshot shows a software window with four tabs: 'Stepper Motor.PcbDoc.htm', 'Main.SchDoc', 'Dev Board.PcbDoc *', and 'Exported Dev Board.Net'. The active tab displays a netlist with the following content:

```
[
U2
FT232RL
FT232RL

]
[
U1
32A_N
ATmega8-16AC

]
[
R16
CR2012-0805
330

]
[
R15
CR2012-0805
330
```

Creating the PCB

- It also contains connection information as shown to the right
- Each Netlabel (either assigned by you or the software, every net has a netlabel) shows which component and pin number connects to that node
- Obviously a connection like GND will have many connections, and netlabels such as MISO might only have a few (3 in this case)

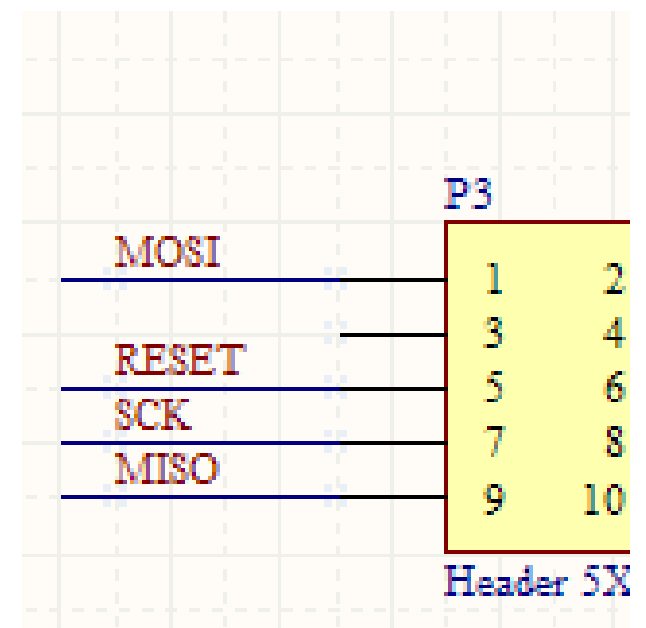
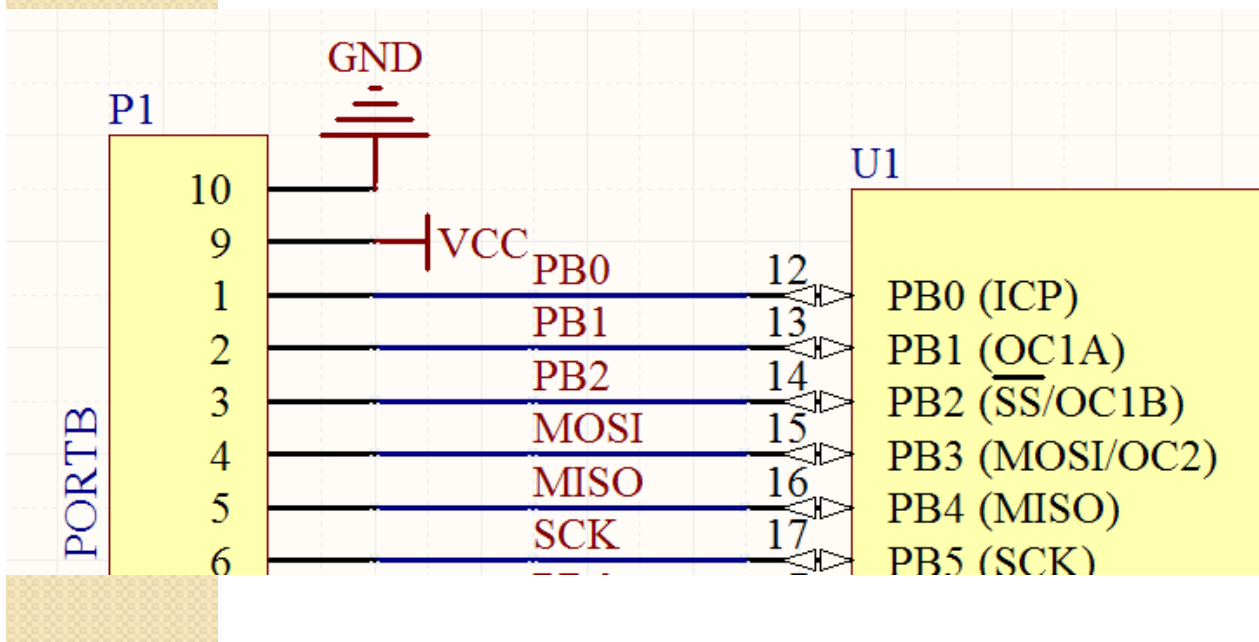
```
Stepper Motor.PcbDoc.htm
(
MISO
P3-9
P1-5
U1-16
)
(
INT1
P4-4
U1-1
)
(
INT0
P4-3
U1-32
)
(
GND
J1-4
C10-1
C11-1
U2-7
U2-18
U2-21
U2-25
U2-26
D3-2
C7-2
D2-2
R14-2
D1-2
C8-2
C9-2
D5-2
D4-2

```

Creating the PCB

- Netlist Connections for Net MISO:
 - P3 pin 9
 - P1 pin 5
 - U1 pin 16
- A quick look at the schematic shows that this net is properly connected

```
(
MISO
P3-9
P1-5
U1-16
)
```



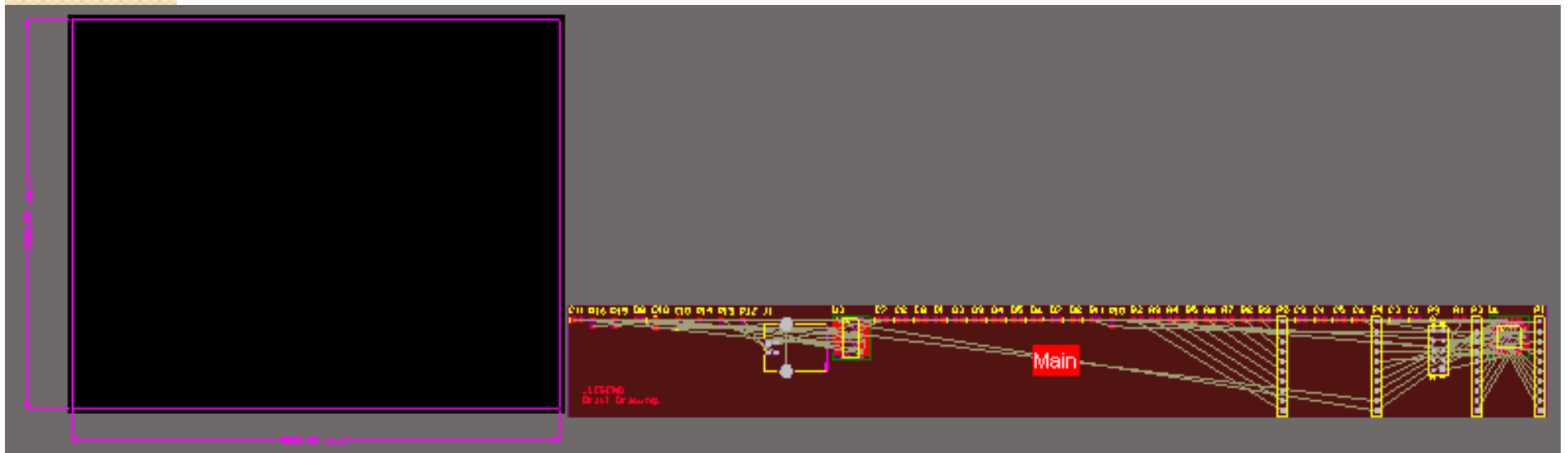


Creating the Netlist

- Fortunately, Altium handles the Netlist transparently to the end user, but it is important to at least be familiar with the term and how the schematic is transferred to the PCB and vice versa
- Any changes that are made must be made to the Netlist.
- The Netlist is the 'link' between the PCB and the schematic. It is important that the schematic and PCB agree on connections, nets, footprints, etc..

Creating the PCB

- After executing changes. If there are errors or not, whatever the software can import into the PCB it will do so
- Notice how the software just sort of ‘dumps’ all your parts (PCB Footprints) off to the side of the board.
- **Does anyone know why the software doesn't place the parts ???**



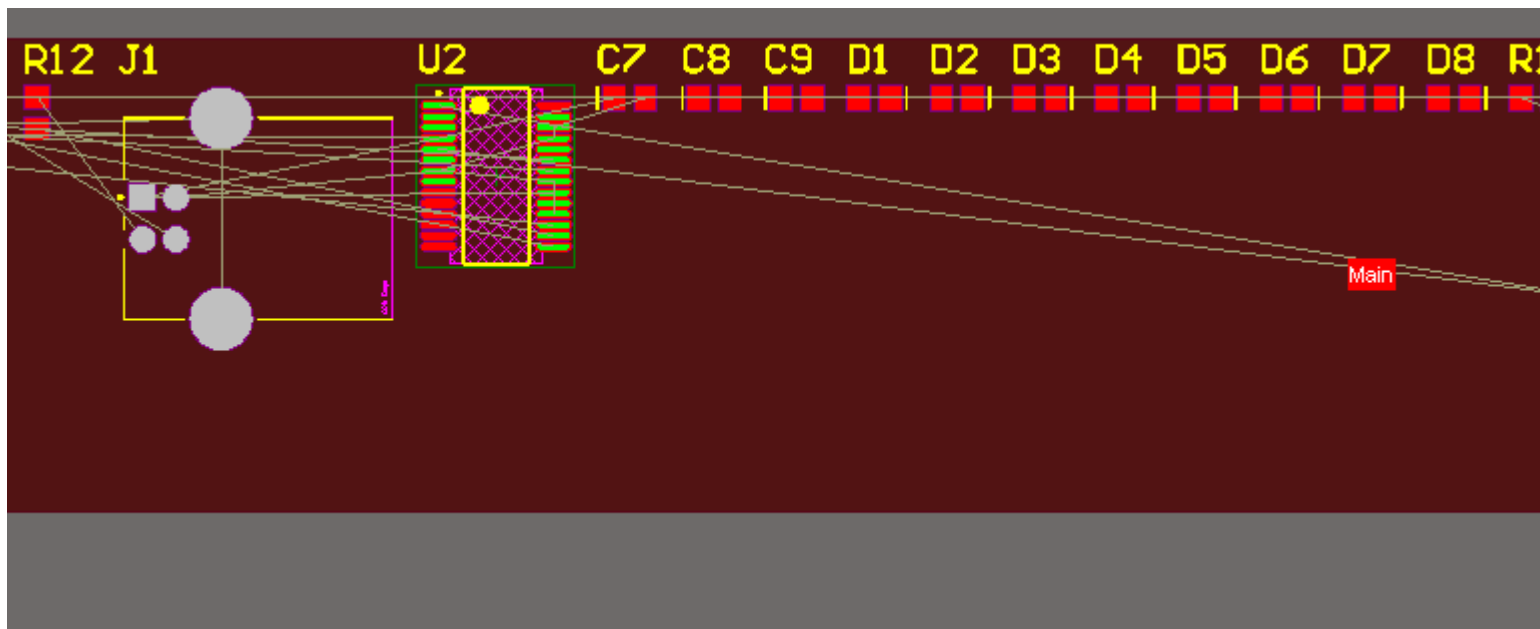
Creating the PCB

- You may have noticed the words on the right on your design. Leave it where it is and don't mess with it.
- We will get to that later what it does
- This is called a **Special String**

**.LEGEND
Drill Drawing.**

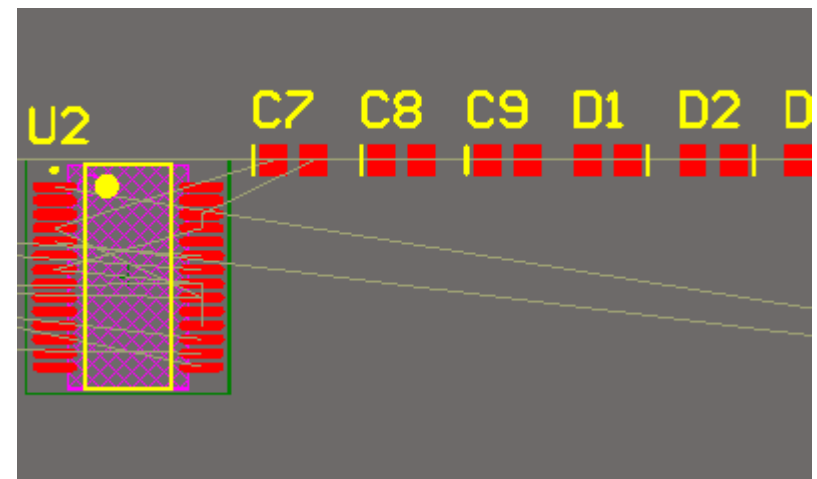
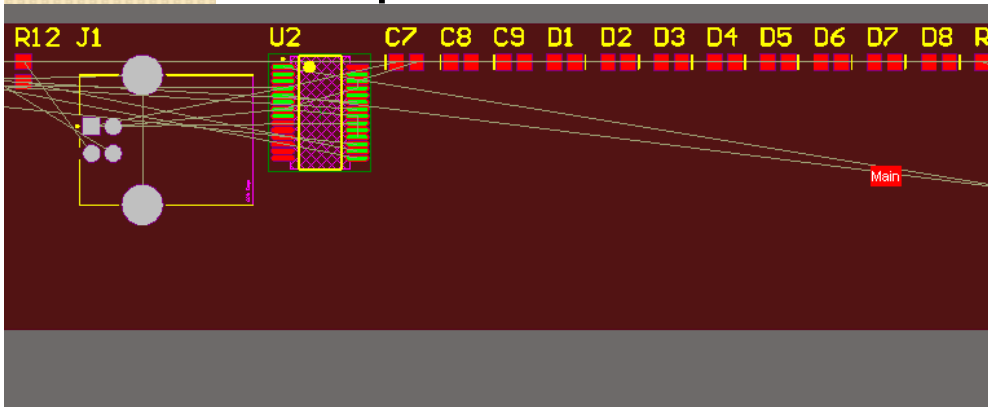
Creating the PCB

- Notice how all the parts are placed inside a colored rectangle box called 'Main'. This is called a ROOM.
- Rooms are used to group common circuit types. A room is created for each schematic page we have
- Since we only have one page, called 'MAIN', there is only one room



Creating the PCB

- Rooms are like borders for your parts. An item inside a room cannot escape without an error message.
- Since we only have one room, we can delete the room without affecting our layout at all.
- Click anywhere in the room and hit the DEL key
- The room will disappear and the components will remain
- NOTE: You can disable the use of rooms in project options



Creating the PCB

- If you hit the 'L' key on the PCB page, it will bring up the following prompt.
- Change the option under mechanical layers that says 'Used Layers Only'

The screenshot shows the 'View Configurations' dialog box in Altium Designer. The 'Select PCB View Configuration' section on the left lists various configurations, with 'Altium Standard 2D' selected. The main area is divided into several sections for configuring layer visibility and colors.

Board Layers And Colors

Signal Layers (S)	Color	Show
Top Layer (T)	Red	<input checked="" type="checkbox"/>
Bottom Layer (B)	Blue	<input checked="" type="checkbox"/>

Internal Planes (P)

Color	Show
	<input type="checkbox"/>

Mechanical Layers (M)

Mechanical Layers (M)	Color	Show	Enable	Single Layer Mode	Linked To Sheet
Mechanical 1	Magenta	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 2	Purple	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 3	Green	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 4	Olive	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 5	Pink	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 6	Purple	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 7	Green	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 8	Olive	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 9	Pink	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 10	Purple	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>
Mechanical 11	Green	<input type="checkbox"/>	<input checked="" type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>

Only show layers in layer stack Only show planes in layer stack Only show enabled mechanical Layers

Mask Layers (A)

Color	Show
Top Paste	<input type="checkbox"/>
Bottom Paste	<input type="checkbox"/>
Top Solder	<input type="checkbox"/>
Bottom Solder	<input type="checkbox"/>

Other Layers (O)

Color	Show
Drill Guide	<input type="checkbox"/>
Keep-Out Layer	<input checked="" type="checkbox"/>
Drill Drawing	<input checked="" type="checkbox"/>
Multi-Layer	<input checked="" type="checkbox"/>

Silkscreen Layers (K)

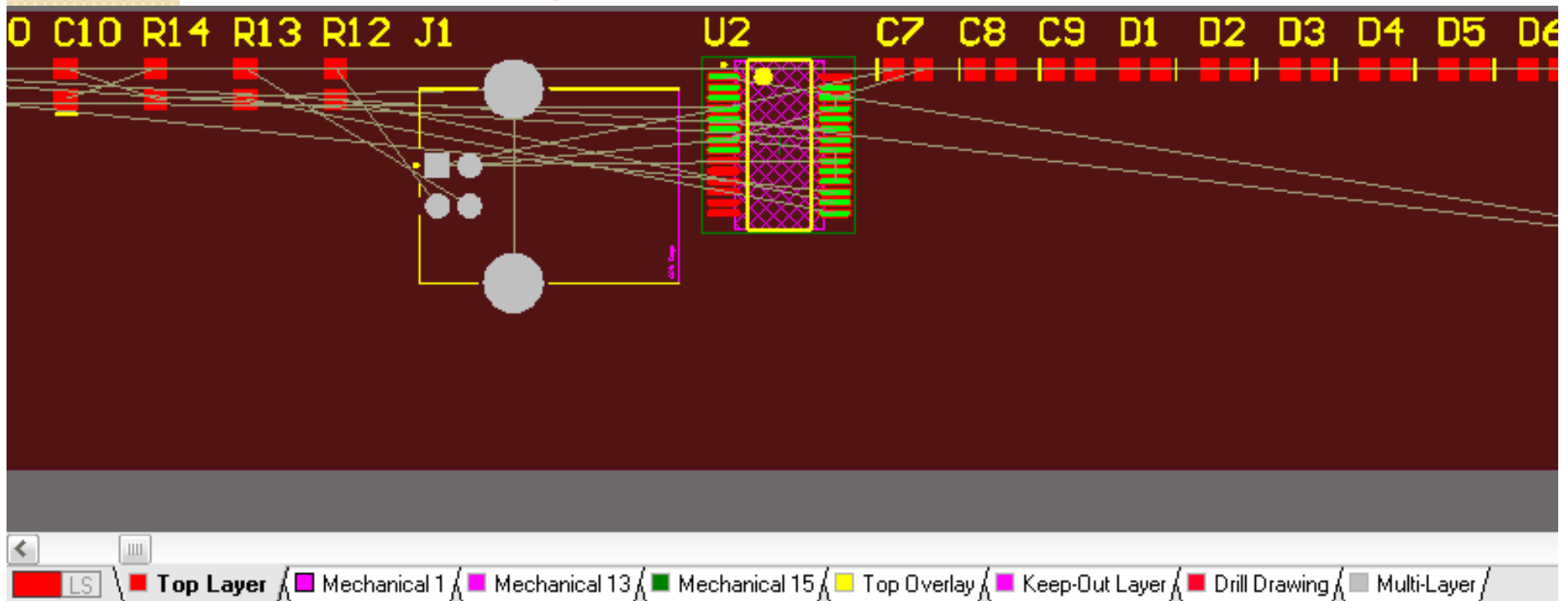
Color	Show
Top Overlay (E)	<input checked="" type="checkbox"/>
Bottom Overlay (R)	<input checked="" type="checkbox"/>

System Colors (Y)

Color	Show
Connections and From Tos	<input checked="" type="checkbox"/>
DRC Error Markers	<input checked="" type="checkbox"/>
Selections	<input type="checkbox"/>
Visible Grid 1	<input checked="" type="checkbox"/>
Visible Grid 2	<input checked="" type="checkbox"/>
Pad Holes	<input type="checkbox"/>
Via Holes	<input type="checkbox"/>
Highlight Color	<input type="checkbox"/>
Board Line Color	<input type="checkbox"/>
Board Area Color	<input type="checkbox"/>
Sheet Line Color	<input type="checkbox"/>
Sheet Area Color	
Workspace Start Color	<input type="checkbox"/>
Workspace End Color	<input type="checkbox"/>

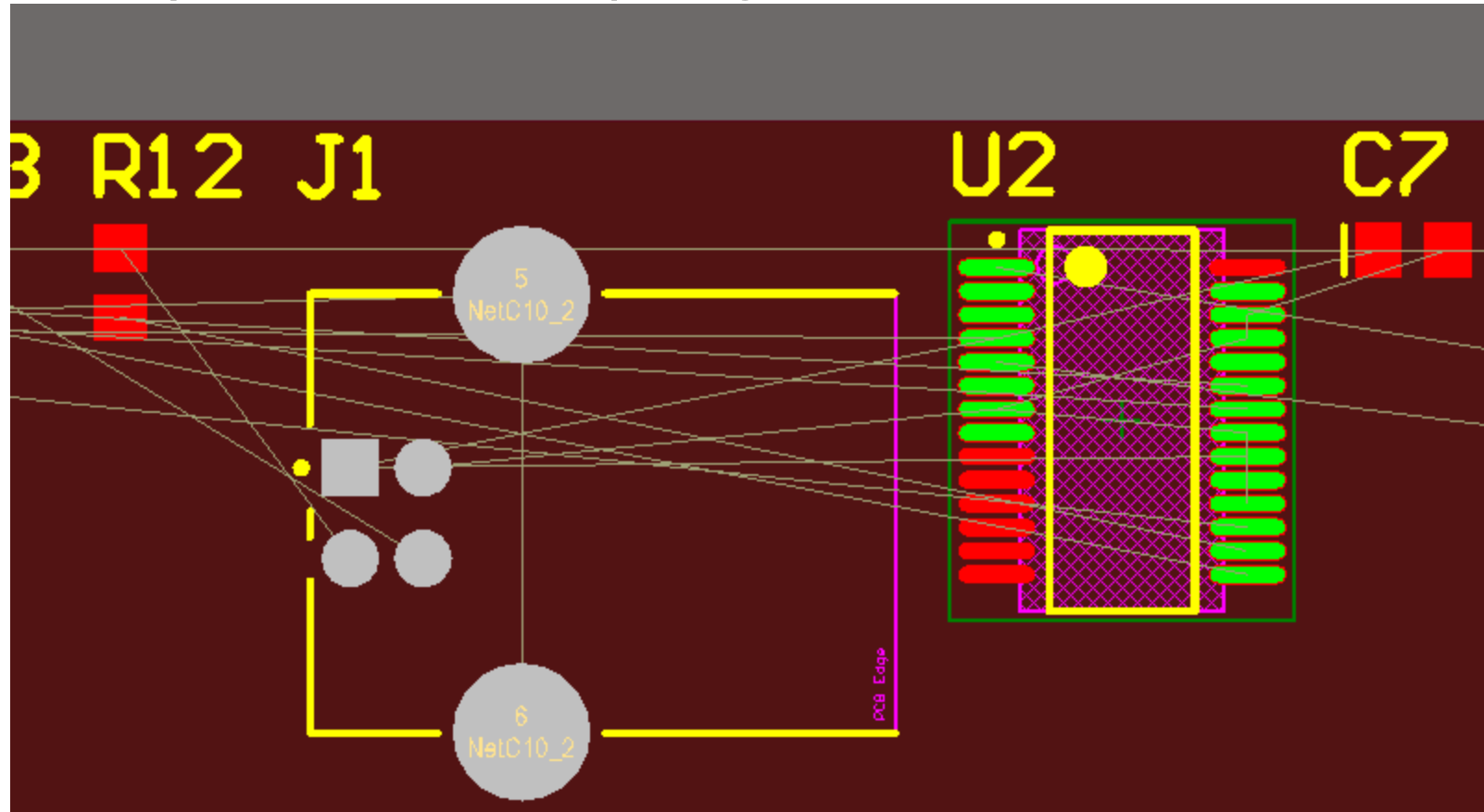
Creating the PCB

- Notice the different tabs at the bottom of the screen. The selected layer will look different than the others (in this case, top layer)
- Unlike a schematic, which is 2D, a PCB is a 3D entity with a whole bunch of layers on top of each other. These layers are all combined to form your PCB.
- **NOTE: These layers are all color coded!!!!**














Creating the PCB

- If we zoom in, notice that U2 (The FT232 chip) has several green pads, and several red pads.
- The red pads represent something on the top layer. But why are some of the pads green ??



Creating the PCB

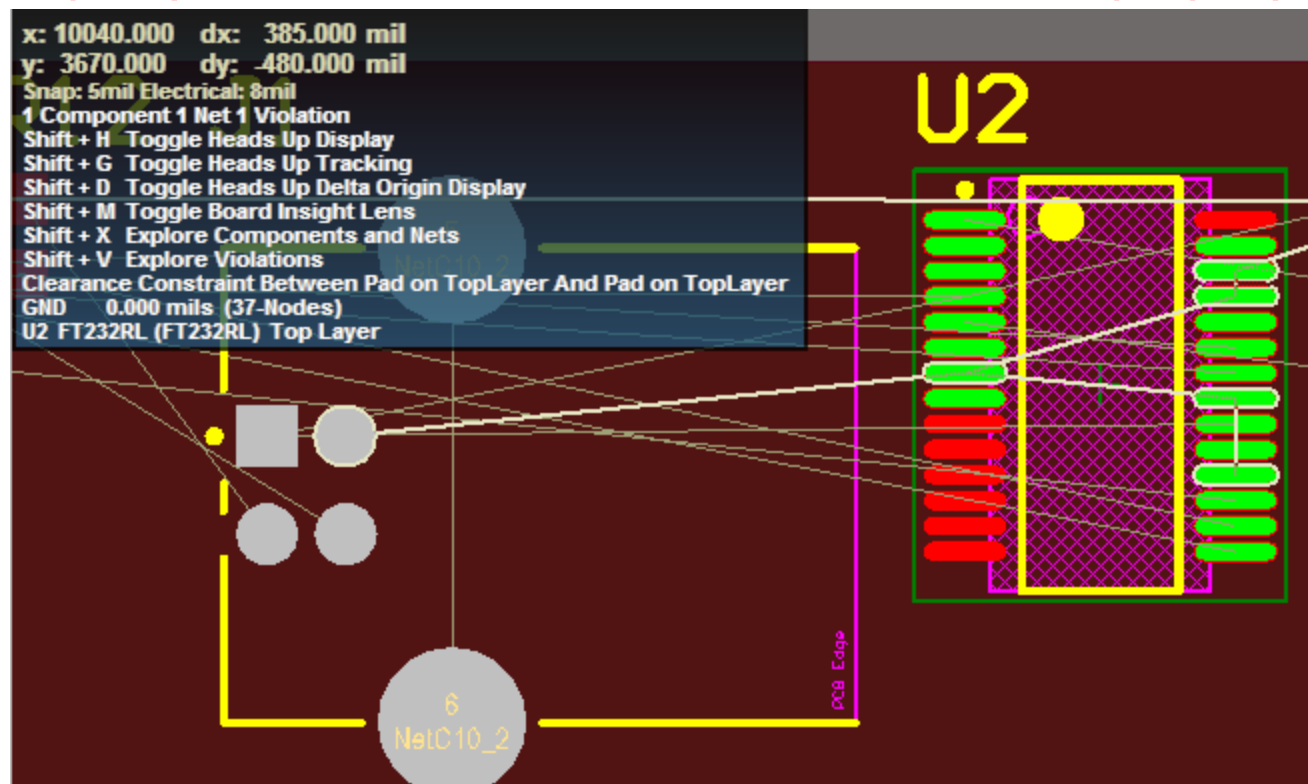
- If we type 'L' again, it will bring up the board layer lists.
- Since the layers are color coded, we can quickly look for the green one and notice that this is a DRC error marker (Design Rule Check)
- For some reason Altium has a problem with our FT232 Chip
- Check the boxes for pad holes and via holes

System Colors (Y)	Color	Show
Connections and From Tos		<input checked="" type="checkbox"/>
DRC Error Markers		<input checked="" type="checkbox"/>
Selections		
Visible Grid 1		<input checked="" type="checkbox"/>
Visible Grid 2		<input checked="" type="checkbox"/>
Pad Holes		<input type="checkbox"/>
Via Holes		<input type="checkbox"/>
Highlight Color		
Board Line Color		
Board Area Color		
Sheet Line Color		
Sheet Area Color		
Workspace Start Color		
Workspace End Color		

All On All Off Used On

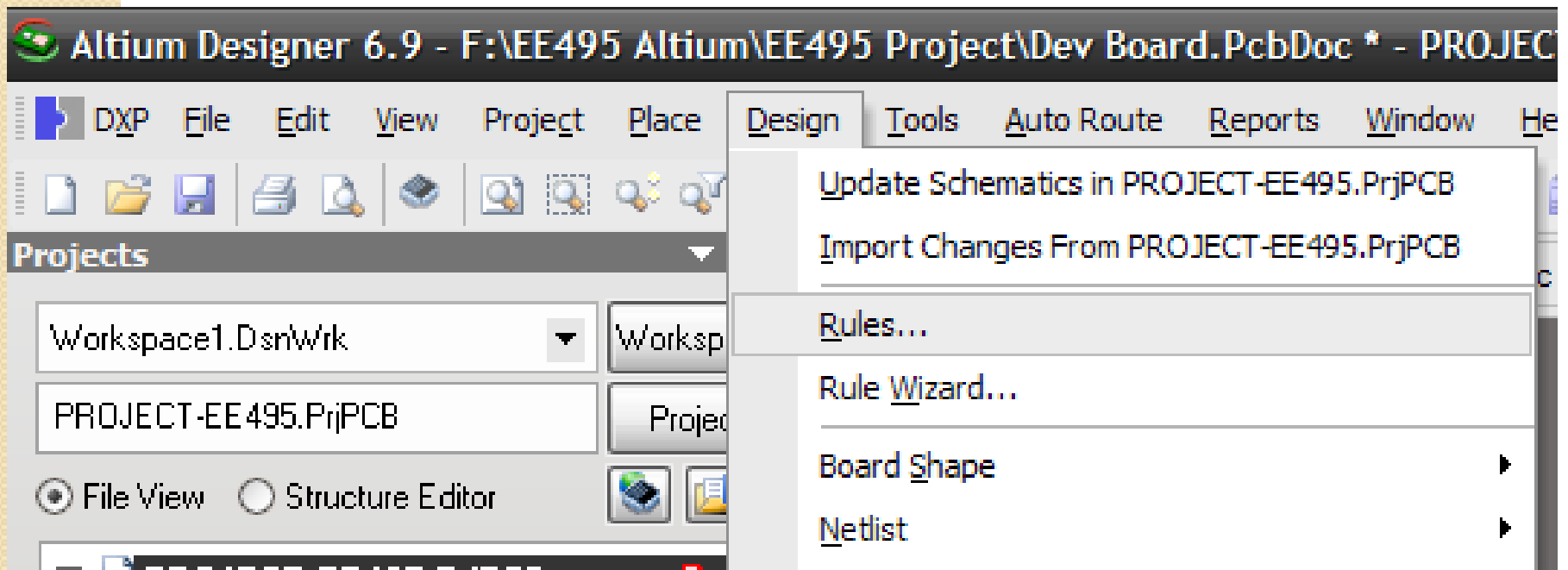
Creating the PCB

- If we put the mouse over one of the green pads, it will tell us information about the pad, most importantly it will tell us what the error is
- For these pads, there is a clearance constraint error between a pad on the top layer, and another pad on the top layer
- **NOTE:** It is up to you to decide if this is a real error, or an improperly set rule



Creating the PCB

- Like the schematic, there are rules we need to set.
- Click Design > Rules (or) Rules wizard to create/modify rules
- **NOTE:** The software only creates errors based on your rules. It is up to you to know if something is OK or not. If you tell the software that 2 mil traces are OK, the software will enforce that.



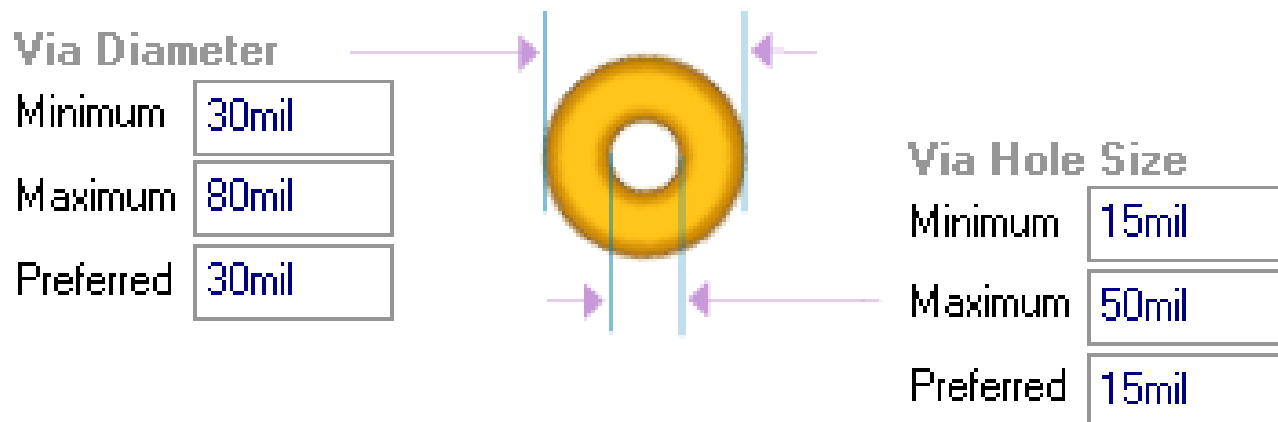
Creating the PCB

- The DRC expects a minimum spacing between different nets of 8 mils.

The screenshot displays the 'PCB Rules and Constraints Editor [mil]' interface. On the left, a tree view shows the hierarchy of Design Rules, with 'Clearance' selected under the 'Electrical' category. The main panel on the right is configured for a 'Clearance' rule. It includes a 'Name' field set to 'Clearance' and a 'Comment' field. Below this, there are two sections for defining object matches: 'Where the First object matches' and 'Where the Second object matches'. Both sections have radio buttons for 'All', 'Net', 'Net Class', 'Layer', 'Net and Layer', and 'Advanced (Query)'. The 'All' option is selected in both. To the right of these sections are 'Full Query' fields, both containing 'All'. At the bottom, a 'Constraints' section features a diagram of a yellow L-shaped trace and a circular pad. A blue arrow points to the gap between the trace and the pad, labeled 'Different Nets Only' and 'Minimum Clearance 8mil'. A purple arrow points to the circular pad.

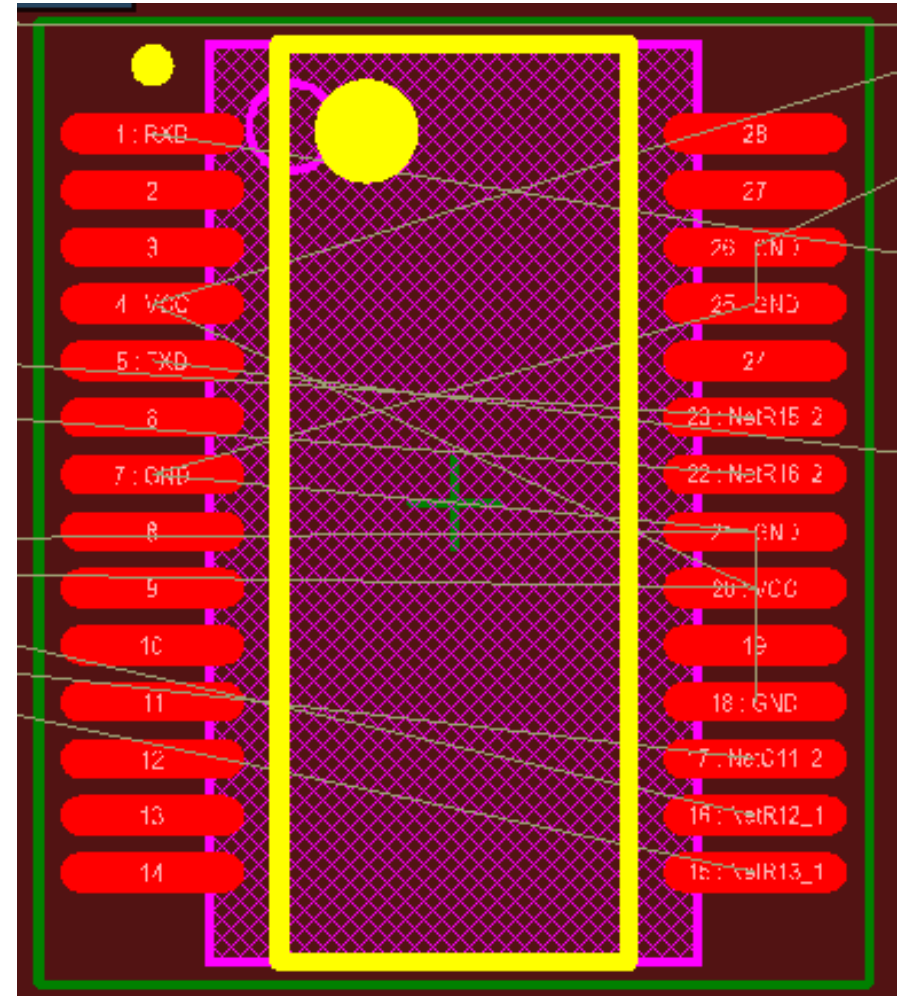
Creating the PCB

- Make the following rule changes (from top of list down):
 - Max Trace Width= 200 mils
 - Minimum Via Size = 15 mils, Min Via diameter = 30 mils
 - Maximum Via size = 50 mils, Max Via diameter = 80 mils
 - Minimum hole size = 15 mils, Max hole size = 200



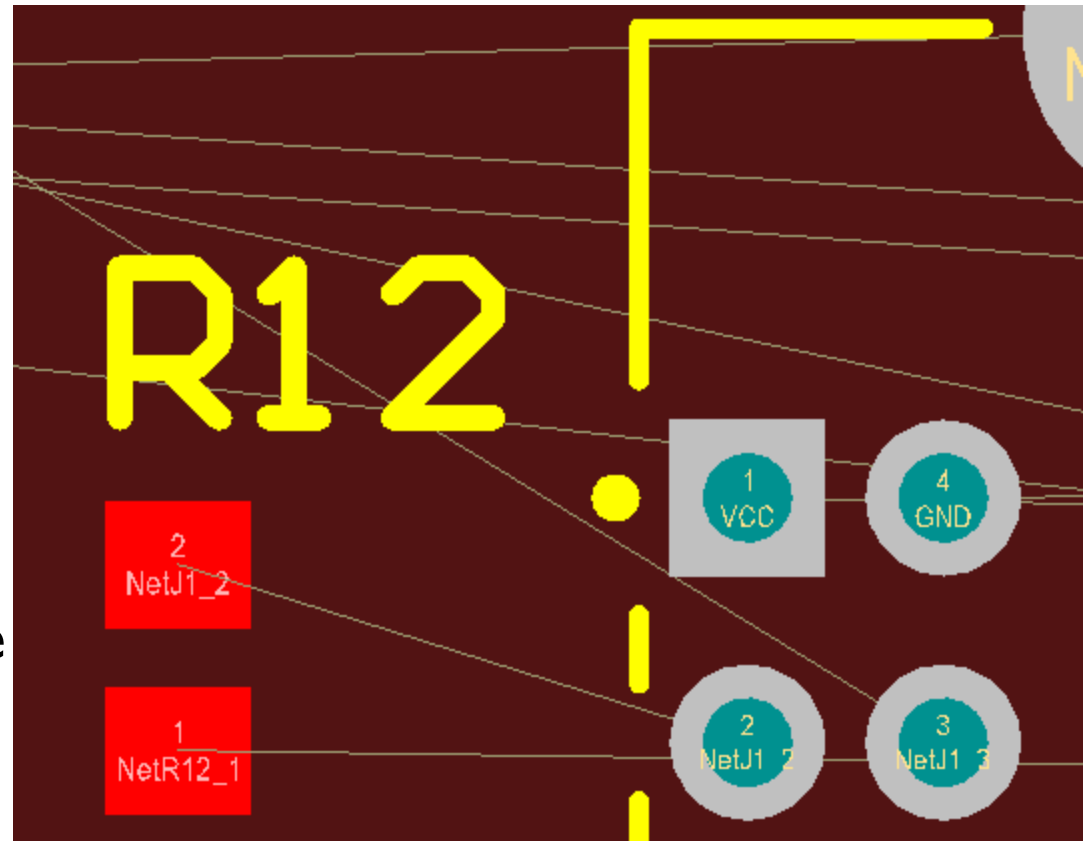
Creating the PCB

- Try using trial and error and change the setting to 7 mils and you'll notice the error markers go away.
- **NOTE:** There are many better ways of determining the actual spacing, but we want to keep things simple for now



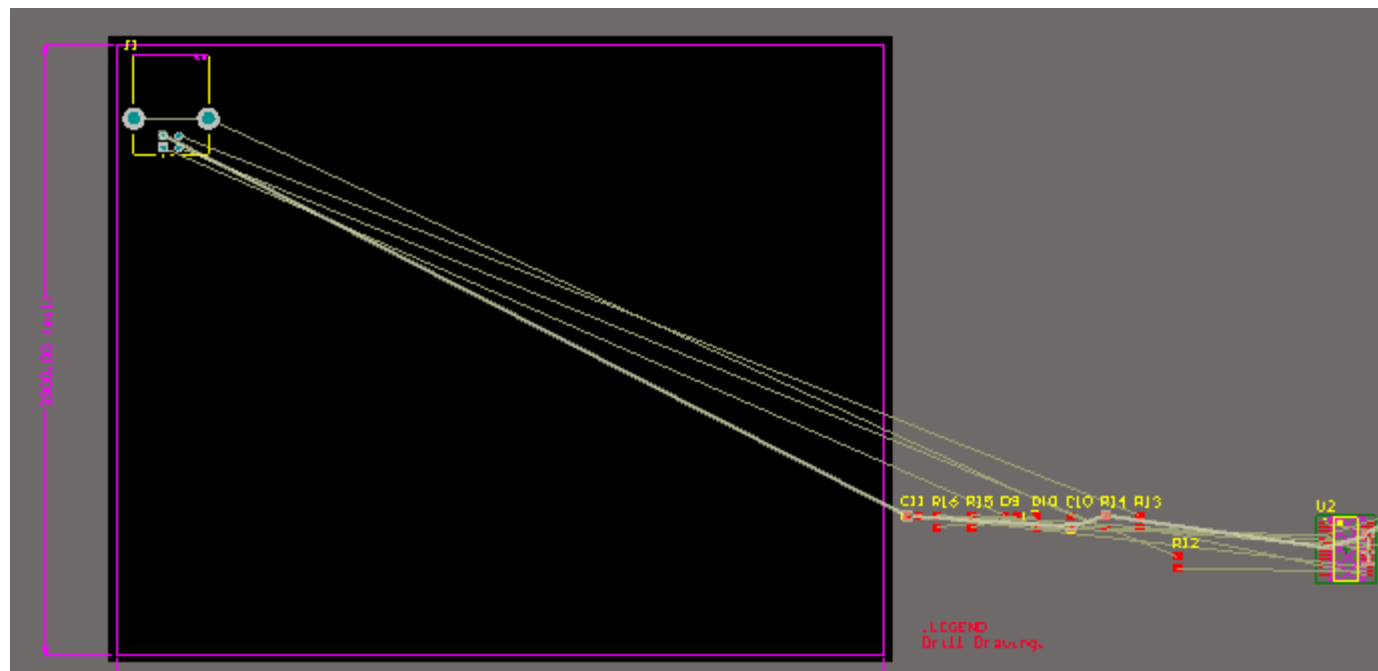
Creating the PCB

- Notice the small lines, these are collectively referred to as a **Ratsnest**.
- **It is a way of telling you which connections need to be made still**
- Example: You can see pin 2 of R12 needs to connect to Pin 2 of the USB connector



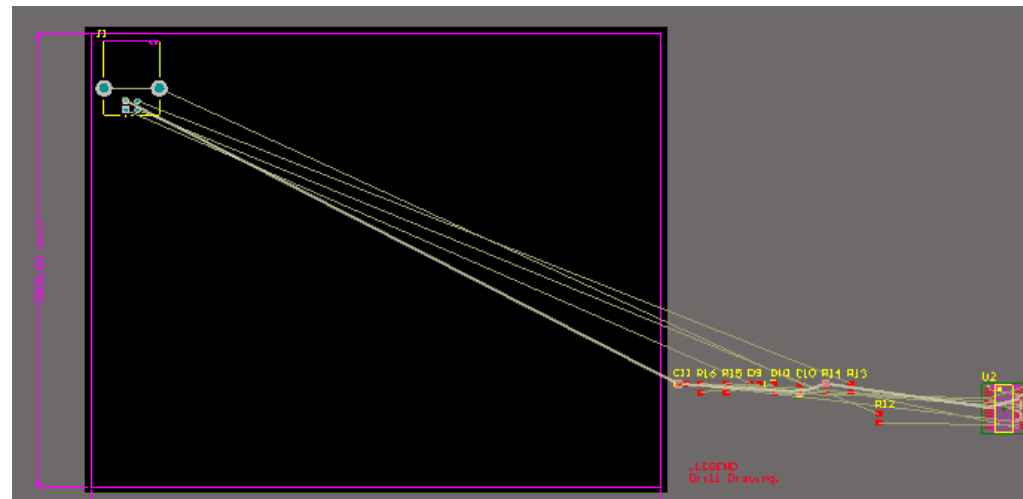
Creating the PCB

- Components are moved in the PCB the same way they are moved in the schematic.
- Notice how the Ratsnest connections remain and will dynamically change based on how a component is rotated.



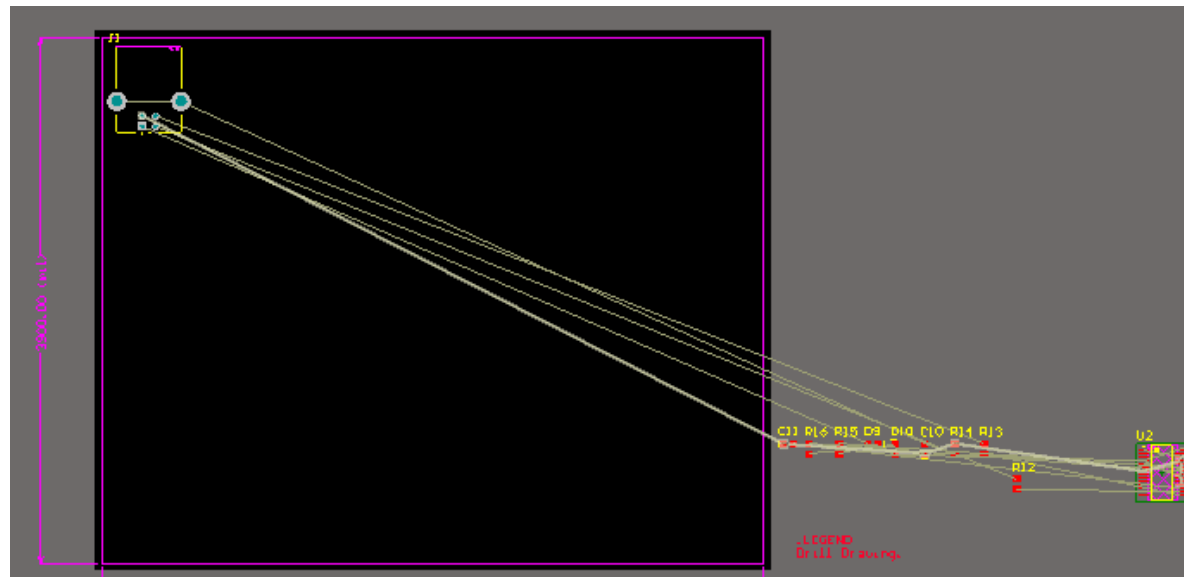
Creating the PCB

- There is usually a tradeoff when placing components. This USB connector might be easier to route if it is in the middle of the board, but will be difficult for a user to use a connector in the middle of the board.
- For this design, do the best you can to make it easier to route, for your project, it should make physical sense to an end user, even if it means harder routing



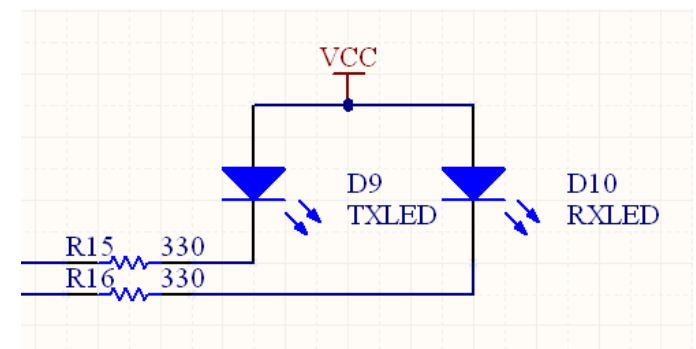
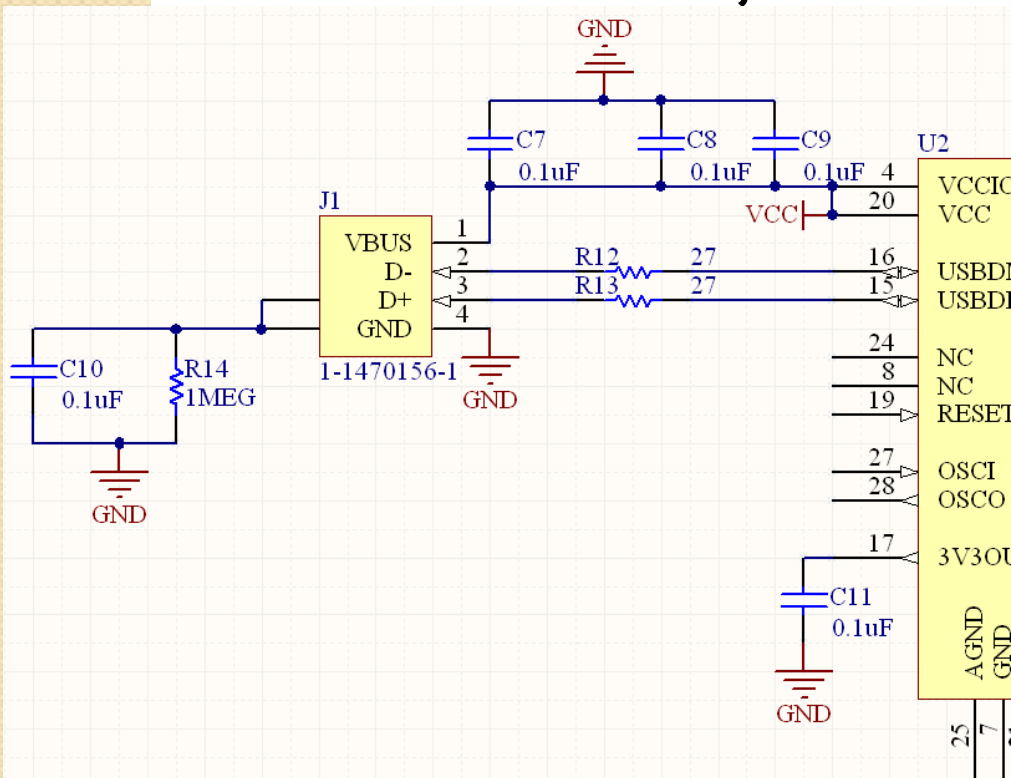
Creating the PCB

- EVERYONE'S PCB WILL BE DIFFERENT!!!
- Do not try to make yours exactly like mine or anyone else's.
- This design I went out of my way to create a more dense design for clarity and to resize the board. Feel free to use the entire board area.



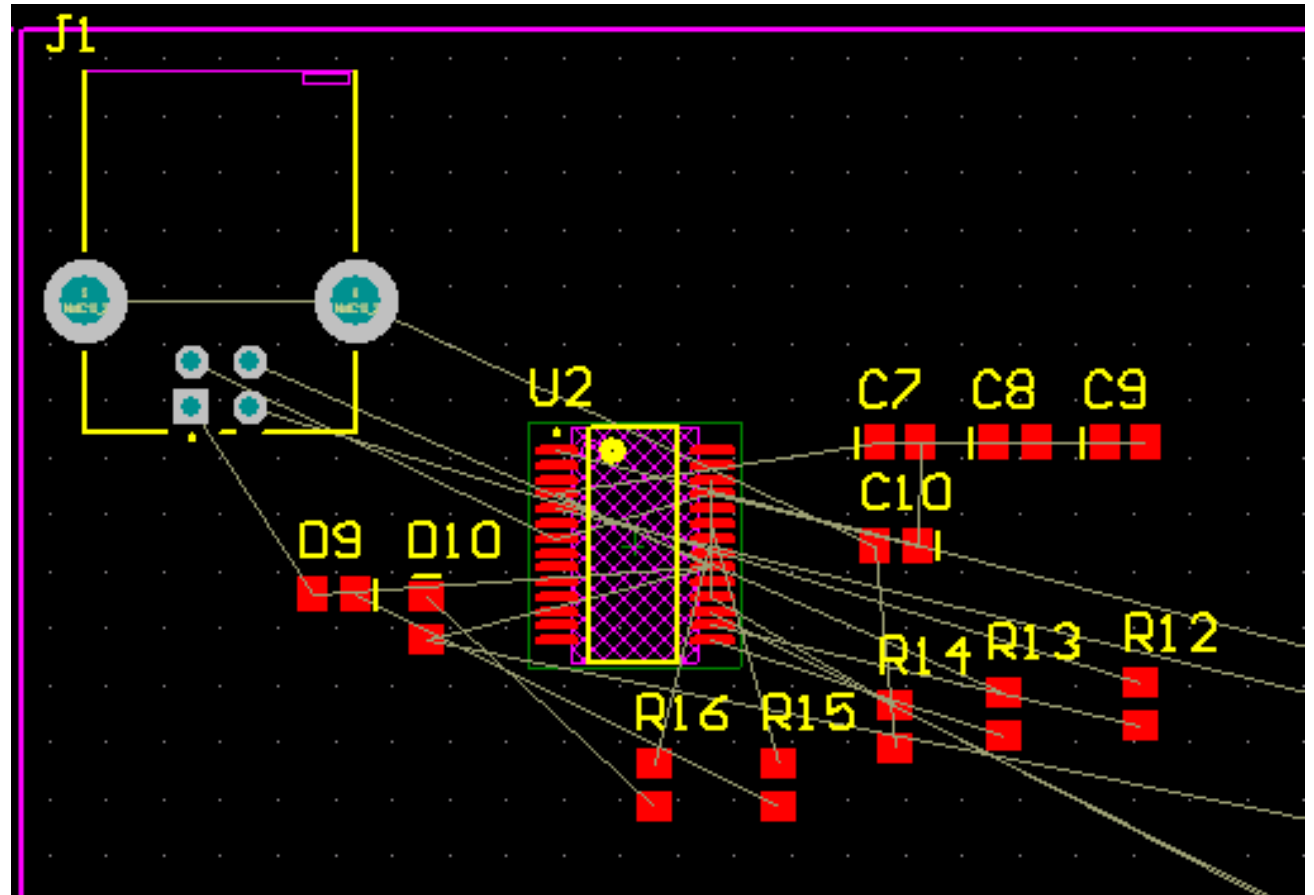
Creating the PCB

- Usually, components that are close together in the schematic are placed close together in the PCB.
- NOTE: If you have two monitors, keep the schematic open on one side, and the PCB on the other side. If you have one monitor, just write down the components



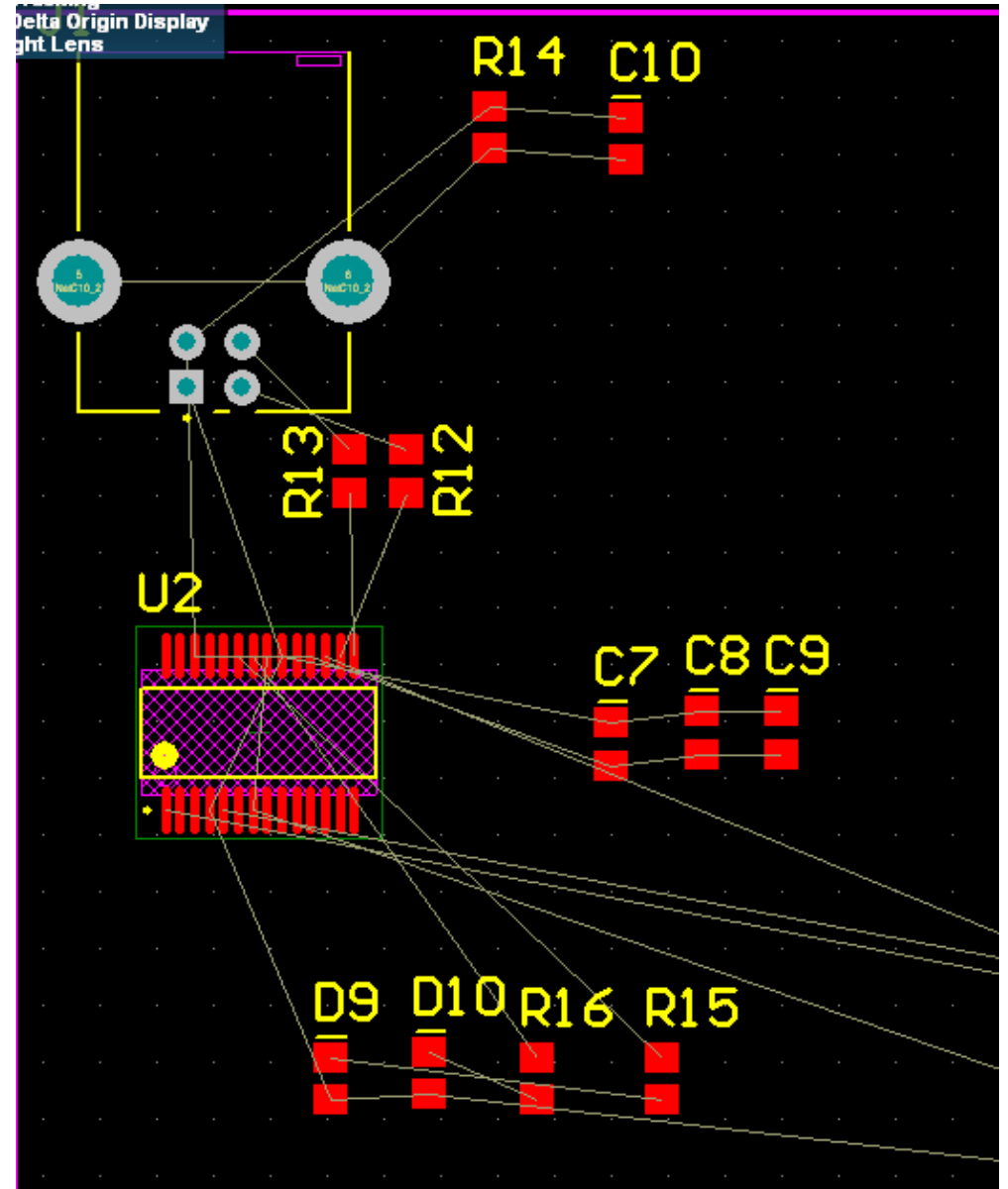
Creating the PCB

- For the USB circuit, we will group the following components
 - C7- C11
 - R12 – R16
 - U2
 - D9, D10



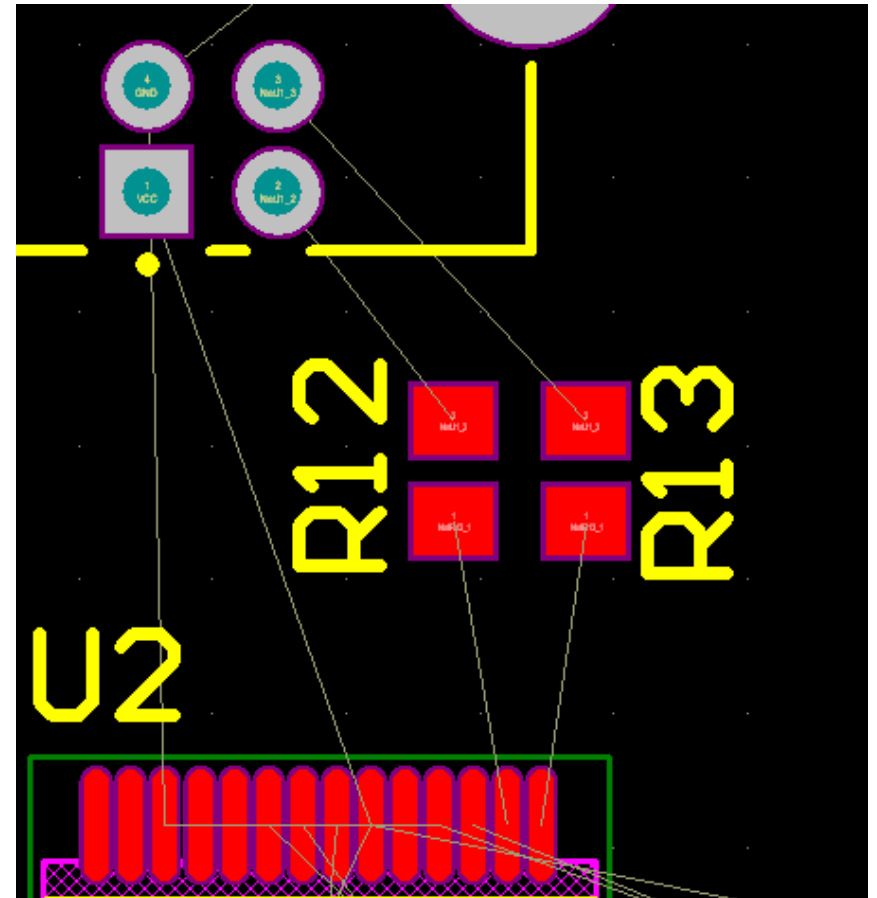
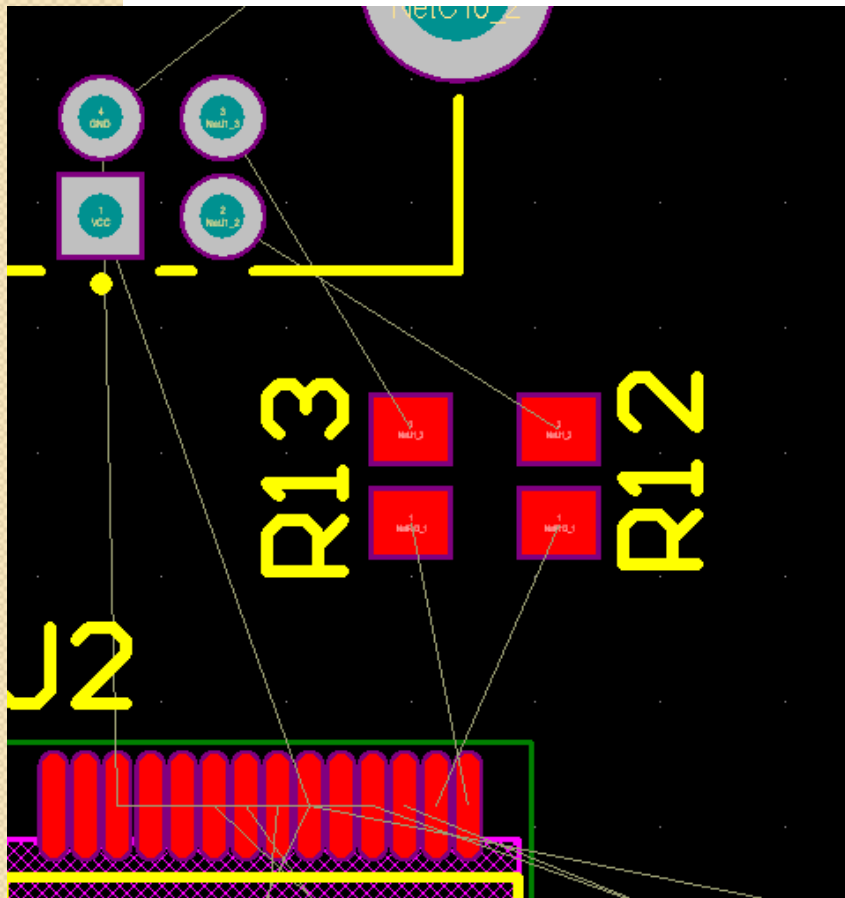
Creating the PCB

- Next group the components as they are arranged in the schematic.
- Prioritize the placement by organizing the high speed or critical components first at closest length



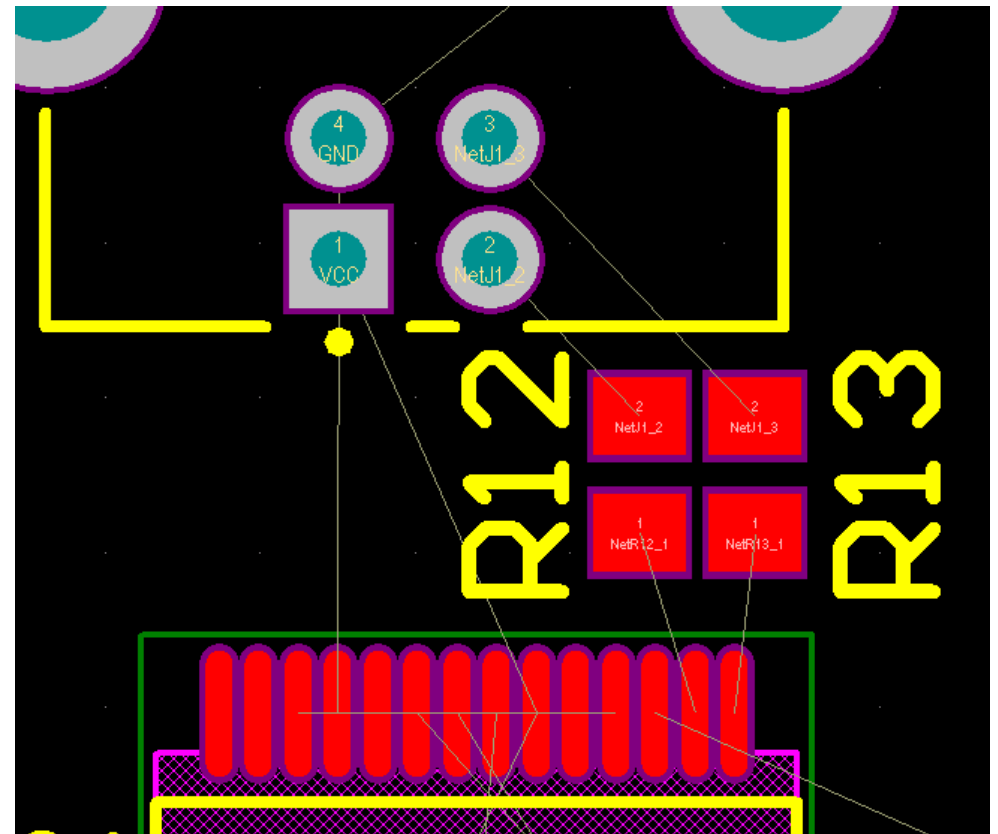
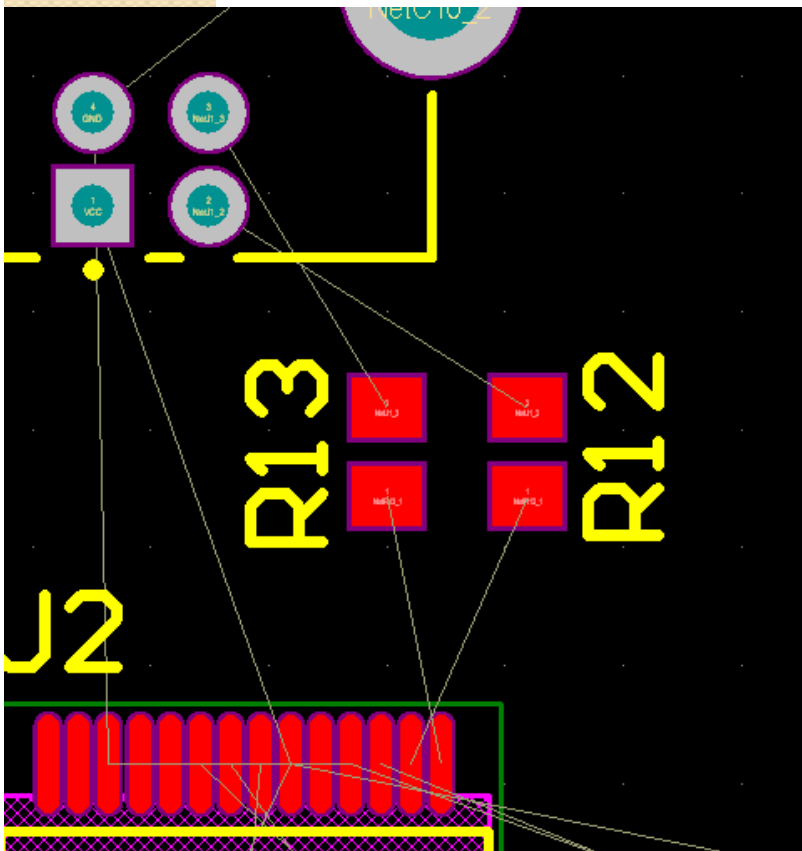
Creating the PCB

- Usually the more time you take in component placement, the less time you will need for routing
- Which placement is better and why ??? (L or R)



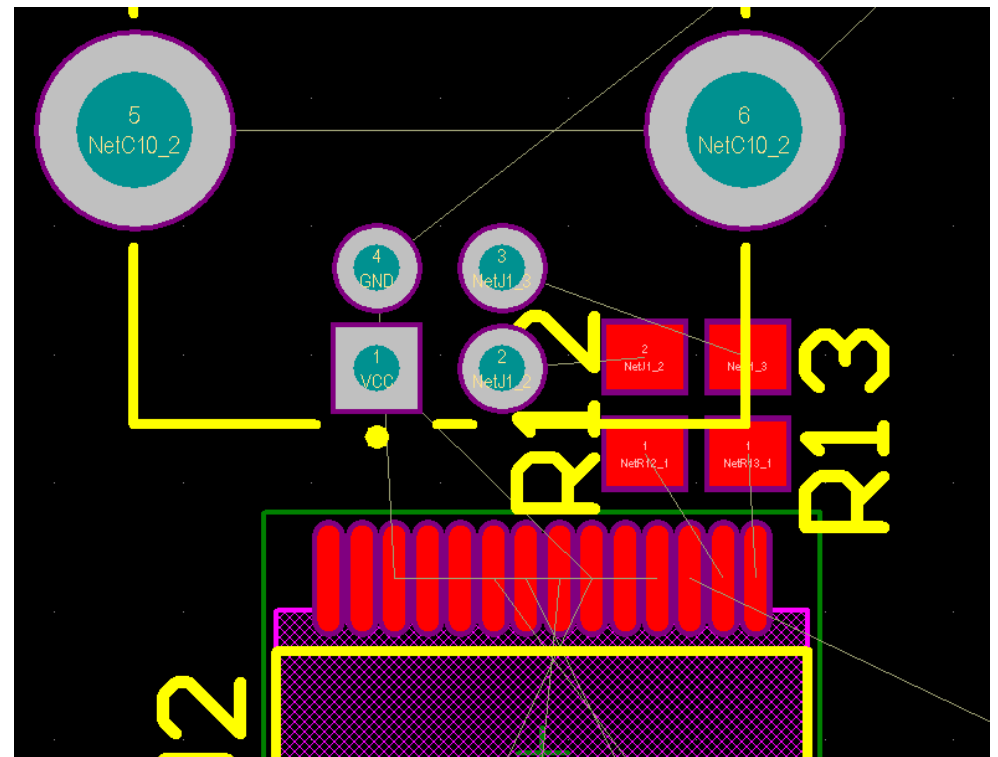
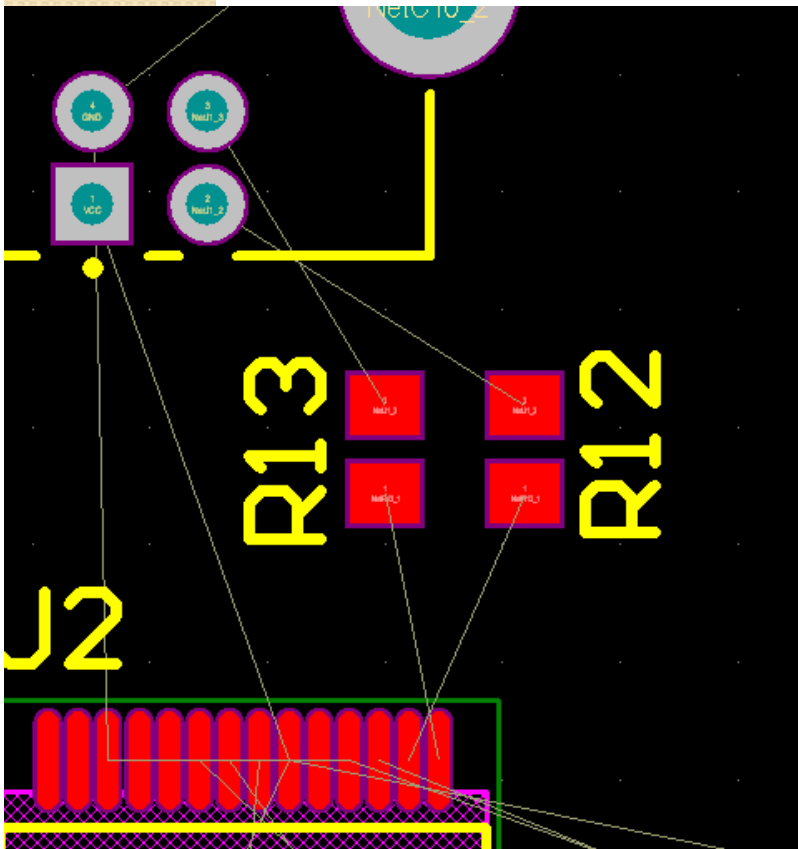
Creating the PCB

- Which placement is better and why ??? (L or R)



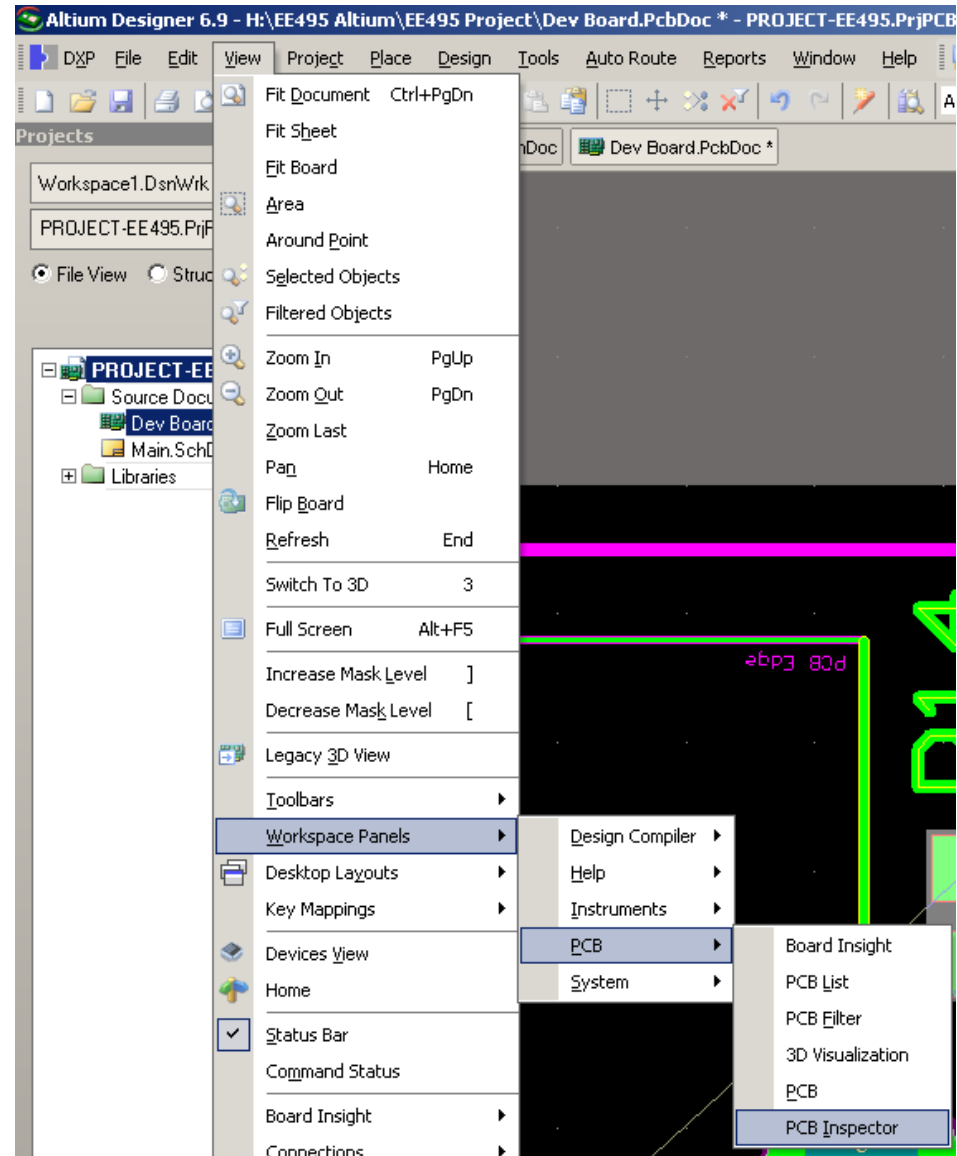
Creating the PCB

- Which placement is better and why ??? (L or R)



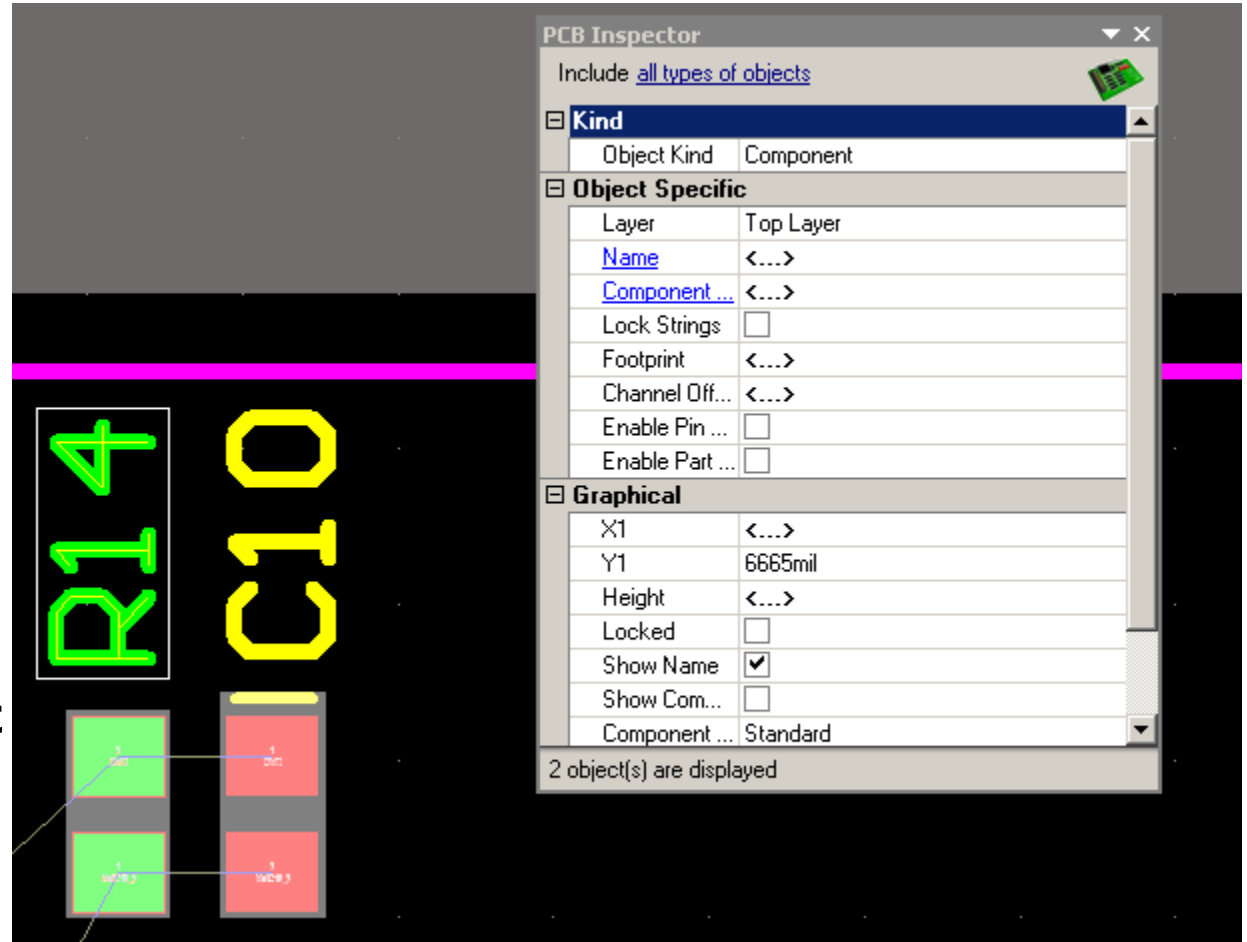
PCB Inspector

- Start the **PCB Inspector**.
- This is one of the most useful and time saving features in Altium



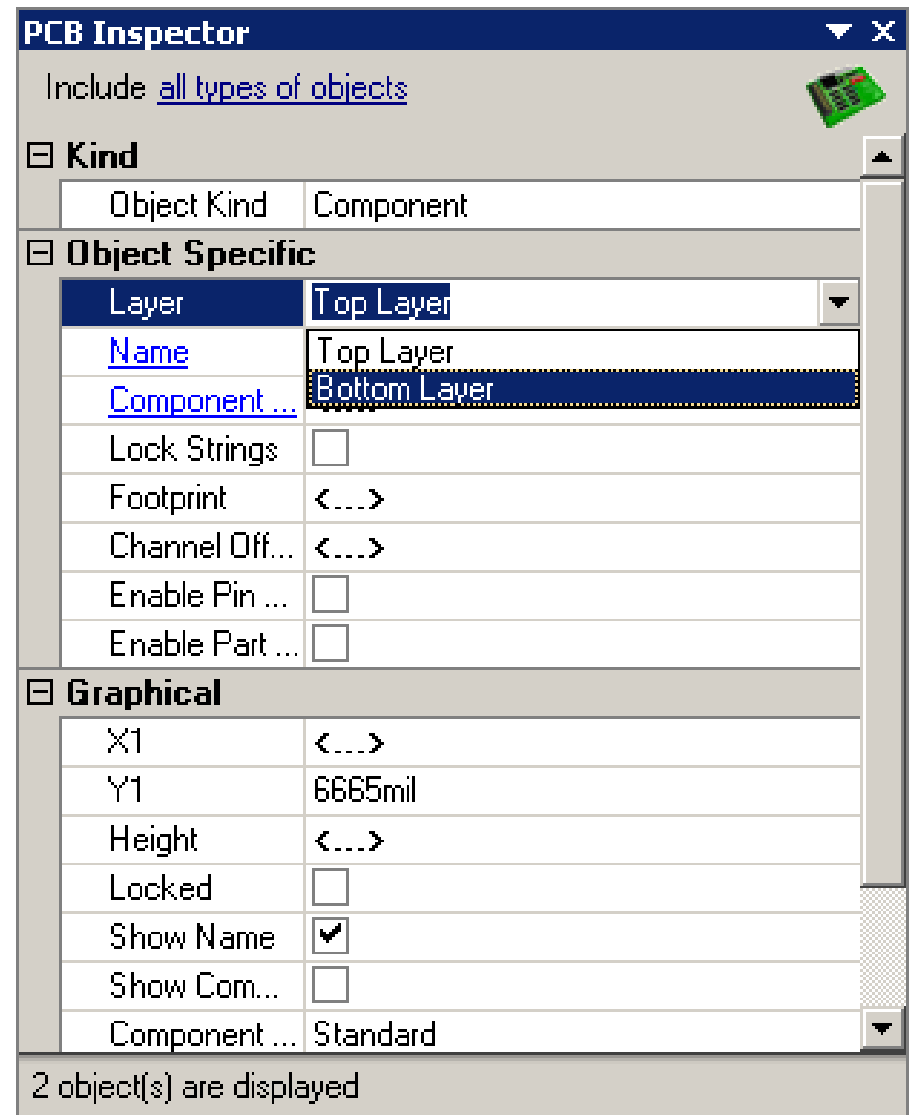
PCB Inspector

- Hold down control and click both R14 and C10
- The PCB inspector shows two objects are displayed/selected.



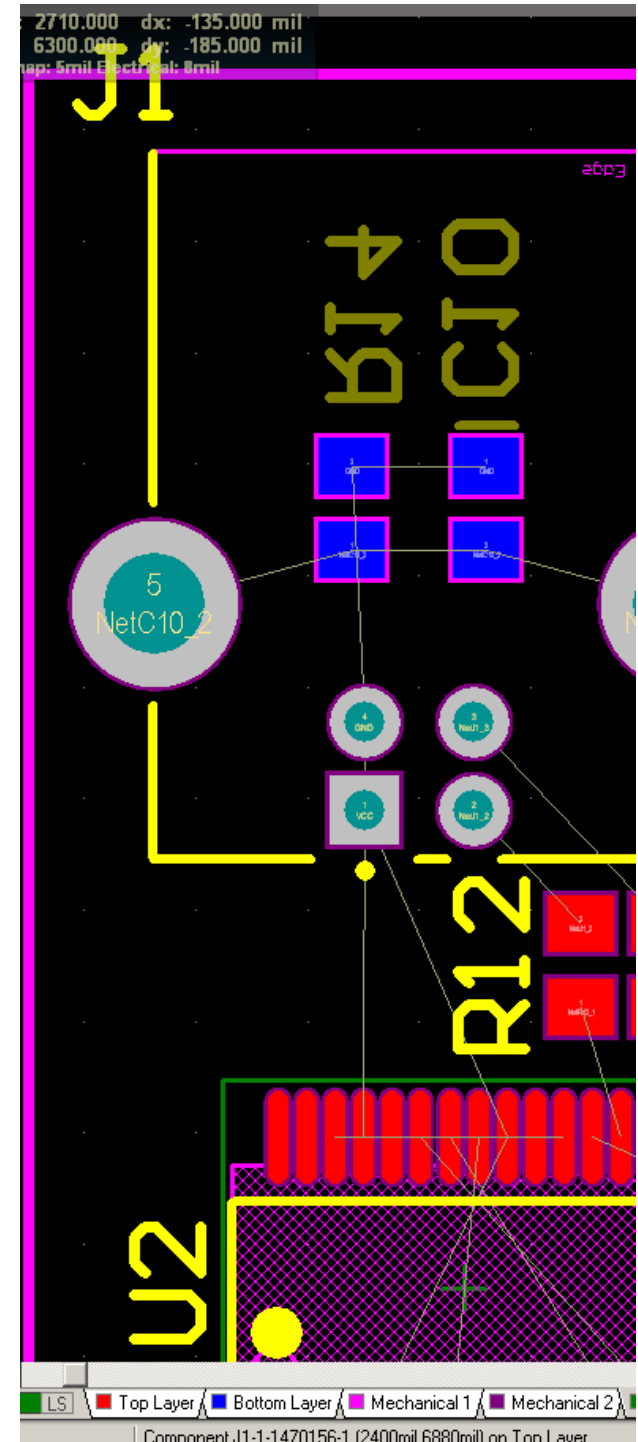
PCB Inspector

- Change the layer from TOP to BOTTOM, and all selected objects are immediately changed to the bottom layer
- This change moved the component pads from the top of the board to the bottom of the board and mirrored them



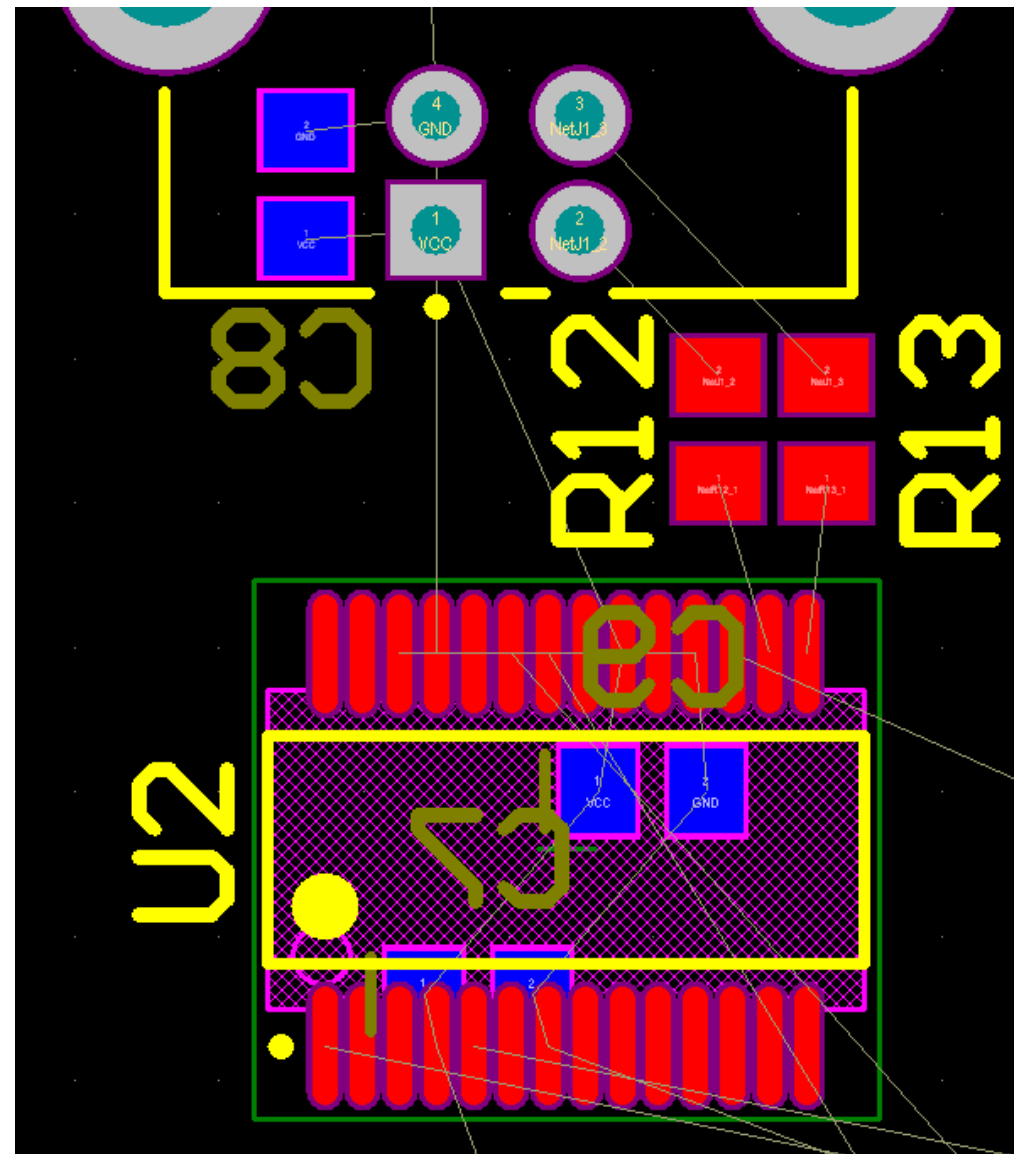
PCB Inspector

- We can now place R14 and C10 underneath the USB connector and make the connection from the bottom of the board.
- Notice the pads are BLUE (color code bottom layer), This lets you know that these components are on the bottom of the board



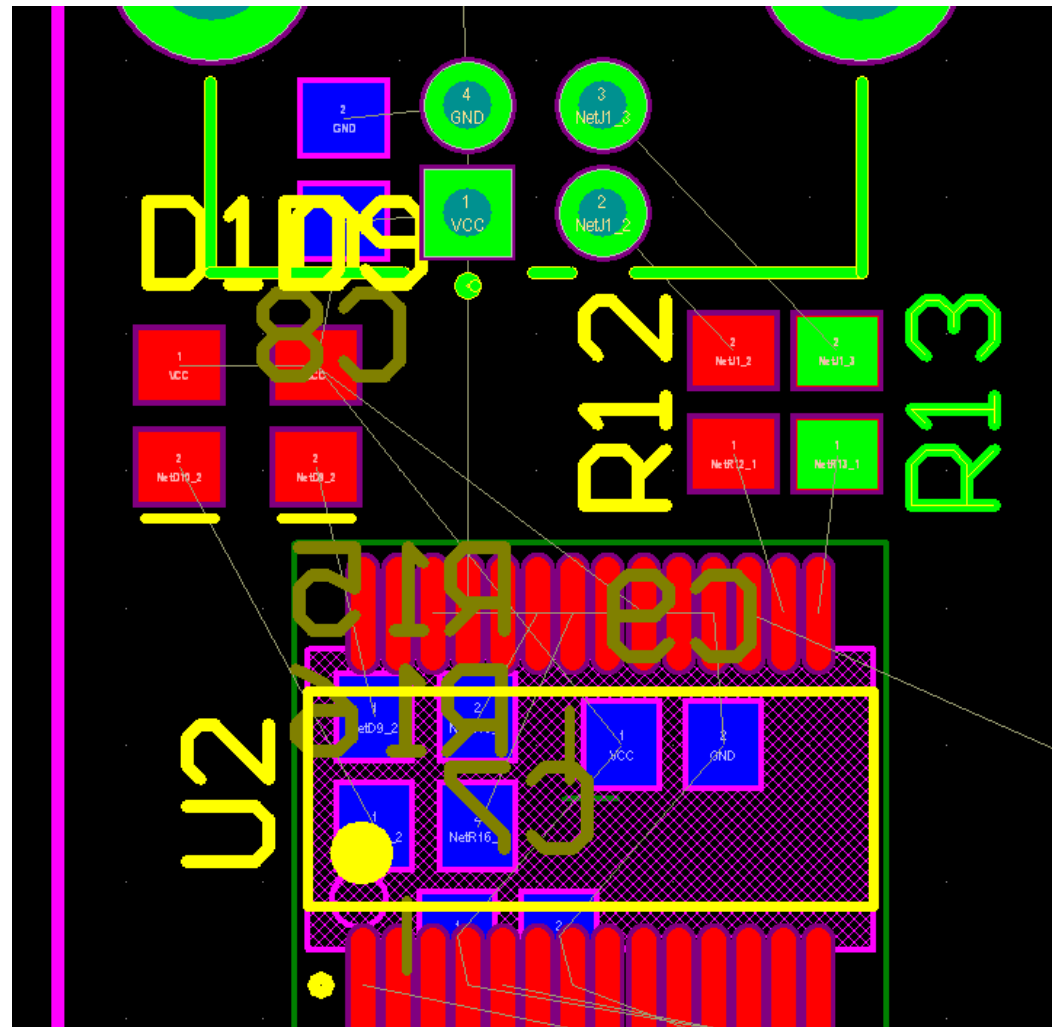
Component Placement

- Place the three decoupling capacitors on the bottom layer, and position them as close to the power pins as possible as shown below



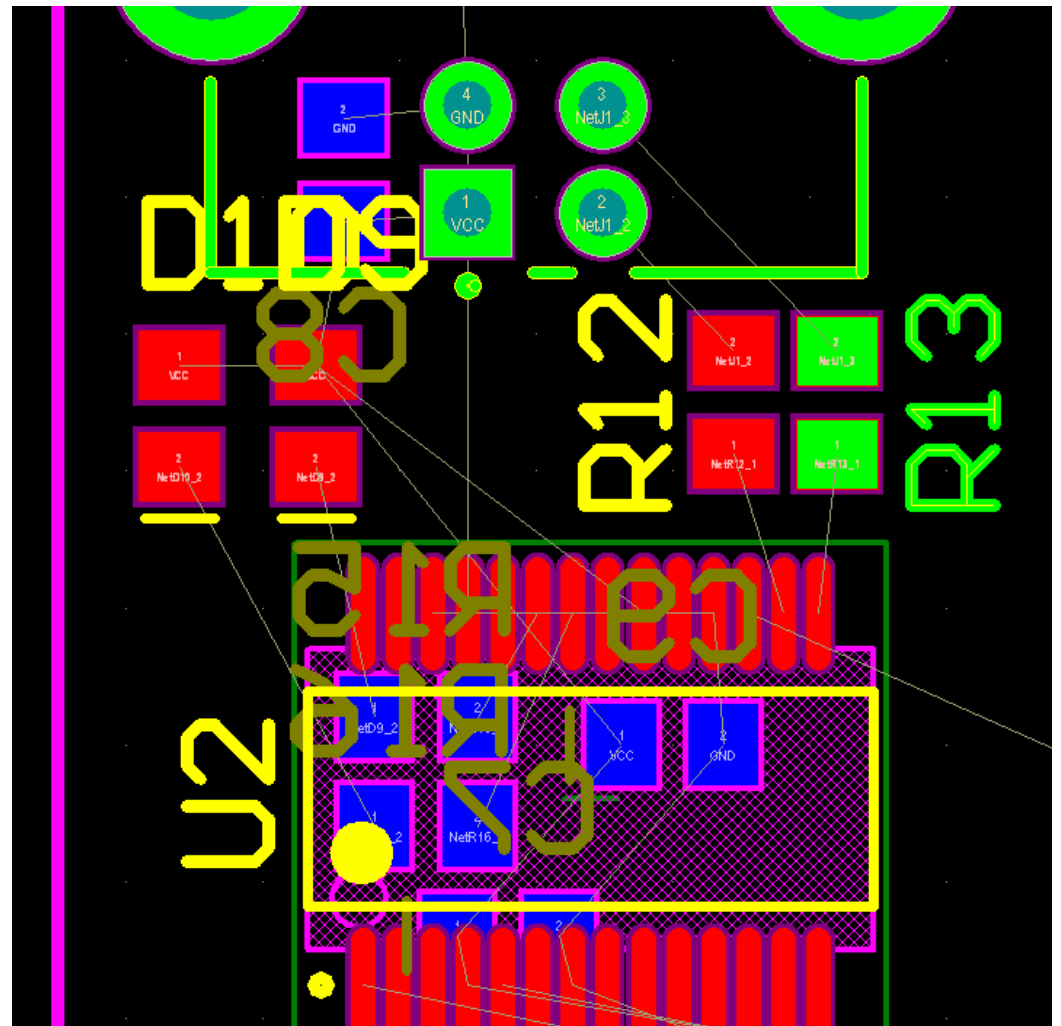
Component Placement

- The current limiting resistors can be placed on the bottom of the board (under the FT232 chip)
- The RX and TX indicator LEDs should be placed on the top of the board



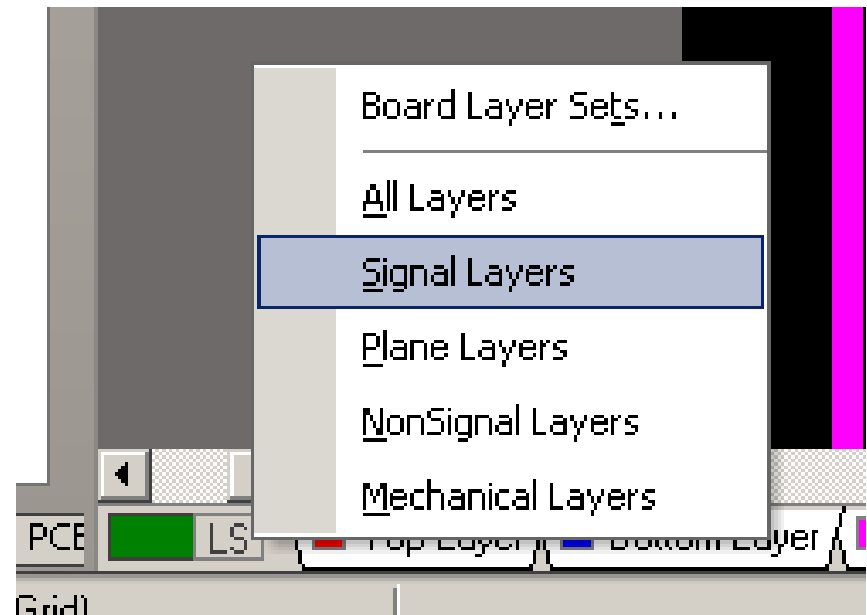
Component Placement

- This is beginning to look very cramped and hard to see things which are overlapping.
- We don't care where the component labels (top overlay) go, and will position them last, so they are just getting in the way right now



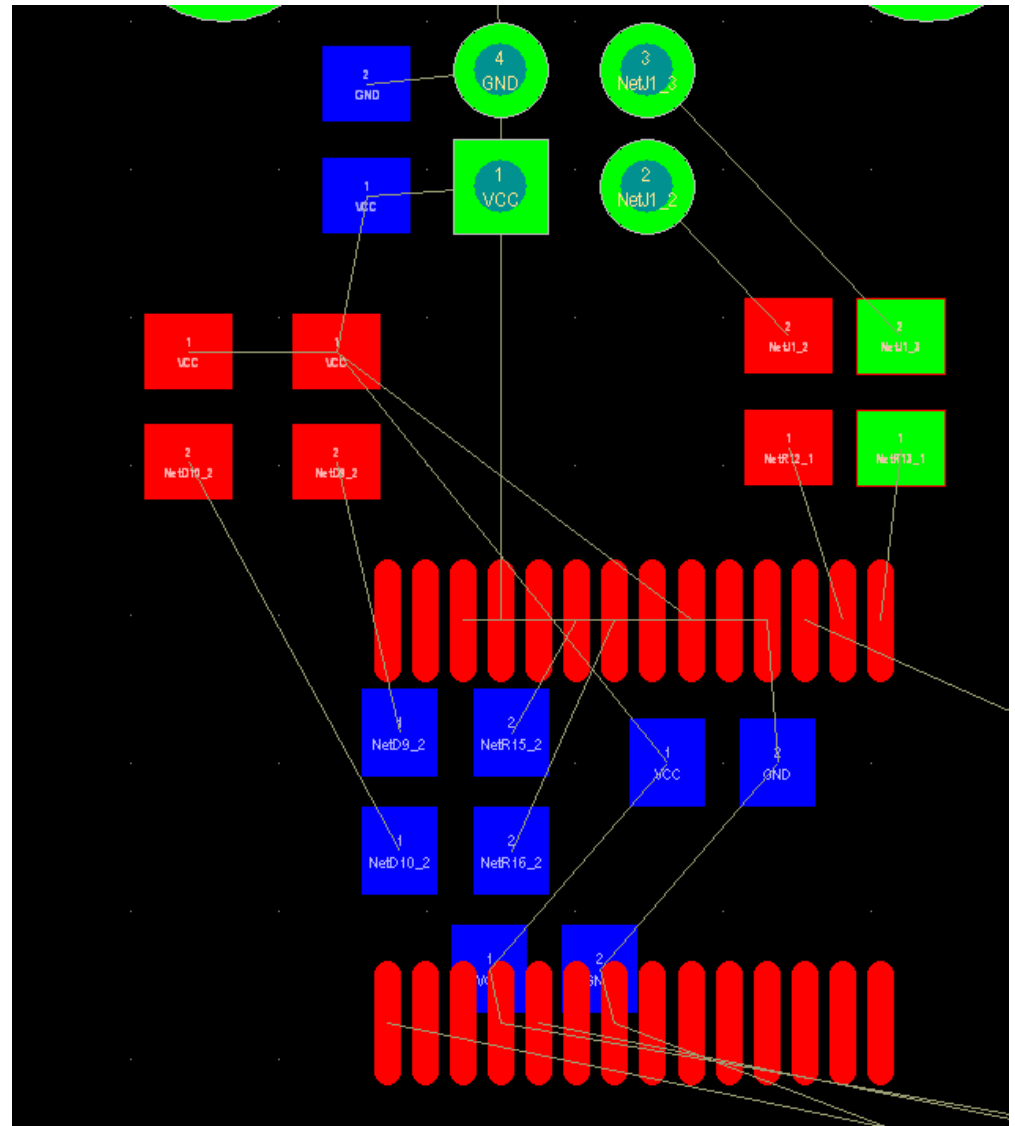
Component Placement

- Click on the LS at the left of the list of layers. This is to change the active **Layer Set (Visible layers)**
- This will bring up a screen like that on the right. Change the setting to 'Signal Layers'



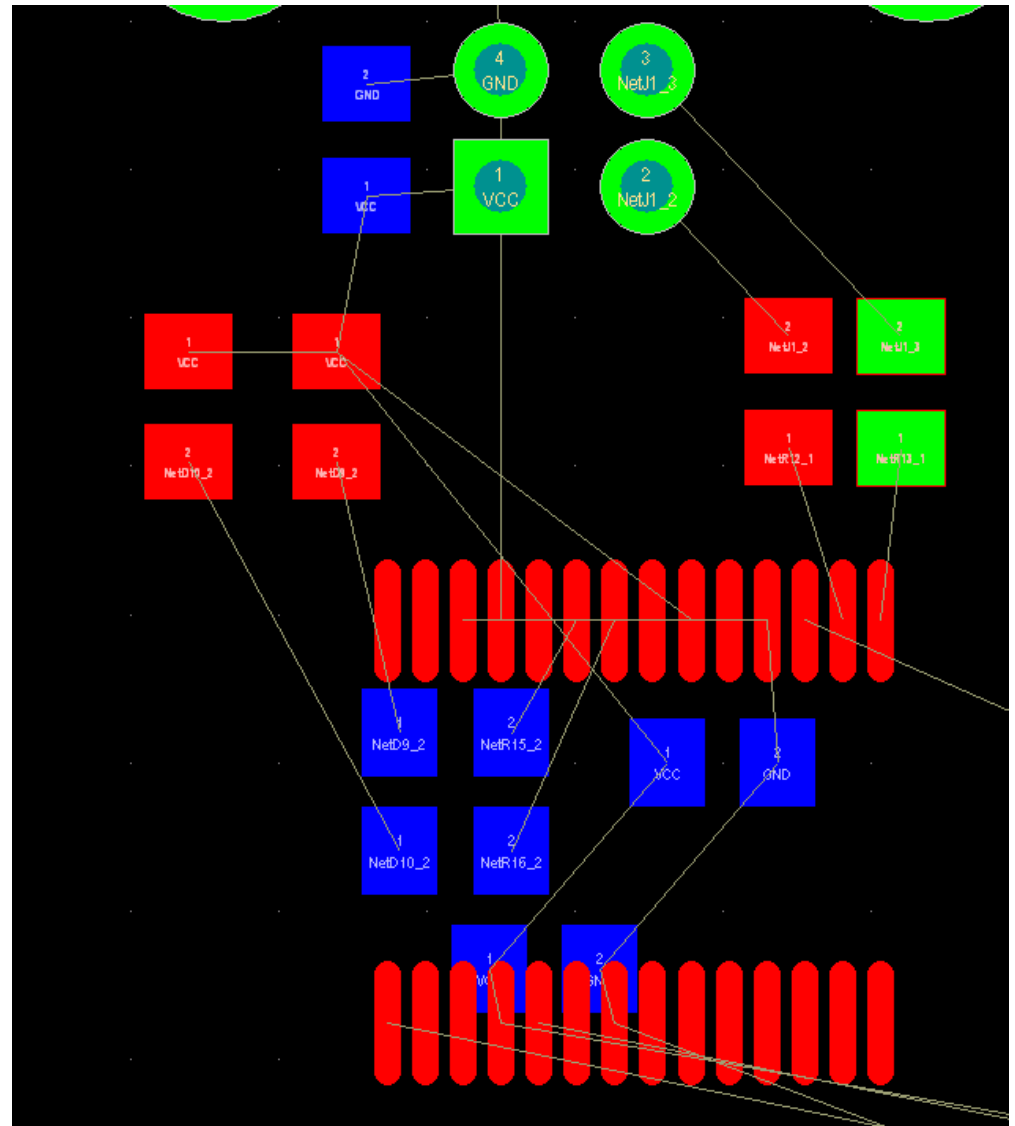
Component Placement

- This shows only **SIGNAL LAYERS** or layers that conduct electricity and hides the rest of the layers that we don't care about at the moment.
- Most of routing is best done with the **Signal Layers** layerset active



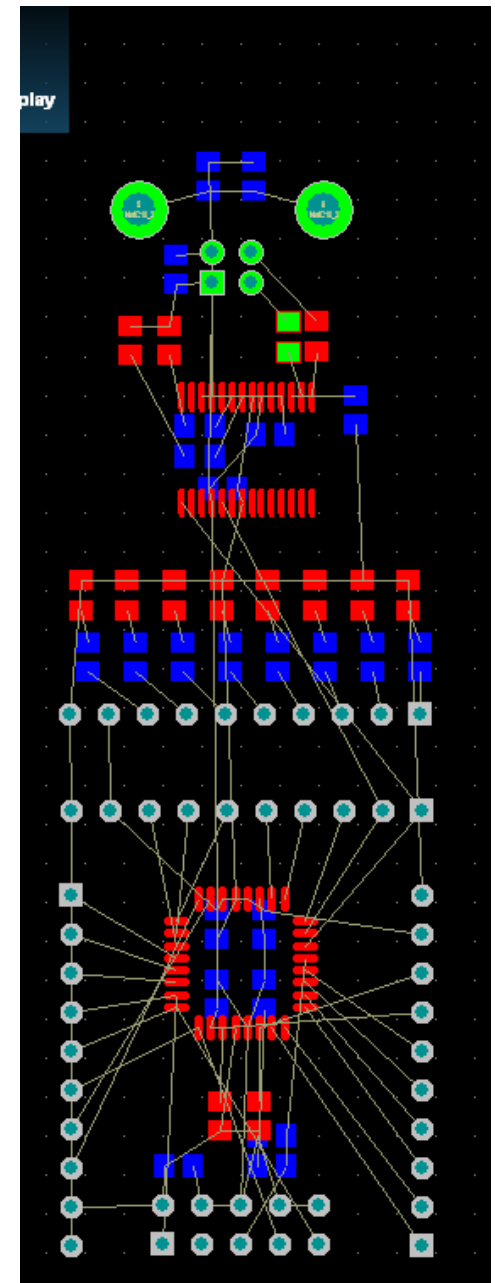
Component Placement

- It is much easier to look for lines that cross over here and make changes with component placement.
- NOTE: Be careful when placing components in this mode. **The top overlay (silkscreen)** is sometimes used for component boundaries and you might place a component inside the boundary of another part



Component Placement

- Place the rest of the components however you want to



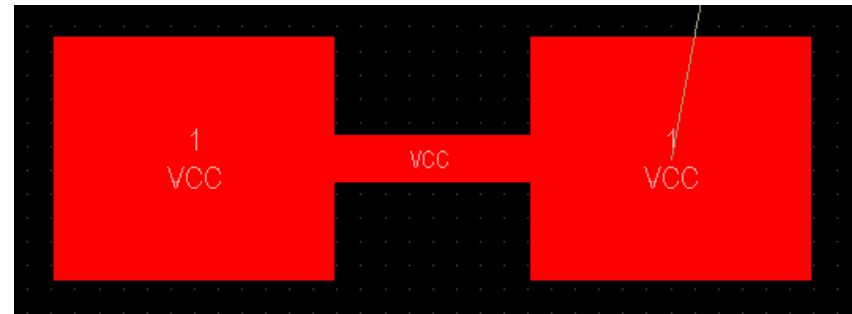
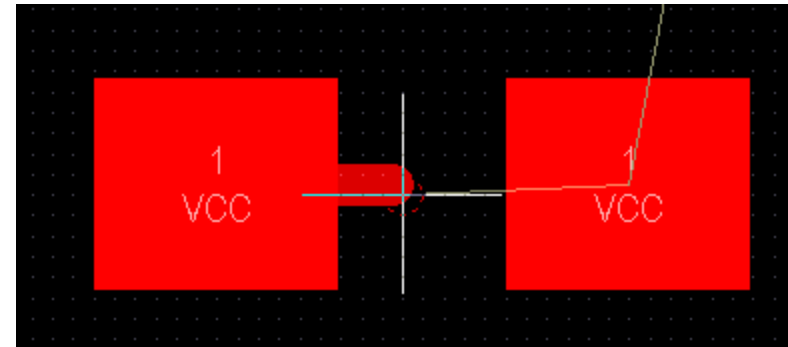
Routing

- Now we need to make electrical connections between the components
- Click the icon to interactively route connections
- NOTE: Whichever layer is active, the trace will be on that layer



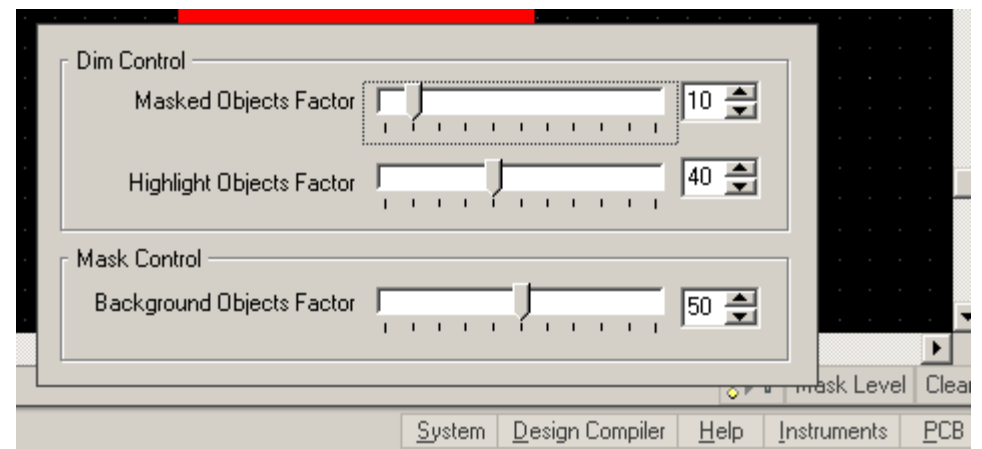
Routing

- Make a connection between two components as shown to the right.
- Notice that the wire itself is associated with the net VCC



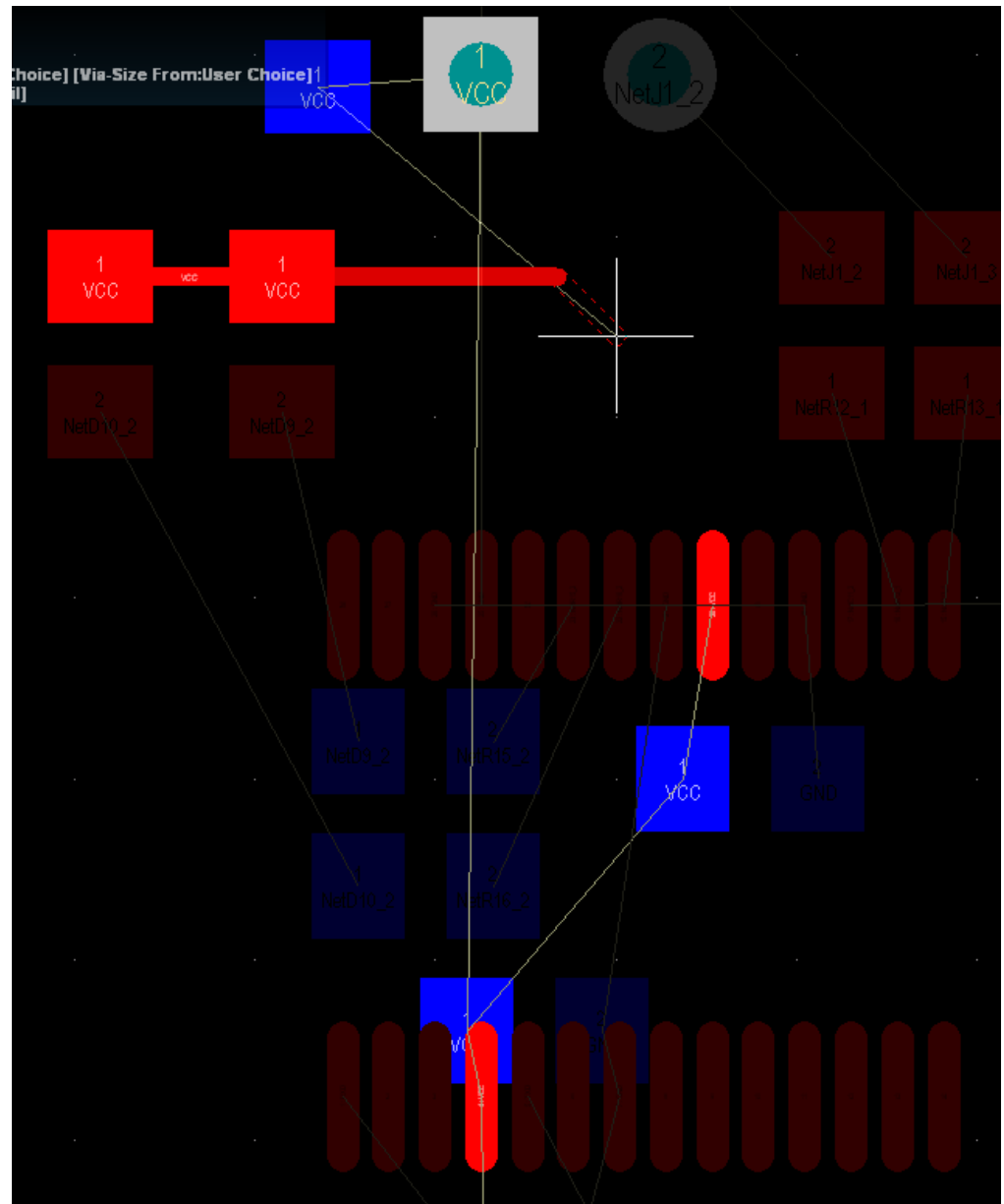
Routing

- By lowering the '**Masked Objects Factor**' in the '**Mask Level**' in the bottom right corner, when you click on a pad to wire, such as VCC, it will dull out the other connections to make it easier to see where the connections need to go



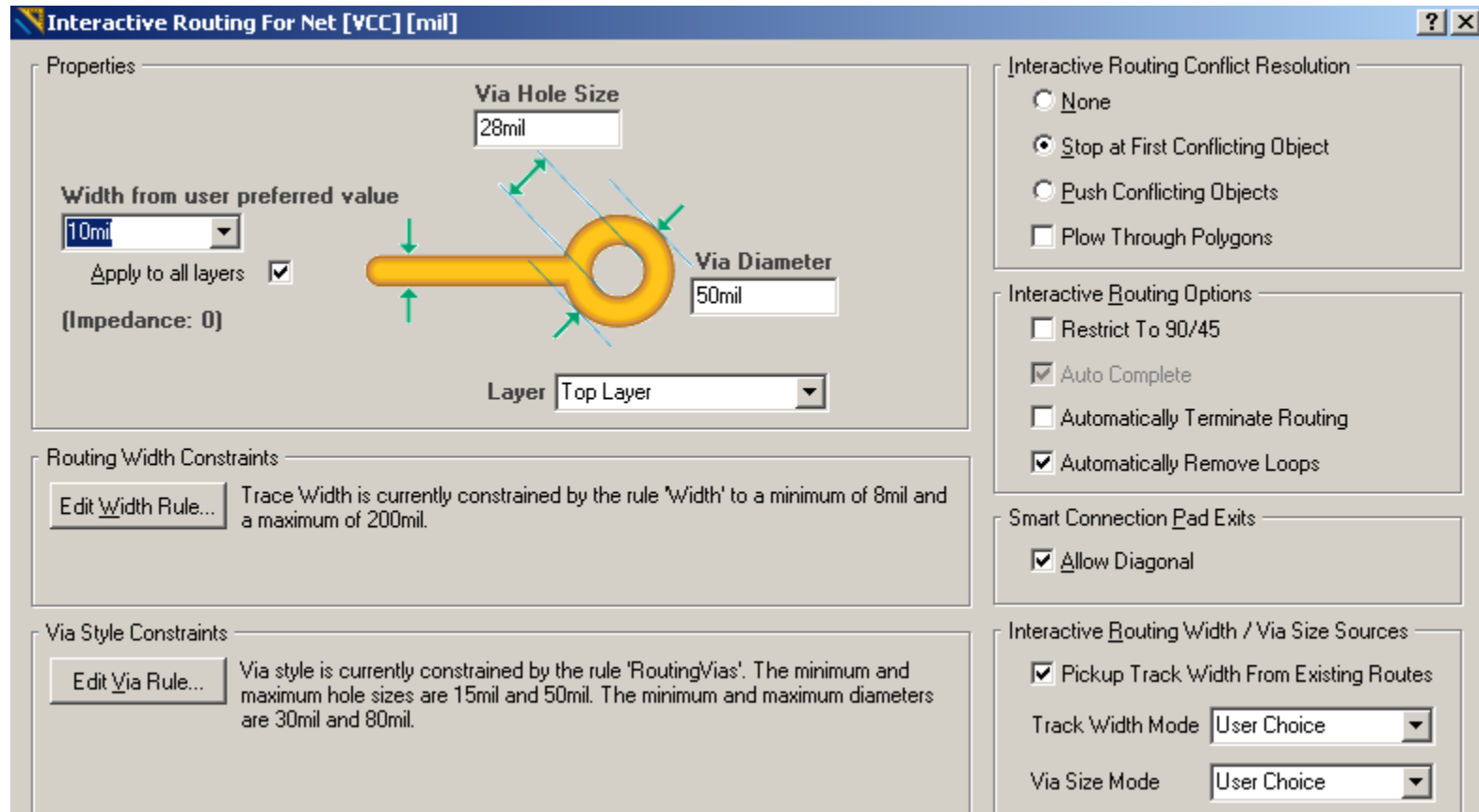
Routing

- You should adjust this so you can easily see the highlighted net, but still see the outlines of the other pads



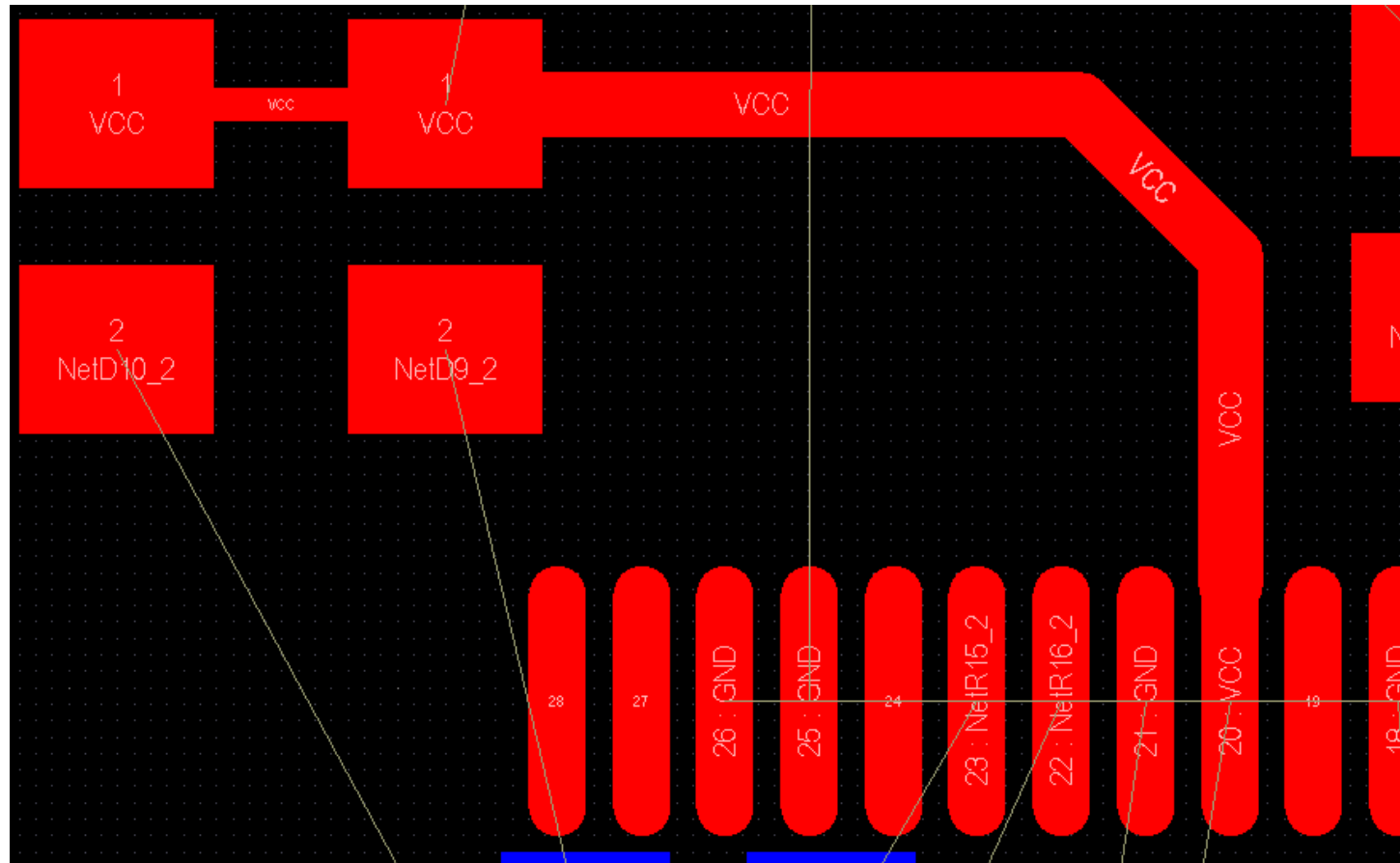
Routing

- Press the TAB key while routing and you can change the width of the trace for that particular net, and specific hole sizes. Make VCC 15 mils



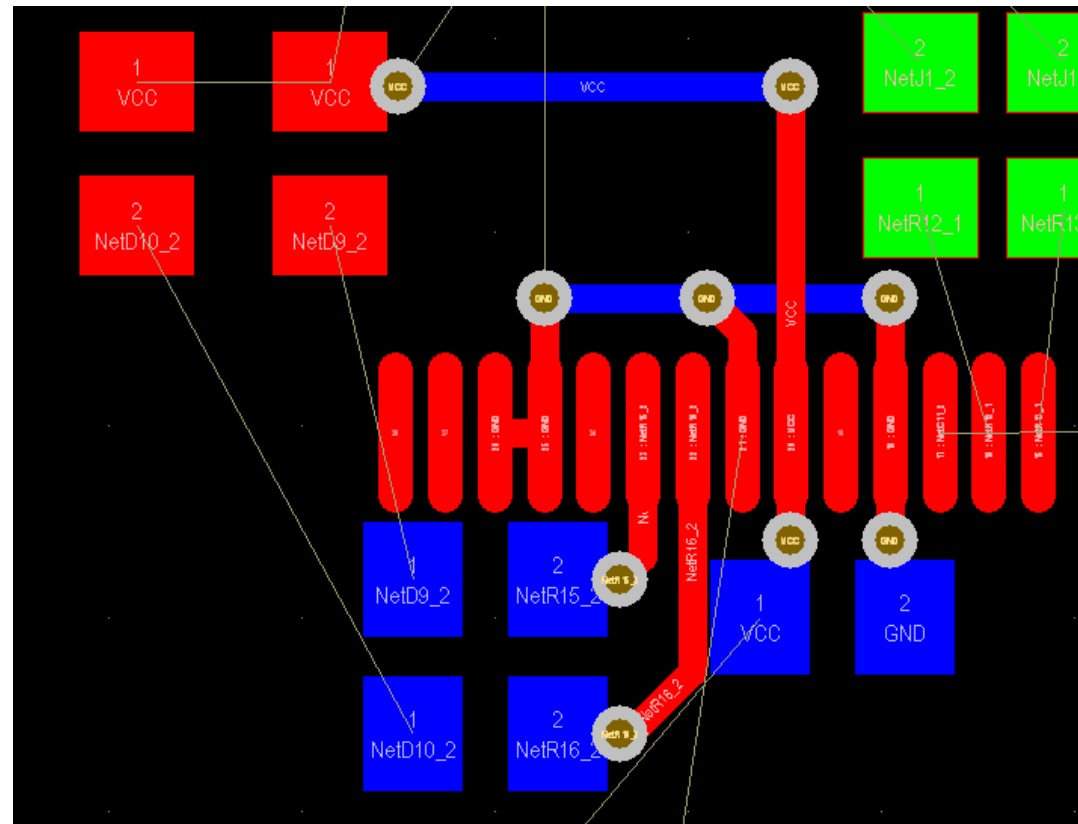
Routing

- Notice the new VCC trace is 15 mils thick



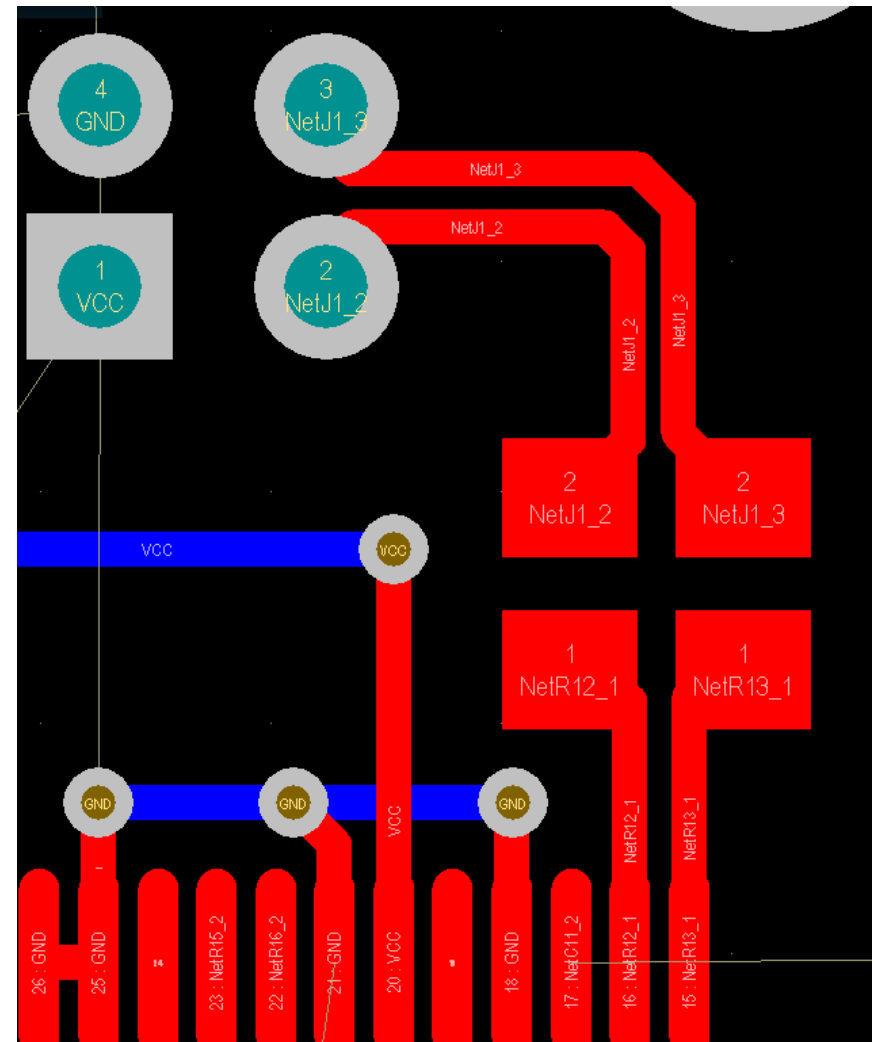
Routing

- Try to route this board using mostly perpendicular traces.
- For example, use the bottom layer for horizontal traces, and the top layer for vertical traces. This will make the routing much easier (at the expense of more holes)



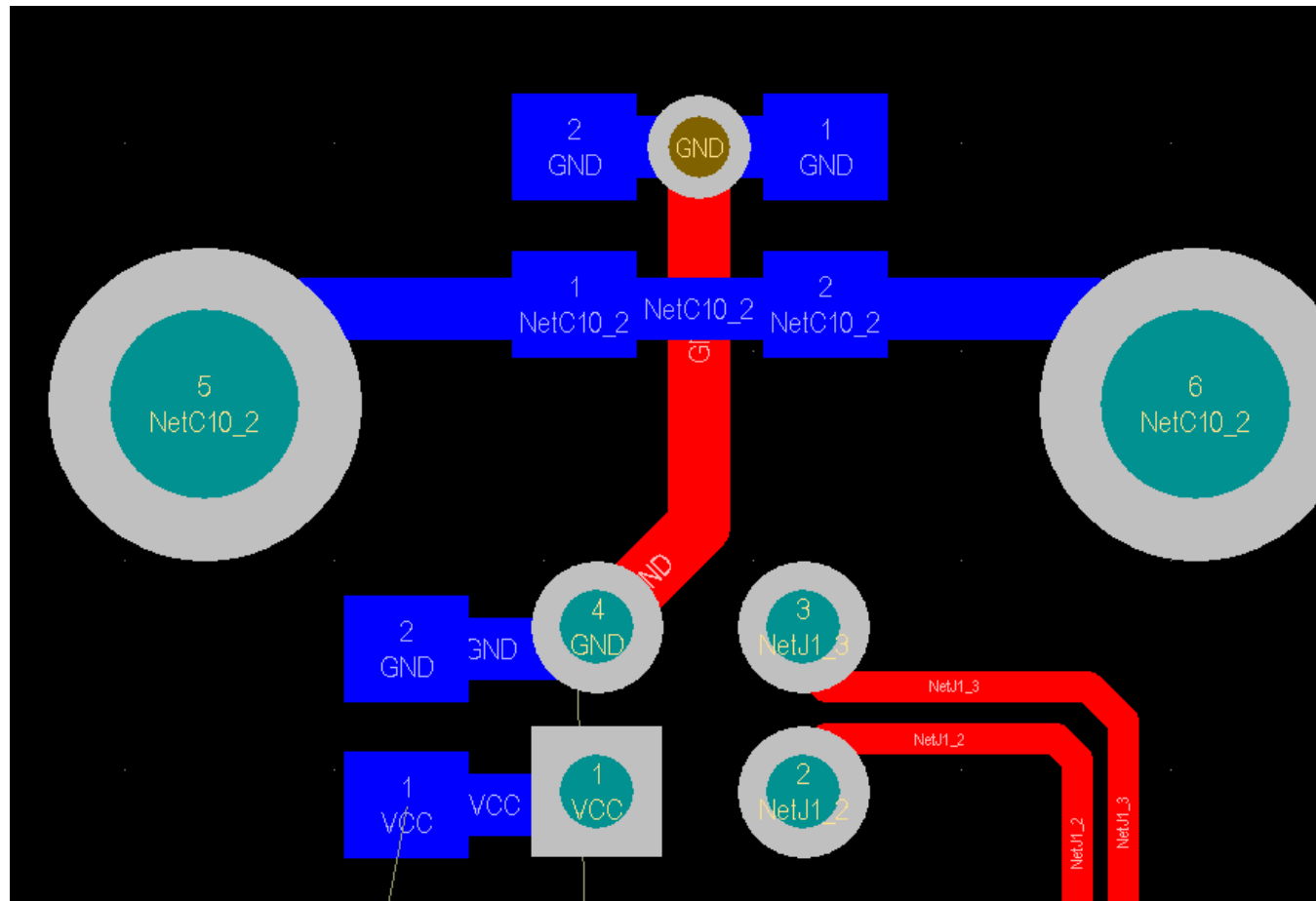
Routing

- The connection on the right is a differential pair at a high frequency.
- Differential pairs should be kept closer together
- Although you are using an orthogonal method of laying out the board, Critical nets do not need to apply.
- Try to keep the layer changes and discontinuities to a minimum for critical lines



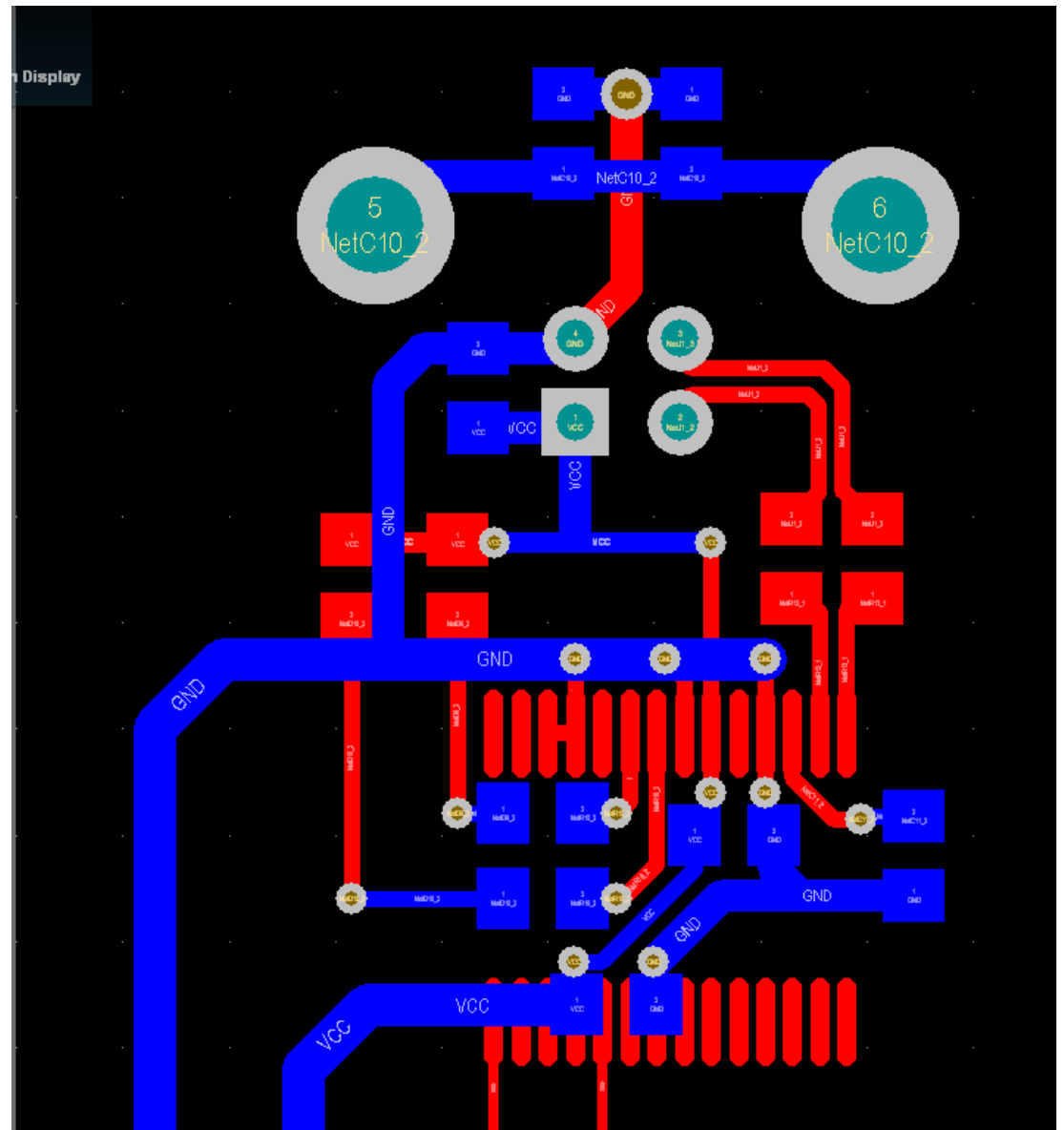
Routing

- As a general rule for right now, you want to make the traces about as large as you can make them.
- **Why ?**



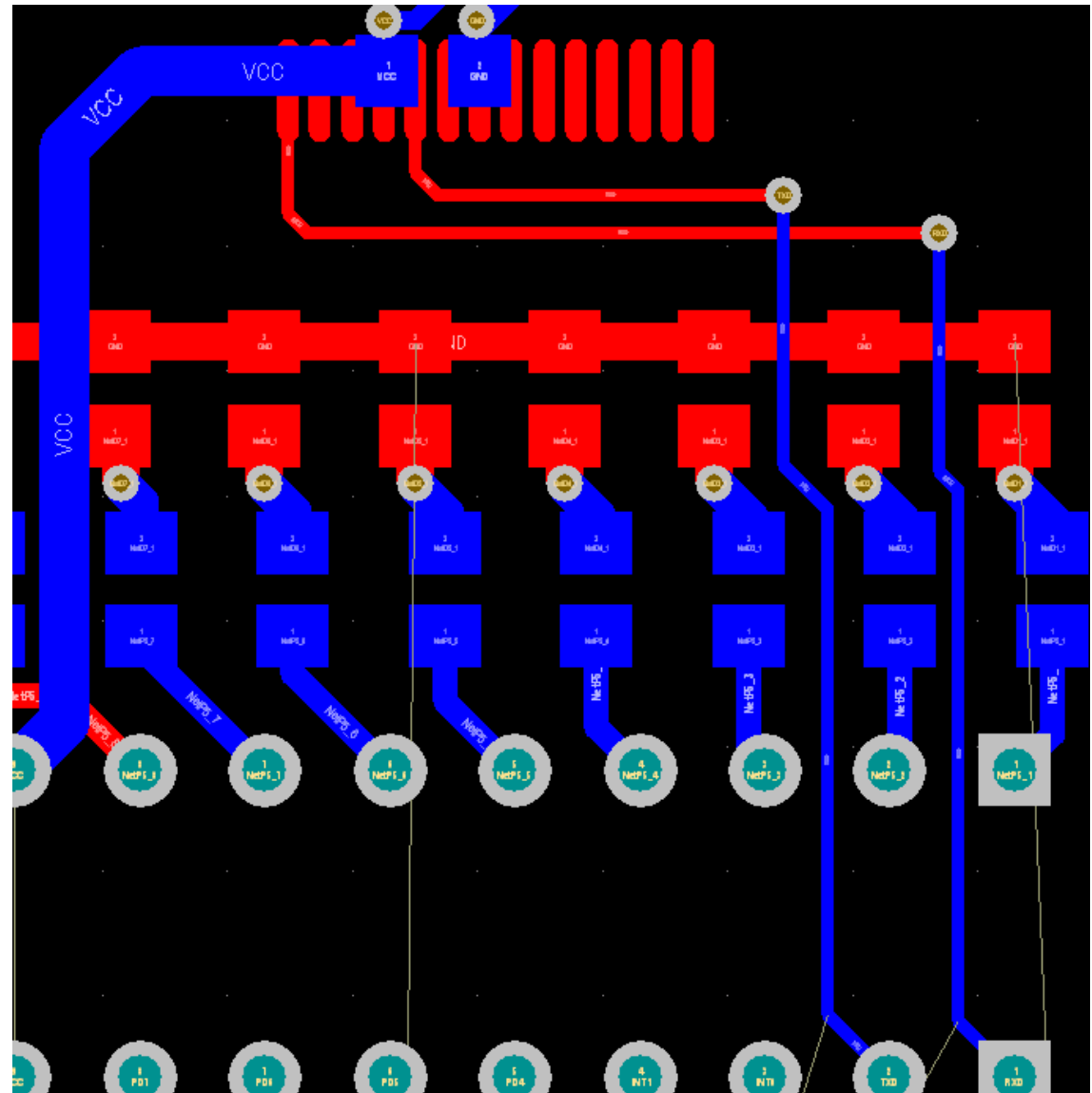
Routing

- When the USB connector is complete, there should be four traces coming from the USB interface to the rest of the circuit
 - VCC
 - GND
 - TX
 - RX



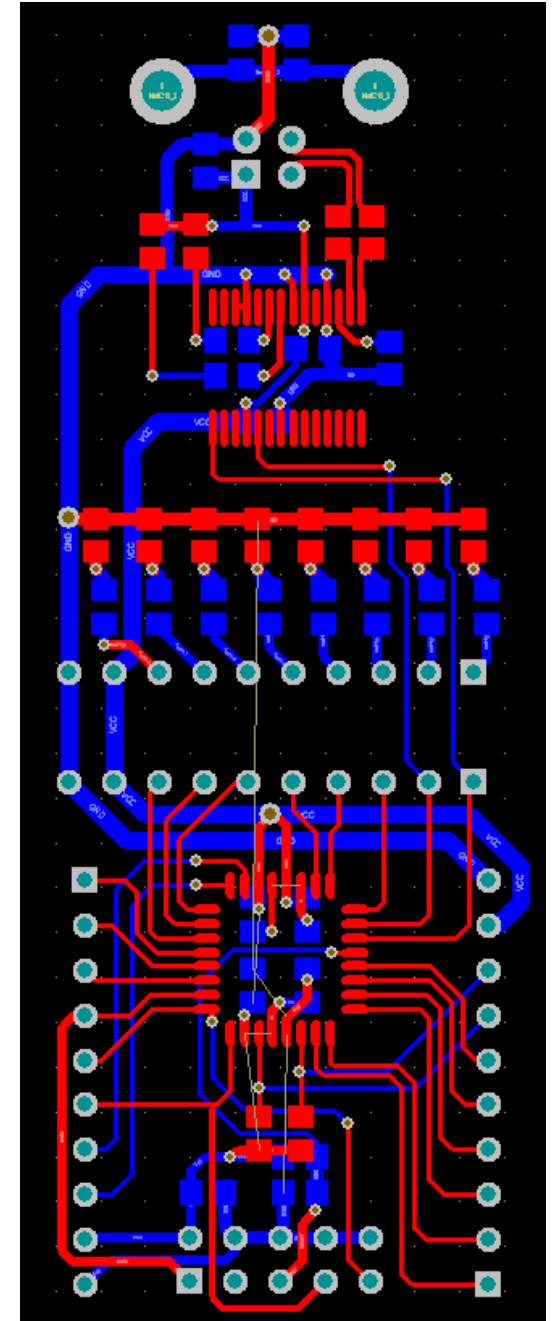
Routing

- The TX and RX traces from the FT232 Chip to the rest of the chip can be relatively high speed (1 MB/s)
- For reasons we will go over later, this means we should try to maintain the same geometry throughout the trace. (Don't vary the width)



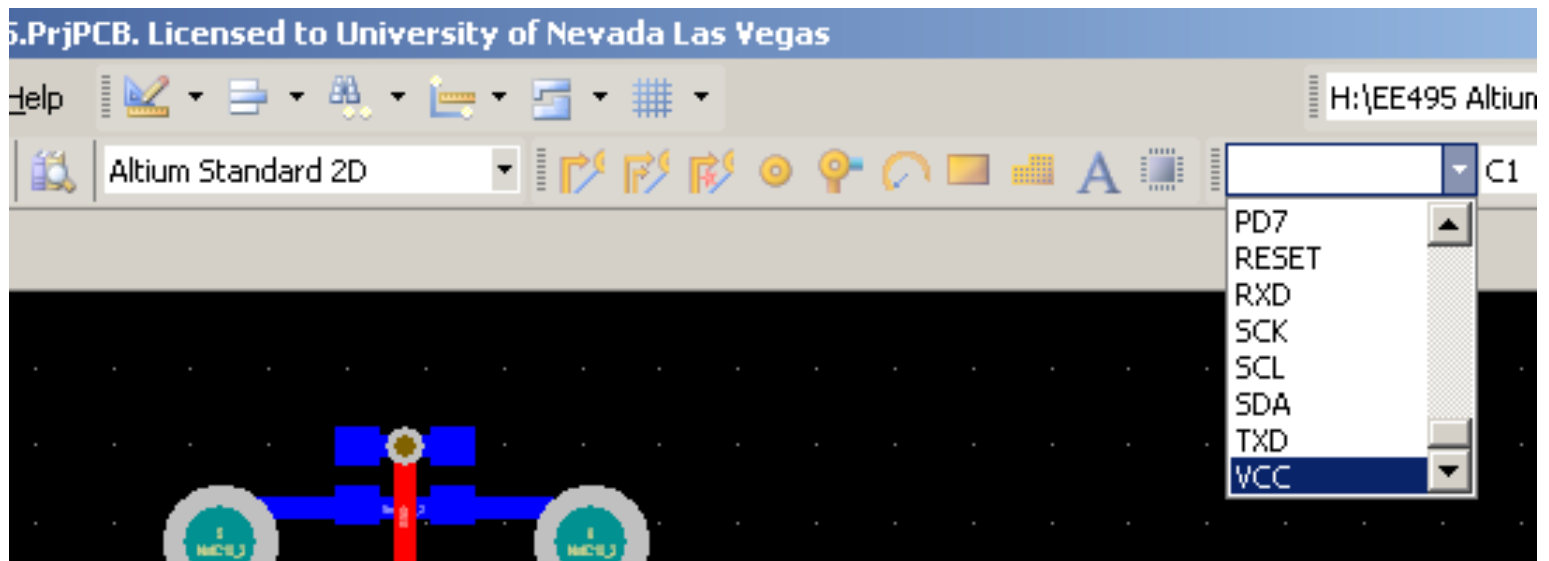
Routing

- When the routing is complete, you should have something that looks like the picture to the right.
- VCC and GND connections are not all complete though.
- We will work on VCC first



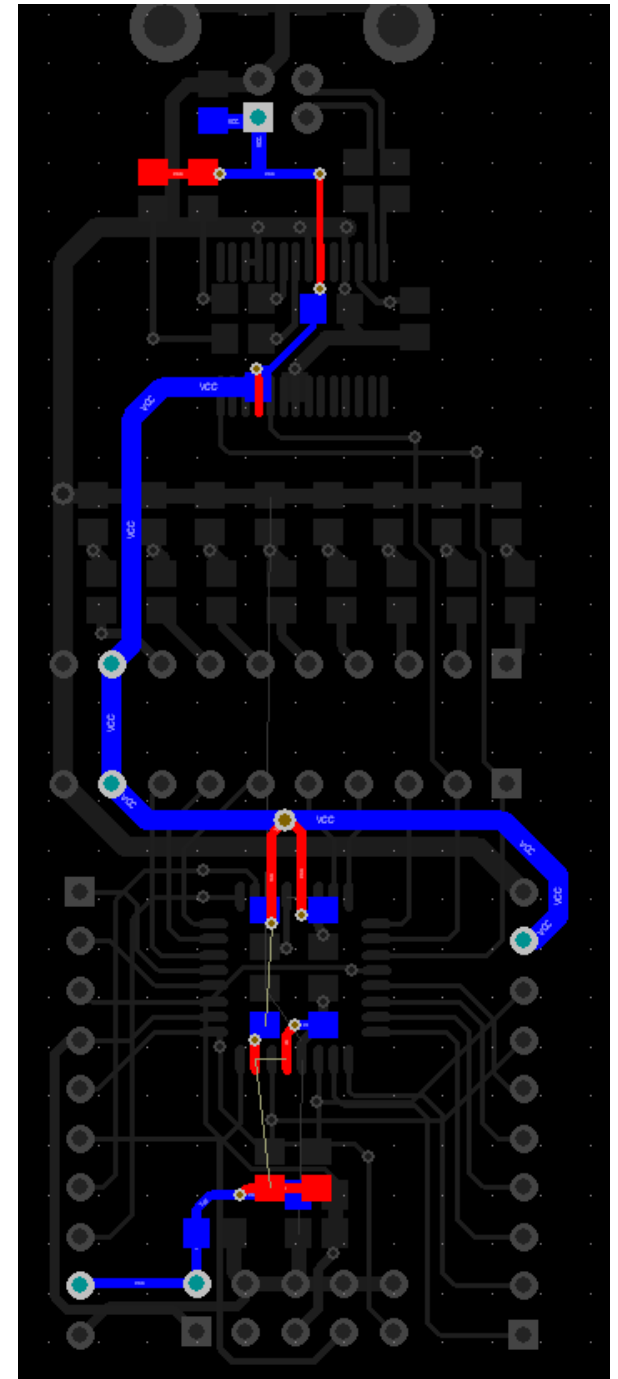
Routing

- In the left-hand drop down box at the top, select VCC. This will highlight the VCC net
- NOTE: The box to the right will highlight and jump to a particular component.



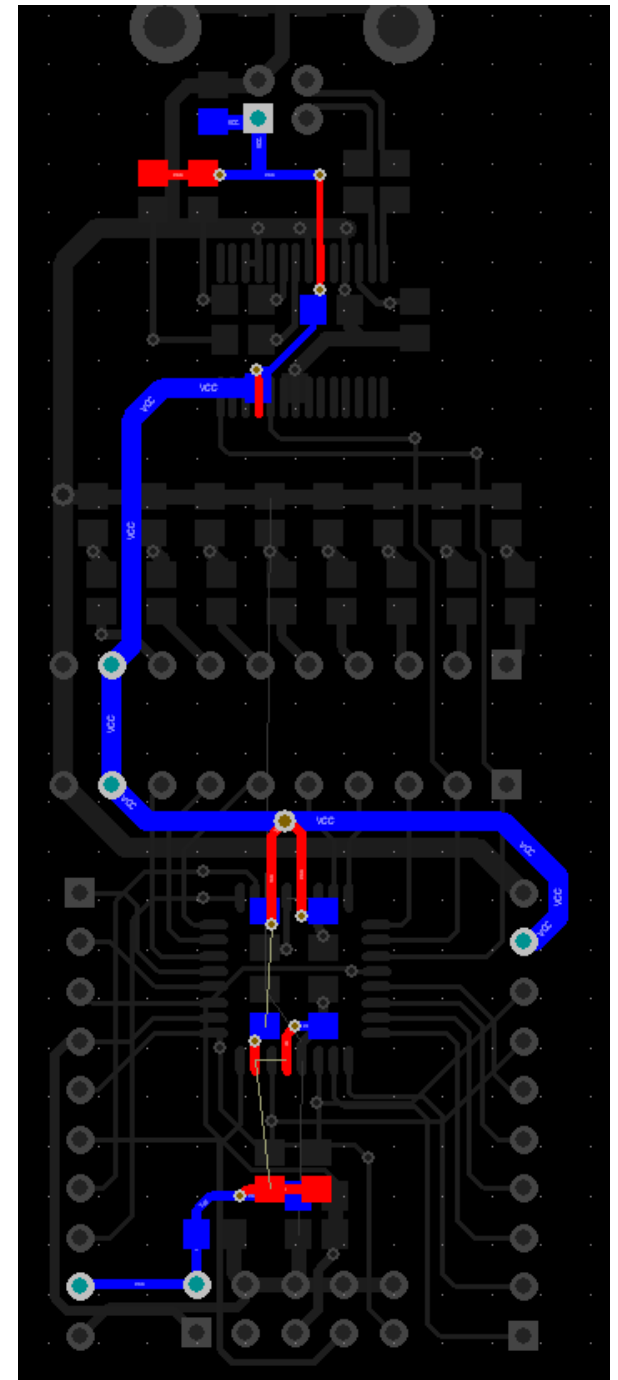
Routing

- The VCC net is highlighted and there are three 'ratsnest' wires that need to be connected.
- Make the connections to connect all the VCC's together



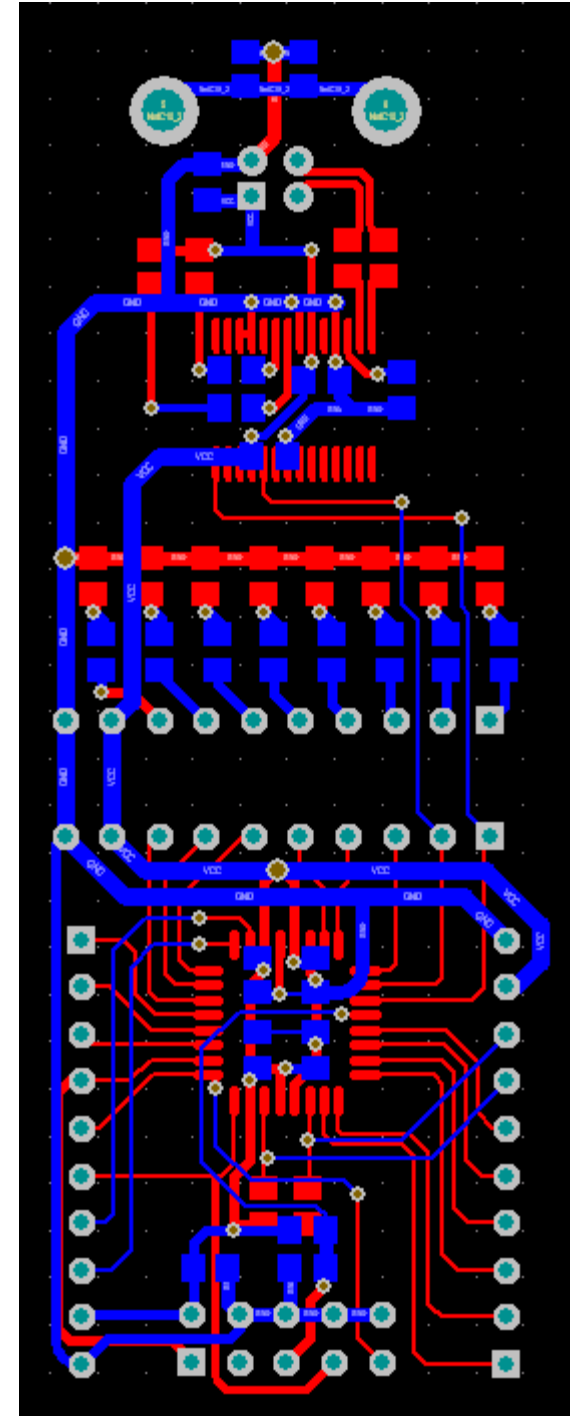
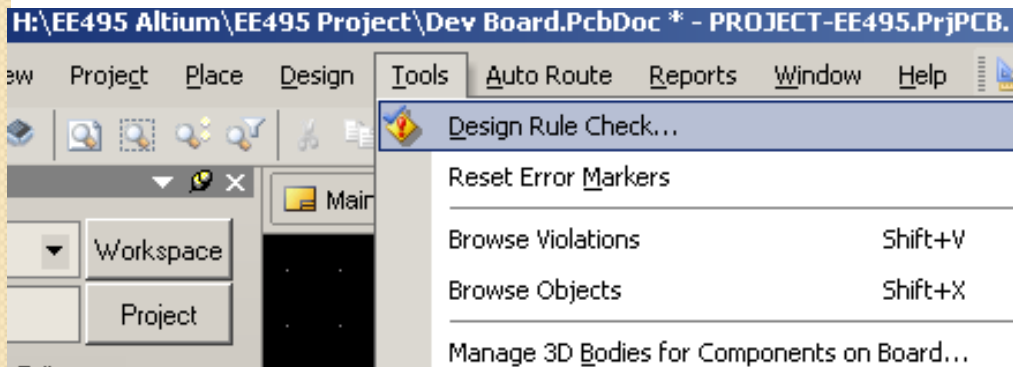
Routing

- The VCC net is highlighted and there are three 'ratsnest' wires that need to be connected.
- Make the connections to connect all the VCC's together
- Repeat the process for GND



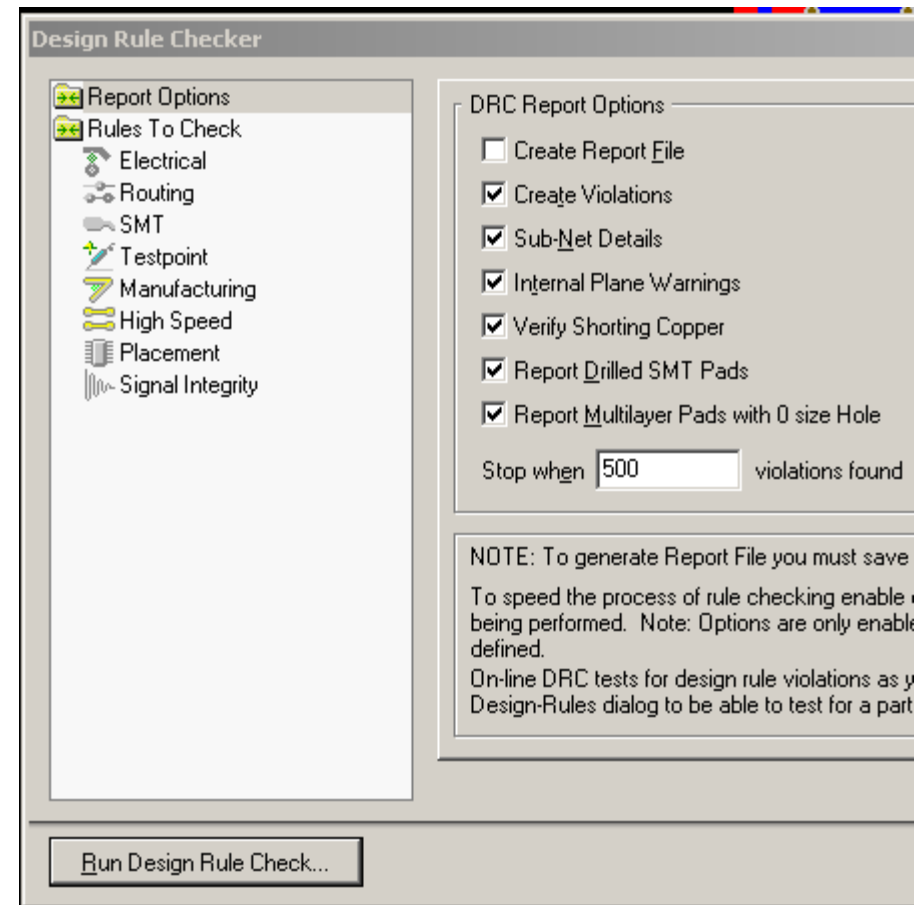
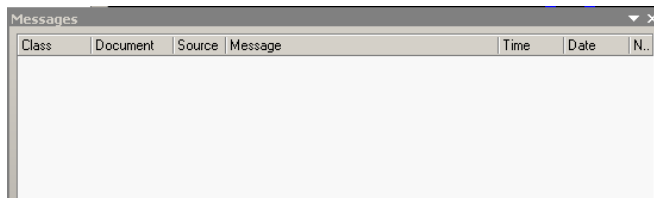
Routing

- When the routing is complete, there should be no more 'ratsnest' wires
- We must run a **DRC (Design Rule Check)** to make sure there are no errors.
- Click TOOLS > Design rule check



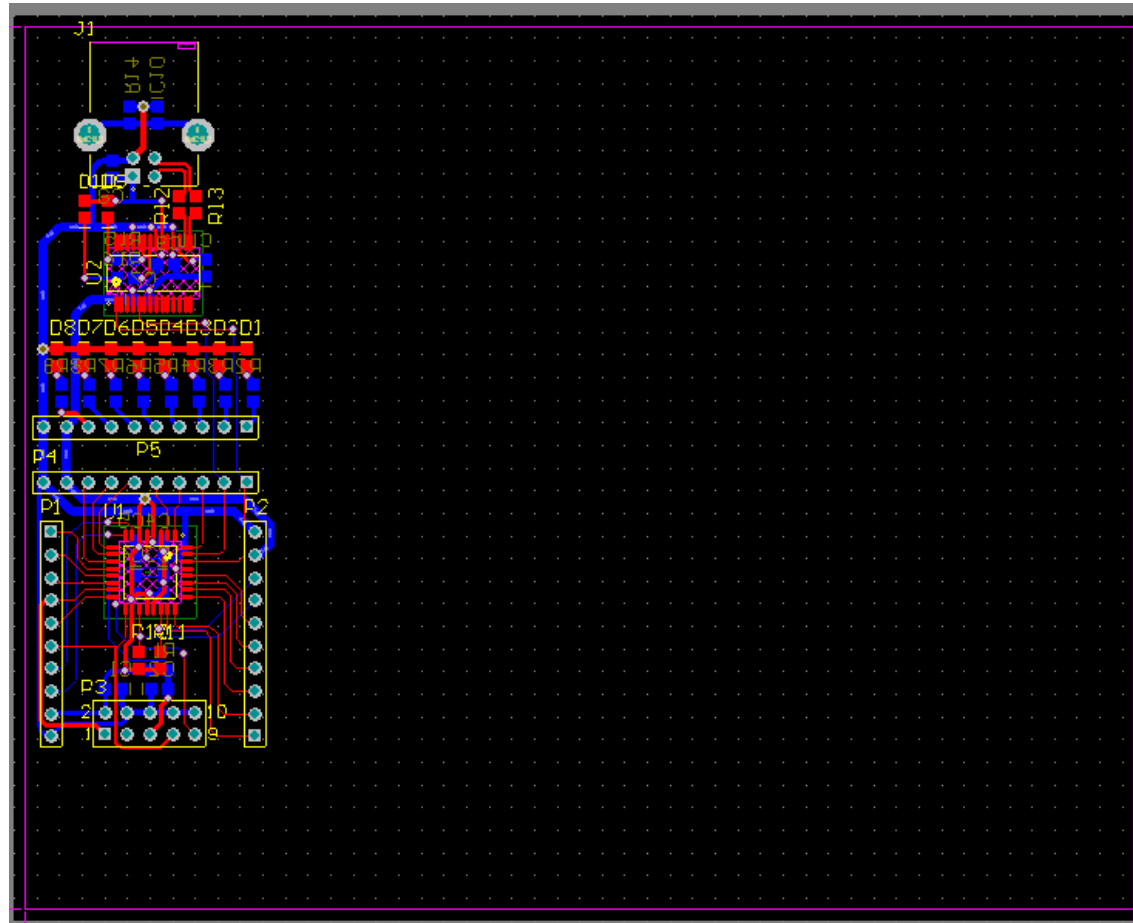
Routing

- Uncheck the box to create a report file
- Click Run Design Rule Check
- If there are no errors, you should get an empty message box like below



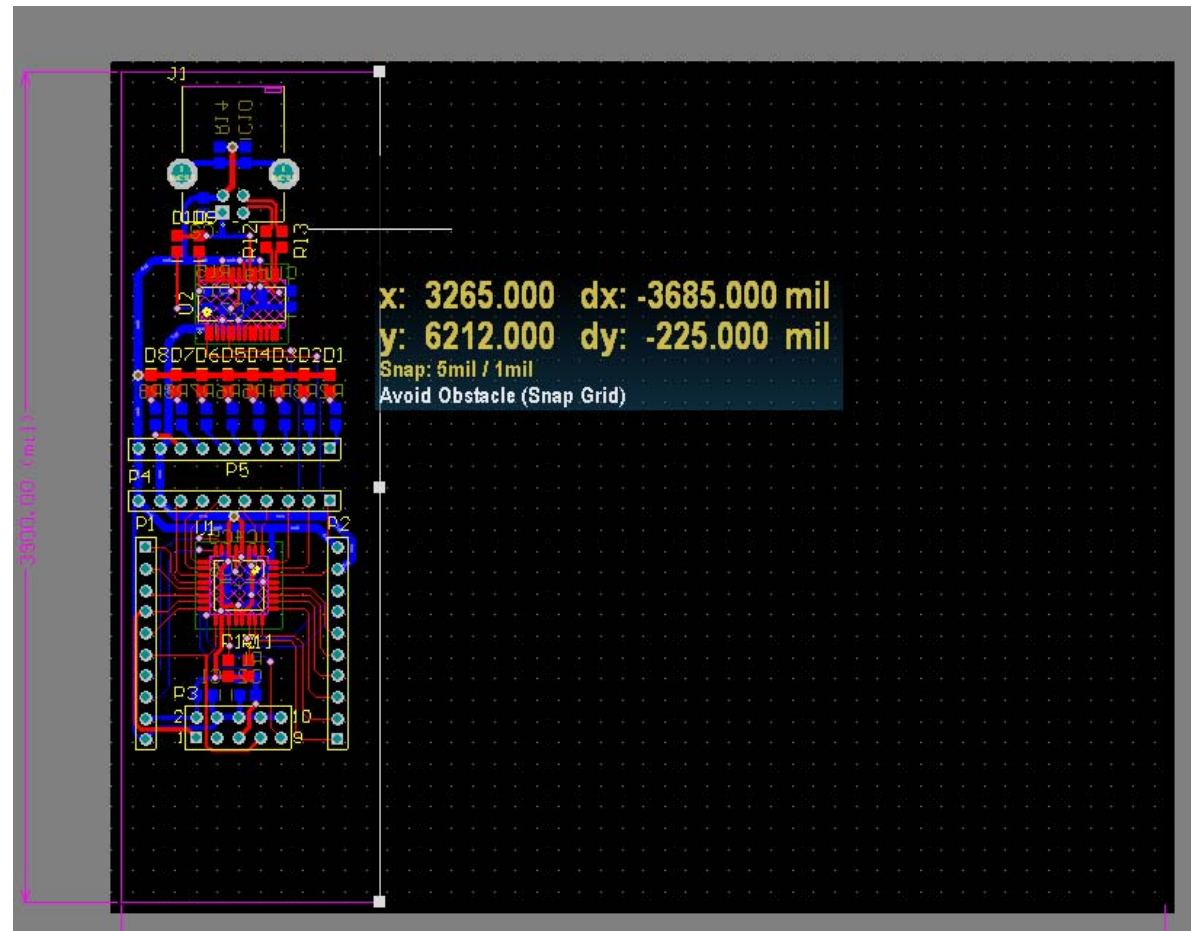
Routing

- Change the Layer Set back to ALL LAYERS and view the entire board.
- As you can see, we have a lot of wasted area, so this gives us a chance to resize the board



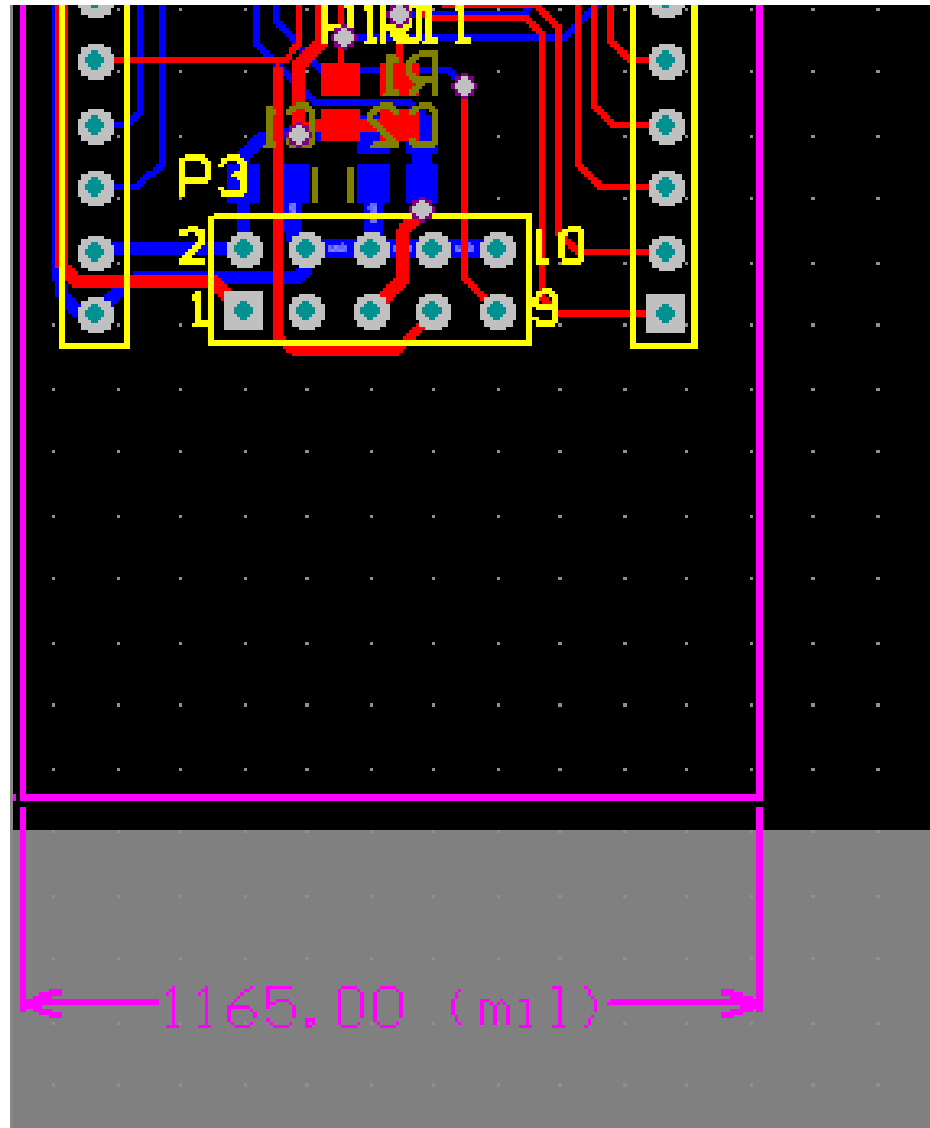
Routing

- Click and drag the pink line to the right to the left, leaving about 50 mils spacing between the pink line and the traces/components



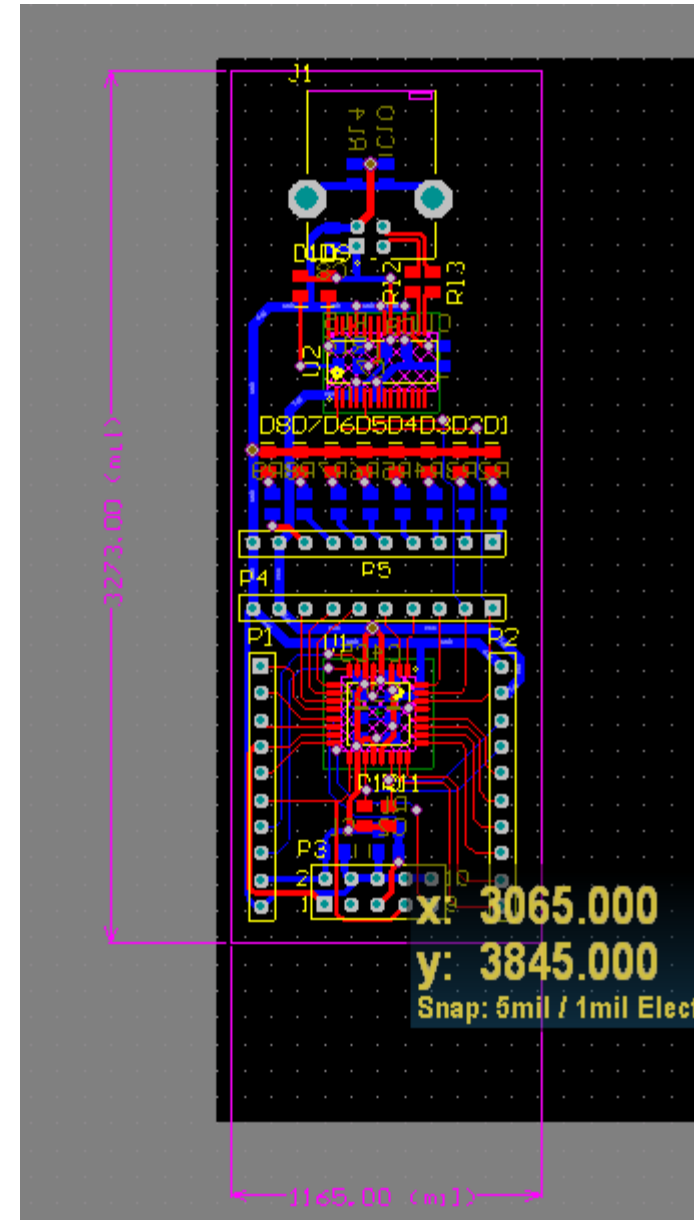
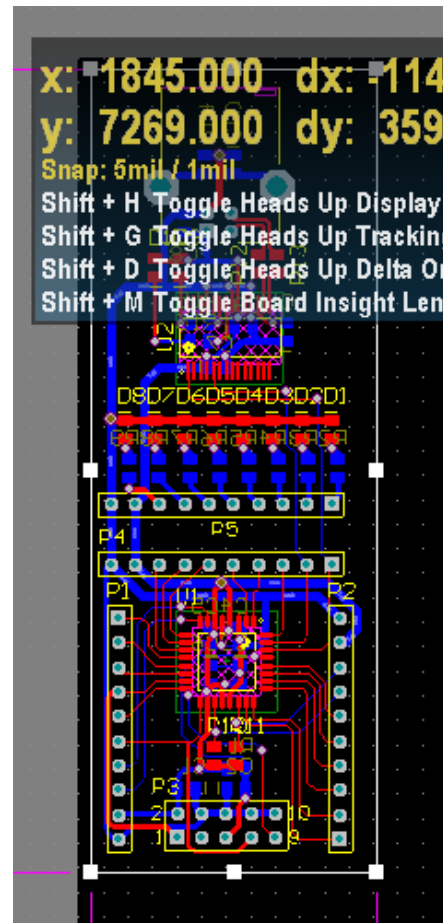
Routing

- Notice the dimensions are automatically fixed to the new distance between the arrows
- Repeat this process with the bottom pink line



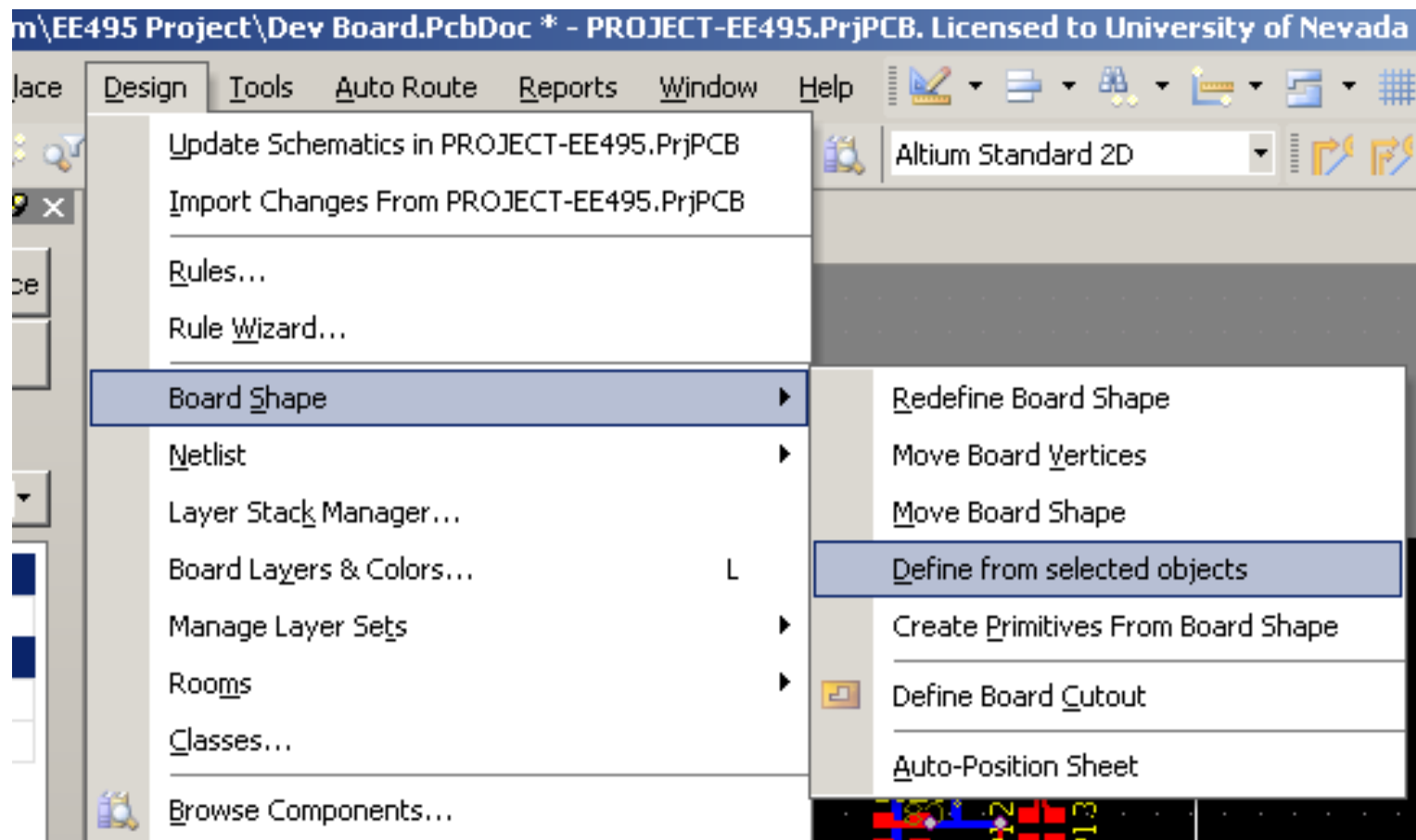
Routing

- The board should look something like this.
- Shift and click to select all four pink lines



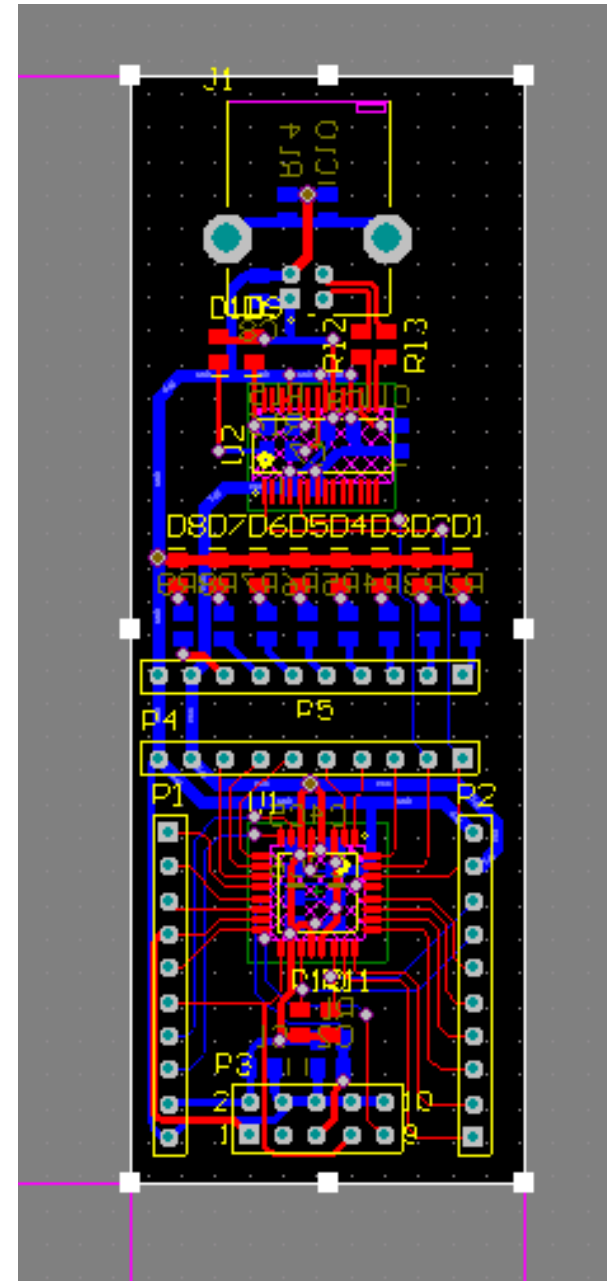
Routing

- Click
 - Design > Board Shape > Define from selected objects



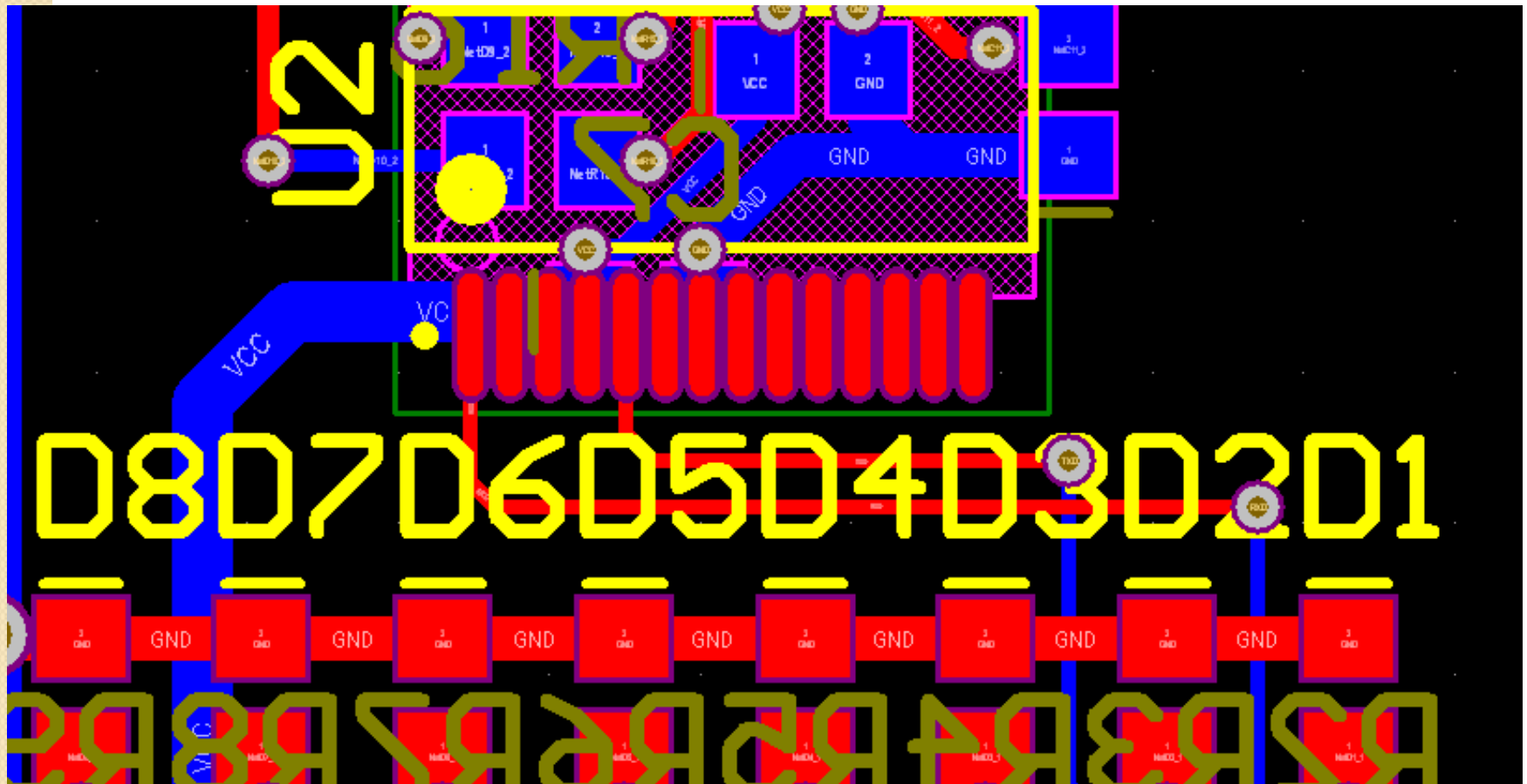
Routing

- The board should now be resized to the new dimensions
- NOTE: The pink line is a special line called a Keep-Out layer. This is to prevent placing components too close to the edge of the board
- This is technically not the smartest way to change the board size, but the simplest to see how it works.
- If you want to change the board size on a real board please let me know and I'll show you better methods



Cleaning Up The Design

- Now we should clean up the design. The top overlay (silkscreen) are overlapping VIAs so they wont show up well, the fonts are too big, and the spacing is too close together

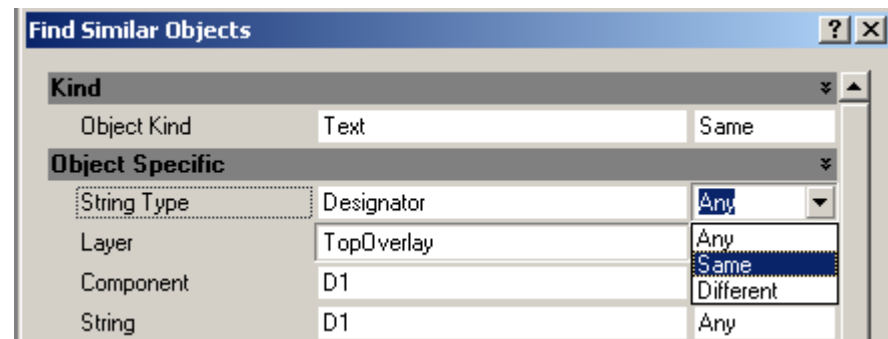


Cleaning Up The Design

- Right click on any of the top overlay (such as D1) and click 'Find Similar Objects'

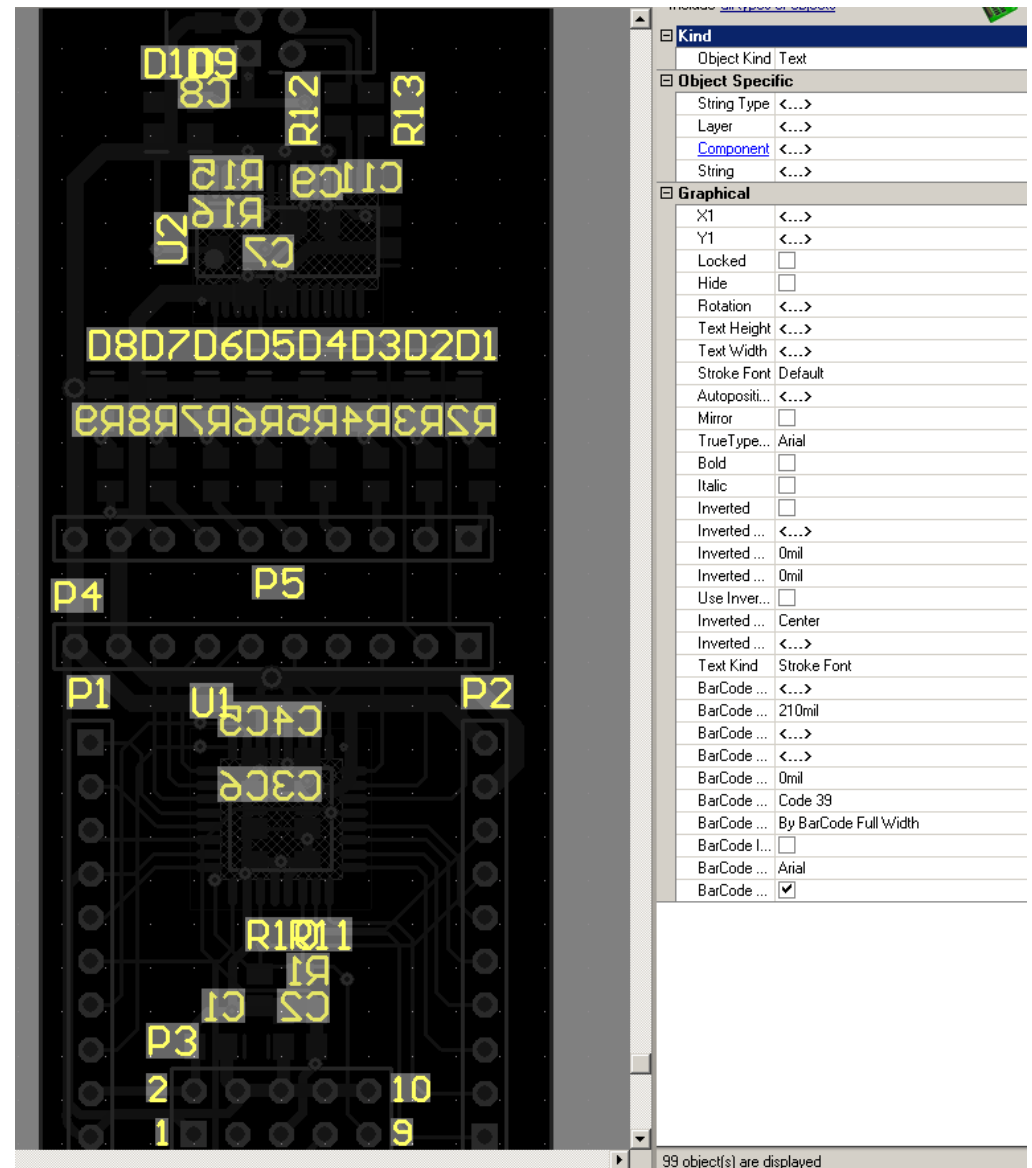


- Under String Type-DESIGNATOR, change from ANY to SAME. This will select all designators on the board



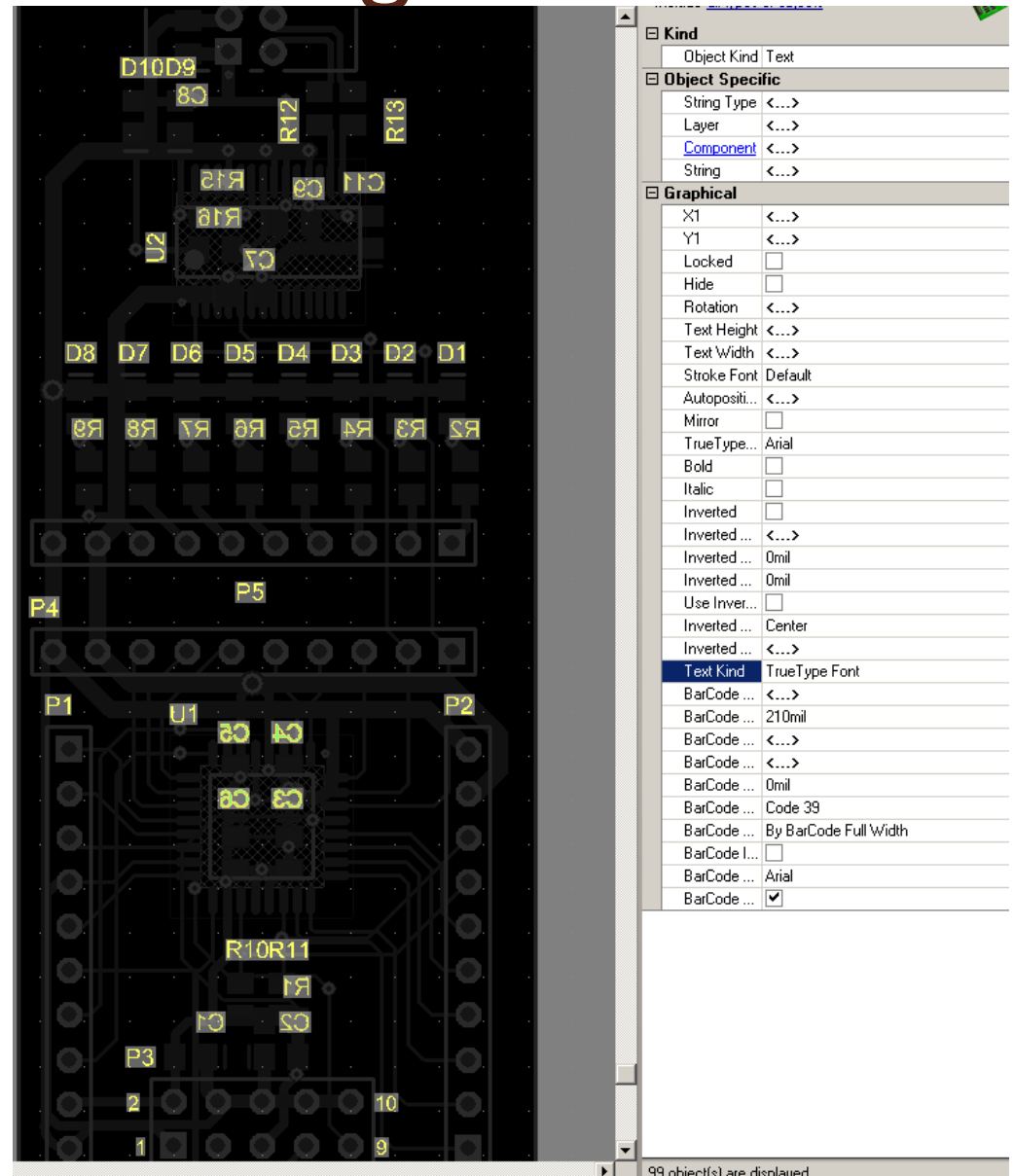
Cleaning Up The Design

- You can see that all the component designators are selected, and nothing else
- The PCB inspector shows that 99 objects are selected
- Change the 'Text Kind' from 'Stroke Font' to 'True Type'



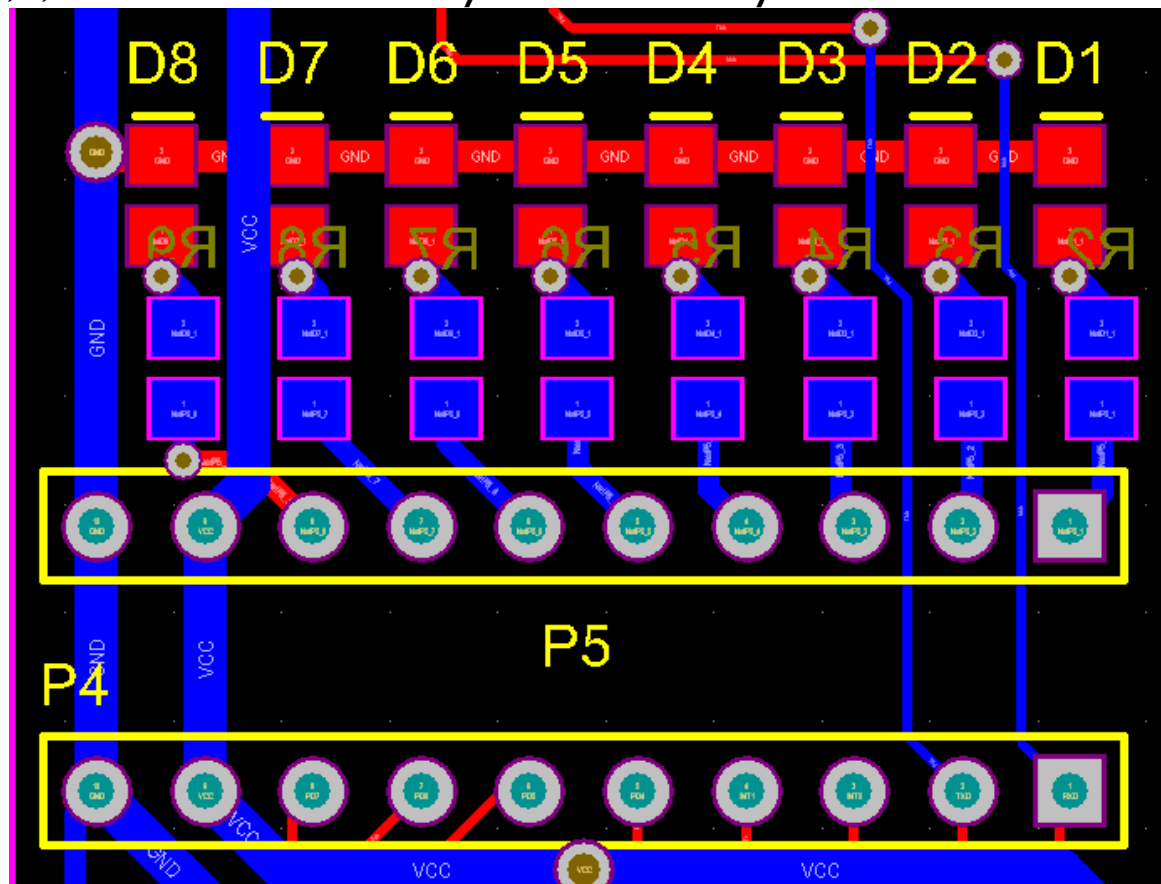
Cleaning Up The Design

- The changes are immediately made and all 99 designators are changed from stroke font to true type font.
- The reduced size and cleaner font type already makes the board look a lot better



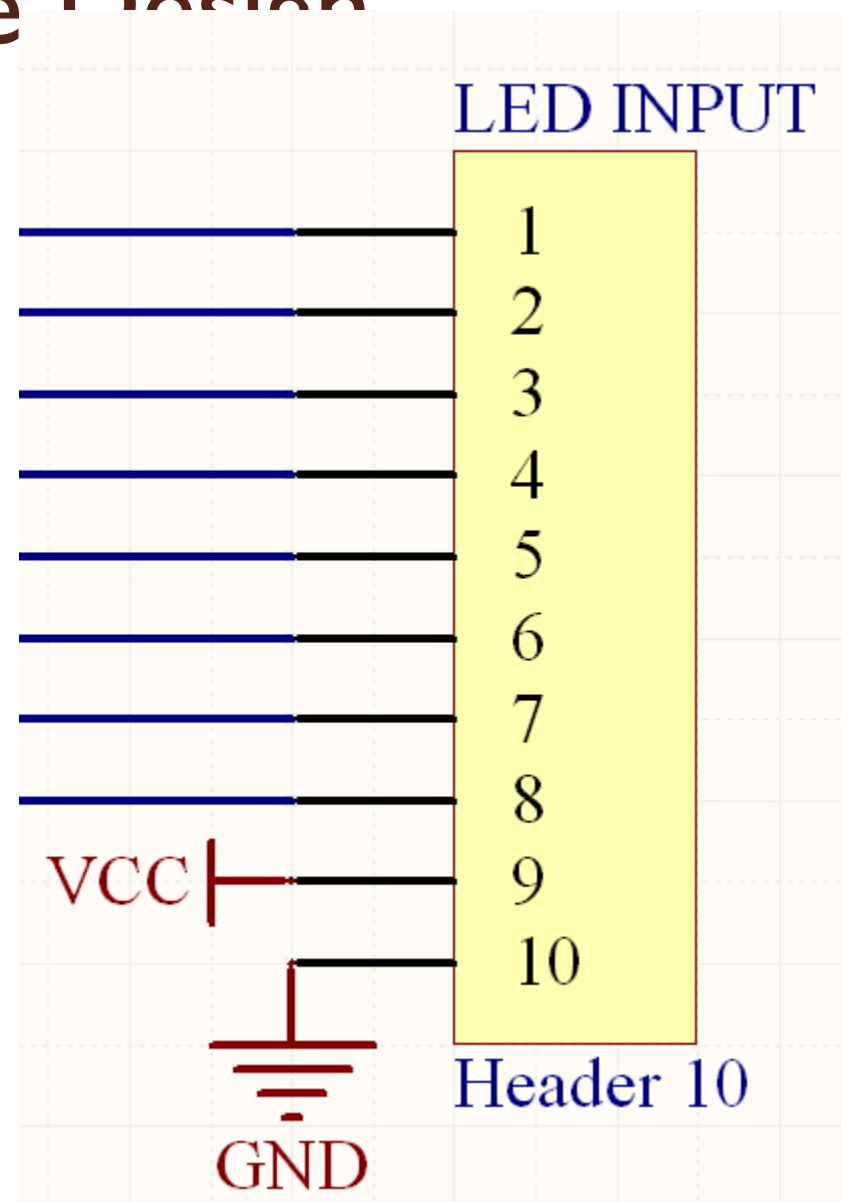
Cleaning Up The Design

- P4 and P5 are not very intuitive to the end user and should probably be changed to something more descriptive.
- Also, the 8-bit LED display can also be changed to Bit0, Bit1, etc... or 1,2,4,8,... so a user can easily add the binary numbers



Cleaning Up The Design

- Go back to the schematic, and change P5 to 'LED INPUT'
- Change P1 to PORTB
- Change P4 to PORTD
- Change P2 to PORTC
- Change P3 to ISP (In circuit serial programming)
- Change D9 to TX
- Change D10 to RX
- We are making 7 changes to component names here



Cleaning Up The Design

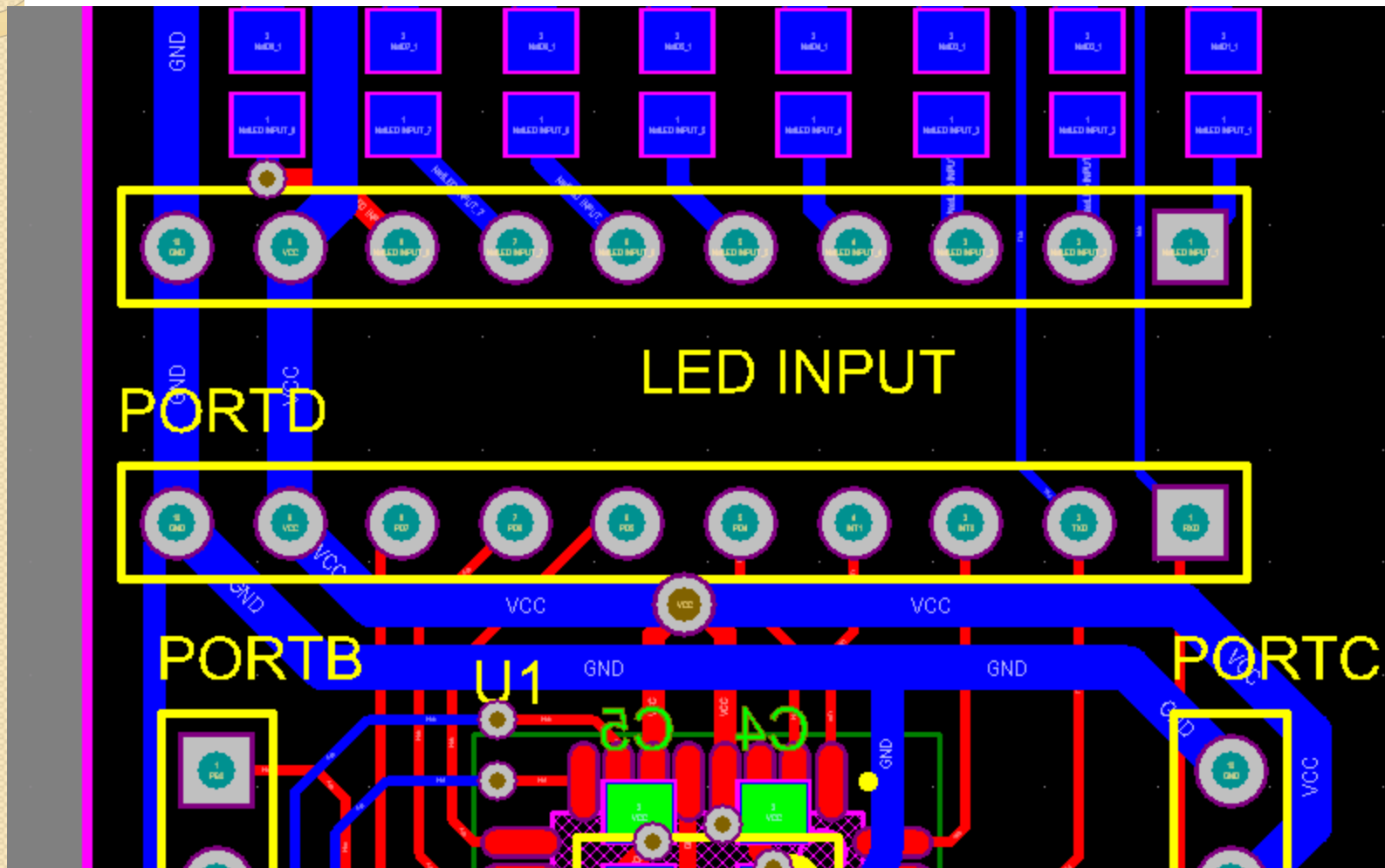
- Click DESIGN > Update PCB Document Dev Board
- As a result of changing the component names, some nets changed also. Apply all changes except adding the room



Engineering Change Order				
Modifications				
Enable	Action	Affected Object	Affected Document	Status
				Check Done Message
<input type="checkbox"/>	Change Component Designators(7)			
<input checked="" type="checkbox"/>	Modify	D9 -> TX	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	D10 -> RX	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	P1 -> PORTB	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	P2 -> PORTC	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	P3 -> ISP	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	P4 -> PORTD	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	P5 -> LED INPUT	In Dev Board.PcbDoc	
<input type="checkbox"/>	Change Net Names(10)			
<input checked="" type="checkbox"/>	Modify	NetD9_2 -> NetR15_1	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetD10_2 -> NetR16_1	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_1 -> NetLED INPUT_1	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_2 -> NetLED INPUT_2	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_3 -> NetLED INPUT_3	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_4 -> NetLED INPUT_4	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_5 -> NetLED INPUT_5	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_6 -> NetLED INPUT_6	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_7 -> NetLED INPUT_7	In Dev Board.PcbDoc	
<input checked="" type="checkbox"/>	Modify	NetP5_8 -> NetLED INPUT_8	In Dev Board.PcbDoc	
<input type="checkbox"/>	Add Rooms(1)			
<input checked="" type="checkbox"/>	Add	Room Main (Scope=InComponentClass('t To	In Dev Board.PcbDoc	

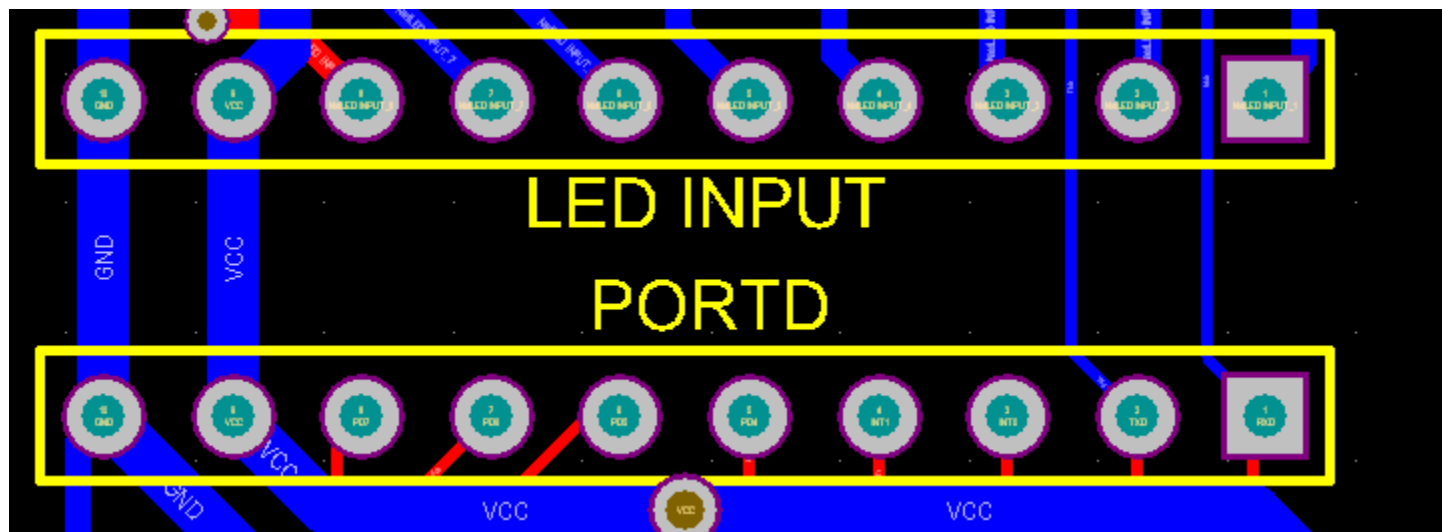
Cleaning Up The Design

- The changes have taken place and now each connector is more descriptive
- This method can also be used to change PCB footprints, etc...



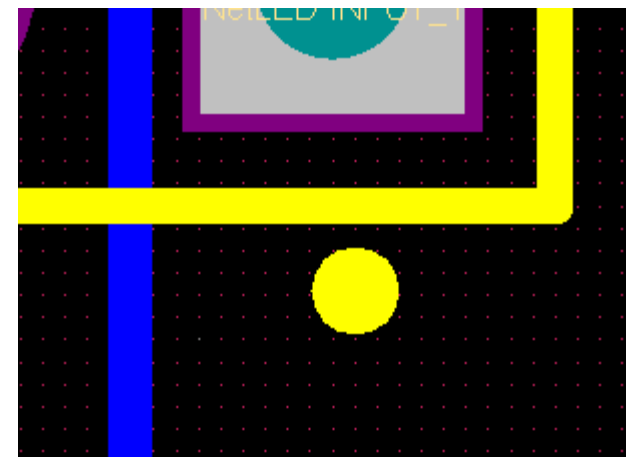
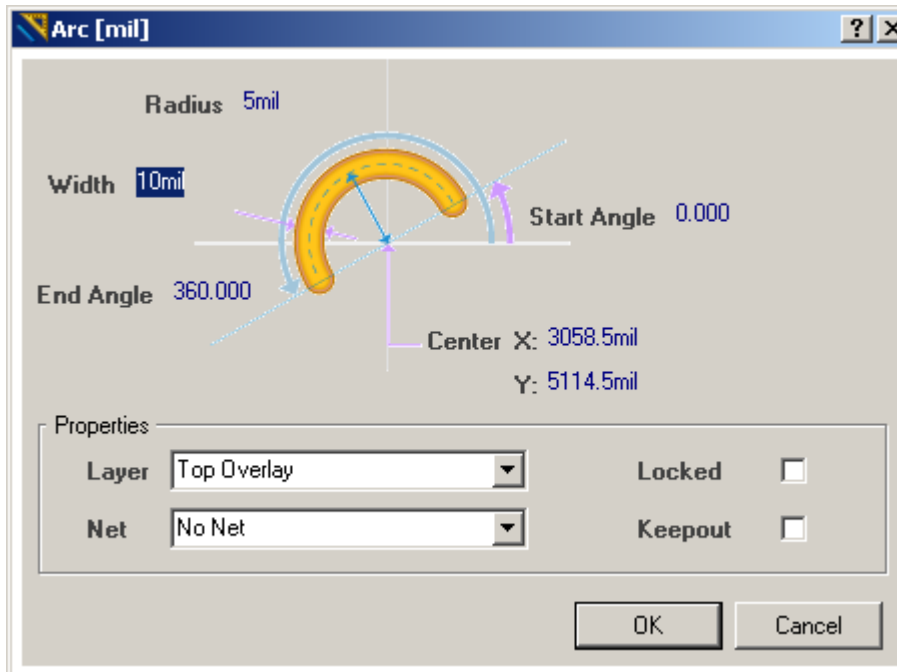
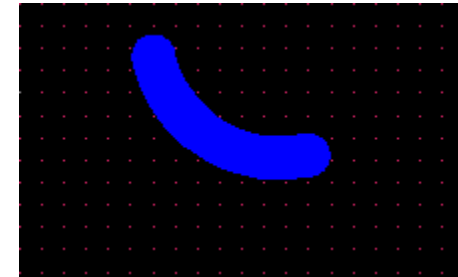
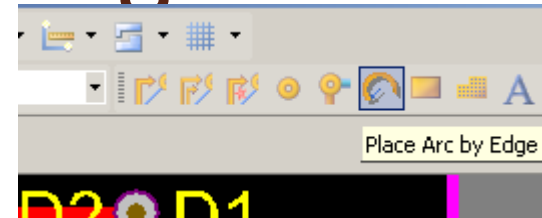
Cleaning Up The Design

- You can make the font larger or in bold, or change the font itself if you want to, just double click the text to bring up the menu
- These connectors will have pin headers on them once assembled, so the Pin 1 indicator with the rectangular pad will be invisible to us. We need to add a better Pin 1 indicator.



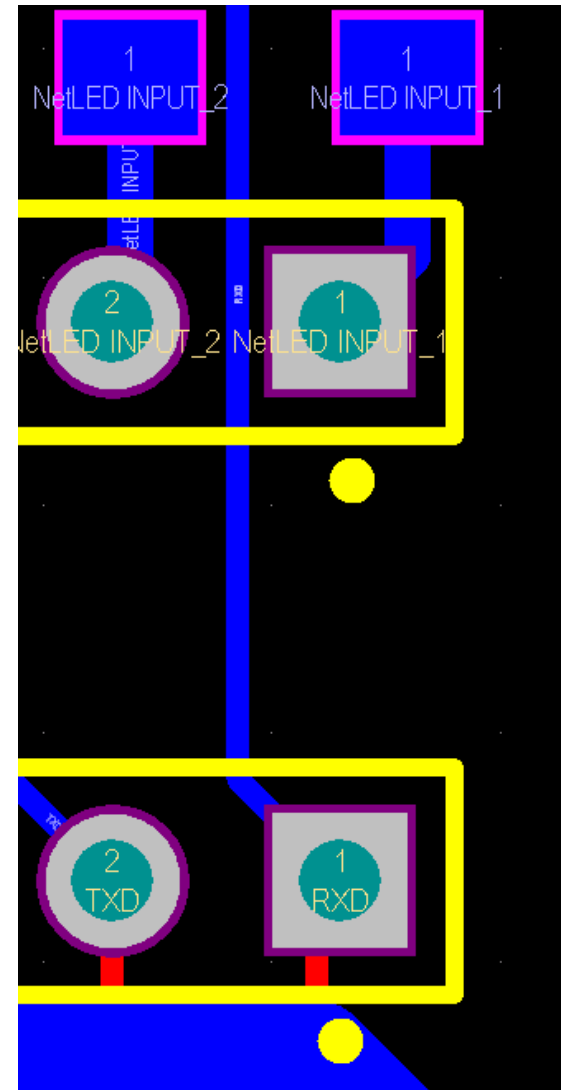
Cleaning Up The Design

- Click the place arc by edge
- Place an arc anywhere on the board
- Double click on the arc to get the prompt below, change:
 - Radius = 5 mil
 - Width = 10 mil
 - Start Angle = End Angle = 0
 - Layer = Top Overlay



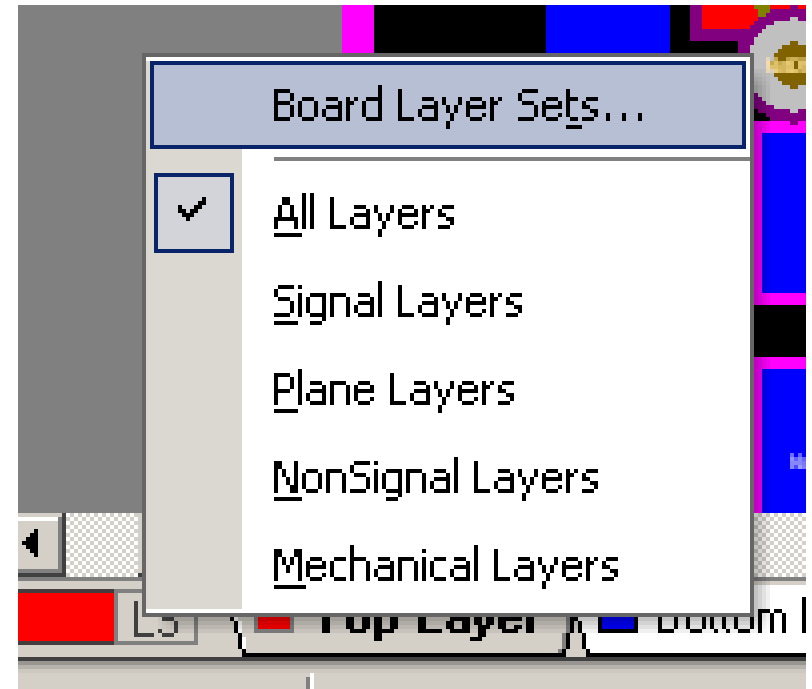
Cleaning Up The Design

- Copy & Paste the arc and place next to the Pin 1's for the 10 pin headers



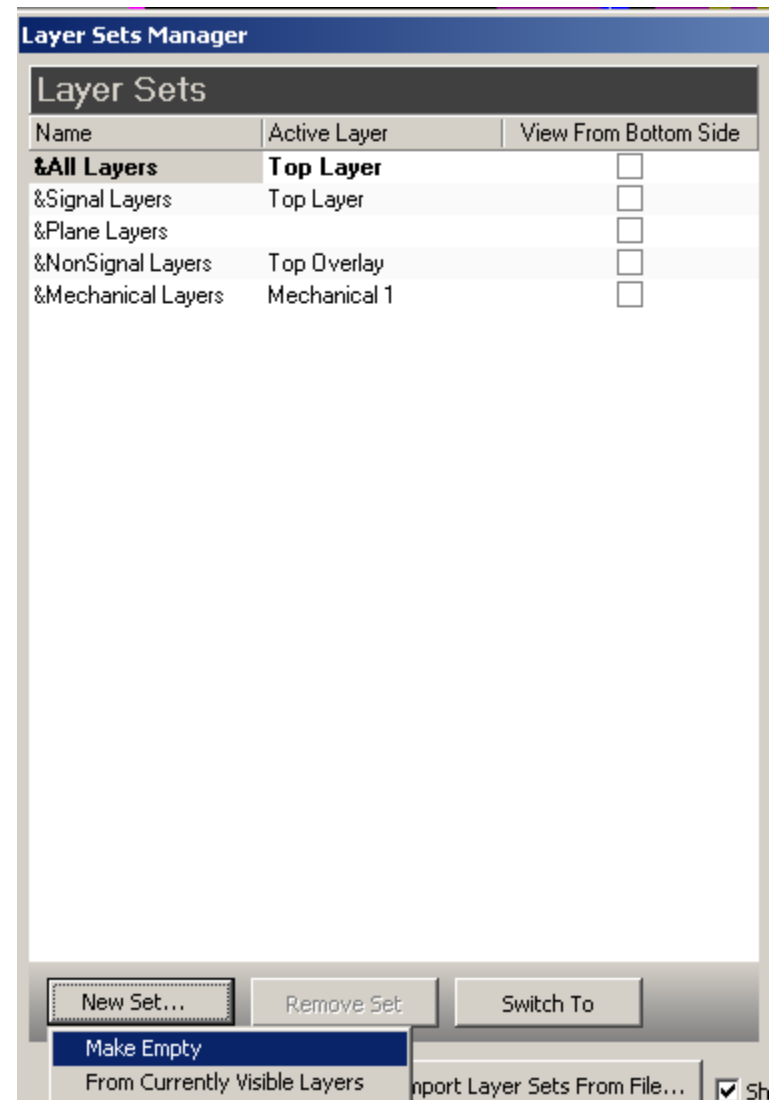
Cleaning Up The Design

- We should create two new board layer sets.
- One where only items on the TOP of the board are visible
- One where items on the BOT of the board are visible
- Click the 'LS' at the bottom of the screen and click 'Board Layer Sets'



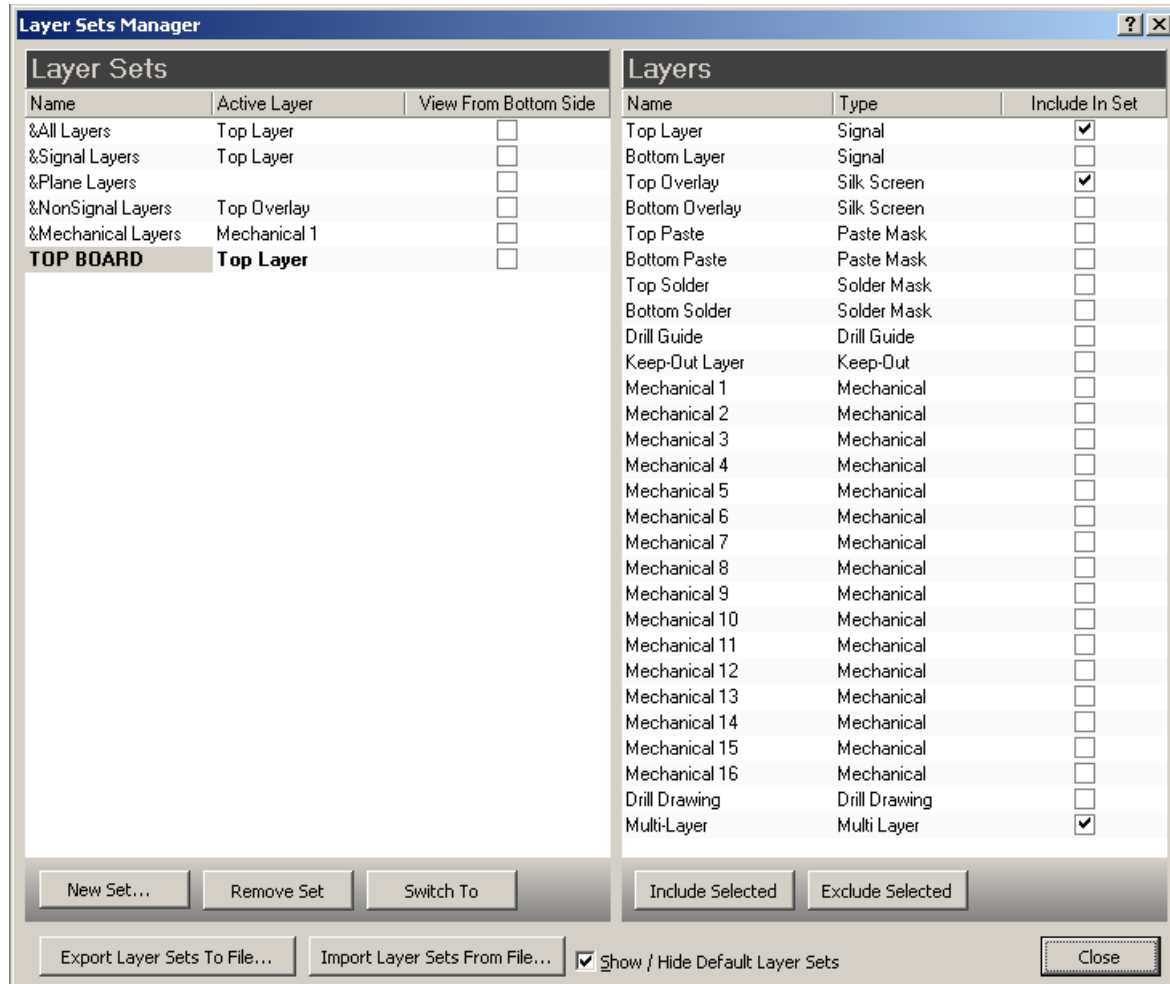
Cleaning Up The Design

- Click 'New Set' in the bottom left
- Click 'Make Empty'



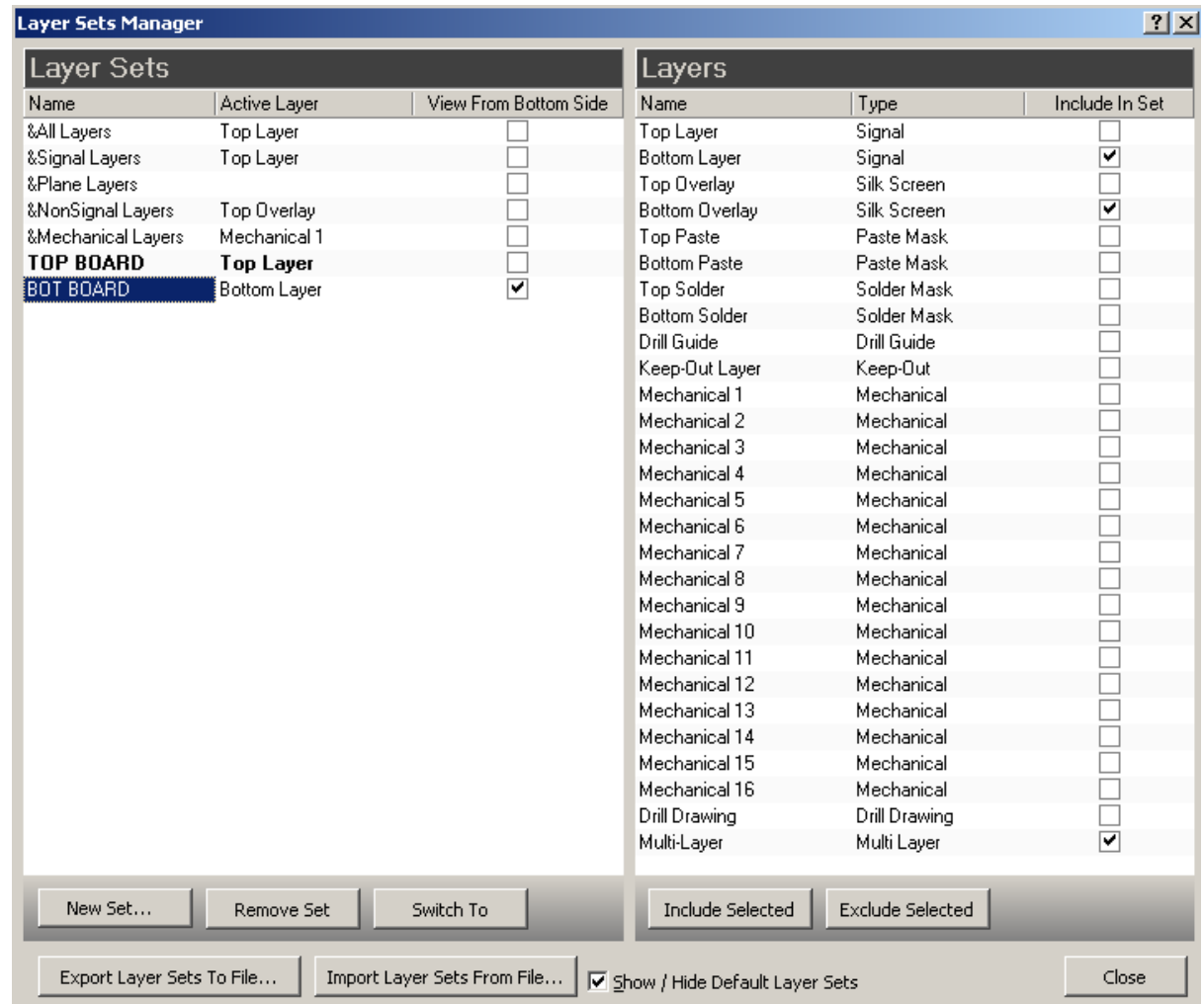
Cleaning Up The Design

- Name the set 'TOP BOARD'
- Check the following boxes:
 - Top Layer
 - Top Overlay
 - Multi-Layer



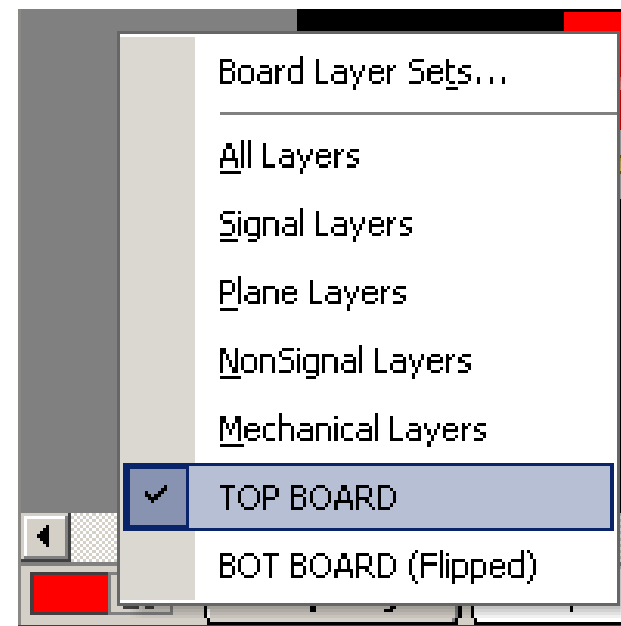
Cleaning Up The Design

- Create another new set called 'BOT BOARD'
- Check the 'View from bottom side'
- Check the following boxes:
 - Top Layer
 - Top Overlay
 - Multi-Layer



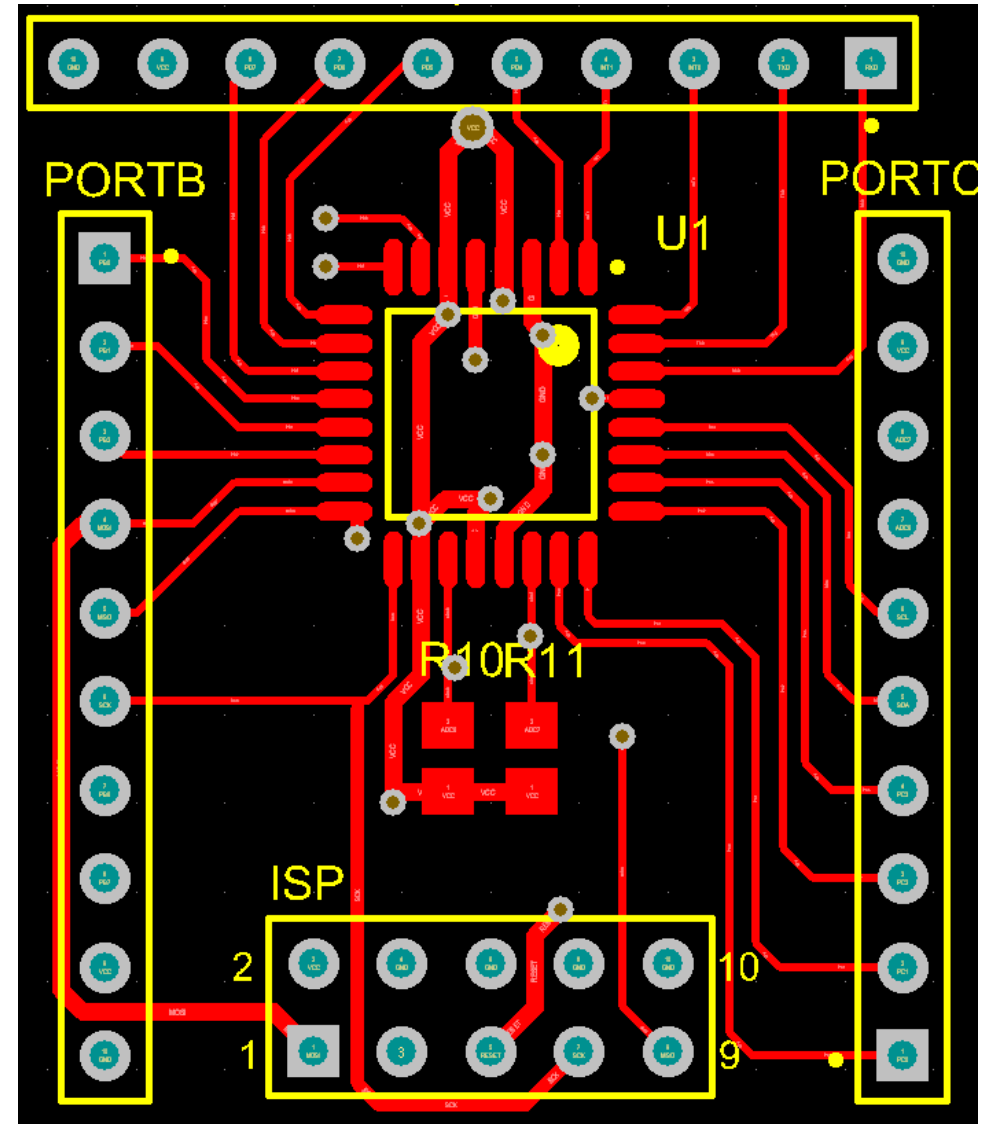
Cleaning Up The Design

- Select the TOP BOARD from the layer set.
- This is what the real board will look like from the top looking down without the default Superman ability to see through all the layers



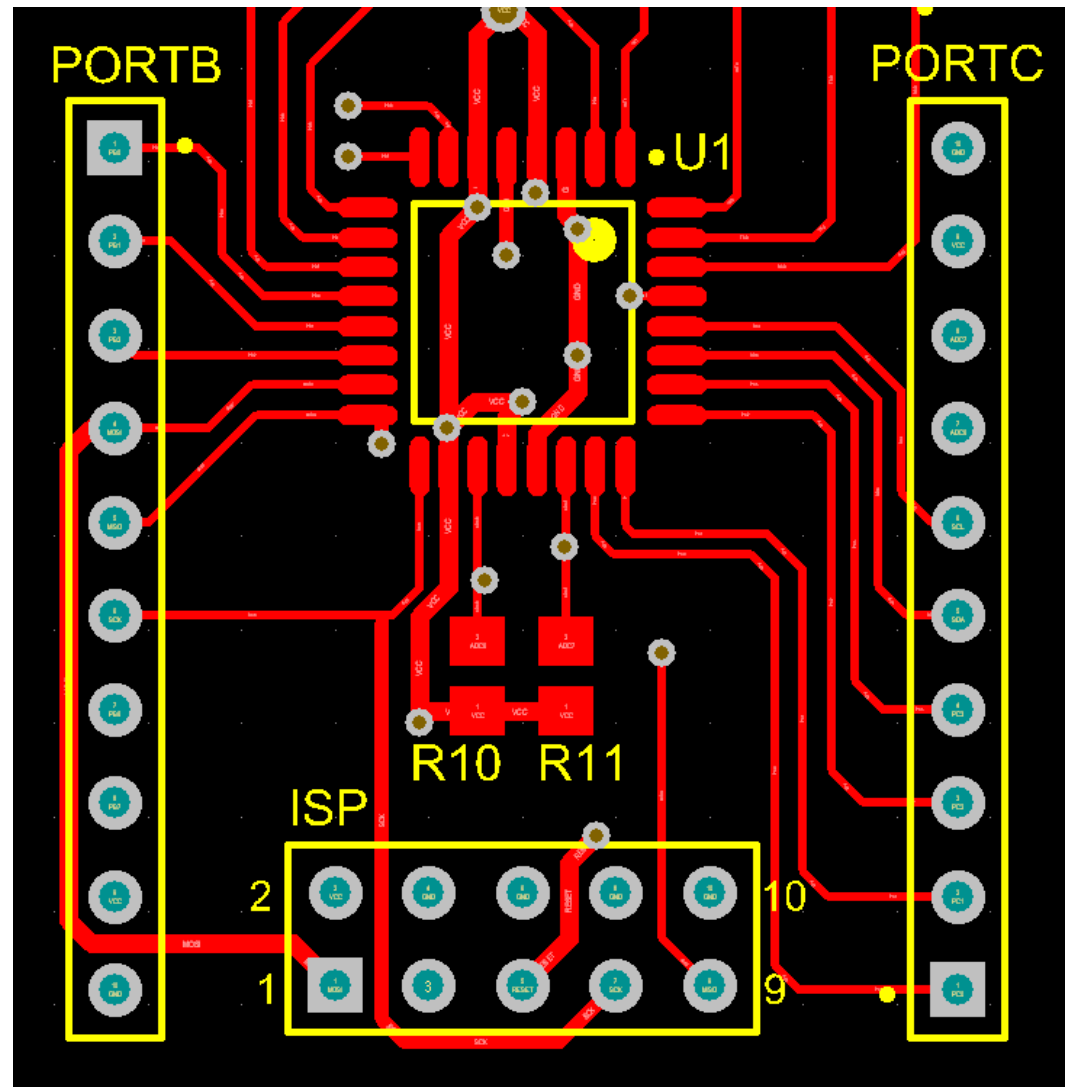
Cleaning Up The Design

- This is the best view to move the top overlay/silkscreen layers around to make sure it will look good when you are done.
- NOTE: You should avoid the holes with the top overlay, but do not worry if the top overlay is on top of a trace



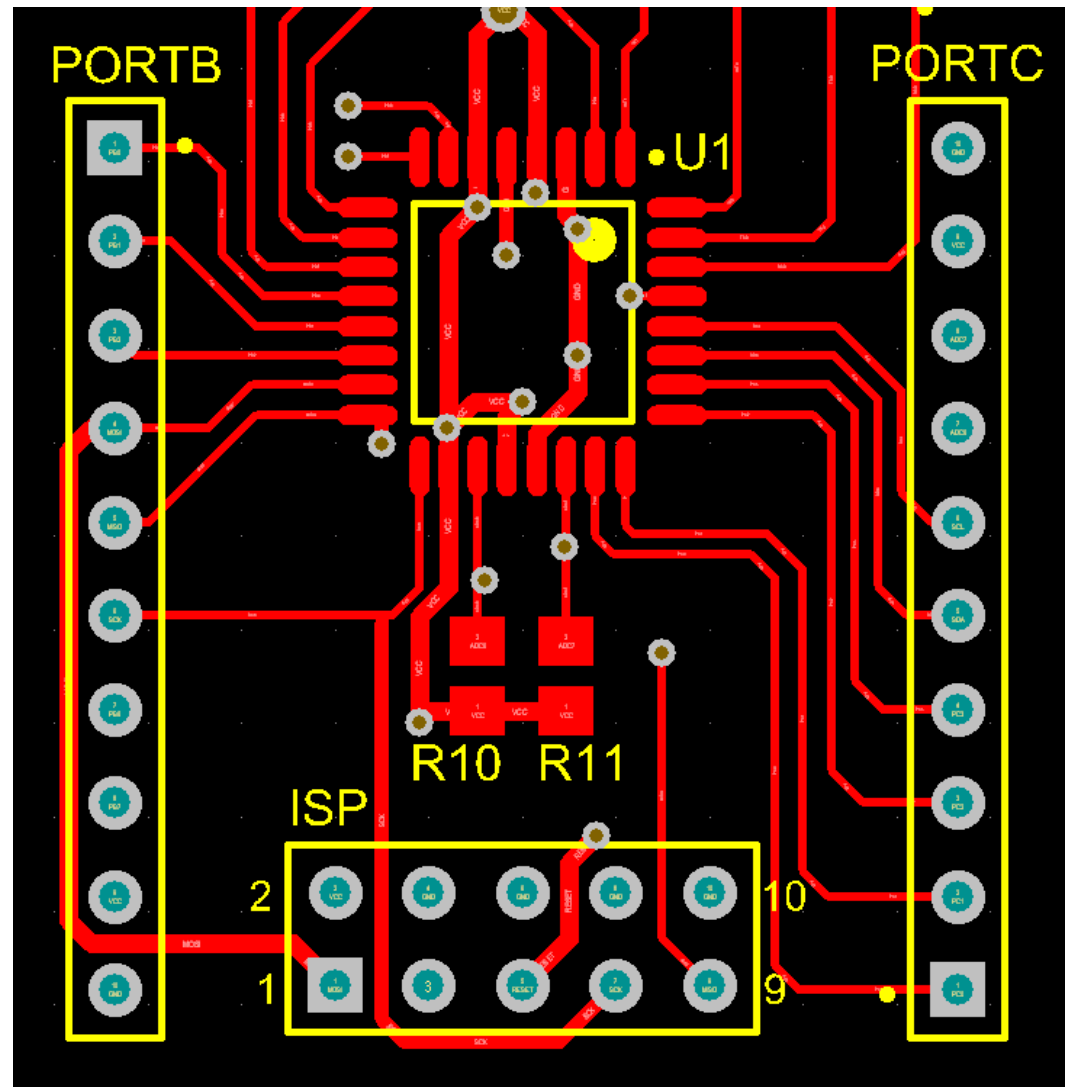
Cleaning Up The Design

- I have moved R10 and R11 indicators to a location with less clutter and without being blocked by a via.



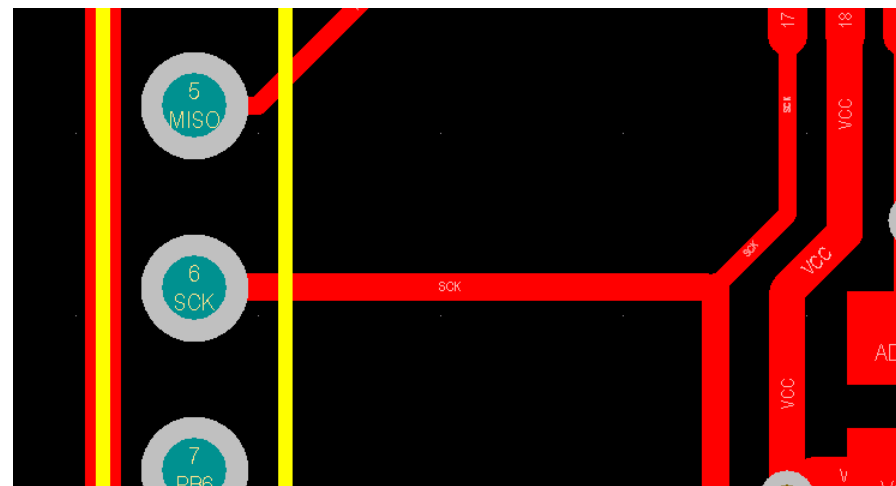
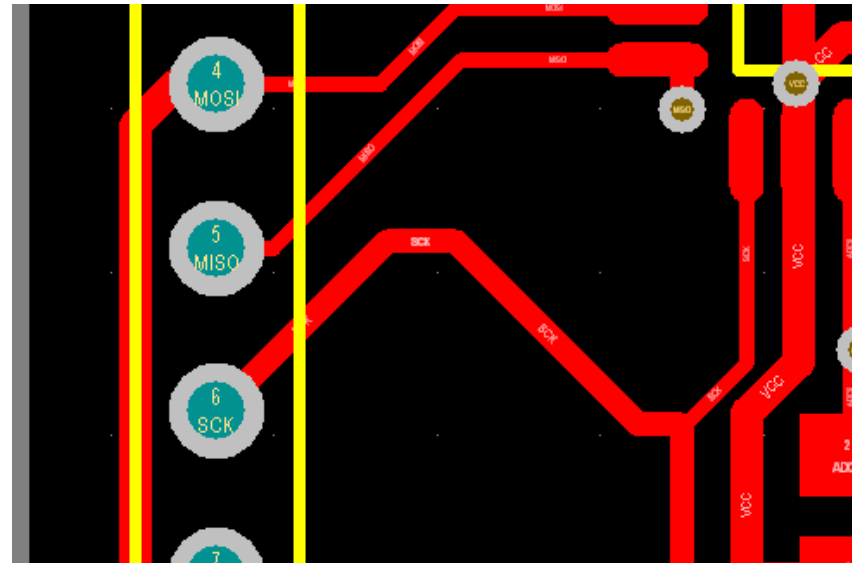
Cleaning Up The Design

- I have moved R10 and R11 indicators to a location with less clutter and without being blocked by a via.



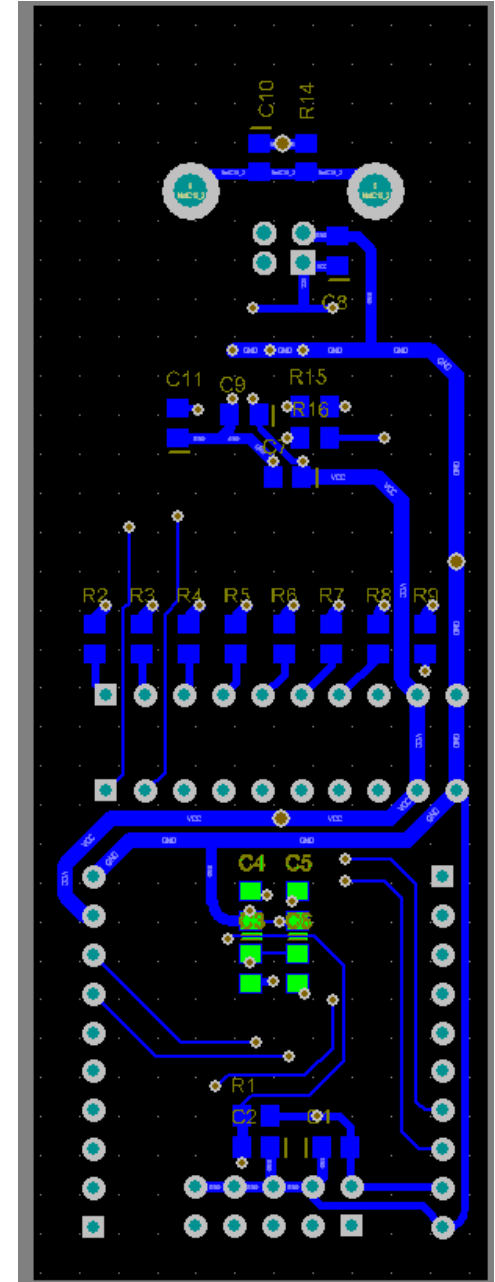
Cleaning Up The Design

- This is also the time where you should optimize your traces.
- The trace on the top image looks silly and was probably done because of optical interference with the bottom layer
- Viewing ONLY the top layer can help spot errors easily and make a clean and professional looking PCB



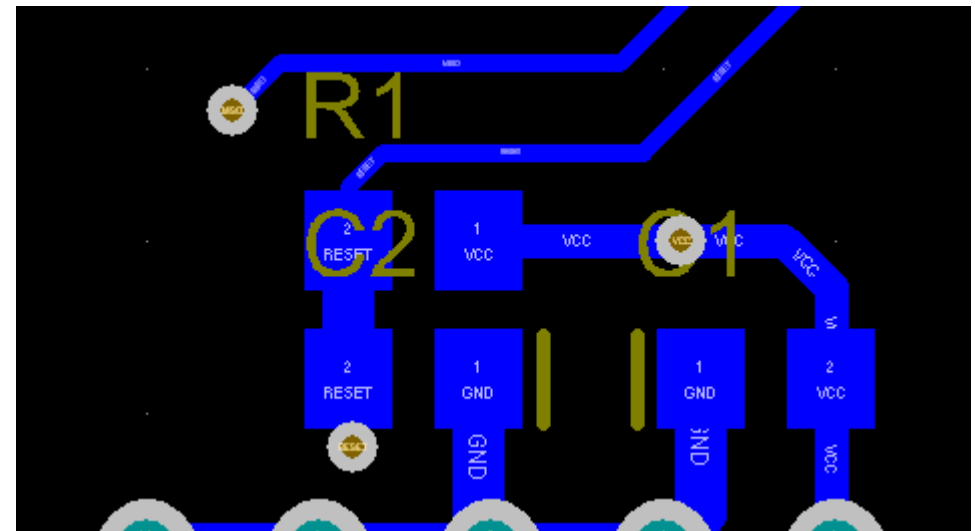
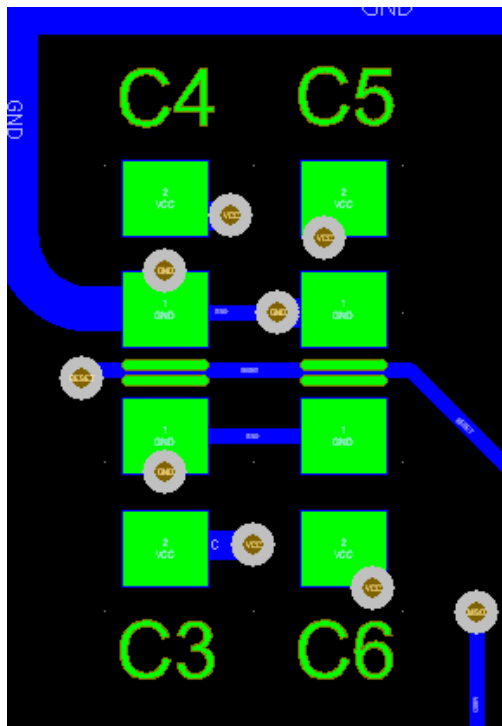
Cleaning Up The Design

- Set the active layer set to the BOT LAYER you made
- Perform the same optimizations as with the top layer



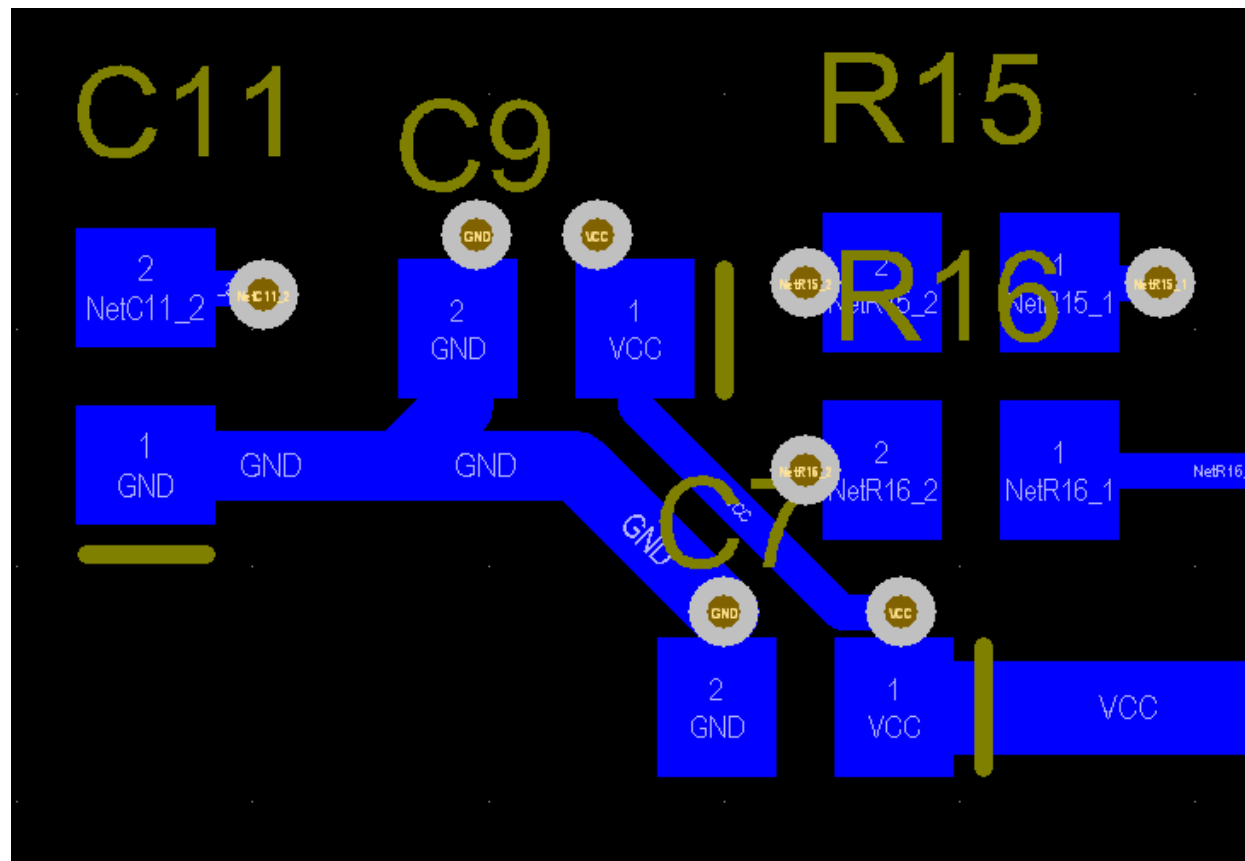
Cleaning Up The Design

- The image to the left shows a DRC error because the polarity marking for the capacitors are too close together. This is not a real error and we don't care about it
- The image on the right does not flag for an error, but the bottom overlay for CI will not show up properly because of the location of the via.
- Does anyone notice anything else wrong with the image to the right ???



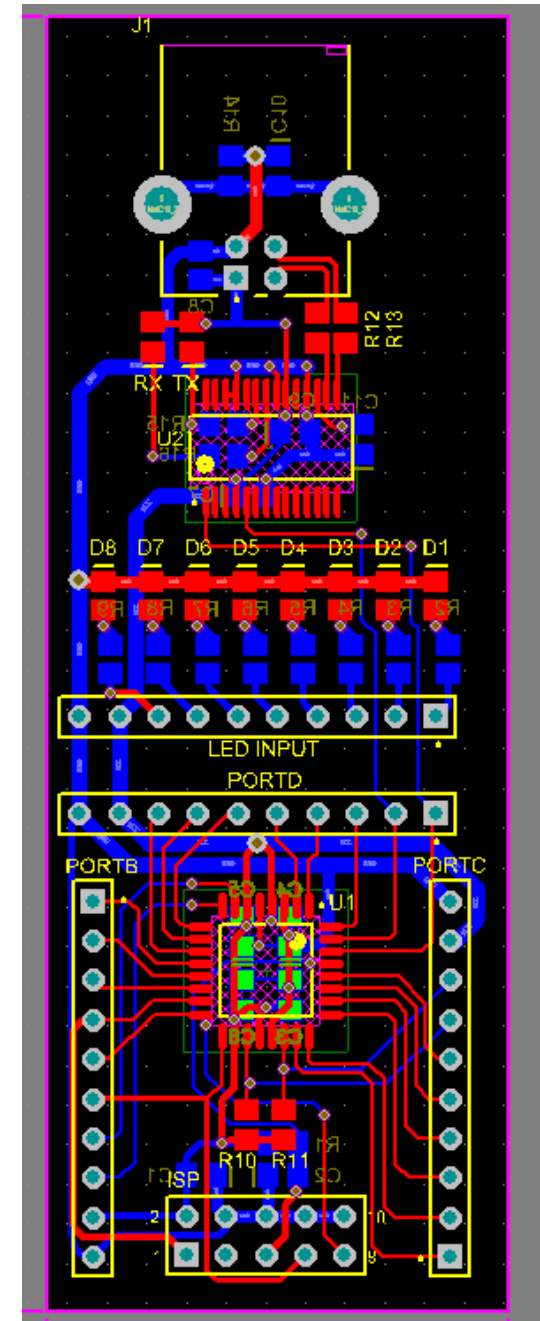
Cleaning Up The Design

- Is anything wrong with the bottom overlay in this picture ?



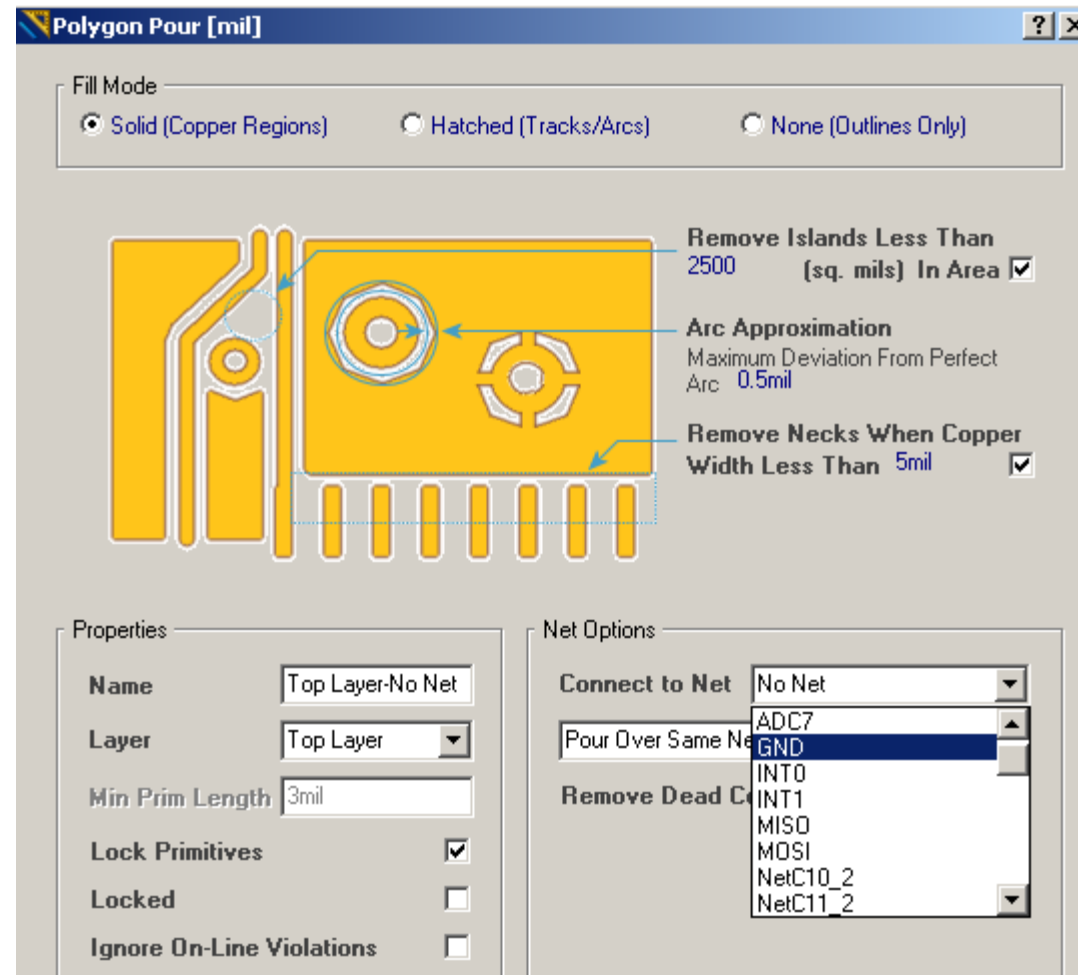
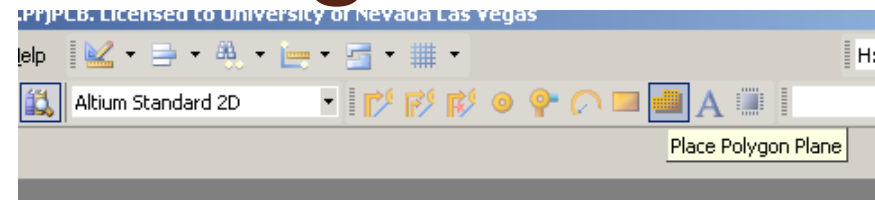
Cleaning Up The Design

- Our board looks OK, but there is a lot of unused space
- We will add what is called a 'copper pour' or 'polygon pour' to fill in the empty areas



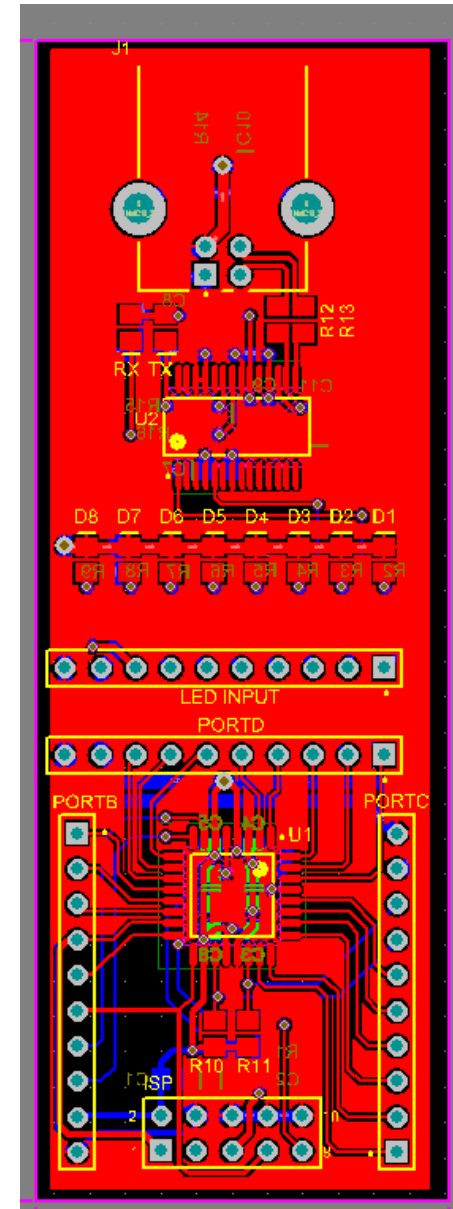
Cleaning Up The Design

- Click the polygon plane icon as shown to the right
- Change net to GND



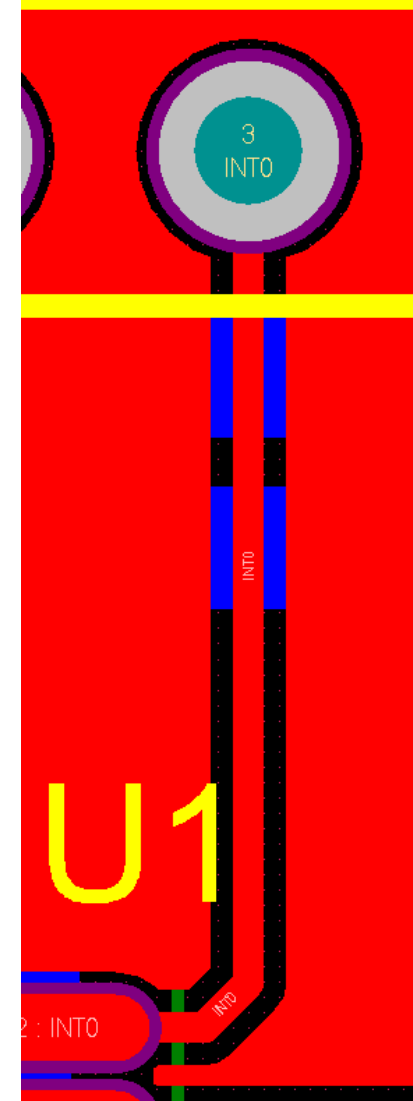
Cleaning Up The Design

- Click about 50-100 mils away from the each corner and then right click
- This will begin the polygon calculation and placement.
- NOTE: For large designs this can take hours



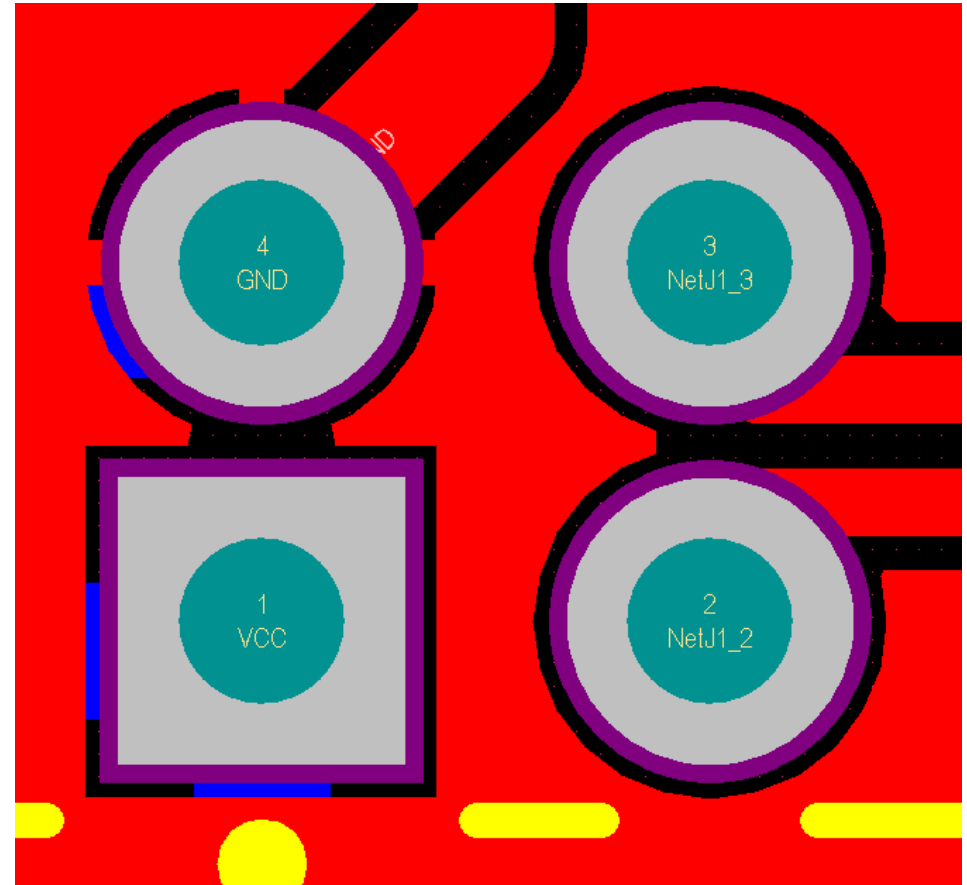
Cleaning Up The Design

- Notice how the polygon pour will avoid a net that is different than the net assigned to it.
- It will put copper everywhere possible that doesn't create a violation



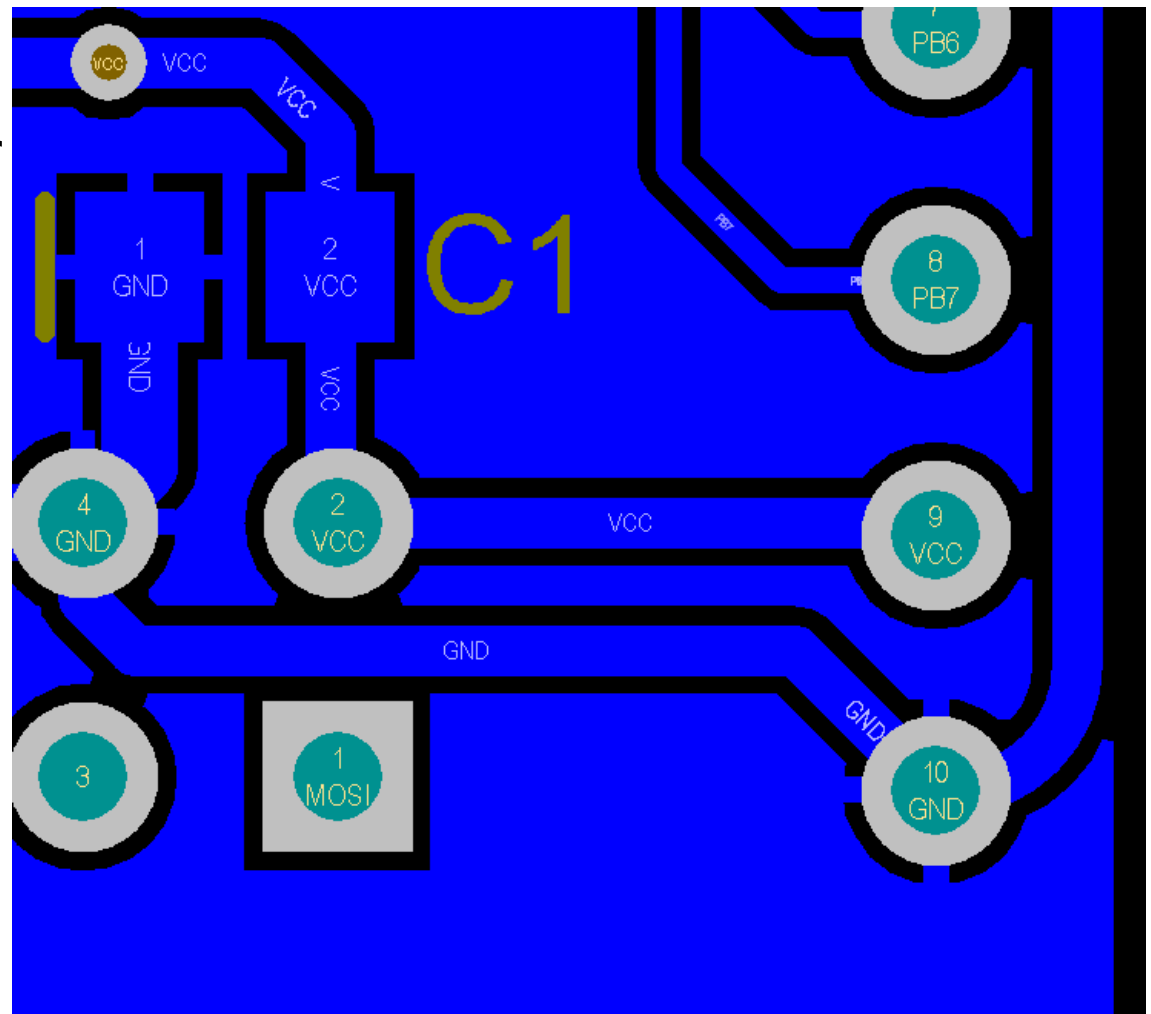
Cleaning Up The Design

- On nets that there is a connection (GND), you can see it handles it exactly the same as any other net, but has a few short connectors that connect the pad/hole to the pour



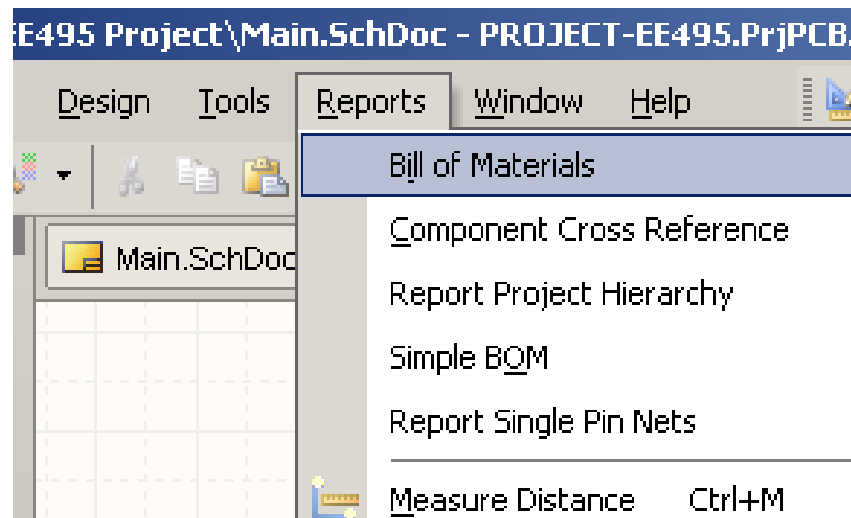
Cleaning Up The Design

- Repeat this process on the bottom layer of the board, also connecting to GND



PCB Complete!

- Rather than trying to manually figure out what to order, we can extract a bill of materials from the design
- This is why it was so important to enter all the details in correctly earlier
- Click 'Bill of Materials' as shown below



PCB Complete!

- Check the box to 'Open Exported' and click 'Export'

Bill of Materials For Project [PROJECT-EE495.PrjPCB] (No PCB Document Selected)

Grouped Columns	Show	Comment	Description	Designator	Footprint	LibRef	Quantity
Comment	<input checked="" type="checkbox"/>	0.1uF	Capacitor	C1, C2, C3, C4, C5	CC2012-0805	Cap	11
Footprint	<input checked="" type="checkbox"/>	BIT0	Typical INFRARED	D1	CD2012-0805	LED0	1
		BIT1	Typical INFRARED	D2	CD2012-0805	LED0	1
		BIT2	Typical INFRARED	D3	CD2012-0805	LED0	1
		BIT3	Typical INFRARED	D4	CD2012-0805	LED0	1
		BIT4	Typical INFRARED	D5	CD2012-0805	LED0	1
		BIT5	Typical INFRARED	D6	CD2012-0805	LED0	1
		BIT6	Typical INFRARED	D7	CD2012-0805	LED0	1
		BIT7	Typical INFRARED	D8	CD2012-0805	LED0	1
		Header 5X2	Header, 5-Pin, Dual	ISP	HDR2X5	Header 5X2	1
		1-1470156-1	USB 1.1, Right Angl	J1	1-1470156	1-1470156-1	1
		Header 10	Header, 10-Pin	LED INPUT	HDR1X10	Header 10	1
		PORTB	Header, 10-Pin	PORTB	HDR1X10	Header 10	1
		PORTC	Header, 10-Pin	PORTC	HDR1X10	Header 10	1
		PORTD	Header, 10-Pin	PORTD	HDR1X10	Header 10	1
		10k	Resistor	R1, R10, R11	CR2012-0805	Res1	3
		330	Resistor	R2, R3, R4, R5, R6	CR2012-0805	Res1	10
		27	Resistor	R12, R13	CR2012-0805	Res1	2
		1MEG	Resistor	R14	CR2012-0805	Res1	1
		RxLED	Typical INFRARED	RX	CD2012-0805	LED0	1
		TxLED	Typical INFRARED	TX	CD2012-0805	LED0	1
		ATmega8-16AC	8-Bit AVR Microcont	U1	32A_N	ATmega8-16AC	1
		FT232RL	FT232RL USB <>	U2	FT232RL	FT232RL	1

All Columns Show

Address1

Address2

Address3

Address4

ApprovedBy

Author

CheckedBy

Code_JEDEC

Comment

CompanyName

Component Kind

ComponentKind

ComponentLink1Des

ComponentLink1URI

ComponentLink2Des

ComponentLink2URI

ContactMaterial

ContactPlating

Export Options

File Format: Microsoft Excel Worksheet (*.xls)

Add to Project Open Exported

Excel Options

Template:

Relative Path to Template File

Menu Export... Force Columns to View Include Parameters From Database Include Parameters From PCB OK

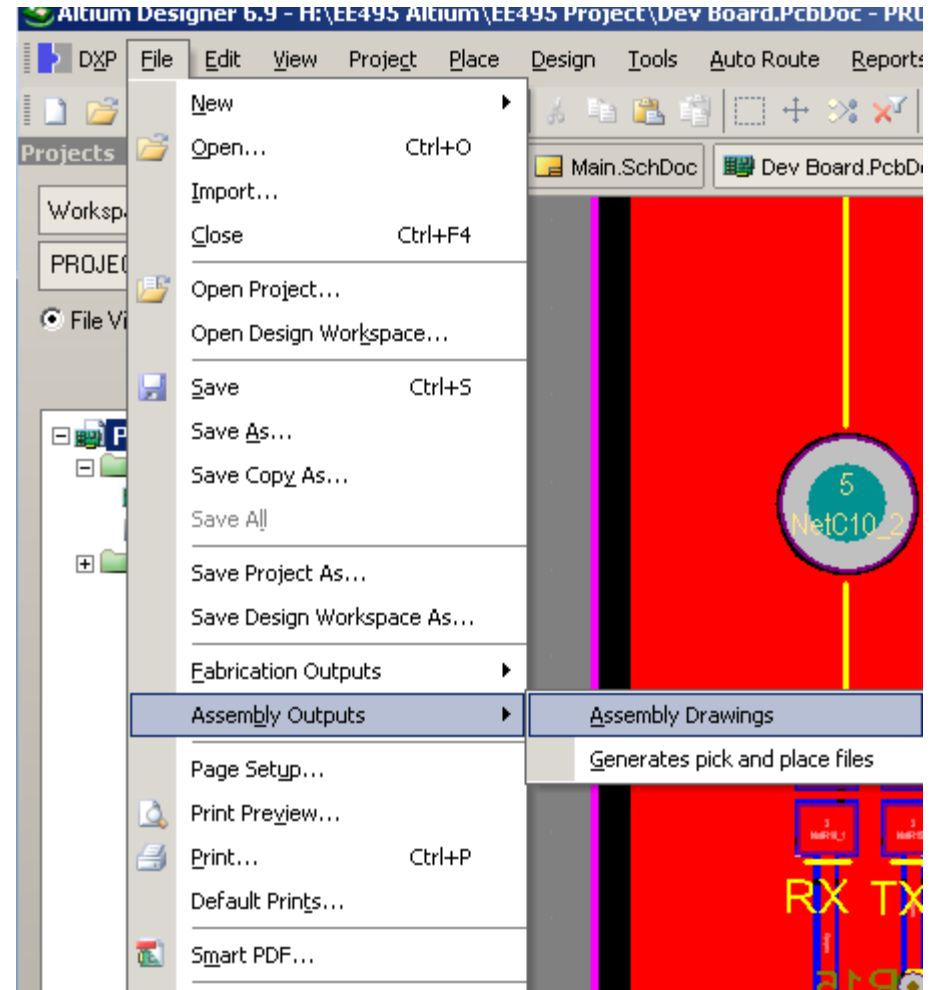
PCB Complete!

- An Excel spreadsheet of the B.O.M. will open with all the information you need to quickly order the parts

Comment	Description	Designator	Footprint	LibRef	Quantity
0.1uF	Capacitor	C1, C2, C3, C4, C5, C6, C7, C8, C9, C10, C11	CC2012-0805	Cap	11
BIT0	Typical INFRARED GaAs LED	D1	CD2012-0805	LED0	1
BIT1	Typical INFRARED GaAs LED	D2	CD2012-0805	LED0	1
BIT2	Typical INFRARED GaAs LED	D3	CD2012-0805	LED0	1
BIT3	Typical INFRARED GaAs LED	D4	CD2012-0805	LED0	1
BIT4	Typical INFRARED GaAs LED	D5	CD2012-0805	LED0	1
BIT5	Typical INFRARED GaAs LED	D6	CD2012-0805	LED0	1
BIT6	Typical INFRARED GaAs LED	D7	CD2012-0805	LED0	1
BIT7	Typical INFRARED GaAs LED	D8	CD2012-0805	LED0	1
Header 5X2	Header, 5-Pin, Dual row	ISP	HDR2X5	Header 5X2	1
1-1470156-1	USB 1.1, Right Angle, Thru-Hole, B Type	J1	1-1470156	1-1470156-1	1
Header 10	Header, 10-Pin	LED INPUT	HDR1X10	Header 10	1
PORTB	Header, 10-Pin	PORTB	HDR1X10	Header 10	1
PORTC	Header, 10-Pin	PORTC	HDR1X10	Header 10	1
PORTD	Header, 10-Pin	PORTD	HDR1X10	Header 10	1
10k	Resistor	R1, R10, R11	CR2012-0805	Res1	3
330	Resistor	R2, R3, R4, R5, R6, R7, R8, R9, R15, R16	CR2012-0805	Res1	10
27	Resistor	R12, R13	CR2012-0805	Res1	2
1MEG	Resistor	R14	CR2012-0805	Res1	1
RXLED	Typical INFRARED GaAs LED	RX	CD2012-0805	LED0	1
TXLED	Typical INFRARED GaAs LED	TX	CD2012-0805	LED0	1
ATmega8-16AC	8-Bit AVR Microcontroller with 8K Bytes of	U1	32A_N	ATmega8-16AC	1
FT232RL	FT232RL USB <> UART	U2	FT232RL	FT232RL	1

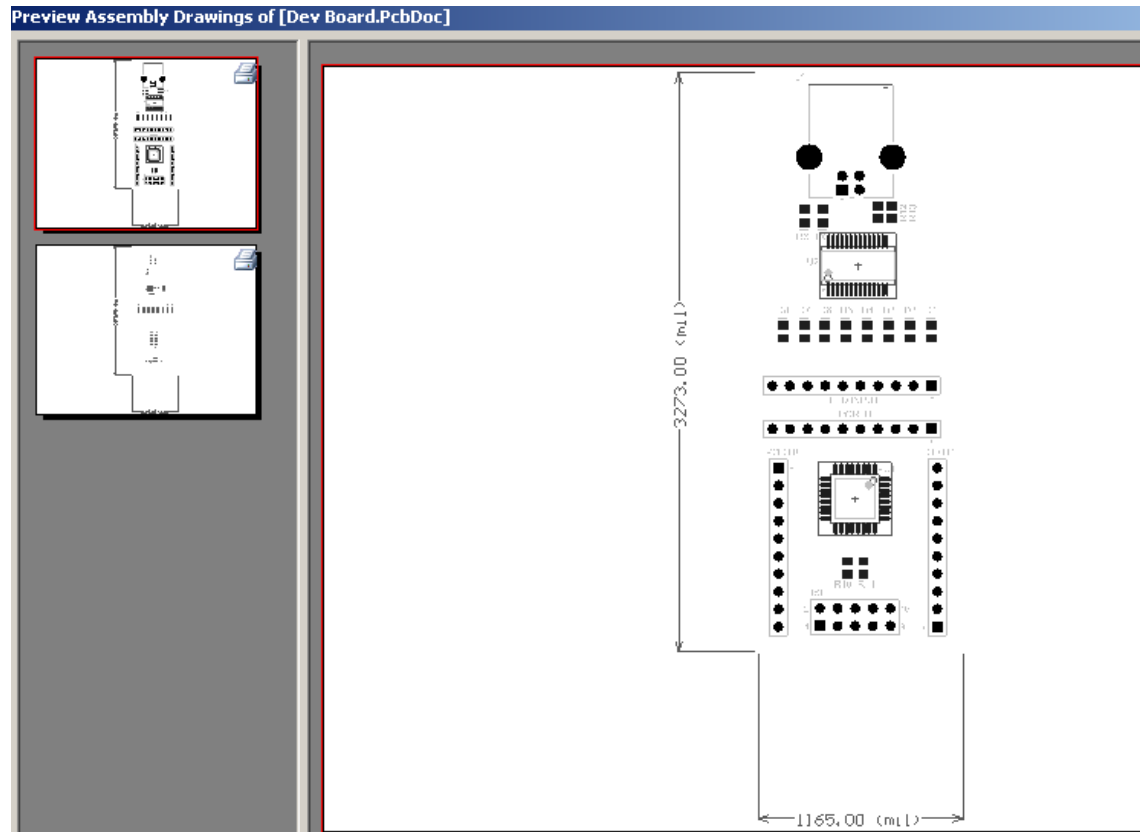
PCB Complete!

- From the PCB, click File > Assembly Outputs > Assembly Drawings



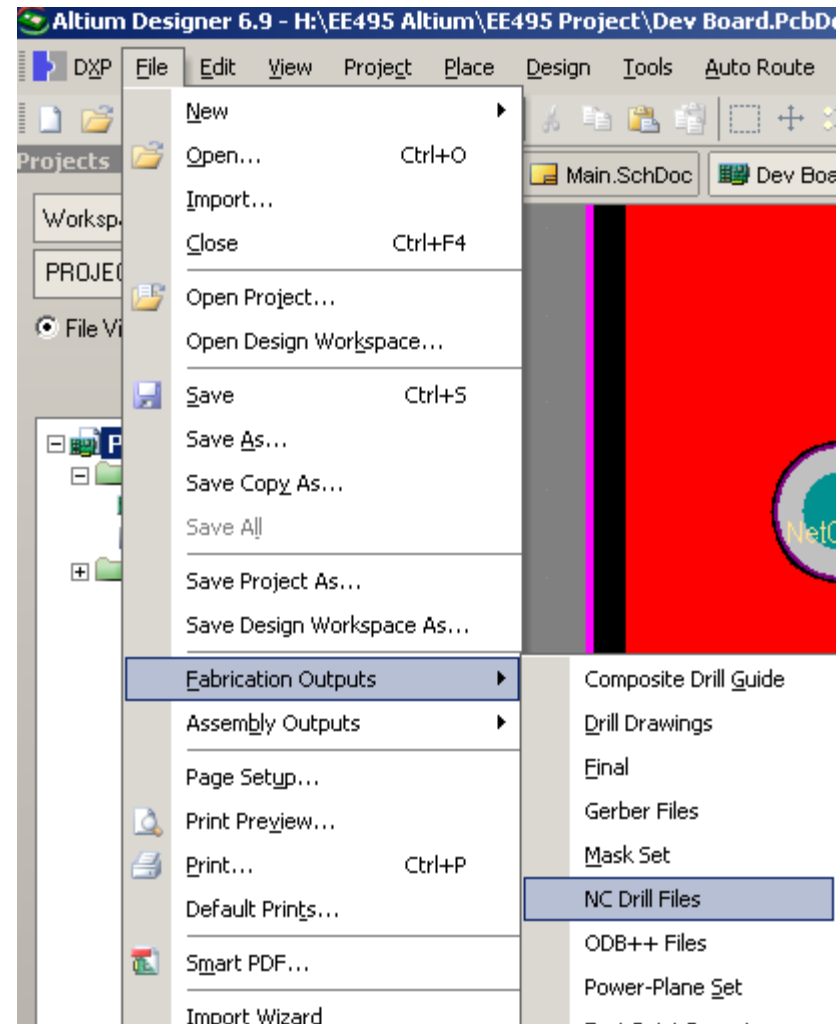
PCB Complete!

- This will create Assembly Drawings that are helpful when assembling the parts.
- Usually I print these and write the value of each component where the component goes



PCB Complete!

- To get the PCB fabricated the board house needs two things:
 - **Gerber Files**
 - **NC Drill Files**
- Follow the menu sequence to obtain the NC Drill files first



PCB Complete!

- Leave these values as default unless you know what you are doing
- Click OK

NC Drill Setup

Options

NC Drill Format
Specify the units and format to be used in the NC Drill output files.

This controls the units (inches or millimeters), and the number of digits before and after the decimal point.

Units
 Inches
 Millimeters

Format
 2:3
 2:4
 2:5

The number format should be set to suit the requirements of your design. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution. If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are holes on a grid finer than 1 mil.

Leading/Trailing Zeroes
 Keep leading and trailing zeroes
 Suppress leading zeroes
 Suppress trailing zeroes

Coordinate Positions
 Reference to absolute origin
 Reference to relative origin

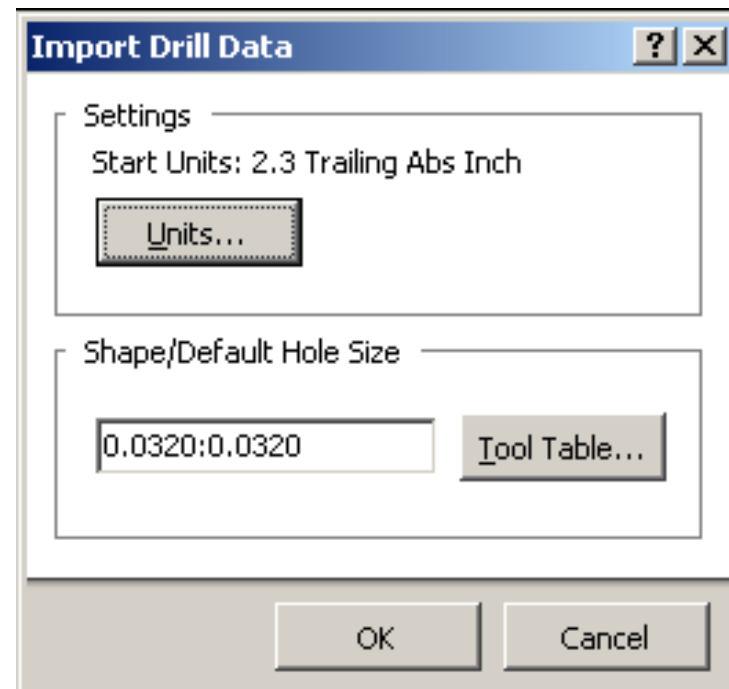
Other
 Optimize change location commands
 Generate separate NC Drill files for plated & non-plated holes
 Use drilled slot command (G85)
 Generate Board Edge Rout Paths

Rout Tool Dia: 200mil

OK Cancel

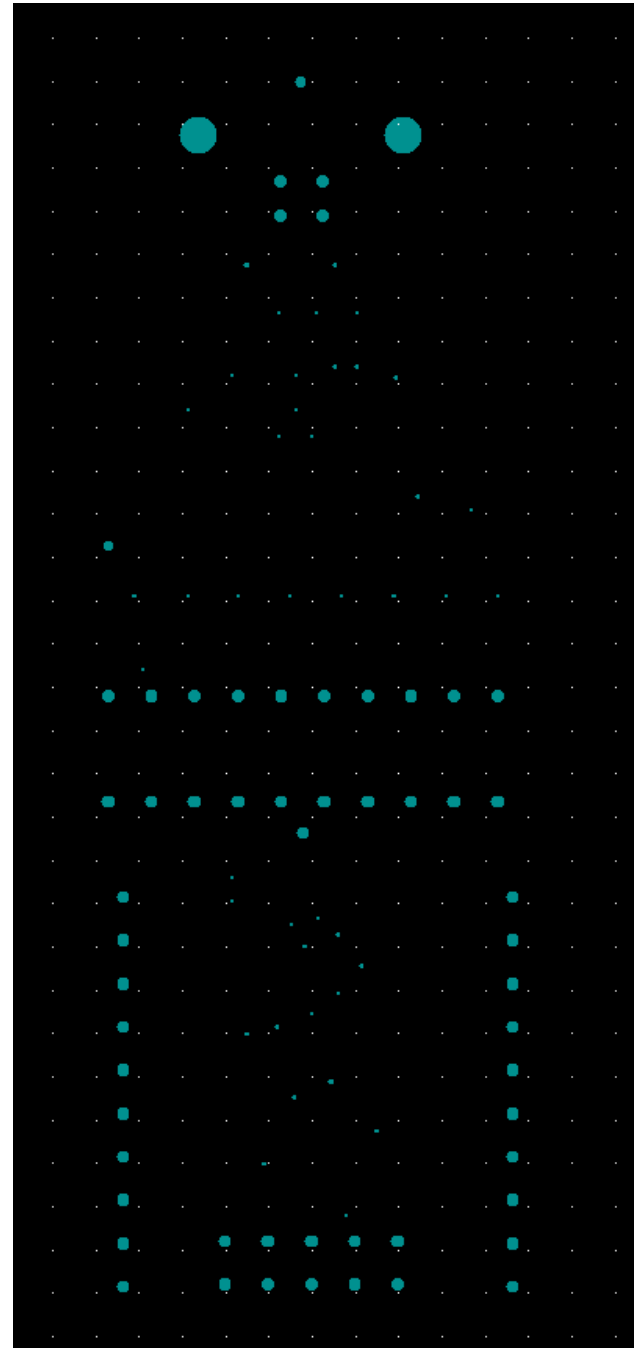
PCB Complete!

- Leave these values as default unless you know what you are doing
- Click OK



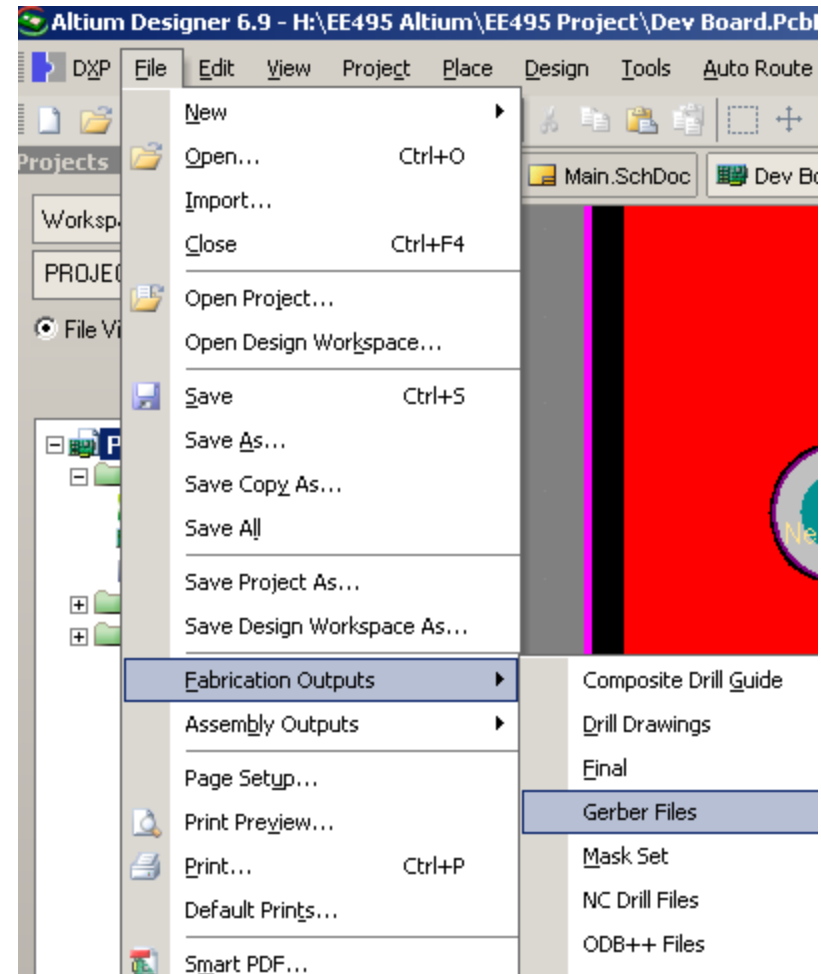
PCB Complete!

- A PCB with only holes in it should appear.
- These holes match the holes of your design
- Go back to the PCB



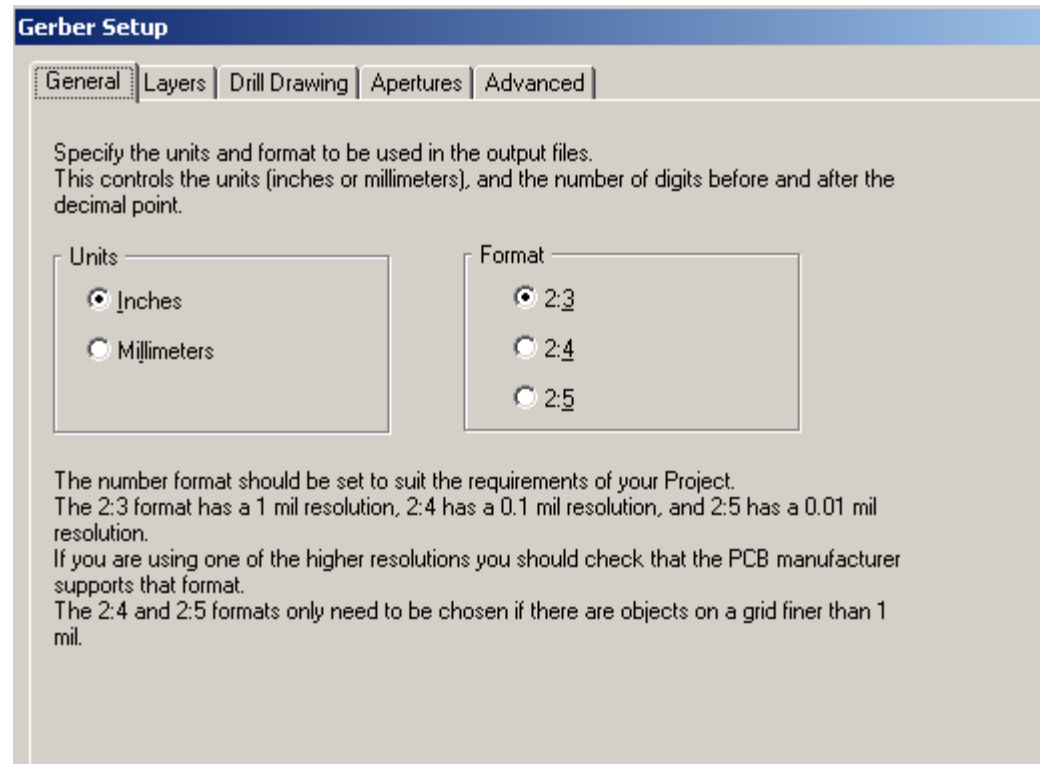
PCB Complete!

- Follow the menu sequence to create the output Gerber files



PCB Complete!

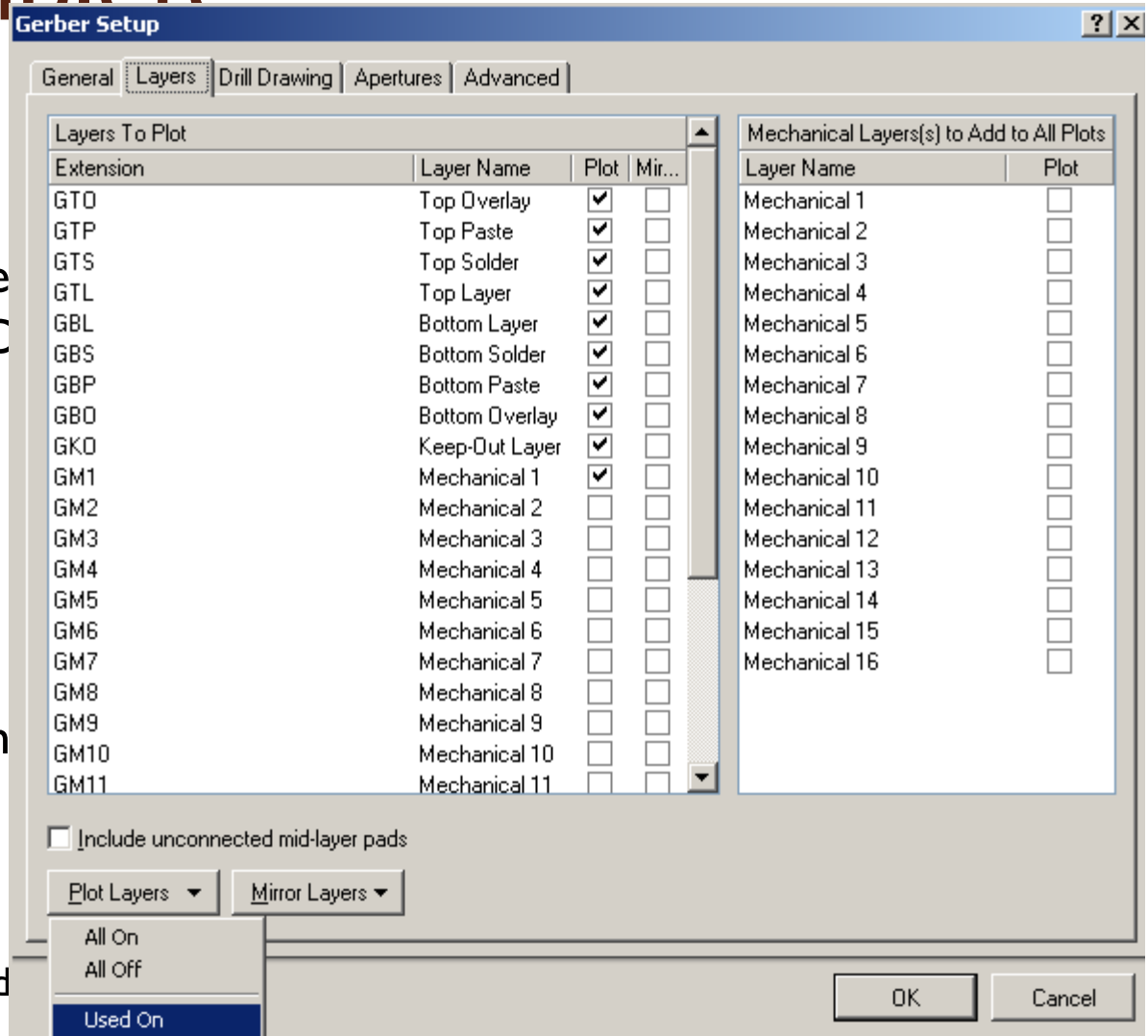
- Leave the values intact under the 'General' tab



The screenshot shows the 'Gerber Setup' dialog box with the 'General' tab selected. The dialog has a blue title bar and a grey background. At the top, there are five tabs: 'General', 'Layers', 'Drill Drawing', 'Apertures', and 'Advanced'. Below the tabs, there is a text box with the following text: 'Specify the units and format to be used in the output files. This controls the units (inches or millimeters), and the number of digits before and after the decimal point.' Below this text are two groups of radio buttons. The first group is labeled 'Units' and contains two options: 'Inches' (selected) and 'Millimeters'. The second group is labeled 'Format' and contains three options: '2:3' (selected), '2:4', and '2:5'. Below these groups is a paragraph of text: 'The number format should be set to suit the requirements of your Project. The 2:3 format has a 1 mil resolution, 2:4 has a 0.1 mil resolution, and 2:5 has a 0.01 mil resolution. If you are using one of the higher resolutions you should check that the PCB manufacturer supports that format. The 2:4 and 2:5 formats only need to be chosen if there are objects on a grid finer than 1 mil.'

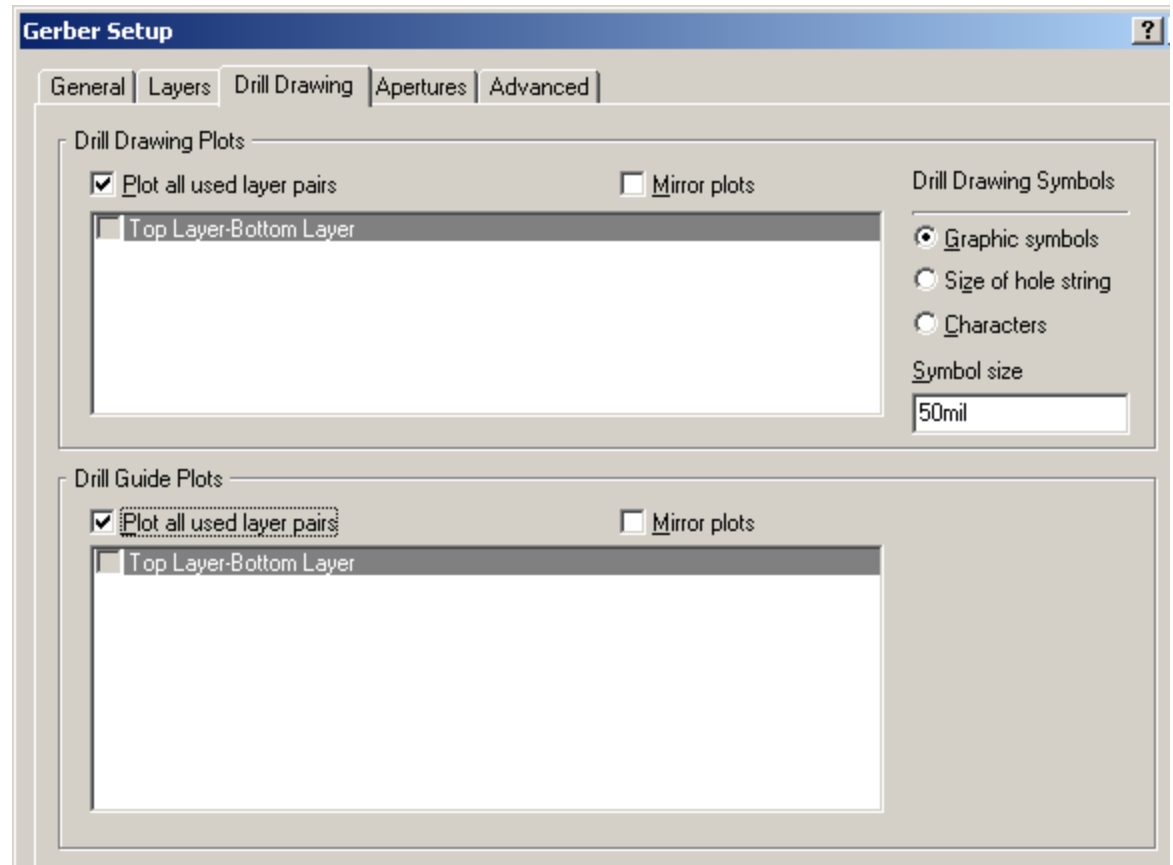
PCB Complete!

- Under the Layers tab, click 'Plot Layers' and you can either choose ALL ON or USED ON.
- If you choose USED ON you should manually check the bottom two layers called
 - GPT – Top Pad Master
 - GPB – Bottom Pad Master



PCB Complete!

- Under 'Drill Drawing' (This is different than NC Drill files) check the 'plot all used layer pairs' in the two locations at the left

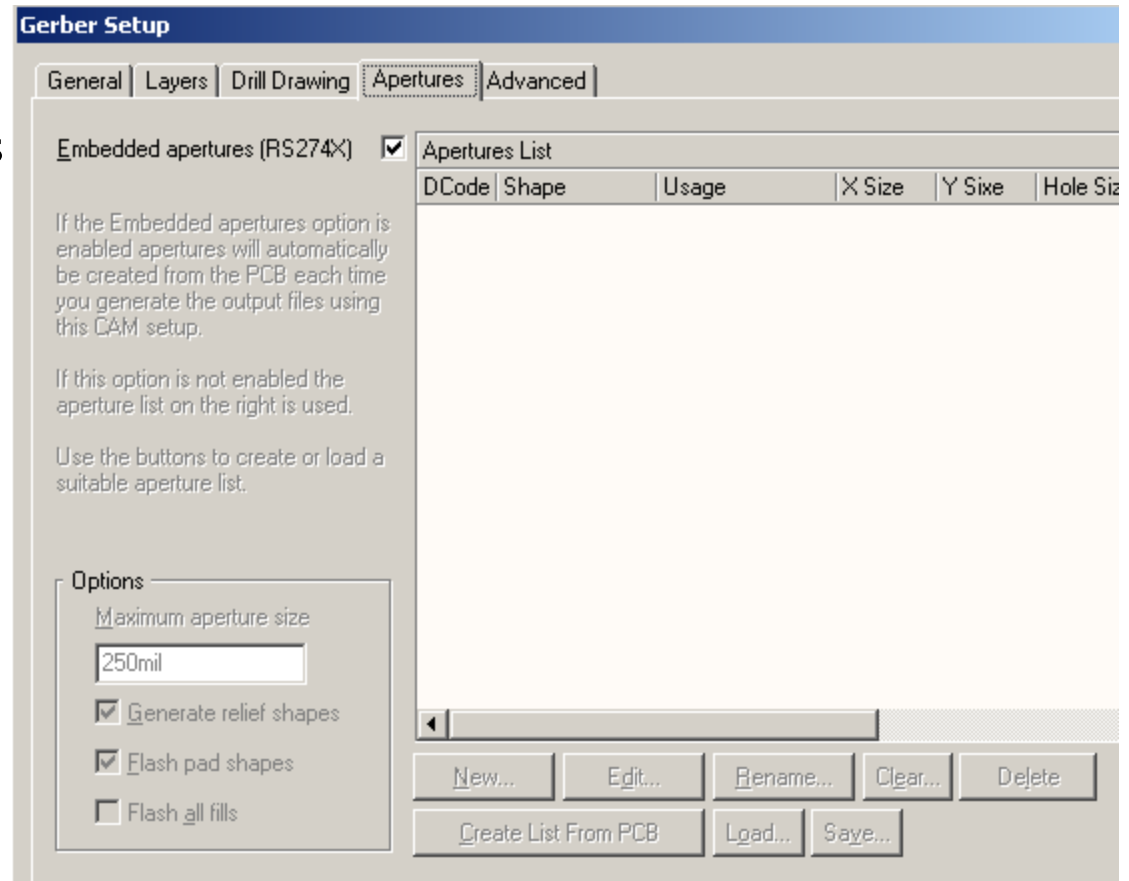


PCB Complete!

- Under 'Apertures' make sure Embedded Apertures are selected (RS274X)

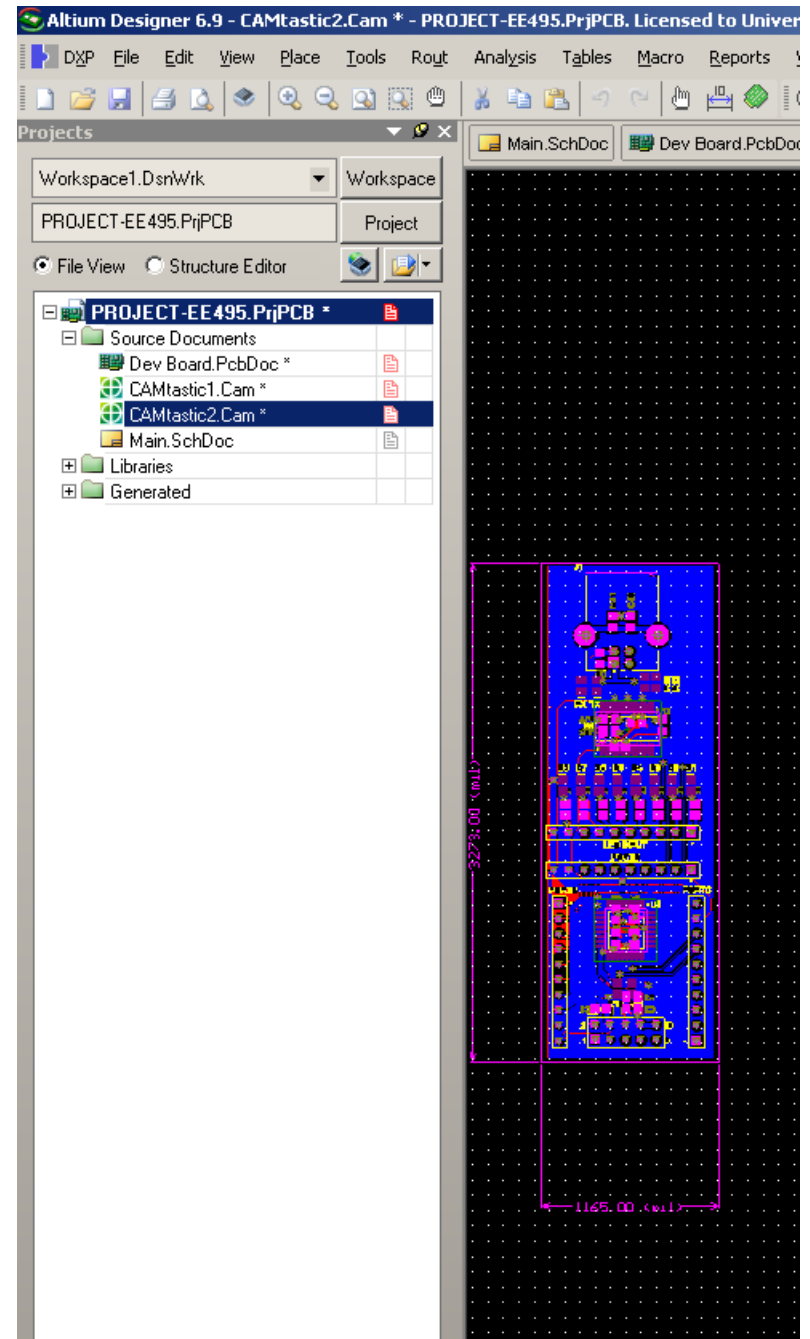
- NOTE: Before RS274X, there was RS274D, which did not contain the embedded Apertures, most everyone requires **RS274-X format**

- Ignore the Advanced tab and click OK



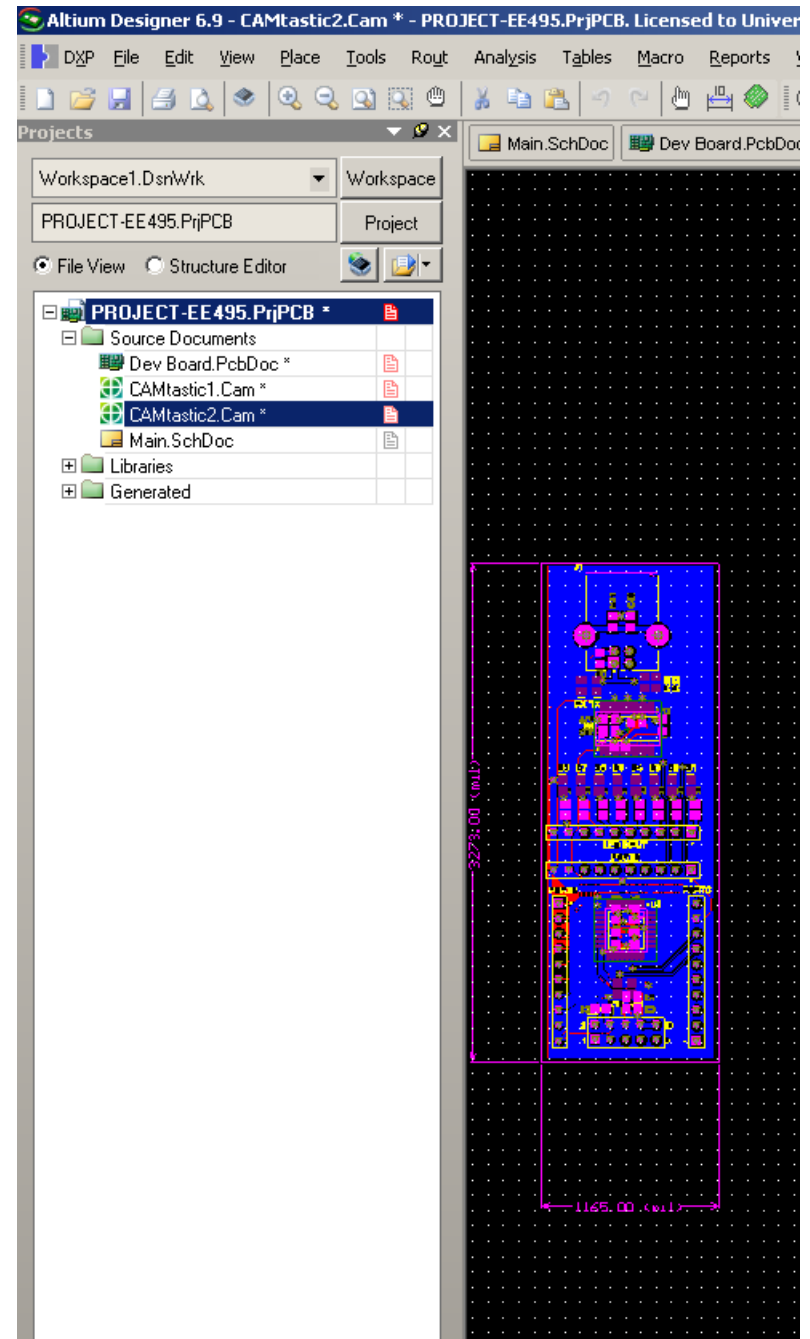
PCB Complete!

- Again, depending on the complexity of the design and the capabilities of your computer, this could take hours.
- This design should take around 10 seconds
- Notice how two new files have been created, These are CAM files



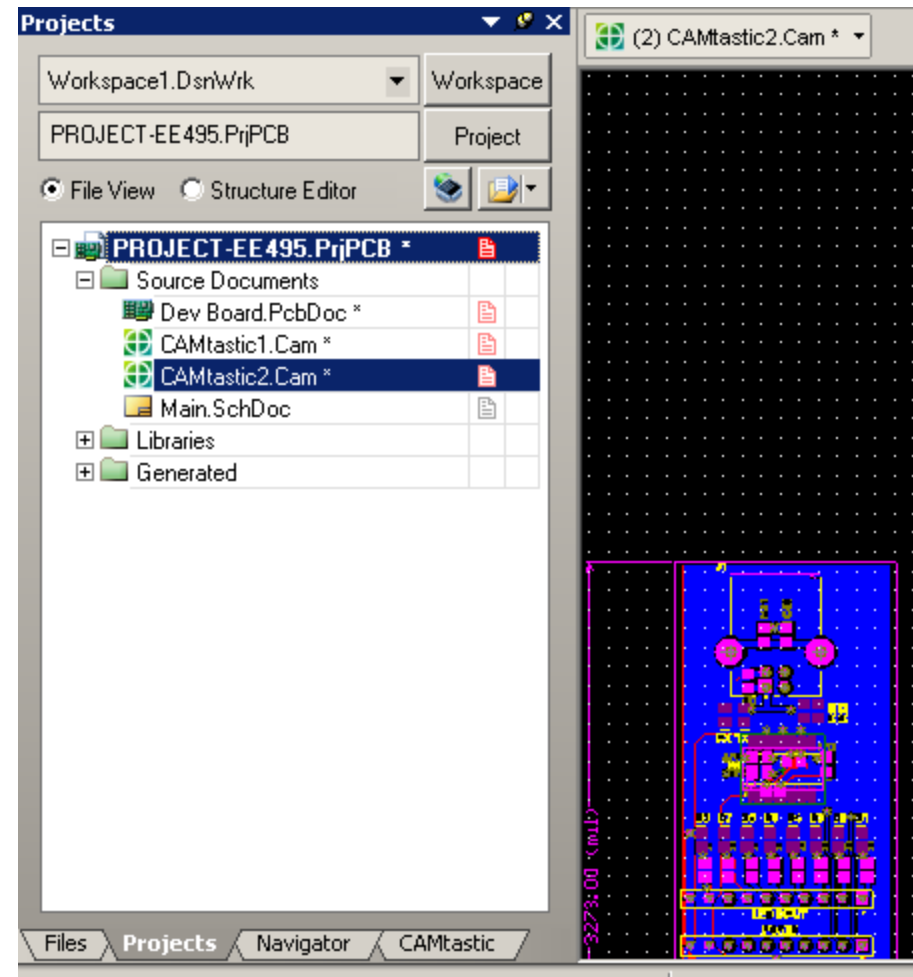
PCB Complete!

- The PCB Document is a single document
- The Gerber files are a list of around 10-40 individual files with information on each layer
- This standardized format of Gerber files means the board house does not need to own every software package imaginable
- CAMViewers are required to open these 'generic' Gerber files. CAMtastic is Altiums included CAM viewer



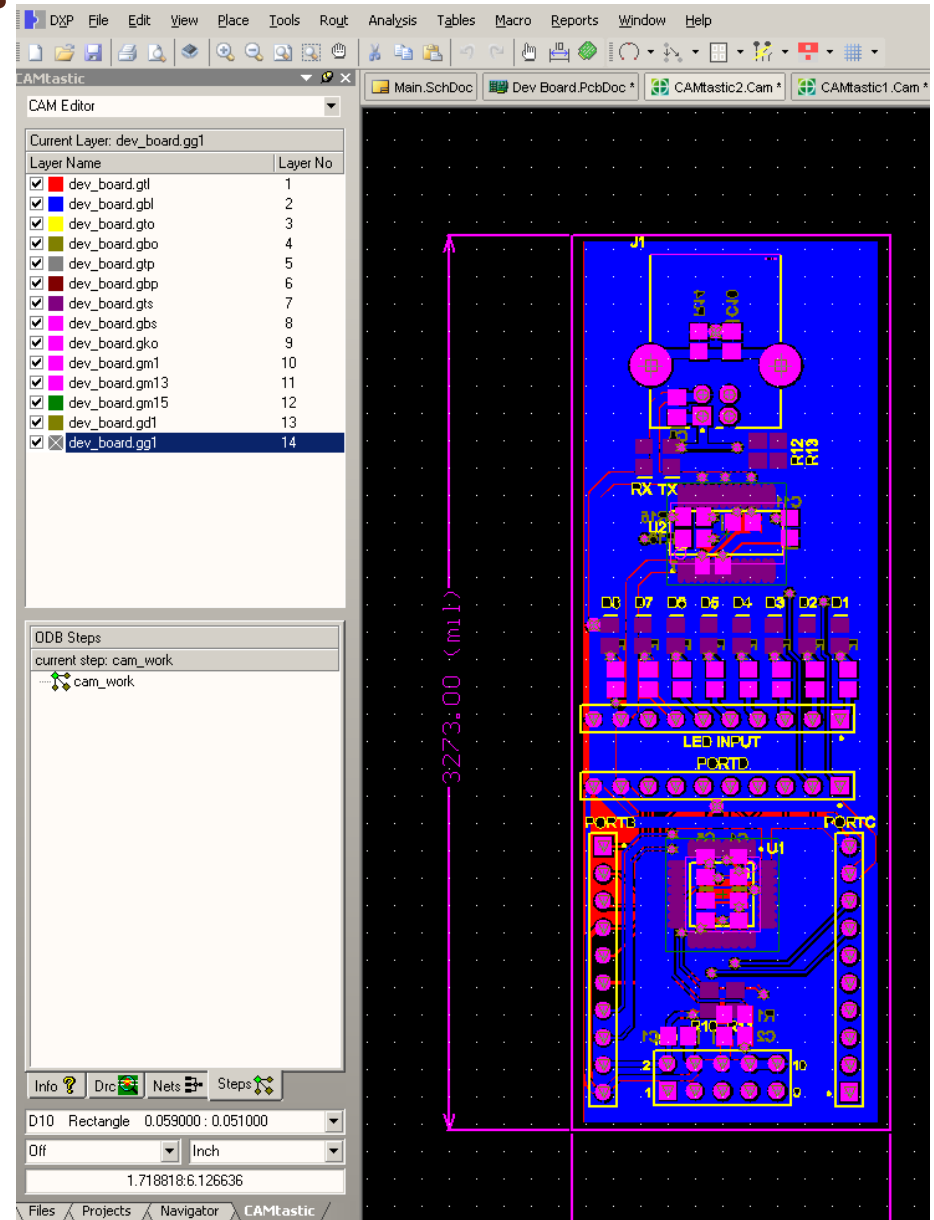
PCB Complete!

- Highlight the CAMtastic file with your board layout on it
- Click the CAMtastic tab



PCB Complete!

- Inside the CAM viewer, you can see a rather crappy rendition of your board and a bunch of color coded files.
- You may also notice there are new buttons in the menu screen. This is so you can modify the output files (Gerber files) directly.



PCB Complete!

- You may notice a new chart that appears in the Gerber files.
- If you remember the .LEGEND from the PCB layout, that is known as a SPECIAL STRING which performs a special function.
- This particular one creates information about the holes and represents this information with symbols

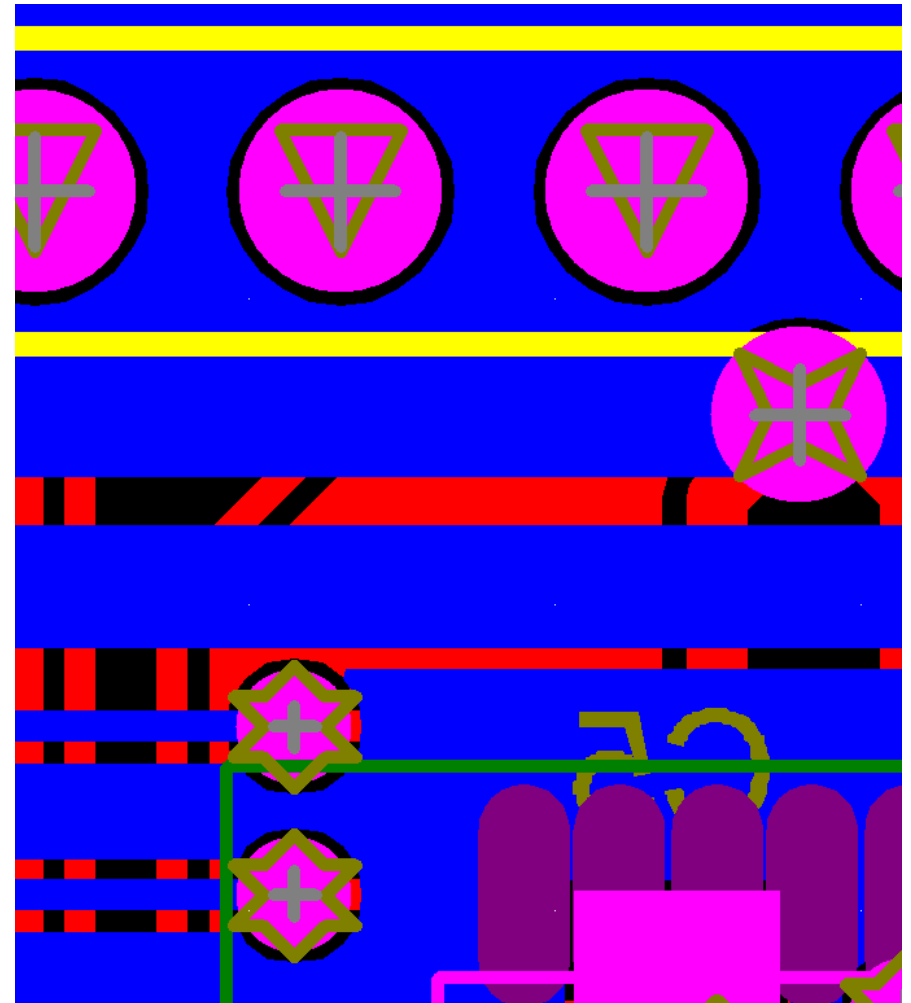
.LEGEND
Drill Drawing.

Symbol	Hit Count	Tool Size	Plated	Hole Type
☆	41	15mil (0.381mm)	PTH	Round
✱	3	30mil (0.762mm)	PTH	Round
▽	50	35.433mil (0.9mm)	PTH	Round
○	4	36.221mil (0.92mm)	PTH	Round
□	2	90.551mil (2.3mm)	PTH	Round
	100 Total			

Drill Drawing.

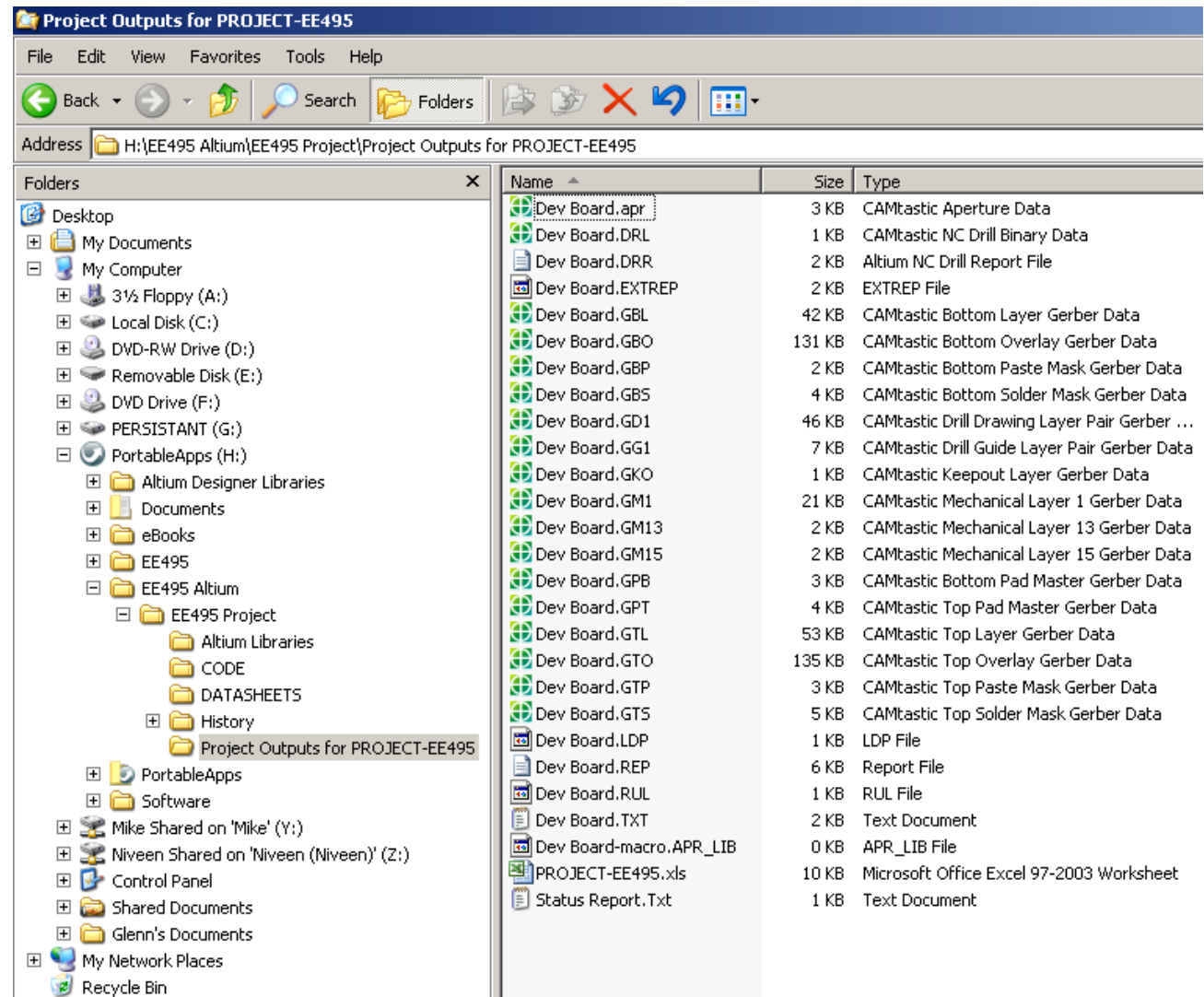
PCB Complete!

- Although it looks like someone did graffiti on your work, these patterns represent the type of hole it is, and the size of the hole
- The + is the center of the hole, and the pattern reveals drill information from the .LEGEND special string
- NOTE: There are much better CAM viewers than Altium's, Some are included in my syllabus



PCB Complete!

- When you created the Gerber files and NC Drill files, a new folder is automatically created in your project folder called 'Project Outputs'
- All these files are what you send to get your board made
- Altium is nice enough to register the file extensions with Windows so you can see which each file is

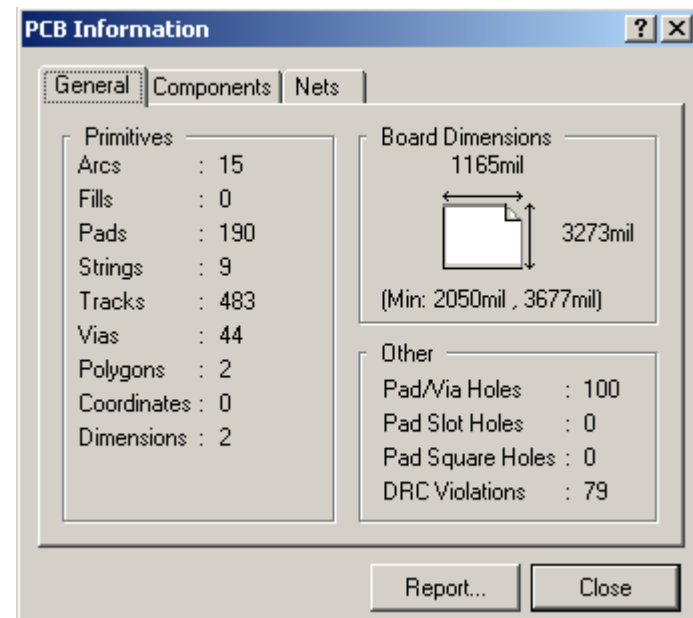
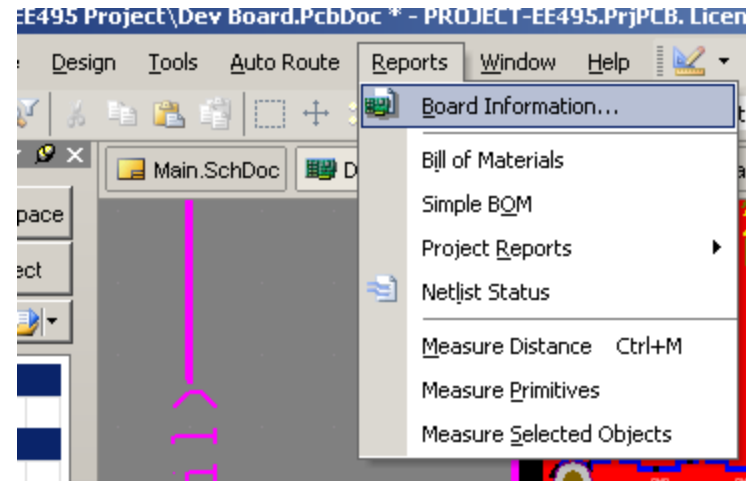


The screenshot shows a Windows Explorer window titled "Project Outputs for PROJECT-EE495". The address bar shows the path: H:\EE495 Altium\EE495 Project\Project Outputs for PROJECT-EE495. The left pane shows the folder structure, with "Project Outputs for PROJECT-EE495" selected. The right pane displays a list of files with columns for Name, Size, and Type.

Name	Size	Type
Dev Board.apr	3 KB	CAMtastic Aperture Data
Dev Board.DRL	1 KB	CAMtastic NC Drill Binary Data
Dev Board.DRR	2 KB	Altium NC Drill Report File
Dev Board.EXTREP	2 KB	EXTREP File
Dev Board.GBL	42 KB	CAMtastic Bottom Layer Gerber Data
Dev Board.GBO	131 KB	CAMtastic Bottom Overlay Gerber Data
Dev Board.GBP	2 KB	CAMtastic Bottom Paste Mask Gerber Data
Dev Board.GBS	4 KB	CAMtastic Bottom Solder Mask Gerber Data
Dev Board.GD1	46 KB	CAMtastic Drill Drawing Layer Pair Gerber ...
Dev Board.GG1	7 KB	CAMtastic Drill Guide Layer Pair Gerber Data
Dev Board.GKO	1 KB	CAMtastic Keepout Layer Gerber Data
Dev Board.GM1	21 KB	CAMtastic Mechanical Layer 1 Gerber Data
Dev Board.GM13	2 KB	CAMtastic Mechanical Layer 13 Gerber Data
Dev Board.GM15	2 KB	CAMtastic Mechanical Layer 15 Gerber Data
Dev Board.GPB	3 KB	CAMtastic Bottom Pad Master Gerber Data
Dev Board.GPT	4 KB	CAMtastic Top Pad Master Gerber Data
Dev Board.GTL	53 KB	CAMtastic Top Layer Gerber Data
Dev Board.GTO	135 KB	CAMtastic Top Overlay Gerber Data
Dev Board.GTP	3 KB	CAMtastic Top Paste Mask Gerber Data
Dev Board.GTS	5 KB	CAMtastic Top Solder Mask Gerber Data
Dev Board.LDP	1 KB	LDP File
Dev Board.REP	6 KB	Report File
Dev Board.RUL	1 KB	RUL File
Dev Board.TXT	2 KB	Text Document
Dev Board-macro.APR_LIB	0 KB	APR_LIB File
PROJECT-EE495.xls	10 KB	Microsoft Office Excel 97-2003 Worksheet
Status Report.Txt	1 KB	Text Document

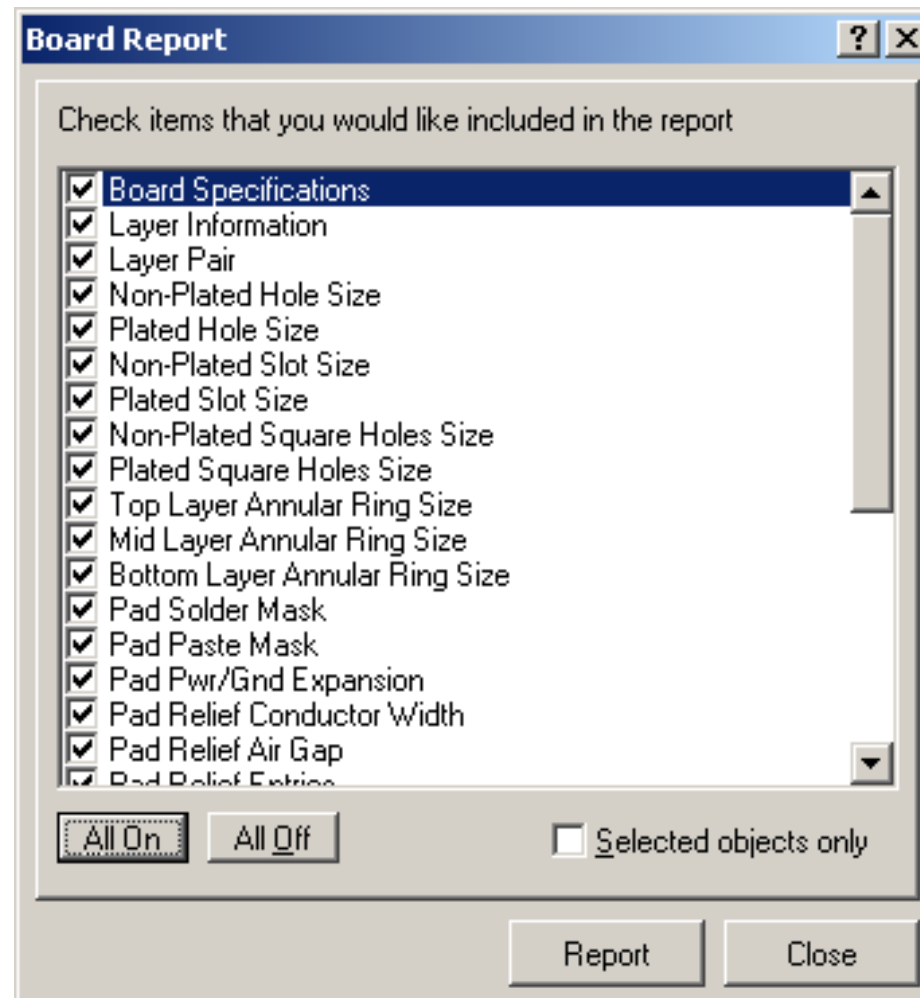
PCB Complete!

- Click Report > Board information for a quick summary of your board
- Click 'REPORT' and include this report file in any project files you send me. **POINTS WILL BE TAKEN OFF IF IT IS NOT INCLUDED**



PCB Complete!

- Click the 'All On' box and click 'Report'



PCB Complete!

- An HTML report file is created (file path is given) and you can select the units.

- This HTML file has hyperlinks so you can quickly view areas

Board Information Report

Date : 6/2/2008
Time : 3:11:42 PM
Elapsed Time : 00:00:00
Filename : [H:\EE495 Altium\EE495 Project\Dev Board.PcbDoc](#)
Units : mm mils

Contents

Contents

Board Specification

[General](#)

Routing

[Routing Information](#)

Layer Information

[Layer Information](#)

[Layer Pairs](#)

Board Holes

[Non-Plated Hole Size](#)

[Plated Hole Size](#)

Board Slots

PCB Complete!

General

Board Size	1165mil x 3273mil
Components on board	45

[Back to top](#)

Routing

Routing Information

Routing completion	100.00%
Connections	122
Connections routed	122
Connections remaining	0

[Back to top](#)

PCB Complete!

Board Holes

Non-Plated Hole Size	Pads	Vias
Total	0	0

[Back to top](#)

Plated Hole Size	Pads	Vias
0mil	134	0
15mil	0	41
30mil	0	3
35.433mil	50	0
36.221mil	4	0
90.551mil	2	0
Total	190	44

[Back to top](#)

Board Slots

Non-Plated Slot Size / Length	Pads
Total	0

[Back to top](#)

Plated Slot Size / Length	Pads
Total	0

[Back to top](#)

PCB Complete!

Board Holes

Non-Plated Hole Size	Pads	Vias
Total	0	0

[Back to top](#)

Plated Hole Size	Pads	Vias
0mil	134	0
15mil	0	41
30mil	0	3
35.433mil	50	0
36.221mil	4	0
90.551mil	2	0
Total	190	44

[Back to top](#)

Board Slots

Non-Plated Slot Size / Length	Pads
Total	0

[Back to top](#)

Plated Slot Size / Length	Pads
Total	0

[Back to top](#)

PCB Complete!

- Be careful with this one, The annular ring size from board vendors is defined differently. **The Ring size is this number divided by 2**

Other

Top Layer Annular Ring Size	Count
15mil	41
20mil	3
23.622mil	50
26.772mil	4
59.055mil	2
Total	100

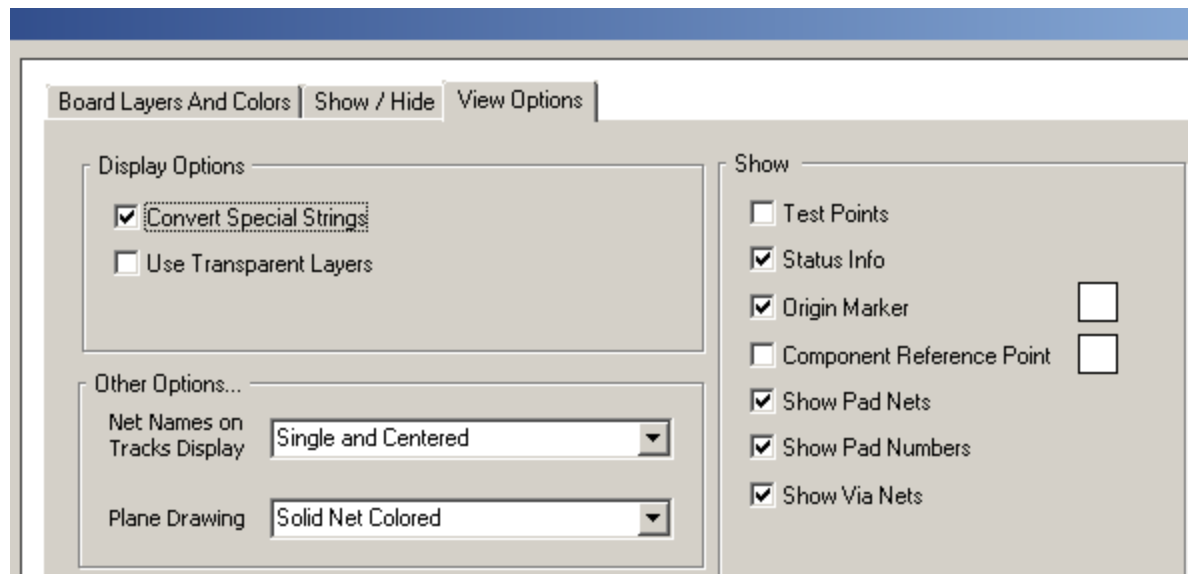
PCB Complete!

- What's wrong with this ????
- We specified a minimum trace width of 10 mils, yes we have 81 tracks with a more narrow width. Is the DRC Broken ?

Track Width	Count
3.937mil	21
7.874mil	60
10mil	162
15mil	61
20mil	86
30mil	69
40mil	24
Total	483

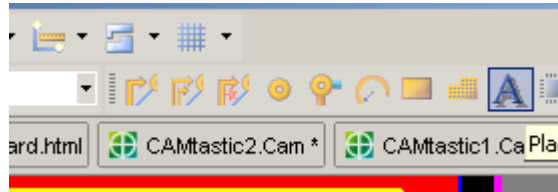
Special Strings

- Type 'L' from the PCB screen and click the 'View Options'
- Check the box that allows you to Convert Special Strings

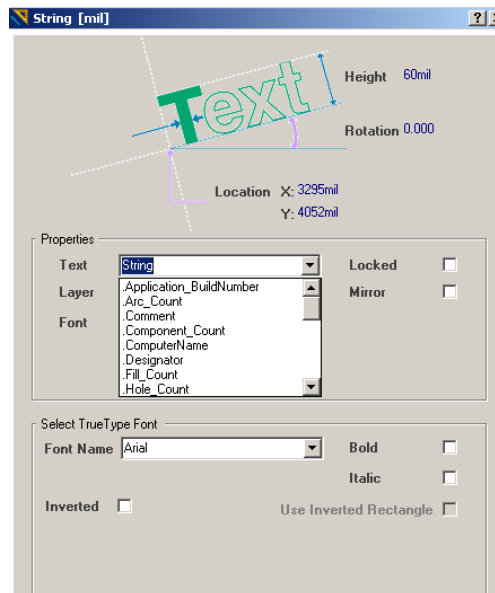


Special Strings

- Click the Text Entry icon from the PCB menu



- Double click on the text you placed to get the following menu

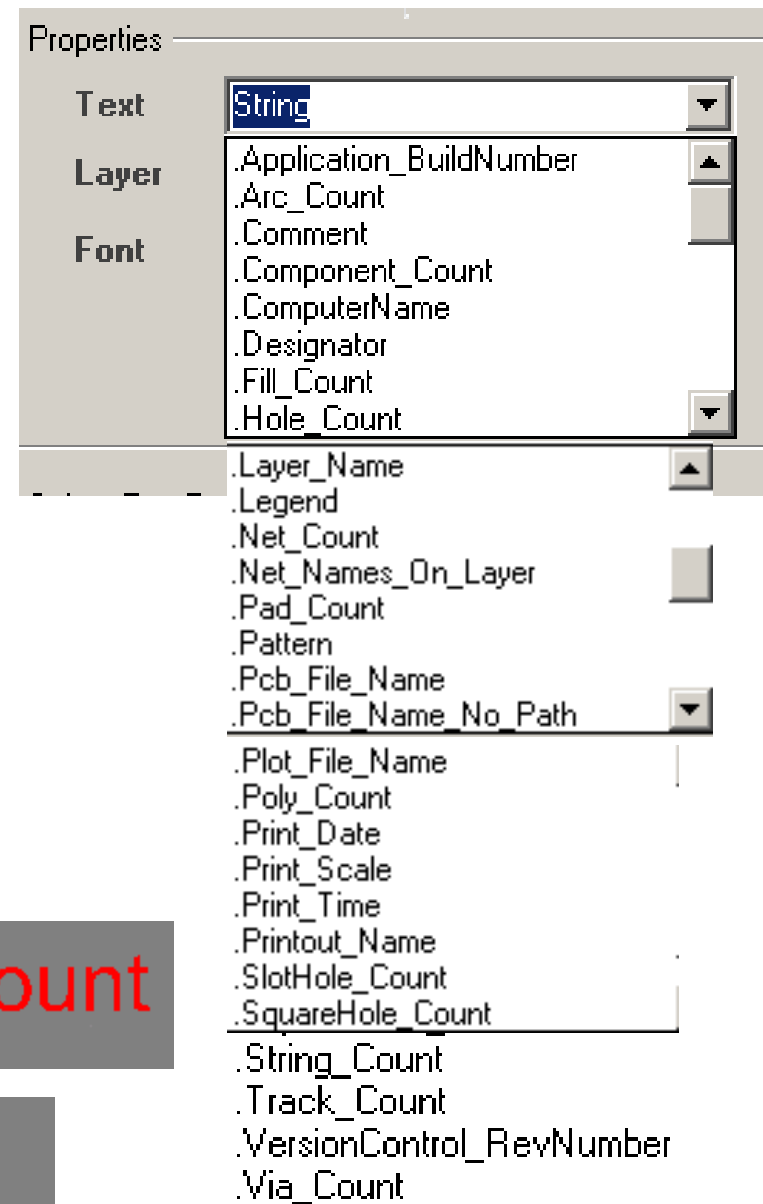


Special Strings

- Shown to the right are all the available Special Strings that can be placed in PCB Entry.
- NOTE: This list usually changes with each revision

Hole Count .Hole_Count

Hole Count 100



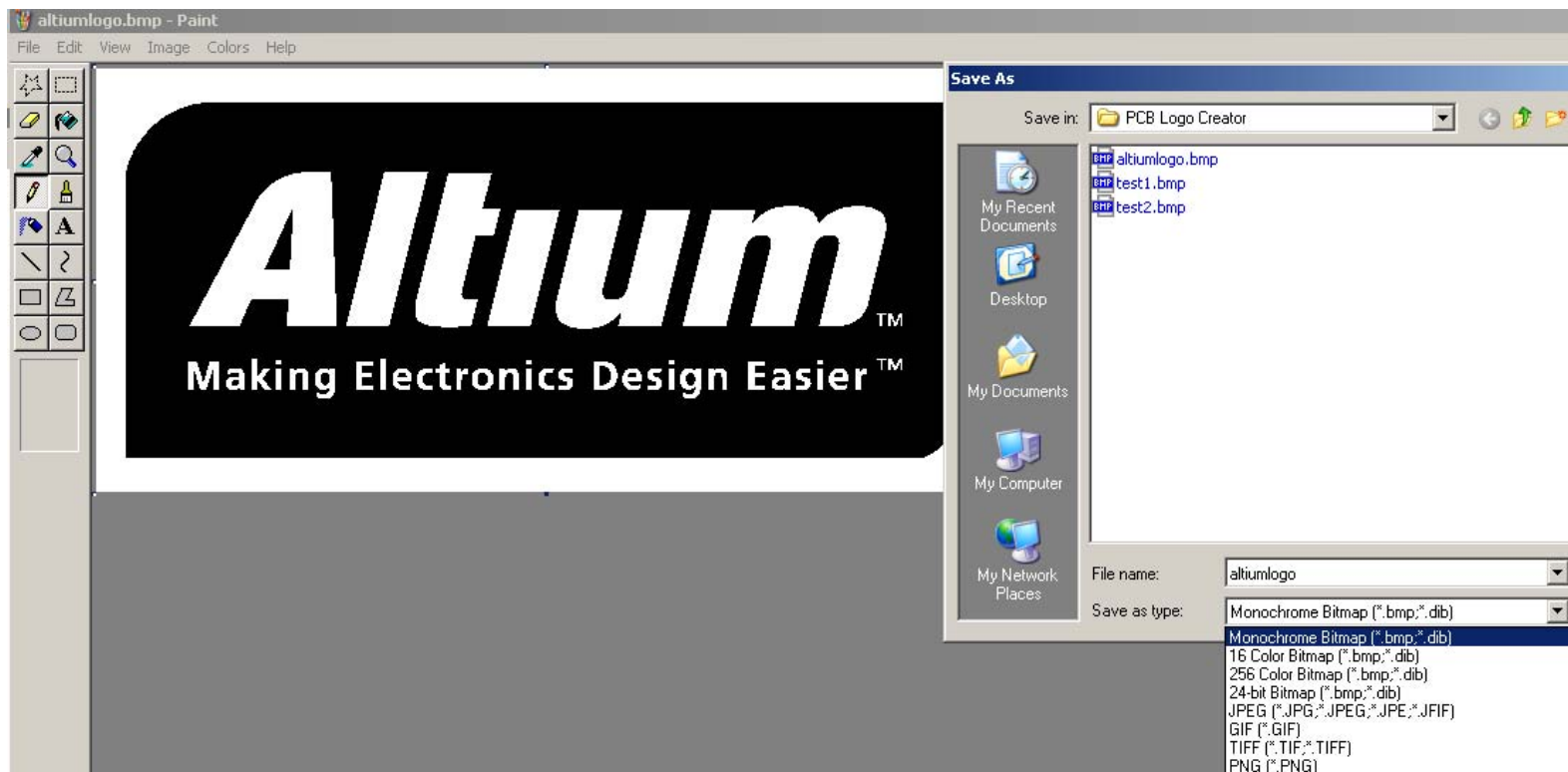


Part 2: Intermediate Guide

- This portion of the guide contains more advanced features as is geared towards:
 - Hardware Engineers
 - Signal Integrity Issues
 - Advanced Altium Designer features

Logos

- The image you want to create a logo out of **MUST** be saved as a **MONOCHROME BITMAP** file. This can be done by opening the file in MS Paint and saving the file as shown in the box below



Logos

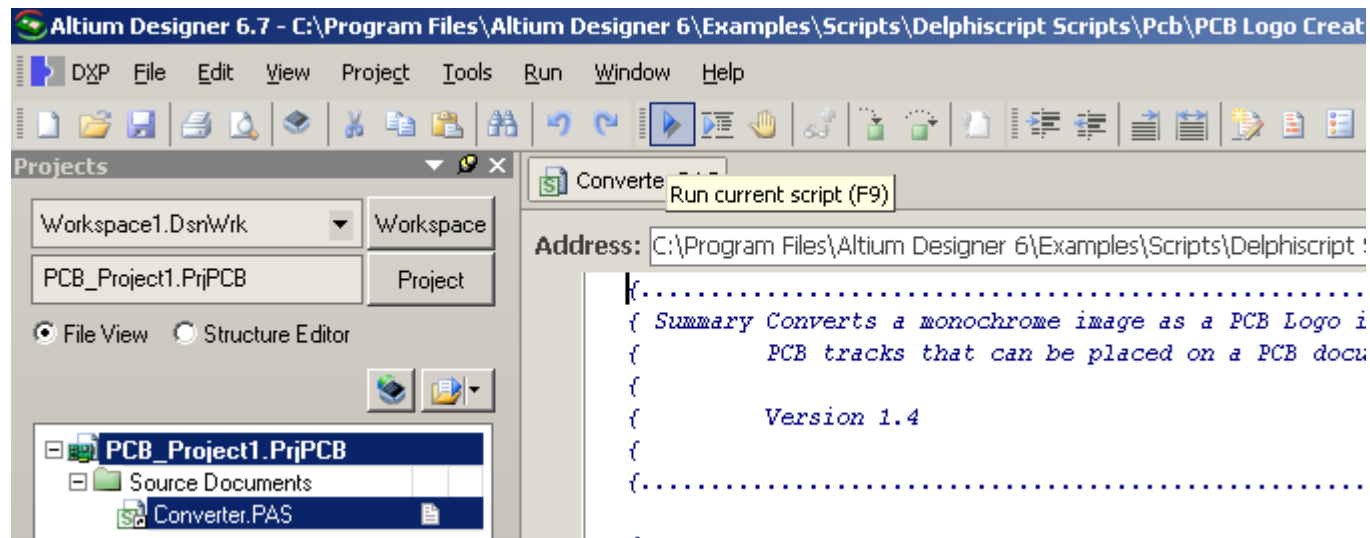
- The most requested feature people want to know is how to implement a logo in their design. There is a script located in this directory

`C:\Program Files\Altium Designer 6\Examples\
Scripts\Delphiscript Scripts\Pcb\PCB Logo Creator`

- Add Converter.Pas to your project. This script will convert any Black and white image to a PCB footprint.

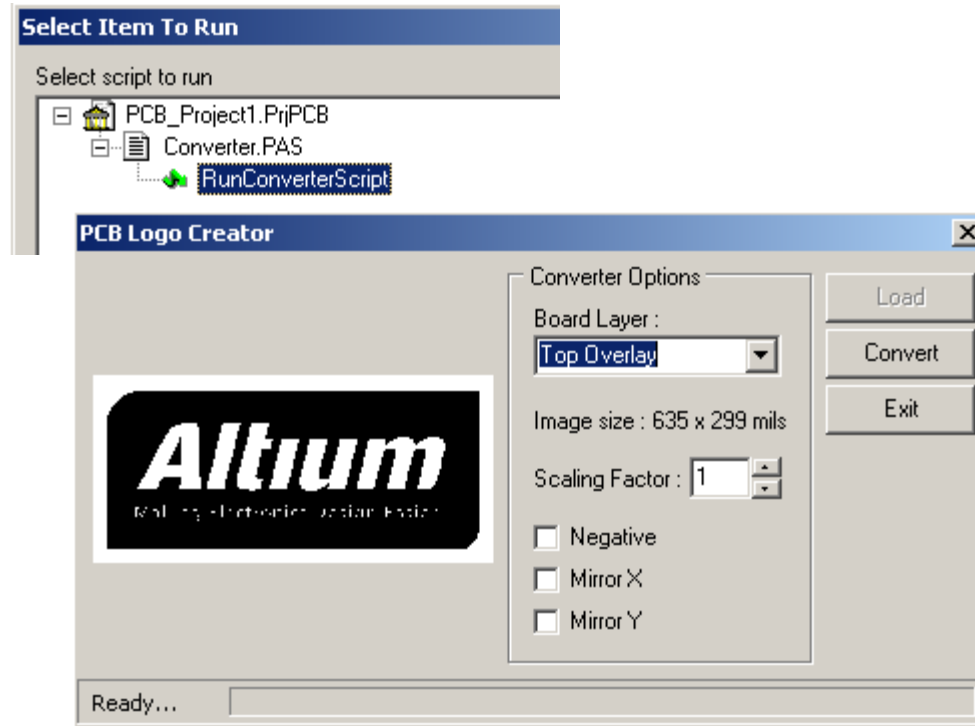
Logos

- Notice the different menu icons when you have a script selected. From here, you can click the small triangle button to execute the script or press F9



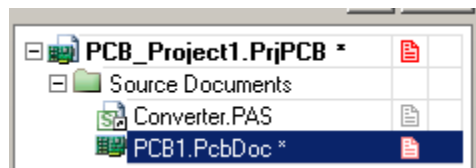
Logos

- Make sure to change the layer to TOP OVERLAY (Silkscreen) and load the .BMP image. Notice there is an option for scaling factor, and it shows image size.
- Image size will be the dimensions of the logo in mils once the conversion is complete. Using the scaling allows you to make the exact dimensions you want for the



Logos

- Depending on many factors, such as complexity, and processing power, the converting time may vary
- When complete, there will be a new PCB document added to your project containing your logo. It is a good idea to copy & paste the logo into your footprint library for future use



Logos

- Logos can also include:
 - Compliance icons such as FCC, ROHS, CE, etc..
 - Warnings such as ESD, High Voltage, Etc
 - Make sure you can legally use any logos you place on your PCB



Atmel_logo.bmp



CE-LOGO.bmp



CE-LOGO.jpg



confidential_stamp.jpg



Copyright.jpg



ESD.bmp



ESD.gif



esd_symbol_l.jpg



FCC-logo.bmp



FCC-logo.gif



Hatsui.jpg



ROHS.jpg



RoHS_logo.bmp



RoHS_logo_small.jpg



Multiple Monitor Setup



Schematic/PCB Interaction with Multiple Monitors



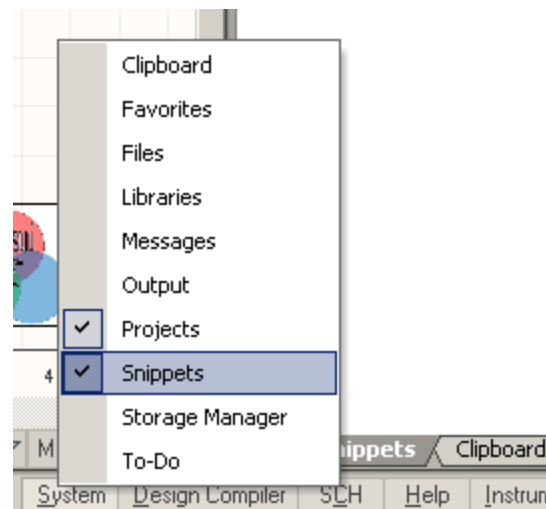
Circuit Simulation



Version Control System

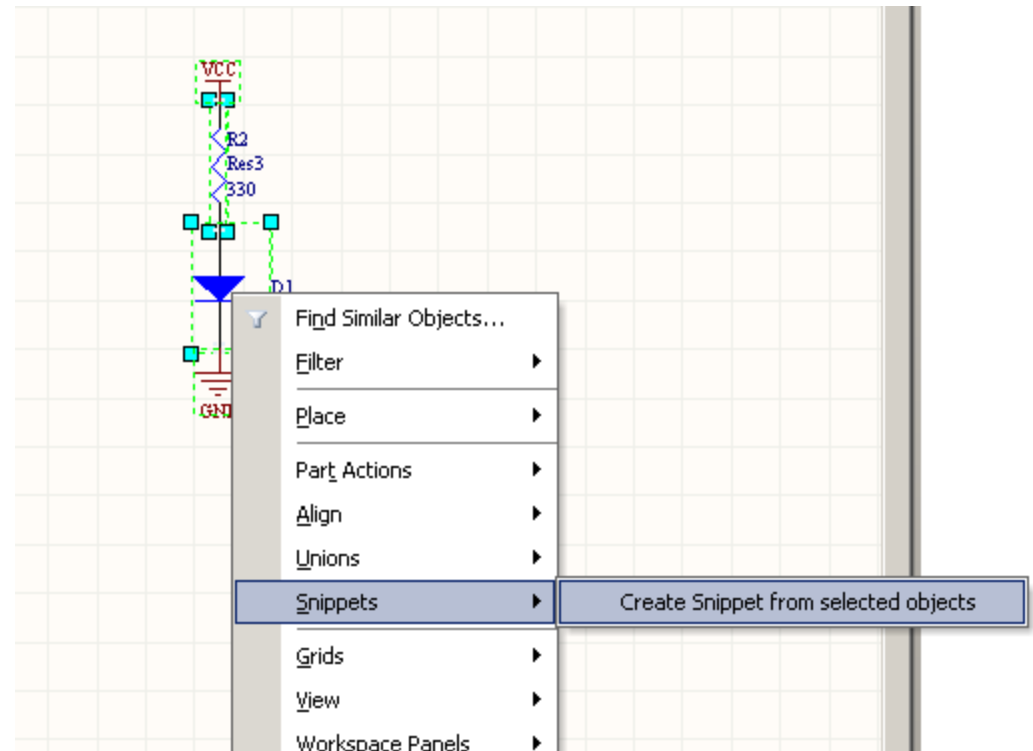
Snippets

- Snippets allow for quick reuse of existing designs.
- Schematic, PCB, Routing can all be saved as snippets.
- Easily found by clicking the 'System' bar in the bottom right corner, then enabling 'Snippets'



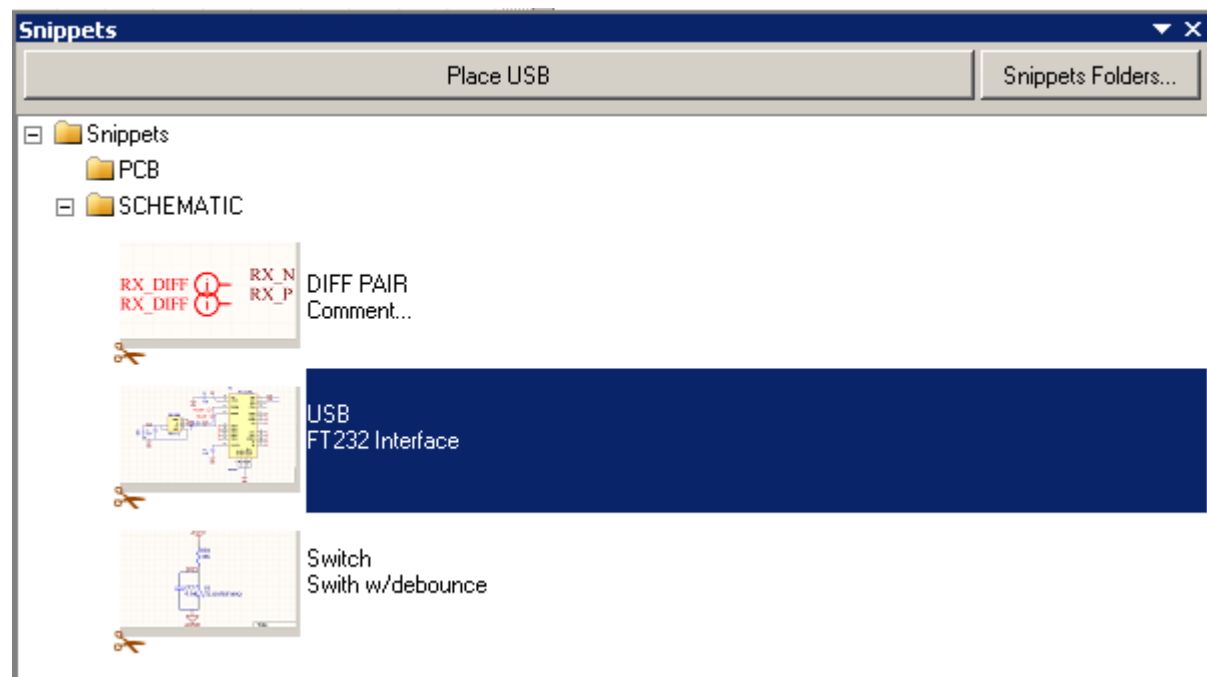
Snippets

- To create a snippet, simply highlight the schematic/pcb information you would like to reuse, right click, then select 'Add to Snippet'



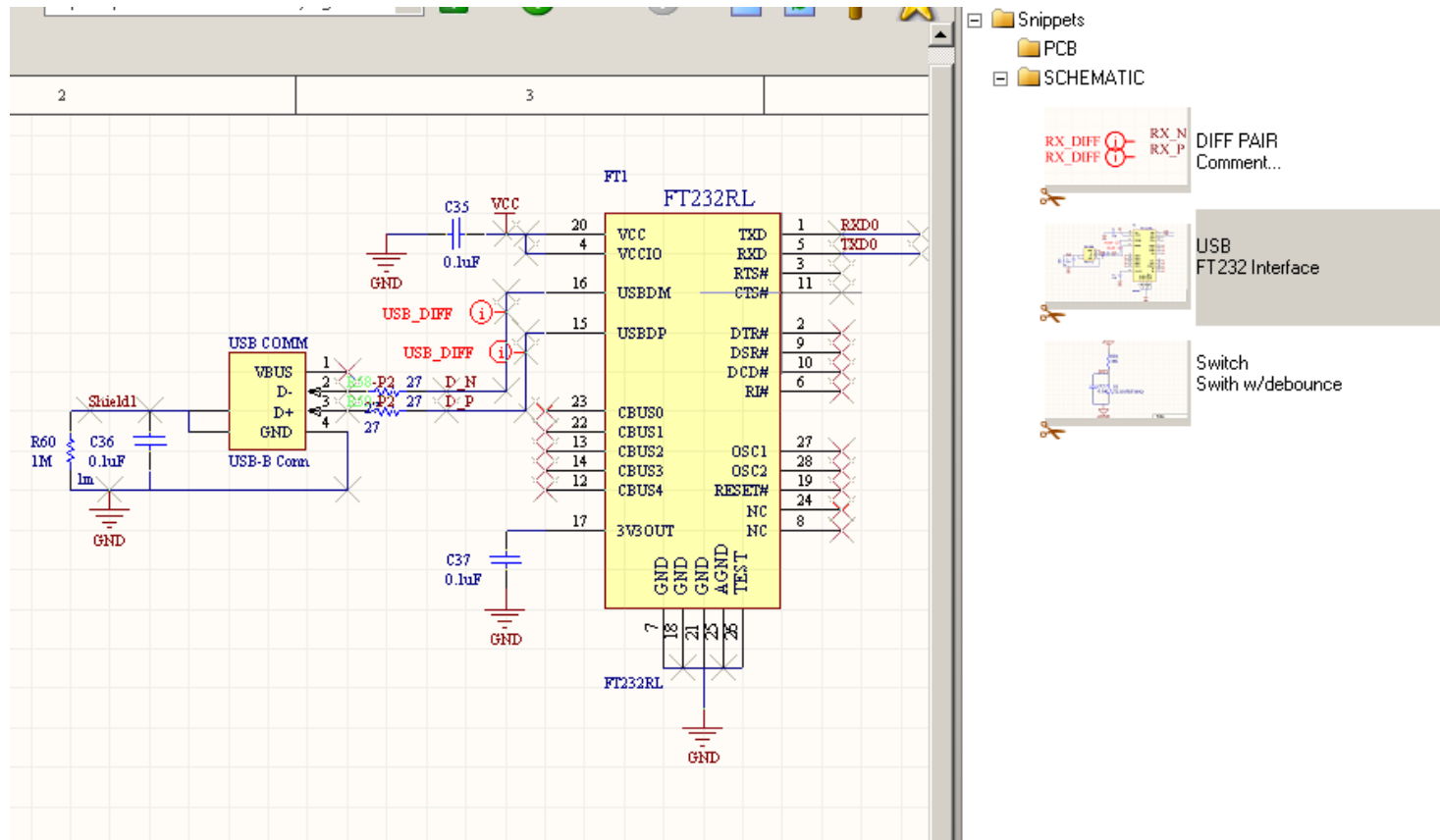
Snippets

- You can create multiple subfolders to keep your snippets organized.
- To place a snippet, simply highlight the snippet of your choice, and click the 'PLACE' button at the top of the snippets box.



Snippets

- When placing a snippet, you will see a rectangular boundary outline of the object. Once the location is held, you will place the ENTIRE snippet as shown below.
- Notice that nodes, net labels, differential pairs, and component ID's are all pasted from the snippet.
- When pasting a PCB snippet, the footprint, layer information, traces, etc..Are all pasted from the snippet
- This powerful, simple to use feature is a great timesaving tool when creating multiple projects



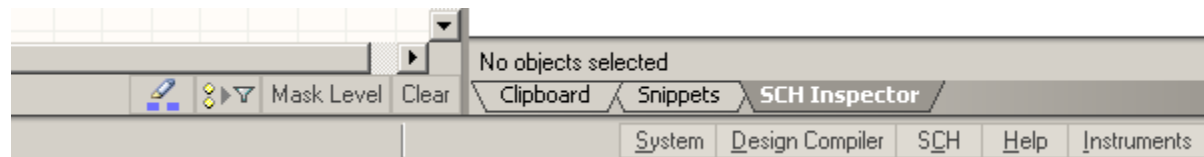
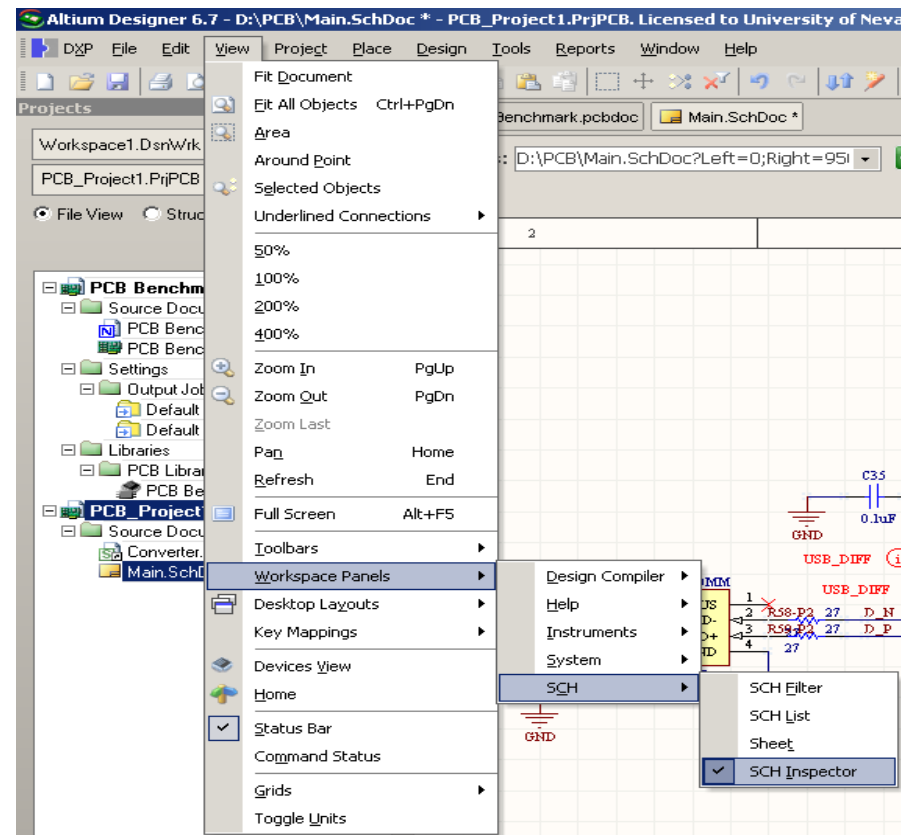


Inspector Tools

- The PCB and Schematic Inspector tools for some reason are hidden a little deep inside the software.
- This feature allows you to easily and quickly make project-wide changes to components, nets, etc.
- Serious users will be amazed at how much time is saved by using the Inspector tools

Schematic Inspector

- Start out by docking the Schematic Inspector. I usually keep it in the side dockbar which is useful for a multiple-screen display.





Schematic Inspector

- By selecting multiple objects (shift & left click), you can see the Schematic Inspector update the display for all selected parts.
- Items showing <...> have varying values for more than one component (Such as location X/Y coordinates)

Schematic Inspector

- In this example, Three capacitors are selected
- Schematic Inspector shows that 3 objects are selected in the bottom of it's taskbar.
- From this stage, any value changed inside the inspector will immediately be changed in ALL SELECTED OBJECTS

The screenshot displays the Schematic Inspector software interface. The main window shows a circuit diagram with a central component labeled FT1 (FT232RL). Three capacitors (C35, C36, and C37) are highlighted with green selection boxes. The Schematic Inspector panel on the right shows the properties for the selected objects. The 'Object Specific' section shows the description 'Capacitor' and the current part 'CC1608-0603'. The 'Parameters' section lists various parameters such as 'Published', 'LatestRevisionDate', 'LatestRevisionNote', 'PackageReference', and 'Publisher'.

Kind	Part
Object Kind	Part

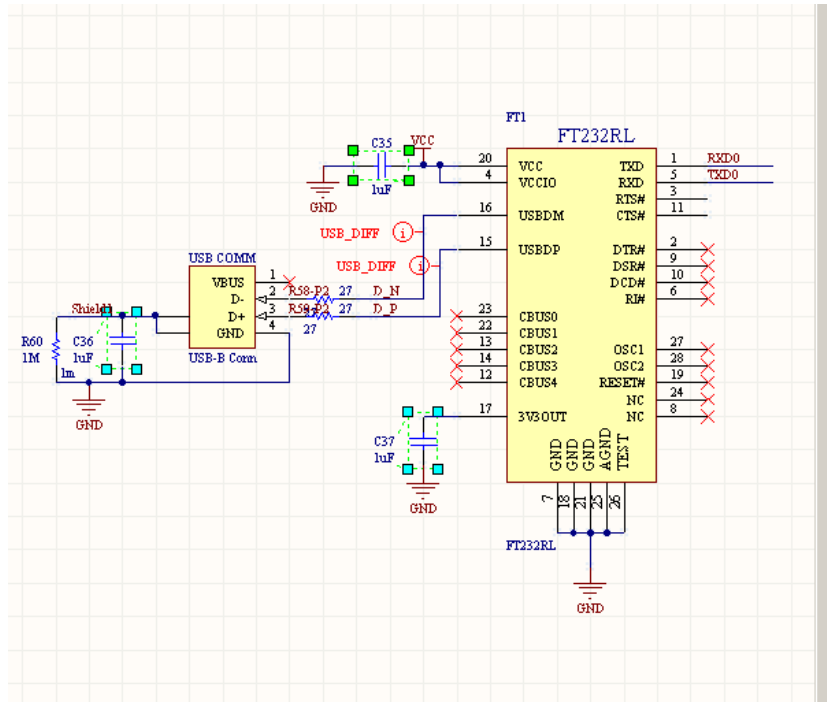
Design	Owner Document	Main.Sch.Doc
Graphical	X1	<...>
	Y1	<...>
	Orientation	<...>
	Mirrored	<input type="checkbox"/>
	Display Mode	Normal
	Show Hidden Pins	<input type="checkbox"/>
	Show Designator	<input checked="" type="checkbox"/>

Object Specific	Description	Capacitor
	Lock Designator	<input type="checkbox"/>
	Lock Part ID	<input type="checkbox"/>
	Pins Locked	<input checked="" type="checkbox"/>
	File Name	*
	Configuration	*
	Library	Miscellaneous Devices.IntLib
	Symbol Reference	Cap
	Component Designator	<...>
	Current Part	
	Part Comment	0.1uF
	Current Footprint	CC1608-0603
	Component Type	Standard
	Database Table Name	
	Use Library Name	<input checked="" type="checkbox"/>
	Use Database Table Name	<input checked="" type="checkbox"/>
	Design Item ID	Cap

Parameters	Value
<input checked="" type="checkbox"/> Published	8-Jun-2000
<input checked="" type="checkbox"/> LatestRevisionDate	17-Jul-2002
<input checked="" type="checkbox"/> LatestRevisionNote	Re-released for DXP Platform.
<input checked="" type="checkbox"/> PackageReference	RAD-0.3
<input checked="" type="checkbox"/> Publisher	Altium Limited
Add User Parameter:	

Schematic Inspector

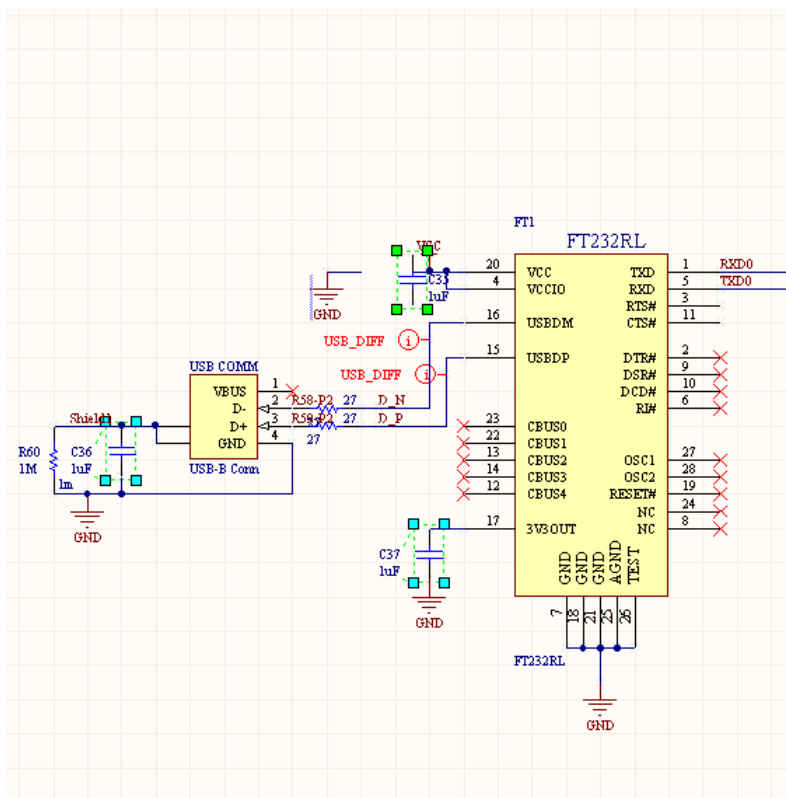
- Changes such as changing the part comment from 0.1uF to 1uF are immediately visible in the schematic



Display Mode	Normal
Show Hidden Pins	<input type="checkbox"/>
Show Designator	<input checked="" type="checkbox"/>
Object Specific	
Description	Capacitor
Lock Designator	<input type="checkbox"/>
Lock Part ID	<input type="checkbox"/>
Pins Locked	<input checked="" type="checkbox"/>
File Name	
Configuration	*
Library	Miscellaneous Devices.IntLib
Symbol Reference	Cap
Component Designator	<...>
Current Part	
Part Comment	1uF
Current Footprint	CC1608-0603
Component Type	Standard
Database Table Name	
Use Library Name	<input checked="" type="checkbox"/>
Use Database Table Name	<input checked="" type="checkbox"/>
Design Item ID	Cap
Parameters	
<input checked="" type="checkbox"/> Published	8-Jun-2000
<input checked="" type="checkbox"/> LatestRevisionDate	17-Jul-2002
<input checked="" type="checkbox"/> LatestRevisionNote	Re-released for DXP Platform.
<input checked="" type="checkbox"/> PackageReference	RA&D-0.3
<input checked="" type="checkbox"/> Publisher	Altium Limited
Add User Parameter:	

Schematic Inspector

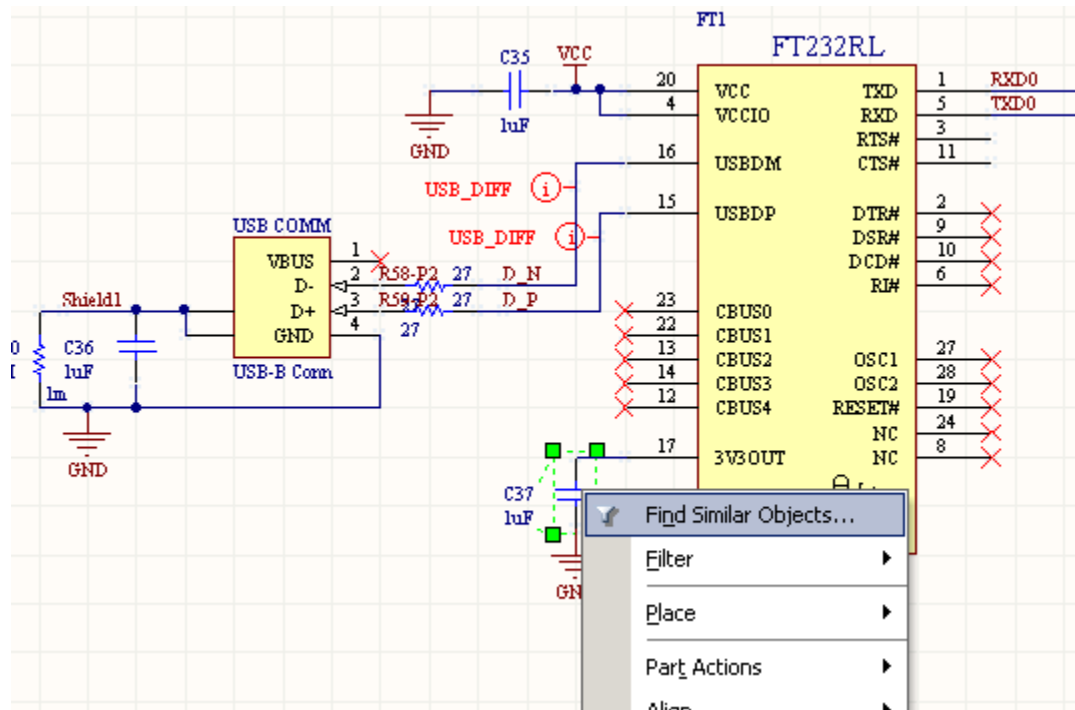
- Allows changing a field with different values (Showing <...>) to make all values uniform
- In the example below, component orientation is changed from <...> to 90 Degrees.
- Immediately each component is rotated to the 90 degree position.
- For this example, the lower two capacitors are unchanged, as their value was already 90 degrees, and the top capacitor is rotated from 0 degrees to 90 degrees.



X1	<...>
Y1	<...>
Orientation	90 Degrees
Mirrored	<input type="checkbox"/>
Display Mode	Normal
Show Hidden Pins	<input type="checkbox"/>
Show Designator	<input checked="" type="checkbox"/>
Object Specific	
Description	Capacitor
Lock Designator	<input type="checkbox"/>
Lock Part ID	<input type="checkbox"/>
Pins Locked	<input checked="" type="checkbox"/>
File Name	
Configuration	*
Library	Miscellaneous Devices.IntLib
Symbol Reference	Cap
Component Designator	<...>
Current Part	
Part Comment	1uF
Current Footprint	CC1608-0603
Component Type	Standard
Database Table Name	
Use Library Name	<input checked="" type="checkbox"/>
Use Database Table Name	<input checked="" type="checkbox"/>
Design Item ID	Cap
Parameters	
<input checked="" type="checkbox"/> Published	8-Jun-2000
<input checked="" type="checkbox"/> LatestRevisionDate	17-Jul-2002
<input checked="" type="checkbox"/> LatestRevisionNote	Re-released for DXP Platform.
<input checked="" type="checkbox"/> PackageReference	RAD-0.3
<input checked="" type="checkbox"/> Publisher	Altium Limited
Add User Parameter:	

Find Similar Objects

- The perfect complimentary tool used with Inspector is the 'Find Similar Objects' right click menu.



Find Similar Objects

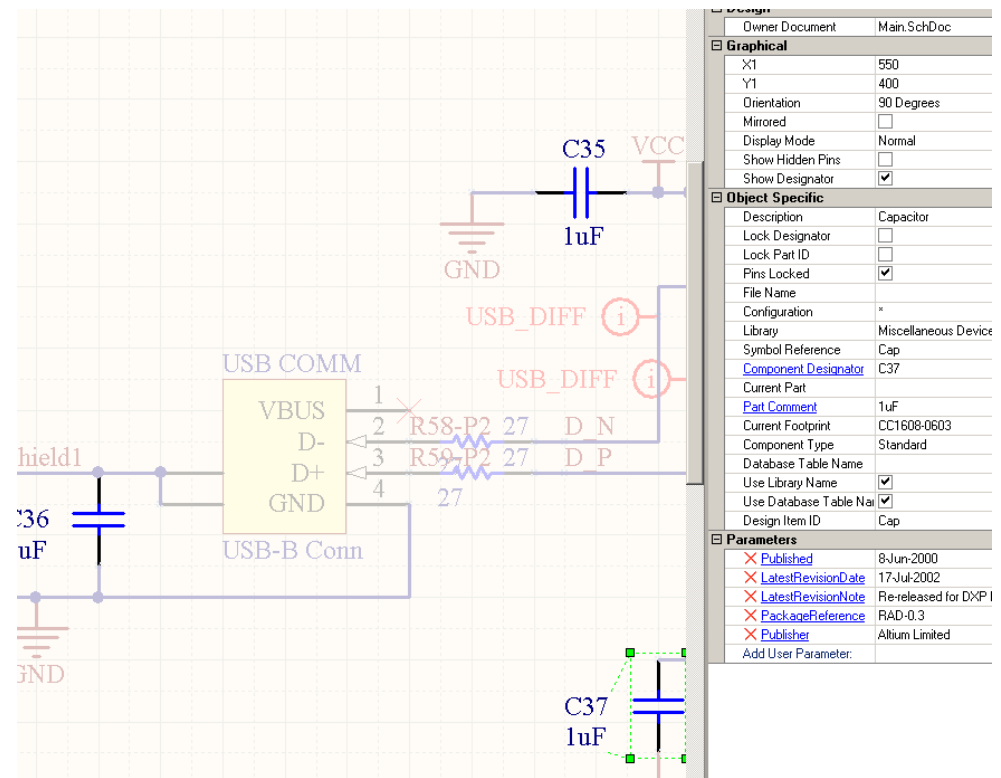
- To find ALL capacitors with a part comment of 1uF we change the following objects from 'ANY' to 'SAME'
 - Description: Capacitor
 - Part Comment: 1uF
- Click OK when finished

Find Similar Objects		
Kind		
Object Kind	Part	Same
Design		
Owner Document	D:\PCB\Main.SchDoc	Any
Graphical		
X1	550	Any
Y1	400	Any
Orientation	90 Degrees	Any
Mirrored	<input type="checkbox"/>	Any
Display Mode	Normal	Any
Show Hidden Pins	<input type="checkbox"/>	Any
Show Designator	<input checked="" type="checkbox"/>	Any
Selected	<input checked="" type="checkbox"/>	Any
Object Specific		
Description	Capacitor	Any
Lock Designator	<input type="checkbox"/>	Any
Lock Part ID	<input type="checkbox"/>	Any
Pins Locked	<input checked="" type="checkbox"/>	Any
File Name		Any
Configuration	*	Any
Library	Miscellaneous Devices.IntLib	Any
Symbol Reference	Cap	Any
Component Designator	C37	Any
Current Part		Any
Part Comment	1uF	Any
<input checked="" type="checkbox"/> Zoom Matching	<input type="checkbox"/> Select Matching	Current Document
<input checked="" type="checkbox"/> Clear Existing	<input type="checkbox"/> Create Expression	
<input checked="" type="checkbox"/> Mask Matching	<input checked="" type="checkbox"/> Run Inspector	



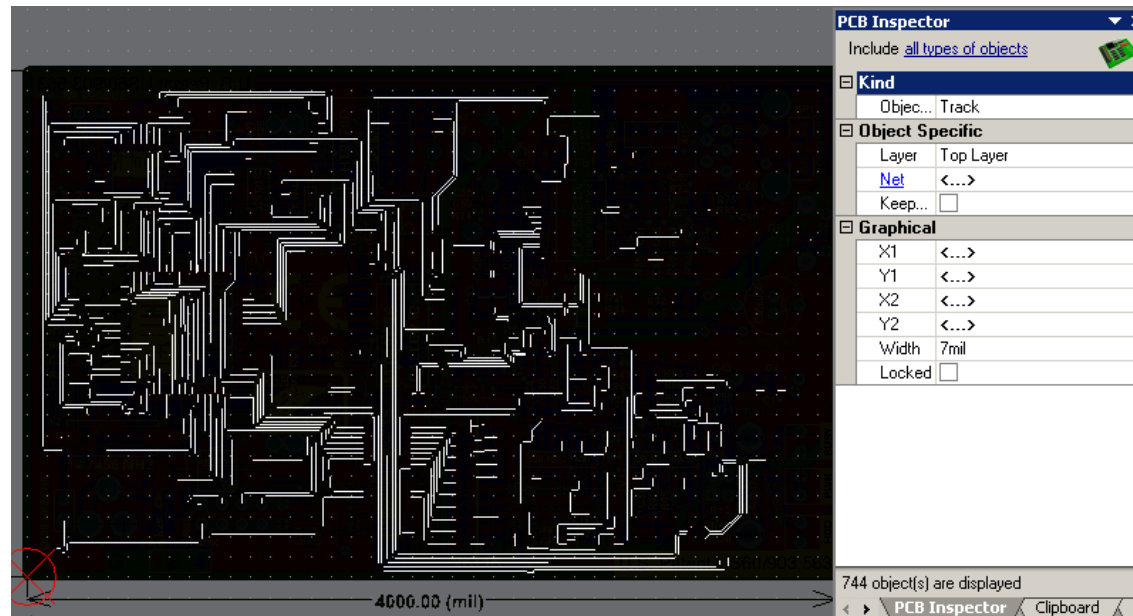
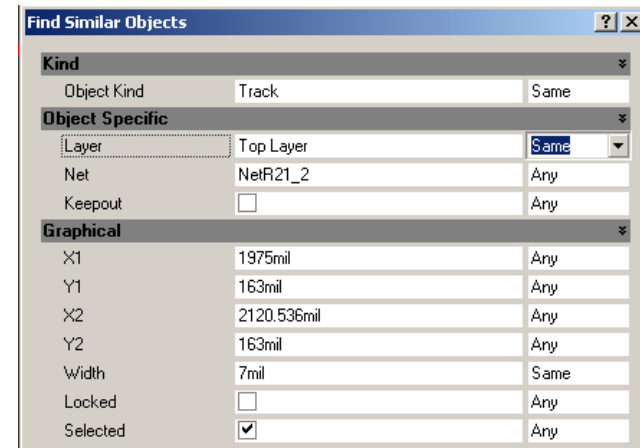
Schematic Inspector- Find Similar Objects

- After hitting 'Apply' or 'OK', the Schematic Inspector immediately finds all matching parts.
- In this case the three capacitors we changed earlier to 1uF are highlighted over the rest of the schematic, and selected in the Inspector.



PCB Inspector

- PCB Inspector works similar to the Schematic Inspector.
- Shown below is an example of finding and selecting all traces that are 7 mils width on the top layer only
- Notice how quickly we selected and can modify 744 objects meeting this criteria!!



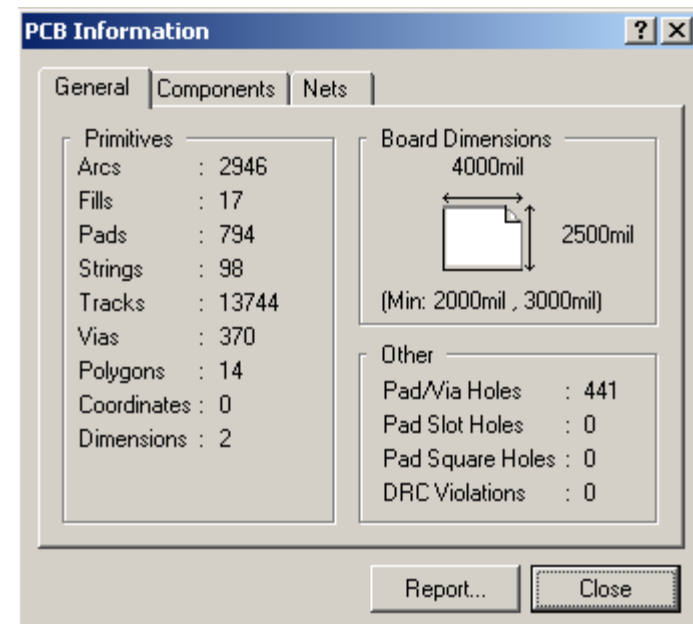
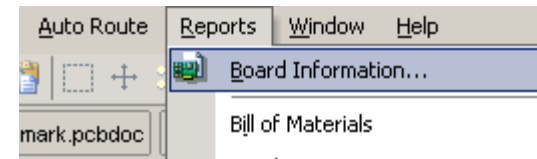


PCB Inspector

- We can almost immediately find any group of objects on the PCB and change values.
- This is an important feature because it will allow you to quickly adhere to changes in project requirements, such as fabrication trace-width parameters

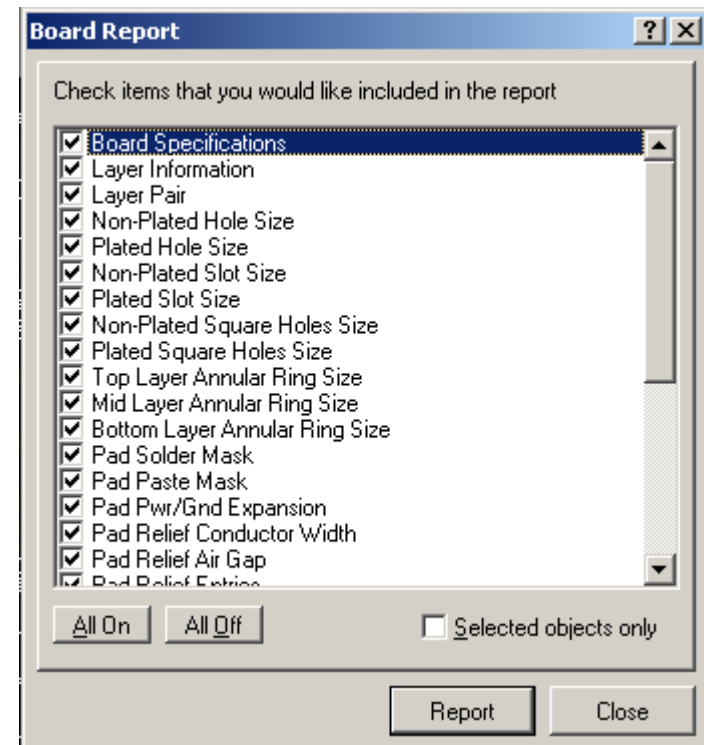
Project Report

- A quick detour from the PCB Inspector allows us to see a general report on the project
- From the main toolbar, click 'Reports' then Board information
- You will see a general summary as shown to the right, click on the 'Report' button



Project Report

- Select the information you want included in the report.
- Usually it is best to select all and then navigate to when you want in the report with hyperlinks
- Click 'Report' when done selecting objects



Project Report

- An HTML based report will be generated with a table of contents and hyperlink navigation available

Contents

Contents

Board Specification

[General](#)

Routing

[Routing Information](#)

Layer Information

[Layer Information](#)

[Layer Pairs](#)

Board Holes

[Non-Plated Hole Size](#)

[Plated Hole Size](#)

Board Slots

[Non-Plated Slot Size](#)

[Plated Slot Size](#)

Board Square Holes

[Non-Plated Square Hole Size](#)

[Plated Square Hole Size](#)

Other

[Top Layer Annular Ring Size](#)

[Mid Layer Annular Ring Size](#)

[Bottom Layer Annular Ring Size](#)

[Pad Solder Mask](#)

[Pad Paste Mask](#)

[Pad Pwr/Gnd Expansion](#)

[Pad Relief Conductor Width](#)

[Pad Relief Air Gap](#)

[Pad Relief Entries](#)

[Via Solder Mask](#)

Project Report

- The report will probably give more information than you will ever want to know, such as a detailed breakdown of the routing, layer information, vias, tracks, etc..

Routing

Routing Information

Routing completion	53.86%
Connections	531
Connections routed	286
Connections remaining	245

[Back to top](#)

Layer Information

Layer	Arcs	Pads	Vias	Tracks	Texts	Fills	Regions	ComponentBodies
Top Layer	1956	550	0	1174	1	1	47	0
GND	0	0	0	15	4	6	0	0
PWR	0	0	0	15	4	6	0	0
Bottom Layer	948	173	0	707	1	0	42	0
Board Information	4	0	0	6	30	0	0	0
Gerber ID	0	0	0	0	1	0	0	0
PCB Boundary	0	0	0	20	3	0	0	0
Printout_Name	0	0	0	0	1	0	0	0
Component Bodies	6	0	0	24	0	0	0	6

Project Report

- Shown below is a part of the report on board holes.
- This contains EVERY hole on the board, and includes the drill size of the hole.

Board Holes		
Non-Plated Hole Size	Pads	Vias
Total	0	0

[Back to top](#)

Plated Hole Size	Pads	Vias
0mil	723	0
15mil	10	351
30mil	10	19
36mil	33	0
47.244mil	2	0
65mil	5	0
80mil	4	0
100mil	5	0
125mil	2	0
Total	794	370



Report and PCB Inspector

- Scenario: Your PCB Manufacturer tells you that your hole size must be 20 mils in diameter, and that the holes in your submitted project are 15 mils (As shown in the previous slide)
- Go to the report to see a detailed analysis of the holes. For this board we can see the smallest hole size we need to change is 15 to 20 mils, every other hole is larger
 - Use 'Find similar objects' to find pads/vias that match 15 mil hole size
 - In the PCB Inspector, we know from the report that we need to change 351 Vias, and 10 pads from 15 mils to 20 mils. We can see in the Inspector window how many objects are selected.
 - Change the hole size from 15 to 20 mils using PCB Inspector
 - Find any clearance conflicts with the online DRC and quickly fix
 - Regenerate output files with smallest hole size = 20 mils
- A process like this, to quickly find and modify 361 holes might take just a few minutes rather than hours of finding and modifying each pad/via manually.



Output Job File



Scale Print (1:1 Footprint Verification)



PCB Tab



The Return Path



Bandwidth



Microstrips



Differential Pairs



Signal Integrity Introduction



Fourier Series and Transform



Physical Resistance



Physical Inductance



Physical Capacitance



Reflections



Multi-Layer Boards

- The term 'Multi-Layer board' usually refers to more than two layers.
- A multi-layer board contains a lot more information such as
 - Layer Stack



Layer Stack

- A Layer Stack is a detailed report that contains information on:
 - Physical Layers and ordering of layers
 - External signal layers
 - Internal signal layers
 - Internal planes
 - Copper thickness per layer (1 oz = 1.4 mil, 1/2 oz = 0.7 mil)
 - Dielectric material and permittivity constant
 - Prepreg thickness
 - Core thickness
 - Total board thickness
- This information is crucially important when implementing transmission lines, microstrips, striplines, etc.



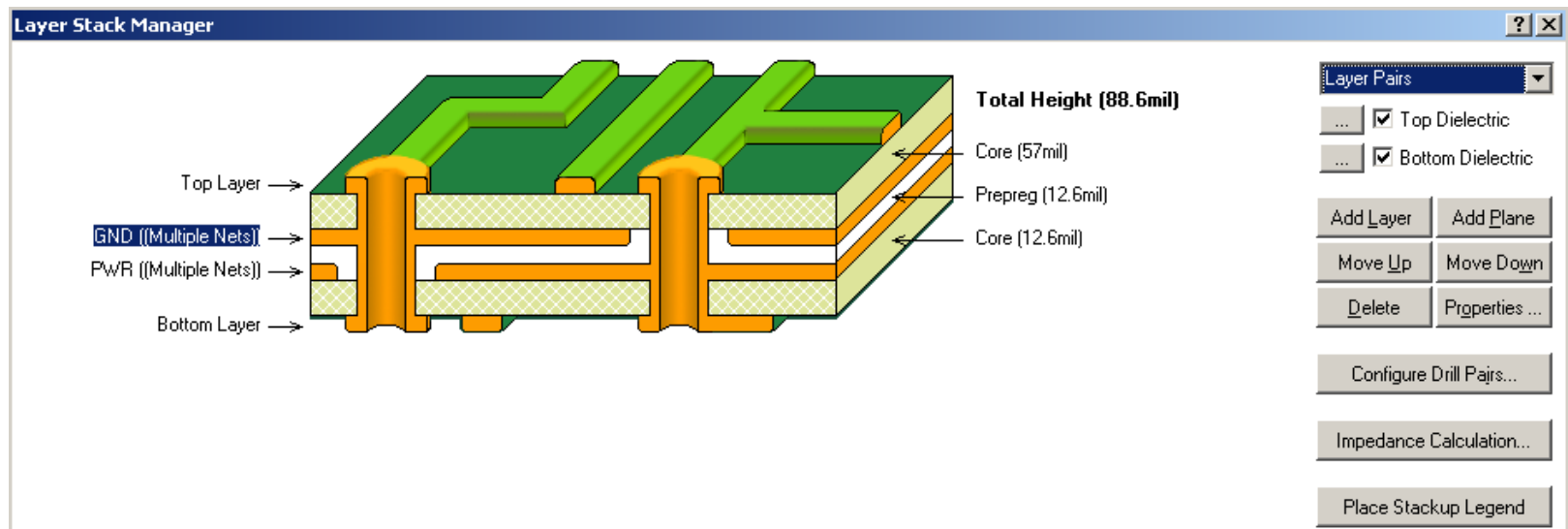
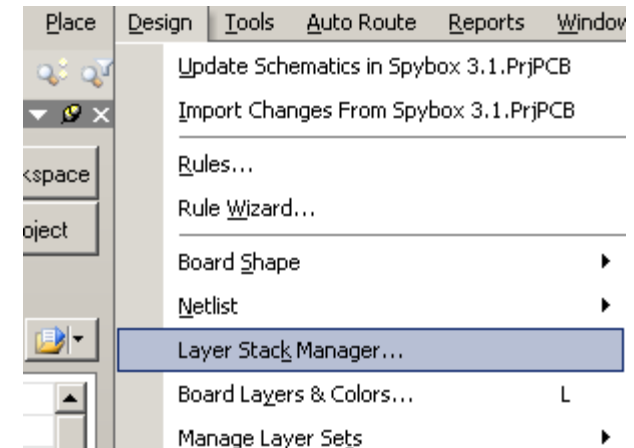
Internal Planes



Internal Signal Layers

Layer Stack

- Design > Layer Stack manager will show a 3D representation of the layer stack





Layer Stack



Impedance Controlled Routing

- Transmission lines usually required if:



Microstrip



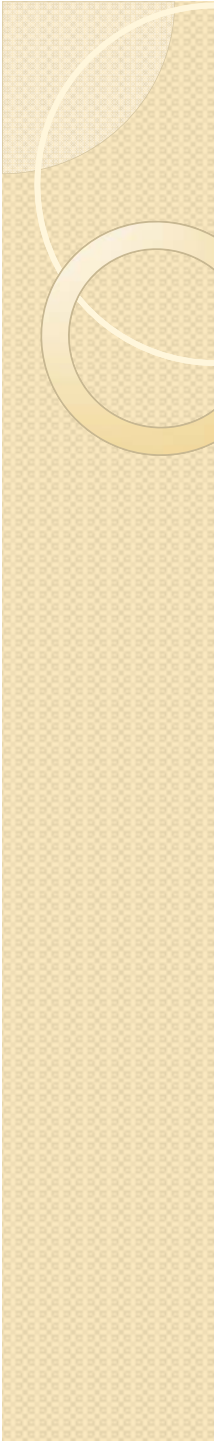
Stripline



Reflections



Cross Talk



EMI



Output Files



CAM Viewers

CAM Viewers allow the re/viewing of Gerber files.

- CAMTastic: Basic, Included in Altium Designer
 - <http://www.altium.com/files/learningguides/AR0119%20CAMtastic%20Introduction.pdf>
- ViewMate- Pentalogix FREE Gerber Viewer
 - <http://www.pentalogix.com/Download/download.html>
- CAMMaster- Pentalogix
 - <http://www.pentalogix.com/Products/CAMMaster/cammaster.html>
- CAM350- Downstream Technology
 - <http://www.downstreamtech.com/cam350.html>
- GC Prevue- FREE Gerber Viewer
 - <http://www.graphicode.com/>
- GerbTool- Wise Software
 - http://www.wssi.com/index.php?option=com_content&task=section&id=16&Itemid=167



IPC: Formerly known as

Institute of Interconnecting and Packaging Electronic Circuits

- **IPC Guides the electronic interconnection industry through dramatic changes. IPC brings together designers, board manufacturers, assembly companies, suppliers, and OEM's.**
- **IPC is responsibly for creating over 200 standards**



IPC: Certification

- IPC also has a certification program which allows one to be certified as:
 - Certified Interconnect Designer (C.I.D.)
 - Proves a designer's knowledge of how to transform a schematic into a reliable rigid PCB design.
 - Advanced Certified Interconnect Designer (C.I.D.+)
 - Recognition of a designer's accomplishments. Exam covers forty key objectives which go beyond the C.I.D. certification.



IPC: C.I.D. Certification

- Interested individuals who wish to be certified should obtain the following articles:
 - IPC-2221A: Generic Standard on Printed Board Design (\$30 members/\$60 nonmembers)
 - IPC-2222: Sectional Standard on Rigid Organic Printed Boards (\$20/\$40)
 - IPC-D-325A: Documentation Requirements for Printed Circuit Boards (\$25/\$50)
 - IPC-T-50G: Terms and Definitions for Interconnecting and Packaging Electronic Circuits (\$40/\$80)
 - PWB-CRT-SGA: PWB Designer Certification Study Guide (\$35/\$70)

IPC: C.I.D. Certification

- Shown to the right are some of the sample questions available on the IPC Website

1. If analog circuits are restricted to the center of a vertically mounted board, what is the effect of placing a power supply in the connector zone, below the analog circuits?

- A. the analog circuit performs better
- B. the heat from the power supply causes instability
- C. the power supply, influenced by the analog circuit, is erratic
- D. the analog circuit requires additional protection to overcome spikes

2. Which two factors are the primary influence for the layer assignment in a multi-layer printed board structure?

- A. the bonding material used
- B. the requirements of solder mask
- C. the direction of conductor routing
- D. the relationship of core to prepreg
- E. the balanced distribution of copper
- F. the characteristics of the surface layer

3. What are the three most common dielectric thicknesses of copper-clad laminate used to produce double-sided printed boards?

- A. .50mm [0.020"]
- B. .75mm [0.030"]
- C. 1.00mm [0.040"]
- D. 1.50mm [0.060"]
- E. 2.00mm [0.080"]
- F. 2.40mm [0.090"]



IPC: For More Information

- IPC Homepage

- <http://www.ipc.org/>

- Online Store – Purchase Articles

- <http://webvision.ipc.org/scripts/mgrqispi.dll?APPNAME=IPCWEB&PRGNAME=TOCFRAME&ARGUMENTS=-N,-N,-A,-A,-N50>

- Certification

- <http://dc.ipc.org/certification.htm>



Keyboard Shortcuts



References

- Bogatin, Eric. *Signal Integrity- Simplified*. Upper Saddle River, NJ: Prentice Hall, 2006
- Schmitt, Ron. *Electromagnetics Explained*. Boston, MA: Newnes, 2002
- Johnson, Howard, and Graham, Martin. *High Speed Signal Propagation- Advanced Black Magic*. Upper Saddle River, NJ: Prentice Hall, 2003
- Altium Designer Learning Guides
<http://www.altium.com/community/support/learningguides/altiumdesigner6learningguides/>