## Introduction

Custom PCB in eagle, no exercise files. We are going to make out own.

## Chapter 1 – Getting Started with PCBS

### 1.1 Manufacturing options

How will you manufacture when design is completed?

Board houses:

Seeed – China - $4.90 cents – 10 copies of PCB

OSH – Park – purple PCB

Think of cost, time and limitations.

Wll come back when design is complete and we are looking to export to manufacturing.

### 1.2 Download and First time set up

Signal Planner Layers

Board area – 80cm2

This is for the free version.

### 1.3 Control Panel

Libraries – common electrical component definitions

Design Blocks – homemade designs that can be used in other designs

Design rules – rules from particular manufacture that is to do with their particular equipment.

Scripts – stuff that you can teach eagle how to do – advanced

SPICE models – imitate electrical activity within a circuit test before produce

Projects – where files are stored – all stored in EAGLE directory inn documents for now. So copy and paste if you need anything else.

## Chapter 2: Your First Board

### 2.1 Annotation Severed Notice

When this error appears close eagle and reload from most recent save point as you might be doing work you cannot save.

### 2.2 Package Types to use

Two types of packages – through hole and surface mount

Through hole – larger elements and often more expensive than SMT

SMT – smaller and fit more on a board – often cheaper

This course we will work with both

### 2.3 Download your libraries

It’s really hard to search for already built in libraries

Let’s download some new libraries – once added in you need to make sure you put it into the libraries folder and click use all. Green dot will let you know it is added in and ready to use.

### 2.4 Add in parts

Schematic and new files are the ones you will be working in – always want schematic to go first

Turn grid on – all parts constructed based on the assumption of 0.1inch grid

Moving and rotating is very cumbersome – have to right click and the select rotate or move. The ESC to move it in the same way

### 2.5 Incorporate Names and Values

To rename – click the rename icon and then click the middle of the item. Do the same for the value of resistors.

To wire – you click the little green thing that looks like wires and then hover over the ports and a green circle should pop up.

### 2.6 Develop the Board layout

Convert abstract info to board info – click into board mode.

Red marks let you know that component do close to edge and it won’t be made correctly.

Layer 20 is the dimension layer – the board size.

### 2.7 Add in routing

Can just type auto into the little bar to show auto trace

Rip up takes all the routes and takes them back to air wires

## Chapter 3 Custom Components

3.1 Create your own library

File -> New -> Library

Library is the accumulation of a symbol and a footprint.

Build new or take copies of parts and put them in easy to find library

3.2 Working with datasheets

Important pieces of information

Download from any manufacturer

Used AP 1117

Need tto make sure you are working with the right package.

Get the overall dimensions of the device as it sits in the circuit board

Then need pin size, pin numbers, name and any values so you can add it all into eagle

3.3 Create Footprints – Pads.

Check your units of measure before making anything!!!

Change the grid size – remember to zoom in if you need to.

Move (>0 0) (0 2.9); – move the pad up 2.9mm on the axis but not the x

3.4 – Create Manufacturing Markings

Still missing info to let the manufacturer or assembler to let you know orientation etc.

Layer 51 (tdocu) – top documentation.

Take line tool – mark the top part of IC and then go to the negative coordinates for the opposite parts.

Put a little dot by pin 1

3.5 Create Footprints – Visible Markings

Layer 25 tNames – give it a place holder name to start with

Layer 27 tValues – give it a placeholder Value so it can be reused.]

View – Layers – Show layers you can see all layers and then can also select particular layers so you can edit really busy boards at a specific layer and place

Add lines to tPlace – silkscreen layer – visual indicator of where part is going to go

3.6 Create Symbols and Pins

Open the library and then add a symbol – give it the name you want

The add the pins in and give it the general shape. Label the pins and add those placeholders for name and value

Don’t have to worry about adding names in etc.

3.7 Create a Device

Open Library and click add device

Then you have to add the device in and then press connect – then you have to link the pins.

Click on when it comes to value

3.8 Placing a Custom Part

You can make stubs and combine them together that way – label each stub trace and it should let you automatically connect them all

3.9 Incorporate custom outlines

3.10 Add custom imagery

Not the best program for making custom stuff for example stands etc.

File -> import there you can import loads of things – this is how you will import the gerber files

3.11 – Custom Imagery

There is some more custom stuff you can do on the silkscreen layer

Import a bitmap – then it will take a little bit of time to load it

22 is the silkscreen layer

Delete text that loads with the image

Images on the back select the layer so you don’t delete anything else. In this case bPlace.

## Chapter 4 Manufacturing

4.1 Electrical rules check

Takes place in schematic view – type in erc

Get errors and warnings

4.2 Design Rule Checking DRC

Same as before but you type in drc – this is done in the board view

Electrical rules are universal – design rules are based on manufacturers

Can download a dru you can download a design rule for a particular board house.

And then just press load and then it will take over the other stuff.

4.3 Generate data and upload to manufacturer.

Load the CAM processor and then you can see all the gerber files.

Sometimes using bitmaps will not show you it on the preview of the board. Pick manufacturer if this is the case