

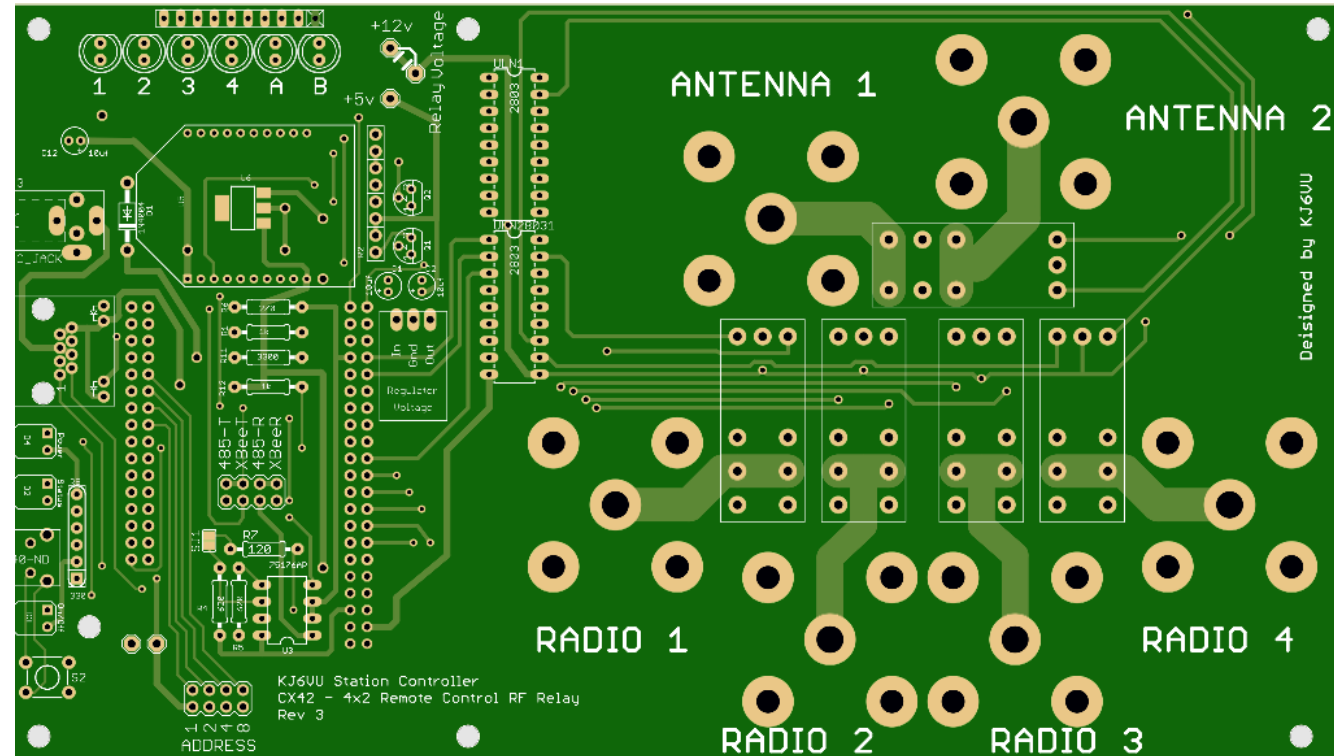
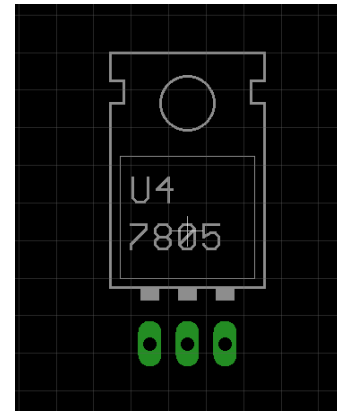
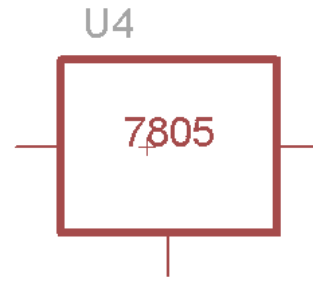
“PCB Design 101”
Session 1 – The Basics
George KJ6VU
ham radio workbench . com

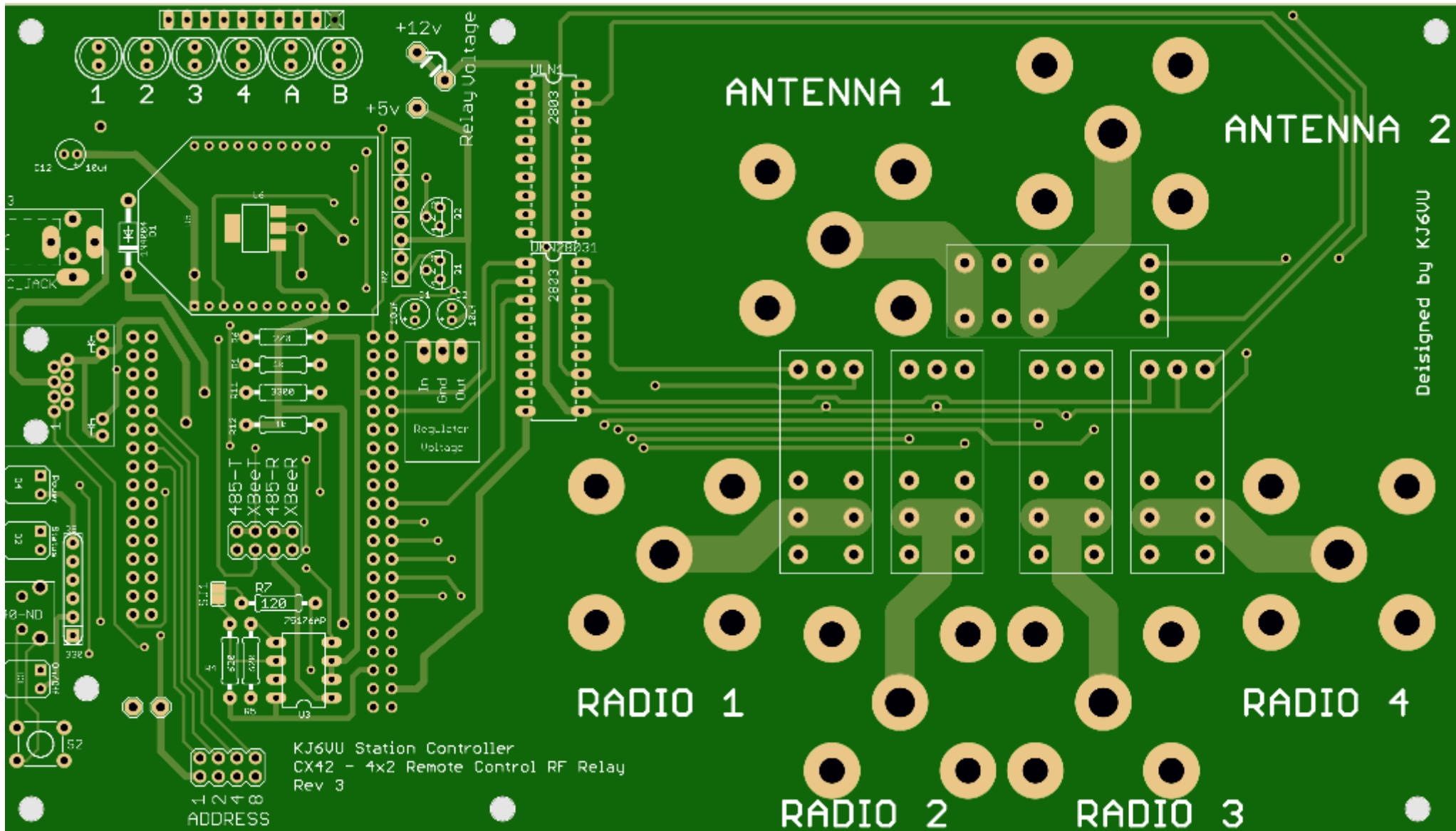
Webinar Goals

- Part 1 - Lightning round - Idea to Fab in 30 minutes
- Part 2 - Making a custom BenchDuino expansion board

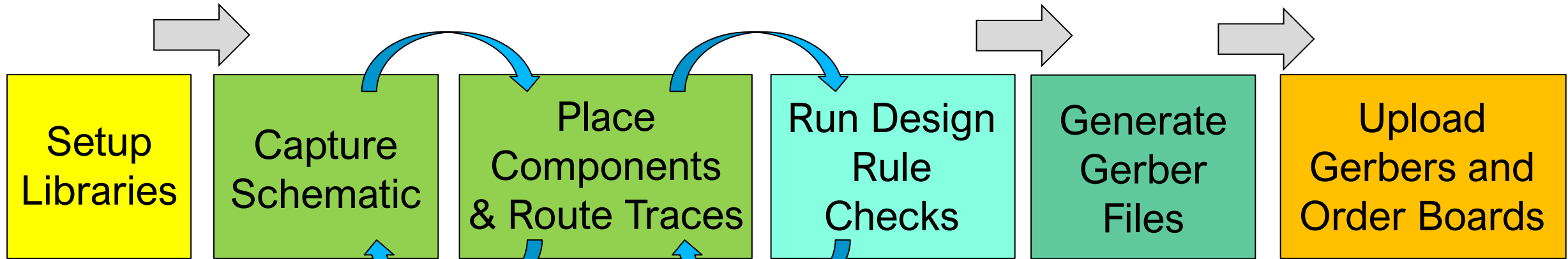
PCB Lingo

- Symbol
- Footprint
- Trace
- Pad
- Via
- Layer
- Silkscreen
- Soldermask
- Gerber files



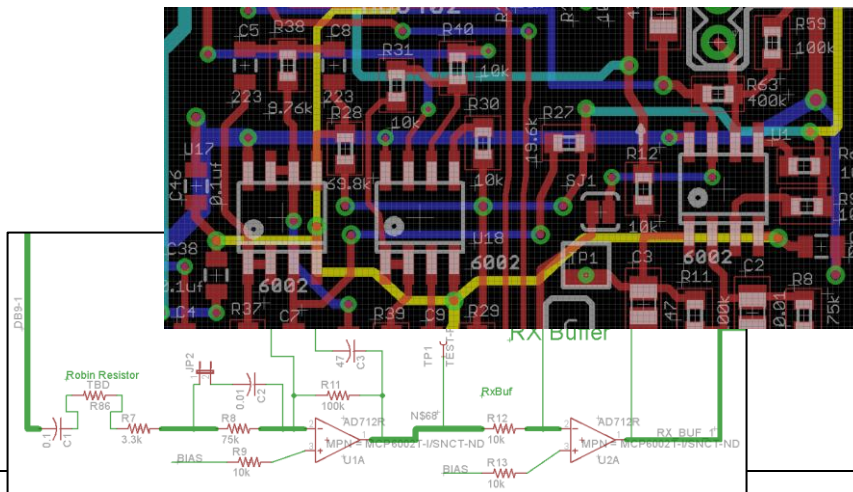


Design Flow

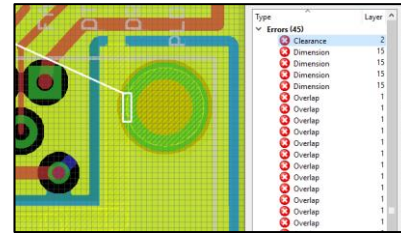


- Install libs
- Sparkfun
- Adafruit
- Seedstudio
- Your library
- Create your own parts

Interactive design
90% of your time



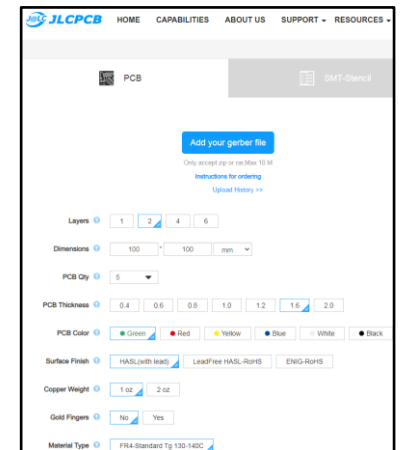
Do this at the end



```

M48
M72
T01C0.0150
T02C0.0236
T03C0.0300
T04C0.0320
T05C0.0360
T06C0.0390
T07C0.0400
%
T01
X20650Y14600
T02
    
```

- JLCPCB
- Seedstudio
- CircuitHub

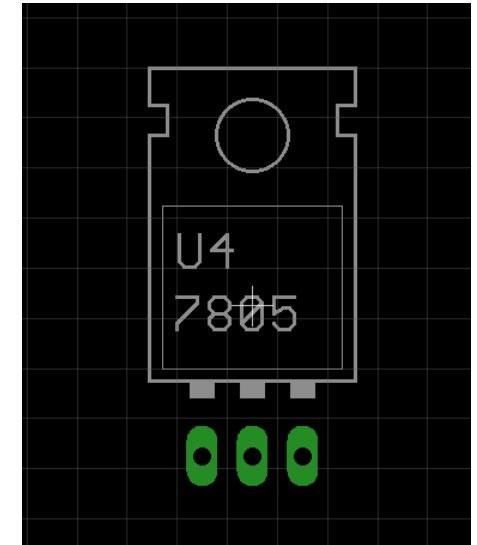
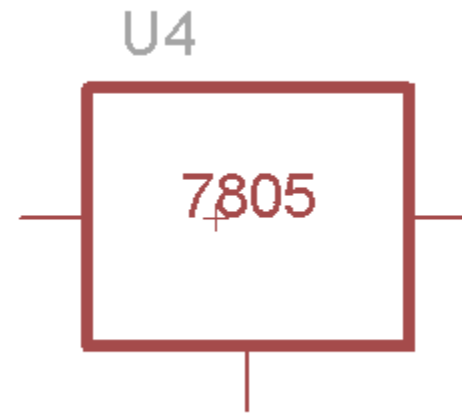


Install Eagle Free Version

- Free version limits
 - 2 schematic sheets
 - 2 layers
 - 12 square inches
- Download Eagle
- Install and create an Autodesk account

Install Libraries

- Library contains components
- A component has 2 views
 - The schematic symbol
 - The physical package footprint
- Popular library sources
 - Sparkfun
 - Adafruit
 - Seedstudio
- Build your own library
 - Uncommon parts
 - Special graphics (logos)
 - PCB modules



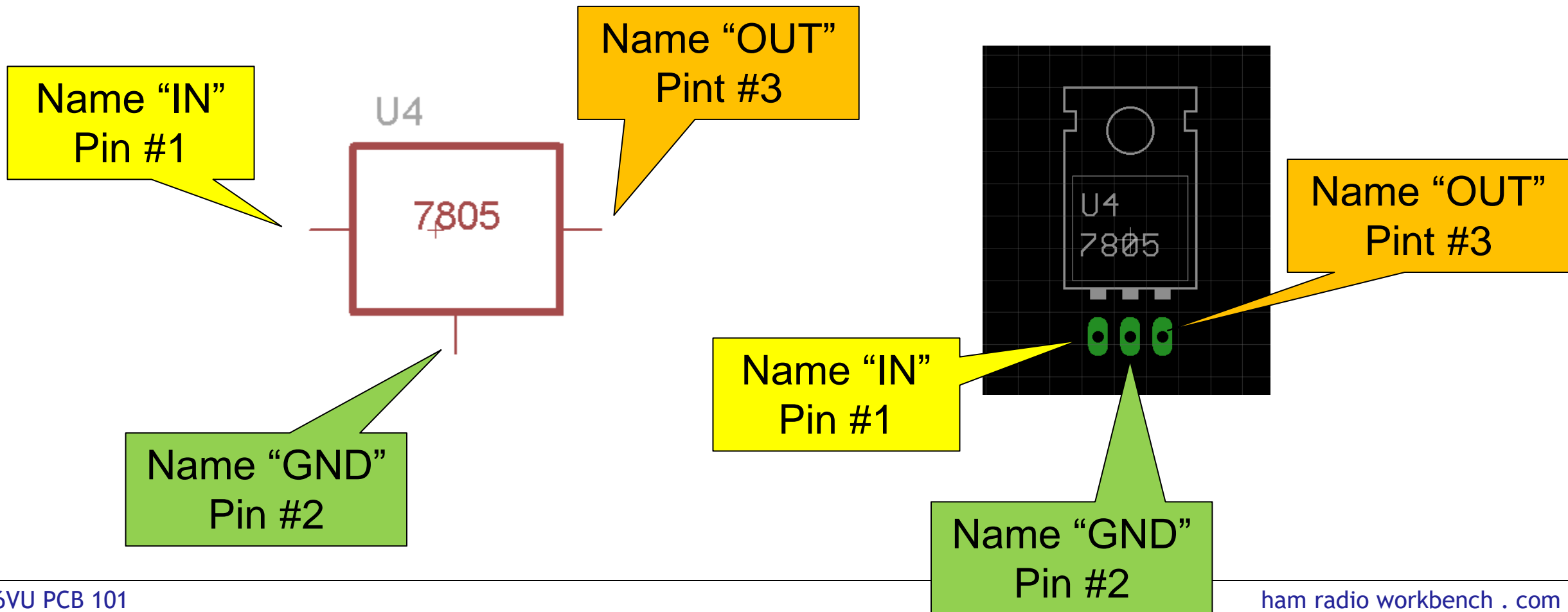
Copy files into `\Program Files\EagleX.X\lib`

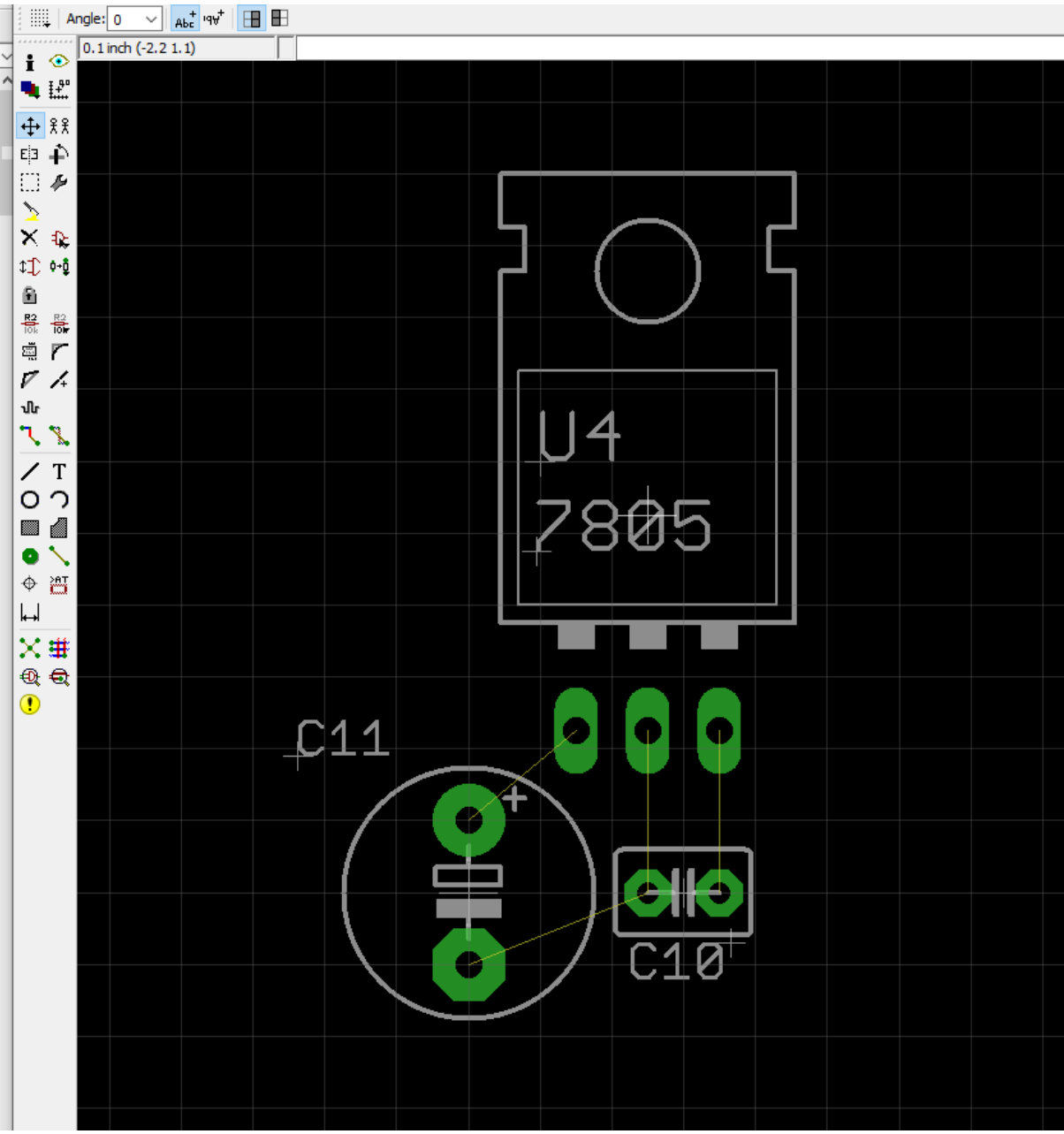
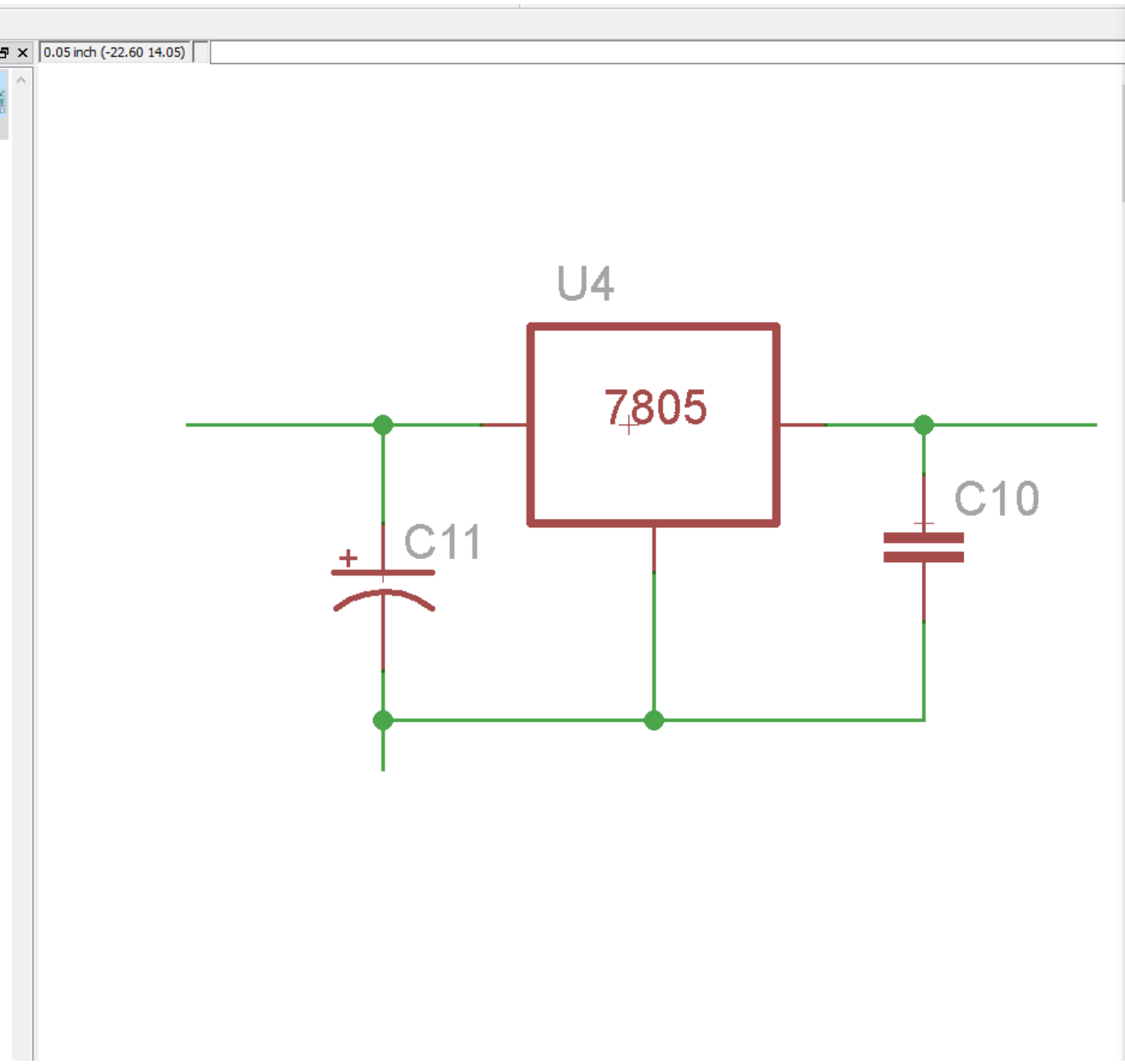
Install CAM Files

- Computer Aided Manufacturing files
 - Tells the Eagle software how to convert the PCB layout into manufacturing files
 - Format is called Gerber Files
- Seedstudio is a good source for typical CAM files
 - [Download 2 layer CAM file](#)
 - [Download 4 layer CAM file](#)
- Copy files into \Program Files\EagleX.X\cam

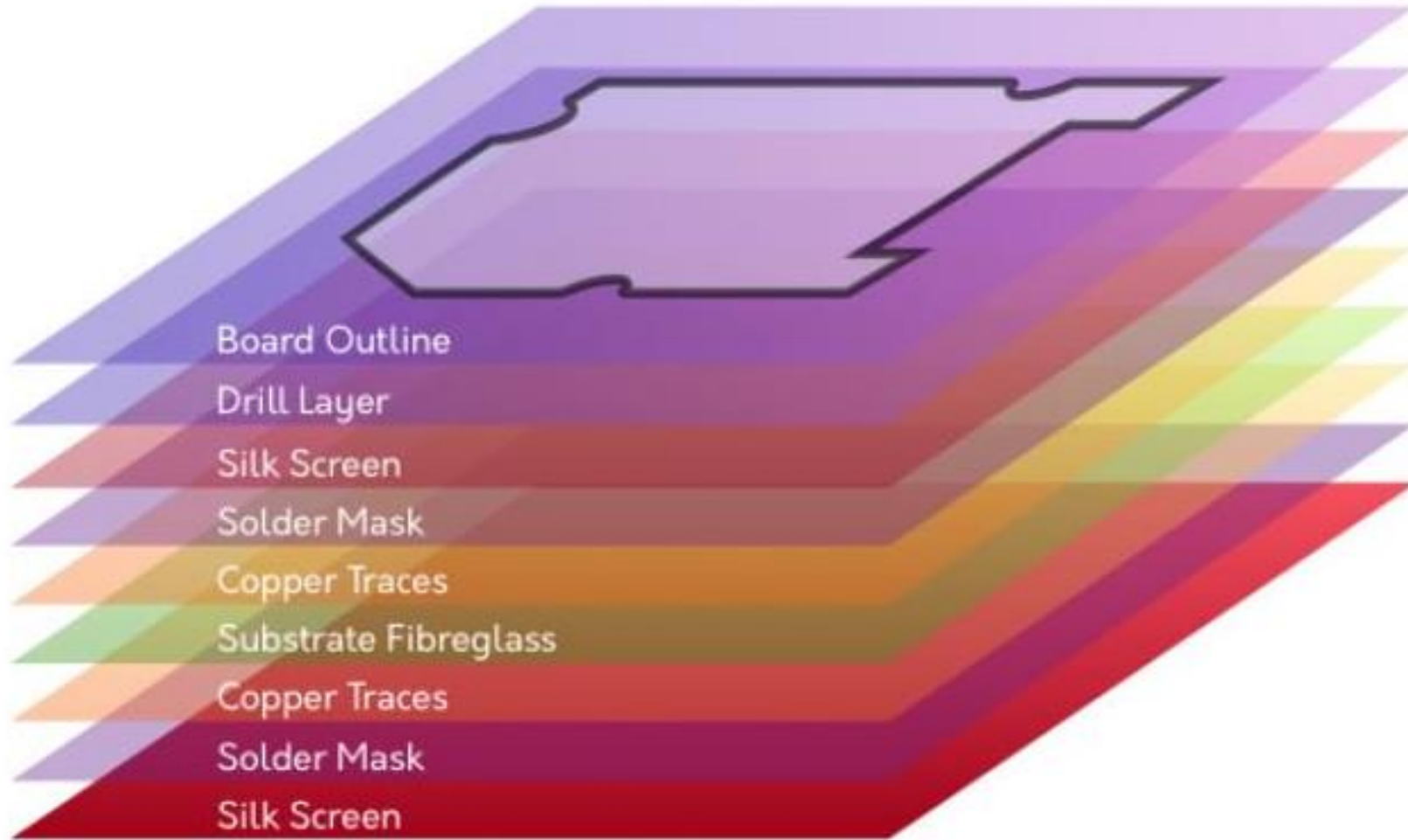
What's the big deal about good PCB tools ???

The data base understands the connection between a schematic pin and a physical pin





Layers



- 1 Top copper traces
- 16 Bottom copper traces
- 20 Board outline
- 21 Top silkscreen
- 22 Bottom silkscreen