A taste of EagleCAD

SDP ‘06

Matt Brennan
mattb@ieee.org
30 March 2006

Note: this will be best viewed in slide-show mode
Terms

• **Footprint**: what a part looks like on the board

• **Airwire**: a line in the layout indicating a connection needing to be made

• **Silkscreen**: notation on PCB (no connection)

• **Net**: connections between a group of pins
Control Panel

- r-click new project
- r-click new->schematic
Making a part

- Symbol + Footprint = Device
- Devices go in to schematic

Create/Edit Device, Footprint, Symbol

Command bar
In command bar:   copy so08@40xx so08
Connect device

Add symbol to device

Make new package, Choose footprint

Connect symbol pins to footprint pads

Save library, and “use” it through control panel
If you decide to make your own footprint:

- Remember to mark pin1 on something that will be printed
- Put >NAME and >VALUE on the respective layers
- Print the footprint in actual size, line up the part to make sure you got it right
Schematic Editor
Toolbar

- **Add part**
- **Name:** it works on wires, too
- **Smash:** move pieces of part
- **Design rule check:** DO IT
- **Label:** wires, again
Some useful libraries

- rcl – resistors, caps, inductors
- con-lstb, con-lsta – standard connectors
- con-subd – DB-# connectors
- con-coax – SMA, etc
- linear – op-amps
- solpad – extraneous connections
Layout

- Red: top
- Blue: bottom
- Green: through-hole
- White: dimensions & silk screen
- Can adjust dimensions w/ Move
Layout Toolbar

Route

Un-Route

Ratsnest – redraw air wires

AutoRoute – not all it’s cracked up to be

DRC – Set & Check clearances/layers
DRC - Layers

Setup: (1*16)
Two Layers

Setup: (1*2+15*16)
Four Layers
DRC - Clearances

- Specified by board house
- Larger => Cheaper
- 6/6 is small, up to 10/10+
- Careful: some SMT packages require small clearances!

Also check: minimum drill size. 12.5 is safe
Grid

Normal snap

Snap while holding ‘alt’

Also set grid in Autoroute options: smaller grid gives router more flexibility, but takes longer to route
Routing

• High-current traces should be WIDE => less resistance => less voltage drop
• Route power & ground first, “by hand”
• Especially if you are using SMD parts, READ the layout section of the data sheet
  – Anecdotal: DC-DC converters can fry if their passive components are too far away, or traces are too small
CAM Processor

Airwires (NOT connected yet)

Top wires (red)
Polygon tool

- For large copper areas. Draw polygon, name (with net name), hit ‘ratsnest’ to fill
CAM Processing: Files for board house
CAM Processing

• Two jobs. File->open
  – gerb274x   (copper data)
  – excellon   (drill data)

• Add (a) section(s) if doing silkscreen on both sides, or additional layers

• Cheap boards have no silkscreen
  – put text on the Top or Bottom copper layers
  – make sure there are no accidental connections
Other notes

• ALWAYS run ERC & DRC. Then double-check by eye
• Rip all routing?

• Power/GND planes (multiple layers):
  – in layout, go to ‘layer setup’ and name a layer $GND or $VCC, or $netname

• Symbol: Multiple pins with same name?
  – VCC$1, VCC$2, VCC$3: anything after $ won’t show in schematic

• Net classes: can define different min sizes for different types of connections (pwr vs data)
• Most commands are available from command bar
• Minimize vias in design
  – less resistance, sometimes cheaper boards
• Check status bar for: “Autorouter: 100% finished.”
  – Otherwise, find what it missed