



A taste of EagleCAD

SDP '06

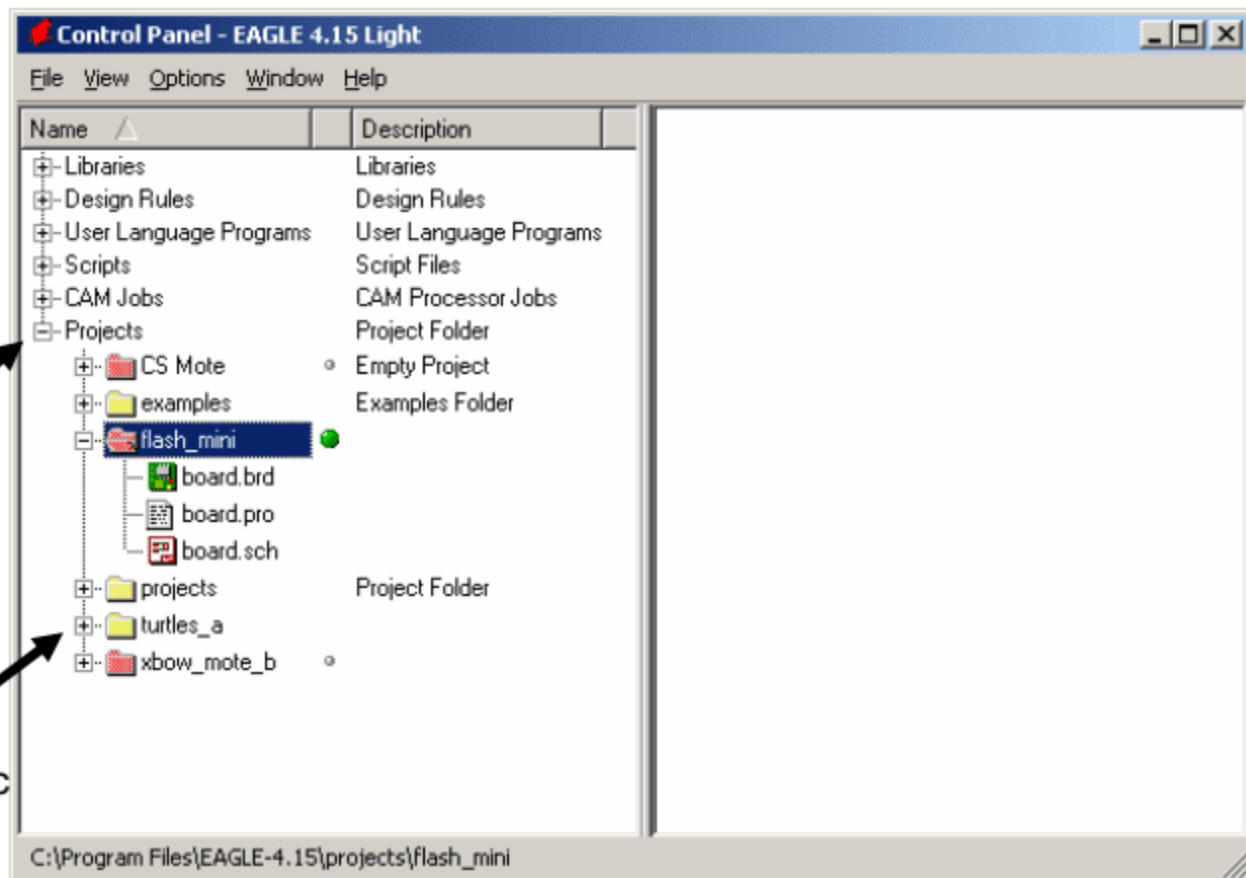
Matt Brennan
mattb@ieee.org
30 March 2006

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

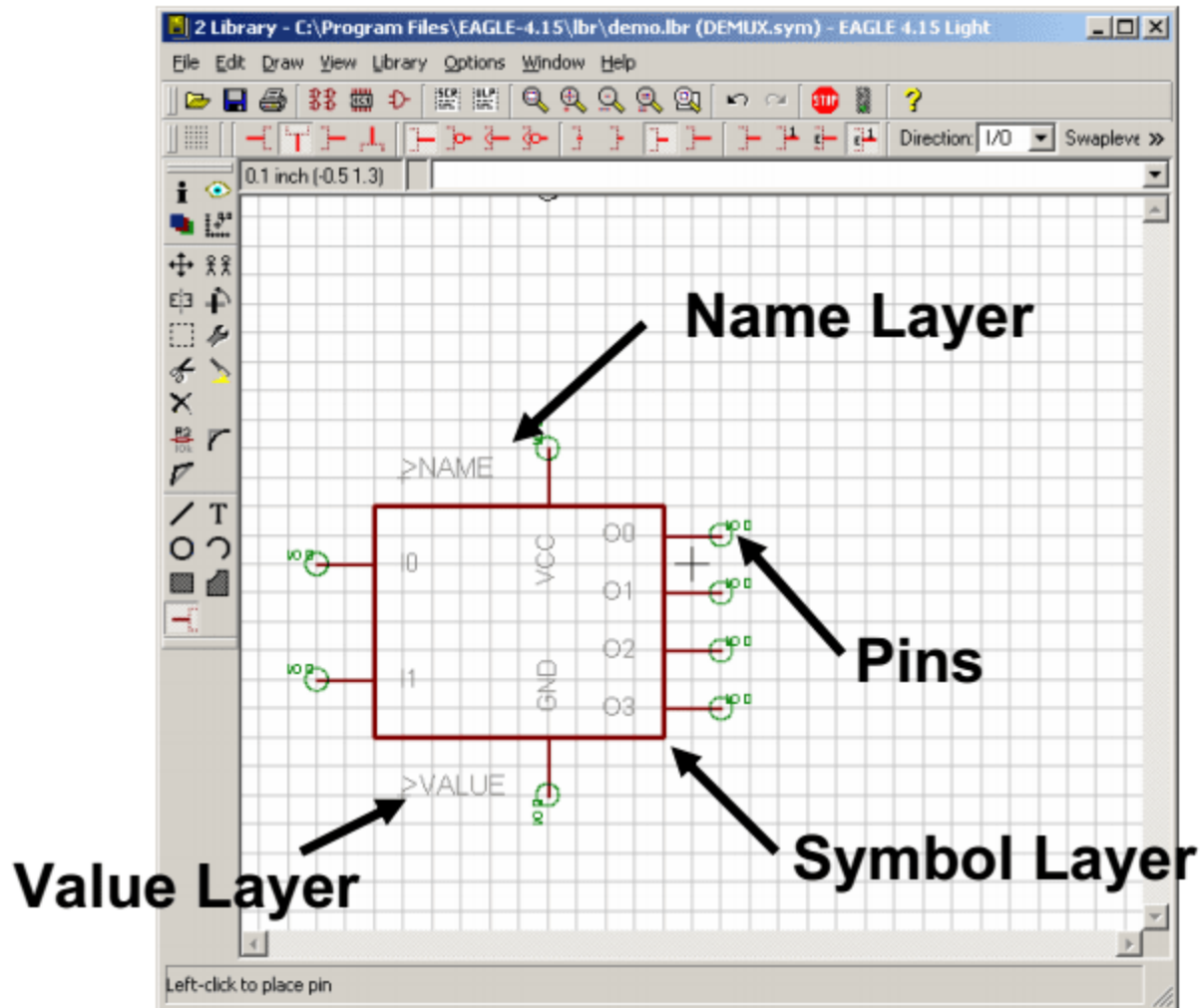
Making a part

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



**Create/Edit
Device, Footprint, Symbol**

Command bar

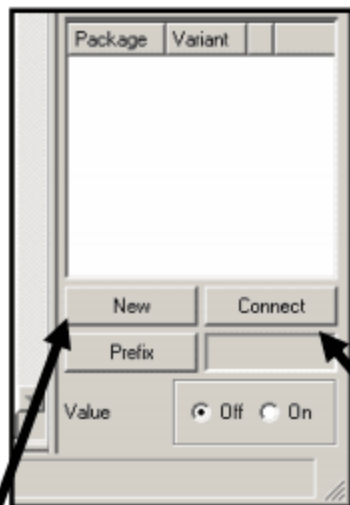


In command bar: `copy so08@40xx so08`

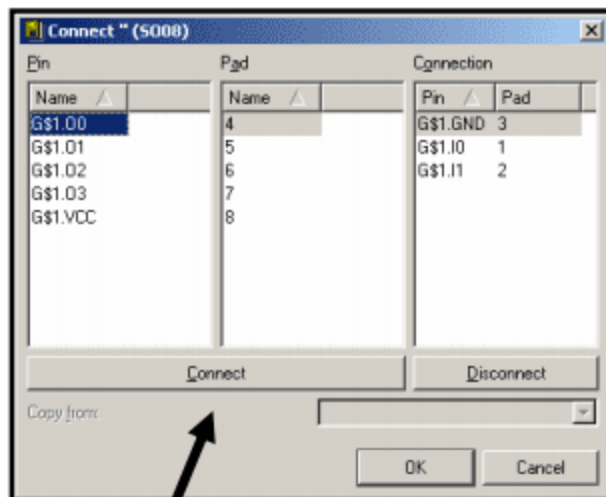
Connect device



Add symbol
to device



Make new package,
Choose footprint



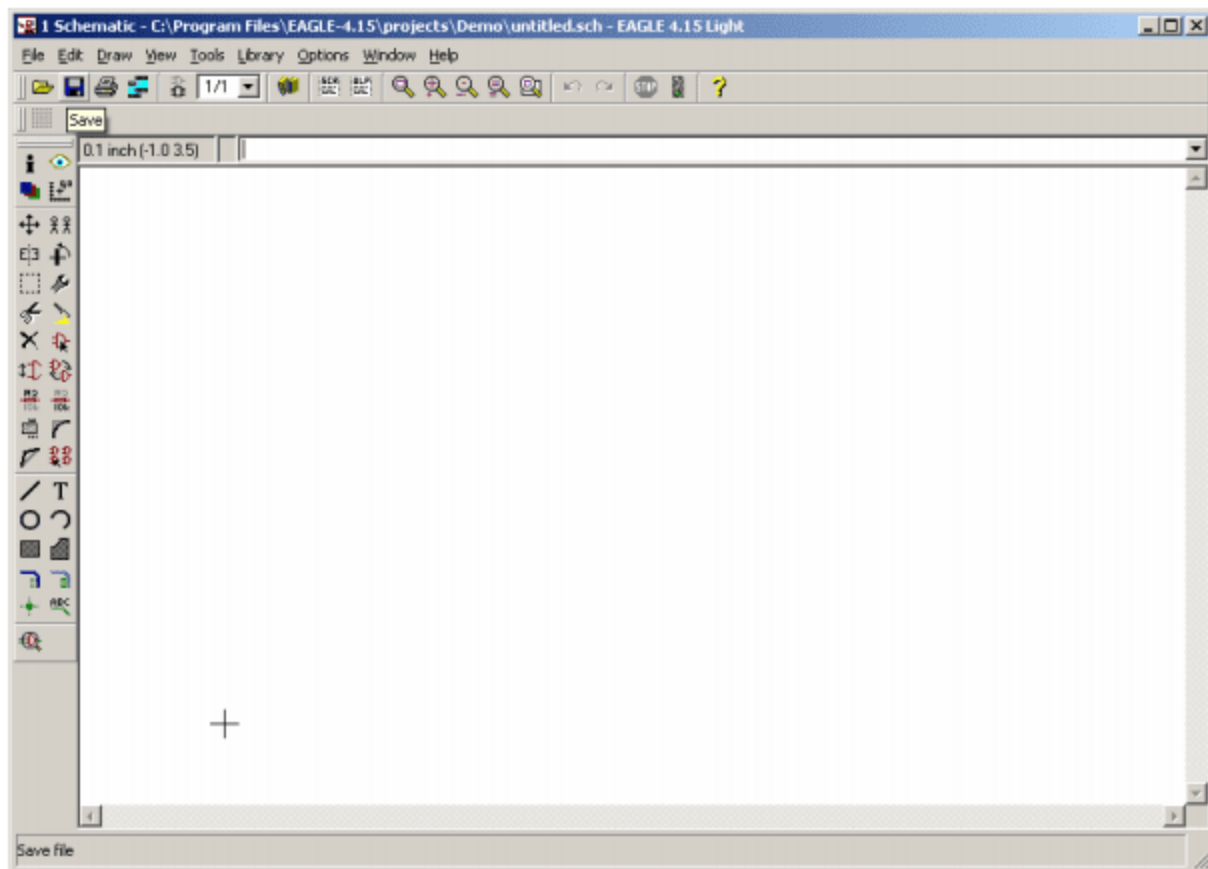
Connect symbol pins
to footprint pads

Save library, and "use" it through control panel

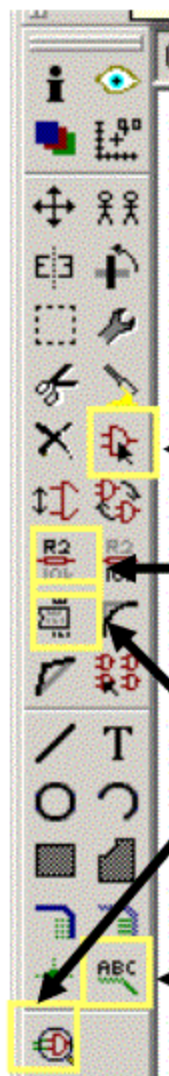
If you decide to make your own footprint:

- 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

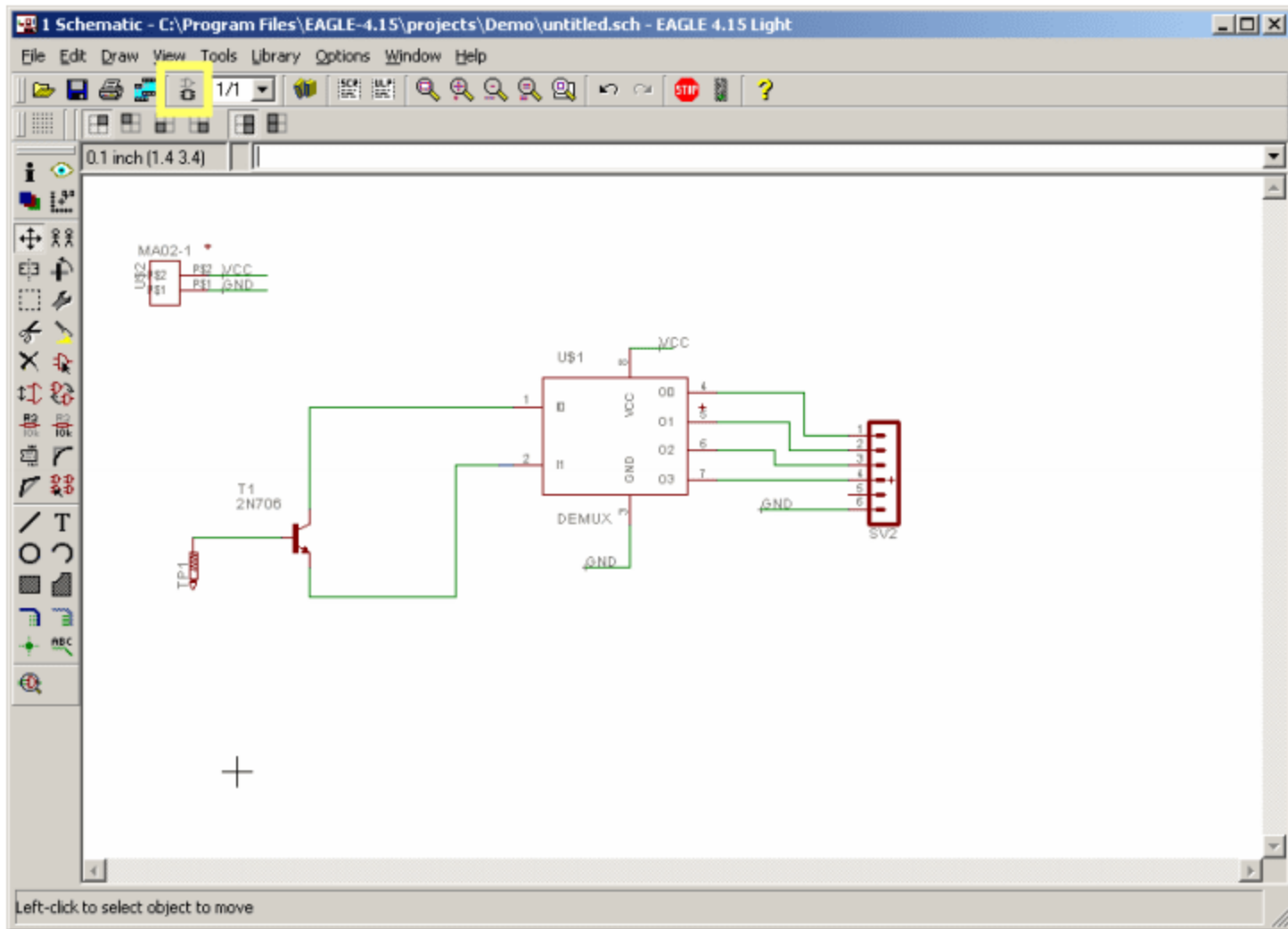
Smash: move pieces of part

Design rule check: DO IT

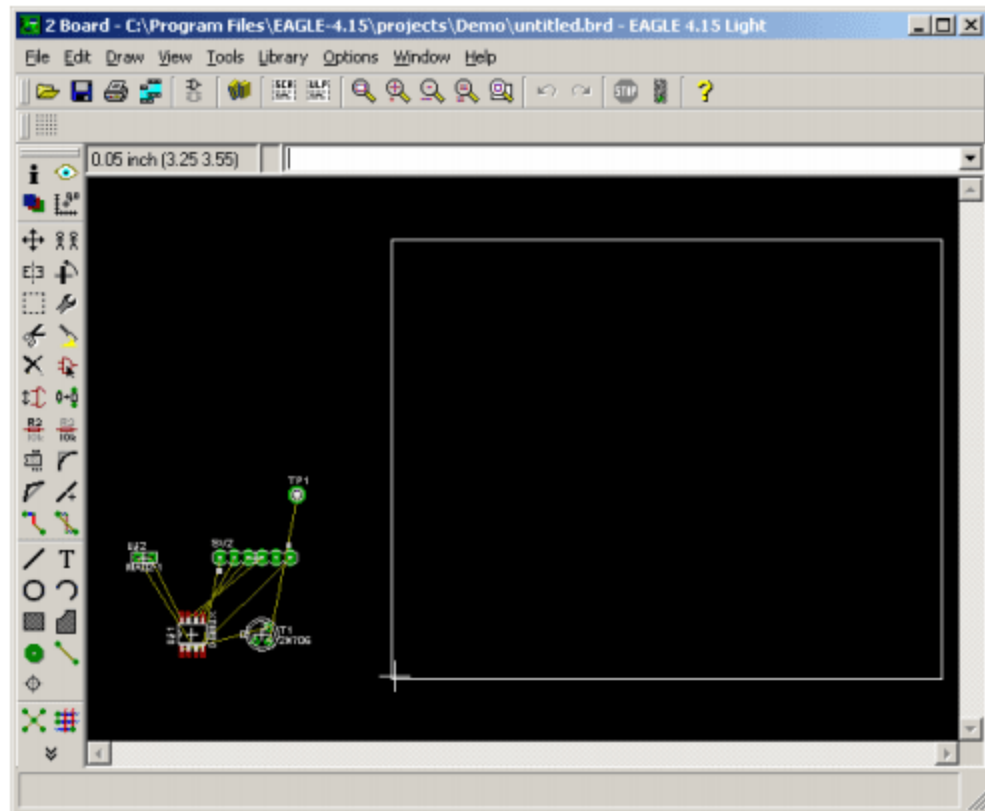
Label: wires, again

Some useful libraries

- rcl – resistors, caps, inductors
- con-1stb, con-1sta – standard connectors
- con-subd – DB-# connectors
- con-coax – SMA, etc
- linear – op-amps
- solpad – extraneous connections

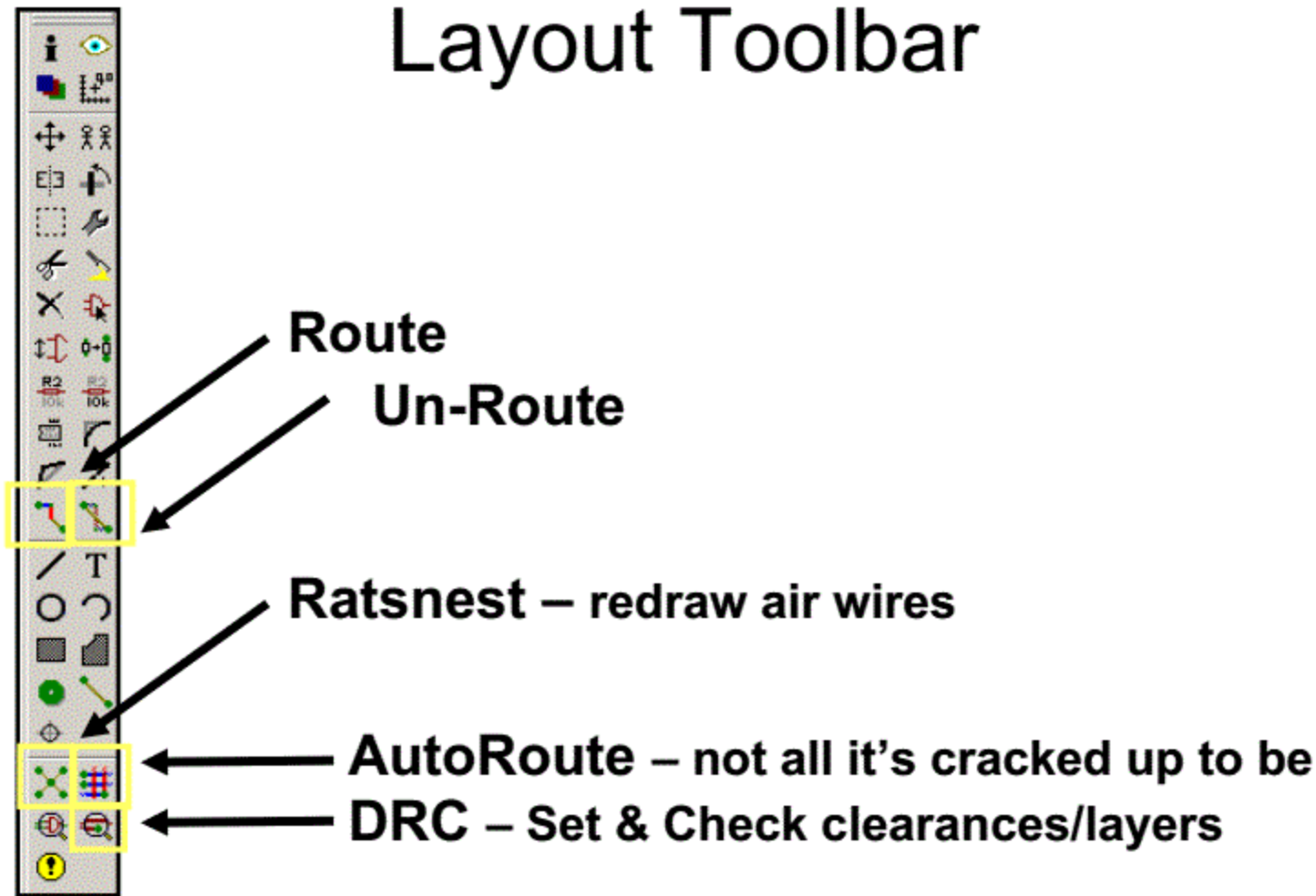


Layout

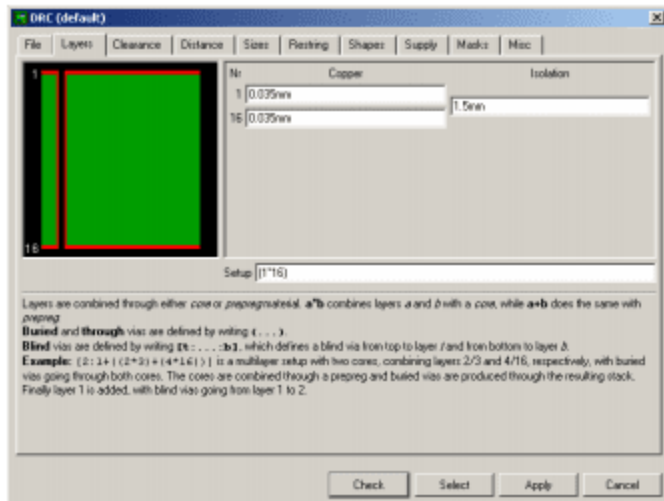


- Red: top
- Blue: bottom
- Green: through-hole
- White: dimensions & silk screen
- Can adjust dimensions w/ Move

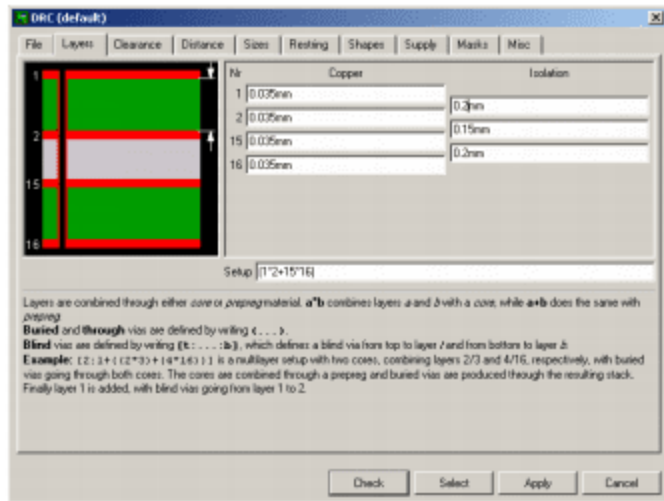
Layout Toolbar



DRC - Layers



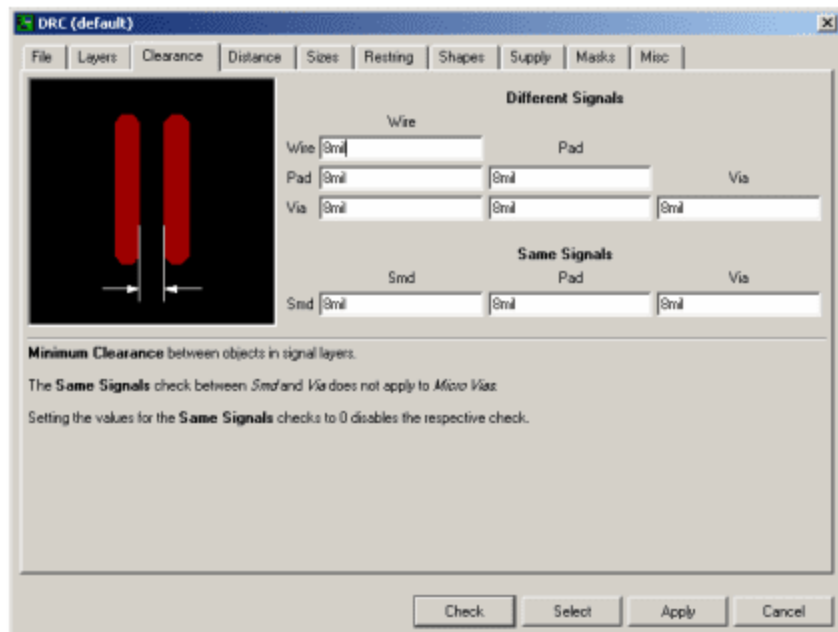
Setup: (1*16)
Two Layers



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

原创力文档
max.book118.com
预览与源文档一致, 下载高清无水印

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



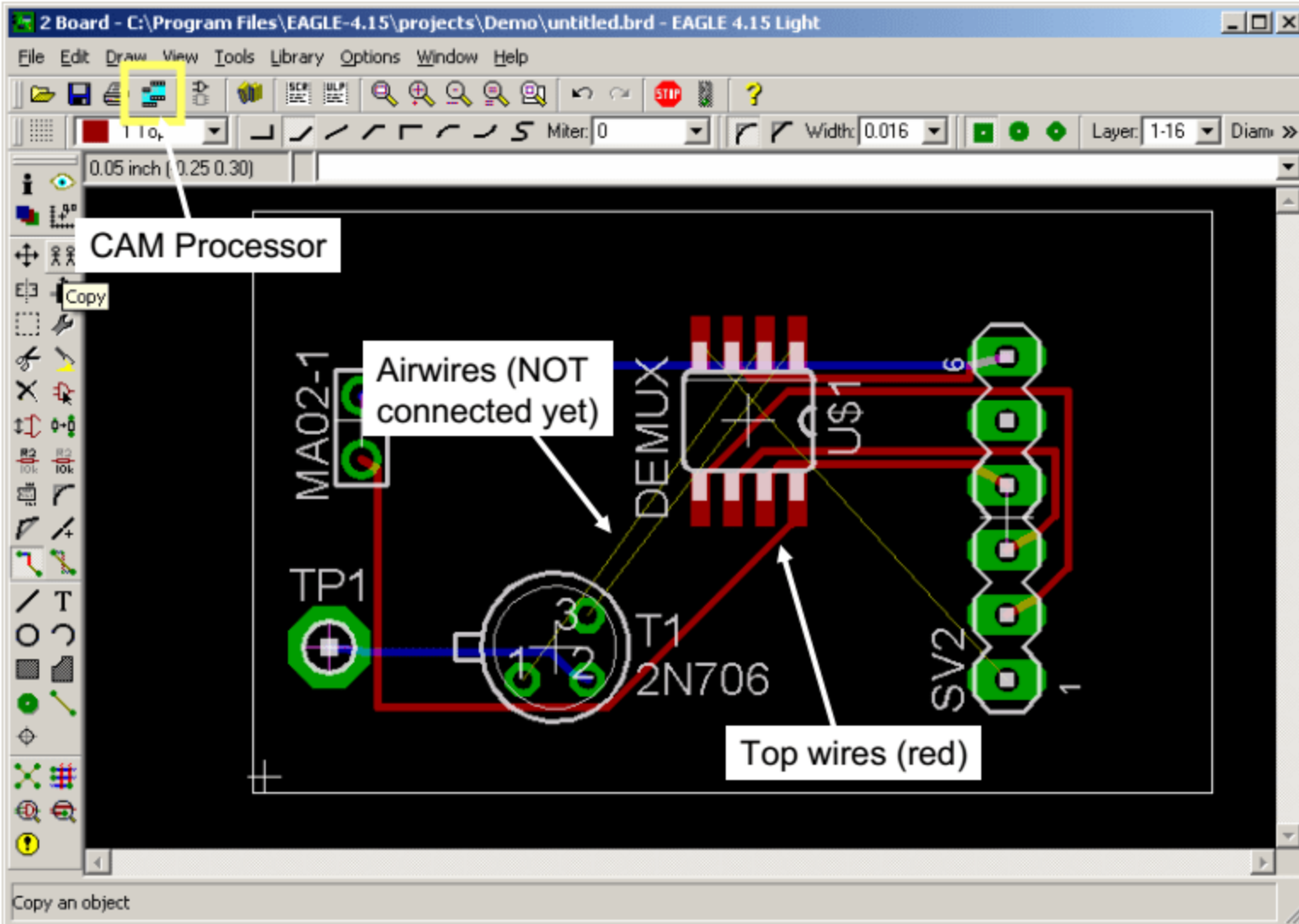
Snap while
holding 'alt'



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

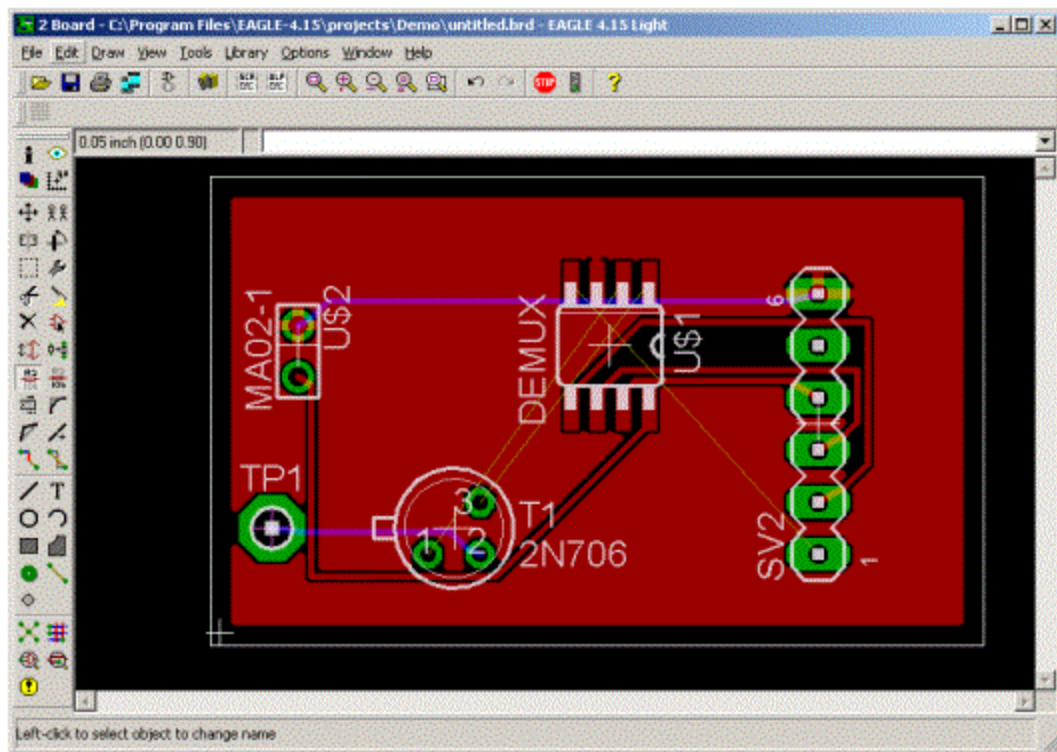
Routing

- 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

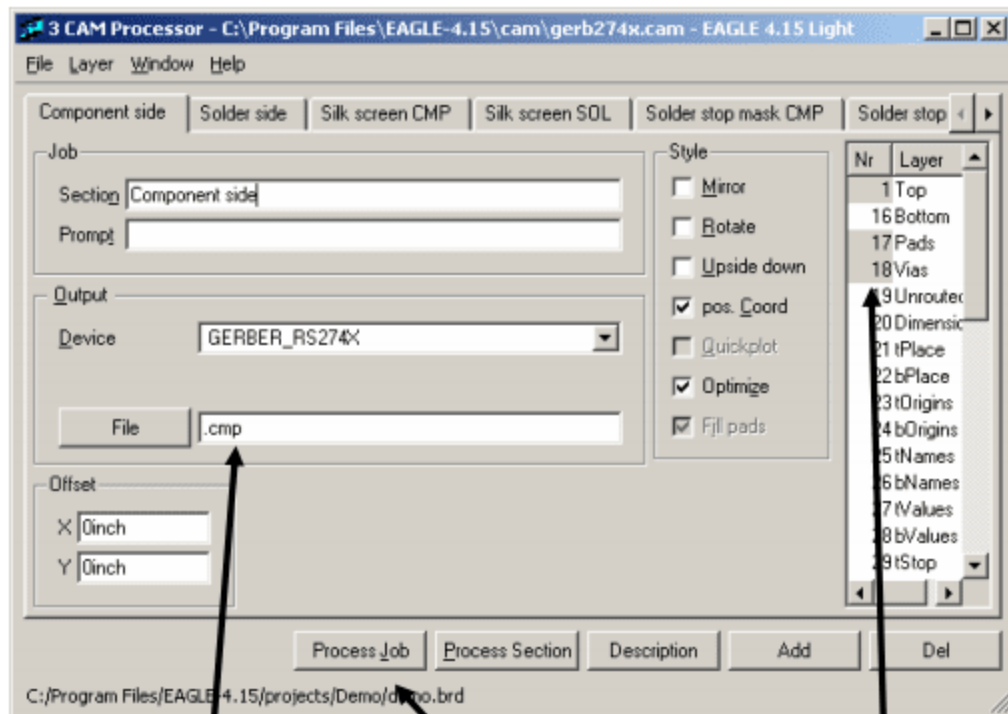


Polygon tool

- 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? The diagram consists of two small circuit board layout images separated by a right-pointing arrow. The left image shows a complex routing with multiple green traces and vias. The right image shows a simplified version of the same area, with a yellow rectangular highlight around a specific component or connection, suggesting a 'ripping' or simplification process.
- 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