
1 PCB design and computers 10

1.1 History 10

1.1.1 PADS evolution 14

1.1.2 Internet and PCB design 15

1.2 Documentation 15

1.3 Photoplotting 16

2 PCB design project flow 18

2.1 What is Schematic Design 18

2.2 Printed circuit board design 19

2.3 Libraries and other databases 21

2.4 Design changes and costs 22

2.5 Schematic capture software 22

2.5.1 Technology independent schematic packages 23

2.5.2 PCB-oriented schematic tools 23

2.5.3 Schematic editors with analog simulators 24

2.6 CAM - Computer Aided Manufacturing 24

2.7 Assembling the board 25

3 PADS General features 27

3.1 PowerLogic and PowerPCB design files 27

3.1.1 PowerLogic 27

3.1.2 PowerPCB and BlazeRouter 28

3.1.3 Components 28

4 Schematic design 29

4.1 Adoption of a schematic tool 29

4.1.1 PowerLogic basics 30

4.1.2 Keyboard commands (Modeless commands) 32

4.1.3 Database and the grid 32

4.1.4 Toolbars 33

4.1.5 Using colors 34

4.2 The database elements in PowerLogic and PowerPCB 35

4.2.1 Pin Decal 37

4.2.2 Decal 37

4.2.3 Parttype 38

- 4.2.4 Connections, Routes/Traces 39
- 4.2.5 Solid Copper (PowerPCB) 40
- 4.2.6 Flood Copper (PowerPCB) 40
- 4.2.7 Split/Mixed Plane (PowerPCB) 41
- 4.2.8 Free Text 41
- 4.2.9 2D-Lines 41

4.3 Implementing PowerLogic 42

- 4.3.1 Sheets 42
 - 4.3.2 Symbol sizes 43
 - 4.3.3 Off-page, Power & Ground Symbols 44
 - 4.3.4 Customising the special symbols 44
-

5 PADS Libraries 48

5.1 Custom Library 49

5.2 Organizing the libraries 51

- 5.2.1 Sharing libraries without a network 52
 - 5.2.2 Combining the libraries 53
 - 5.2.3 Maintaining libraries in multiuser environment 54
-

6 Designing schematic and pcb 55

6.1 Adding parts to schematic 56

- 6.1.1 Searching parts from the library 58

6.2 Adding connections 62

6.3 Off Page 63

6.4 Bus 64

- 6.4.1 Adding sheets 65
- 6.4.2 Hierarchical components 66

6.5 Creating parts 66

- 6.5.1 Sample: mounting hole 67
- 6.5.2 Sample part: Optoisolator 76
- 6.5.3 Connectors 85
- 6.5.4 Mounting holes and test pins 85
- 6.5.5 Decoupling capacitors 86

6.6 Component build memo 86

6.7 Layer stackup definition 87

6.8 Defining trace widths and spacings 88

- 6.8.1 Routing rules 89
- 6.8.2 How to select trace widths 90
- 6.8.3 Good schematic drawing practices 92

-
- 6.8.4 Copying blocks or moving to another page 93
 - 6.8.5 Copying schematic to other applications for documentation 94
 - 6.8.6 Bills of Materials listing 96
 - 6.8.7 Printing the schematic 98
 - 6.8.8 Changing line widths 100
 - 6.8.9 Sending data from schematic to pcb 101
 - 6.8.10 Crossprobing between applications 104
-

7 PCB Design 106

- 7.1 Modelling the board in PowerPCB 106**
 - 7.2 Board outline 106**
 - 7.3 Layers and documentation 108**
 - 7.4 Layer count and startup files 109**
 - 7.4.1 One layer boards 109
 - 7.4.2 Two layer boards 112
 - 7.4.3 PCB components 126
 - 7.4.4 Connections and traces 128
 - 7.4.5 Drafting lines (2D-lines) 128
 - 7.4.6 Free text 129
 - 7.4.7 Solid Copper 129
 - 7.4.8 Pour Copper, Flood Copper 130
 - 7.4.9 Split/Mixed Plane 130
 - 7.4.10 Libraries in PowerPCB 131
 - 7.5 Importing data from schematic 131**
 - 7.6 Placing components 132**
 - 7.6.1 How soldering process affects to placement 133
 - 7.6.2 Busses and placement 134
 - 7.6.3 Testing the placement 134
 - 7.6.4 Placing decoupling capacitors 135
 - 7.6.5 Automatic placement 135
 - 7.6.6 Checklist for component placement 136
 - 7.7 Routing the board 137**
 - 7.7.1 Autorouting 138
-

8 PowerPCB features 139

- 8.1 PowerPCB Graphical User Interface 139**
 - 8.1.1 PowerPCB toolboxes 140
 - 8.1.2 Display colors 146
- 8.2 Setup/Preferences in PowerPCB 148**

- 8.2.1 PowerPCB Preferences/Global 148
- 8.2.2 PowerPCB Preferences/Design 150
- 8.2.3 PowerPCB Preferences/Routing 152
- 8.2.4 PowerPCB Preferences/Thermals 153
- 8.2.5 PowerPCB Preferences/Auto Dimensioning 155
- 8.2.6 PowerPCB Preferences/Teardrops 156
- 8.2.7 PowerPCB Preferences/Drafting 157
- 8.2.8 PowerPCB Preferences/Grids 158
- 8.2.9 PowerPCB Preferences/Split/Mixed Plane 159

8.3 Startup settings 160

- 8.3.1 Library paths, establishing a custom library 164
-

9 PCB design using PowerPCB/BlazeRouter 165

9.1 Transferring data from schematic to pcb 165

9.2 Defining the board outline 166

- 9.2.1 Using DXF to import the board outline 169
- 9.2.2 Adding the mounting holes, ECO Registration 170
- 9.2.3 Mounting hole sizes 173

9.3 Net Display 174

- 9.3.1 Length minimisation rules 176
- 9.3.2 Spacing and trace widths in PowerPCB 176

9.4 Component placement 178

- 9.4.1 Using cross-probing in component placement 178
- 9.4.2 Decoupling capacitors on two layer boards 181
- 9.4.3 Routing Power and Ground on a multilayer board 182
- 9.4.4 Topology of Power and Ground (two layer boards) 183

9.5 Automatic component placement 183

- 9.5.1 Other tips for placement 185

9.6 BlazeRouter 186

- 9.6.1 Launching the BlazeRouter 186
- 9.6.2 BlazeRouter GUI 187
- 9.6.3 Interactive routing using FIRE 188
- 9.6.4 Batch routing in BlazeRouter 195

9.7 Interactive routing in PowerPCB 200

- 9.7.1 Using grid and optimum pad sizes 200
- 9.7.2 Gridless routing 202
- 9.7.3 Routing stages 202
- 9.7.4 Hints for modifying traces in PowerPCB 203
- 9.7.5 Interactive routing using Dynamic Route Editor 205
- 9.7.6 Editing a trace 206

9.8 Dynamic Copper areas 209

9.8.1 Copper pour 209

9.8.2 Split/Mixed Planes 216

9.9 Testpoints 222**9.10 Final tune-up and checking 223**

9.10.1 Comparing the schematic and pcb 223

9.10.2 Copper balancing 224

9.10.3 Checking the connectivity and spacings 225

9.10.4 Spacing checks 225

9.10.5 Design For Fabrication checks 227

9.10.6 Documentation markings on pcb 231

10 Documentation for PCB Fabrication - CAM 238

10.1 Gerber format 239**10.2 Gerber parameters in PowerPCB 243****10.3 Excellon format 245****10.4 The Excellon parameters in PowerPCB 246****10.5 Producing Gerber files from PowerPCB 248****10.6 Exporting the Drill files (Excellon) 258****10.7 A sample list of fabrication files 258****10.8 Assembly arrays 260**

11 General tips 262

11.0.1 Default Text Editor 262

11.0.2 Explorer settings 263

11.0.3 Opening text files without assigning the file extension 263

11.0.4 Tweak UI 264

11.0.5 Copy to clipboard as Name 264

12 Viewing and printing the Gerber files. 266

12.1 Loading the files 267**12.2 Modifying the Gerber files for lab use 267**12.2.1 Drilling 268

13 Fabrication of a Printed Wiring board 270

13.1 Checking the files 270**13.2 Creating the phototools from Gerber files 271**

- 13.3 Drilling 271**
- 13.4 Hole treatment for plating 272**
- 13.5 Transferring the images 272**
- 13.6 Adding the electrolytic copper 272**
- 13.7 Etching 272**
- 13.8 Solder resist 273**
- 13.9 Silk screen 273**
- 13.10 Milling 273**
- 13.11 Bare board testing 273**