Complete PCB Design Using OrCAD® Capture and PCB Editor

Kraig Mitzner
INTRODUCTION xi
ACKNOWLEDGMENTS xv

CHAPTER 1  
Introduction to PCB Design and CAD ........................................... 01
  Computer-Aided Design and the OrCAD Design Suite .................. 01
  Printed Circuit Board Fabrication ........................................... 02
  PCB Cores and Layer Stack-Up ............................................. 02
  PCB Fabrication Process .................................................... 04
  Photolithography and Chemical Etching .................................. 05
  Mechanical Milling ............................................................ 07
  Layer Registration ............................................................ 07
  Function of OrCAD PCB Editor in the PCB Design Process .......... 09
  Design Files Created by PCB Editor ....................................... 12
    PCB Editor Format Files .................................................. 12
    Artwork (Gerber) Files .................................................... 12
    PCB Assembly Layers and Files ........................................ 13

CHAPTER 2  
Introduction to the PCB Design Flow by Example ....................... 15
  Overview of the Design Flow ............................................. 15
    Creating a Circuit Design with Capture ................................ 15
  Designing the PCB with PCB Editor ..................................... 22
    The PCB Editor Window .................................................. 22
    Drawing the Board Outline ............................................. 24
    Placing Parts ............................................................. 25
    Moving and Rotating Parts ............................................. 25
    Routing the Board ....................................................... 27
    Creating Artwork for Manufacturing ................................... 31

CHAPTER 3  
Project Structures and the PCB Editor Tool Set ......................... 33
  Project Setup and Schematic Entry Details ................................ 33
    Capture Projects Explained ............................................. 33
    Capture Part Libraries Explained ..................................... 36
  Understanding the PCB Editor Environment and Tool Set ............. 38
    Terminology .............................................................. 38
    PCB Editor Windows and Tools ........................................ 39
    The Design Window ..................................................... 39
    The Toolbar Groups .................................................... 39
    Control Panel with Foldable Window Panes ......................... 44
    Command Window Pane ................................................ 46
<table>
<thead>
<tr>
<th>CHAPTER 7</th>
<th>Making and Editing Capture Parts</th>
<th>159</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Capture Part Libraries</td>
<td>159</td>
<td></td>
</tr>
<tr>
<td>Types of Packaging</td>
<td>160</td>
<td></td>
</tr>
<tr>
<td>Homogeneous Parts</td>
<td>160</td>
<td></td>
</tr>
<tr>
<td>Heterogeneous Parts</td>
<td>160</td>
<td></td>
</tr>
<tr>
<td>Pins</td>
<td>161</td>
<td></td>
</tr>
<tr>
<td>Part Editing Tools</td>
<td>162</td>
<td></td>
</tr>
<tr>
<td>The Select Tool and Settings</td>
<td>162</td>
<td></td>
</tr>
<tr>
<td>The Pin Tools</td>
<td>162</td>
<td></td>
</tr>
<tr>
<td>The Graphics Tools</td>
<td>163</td>
<td></td>
</tr>
<tr>
<td>The Zoom Tools</td>
<td>163</td>
<td></td>
</tr>
<tr>
<td>Methods of Constructing Capture Parts</td>
<td>164</td>
<td></td>
</tr>
<tr>
<td>Method 1. Constructing Parts Using the New Part Option (Design Menu)</td>
<td>164</td>
<td></td>
</tr>
<tr>
<td>Method 2. Constructing Parts with Capture Using the Design Spreadsheet</td>
<td>176</td>
<td></td>
</tr>
<tr>
<td>Method 3. Constructing Parts Using Generate Part from the Tools Menu</td>
<td>179</td>
<td></td>
</tr>
<tr>
<td>Method 4. Generating Parts with the PSpice Model Editor</td>
<td>180</td>
<td></td>
</tr>
<tr>
<td>Constructing Capture Symbols</td>
<td>194</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CHAPTER 8</th>
<th>Making and Editing Footprints</th>
<th>197</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction to PCB Editor's Symbols Library</td>
<td>197</td>
<td></td>
</tr>
<tr>
<td>Symbol Types</td>
<td>198</td>
<td></td>
</tr>
<tr>
<td>Composition of a Footprint</td>
<td>199</td>
<td></td>
</tr>
<tr>
<td>Padstacks</td>
<td>200</td>
<td></td>
</tr>
<tr>
<td>Graphical Objects</td>
<td>201</td>
<td></td>
</tr>
<tr>
<td>Text</td>
<td>202</td>
<td></td>
</tr>
<tr>
<td>Minimum Footprint Requirements</td>
<td>202</td>
<td></td>
</tr>
<tr>
<td>Optional Footprint Objects</td>
<td>202</td>
<td></td>
</tr>
<tr>
<td>Introduction to the Padstack Designer</td>
<td>203</td>
<td></td>
</tr>
<tr>
<td>Padstack Designer Parameters Tab</td>
<td>203</td>
<td></td>
</tr>
<tr>
<td>Padstack Designer Layers Tab</td>
<td>204</td>
<td></td>
</tr>
<tr>
<td>Footprint Design Examples</td>
<td>205</td>
<td></td>
</tr>
<tr>
<td>Example 1. Design of a Through-Hole Device from Scratch</td>
<td>206</td>
<td></td>
</tr>
<tr>
<td>Example 2. Design of Surface-Mount Device from an Existing Symbol</td>
<td>212</td>
<td></td>
</tr>
<tr>
<td>Example 3. PGA Design Using the Symbol Wizard</td>
<td>216</td>
<td></td>
</tr>
<tr>
<td>Flash Symbols for Thermal Reliefs</td>
<td>219</td>
<td></td>
</tr>
<tr>
<td>Mechanical Symbols</td>
<td>222</td>
<td></td>
</tr>
<tr>
<td>Mounting Holes</td>
<td>223</td>
<td></td>
</tr>
<tr>
<td>Creating Mechanical Drawings</td>
<td>225</td>
<td></td>
</tr>
<tr>
<td>Placing Mechanical Symbols on a Board Design</td>
<td>226</td>
<td></td>
</tr>
<tr>
<td>Blind, Buried, and Microvias</td>
<td>227</td>
<td></td>
</tr>
<tr>
<td>Using the IPC-7351 Land Pattern Viewer</td>
<td>229</td>
<td></td>
</tr>
<tr>
<td>References</td>
<td>232</td>
<td></td>
</tr>
</tbody>
</table>
# CONTENTS

## CHAPTER 9  
**PCB Design Examples**

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction</td>
<td>233</td>
</tr>
<tr>
<td>Overview of the Design Flow</td>
<td>234</td>
</tr>
<tr>
<td>Example 1. Dual Power Supply, Analog Design</td>
<td></td>
</tr>
<tr>
<td>Initial Design Concept and Preparation</td>
<td>237</td>
</tr>
<tr>
<td>Setting Up the Project in Capture</td>
<td>238</td>
</tr>
<tr>
<td>Preparing the Design for PCB Editor</td>
<td>245</td>
</tr>
<tr>
<td>Setting Up the Board</td>
<td>253</td>
</tr>
<tr>
<td>Design Rule Check and Status</td>
<td>276</td>
</tr>
<tr>
<td>Defining the Layer Stack-Up</td>
<td>278</td>
</tr>
<tr>
<td>Pouring Copper Planes</td>
<td>282</td>
</tr>
<tr>
<td>Verifying Connectivity between Pins and Planes</td>
<td>285</td>
</tr>
<tr>
<td>Defining Trace Width and Spacing Rules</td>
<td>289</td>
</tr>
<tr>
<td>Prerouting the Board</td>
<td>292</td>
</tr>
<tr>
<td>Manually Routing Traces</td>
<td>295</td>
</tr>
<tr>
<td>Finalizing the Design</td>
<td>298</td>
</tr>
<tr>
<td>Mixed-Signal Circuit Design in Capture</td>
<td>304</td>
</tr>
<tr>
<td>Defining the Layer Stack-Up for Split Planes</td>
<td>310</td>
</tr>
<tr>
<td>Setting Up Routing Constraints</td>
<td>316</td>
</tr>
<tr>
<td>Adding Ground Planes to Routing Layers</td>
<td>322</td>
</tr>
<tr>
<td>Example 3. Multipage, Multipower, and Multiground Mixed A/D PCB Design</td>
<td></td>
</tr>
<tr>
<td>Capture Project Setup for PSpice Simulation and Board Design</td>
<td>326</td>
</tr>
<tr>
<td>Multiplane Layer Methodologies</td>
<td>327</td>
</tr>
<tr>
<td>Designing the Board with PCB Editor</td>
<td>340</td>
</tr>
<tr>
<td>Assigning Vias to Nets</td>
<td>348</td>
</tr>
<tr>
<td>Alternate Methods of Connecting Separate Ground Planes</td>
<td>354</td>
</tr>
<tr>
<td>Example 4. High-Speed Digital Design</td>
<td></td>
</tr>
<tr>
<td>Layer Setup for Microstrip Transmission Lines</td>
<td>360</td>
</tr>
<tr>
<td>Constructing a Heat Spreader with Copper Pours and Vias</td>
<td>362</td>
</tr>
<tr>
<td>Determining Critical Trace Length of Transmission Lines</td>
<td>369</td>
</tr>
<tr>
<td>Moated Ground Areas for Clock Circuits</td>
<td>373</td>
</tr>
<tr>
<td>Gate and Pin Swapping</td>
<td>375</td>
</tr>
<tr>
<td>Using Swap Options</td>
<td>377</td>
</tr>
<tr>
<td>Using the Autoswap Option</td>
<td>380</td>
</tr>
<tr>
<td>Positive Planes</td>
<td>383</td>
</tr>
<tr>
<td>Positive Plane Artwork Production</td>
<td>389</td>
</tr>
<tr>
<td>Positive vs. Negative Plane File Sizes</td>
<td>389</td>
</tr>
<tr>
<td>Pros and Cons of Using Positive vs. Negative Planes</td>
<td>389</td>
</tr>
<tr>
<td>Design Templates</td>
<td></td>
</tr>
<tr>
<td>Making a Custom Capture Template</td>
<td>391</td>
</tr>
<tr>
<td>Making a Custom PCB Editor Board Template</td>
<td>391</td>
</tr>
<tr>
<td>Making a Custom PCB Editor Technology Template</td>
<td>392</td>
</tr>
</tbody>
</table>
CHAPTER 10
Artwork Development and Board Fabrication
Schematic Design in Capture
The Board Design with PCB Editor
Routing the Board
Placing Mechanical Symbols
Generating Manufacturing Data
Generating the Artwork Files
Generating Drill Files
Generating Route Path Files
Generating the Route File
Verifying the Artwork
Using CAD Tools to 3-D Model the PCB Design
Fabricating the Board
Receipt Inspection and Testing
Generating Pick and Place Files
References
APPENDICES
APPENDIX A List of Design Standards
APPENDIX B Partial List of Packages and Footprints and Some of the Footprints Included in OrCAD Layout
APPENDIX C Rise and Fall Times for Various Logic Families
APPENDIX D Drill and Screw Dimensions
APPENDIX E References by Subject
INDEX