

# Altium Designer 18 Essentials

**Altium**

## Altium Designer Essentials Lessons



Software, documentation and related materials:

Copyright © 2018 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties. Altium®, Altium Designer®, Altium Vault®, Autotrax®, Camtastic®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, DXP™, Easytrax®, LiveDesign®, NanoBoard®, Octopart®, PCBWORKS®, P-CAD®, Protel®, TASKING® and their respective logos are trademarks or registered trademarks of Altium LLC or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.



# Table of Contents

<b>Module 1: Getting Started</b>	<b>21</b>
<b>1.1 Altium Live</b>	<b>21</b>
1.1.1 Creating an AltiumLive Account .....	21
1.1.2 Signing in to AltiumLive.....	21
<b>1.2 Products Menu</b>	<b>23</b>
1.2.1 Software Downloads .....	23
<b>1.3 Community Menu</b>	<b>24</b>
1.3.1 Forums .....	24
1.3.2 BugCrunch.....	25
1.3.3 Design Ideas .....	26
1.3.4 Altium Blogs .....	26
<b>1.4 Resources Menu</b>	<b>27</b>
1.4.1 Video Library.....	27
1.4.2 Design Content .....	28
1.4.3 Support Center .....	31
1.4.4 Documentation .....	34
<b>Module 2: The Altium Designer Environment</b>	<b>37</b>
<b>2.1 A Unified Design Platform</b>	<b>37</b>
<b>2.2 The Editor View</b>	<b>38</b>
2.2.1 Document Tabs in the Documents Bar .....	39
<b>2.3 Panels</b>	<b>40</b>
2.3.1 Accessing Panels .....	40
2.3.2 Panel Display Modes .....	41
2.3.3 Grouping Panels .....	44
2.3.4 Moving Panels .....	47
<b>2.4 Active Bar</b>	<b>48</b>
2.4.1 Expanding Tool Buttons .....	48
<b>2.5 Search</b>	<b>49</b>
<b>2.6 Menus</b>	<b>49</b>
2.6.1 Right-click Popup Menus.....	49



2.6.2 Shortcut Keys .....	50
<b>2.7 Command Status and Status Bar</b>	<b>51</b>
<b>2.8 Dialogs</b>	<b>52</b>
<b>2.9 Undo/Redo</b>	<b>52</b>
<b>2.10 Multi-Monitor Support</b>	<b>53</b>
<b>2.11 Saving Desktop Layouts</b>	<b>53</b>
<b>2.12 Saving Preferences to a File</b>	<b>53</b>
<b>Module 3: Design Documents</b>	<b>55</b>
<b>3.1 Creating a new Document</b>	<b>55</b>
<b>3.2 Opening and Displaying Documents</b>	<b>56</b>
3.2.1 Document Insight .....	57
3.2.2 Connectivity Insight .....	57
3.2.3 Document's State Display .....	58
3.2.4 Document Window Management.....	59
3.2.5 Document Storage.....	60
<b>3.3 Different Types of Design Documents</b>	<b>61</b>
3.3.1 Project Files .....	61
3.3.2 Project Collection Files .....	61
3.3.3 Common PCB Design Files .....	61
3.3.4 Automation Script Files .....	62
3.3.5 Miscellaneous Files.....	62
3.3.6 Importable Binary Files .....	62
3.3.7 3D Model Files .....	63
<b>3.4 Exercise</b>	<b>64</b>
<b>Module 4: It Starts with a Project</b>	<b>65</b>
<b>4.1 Types of Design Projects</b>	<b>65</b>
4.1.1 PCB Project (*.PrjPcb).....	65
4.1.2 Integrated Library (*.LibPkg) & (*.IntLib) .....	66
4.1.3 Script Project (*.PrjScr) .....	66
<b>4.2 Working with Projects</b>	<b>66</b>
4.2.1 The Projects panel.....	66



4.2.2 Creating a new Project .....	67
4.2.3 Adding Documents to a Project .....	69
4.2.4 Removing a Document from a project .....	70
4.2.5 Specifying the Order of Documents in a Project .....	70
4.2.6 Sharing Documents between Projects .....	71
4.2.7 Saving a Project with a New Name.....	71
4.2.8 Making Changes to the Project .....	71
4.2.9 Renaming Projects - Save Project As .....	72
<b>4.3 Transfer / Copy Projects - the Project Packager</b>	<b>73</b>
<b>4.4 Exercise</b>	<b>74</b>
<b>Module 5: The Schematic Editor</b>	<b>75</b>
<b>5.1 The Schematic Editor User Interface</b>	<b>75</b>
5.1.1 The Design Insight System .....	76
<b>5.2 Navigating the Schematic Editor Workspace</b>	<b>77</b>
5.2.1 Navigating with the Mouse .....	77
5.2.2 Navigating with the Keyboard .....	77
5.2.3 Navigating via the View menu .....	77
5.2.4 Navigating with the Navigator Panel.....	78
5.2.5 Using the Jump menu .....	80
<b>5.3 Schematic Preferences</b>	<b>80</b>
<b>5.4 Document Options</b>	<b>81</b>
5.4.1 General Document Options .....	81
5.4.2 Parameters .....	85
<b>5.5 Exercise</b>	<b>87</b>
<b>Module 6: Placing Schematic Objects</b>	<b>89</b>
<b>6.1 Common Elements of Object Placement</b>	<b>89</b>
6.1.1 Object Editing Options.....	89
6.1.2 Using Over-scores.....	90
6.1.3 Getting Help when Placing Objects .....	90
<b>6.2 Schematic Grids and Cursors</b>	<b>91</b>
6.2.1 Different Types of Grids.....	91
6.2.2 Setting the Schematic Cursor Appearance .....	91



<b>6.3 Graphic Objects</b>	<b>92</b>
6.3.1 Lines <PDL>.....	93
6.3.2 Polygons <PDY>.....	93
6.3.3 Arcs <PDA>.....	94
6.3.4 Full Circle <PDU>.....	94
6.3.5 Text Strings <PT>.....	94
6.3.6 Text Frames <PF>.....	94
6.3.7 Notes.....	95
6.3.8 Rectangles <PDR>.....	96
6.3.9 Round Rectangles <PDO>.....	96
6.3.10 Ellipses <PDE>.....	96
6.3.11 Graphic images <PDG>.....	96
<b>6.4 Electrical objects</b>	<b>97</b>
6.4.1 Power Ports <PO>.....	98
6.4.2 Parts <PP>.....	99
6.4.3 Sheet Symbols <PS>.....	102
6.4.4 Sheet Entries <PE>.....	103
6.4.5 Place Device Sheet Symbols<PI>.....	104
6.4.6 Ports <PR>.....	105
6.4.7 Off Sheet Connectors <PC>.....	106
<b>6.5 Exercise</b>	<b>107</b>
<b>Module 7: Schematic Connections</b>	<b>109</b>
<b>7.1 Creating Connections</b>	<b>109</b>
7.1.1 Wires <PW>.....	109
7.1.2 Buses <PB>.....	110
7.1.3 Bus Entries <PU>.....	110
7.1.4 Signal Harnesses <PHH>.....	111
7.1.5 Harness Connector <PHC>.....	112
7.1.6 Harness Entry <PHE>.....	113
7.1.7 Net Labels <PN>.....	115
7.1.8 Automatic Junctions.....	116
<b>7.2 Editing Connections</b>	<b>116</b>
7.2.1 Common Move operations.....	116
7.2.2 Common Drag Operations.....	117



7.2.3 Editing Placed Wires .....	117
<b>7.3 Exercise</b>	<b>120</b>
<hr/>	
<b>Module 8: Schematic Annotation</b>	<b>121</b>
<hr/>	
<b>8.1 Annotation Overview</b>	<b>121</b>
<hr/>	
<b>8.2 Schematic Level Annotation</b>	<b>122</b>
8.2.1 Order of Processing .....	123
8.2.2 Matching Options .....	124
8.2.3 Schematic Sheets to Annotate .....	127
8.2.4 Proposed Change List.....	127
<hr/>	
<b>8.3 Additional Schematic Annotation Commands</b>	<b>129</b>
8.3.1 Reset Schematic Designators .....	129
8.3.2 Reset Duplicate Schematic Designators .....	129
8.3.3 Annotate Schematics Quietly .....	129
8.3.4 Force Annotate All Schematics .....	129
8.3.5 Back Annotate Schematics.....	130
8.3.6 Number Schematic Sheets .....	130
8.3.7 Schematic Sheet Numbering and Device Sheets .....	132
<hr/>	
<b>8.4 Exercise</b>	<b>133</b>
<hr/>	
<b>Module 9: Global Editing</b>	<b>135</b>
<hr/>	
<b>9.1 Altium Designer's Data Editing Paradigm</b>	<b>135</b>
9.1.1 Selecting Objects .....	135
9.1.2 Inspecting Objects.....	136
9.1.3 Editing Objects .....	136
9.1.4 Masking .....	137
<hr/>	
<b>9.2 Selection Methods</b>	<b>138</b>
9.2.1 Selection Filter in the Properties Panel .....	138
9.2.2 Select Menu.....	139
9.2.3 Find Similar Objects (FSO) .....	139
<hr/>	
<b>9.3 Properties Panel</b>	<b>141</b>
9.3.1 Opening the Properties Panel .....	141
9.3.2 Properties Panel Access.....	142
9.3.3 Panel Display Scope.....	142



<b>9.4 List Panel</b>	<b>143</b>
9.4.1 List Panel - Inspect and Edit.....	143
9.4.2 SCH List Panel Access.....	143
9.4.3 Defining Panel Display Scope .....	143
9.4.4 Making Selections from the List Panel .....	144
9.4.5 Inspecting and Editing Object Attributes .....	144
9.4.6 Smart Grid Tools .....	145
<b>Module 10: Compile and Verify the PCB Project</b>	<b>147</b>
<b>10.1 Compiler settings</b>	<b>147</b>
10.1.1 Error Reporting tab.....	147
10.1.2 Connection Matrix tab .....	148
<b>10.2 Interpreting messages and locating errors</b>	<b>150</b>
10.2.1 Getting More Error/Warning Information.....	150
10.2.2 Clearing Messages .....	151
10.2.3 Allocating “No ERC” Directive .....	151
10.2.4 Project Compiler Error Reference.....	152
<b>10.3 Exercise</b>	<b>154</b>
<b>Module 11: Multi-Sheet Design &amp; Design Hierarchy</b>	<b>155</b>
<b>11.1 Defining Sheet Structure</b>	<b>155</b>
<b>11.2 Structuring a Multi-Sheet Design</b>	<b>156</b>
11.2.1 Building Hierarchical Structure.....	157
<b>11.3 Multi-Sheet Design Connectivity</b>	<b>158</b>
11.3.1 Net Identifiers.....	158
11.3.2 Net Identifier Scope .....	159
<b>11.4 Assigning Sheet Numbers</b>	<b>161</b>
<b>11.5 Checking Sheet Synchronization</b>	<b>163</b>
<b>11.6 Exercise</b>	<b>164</b>
<b>Module 12: Design Transfer to PCB</b>	<b>165</b>
<b>12.1 Synchronization - Overview</b>	<b>165</b>
<b>12.2 Detecting the Differences</b>	<b>166</b>
12.2.1 Setting up the Comparator .....	166





12.2.2 Setting up ECO Options .....	167
<b>12.3 Transferring the Design</b> .....	<b>168</b>
12.3.1 Pre-transfer Checks.....	168
12.3.2 Direct Synchronization Commands .....	169
12.3.3 Generating the ECO.....	170
12.3.4 Proceeding with the Transfer .....	171
<b>12.4 Manual Compare Methods</b> .....	<b>173</b>
12.4.1 Running the Comparator .....	173
12.4.2 Showing Physical Differences – Going Graphical.....	176
<b>12.5 Resolving Synchronization Errors</b> .....	<b>180</b>
12.5.1 Component Linking through Unique IDs.....	180
<b>12.6 Exercise</b> .....	<b>182</b>
<b>Module 13: The PCB Editor</b> .....	<b>183</b>
<b>13.1 The PCB Editor User Interface</b> .....	<b>183</b>
13.1.1 Three PCB Editor Modes.....	184
<b>13.2 PCB Preferences</b> .....	<b>185</b>
13.2.1 PCB General Preferences .....	185
13.2.2 PCB Board Insight Display Preferences.....	186
13.2.3 PCB Board Insight Mode Preferences .....	187
<b>13.3 Navigating the PCB Editor Workspace</b> .....	<b>188</b>
13.3.1 Navigating with the Mouse .....	188
13.3.2 Navigating with a Keyboard.....	189
13.3.3 Navigating via the View or Zoom menu .....	189
13.3.4 Using the Jump menu .....	190
13.3.5 3D Navigation.....	191
13.3.6 Navigating with a 3D Mouse .....	192
<b>13.4 View Configurations</b> .....	<b>193</b>
13.4.1 Layers and Colors Tab.....	193
13.4.2 View Options Tab.....	197
<b>13.5 Exercise</b> .....	<b>203</b>
<b>Module 14: Working with the PCB Panel</b> .....	<b>205</b>
<b>14.1 Navigate using the PCB Panel</b> .....	<b>205</b>



14.1.1 Browse mode selection list .....	205
14.1.2 Panel Mini-Viewer .....	206
<b>14.2 Browsing nets and net classes</b>	<b>207</b>
<b>14.3 Connection Lines and Trace Colors</b>	<b>209</b>
14.3.1 Changing the Connection Line Color .....	209
14.3.2 Changing Trace Colors .....	212
<b>14.4 Browsing components and component classes</b>	<b>215</b>
<b>14.5 Browsing xSignals</b>	<b>217</b>
<b>14.6 From-To Editor</b>	<b>218</b>
<b>14.7 Split Plane Editor</b>	<b>220</b>
<b>14.8 Differential Pair Editor</b>	<b>221</b>
<b>14.9 Browsing Polygons</b>	<b>222</b>
<b>14.10 Hole Size Editor</b>	<b>223</b>
<b>14.11 Browsing 3D Models</b>	<b>224</b>
<b>14.12 Browsing Structure Classes</b>	<b>226</b>
<b>14.13 Browsing Layer Stack Regions</b>	<b>228</b>
<b>14.14 Unions</b>	<b>230</b>
<b>14.15 Pad &amp; Via Templates</b>	<b>232</b>
<b>Module 15: Introduction to PCB Global Editing</b>	<b>233</b>
<b>15.1 Altium Designer's Data Editing Paradigm</b>	<b>233</b>
15.1.1 Selecting Objects .....	233
15.1.2 Inspecting Objects.....	234
15.1.3 Editing Objects .....	234
15.1.4 Masking .....	235
<b>15.2 Selection Methods</b>	<b>236</b>
15.2.1 Selection Filter in the Properties Panel .....	236
15.2.2 Filter Panel .....	237
<b>15.3 Editing Objects</b>	<b>239</b>
<b>15.4 Exercise</b>	<b>240</b>



<b>Module 16: PCB Layers</b>	<b>241</b>
<b>16.1 Layer Stack Manager</b>	<b>241</b>
16.1.1 Defining the Layer Stack.....	241
16.1.2 Configuring the Layer Properties.....	245
16.1.3 Documenting the Layer Stack .....	247
16.1.4 Configuring the Drill Pairs .....	248
16.1.5 Including a Drill Table .....	249
<b>16.2 Layer Definitions</b>	<b>250</b>
16.2.1 Signal Layers .....	251
16.2.2 Internal Plane Layers.....	251
16.2.3 Silkscreen Layers .....	251
16.2.4 Mechanical Layers.....	251
16.2.5 Solder Mask Layers .....	251
16.2.6 Paste Mask Layers.....	251
16.2.7 Drill Drawing Layer .....	251
16.2.8 Drill Guide Layer .....	252
16.2.9 Keep-Out Layer .....	252
16.2.10 Multi-Layer .....	252
<b>16.3 PCB Layer Display in the Workspace</b>	<b>252</b>
16.3.1 Changing the Current Layer .....	252
16.3.2 Layer Sets .....	253
<b>16.4 Exercise</b>	<b>254</b>
<b>Module 17: Defining the Board Shape</b>	<b>255</b>
<b>17.1 The Board Shape</b>	<b>255</b>
<b>17.2 Defining the Board Shape</b>	<b>256</b>
17.2.1 Defining the Board Shape from Existing Primitives .....	256
17.2.2 Defining the Board Shape from a DXF File.....	257
17.2.3 Defining the Board Shape Using a 3D Body .....	257
<b>17.3 Creating a Keepout from the Board Shape</b>	<b>259</b>
<b>17.4 Adding Cutouts to the Board Shape</b>	<b>259</b>
<b>17.5 Board Planning Mode</b>	<b>261</b>
17.5.1 Editing the Board Shape .....	261



17.5.2 Modifying the Board Shape .....	263
17.5.3 Redefining the Board Shape.....	263
17.5.4 Tips when defining/redefining the board shape .....	264
<b>17.6 Exercise</b> .....	<b>265</b>
<b>Module 18: PCB Grids, Guides &amp; Snap Options</b> .....	<b>267</b>
<b>18.1 Accessing the Unified Cursor-Snap System</b> .....	<b>267</b>
<b>18.2 User-Definable Grids</b> .....	<b>269</b>
18.2.1 Accessing the Grid Manager .....	269
18.2.2 Creating and Defining a Cartesian Grid.....	269
18.2.3 Creating and Defining a Polar Grid .....	272
18.2.4 Default Snap Grid .....	274
18.2.5 Defining Grid Purpose.....	274
18.2.6 Defining Grid Display .....	275
18.2.7 Nesting and Grid Priority .....	276
18.2.8 Disabling a Grid .....	277
18.2.9 Deleting a Grid .....	278
18.2.10 Changing Grid Settings in the PCB.....	278
<b>18.3 Snap Guides</b> .....	<b>279</b>
18.3.1 Defining a Snap Guide .....	280
18.3.2 Disabling a Snap Guide.....	280
18.3.3 Deleting a Snap Guide.....	280
<b>18.4 Object Snap-Points</b> .....	<b>280</b>
18.4.1 Snap to Object Hotspots.....	281
18.4.2 Snap to Object Axis.....	281
<b>18.5 Other Board Options</b> .....	<b>282</b>
18.5.1 Measurement Unit.....	282
18.5.2 Polygon Naming Scheme.....	283
18.5.3 Designator Display .....	283
18.5.4 Route Tool Path .....	283
<b>18.6 Exercise</b> .....	<b>283</b>
<b>Module 19: Introduction to PCB Design Rules</b> .....	<b>285</b>
<b>19.1 PCB Rules System - Fundamentals</b> .....	<b>285</b>



<b>19.2 Defining and Managing Design Rules</b>	<b>286</b>
19.2.1 Creating a New Rule .....	286
19.2.2 Scoping a Rule.....	289
19.2.3 Prioritizing the Rules.....	293
19.2.4 Modifying Existing Rules .....	294
19.2.5 Deleting Rules.....	295
19.2.6 Creating Rules from the Filter Panel.....	298
<b>19.3 Checking Rule Application</b>	<b>299</b>
19.3.1 From an Object's Perspective .....	299
19.3.2 From a Rule's Perspective .....	300
<b>19.4 Exporting and Importing Rules</b>	<b>302</b>
<b>19.5 Design Rule Reports</b>	<b>302</b>
<b>19.6 Exercise</b>	<b>303</b>
<b>Module 20: Introduction to Classes</b>	<b>305</b>
<b>20.1 Object Classes</b>	<b>305</b>
20.1.1 Object Class Explorer .....	306
20.1.2 Creating a New Class in the Class Explorer.....	308
<b>20.2 Creating a Net Class from Schematic</b>	<b>309</b>
20.2.1 Using Directives.....	309
20.2.2 Automatic Component Class Generation.....	311
20.2.3 User-Defined Component Classes .....	311
20.2.4 Component Class Generator .....	312
<b>Module 21: Positioning Components on the PCB</b>	<b>315</b>
<b>21.1 Placing Components into the Workspace</b>	<b>315</b>
<b>21.2 Initial Component Placement</b>	<b>315</b>
21.2.1 Cluster Components by Schematic Sheet.....	315
<b>21.3 Rooms</b>	<b>317</b>
21.3.1 Defining Rooms .....	317
<b>21.4 Finding a Component for Placement</b>	<b>318</b>
21.4.1 Working from the Schematic to the PCB.....	318
21.4.2 Working in the PCB Panel.....	319
<b>21.5 Moving a Component</b>	<b>320</b>



21.5.1 Moving Basics .....	320
21.5.2 Moving in the PCB Editor.....	321
21.5.3 Drag Tracks while Moving a Component.....	322
21.5.4 Component Unions.....	323
21.5.5 Component Grids .....	323
<b><u>21.6 Interactive Placement Commands</u></b>	<b>325</b>
21.6.1 Align Commands .....	325
21.6.2 Component Placement Commands .....	325
21.6.3 Component Alignment .....	326
<b><u>21.7 Exercise</u></b>	<b>329</b>
<b><u>Module 22: PCB Pad-Via Library</u></b>	<b>331</b>
<b><u>22.1 Pad and Via Templates</u></b>	<b>331</b>
<b><u>22.2 PCB Pad Via Templates Panel</u></b>	<b>333</b>
22.2.1 Local Pad & Via Library.....	334
22.2.2 Pad & Via Template Libraries .....	334
22.2.3 Pad Via Library Template to Local .....	335
22.2.4 Replace a Local Template .....	335
22.2.5 Update from Pad Via Library .....	335
22.2.6 Loading a Pad Via Library.....	337
<b><u>22.3 Editing a placed Library Template Pad/Via</u></b>	<b>337</b>
<b><u>22.4 PCB Panel - Pad &amp; Via Templates</u></b>	<b>339</b>
22.4.1 Libraries section .....	340
22.4.2 Templates section .....	340
22.4.3 Pad/Via section .....	341
<b><u>Module 23: Interactive Routing</u></b>	<b>343</b>
<b><u>23.1 Interactively Routing a Net</u></b>	<b>343</b>
<b><u>23.2 Understanding Connectivity</u></b>	<b>344</b>
<b><u>23.3 Setting up Routing Design Rules</u></b>	<b>346</b>
23.3.1 Routing Width Rule .....	347
23.3.2 Routing Via Style Rule .....	348
<b><u>23.4 Interactive Routing Options and Features</u></b>	<b>349</b>
23.4.1 The Routing Conflict Resolution Mode.....	351



23.4.2 Controlling the Routing Width and Via Size .....	352
23.4.3 Changing Layers while routing .....	354
23.4.4 Routing Gloss Effort.....	354
23.4.5 Restrict to 90/45 .....	355
23.4.6 Follow Mouse Trail .....	355
23.4.7 Automatically Terminate Routing.....	355
23.4.8 Loop Removal .....	355
23.4.9 Allow Via Pushing.....	356
23.4.10 Display Clearance Boundaries .....	356
23.4.11 Look Ahead Behavior .....	356
<b><u>23.5 Important Shortcuts available during Interactive Routing</u></b>	<b>357</b>
<b><u>23.6 ActiveRoute</u></b>	<b>359</b>
<b><u>23.7 Glossing</u></b>	<b>360</b>
<b><u>23.8 Dragging Routes</u></b>	<b>360</b>
<b><u>23.9 Un-Routing</u></b>	<b>361</b>
<b><u>23.10 Exercise</u></b>	<b>363</b>
<b><u>Module 24: Polygons</u></b>	<b>365</b>
<b><u>24.1 Overview</u></b>	<b>365</b>
<b><u>24.2 Placing a Polygon Pour</u></b>	<b>365</b>
24.2.1 Defining Polygon Pour Properties .....	366
24.2.2 Editing Polygon Pour Properties.....	369
24.2.3 Defining the Shape of the Polygon by drawing it .....	369
24.2.4 Shelved Polygons.....	370
24.2.5 Repour Polygons .....	370
<b><u>24.3 Selecting a Polygon</u></b>	<b>370</b>
<b><u>24.4 Deleting a Polygon Pour</u></b>	<b>370</b>
<b><u>24.5 Editing a Polygonal Object</u></b>	<b>370</b>
24.5.1 Active Mouse Hover .....	371
24.5.2 Polygon Editing Modes .....	371
24.5.3 Editing a Corner Vertex.....	372
24.5.4 Editing a Center Vertex .....	373
24.5.5 Adding a Vertex .....	375



24.5.6 Deleting a Vertex.....	375
24.5.7 Combining, Subtracting and Modifying.....	375
24.5.8 Alignment Guides .....	375
<b>24.6 Detecting Unpoured Polygons</b>	<b>376</b>
<b>24.7 The Polygon Pour Manager</b>	<b>378</b>
<b>24.8 Exercise</b>	<b>381</b>
<b>Module 25: DRCs on PCBs</b>	<b>383</b>
<b>25.1 Configuring the DRC</b>	<b>383</b>
<b>25.2 Using Online DRC</b>	<b>384</b>
<b>25.3 Using Batch DRC</b>	<b>385</b>
25.3.1 DRC Reports .....	386
<b>25.4 Violation Display Options</b>	<b>387</b>
25.4.1 Custom Violation Graphics.....	387
25.4.2 Violation Overlay .....	387
25.4.3 Setting Violation Display Preferences .....	388
25.4.4 Defining Violation Colors.....	389
<b>25.5 Resolving Design Violations</b>	<b>389</b>
25.5.1 Interrogating Violations .....	390
<b>25.6 DRC Validation in an Output Job</b>	<b>394</b>
<b>25.7 Exercise</b>	<b>394</b>
<b>Module 26: Generating the BOM</b>	<b>395</b>
<b>26.1 Creating a BOM</b>	<b>395</b>
26.1.1 Layout and Grouping of Data .....	396
26.1.2 Adding Database Information Directly to the BOM.....	399
26.1.3 Adding PCB Information to the BOM .....	400
26.1.4 Including Supplier Data in the BOM.....	401
26.1.5 Catering to Variants.....	404
<b>26.2 Exporting the Report</b>	<b>406</b>
26.2.1 Using Excel Templates.....	406
<b>Module 27: Draftsman</b>	<b>411</b>
<b>27.1 Key Features</b>	<b>411</b>





<b>27.2 Working with Documents</b>	<b>412</b>
27.2.1 Create a New Document.....	412
27.2.2 Document Options.....	413
<b>27.3 Working with Drawing Views</b>	<b>417</b>
27.3.1 Draftsman Properties Panel.....	417
27.3.2 Assembly View .....	418
27.3.3 Fabrication View .....	422
27.3.4 Drill Drawing View .....	424
27.3.5 Detail View.....	426
27.3.6 Section View .....	427
27.3.7 Isometric View .....	429
27.3.8 Layer Stack Legend.....	430
27.3.9 Back Drilling.....	433
<b>27.4 Geometric Dimensioning and Tolerances</b>	<b>434</b>
27.4.1 Datum Feature .....	434
27.4.2 Feature Control Frame .....	435
<b>27.5 Annotation and Drawing Tools</b>	<b>437</b>
27.5.1 Dimensions .....	437
27.5.2 Callouts .....	441
27.5.3 Surface Finish Indicators.....	442
27.5.4 Notes.....	444
27.5.5 BOM, Table and Drill Tables .....	445
27.5.6 DXF Import .....	452
27.5.7 Documentation Outputs.....	452
27.5.8 Variant Parameters and OutJobs .....	453
<b>27.6 Exercise</b>	<b>454</b>
<b>Module 28: Outputs and Fabrication Files</b>	<b>455</b>
<b>28.1 Adding &amp; Defining an OutJob - Overview</b>	<b>455</b>
<b>28.2 Adding Outputs to an OutJob</b>	<b>457</b>
28.2.1 Configuring Outputs.....	458
<b>28.3 Defining Output Format</b>	<b>459</b>
28.3.1 Output Containers .....	459
28.3.2 Hard Copy - Print Jobs.....	463



28.3.3 Linking Outputs to Output Containers and Print Jobs.....	464
<b>28.4 Variant Choice</b>	<b>465</b>
<b>28.5 Output Generation</b>	<b>468</b>
28.5.1 From Output Containers .....	468
28.5.2 From Print Jobs .....	470
<b>28.6 Exercise</b>	<b>472</b>
<b>Module 29: Schematic Symbol Creation</b>	<b>473</b>
<b>29.1 The Dual Nature of the Schematic Symbol</b>	<b>473</b>
<b>29.2 Creating a new Symbol Library and an Empty Component Symbol</b>	<b>474</b>
<b>29.3 Creating the Component Symbol</b>	<b>474</b>
29.3.1 The Component Symbol Design Space .....	474
29.3.2 The Component Symbol Body .....	475
29.3.3 The Component Symbol Pins .....	476
29.3.4 The Component Symbol Properties .....	478
29.3.5 Adding Parameters .....	480
29.3.6 Creating a Multi-Part Component.....	483
<b>29.4 Creating an Alternate View Mode for a Part</b>	<b>483</b>
<b>29.5 Using the SCH Library Panel</b>	<b>484</b>
29.5.1 Components Section.....	484
29.5.2 Supply Links and Supplier Information .....	485
<b>29.6 Library and Component Management Techniques</b>	<b>485</b>
29.6.1 Creating a Library of Components from an Existing Project.....	485
29.6.2 Copying Existing Components to a New Symbol Library .....	485
29.6.3 Copying Components from other Libraries .....	485
<b>29.7 Symbol Wizard</b>	<b>486</b>
29.7.1 Creating a Symbol .....	486
<b>29.8 Exercise</b>	<b>488</b>
<b>Module 30: PCB Component Creation</b>	<b>489</b>
<b>30.1 Manually Creating a Footprint</b>	<b>489</b>
<b>30.2 Creating a Component using IPC® Compliant Footprint Wizard</b>	<b>491</b>
30.2.1 Creating a Component using IPC Footprints Batch Generator .....	492



30.2.2 Creating a Component using the Component Wizard.....	493
30.2.3 Adding a 3D Model to the Component.....	493
<b>30.3 Glossary</b>	<b>494</b>
<b>30.4 Mechanical Layers</b>	<b>495</b>
<b>30.5 Exercise</b>	<b>496</b>
<b>Module 31: Component Creation</b>	<b>497</b>
<b>31.1 Adding Links to Other Domain Models</b>	<b>497</b>
31.1.1 Adding Models .....	498
31.1.2 Defining Model Links.....	499
31.1.3 Configuring Model Link Search Resolution .....	503
<b>31.2 Library Reports</b>	<b>503</b>
<b>31.3 Exercise</b>	<b>505</b>
<b>Module 32: Integrated Libraries</b>	<b>507</b>
<b>32.1 Creating an IntLib using a Library Package</b>	<b>507</b>
32.1.1 Creating the Source Library Package .....	508
32.1.2 Creating and Adding a Source Schematic Library .....	508
32.1.3 Creating and Adding Domain Model Files .....	509
32.1.4 Compiling the Library Package .....	511
<b>32.2 Creating an IntLib from Project Documents</b>	<b>513</b>
<b>32.3 Placing Components from an Integrated Library</b>	<b>513</b>
<b>32.4 Modifying an Integrated Library</b>	<b>514</b>
32.4.1 Decompiling an Integrated Library.....	514
<b>32.5 Exercise</b>	<b>516</b>



