

# Altium Designer

**Altium**

## Altium Designer 18

### Advanced 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: Schematic Design Directives</b>	<b>17</b>
<b>1.1 Compilation Related Directives - No ERC</b>	<b>17</b>
1.1.1 Editing No ERC directives after placement .....	19
1.1.2 Right-click to Place a Specific No ERC Directive.....	20
<b>1.2 Controlling the Printing of No ERC Directives</b>	<b>21</b>
1.2.1 NoERC Behavior in Older Versions of Altium Designer.....	21
<b>1.3 Compilation Related Directives - Compile Mask</b>	<b>21</b>
1.3.1 Disabling Compile Masks .....	23
1.3.2 Creative Compile Mask Applications .....	23
<b>1.4 PCB Related Directives</b>	<b>24</b>
1.4.1 Parameter Set Directives .....	24
1.4.2 PCB Layout directive.....	24
1.4.3 Net Class Directive .....	27
1.4.4 Differential Pair Directive .....	28
<b>1.5 Using Blankets with Directives</b>	<b>29</b>
1.5.1 Highlighting Nets Covered by a Blanket Directive.....	30
<b>1.6 Indirect (Parameter-Based) Design Directives</b>	<b>30</b>
1.6.1 Specifying component classes .....	31
<b>Module 2: Classes</b>	<b>33</b>
<b>2.1 Class Concept</b>	<b>33</b>
<b>2.2 Viewing Classes</b>	<b>33</b>
2.2.1 Object Class Explorer .....	33
2.2.2 Default Classes .....	34
2.2.3 Viewing Classes in PCB Panel .....	35
<b>2.3 Using Classes in Logical Queries</b>	<b>35</b>
<b>2.4 Creating Classes in Schematic Editor</b>	<b>36</b>
2.4.1.1 Net Class Directive .....	36
2.4.1.2 Net Class Blanket Directive .....	37
<b>2.4.2 Enabling Automatic Classes</b> .....	<b>38</b>
2.4.2.1 Net Classes for Components .....	38
2.4.2.2 Net Classes for Buses .....	39



2.4.2.3 Net Classes for Schematic Sheets.....	39
<b>2.4.3 Generating Component Classes .....</b>	<b>40</b>
2.4.3.1 Default Component Classes per Page.....	41
2.4.3.2 Adding Classes in Parameter Manager.....	42
<b><u>2.5 Structure Classes .....</u></b>	<b><u>43</u></b>
<b>2.5.1 Creating Structure Classes.....</b>	<b>44</b>
2.5.1.1 Manually Creating Structure Classes.....	44
2.5.1.2 Automatically Creating Structure Classes.....	44
<b>2.5.2 Browsing the Class Structure Hierarchy .....</b>	<b>45</b>
<b>2.5.3 Using Structure Classes in Logical Queries .....</b>	<b>46</b>
<b><u>2.6 Creating Classes in the PCB .....</u></b>	<b><u>46</u></b>
<b>2.6.1 Creating New Class in Object Class Explorer .....</b>	<b>46</b>
<b>2.6.2 Adding Members to a New Class .....</b>	<b>47</b>
2.6.2.1 Adding Members from Non-Members Column.....	47
2.6.2.2 Adding / Removing Members from PCB .....	47
2.6.2.3 Net Classes from Selected Objects.....	48
2.6.2.4 Adding Members from Component Class Generator .....	48
<b><u>Module 3: PCB Queries &amp; PCB Filter Panel .....</u></b>	<b><u>49</u></b>
<b><u>3.1 Panel Access .....</u></b>	<b><u>49</u></b>
<b><u>3.2 Panel Makeup .....</u></b>	<b><u>49</u></b>
<b><u>3.3 Defining the Filter Scope .....</u></b>	<b><u>50</u></b>
<b><u>3.4 Defining the Filter Query .....</u></b>	<b><u>50</u></b>
3.4.1 The Query Helper.....	51
3.4.2 The Query Builder.....	51
3.4.3 Query Expression Operator Precedence.....	52
<b><u>3.5 Executing the Filter .....</u></b>	<b><u>53</u></b>
<b><u>3.6 Reusing Past Filter Expressions .....</u></b>	<b><u>54</u></b>
<b><u>3.7 Creating Design Rules .....</u></b>	<b><u>55</u></b>
<b><u>3.8 Clearing the Current Filter .....</u></b>	<b><u>55</u></b>
<b><u>3.9 Additional non-panel Filtering .....</u></b>	<b><u>55</u></b>
<b><u>Module 4: Clearance Checking in 3D .....</u></b>	<b><u>57</u></b>
<b><u>4.1 Rules for Checking Component Clearances .....</u></b>	<b><u>57</u></b>
<b>4.1.1 The Component Clearance Rule.....</b>	<b>57</b>
4.1.1.1 Constraints .....	58



4.1.2 Applying Clearance Rules .....	59
<b>4.2 Actively Checking Clearances - DRC</b>	<b>60</b>
<b>4.3 Resolving Clearance Violations</b>	<b>63</b>
<b>4.4 Need More Checking?</b>	<b>63</b>
<b>Module 5: Differential Pair Routing</b>	<b>65</b>
<b>5.1 Defining the Differential Pairs on the Schematic</b>	<b>65</b>
<b>5.2 Viewing and Managing Differential Pairs on the PCB</b>	<b>66</b>
5.2.1 Defining Differential Pairs on the PCB.....	67
5.2.2 Create Differential Pairs from Nets .....	67
<b>5.3 Applicable Design Rules</b>	<b>68</b>
5.3.1 Setting the Scope of the Design Rules .....	68
5.3.2 Using the Differential Pair Wizard to Define Rules .....	69
<b>5.4 Routing a Differential Pair</b>	<b>69</b>
<b>Module 6: Re-Routing</b>	<b>71</b>
<b>6.1 Rerouting an Existing Route - Loop Removal</b>	<b>71</b>
6.1.1 Protecting an Existing Route from Loop Removal .....	71
6.1.2 Protecting Existing Routes from Pushing and Shoving .....	72
<b>6.2 Track Dragging with Angle Preservation</b>	<b>73</b>
<b>6.3 Track Slicing</b>	<b>74</b>
<b>6.4 Incremental Unroute</b>	<b>75</b>
6.4.1 Ctrl + Delete.....	75
6.4.2 Backspace.....	76
6.4.3 Tab .....	76
<b>Module 7: Length Tuning Routing</b>	<b>78</b>
<b>7.1 Interactive Length Tuning</b>	<b>78</b>
<b>7.2 Configuring the Design Rules</b>	<b>80</b>
7.2.1 Matched Net Lengths design rule .....	81
7.2.2 Length design rule.....	81
7.2.3 Converting Time to Length .....	81
<b>7.3 Using the Net Length Indicator</b>	<b>81</b>



7.3.1.1 Net Length Indicator example .....	82
<b>7.3.2 Re-Routing Accordion Routes.....</b>	<b>82</b>
<b>7.4 Further information</b>	<b>83</b>
7.4.1 See Also .....	83
<b>Module 8: ActiveRoute</b>	<b>84</b>
<b>8.1 ActiveRoute Process</b>	<b>84</b>
<b>8.1.1 Setup of Active Route .....</b>	<b>84</b>
8.1.1.1 Tune Selected option.....	85
8.1.1.2 Pin Swap Routing Section.....	86
8.1.1.3 Gloss Results .....	86
8.1.1.4 Layer Section.....	86
8.1.1.5 Additional ActiveRoute Setup Helps and Hints .....	86
<b>8.1.2 Selection.....</b>	<b>87</b>
<b>8.1.3 Placement of Route Paths/Guides .....</b>	<b>88</b>
<b>8.1.4 Running ActiveRoute .....</b>	<b>89</b>
<b>8.1.5 ActiveRoute Limitations.....</b>	<b>90</b>
<b>Module 9: High Speed Design with xSignals</b>	<b>92</b>
<b>9.1 Creating New xSignals</b>	<b>93</b>
9.1.1 xSignal Creation Commands .....	93
9.1.2 Creating xSignals from Selected.....	94
9.1.3 Creating xSignals Between Components.....	95
9.1.4 Create xSignals From Connected Nets Dialog.....	96
<b>9.2 The Role of the Net Topology</b>	<b>96</b>
9.2.1 Defining the Branch Point in a Balanced T Pattern .....	97
<b>9.3 Working in the xSignal Panel</b>	<b>98</b>
9.3.1 Displaying and Hiding the xSignal Lines.....	98
9.3.2 Deleting an xSignal.....	98
<b>9.4 xSignal Query Keywords</b>	<b>99</b>
9.4.1 Membership Check Type Keywords .....	99
9.4.2 Attribute Check Type Keywords.....	99
<b>9.5 Design Rule Support for xSignals</b>	<b>99</b>
9.5.1 Matched Length Rule .....	99
9.5.2 Length Rule.....	100
<b>9.6 Accurate Length Calculations</b>	<b>101</b>



<b>9.7 Net-related Terminology</b>	<b>102</b>
<b>Module 10: Back Drill</b>	<b>104</b>
<b>10.1 Targeting the Holes to be Back Drilled</b>	<b>105</b>
<b>10.2 Defining the Drill Pair layers for Back Drilling</b>	<b>105</b>
10.2.1.1 Accessing the Drill-Pair Manager Window .....	105
<b>10.3 PCB Hole Size Editor</b>	<b>106</b>
<b>10.4 Viewing the resulting Back Drilled Vias using 3D</b>	<b>108</b>
<b>Module 11: Via Stitching</b>	<b>110</b>
<b>11.1 Adding Stitching Vias to a Net</b>	<b>110</b>
11.1.1 Stitching Parameters .....	111
11.1.2 Same Net Clearances .....	111
11.1.3 Via Style .....	113
11.1.4 Via Template .....	113
11.1.5 Properties .....	114
11.1.6 Solder Mask Expansions .....	114
<b>11.2 Stitching Algorithm</b>	<b>114</b>
<b>11.3 Removing Stitching Vias</b>	<b>114</b>
<b>Module 12: Teardrops</b>	<b>116</b>
<b>12.1 Adding and Removing Teardrops</b>	<b>116</b>
<b>Module 13: PCB Layer Stack Management</b>	<b>120</b>
<b>13.1 A Single Design with Multiple Layer Stacks</b>	<b>121</b>
<b>13.2 Defining the Layer Stack in the Layer Stack Manager</b>	<b>122</b>
13.2.1.1 Adding and Removing Layers from a Stack .....	123
13.2.1.2 Adding, Removing and Configuring the Layer Stacks .....	124
<b>13.2.2 Configuring the Layer Properties</b> .....	<b>125</b>
13.2.2.1 Layer Types, Properties, and Function .....	125
<b>13.3 Defining the Overall Board Shape</b>	<b>126</b>
<b>13.3.1 Defining a Board Region and Assigning a Sub-Stack</b> .....	<b>127</b>
<b>13.4 Documenting the Layer Stack</b>	<b>127</b>
13.4.1.1 Capturing the Layer Stack for Documentation .....	127
13.4.1.2 Creating a Layer Stack Legend .....	127
<b>13.5 Configuring the Drill Pairs</b>	<b>128</b>



13.5.1 Configuring Drill Pairs for the selected stack .....	129
<b>Module 14: Flex and Rigid-Flex Printed Circuit Design</b>	<b>131</b>
<b>14.1 What is Rigid-Flex?</b>	<b>131</b>
14.1.1 Rigid-Flex Design .....	131
14.1.2 Materials Used in Flexible Circuit Manufacture .....	132
<b>14.2 Flex and Rigid-Flex Layer Stackup Types</b>	<b>132</b>
14.2.1 Type 1 - Single Layer .....	133
14.2.2 Type 2 - Double Layer .....	133
14.2.3 Type 3 - Multilayer .....	133
14.2.4 Type 4 - Multilayer Rigid-Flex .....	134
<b>14.3 How Rigid-flex is supported in Altium Designer</b>	<b>134</b>
14.3.1 Multiple Layer Stacks .....	135
14.3.2 The Board Shape .....	136
14.3.2.1 Assigning a Layer Stack to a Region of the Board.....	136
14.3.2.2 Placing and Managing Flex Bend Lines.....	137
14.3.2.3 Folding the Board .....	137
<b>Module 15: Snippets</b>	<b>139</b>
<b>15.1 General Description</b>	<b>139</b>
<b>15.2 Panel Access</b>	<b>140</b>
<b>15.3 Snippet Organization</b>	<b>140</b>
15.3.1 Adding and removing Snippets Folders.....	140
15.3.2 Adding and Removing Sub-Folders.....	141
<b>15.4 Creating Snippets</b>	<b>142</b>
<b>15.5 Placing Snippets</b>	<b>145</b>
15.5.1 Right-Click Menus .....	145
<b>Module 16: Multi-Channel Design</b>	<b>147</b>
<b>16.1 Multi-Channel Connectivity</b>	<b>147</b>
<b>16.2 Channel Designations</b>	<b>149</b>
<b>16.3 Displaying Designators on the PCB</b>	<b>151</b>
<b>16.4 Parametric Multi-Channel Design</b>	<b>151</b>
<b>16.5 Room for PCB Designers</b>	<b>152</b>





<b>Module 17: Board Level Annotation</b>	<b>153</b>
<b>17.1 Navigating Board Level Annotation</b>	<b>153</b>
17.1.1 Editing Physical Instances of a Component (Designator) .....	154
<b>17.2 The Board Level Annotate Dialog</b>	<b>155</b>
<b>17.2.1 Filter Options</b> .....	<b>155</b>
17.2.1.1 Schematic Sheet .....	155
17.2.1.2 Channel Name.....	155
17.2.1.3 Enabled .....	155
17.2.1.4 Annotation Scope .....	155
17.2.1.5 All On, All Off Buttons.....	156
<b>17.2.2 Proposed Change List</b> .....	<b>156</b>
17.2.2.1 Schematic Source Component.....	156
17.2.2.2 Calculated Design Data.....	156
17.2.2.3 Naming Scheme .....	157
17.2.2.4 PCB Component Instance .....	157
<b>17.2.3 Annotate Options</b> .....	<b>157</b>
<b>17.2.4 Annotate</b> .....	<b>157</b>
<b>17.2.5 Reset All</b> .....	<b>157</b>
<b>17.2.6 Back Annotation</b> .....	<b>157</b>
<b>17.3 Annotate Options</b>	<b>158</b>
<b>17.3.1 Naming Scheme</b> .....	<b>158</b>
17.3.1.1 Custom Naming Schemes.....	159
<b>17.4 Default Names for Compiled Components</b>	<b>160</b>
<b>17.4.1 Using Rooms to Create Unique Physical Designators</b> .....	<b>160</b>
<b>17.4.2 Component Naming</b> .....	<b>160</b>
17.4.2.1 Defining Custom Designator Formats .....	161
<b>17.5 Annotating Compiled Sheets</b>	<b>161</b>
<b>17.5.1 Annotate Sheet Options</b> .....	<b>162</b>
17.5.1.1 Annotate Sheet.....	163
17.5.1.2 Move Up / Move Down .....	163
<b>17.5.2 Applying Custom Sheet Numbers</b> .....	<b>164</b>
<b>17.5.3 Using Compiled Sheet Annotation in Board Level Annotation</b> .....	<b>164</b>
<b>17.6 Displaying Physical Names.</b>	<b>164</b>
<b>Module 18: Variants</b>	<b>167</b>
<b>18.1 Overview</b>	<b>167</b>
<b>18.1.1 Driving Fabrication Outputs</b> .....	<b>167</b>
<b>18.2 Variant Management</b>	<b>168</b>



18.2.1 Adding a Variant .....	169
18.2.2 Selecting a Variant.....	169
18.2.3 Allowing Variation of Fabrication Outputs .....	170
18.2.4 Specifying Variant Parameters .....	170
18.2.5 Renaming a Variant .....	171
18.2.6 Duplicating a Variant .....	171
18.2.7 Changing Variant Order .....	171
18.2.8 Removing a Variant .....	172
<b>18.3 Specifying Components for a Variant</b> .....	<b>172</b>
18.3.1 Varying Components.....	172
18.3.2 Varying Component Parameters .....	173
18.3.3 Update Parameters for multiple Component Selections.....	174
18.3.4 Batch Update of Parameters from a Library Component.....	175
18.3.5 True Variants or Alternate Components .....	178
18.3.5.1 Selection of Alternate Parts .....	178
18.3.5.2 Updating Alternate Parts on the Schematic .....	179
18.3.5.3 Alternate Parts, Design Transfer, and PCB Footprints .....	180
18.3.6 Editing Techniques.....	181
18.3.6.1 Selection.....	181
18.3.6.2 Filtering Varied Components.....	181
18.3.6.3 Type-ahead Data Filtering.....	182
<b>18.4 Reporting / Saving Variant Information</b> .....	<b>182</b>
18.4.1 Exporting ASCII .....	182
18.4.2 Basic Report .....	182
18.4.2.1 Detailed Report.....	183
<b>18.5 Accessing a Variant from the Schematic</b> .....	<b>184</b>
18.5.1 Cross Probing.....	186
<b>18.6 Graphical Editing of Variants</b> .....	<b>186</b>
18.6.1 Choosing which Variant to Use .....	187
18.6.2 Graphically Editing Component Fitted / Not Fitted Status .....	187
18.6.3 Graphically Varying Component Parameters.....	188
<b>18.7 Variant Parameter Non-Graphical Editing</b> .....	<b>189</b>
18.7.1 Component Parameter Editing .....	189
<b>18.8 Visualization of Variant Information on the PCB</b> .....	<b>190</b>
18.8.1 Viewing a Varied Component Comment.....	190
18.8.2 Viewing Variants in 3D .....	190



<b>18.9 Generating Output for a Variant</b>	<b>191</b>
<b>18.9.1 Handling Non-Fitted Components on Print Output</b>	<b>192</b>
18.9.1.1 Schematic Drawing Options	193
18.9.1.2 PCB Drawing Options	193
<b>18.9.2 Labeling Print Output with the Variant Name</b>	<b>194</b>
<b>18.10 Impact of Design Updates on Variants</b>	<b>195</b>
<b>Module 19: Pin Swapping</b>	<b>197</b>
<b>19.1 Configuring Swap Groups</b>	<b>197</b>
<b>19.1.1 Pin Groups</b>	<b>198</b>
<b>19.1.2 Part Groups and Sequence IDs</b>	<b>198</b>
<b>19.1.3 Differential Pair Groups</b>	<b>199</b>
<b>19.1.4 Controlling How the Swaps are Performed on the Schematic</b>	<b>199</b>
19.1.4.1 Swapping Pins	200
19.1.4.2 Swapping Net Labels	200
<b>19.2 Enabling Pin, Pair and Part Swapping on the PCB</b>	<b>201</b>
<b>19.2.1 Swap Manager Dialog</b>	<b>201</b>
<b>19.3 Performing Pin, Pair and Part Swaps</b>	<b>202</b>
<b>19.3.1 Interactive Pin, Pair and Part Swapping</b>	<b>202</b>
<b>19.3.2 Automatic Pin/Net Optimizer</b>	<b>203</b>
<b>19.4 Passing Changes Back to the Schematic</b>	<b>203</b>
<b>19.4.1 Pushing the Changes from the PCB to the Schematic</b>	<b>203</b>
<b>19.5 Taking Advantage of Pin/Part Swap System</b>	<b>204</b>
<b>Module 20: Schematic Templates</b>	<b>207</b>
<b>20.1 Schematic Document Options</b>	<b>207</b>
<b>20.1.1 General section</b>	<b>207</b>
20.1.1.1 Schematic Units	207
20.1.1.2 Grids	207
20.1.1.3 Sheet and Border Colors	208
<b>20.1.2 Page Options</b>	<b>208</b>
20.1.2.1 Formatting and Size	208
20.1.2.2 Standard Style	209
20.1.2.3 Custom Style	209
20.1.2.4 Template Style	209
20.1.2.5 Margin and Zones	210
<b>20.2 Drawing Objects and Graphics</b>	<b>210</b>
<b>20.2.1 PolyLine</b>	<b>210</b>



20.2.2 Graphics .....	211
20.2.3 Text .....	212
<b>20.3 Special Strings</b> .....	<b>214</b>
20.3.1 Document Parameters.....	214
20.3.2 Project Parameters .....	215
<b>20.4 Bringing It All Together</b> .....	<b>216</b>
<b>20.5 Assigning a Template to a Schematic Sheet</b> .....	<b>217</b>
<b>Module 21: PCB and PCB Library Templates</b> .....	<b>219</b>
<b>21.1 PCB Templates</b> .....	<b>219</b>
21.1.1 Mechanical Layer Usage .....	219
21.1.2 Component Layer Pairs .....	222
21.1.3 Title Block and Reference Zones .....	222
21.1.3.1 DXF Import .....	222
21.1.3.2 Title Block Mirrored.....	225
21.1.4 Additional Documentation .....	227
21.1.4.1 Fabrication and Assembly Placed as Text .....	227
21.1.4.2 Fabrication and Assembly Placed from File .....	229
21.1.4.3 Layer Stack Table.....	230
21.1.4.4 Drill Table .....	230
<b>21.2 PCB Library Templates</b> .....	<b>231</b>
<b>Module 22: Bill of Material Templates</b> .....	<b>233</b>
<b>22.1 The BOM Template</b> .....	<b>233</b>
22.1.1 Formatting Template Fields .....	233
22.1.2 Formatting Template Columns .....	234
<b>Module 23: Draftsman Templates</b> .....	<b>237</b>
<b>23.1 Draftsman Properties</b> .....	<b>237</b>
23.1.1 Draftsman Templates .....	237
23.1.2 Draftsman Default Preferences .....	238
23.1.3 The Properties Panel General Section.....	238
23.1.4 The Properties Panel Page Options Section .....	239
23.1.4.1 Template.....	239
23.1.4.2 Standard Mode.....	239
23.1.4.3 Custom Mode .....	239
23.1.4.4 Margin and Zones.....	240
<b>23.2 Drawing Objects and Graphics</b> .....	<b>240</b>



23.2.1 PolyLine.....	240
23.2.2 Graphics.....	240
23.2.3 Text .....	242
<b>23.3 Special Strings</b>	<b>243</b>
23.3.1 Document Parameters.....	243
23.3.2 Project Parameters .....	244
<b>23.4 Using a Template for a Draftsman Document</b>	<b>245</b>
<b>Module 24: PCB Project Template</b>	<b>247</b>
<b>24.1 The PCB Project Document Template</b>	<b>247</b>
24.1.1 Project Options .....	248
24.1.2 Project Parameters Tab.....	249
24.1.2.1 Displaying Special Strings.....	250
<b>24.2 Adding Schematics</b>	<b>252</b>
<b>24.3 Adding a PCB Template</b>	<b>255</b>
<b>24.4 Adding Output Job Files</b>	<b>257</b>
24.4.1 Adding Outputs to an OutJob.....	258
24.4.1.1 Adding a Fabrication Drawing to the OutJob .....	258
<b>24.5 Adding BOM Template</b>	<b>263</b>
<b>24.6 Putting it All Together</b>	<b>264</b>
24.6.1 Targeting New Document Defaults and Templates Directory.....	264
24.6.2 Using the Template.....	265
<b>Module 25: Defining the Board Shape (Review)</b>	<b>267</b>
<b>25.1 The Board Shape</b>	<b>267</b>
<b>25.2 Defining the Board Shape</b>	<b>267</b>
25.2.1 Defining the Board Shape from Existing Primitives .....	267
25.2.2 Defining the Board Shape from a DXF File.....	269
25.2.3 Defining the Board Shape Using a 3D Body .....	269
<b>25.3 Creating a Keepout from the Board Shape</b>	<b>270</b>
<b>25.4 Adding Cutouts to the Board Shape</b>	<b>271</b>
<b>25.5 Board Planning Mode</b>	<b>272</b>
25.5.1 Editing the Board Shape .....	272



25.5.1.1 Editing Board Vertices .....	273
25.5.1.2 Moving Solid Vertices .....	273
25.5.1.3 Moving Non-Solid Vertices or Edges.....	273
25.5.1.4 Inserting new Vertex.....	273
<b>25.5.2 Modifying the Board Shape .....</b>	<b>274</b>
<b>25.5.3 Redefining the Board Shape.....</b>	<b>274</b>
<b>25.5.4 Tips when defining/redefining the board shape .....</b>	<b>275</b>
<b><u>Module 26: Testpoints</u></b>	<b><u>277</u></b>
<b><u>26.1 Testpoint Strategy</u></b>	<b><u>277</u></b>
<b>26.1.1 Fabrication Testing.....</b>	<b>278</b>
26.1.1.1 Testpoint Location Constraints.....	278
<b>26.1.2 Assembly Testing .....</b>	<b>278</b>
26.1.2.1 Testpoint Location Constraints.....	279
<b><u>26.2 Pad and Via Testpoint Support</u></b>	<b><u>279</u></b>
<b><u>26.3 Testpoint Design Rules</u></b>	<b><u>281</u></b>
<b><u>26.4 Managing Testpoints</u></b>	<b><u>282</u></b>
<b><u>26.5 Checking the Validity of Testpoints</u></b>	<b><u>283</u></b>
<b><u>26.6 Testpoint-related Query Fields</u></b>	<b><u>284</u></b>
<b><u>26.7 Generating Testpoint Reports</u></b>	<b><u>284</u></b>
26.7.1 Role of the IPC-D-356A Netlist File.....	285
<b><u>Module 27: Embedded Board Array</u></b>	<b><u>287</u></b>
<b><u>27.1 Placement</u></b>	<b><u>287</u></b>
27.1.1 OPTIONS/CONTROLS .....	288
<b><u>27.2 Editing an Embedded Board Array</u></b>	<b><u>289</u></b>
27.2.1 Editing pre-placement .....	289
27.2.2 Editing during/post placement .....	289
27.2.3 Editing via the PCB list panel .....	290
27.2.4 Graphical Editing .....	290
<b><u>27.3 Embedded Board Array (Panel) Templates</u></b>	<b><u>291</u></b>
<b><u>27.4 Route Tool Path</u></b>	<b><u>291</u></b>
<b><u>Module 28: Design Views</u></b>	<b><u>297</u></b>
<b><u>28.1 Placing a Design View</u></b>	<b><u>297</u></b>



28.1.1 Defining the View Area.....	298
28.1.2 Setting the Location and Scale of the Design View.....	299
28.1.3 Defining the Title.....	299
<b><u>28.2 Interactively adjusting the Scale, Size and Focus of the Design View</u></b>	<b>299</b>
<b><u>28.3 Placing a View of the Bottom Side of the Board</u></b>	<b>300</b>
<b><u>28.4 Placing Design View of Different PCB Document</u></b>	<b>301</b>
<b><u>28.5 Including Design View in Printed &amp; PDF Output</u></b>	<b>302</b>



