
Table of Contents

Module 1: Getting Started With Altium Designer

1.1	Introduction to Altium Designer	1-1
1.2	The Altium Designer environment.....	1-2
1.3	Document Editor Overview.....	1-6
1.4	Working with projects and documents	1-10

Module 2: Help and DXP system menu

2.1	Using the Help system	2-1
2.2	Using the Altium website	2-3
2.3	DXP System menu.....	2-10

Module 3: Schematic Editor Basics

3.1	Schematic Editor basics	3-1
3.2	Schematic graphical objects.....	3-4
3.3	Schematic electrical objects	3-9

Module 4: Schematic Capture

4.1	Introduction to Schematic Capture.....	4-1
4.2	The Schematic Editor workspace.....	4-2
4.3	Libraries and components	4-9
4.4	Placing and wiring.....	4-17

Module 5: Multi-Sheet Design

5.1	Multi-Sheet Design.....	5-1
-----	-------------------------	-----

Module 6: Building the Project

6.1	Assigning designators.....	6-1
6.2	Compiling and verifying the project	6-5

Module 7: Setting Up for Transfer to PCB and Importing Data

7.1	Setting Up for Transfer to PCB.....	7-1
-----	-------------------------------------	-----

Module 8: PCB Editor Basics

8.1	PCB Editor Basics.....	8-1
-----	------------------------	-----

8.2	PCB design objects.....	8-4
-----	-------------------------	-----

Module 9: Setting up the PCB

9.1	Setting up the PCB.....	9-1
9.2	Creating a new PCB	9-6
9.3	Setting up the PCB layers.....	9-10

Module 10: Global Editing

10.1	Editing Multiple Text Objects	10-1
10.2	The Data Editing System	10-4

Module 11: PCB Design Flow, Transferring a Design and Navigation

11.1	PCB design process	11-1
11.2	Transferring design information to the PCB.....	11-3
11.3	Using the PCB Panel	11-7
11.4	Project Navigation and Cross Probing.....	11-16

Module 12: Design Rules

12.1	Design rules and design rule checking	12-1
------	---	------

Module 13: Classes and Rooms

13.1	Object classes	13-1
13.2	Rooms	13-3

Module 14: Placement and Re-annotation

14.1	Component Placement tools	14-1
14.2	Re-Annotation and back annotate.....	14-3

Module 15: Schematic Library Editor

15.1	Introduction to Library Editing	15-1
15.2	Schematic Library Editor.....	15-2

Module 16: PCB Library Editor

16.1	PCB Library Editor	16-1
16.2	3D dimensional component detail.....	16-14

Module 17: Linking Models, Parameters, Library Package and Updates

17.1	Adding Model and Parameter Detail to a Component	17-1
17.2	Component Auditing	17-8
17.3	3D PCB Components	17-14
17.4	Library Package types	17-18
17.5	Library Reports.....	17-20

Module 18: Routing and Polygons

18.1	Routing	18-1
18.2	Testpoint System	18-10
18.3	Adding and removing teardrops	18-15
18.4	Automatic routing	18-16
18.5	Polygons and the Polygon Manager	18-19

Module 19: Output Generation and CAM File Editing

19.1	Bill of Materials	19-1
19.2	Output Generation.....	19-8
19.3	CAM Editor	19-20

Module 20: Interfacing to 3D Mechanical CAD

20.1	Interfacing to 3D Mechanical CAD	20-1
------	--	------