

TABLE OF CONTENTS

•	Page	•	Page
• Objectives	3	• PCB Design	202
• Review Fundamentals	4	• Design Tools	204
• Reflections	8	• Gathering Design Data	209
• Transmission Line Definition	10	• Designing the PCB Stackup	212
• Terminations	21	• Creating an EMI Containment Strategy	217
• Types of Transmission Lines	32	• Developing Routing Rules	228
• Calculating Impedance	33	• Supervising PCB Routing	232
• PCB Structures, right angle bends, vias, etc.	47	• Post Layout Checking	232
• Crosstalk	58	• Testing Fabricated PCBs	233
• Differential Signaling	68	• EMI Testing	2327
• IC Package Effects	78	• What Happens When Signals Get Very Fast	239
• Power Delivery System Design	90	• PCB Fabrication	262
• Bypass Capacitors and Inductors	105	• Low Loss/High Speed Materials	269
• Plane Capacitance	153	• Blind and Buried Vias	277
• Modeling the PDS	171	• Summary	279
• Noise Margin Analysis and Design Rule Creation	180	• References	280

ISSUES TO BE COVERED

- **Power Integrity**
- **EMI Containment**
- **Transmission Line Rule Generation**
- **IC Package Integrity**
- **Routing Rule Generation**
- **Verifying Design Is Correct**
- **PCB Fabrication and Materials**
- **What Happens When Signals Become Very Fast**

OBJECTIVES

- **Create stable transmission lines.**
- **Create a stable power subsystem.**
- **Select IC packages that are capable of housing the circuits to be used.**
- **Design PCBs that work the first time.**
- **Pass EMC Tests**