Cherry Clough
Consultants

9 Bracken View, Brocton
Stafford, Staffs, UK

tel: +44 (0)1785 660 247
fax +44 (0)1785 660 247
email: keith.armstrong@cherryclough.com
web: www.cherryclough.com

EMC for Printed Circuit Boards

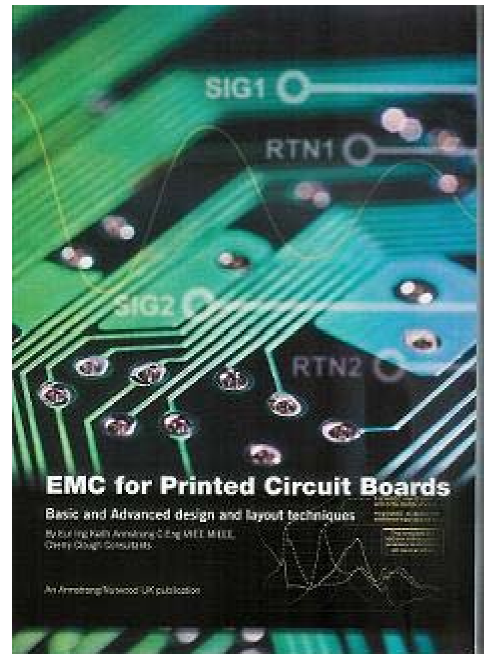
Basic and Advanced Design and Layout Techniques

Keith Armstrong First published February 2007

Perfect bound (with titled spine): ISBN 978-0-9555118-1-3

Spiral bound (lays flat): ISBN 978-0-9555118-0-6

To order, visit: http://www.compliance-club.com



Overview, and complete list of contents

This book is for electronic circuit designers, as well as for PCB designers themselves, and has full-colour figures throughout.

All application areas are covered, from household appliances, commercial, industrial and medical equipment, through automotive to aerospace and military.

The techniques it describes help you to…

- Improve signal integrity (SI), signal/noise ratio (S/N), especially in mixed technologies
- Comply with EMC Directive, FCC, etc. with the lowest cost of manufacture
- Reduce the number of iterations of hardware and software to reduce time-to-market whilst also reducing financial risks
- Improve the reception range of co-located wireless voice or data communications (GSM, PCS, GPRS, EDGE, CDMA2000, UMTS, Bluetooth, Wi-Fi, UWB, etc.)
- Improve GPS or Galileo reception when using co-located antennas
- Save cost, size and weight by reducing (or eliminating) shielding/filtering of the overall enclosure
- Improve reliability, reduce warranty costs without adding significantly to cost of manufacture
- Use very high-speed devices, high-power digital signal processing (DSP), latest IC technologies (90 or 65nm), and/or latest packaging technologies (chip scale, flip-chip, micro-BGA, etc.)

Its eight chapters cover…

1) Saving time and cost overall
2) Segregation and interface suppression
3) PCB-chassis bonding
4) Reference planes for 0V and power
5) Decoupling, including buried capacitance technology
6) Transmission lines (and any traces carrying high-speed signals or noise)
7) Routing and layer stacking, including microvia technology
8) A number of miscellaneous issues (heatsinks, in-circuit testing, etc.)

This book describes the techniques, and when they are appropriate, in practical engineering language. It does not describe *why* they work in great detail, and only uses a few simple maths formulas where they are practically useful. However, these techniques *are* very well proven in practice and the reasons why they work *are* well understood.

The many web-based references lead to detailed explanations and mathematical foundations. It is difficult for textbooks to keep up to date with PCB technology and EMC techniques, which is why most of the references are conference papers and articles written during the last few years.

Although the subject is EMC, many of the techniques are essential for achieving good SI or S/N and such issues are often discussed – especially in the few areas where EMC and SI requirements could conflict.

Cherry Clough Consultants

Complete list of contents

	Introduction		**6**
Chapter 1	**Saving Time and Cost Overall**		**8**
1.1	Reasons for using these EMC techniques		8
	1.1.1	Development – reducing costs and getting to market on time	8
	1.1.2	Reducing unit manufacturing costs	9
	1.1.3	Enabling wireless datacommunications	10
	1.1.4	Enabling the use of the latest ICs and IC packages	10
	1.1.5	Easier compliance for high-power DSP	11
	1.1.6	Improving the immunity of analogue circuits	11
1.2	What do we mean by “high speed”		11
1.3	Electronic trends, and their implications for PCBs		12
	1.3.1	Shrinking silicon	12
	1.3.2	Shrinking packaging	14
	1.3.3	Shrinking supply voltages	14
	1.3.4	PCBs are becoming as important as hardware and software	15
	1.3.5	EMC testing trends	15
	1.3.6	Frequency, velocity and wavelength	15
1.4	Designing to reduce project risk		15
	1.4.1	Guidelines, maths formulae, and field solvers	15
	1.4.2	Virtual design	16
	1.4.3	Experimental verification	17
1.5	Responsibility for EMC		18
1.6	EMC-competent QA, change control, cost-reduction		19
1.7	Compromises		19
Chapter 2	**Segregation and Interface Suppression**		**20**
2.1	The Basics of Segregation and Interface Suppression		20
	2.1.1	Segregating the ‘Inside World’ from the ‘Outside World’	20
	2.1.2	Segregation inside the Inside World	21
	2.1.3	Implementing segregation on a PCB	21
	2.1.4	Interface suppression	22
	2.1.5	Implementing interface suppression on a PCB	23
	2.1.6	The synergy of shielding and filtering	24
2.2	PCB-level shielding		24
	2.2.1	Reasons for shielding on the PCB	24
	2.2.2	Overview of shielding at PCB level	25
	2.2.3	Types of PCB shielding-can	26
	2.2.4	Attaching shielding-cans to PCBs	27
	2.2.5	PCB shielding-can materials	27
	2.2.6	Apertures and gaps in shielding-cans	27
	2.2.7	Waveguide-below-cutoff methods	28
	2.2.8	Near field effects on shielding	29
	2.2.9	Cavity resonances	29
2.3	Interconnections and shielding		30
	2.3.1	Combining PCB shielding with filtering	31
2.4	Combining shielding with heatsinking		34
2.5	Environmental issues		34
2.6	PCB-level filtering		34
	2.6.1	Reasons for filtering on the PCB	34
	2.6.2	Overview of PCB filtering	34
	2.6.3	High-performance filtering requires a good quality RF reference	35
	2.6.4	Design of single-stage low-power and signal PCB filters	35
	2.6.5	Power filtering on PCBs	39
	2.6.6	Filtering for shielded connectors	39
2.7	Placement of off-board interconnections		39
Chapter 3	**PCB-to-Chassis Bonding**		**41**

3.1 Introduction to PCB-to-chassis bonding 41
3.1.1 What do we mean by 'chassis'? 41
3.1.2 What do we mean by 'bonding'? 41
3.1.3 Hybrid bonding 44
3.1.4 'Ground loops' and religion 44
3.2 Why bond PCB 0V planes to chassis anyway? 44
3.2.1 Reduced transfer impedance 44
3.2.2 Better control of common-mode 'leakage' 44
3.3 Benefits of closer spacing between a PCB and its chassis 46
3.4 The 'highest frequency of concern' 46
3.5 Controlling resonances in the PCB-chassis cavity 47
3.5.1 Why and how the cavity resonates 47
3.5.2 Wavelength rules 48
3.5.3 Increasing the number of bonds to increase resonant frequencies 48
3.5.4 What if we can't use enough bonds? 49
3.5.5 Spreading the resonances more widely to reduce peak amplitude 50
3.5.6 Designing resonances to miss problem frequencies 50
3.5.7 Being clever with capacitors 50
3.5.8 Using resistors to 'dampen' cavity resonances 50
3.5.9 Using absorber to 'dampen' cavity resonances 51
3.5.10 Reducing the impedance of capacitive bonds 51
3.5.11 Using shielding techniques 52
3.5.12 Using fully shielded PCB assemblies 52
3.6 Daughter and mezzanine boards 52

Chapter 4 Reference Planes for 0V and Power 54
4.1 Introduction to Reference Planes 54
4.2 Design issues for reference planes 55
4.2.1 Plane dimensions 55
4.2.2 Dealing with gaps and holes in planes 55
4.2.3 Cross-hatching and copper fills 58
4.2.4 Connecting devices to planes 59
4.2.5 Thermal breaks 60
4.2.6 Device placement 60
4.2.7 Fills and meshes 61
4.2.8 Resonances in the 0V plane 61
4.2.9 Cavity resonances in plane pairs 62
4.2.10 Reducing the 'edge-fired' emissions from plane pairs 63
4.2.11 Locating via holes for aggressive signals or power 64
4.2.12 When traces change layers 65
4.2.13 Component-side planes for DC/DC converters and clocks 65
4.3 Splitting a 0V plane is not generally a good idea any more 65
4.4 When traces must cross a 0V or power plane split 67
4.5 Advantages of 'High Density Interconnect' (HDI), 'build-up' and 'microvia' PCB technologies 67
4.6 The totally shielded PCB assembly 68

Chapter 5 Decoupling, including Buried Capacitance Technology 70
5.1 Introduction to decoupling 70
5.2 Decoupling with discrete capacitors 71
5.2.1 Which circuit locations need decaps? 71
5.2.2 The benefits of decaps in ICs and MCMs 71
5.2.3 How much decoupling capacitance to use? 72
5.2.4 Types of decaps 72
5.2.5 Layouts that reduce the size of the current loop 73
5.2.6 Series resonances in decaps 75
5.2.7 Using ferrites in decoupling 76
5.2.8 Splitting the decap into two 77
5.2.9 Using multiple decaps in parallel 77
5.2.10 Other ways to reduce decap ESL 80
5.3 Decoupling with 0V/Power plane pairs 81
5.3.1 Introduction to the decoupling benefits of 0V/Power plane pairs 81
5.3.2 The distributed capacitance of a 0V/Power plane pair 81
5.3.3 PCB 0V and power routing with 0V/Power plane pairs 82

5.3.4 Location of decaps 83
5.3.5 Defeating parallel decap resonances when using 0V/Power plane pairs 84
5.3.6 'Cavity resonances' in 0V/Power plane pairs 84
5.3.7 Bonding planes with decaps to increase resonant frequencies 85
5.3.8 Power plane islands fed by π filters 85
5.3.9 Damping cavity resonance peaks 86
5.3.10 The spreading inductance of planes 86
5.3.11 The 20-H rule 87
5.3.12 Taking advantage of decap series resonances 87
5.3.13 Decap walls 87
5.3.14 Other 0V/Power plane pair techniques to reduce emissions 87
5.3.15 The buried capacitance technique 88
5.4 Field solvers for power bus impedance simulations 89

Chapter 6 Transmission lines (and any traces carrying high-speed signals or noises) 91
6.1 Matched transmission lines on PCBs 91
6.1.1 Introduction 91
6.1.2 Propagation velocity, V and characteristic impedance, Z0 92
6.1.3 The effects of impedance discontinuities 92
6.1.4 The effects of keeping Z0 constant 94
6.1.5 Time Domain Reflectometry (TDR) 94
6.1.6 When to use matched transmission lines 95
6.1.7 Increasing importance of matched transmission lines for modern products 97
6.1.8 It is the real rise/fall times that matter 97
6.1.9 Noises and immunity should also be taken into account 98
6.1.10 Calculating the waveforms at each end of a trace 99
6.1.11 Examples of two common types of transmission lines 99
6.1.12 Coplanar transmission lines 100
6.1.13 The effects of capacitive loading 100
6.1.14 The need for PCB test traces 102
6.1.15 The relationship between rise/fall-time and frequency 102
6.2 Terminating transmission lines 102
6.2.1 A range of termination methods 103
6.2.2 Difficulties with drivers 105
6.2.3 Compromises in line matching 106
6.2.4 ICs with 'smart' terminators 106
6.2.5 Bi-directional terminations 106
6.2.6 Non-linear termination techniques 106
6.2.7 'Equalising' terminations 107
6.2.8 Location of terminations at the ends of transmission-lines 107
6.3 Transmission line routing constraints 107
6.3.1 General routing guidelines 107
6.3.2 A transmission line exiting a product via a cable 108
6.3.3 Interconnections between PCBs inside a product 109
6.3.4 Changing plane layers within one PCB 110
6.3.5 Crossing plane breaks or gaps within one PCB 111
6.3.6 Avoid sharp corners in traces 112
6.3.7 Linking return current planes with vias or decaps 112
6.3.8 Effects of via stubs 112
6.3.9 Effects of routing around via fields 113
6.3.10 Other effects of the PCB stack-up and routing 113
6.3.11 Some issues with microstrip 114
6.4 Differential matched transmission lines 115
6.4.1 Introduction to differential signalling 115
6.4.2 CM and DM characteristic impedances in differential lines 116
6.4.3 Exiting PCBs, or crossing plane splits with differential lines 117
6.4.4 Controlling imbalance in differential signalling 118
6.4.5 Routing asymmetry 120
6.5 Choosing a dielectric 121
6.5.1 Effects of woven substrates (like FR4 and G-10) 121
6.5.2 Other types of PCB dielectrics 122
6.6 Matched-impedance connectors 123
6.7 Shielded PCB transmission lines 124
6.7.1 'Channelised' striplines 124
6.7.2 Creating fully shielded transmission lines inside a PCB 124

6.8	Miscellaneous related issues	125
6.8.1	Impedance matching, transforming and AC coupling	125
6.8.2	A ‘safety margin’ is a good idea	125
6.8.3	Filtering	126
6.8.4	CM chokes	126
6.8.5	Replacing parallel busses with serial	127
6.8.6	The lossiness of FR4 and copper	127
6.8.7	Problems with coated microstrip	127
6.8.8	The effects of bond-wires and leads	127
6.9	Simulators and solvers help design matched transmission lines	127
6.10	Some useful sources of further information on PCB transmission lines	128
Chapter 7	**Routing and Layer Stacking, including Microvia Technology**	**130**
7.1	Routing and layer stacking techniques, and microvia technology	130
7.2	Routing	130
7.3	Stack-ups	130
7.3.1	The benefits of closer trace-plane spacing	130
7.3.2	The benefits of closer component-plane spacing	131
7.3.3	Copper balancing	131
7.3.4	Single-layer PCBs	132
7.3.5	Two-layer PCBs	133
7.3.6	Four-layer PCB stack-ups	133
7.3.7	Six layer PCBs	135
7.3.8	Eight layer PCBs	136
7.3.9	PCBs with more than eight layers	137
7.3.10	Number of PCB layers and cost-effective design in real-life	137
7.3.11	Shielding power planes with different voltages	138
7.4	EMC issues with copper balancing using area fills or cross-hatches	138
7.5	HDI PCB technology	139
7.5.1	What is HDI?	139
7.5.2	The EMC benefits of HDI	139
7.5.3	HDI suppliers and costs	140
7.5.4	HDI PCD design issues	140
7.5.5	More information on HDI	140
7.6	Current capacity of traces	140
7.6.1	Handling surge and transient currents	140
7.6.2	Maximum continuous DC and low frequency current handling	141
7.6.3	Voltage drops in the PCB’s power distribution	142
7.6.4	Handling continuous RF currents	142
7.6.5	A note on accuracy	142
7.7	Transient and surge voltage capacity of layouts	142
7.7.1	Trace-trace and trace-metal spacing	142
7.7.2	The EMC and safety problems caused by compliance with the RoHS directive	143
Chapter 8	**A Number of Miscellaneous Final Issues**	**144**
8.1	Power supply connections to PCBs	144
8.2	Low-K dielectrics	144
8.3	Chip-scale packages (CSPs)	145
8.4	Chip-on-board (COB)	145
8.5	Heatsinks on PCBs	146
8.5.1	EMC effects of heat sinks	146
8.5.2	Heat sink RF resonances	146
8.5.3	Bonding heatsinks to a PCB plane	149
8.5.4	Combining shielding with heatsinking	152
8.5.5	Other heatsink techniques that may help	152
8.5.6	Heatsinks for power devices	153
8.6	Package resonances	154
8.7	Eliminate the test pads for bed-of-nails or flying probe testing	154
8.8	Unused I/O pins	155
8.9	Crystals and oscillators	155
8.10	IC tricks	155

8.11	Location of terminations at the ends of transmission-lines	156
8.12	Electromagnetic Band Gap (EBG)	156
8.13	Some final PCB design issues	157
8.14	Beware board manufacturers changing layouts or stack-ups	157
8.15	Future-proofing the EMC design	158
8.15.1	Marking EMC design features or critical parts on the design drawings	158
8.15.2	A quality-controlled procedure for EMC design	158
	References	**159**
	Glossary of Terms and Abbreviations	**165**
	Author, Keith Armstrong's biography	**166**