Another EMC resource
from EMC Standards

EMC techniques for PCBs book, overview & contents

Helping you solve your EMC problems

9 Bracken View, Brocton, Stafford ST17 0TF T:+44 (0) 1785 660247 E:info@emcstandards.co.uk

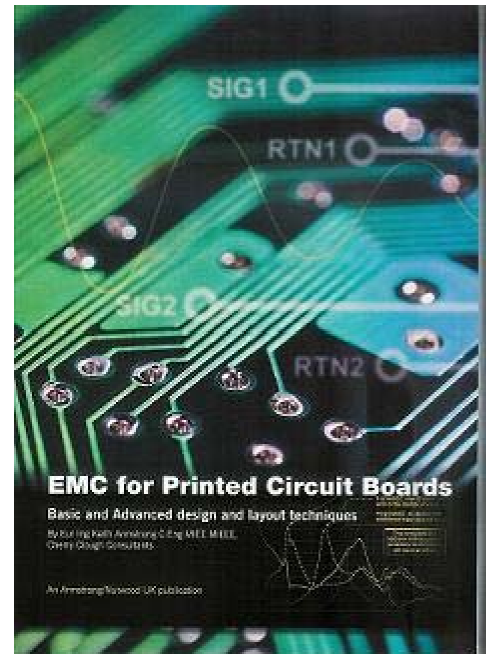
Cherry Clough Consultants

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