Drop-impact analysis of packaged electronic products containing nonlinear cushion buffers and multilayer circuit boards

Wang, Yuqi

2008

Wang, Y. (2008). Drop-impact analysis of packaged electronic products containing nonlinear cushion buffers and multilayer circuit boards. Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/5368>

<https://doi.org/10.32657/10356/5368>

Nanyang Technological University

Downloaded on 26 Apr 2023 19:23:48 SGT

Drop-Impact Analysis of Packaged Electronic Products Containing Nonlinear Cushion Buffers and Multilayer Circuit Boards

Wang Yuqi

School of Mechanical & Aerospace Engineering

A thesis submitted to the Nanyang Technological University

in fulfillment of the requirements for the degree of

Doctor of Philosophy

2008

Statement of Originality

I hereby certify that the content of this thesis is the result of work done by me and has not been submitted for a higher degree to any other University or Institution.

.....

Date

.....

Wang Yuqi

Acknowledgments

I would like to express my appreciation and gratitude to my supervisor, Prof. K. H. Low, for his advice, encouragement, and patience throughout the course of my research and graduate studies. I also wish to thank Prof. K. H. Hoon, Prof. H. L. Pang, Prof. V. Nader, Prof G. Chen and Prof Gerald Seet for their advice and guidance. Their insight into drop impact vibration and advanced electronic packaging was extremely helpful in interpreting experimental results and developing mathematical models. I would also like to thank all the technicians of Robotic Research Centre, Strength Material Lab, Material Lab A, Mechanics of Machine Lab, and Design and Research Center in Nanyang Technological University, for their kind assistance with the use of equipment and facilities.

Finally, I wish to thank my friends and fellow graduate students in Robotics Research Center, Material Lab A, Mechanics of Machine Lab, and Centre for Advanced Numerical Engineering Simulations, for helpful discussion about the experiments and numerical analysis. I would also like to thank my FYP students and Master students Simon Wai, Keith Yong, Chee How Tan, Wilman Nah, etc. for their support to my PhD work.

Abstract

Modern electronic products are becoming lighter in weight and smaller in size as well as its increasing functionality. They are often experiencing the severe vibration and shock environments in many applications. Drop/impact tolerance, or the ability to safely withstand accidental drop/impact, is becoming an increasingly important aspect of the reliability of electronic products. And shock analysis and corresponding protection methods are quite meaningful to the development of electronic products. Thus, the objective of this work is to realize quick drop-impact analysis of packaged electronic products.

The focus of this work is to develop a full-scale virtual buffer model to realize a high efficient analysis for large-scale packaged electronic product model. Two steps are performed to achieve this virtual buffer model: first, establish virtual buffer model; second, create objective model and incorporate into virtual buffer model. The proposed local linearization method is contributed to the establishment of virtual buffer model. This work can save the computational time on buffer modeling and contact simulation. In addition, the hybrid laminate modeling method is applied in the creation of multilayer circuit board, which is objective model in the virtual buffer model. For complex multilayer model, the hybrid laminate modeling method can realize high-efficient two-dimensional simulations without numerous simulations of three-dimensional unit-cell model and laborious physical measurement of various material properties.

We conclude that virtual buffer model provides a platform for the applica-

tion of local linearization method and hybrid laminate modeling model, and these two methods increase the efficiency of drop-impact analysis for packaged electronic products. The results presented in this PhD thesis proven that virtual buffer model can be effective to reduce the product development cycle and improve the flexibility of optimization design.

Table of Contents

Acknowledgments	i
Abstract	ii
List of Figures	x
List of Tables	xviii
List of Notations	xx
List of Abbreviations	xxiii
1 Introduction	1
1.1 Background	1
1.1.1 Role of Drop-Impact Analysis in Electronic Products Design	2
1.1.2 Need for Numerical Analysis in Drop-Impact Analysis	4
1.1.3 Methodologies to Improve the Efficiency of Numerical Analysis	5
1.2 Objectives and Contributions	7
1.3 Thesis Organization	9
2 Literature Reviews	12

TABLE OF CONTENTS

v

2.1	Drop-Impact Tests for Electronic Products	12
2.1.1	Drop Tests of Full-Scale Electronic Products	13
2.1.2	Drop Tests for Printed Circuit Boards	15
2.1.3	Drop Tests for Cushion Buffers	18
2.2	Drop-Impact Simulation for Electronic Products	21
2.2.1	Basic Techniques and Procedure of Drop-Impact Simulation for Electronic Devices	22
2.2.2	Time Integration Scheme of FEM Simulation for Drop-Impact Analysis	25
2.2.3	Drop Impact Simulation of PCB with Foam Packaging by Using Explicit Software	27
2.3	Efficiency of Methodologies for Large-Scale Simulation	28
2.3.1	Mesh Control Methods	29
2.3.2	Initial Investigation of Hybrid Efficiency Methods	38
2.4	Dynamic Analysis of Nonlinear Drop Impact Cushion Systems . . .	42
2.4.1	Relationship Between Impact Stress-Strain Curve and Cush- ion Curve	43
2.4.2	Application of Elliptic Function into Nonlinear Drop Impact System	45
2.4.3	Hybrid Analytical-Numerical Analysis for Nonlinear Damped System	48
2.5	Modeling Techniques of Printed Circuit Boards	52
2.5.1	Classification of Printed Circuit Boards	53
2.5.2	Structure of Printed Circuit Boards	53
2.5.3	Material Buildup of Multilayer Circuit Boards	56

TABLE OF CONTENTS

vi

2.5.4	Previous Modeling of Multilayer and Woven Fiber Reinforced Composites	61
2.5.5	Previous Modeling of Printed Circuit Boards	68
3	Initial Development of Virtual Buffer Model for Quick Drop-Impact Analysis of PCB in Electronic Products	72
3.1	Introduction of Chapter	73
3.2	Methodology of Virtual Boundary Method	76
3.2.1	Definitions in Virtual Boundary Method	76
3.2.2	Solution Scheme	79
3.2.3	Comparison of Methods	80
3.3	Application of Virtual Boundary Method in Virtual Buffer Model .	82
3.3.1	Establishment of Virtual Buffer Model	82
3.3.2	Model Verification	84
3.3.3	Analysis of Objective Model	92
3.4	Summary of Chapter	96
3.5	Development of Full-Scale Virtual Buffer Model	100
4	Further Development of Virtual Buffer Model by Using a Local Linearization Method for Quick Extrication of Boundary Information in Nonlinear Cushion Systems	103
4.1	Introduction of Chapter	104
4.2	Methodology of Local Linearization Method	106
4.3	Mathematical Analysis	110
4.3.1	Linearized Differential Equation of Motion for Nonlinear Systems	110

TABLE OF CONTENTS

vii

4.3.2	Dynamic Loading Analysis of Damped Nonlinear Systems	113
4.3.3	Transient Response Analysis of Damped Nonlinear Systems	118
4.4	Application of Local Linearization Method	123
4.4.1	Dynamic Loading	124
4.4.2	Transient Response	125
4.5	Experimental Verification	126
4.5.1	Experimental Setup	127
4.5.2	Dynamic Loading Verification	127
4.5.3	Transient Response Verification	132
4.6	Summary of Chapter	136
4.7	Furtherer Development of Full-Scale Virtual Buffer Model with the Contribution from Local Linearization Method	136
5	Final Development of Full-Scale Virtual Buffer Model by Using a Hybrid Laminate Modeling Method to Model Objective Model: Multilayer Circuit Boards	139
5.1	Introduction of Chapter	140
5.2	Physical Characterization of Multilayer Composites	141
5.3	Presentation of Multilayer Finite Element	142
5.3.1	Elastic-Plastic Damaging Ply Model	145
5.3.2	Bi-phase Ply Model	145
5.4	Modeling Methodologies for Static Analysis	152
5.4.1	Direct Bi-Phase Multilayer Model	153
5.4.2	EQLAM Multilayer Model	155
5.4.3	Integrated Orthotropic Model	174

TABLE OF CONTENTS

viii

5.5	Dynamic Analysis for Multilayer Circuit Boards	179
5.5.1	Experimental Setup for Dynamic Tests	180
5.5.2	Dynamic Simulation	183
5.5.3	Comparison of Simulation and Test Results	184
5.5.4	Comparison of CPU Time in Different Models	184
5.5.5	Discussion of Modeling Techniques	186
5.5.6	Stress and Strain Analysis	187
5.6	Summary of Chapter	192
5.7	Final Development of Full-scale Virtual Buffer Model with the Con- tribution from Hybrid Modeling of Multilayer Circuit Boards	192
6	Application of Full-Scale Virtual Buffer Model in a General Pro- totype for Packaged Electronic Products	195
6.1	Introduction of Chapter	196
6.2	Drop-Test Arrangement and Parameters	197
6.3	Application of Full-Scale Virtual Buffer Model	201
6.3.1	Establishment of Virtual Buffer Model	202
6.3.2	Creation of Objective Model for Incorporation into Virtual Buffer Model	204
6.4	Verification of Full-Scale Virtual Buffer Model by Experimental Re- sults	208
6.4.1	Verification of Boundary Field in Virtual Buffer Model . . .	208
6.4.2	Verification of Shock Response of Objective Model in Virtual Buffer Model	208
6.5	Summary of Chapter	213

TABLE OF CONTENTS ix

7	Conclusions and Recommendations	215
7.1	Summary of Thesis	215
7.2	Summary of Contributions	216
7.3	Suggestions for Future Work	219
	Bibliography	222
	Author's Publications	235

List of Figures

1.1	Schematic structure of PhD thesis	11
2.1	Experimental set-up for general drop test	14
2.2	Sensor layout on TFT-LCD monitor, where A denotes accelerom- eter, and S denotes strain gauge. (a) on panel; (b) on housing (B indicates fastening point of LCD panel to housing)	14
2.3	Experimental setup of drop laboratory	15
2.4	Drop tester conducting drop impact test of a mobile phone	16
2.5	Schematic diagrams showing impact orientations for both test methods	17
2.6	Cushion curves for foams made of EPS 20	20
2.7	Cushion tester	20
2.8	Structure and finite element model of TV set with packaging material	23
2.9	Finite element model of PCB and packaging material	28
2.10	Flow chart of global-local finite element method	31
2.11	Models with different meshes for global-local analysis	33
2.12	Vertical displacement of node 1 for three different models	34
2.13	Two-way stiffened plate	36
2.14	Differences between GL and super element approaches	37

LIST OF FIGURES

xi

2.15 Schematic diagram of composite cylinder	38
2.16 Stages of specified boundary method	40
2.17 Global-local finite element model of specified boundary method (a) finite element mesh of global model; (b) finite element mesh of local model	40
2.18 Global-local finite element model of multi-point constraint mode . .	41
2.19 Vibration model	45
2.20 Classification flow chart of printed circuit boards	54
2.21 Four-layer multilayer using foil lamination technique	55
2.22 Foil capped stack-up for a six-layer board	56
2.23 Buildup diagram for a six-layer multilayer circuit board	57
2.24 Component capped multilayer board	57
2.25 Fibers in plain weave pattern	59
2.26 Epoxy-resin-impregnated woven glass cloth	60
2.27 Photomicrographs of 7628 laminate under 67 times magnification a) Warp yarns parallel to the page, fill yarns perpendicular. b) Fill yarns parallel to the page, warp yarns perpendicular	60
2.28 Orientation and stacking of aluminum and E-glass / epoxy composite	62
2.29 Global-local multilayer model	62
2.30 Sections of symmetric cross-ply laminates	63
2.31 Unit cell of plain unit cell of plain-weave textile composite	64
2.32 Woven composite interlacing pattern	65
2.33 Micro-mechanical model	65
2.34 Types of fiber reinforced composite failures	66

LIST OF FIGURES

xii

2.35	Global composite ply damages in Ladevze model	67
2.36	Twist test set-up for determining shear modulus of PCB	69
2.37	Thermal and mechanical properties of PCB	70
2.38	3D FEM Model of BGA substrate with orthogonal copper layout . .	70
3.1	FEA model of entire TV with packaging materials and rigid floor .	75
3.2	FEA model of PCB board and chassis	75
3.3	Stages in virtual boundary method	77
3.4	Physical boundary and virtual boundary models	78
3.5	Comparison of computation time between virtual boundary and global/local methods	81
3.6	Establishment of virtual buffer model	83
3.7	Locations of nodes on bottom cabinet (top view)	86
3.8	Deflection of nodes at specified locations in physical buffer model .	86
3.9	Percentage error distributions of deflection for entire time domain .	87
3.10	Stress distribution of bottom cabinet at $t = 18.4$ ms	88
3.11	Equivalent von Mises stress profiles at points HS and HG given in Figure 3.10	89
3.12	Equivalent von Mises stress at rebound (a) physical buffer model and (b) virtual buffer model	90
3.13	Percentage error distributions at points HS and HG given in Fig- ure 3.10	91
3.14	Application of PCB into virtual buffer model	92
3.15	Connection of objective model with virtual buffer model	93

LIST OF FIGURES

xiii

3.16 Von Mises stress contour of PCB at $t = 8.8$ ms (rebound moment illustrated in Figure 3.15)	94
3.17 Deflection of central Node 5681 (PCB's central)	95
3.18 Screen capture of PCB y -translation results at $t = 25.6$ ms	95
3.19 Schematic of PCB in four different positions	97
3.20 Comparison of deflection at Node 4689 (point of high stress)	97
3.21 Comparison of Von Mises stress at Node 4689 (point of high stress)	98
3.22 Comparison of deflection at Node 5681 (PCB's central)	98
3.23 Comparison of Von Mises stress at Node 5681 (PCB's central)	99
3.24 Flow chart of full-scale virtual buffer model	102
4.1 Spring force characteristic curve for a strong nonlinear vibration system	109
4.2 Overview of local linearization method	109
4.3 Single-degree-of-freedom drop-impact system	111
4.4 Stress-strain curve and strain energy for cushion buffer	112
4.5 Strain energy curve and static load strain energy curve	119
4.6 Local calculation time parameters in transient response process with drop $h=50$ cm	122
4.7 Local calculation time parameters in transient response process with $W=280$ N	122
4.8 Comparison of mathematical and experimental results for transient response verification under condition: 400 N weight, 50 mm cushion thickness, 500 mm drop height and 400 cm ² contact area	126
4.9 Setup for cushion drop tests	127

LIST OF FIGURES

xiv

4.10 Comparison of mathematical and experimental results with $H/b=10$ for dynamic loading under different damping ratios	128
4.11 Comparison of mathematical and experimental results with $H/b=16$ for dynamic loading under different damping ratios	129
4.12 Comparison of mathematical and experimental results with $H/b=20$ for dynamic loading under different damping ratios	129
4.13 Acceleration error of mathematical models under various damping ratios with $H/b=10$ for dynamic loading under different damping ratios	130
4.14 Acceleration error of mathematical models under various damping ratios with $H/b=16$ for dynamic loading under different damping ratios	131
4.15 Acceleration error of mathematical models under various damping ratios with $H/b=20$ for dynamic loading under different damping ratios	131
4.16 Photomicrographs of EPS foam under 200 times magnification . . .	132
4.17 Comparison of mathematical and experimental results for transient response verification under condition: 200 N weight, 25 mm cushion thickness, 500 mm drop height and 400 cm ² contact area	133
4.18 Compression test for cushion buffer	134
4.19 Compressing and unloading stress-strain curves for cushion buffers .	134
4.20 An arbitrary unloading curve and its quadratic unloading expression	135
4.21 Operation of local linearization method in Visual Basic interface . .	137
4.22 Flow chart of full-scale virtual buffer model with the combination of local linearization method	138

LIST OF FIGURES

xv

5.1	Schematic graph of six-layer circuit board and structure of woven composite ply	143
5.2	Schematic graph of F6.1 multilayer circuit board	143
5.3	Theoretical description of multilayered shell element	145
5.4	Polygonal stress-strain curve for metallic plies	146
5.5	Bi-phase rheological composite model	146
5.6	Calibrated tensile or compressive damage law stress-strain curve . .	151
5.7	Schematic graph of direct bi-phase multilayer model	154
5.8	Damage stress strain curve of ED copper	155
5.9	Damage stress strain curve of brittle glass fiber	156
5.10	Damage stress strain curves of epoxy resin	156
5.11	Schematic graph of plain weave geometry	157
5.12	Process of improved EQLAM modeling method	159
5.13	Schematic graph of undulating bundle	159
5.14	Structure of EQLAM multilayer model in multilayer finite element code	171
5.15	Tensile test to measure damage properties of 7628 composite ply . .	172
5.16	Damage tensile test samples of 7628 prepreg	173
5.17	Stress-strain curves of 7628 prepreg along warp direction	173
5.18	Stress-strain curves of 7628 prepreg along fill direction	174
5.19	Tensile test for a six-layer circuit board sample	176
5.20	Stress-strain relationship for six-layer circuit board	177
5.21	Damage stress-strain curves of six-layer PCB along warp direction .	178
5.22	Damage Stress-strain curves of six-layer PCB along fill direction . .	178

LIST OF FIGURES

xvi

5.23	Drop impact experiment setup	181
5.24	A three-element strain rosette	182
5.25	Drop impact simulation model	182
5.26	Meshed two-dimensional circuit board model	183
5.27	Peak values of principal strains for different models	185
5.28	Strain contours of whole PCB at the instant of 6.6 ms (maximum principle strain)	188
5.29	Maximum principal strain profiles at Points A, B and C	189
5.30	Stress contours of whole PCB at the instant of 6.6 ms (maximum principle stress)	190
5.31	Maximum principal stress profiles at Points A, B and C	191
5.32	The transient response of maximum principle stress and strain at the central position B in Figures 5.28 and 5.32	191
5.33	Flow chart of full-scale virtual buffer model	194
6.1	PDT-56ED drop tester and test specimen	198
6.2	0°, 45° and 90° orientations of fixtures	198
6.3	Improved experimental set-up for 0° orientation of fixture	200
6.4	Improved experimental set-ups for 45° and 90° orientations of fixture	200
6.5	Drop test set up to measure the impact force between metal base and PCB	201
6.6	Establish virtual buffer model for prototypes	203
6.7	Transient acceleration response for different drop heights (from local linearization model)	204
6.8	Modeling process of EQLAM multilayer modeling method	205

LIST OF FIGURES

xvii

6.9	Connection of objective model with full-scale virtual buffer model .	206
6.10	Dynamic response of horizontal dropped PCB under different drop heights (acceleration along vertical drop direction)	207
6.11	Dynamic response of PCB under different drop angles (acceleration vertical to PCB plane)	207
6.12	Verification of boundary field for virtual buffer model under the drop height of 0.5 meter	209
6.13	Verification of boundary field for virtual buffer model under the drop height of 0.8 meter	209
6.14	Verification of boundary field for virtual buffer model under the drop height of 1.0 meter	210
6.15	Verification of dynamic response of objective model under the drop height of 0.5 meter	210
6.16	Verification of dynamic response of objective model under the drop height of 0.8 meter	211
6.17	Verification of dynamic response of objective model under the drop height of 1.0 meter (Verification of dynamic response of objective model under the drop angle of 0°)	211
6.18	Verification of dynamic response of objective model under the drop angle of 45°	212
7.1	Summary of the main part of thesis	217

List of Tables

2.1	Typical G -factors for various product types	19
2.2	Comparison of different meshed models illustrated in Figure 2.11 .	34
2.3	Different grades of laminates	59
3.1	An overview of finite element mesh of components	74
3.2	Comparison of different meshed models in TV structure	81
3.3	Maximum error and time of occurrence of each node for $t > 8$ ms	87
3.4	Summary of CPU time and space consumed for bottom drop simulation	91
3.5	Summary of the responses of PCB in four designed positions	96
5.1	Material properties and structure characterizations of F6.1 multilayer board (Material properties for direct bi-phase multilayer model)	144
5.2	Measurements of bundle sizes, crimp, and fiber volume fractions . .	157
5.3	Comparison of elastic properties for 7628 composite material	170
5.4	Orthotropic material properties of 7628 composite material	170
5.5	Damage parameters for 7628 composite	174
5.6	Material properties for EQLAM multilayer model	175
5.7	Material properties of integrated orthotropic model	175

LIST OF TABLES

xix

5.8	Comparison of elastic properties for six-layer circuit board	177
5.9	Damage parameters for six-layer circuit board	179
5.10	Material properties for integrated orthotropic model	180
5.11	Comparison of experimental and numerical peak values of maximum principal strain at central point when the drop height of 0.1 meter .	185
5.12	Comparison of experimental and numerical peak values of minimum principal strain at central point when the drop height of 0.1 meter .	185
5.13	Comparison of experimental and numerical peak values of maximum principal strain at central point when the drop height of 1 meter . .	186
5.14	Comparison of experimental and numerical peak values of minimum principal strain at central point when the drop height of 1 meter . .	186
5.15	Comparison of CPU time	187

List of Notations

τ_i	Local calculation time
$\lambda_{\max}$	Element maximum eigenvalue
L_e	Characteristic element dimension
c_d	Dilation wave velocity of material
Δt	Time step size
q	Total number of time steps
L	Characteristic length of elements
σ	Stress
ε	Strain
F	External force
Fc	Scale factor for stability reasons in simulation
c	Damping coefficient
H	Drop height
A	Contact area between cushion buffer and drop item
b	Undeformed thickness of cushion buffer
x	Deformation of cushion buffer
X	Maximum deformation amplitude
M	Mass
W	Weight of Package, $W = M \times g$
g	Acceleration of gravity
G	G-factor, dimensionless ratio between specific acceleration and the acceleration due to gravity

α, μ	Parameter of nonlinearity
β	Fitting coefficient in cubic curve fitting equation
P	Nonlinear curvature coefficient
p_i	Fitting coefficient in 9 th polynomial curve fitting equation
ω_0	Natural frequency of vibration
$\omega(x_i)$	Frequency of vibration at a local deformation x_i
$U(\sigma_m)$	Energy density function
U_i	Static load strain energy
K	Linear stiffness coefficient
K_E	Stiffness coefficient in linear spring
$K(x_i), K_i$	Stiffness coefficient at a local deformation x_i
T	Period of motion
$El\left(\frac{\alpha}{\omega_0}x^2(0)\right)$	Complete elliptic integral of the first kind
D, B	Nonlinear differential operators
γ, γ_1	Linear damping parameters
E	Young's modulus of cushion buffer
E_m	Young's modulus in stiffness matrix, ($m = 1, 2, 3$ or x, y, z)
G_{mn}	Shear modulus in stiffness matrix, ($m, n = 1, 2, 3$ or x, y, z)
δ	Deflection of plane
θ	Force-displacement angle
ζ	Damping ratio
$\bar{x}$	Displacement in the linearized nonlinear equation of motion
$\bar{\omega}$	Vibration frequency in the linearized nonlinear equation of motion
$\bar{K}$	Stiffness coefficient in the linearized nonlinear equation of motion
E_d	Energy dissipated by damping
E_i	Strain energy of buffer element
k'	Modulus of elliptic function
ν	Poisson's ratio
α_f	Fiber volume fraction
α_m	Matrix volume fraction

C^f	Stiffness matrix of glass fiber
C^m	Stiffness matrix of matrix
C^{UD}	Stiffness matrix of unidirectional (UD) ply
E^{UD}	Young's modulus of unidirectional (UD) ply
G^{UD}	Shear modulus of unidirectional (UD) ply
$C(d)$	Instantaneous modulus matrix
C_0	Undamaged initial modulus matrix
d	Scalar damage parameter
d_V	Scalar volumetric damage parameter
d_S	Scalar shear damage parameter
e_{ij}	Components of deviatoric strain tensor, $(i, j = 1, 2, 3 \text{ or } x, y, z)$
σ_i, ε_i	Initial threshold stress-strain
σ_1, ε_1	Intermediate stress-strain
σ_u, ε_u	Ultimate stress-strain
$E(\varepsilon)$	Modulus damage
E_0	Initial elastic modulus
E_1	Intermediate modulus
E_u	Ultimate modulus
d_1	Modulus damage
d_u	Ultimate damage
b_f	Thickness of fill ply
b_w	Thickness of warp ply
S	Compliance matrix
γ_{xy}	Shearing strain

List of Abbreviations

PCB	Printed Circuit Board
MEMS	Micro Electro Mechanical Systems
IC	Integrated Circuit
FEM	Finite Element Method
ASTM	American Society for Testing and Materials
FEA	Finite Element Analysis
GL	Global-Local
SBM	Specified Boundary Method
MPC	Multi-Point Constraint method
SDOF	Single Degree of Freedom
PTH	Plated Through Hole
SSB	Single Sided Board
DSB	Double Sided Board
VBC	Virtual Boundary Condition
3D, 2D	Three Dimensional, Two Dimensional
EQLAM	Equivalent Laminates
LSIs	Large Scale Integrated Circuits
ED	Electrodeposited
UD	Unidirectional
SEM	Scanning Electron Microscopes
SL	Static load (load caused by gravity of drop item in static state)

Chapter 1

Introduction

1.1 Background

Drop/impact induced damage has always been one of the most dominate failure modes experienced by electronic products [1]. The different forms of damage include housing fracture, liquid crystal display (LCD) cracking, solder joint breaking, battery cover pop-off [2], component failure, and chip interconnection disconnection [3][4]. Drop/impact performance testing is usually conducted to investigate and understand the detailed impact behavior and damage mechanics of products. Drop tests can be classified as impact-contact problems where the dynamic contact duration is short and all solution variables are time dependent which are determined only when both boundary conditions and initial conditions are specified [5].

Physical drop tests typically produce few useful results as they usually only identify whether the product did or did not survive a fall. There is no easy way to observe the fall at the moment of impact. In particular there is no easy way to observe the effects of such a crash from within devices to identify key weaknesses in product design. The difficulties worsen with the development of smaller and more complex components in electronic devices. Therefore, computer simulation

approach is of paramount importance in modern drop-impact analysis.

However, using computer simulation approach, the data obtained can be used to visualize the effects of impact. The simulation is capable of registering any mechanical information at any time step on analyzed entity. Nevertheless, the existing computer aided analysis is usually not satisfactory because the reliability and accuracy of the results depends largely on the modeling of the object and the expertise of analysts. In addition, a global fine mesh with the appropriate element type will generally produce a model with the best results. This will however, cost tremendous computational time and resources. The existence of small substructures in electronic devices make the adoption of a global fine mesh seems inevitable. The application of efficiency methodologies in numerical simulation can solve this problem. An efficiency methodology is expected to speed up simulation process, reduce development time, stabilize repeated simulation results and make optimization flexible.

The present research work proposes efficiency methodologies to realize high-efficient drop-impact analysis for packaged electronic products with tiny and complex components. In the following, some surveys for electronic products experienced drop-impact will be reviewed, the reasonable usage of physical tests and numerical analysis will be discussed, and previous work on efficiency analysis will also be surveyed.

1.1.1 Role of Drop-Impact Analysis in Electronic Products Design

Electronic products are becoming lighter in weight and smaller in size as well as of increasing functionality nowadays. In order to meet the rapid increase of competitiveness among worldwide companies, consumer electronic and electrical product manufacturers need not only improve the aesthetic appearance of their products, but also consider reliability, robustness as well as shock resistance. Thus,

mechanical reliability of electronic products is becoming an essential factor. This situation has driven some leading manufacturers to consider and investigate the response of their products subjected to impacts, shocks and vibrations [6][7] caused by drops or free falls onto ground.

To determine the reliability of newly designed products, most manufactures would perform drop-impact analysis on prototypes to examine their robustness [8]. This work also can provide some useful advices for optimization design. In drop-impact analysis, two important issues are normally concerned: one is *dynamic response of whole products*; another one is *dynamic response of sensitive components attached in electronic products*.

For *dynamic response analysis of whole electronic products*, designers must know what level of mechanical shock causes damage. Damage may be cosmetic, functional, or both. In any case, it represents an unacceptable change in products. Mechanical shock fragility is normally expressed by impact force. For example, the impact force sometimes is presented as G-factor of whole product, where G is the dimensionless ratio between specific acceleration and acceleration due to gravity. It is an approximate fragility evaluation of products such as a table provided in Reference [9]: various typical products are classified by different sensitive level with different fragilities, which is described by the maximum G-factor that products can suffer.

Beside the fragility evaluation of whole, *impact response of detailed components in electronic products* is also important. In micro-electro-mechanical systems (MEMS), drop-impact analysis for electronic packaging is highly considered. When an electronic product undergoes drop impact, it can experience large forces that can exceed the design limits of products. These forces are transmitted to internal components such as integrated circuit (IC) packages/printed circuit boards (PCB). The reliability performance of IC packages and vibration of PCBs during drop impact has become a great concern, due to increasing demand and popularity of handheld or portable telecommunication devices such as mobile phone and PDA.

The mechanical shock resulted from mishandling during transportation or customer usage may cause solder joint failure, which leads to malfunction of product. When an electronic product is dropped onto ground, the impact force is transmitted to PCB, solder joints and packages. Some works on drop test simulation [10][11][12][13][14][15] and experiments [16][17] were reported over past few years. For examples, Tee et al. [18][19][20] established impact life prediction models for TFBGA (Thin-profile Fine-pitch BGA) and QFN (Quad Flat No-lead) packages with good correlation between simulation and drop testing. For microelectronic packaging components and devices, dynamic responses of PCB, such as input acceleration, output acceleration, PCB deflection and PCB strain, are becoming important [21]. The functionality and operation of every electronic device depends on PCBs, the warpage and deflections of boards can cause mounted electronic components to dismount or failure of component lead wire, especially in drop impact conditions [4].

1.1.2 Need for Numerical Analysis in Drop-Impact Analysis

At component level, the necessary strength of a component is defined in an empirical manner without scientific and quantitative analysis. Sometimes, electronic functions are hampered by lack of a suitable mechanical or structural design. To obtain the detailed understanding of impact behavior and damage mechanism of electronic product, there are two approaches: physical test and numerical simulation.

In the traditional method of product design, prototypes are fabricated to test experimentally for drop, vibration and shock. Physical drop test can detect failure mechanism by collecting impact acceleration, contact force and strain/stress data in both package and component level tests. These results in high expenditure of manpower, time and cost. Because of the iterative process of product design,

physical prototyping and testing must be repeated until final product is obtained. Usually, the product durability upon drop impact is determined by designer's experience in design-prototype-test-redesign paradigm that is time-consuming and expensive. In addition, in most experiments, the physics of failures and their causes are always difficult to identify due to spatial restriction for fixing equipment and sensors.

With more than fifty years of development for finite element methods (FEM), and with the help of high performance computers, computer simulation based on FEM has been widely applied in industry [22]. Computer simulation based on FEM can be carried out in every design stage in a timely, efficient manner. This is a successful combination of mathematical analysis with computer engineering.

Comparatively, reliable computer simulation allows designers to determine complete mechanical information, compared with only segmental messages by sensors in a test, to predict potential failures. This will not only save both time and cost in terms of cutting down numbers of prototype to build and test, but will also help to shorten product development cycle [23][24][25]. Having a computational model of a product allows quick changes made to design and drop tests simulated to examine any further improvements in the robustness of products. Once the design is decided upon, the prototype can be fabricated to determine the actual fragility of products. At the same time, simulation provides explicit scientific judgment for evaluation of alternative designs.

1.1.3 Methodologies to Improve the Efficiency of Numerical Analysis

Using computer simulation approach through FEM, the data obtained can be used to visualize the effects of impact. The simulation is capable of registering any mechanical information at any time step on analyzed entity. Nevertheless, the reliability and accuracy of the results depends largely on the modeling of object

and expertise of analysts. Generally, a global fine mesh with appropriate element type will produce a model with good results. This will, however, result in a mesh with an excessive number of degrees of freedom and costly computational resources. The existence of small substructures in electronic devices makes the adoption of a global fine mesh seems inevitable. Some efficiency methodologies have been tried to solve this problem, especially in multi-structure analysis, as it is possible to divide the whole multi-structure model simulation into pre-process and post-process.

In order to realize efficiency simulation, the number of elements and degrees of freedom of elements are to be decreased. Two ways are considered in this work: optimization of simulation process and incorporation of non-simulation analysis into simulation.

Among all the optimization methodologies of simulation process, global/local method is one of the most popular methods [26]. The biggest contribution of global local method is that, with successful combination of global and local models and through application of different meshes (ie; fine and coarse meshes) in different stages in whole analysis process, computational time and cost is reduced yet retaining the integrity and accuracy of model. Global-local FEM basically uses global displacement solutions as boundary conditions for local area of interest. At the same time, a refine mesh will be used for local area to achieve more accurate results such as detailed stress states for specific structural regions. Using this method, drop/impact simulation on tiny structures can be carried out more efficiently without compromising any accuracy. In drop-impact analysis, the efficiency methodology in FEM process have the same aim to save time and money in product design, and help to shorten product development cycle. Also, the basic concept in the method of multi-structure analysis is to avoid calculation of large-scale model or complex structure that has too many elements which require too much CPU space. The quick numerical study becomes more and more important in modern industry.

However, sometimes, the application of simulation process optimization is

insufficient. The error from coarse meshed global model is not small enough. Bossak and Kaczowski [27] successfully used global/local method to finish drop-impact simulation of light aircraft with large composite components. This is a breakthrough in the simulation analysis for this large-scale model, since this work seems impossible to be finished with fully fine meshed model till now. However, through the comparison with experimental results, some error of simulation even reached 62% [27]. It is found that the accuracy and efficiency could be further improved if some non-simulation methods are hybridized into simulation, such as analytical calculation for vibration or mechanics analysis for composite components. Because non-simulation methods can be used to obtain some effective boundary conditions or material properties that can be used in local simulation. And there is no computational resource spent by non-simulation methods or error from element meshing.

1.2 Objectives and Contributions

The focus of this PhD work is mainly on the impact response of multilayer printed circuit boards and their interaction with nonlinear cushion material. With the help of platform provided by virtual buffer model, these two studies will contribute to realize efficiency analysis for packaged electronic products. To achieve this aim, the following objectives were identified, and some contributions were made in the course to realize these objectives.

1. To identify and classify the characterizations of experimental, numerical, and analytical methods in drop impact analysis and hence find the possibility to apply these methods in high efficient design.
2. To show the adoption of high efficient methodologies for the analysis of multi-scale models and hence identify the availability of efficiency methodologies for quick analysis of packaged product model, which is one kind of multi-scale models.

3. To establish an initial virtual buffer model with only numerical analysis inside. It is proposed to provide efficiency analysis for packaged electronic products.
4. To develop initial virtual buffer model to full-scale virtual buffer model with hybrid analysis methodologies. Full-scale virtual buffer model has higher efficiency than initial virtual buffer model, and it can provide a platform for the application of nonlinear local linearization methods for cushion buffers and hybrid modeling methods for multilayer circuit boards.
5. To evaluate the efficiency of analytical methods for nonlinear analysis. It is proposed to show an avenue to quick analysis for nonlinear drop impact systems.
6. To innovate a local linearization method for nonlinear analysis. It is an analytical method. This method can quickly predict the drop impact response of nonlinear cushion system with analytical method quantitatively, and provide stimulation to multilayer circuit board model that is a sub-model in virtual buffer model.
7. To identify the characterizations of mechanics/FEM modeling methodologies for multilayer multimaterial composites that is used as basic material of multilayer circuit boards and hence to figure out the advantages of each method.
8. To propose a hybrid modeling method which works with the combination of 3D mechanics unit-cell model and 2D multilayer FEM shell elements. It is aimed to performed static and dynamic analysis of multilayer circuit boards on both micro-structure and board levels.
9. To apply full-scale virtual buffer model with the combination of local linearization method and hybrid modeling method to the analysis of a general prototype for packaged electronic products and hence shows its efficiency and practicability.

As a conclusion, the innovated virtual buffer model, local linearization method and hybrid modeling method are three major contributions of this PhD work. The application of each of them can realize high efficient drop-impact analysis for packaged electronic products.

1.3 Thesis Organization

The present work is organized into seven chapters:

Chapter 1 (current chapter) introduces the practical background of drop-impact analysis in electronic products design, the effect of numerical methods, and efficiency methodologies. It seeks to define the motivation of present research as the need to improve the efficiency of product design.

Chapter 2 reviews previous related research for experimental, numerical, and analytical methods in drop impact analysis. Focusing on multi-scale simulation, efficiency methodologies, nonlinear analysis, and modeling techniques for PCBs, different methods used in these fields were introduced and compared. Their relationships to the drop impact analysis of packaged electronic products were also introduced.

In *Chapter 3*, an initial virtual buffer model is proposed to realize high-efficient drop-impact analysis for a packaged TV product. A packaged TV model is identified as multi-scale model, and virtual buffer model is established for this multi-scale model. Full-scale virtual buffer is provided at the end of this chapter, the achievement of this full-scale model becomes the backbone of present work: quick extrication of boundary condition applied on the objective models and efficiency modeling objective models, which will be introduced at the end of Chapter 3.

In addition, the achievement of full-scale virtual buffer model can focus on the shock response of multilayer printed circuit boards and their interaction with

nonlinear cushion material, which is the major aim of this PhD work as mentioned in Abstract. Therefore, the full-scale virtual buffer model not only improves the design efficiency but also provides a platform for the shock response of multilayer printed circuit boards in nonlinear drop-impact cushion systems. It will be finished in Chapters 4 to 6.

In **Chapter 4** a local linearization method is innovated, it is aimed to realize quick quantitative dynamic analysis for nonlinear vibration systems. In this work, it is used to predict the dynamic response of nonlinear cushion system in drop-impact condition quickly. The calculated dynamic response will be used as boundary condition in virtual buffer model. Because it is an analytical method, quantitative results are available and quickly achieved.

In **Chapter 5** presents a hybrid modeling method for multilayer circuit boards. In full-scale virtual buffer model, multilayer circuit boards are selected as objective model, and the boundary condition extracted from Chapter 4 will be applied on multilayer circuit board models. The proposed model can analyze multi-layer PCBs on both micro-structure and board levels with less computation.

In **Chapter 6**, a general prototype of packaged electronic products is presented. The local linearization method and hybrid modeling methods are used into full-scale virtual buffer model for drop-impact analysis of this prototype.

Chapter 7 summarizes key contributions and conclusions; that is, a development of virtual buffer model for efficient drop-impact analysis of packaged electronic products.

Figure 1.1 summaries the contents and structure of this thesis.

Figure 1.1: Schematic structure of PhD thesis

Chapter 2

Literature Reviews

This chapter will review some basic research in drop-impact analysis and simplified models generally used in studies of drop items, provide a preliminary description of efficiently methodologies for large-scale simulation, and review past research on nonlinear cushion model and modeling techniques of printed circuit boards.

2.1 Drop-Impact Tests for Electronic Products

Physical test and simulation are two general methods in drop-impact analysis for electronic products. Physical test is a traditional way with high trustworthiness, while simulation is quite useful for the collection of detailed information and optimization design. Nowadays, simulation is more and more preferred, one of the main reasons is that electronic components become smaller and lighter, but the dynamic response of tiny components is not easy measured in test. However, physical drop tests are still often used in industry, since it is fast and direct for some simple tests when equipment and prototype are both well prepared. For research and design, although it has some limitations, its advantage of high trustworthiness can be utilized to verify theoretical models.

First, some general drop tests for full-scale electronic products are reviewed.

Next, because the focus of this work is mainly on the shock response of multilayer printed circuit boards applied in electronic products and their interaction with nonlinear cushion material, drop tests for printed circuit boards and cushion buffers will be resurveyed individually.

2.1.1 Drop Tests of Full-Scale Electronic Products

In order to meet the requirements from industry, some authors did drop-impact analysis for mature electronic products. In Low et al. [26], Y. Wang [28] and Y. Y. Wang et al.'s [29] work, a 29-inch TV was dropped from different sides (six-face drop orientations) and different heights to collect numerous test results. Enough test results successfully proved the reliability of simulation model. In test, a precision Lansmont drop tester with 79 kg capacity was utilized. The experimental set-up is shown in Figure 2.1. There were two parts: (1) Test Partner 3, it is a powerful combination of computer software and hardware specifically tailored to capture and analyze transient hock events; (2) Drop tester, it can make a package drop from a given height. In this drop test, acceleration sensor was attached at the center of TV glass tube. Although TV was dropped by many times, only the acceleration at the center of TV could be detected. The stress and strain conditions on TV sample were not be measured in tests, the damage at sensitive part couldn't be illustrated. This is the limitation in drop tests. Simulation becomes a necessary tool in reliability design.

In order to collect more detailed information in drop tests, compared to above drop tests, Pan et al. [30] used more test equipment in drop test for LCD monitor: (1) more accelerometers were applied as shown in Figure 2.2, peak value and duration of structural shock acceleration on main parts, such as LCD panel and housing frame, are acquired; (2) strain gauges were used to require dynamic strain shown in Figure 2.2, the transient deformation on structure was measured; (3) Two CCD cameras were applied to ensure the set of measurement data resulting from a correct orientation shown in Figure 2.3, all one-corner/three-edge/six-face drop

Figure 2.1: Experimental set-up for general drop test [28]

orientations were taken. With these three improvements, the drop tests of LCD are more comprehensive. However, as mentioned in their work, the stress of some tiny part, such as fastening points, still need the analysis from simulation. Measurement method is still a bottleneck in drop tests experienced transient impact damage. At the same time, the high-cost in damage test is another issue for the application of physical drop tests.

Figure 2.2: Sensor layout on TFT-LCD monitor, where A denotes accelerometer, and S denotes strain gauge. (a) on panel; (b) on housing (B indicates fastening point of LCD panel to housing) [30]

In the mentioned work above, full-scale electronic products were dropped. All of the study focus on the analysis of damage conditions on cover, such as stress/strain contour on the housing of LCD. Actually, engineers most concern

Figure 2.3: Experimental setup of drop laboratory [30]

operational drop failure [4]. In next section, the work about drop tests for printed circuit boards will be reviewed.

2.1.2 Drop Tests for Printed Circuit Boards

Based on throughout survey of previous work, Ong et al. [31] systematically concluded that there were two categories in mechanical drop tests for printed circuit boards: board-level and product-level. In product-level tests [32], entire product is gripped at a specific orientation, allowed to fall under gravity and released from gripper to strike a desired surface. The product is free to rebound and experience subsequent impacts. This mimics the dynamic response of an actual drop impact experienced by products. However, it is complicated because it depends on many factors such as drop height, drop orientation, components layout on PCB, product design, and etc. In board-level tests, a test board with electronic components mounted on it is fixed to a drop platen; this enables the assembly to be subjected to controlled drop impact onto a rigid surface. Board level drop test is convenient to characterize solder joint performance, because it is more controllable than product level drop test. The purpose of board-level tests is to simulate using a

test board, the same mechanical effects that are observed with respect to a PCB inside an electronic product during a drop impact. Most of standardized test protocols including JEDEC standard for handheld electronic products [33][34][35][36] are essentially based on board level tests.

In order to understand and analyze differences in mechanical responses of PCBs undergoing product-level and board-level shock tests, the test subjects for product-level and board-level tests were respectively a Nokia Model 3210 mobile phone and its PCB. As shown in Figure 2.4, product-level drop testing was effected using a proprietary orientation control drop tester that has a gripper attached to a sliding block which is elevated to the desired height before being released to undergo free fall. The gripper enables test sample to be held at any desired impact orientation. Just prior to impact against the target surface of a load cell, the test sample is released from grippers so that the ensuing impact and subsequent motion are unrestrained. For board-level tests, the PCB extracted from mobile phone (as shown in Figure 2.4) was mounted onto a fixture via the same locations where it was originally secured to the phone casing as shown in Figure 2.5. Different fixtures were designed to facilitate impact at various orientations. The fixture together with PCB is then secured rigidly onto a drop platen in a drop tower. This drop assembly is then raised to the desired drop height and released.

Figure 2.4: Drop tester conducting drop impact test of a mobile phone [37]

In their research, several concerns [38] about how accurate board-level testing is in duplicating the state of loading experienced by electronic components were

Figure 2.5: Schematic diagrams showing impact orientations for both test methods [31]

carried out- namely: (i) Since a test board is mounted firmly onto drop platen, its orientation at impact is fixed and it is unable to execute its natural dynamics during impact, such as clattering or multiple impacts. (ii) Interaction between PCB and other internal components are not taken into consideration. (iii) It is difficult to subject a test board to direct impact against its vertices and edges because it is usually mounted onto drop platen using columns supports at its corners. (iv) Impact occurs between metallic drop platen and target surface rather than between plastic product casing and impact surface. (v) A test board is usually mounted onto drop platen using discrete columns but in complete products, the PCB is supported internally within product casing.

The experimental study shows that the mechanical response of a PCB for product-level and board-level testing differs in terms of impact contact sequences, as well as accelerations and strains induced. Through comprehensive discussion, some significative conclusions were provided. In general, current board-level test standards do provide a common basis to evaluate the impact tolerance of electronic components, however the conditions during drop impact loading might not be reproduced comprehensively using a test board when the product casing is not rigid enough. Therefore, board-level tests are more suitable for academic study since it is easier to control and not so much noise, unexpected condition, will happen in test. Product-level is also necessary in final prototype test, it is closer to real cases, all the drop test examples in Section 2.1.1 are product-level tests.

2.1.3 Drop Tests for Cushion Buffers

Cushion buffers are popularly applied in industry to protect packaged electronic products out of impact damage. Cushion buffers are quite practical since it is cheap, light and high energy-absorbed. The cushion effect in drop tests is also comprehensive studied in test [9]. In drop-impact process, the dynamic responses in rebound time are mostly concerned, such as maximum acceleration, etc. The maximum acceleration is generally evaluated by G-factors, which is defined as the

Table 2.1: Typical G -factors for various product types [9]

G -factor	Sensitivity	Product Type
≤ 20	Extremely sensitive	Precision instruments with fine bearings
21 - 40	Very sensitive	Electro-mechanical instruments
41 - 60	Sensitive	Electro-mechanical equipment
61 - 84	Fairly sensitive	TV receivers, optical equipment
85 - 110	Moderately sensitive	Domestic equipment, e.g. glassware
> 110	Insensitive	Industrial machines

ratio of the deceleration of the object after impact to acceleration of gravity during free fall drop. Table 2.1 shows typical G -factors for various products [9]. Thus, the prediction of G -factors under different cushion conditions is crucial in packaging design. It can evaluate the cushion effect and help to select suitable buffer designs for various products. The most typical study to predict G -factors is the derivation of cushion curve.

The ability of a cushion to cushion shock is usually presented as a cushion curve. A cushion curve shows how a particular cushioning material of a given thickness behaves at different impact conditions. Cushion curves are generated by dropping a series of known weights onto a cushion sample from a specified height and measuring the amount of shock experienced by the weights as they impact cushion. Cushion curves are available for a variety of drop heights, thickness and usually for single and multiple impacts. ASTM D1596 method specifies how the test method must be performed [39]. The cushion curves for EPS 20 provide by BASF [40] illustrate general shape as shown in Figure 2.6.

As per ASTM D1596, a procedure developed by American Society for Testing and Materials (ASTM), cushion curves are developed by dropping predetermined weights onto a block of cushion of known thickness and area from a given drop height as shown in Figure 2.7. This procedure recommends that the test specimens be right square prisms or other configuration as desired with maximum length and width dimensions to be less than the corresponding drop platen dimension. Manufacturers of cushion materials use this procedure and publish cushion curves; the transmissibility and stress-strain curves of material could also represent

Figure 2.6: Cushion curves for foams made of EPS 20 [40]

the vibration transmission and compression properties of cushion material respectively. A cushion package design process begins with determining the thickness of cushion material necessary to achieve desired shock protection. Cushion curves are also used to determine the optimal area and thickness of cushion material.

Figure 2.7: Cushion tester [41]

2.2 Drop-Impact Simulation for Electronic Products

In order to ensure the products quality and reliability, the traditional way is to carry out physical tests using prototype. However, it is difficult to catch drop phenomenon, especially inside response, as the impact is of quite short duration. When damage/failure occurs, engineers can only analyze the causes based on the final failure and hence to improve design. The general ways to modify design are to increase the strength of products and/or to improve the shock absorption ability of packaging. However such design entirely depends on the experience of engineers and undergoes lots of trial and error testing, which causes a delay of product development and uncertain quality of products. As rapid development of state-of-art computer hardware and software, simulation-based virtual test can be performed for electronic devices, which can be used to replace expensive and time-consuming physical tests. More importantly, virtual test can be carried out during design stage and be used to investigate insight failure mechanism. Finite element analysis (FEA) has been applied to simulate the drop tests of various products [42][43], packaging material are also considered in drop test simulations [44]. Most of commercial software can be applied to simulate drop impact responses.

In this section, several drop impact simulation projects will be implemented to show their contribution to industry. Basic techniques and procedure of simulation will be first introduced. Next, two basic algorithms, implicit and explicit integrations, are compared and discussed. Explicit algorithm is regarded to be more suitable for drop-impact simulation. Finally, an initial work about drop-impact simulation of PCB with foam packaging is reviewed.

2.2.1 Basic Techniques and Procedure of Drop-Impact Simulation for Electronic Devices

To replace physical test, it is required that drop test simulation supplies reliable results and sufficient information. Thus, it is a vital step to set up simulation model properly, which includes geometry simplification, FE model, material model and boundary conditions, etc. Another challenging requirement is to detect the failure inside small components during drop test simulation in product level. The difficulty is to obtain drop-induced response in components, which needs fine mesh for tiny structures. Fine mesh may cause simulation expensive and even make simulation impossible.

In Wang et al.'s work [29], FEM was applied to the numerical simulation of drop test. Detail discussion was presented to imply method properly. Emphasis was on the methodology of analyzing the reliability of electronic devices under drop impact. Modeling and simulation method for such kind of complex structure was discussed. Some important issues, such as control of simulation, material model, etc. were addressed. Numerical examples were presented to demonstrate the application of FEM on virtual product development. Firstly, the drop test simulation for a TV set including packaging material was considered as shown in Figure 2.8. In this application, the packaging material took a crucial function during drop impact. As packaging material (foam) is much softer than TV cabinet, the contacts between foam and other structures, TV cabinet and ground, becomes quite complicated. Therefore, drop test simulation for TV set with packaging material is a challenge and cannot be analyzed by general drop test modules. Another crucial part is the connection structure between heavy tube and cabinet, which was also discussed. Secondly, the reliability and robustness under consecutive drops was illustrated for an electronic device with aluminum housing. The procedure was described to simulate the damage accumulation in the part.

In Wang et al.'s work [29], the basic procedures of drop impact simulation

Figure 2.8: Structure and finite element model of TV set with packaging material [29]

were summarized as: (1) understanding problem and requirements, (2) building up geometry model and FE model, (3) applying proper loading and boundary conditions, (4) selecting suitable material constitutive model according to material type and working condition, inputting proper material properties, (5) selecting control parameters and run simulation, and (6) analyzing results.

Meanwhile, FE model will affect accuracy and CPU time, thus some basic rules for FE model construction were summarized: (1) using shell elements instead of solid elements whenever possible, (2) avoiding small elements and select proper mesh density, (3) simplifying interconnections as possible, such as using nodal constrain to define glue interface or catcher fixation instead of contact constrain.

For the time control of simulation, the CPU time needed for a drop simulation is concerned with the duration of simulation, element number, time step of explicit integration, certainly computer speed which affects CPU time needed per finite element and per time step. The decrement of mesh density will result in a reduction on CPU time. However, it has been mentioned that time step is controlled by the smallest element, in some case, structural and accuracy requirement

make it difficult to avoid using small elements. Therefore other methods must be sought to control time step. There are three methods to control time step: (1) Subcycling: In some cases, subcycling can lead to saving in CPU time, but it is not always true. Subcycling is most beneficial in finite element meshes where finite element sizes change rather smoothly and regularly from small to large element. If, on the contrary, the element size distribution is quite irregular, i.e., when small and large elements are distributed rather randomly the gain of scheme on CPU time reduction may be reduced. In some extreme case, this method even leads to an increase in CPU time. (2) Minimum time step option: Users can control time step by specifying minimum time step $t_{\min}$. If the time step of an element is found to be smaller than $t_{\min}$, the program adjusts internal element parameters such that its stable time step equals $t_{\min}$. Generally, the minimum time step option is used as a safeguard against excessive reduction of stable solution time step. Therefore the minimum time step option is less used to reduce CPU time, but specified as a stable minimum value of time step. (3) Mass scaling: mass scaling option is useful when seeking a quasi-static solution to a structural problem. It can also be used just for raising initial time step for a dynamic run where the sum of adjusted masses should remain small. In the case of element sizes varying or element size distribution is quite irregular, i.e., when small and large elements are distributed rather randomly. This option is particularly useful in saving CPU time.

Finally, some key points for efficient modeling and simulation of drop test were summarized in Wang et al.'s work [29]: (1) Most of electronic devices are designed with complex structure, simplification on geometry and interconnections of components are quite important. The simplification should be made based on simulation requirements. (2) Finite element mesh should be controlled properly. Poor element shape and small elements should be avoided as possible, which is a benefit for accuracy and simulation cost. (3) CPU time for drop test simulation is still relative-longer. For the cases discussed above which are run in SGI Origin2000 system, CPU time for TV drop test is about 70 h for the duration of 25 ms, about 20 h for the assembly device with duration of 10 ms, and 3 h for Al casing with 5ms

duration. The duration, element size, material nonlinearly, etc. are all affected by CPU time. Thus control of simulation is an important issue. (4) Contact definition for the drop test of electronic devices is one of the critical steps. Special attention should be paid to the contact of two materials with quite different properties, such as the casing with packaging material and resistant rubber. (5) Defining material properties is another important issue. Proper constitutive model should be selected. (6) Analyzing simulation results carefully to develop reliable products. The results to identify reliability.

2.2.2 Time Integration Scheme of FEM Simulation for Drop-Impact Analysis

In the transient analysis by FEM, there are two basic algorithms for time integration: implicit and explicit formulation [1][45]. Understanding the nature of these algorithms will definitely assist analysts to adopt correct algorithm for a particular problem.

In the particular case of a drop/impact analysis, two algorithms are expressed in governing equations and comparisons are made to determine appropriate formulation. The governing equation of a dynamic system can be expressed as [1]

$$M\ddot{x} + c\dot{x} + Kx = F \quad (2.1)$$

where x is nodal displacement vector and the dots denote its differentiation with respect to time. Also, mass, damping and stiffness matrices are denoted by M , c and K respectively. Equation (2.1) represents the dynamic equilibrium of a system.

In a transient analysis, a dynamic process is divided into many time steps. The time step integration is an algorithm to calculate an unknown state at time

step $t + \Delta t$ from a known state at moment t . For explicit formulation, a central difference time integration method is used [46]. Equation (2.1) can be written as

$$M\ddot{x}_t + c\dot{x}_t + Kx_t = F_t \quad (2.2)$$

when the equilibrium state is considered at moment t . The explicit integration operator is conditionally stable, so that the time increments must satisfy

$$\Delta t \leq \frac{2}{\lambda_{\max}} \quad (2.3)$$

where $\lambda_{\max}$ is element maximum eigenvalue [47]. In order to achieve stability, Equation (2.3) can be expressed as

$$\Delta t = \min \left(\frac{L_e}{c_d} \right) \quad (2.4)$$

where L_e is characteristic element dimension and c_d is dilation wave velocity of material. The nature of explicit formulation limits it to the analysis of short transient problems. In contrast, implicit formulation considers its state of equilibrium at time $t + \Delta t$ and the governing equation can be written as

$$M\ddot{x}_{t+\Delta t} + c\dot{x}_{t+\Delta t} + Kx_{t+\Delta t} = F_{t+\Delta t} \quad (2.5)$$

For linear problems, the solution is unconditionally stable when the stiffness matrix K is linear, and large time steps can be taken. Whereas for a case where nonlinearity is of great concern in analysis, the small iterative steps are required to achieve convergence.

Comparison between two formulations revealed that explicit formulation is better suited to solve drop-impact analysis. This is due to three main reasons: (1) impact occurs and the consequential response happens within a short transient

period; (2) contact will inevitably occur during impact and this will undoubtedly introduce nonlinearity in analysis. Hence, implicit method is deemed undesirable; (3) two physical factors limit time step in impact analysis. If a large time step is adopted, too much strain will be introduced in one time step and this will cause divergence in such large deformation analysis. The second restriction is due to contact surface algorithm. Similarly, a large time step will cause a large contact force at contact interface, which will result in local distortion and analysis failure. In view of the two stated reasons, the advantage of larger time step allowable in implicit formulation disappears.

Sun et al. [47] investigated linear dynamic problems using two formulations in a fast and slow case. Results obtained were as predicted using mathematical formulation. For fast case, the advantages of explicit method are apparent within desirable tolerance. The cost is also much less than that of implicit method. For slow case, CPU time for explicit method is quite long as it took many time increments to complete analysis. In addition, as time step in explicit method is controlled by the smallest element, thus, the unimportant small parts and features should be removed or simplified to avoid small elements.

2.2.3 Drop Impact Simulation of PCB with Foam Packaging by Using Explicit Software

Because the focus of this work is mainly on the shock response of multilayer printed circuit boards applied in electronic products and their interaction with nonlinear cushion material. An initial simulation work focusing on the interactive response between printed circuit boards and cushion buffers is independently reviewed in this section. This analysis involved drop test of a circuit board and foam assemblage onto a flat, rigid floor from a virtual drop height of 1 meter [48]. The objective of this simulation was to assist the design of packaging material (a crushable foam), which was intended to protect circuit board from damage in such an

event. Only results on the accelerations experienced by the circuit board were being post processed.

This simulation was performed by ABAQUS, INC using ABAQUS/Explicit software. Material properties such as high volumetric compressibility, inelasticity and strain rate dependent effects have been implemented for use with foamed, lightweight and energy absorbing materials. The packaging was modeled with 1200 brick elements. The circuit board was modeled as a rigid body with appropriate mass and rotary inertial. The model is shown in Figure 2.9. The accelerations of circuit board in X , Y and Z directions were obtained and plotted on graph in [48].

Figure 2.9: Finite element model of PCB and packaging material [48]

2.3 Efficiency of Methodologies for Large-Scale Simulation

Traditionally, the modeling and meshing of structure is straightforward. The whole system is treated as a single entity. The entire model can either be coarsely meshed or otherwise. The accuracy of model generally improves with a finer mesh of model.

On the other hand, CPU cost and time increases with finer mesh. Therefore, the difficulty is to obtain drop-induced response in components, which needs fine mesh for tiny structures. The fine mesh may cause simulation expensive and even make simulation impossible. In addition, the reliability and accuracy of results depends largely on the modeling of object and the expertise of analysts. This makes the repeated simulation results unstable. And this vibration causes low credit, and too much dependence on verification tests. All of these restrict the practical application of simulation methods. Therefore, high-efficient methodologies are proposed to solve these issues. Less computational time without the lost of accuracy, more stable results in the repeatable simulation, and easy operation are the objectives of efficiency techniques.

In this section, some efficiency methodologies and their theory background will be reviewed. It will make us understand the effect of high-efficiency methodologies. Through this review, it is necessary to know the strategy on the control of mesh size is an efficiency avenue, in addition, hybrid of other non-simulation methods sometimes can further achieve some effect beyond the efficiency of mesh control methods. This is one of the important objectives in this PhD project.

2.3.1 Mesh Control Methods

The basic idea of mesh control methods is fine mesh sensitive parts with high concern, tiny complex structure, or high error estimation. In this, three popularly applied mesh control methods are introduced: global-local method, super element method and adaptive meshing method.

Global-Local (GL) Method

As one of the representative mesh control methods, global-local method is commonly used in multi-scale simulation. Many engineering applications in structural design involve situations where the detailed response of a segment of a large struc-

ture is of interest to designers. In such circumstances, creation of a fine mesh is necessary to capture detailed stress and strain behavior at local area of interest. However, the creation of fine mesh greatly increases the number of degree of freedom and hence sacrificing much CPU computation and cost. To solve such problems with minimal computing resources, the use of special FEM solution techniques commonly known as global-local methods are often adopted.

Global-local FEM is defined as a procedure to determine local, detailed dynamic responses for specific structural areas using the information obtained from an independent global analysis [49]. Generally, there are three main steps in typical global-local analysis procedure shown in Figure 2.10: (1) *Global analysis*: A global analysis is performed on FE model with a coarse mesh to obtain the physical response of entire structure. For the nodes along the interface of global and local models, their results of displacements against time domain are extracted and stored as arrays for use in subsequent steps. (2) *Interpolation of the displacements from global solution*: Often, the node-to-node connectivity between the nodes along the common interface of two models may not be complete due to the necessity for a more refined mesh local model. In such cases, interpolation of the displacement results of unconnected nodes needs to be performed. (3) *Local analysis*: In the area of interest where accurate dynamic responses are expected, a refined mesh is used to perform transient analysis using FEM. With interpolated displacements applied as boundary conditions and the input of necessary initial conditions and loads, the dynamic problem is solved within the local area of interest.

In fact, sometimes global and local model can actually be solved in two steps [50]: first-*global*, when calculations are performed using the model of whole structure, and second-*local*, when calculations are performed using model of particular structure's part. GL method is not connected with any computer aided design system.

In Low et al. [26] and Bossak and Kaczkowski's [27] work, the theory of global-local has been discussed in detail. The discussion focus on the way out to

Figure 2.10: Flow chart of global-local finite element method [49]

reduce computation time in simulation. Most FE models developed in industry for drop simulation of electronic products are generally quite large, at the same time, some detailed and tiny components cannot be ignored. These are two main factors that cause a long computation process. It is therefore necessary to reduce actual computation time without ignoring the analysis of certain tiny elements. It is necessary to explore this possibility by understanding more about the factors affecting computation time.

First in simulation, the time step of model is automatically defined by [51],

$$\Delta t_i = Fc \times \min\left(\frac{L}{\sqrt{E/\rho}}\right) \quad (2.6)$$

Note that time step size Δt_i is determined by taking minimum value over all elements, and it is to ensure solution stability for elements. Also, Fc in Equation (2.6) is a scale factor for stability reasons and L is the characteristic length of each element.

Next, one would set a termination time for whole drop-impact analysis. The *input termination time* can be expressed as

$$T_{input_term} = \sum_{i=1}^{q-1} \Delta t_i + \delta, \quad \delta \leq \Delta t_q \quad (2.7)$$

in which q is the total number of time steps over whole simulation. Note that Δt_i is the time step used in simulation computational time, T_{actual} .

However, the actual time taken to complete analysis process will be much more longer than the time prescribed, T_{input_term} . Let us assume that actual computation time is given by

$$T_{actual} = \sum_{j=1}^q \Delta t_j \quad (2.8)$$

where q is again the total number of time steps, while Δt_j is the actual calculation time taken for respective time step size Δt_i , which has been defined by Equation (2.6), and it depends on the complexity of model (and the computer system as well). It is seen that the step number q in Equation (2.7) will decrease as Δt_i is increased, since the value of T_{input_term} is constant and fixed by users. Furthermore, in Equation (2.8), the reduction of step number can cause the decrease of actual calculation time T_{actual} . Therefore, two approaches are suggested to reduce the actual computation time, T_{actual} :

- i. *Methods for reducing number of operations:* One of such methods is deleting some tiny parts, and it should not affect too much the accuracy of result [27]. This simplification will reduce actual computational time for every respective time step Δt_j . Another of such methods, most frequently used during explicit integrations, allows increasing stable time steps through elimination of tiny elements. This will in turn increase the time step size (Δt_i) through a reasonable coarser mesh, which implies a larger L and smaller q . As simulations had shown, this approach is currently the most effective one, especially for complicated structures with small mesh elements, because time step size Δt_i can be increased significantly through increasing the characteristic length L in Equation (2.6). Furthermore, owing to the reducing number of meshed elements, the total number of time steps q is smaller, and it thus contributes to the reduction of *actual* computation time, T_{actual} .

- ii. *Parallel computing methods*: These methods utilize a wide range of different approaches to parallel computations. One of them is GL method [27]. global analysis was utilized to find the behavior of whole system during impact, while local analysis was used for finding behavior of detailed components. The proposed scheme can reduce the *actual* computational time for every respective time step, Δt_j .

A case considered in reference [49] is now used to illustrate the efficiency of GL method. As shown in Figure 2.11, a plate with convex tip on the right side undergoes free drop. The following material constants were assumed: Young's modulus of 210 GPa, density of 7800 kg/m³, Poisson ratio of 0.3. The initial velocity is set for 4 m/s and the termination time is taken as 0.24 ms.

Figure 2.11: Models with different meshes for global-local analysis [26]

As depicted in Figure 2.11, entire plate is uniformly divided into three different meshes. Four node membrane elements (no bending or transverse shear effects) were used to mesh the plate. local model (c) was extracted from global model (b), and adopted the same fine mesh size as that of original model (a). The global model adopts coarse mesh, while the original and local models are formed by fine meshing. In local analysis, the extracted and interpolated displacements are both from global model.

The results of displacement of tip node 1 associated to the global, original and local models are presented in Figure 2.12. It is seen that the results from GL

Table 2.2: Comparison of different meshed models illustrated in Figure 2.11[26]

	<i>Original Model</i>	<i>Global Model</i>	<i>Local Model</i>
No. of element	512	128	80
Element area (mm ²)	5.0×5.0	5.0×5.0	2.5×2.5
Input termination time, T_{input_term} (ms)	0.24	0.24	0.24
Average time step, Δt_i (ms)	0.1248×10^{-3}	0.4913×10^{-3}	0.1245×10^{-3}
No. of time step, q	1924	489	1928
Average actual time step, Δt_j (ms)	4.0879	2.025	1.1838
Actual computation time, T_{actual} (s)	7.861	0.9893	2.282
Time saving*	58.39 %		

$$* \text{Time saving} = \frac{T_{actual_original} - (T_{actual_global} + T_{actual_local})}{T_{actual_original}} = \frac{7.861 - (0.9893 + 2.282)}{7.861}$$

model is reliable. The comparisons of T_{actual} , Δt_i , Δt_j , T_{input_term} , q for respective models are also given in Table 2.2. It is seen that GL method with models (b) and (c) can save about 58 % of computational time if compared to that of original model (a). Also illustrated in Table 2.2, the major contribution to computational time comes from actual computational time associated to various time steps, Δt_j .

Figure 2.12: Vertical displacement of node 1 for three different models [26]

Super Element Method

Global-local method comes from super element methods. Super element method is developed by Koko and Olson [52][53] to speed up simulation for complex structure with large number of elements. They developed special plate and beam elements called super finite element for the design-analysis of air blast loaded stiffened plate structures. The displacement fields for these elements are represented by analytical as well as polynomial functions in both in-plane directions, and have been specially designed so that only one plate element per bay and one beam element per span are needed to model the response. The resulting modeling is quite efficient in that both required input data and CPU cost are significantly reduced as compared to conventional finite element method.

In one of Koko and Olson's studies [54], a schematic representation of a typical orthogonally stiffened plate is shown in Figure 2.13. Each panel (e.g. ABCD) is treated as basic plate element and stiffeners (such as AB, BC) are treated as beam elements running along the edges of panels. The displacement fields for plate and beam elements have been carefully chosen to simulate all possible linear and non-linear deformation modes for the elements acting together as in stiffened plates or separately as simple beams or plates with all possible boundary conditions. The elements are termed "super finite elements" since only a single element is required to model basic response.

As mentioned above, global-local method comes from super element method. The difference between these two approaches is discussed by Bossak and Kaczkowski [27] their light aircraft crash landing simulation by the way of global/local analysis. Figure 2.14 is shown in their paper to illustrate this difference. As shown in this figure, the main difference between the two approaches is derived from the way the model is partitioned. In super element method, the whole model is divided into sub-models. The sub-models degrees of freedom are divided into masters (retained) and slaves (condensed). In the first step, the calculations are performed using only master degrees of freedom. Using the results of these calculations, dur-

Figure 2.13: Two-way stiffened plate [54]

ing the -next step the calculations for slave degrees of freedom are performed. In G/L method there are at least two different models. The first one global is used for generating adequate boundary conditions-external loads for next local models. Then local models are used for obtaining the main results of analysis.

Adaptive Meshing Method

In metal forming and high-speed impact analysis, a body may experience large amounts of plastic deformation. Single point integration explicit elements, which are usually robust for large deformations, may give inaccurate results in these situations due to inadequate element aspect ratios [46]. Applications involving unconstrained flows of material are amendable to a fully Lagrangian finite element simulation provided that the inevitable deformation-induced distortion is eliminated by recourse to continuous remeshing [55]. This remeshing capability, commonly known as adaptive meshing, is able to improve the meshed entities' integrity during analysis. This technique is usually adopted to reduce element

Figure 2.14: Differences between GL and super element approaches [27]

distortion or refine a mesh in areas where error estimations are highest.

Adaptive meshing works on the principle of minimizing discretization error. Using this measure of mesh discretization error, FEM program can then determine if a particular mesh is fine enough. If it is not, the program will automatically refine mesh so that the measured energy error norm will decrease to some acceptable limits as set by analysts. Convergence is determined by comparing the energy error norm in the model to the energy error norm that was specified previously. The solution is considered as converged when the model energy error norm is equal to or below specified energy error norm.

The energy error norm is based on strain energy error. This is computed from elemental strain energy errors and model's total strain energy. The expression for energy error norm can be written as [56]:

$$\text{Energy_error_norm} = \sqrt{\left[\frac{\sum \text{elemental strain energy errors}}{\text{total strain energy}} \right]} \times 100\% \quad (2.9)$$

2.3.2 Initial Investigation of Hybrid Efficiency Methods

Mesh control is a normal way to improve the efficiency of simulation. However, it is realized that simulation efficiency can be further improved if non-simulation methods are incorporated. For example, a composite is analyzed in this. Composite materials are simply a combination of multi sub-phases of different materials. If only simulation methods are applied, each sub-phase should be modeled and the size of element cannot be larger than the size of sub-phase, meanwhile the interface between phases is not easy to be modeled. But this restriction can be coped with by hybrid methods.

Voleti et al. [50] made a comprehensive review to the current study on global-local methods. In addition, he proposed a possible hybrid of mechanics analysis into the process of global-local method. With the help of hybrid method, the size of global element could be larger than the size of composite phases. This cannot be achieved by a simple global-local method. In their work, two methods were used: specified boundary method (SBM) and multi-point constraint method (MPC).

The example problem was an annular cylinder as shown in Figure 2.15, composed of four "unit cells" periodically arranged throughout the domain. Each unit cell was a composite, consisting of four fibers embedded in the matrix. The entire cylinder was subjected to an internal pressure.

Figure 2.15: Schematic diagram of composite cylinder [50]

Specified Boundary Method

In this method the complete structure, called global model is created and this model uses effective material properties. The solution of this mesh determines overall response of structure under load. The displacements from global solution are extracted and applied to the boundary of local mesh. The local mesh models the area of interest and uses individual material properties. The following are the stages in the procedure of SBM: (1) Calculation of effective material properties: the effective material properties of multi-material unit cell (that is repeated throughout the domain of global structure) are calculated. *Any suitable technique, for example rule of mixtures, Tsai-Halpin equations and FEM-based methods, can be used for this purpose* [50]. These homogeneous properties are then used in global model. (2) Creating global and local meshes: global mesh refers to the mesh that models the entire structure. Local mesh refers to the mesh that models the local area of interest and is a more detailed mesh, designed to capture local variations in stress/strain. This mesh uses individual material properties of different phases present in local area, unlike global mesh. (3) Extraction and interpolation of displacements from global solution: the extraction and interpolation of data from global FEM solution are done here. Typically, the axial and radial displacement data along the boundary of local area in global mesh are interpolated. Interpolation of data from global mesh is necessary because there are fewer global nodes in global-local boundary as compared to the number of nodes on the boundary of local mesh. (4) Solution to local problem: the interpolated displacements are now applied to the boundaries of local model and problem is solved. These steps are schematically shown in Figure 2.16.

After the smeared properties of unit-cell are applied to global finite element mesh as described in Figure 2.16, global-local analysis was successfully performed as shown in Figure 2.17, and the size of global element was bigger than the size of fiber phases. If the properties of unit cell were calculated by mechanics analysis, the efficiency must be higher than pure finite element method.

Figure 2.16: Stages of specified boundary method

Figure 2.17: Global-local finite element model of specified boundary method (a) finite element mesh of global model; (b) finite element mesh of local model [50]

Multi-Point Constraint Method

If the local area of interest is known in advance, the efficiency could be further improved. This is the objective of multi-point constrain method. This method consists of one step less than SBM technique; the extraction and interpolation of displacements are avoided. Both the global and local areas of structure are modeled in one single mesh. The disadvantage to this procedure is that the user must have a priori knowledge of the location of the region of interest for further refinement. Global mesh of structure is created incorporating effective material properties. The local area of interest, whose location is known a priori, is more finely discretized and individual material properties are used here. Since local mesh has a greater number of nodes at interface than global mesh, there is an inter-element incompatibility at interface. This incompatibility is resolved by appropriately tying the "extra" nodes of local mesh to global mesh.

The calculated material properties for unit-cell can be directly used here. Thus the hybrid process also can be applied in this method. The MPC mesh for illustrative example is shown in Figure 2.18. The local region is meshed much more finely than the rest of the structure. This mesh contains two types of material properties: the coarse portion of mesh uses effective properties of unit-cell and the fine portion of mesh uses individual properties of two materials (fibers and matrix).

Figure 2.18: Global-local finite element model of multi-point constraint model [50]

It is concluded that although Voleti et al. [50] didn't use hybrid method in their model, at least they provided this basic idea to improve the efficiency of simulation to a new level that normal mesh-control methods couldn't achieve. In this PhD work, hybrid method will be used.

2.4 Dynamic Analysis of Nonlinear Drop Impact Cushion Systems

If a hybrid method is applied, that means it is necessary to do some non-simulation analysis. It is hoped that this analysis can take the place of simulation work or realize coarser mesh as shown in Section 2.3.2. Through these efforts, efficiency is hoped to be greatly improved and solve the limitations that happen in simulation work. In this section, all the work reviewed will not be performed by simulation. It is hoped that FE simulation can be replaced by other method. The computation time on the simulation of cushion buffers can be saved.

In drop-impact system, sometimes cushion material is simply taken as linear spring system [57][58]. However, polystyrene foam systems usually illustrate strong nonlinear stiffness property. In order to make an accurate prediction of drop impact response for packaged products, nonlinear analysis is inevitable. In nonlinear analysis, various numerical methods are performed previously. A lot of numerical calculation is used in nonlinear FE simulation, and this large-scale calculation makes nonlinear analysis inefficient. Therefore, analytical methods are recommended to improve the efficiency of nonlinear analysis. Because, normally, analytical results are not only quantitative but also quickly achieved.

In this section, previous studies related to analytical dynamic analysis for nonlinear drop impact system are reviewed focusing on the prediction of impact force and transient response. As mentioned in Section 2.1.3, the impact force is evaluated by cushion curve, cushion curve are plotted by numerous experimental

results. Now a theoretical method will be reviewed to show how to calculate cushion curves. After that, two analytical algorithms for general nonlinear analysis are introduced: elliptic function [59][60] and a hybrid analytical-numerical technique [61].

2.4.1 Relationship Between Impact Stress-Strain Curve and Cushion Curve

Loveridge and Mills [62] showed how to calculate cushion curves from an impact stress-strain curve, impact stress-strain curve is nominated as master curve. Master curves exist for all foams, which have a marked visco-plastic response.

The first step of this method is the experimental measurement of impact stress-strain curve, rather than static stress-strain curve. The measurement of the compressive stress strain behavior of foams has been described by Gale and Mills in 1985. An impact machine like cushion tester shown in Figure 2.7 was used. In test, the drop height of falling platen was varied from 0.25 meter to 1 meter. The acceleration of platen is measured by a mounted accelerometer. Its value is integrated twice to obtain the deflection of the top of cushion foam, and hence the strain in the foam. The compressive stress in the foam is calculated from the platen force ($= \text{platen mass} \times \text{acceleration}$) divided by foam area.

Then it is assumed that the impact stress-strain behavior follows a master curve. For any impact energy resulting force-deflection graph, when plotted in stress-strain terms, will follow master curve up to a point σ_m . For a drop height H the kinetic energy of falling platen at the moment of impact is equal the initial potential energy MgH . When the striker has been arrested, all of this energy has been applied to the block of foam of initial volume Ab . Therefore the energy

density input into the foam U is given by

$$U = \frac{MgH}{Ab} = \int_0^{\sigma_m} \sigma \times d\varepsilon \quad (2.10)$$

The integral, representing the area under impact stress-strain curve up to a maximum stress σ_m , can be integrated. The basic stress, defined as Mg/A , is therefore given by Equation (2.10) as a function of input energy density

$$\sigma_s = \frac{b}{H} \times U(\sigma_m) \quad (2.11)$$

For a given foam thickness and drop height Equation (2.11) says that the static stress is a function of input energy density and hence of maximum stress. The maximum acceleration in impact G occurs when the compressive stress has value σ_m . The acceleration is $1\ g$ upwards (in a freely falling reference frame) when static stress is applied via foam to platen mass M . Hence the ratio of accelerations gives

$$G = \frac{\sigma_m}{\sigma_s} = \frac{H\sigma_m}{bU(\sigma_m)} \quad (2.12)$$

Therefore both parameters of cushion curve ($\frac{b}{H}$ and $\frac{Mg}{A}$) can be calculated from energy density function $U(\sigma_m)$ via Equations (2.10) to (2.12).

In their study, Loveridge and Mills tried to do some experiments to verify this method. The mathematical results illustrated high reliability with average error of 12 % (3.2 % to 16.7 %). Before their work, Gruenbaum and Miltz also used the same theory to derive cushion curve in 1985. However, the error of their results was approximately 50%. As explained by Loveridge and Mills, this large error was caused by the application of static stress-strain curve, rather than impact stress-strain curve.

2.4.2 Application of Elliptic Function into Nonlinear Drop Impact System

Base on this concern of nonlinear property of cushion buffers, Suhir [59] introduced a nonlinear spring as cushion material in his numerical analysis. Using the simplest case of a nonlinear spring with a cubic restoring force, Suhir showed two basic nonlinear characterizations: hard and soft characteristics. A hard characteristic might be advisable for the structural elements that are able to withstand high accelerations, while the maximum displacement has to be made small by any means. Application of a spring with a soft characteristic can result in appreciably lower maximum accelerations than in a linear system. Therefore, the application of soft characteristic can be recommended in the case when the requirement for the lowest displacement possible is not stringent.

Figure 2.19: Vibration model [59]

In Suhir's nonlinear system, a single-degree-of-freedom (SDOF) model with nonlinear shock protector (spring) is employed as shown in Figure 2.19. This model is a general model in drop-impact analysis, and a drop-impact formulation is used in this model:

$$M\ddot{x} + c\dot{x} + (Kx + M\alpha x^3) = Mg \quad (2.13)$$

where, x is the deformation of cushion buffers, M is the mass of drop item, g is the acceleration of gravity, α is the parameter of nonlinearity. For the case of simplicity, damping coefficient of c and gravity force of Mg are ignored in nonlinear research, then the equation of motion becomes:

$$\ddot{x} + \omega_0^2 x + \alpha x^3 = 0 \quad (2.14)$$

where $\omega_0 = \sqrt{\frac{K}{M}}$ is vibration frequency. The parameter of nonlinearity α is used to analyze whether application of a nonlinear spring can lead to an appreciable reduction in the product of maximum displacement and maximum acceleration.

In the case of hard spring, $\alpha > 0$; while in the case of soft spring, $\alpha < 0$. The concept of "soft spring" and "hard spring" was discussed. The effect of these two nonlinear systems was compared with that of linear spring in his study.

1. *Application of a nonlinear spring with a hard characteristic.* It results in an appreciable decrease in maximum displacement and a substantial increase in maximum acceleration, in comparison with linear case. The accumulative effect of a hard spring is such that the product of maximum displacement and maximum acceleration increase with an increase in the degree of nonlinearity. Therefore, employment of such a spring can become feasible for elements that are able to withstand high accelerations, while the maximum deformation has to be made small by any means.
2. *Application of a nonlinear spring with a soft characteristic.* It results in an appreciable increase in the maximum displacement and a substantial decrease in maximum acceleration, in comparison with linear case. The accumulative effect of a soft spring is such that the product of maximum displacement and maximum acceleration decrease with an increase in the degree of nonlinearity. Therefore, employment of such a spring can become feasible for elements that cannot withstand high accelerations, while maximum deformation allowed being larger than that of linear case.

Equation (2.14) is the canonical form of Duffing's equation [60], an analytical solution is available for this formula [63]. By reviewing Equation (2.14) using the chain rule for $\ddot{x}$,

$$\dot{x} \frac{d\dot{x}}{dx} = -\omega_0 x - \alpha x^3 \quad (2.15)$$

and then separating variables and integrating,

$$\int_{\dot{x}(0)}^{\dot{x}} \dot{x} d\dot{x} = \int_{x(0)}^x (-\omega_0 x - \alpha x^3) dx \quad (2.16)$$

$$\frac{\dot{x}^2}{2} - \frac{\dot{x}^2(0)}{2} = - \left(\frac{\omega_0 x^2}{2} + \frac{\alpha x^4}{4} \right) + \left(\frac{\omega_0 x^2(0)}{2} + \frac{\alpha x^4(0)}{4} \right) \quad (2.17)$$

if $x(0) = 0$ and $H = \frac{\dot{x}^2(0)}{2}$, the following expression for the integral of motion in phase plane is obtained:

$$\frac{\dot{x}^2}{2} + \left(\frac{\omega_0 x^2}{2} + \frac{\alpha x^4}{4} \right) = H \quad (2.18)$$

$$\dot{x} = \pm \sqrt{\left(2H - \omega_0 x^2 - \frac{\alpha x^4}{2} \right)} \quad (2.19)$$

$$dt = \frac{dx}{\pm \sqrt{2H - \omega_0 x^2 - \frac{\alpha x^4}{2}}} \quad (2.20)$$

Equation (2.19) describes phase plane trajectory of the system in Equation (2.14) for a given set of initial conditions.

If a change of variables is implemented, $x = -x(0) \cos(\theta)$, then $H = \frac{\omega_0 x^2(0)}{2} + \frac{\alpha x^4(0)}{4}$ and the integral in Equation (2.20) can be evaluated as follows:

$$t = \frac{1}{\sqrt{\omega_0 + \alpha x^2(0)}} \int_{\theta(0)}^{\theta} \frac{d\theta}{\sqrt{1 - l \sin^2(\theta)}} \quad (2.21)$$

where $l = \frac{\alpha x^2(0)}{2(\omega_0 + \alpha x^2(0))}$. The period of motion over period is thus given by

$$T = \frac{4}{\sqrt{\omega_0 + \alpha x^2(0)}} \int_0^{\frac{1}{2\pi}} \frac{d\theta}{\sqrt{1 - l \sin^2(\theta)}} = \frac{4}{\sqrt{\omega_0 + \alpha x^2(0)}} El \left(\frac{\alpha}{\omega_0} x^2(0) \right) \quad (2.22)$$

in which $El \left(\frac{\alpha}{\omega_0} x^2(0) \right)$ is the complete elliptic integral of the first kind. Therefore, if Duffing's equation (2.14) can be applied for nonlinear drop-impact analysis and can achieve an analytical Elliptic Function solution.

2.4.3 Hybrid Analytical-Numerical Analysis for Nonlinear Damped System

In Asir et al.'s work, a developed hybrid analytical-numerical method was mentioned [61]. This is a technique developed by Mansour and Hussein. This is used to solve nonlinear differential equation model that describes underdamped and overdamped motion of systems. And it has been applied efficiently to solve highly nonlinear systems of both partial and ordinary differential equation models as mentioned by authors. In this part, the basic process of this method will be introduced. In addition, an application example will also be reviewed.

General Process of High-Efficiency Hybrid Analytical-Numerical Technique

First, the basic process of this technique is introduced. In this, consider a general nonlinear problem described by differential equation

$$D(x, t) = 0 \quad \text{in } S \quad (2.23)$$

with boundary conditions

$$B(x, t) = 0 \quad \text{in } R \quad (2.24)$$

where D and B are non-linear differential operators, S is the domain of the problem and R its boundary.

The following steps can describe the application of present hybrid analytical-numerical technique: (1) linearization of the nonlinear terms in Equation (2.23), which will become of linear form

$$\bar{D}(x, t) = 0 \quad (2.25)$$

where $\bar{D}$ is a linear differential operator; (2) if any x terms remain nonlinear after linearization step, they are assumed temporarily constant to permit linear solution of Equation (2.25); (3) Equation (2.25) is solved by one of standard methods used for linear differential equations, and the solution obtained is put in the form

$$x_{i+1} = f(x_i, t) \quad (2.26)$$

where x_i is the X -value at the i th iteration level, which has been assumed constant in Step (2), and x_{i+1} is the X -value at $(i+1)$ th level iteration, which represents

the dependent variable solution of Equation (2.26); (4) an iterative procedure is performed until convergence between x_{i+1} and x_i is obtained at each t -value in the interval of interest; an iterative solution is said to be convergent if $|x_{i+1} - x_i| < \delta$, where δ is tolerance error.

Practical Application of Hybrid Analytical-Numerical Technique

The equation of motion of a nonlinear damped system of a single degree of freedom subjected to external force F can be expressed as

$$\ddot{x} + (2\gamma + \alpha\gamma_1 x^2) \dot{x} + \omega_0^2 x + \alpha x^3 = F \quad (2.27)$$

where 2γ and γ_1 are linear damping parameters, α is nonlinearity parameter, ω_0 is the frequency of underdamped motion. The system is assumed to be at rest initially:

$$x(0) = \dot{x}(0) = 0 \quad (2.28)$$

Now it assumes

$$x^2 = \beta^2 \quad (2.29)$$

and the substitution of Equation (2.29) into Equation (2.27) yields

$$\ddot{x} + (2\gamma + \alpha\gamma_1\beta^2) \dot{x} + (\omega_0^2 + \alpha\beta^2) x = F \quad (2.30)$$

or

$$\ddot{x} + \mu_1 \dot{x} + \mu_2 x = F \quad (2.31)$$

Equation (2.31) is a non-homogeneous constant coefficient ordinary differential equation that has general solution

$$x(t) = c_1 e^{m_1 t} + c_2 e^{m_2 t} + F/\mu_2 \quad (2.32)$$

where

$$m_1 = \left[-\mu_1 + (\mu_1^2 - 4\mu_2)^{1/2} \right] / 2 \quad (2.33)$$

$$m_2 = \left[-\mu_1 - (\mu_1^2 - 4\mu_2)^{1/2} \right] / 2 \quad (2.34)$$

$$\mu_1 = 2\gamma + \alpha\gamma_1\beta^2 \quad (2.35)$$

$$\mu_2 = \omega_0^2 + \alpha\beta^2 \quad (2.36)$$

Upon substituting the initial conditions given in Equation (2.28) into Equation (2.32) the complete analytical solution becomes

$$x(t) = \frac{F}{\mu_2} \left[1 + \frac{m_2 e^{m_1 t} - m_1 e^{m_2 t}}{m_1 - m_2} \right] \quad (2.37)$$

Based on the method of solution, Equation (2.37) can be put in general iterative form

$$x_{i+1}(t) = f(t, \beta) = f(t, x_i) \quad (2.38)$$

The most important contribution of this work is the successful application of linear differential algorithms to solve nonlinear problem. The efficiency of linear algorithms is definitely higher than nonlinear algorithms.

2.5 Modeling Techniques of Printed Circuit Boards

The focus of this work is mainly on shock response of multilayer printed circuit boards and their interaction with nonlinear cushion material. Therefore, after review work for nonlinear cushion system, previous work on the analysis of multilayer circuit boards is reviewed in this section. In order to understand the response of multilayer circuit boards, PCB should be successfully modeled. Multilayer circuit board is a multilayer multimaterial composite with woven fiber reinforced composite plies and metallic plies. In order to obtain detailed material properties, unit-cell modeling method is used in micro-level [64][65]. However, it is not easy to expand analysis to board-level sometimes by using this method. If experimental method is used to measure material properties in board-level, the material properties is not enough to built up a sufficient stiffness/compliance matrix to describe orthotropic material properties of PCB[66] because of the limit measurements in physical tests. If finite element simulation is used, contact interface is not easy to define between layers without a developed multilayer model.

This section is presented to understand the material structure of printed circuit board and review modeling technique for printed circuit boards. There are five subsections in this section: The first subsection describes classification of different PCBs; The second subsection discusses the material structure and layout of different PCBs; The third subsection covers the material characteristics of each type of material that makes up multilayer PCB; The fourth subsection surveys some modeling methods for multilayer laminates and woven fiber reinforced composites that consists of multilayer circuit boards; The last subsection presents previous PCB modeling methodologies.

2.5.1 Classification of Printed Circuit Boards

The classification of printed circuit boards is a broad process, taking many factors such as substrate materials and fabrication processes into consideration. There are basically six main classification considerations as follows:

- (1) Nature of substrate
- (2) Graphical or Discrete Wiring Boards
- (3) Rigid or Flexible Boards
- (4) Subtractive or Additive Process
- (5) Structure of PCB: Single-sided, Double-sided or Multilayer
- (6) Plated-Through-Hole (PTH) or non-PTH method
- (7) Process: Print and Etch, Pattern Plating or Panel Plating

The flow chart in Figure 2.20 [67] shows broad classification of PCBs.

In present work, the main class of consideration will be on the structure of PCBs: Single-sided, Double-sided and Multilayer boards. In particular, the multilayer structure will be studied in detail and will be used for the modeling and simulation of PCB.

2.5.2 Structure of Printed Circuit Boards

Single Sided Boards

Single-sided boards (SSB) have circuitry on only one side of board and are often referred to as Print and Etch boards because etch resist is usually printed on by screen-printing techniques and conductor pattern is subsequently formed by chemically etching the exposed, and unwanted copper foil. Single-sided boards are

Figure 2.20: Classification flow chart of printed circuit boards [67]

usually used for low-cost, high-volume, simple circuitry and relatively low functionality boards. They are applicable when circuit types and speeds do not demand unusual electrical characteristics.

Double Sided Boards

Double-sided boards (DSB) have circuitry on both sides of boards and are further classified into two categories: (1) Plated-Through-Hole (PTH) and (2) Non-PTH. The application of double-sided boards are extended to high-volume, low-cost products such as audio equipment, floppy disk controllers, car radios, remote controls etc.

Multilayer Boards

Multilayer PCBs have at least three layers of conductors, two of which are on the outer surfaces of board, while the remainder layers are incorporated into insulating board. Multilayer boards (MLB) are the mainstreams of all electronic devices

such as computers, military equipment and consumer products such as televisions, digital cameras, portable video cameras, cellular phones and audio discs etc. It has become a dominant board design since electronic products are becoming smaller, faster and more powerful.

A standard multilayer board contains internal and external circuits, interconnected by plated through holes (PTHs) drilled through entire board. The multilayer board is fabricated by gluing together or technically called laminating, pre-patterned, double-sided inner layer panels called components. These components include one or more board images. The starting material used to make the component is called C-stage, which indicates that it is fully cured prior to lamination. The glue layer is called B-stage, which indicates that it is partially cured prior to lamination. After lamination, composite panel is drilled, patterned and finished. Figure 2.21 illustrates a typical four-layer, depicting the metamorphosis of laminate and the surface of PCB as a multilayer is processed from material preparation through solder coat operation.

Figure 2.21: Four-layer multilayer using foil lamination technique [68]

For multilayer boards, there are mainly two different stack-up configurations and construction: (1) Foil-capped Stack-up and (2) Component-capped stack-up.

A foil capped board is fabricated from one or more patterned inner layer components and two copper sheets. The copper sheets form outer layers of fab-

ricated multilayer board. Figure 2.22 [67] shows a foil capped stack-up for a six-layer multilayer board containing two component boards, three glue layers and two sheets of copper. The copper layers are numbered from 1 to 6, beginning from top foil layer. Hence, when someone describes a multilayer PCB as a six-layer board, it really means that there are six layers of copper foil in the buildup of multilayer PCB and not six physical layers per se. Figure 2.23 [68] shows a buildup diagram of a typical six-layer multilayer board.

Figure 2.22: Foil capped stack-up for a six-layer board [67]

A component-capped multilayer board does not use copper foil as outer layers. It is fabricated from inner layer components and B-stage. Figure 2.24 [67] shows a six-layer component capped board containing three component boards (as compared to two component boards in foil-capped stack-up) and two glue layers.

2.5.3 Material Buildup of Multilayer Circuit Boards

Material of Laminates

The base material of most PCBs comprises of reinforced plastic laminate. Laminates for PCBs are composite materials. They are made of an insulating base, which is made of reinforced plastic and one or two thin copper foils bonded to insulating base.

Figure 2.23: Buildup diagram for a six-layer multilayer circuit board [68]

Figure 2.24: Component capped multilayer board [67]

There are many different base materials that are used to fabricate multilayer boards. With the exception of metallic and ceramic bases, these base materials are all based on synthetic resins, either thermoplastic or thermosetting, produced by means of a polymerization process. The selection of the type of resin to be used takes into account many characteristics such as dielectric constant, tensile and compression strength, heat resistance, machinability, etc. These characteristics have varying degrees of importance depending on the specific application of PCB.

There are a few types of resin and these are mentioned in order of decreasing importance: epoxy resins, phenolic resin, polyester resin, melamine resin, silicon resin and polytetrafluoroethylene resin [69]. Out of these, epoxy resins are the most commonly used.

There are also many different types of reinforcing materials. These may include paper, chopped glass filaments, glass cloth, glass mat, asbestos and so on.

With the different combinations of base material and reinforcing material, different grades of laminates exist. Table 2.3 shows a summary of different combinations of resin and reinforcing materials and their NEMA classification [69].

Since the material properties for PCB model used in drop-impact analysis will be based on FR-4, a more detailed presentation of the structure of FR-4 is as described in the next section.

Material of FR4

FR-4 laminates are constructed with multiple plies of epoxy-resin-impregnated woven glass cloth. The following sections briefly describe the materials that make up FR-4 laminates.

The glass cloths used for reinforcing epoxy resins are produced from long glass fibers or filaments. A large number of these filaments are bunched together and twisted to form a thread. The threads are subsequently woven to produce glass fabric. The mesh of cloth depends upon the spacing of threads and usually, the

Table 2.3: Different grades of laminates [69]

Resin	Reinforcing material		NEMA classification
	Type	State	
Phenolics	NA	NA	ES-2
	Paper	Sheet	X, XP, XPC, XX, XXP, XXPC, XXX, XXXP, XXXPC, FR-2
	Cotton	Fabric	C, CE, L, LE
	Asbestos	Sheet	A
	Asbestos	Fabric	AA
	Glass	Staple Fibers	G-2
	Glass	Cloth	G-3
	Nylon	NA	N-1
Aminos	NA	NA	ES-1, ES-3
	Glass	Cloth	G-5, G-9
Epoxyes	Paper	Sheet	FR-3
	Glass	Cloth	G-10, G-11, FR-4, FR-5
Alkyds	Glass	Mat	GPO-1, GPO-2
Silicons	Glass	Cloth	G-7

spacing is different for the two orthogonal directions of threads. The two thread directions are called Warp yarns and Fill yarns that are 90° to warp. Figure 2.25 shows the woven pattern of glass cloth. One of the most commonly used glass cloth for PCB laminates is Type 7628.

Figure 2.25: Fibers in plain weave pattern [70]

The base raw material, which in this case is woven glass cloth, is impreg-

nated or coated with epoxy resin. The resin is polymerized to what is called a B-stage. This semi-cured material is also known as prepreg. Next, typically one or two layers of B-stage prepreg will be consolidated between two layers of copper foil in a hot press to form a C-stage core. This C-stage core is a fully cured layer and is the starting material in making multilayer PCBs as described in Section 2.5.2.

Zhu et al. [71] have described epoxy-resin-impregnated woven glass cloth with a model as shown in Figure 2.26. The fill and warp bundles are two orthogonal directions of glass cloth threads while the matrix layer represents epoxy resin impregnated into glass cloth. Figure 2.27 shows a microscopic view of epoxy-resin-impregnated woven glass cloth as described by Zhu et al.

Figure 2.26: Epoxy-resin-impregnated woven glass cloth [71]

Figure 2.27: Photomicrographs of 7628 laminate under 67 times magnification a) Warp yarns parallel to the page, fill yarns perpendicular. b) Fill yarns parallel to the page, warp yarns perpendicular [71]

2.5.4 Previous Modeling of Multilayer and Woven Fiber Reinforced Composites

As the review of structure of PCB, multilayer laminate and woven composite are two main composite phases consisting of multilayer circuit boards. This section aims to review on past composite modeling methods for these two kinds of composite.

Modeling of Multilayer Laminates

In Coutellier and Rozycki's [72] study of multilayer multimaterial composites, PAM-Crash software was used to take into account elastic plastic damaging behavior for metallic plies and elastic damaging behavior for composite plies. The two simulation models used in this work consist of a homogeneous model and a bi-phase model where the behavior of fibers and matrix are distinguished separately. In the homogeneous model, the composite behavior of fibers was assumed to be the same in both tension and compression. Damage parameters were taken into account in this modeling. The bi-phase model [73] [74] was first introduced due to a crash behavior study of a car made entirely in composite [75]. This model distinguishes two elementary layer components: fibers (uni-dimensional phase) and matrix (orthotropic phase). A three point dynamic bending experiment was carried out to validate multilayer multimaterial finite element model. The model includes a combination of aluminum and E-glass / epoxy composite (30% of fibers). The orientation and stacking of plies used in this paper is described in Figure 2.28. The numerical results derived from these models are close to experimental results as real multilayered multi-material behavior is more suitably taken into account.

Yu et al. [76] attempted to develop an accurate, efficient and reliable stress analysis procedure for laminated composites. The finite element model used was based upon global-local model proposed by Pagano and Soni [77], in which the entire laminate thickness can be divided into two convenient regions as illustrated

Figure 2.28: Orientation and stacking of aluminum and E-glass / epoxy composite [72]

in Figure 2.29. The first region is called global region in which is assumed to be a single unit while the other region is called local region that regards each inter-laminar plane as distinct within its boundaries. The numerical analysis was carried out using an out-of-core linear equation solver which was developed by Yu et al. to solve this global-local model.

Figure 2.29: Global-local multilayer model [76]

Nakamura and Wu [78] presented comprehensive failure mechanisms of composite structures consisting of many laminates using detailed finite element models. The objective of this research was to identify the effects of delamination and ply arrangement on multilayer structures. This paper [78] presented possible arrangements and stacking sequence for laminates multilayer composite panel. In this analysis, $[0/90]_{ns}$ and $[90/0]_{ns}$ stacking sequence was considered in cross-ply lam-

inates as shown in Figure 2.30. Figure 2.30 also shows that the fibers in each lamina are orientated either horizontally (0°) or out-of-plane (90°) direction. In all models used in this paper, the overall thickness of laminates were kept constant while the thickness of each layer is varied proportionally (i.e.: each ply thickness of $[0/90]_{2s}$ is one-half of that $[0/90]_s$) to allow direct comparison of various models with different number of layers.

Figure 2.30: Sections of symmetric cross-ply laminates [78]

Modeling of Fiber Reinforced Composites

A study of a fully three dimensional viscoelastic model of a plain weave composite with accurate characterization of the woven geometry was presented by Zhu et al. [71]. The woven composite substrate for multilayer circuit boards was modeled in detail with discussions on fabric, matrix properties and fiber bundles, both in fill and warp directions. The viscoelastic model was utilized to predict the creep compliance of 7628 style composite substrate for multilayer circuit board applications and result showed that the flexural deformations and response of woven fabric bundles play a major role in viscoelastic behavior of composite.

Brown and Sottos [79] have characterized the thermoelastic properties of several common woven glass / epoxy substrate materials in both warp and fill directions. The thermoelastic properties of different plain weave fabric styles (1080, 2116, 2313, 3313, and 7628) used in multilayered PCBs were investigated to predict the warpage of boards due to residual stresses developed during the manufacturing process of these laminated structure.

3D unit-cell model by solid finite element method is one of common method for the study of this fiber-reinforced material. As shown in Figure 2.31, Kim and Swan [80] provided a unit-cell model to developed unit-cell analysis as a tool to study property-structure characteristics of Plain-weave textile composites. In their work, they modeled finite deformation behavior at fiber diameter scale; performed hyper-elastic modeling of aligned fiber and matrix yarns, computed onset of compression buckling in textile composite; computed strong tension-stiffening at large stretches. Also some specific challenges about finite element unit-cell cell modeling is also presented: (1) modeling/meshing of material structure; (2) high cost of unit-cell analysis; (3) finite deformation homogenization; (4) Constitutive modeling. Also in Tang and Whitcomb's [65] work, solid unit-cell model was applied in damage analysis of fiber-reinforced composites. Exclusive plain weaves, 4-, 5-, 8-harness satin and twill weaves were also considered. The flexibility of unit-cell model in microanalysis is highly presented.

Figure 2.31: Unit cell of plain unit cell of plain-weave textile composite [80]

Tabiei and Ivanov [81] developed a computational micro-mechanical model of woven fabric composite material to simulate failure. The material model was based on a repeated unit cell approach. The micro-mechanical material model [82] involves using the representative volume cell of woven fabric composite material that is extracted from the deformed material pattern as shown in Figures 2.32 and

2.33. The described model was first programmed by using MATLAB software and validation example considered was Graphite / Epoxy plain-woven fabric composite material (AS4 / 3501-6) as described in [83][84]. It was subsequently modeled as shell elements in LS-DYNA commercial finite element code. The strength and elastic properties of fabric yarns and epoxy resin can be found in [81].

Figure 2.32: Woven composite interlacing pattern [81]

Figure 2.33: Micro-mechanical model [81]

Exclusive elastic analysis, damage behavior is also an important part for study. As shown in Figure 2.34, failure of fiber reinforced composites may be caused by a) Fiber buckling, b) Fiber breakage, c) Matrix cracking, d) Delaminating, or by a combination of these factors. Buckling causes the reduction in the stiffness of fibers. One of the main roles of fibers is to carry tensile loads. When the fiber without matrix fails, it can no longer carry loads. In case of fibers surrounded by matrix, matrix transfers load across this gap. This is the reason fiber-reinforced matrix is of higher strength than fibers alone. Matrix cracking, which is another way of failure of a fiber-reinforced matrix, have some detrimental

effects: they facilitate moisture absorption, reduce the stiffness of laminate and they also initiate delamination. Delamination is the separation of adjacent layers that may be introduced either during manufacture or subsequently by loads applied to laminate. Delamination reduces bending stiffness and strength as well as load carrying capacity of laminate under compression. As in the case of metals, when the critical size of delamination is reached the loss of compressive strength becomes highly significant.

Figure 2.34: Types of fiber reinforced composite failures

Tang and Whitcomb's [65] used unit-cell models for the damage analysis of fiber-reinforced composites. Rather than developing a new progressive failure analysis approach, the focus is placed on comparing the damage behaviors of various weaves predicted by the selected failure criterion and property degradation model. The loading conditions include uniaxial tension and compression. The discussions focus on (1) the effect of woven architecture on predicted progressive failure behaviors (2) the similarities and difference in damage initiation and evolution mechanisms between plain and satin weaves and (3) the sensitivity of predicted progressive failure behavior to assumptions about geometric nonlinearity and property degradation model.

Beside unit-cell damage model, some mechanics models were provided. Ladeveze model [85][86] is one of them. The obvious advantage of this model is that detailed failure information in a fiber-reinforced composite can be predicted in a

homogeneous model, rather than multi-scale unit-cell model. Thus this model has a high computational efficiency. Ladevze model was already applied in simulation [51] [72], it is dedicated to the numerical simulation of unidirectional continuous fiber reinforced composite materials. Other than heterogeneous "bi-phase" model, Ladevze model does not treat two phases separately (fibers and matrix). Instead composite ply is described using homogeneous continuum mechanics. The elementary layer behaviour law can be elastic or elastic/plastic and includes some damages related to experimentally observed phenomena, Figures 2.35, attributed to matrix microcracking which appears under a transverse tensile loading, fiber-matrix rebounding which is generally due to a shear loading. The model takes also into account fiber failure for tensile and compressive loads. Finally, a nonlinear compressive behaviour in fiber direction is available if fiber misalignments or micro buckling phenomena prescribe a longitudinal compressive behaviour different from tensile. The modified Ladevze parameters from following experimental test series. (1) Simple tension test on $[0]_8$ laminates, (2) simple tension test with "load/unload" cycles on $[45]_{2S}$ laminates, (3) simple tension test with "load/unload" cycles on $[+45]_8$ laminates, (4) simple tension test with "load/unload" cycles on $[67.5]_{2S}$ laminates, (5) simple compression test on $[0]_8$ laminates. Each test allows to find a given subset of material constants that may be needed in subsequent tests to calibrate further material constants.

Figure 2.35: Global composite ply damages in Ladevze model [51]

2.5.5 Previous Modeling of Printed Circuit Boards

Printed circuit boards are usually modeled as 2D and 3D models. For 2D model, PCB is usually homogeneously modeled. This modeling method has high computational efficiency. However in 2D models, material properties are usually obtained from physical tests and cannot be comprehensively described. 3D models can describe all detailed information of material properties, however, simulation is time-consuming. One of the objectives of this PhD work is the combination of 2D modeling and 3D modeling: high efficient modeling with comprehensive description of material properties. In this part, previous modeling work for PCBs will be reviewed.

Two-Dimensional Homogeneous Modeling of Printed Circuit Boards

Two-dimensional modeling of printed circuit board is normally used. The simple operation and high computational efficiency make it becomes first choice sometimes. In the work by Wu et al. [87], a finite element model of PCB was established with use of ANSYS based on design specifications and static tests. The model is calibrated with various static and dynamic tests to verify that static displacements at selected locations, PCB twist angles, and first three natural frequencies predicted by FEA model are consistent with those obtained by tests. With the application of tensile tests and shear tests, in-plane (x - z plane) material properties (E_x , E_y , G_{xy} , ν_{xz}) were obtained. For example, shear modulus of PCB was determined by a twist test as shown in Figure 2.36. In Wang et al.'s work [66], similar modeling methods were also performed.

Three-Dimensional Modeling of Printed Circuit Boards

The advantage of 3D model is that this model has enough degrees of freedom to describe composite material properties, which can improve the accuracy of modeling in theory. Lee [88] carried out 3D finite element modeling of PCBs for structural

Figure 2.36: Twist test set-up for determining shear modulus of PCB [87]

analysis. The objective of this work was to establish main causes of warpage or out-of-plane PCB deflections and to construct a finite element model of board deflections. The software used in simulation was MARC, a commercial finite element model program for predicting the deflection of PCB when subjected to thermal shock. The PCB model was a Cem-3 board, a commercial name of woven glass fiber in phenolic resin and is made up of two to three plies of fine glass and 50-65 percent weight resin content. The PCB material was assumed to be orthotropic Cem-3 and temperature dependent thermal and mechanical properties used in FEM structure analysis are shown in Figure 2.37. As shown in Figure 2.33, the material properties of all composite phases were used in 3D modeling. In this research, the empirical measurement of PCB deflection and numerical analysis showed a 6% difference in maximum deflection.

In Moore and Jarvis's work [89], difference between the properties of printed circuit boards in out-of-substrate direction and the properties in the plane of substrate was concerned. The variation of in-substrate-plane properties was examined in mechanical testing and in finite element analysis. It was shown that the in-substrate-plane coefficient of thermal expansion and modulus of elasticity are influenced significantly by the direction of the traces in the copper layers of substrate. The impact of these on warpage was explored for a BGA assembly. Therefore, for multichip products it is necessary to consider the effects that differences in in-

Property	Phenolic fibreglass			Copper
$T(^{\circ})$	25	100	200	
$\rho \text{ (kg/m}^3\text{)}$	1,814	1,814	1,814	8,940
$E_x \text{ (Gpa)}$	19.42	15.14	14.42	103.42
$E_y \text{ (Gpa)}$	19.42	15.14	14.42	103.42
$E_z \text{ (Gpa)}$	3.42	0.864	0.096	103.42
ν_{xy}	0.25	0.27	0.46	0.3
ν_{xz}	0.16	0.18	0.48	0.3
ν_{yz}	0.16	0.18	0.48	0.3
$G_{xy} \text{ (Gpa)}$	6.8	4.6	4.33	39.71
$G_{xz} \text{ (Gpa)}$	1.44	0.44	0.133	39.71
$G_{yz} \text{ (Gpa)}$	1.44	0.44	0.133	39.71
$\alpha_x \text{ (ppm/}^{\circ}\text{C)}$	19.48	15.52	12.5	17.0
$\alpha_y \text{ (ppm/}^{\circ}\text{C)}$	19.48	15.52	12.5	17.0
$\alpha_z \text{ (ppm/}^{\circ}\text{C)}$	37.68	50.16	260.8	17.0

Figure 2.37: Thermal and mechanical properties of PCB [88]

substrate-plane mechanical properties may have on finished products. In order to investigate the effects of the orientation of copper traces, a simple 3-D finite element model was made of a square two-layer substrate as shown in Figure 2.38. The software used was ANSYS 5.6. In this model the copper was laid out to run continuously from end to end in X -direction, while in Z -direction the copper was divided into parallel lines occupying 50% of total area. Three substrate models can be combined into a four-layer PCB model.

Figure 2.38: 3D FEM Model of BGA substrate with orthogonal copper layout [89]

If the fiber phases in FR4 composite are expected to layout as the way shown in Figure 2.38, it is necessary to use the unit-cell model shown in Figure 2.26. However, the requirement of computational resource will become too large

to be suffered by normal computer. Till now, we didn't find any work that directly use 3D unit cell models to perform simulation in board-level. This is challenge to perform micro-structure and board-level analysis for PCB at the same time, if the woven fibers are expected to be modeled by using finite element methods.

Chapter 3

Initial Development of Virtual Buffer Model for Quick Drop-Impact Analysis of PCB in Electronic Products

A virtual boundary model is presented as virtual buffer model in this study; it is initially established to achieve quick finite element analysis for components in packaged electronic products. Furthermore, this model illustrated one more advantage: it can provide a platform to hybridize FE simulation and methods without FE together in multi-scale models. Multi-scale models are usually large-scale with complex finite elements simulation. The incorporation of non-FE methods not only can reduce computational time, but also can improve the flexibility of optimization analysis.

Usually, drop-impact simulation for packaged electronic products is time-consuming and complicated [26][29]. However, the simulation of some small but important components in the electronic products doesn't need too much computational resource; most resource is used to calculate contact arithmetic and modeling

of large components rather than some important small parts. Virtual boundary method can divide the simulation on small and large components, and boundary field from large components is calculated based on coarse meshing model or non-FE methods. Through this way, most computation resource is used on small components; thus high computational efficiency is realized. For the modeling of packaged electronic products, boundary field is from packaging material-buffer on TV, therefore this method is also called as virtual buffer method. In this chapter, the reliability and advantage of virtual buffer method will be presented through the analysis of a packaged TV model. The application of the method to design and optimize an electronic component (PCB)'s in this TV model is briefly discussed at the same time.

3.1 Introduction of Chapter

The concept phase of product development is characterized by the need to evaluate and optimize electronic devices as fast as possible. Numerical simulations allow optimization design in a faster and more economical way as compared to real tests in crash-worthiness evaluation and optimization [24][90][91]. During last few years, with numerical simulations, absolute simulation time for modeling, computing and investigating a complete crash model have been reduced significantly. However, it is noticed that the simulation stage in packaged electronic devices turned out to be time consuming because of cushioning material's complex and solid structure [2][1][26]. Therefore, reducing analysis on cushioning package can accelerate the process of crash-worthiness simulation. With this in mind, an object-oriented, event-driven virtual boundary condition (VBC) concept for FEM simulation of tiny flexible multi-body systems is provided to achieve this. Through the proposed virtual boundary method, the boundary field, which only accounts for about 10% of original CPU time, will replace those external components. Accordingly, the substantial CPU time on those components and elements will be saved, and it can

Table 3.1: An overview of finite element mesh of components

	TV Cabinet	Glass Tube	Buffers	PCB/ Electronics components	Chassis	Floor
Element Type	SHELL	SOLID	SOLID	SHELL / SOLID	SOLID	SOLID
No. of Elements	1893	3648	26666	537/132	5548	18
No. of Nodes	1848	4420	7922	839	9214	56

speed up development process and would provide stable optimization results. In this chapter, VBC is the boundary conditions derived from TV buffers. Hence, virtual boundary can also be described as virtual buffer in the present work.

This chapter describes a study, which is initiated by a local electronics company to perform a drop test for a given packaged 29" TV structure using computer simulation. The FEM TV assembly model showed in Figure 3.1 consists of TV cabinet, glass tube, bottom buffers, and top buffers. PCB is tiny component mounted on the bottom of TV cabinet by chassis as shown in Figure 3.2. An overview of finite element mesh of components is described in Table 3.1. Referring to Table 3.1, the number of SOLID elements used to mesh the buffers is 26666 compared to the mere 1893 SHELL elements used in TV structure. More often than not, only PCB on TV structure are studied for product development. This indirectly suggests that the tremendous amount of CPU resources used to process buffer models is not fully utilized. Huge amounts of CPU costs will be incurred unnecessarily if repeated testing were to be performed. As such, the concept of virtual boundary (or virtual buffer) could be adopted to eliminate redundant computation.

The basic concept of virtual buffer in this study refers to the replication of actual cushioning buffers by displacement field applied onto TV cabinet to represent actual contact conditions. This way, there is no requirement to process buffer models as they are removed from analysis. Only a displacement field is applied to represent contact conditions. This greatly reduces the number of elements and hence, CPU resources needed. In addition, the required contact algorithm usually

Figure 3.1: FEA model of entire TV with packaging materials and rigid floor

Figure 3.2: FEA model of PCB board and chassis

accounts for about 10% of total CPU time and hence, this portion of time will be saved in virtual buffer analysis.

3.2 Methodology of Virtual Boundary Method

The basic idea of virtual boundary method is to use boundary field to replace the components that generate this boundary field. Boundary field is boundary condition or loading in FEM concept. In current study, we use time-domain displacement load on TV cabinet. This method consists in two steps as shown in Figure 3.3: (1) Establish a virtual boundary model, (2) Create an objective model and incorporate it into virtual boundary model. As a multi-body system, TV model is presented to help explaining the application of this method as long as its terminologies. In this, PCB in TV model is selected as the component concerned, which is called objective model.

3.2.1 Definitions in Virtual Boundary Method

1. *Physical boundary model.* Physical boundary model is defined with respect to virtual boundary model. In this study, physical buffer model is the assembly of buffers, TV cabinet and TV tube as shown in Figure 3.4(b). Its task is to output boundary field (displacement field) to virtual boundary model.
2. *Objective model.* Objective model is usually the tiny part needed to investigate. In our study, the final aim is to analyze the dynamic response of PCB; therefore PCB with chassis as shown in Figure 3.2 is objective model .
3. *Virtual boundary model.* Virtual boundary model is the combination of boundary field and boundary entity model. As shown in Figure 3.4(a), boundary field is displacement field applied on contact area between buffers and TV cabinet; while boundary entity model is the assembly of TV cabinet and TV tube. Virtual boundary model is meant to substitute the actual

Figure 3.3: Stages in virtual boundary method

cushioning buffers by means of boundary field (displacement field). The contact interaction is transferred to objective model (PCB) in the second step of virtual boundary method shown in Figure 3.3.

Figure 3.4: Physical boundary and virtual boundary models

The difference between physical boundary model and virtual boundary models is that: in physical boundary model, buffer elements exist in FE model as physical state; while in virtual boundary model, buffer elements are replaced by the displacement field as virtual state. Thus in this study, the physical boundary model can be defined as physical buffer model, and virtual boundary model as virtual buffer model as shown in Figure 3.4.

As a conclusion, in this study, whole TV structure (TV cabinet, TV tube and PCB) is divided into two parts: virtual buffer model (TV cabinet, TV tube and boundary field) and objective model (PCB), they are analyzed in two stages. Moreover buffers are the cushion materials outside of TV structure, after removing them and replacing displacement fields, physical buffer model becomes a virtual buffer model.

3.2.2 Solution Scheme

The detailed solution process with virtual boundary method has been shown in Figure 3.3. The steps are listed as follows:

1. *Establish a virtual boundary model*: boundary field and virtual boundary entity model are two parts of virtual boundary model, and they are all extracted from physical boundary model. In this study, physical boundary model is the combination of TV structure and cushion materials, and the displacement field along the boundary of TV structure in physical buffer model is interpolated. The interpolated displacement field is applied to the boundary of TV structure, then the buffers that are outside of TV structure are replaced. The virtual boundary model is established, and these stages are schematically shown in Step 1 on Figure 3.3.
2. *Create objective model and incorporate it into virtual boundary model*: after establishing virtual boundary model, a "container" structure is set up. Outside of "container", buffers are removed and replaced by displacement field, while inside "container", the objective model (PCB with chassis) is created and mounted. Then the PCB can be modified again and again in the virtual boundary model set in Step 1, and cushion material outside of TV structure need not be recalculated. These stages are also schematically shown in Step 2 on Figure 3.3.

There is another point of concern: as described previously, the objective model does not exist in boundary model and the first step of virtual boundary method. Owing to two reasons: (1) During simulation, tiny structure is too small to influence cushioning material's impact behavior, while the influence from cushioning material is major contribution to tiny structure's impact behavior. Therefore, in the process of establishing boundary model, the influence from objective model can be ignored, and it is also not required in boundary model. (2) Any modification of PCB cannot bring noticeable influence to the displacement field

from cushion buffers because of first reason, thus in Step 2, PCB can be placed in any position freely, and these optimization designs of PCB need not repeat the calculation of cushion materials again. It provides efficiency and flexibility to this method under a reasonable accuracy.

3.2.3 Comparison of Methods

In order to further understand virtual boundary method, the difference between virtual boundary method and global/local method is given in Table 3.2 and Figure 3.5. A case study considered in the present work is used to illustrate the efficiency of virtual boundary method. Note that the operation of calculation time quantity cannot represent exact calculation time on computer in reality, but it can be used to compare calculation time qualitatively. As can be seen in Figure 3.5, the main difference between the two approaches is derived by the fact that the sub-models are combined in different stages. In virtual boundary method, pre-processing and optimization design processing are relatively independent of each other, hence the calculation time on physical buffers is saved in post-processing stage. As compared to global/local method, physical buffers are analyzed repetitively, making it a time-consuming process. The efficiency of virtual boundary method relative to global/local method is described by:

$$\text{Efficiency}_{\text{GL-VB}} = \frac{T_{\text{GL-optimization}}}{T_{\text{VB-optimization}}} = \frac{223n}{103 + 83n} = \frac{223}{103/n + 83} \quad (3.1)$$

which illustrates that with the increasing design's optimization times, n , the virtual boundary method is more effective than global/local method.

3.2 Methodology of Virtual Boundary Method

81

$$T_{VBI} = 103 + 83 = 186 \text{ units}$$

$$T_{VB_optimization} = 103 + 83n \text{ units}$$

(a) Virtual boundary method

$$T_{GLI} = 143 + 80 = 223 \text{ units}$$

$$T_{GL_optimization} = T_1 \times n = 223n \text{ units}$$

(b) Global/local method

*Note: refer to Table 2 for Models A, B, C, D and the associated time units

Figure 3.5: Comparison of computation time between virtual boundary and global/local methods

Table 3.2: Comparison of different meshed models in TV structure

	Model A (buffer only)	Model B (cabinet and tube only)	Model C (PCB only with coarse mesh)	Model D (PCB only with fine mesh)
Geometric Entities	Buffer	TV cabinet and tube	PCBcoarse	PCBfine
Mesh quality	Coarse mesh	Coarse mesh	Coarse mesh	Fine mesh
Calculation time quantity	100 units*	3 units	40 units	80 units

*The calculation time quantity on buffer is referred to as 100 units

3.3 Application of Virtual Boundary Method in Virtual Buffer Model

As mentioned in Section 3.2, in the analysis of packaged electronic products, virtual boundary method will be used in virtual buffer model. The stages are shown in Figure 3.3, there are two steps: (1) establish virtual buffer model; (2) create objective model and place it into virtual buffer model.

3.3.1 Establishment of Virtual Buffer Model

Establishment of virtual buffer model is the first step. As described previously, in order to establish a virtual buffer model, boundary field should be obtained first. In this study, the displacement field is the boundary field that is extracted from physical buffer model. Also, the entity part of virtual buffer model needs to be extracted from physical buffer model. The following are the stages in the establishment of virtual buffer model, and the stages are graphically shown in Figure 3.6.

Creation and Solution of Physical Buffer Model

The physical buffer model is the combination of TV cabinet, glass tube and cushion buffers. All the drawings were created using 3D CAD software, Pro/E. The geometries (lines and surfaces) were then imported into commercial FE software, ANSYS, where the bottom drop simulation is conducted, as shown in Figure 3.1. It is made up entirely with solid element type, SOLID164, except for TV cabinet, which adopted shell element type, SHELL163 in ANSYS. The complete system model contains 1893 shell elements, 30332 solid elements and 14246 nodes. Among them, buffer has 26666 solid elements. The mass of buffers is 1.2 kg (with a density of 35 kg/m^3). The displacement on the bottom cabinet (see Figure 3.4) is the important area of concern when the bottom buffers first hit floor. The initial

Figure 3.6: Establishment of virtual buffer model

velocity when the set hits ground is defined as 2.42 m/s, which is equivalent to a 0.3 m free drop, and a solution time of 60 ms was input.

The main objective of this step is to obtain the displacement field of nodes along the contact area of TV structure; the displacement will be used to replace real buffers that are removed in virtual buffer model. As shown in Figure 3.6, the contact area is the region where bottom cabinet contacts the ribs of bottom buffers, and the arrows represent the displacement field from physical buffer model's simulation.

Extraction and Interpolation of Displacement Field

After the simulation of physical buffer model as Figure 3.6(a), the displacement field along boundary is obtained; remove buffers from physical buffer model, TV structure model is obtained as shown in Figure 3.6(b). TV structure is the entity part of virtual buffer model. After applying the displacement field to the contact regions as shown in Figure 3.6(c), boundary field and virtual buffer entity model are combined. At the end of this step, virtual buffer model is established as shown in Figure 3.6(d). In the following work, all the analysis on objective model (PCB) can be processed in virtual buffer model.

3.3.2 Model Verification

The focus of this section is to determine the reliability and effectiveness of the virtual buffers concept based on bottom drop simulation. As such, the bottom drop analysis will be discussed in detail and comparisons between physical buffers and virtual buffer models will be carried out. To compare the accuracy of two models, a "true" solution is required. Since an analytical solution cannot be found for this problem, the FE solution of physical buffer model is used as "basis" solution [50]. Basically, three parameters are identified for comparison and they are: (i) deflection, δ ; (ii) stress, ε ; and (iii) computational time and space. Logically,

the largest deformation and high stress concentration will occur at the bottom of TV cabinet. Thus, retrieving the necessary results from bottom cabinet will be more appropriate and sufficient to assess the applicability of this concept.

Comparison of Deflection Results

The deflection results of eleven nodes on the bottom cabinet are extracted from both models for comparison. The locations of these nodes are shown in Figure 3.7. The behavior of bottom cabinet is expected to be symmetrical about y - z plane in Figure 3.7. Thus, deflection results on only half of cabinet are considered. The nodes are positioned in such a way so that the deflection trend along y -axis can be found. Figure 3.8 presents the deflection of nodes in physical buffer model. Note that the deflection in the first stage that is less than an absolute value of 0.1 mm is taken as zero. A percentage difference of the deflections for each point throughout entire time domain was calculated, thus:

$$\%error = \left| \frac{\delta_{physical_sol} - \delta_{virtual_sol}}{\delta_{physical_sol}} \right| \quad (3.2)$$

The percentage error distribution of entire time domain is illustrated in Figure 3.9. The error distribution is zero for the initial 6 ms because the deflection results in both models are less than 0.1 mm and are assumed to be zero. Immediately after this time, the deflections are still relatively small ($\delta \approx 0.15$ mm) and this caused error to peak. This peak arises because a small difference in deflection from the two models over a small denominator (deflection in physical buffer model) will give a reasonably high value. Therefore, the large errors reflected just after 6 ms may be misleading to suggest that the correlation is poor. As such, the correlation in this study compares error results obtainable after $t = 8$ ms where the deflections are shown in Table 3.3.

From the oscillatory behavior of percentage error distribution shown in

Figure 3.7: Locations of nodes on bottom cabinet (top view)

Figure 3.8: Deflection of nodes at specified locations in physical buffer model

Figure 3.9: Percentage error distributions of deflection for entire time domain

Table 3.3: Maximum error and time of occurrence of each node for $t > 8$ ms

	A1	A2	A3	B1	C1	C2	C3	D1	E1	E2	E3
Maximum percentage error (%)	2.1	2.45	0.34	2.15	1.2	1.3	1.1	2.35	4	3	0.1
Time of occurrence (ms)	40.4	58	57.6	49.2	49.4	44	32.8	56	56.8	57.2	44

Figure 3.9, it implies that the deflection curves of the physical and virtual buffer models are constantly "crisscrossing" each other. Moments where the deflection curves intersect, the error is minimal and vice versa. A trend in error distribution can be seen as well. The errors of all the nodes appear to be minimal at the stage of rebound at $t = 18.8$ ms. Subsequently, comparisons of Figures 3.9 and 3.10 show the errors increase gradually as the amplitudes of deflections are reduced. A summary of the maximum error of each node and time ($t > 8$ ms) in which it occurs is tabulated in Table 3.3.

Figure 3.10: Stress distribution of bottom cabinet at $t = 18.4$ ms

It is noted that the error at locations A3 and E3 is small. In fact, the error at these two locations is quite small for entire time domain unlike other nodes. This is because the displacement results of nodes A3 and E3 in virtual buffer model are in fact the displacement resulted from physical buffer model input as contact conditions. So, it is not surprising that error is small. Without considering the errors at A3 and E3, a trend in the error distribution is seen. The trend shows

that nodes nearer to the front and rear cabinet actually have higher error than those nodes along the center (along C's locations) of bottom cabinet. On the whole, comparison of deflection results on the bottom cabinet is encouraging as the maximum percentage error is only 4%.

Comparison of Stress Results

From the summary result of TV cabinet, the highest equivalent von Mises stress is obtained near to moment $t = 18.4$ ms, as shown in Figures 3.11. Figure 3.12 presents Von Mises stress concentration of two models at this instant. Comparing Figures 3.12(a) and 3.12(b), it shows that the general stress distribution of both models is quite similar. Regions of high and low stress concentration match nicely.

Figure 3.11: Equivalent von Mises stress profiles at points HS and HG given in Figure 3.10

Figure 3.12: Equivalent von Mises stress at rebound (a) physical buffer model and (b) virtual buffer model

Based on the stress distribution in Figure 3.12, it clearly indicates the majority of impact is sustained at bottom cabinet, thus a detailed contour of the stress distribution at bottom cabinet in physical buffer simulation is performed and presented in Figure 3.10. Two points of interests are selected for comparison. The first point, labeled HS in the figure, indicates area of high stress concentration but low stress gradient [27]. The second point, HG, indicates region of moderate stress concentration but high stress gradient. Based on error Equation (3.2), the percentage error distribution of about these two points in both physical and virtual buffer model is computed for entire time domain in Figure 3.13. As can be seen, the error at HG is higher than HS except for time from $14 \text{ ms} < t < 23 \text{ ms}$. The highest error at HG and HS are 16% and 10% respectively. The percentage error at HS is generally less than 5%. However, the percentage error at HG is not quite encouraging as it varies at around 10%. This finding implies that regions of high stress gradient contribute a higher degree of inaccuracy over regions of high stress concentration and low stress gradient [27].

So far, based on the comparison of deflection and stress concentration results at bottom cabinet, adopting the concept of virtual buffer for such a drop/impact simulation is able to yield relatively accurate results. This is especially true for results on deflection.

Figure 3.13: Percentage error distributions at points HS and HG given in Figure 3.10

Table 3.4: Summary of CPU time and space consumed for bottom drop simulation

	Physical buffer simulation	Virtual buffer simulation	Percentage of resources saved (%)
CPU time	4 hours 57 minutes 7 seconds	8 minutes 16 seconds	97.2
CPU space (GB)	1.96	0.99	49.5

Comparison of Computation Time and Space

The amount of CPU time and space consumed by both models in bottom drop simulation are summarized in Table 3.4. It shows that the amount of CPU time saved by adopting the concept of virtual buffer is an amazing 97.2 %. The required CPU space for simulation is almost half of physical buffer model with real buffers. This clearly demonstrates the efficiency of virtual buffer concept in conserving CPU resources and yet maintaining a relatively high degree of accuracy in the model.

3.3.3 Analysis of Objective Model

In the previous section, a virtual buffer model was set up by incorporating displacement boundary condition into TV structure. In the following step, the objective model, PCB, is created and mounted on the virtual buffer model with chassis as shown in Figure 3.14. In this section, the analysis of PCB by virtual buffer method is described. At the same time, the advantages of virtual buffer method will be presented. In the beginning stages of analysis, the PCB was located at the original position of the design provided by manufacturers.

Figure 3.14: Application of PCB into virtual buffer model

Geometric Structure of Objective Model

In this study, PCB structure is created independently from TV assembly structure shown in Figure 3.1. As shown in Figure 3.2, the PCB model consists of 537 shell elements. Through simplification, only two main chips were left on PCB board, and these chips were meshed as 132 solid elements. As a connecting component, chassis was put between PCB board and TV bottom cabinet, and it was meshed as 5548 solid elements. In Figure 3.2, the mesh of PCB had been made semi-transparent to better illustrate the chassis's structure below it.

After setting up objective model, the PCB model will be placed into virtual

buffer model as shown in Figure 3.14. Also, their connection relationship is shown in Figure 3.15 at detail. As shown in Figure 3.15, through stoppers of chassis, the PCB board is fixed on chassis; through joints, the chassis is mounted on TV cabinet. With the help of chassis connection, PCB was mounted on TV cabinet at last.

Figure 3.15: Connection of objective model with virtual buffer model

At the end of this step, the objective model was created and inserted into virtual buffer model successfully. As the virtual buffer model only cost about 8 minutes CPU time as shown in previous section, in the following simulation, almost all the CPU space will be spent on objective model.

Analysis and Optimization of Objective Model

To study drop/impact behavior of PCB under virtual buffer method, results are retrieved from Node 5681 located at the central portion of PCB. Although the nature of drop phenomenon cannot be determined with complete certainty based on results retrieved at a single point, a good estimate can be made. Figure 3.16 illustrates the location of central Node 5681. The time history of deflection U_y of Node 5681 is shown in Figure 3.17, where U_y means the deflection along

drop direction, y . Figure 3.17 shows a negative and positive peak, indicating PCB experiences a large degree of downward and upward flexing. The large degree of downward flexing of -10.9 mm at $t = 10$ ms is expected due to the inertial forces acting downward. On the other hand, at the moment of another large amount of upward flexing of 5.15 mm at $t = 25.6$ ms, the knocking of PCB against chassis, which caused PCB to rebound and flex significantly upward. Figure 3.18 clearly illustrates damaged stoppers on chassis at $t = 25.6$ ms. The PCB at this instant has detached itself from the chassis and is actually flexing upward. The state of bending of PCB in longitudinal and transverse directions is also calculated.

Figure 3.16: Von Mises stress contour of PCB at $t = 8.8$ ms (rebound moment illustrated in Figure 3.15)

The drop/impact performance reliability of the PCB depends largely on the survivability of the electronic components that are mounted on it. Therefore, a study of the stress distribution in PCB has been carried out to investigate this area. It was noted that the maximum equivalent von Mises stress was retrieved at the occurrence of rebound. So logically, the initial search path for the maximum

Figure 3.17: Deflection of central Node 5681 (PCB's central)

Figure 3.18: Screen capture of PCB y -translation results at $t = 25.6$ ms

Table 3.5: Summary of the responses of PCB in four designed positions

	Position 1		Position 2		Position 3		Position 4	
Node	4689	5681	4689	5681	4689	5681	4689	5681
Max deflection (mm)	5.9	9.5	4.2	7.3	5.8	8.3	4.5	6.4
Max stress (MPa)	18	4.5	12	3.8	19	5	15	4.2

stress concentration in PCB should be at the time of rebound. The equivalent von Mises stress contour of PCB at time of rebound is presented in Figure 3.16. As can be seen in Figure 3.16, the highest stress concentration in PCB is found at a region where the edges of two components are closest, and Node 4689 is located at the component's edge. Therefore in following optimization design, the deflection and stress on central node 5681 and high stress node 4689 are selected to evaluate the reliability of PCB design. Due to the use of virtual buffer method, physical buffer model need not be calculated again. The time used to evaluate optimization is reduced greatly. In this study, three different positions of PCB on bottom cabinet from original model are provided as optimization plan, as shown in Figure 3.19. Among them, the positions are named as P1, P2, P3 and P4. Location P1 is the original position while P2, P3 and P4 are optimization plans. After analyzing PCB in virtual buffer model, we can evaluate optimization plans through comparing the stress and deflection of key points, Node 5681 and Node 4689 in those 4 conditions respectively. The comparisons are presented in Figures 3.20, 3.21, 3.22 and 3.23. Finally, all the results are summarized from four group curves in Table 3.5. From Table 3.5, Positions 2 and 4 are proven to be successful in three optimization plans.

3.4 Summary of Chapter

A virtual boundary method has been described and used for analysis of a PCB in a packaged TV model as the form of virtual buffer model. The method provided an effective optimization design to PCB's position and speed up optimization process

Figure 3.19: Schematic of PCB in four different positions

Figure 3.20: Comparison of deflection at Node 4689 (point of high stress)

Figure 3.21: Comparison of Von Mises stress at Node 4689 (point of high stress)

Figure 3.22: Comparison of deflection at Node 5681 (PCB's central)

Figure 3.23: Comparison of Von Mises stress at Node 5681 (PCB's central)

greatly. Note that the methodology can be extended to other areas outside of electronic devices components' design when the physical boundary model outside of concerned parts consumed too much calculation space in FEA field, such as hydro analysis or thermal analysis in mega boundary. And because this method is flexible and effective, which has been illustrated in this chapter, it will provide a good solution to engineers in many fields.

The electronic components in modern electronic devices are usually sensitive and becoming more and more tiny. In such cases, the use of virtual buffers appears to be useful in reality. This method is especially suited for the optimization design of tiny structures. In these cases, virtual boundary method will become much more effective than traditional GL method. However, there are some restrictions to this method as tiny structure's influence to the whole model is ignored. Thus, the error is larger as tiny structure becomes bigger. Even though the tiny structure is not small enough, this method can at least provide an approximate optimization design to engineers.

3.5 Development of Full-Scale Virtual Buffer Model

The initial part of this PhD work is performing physical tests for Philips Company. Some drop tests were done in the drop-impact of TV sets, including side drop, base drop, front drop and back drop [26][28]. This work is time consuming and expensive, especially when the optimization design is performed. Since the limitations and undesirable results of physical tests as concluded in Section 2.1, we tried to use simulation software in this study. At first, the model is put into simulation without using any efficient method. In order to simulate 40 *ms* in visualization, it takes about 40 days in Sun Workstation that has only 526M EMS memory and 360 MHz CPU speed. Although all the possible simplification is performed, it is still too time-consuming to be acceptable. The use of a high performance computer will be helpful. As mentioned in Section 2.2, in a similar project, Wang et al. [29] used a supercomputer with SGI Origin 2000 system, CPU time for TV drop test was still about 70 *h* for the duration of 25 *ms*. The duration, element size, material nonlinearly, etc. are all affected by CPU time. Thus control of simulation is an important issue, this work needs a high requirement for simulation engineers. Therefore, some quick design methods are considered into application. As one of the most popular efficient method, global-local method is selected to analyze tiny components in TV set [26]. During analysis, global/local expressed its advantages in large-scale model design: tiny components in complex model are investigated in detail in shorter calculation time. At the same time, it is realized that the most contribution of global/local method comes from the method of dividing the whole analysis into pre-process and post-process. The object of pre-process is to provide boundary field to the analysis of concerned models in post-process. If boundary field can be well calculated by using less computation resource in pre-process, then much CPU space can be saved without lost of simulation accuracy.

How can we combine FE simulation and methods without FE simulation? What is the benefit from it? The answer is in the first step of virtual buffer method, virtual buffer model is used to output boundary field without detailed modeling

work for objective model. This provides possibility to simplify virtual buffer model into a single degree of freedom (SDOF) drop-impact system. The dynamic response of this SDOF system can be calculated by other numerical or analytic methods rather than simulation [59][92][93]. Then all the simulation resource is focused on the analysis of the concerned parts. We call this hybrid virtual buffer model as full-scale virtual buffer model. The process of full-scale virtual buffer model is shown in Figure 3.24. A SDOF drop impact system will be used to establish virtual buffer model and to output boundary field, which will next be applied on the boundary of objective model for the following detailed analysis. The concerned components in this work are multilayer circuit boards.

Figure 3.24: Flow chart of full-scale virtual buffer model

Chapter 4

Further Development of Virtual Buffer Model by Using a Local Linearization Method for Quick Extrication of Boundary Information in Nonlinear Cushion Systems

As described in Figures 3.24, in order to establish a full-scale virtual buffer model, an analytical virtual buffer model is proposed to simulate buffer effect to output boundary field. Boundary field is dynamic response of cushion buffers in drop impact condition. In cushion buffer systems, nonlinear characteristic is too important to be ignored. This nonlinear characteristic of a cushion buffer would affect crashworthiness of packaged goods. Therefore, the prediction of drop-impact response of nonlinear systems becomes one of the important concerns in the establishment of virtual buffer model.

In the description of natural phenomena, the process of linearization has become an intrinsic part of rational analysis of physical problems. Quantitative solution and easy solubility provide high practicality for this methodology. However, in many instances traditional linear analysis is insufficient to adequately describe nonlinear behavior of physical system. Therefore, nonlinear analysis is of main concern in this case. Meanwhile, quantitative analysis in nonlinear system is not easy to obtain, because there are some difficulties to exactly describe and solve physical nonlinear characteristic by nonlinear ordinary differential equations. In this chapter, a local linearization method is proposed to obtain quantitative solution in nonlinear analysis practically.

A local linearization method will be applied in nonlinear cushion system that illustrates strong nonlinear relationship between spring force and deformation. Dynamic loading will first be calculated by local linearization method. Transient response is then obtained with the same process. Nonlinear characterization of cushion buffers is studied. The effect of model simplification is also discussed. For consolidation, drop-impact tests based on ASTM standard are performed to verify local linearization model, and some mature analytical and numerical methods are provided for the same purpose. The results from local linearization model have been shown to be accurate in comparison.

4.1 Introduction of Chapter

Non-linearity is ubiquitous in nature, while linearity is an approximation to reality [94]. However, in many instances traditional linear analysis is insufficient to describe nonlinear behavior of physical system adequately [95]. Therefore, nonlinear analysis is required in most studies. In nonlinear analysis, various vibration equations were developed for dynamic analysis of single-degree-of-freedom (SDOF) system, namely, Duffing equation [96][97][59], Lienard equation, Hill equation, Mathieu equation, etc. [98]. In practical usage, curve-fitting work has to be done first to

establish these equations, after that large-scale numerical computations or calculus of high transcendental functions are usually performed, such as Runge-Kutta method [99] or elliptical function [59][60]. Quantitative solution, however, is not easy to be obtained based on these methods [100].

In this study, practical application of quantitative nonlinear analysis will be performed in polystyrene foam system that has strong nonlinear stress-strain property [101]. In previous research, the simplification of cushion buffers into a linear spring will underestimate cushion effect greatly in drop impact case [93][59]. Therefore, nonlinear analysis becomes critical in cushion design and product reliability evaluation, and high accuracy is needed also. In order to achieve this objective, a local linearization method is proposed as quantitative analytical method. In this method, qualitative curve-fitting formula for the description of nonlinear characteristic is not required. Through extracting local calculation time parameters in nonlinear deformation process, acceleration at each deformation point can be extracted individually, thus large-scale numerical iterative calculation can be avoided. Furthermore, maximum acceleration and maximum deformation can be quantitatively and quickly calculated without transient response calculation. This is quite useful for industry application when dynamic loading (maximum acceleration and maximum deformation) is only needed in drop impact analysis [9].

In this chapter, the methodology of proposed local linearization method is described first. With this method, quantitative analysis of dynamic loading in nonlinear system is presented. Dynamic loading will then be used as boundary conditions to calculate transient response in whole drop impact processes. The accuracy of local linearization method will be verified by comparison with physical drop impact experiments. Its practical usage is also concluded in real application. Finally, further discussions are performed for soft nonlinear characteristic of cushion material, computation methodology in local linearization model and damping ratio characteristic in compressing polystyrene foam.

4.2 Methodology of Local Linearization Method

In this work, a local linearization method is proposed to obtain a quantitative solution for nonlinear systems, which is different to numerical nonlinear analysis methods [100][102]. This method uses the common point between linear and nonlinear analysis: differential relationship between displacement, velocity and acceleration at a transient instant [96][97][61]. Based on this point, linear differential relationship is applicable in nonlinear model at a transient displacement point. Therefore, nonlinear case also can be solved by using quantitative analytical calculation as linear system.

A spring force with nonlinear stiffness characteristic curve is shown in Figure 4.1. An arbitrary deformation point x_i is selected, the corresponding spring force is $F(x_i)$. At this deformation point, the stiffness coefficient at this deformation point x_i is defined as

$$K(x_i) = \tan^{-1} \theta = \frac{F(x_i)}{x_i} \quad (4.1)$$

where θ is force-displacement gradient shown in Figures 4.1, this value varies at different deformation point x_i , $K(x_i)$ is a variable in nonlinear stiffness system. The differential equation of motion of a viscously damped SDOF system can be written as

$$\ddot{x}_i + 2\zeta\omega(x_i)\dot{x}_i + \omega^2(x_i)x_i = \frac{F_0}{M} \quad (4.2)$$

$$\omega(x_i) = \sqrt{\frac{K(x_i)}{M}} \quad (4.3)$$

in which $2\zeta\omega(x_i)$ and $\omega^2(x_i)$ are functions of system parameters, ζ damping factor, and M mass of system. F_0 external force applied on spring system, if there is only

gravity exist, F_0 is equal to Mg . undamped natural frequency $\omega(x_i)$ is a variable with respect to x_i . Based on linear/nonlinear sameness: linear dynamic differential exists at a deformation point in nonlinear system, a linear equation of motion at arbitrary local deformation point x_i can be obtained,

$$\ddot{\bar{x}} + 2\zeta\bar{\omega}\dot{\bar{x}} + \bar{\omega}^2\bar{x} = \frac{F_0}{M} \quad (4.4)$$

$$\bar{\omega} = \sqrt{\frac{\bar{K}}{M}} \quad (4.5)$$

where all the parameters with upper bar is taken as linear operation parameters at a local deformation point x_i . Therefore, the general solution for ordinary differential equation can be expressed as

$$\bar{x} = e^{-\zeta\bar{\omega}t_{local}}(D_1e^{\bar{\omega}t_{local}\sqrt{\zeta^2-1}} + D_2e^{-\bar{\omega}t_{local}\sqrt{\zeta^2-1}}) + \frac{F_0}{M\bar{\omega}^2} \quad (4.6)$$

$$\begin{aligned} \dot{\bar{x}} = & D_1(-\zeta\bar{\omega} + \bar{\omega}\sqrt{\zeta^2-1})e^{-\zeta\bar{\omega}t_{local}+\bar{\omega}t_{local}\sqrt{\zeta^2-1}} \\ & + D_2(-\zeta\bar{\omega} - \bar{\omega}\sqrt{\zeta^2-1})e^{-\zeta\bar{\omega}t_{local}-\bar{\omega}t_{local}\sqrt{\zeta^2-1}} \end{aligned} \quad (4.7)$$

$$\begin{aligned} \ddot{\bar{x}} = & D_1(-\zeta\bar{\omega} + \bar{\omega}\sqrt{\zeta^2-1})^2e^{-\zeta\bar{\omega}t_{local}+\bar{\omega}t_{local}\sqrt{\zeta^2-1}} \\ & + D_2(\zeta\bar{\omega} + \bar{\omega}\sqrt{\zeta^2-1})^2e^{-\zeta\bar{\omega}t_{local}-\bar{\omega}t_{local}\sqrt{\zeta^2-1}} \end{aligned} \quad (4.8)$$

where D_1 and D_2 are arbitrary constants that is defined by initial conditions. Since this solution is only available at the local deformation point, time parameter t_{local} cannot cover the whole nonlinear vibration progress. This parameter only can be used to describe local calculation time. In order to avoid confusion in following

analysis, local time parameter is replaced by local calculation time τ_i as

$$t_{local} = \tau_i \quad (4.9)$$

where τ_i is set as the local calculation time at local deformation point x_i . Next, by converting Equations (4.6) to (4.8) back to variable coefficient form, motion of whole nonlinear system is expressed as

$$x_i = D_1 e^{-\zeta\omega(x_i)\tau_i + \omega(x_i)\tau_i\sqrt{\zeta^2-1}} + D_2 e^{-\zeta\omega(x_i)\tau_i - \omega(x_i)\tau_i\sqrt{\zeta^2-1}} + \frac{F_0}{M\omega^2(x_i)} \quad (4.10)$$

$$\dot{x}_i = D_1 \left[-\zeta\omega(x_i) + \omega(x_i)\sqrt{\zeta^2-1} \right] e^{-\zeta\omega(x_i)\tau_i + \omega(x_i)\tau_i\sqrt{\zeta^2-1}} + D_2 \left[-\zeta\omega(x_i) - \omega(x_i)\sqrt{\zeta^2-1} \right] e^{-\zeta\omega(x_i)\tau_i - \omega(x_i)\tau_i\sqrt{\zeta^2-1}} \quad (4.11)$$

$$\ddot{x}_i = D_1 \left[-\zeta\omega(x_i) + \omega(x_i)\sqrt{\zeta^2-1} \right]^2 e^{-\zeta\omega(x_i)\tau_i + \omega(x_i)\tau_i\sqrt{\zeta^2-1}} + D_2 \left[\zeta\omega(x_i) + \omega(x_i)\sqrt{\zeta^2-1} \right]^2 e^{-\zeta\omega(x_i)\tau_i - \omega(x_i)\tau_i\sqrt{\zeta^2-1}} \quad (4.12)$$

Equation (4.12) can be expressed furthermore as:

$$\ddot{x}_i = g(\omega(x_i), \tau_i) = f(K(x_i), \tau_i) \quad (4.13)$$

It is found from Equation (4.13) that the acceleration at an arbitrary deformation point x_i can be expressed in terms of variable stiffness $K(x_i)$ and local calculation time τ_i . This is algorithm of local linearization method as depicted in Figure 4.2: local calculation time τ_i and stiffness coefficient $K(x_i)$ are two necessary conditions for nonlinear analysis with local linearization method.

Figure 4.1: Spring force characteristic curve for a strong nonlinear vibration system

Figure 4.2: Overview of local linearization method

4.3 Mathematical Analysis

In this section, drop impact analysis of a nonlinear cushion system will be performed by using local linearization method. The general algorithm shown in Figure 4.2 is the backbone in following analysis. There are four steps: (1) establish linearized different equation of motion for nonlinear systems $f(K(x_i), \tau_i)$; (2) calculate calculation time τ_i ; (3) extract stiffness at local deformation point $K(x_i)$; (4) substitute results obtained in Steps 2 and 3 into Step1, acceleration can be achieved $\ddot{x}_i = f(K(x_i), \tau_i)$.

4.3.1 Linearized Differential Equation of Motion for Nonlinear Systems

Drop-impact response of drop item with nonlinear cushion can be simplified as a SDOF drop-impact system as shown in Figure 4.3. The system includes drop items and cushion buffer. It is established when cushion buffer initially contact ground, cushion buffer is idealized as spring which connect drop item and fixed ground. In this figure, M is mass of drop item and cushion, K_i is stiffness coefficient of spring-like cushion buffer, c_i is damping coefficient. The stress-strain curve of cushion material is shown in Figure 4.4, which follows cushion properties of EPS20 [40]. The following equations will help to transfer cushion properties to spring properties for calculation:

$$F = F(\varepsilon_i) = \sigma(\varepsilon_i)A \quad (4.14)$$

$$x_i = \varepsilon_i b \quad (4.15)$$

$$K_i = K(x_i) = \frac{F(x_i)}{x_i} = \frac{\sigma(\varepsilon_i)A}{\varepsilon_i b} = K(\varepsilon_i) \quad (4.16)$$

where F = spring force applied on cushion buffer

$\sigma(\varepsilon_i)$ = stress of cushion buffer

A = contact area between cushion buffer and drop item

x_i = deformation of cushion buffer along drop direction

ε_i = strain of cushion buffer

b = undeformed thickness of cushion buffer

Figure 4.3: Single-degree-of-freedom drop-impact system

initial displacement and velocity conditions of the SDOF system in Figure 4.3 are given by

$$x_i(0) = 0 \quad \text{and} \quad \dot{x}_i(0) = \dot{x}_{\max} = \sqrt{2gH} \quad (4.17)$$

where g is acceleration of gravity, H is free-drop height as shown in Figure 4.3. Equation of motion of SDOF system can be described by Equations (4.2) and (4.3). The equation of motion can be written as

$$\ddot{x}_i + 2\zeta\omega(\varepsilon_i)\dot{x}_i + \omega^2(\varepsilon_i)x_i = g \quad (4.18)$$

Figure 4.4: Stress-strain curve and strain energy for cushion buffer

$$\omega(\varepsilon_i) = \sqrt{\frac{K(\varepsilon_i)}{M}} \quad (4.19)$$

in Equation (4.18), damping ratio is assumed to be a constant less than 1, $\zeta < 1$, it is common in many machines and structural systems [42]. Based on local linearization method, Equation (4.18) is solved as

$$x_i = X_i e^{-\zeta \omega(\varepsilon_i) \tau_i} \sin \left[\omega(\varepsilon_i) \tau_i \sqrt{1 - \zeta^2} + \phi \right] + \frac{Mg}{K(\varepsilon_i)} \quad (4.20)$$

$$\dot{x}_i = X_i \omega_i(\varepsilon_i) e^{-\zeta \omega(\varepsilon_i) \tau_i} \cos \left[\omega(\varepsilon_i) \tau_i \sqrt{1 - \zeta^2} + \phi + \phi_1 \right] \quad (4.21)$$

$$\ddot{x}_i = -X_i \omega_i^2(\varepsilon_i) e^{-\zeta \omega(\varepsilon_i) \tau_i} \sin \left[\omega(\varepsilon_i) \tau_i \sqrt{1 - \zeta^2} + \phi + 2\phi_1 \right] \quad (4.22)$$

where

$$X_i = \sqrt{\frac{Mg}{K(\varepsilon_i)(1 - \zeta^2)} \left(\frac{Mg}{K(\varepsilon_i)} + 2H - \frac{2\zeta \sqrt{2MgHK(\varepsilon_i)}}{K(\varepsilon_i)} \right)} \quad (4.23)$$

$$\phi = \tan^{-1} \left(\frac{\sqrt{(1 - \zeta^2)Mg}}{\zeta \sqrt{Mg} - \sqrt{2HK(\varepsilon_i)}} \right) \quad (4.24)$$

$$\phi_1 = \tan^{-1} \left(\frac{\zeta}{\sqrt{1 - \zeta^2}} \right) \quad (4.25)$$

The above equations are the solution of linearized differential equations of motion as mentioned in Figure 4.2. Substitution of calculation time τ_i and stiffness coefficients $K(\varepsilon_i)$ into these solutions will produce dynamic response in nonlinear system. In following analysis, dynamic loading and transient response calculations will be presented individually.

4.3.2 Dynamic Loading Analysis of Damped Nonlinear Systems

Dynamic loading includes maximum acceleration and maximum deformation that occur at the instant of rebound. With help of local linearization method, analysis in nonlinear system can be performed quickly.

At the instant of rebound, velocity is zero, while the acceleration and deformation are assumed to achieve maximum value. Local instant parameter and stiffness coefficient are two necessary conditions for nonlinear analysis.

calculation time Parameter at the Instant of Rebound

At the instant of rebound, velocity is zero:

$$\begin{aligned} \dot{x}_{rebound} &= X_{rebound}\omega_{rebound}(\varepsilon_{rebound})e^{-\zeta\omega(\varepsilon_{rebound})\tau_{rebound}} \\ &\times \cos \left[\omega(\varepsilon_{rebound})\tau_{rebound}\sqrt{1-\zeta^2} + \phi + \phi_1 \right] = 0 \end{aligned} \quad (4.26)$$

where $\tau_{rebound}$ is local calculation time at rebound instant, it can be solved as

$$\omega(\varepsilon_{rebound})\tau_{rebound}\sqrt{1-\zeta^2} + \phi + \phi_1 = \frac{\pi}{2} + (n-1)\pi, \quad n = 1, 2, \dots, N \quad (4.27)$$

or

$$\tau_{rebound} = \frac{\frac{\pi}{2} + (n-1)\pi - \phi - \phi_1}{\omega(\varepsilon_{rebound})\sqrt{1-\zeta^2}}, \quad n = 1, 2, \dots, N \quad (4.28)$$

where n is number of cycles for oscillation. It was found that maximum acceleration occurred in the first cycle in a free fall drop process under the protection of cushion buffers [1]. Therefore, in our study, it is enough only to consider first cycle ($n=1$). Hence, it is found

$$\tau_{rebound} = \frac{\frac{\pi}{2} - \phi - \phi_1}{\omega(\varepsilon_{rebound})\sqrt{1-\zeta^2}} \quad (4.29)$$

Stiffness Coefficient at the Instant of Rebound

At the instant of maximum deflection and acceleration, there is an energy conservation at a moment when kinetic energy at initial condition is completely transferred to deformation energy and energy dissipated by viscous damping.

$$\frac{1}{2}M\dot{x}^2(0) + Mgx_{rebound} = \int_0^{x_{rebound}} K(x)xdx + E_d \quad (4.30)$$

where E_d is energy dissipated by damping, Mg_0x_{rebound} is potential energy caused by the deformation of impressed cushion. Substituting Equations (4.17) into (4.30), it is obtained

$$MgH + Mg x_{\text{rebound}} = \int_0^{x_{\text{rebound}}} K(x)x dx + E_d \quad (4.31)$$

By virtue of Equations (4.15), (4.16) and (4.31), it can be obtained

$$\int_0^{\varepsilon_{\text{rebound}}} \sigma(\varepsilon) d\varepsilon = \frac{WH}{Ab} + \frac{Wb\varepsilon_{\text{rebound}}}{Ab} - \frac{E_d}{Ab} \quad (4.32)$$

where W is weight of system, which is equal to Mg . $E_i = \int_0^{\varepsilon_i} \sigma(\varepsilon) \times d\varepsilon$ is defined as strain energy of buffer element as shown in Figure 4.4, the area below stress-strain curve is taken as strain energy. The definition $\int_0^{\varepsilon_{\text{rebound}}} \sigma(\varepsilon) d\varepsilon = E_{\text{rebound}}$ is the strain energy at the instant of rebound. Therefore, Equation (4.32) can be rewritten as

$$E_{\text{rebound}} = \frac{WH}{Ab} + \frac{Wb\varepsilon_{\text{rebound}}}{Ab} - \frac{E_d}{Ab} \quad (4.33)$$

in which E_d is an unknown parameter that can be evaluated by

$$E_d = \int_0^{x_{\text{rebound}}} c\dot{x} dx = \int_0^{\tau_{\text{rebound}}} c\dot{x} \frac{dx}{d\tau} d\tau = \int_0^{\tau_{\text{rebound}}} c\dot{x}^2 d\tau \quad (4.34)$$

where c is damping coefficient,

$$c(\varepsilon_i) = 2\zeta \sqrt{K(\varepsilon_i)M} \quad (4.35)$$

By substituting Equations (4.21) and (4.35) into (4.34), energy dissipation can be expressed as follows:

$$\begin{aligned}
 E_d &= \int_0^{\tau_{\text{rebound}}} \zeta X^2 \omega^2(\varepsilon) \sqrt{K(\varepsilon)M} \left\{ e^{-2\zeta\omega(\varepsilon)\tau} \left[1 + \cos(2\omega(\varepsilon)\tau \sqrt{1-\zeta^2} + 2\phi + 2\phi_1) \right] \right\} d\tau \\
 &= \zeta X^2 \omega^2(\varepsilon) \sqrt{K(\varepsilon)M} \int_0^{\tau_{\text{rebound}}} e^{-2\zeta\omega(\varepsilon)\tau} \left[1 + \cos(2\omega(\varepsilon)\tau \sqrt{1-\zeta^2} + 2\phi + 2\phi_1) \right] d\tau \quad (4.36) \\
 &= \frac{K(\varepsilon_{\text{rebound}})}{2} (Y_1^2 + Y_2^2) [1 - e^O + \zeta^2 e^O - \zeta \sin(2\phi + \phi_1)]
 \end{aligned}$$

where

$$Y_1 = -\frac{Mg}{K(\varepsilon_{\text{rebound}})} \quad (4.37)$$

$$Y_2 = \sqrt{\frac{Mg}{K(\varepsilon_{\text{rebound}})(1-\zeta^2)}} \left(\sqrt{2H} - \zeta \sqrt{\frac{Mg}{K(\varepsilon_{\text{rebound}})}} \right) \quad (4.38)$$

$$O = -\frac{\zeta}{\sqrt{1-\zeta^2}} (\pi - 2\phi - 2\phi_1) \quad (4.39)$$

Since $H \gg \frac{Mg}{K(\varepsilon_{\text{rebound}})}$, $\frac{Mg}{K(\varepsilon_{\text{rebound}})}$ can be ignored. And after implement of ϕ and ϕ_1 from Equations (4.24) and (4.25), Equation (4.36) is simplified as

$$E_d = WH \left[1 - e^{-\frac{\zeta}{\sqrt{1-\zeta^2}} (\pi - 2\phi_1)} \right] \quad (4.40)$$

Next, substitution of Equations (4.40) into (4.33) yields the strain energy at instant of rebound,

$$\begin{aligned}
 E_{\text{rebound}} &= \int_0^{\varepsilon_{\text{rebound}}} \sigma(\varepsilon) \times d\varepsilon = \frac{WH}{Ad} - \frac{E_d}{Ad} + \frac{W\varepsilon_{\text{rebound}}}{A} \\
 &= \frac{WH}{Ad} e^{-\frac{\zeta}{\sqrt{1-\zeta^2}} (\pi - 2\phi_1)} + \left(\frac{W}{A} \right) \varepsilon_{\text{rebound}} \quad (4.41)
 \end{aligned}$$

where (W/A) is static load (SL). Equation (4.41) can be rearranged as

$$\begin{aligned} \int_0^{\varepsilon_{\text{rebound}}} \sigma(\varepsilon) \times d\varepsilon - \left(\frac{W}{A}\right) \varepsilon_{\text{rebound}} &= \frac{WH}{Ad} - \frac{E_d}{Ad} \\ &= \frac{WH}{Ad} e^{-\frac{\zeta}{\sqrt{1-\zeta^2}}(\pi-2\phi_1)} = U_{\text{rebound}} \end{aligned} \quad (4.42)$$

in which, the left part $\int_0^{\varepsilon_i} \sigma(\varepsilon) \times d\varepsilon - (W/A) \varepsilon_i$ is set as static load (SL) strain energy U_i shown in Figure 4.5.

In Equation (4.42), the value of $\frac{WH}{Ad} e^{-\frac{\zeta}{\sqrt{1-\zeta^2}}(\pi-2\phi_1)}$ is calculated first, then rebound strain $\varepsilon_{\text{rebound}}$ can be found through SL strain energy curve in Figure 4.5. After that, rebound stress σ_{rebound} is obtained using stress-strain curve in Figure 4.4. Finally, by substituting strain $\varepsilon_{\text{rebound}}$ and stress σ_{rebound} obtained into Equation (4.16), the stiffness at instant of rebound can be determined. Based on the characteristic of rebound instant, acceleration and compressed strain achieve their peak at the same time, there exists

$$\varepsilon_{\text{rebound}} = \varepsilon_{\text{max}} \quad (4.43)$$

It is found from analysis that: beside stress-strain relationship, position of rebound instant is defined by static load (W/A) and the ratio of drop height and cushion thickness (H/b) .

Maximum Acceleration

In above two sections, local calculation time and stiffness coefficient at the instant of rebound are obtained individually. As illustrated in Figure 4.2, local instant parameter and stiffness coefficient will be substituted into local linearized differential equation of motion to obtain acceleration. By substituting Equations (4.29)

into (4.22), it is obtained

$$\ddot{x}_{\text{rebound}} = -\sqrt{g \times g + \frac{2K(\varepsilon_i)Hg}{M} - 2\zeta g \sqrt{\frac{2gHK(\varepsilon_i)}{M}}} \times \exp \left[\frac{\zeta}{\sqrt{1-\zeta^2}} \left(\phi + \phi_1 - \frac{\pi}{2} \right) \right] \quad (4.44)$$

With help of Equation (4.16), final maximum acceleration solution can be written as

$$\ddot{x}_{\text{max}} \simeq |\ddot{x}_{\text{rebound}}| = g \sqrt{1 + \frac{2 \left(\frac{\sigma(\varepsilon_{\text{rebound}})}{\varepsilon_{\text{rebound}}} \right) \left(\frac{H}{b} \right)}{\left(\frac{W}{A} \right)} - 2\zeta \sqrt{\frac{2 \left(\frac{\sigma(\varepsilon_{\text{rebound}})}{\varepsilon_{\text{rebound}}} \right) \left(\frac{H}{b} \right)}{\left(\frac{W}{A} \right)}}} \times \exp \left[\frac{\zeta}{\sqrt{1-\zeta^2}} \left(\phi + \phi_1 - \frac{\pi}{2} \right) \right] \quad (4.45)$$

where

$$\phi = \tan^{-1} \left[\frac{\sqrt{1-\zeta^2} \sqrt{\left(\frac{W}{A} \right)}}{\zeta \sqrt{\left(\frac{W}{A} \right)} - \sqrt{2 \left(\frac{\sigma(\varepsilon_{\text{rebound}})}{\varepsilon_{\text{rebound}}} \right) \left(\frac{H}{b} \right)}} \right] \quad (4.46)$$

It is found from above analysis that the values $\left(\frac{W}{A} \right)$ and $\left(\frac{H}{b} \right)$ can be used to define maximum acceleration. This is the reason why cushion curve can be generated based on these two coefficients [9][39].

4.3.3 Transient Response Analysis of Damped Nonlinear Systems

Compared to dynamic loading analysis, transient response analysis is more time-consuming, since the local calculation time at each deformation point has to be calculated.

In industry application, prediction of dynamic loading is required while

Figure 4.5: Strain energy curve and static load strain energy curve

transient response is not necessary sometimes [9][40]. That is the reason why application of cushion curve is quite practical so far. However, if dynamic response of complex model is analyzed, transient response becomes quite meaningful [58]. For instance, if shock wave is close to natural frequency of electronic components, vigorous damage will happen even if amplitude of stimulation is not significant.

In nonlinear cushion system, stimulation comes from the cushion buffer that is taken as a shock absorber. Strong nonlinear relationship between spring force and displacement in absorber will influence transient response greatly. Soft characteristic of nonlinear shock absorber not only decreases acceleration amplitude, but also increases the cycle of vibration compared to linear system [59].

In this section, local linearization method is still performed. local calculation time and stiffness coefficient at each deformation point are crucial in transient analysis. Compared to dynamic loading analysis, there are differences in transient analysis:

1. The value of velocity is used to obtain local calculation time. In dynamic loading analysis, velocity is known to be zero and it can be used directly; in

transient response analysis, velocity is unknown and energy conservation is applied for it.

2. The position of local deformation point is used to obtain stiffness coefficient. In dynamic loading analysis, the position of local deformation point is unknown and energy conservation is applied for it; in transient response analysis, the position of local deformation point is known and it can be used directly.

At arbitrary deformation points, stiffness coefficient can be obtained by Equation (4.16) directly. The energy conservation is used to determine local calculation time at arbitrary deformation points.

$$MgH + Mgx_i = \frac{1}{2}M\dot{x}_i^2 + \int_0^{x_i} K(x)xdx + E_d(x_i) \quad (4.47)$$

The velocity expression can then be written as

$$\dot{x}_i^2 = 2gH + 2gx_i - \frac{2}{M} \left(\int_0^{x_i} K(x)xdx + E_d(x_i) \right) \quad (4.48)$$

or

$$\dot{x}_i^2 = 2gH + 2g_0b\varepsilon_i - \frac{2}{M} [AbE_i + E_d(\varepsilon_i)] \quad (4.49)$$

where E_i is strain energy at arbitrary deformation point, damping energy $E_d(\varepsilon_i)$ is also a variable. Based on Equation (4.40), it is assumed to be

$$E_d = WH \left[1 - e^{-\frac{\varepsilon_i}{\varepsilon_{\max}} \times \frac{\zeta(\pi - 2\phi_1)}{\sqrt{1 - \zeta^2}}} \right] \quad (4.50)$$

where $\varepsilon_{\max}$ is the maximum strain as mentioned in Equation (4.43). Next, by

virtue of Equations (4.21), (4.49) and (4.50), local calculation time parameter τ_i can be derived as follows:

$$\begin{aligned} \dot{x}_i &= X_i \omega_i(\varepsilon_i) e^{-\zeta \omega(\varepsilon_i) \tau_i} \cos \left[\omega(\varepsilon_i) \tau_i \sqrt{1 - \zeta^2} + \phi + \phi_1 \right] \\ &= \sqrt{2gH + 2gb\varepsilon_i - \frac{2}{M_0} \left[AbE_i + WH \left(1 - e^{-\frac{\varepsilon_i}{\varepsilon_{\max}} \times \frac{\zeta(\pi - 2\phi_1)}{\sqrt{1 - \zeta^2}}} \right) \right]} \end{aligned} \quad (4.51)$$

which can be solved as

$$\tau_i = f_1(\varepsilon_i) \quad (4.52)$$

Six local calculation time parameter curves with variable ε_i are calculated through Equation (4.52) as shown in Figures 4.6 and 4.7 for example. In Figure 4.6, the weights of drop items are selected as 40 N, 280 N and 520 N, while the ratio between drop height and cushion thickness is fixed as 10. As in Figure 4.7, static load is fixed as 0.7 N/cm², while drop heights are selected as 50 cm, 80 cm and 100 cm. The cushion area is 400 cm², and its thickness is 5 cm.

Substituting local calculation time parameter and stiffness coefficient into Equation (4.22), the relationship between displacement and acceleration is established.

$$\ddot{x}_i = f(K(\varepsilon_i), \tau_i) = f(K(\varepsilon_i), f_1(\varepsilon_i)) = f_2(\varepsilon_i) \quad (4.53)$$

Till this step, relationship between displacement and time is required to solve, then transient acceleration curve can be obtained. Since velocity at each deformation point can be obtained in Equation (4.49), time parameter can be

Figure 4.6: Local calculation time parameters in transient response process with drop $h=50$ cm

Figure 4.7: Local calculation time parameters in transient response process with $W=280$ N

calculated as

$$\begin{aligned}
 t_i &= \int_0^{t_i} dt = \int_0^{x_i} \frac{dx}{\dot{x}} \\
 &= \int_0^{\varepsilon_i} \frac{b \times d\varepsilon}{\sqrt{2gH + 2gb\varepsilon_i - \frac{2}{M_0} \left[AbE_i + WH \left(1 - e^{-\frac{\varepsilon_i}{\varepsilon_{\max}} \times \frac{\zeta(\pi - 2\phi_1)}{\sqrt{1-\zeta^2}}} \right) \right]}} = t(\varepsilon_i)
 \end{aligned} \tag{4.54}$$

where $d\varepsilon$ is distance between adjacent deformation points, this value can be settled arbitrarily. Substitute Equations (4.54) to (4.53), transient acceleration can be obtained as

$$\ddot{x}_i = f_2(\varepsilon_i) = f_2(f_3^{-1}(t_i)) = f_4(t_i) \tag{4.55}$$

4.4 Application of Local Linearization Method

An example is now presented to illustrate the application of local linearization method to nonlinear cushion system. A packaged component with the protection of cushion undergoes a free drop. The following characteristics are assumed in an example: packaged component weight W of 400 N, cushion area A of 400 cm^2 , cushion thickness b of 5 cm, drop height H of 0.5 m, density of 20 kg/m^3 for EPS 20 buffer. Nonlinear stress-strain curve for EPS 20 buffer is shown in Figure 4.4. Note that strains of buffer are all less than 72 % as shown in Figure 4.4. In following work, local linearization method will be used to predict dynamic loading and transient response in nonlinear drop impact process.

4.4.1 Dynamic Loading

In dynamic loading analysis, the static surface load W/A , ratio between drop height and cushion thickness H/b , and damping ratio ζ are taken as

$$W/A = 1\text{N/cm}^2, \quad H/b = 10, \quad \text{and} \quad \zeta = 0.1 \quad (4.56)$$

Local calculation time is the result of Equation (4.29), which is substituted into acceleration equation to form Equations (4.44) and (4.45) and used in this example. Now we calculate the position of rebound instant, at first, Equations (4.25) and (4.42) are performed as

$$\phi_1 = \tan^{-1} \left(\frac{\zeta}{\sqrt{1-\zeta^2}} \right) \approx 0.1 \quad (4.57)$$

$$U_i = \frac{WH}{Ad} e^{-\frac{\zeta}{\sqrt{1-\zeta^2}}(\pi-2\phi_1)} \approx 74.41\text{kPa} \quad (4.58)$$

It is found from Figure 4.5 that the strain $\varepsilon_{\max_acc}$ according to SL strain energy of 74.41 kPa is 48 %, this is less than 46 % in pure strain energy curve. That is also the maximum strain $\varepsilon_{\max}$ based on Equation (4.43). From stress-strain curve in Figure 4.4, corresponding maximum stress $\sigma_{\max_acc}$ is 220 kPa. Substituting the value of $\varepsilon_{\max_acc}$ and $\sigma_{\max_acc}$ into Equations (4.45) and (4.46), the maximum acceleration is for

$$\begin{aligned} \phi &= \tan^{-1} \left[\frac{\sqrt{1-\zeta^2} \sqrt{\left(\frac{W}{A}\right)}}{\zeta \sqrt{\left(\frac{W}{A}\right)} - \sqrt{2 \left(\frac{\sigma(\varepsilon_{\max_acc})}{\varepsilon_{\max_acc}} \right) \left(\frac{H}{b} \right)}} \right] \\ &= -0.00329 \end{aligned} \quad (4.59)$$

to be

$$\ddot{x}_{\max} = g \sqrt{1 + \frac{2\left(\frac{\sigma(\varepsilon_{\max_acc})}{\varepsilon_{\max_acc}}\right)\left(\frac{H}{b}\right)}{\left(\frac{W}{A}\right)}} - 2\zeta \sqrt{\frac{2\left(\frac{\sigma(\varepsilon_{\max_acc})}{\varepsilon_{\max_acc}}\right)\left(\frac{H}{b}\right)}{\left(\frac{W}{A}\right)}} \times \exp\left[\frac{\zeta}{\sqrt{1-\zeta^2}}\left(\phi + \phi_1 - \frac{\pi}{2}\right)\right] \approx 25.95g \quad (4.60)$$

The corresponding experimental result of maximum G-force (acceleration/ g) provided by BASF is 26.6 g [40]. The error is thus

$$\begin{aligned} \text{error}(\%) &= \left| \frac{G_{\text{analytical}} - G_{\text{experimental}}}{G_{\text{experimental}}} \right| \\ &= \left| \frac{25.95g - 26.6g}{26.6g} \right| \times 100\% = 2.44\% \end{aligned} \quad (4.61)$$

which is acceptable in analytical analysis. Also, the analysis illustrates high efficiency and convenience.

4.4.2 Transient Response

In transient response analysis, four steps will be adopted: (1) The local calculation time parameter at each deformation point will be determined as in examples shown in Figures 4.6 and 4.7; (2) The stiffness coefficient at each deformation point is calculated from stress strain curve; (3) The local calculation time parameter and stiffness coefficient obtained in above two steps are substituted into Equation (4.22), to obtain the acceleration at each instant; (4) With help of equation (4.54), time at each deformation point is obtained. Till now, transient acceleration response is calculated. Figure 4.8 shows the transient response obtained by local linearization method.

However, as shown in Figure 4.8, predicted transient response at unloading stage shows much difference to testing results, this is caused by: stress-strain curve changing in elastic unloading stage of polystyrene foam [103]. Therefore, as shown in Figure 4.8, if plastic unloading stress-strain curve is considered, transient

response at unloading stage is accurately predicted.

Figure 4.8: Comparison of mathematical and experimental results for transient response verification under condition: 400 N weight, 50 mm cushion thickness, 500 mm drop height and 400 cm² contact area

4.5 Experimental Verification

Test method ASTM D1596 [39] is applied to verify mathematical results in this study. This method is used to determine shock absorbing characteristics of cushion buffer. It is an experimental method for developing dynamic cushioning curves, which is used to verify impact loading results in this thesis. Furthermore, transient acceleration response is also obtained for the verification of calculated transient response results.

4.5.1 Experimental Setup

Figure 4.9 shows equipment, Lansmont shock table, used for the development of dynamic cushioning curves. Note that a thimble-like device is mounted on the top of metal platen. This is an accelerometer that is connected by a thin wire to a computer. The accelerometer measures the mechanical shock experienced by the platen as it impacts the test sample. In order to measure initial velocity of drop, a U shape velocity sensor is mounted at the bottom of guide bars, which is also used to restrict drop direction vertical. Weights can be added to plate to provide different static loadings.

Figure 4.9: Setup for cushion drop tests

4.5.2 Dynamic Loading Verification

With the same test method, BASF [40] developed a series cushion curves for EPS20 material as shown in Figure 2.6. Dynamic cushion curves are developed with the

level of shock, maximum acceleration, transmitted at various static loadings as a function of cushion thickness for particular drop heights.

For comparison shown in Figures 4.10 to 4.12, the value of H/b is fixed as 10, 16 and 20, maximum acceleration is plotted with respect to the value of W/A , the values of W/A start from 0.10 N/cm². Damping ratio ζ is an unknown constant and varies from 0 to 0.15 (0, 0.05, 0.1 and 0.15) as parametric study. Under different damping ratios, cushion curves are plotted in Figures 4.10 to 4.12. It shows that accuracy is satisfactory for dynamic loading prediction.

Figure 4.10: Comparison of mathematical and experimental results with $H/b=10$ for dynamic loading under different damping ratios

Meanwhile error between mathematical model and experimental results is plotted in Figures 4.13 to 4.15. As referring to ASTM standard [39], maximum laboratory repeatability deviation ranges from 5% to 18% of average value. Therefore, if error is less than 11%, prediction can be regarded satisfactory. When damping ratio achieves 0.1, all the error is no more than 11%, this accuracy is satisfactory for dynamic loading prediction.

At the same time, it is found from Figures 4.13 to 4.15 that error of all the mathematical models increases in high static load stage. We called this stage

Figure 4.11: Comparison of mathematical and experimental results with $H/b=16$ for dynamic loading under different damping ratios

Figure 4.12: Comparison of mathematical and experimental results with $H/b=20$ for dynamic loading under different damping ratios

as failure zone caused by high deformation of buffer. EPS cushion is a material that most faces of microstructures have six sides as shown in Figure 4.16. The microstructures are honey cone like cells, and walls of cells are fully intact as shown in Figure 4.16. The function of these walls is to protect cell from rupturing. But under heavy impact, cell wall will be ruptured. To compare microstructures, a drop test of static load 1.3 N/cm^2 and 0.5 meter drop height is conducted on cushion buffer. After drop test, specimen is cut into small pieces to be observed under SEM (Scanning Electron Microscopes). Results are shown in Figure 4.16 and compared with un-deformed EPS foam specimen. From Figure 4.16, it can be observed that some of cells at boundary have undergone deformation. Cells are shocked and compressed against each other in direction of impact. Cell walls are ruptured. This causes cushion foam lose its cushioning ability. Current mathematical model is unable to predict how fragility factor in large deformation condition. Therefore, error of mathematical models increases in failure zone as shown in Figures 4.13 to 4.15.

Figure 4.13: Acceleration error of mathematical models under various damping ratios with $H/b=10$ for dynamic loading under different damping ratios

Figure 4.14: Acceleration error of mathematical models under various damping ratios with $H/b=16$ for dynamic loading under different damping ratios

Figure 4.15: Acceleration error of mathematical models under various damping ratios with $H/b=20$ for dynamic loading under different damping ratios

(a) The cells are hexagonal in shapes before drop test

(b) The cell walls are ruptured after shock test

Figure 4.16: Photomicrographs of EPS foam under 200 times magnification

4.5.3 Transient Response Verification

With the application of experimental method as shown in Section 4.5.1, transient acceleration curve is recorded by accelerometer. Two cases are performed: (1) platen weight W of 400 N, cushion thickness b of 50 mm, drop height H of 0.5 m, cushion area A of 400 cm²; (2) platen weight W of 200 N, cushion thickness b of 25 mm, drop height H of 0.5 m, cushion area A of 400 cm².

As shown in Figures 4.8 and 4.17, experimental transient acceleration result is compared with mathematical transient acceleration result for these two cases individually. The results in compressing stage well match each other. However, at unloading stage as noted in Figures 4.8 and 4.17, bigger error exists, which is caused by the reason that stress-strain curve is changed in plastic unloading stage for polystyrene foam [103]. Since stress-strain curve in compressing stage is still used at unloading stage calculation, error happens. In order to prove this point, a compressing test is performed for EPS cushion buffer.

As shown in Figure 4.18, Instron 5500 compression tester is used for compression test and unloading test. The test standard and procedure follow ASTM designation D1621 [104]. Merlin software is fully equipped in a computer with all instrument control and data recording. Maximum compression strain is set from 10% to 70% with 10% step. Compressing test is first performed, unloading

Figure 4.17: Comparison of mathematical and experimental results for transient response verification under condition: 200 N weight, 25 mm cushion thickness, 500 mm drop height and 400 cm² contact area

test is performed when compressing strain achieve its maximum value. Based on ASTM standard, compressing and unloading speeds are set as 10% per minute, and compressing/unloading results are plotted as shown in Figure 4.19. Two material behaviors of cushion buffer is investigated: (1) curve traces a different path during unloading stage as compared to that during compressing stage; (2) compressing/unloading stress-strain curves are almost parallel between different compression cases.

Since unloading curves are almost parallel under different loading cases, it is possible to generate a general equation to describe this behavior. As shown in Figure 4.20, an arbitrary unloading is plotted. In this, a quadratic formula is used to fit this unloading curve and plotted in the same figure. Quadratic formula can be expressed as a step function as

$$\sigma_{unloading} = \begin{cases} P(\varepsilon_{unloading} - \varepsilon_P)^2, & \varepsilon_{unloading} > \varepsilon_P \\ 0 & , \quad \varepsilon_{unloading} \leq \varepsilon_P \end{cases} \quad (4.62)$$

Figure 4.18: Compression test for cushion buffer

Figure 4.19: Compressing and unloading stress-strain curves for cushion buffers

where P is curvature coefficient, ε_P is permanent deformation after unload. As shown in Figure 4.20, quadratic unloading curve starts from maximum stress/strain point, ends at origin point, and fits with original unloading stress-strain curve. Fitting accuracy is depended on P , and ε_P is depended on P and maximum stress/strain $\sigma_{\max}/\varepsilon_{\max}$ as

$$\varepsilon_P = \varepsilon_{\max} - \sqrt{\frac{\sigma_{\max}}{P}} \quad (4.63)$$

Figure 4.20: An arbitrary unloading curve and its quadratic unloading expression

Curvature coefficient P is defined by users, but maximum stress and strain can be automatically obtained at the end of calculation for compressing response. Finally as shown in Figures 4.8 and 4.17, as far as plastic unloading stress-strain curve is considered, transient responses at compressing and unloading stages are both accurately predicted. It is proven that local linearization method can predict transient response of nonlinear drop impact cushion systems.

4.6 Summary of Chapter

The objective of this study is to present a local linearization method for nonlinear vibration analysis of drop impact case. The basic idea and general algorithm of the local linearization method are presented in this chapter. Following the methodology presentation, the analytical analysis of dynamic response for nonlinear cushion system is calculated by using this method. At the same time, the comparison with testing results shows local linearization model is reliable. Furthermore, based on developed local linearization model, the plastic unloading characterization of nonlinear cushion buffer is also discussed.

4.7 Furtherer Development of Full-Scale Virtual Buffer Model with the Contribution from Local Linearization Method

With the successful establishment of local linearization model, the drop-impact response of packages with nonlinear cushion buffers can be predicted. Further on, the local linearization model is programmed by using Visual Basic as shown in Figure 4.21. Since large-scale numerical method is not used, the programming can be simply finished without high demand of computation resource. When all the initial drop-impact conditions and material properties of cushion buffers are input into the Visual Basic package, the dynamic loading and transient response can be quickly obtained. These results will help engineer to do cushion design and optimization. Finally, as shown in Figure 4.22, the full-scale virtual buffer model is further developed with the use of local linearization method. The characterizations of quick evaluation and easy programming from local linearization model further improve the efficiency of virtual buffer model. It not only helps to establish virtual buffer model without FE simulation, but also help to increase buffer design optimization efficiency.

4.7 Further Development of Full-Scale Virtual Buffer Model with the Contribution from Local Linearization Method

137

Figure 4.21: Operation of local linearization method in Visual Basic interface

4.7 Furtherer Development of Full-Scale Virtual Buffer Model with the Contribution from Local Linearization Method

138

Figure 4.22: Flow chart of full-scale virtual buffer model with the combination of local linearization method

Chapter 5

Final Development of Full-Scale Virtual Buffer Model by Using a Hybrid Laminate Modeling Method to Model Objective Model: Multilayer Circuit Boards

In this PhD work, multilayer circuit board is selected as objective model, because it contains multilayer structure that is suitable for high speed and high frequency applications. In this chapter, the hybrid modeling methods consist of three-dimensional (3D) mechanics model and two-dimensional (2D) multilayer finite element code. This modeling techniques can perform static and dynamic analysis for multilayer circuit boards in micro-structure and board levels.

5.1 Introduction of Chapter

Great progress has been made during the last decade to increase the operational speed of large-scale integrated circuits (LSIs)[105]. The demand for microelectronic packaging technology to make use of LSIs has been increased. Meanwhile the size of microelectronic packaging tends to be small [106]. This increases the requirements of high density and high frequency in semiconductor industry. Hence multi-layer printed circuit boards (PCBs) are developed to meet these specific requirements. This kind of PCB can support more complicated chips compared to single- and double-sided circuit boards [106]. PCB can also provide higher circuitry density for the integration of more components to achieve more functions.

With the increasing application of multi-layer PCBs, the mechanical reliability of circuit board is becoming important for devices [107]. The functionality and operation of every electronic device depends on PCBs. The warpage and deflections of boards can cause mounted electronic components dismount or failure of component lead wire, especially in drop impact conditions [108]. A multi-material laminate composite structure is developed [72] to reinforce multi-layer PCBs. The structure is a combination of metallic and composite plies: consist of multiple layers of woven glass/epoxy composite substrate sandwiched between copper foils [71]. However, numerous composite parts and complex material properties of multi-layer PCBs complicate the reliability of simulation. Therefore, the multi-layer PCBs have often been simplified to single layer boards as isotropic models [66][109][70] or orthotropic models [88]. Note that the simplified model does not investigate the microstructure of multi-layer PCBs. Another commonly-used method is unit-cell method [84][65]. With detailed geometry modeling, the woven structure of cross fiber and impregnated resin component are described individually. The unit-cell method requires extensive computation for the analysis of circuit boards.

A multi-layer, bi-phase shell element is therefore introduced in the present work to solve the above-mentioned problems. This chapter presents three hy-

brid modeling methods based on this element: direct bi-phase multilayer mode, equivalent laminate (EQLAM) multilayer model and integrated orthotropic model. Multi-linear law and scalar damage parameters are applied for the description of damage properties. Based on these three methods, the static and dynamic analysis of this composite can be performed at both micro-structure and board levels. These hybrid damage models illustrate a good agreement with experimental results and good computational efficiency in finite element simulation. Conceptually, this study is aimed to provide an efficient damage modeling techniques for laminate composites and flexible modeling methodology for further development of new composite material systems.

5.2 Physical Characterization of Multilayer Composites

A six-layer circuit board is selected as an example of this multilayer composite. As shown in Figure 5.1, the multilayer circuit board consists of multiple layers of woven glass/epoxy composite pre-preg sandwiched between copper foils, in which copper layers numbering from 1 to 6. The manufacturing of multi-layer circuit boards is briefly mentioned for the understanding of their composite structures. In the first stage, glass fibers are woven into plain weave fabric consisting of two fiber bundles in orthogonal directions, which alternately pass over and under each other. The bundles that emerge perpendicular to weaving looms are known as warp bundles, while parallel bundles are fill [21]. Glass fiber fabric is impregnated with FR4 epoxy resin to form a B-staged pre-preg and typically one B-stage pre-preg layer is then consolidated between two layers of copper foil in a hot press to form a C-staged core. Cores go through a series of etching and screening processes. Therefore, multilayer circuit boards are fabricated by relaminating alternating layers of B-staged pre-preg, which act as bonding sheets, and C-stage cores.

A large number of plain weave fabric styles are used in circuit board design.

The fabric styles are often unbalanced, i.e. warp and fill directions include different numbers of fiber bundles or different size of fiber diameters as shown in Figure 5.2. Because of the variation in fiber bundle sizes, geometry of undulating fiber bundles is different and depends on fabric styles. Hence, properties of composite substrates with various fabric styles can be quite different. Furthermore, the same fabric style has different properties in warp and fill directions. In this study, all of these properties will be considered in modeling process.

A F6.1 multilayer circuit board, produced by a printed wiring company in UK, is considered in this study. 1-mm circuit board contains 6 copper layers and 5 composite layers, and the thickness of each respective layer is listed in Figure 1 in micro-structure level. As for board level, the length of board is 200 mm and the width is 160 mm as shown in Figure 5.2. Two pieces of samples, each was cut across its length and width, respectively. After a careful polishing and examination under microscope, the photographs of woven pre-preg were taken as shown in Figure 5.2. It is found that warp fiber is along the length direction and fill is along the width at each pre-preg ply. For F6.1 boards, copper foil is electrodeposited (ED) copper foil, and the associated pre-preg composite is 7628 fabric style. All the material properties and structure characterizations are listed in Table 5.1.

5.3 Presentation of Multilayer Finite Element

The proposed models in this study were implemented in PamCrash multilayer shell element, Material type 131 [51]. This element is a three- or four-node 2D explicit element. The assumptions mainly impose a state of plane stresses and an invariance of deformation through thickness. In addition, transverse and shear effects are taken into account. The interpolation functions relating to finite element are bilinear. Owing to ply concept, the stacking and the nature of materials composing the structure to be designed (multi-layered multi-material, laminated composites,

Figure 5.1: Schematic graph of six-layer circuit board and structure of woven composite ply

Figure 5.2: Schematic graph of F6.1 multilayer circuit board

Table 5.1: Material properties and structure characterizations of F6.1 multilayer board (Material properties for direct bi-phase multilayer model)

	ED copper foil ply	E-glass fiber / Epoxy resin ply (7628)		
Density	$8.92 \times 10^{-6}\text{kg/mm}^3$	$1.2 \times 10^{-6}\text{kg/mm}^3$		
Ply thickness	0.0175 mm	Fill (0.18 / 4 mm)/(0.20 / 4 mm) Wrap (0.18 / 2 mm)/(0.20 / 2 mm)		
		Glass fiber	Resin matrix	
Young's modulus	110 GPa	72.35 GPa	3.05 GPa	
Poisson's ratio	0.3			0.33
Volume ratio		Fill 0.194	Warp 0.268	
Yield stress	310 MPa			
Multi-linear law for metallic ply and damage parameters for composite ply	$E_1=469\text{MPa}, \sigma_1=315\text{MPa}$ $E_2=469\text{MPa}, \sigma_2=335\text{MPa}$ $E_3=469\text{MPa}, \sigma_3=355\text{MPa}$ $E_4=469\text{MPa}, \sigma_4=355\text{MPa}$ $E_5=-72.4\text{MPa}, \sigma_5=279\text{MPa}$ $E_6=-72.4\text{MPa}, \sigma_6=179\text{Mpa}$ $E_7=-72.4\text{MPa}, \sigma_7=17 \text{ MPa}$	$\varepsilon_i^f=4.77\%$ $\varepsilon_1^f=4.8\%$ $\varepsilon_u^f=4.85\%$ $d_1^f=0.001$ $d_u^f=0.999$	$\varepsilon_{si}^m=1.536\%$, $\varepsilon_{s1}^m=2.073\%$, $\varepsilon_{su}^m=2.077\%$, $d_{s1}^m=0.136$, $d_{su}^m=0.988$,	$\varepsilon_{vi}^m=0.680\%$ $\varepsilon_{v1}^m=0.918\%$ $\varepsilon_{vu}^m=0.920\%$ $d_{v1}^m=0.136$ $d_{vu}^m=0.988$

etc.) are reproduced. The materials can be only of metallic, composite or foam type.

For this multilayered composite shell, the program assigns one integration point per layer, and the shell thickness is computed from layer thickness. As shown in Figure 5.3, there is a 2D composite shell with n layers with n integration points located in the middle section of ply. In theory, three integration points are sufficient in most cases to integrate bending and membrane effects. Use 4 or 5 points for added precision in exceptional cases. Of course, laminate layers will increase as long as the number of integration points, and this will cause the associated increase in CPU time.

PamCrash finite element 131 allows four types of ply: bi-phase ply, global ply, elastic-plastic damaging ply and hyper-elastic Mooney-Rivlin ply. In present study, only elastic-plastic damaging ply and bi-phase model are used, since multilayer circuit board basically consists of copper foil and alternating prepreg.

Figure 5.3: Theoretical description of multilayered shell element

5.3.1 Elastic-Plastic Damaging Ply Model

Elastic-plastic damaging ply [51] may display an elastic-plastic behavior, initially isotropic. The elastic behavior is defined via Young's modulus, Poisson's ratio, shear modulus, and thickness of ply. On the other hand, the plastic behaviour is described by a multi-linear stress-strain curve, as shown in Figure 5.4. The curve is defined via seven points, for each of them the plastic tangent modulus and the plastic stress are defined. Extrapolation beyond the last specified point is horizontal, which is used to describe break phenomena. Based on material properties provided in reference [110], the damage parameters of ED copper foil are calculated and listed in Table 5.1. All the coefficients in the table will be used in modeling.

5.3.2 Bi-phase Ply Model

Bi-phase model [51] is a model of heterogeneous materials adapted to unidirectional composite materials reinforced by continuous fibers. This model was used in a

Figure 5.4: Polygonal stress-strain curve for metallic plies [51]

crash behavior study of car made entirely in composites [72]. Essentially, the key lies in distinguishing two elementary layer components: fibers (uni-dimensional phase) and matrix (orthotropic phase) as shown in Figure 5.5. Consequently, each phase features its specific rheological characteristics: fibers have a unidirectional behavior that is brittle isotropic elastic with damage, whereas matrix has behavior that is brittle orthotropic elastic with plastic damage. A description of each phase features is given in tensile and compressive responses when considering composite asymmetric behavior. The superposition of the effect of each phase allows the formulation of material property. However, stresses are calculated separately for each of these phases and damage can be propagated independently according to the selected criteria.

Figure 5.5: Bi-phase rheological composite model [51]

The description of behavior of phases requires the knowledge of intrinsic

composite elastic properties in three orthogonal directions (Young's modulus, shear modulus, Poisson's ratio). Elastic data of each phase are obtained from the intrinsic elastic properties of constituents and fiber volume fraction α_f . The independent parameters in transversely isotropic model are less than ordinary orthotropic model with the hypothesis of transversely isotropic behavior [111].

In bi-phase rheological composite model, the known or measured unidirectional ply material properties are:

$$\begin{aligned} E_{11}^{UD}, E_{22}^{UD}, G_{12}^{UD}, \nu_{12}^{UD} &= \text{in-plane UD material constants} \\ E_{true}^f &= \text{true fiber modulus} \\ \alpha_f &= \text{fiber volume fraction} \end{aligned} \quad (5.1)$$

The expression for plane stress elasticity matrices is $\sigma = C\varepsilon$, for fibers (f), matrix (m) and tested unidirectional ply (UD), becomes for bi-phase material

$$C^{UD} = C^f + C^m \quad (5.2)$$

where

$$C^{UD} = \begin{bmatrix} E_{11}^{UD}/(1 - \nu_{12}^{UD}\nu_{21}^{UD}) & \nu_{12}^{UD}E_{22}^{UD}/(1 - \nu_{12}^{UD}\nu_{21}^{UD}) & 0 \\ \nu_{21}^{UD}E_{11}^{UD}/(1 - \nu_{12}^{UD}\nu_{21}^{UD}) & E_{22}^{UD}/(1 - \nu_{12}^{UD}\nu_{21}^{UD}) & 0 \\ 0 & 0 & G_{12}^{UD} \end{bmatrix} \quad (5.3)$$

$$C^f = \begin{bmatrix} E_{11}^f & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \quad (5.4)$$

$$C^m = \begin{bmatrix} E_{11}^m/(1 - \nu_{12}^m \nu_{21}^m) & \nu_{12}^m E_{22}^m/(1 - \nu_{12}^m \nu_{21}^m) & 0 \\ \nu_{21}^m E_{11}^m/(1 - \nu_{12}^m \nu_{21}^m) & E_{22}^m/(1 - \nu_{12}^m \nu_{21}^m) & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \quad (5.5)$$

$$E_{11}^f = \alpha_f E_{true}^f \quad (5.6)$$

$$\nu_{21}^{UD} = \nu_{12}^{UD} E_{22}^{UD} / E_{11}^{UD} \quad \text{and} \quad \nu_{21}^m = \nu_{12}^m E_{22}^m / E_{11}^m \quad (5.7)$$

The above equations will be used for the integration of material properties of composite ply in direct bi-phase multilayer model. In addition, note that if the fiber phase is neglected, matrix properties can be used to model a quasi-homogeneous orthotropic material behavior that can be brittle-elastic and elastic-fracturing. The orthotropic model is defined as degenerated bi-phase models, which will be applied in EQLAM multilayer modeling method to model an orthotropic ply.

Both matrix and fibers of bi-phase law may undergo intermediate damage after an initial linear elastic phase according to

$$C(d) = C_0(1 - d) \quad (5.8)$$

where $C(d)$ is instantaneous modulus matrix, C_0 is undamaged initial modulus matrix and d is a scalar damage parameter that depends on strain. Damage parameter, d , can be expressed as

$$d(\varepsilon) = d_V(\varepsilon_V) + d_S(\varepsilon_S) \quad (\text{fiber phase } d(\varepsilon) = d_V(\varepsilon^f)) \quad (5.9)$$

where d_V is a volumetric damage due to a volumetric equivalent strain, ε_V , and d_S is a shear induced damage due to an equivalent shear strain, ε_S . The volumetric

and equivalent shear strains are given by (ε^f = fiber strain)

$$\varepsilon_V = \varepsilon_{kk} \quad (\text{fiber phase : } \varepsilon_V = \varepsilon^f) \quad (5.10)$$

$$\varepsilon_S = (1/2 e_{ij}^2)^{1/2}, \text{ with } e_{ij} = \varepsilon_{ij} - 1/3 \varepsilon_{kk} \delta_{ij} \quad (\text{fiber phase } \varepsilon_S = 0) \quad (5.11)$$

where ε_{kk} is the trace of total strain tensor and e_{ij} are components of deviatoric strain tensor [112]. Scalar ε_S can be interpreted as the second invariant of deviatoric strain tensor, while scalar ε_V is the first invariant of direct strain tensor.

The implemented law implies that damage parameter $d(\varepsilon)$ is zero for equivalent strain $0 \leq \varepsilon \leq \varepsilon_i$, where ε_i is an initial equivalent threshold strain (volume and shear). Parameter $d(\varepsilon)$ grows linearly between $\varepsilon_i < \varepsilon < \varepsilon_1$ and $\varepsilon_1 < \varepsilon < \varepsilon_u$, where ε_1 is an intermediate equivalent strain with intermediate damage d_1 , while ε_u is ultimate equivalent strain at which ultimate damage d_u is reached. For matrix phase, equivalent strains ε_i , ε_1 , ε_u and damages d_1 , d_u must be specified for shear and volume damage for (fiber) tension and compression loading. For fibers, no shear damage exists and direct strains ε_i , ε_1 , ε_u and damages d_1 , d_u must be specified for tension and compression loadings in fiber direction.

The physical meaning of damage parameters can be expressed by calibrated tensile or compressive damage law stress-strain curve, as shown in Figure 5.6. It is obtained from modulus damage law by making use of measured points: initial threshold stress-strain point (σ_i , ε_i), intermediate stress-strain point (σ_1 , ε_1) and ultimate stress-strain point (σ_u , ε_u). damage modulus values can be described as a consequence of specified function

$$E(\varepsilon) = [1 - d(\varepsilon)] E_0 \quad (5.12)$$

There are three cases for $d(\varepsilon)$ in Equation (5.12):

(i) $\varepsilon_i < \varepsilon < \varepsilon_1$

$$\begin{aligned} d(\varepsilon) &= \frac{\varepsilon - \varepsilon_i}{\varepsilon_1 - \varepsilon_i} d_1 = \frac{\varepsilon - \varepsilon_i}{\varepsilon_1 - \varepsilon_i} \left(1 - \frac{E_1}{E_0}\right) \\ &= \frac{\varepsilon - \varepsilon_i}{\varepsilon_1 - \varepsilon_i} \left(1 - \frac{\sigma_1/\varepsilon_1}{E_0}\right) \end{aligned} \quad (5.13)$$

(ii) $\varepsilon_1 < \varepsilon < \varepsilon_u$

$$\begin{aligned} d(\varepsilon) &= \frac{\varepsilon - \varepsilon_1}{\varepsilon_u - \varepsilon_1} (d_u - d_1) + d_1 = \frac{\varepsilon - \varepsilon_1}{\varepsilon_u - \varepsilon_1} \left(1 - \frac{E_u}{E_0} - d_1\right) + d_1 \\ &= \frac{\varepsilon - \varepsilon_1}{\varepsilon_u - \varepsilon_1} \left(1 - \frac{\sigma_u/\varepsilon_u}{E_0} - d_1\right) + d_1 \end{aligned} \quad (5.14)$$

(iii) $\varepsilon_u < \varepsilon < \infty$

$$d(\varepsilon) = 1 - (1 - d_u)\varepsilon_u/\varepsilon = 1 - \frac{E_u\varepsilon_u}{E_0\varepsilon} = 1 - \frac{\sigma_u}{E_0\varepsilon} \quad (5.15)$$

where $E(\varepsilon)$ = intermediate damage d_1 = intermediate damage

E_0 =initial elastic modulus d_u =ultimate damage

E_1 =intermediate modulus E_u =ultimate modulus

The material properties of glass fibers are provided in reference [113]. With the help of Equations (5.12) to (5.15), critical damage parameters of fibers are calculated and listed in Table 5.1. However, for matrix ply, the strains in Figure 5.6, either equivalent volume or shear strains, need to be calculated from the results of uniaxial tests.

In uniaxial tests, the strain-stress matrix relation is described by neglecting irrelevant shear submatrices,

$$\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \end{Bmatrix} = \begin{Bmatrix} 1/E_{11} & -\nu_{21}/E_{22} & -\nu_{31}/E_{33} \\ -\nu_{12}/E_{11} & 1/E_{22} & -\nu_{32}/E_{33} \\ -\nu_{13}/E_{11} & -\nu_{23}/E_{22} & 1/E_{33} \end{Bmatrix} \begin{Bmatrix} \sigma_{11} \\ 0 \\ 0 \end{Bmatrix} \quad (5.16)$$

where the uniaxial tension direction is along direction 1. Equation (5.16) leads the

Figure 5.6: Calibrated tensile or compressive damage law stress-strain curve [51]

unidirectional strain tensor as

$$\varepsilon_{ij}^{unid} = \begin{bmatrix} \varepsilon_{11} & 0 & 0 \\ 0 & -\nu_{12}\varepsilon_{11} & 0 \\ 0 & 0 & -\nu_{13}\varepsilon_{11} \end{bmatrix} \quad (5.17)$$

The equivalent scalar volumetric strain is then calculated as given by Equation (5.10)

$$\varepsilon_V = \varepsilon_{kk}^{unid} = (1 - \nu_{12} - \nu_{13})\varepsilon_{11} \quad (5.18)$$

Meanwhile, Equation (5.11) is expressed as

$$e_{ij} = \varepsilon_{ij}^{unid} - 1/3\varepsilon_{kk}^{unid}\delta_{ij} = \begin{bmatrix} \varepsilon_{11} - \varepsilon_V/3 & 0 & 0 \\ 0 & -\nu_{12}\varepsilon_{11} - \varepsilon_V/3 & 0 \\ 0 & 0 & -\nu_{13}\varepsilon_{11} - \varepsilon_V/3 \end{bmatrix} \quad (5.19)$$

The square of strain-tensor e_{ij} is then

$$e_{ij}^2 = \begin{bmatrix} e_{11}^2 & 0 & 0 \\ 0 & e_{22}^2 & 0 \\ 0 & 0 & e_{33}^2 \end{bmatrix} = \begin{bmatrix} (\varepsilon_{11} - \varepsilon_V/3)^2 & 0 & 0 \\ 0 & (\nu_{12}\varepsilon_{11} + \varepsilon_V/3)^2 & 0 \\ 0 & 0 & (\nu_{13}\varepsilon_{11} + \varepsilon_V/3)^2 \end{bmatrix} \quad (5.20)$$

By virtue of Equations (5.18) and (5.20)

$$e_{kk}^2 = \frac{2}{3} \times \varepsilon_{11}^2 (1 + \nu_{12} + \nu_{13} - \nu_{12}\nu_{13} + \nu_{12}^2 + \nu_{13}^2) \quad (5.21)$$

By virtue of Equation (5.11), the equivalent scalar shear strain is given by

$$\varepsilon_S = \left(\varepsilon_{11} / \sqrt{3} \right) (1 + \nu_{12} + \nu_{13} - \nu_{12}\nu_{13} + \nu_{12}^2 + \nu_{13}^2)^{1/2} \quad (5.22)$$

If the strain in uniaxial tensile test is measured and Poisson's ratios are known, the equivalent scalar strains (volume and shear) can be calculated based on Equations (5.18) and (5.22). In this work, matrix, epoxy resin, is taken as a brittle isotropic material. Based on original material properties of epoxy resin, the equivalent scalar volume and shear stress-strain relations are given in Table 5.1.

5.4 Modeling Methodologies for Static Analysis

By virtue of multilayer finite element code, the geometry model of multilayer circuit board is established as a 2D plane in board-level. All the composite substrate characterization in micro-level can be described by mathematics model and input into finite element code as material property. Therefore, 3D solid element and unit cell are not required, where the composite substrate characterization is described by geometry model. Accordingly, a large-scale finite element model is simplified, and computation cost is then reduced greatly. In this section, two modeling methods

are provided: (1) direct bi-phase multilayer model, (2) EQLAM multilayer model. The damage behaviors of composite phases are also described.

5.4.1 Direct Bi-Phase Multilayer Model

Wang et al. [70] developed a multi-layer fill-warp PCB model, which illustrated high accuracy in PCB modeling work. Based on this model, a direct bi-phase multilayer is further developed. In this modeling technique, an elastic-plastic damage model is used to represent copper ply. Multi-linear curve is applied to describe the damage behavior of copper ply, while bi-phase model represents glass/epoxy composite ply. Scalar damage parameters are used to describe the damage behaviors of glass fiber and epoxy resin individually. Note that in the modeling of composite ply, woven fabric is simplified as unidirectional laminate ply. The model is named as direct bi-phase multilayer model as it is a one-to-one modeling method.

For composite plies, fibers (uni-dimensional phase) and matrix (orthotropic phase) in bi-phase model are distinguished to represent glass fiber and epoxy resin, respectively. As shown in Figures 5.1 and 5.2, real composite model is a woven fabric laminate. However, bi-phase model in PamCrash code is uni-dimensional fiber model, and all the fibers are parallel to each other. Therefore, woven fabric laminate is simplified as a symmetric three-ply cross-ply laminate in analysis. The fibers along warp and fill directions of unit cell are separated and represented as three individual plies. The plies are arranged as a symmetric cross-ply laminate with the fill ply of thickness b_f , sandwiched between two warp plies of thickness $b_w/2$ each. Figure 5.7 illustrates how each layer of prepreg is modeled and the layers of prepreg are stacked to form a six-layer circuit board. Through three-ply cross-ply laminate, 21 plies (6 elastic-plastic damage plies and 15 bi-phase plies) are required for the configuration of six-layer boards. Meanwhile, all the material properties required in direct multilayer model are listed in Table 5.1. Note that as the thickness of fill ply is half and warp ply is a quarter of prepreg thickness in direct multilayer model, the volume fraction of glass fiber is twice of the separated

cross ply accordingly.

Figure 5.7: Schematic graph of direct bi-phase multilayer model

Electrodeposited (ED) copper foil is modeled as isotropic material. Its stress-strain curve is plotted as shown in Figure 5.8. From this figure, Young's modulus, yield stress can be obtained, and damage material behavior is also described in Figure 5.6 as seven plastic stress-strain points curve. The material properties of ED copper will be input into PamCrash as elastic-plastic damage ply as shown in Figure Figure 5.10.

In direct bi-phase multilayer model, the material properties of glass fiber and epoxy matrix will be independently described. The elastic properties of glass fiber and epoxy resin will also be presented as Young's modulus and Poisson ratio. For damage behavior, fibers exhibit a unidirectional damage behavior, they can directly use scalar damage parameters; whereas the matrix has orthotropic damage behavior with volume damage and shear damage, the volume damage and shear

Figure 5.8: Damage stress strain curve of ED copper

damage curves need to be measured before the use of scalar damage parameters. The stress strain curve of glass fiber is shown in Figure 5.9. With the help of Equations (5.12) to (5.15), scalar damage parameters of glass fiber is calculated and shown in Table 5.1. In addition, the stress strain curve of epoxy resin is plotted shown in Figure 5.10. With the help of Equations (5.18) and (5.22), the volume damage and shear damage curves are presented in Figure 5.10 at the same time. The scalar damage parameters of epoxy resin were calculated based on these two damage curves, they are listed in Table 5.1.

5.4.2 EQLAM Multilayer Model

With the combination of mechanics EQLAM model and multilayer finite element code, an EQLAM multilayer model is devised. The original EQLAM model is only developed for static analysis and only the in-plane material properties are investigated. In EQLAM multilayer model, static analysis is expanded to dynamic

Figure 5.9: Damage stress strain curve of brittle glass fiber

Figure 5.10: Damage stress strain curves of epoxy resin

Table 5.2: Measurements of bundle sizes, crimp, and fiber volume fractions [50]

	Aspect ratio a/b	Crimp h/d	Fiber volume fraction in bundle	Fiber volume fraction in composite
Fill	7.26	0.083	0.75	0.19
Wrap	5.26	0.053	0.75	0.26

analysis, and out-of-plane material properties are also simulated. First, static analysis based on 3D mechanics unit-cell model is performed. In the composite ply as shown in Figure 5.1, glass fiber is woven as undulated structure. In direct bi-phase multilayer model, it is simplified as linear structure without the considerations of fibers' weave geometry. Therefore, EQLAM multilayer model is provided, which considers the woven structure of reinforced fiber. Before mechanics analysis, the undulated structure of glass fiber should be quantitatively described. In a previous work, Shrotriya and Sottos [79] characterized the fabric architecture and viscoelastic properties of 7628 style composite substrate. Based on the photomicrographs taken in warp and fill directions of 7628 laminate. The warp and fill bundles have different sizes and crimp angles, which are described by a , b , d and h (see Figure 5.11 and Table 5.2)

Figure 5.11: Schematic of plain weave geometry [79]

A composite model developed by EQLAM theory is combined with Pam-Crash multilayer finite element to form an EQLAM multilayer model. In this model, elastic-plastic damage ply and bi-phase ply are both applied. To compare with direct bi-phase multilayer model, composite ply is integrated into a whole orthotropic ply: (1) orthotropic ply is developed by EQLAM theory and simulated

by using a degenerate bi-phase model, which considers woven fabric structure;(2) only scalar damage parameters for matrix ply are used to describe the damaging behavior of whole composite ply. Note that the elastic-plastic damage ply is still used to model copper ply. In this section, damage modeling will be performed after elastic modeling process.

EQLAM model, as shown schematically in Figure 5.12, idealizes woven fabric laminate as a symmetric three-ply cross-ply laminate. The properties of outer plies are first calculated by combining warp fiber and resin properties via self-consistent micromechanics. Average ply stiffness properties for undulating warp fibers are obtained by integrating over the length of unit cell. The inner ply properties are calculated in a similar fashion by averaging the properties of fibers in fill direction. The overall properties of laminate are then obtained using the classical lamination theory and sublamine model method [111]. Compared to original EQLAM model [21], the improved EQLAM model simulates the out-of-plane material properties by using sub-lamine model method.

At first, self-consistent relationships are used to calculate the transversely isotropic properties of a small segment dx as shown in Figure 5.13, which illustrates the structure of an undulating bundle. Fibers are assumed straight throughout small segment and oriented at an angle θ to xy coordinate system.

The following expressions are used to predict the effective properties of unidirectional fiber composites by the modified rule of mixtures [111].

$$\text{Fiber shear modulus} \quad G_f = \frac{E_f}{2(1 + \nu_f)} \quad (5.23)$$

$$\text{Matrix shear modulus} \quad G_m = \frac{E_m}{2(1 + \nu_m)} \quad (5.24)$$

Figure 5.12: Process of improved EQLAM modeling method

Figure 5.13: Schematic graph of undulating bundle [21]

$$\text{Young's modulus (longitudinal)} \quad E'_1 = \alpha_f E_f + \alpha_m E_m \quad (5.25)$$

$$\begin{aligned} &\text{Young's modulus (transverse)} \\ E'_2 = E'_3 &= \left(\frac{\sqrt{\alpha_f}}{\sqrt{\alpha_f} E_f + (1 - \sqrt{\alpha_f}) E_m} + \frac{1 - \sqrt{\alpha_f}}{E_m} \right)^{-1} \end{aligned} \quad (5.26)$$

$$\begin{aligned} &\text{Shear modulus (longitudinal)} \\ G'_{12} = G'_{13} &= \left(\frac{\sqrt{\alpha_f}}{\sqrt{\alpha_f} G_f + (1 - \sqrt{\alpha_f}) G_m} + \frac{1 - \sqrt{\alpha_f}}{G_m} \right)^{-1} \end{aligned} \quad (5.27)$$

$$\begin{aligned} &\text{Shear modulus (transverse)} \\ G'_{23} &= \left(\frac{\sqrt{\alpha_f}}{\sqrt{\alpha_f} G_f + (1 - \sqrt{\alpha_f}) G_m} + \frac{1 - \sqrt{\alpha_f}}{G_m} \right)^{-1} \end{aligned} \quad (5.28)$$

$$\text{Poisson ratio (longitudinal)} \quad \nu'_{12} = \nu'_{13} = \alpha_f \nu_f + \alpha_m \nu_m \quad (5.29)$$

$$\text{Poisson ratio (longitudinal)} \quad \nu'_{21} = \nu'_{31} = \frac{E'_2 \nu'_{12}}{E'_1} \quad (5.30)$$

$$\text{Poisson ratio (transverse)} \quad \nu'_{23} = \nu'_{32} = \frac{E'_2}{2G'_{23}} - 1 \quad (5.31)$$

where E_f = Young's modulus of fiber E_m = Young's modulus of matrix

ν_f = Poisson ratio of fiber ν_m = Poisson ratio of matrix Note that Equations (5.23) and (5.31) follow the coordinate system shown in Figure 5.13: 1, 2, 3 axes represent x , y , z axes individually. The self-consistent coefficients obtained

from Equations (5.23) and (5.31) consist of stiffness matrix for small segment dx , as shown in Figure 5.13. This is a transversely isotropic structure, and it can be written as

$$[C'] = \frac{1}{D} \begin{bmatrix} E'_1 (1 - \nu'^2_{23}) & E'_2 \nu'_{12} (1 + \nu'_{23}) & E'_2 \nu'_{12} (1 + \nu'_{23}) & 0 & 0 & 0 \\ E'_2 \nu'_{12} (1 + \nu'_{23}) & E'_2 \left(1 - \frac{E'_2}{E'_1} \nu'^2_{12}\right) & E'_2 \left(\nu'_{23} + \frac{E'_2}{E'_1} \nu'^2_{12}\right) & 0 & 0 & 0 \\ E'_2 \nu'_{12} (1 + \nu'_{23}) & E'_2 \left(\nu'_{23} + \frac{E'_2}{E'_1} \nu'^2_{12}\right) & E'_2 \left(1 - \frac{E'_2}{E'_1} \nu'^2_{12}\right) & 0 & 0 & 0 \\ 0 & 0 & 0 & DG'_{23} & 0 & 0 \\ 0 & 0 & 0 & 0 & DG'_{13} & 0 \\ 0 & 0 & 0 & 0 & 0 & DG'_{12} \end{bmatrix} \quad (5.32)$$

$$\text{where} \quad D = 1 - \nu'^2_{23} - 2(1 + \nu'_{23}) \frac{E'_2}{E'_1} \nu'^2_{12} \quad (5.33)$$

The models presented in this work represent undulated fiber bundles. Fiber bundles undulate in xz plane following a cosine curve described by

$$z(x) = \frac{h}{2} \cos\left(\frac{\pi}{d}x\right) \quad (5.34)$$

where $2h$ and $2d$ represent, respectively, the thickness and length of unit composite cell as shown in Figure 5.11. If fibers are assumed straight and oriented at angle θ over a small section dx of bundle, angle can be given by the slope of curve as

$$\theta(x) = -\tan^{-1} \left(\frac{dz}{dx} \right) \quad (5.35)$$

As presented by process in Figure 5.12, next, the average ply stiffness matrix $[\bar{C}]$ for undulating warp fibers is obtained by integrating local stiffness components $[\bar{C}']$ over the length of undulation. The local stiffness matrix $[\bar{C}']$ of fiber bundles in x - z coordinated system are found by performing coordinate transformations of stiffness components, C'_{ij} , on small section dx of bundle following procedure outline

in Lekhinitzki's work [114]. When a new coordinate system is rotated a certain angle θ , new stiffness components follow transformation form,

$$\bar{C}'_{ij} = \sum_{m=1}^6 \sum_{n=1}^6 C'_{mn} q_{im} q_{jn} \quad (5.36)$$

where $i, j = 1$ to 6; the indices m and n are to be summed. q_{ij} is transformation matrix, it follows

$$[q] = \begin{bmatrix} l_{11}^2 & l_{12}^2 & l_{13}^2 & l_{12}l_{13} & l_{13}l_{11} & l_{12}l_{11} \\ l_{21}^2 & l_{22}^2 & l_{23}^2 & l_{23}l_{22} & l_{23}l_{21} & l_{22}l_{21} \\ l_{31}^2 & l_{32}^2 & l_{33}^2 & l_{33}l_{32} & l_{33}l_{31} & l_{32}l_{31} \\ 2l_{31}l_{21} & 2l_{32}l_{22} & 2l_{33}l_{23} & l_{33}l_{22} + l_{32}l_{23} & l_{33}l_{21} + l_{31}l_{23} & l_{31}l_{22} + l_{32}l_{21} \\ 2l_{31}l_{11} & 2l_{32}l_{13} & 2l_{33}l_{13} & l_{33}l_{12} + l_{32}l_{13} & l_{33}l_{11} + l_{31}l_{13} & l_{31}l_{12} + l_{32}l_{11} \\ 2l_{21}l_{11} & 2l_{12}l_{22} & 2l_{13}l_{23} & l_{13}l_{22} + l_{12}l_{23} & l_{13}l_{21} + l_{11}l_{23} & l_{11}l_{22} + l_{12}l_{21} \end{bmatrix} \quad (5.37)$$

Axes of fiber bundle rotates along y direction, l_{ij} shown as above is presented in matrix form as

$$[l] = \begin{bmatrix} \cos \theta & 0 & \sin \theta \\ 0 & 1 & 0 \\ -\sin \theta & 0 & \cos \theta \end{bmatrix} \quad (5.38)$$

By rotation of transversely isotropic stiffness components C'_{ij} through a certain angle θ along y direction, Equation (5.36) can be completely expanded as,

$$\bar{C}'_{11} = C'_{11} \cos^4 \theta + (2C'_{13} + C'_{55}) \sin^2 \theta \cos^2 \theta + C'_{33} \sin^4 \theta \quad (5.39)$$

$$\bar{C}'_{12} = \bar{C}'_{21} = C'_{12} \cos^2 \theta + C'_{32} \sin^2 \theta \quad (5.40)$$

$$\bar{C}'_{13} = \bar{C}'_{31} = (C'_{11} + C'_{33} - 2C'_{13} - C'_{55}) \sin^2 \theta \cos^2 \theta + C'_{13} \quad (5.41)$$

$$\bar{C}'_{22} = C'_{22} \quad (5.42)$$

$$\bar{C}'_{32} = \bar{C}'_{23} = C'_{12} \sin^2 \theta + C'_{32} \cos^2 \theta \quad (5.43)$$

$$\bar{C}'_{33} = C'_{11} \sin^4 \theta + (2C'_{13} + C'_{55}) \sin^2 \theta \cos^2 \theta + C'_{33} \cos^4 \theta \quad (5.44)$$

$$\bar{C}'_{44} = C'_{44} \cos^2 \theta + C'_{66} \sin^2 \theta \quad (5.45)$$

$$\bar{C}'_{55} = 4(C'_{11} + C'_{33} - 2C'_{13} - C'_{55}) \sin^2 \theta \cos^2 \theta + C'_{55} \quad (5.46)$$

$$\bar{C}'_{66} = C'_{44} \sin^2 \theta + C'_{66} \cos^2 \theta \quad (5.47)$$

All the other unlisted elements in stiffness matrix $\bar{C}'_{ij}$ are equal to zero. As shown in Equation (5.35), function $\theta(x)$ depends on variable x . Therefore the transformed stiffness matrix is also function depended on variable x , it can be described as

$$\bar{C}'_{ij}(\theta) = \bar{C}'_{ij}(\theta(x)) = \bar{C}'_{ij}(x) \quad (5.48)$$

Rotated stiffness is then averaged over the length of fiber as by the form of

$$\bar{C}_{ij} = \frac{1}{2d} \int_0^{2d} \bar{C}'_{ij}(x) dx \quad (5.49)$$

By the virtue of Equations (5.34)(5.35) and (5.49), resulting average stiffness of woven bundles is

$$\bar{C}_{11} = \frac{C'_{11} \left(1 + \frac{\alpha}{2}\right) + (2C'_{13} + C'_{55}) \left(\frac{\alpha}{2}\right) + C'_{33} \left[(1 + \alpha)^{\frac{3}{2}} - \left(1 + \frac{3}{2}\alpha\right)\right]}{(1 + \alpha)^{\frac{3}{2}}} \quad (5.50)$$

$$\bar{C}_{12} = \bar{C}_{21} = \frac{C'_{12} + C'_{32} (\sqrt{1 + \alpha} - 1)}{\sqrt{1 + \alpha}} \quad (5.51)$$

$$\bar{C}_{13} = \bar{C}_{31} = \frac{(C'_{11} + C'_{33} - 2C'_{13} - C'_{55}) \left(\frac{\alpha}{2}\right) + C'_{13} (1 + \alpha)^{\frac{3}{2}}}{(1 + \alpha)^{\frac{3}{2}}} \quad (5.52)$$

$$\bar{C}_{22} = C'_{22} \quad (5.53)$$

$$\bar{C}_{23} = \bar{C}_{32} = \frac{C'_{12} (\sqrt{1 + \alpha} - 1) + C'_{32}}{\sqrt{1 + \alpha}} \quad (5.54)$$

$$\bar{C}_{33} = \frac{C'_{11} \left[(1 + \alpha)^{\frac{3}{2}} - \left(1 + \frac{3}{2}\alpha\right)\right] + (2C'_{13} + C'_{55}) \left(\frac{\alpha}{2}\right) + C'_{33} \left(1 + \frac{\alpha}{2}\right)}{(1 + \alpha)^{\frac{3}{2}}} \quad (5.55)$$

$$\bar{C}_{44} = \frac{C'_{44} + C'_{66} (\sqrt{1 + \alpha} - 1)}{\sqrt{1 + \alpha}} \quad (5.56)$$

$$\bar{C}_{55} = \frac{2(C'_{11} + C'_{33} - 2C'_{13} - C'_{55})\alpha + C'_{55}(1 + \alpha)^{\frac{3}{2}}}{(1 + \alpha)^{\frac{3}{2}}} \quad (5.57)$$

$$\bar{C}_{66} = \frac{C'_{44} (\sqrt{1 + \alpha} - 1) + C'_{66}}{\sqrt{1 + \alpha}} \quad (5.58)$$

$$\text{where} \quad \alpha = \left(\frac{\pi h}{2d} \right)^2 \quad (5.59)$$

Note that all the other unlisted elements in averaged stiffness matrix are equal to zero. The averaged stiffness matrix for warp ply and fill ply, $[\bar{C}]^w$ and $[\bar{C}]^f$, can now be established. The woven structure of glass fiber is idealized as crossed unidirectional three layers ply, in which the corresponding average stress and strain are respectively described by $\bar{\sigma}$ and $\bar{\varepsilon}$. Because bundle direction along fill direction and warp direction are vertical to each other, the averaged stiffness matrix is required to rotate 90 degree about z -axis as [111]

$$[\bar{C}]^{90^\circ} = [\hat{T}_\sigma^z]^{90^\circ} [\bar{C}]^{0^\circ} \left\{ [\hat{T}_\varepsilon^z]^{90^\circ} \right\}^{-1} \quad (5.60)$$

where

$$\begin{bmatrix} \hat{T}_\sigma^z \end{bmatrix}^{90^\circ} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 \end{bmatrix} \text{ and } \begin{bmatrix} \hat{T}_\epsilon^z \end{bmatrix}^{90^\circ} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 \end{bmatrix} \quad (5.61)$$

In the following, the classical lamination theory is applied to obtain overall properties of a laminate ply. However, this theory can only be used to obtain some crucial in-plane stiffness coefficients. Therefore, a sublamine model is also provided to establish the whole stiffness matrix, including in-plane characteristics and out-of-plane characteristics. This model is established based on three-dimensional elements that may result in a large number of elements to make numerical computation difficult and infeasible [111]. The sublamine model overcomes these difficulties by dividing the laminate into sublaminae. Each layer in sublamine may be monoclinic, orthotropic, transversely isotropic, or isotropic. The thickness of each element is the same as the thickness of the corresponding sublamine. In this study, the thickness direction is along z -axes, the stresses σ^c for the whole composite ply are defined as

$$\begin{aligned} \sigma_{xx}^c &= \frac{1}{t} \int_t \bar{\sigma}_{xx} dz & \sigma_{zz}^c &= \bar{\sigma}_{zz} \\ \sigma_{yy}^c &= \frac{1}{t} \int_t \bar{\sigma}_{yy} dz & \sigma_{yz}^c &= \bar{\sigma}_{yz} \\ \sigma_{xy}^c &= \frac{1}{t} \int_t \bar{\sigma}_{xy} dz & \sigma_{xz}^c &= \bar{\sigma}_{xz} \end{aligned} \quad (5.62)$$

Meanwhile, the strains ε^c for whole composite ply are given as

$$\begin{aligned}\varepsilon_{zz}^c &= \frac{1}{t} \int_t \bar{\varepsilon}_{zz} dz & \varepsilon_{xx}^c &= \bar{\varepsilon}_{xx} \\ \varepsilon_{yz}^c &= \frac{1}{t} \int_t \bar{\varepsilon}_{yz} dz & \varepsilon_{yy}^c &= \bar{\varepsilon}_{yy} \\ \varepsilon_{xz}^c &= \frac{1}{t} \int_t \bar{\varepsilon}_{xz} dz & \varepsilon_{xy}^c &= \bar{\varepsilon}_{xy}\end{aligned}\quad (5.63)$$

$\bar{\sigma}_{ij}$ and $\bar{\varepsilon}_{ij}$ are from Matrix $[\bar{C}]$ in Equation (5.49), bar above variables at the right side of Equations (5.62) and (5.63) means strain average along undulating fiber direction of each composite ply. While σ_{ij}^c and ε_{ij}^c at left side of Equations (5.62) and (5.63) are from stiffness matrix $[C]^c$ (compliance matrix $[S]^c$), which is the combination stiffness of whole composite laminate. Before we calculate σ_{ij}^c and ε_{ij}^c , we list in-plane compliance elements of composite ply in Matrix $[\bar{Q}]$, which is derived from Matrix $[\bar{C}]$.

$$\bar{Q} = \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix} = \begin{bmatrix} \bar{S}_{11} & \bar{S}_{12} & \bar{S}_{16} \\ \bar{S}_{12} & \bar{S}_{22} & \bar{S}_{26} \\ \bar{S}_{16} & \bar{S}_{26} & \bar{S}_{66} \end{bmatrix}^{-1} \quad (5.64)$$

in which elements $\bar{S}_{ij}$ come from compliance matrix $[\bar{S}]$, which is the inverse matrix of average ply stiffness matrix $[\bar{C}]$. As shown in Figure 5.12, a composite ply consists of two warp average plies with thickness of $4/t$ and one fill average ply with thickness of $2/t$. Therefore, the laminate stiffness matrix $[A]^c$ can be obtained as

$$[A]^c = [\bar{Q}]^w \frac{t}{4} + [\bar{Q}]^f \frac{t}{2} + [\bar{Q}]^w \frac{t}{4} \quad (5.65)$$

In-plane compliance coefficients can then be described as

$$\begin{bmatrix} S_{11}^c & S_{12}^c & S_{16}^c \\ S_{21}^c & S_{22}^c & S_{26}^c \\ S_{61}^c & S_{62}^c & S_{66}^c \end{bmatrix} = t ([A]^c)^{-1} \quad (5.66)$$

$$\begin{aligned} \begin{bmatrix} S_{31}^c & S_{32}^c & S_{36}^c \end{bmatrix} &= \left(\begin{bmatrix} \bar{S}_{13}^w & \bar{S}_{23}^w & \bar{S}_{36}^w \end{bmatrix} \frac{t}{4} [Q]^w \right) ([A]^c)^{-1} \times 2 \\ &+ \left(\begin{bmatrix} \bar{S}_{13}^f & \bar{S}_{23}^f & \bar{S}_{36}^f \end{bmatrix} \frac{t}{2} [Q]^f \right) ([A]^c)^{-1} \end{aligned} \quad (5.67)$$

For out-of-plane compliance elements, it is obtained

$$\begin{aligned} \begin{bmatrix} S_{13}^c \\ S_{23}^c \\ S_{63}^c \end{bmatrix} &= - \begin{bmatrix} S_{11}^c & S_{12}^c & S_{16}^c \\ S_{21}^c & S_{22}^c & S_{26}^c \\ S_{61}^c & S_{62}^c & S_{66}^c \end{bmatrix} \frac{1}{t} \left(\begin{bmatrix} \bar{C}_{13}^w \\ \bar{C}_{23}^w \\ \bar{C}_{63}^w \end{bmatrix} \frac{t}{4} \frac{1}{\bar{C}_{33}^w} \right) \times 2 \\ &- \begin{bmatrix} S_{11}^c & S_{12}^c & S_{16}^c \\ S_{21}^c & S_{22}^c & S_{26}^c \\ S_{61}^c & S_{62}^c & S_{66}^c \end{bmatrix} \frac{1}{t} \left(\begin{bmatrix} \bar{C}_{13}^f \\ \bar{C}_{23}^f \\ \bar{C}_{63}^f \end{bmatrix} \frac{t}{2} \frac{1}{\bar{C}_{33}^f} \right) \end{aligned} \quad (5.68)$$

$$\begin{aligned} S_{33}^c &= \left[\frac{1}{t} \left(\frac{t}{4} \frac{1}{\bar{C}_{33}^w} \right) \times 2 + \frac{1}{t} \left(\frac{t}{2} \frac{1}{\bar{C}_{33}^f} \right) \right] - \begin{bmatrix} S_{31}^c & S_{32}^c & S_{36}^c \end{bmatrix} \\ &\times \frac{1}{t} \left[\left(\begin{bmatrix} \bar{C}_{13}^w \\ \bar{C}_{23}^w \\ \bar{C}_{63}^w \end{bmatrix} \frac{t}{4} \frac{1}{\bar{C}_{33}^w} \right) \times 2 + \left(\begin{bmatrix} \bar{C}_{13}^f \\ \bar{C}_{23}^f \\ \bar{C}_{63}^f \end{bmatrix} \frac{t}{2} \frac{1}{\bar{C}_{33}^f} \right) \right] \end{aligned} \quad (5.69)$$

$$\begin{bmatrix} S_{44}^c & S_{45}^c \\ S_{54}^c & S_{55}^c \end{bmatrix} = \frac{1}{t} \left(\frac{t}{4} \begin{bmatrix} \bar{S}_{44}^w & \bar{S}_{45}^w \\ \bar{S}_{45}^w & \bar{S}_{55}^w \end{bmatrix} \right) \times 2 + \frac{1}{t} \left(\frac{t}{2} \begin{bmatrix} \bar{S}_{44}^f & \bar{S}_{45}^f \\ \bar{S}_{45}^f & \bar{S}_{55}^f \end{bmatrix} \right) \quad (5.70)$$

With the combination of in-plane stiffness obtained from Equations (5.66) to (5.67) and out-of-plane stiffness obtained from Equations (5.68) to (5.70), the overall stiffness matrix for the composite ply is obtained in form

$$[S]^c = \begin{bmatrix} S_{11}^c & S_{12}^c & S_{13}^c & 0 & 0 & S_{16}^c \\ S_{21}^c & S_{22}^c & S_{23}^c & 0 & 0 & S_{26}^c \\ S_{31}^c & S_{32}^c & S_{33}^c & 0 & 0 & S_{36}^c \\ 0 & 0 & 0 & S_{44}^c & S_{45}^c & 0 \\ 0 & 0 & 0 & S_{54}^c & S_{55}^c & 0 \\ S_{61}^c & S_{62}^c & S_{63}^c & 0 & 0 & S_{66}^c \end{bmatrix} \quad (5.71)$$

where the right-hand part is a monoclinic stiffness matrix. Because there is only 90° rotation between warp ply and fill ply, there exists

$$S_{61}^c = S_{62}^c = S_{63}^c = S_{54}^c = S_{45}^c = S_{16}^c = S_{26}^c = S_{36}^c = 0 \quad (5.72)$$

Therefore, the stiffness matrix $[S]^c$ for composite ply is an orthotropic matrix. In this study, we set that x axis is along warp fiber direction, while y axis is along fill fiber direction. The Young's modulus along these two directions can be calculated as

$$E_{wrap} = E_1^c = \frac{1}{S_{11}^c} \quad \text{and} \quad E_{fill} = E_2^c = \frac{1}{S_{22}^c} \quad (5.73)$$

The calculated modulus along warp and fill directions for 7628 composite are listed in Table 5.3. Meanwhile, the modulus measured from physical tests is also listed in Table 5.3 for comparison. The mathematical results of 7628 composite elastic properties are calculated based on the material properties of glass fibers and resin matrix listed in Table 5.1, while experimental results were obtained by Shrotriya [21] with uniaxial tensile test method. It is found from the compari-

Table 5.3: Comparison of elastic properties for 7628 composite material

	E_{wrap} (GPa)	E_{fill} (GPa)
Experiment [9]	24.5 ± 0.5	19.8 ± 0.5
EQLAM	23.7	19.7
Direct bi-phase	22.7	17.4

Table 5.4: Orthotropic material properties of 7628 composite material

Young's modulus (GPa)	Shear modulus (GPa)	Poisson ratio
$E_{11}^c = 23.7$	$G_{23}^c = 3.8$	$\nu_{12}^c = 0.12$
$E_{22}^c = 19.7$	$G_{13}^c = 3.4$	$\nu_{21}^c = 0.10$
$E_{33}^c = 8.7$	$G_{12}^c = 8.7$	$\nu_{13}^c = 0.32$
		$\nu_{31}^c = 0.12$
		$\nu_{23}^c = 0.34$
		$\nu_{32}^c = 0.15$

son that small difference exists between analytical and experimental results, the mathematical results are satisfying and EQLAM model is proven to be reliable: Furthermore, the calculated modulus of 7628 composite from direct bi-phase multilayer model is also listed in Table 5.3. Note that the elastic properties of composite ply for direct bi-phase multilayer model are obtained based on PamCrash bi-phase Equations (5.2) to (5.7). Through comparison, EQLAM model illustrates higher accuracy than direct bi-phase model because the woven structure of glass fiber is considered in EQLAM model. At the same time, the detailed material properties of 7628 composite are listed in Table 5.4. They are required in numerical modeling of orthotropic composite ply for EQLAM multilayer model.

As shown in Figure 5.14, the whole EQLAM multilayer model can be modeled as 11 plies (6 elastic-plastic plies and 5 degenerated bi-phase plies) as depicted in Figure 5.14. With the decreasing of layers from 21 to 11, less layers are required to built into multilayer finite element, therefore the computational efficiency of EQLAM multilayer model is higher than that of direct bi-phase multilayer model.

Figure 5.14: Structure of EQLAM multilayer model in multilayer finite element code

After elastic modeling, damage modeling is presented. Since the integrated composite ply is an orthotropic ply, the damage behavior can be described by scalar damage parameters. A uniaxial tensile test is performed for 7628 prepreg ply, and the test results will be used to calculate the damage parameters along tensile direction. The test standard and procedure follow ASTM designation D3039/D3039M [115]. As shown in Figure 5.15, Instron 5569 mechanical tester is used for tensile test, instrument control and data record. Two lines are marked on the surface of test samples. The length between these lines is recorded by an optical camera, while the extensional length is measured and transferred to tensile strain by using Instron's Merlin software. This optical measuring method provides higher accuracy than the traditional method by using mounted strain gauges. Furthermore, it is useful for slim prepreg test sample whose thickness is only 0.2 mm, the case in this study.

7628 prepreg sample is separately tested along warp direction and fill direction. A crack line exhibits on test sample. The line is oblique along tensile direction as shown in Figure 5.16. The stress-strain curves for warp direction and fill direction are plotted as shown in Figures 5.17 and 5.18. As shown in Figure 5.16, matrix cracking causes marked lines on test sample flakes off and fiber break causes tensile force suddenly reduces at point *B* as shown in Figures 5.17 and 5.18. Bi-linear stress-strain curves are obtained based on idealization of original

Figure 5.15: Tensile test to measure damage properties of 7628 composite ply result curves [26][115]. Note that in reference [65], similar bi-linear stress-strain curve is obtained for woven fiber reinforced polymer matrix composite. As shown in Table 5.4, all Poisson ratios (ν_{12}^c , ν_{21}^c , ν_{13}^c , ν_{31}^c , ν_{23}^c and ν_{32}^c) for 7628 composites are already known. Next by the virtue of Equations (5.18) and (5.22), the equivalent scalar stress-strain curves (volume damage curves and shear damage curves) are also calculated from original stress-strain curves and plotted in Figures 5.17 and 5.18, which are used to obtain scalar damage parameters through Equations (5.12) and (5.15). The equivalent scalar damage parameters are listed in Table 5.5. They will be applied in degenerated bi-phase model to finish damage modeling of EQLAM multilayer model.

As a conclusion, EQLAM multilayer model use 11 layers in modeling process, rather than 21 layers in direct bi-phase multilayer model. This model still considers metallic plies and composite plies, and it use the same modeling method for metallic plies in direct bi-phase multilayer model. But for composite plies, with the application of EQLAM method, composite structure is combined into a ho-

Figure 5.16: Damage tensile test samples of 7628 prepreg

Figure 5.17: Stress-strain curves of 7628 prepreg along warp direction

Figure 5.18: Stress-strain curves of 7628 prepreg along fill direction

Table 5.5: Damage parameters for 7628 composite

Warp direction		Fill direction	
$\varepsilon_{si}^{wrap} = 0.355 \%$	$\varepsilon_{vi}^{wrap} = 0.280 \%$	$\varepsilon_{si}^{fill} = 0.268 \%$	$\varepsilon_{vi}^{fill} = 0.210 \%$
$\varepsilon_{s1}^{wrap} = 1.975 \%$	$\varepsilon_{v1}^{wrap} = 1.555 \%$	$\varepsilon_{s1}^{fill} = 2.162 \%$	$\varepsilon_{v1}^{fill} = 1.694 \%$
$\varepsilon_{su}^{wrap} = 2.012 \%$	$\varepsilon_{vu}^{wrap} = 1.584 \%$	$\varepsilon_{su}^{fill} = 2.199 \%$	$\varepsilon_{vu}^{fill} = 1.724 \%$
$d_{s1}^{wrap} = 0.526$	$d_{v1}^{wrap} = 0.526$	$d_{s1}^{fill} = 0.550$	$d_{v1}^{fill} = 0.550$
$d_{su}^{wrap} = 0.763$	$d_{vu}^{wrap} = 0.763$	$d_{su}^{fill} = 0.798$	$d_{vu}^{fill} = 0.798$

homogeneous ply. At the same time, the woven structure of reinforced fibers is also analyzed, which is simplified as uni-directional linear structure in direct bi-phase multilayer model. The total material properties of EQLAM multilayer model is summarized in Table 5.9.

5.4.3 Integrated Orthotropic Model

Integrated orthotropic model is an extended model based on EQLAM multilayer model. With the help of classical lamination theory and sublaminates modeling method, eleven-layer EQLAM multilayer model is integrated as a one-layer orthotropic model. Detailed material properties of whole multilayer circuit board

Table 5.6: Material properties for EQLAM multilayer model

	ED copper foil ply	E-glass fiber / Epoxy resin ply (7628)
Density	$8.92 \times 10^{-6} \text{kg/mm}^3$	$1.2 \times 10^{-6} \text{kg/mm}^3$
Ply thickness	0.0175 mm	0.18mm and 2 mm
Young's modulus	110 GPa	$E_{11}^c = 23.7 \text{ GPa}$, $E_{22}^c = 19.7 \text{ GPa}$, $E_{33}^c = 8.7 \text{ GPa}$
Shear modulus		$G_{23}^c = 3.8 \text{ GPa}$, $G_{13}^c = 3.4 \text{ GPa}$, $G_{12}^c = 8.7 \text{ GPa}$
Poisson's ratio	0.3	$\nu_{12}^c = 0.12$, $\nu_{21}^c = 0.10$, $\nu_{13}^c = 0.32$ $\nu_{31}^c = 0.12$, $\nu_{23}^c = 0.34$, $\nu_{32}^c = 0.15$
Yield stress	310 MPa	
Multi-linear law for metallic ply and damage parameters for composite ply	$E_1=469\text{MPa}$, $\sigma_1=315\text{MPa}$ $E_2=469\text{MPa}$, $\sigma_2=335\text{MPa}$ $E_3=469\text{MPa}$, $\sigma_3=355\text{MPa}$ $E_4=469\text{MPa}$, $\sigma_4=355\text{MPa}$ $E_5=-72.4\text{MPa}$, $\sigma_5=279\text{MPa}$ $E_6=-72.4\text{MPa}$, $\sigma_6=179\text{MPa}$ $E_7=-72.4\text{MPa}$, $\sigma_7=17 \text{ MPa}$	$\varepsilon_{si}^{wrap} = 0.355 \%$, $\varepsilon_{si}^{fill} = 0.268 \%$ $\varepsilon_{s1}^{wrap} = 1.975 \%$, $\varepsilon_{s1}^{fill} = 2.162 \%$ $\varepsilon_{su}^{wrap} = 2.012 \%$, $\varepsilon_{su}^{fill} = 2.199 \%$ $\varepsilon_{vi}^{wrap} = 0.280 \%$, $\varepsilon_{vi}^{fill} = 0.210 \%$ $\varepsilon_{v1}^{wrap} = 1.555 \%$, $\varepsilon_{v1}^{fill} = 1.694 \%$ $\varepsilon_{vu}^{wrap} = 1.584 \%$, $\varepsilon_{vu}^{fill} = 1.724 \%$ $d_{s1}^{wrap} = 0.526$, $d_{s1}^{fill} = 0.550$ $d_{su}^{wrap} = 0.763$, $d_{su}^{fill} = 0.798$ $d_{v1}^{wrap} = 0.526$, $d_{v1}^{fill} = 0.550$ $d_{vu}^{wrap} = 0.763$, $d_{vu}^{fill} = 0.798$

Table 5.7: Material properties of integrated orthotropic model

Young's modulus (GPa)	Shear modulus (GPa)	Poisson ratio
$E_{11} = 32.6$	$G_{23} = 4.0$	$\nu_{12} = 0.19$
$E_{22} = 29.1$	$G_{13} = 3.6$	$\nu_{21} = 0.17$
$E_{33} = 9.7$	$G_{12} = 7.2$	$\nu_{13} = 0.30$
		$\nu_{31} = 0.10$
		$\nu_{23} = 0.32$
		$\nu_{32} = 0.12$

are listed in Table 5.7, and only degenerated bi-phase model is applied for numerical modeling.

In order to verify calculated material property, a tensile test is applied to measure Young's Modulus of whole six-layer circuit board. The test standard and procedure follow ASTM designation D3552/D3552M [116]. The measured range of elastic strain is set around 0.15% [21].

As shown in Figure 5.19, Instron 5569 mechanical tester is used for tensile test, instrument control and data record. Meanwhile, analysis is performed through

an Instron's Merlin software. Circuit board samples along warp direction and fill direction are tested separately. The stress-strain points are plotted and linearized as two-order trendlines shown in Figure 5.20, and the equations of these trendlines are also presented. The slopes of trendlines prove Young's Modulus, which are compared with the results from three mathematical models. The results are listed in Table 5.8. Note that the elastic properties of whole circuit boards for direct bi-phase multilayer model and EQLAM multilayer model are obtained based on the assumption that PamCrash multilayer model also follows classical laminate theory [111]. High reliability exists in static test verification.

Figure 5.19: Tensile test for a six-layer circuit board sample

After elastic modeling, damage modeling is presented. Since integrated orthotropic ply also can be described by scalar damage parameters. In the above uniaxial tensile tests as shown in Figure 5.19, damage stress-strain curve of whole multilayer circuit board can be plotted. PCB sample is separately tested along warp direction and fill direction. Stress-strain curves for warp direction and fill direction are plotted as shown in Figures 5.17 and 5.18. As shown in the figures,

Figure 5.20: Stress-strain relationship for six-layer circuit board

Table 5.8: Comparison of elastic properties for six-layer circuit board

	E_{wrap} (GPa)	E_{fill} (GPa)
Experiment	32.6	28.2
Direct bi-phase multilayer	31.9	27.2
EQLAM multilayer	32.6	29.1
Integrated orthotropic	32.6	29.1

the crack of PCB sample causes tensile force suddenly reduces at point B . Bi-linear stress-strain curves are obtained based on the idealization of original result curves [26][115]. As shown in Table 5.7, all the Poisson ratios (ν_{12} , ν_{21} , ν_{13} , ν_{31} , ν_{23} and ν_{32}) for 7628 composites are already known. Next by the virtue of Equations (5.18) and (5.22), the equivalent scalar stress-strain curves (volume damage curves and shear damage curves) are also calculated from original stress-strain curves and plotted in Figures 5.21 and 5.22, which are used to obtain scalar damage parameters through Equations (5.12) and (5.15). The equivalent scalar damage parameters are listed in Table 5.9. They will be applied in the degenerated bi-phase model to finish the damage modeling of integrated orthotropic model.

Figure 5.21: Damage stress-strain curves of six-layer PCB along warp direction

Figure 5.22: Damage Stress-strain curves of six-layer PCB along fill direction

Table 5.9: Damage parameters for six-layer circuit board

Warp direction		Fill direction	
$\varepsilon_{si}^{wrap} = 0.171 \%$	$\varepsilon_{vi}^{wrap} = 0.121 \%$	$\varepsilon_{si}^{fill} = 0.177 \%$	$\varepsilon_{vi}^{fill} = 0.125 \%$
$\varepsilon_{s1}^{wrap} = 1.649 \%$	$\varepsilon_{v1}^{wrap} = 1.167 \%$	$\varepsilon_{s1}^{fill} = 1.641 \%$	$\varepsilon_{v1}^{fill} = 1.158 \%$
$\varepsilon_{su}^{wrap} = 1.664 \%$	$\varepsilon_{vu}^{wrap} = 1.177 \%$	$\varepsilon_{su}^{fill} = 1.660 \%$	$\varepsilon_{vu}^{fill} = 1.171 \%$
$d_{s1}^{wrap} = 0.355$	$d_{v1}^{wrap} = 0.355$	$d_{s1}^{fill} = 0.426$	$d_{v1}^{fill} = 0.426$
$d_{su}^{wrap} = 0.621$	$d_{vu}^{wrap} = 0.621$	$d_{su}^{fill} = 0.673$	$d_{vu}^{fill} = 0.673$

The integrated orthotropic model also can be used in simulation with the help of degenerated bi-phase model. However, integrated orthotropic model is not a multilayer model, it is a one-layer homogeneous model. There will be some error from single-layer model if the bending analysis is performed. As mentioned in Section 3, three integration points is enough to integrate bending and membrane effects in multilayer model. In direct bi-phase multilayer model, there are 21 layers with 21 integration points; and EQLAM multilayer model with 11 layers and 11 integration points. The precision of bending moments can be kept. However, in the integrated orthotropic model, the associated simulation model has less than three integration points. This definitely will cause some limitations in the simulation including bending analysis. On the other hand, since only one orthotropic layer is modeled in integrated orthotropic model, rather than 21 layers in direct bi-phase multilayer model and 11 layers in EQLAM multilayer model, simulation based on this modeling method will numerously reduce associated CPU time. This will be proven in the following simulation analysis. Finally, the totally material properties of integrated orthotropic model is summarized in Table Table 5.10.

5.5 Dynamic Analysis for Multilayer Circuit Boards

In the modeling of multilayer circuit board, static analysis is performed at the same time in above. Both of them are successfully performed at micro-structure and board levels, and have a good agreement with experimental results. After mod-

Table 5.10: Material properties for integrated orthotropic model

Density	$1.976 \times 10^{-6} \text{ kg/mm}^3$			
Ply thickness	1.045 mm			
Young's modulus	$E_{11}^c = 32.6 \text{ GPa}, E_{22}^c = 29.1 \text{ GPa}, E_{33}^c = 9.7 \text{ GPa}$			
Shear modulus	$G_{23}^c = 4.0 \text{ GPa}, G_{13}^c = 3.6 \text{ GPa}, G_{12}^c = 7.2 \text{ GPa}$			
Poisson's ratio	$\nu_{12}^c = 0.19, \nu_{21}^c = 0.17, \nu_{13}^c = 0.30$ $\nu_{31}^c = 0.10, \nu_{23}^c = 0.32, \nu_{32}^c = 0.12$			
Damage parameters	$\varepsilon_{si}^{wrap} = 0.171 \%$, $\varepsilon_{s1}^{wrap} = 1.649 \%$, $\varepsilon_{su}^{wrap} = 1.664 \%$, $d_{s1}^{wrap} = 0.355$, $d_{su}^{wrap} = 0.621$,	$\varepsilon_{vi}^{wrap} = 0.121 \%$, $\varepsilon_{v1}^{wrap} = 1.167 \%$, $\varepsilon_{vu}^{wrap} = 1.177 \%$, $d_{v1}^{wrap} = 0.355$, $d_{vu}^{wrap} = 0.621$	$\varepsilon_{si}^{fill} = 0.177 \%$, $\varepsilon_{s1}^{fill} = 1.641 \%$, $\varepsilon_{su}^{fill} = 1.660 \%$, $d_{s1}^{fill} = 0.426$, $d_{su}^{fill} = 0.673$,	$\varepsilon_{vi}^{fill} = 0.125 \%$, $\varepsilon_{v1}^{fill} = 1.158 \%$, $\varepsilon_{vu}^{fill} = 1.171 \%$, $d_{v1}^{fill} = 0.426$, $d_{vu}^{fill} = 0.673$

eling and static analysis, dynamic analysis is performed based on established models (direct bi-phase model, EQLAM multilayer model, and integrated orthotropic model). In this section, dynamic analysis will be performed in simulation and physical tests. Simulation is mainly performed to investigate the drop-impact response of multilayer circuit board. Meanwhile, some experimental results will be used to verify the reliability of three modeling method in dynamic simulation.

5.5.1 Experimental Setup for Dynamic Tests

In order to investigate dynamic response of multilayer circuit boards in drop impact condition, a free-fall drop test of circuit boards was conducted by using Lansmont Model 65/81-shock table, as shown in Figure 5.23. The shock table was made of aluminum and has an assembled mass of approximately 190.5kg. It has a 650mm x 810mm top-mounting surface, which was drilled and fitted with threaded inserts for mounting of test specimens. As shown in Figure 5.23, a six-layer circuit board is mounted on drop machine through an aluminum fixture in experiment. The circuit board was clamped at two edges, 5mm away from each edge. Drop height of shock table is set as 0.1m and 1m. And drop impact response in these two heights was used to verify numerical models. During drop test, a three-element strain rosette was mounted at the center of circuit board as shown in Figure 5.23.

Figure 5.23: Drop impact experiment setup

It is a rectangular (0-45-90 degree) strain gage as shown in Figure 5.24. With the measurement of three-axis strains (ε_x , ε_y and ε_{45°), the principal strain can be calculated [112]. Strain components ε_x and ε_y can be determined at a given point of free surface of a material by measuring normal strain along x - and y -axes drawn through the point. The shearing strain γ_{xy} can be determined by measurement of normal strain along 45° bisector of x and y axes using

$$\gamma_{xy} = 2\varepsilon_{45^\circ} - (\varepsilon_x + \varepsilon_y) \quad (5.74)$$

After calculating γ_{xy} , maximum and minimum principal strain, $\varepsilon_{\max}$ and $\varepsilon_{\min}$, can be calculated using the following equations:

$$\varepsilon_{\max} = \frac{\varepsilon_x + \varepsilon_y}{2} + \sqrt{\left(\frac{\varepsilon_x - \varepsilon_y}{2}\right)^2 + \left(\frac{\gamma_{xy}}{2}\right)^2} \quad (5.75)$$

$$\varepsilon_{\min} = \frac{\varepsilon_x + \varepsilon_y}{2} - \sqrt{\left(\frac{\varepsilon_x - \varepsilon_y}{2}\right)^2 + \left(\frac{\gamma_{xy}}{2}\right)^2} \quad (5.76)$$

In order to investigate the strain along three directions, three strain meters were applied. Through three 350B strain boxes, strain meters and strain gage were connected together. Finally, strain can be captured by a digital oscilloscope that is connected with strain meters. Meanwhile, an accelerometer was mounted on the fixture as shown in Figure 5.23. The measured transient acceleration will be applied in simulation as a boundary condition along fixed edges of circuit board as shown in Figure 5.25.

Figure 5.24: A three-element strain rosette

Figure 5.25: Drop impact simulation model

5.5.2 Dynamic Simulation

As shown in Figure 5.25, the acceleration boundary applied to two longer clamped edges of circuit board is obtained from the accelerometer measurements mentioned in previous section. This approach avoids simulation of drop table structure and the contact effects of table-to-foundation contact event, as shown in Figure 5.23. It minimizes errors when comparing simulation results to full-scale simulation including shock dynamic response at the central point will be calculated and compared with experimental results.

Finite element circuit board model consists of 5120 shell elements and 5265 nodes as shown in Figure 5.26. Element 2520 is the central element according to the central point of circuit board. Only two-dimensional shells exist in this model, multilayer and orthotropic properties will be simulated as material type model that is already established in PamCrash code. This method can improve computational efficiency and simplify simulation process since no complex three-dimensional cell-element model is required.

Figure 5.26: Meshed two-dimensional circuit board model

5.5.3 Comparison of Simulation and Test Results

In numerical simulation, three models (direct bi-phase multilayer model, EQLAM multilayer model and integrated orthotropic model) are applied in meshed 2D finite element model. As shown in Figure 5.25, longer length is fixed in test, which is along warp direction (see Figures 5.2 and 5.25). In drop impact test, vibration dominantly takes place along shorter direction, which is along fill direction. Therefore, the damage behavior along fill direction is considered for composite ply in simulation with EQLAM multilayer model.

The amplitude of maximum and minimum principal strains at central point are compared between simulation results of three models and experimental as shown in Tables 5.11 and 5.14. For Tables 5.11 and 5.12, the comparison is performed based on the drop height of 0.1 meter; in Tables 5.13 and 5.14, the drop height was set as 1 meter. The percentage error is also given as a comparison with experimental results. Figure 5.27 also illustrates the comparison of principal strains between three numerical models and experimental results. It is found from comparison that, both direct bi-phase multilayer model and EQLAM multilayer model show a good agreement with experimental results. However, integrated orthotropic model shows larger error than multilayer model. As explained in Section 5.4.3, in integrated orthotropic model, the integration points spaced along shell thickness is insufficient to describe bending moments exactly, this cause large error in the simulation of dynamic behavior that includes bending behavior. Furthermore, between multilayer models, EQLAM multilayer model illustrates higher accuracy than direct bi-phase multilayer model. This is the same as static test verification shown in Table 5.3.

5.5.4 Comparison of CPU Time in Different Models

All three numerical models were performed on a Sun Workstation (ULTRA 10 with 360 MHz). The total simulation time for each model is concluded in Table 5.15.

Table 5.11: Comparison of experimental and numerical peak values of maximum principal strain at central point when the drop height of 0.1 meter

	Peak values of maximum principal strain (%)	Error (%)
Experimental	0.069	
Direct bi-phase multilayer	0.073	5.80
EQLAM multilayer	0.071	2.90
Integrated orthotropic	0.055	20.29

Table 5.12: Comparison of experimental and numerical peak values of minimum principal strain at central point when the drop height of 0.1 meter

	Peak values of minimum principal strain (%)	Error (%)
Experimental	- 0.0430	
Direct bi-phase multilayer	- 0.0292	32.09
EQLAM multilayer	- 0.030	30.23
Integrated orthotropic	- 0.011	74.72

Figure 5.27: Peak values of principal strains for different models

The CPU time taken increases with greater number of elements and increased modeling layers of circuit board model. As we know, nowadays, the CPU speed

Table 5.13: Comparison of experimental and numerical peak values of maximum principal strain at central point when the drop height of 1 meter

	Peak values of maximum principal strain (%)	Error (%)
Experimental	0.158	
Direct bi-phase multilayer	0.165	4.43
EQLAM multilayer	0.153	3.16
Integrated orthotropic	0.133	15.82

Table 5.14: Comparison of experimental and numerical peak values of minimum principal strain at central point when the drop height of 1 meter

	Peak values of minimum principal strain (%)	Error (%)
Experimental	-0.0578	
Direct bi-phase multilayer	-0.0666	15.22
EQLAM multilayer	-0.0595	2.94
Integrated orthotropic	-0.0360	37.72

of a normal workstation usually can achieve more than 3 GHz, 5-hour simulation in 360 MHz workstation can be easily finished in a new workstation much faster. Therefore, hybrid modeling method presented in this work is practical.

5.5.5 Discussion of Modeling Techniques

With three developed models, the analysis of multilayer circuit boards can be performed at both micro-structure and board levels. All the models have high flexibility and efficiency for circuit board evaluation and design: the microstructure and properties of glass fiber, resin matrix and copper foil can be changed as designers' requirement. 2D structure of geometry model provides a high computational efficiency in numerical simulation. Furthermore, the developed circuit boards model provides a good platform for detailed drop impact analysis of electronic components attached on circuit boards. Chips and modules can be simulated into circuit board model, the stress/strain and failure in a given package can then be predicted.

Table 5.15: Comparison of CPU time

Material model	CPU time
Direct bi-phase multilayer model	5 hr 19min 30 sec
EQLAM multilayer model	3 hr 16 min 30 sec
Integrated orthotropic model	1 hr 6 min 58 sec

Based on present study, EQLAM multilayer model is recommended if efficient modeling is required: it not only illustrates accuracy in static tests and dynamic tests among three models, but also shows higher computational efficiency than direct bi-phase multilayer model. However, if a easy-handling modeling method is needed, direct bi-phase multilayer model is preferred, the simplification of woven fiber to unidirectional fiber avoids complex mechanics analysis, and error is acceptable in presented circuit board model. For integrated orthotropic model, although it doesn't consider bending moments, it is easily solved in simulation.

5.5.6 Stress and Strain Analysis

To facilitate the study of potential PCB failures due to over flexures and stresses, stress and strain on PCB is investigated during whole drop event. EQLAM multilayer model is used in this study, and one meter drop height is selected. Figure 5.28 shows that the strain concentration area (hotspot) on multilayer circuit board, denoted by high strain contours at the center of PCB. Meanwhile, low strain areas (blue) on multilayer circuit board, denoted by low strain contours at clamped sides. The strain of element 4881 (located in low strain zone) has the minimum strain that is closed to zero. While, the strain of element 2640 (located in high strain zone) has maximum strain (0.165%) that is closed to strain (0.153%) at central element 2520. The central element 2520 is also located in high strain area. The axis of symmetry for this strain contours is along the center of PCB Therefore, three positions along z-axis is selected for further study: Positions A, B and C. Position B is the central area containing Element 2520 and Node 2777. The transient

Figure 5.28: Strain contours of whole PCB at the instant of 6.6 ms (maximum principle strain)

response of strain at these three positions is illustrated as shown in Figure 5.29. At the instant of 6.6 ms, the strains at three positions achieve their peaks at the same time, and this instant is selected as the moment of strain contour shown in Figure 5.28. In addition, the strain responses at Positions A and C are the same to each other, this is because of symmetry distribution of strain on PCB. And the strain at center area, Position B, is obviously higher than other two side positions, Positions A and C.

Figure 5.29: Maximum principal strain profiles at Points A, B and C

Besides strain, stress on PCB is also investigated. As shown in Figure 5.30, The maximum principal stress contour at the instant of 6.6 ms is presented. The stress contour is similar to the strain contour shown in Figure 5.30. High stress (hotspot area) concentrates at the central line of PCB, while low strain (blue area) is located at clamping edges of PCB. The stress of Element 2640 is highest, which is in high stress zone. In addition, the stress of Element 164 is minimum, which is located in low stress zone. The axis of symmetry for this stress contours is along the center of PCB (x -axis). The maximum principle stress on Positions A, B and C are plotted and compared in Figure 5.31. The stress at Position B is obviously

higher than the stresses on Positions A and C, while the stresses of Positions A and C are similar to each other.

Figure 5.30: Stress contours of whole PCB at the instant of 6.6 ms (maximum principle stress)

Note that in Pam-Crash code, the stresses and strain are calculated separately for each of phases, such as bi-phase ply and elastic-plastic damage ply [72]. Only the strains and stresses of the outer layers (copper layers in this work) will be presented in the plot of result. For initial concern, EQLAM multilayer model consists of elastic-plastic damage plies (for copper layers) and degenerated bi-phase plies (for composite layers). It is found from Table 5.9 that, the stiffness of copper layers is much higher than other composite plies. Therefore, if the stress value presented is from copper plies, it will be much higher than equivalent stress of whole composite model under the same strain effect. In order to prove this point, the maximum principle stress and strain of Element 2520 (central position B) is plotted in Figure 5.32. As shown in Figure 5.32, the peak values of maximum stress (0.181 GPa) and strain (0.153) happen at the same time, and their ratio is

Figure 5.31: Maximum principal stress profiles at Points A, B and C

118 GPa. Omit the error from finite element integration, this ratio is closed to 110 GPa, the Young's modulus of copper foil. Because it is much higher than the Young's modulus of composite ply shown in Table 5.9, the illustrated stress and strain at the peak point is from copper foil, not equivalent stress and strain.

Figure 5.32: The transient response of maximum principle stress and strain at the central position B in Figures 5.28 and 5.32

5.6 Summary of Chapter

Three modeling methods are proposed in this paper to fulfill the requirement related to research and the utilization of multilayer circuit board with multilayer multimaterial composite material. The three are direct bi-phase multilayer model, EQLAM multilayer model and integrated orthotropic model. In comparison, the EQLAM multilayer model is recommended if efficient modeling is required as it not only demonstrates good accuracy in the static tests and dynamic tests, but also shows a higher computational efficiency than direct bi-phase multilayer model. With the innovation of these hybrid-modeling techniques, the analysis of multilayer circuit board consisting of woven fabric-reinforced polymer matrix and metal plies can be performed at both micro-structure and board levels. The models have high flexibility and efficiency: (1) the micro structure and properties of reinforced fibers, polymer matrix and metallic plies can be changed conveniently in 3D mechanics unit-cell model; (2) The 2D structure of geometry model provides a high computational efficiency in the numerical simulation. The work presented also provides the damage modeling methods, multi-linear damage law and scalar damage parameters, to simulate damage behavior after impact.

5.7 Final Development of Full-scale Virtual Buffer Model with the Contribution from Hybrid Modeling of Multilayer Circuit Boards

The objective of this PhD work is to build a virtual buffer model to realize efficient drop-impact simulation. This work can be finally complete in full-scale virtual buffer model as shown in Figures 3.24: multilayer circuit board is objective model, while the boundary field obtained from first step can be used to build virtual buffer model. In Chapter 4, the boundary field from nonlinear drop-impact cushion system has been successfully derived from local linearization model. Next

5.7 Final Development of Full-scale Virtual Buffer Model with the Contribution from Hybrid Modeling of Multilayer Circuit Boards 193

in Chapter 5, with the combination of 3D mechanics unit-cell model and 2D multilayer finite element model, multilayer circuit board can be efficiently modeled by using hybrid-modeling techniques, which improve the efficiency of virtual buffer model and finally make it complete.

5.7 Final Development of Full-scale Virtual Buffer Model with the Contribution from Hybrid Modeling of Multilayer Circuit Boards

194

Figure 5.33: Flow chart of full-scale virtual buffer model

Chapter 6

Application of Full-Scale Virtual Buffer Model in a General Prototype for Packaged Electronic Products

Full-scale virtual buffer model is complete as shown in Section 5.6. In this chapter, a general drop impact prototype devised for packaged electronic products is presented to illustrate the application procedure of full-scale virtual buffer model. The developed local linearization model is utilized to obtain the transient response from nonlinear cushion system. This transient response is taken as boundary field to establish virtual buffer model. The CPU time spent on cushion components can be totally saved. In addition, a multi-layer circuit board is efficiently modeled by hybrid modeling techniques and settled into virtual buffer model as an objective model. Through this work, the shock response of multilayer printed circuit boards and their interaction with nonlinear cushion material are fully investigated. Finally, the verification tests and discussion for method efficiency are presented. The full-scale virtual boundary model illustrates a good agreement with experimental results and good computational efficiency needed for finite element simulation.

6.1 Introduction of Chapter

The experimental drop-impact test is widely used for the quality assurance of existing electronic products. However, physical tests are expensive and the preparation is quite time-consuming, the data collected is also limit. Therefore, finite element simulation is usually provided to solve these problems, especially for the quality examination of prospective product design. Usually the complex structure of electronic products causes large-scale finite element model, which includes excessive simulation elements and degrees of freedom, it can burden too much computing resources to make simulation cannot be implemented. Therefore, several special techniques were utilized to achieve high-efficiency for large-scale simulation, as reviewed in Section 2.3, super element method, global/local method, virtual boundary method, etc. Through the reduction of computational resource on the external components outside of the region of interest, these techniques can achieve solution for large-scale finite element model with reasonable computational time.

In previous study, global/local methodology has been successfully applied to realize a quick simulation in finite element analysis of packaged TV model [26]. Furthermore, a virtual boundary model was developed based on the resurvey of global/local technique [117]. This method illustrated higher efficiency than global/local method, especially in optimization analysis. In this chapter, a full-scale virtual buffer model will be used: local linearization model will be used to build virtual buffer model, and Multilayer EQALM model is used to model objective model that is inserted into virtual buffer model.

In this chapter, a general prototype for packaged electronic products will be presented in product-level drop tests. In this prototype, nonlinear cushion buffers and multilayer circuit boards are both concerned. A full-scale virtual buffer model will be used to simulate drop tests for this prototype and predict the dynamic response of multilayer circuit board in nonlinear drop-impact condition. After that, experimental results will be presented to verify the reliability of this full-

scale virtual buffer model. Finally, some discussions focusing on the efficiency and accuracy of full-scale virtual boundary model are performed, it is proposed to guide further study on high-efficient techniques.

6.2 Drop-Test Arrangement and Parameters

As reviewed in Section 2.1, an important requirement by many manufactures of electronic products is the ability to satisfy mechanical shock tests that generally fall into two categories: board-level and product-level. In board-level tests, a test board with electronic components mounted on it is fixed to a drop platen; this enables the assembly to be subjected to controlled drop impact onto rigid surface. In product-level tests, the entire product is gripped at a specific orientation, allowed to fall under gravity and released from gripper to strike a desired surface. The product is free to rebound and experience subsequent impacts. This mimics the dynamic response of an actual drop impact.

The purpose of board-level test is to simulate using a test board, the same mechanical effects that are observed with respect to a PCB inside an electronic product during a drop-impact. In previous sections, all dynamic tests, cushion tests and PCB tests, are performed on board-level. However, board-level testing cannot accurately duplicate the state of loading experienced by electronic components, since a test board is mounted firmly onto drop platen using columns supports, its orientation at impact is fixed and it is unable to execute its natural dynamics during impact. In short, product-level test is more close to practical situation, while board-level test is under idealized condition. At the last part of this report, full-scale virtual buffer model is used to predict dynamic response in product-level drop tests.

Product-level testing is effected using a precision drop tester, Lansmont PDT-56ED drop tester, as shown in Figure 6.1. Drop tester allows drop leaf to pick up heavy, cumbersome packages and position them accurately at a chosen drop

height. In drop test, drop leaf falls straight down at acceleration levels greater than $1g$, imparting no rotational forces on package and assuring accurate flat free-fall drops.

Figure 6.1: PDT-56ED drop tester and test specimen

Figure 6.2: 0° , 45° and 90° orientations of fixtures

For test specimens, a PCB is mounted onto a metal base via fixtures as

shown in Figure 6.1. Between metal base and drop leaf, cushion foam is used to reduce drop impact force. In this drop test, the response of PCB with various impact orientations is expected to investigate. However, PDT-56ED drop tester only can provide flat free-fall drops. Therefore, different fixtures are designed to facilitate various impact orientations to PCB. As shown in Figure 6.2, three fixtures are provided, the orientations of 0° , 45° and 90° are defined by the angle between PCB and horizontal axis.

In initial test, three-element strain rosette is applied to measure the response of PCB under drop impact as shown in Figure 6.2. As mentioned in Chapter 5, the application of strain gauge can avoid the error influence from sensors whose mass can change the vibration response of PCB with thin-wall structure. However, there are also some problems arising from this measuring method. In strain measurement, all reading data is voltage information shown in oscilloscope that can record the change of electrical resistance of strain gauge. The resistance of strain gauge varies in proportion to the amount of strain on PCB. The current is delivered between different devices, meanwhile, the inductive noise and resistance noise happens. In this test, the electronic noise level is around 0.25 Volt. After conversion, the strain noise level is around 0.025%. That is to say, even there is no strain on PCB, oscilloscope will still illustrate the strain value of 0.025%. The trigger function in recorder is designed to solve this problem. If trigger level is higher than noise level, the oscilloscope will not record any data until the strain on PCB exceeds the noise level. However, if trigger function is applied, another problem happens: if the maximum strain of PCB is less than 0.025% at the instant of impact, the recorder cannot be triggered and record any data when impact happens. In previous shock table test, the maximum strain normally can achieve 0.1%, which is larger than trigger level 0.025%. Therefore, the strain can be measured at the instant of impact. But in drop test, cushion buffers are applied to reduce the impact force on metal base. Because the stimulation is greatly reduced, the maximum strain on PCB in drop test is less than trigger level, then recorder cannot work. Thus current strain gauge is not suitable for this test.

Accelerometer is used, since the accuracy of strain gauge is not acceptable in this test. As shown in Figures 6.3 and 6.4, an accelerometer is mounted at center of PCB that is fixed in the fixtures of 0° , 45° and 90° respectively, the weight of sensor is only 9.2 grams. Dynamic response of PCB will be measured by an accelerometer which is connected to a Nicolet analyzer. Besides the measurement of acceleration on PCB, the stimulation applied on the edges of PCB is also measured as shown in Figure 6.5, where the sensor is attached on metal fixture directly.

Figure 6.3: Improved experimental set-up for 0° orientation of fixture

Figure 6.4: Improved experimental set-ups for 45° and 90° orientations of fixture

Drop heights were set as 0.5m, 0.8m and 1m in test, and F6.1 multilayer

Figure 6.5: Drop test set up to measure the impact force between metal base and PCB

circuit boards and EPS20 polyfoam were used. In this, the drop impact conditions for multilayer circuit board under different angles and different drop height will be investigated.

6.3 Application of Full-Scale Virtual Buffer Model

As show in Figure 5.33, full-scale virtual buffer model is finally achieved for packaged electronic product models: all the CPU resource will be used in the analysis of concerned parts (multilayer circuit board) with a high efficient way (2D simulation). The original time-consuming work (3D simulation for cushion buffers and multilayer multimaterial composite) is avoided by the replacement of efficient non-simulation work. In this chapter, a general prototype for packaged electronic products is presented as shown in Section 6.2. As this prototype as an example, efficient analysis will be performed by using full-scale virtual buffer method. Shown in Figure 5.33, two steps will be adopted: (1) Establish virtual buffer model; (2) Create objective model and place it into virtual buffer model.

6.3.1 Establishment of Virtual Buffer Model

The establishment of virtual buffer model is based on the application of local linearization method for dropped nonlinear cushion systems. As shown in Figure 6.6, all the drop-impact prototypes can be simplified as nonlinear one DOF systems. Since the weight of PCB is only 0.085 kg, while the weight of steel base is around 20 kg, the structure and fix characterizations of PCB on the metal base has a slim influence to the interaction response between cushion buffer and metal base. The simplification to one DOF systems is feasible. Also this single-effect is an important pre-condition for the use of virtual buffer model as mentioned in the conclusion part of Chapter 3.

For the prototype presented in this work, as shown in Figure 6.6, the virtual buffer model is just boundary field, which is dynamic response from nonlinear one DOF systems. This is different to the virtual buffer model presented in Chapter 3, where virtual buffer model is the combination of TV base and boundary field. Because the TV base in Chapter 3 is shell structure, the flexibility of shell model cause the different boundary field between buffer/base and base/PCB. However, for rigid solid base, the stimulation applied on solid base can be exactly transferred to PCB, the difference mentioned above doesn't exist. Therefore, the rigid solid base can be simplified and removed in virtual buffer model, only boundary field exists.

In the calculation of boundary field, it is necessary to use local linearization method. In this project, the weight of solid base is 21 kg; the drop heights are set as 0.5 meter, 0.8 meter and 1 meter; the cushion buffer is EPS 20 polyfoam with the size of 50mm $\times$ 200mm $\times$ 200mm. Local linearization model is already programmed into Visual Basic package as shown in Figure 4.21. The dynamic responses (dynamic loading and transient acceleration response) for three drop-heights are calculated and plotted shown in Figure 6.7. The dynamic responses are the boundary field required in virtual buffer model. Till now, the first step for full-scale virtual buffer model is finished, the boundary field will be applied on

Figure 6.6: Establish virtual buffer model for prototypes

objective models as shown in Figure 6.6 in next Step 2.

Figure 6.7: Transient acceleration response for different drop heights (from local linearization model)

6.3.2 Creation of Objective Model for Incorporation into Virtual Buffer Model

After the establishment of virtual buffer model, the objective model (multilayer circuit board) is modeled and placed into virtual buffer model. Because virtual buffer model is only boundary field as mentioned above, transient dynamic response obtained from nonlinear drop impact models will be directly applied on the edges of multilayer PCB model. As mentioned in Chapter 5, EQLAM multilayer modeling method is recommended shown in Figure 6.8.

Firstly, the dynamic response of multilayer circuit boards under different drop heights is studied. As shown in Figure 6.7, the boundary fields under different drop heights (0.5 meter, 0.8 meter and 1 meter) are extracted and applied to PCB model. The direction of PCB is set as 0 degree to simulate horizontal drop cases. As shown in Figure 6.3, an accelerometer is mounted on PCB, the weight of this sensor

Figure 6.8: Modeling process of EQLAM multilayer modeling method

is 9.8 gram, which cannot be ignored since the weight of PCB is 85 gram. Therefore as shown in Figure 6.9, the geometry model of objective model is presented as the combination of accelerometer model and PCB model. The acceleration of chip (accelerometer) is calculated and plotted in Figure 6.10. As shown in Figure 6.3, the acceleration vertical to PCB plane is measured, which is along drop direction. Thus the acceleration along drop direction is provided in the present study.

Figure 6.9: Connection of objective model with full-scale virtual buffer model

The dynamic response of PCB under different drop angles is studied next. As shown in Figure 6.6, drop angles are selected as 0° , 45° and 90° . This is proposed to simulate horizontal, inclined and vertical drop cases. The drop height is fixed as 1 meter, and only the acceleration vertical to PCB plane is measured as shown in Figures 6.3 and 6.4. Therefore, only out-of-plane acceleration is provided for following verification work. The calculated accelerations are plotted as shown in Figure 6.11. In this, the second step for full-scale virtual buffer model is finished. The final results shown in Figures 6.10 and 6.11 will be compared

Figure 6.10: Dynamic response of horizontal dropped PCB under different drop heights (acceleration along vertical drop direction)

with experimental results from the drop tests mentioned in Section 6.2 for the verification of full-scale virtual buffer model.

Figure 6.11: Dynamic response of PCB under different drop angles (acceleration vertical to PCB plane)

6.4 Verification of Full-Scale Virtual Buffer Model by Experimental Results

Two groups of results are outputted from full-scale virtual buffer model: (1) The calculated boundary field from local linearization model, this is used to establish virtual buffer model in Step 1; (2) The drop impact response of multilayer circuit boards in packaged electronic products. The set-ups of verification testing are listed in Section 6.2.

6.4.1 Verification of Boundary Field in Virtual Buffer Model

Firstly, the acceleration of metal base is used to verify the boundary field calculated from local linearization model. The experimental set-ups are shown in Figure 6.5. The prototype is respectively dropped from the heights of 0.5 meter, 0.8 meter and 1.0 meter. The experimental transient acceleration results (twice drops - Drops 1 and 2) are plotted against the mathematical transient acceleration result for the above drop heights individually as shown in Figures 6.12 to 6.14. All three group results are matching and exhibited similar trends. Hence local linearization method is proven to be able to predict the transient response in nonlinear cushion system and establish a reliable virtual buffer model sufficiently.

6.4.2 Verification of Shock Response of Objective Model in Virtual Buffer Model

As mentioned in Section 6.3, the acceleration at PCB center is used to verify established objective model. Firstly, the dynamic response of multilayer circuit boards under different drop heights is verified. The prototype is respectively dropped from the heights of 0.5 meter, 0.8 meter and 1.0 meter. The acceleration vertical to PCB plane under different drop heights is calculated as shown in Figure 6.10. The experimental transient acceleration results are plotted against mathematical

Figure 6.12: Verification of boundary field for virtual buffer model under the drop height of 0.5 meter

Figure 6.13: Verification of boundary field for virtual buffer model under the drop height of 0.8 meter

Figure 6.14: Verification of boundary field for virtual buffer model under the drop height of 1.0 meter

acceleration results for the above drop heights individually as shown in Figures 6.15 to 6.17. All three group results are matching and exhibited similar trends.

Figure 6.15: Verification of dynamic response of objective model under the drop height of 0.5 meter

Next, the dynamic response of PCB under different drop angles is verified.

Figure 6.16: Verification of dynamic response of objective model under the drop height of 0.8 meter

Figure 6.17: Verification of dynamic response of objective model under the drop height of 1.0 meter (Verification of dynamic response of objective model under the drop angle of 0°)

6.4 Verification of Full-Scale Virtual Buffer Model by Experimental Results 212

As shown in Figure 6.11, the dynamic response of PCB under drop angles of 0° , 45° and 90° is calculated and plotted. All the acceleration obtained is vertical to PCB plane, and compared with experimental results. As shown in Figures 6.17 and 6.18, the experimental transient acceleration results are individually plotted against the mathematical acceleration results for the drop angles of 0° and 45° . The verification work for different drop angles also illustrates high accuracy. Note that, the acceleration under drop height of 90 degree is not compared in this section, because the acceleration vertical to PCB plane is quite small under vertical drop angle as shown in Figure 6.11.

Figure 6.18: Verification of dynamic response of objective model under the drop angle of 45°

Through the verification work for virtual buffer model and dynamic response of objective model, full-scale virtual buffer model is proven to be reliable. However, there is one limitation in dynamic response analysis of PCB: as shown in Figures 6.15 and 6.18, after first cycle, slim damping effect in simulation model causes less vibration period and less amplitude decline. Because PamCrash is an explicit finite element code, it is developed for impact analysis when large deformation happens in short transient period. In the built-up of this finite element code, the damping effect is calculated based on stress velocity [51], thus it only

obviously works when vigorous strain happens. However, as mentioned in Section 6.2, strain on PCB is too slim because of cushion effect from buffer package. This low damping effect in simulation causes less vibration period and higher vibration amplitude compared with experimental results. This has also been mentioned in previous work [118][119]. Therefore, more reliable damping calculation under low strain velocity conditions in finite element code may improve the accuracy of present models. But till present work, the improvement of algorithm of finite element code is beyond the scope of this thesis.

6.5 Summary of Chapter

In this chapter, full-scale virtual buffer model is presented to realize an efficient drop-impact analysis for a general prototype of packaged electronic products. The prototype is assembled with nonlinear cushion buffers and multilayer circuit boards, both of which require large computational resource to simulate by using traditional methods. Multilayer circuit board is objective model, whose dynamic response is quickly predicted by using full-scale virtual buffer model. In addition, the results from full-scale virtual buffer model illustrates good agreement with experimental results: first, the dynamic response of nonlinear cushion is accurately predicted and used to establish a reliable virtual buffer model; next, multilayer circuit board is efficiently modeled with hybrid laminate modeling method and inserted into virtual buffer model for dynamic analysis.

As a conclusion, virtual buffer model is initially developed in Chapter 3 as shown in Figure 3.24. Next, an innovated local linearization method and hybrid laminated modeling method are proposed in Chapters 4 and 5 to improve initial virtual buffer model to full-scale virtual buffer model as shown in Figures 4.22 and 5.33. Till Chapter 5, full-scale virtual buffer model is only theoretically presented in Figure 5.33 without practical application. Furthermore, in Chapter 6, a general prototype is provided and full-scale virtual buffer model is practically used for the

analysis of this prototype. In this application example, the efficiency and accuracy of virtual buffer model are illustrated.

Chapter 7

Conclusions and Recommendations

The objective of this PhD work is to generate some new ideas for general drop-impact system, rather than investigation to some specific objects. The packaged electronic product model provides a practical platform to perform these ideas. When a new method is innovated, experimental verifications are presented to consolidate the reliability of new ideas. Through the reiterative survey and comparison with other established methods, the new methods illustrate good performance in practical, such as high efficiency and flexibility.

This chapter concludes the dissertation with three parts: conclusion of thesis, summary of contributions and suggestions for future work.

7.1 Summary of Thesis

This work mainly focuses on the shock response of multilayer printed circuit boards and their interaction with nonlinear cushion material. These two components can be used to establish a packaged electronic product model. In order to realize the efficient drop-impact analysis, simulation methodologies, nonlinear analysis for

cushion buffers and modeling techniques for multilayer circuit boards are concerned in this project.

This thesis starts with a background introduction of drop-impact analysis for packaged electronic products. The application and theoretical background are respectively introduced in Chapters 1 and 2. In Chapter 1, the practical motivation of present work is presented. After that, the backdrop of research methodologies used in this PhD work is briefly reviewed in Chapter 2. The literature review in Chapter 2 covers physical drop test, numerical simulation, nonlinear analysis method for cushion buffers, and modeling techniques for printed circuit boards. Through the re-survey of research target and established research methods, the challenging points for efficiency analysis are stated and solved one by one in Chapters 3 to 6.

Chapters 3 to 6 are the main parts of this thesis as shown in Figure 7.1, they focus on the ideas innovation and problem solution. In Chapter 3, a virtual boundary method is generated for drop-impact analysis of packaged electronic products. This established virtual buffer model successfully combined all the research works in this thesis: In order to realize higher efficiency for packaged electronic products, virtual buffer model is initially generated with only simulation techniques in Chapter 3 and continuously improved to a full-scale virtual buffer model in Chapter 6. In the improvement process between Chapters 3 and 6, local linearization model in Chapter 4 and hybrid laminate modeling technique in Chapter 5 are created and inducted into virtual buffer model to drive the analysis efficiency to a high level beyond pure FEA simulation methods.

7.2 Summary of Contributions

Three main contributions are concluded in this PhD work: (1)virtual buffer method, (2)local linearization method and (3)hybrid laminate modeling technique. Although local linearization method and hybrid laminate modeling techniques are

Figure 7.1: Summary of the main part of thesis

used to improve virtual buffer model in this thesis. These two methods, especially local linearization method, are quite meaningful from technique point of view. Ordered from most to least importance, the contributions of new knowledge in this thesis are listed as short numbered paragraphs as following:

1. **Local linearization method:** The objective of local linearization method is to realize a quick quantitative dynamic analysis for nonlinear vibration systems. Nonlinearity is ubiquitous in nature, however, nonlinear differential calculation requires laborious numerical analysis usually, and sometimes the accuracy of approximation is not satisfactory. In this PhD work, an innovative local linearization method is proposed. As shown in Figure 4.2, through inducing local transient parameters and tracing nonlinear stiffness coefficient, the dynamic response of nonlinear vibration can be accurately predicted by linear equation of motion. The efficiency and accuracy are fully illustrated though the application examples in Section 4.4. In addition, this method is easy for programming, since no curve-fitting work and advanced function are required in use. In this PhD work, this method is used to predict the dynamic response of nonlinear drop-impact cushion system. This work was carried out to establish virtual buffer model in Step 1 of virtual buffer method as shown in Figure 4.22.
2. **Hybrid laminate modeling technique:** The objective of hybrid modeling techniques is to perform static and dynamic analysis of multilayer multimaterial composites in micro-structure and board levels. Laminate composites modeling is quite challenging: micro-structure and board-level analysis cannot be performed at the same time by using FE simulation method only; while dynamic analysis cannot be performed by using mechanics method only. Now three methods are developed with combination of 3D mechanics unit-cell model and 2D multilayer finite element. Among them, EQLAM multilayer method, as shown in Figure 6.8, illustrates the best performance in comparison with other methods. This method not only can realize quick

dynamic simulation and accurate mechanics analysis, but also illustrates high flexibility in the optimization design for multilayer multimaterial composites. In this PhD work, it is used to model multilayer circuit boards consisting of woven fabric-reinforced polymer matrix and metal plies. This work can create objective model in Step 2 of virtual buffer method as shown in Figure 5.33.

3. **Virtual buffer model:** virtual buffer model is the practical application of virtual boundary method for drop impact analysis of packaged electronic products. The virtual boundary method is used to realize efficient analysis for tiny components in large-scale models. This method is developed based on global-local method, and further illustrates higher efficiency. In this PhD work, it has another function: provides an application platform to combine local linearization method and hybrid laminate modeling technique. And thanks to this combination, virtual buffer model can be fully developed to achieve higher efficiency than initial model that only depend on FE simulation techniques.

As a conclusion, the development process of virtual buffer method integrated all the works in this thesis. Moreover, these three contributions can be used in other field rather than drop-impact analysis: (1) Local linearization method is suitable for the dynamic analysis of all nonlinear system, not only limited for cushion buffers; (2) Hybrid laminate modeling technique can be used to model various multimaterial multilayer composites in both micro-structure and board levels, not only for multilayer PCB; (3) The original version of virtual buffer model, virtual boundary model, could have a wide application field as global-local method.

7.3 Suggestions for Future Work

Three proposed methods (virtual boundary method, local linearization method, and hybrid modeling technique) have illustrated their advantages in the analysis

of packaged electronic products model, and each method can be further used for the analysis of other general systems. But there are still some limitations. Future work is suggested to cope with these limitations and extend application field.

Local linearization method. In this work, local linearization method successfully realizes quick analysis of nonlinear vibration system by using linear differential algorithm. The laborious numerical calculation and abstruse advanced functions are avoided. Especially when velocity is known, local transient parameter can be quickly obtained to calculate acceleration. However as shown in Figure 4.2, local transient parameter used in local linearization method only represents local calculation time, rather than global time. This troubles transient response analysis. It is believed that the direct calculation of global time parameter can greatly improve practicability of local linearization method.

Damping ratio of cushion buffers. As mentioned in Chapter 4, damping ratio is not constant. It increases along with strain, especially when cushion buffer is quickly compressed in drop impact condition. The study of material property of buffer material is beyond the scope of this work. It is, however, an interesting problem. If some calculation method of transient damping ratio can be implemented in local linearization model, the accuracy of transient analysis for nonlinear drop-impact cushion system will be further improved.

Hybrid laminate modeling technique. Hybrid laminate modeling technique is a method with the combination of 3D composite mechanics model and 2D multilayer finite element model. It is used to perform static and dynamic analysis for multilayer multimaterial composites at both micro-structure and board levels. The finite element code has some limitations: deficient damping effect in simulation, which reduces accuracy of dynamic analysis. Any finite element software has 2D multilayer codes can be used to perform hybrid laminate modeling techniques. We select PamCrash, because this software has a 2D multilayer code. However, PamCrash is an explicit finite element code, it is developed for impact analysis with large deformation. In the built-up of this finite element code, the

damping effect only obviously works when vigorous strain happens. However, as mentioned in Chapter 6, the deflection on PCB is small, therefore damping effect is insufficiently simulated. This causes less vibration period and higher vibration amplitude compared with experimental results. More reliable damping calculation under low strain velocity conditions in finite element code will improve the accuracy of present models.

Bibliography

- [1] K. H. Low, A. Yang, K. H. Hoon, X. Zhang, J. K. T. Lim and K. L. Lim, “Initial study on the drop-impact behavior of mini Hi-Fi audio products”, *Advances in Engineering Software*, vol. 32, no. 9, pp. 683–693, 2001.
- [2] K. H. Low, “Drop-impact cushioning effect of electronics products formed by plates”, *Advances in Engineering Software*, vol. 34, no. 1, pp. 31–50, 2003.
- [3] S. K. W. Seah, C. T. Lim, E. H. Wong, V. B. C. Tan and V. P. W. Shim, “Mechanical response of PCBs in portable electronic products during drop impact”, in *4th EPTC Conference Proceeding*, 2002, vol. 1, pp. 120–125.
- [4] E. H. Wong, K. M. Lim, N. Lee, S. Seah, C. Hoe and J. Wang, “Drop impact test - Mechanics and physics of failure”, in *4th EPTC Conference Proceeding*, 2002, vol. 1, pp. 327–333.
- [5] H. Ho, “A finite element analysis of the impact-contact problem of a portable telephone using I-DEAS and ADINA”, *Computers and Structures*, vol. 56, no. 2, pp. 397–409, 1995.
- [6] J. Wu, R. R. Zhang, S. Radons, X. Long and K. K. Stevens, “Vibration analysis of medical devices with a calibrated FEA model”, *Computers and Structures*, vol. 80, pp. 1081–1086, 2002.
- [7] J. M. Pitarresi and A. V. D. Edwardo, “A design approach for the systematic improvement of support locations for vibrating circuit cards”, *ASME, Journal of Electronic Packaging*, vol. 115, pp. 118–123, 1993.

- [8] K. Mishiro, S. Ishikawa, M. Abe, T. Kumai, Y. Higashiguchi and K. Tsubone, “Effect of the drop impact on BGA/CSP package reliability”, *Microelectronics Reliability*, vol. 42, pp. 77–82, 2002.
- [9] M. Barker and D. Eckroth, *The Wiley Encyclopedia of Packaging Technology*, New York: John Wiley and Sons, 1986.
- [10] T. Y. Tee, H. S. Ng, C. T. Lim, E. Pek, and Z. W. Zhong, “Application of drop test simulation in electronic packaging”, in *4th ASEAN ANSYS Conference Proceeding, Singapore*, 2002.
- [11] T. Y. Tee, H. S. Ng and Z. W. Zhong, “Design for enhanced solder joint reliability of integrated passives device under board level drop test and thermal cycling test”, in *5th EPTC Conference Proceeding, Singapore*, 2003.
- [12] J. Wu, G. Song, C. Yeh and K. Wyatt, “Drop/impact simulation and test validation of telecommunication products”, in *InterSociety Conference on Thermal Phenomena Conference Proceeding*, 1998, pp. 330–336.
- [13] J. Wu, “Global and local coupling analysis for small components in drop simulation”, in *6th International LSDYNA Users Conference Proceeding*, 2000, pp. 1117–1126.
- [14] L. Zhu, “Submodeling technique for BGA reliability analysis of CSP packaging subjected to an impact loading”, in *InterPACK Conference Proceeding*, 2001.
- [15] L. Zhu, “Modeling technique for reliability assessment of portable electronic product subjected to drop impact loads”, in *53rd ECTC Conference Proceeding*, 2003, pp. 100–104.
- [16] T. Sogo and S. Hara, “Estimation of fall impact strength for BGA solder joints”, in *ICEP Conference Proceeding*, 2001, vol. 1, pp. 369–373.

- [17] D. Xie, M. Arra, S. Yi and D. Rooney, “Solder joint behavior of area array packages in board level drop for handheld devices”, in *53rd ECTC Conference Proceeding*, 2003, vol. 1, pp. 130–135.
- [18] T. Y. Tee, H. S. Ng, C. T. Lim, E. Pek and Z. W. Zhong, “Board level drop test and simulation of TFBGA packages for telecommunication applications”, in *53th ECTC Conference Proceeding*, 2003, vol. 1, pp. 121–129.
- [19] T. Y. Tee, H. S. Ng, C. T. Lim, E. Pek and Z. W. Zhong, “Board level drop test and simulation of QFN packages for telecommunication applications”, in *ICEP Conference Proceeding, Japan*, 2003, pp. 221–226.
- [20] T. Y. Tee, H. S. Ng, C. T. Lim, E. Pek and Z. W. Zhong, “Drop test and impact life prediction model for QFN packages”, *Surface Mount Technology*, vol. 16, no. 3, pp. 31–39, 2003.
- [21] P. Shrotriya, *Dimensional Stability of Multilayer Circuit Boards*, PhD Thesis, University of Illinois at Urbana-Champaign, Chicago, Illinois, USA, 2001.
- [22] http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html 2005.
- [23] M. Schulz, T. Ertl and T. Reuding, “From high-end VR to PC-based VRML viewing: supporting the car development process by adapted virtual environments”, in *IATED CGIM’98 conference Proceeding, Halifax, Nova Scotia, Canada*, 1998.
- [24] S. Kuschfeldt, M. Holzner, O. Sommer and T. Ertl, “Efficient visualization of crash-worthiness simulations”, *IEEE Computer Graphics and Applications*, vol. 18, pp. 60–65, 1996.
- [25] T. Pepelnjak, G. Gantar and K. Kuzman, “Numerical simulations in optimisation of product and forming process”, *Journal of Materials Processing Technology*, vol. 115, pp. 122–126, 2001.

- [26] K. H. Low, Y. Wang, K. H. Hoon and N. Vahdati, “Initial global-local analysis for drop-impact effect study of TV products”, *Advances in Engineering Software*, vol. 35, pp. 179–190, 2004.
- [27] M. Bossak and J. Kaczkowski, “Global/local analysis of composite light aircraft crash landing”, *Computers and Structures*, vol. 81, pp. 503–514, 2003.
- [28] Y. Wang, *Drop-impact analysis of packaged electronic products using a virtual boundary model*, PhD First Year Confirmation Report, Nanyang Technological University, Singapore, 2003.
- [29] Y. Y. Wang, C. Lu, J. Li, X. M. Tan, Y. C. Tse, “Simulation of drop/impact reliability for electronic devices”, *Finite Elements in Analysis and Design*, vol. 41, pp. 667–681, 2005.
- [30] M. C. Pan, P. C. Chen and J. Ho, “Drop simulation and experimental validation of TFT-LCD monitors”, in *5th EPTC Conference Proceeding, Singapore*, 2003, vol. 1, pp. 269–274.
- [31] Y. C. Ong, V. P. W. Shim, T. C. Chai and C. T. Lim, “Comparison of mechanical response of PCBs subjected to product-level and board-level drop impact tests”, in *5th EPTC Conference Proceeding, Singapore*, 2003, vol. 1, pp. 223–227.
- [32] C. T. Lim and Y. J. Low, “Investigation of drop impact of portable electronic products”, in *52th ECTC Conference Proceeding*, 2002, vol. 1, pp. 1270–1274.
- [33] IEC 68-2-27, *International Standard: Basic Environmental Testing Procedures*, International Electrotechnical Commission, Geneva, Switzerland, 1987.
- [34] JESD22-B104-B, *Mechanical Shock Test Method*, JEDEC Solid State Technology Association, Arlington, 2001.

- [35] JESD22-B110, *Subassembly Mechanical Shock Test Method*, JEDEC Solid State Technology Association, Arlington, 2001.
- [36] JESD22-B111, *Board Level Drop Test Method of Components for Handheld Electronic Products*, JEDEC Solid State Technology Association, Arlington, 2003.
- [37] <http://www.wsu.edu:8080/gmhyde/ImpactProperties.html>, 2005.
- [38] S. K. W. Seah, C. T. Lim, E. H. Wong, V. B. C. Tan, V. P. W. Shim, “Mechanical response of PCBs in portable products during drop impact”, in *52th ECTC Conference Proceeding*, 2002, vol. 1, pp. 120–125.
- [39] ASTM D 1596, *Standard Test Methods for Dynamic Shock Cushioning Characteristics of Packaging Material*, American Society of Testing and Materials, Philadelphia, Pa., 1998.
- [40] Styropor, expandable polystyrene (EPS), packaging with styropor, Tech. Rep., BASF, 2003.
- [41] <http://www.sealedair.com/library/articles/article4.html>, 2005.
- [42] Y. Y. Wang, G. R. Liu, K. Y. Lam and L. Hua, “Numerical simulation of consumer products drop”, in *HPCASIA Conference Proceeding, Singapore*, 1998, vol. 1, pp. 181–185.
- [43] Y. Y. Wang, T. Y. Lin and L. Hua, “Drop-impact simulation and experimental verification for spindle fixation of video and audio module (VAM)”, *Mechatronics*, vol. 13, no. 5, pp. 427–440, 2003.
- [44] H. Kim, J. Lee and S. Park, “Application of drop-impact simulation for package cushioning material reduction”, in *PUCA'97 Conference Proceeding*, 1997, vol. 1, pp. 299–303.

- [45] K. Farahani, M. Mofid and A. Vafai., “A solution method for general contact-impact problems”, *Computer Methods in Applied Mechanics and Engineering*, vol. 187, no. 5, pp. 69–77, 2000.
- [46] ANSYS/LS-DYNA, *ANSYS/LS-DYNA Theoretical and Manual for Release 5.6*, ANSYS, Inc, Canonsburg, Pa., 1999.
- [47] J. S. Sun, K. H. Lee and H. P. Lee, “Comparison of implicit and explicit finite element methods for dynamic problems”, *Journal of Materials Processing Technology*, vol. 105, no. 1, pp. 110–118, 2000.
- [48] http://www.hks.com/solutions/sol_circuit_board.html, 2005.
- [49] K. H. Low and X. Zhang, “Global-local finite element method for the drop/impact analysis on the electronic products with small components”, in *4th ISIE Conference Proceeding, Kumamoto, Japan*, 2001, vol. 2, pp. 975–980.
- [50] S. R. Voleti, N. Chandra and J. R. Miller, “Global-local analysis of large-scale composite structure using finite element methods”, *Computers and Structures*, vol. 58, pp. 453–464, 1996.
- [51] ESI, *PAM-Crash Solver Notes Manual*, PAM System International SA, France, 1998.
- [52] T. S. Koko and M. D. Olson, “Vibration analysis of stiffened plates by super elements”, *Journal of Sound and Vibration*, vol. 158, no. 1, pp. 149–167, 1992.
- [53] T. S. Koko and M. D. Olson, “Non-linear analysis of stiffened plates using super elements”, *International Journal for Numerical Methods in Engineering*, vol. 31, no. 2, pp. 319–343, 1991.
- [54] T. S. Koko and M. D. Olson, “Nonlinear transient response of stiffened plates to air blast loading by super element approach”, *Computer Methods in Applied Mechanics and Engineering*, vol. 90, no. 3, pp. 737–760, 1991.

- [55] R. Radovitzky and M. Ortiz, “Error estimation and adaptive meshing in strongly nonlinear dynamic problems”, *Computer Methods in Applied Mechanics and Engineering*, vol. 172, pp. 203–240, 1999.
- [56] [http : //www.cesup.ufrgs.br/ansys/guide_55/g – adv/GADV3.htm](http://www.cesup.ufrgs.br/ansys/guide_55/g-adv/GADV3.htm), 2004.
- [57] S. Goyal, J. M. Papadopoulos and P. A. Sullivan, “Shock protection of portable electronic products: Shock response spectrum, damage boundary approach, and beyond”, *Shock and Vibration*, vol. 4, pp. 169–191, 1997.
- [58] D. S. Steinberg, *Vibration analysis for electronic equipment*, Wiley-Interscience Publication, 2nd edition, 1988.
- [59] E. Suhir, “Shock protection with a nonlinear spring”, *IEEE Trans. Comp. Packag. Manufact. Technol.*, vol. 18, no. 2, pp. 430–437, 1995.
- [60] D. F. Lawden, *Elliptic Functions and Applications*, Springer-Verlag, New York, 1989.
- [61] B. E. Asir, A. Mansour, A. A. Mowis and N. Kassab, “A new approximate analytical analysis of the response of non-linear damped systems subjected to step function excitation”, *Journal of Sound and Vibration*, vol. 174, pp. 115–121, 1994.
- [62] P. L. Loveridge and N. J. Mills, “prediction of packaging cushion curves and helmet linear response”, in *2nd Cellular Polymers Conference Proceeding, Edinburgh, RAPRA technology*, 1993, p. paper 21.
- [63] http://www.sdrl.uc.edu/sdrl_jscript_homepage.html, 2004.
- [64] D. M. Blacketter, D. E. Walrath and A. C. Hansen, “Modeling damage in a plain weave fabric-reinforced composite material”, *Journal of Composite Technology and Research*, vol. 15, pp. 136–142, 1993.
- [65] X. Tang and J. D. Whitcomb, “Progressive failure behaviors of 2D woven composites”, *Journal of Composite Materials*, vol. 37, pp. 1239–1259, 2003.

- [66] Y. Q. Wang, K. H. Low, F. X. Che, H. L. J. Pang and S. P. Yeo, “Modeling and simulation of printed circuit board drop test”, in *5th EPTC Conference Proceeding, Singapore*, 2003, vol. 1, pp. 275–281.
- [67] C. F. Coombs, *Printed Circuits Handbook*, McGraw-Hill, New York, 4th edition, 1995.
- [68] M. Flatt, *Printed Circuit Board Basics*, Miller Freeman Inc., 2nd edition, 1992.
- [69] G. Leonida, *Handbook of Printed Circuit Design, Manufacture, Components and Assembly*, Electrochemical Publications Ltd., 1981.
- [70] Y. Wang, K. H. Low, H. L. J. Pang, K. H. Hoon, F. X. Che, Y. S. Yong, “Modeling and simulation for a drop-impact analysis of multi-layered printed circuit boards”, *Microelectronics Reliability*, vol. 46, no. 2-4, pp. 558–573, 2005.
- [71] Q. Zhu, P. Shrotriya, N. R. Sottos and P. H. Geubelle, “Three-dimensional viscoelastic simulation of woven composite substrates for multilayer circuit boards”, *Composites Science and Technology*, vol. 63, pp. 1971–1983, 2003.
- [72] D. Coutellier and P. Rozycki, “Multi-layered multi-material finite element for crashworthiness studies”, *Composites: Part A: Applied science and manufacturing*, vol. 31, pp. 841–851, 2000.
- [73] E. Haug and A. R. De, *Crash Response of Composite structures, Structural Crashworthiness and Failure*, Amsterdam: Elsevier, 1992.
- [74] E. Haug, O. Fort, A. Tramecon, M. Watanabe, I. Nakada, “Numerical crashworthiness simulation of automotive structures and components made of continuous fiber reinforced composite and sandwich assemblies”, in *S.A.E. International Congress and Exhibition, Detroit*, 1991, pp. 245–258.

- [75] A. R. De and E. Haug, *Failure of Brittle and Composite Materials by Numerical Methods*, In: Wierzbicki T, editor. *Structural Failure*, New York: John Wiley and Sons, 1989.
- [76] G. Yu, G. Y. Tzeng, S. Chaturvedi, H. deli and S. Q. Zhang, “A finite element approach to global-local modeling in composite laminate analysis”, *Computers and structures*, vol. 57, no. 6, pp. 1035–1044, 1995.
- [77] N. J. Pagano and S. R. Soni, “Global-local laminate variational model”, *International Journal of Solids and Structures*, vol. 19, no. 3, pp. 207–228, 1983.
- [78] T. Nakamura and L. C. Wu, “Effects of ply-arrangement on compressive failure of layered structures”, *Eng. fracture mechanics*, vol. 67, pp. 421–443, 2000.
- [79] E. N. Brown and N. R. Sottos, “Thermoelastic properties of plain weave composites for multilayer circuit board applications”, *Theoretical and Applied Mechanics Report, University of Illinois at Urbana-Champaign*, vol. 878, pp. 1–17, 1997.
- [80] H. J. Kim and C. C. Swan, “Multi-scale computational homogenization of plain-weave textile composites”, in *15th ASCE Engineering Mechanics Division Conference Poster*, 2002.
- [81] A. Tabiei and I. Ivanov, “Materially and geometrically non-linear woven composite micro-mechanical model with failure for finite element simulations, international journal of non-linear mechanics”, *Theoretical and Applied Mechanics Report, University of Illinois at Urbana-Champaign*, vol. 39, no. 2, pp. 175–188, 2004.
- [82] I. Ivanov and A. Tabiei, “Three-dimensional computational micro-mechanical model for woven fabric composites”, *Composite Structures*, vol. 54, no. 4, pp. 489–496, 2001.

- [83] R.A. Naik, “Failure analysis of woven and braided fabric reinforced composites”, *Journal of Composite Materials*, vol. 29, pp. 2334–2363, 1995.
- [84] D. M. Blacketter, D. E. Walrath and A. C. Hansen, “Modeling damage in a plain weave fabric-reinforced composite material”, *Journal of Composite Technology and Research*, vol. 15, no. 2, pp. 136–142, 1993.
- [85] P. Ladeveze and E. L. Dantec, “Damage modelling of the elementary ply for laminated composites”, *Composites Science and Technology*, vol. 43, pp. 257–267, 1992.
- [86] M. L. James, G. M. Smith, J. C. Wolford, P. W. Whaley, *C. T. Herakovich*, New York: John Wiley and Sons, 1st edition, 1998.
- [87] J. Wu, R. R. Zhang, S. Radons, X. Long and K. K. Stevens, “Vibration analysis of medical devices with a calibrated FEA model”, *Computers and Structures*, vol. 80, pp. 1081–1086, 2002.
- [88] M. Lee, “Finite element modeling of printed circuit boards (PCBs) for structural analysis”, *Journal of Circuit World*, vol. 26, no. 3, pp. 24–29, 2000.
- [89] T. D. Moore and J. L. Jarvis, “The effects of in-plane orthotropic properties in a multi-chip ball grid array assembly”, *Microelectronics Reliability*, vol. 42, no. 6, pp. 943–949, 2002.
- [90] E. Makiewicz, P. Ducrocq and P. Drazetic, “An inverse approach to determine the constitutive model parameters from axial crushing of thin-walled square tubes”, *International Journal of Impact Engineering*, vol. 21, pp. 433–449, 1998.
- [91] M. A. McCarthy and J. F. M. Wiggenraad, “Numerical investigation of a crash test of a composite helicopter subfloor structure”, *Computers and Structures*, vol. 51, pp. 345–359, 2001.

- [92] M. L. James, G. M. Smith, J. C. Welford, P. W. Whaley, *Vibration of Mechanical and Structural systems with microcomputer applications*, Harper Collins College Publisher, 2nd edition, 1994.
- [93] Y. Wang and K. H. Low, “Damped response analysis of nonlinear cushion systems by a linearization method”, *Computers and Structures*, vol. 83, pp. 1584–1594, 2005.
- [94] N. C. Shekhar, H. Hatwal and A. K. Mallik, “Response of non-linear dissipative shock isolators”, *Journal of Sound and Vibration*, vol. 214, pp. 589–603, 1998.
- [95] C. M. Harris and C. E. Crede, *Harris’ Shock and Vibration Handbook*, New York: McGraw-Hill, 2nd edition, 1976.
- [96] H. T. Davis, *Introduction to Nonlinear Differential and Integral Equations*, New York: Dover Publications, 1962.
- [97] M. Cartmell, *Introduction to Linear, Parametric and Nonlinear Vibrations*, Chapman and Hall, 1st edition, 1990.
- [98] Q. Lu, *Qualitative Method and Bifurcation of Ordinary Differential Equations*, Beijing: Beijing University of Aeronautics and Aeronautics Press, 2002.
- [99] P. Calter, *Technical Mathematics with Calculus*, New Jersey: Prentice-Hall, 2nd edition, 1990.
- [100] W. S. Stupnicka, *The Behavior of Nonlinear Vibration System I: Fundamental Concepts and Methods: Applications to Single-Degree-of-Freedom Systems*, Boston: Kluwer Academic Publisher, 1st edition, 1990.
- [101] M. Avalu, G. Belingardi and R. Montanini, “Characterization of polymeric structural foams under compressive impact loading by means of energy-absorption diagram”, *International Journal of Impact Engineering*, vol. 25, pp. 455–472, 2001.

- [102] H. J. Pradlwarter, G. I. Schueller and C. A. Schenk, “A computational procedure to estimate the stochastic dynamic response of large non-linear FE-models”, *Computer Methods Applied Mechanics and Engineering*, vol. 192, pp. 777–801, 2003.
- [103] Y. M. Moreu and N.J. Mills, “Rapid hydrostatic compression of low-density polymeric foams”, *Polymer Testing*, vol. 23, pp. 313–322, 2004.
- [104] ASTM D1621, *Standard Test Method for Compressive Properties Of Rigid Cellular Plastics*, American Society of Testing and Materials, Philadelphia, Pa., 1998.
- [105] A. A. El-Kheshen and M. F. Zawrah, “Sinterability microstructure and properties of glass/ceramic composites”, *Ceramics International*, vol. 29, pp. 251–257, 2003.
- [106] R. R. Tummala, *Fundamentals of Microsystems Packaging*, Singapore: McGraw-Hill, 2nd edition, 2001.
- [107] C. M. Harris and A. G. Piersol, *Harris’ Shock and Vibration Handbook*, New York: McGraw-Hill, 5th edition, 2002.
- [108] J. Luan, T. Y. Tee, E. Pek, C. T. Lim and Z. Zhong, “Sinterability microstructure and properties of glass/ceramic composites”, *Ceramics International*, vol. 29, pp. 251–257, 2003.
- [109] T. R. Chandrupatla and A. D. Belegundu, *Introduction to Finite Elements in Engineering*, New Jersey: Prentice Hall, 3rd edition, 1997.
- [110] H. Conrad and D. Yang, “Effect of an electronic field on the plastic deformation kinetics of electrodeposited Cus at low and intermediate temperatures”, *Acta Materialia*, vol. 50, pp. 2851–2866, 2002.
- [111] L. P. Kollar and G. S. Springer, *Mechanics of Composite Structure*, New York: Cambiedge University Press, 1st edition, 2003.

- [112] I. H. Shames and F. A. Cozzarelli, *Elastic and Inelastic Stress Analysis*, Washington DC: Taylor and Francis, 2nd edition, 1997.
- [113] R. J. Hannemann, A. D. Kraus and M. Pecht, *Semiconductor Packaging, A Multidisciplinary Approach*, John Wiley and Sons, 1994.
- [114] S. G. Lekhnitskii, *Theory of Elasticity for an Anisotropic Body*, San Francisco: HoldenDay, 1963.
- [115] ASTM D3039/D3039M, *Standard Test Method for Tensile Properties of Polymer Matrix Composite Materials*, American Society of Testing and Materials, Philadelphia, Pa., 1998.
- [116] ASTM D3552/D3552M, *Test Method for Tensile Properties of Fiber Reinforced Metal Matrix Composite*, American Society of Testing and Materials, Philadelphia, Pa., 1998.
- [117] K. H. Low, Y. Wang, K. H. Hoon and W. K. Wai, “a virtual boundary model for a quick drop-impact analysis of electronic components in TV model”, *Advances in Engineering Software*, vol. 35, pp. 537–551, 2004.
- [118] Y. Wang, K. H. Low and K. H. Hoon, “Two-dimensional modeling of multilayer multimaterial circuit boards by using an EQLAM multilayer model”, in *Proceeding of 2005 ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conference, Long Beach, California USA*, DETC2005-84620, 2005.
- [119] Y. Wang, K. H. Low and K. H. Hoon, “Damage analysis of multilayer multimaterial circuit boards by using a bi-phase multilayer model”, in *Proceeding of 2005 ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conference, Long Beach, California USA*, DETC2005-84621, 2005.

Author's Publications

Journal Papers

- (1) K. H. Low, Y. Wang, K. H. Hoon and N. Vahdati, "Initial global-local analysis for drop-impact effect study of TV products", *Advances in Engineering Software*, Vol. 35, pp. 179-190, 2004.
- (2) K. H. Low, Y. Wang, K. H. Hoon and W. K. Wai, "A virtual boundary model for a quick drop-impact analysis of electronic components in TV model", *Advances in Engineering Software*, Vol. 35, pp. 537-551, 2004.
- (3) Y. Wang and K. H. Low, "Damped response analysis of nonlinear cushion systems by a linearization method", *Computers and Structures*, Vol 83, pp. 1584-1594, 2005.
- (4) Y. Wang, K. H. Low, H. L. J. Pang, K. H. Hoon, F. X. Che, and Y. S. Yong, "Modeling and simulation for a drop-impact analysis of multi-layered printed circuit boards", *Microelectronics Reliability*, Vol. 46, pp. 558-573, 2006.
- (5) K. H. Low and Y. Wang, "Modeling of multilayer circuit boards by using a multilayer model with bi-phase and elastic-plastic plies", *Circuit World*, Vol 33, Number 3, pp. 9-20, 2007.

Conference Papers

- (1) K. H. Low, Y. Wang, K. H. Hoon and W. K. Wai, "A virtual boundary model for a quick drop-impact analysis of PCB in electronic products", *Proceedings of 5th IEEE International Conference on Electronics Packaging Technology*, ISBN 0780382056, Singapore, pp. 275-281, 2003.
- (2) Y. Wang, K. H. Low, F. X. Che, H. L. J. Pang and S. P. Yeo, "Modeling and simulation of printed circuit board drop test", *Proceedings of 5th IEEE International Conference on Electronics Packaging Technology*, ISBN 0780382056, Singapore, pp. 263-268, 2003. (Won IEEE Outstanding Student Paper Award for best paper at 5th IEEE International Conference on Electronics Packaging Technology in 2003.)
- (3) Y. Wang, K. H. Low and K. H. Hoon "Two-dimensional modeling of multilayer multimaterial circuit boards by using an EQLAM multilayer model", *Proceeding of 2005 ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conference*, DETC2005-84620, Long Beach, California USA, 2005.
- (4) Y. Wang, K. H. Low and K. H. Hoon "Damage analysis of multilayer multimaterial circuit boards by using a bi-phase multilayer model", *Proceeding of 2005 ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conference*, DETC2005-84621, Long Beach, California USA, 2005.
- (5) Y. Wang and K. H. Low, "Quick evaluation of crashworthiness for nonlinear damped systems by using an analytical linearization method", *Proceedings of 2005 ASME International Mechanical Engineering Congress and Exposition*, IMECE2005-80709, Orlando, Florida USA, 2005.
- (6) Y. Wang and K. H. Low, "Hybrid modeling of multilayer composite consisting of woven fabric-reinforced polymer matrix and metal plies", *Present at 2005 ASME International Mechanical Engineering Congress and Exposition*, IMECE2005-80700, Orlando, Florida USA, 2005.