

IPC/JEDEC-9301

2018 - December

Numerical Analysis Guidelines for Microelectronics Packaging Design and Reliability

An international standard developed by IPC

Association Connecting Electronics Industries



The Principles of Standardization

In May 1995 the IPC's Technical Activities Executive Committee (TAEC) adopted Principles of Standardization as a guiding principle of IPC's standardization efforts.

Standards Should:

- Show relationship to Design for Manufacturability (DFM) and Design for the Environment (DFE)
- Minimize time to market
- Contain simple (simplified) language
- Just include spec information
- Focus on end product performance
- Include a feedback system on use and problems for future improvement

Standards Should Not:

- Inhibit innovation
- Increase time-to-market
- Keep people out
- Increase cycle time
- Tell you how to make something
- Contain anything that cannot be defended with data

Notice

IPC Standards and Publications are designed to serve the public interest through eliminating misunderstandings between manufacturers and purchasers, facilitating interchangeability and improvement of products, and assisting the purchaser in selecting and obtaining with minimum delay the proper product for his particular need. Existence of such Standards and Publications shall not in any respect preclude any member or nonmember of IPC from manufacturing or selling products not conforming to such Standards and Publication, nor shall the existence of such Standards and Publications preclude their voluntary use by those other than IPC members, whether the standard is to be used either domestically or internationally.

Recommended Standards and Publications are adopted by IPC without regard to whether their adoption may involve patents on articles, materials, or processes. By such action, IPC does not assume any liability to any patent owner, nor do they assume any obligation whatever to parties adopting the Recommended Standard or Publication. Users are also wholly responsible for protecting themselves against all claims of liabilities for patent infringement.

IPC Position Statement on Specification Revision Change

It is the position of IPC's Technical Activities Executive Committee that the use and implementation of IPC publications is voluntary and is part of a relationship entered into by customer and supplier. When an IPC publication is updated and a new revision is published, it is the opinion of the TAEC that the use of the new revision as part of an existing relationship is not automatic unless required by the contract. The TAEC recommends the use of the latest revision. Adopted October 6, 1998

Why is there a charge for this document?

Your purchase of this document contributes to the ongoing development of new and updated industry standards and publications. Standards allow manufacturers, customers, and suppliers to understand one another better. Standards allow manufacturers greater efficiencies when they can set up their processes to meet industry standards, allowing them to offer their customers lower costs.

IPC spends hundreds of thousands of dollars annually to support IPC's volunteers in the standards and publications development process. There are many rounds of drafts sent out for review and the committees spend hundreds of hours in review and development. IPC's staff attends and participates in committee activities, typesets and circulates document drafts, and follows all necessary procedures to qualify for ANSI approval.

IPC's membership dues have been kept low to allow as many companies as possible to participate. Therefore, the standards and publications revenue is necessary to complement dues revenue. The price schedule offers a 50% discount to IPC members. If your company buys IPC standards and publications, why not take advantage of this and the many other benefits of IPC membership as well? For more information on membership in IPC, please visit www.ipc.org or call 847/597-2809.

Thank you for your continued support.



IPC/JEDEC-9301



Numerical Analysis Guidelines for Microelectronics Packaging Design and Reliability

Developed by the JEDEC Reliability Test Methods for Packaged Devices Committee (JC-14.1) and the SMT Attachment Reliability Test Methods Task Group (6-10d) of the Product Reliability Committee (6-10) of IPC

Users of this publication are encouraged to participate in the development of future revisions.

Contact:

IPC
3000 Lakeside Drive, Suite 105N
Bannockburn, Illinois
60015-1249
Tel 847 615.7100
Fax 847 615.7105

This Page Intentionally Left Blank

Acknowledgment

Members of the JEDEC Reliability Test Methods for Packaged Devices Committee (JC-14.1) and the SMT Attachment Reliability Test Methods Task Group (6-10d) of the IPC Product Reliability Committee (6-10) have worked together to develop this document. We would like to thank them for their dedication to this effort. Any document involving a complex technology draws material from a vast number of sources. While the principal members of the SMT Attachment Reliability Test Methods Task Group are shown below, it is not possible to include all of those who assisted in the evolution of this standard. To each of them, the members of JEDEC and IPC extend their gratitude.

Product Reliability Committee	SMT Attachment Reliability Test Methods Task Group	Technical Liaison of the IPC Board of Directors
Chair James Monarchio TTM Technologies	Chair Reza Ghaffarian Jet Propulsion Laboratory	Bob Neves Microtek Laboratories
Vice Chair Vasu Vasudevan Dell Inc.	Vice Chair Vasu Vasudevan Dell Inc.	

SMT Attachment Reliability Test Methods Task Group

Mudasir Ahmad, Cisco Systems Inc	Reza Ghaffarian, Jet Propulsion Laboratory	Clarence Nichols, Gables Engineering, Inc.
Aileen Allen, Hewlett-Packard Inc.	Cynthia A. Gomez, Continental Temic SA de CV	David Oris, Gables Engineering, Inc
Dudi Amir, Intel Corporation	Allen Green, Acoustic Technology Group	Chandradip Patel, Schlumberger Well Services
Martin K. Anselm, Rochester Institute of Technology	Bill Hargin, Nan Ya Plastics Corporation	Jagadeesh Radhakrishnan, Intel Corporation
Michael Azarian, University of Maryland	Kayleen Helms, Intel Corporation	Gnyaneshwar Ramakrishna, Cisco Systems Inc.
Elizabeth Elias Benedetto, Hewlett-Packard Inc.	Gaston Hidalgo, Toyota Motor North America	Edda Rivera, Gables Engineering, Inc.
Nicole Butel, Broadcom Limited	Ife Hsu, Intel Corporation	Jose A. Sanchez, Gables Engineering, Inc.
William P. Cardinal, UTC Aerospace Systems	Christopher Hunt, Pireta	Martin Scionti, Raytheon Missile Systems
Beverley Christian, HDP User Group	Jeffrey ChangBing Lee, iST Integrated Service Technology	Jose Ma Servin-Olivares, Continental Temic SA de CV
Jean-Paul Clech, EPSI	Anna Lifton, Alpha Assembly Solutions	Russell S. Shepherd, NTS Anaheim
Gino Cochella, Northrop Grumman Aerospace Systems	Rene R. Martinez, Northrop Grumman Aerospace Systems	Julie Silk, Keysight Technologies
Richard J. Coyle, Nokia	Daniel McCormick, Naval Surface Warfare Center	Bhanu Sood, NASA Goddard Space Flight Center
William C. Dieffenbacher, BAE Systems	Steve Minich, Amphenol InterCon Systems	Jorge Suarez, Gables Engineering, Inc.
Harold O. Ellison, Quantum Corporation	Jim Mulvey, Lockheed Martin Corporation	Taylor J. Swanson, Rochester Institute of Technology
Gerd Fischer, NASA Goddard Space Flight Center	David R. Nelson, Raytheon Company	John Paul Thompson, FCI USA, Inc.
Dennis D. Fritz, SAIC	Keith G. Newman, AMD	
Enrico Galbiati, SEM Communication & GESTLABS Srl		
Phil Geng, Intel Corporation		

Kristen K. Troxel, Hewlett-Packard
Inc.
Vasu S. Vasudevan, Dell Inc.

Milena Vujosevic, Intel Corporation
Michael Wolverton, Raytheon
Systems Company

Fonda B. Wu, Raytheon Company
Andy Zhang, Texas Instruments

SPECIAL ACKNOWLEDGMENT

JC-14.1 JEDEC Reliability Test Methods for Packaged Devices Committee

Ife Hsu, Intel Corporation

Mudasir Ahmad, Cisco Systems Inc

Table of Contents

1 SCOPE	1	11.2.2 Stresses	14
1.1 Introduction	1	11.3 Volume Averaging	14
1.2 Purpose	1	11.4 Temperature Dependent Results	14
2 APPLICABLE DOCUMENTS	4	11.5 Correlation with Experimental Data	14
2.1 IPC	4	11.5.1 Nomenclature	14
2.2 ASTM	4	11.5.2 Experimental Statistical Analysis	15
3 TERMS AND DEFINITIONS	4	11.5.3 Comparison of Results	15
4 RESULTS REPORTING	5	12 REFERENCES	16
5 BASIC ANALYSIS INFORMATION	6	13 APPENDICES	17
5.1 Analysis Type	6	APPENDIX A Material Property Modeling	18
5.2 Physics Type	6	APPENDIX B DMA/TMA Fitting Algorithms	33
5.3 Software	7	APPENDIX C Experimental Benchmarking Model Case Study (Thermomechanical Board Level Joint Reliability)	38
5.3.1 Software Version	7	APPENDIX D Warpage	50
5.4 Analysis Files and Scripts	7	APPENDIX E Experimental Benchmarking Model Case Study (Transient Bend Testing)	52
6 BASIC MODEL INFORMATION	8		
6.1 Boundary Conditions	8		
6.2 Model Stackup Schematics	8		
6.3 Boundary Condition Schematics	8		
7 MATERIAL PROPERTY INFORMATION	10		
7.1 Material Properties	10		
7.2 Reference Temperature	10		
7.3 Temperature Dependent Properties	10		
8 MESH INFORMATION	10		
8.1 Mesh Selection and Information	10		
8.2 Mapped or Tetrahedral Mesh	10		
8.3 Dissimilar Meshes	11		
8.4 Mesh Sensitivity Analysis	11		
9 LOADING INFORMATION	11		
9.1 Ramped vs. Stepped Loading	11		
10 SOLUTION INFORMATION	12		
10.1 Operating System	12		
10.2 High Performance Computing (HPC)	12		
10.3 Hardware	12		
10.4 Solution Algorithms and Settings	13		
10.4.1 Time Steps	13		
10.4.2 Solution Time	13		
11 POST PROCESSING	13		
11.1 Post Processing	13		
11.2 Results Parameters	13		
11.2.1 Fatigue	13		
		Figures	
		Figure 1-1 Building Blocks of a Typical Finite Element Analysis Problem	2
		Figure 1-2 Expected Modular Structure of Numerical Analysis Reliability Guideline Documents	3
		Figure 6-1a Examples of Model Schematics	8
		Figure 6-1b Examples of Model Schematics	8
		Figure 6-1c Examples of Model Schematics	8
		Figure 6-2 Quarter Symmetric Model	8
		Figure 6-3 Symmetry Boundary Conditions Added	9
		Figure 6-4 Zoom in View of Symmetry Boundary Conditions	9
		Figure 6-5 Displacement Boundary Conditions to Constrain All Rigid Body Displacement	9
		Figure 8-1 Example of Multi-Point Constraint to "Tie" Dissimilar Mesh Regions	11
		Figure 8-2 Example of Mesh Sensitivity Analysis	11
		Figure 10-1 Computational Time vs. Number of Cores Used (PBGA 680 Solder Joint Model)	12
		Figure 10-2 Octant PBGA680 Solder Joint Reliability Model	12
		Figure 10-3 Example of Time Step Too Big to Capture Fluctuation in Loading History	13
		Figure A2-1a Isotropic and Kinematic Hardening Rules ...	21
		Figure A2-1b Isotropic and Kinematic Hardening Rules ...	21
		Figure A2-1c Isotropic and Kinematic Hardening Rules ...	21

Figure A2-2a	Examples of One-Dimensional Stress Strain Relationships	21	Figure A4-9b	Measurement and the Simulation of the Deflection Over Temperature of a Bi-material Beam Consisting of Molding Compound and Silicon. The Linear-elastic (LE) Model Clearly Overestimates the Deflection Whereas the LVE Model can Reproduce the Deflection Correctly	28
Figure A2-2b	Examples of One-Dimensional Stress Strain Relationships	21	Figure A4-10a	Schematic Representation of the Tests Until Failure: a) Tensile Test Until Failure, b) Tensile Test Until Failure with Relaxation Segments	29
Figure A2-2c	Examples of One-Dimensional Stress Strain Relationships	21	Figure 4-10b	Schematic Representation of the Tests Until Failure: a) Tensile Test Until Failure, b) Tensile Test Until Failure with Relaxation Segments	29
Figure A2-2d	Examples of One-Dimensional Stress Strain Relationships	21	Figure A4-11a	Results of the Tensile Tests Until Failure at Different Temperatures	30
Figure A3-1a	Characteristic Measurements of the Storage Modulus and the Thermal Expansion of a Thermosetting Polymer Over Temperature.	22	Figure A4-11b	Stress Relaxation of the Investigated Molding Compound	30
Figure A3-1b	Characteristic Measurements of the Storage Modulus and the Thermal Expansion of a Thermosetting Polymer Over Temperature.	22	Figure A4-12	Representation of the Bergstrom-Boyce Model	30
Figure A3-2a	Extraction of T_g by Tangents from TMA Measurement.	23	Figure A4-13	Results of Experiment and Simulation of the Tensile Test Until Failure at 125 °C and the Load Rate of 1.0 mm/min	30
Figure A3-2b	Extraction of T_g by Tangents from TMA Measurement.	23	Figure A4-14	Scheme of Curing Shrinkage During Polymerization Process of Molding Compound [7]	31
Figure A4-1a	Tri-linear (left) and Bi-linear (right) Interpolating Functions for Fitting of DMA/TMA Data Respectively	25	Figure A4-15	Constitutive Law for an Orthotropic Material	32
Figure A4-1b	Tri-linear (left) and Bi-linear (right) Interpolating Functions for Fitting of DMA/TMA Data Respectively	25	Figure C1-1a	Four Packages Selected for Case Study	38
Figure A4-2a	Example of Best Fit for DMA (left) and TMA (right) Data	25	Figure C1-1b	Four Packages Selected for Case Study	38
Figure A4-2b	Example of Best Fit for DMA (left) and TMA (right) Data	25	Figure C1-1c	Four Packages Selected for Case Study	38
Figure A4-3	Thermal Expansion Measurement of a Thermosetting Polymer and the Extraction of Instantaneous CTEs from Measured Thermal Strain Data	26	Figure C1-1d	Four Packages Selected for Case Study	38
Figure A4-4	Extracted Instantaneous CTE $\alpha_{th,inst}$ Over Temperature. The instantaneous CTE $\alpha_{th,inst}$ Changes at the Glass Temperature T_g	26	Figure C1-2	Thermal Cycling Test Profiles	39
Figure A4-5	Extraction of the Secant CTE $\alpha_{th,secant}$ from Measured Thermal Strain Data for the Reference Temperature of 25 °C	26	Figure C1-3a	Different Model Type Schematics	40
Figure A4-6	Extracted Secant CTE $\alpha_{th,secant}$ Over Temperature for a Reference Temperature of 25 °C	27	Figure C1-3b	Different Model Type Schematics	40
Figure A4-7	Overview Viscoelastic Modeling	27	Figure C1-4a	Strip Model	41
Figure A4-8	Illustration of the Linear-Viscoelastic Material Modeling Approach Using the Generalized Maxwell Model Consisting of Several Spring-Damper Elements	28	Figure C1-4b	Strip Model	41
Figure A4-9a	Measurement and the Simulation of the Deflection Over Temperature of a Bi-material Beam Consisting of Molding Compound and Silicon. The Linear-elastic (LE) Model Clearly Overestimates the Deflection Whereas the LVE Model can Reproduce the Deflection Correctly	28	Figure C1-5	Example of Full Model Boundary Conditions for Component Level Temperature Cycling	41
			Figure C1-6a	Example of Boundary Conditions for Multichip Modules	42
			Figure C1-6b	Example of Boundary Conditions for Multichip Modules	42
			Figure C1-6c	Example of Boundary Conditions for Multichip Modules	42
			Figure C1-7a	Strip Model Boundary Conditions	43
			Figure C1-7b	Strip Model Boundary Conditions	43
			Figure C1-8	Critical Solder Joint Dimensions for Analysis	44
			Figure C1-9	CBGA400 Octant Model and Joint Mesh	45
			Figure D1-1a	Experiment vs. FEA Warpage Measurements Across Different Reference Planes	50

Figure D1-1b	Experiment vs. FEA Warpage Measurements Across Different Reference Planes	50	Figure E3-4a	Examples of the DIC Results Showing: (a) In-Plane, U_x ; and (b) Out-of-Plane, U_z , Displacement Components During the TB Testing of Bare-Aluminum Boards	62
Figure E-1	Typical System Manufacturing Process Flow	52	Figure E3-4b	Examples of the DIC Results Showing: (a) In-Plane, U_x ; and (b) Out-of-Plane, U_z , Displacement Components During the TB Testing of Bare-Aluminum Boards	62
Figure E1-2a	Typical Spherical Bend Test Set-Up	53	Figure E3-5	Finite Element TB Model of the Bare-Aluminum Board	63
Figure E1-2b	Typical Spherical Bend Test Set-Up	53	Figure E3-6	Comparison of the Force vs. Displacement Response from the Model and Experiment for a 32mil Thick Bare-Aluminum Board	64
Figure E2-1	A Typical Quarter Symmetric TB Analysis Model Created in Abaqus® CAE	54	Figure E3-7	Computational Model vs. Experimental Board Strains Comparison for the Strains on the Primary Board Side in a 32mil Thick Bare-Aluminum Board	64
Figure E2-2	Example of Solder Joint Mesh	55	Figure E3-8	Computational Model vs. Experimental Board Strains Comparison for the Strains on the Secondary Board Side in a 32mil Thick Bare-Aluminum Board	64
Figure E2-3	An Example of the Boundary Conditions in a Typical Quarter Symmetric TB Model ...	55	Figure E3-9	A Comparison of the Computational Model vs. DIC Experimental Results for the Board Displacements for a 32mil Thick Bare-Aluminum Board	65
Figure E2-4	A Schematic of the Post-Processed SJ Forces for a Given SJ Set in the Local Coordinate System	57	Figure E3-10	Comparison of the Force vs. Displacement Response from the Model and Experiment for a 62mil Thick Bare-Aluminum Board	65
Figure E2-5	Effect of Computational Model Element Type on the Board Strains and SJ Force	57	Figure E3-11a	Computational Model vs. Experimental Board Strains Comparison for the Strains on: (a) Primary Board Side; and (b) Secondary Board Side in a 62mil Thick Bare-Aluminum Board	66
Figure E2-6a	ETB Mesh Size Effect on Board Strains and SJ Force due to Variation in Mesh Size/Seed in the: (a) ETB Thickness Direction; and (b) ETB In-Plane Direction	58	Figure E3-11b	Computational Model vs. Experimental Board Strains Comparison for the Strains on: (a) Primary Board Side; and (b) Secondary Board Side in a 62mil Thick Bare-Aluminum Board	66
Figure E2-6b	ETB Mesh Size Effect on Board Strains and SJ Force due to Variation in Mesh Size/Seed in the: (a) ETB Thickness Direction; and (b) ETB In-Plane Direction	58	Figure E3-12	Comparison of the Force vs. Displacement Response from the Model and Experiment for a 62mil Thick Bare FR4 Board	66
Figure E2-7a	The Three Types of SJ Mesh Sizes/Seeds: (a) Coarse Mesh; (b) Medium Mesh; and (c) Fine Mesh	59	Figure E3-13	Comparison of the TB Model vs. Experimental Results for the Strains on the Primary Board Side for a 62mil Thick Bare FR4 Board	67
Figure E2-7b	The Three Types of SJ Mesh Sizes/Seeds: (a) Coarse Mesh; (b) Medium Mesh; and (c) Fine Mesh	59	Figure E3-14	BGA TV with the Board Strain Gauges Attached	67
Figure E2-7c	The Three Types of SJ Mesh Sizes/Seeds: (a) Coarse Mesh; (b) Medium Mesh; and (c) Fine Mesh	59	Figure E3-15	Comparison of the Force vs. Displacement Response from the Model and Experiment for BGA TV on a 40mil Thick ETB	68
Figure E2-8a	Package Mesh Size/Seed Effect on Board Strains and SJ Force Due to Change in Mesh Size/Seed in the: (a) Package Thickness Direction; and (b) Critical SJs	59	Figure E3-16a	Package Corner-to-Corner (SW, NW, NE and SE) Board Strain Comparison Between the Computational Model and Test Results: (a) e_2 Strain Component; and (b) e_4 Strain Component	69
Figure E2-8b	Package Mesh Size/Seed Effect on Board Strains and SJ Force Due to Change in Mesh Size/Seed in the: (a) Package Thickness Direction; and (b) Critical SJs	59	Figure E3-16b	Package Corner-to-Corner (SW, NW, NE and SE) Board Strain Comparison Between the Computational Model and Test Results: (a) e_2 Strain Component; and (b) e_4 Strain Component	69
Figure E2-9	Effect of Friction Coefficient on the Board Strains and SJ Forces	60	Figure E4-1	Board Level Test to Fail Data	70
Figure E3-1	A Typical TB Test Setup (Transient Bend Application of Quasi-Static DIC BKM, 2010) used for the Model Validation Process	61			
Figure E3-2a	Examples of (a) Schematic of the TB Test Set-up with STB (Shock Test Board); and (b) Position of the Strain-Gauges on the Primary Side of the Board	61			
Figure E3-2b	Examples of (a) Schematic of the TB Test Set-up with STB (Shock Test Board); and (b) Position of the Strain-Gauges on the Primary Side of the Board	61			
Figure E3-3	An Example of the Board-Strain Components (e_2 , e_4) at Three of the Strain Gauges from the TB Testing of Bare-Aluminum Boards ...	62			

Figure E4-2	Model Calibration Board Level Test Data for 40mil Thick Board	70
Figure E5-1	Model Prediction Results for TV (pitch=0.4mm and board thickness=28mil)	71
Figure E9-1	SJ Loading Angle	77

Tables

Table 4-1	Sample Final FEA Report Questionnaire	5	Table C1-7	Octant Model 1 Results and Comparison with Experimental Data	45
Table 5-1	Example of Physics, Loads and Degrees of Freedom	7	Table C1-8	Octant Model 2 Results and Comparison with Experimental Data	45
Table 11-1	Examples of Warpage and Fatigue Life Results Comparison Data	15	Table C1-9	Effect of "crplm" Command	47
Table A1-1	Typical Material Properties of Silicon	18	Table D1-1	Example of Warpage Results Comparison Data Across Different Reference Planes	51
Table A1-2	Typical CTE Properties of Silicon	19	Table E1-1	Parameters Defining Scope of Modeling Approach Applicability	53
Table A3-1	Recommended TMA Parameters	23	Table E2-1	Summary of the Finite-Element Mesh in the TB Model	54
Table A3-2	Recommended DMA Parameters	24	Table E2-2	Summary of the Contact Interactions in the TB Model	56
Table C1-1	Test Package Detail Information	38	Table E2-3	Summary of the TB Analysis Model Outputs	56
Table C1-2	Experimental Thermal Cycling Data	39	Table E2-4	Typical TB Analysis Model Run-Times	56
Table C1-3	Material Properties	39	Table E2-5	Details of the ETB Mesh Size/Seed Study	58
Table C1-4	3D Model Types and Tradeoffs	40	Table E2-6	Details of the Substrate Mesh Size/Seed Study	59
Table C1-5	Modeling Constants for Three Different Methodologies	43	Table E3-1	Summary of the TB Testing Used for Model Validation Purpose	60
Table C1-6	Strip Model Results and Comparison with Experimental Data	44	Table E5-1	Summary of Transient Bend Test Data for TV Used for Validation	70
			Table E5-2	Comparison Between Model and Experiment	71
			Table E5-3	General Reasons for Difference Between Model and Test Data	71

Numerical Analysis Guidelines for Microelectronics Packaging Design and Reliability

1 SCOPE

1.1 Introduction This document is an effort to standardize and document some of the basic tenets of a typical Finite Element Analysis (FEA) model. The intent of this document is to help educate new designers (and in some cases even experienced designers) on the basic information and best practices that should be captured and provided to technical reviewers of the results of FEA data.

FEA has been used in the microelectronics industry for several decades. Numerical models offer several advantages in the design of microelectronics devices. If used correctly, they can provide guidance on:

1. Potential tradeoffs to be made in achieving the optimal design of microelectronic devices.
2. Performance and reliability of microelectronics devices will be evaluated by including the impact of multiple design, material, geometry, and process parameters.

With the exponential reduction in the cost of computing, numerical models can save significant time and expense because they can be used to reduce the number of iterations of experimental tests that may be needed to finalize and optimize a design. With ever increasing cost pressure, shrinking design margins and faster time to market, there is a need to rely more on numerical models rather than running multiple time consuming and expensive experiments.

However, to achieve all these benefits, numerical models must be used correctly. This involves setting up the problem correctly, obtaining the input parameters accurately, running the analysis properly and outputting and interpreting the results correctly.

There is considerable “tribal” knowledge across the industry where individual designers have their own best practices on how they run their models. However, there is a wide spectrum of knowledge proficiency. At one end, there are engineers who design and develop their own models, refined over decades of experience. At the other end of the spectrum, there are engineers who have very little experience, and use ad-hoc information to develop models that are incomplete and could result in even more expense and time-consuming losses if implemented incorrectly.

Across the industry, there is a need to standardize the basic information, terminology and methodologies used to develop these numerical models. While individual models can be quite complex and customized, some best practices could help make model comparison across different players in the microelectronics supply chain much easier and scalable. The basic structure of this document is outlined in the schematic in Figure 1-1.

1.2 Purpose FEA is versatile and is used to solve several complex problems in microelectronics. It is impossible to standardize all applications and flavors in which FEA is performed. This document is not intended to create device release requirements or to imply that simulation or design information is to be provided to customers as evidence for or as a condition of device suitability for any given application. It is expected that this document will be the first of a series of numerical analysis guideline documents. This will serve as a foundational document for other domain specific guidelines that delve deeper into best practices and guidelines for specific applications of FEA. The expected modular structure of the numerical analysis reliability guidelines documents is shown in Figure 1-2.

The objective of this document is to provide some guidelines for the basic information that should be provided when FEA is performed in most cases. Examples are provided for some select cases (illustrated as pattern filled topics in Figure 1-2) to help users understand why certain information should be captured and reported. In addition, some examples of best practices are provided, to help guide users on best practices that could be implemented to avoid loss of information or accuracy. The examples provided in this document are not meant to be exhaustive or all encompassing. They are meant to highlight examples of key aspects that need to be taken into account when an FEA analysis is set up and the results reported.

Several commercial FEA software are widely used across the industry. Due to the nature of the analysis, some recommendations may be specific to certain software and may be articulated in the context of a specific FEA software code. However, these recommendations are not exhaustive and do not imply endorsement of any one software over another.

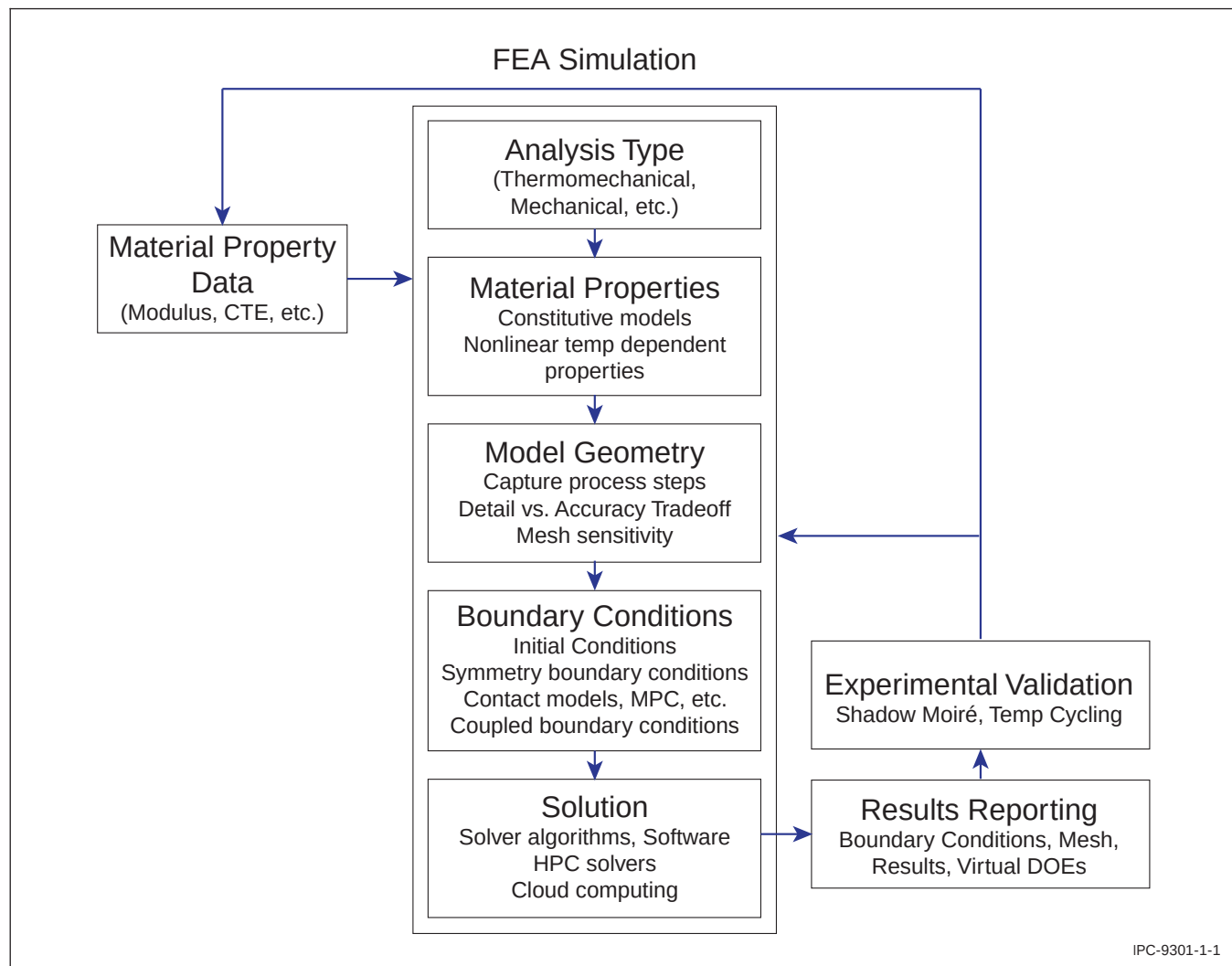


Figure 1-1 Building Blocks of a Typical Finite Element Analysis Problem

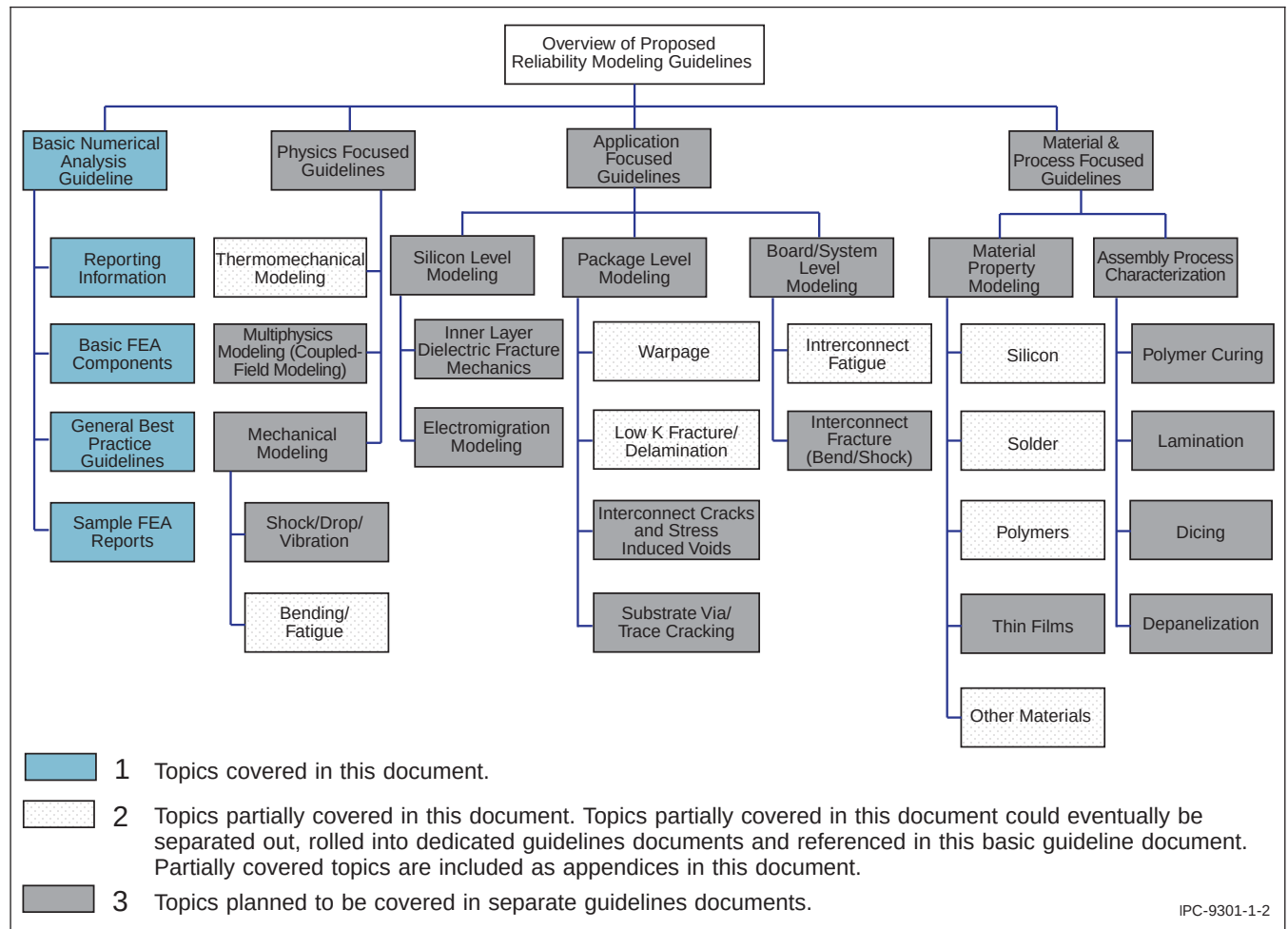


Figure 1-2 Expected Modular Structure of Numerical Analysis Reliability Guideline Documents

2 APPLICABLE DOCUMENTS

2.1 IPC¹

IPC-T-50 Terms and Definitions for Interconnecting and Packaging Electronic Circuits

IPC/JEDEC-9707 Spherical Bend Test Method for Characterization of Board Level Interconnects

2.2 ASTM²

ASTM B311-13 Standard Test Method for Density of Powder Metallurgy (PM) Materials Containing Less Than Two Percent Porosity

ASTM B962-14 Standard Test Method for Density of Compacted or Sintered Powder Metallurgy (PM) Products using Archimedes' Principle

ASTM D638-10 Standard Test Method for Tensile Properties of Plastics

ASTM D696 Standard Test Method for Coefficient of Thermal Expansion of Plastics Between -30 °C and 30 °C with a Vitreous Silica Dilatometer

ASTM D790 Standard Test Method for Flexural Properties of Unreinforced and Reinforced Plastics and Electrical Insulating Materials

ASTM D5023 Standard Test Method for Plastics: Dynamic Mechanical Properties: in Flexure (Three-Point Bending)

ASTM D5024 Standard Test Method for Plastics: Dynamic Mechanical Properties: In Compression

ASTM D5026 Standard Test Method for Plastics: Dynamic Mechanical Properties: In Tension

ASTM D5418 Standard Test Method for Plastics: Dynamic Mechanical Properties: In Flexure (Dual Cantilever Beam)

ASTM D7028 Standard Test Method for Glass Transition Temperature (DMA T_g) of Polymer Matrix Composites by Dynamic Mechanical Analysis (DMA)

ASTM E132-04 Standard Method for Poisson's Ratio at Room Temperature

ASTM E831 Standard Test Method for Linear Thermal Expansion of Solid Materials by Thermomechanical Analysis

ASTM E1545 Standard Test Method for Assignment of the Glass Transition Temperature by Thermomechanical Analysis

ASTM E1640 Standard Test Method for Assignment of the Glass Transition Temperature by Dynamic Mechanical Analysis

3 TERMS AND DEFINITIONS

The definition of all terms used herein **shall** be as specified in IPC-T-50.

1. www.ipc.org
2. www.astm.org

4 RESULTS REPORTING

To ensure that all pertinent and useful information is captured and communicated, a comprehensive final FEA report should be prepared and provided for all FEA analyses. The final FEA report provided should include details as outlined in all the applicable sections of this guideline. The details of each item are outlined in Table 4-1.

Table_4-1 Sample Final FEA Report Questionnaire

Parameter	Example	Value	Units	Description
Basic Analysis Information				
Analysis Type	Static, Transient etc.			
Physics Type	Mechanical Shock, Bend, Vibration, Thermomechanical, Multiphysics etc.			
Software	ANSYS® Mechanical, Multiphysics, ABAQUS®, COMSOL®, LS-DYNA®, etc.			
Software Version				
Analysis Script. If so, include in report	APDL, Log File, Model Files etc.			
Basic Model Information				
Model Category Levels	Silicon Level, Package Level, Board Level etc.			
Symmetry Conditions	Quarter, Octant etc.			
Schematic showing model stackup				
Geometric Information (Dimensions)				
Schematic showing boundary conditions	Show faces where constraints are set to zero. Include contour plots of high risk failure locations and plots of displacement from global/sub model if submodeling is used			
Material Property Information				
Reference Temperature			Celsius	
Temperature dependent properties used? Provide in report		Yes/No		
Glass Transition Temperature (T_g)	Materials 1, 2, 3 etc. Values per ASTM XXX		C	
Modulus below T_g	Materials 1, 2, 3 etc. Values per ASTM XXX		GPa, MPa	
Modulus above T_g	Materials 1, 2, 3 etc. Values per ASTM XXX		GPa, MPa	
CTE below T_g	Materials 1, 2, 3 etc. Values per ASTM XXX		ppm/C	
CTE above T_g	Materials 1, 2, 3 etc. Values per ASTM XXX		ppm/C	
Poisson's Ratio				
Viscoplastic Model & Constants	Materials 1, 2, 3 etc. Garofalo Creep, Anand Constants etc.			
Viscoelastic Model & Constants	Materials 1, 2, 3 etc. Maxwell Constants, Mooney Rivlin etc.			
Nonlinear Model & Constants	Materials 1, 2, 3 etc. Bilinear, Kinematic Hardening etc.			
Mesh Information				
Mapped or Tetrahedral?	Free or structured mesh			
Dissimilar Meshes? If so, specify interfaces	Different meshes joined with contact elements or constraint equations. Substrate/Substrate Interface			
Element Type	Provide the name and number of the element type used.			
Mesh Sensitivity Analysis?	Was effect of mesh density on results captured? If so, include results			
Loading Information				
Temperature dependent loading? If so, provide loading conditions	Temperature vs. Time data			
Ramped or Step Loading? If so, provide loading rate	The rate at which load is applied			



Parameter	Example	Value	Units	Description
Rate Dependent Loading? If so, provide loading rate	Force, Displacement, Energy applied as a function of time			
Solution Information				
Operating System	Win7, 8, IOS etc.			
Workstation or Cluster?				
High Performance Computing Used?	HPC licenses used?	Yes/No		
No. of Nodes Used (HPC)				
No. of Processors				
Processor Name	I7 etc.			
RAM Memory Size	96Gb, 128Gb, etc.			
Solution Algorithm	Sparse Solver, PCG, etc.			
Number of Load Steps				
Number of Time Steps				
Solution Time			Hours	
Post Processing				
Results Parameters	Warping, Fatigue Life, etc.			
Was volume averaging used? If so, provide specifics	Volume Averaging means averaging results over multiple elements. Show elements over which averaging was used	Yes/No		
Are the results temperature dependent? If so, provide at least 10 points over temperature	E.g. Reflow Profile, Thermal Cycling etc.	Yes/No		
Experimental Benchmark Data Available? Provide in report	Provide benchmark data to show correlation with model	Yes/No		
Experimental Calibration Data Available? Provide in report	Calibration data is used to derive the FEA model, whereas benchmark data is strictly for comparison purposes	Yes/No		
Tabulated Comparison of Results	Show percent difference between benchmark and model results			

5 BASIC ANALYSIS INFORMATION

5.1 Analysis Type Analysis type indicates the nature of the analysis performed. Static analysis is performed to analyze invariant solutions for the dependent variable(s) (which are typically displacements or temperatures). In static analysis, time derivatives of the dependent variable(s) are ignored.

Transient analysis, on the other hand, calculates the time-dependent response of the dependent variable(s). Consequently, inertial, damping and capacitance terms need to be included and impact the solution. A time integration technique is inherent in the finite element calculation. It should be noted that some FEA software (e.g., ANSYS®) uses the variable “time” to specify both actual time in a transient analysis and as a load index in static analysis.

It should be noted that time can also be modeled as discrete analysis steps, without the use of a transient method. In this simplified approach, some assumptions may be made, such as, for example, in the case of a thermally-cycled component. As thermal loads are applied in the form of load steps in lieu of a transient model, temperature variations across the component are assumed to be negligible.

Finally, modal analysis is used to calculate system natural frequencies and mode shapes under the presumption that the dependent variable has a solution which is harmonic in time. Modal structural analysis requires a mass matrix.

5.2 Physics Type Numerical analysis can be utilized to analyze several complex physical phenomena. It is important to document the underlying physics being modeled, because the type of physics dictates the types of boundary conditions, solution algorithms and loads that are applied to study the problem. Typical examples of physics type are shown in Table 5-1.

Since the underlying physics establish the framework for the analysis, it is important to explicitly state and explain the physics type being modeled.

Table 5-1 Example of Physics, Loads and Degrees of Freedom

Physics	Response	Loads	Degrees of Freedom
Mechanical Shock	Structural Deformation	Input G Excitation, Force, Displacement	Displacement
Mechanical Bend	Structural Deformation	Force, Displacement	Displacement
Mechanical Vibration	Structural Deformation	Force	Displacement
Thermomechanical	Temperature induced stress	Temperature	Displacement
Thermal-Electric (coupled field)	Thermal and Electric	Temperature, Current	Temperature, Voltage
Structural-Thermal-Electric (coupled field)	Structural, Thermal and Electrical Analysis	Force, Temperature and Current	Displacement, Temperature, Voltage

5.3 Software Several commercially available and proprietary codes can be used to perform numerical analysis. It is important to document the precise name of the software used, because variations in analysis settings and solution algorithms can vary significantly across software, which can impact results. Typical examples of software include (but are not limited to): ANSYS®, ABAQUS®, COMSOL®, LS- DYNA® etc.

Often, user defined functions are used to enhance the capabilities or simplify the analysis workflow within the commercially available software. If used, it should be explicitly stated that custom macros or user defined functions were used which could potentially impact the results and could not be replicated by anyone else that uses the same software. If non-commercial, proprietary software is used for the entire analysis, then it should be stated explicitly that “non-commercial, proprietary numerical analysis software” was used for performing the analysis.

5.3.1 Software Version Numerical analysis software is continuously updated, revised and modified as new solution algorithms are developed, errors are identified and new capabilities are added.

Variations in software versions can also result in markedly different results. Refer to the Appendix C, Section C1.3 for examples showing the effect of software version on analysis results.

Consequently, when reporting the software used for the analysis, the software version and release date should be explicitly included. For example: “ANSYS®, v15.0, 2014

5.4 Analysis Files and Scripts Numerical analysis software uses a wide variety of file types and structures. Even within the same software, several variations of file types can be used to perform the same analysis.

Often, numerical analysis is also performed using scripts, which are simple text files that store all the model building steps, analysis and post processing instructions. The scripts can be an excellent way to store information about the model because they show, line by line, each step taken to build the model and obtain results. Scripts can also be used for parametric analysis. Several macros can also be built using scripts and called as functions or libraries into other scripts to build the complete model.

A key benefit of scripts is their small file size. Since they only contain the model building instructions and not the actual model results, the scripts can be quite small. A user can easily run the script to generate the results for detailed review. Where possible, scripts can be shared in entirety for users and customers to review.

6 BASIC MODEL INFORMATION

6.1 Boundary Conditions The modeling approach and boundary conditions for the model can significantly impact the results obtained. The boundary conditions applied in the model should be clearly shown in the final report. For an example of boundary conditions applied to thermomechanical reliability, refer to Appendix C.

6.2 Model Stackup Schematics Schematic images should be included in the final report, illustrating the details of the model, mesh and the features included in the model. Examples of such schematic images are shown in Figure 6-1a, Figure 6-1b and Figure 6-1c.

6.3 Boundary Condition Schematics In the case of symmetric models, symmetric boundary conditions can be used. Figure 6-2 illustrates a quarter symmetric model. Symmetry boundary conditions are applied on both symmetric surfaces, as shown in Figure 6-3. From the zoomed in view in Figure 6-4, in the Y-Z cut plane, it can be seen that X displacement and Y, Z rotation degree of freedoms are constrained. In the case of 3D models where the node does not have a rotation of degree of freedom, only the X displacement boundary condition is in effect. The symmetric boundary condition on both Y-Z plane and X-Z plane constrains all the rigid body degrees of the freedoms, with the exception of the Z direction displacement. As seen in Figure 6-5, Z displacement was constrained on one node at the center corner of the model.

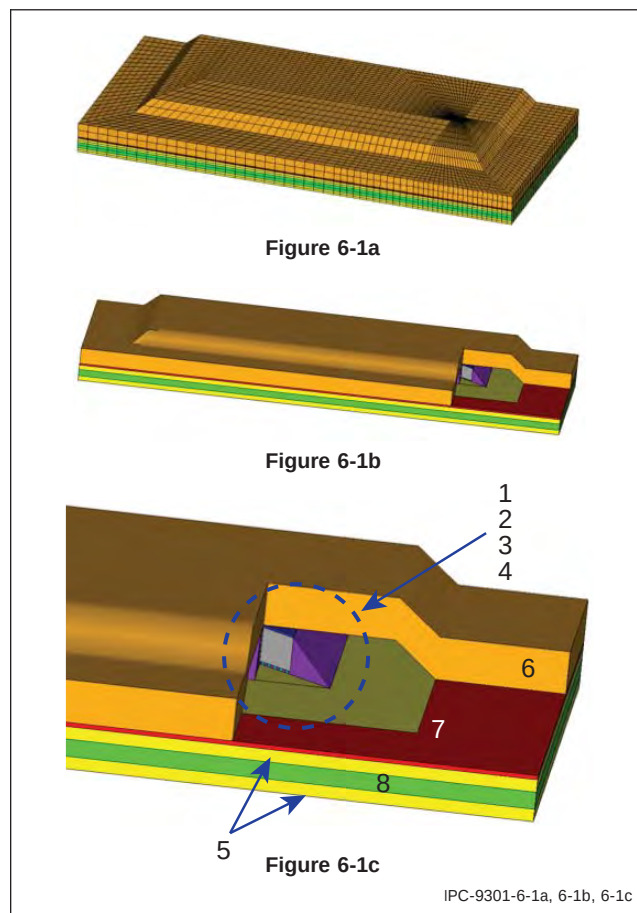


Figure 6-1a, Figure 6-1b and Figure 6-1c Examples of Model Schematics

- Note 1. Underfill (purple)
- Note 2. Silicon Die (gray)
- Note 3. Thermal Interface Material (blue)
- Note 4. Bumps
- Note 5. Upper and Lower Buildup Layers (yellow)
- Note 6. Lid
- Note 7. Lid Adhesive
- Note 8. Substrate Core (green)

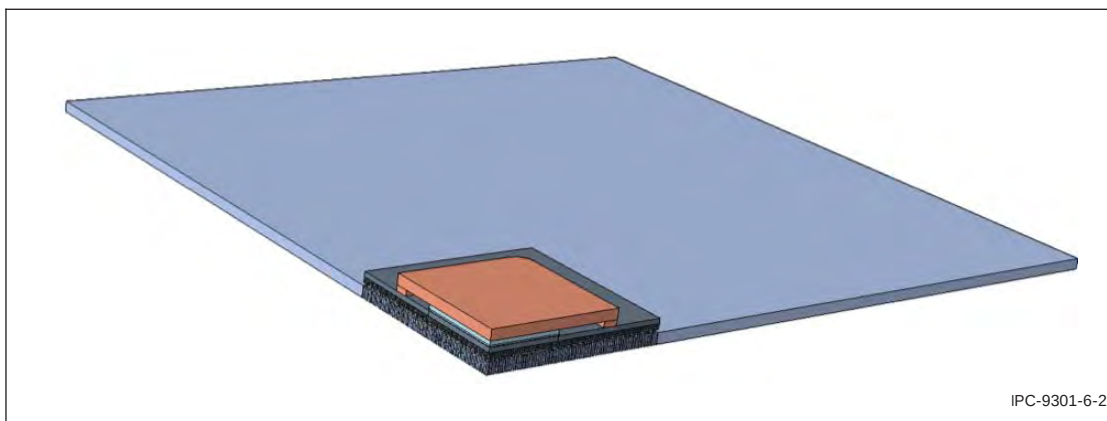


Figure 6-2 Quarter Symmetric Model

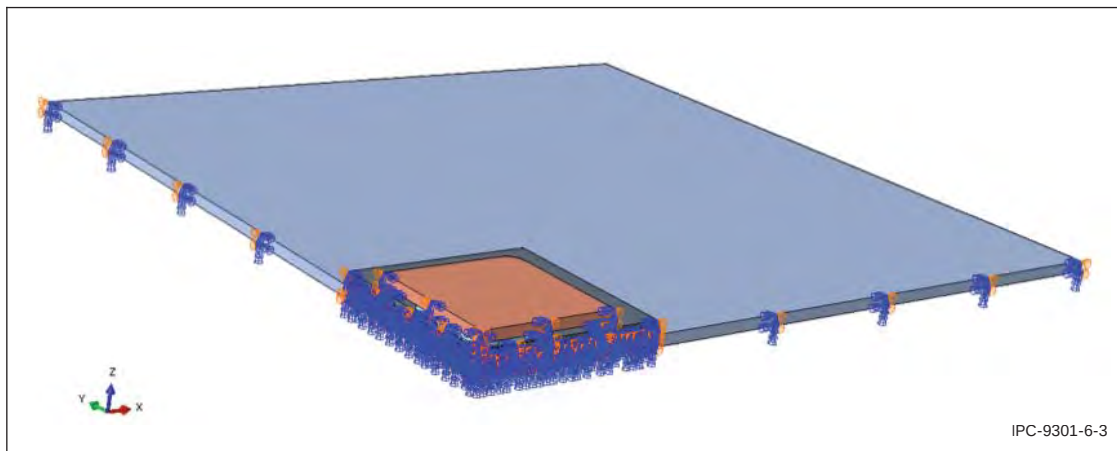


Figure 6-3 Symmetry Boundary Conditions Added

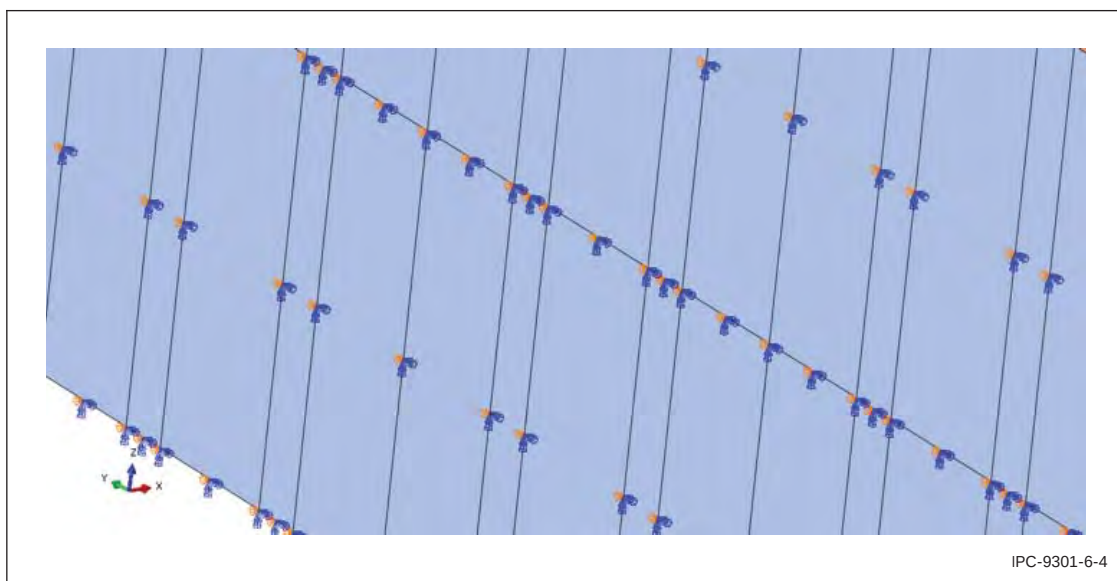


Figure 6-4 Zoom in View of Symmetry Boundary Conditions

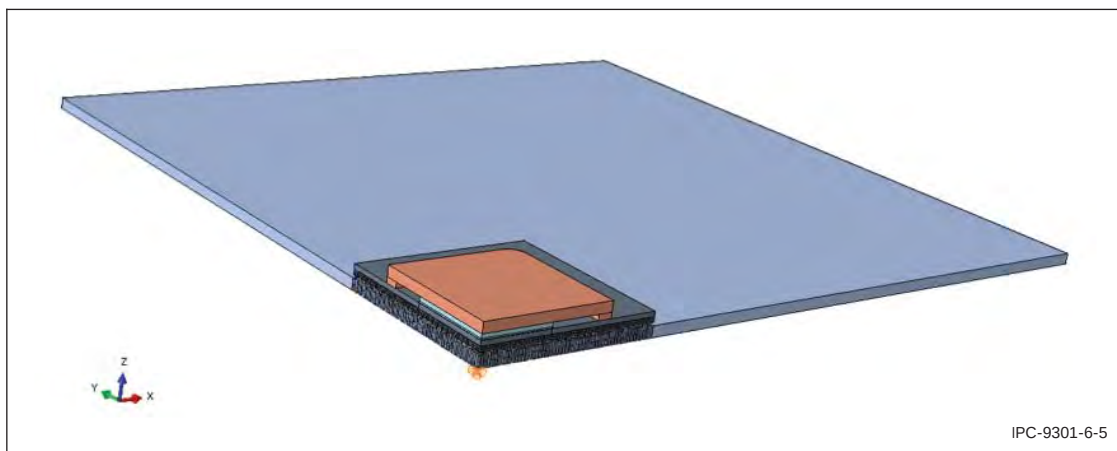


Figure 6-5 Displacement Boundary Conditions to Constrain All Rigid Body Displacement

7 MATERIAL PROPERTY INFORMATION

7.1 Material Properties Material properties can be used from known published literature and referenced properly. To ensure that the results predicted by the model are accurate and can be replicated by others; all material properties used in the model should be explicitly listed and the sources of the data explicitly referenced. For more specific guidelines on material properties for different material types, refer to Appendix A.

7.2 Reference Temperature The reference temperature is the temperature at which the material is considered to be at zero thermal strain. Generally, the reference temperature is the temperature that a material is “cured” or solidified between two different materials, e.g., a polymer cured after bonding between two materials. For solder joints, the reference temperature is the solidification temperature of the solder as the assembly is cooled from reflow to room temperature or lower. Most software packages provide two methods of specifying a reference temperature: global reference temperatures or reference temperatures specified as part of the material properties of each of the materials in the model.

Frequently, semiconductor assemblies may be modeled as having different reference temperatures for various constituents, but analyzed within a single linear analysis. Some analyses may benefit from the sequential assembly of a structure composed of subassemblies with different stress-free temperatures in multi-step nonlinear analysis. This procedure is sometimes described as element “birth or death” to signal the activation or de-activation of certain elements, simulating sequential assembly steps. If element “birth/death” is used, it should be explicitly stated in the final report.

7.3 Temperature Dependent Properties Temperature dependent properties need to span the range of temperatures the structure would be exposed to, in test conditions and use conditions. If temperature dependent properties are used, they should be explicitly reported in the final report, and tabulated data should be provided, outlining the properties at set temperature intervals.

8 MESH INFORMATION

8.1 Mesh Selection and Information A characteristic of the finite element method is that typically the numerical solution for dependent variables approaches the exact solution as the number of elements increases [1]. While dependent variables, such as displacements, approach the exact solution, derived results, such as stresses, may still deviate from the exact solution.

Consequently, the choice of mesh, both in terms of spatial discretization and element formulation, materially influences the accuracy and utility of a numerical solution. Definition of desirable meshing practices is a broad topic, which can be compounded by having different answers based on the nature of the problem to be solved. As a result, this document does not attempt to describe the attributes of “good” or “bad” meshes. It is presumed that the analyst is familiar with the theoretical background needed to define good meshes [e.g., [2] [3]] and performs convergence studies on a model to assure solution stability and correspondence to known analytic or experimental results. The transient bend discussion in Appendix D provides an excellent example of model tuning and validation through test.

Meshing is critical because it impacts the accuracy and sensitivity of a model. If a combination of meshing techniques and elements types are used, a picture of the whole model with the mesh should be provided. Detailed pictures should be shown of the mesh in high risk areas or critical locations. For example, if a model is focused on board level solder interconnect risk assessment, pictures of the solder joints mesh and the area around them should be shown.

The reason for element selection should be briefly discussed to explain what main physical phenomena was the target of the analysis and why this element type is capable of capturing it. For example, board or package bending is the main deformation factor for many failure mechanisms. In this case, element types with shear locking should be avoided, especially with limited layers in the thickness direction.

If multiple element types (e.g., shell and solid elements or linear and quadratic elements) are used in the same model, usually there will be deformation mismatch in the interfaces. These interfaces should be kept as far away from the points of interest as possible.

8.2 Mapped or Tetrahedral Mesh Generally, two different types of meshes are available, depending on the complexity of the geometry. Mapped meshes are more structured meshes with regularly or proportionately spaced elements, which are usually of the same shape. This method will generate higher mesh quality with uniform mesh size. When structured or mapped mesh cannot be achieved for complex geometry, “free meshing” can be adopted. This method may cause lower quality elements and discontinuities in the mesh. In general, effort should be made to use mapped meshes whenever possible.

Generally speaking, higher order shape functions will generate smoother results with lower element count. Linear brick elements have linear strain change from node to node, while tetrahedral elements have constant strain. So brick elements are suggested whenever possible.

8.3 Dissimilar Meshes Geometries with significant differences in feature size could require a large number of elements with mapped meshing. The higher the element count, the longer the computation time. In some instances, the computation time could become untenable as a result of the mesh. In such instances, different features could be meshed differently and then “joined” together by using contact elements or constraint equations.

If contact elements or “constraint equations” are used in the model to “join or tie” the dissimilar mesh elements together, the point of discontinuity at which the contact elements are set up should be kept as far away from the region(s) of interest as possible. An example of such an application is shown in Figure 8-1.

Moreover, care should be taken to ensure that the correct contact settings are used to minimize additional stiffness being introduced into the model due to the contact elements. The procedure used to tie dissimilar meshes affects accuracy. Methods that rely on constraint equations or contact elements enforced with penalty function or Lagrange multiplier techniques may permit geometric discontinuities at the contact surface. Alternately, when contacting surfaces only share certain nodes, the displacement of the unshared nodes are generally interpolated with the shape functions of the coarser mesh, unduly stiffening the finer mesh at the interface.

Most commercial FEA software allow for multiple contact behaviors, such as sliding, bonding etc. In situations where contact elements are used to join discontinuous meshes, bonding contact conditions are typically the best conditions to use. The contact or constraint conditions (sliding, bonding etc.) used in the models should be explicitly outlined in the report.

8.4 Mesh Sensitivity Analysis Complex geometries are particularly sensitive to the mesh used. Wherever possible, it is important to perform a mesh sensitivity analysis of the critical variables being studied (displacement, stress, strain energy density, creep etc.) to mesh density. Higher order node elements could be more sensitive than lower order node elements.

Tabulated results should be provided, showing the sensitivity of the key variables to mesh density. The results of at least three different meshes of varying densities should be analyzed and presented. Results provided without mesh sensitivity analysis can only be used for limited relative comparison purposes. An illustrative example of a mesh sensitivity analysis is outlined in Figure 8-2.

This example showed how creep strain energy density and creep strain range per cycle varied as function of number of elements used through the thickness of a thin layer (25-30µm) on the critical solder joint near the component pad interface. More elements through the thickness may increase the solution time significantly without improving the simulation results.

9 LOADING INFORMATION

9.1 Ramped vs. Stepped Loading How the load is applied in a problem depends on the problem being solved and the software being used. The loading rate is applicable specifically for nonlinear problems. Many software codes offer automated loading control to control the loading rate and achieve convergence. The convergence tolerance values can be adjusted for selected damage metrics (force, deformation, strain, L2 norm, etc.).

Loading type/rate will be important in cases where nonlinear time dependent material behavior is involved (e.g., viscoelastic, viscoplastic materials). Otherwise, the load type/rate is typically inconsequential to the end result (stress/strain/energy).

It should be clearly stated in the final report if non-default convergence tolerance values are used, and sensitivity analysis should be performed to understand the sensitivity of the convergence criteria on the solution results.

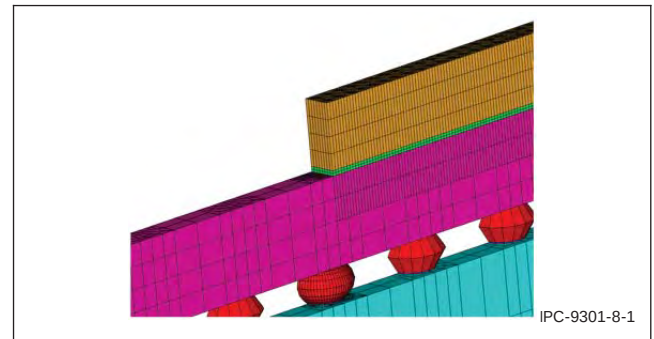


Figure 8-1 Example of Multi-Point Constraint to “Tie” Dissimilar Mesh Regions

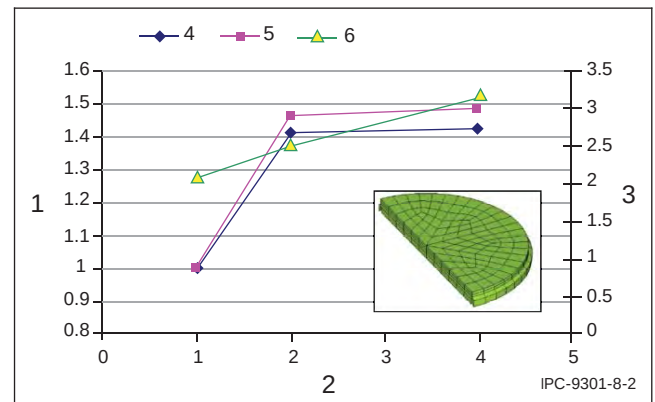


Figure 8-2 Example of Mesh Sensitivity Analysis

Note 1. Normalized Creep Strain/Energy Density
 Note 2. Element Division Number
 Note 3. Solving Time, Hour
 Note 4. Crp str
 Note 5. Crp eng den
 Note 6. Solve time

10 SOLUTION INFORMATION

10.1 Operating System The operating system should not impact the results, but it is important to document the type of operating system used (e.g., Windows® 32bit, Windows® 64bit, REHL® etc.). Documenting the operating system helps in benchmarking the computation time required to run the analysis.

10.2 High Performance Computing (HPC) The advance of computing technologies makes High-Performance Computing more cost attractive and accessible for general purpose. By processing the information in parallel using multicore/multiprocessors, the total solution time could be significantly reduced Figure 10-1 shows the total solution time in hours for a solder joint reliability model (Figure 10-2) as a function of the number of cores used for the analysis. In this particular benchmarking study case, the model consists of 249993 elements and 283871 nodes. The study was performed on clusters (each cluster node has 2 Intel® Xeon® Quad-core processors and 64GB RAM).

It is clear that the reduction in solution time is not linearly scalable to the number of CPUs/Cores used for the analysis. For this particular example, there is no benefit seen when the number of cores used is beyond 48. The reason for that is the available RAM from 6 cluster nodes (48 cores) is large enough to hold the entire model in memory, so the processor speed is becoming the gating item for further improvement. Furthermore, some software relies on the first node (master node), to “book keep” the information distributed among all CPUs/Cores, therefore more resources would be occupied for such a purpose instead of computation. There are a lot of factors that could impact the scalability of HPC performance. Listed below are the most common factors:

- The model size (total Degrees of Freedom (DOF), contact regions and current wave front width);
- Available physical memory (about 10GB RAM needed for 1 million DOFs);
- Problem nature (material nonlinearity prefers higher CPU clock frequency (higher mflops) over RAM size);
- Hardware efficiency (CPU clock frequency, and data bus speed);
- Software efficiency (ANSYS® recommends up to 8 cores for Shared Memory (SMP), and 32 cores for Distributed Memory Parallel (DMP));
- Interconnect speed (1000Mb/sec minimum interconnect speed recommended per ANSYS®);
- Scratch space location (in memory, SSD, HDD, NAS);
- Parallel processing type (SMP, distributed multiprocessor, GPU).

10.3 Hardware Parameters The hardware used to perform the analysis can impact the speed and performance of the analysis. In some cases, information about the hardware used can provide forensic insight on any potential problems/errors in the analysis. For instance, if an analysis takes 8 hours on a given hardware specification, but the same analysis takes 12 hours to run on different hardware, knowing the differences between the hardware can help determine the cause of the significant difference in analysis times.

The following key hardware parameters should be recorded and outlined:

1. Number of Processors/Cores used in the analysis
2. Memory (GB)

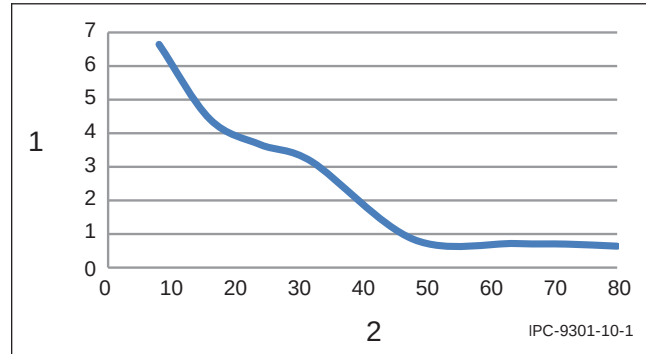


Figure 10-1 Computational Time vs. Number of Cores Used (PBGA 680 Solder Joint Model)

Note 1. Computational Time, Hour

Note 2. Number of CPU Cores

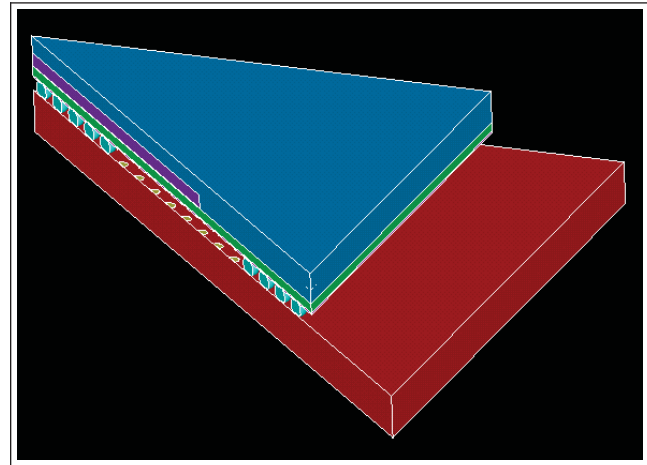


Figure 10-2 Octant PBGA680 Solder Joint Reliability Model

If using server cluster:

1. Number of nodes
2. Interconnect type between nodes e.g., FCoE, Infiniband® etc.
3. Typical interconnect speed between nodes

10.4 Solution Algorithms and Settings The algorithms and settings used depend on the software used for the analysis. Some best practices for some of the software commonly used in the industry for microelectronics modeling applications are provided in this section.

It is important to note that any software related recommendations should NOT be construed as endorsements of the use of the software in any way. These are only best practices consolidated from the general knowledge base of advanced users of software applications in the microelectronics industry. The user is encouraged to identify and develop similar best practices for the software they would be using for their applications.

10.4.1 Time Steps One condition that should be avoided is to allow the time increment to be much longer than the variations in loading history. Some FEA software allows users to input an amplitude time history profile to describe the loading history. Caution should be used to not let the time step become too big to capture some key changes in the loading step. One hypothetical example is shown in Figure 10-3. In this case, one possible solution is to manually setup stop points on the major turning points of the loading history.

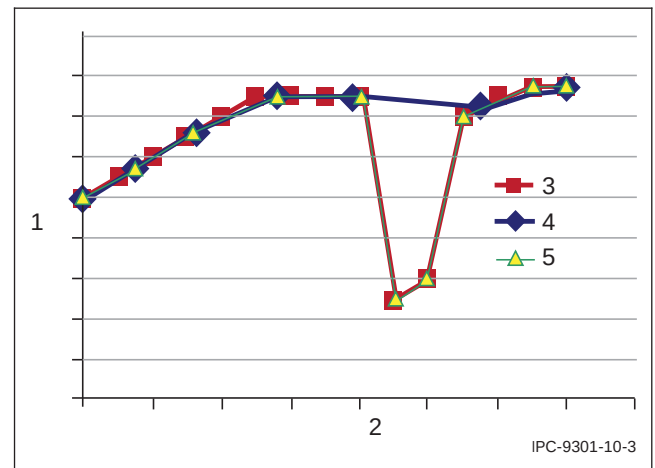


Figure 10-3 Example of Time Step Too Big to Capture Fluctuation in Loading History

Note 1. Load Level

Note 2. Time

Note 3. Loading Profile

Note 4. Time Step Too Big

Note 5. Reasonable Time Step

10.4.2 Solution Time The time taken by the simulation to solve and process should be included in the final report. Reporting solution times can help in understanding the degree of complexity of the model, and potentially help in debugging issues if the model is run at a different location. Differences in solution times can indicate potential convergence issues, boundary condition errors and even the type of material properties used. Temperature dependent, nonlinear material properties lead to time consuming models. Contact elements can also result in longer solution times. Errors in setting up the model are not easy to capture, but solution time can be a critical indicator of potential issues. The user can also review the solution output log (or “solve.out”) files to understand the causes of convergence issues.

11 POST PROCESSING

11.1 Post Processing Post processing involves analyzing the solution and extracting relevant results to be further analyzed for design decision making. Post processing in numerical modeling is not merely reviewing results. In some cases, the kind of post processing performed can significantly change the results output. For instance, displacement in finite element models is output as a nodal result. Stress on the other hand is frequently calculated at element integration points followed by extrapolation to the nodes with the element’s shape functions. Often averaging at contiguous element boundaries is used to produce aesthetically pleasing contour plots.

11.2 Results Parameters The following parameters are typically output from models and reported as results. The guidelines below highlight key practices when reporting these specific parameters.

11.2.1 Fatigue Fatigue life is an estimation of the number of cycles it will take for the structure to fail under cyclic loading. The cyclic loading could be thermal (as in thermal cycling) or mechanical (as in mechanical shock/drop). The structure in focus could be a solder joint, a trace or other structure.

The output of numerical models is never in terms of cycles. The output of numerical models is usually in the form of a damage metric (plastic work, strain energy density, creep strain, total strain etc.). The output metric should be explicitly identified and the metric values outlined.

The equations used to translate the damage metric into fatigue life (in cycles) should be explicitly stated and the transformed results included in the final report. For example, Darveaux's 4-constant model is commonly used in the microelectronics industry. [4]

11.2.2 Stresses In many instances, stresses and strains are compared when relative comparisons are required. A key challenge with finite element models is that the stresses or strains are extremely mesh density specific. That is because the formulation of elements that calculate stress (or strain) is based on a weighted residual within the entire element and, in some cases, because of permissible singular displacement fields. Further mesh refinement can result in markedly different stresses.

Comparisons of stresses in different loading/geometry scenarios should always be done with these two factors taken into consideration:

1. Only element output should be compared, not nodal stress output. Many commercial FEA software packages output stresses as nodal solutions, even though stresses are elemental solutions due to their dependence on area. Stress values and other derivative values are usually calculated at the integration point or the centroid of elements, depending on the element type. Nodal results are produced by averaging the stresses across the nodes of the element or extrapolated from the element, and averaged over the elements that share the same node. Element solutions (or Element Table solutions) should be used for comparisons.
2. The mesh density of the two scenarios being compared should be comparable. In addition, mesh sensitivity analysis should be performed as described in Section 8-3.

Volume averaging (as described in the next section) is also an effective technique to reduce the sensitivity of results on the mesh density.

11.3 Volume Averaging Finite element models inherently function by discretizing the domain into smaller elements. Consequently, the results depend significantly on the number of elements in a specific region of focus. Sharp corners, edges or regions of extremely high stress gradient could produce results that are so mesh dependent that they cannot be accurately quantified. This creates a "singularity," which can result in inordinately high stresses, strains, current density etc. in a small region. Any output parameter that is strongly dependent on area can be significantly impacted by singularities.

Volume averaging essentially distributes the results over a range of elements as opposed to using the output from a single element. The size variation of a single element can significantly skew results, whereas volume averaging the results over several elements tends to reduce the effect of the size of any single element. The volume averaging equation is:

$$E_{eff} = \frac{\sum VE}{\sum V}$$

Where E is the output parameter (stress, current density, strain energy density etc.) and V is the volume of the selected elements.

11.4 Temperature Dependent Results If temperature dependent material properties are used in the model, the results output of the model is also temperature dependent. The results extracted from the model should be at specified temperatures. For example, if stresses are extracted from the model, the stresses should be extracted at specific temperatures and those extraction temperature values should be included in the final report. Moreover, for any output parameters that are temperature dependent, the output parameters (such as stress, warpage etc.) should be reported for at least 2 distinct temperature values. Reporting results at two different temperatures facilitates a sanity check that can indicate whether the temperature dependent properties and boundary conditions have been correctly applied.

11.5 Correlation with Experimental Data

11.5.1 Nomenclature Numerical models are most effective when they are correlated well with experimental data. Correlating models with experimental data is a complex process, involving multiple stages with different semantics. To enable clear understanding, a nomenclature is outlined for each of the different steps involved in the correlation process:

1. **Validation:** This involves comparing parameters that are directly measured experimentally. For example, in monotonic bend tests, the experimentally measured strain and displacement for an applied force can be compared with model output data. To ensure good correlation, simpler, repeatable materials (like aluminum plates in monotonic bend testing) can be used for the experimental validation. The goal of validation is to confirm that the underlying physics of the model is

correct. Validation setups typically utilize simple, repeatable materials and geometries that closely capture the intended boundary conditions of the actual problem in focus.

2. **Calibration:** This involves using the numerical models to derive parameters that are not directly measurable in experiments and correlating them with model derived parameters. For example, determining the stress at which solder joints fail in monotonic bend tests. While PCB strain and displacement can be measured experimentally, the strain in the solder joint at which failure occurs cannot be measured experimentally. In this case, the solder joint strain output from the model for the same applied displacement and PCB strain can be correlated with the occurrence of failure. While the “true” strain in the solder joint is unknown, the model-derived joint strain can be “ascribed” to be the strain at which experimentally observed failure occurred. Experimental set ups should be first validated before they are used for correlation analysis.
3. **Benchmarking:** This involves comparing models that have been correlated with some experimental data with a *different* set of experimental data. It is imperative to ensure that the experimental data used to calibrate the numerical models is NOT the same data that is used to benchmark the models. After a model has been calibrated with select experimental data, the model should be compared with separate, independent experimental data to determine how well the model correlates with experimental data. The objective of benchmarking is to determine the versatility and range of applicability of the model. Only models that have been validated and calibrated should be used for benchmarking.
4. **Prediction/Extrapolation:** After completion of validation, calibration and benchmarking, the models are used to make predictions of future conditions (for which experimental data is not yet available) with a known range of accuracy. Only models that have been validated, calibrated and benchmarked should be used for prediction/extrapolation.

The final report should clearly show the experimental data used for calibration and the experimental data used for benchmarking the model. The final report should also indicate clearly when a model is used for prediction/extrapolation vs. benchmarking, correlation or validation.

11.5.2 Experimental Statistical Analysis Regardless of correlation stage, any experimental data used for correlation with models should be statistically analyzed before comparing with modeling data. All experiments have varying degrees of uncertainty due to variations in test set up, materials, test sample size, measurement data and failure thresholds. These uncertainties need to be accounted for in statistical terms before the experimental data can be compared with model output data. Refer to the appendix for an example on how statistical analysis is used to analyze experimental data before comparing with modeling data.

11.5.3 Comparison of Results The relevant result variables (e.g., warpage, fatigue, strain, stress etc.) should be reported in clear tabulated data. In addition, graphs and/or contour plots should be included to articulate the results of critical parameters (e.g., high stress and strain regions). If comparisons with experimental data are provided, the data should be clearly labeled to indicate validation, correlation, benchmarking or prediction data. Wherever applicable, differences between different model and/or experimental data should include percentage differences. An illustrative example of a reported results table is shown in Table 11-1.

Table 11-1 Examples of Warpage and Fatigue Life Results Comparison Data

SN	Fatigue Life Results (Cycles)						
	Validation (Warpage, μm)			Calibration (Plastic Work)	Benchmarking (Fatigue Life)		
	Model	Experiment	% Error	Model	Model	Experiment	% Error

In addition to comparing with experimental data, benchmarked FEA models are also very effective in performing virtual Design of Experiments (DOEs) or Design of Simulation (DOS). Virtual DOEs (or DOS) can be used to run numerical models to determine the effect of various parameters on the design. For instance, to determine the effect of modulus or CTE of a certain substrate material on the warpage of a package, the CTE and modulus of the substrate could be incrementally changed and the resulting warpage derived from the FEA model.

Just as in experimental DOEs, the results output from virtual DOEs run with FEA models can be used to determine the optimal combination of modulus and CTE to achieve a target minimum warpage. Various statistical software can be used in conjunction with FEA software to build factorial DOE matrices and analyze the output from the FEA models. Refer to the Flexural Bend case study in the appendix for an example of the application of virtual DOEs.

12 REFERENCES

- [1] M.J. Turner, R.W. Clough, H. C. Martin and L. J. Topp, Stiffness and Deflection Analysis of Complex Structures, J. of Aero. Sci., 23 (9), 1956.
- [2] J. N. Reddy, An Introduction to the Finite Element Method, New York: McGraw-Hill 2nd ed., 1993.
- [3] R. D. Cook, D. S. Malkus and M. E. Plesha, Concepts and Applications of Finite Element Analysis, New York: Wiley, 3rd ed., 1989.
- [4] R. Darveaux, "Effect of Simulation Methodology on Solder Joint Crack Growth Correlation," in *Electronics Components Technology Conference*, 2000.
- [5] D. Kiener and others, "A further step towards an understanding of size-dependent crystal plasticity: In situ tension experiments of miniaturized single-crystal copper samples," *Acta Materialia*, vol. 56, no. 3, pp. 580-592, 2008.
- [6] D. Kiener and others, "Determination of Mechanical Properties of Copper at the Micron Scale," *Advanced Engineering Materials*, vol. 8, no. 11, pp. 1119-1125, 2006.
- [7] D. Kiener and others, "Overview on established and novel FIB based miniaturized mechanical testing using in-situ SEM," *International Journal of Materials Research*, vol. 100, no. 8, pp. 1074-1087, 2009.
- [8] G. Dehm and others, "Can micro-compression testing provide stress-strain data for thin films?: A comparative study using Cu, VN, TiN and W coatings," *Thin Solid Films*, vol. 518, no. 5, p. 1517-1521, 2009.
- [9] D. R. Askeland and P. P. Phulé, The science and engineering of materials (5th ed), Cengage Learning. p. 198. ISBN 978-0-534-55396-8., 2006.
- [10] M. A. Hopcroft, W. D. Nix and T. W. Kenny, "What is the Young's Modulus of Silicon?," *Jour. Microelectromech. Sys.*, vol. Vol. 19, no. No. 2, p. 229 - 238, 201.
- [11] Y. S. Touloukian, R. K. Kirby, R. E. Taylor and T. R. Lee, Thermophysical Properties of Matter, 14 volumes, 1970 - 1978.
- [12] F. P. Incropera and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, Wiley, 1981.
- [13] ANSYS® *Mechanical Theory Reference*.
- [14] M. Motalab and others, "Creep Test Method for Determination of Anand Parameters for Lead Free Solders and their Variation with Aging," in *14th IEEE ITherm Conference*, 2014.
- [15] J. Eckermann and a. others, "Computational modelling of creep-based fatigue as a means of selecting lead-free solder alloys," in *14th IEEE International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Microelectronics and Microsystems, EuroSime*, 2013.
- [16] D. Bhate and others, "Constitutive Behavior of Sn3.8Ag0.7Cu and Sn1.0Ag0.5Cu Alloys at Creep and Low Strain Rate Regimes," in *IEEE Transactions On Components And Packaging Technologies*, Vol. 31, No. 3, 2008.
- [17] K. Mysore and others, "Constitutive and Aging Behavior of Sn3.0Ag0.5Cu Solder Alloy," in *IEEE Transactions On Electronics Packaging Manufacturing*, Vol. 32, No. 4, 2009.
- [18] C. Norris and A. H. Landzberg, "Reliability of Controlled Collapse Interconnections," *IBM J. Res. Development*, vol. 13, p. 266 - 71, 1964.
- [19] J. Bergstrom, Large Strain Time-Dependent Behavior of Elastomeric Materials, Massachusetts Institute of Technology, 1999.
- [20] J. Bergström and M. C. Boyce, "Constitutive modelling of the large strain time-dependent behavior of elastomers," *Journal of the Mechanics and Physics of Solids*, vol. 46(5), pp. 931 - 954, 1998.
- [21] P. Gromala, J. Duerr, M. Dressler, K. Jansen, M. Hawryluk and J. de Vreugd, "Comprehensive material characterization and method of its validation by means of FEM simulation," in *EuroSimE*, Linz, Austria, 2011.
- [22] H. Shirangi, B. Wunderle, O. Wittler, H. Walter and B. Michel, "Modeling Cure Shrinkage and Viscoelasticity to Enhance the Numerical Methods for Predicting Delamination in Semiconductor Packages," in *EuroSimE*, Delft, The Netherlands, 2009.
- [23] K. Jansen, M. Hawryluk and P. Gromala, "Cure dependent characterisation of moulding compounds," in *EuroSimE*, Linz, Austria, 2011.
- [24] P. Gromala, S. Fischer, T. Zoller, A. Andreescu, J. Duerr, M. Rapp and J. Wilde, "Internal stress state measurements of the large molded electronic control units," in *EuroSimE*, Wroclaw, Poland, 2013.

- [25] W. Dauksher and J. Lau, "A Finite Element-Based Solder Joint Fatigue Life Prediction Methodology for Sn-Ag-Cu Ball Grid Array Packages," *IEEE TDMR*, vol. 9, no. 2, pp. 231-236, 2009.
- [26] T. Anderson, A. Barut and E. I. Guven, "Revisit of Life Prediction Models for Solder Joints," in *IEEE Electronics Components and Technology Conference*, 2000.
- [27] J. Lau and Y. Pao, *Solder Joint Reliability of BGA, CSP, Flip Chip, and Fine Pitch SMT Assemblies*, McGraw-Hill, 1997.
- [28] *Abaqus® User Manual*.
- [29] J. Pan, "Comparing Techniques for Temperature-Dependent Warpage Measurement," *Forschung & Technologie, PLUS*," 2007. [Online]. Available: [http://www.akrometrix.com/whitepapers/Comparing%20Techniques%20\(10-07\).pdf](http://www.akrometrix.com/whitepapers/Comparing%20Techniques%20(10-07).pdf).
- [30] ASTM-D790, D5023, D5024, D5026, D5418.
- [31] R. Han and others, "A Preliminary Solder Joint Life Prediction Model by Experiment and Simulation for Translation of Use Condition to Temperature Cycling Test Condition," in *Electronics Components and Technology Conference*, 2013.

13 APPENDICES

The appendix covers the following relevant topics:

Appendix A – Material Property Modeling

Appendix B – DMA/TMA Fitting Algorithms

Appendix C – Experimental Benchmarking Model Case Study (Thermomechanical Reliability)

Appendix D – Warpage

Appendix E – Modeling Methodology Development for BGA Risk Assessment in Board Flexure Tests

APPENDIX A

Material Property Modeling

A1 MATERIAL PROPERTIES AND CONSTITUTIVE MODELS

A1.1 Material Properties and Constitutive Models Material properties and the constitutive models used in numerical analysis are critical in ensuring that the problem is accurately captured and the results are meaningful. This section provides some basic guidelines on how to incorporate material properties based on the types of materials used. Material models can be quite complicated and application dependent. Only general guidelines and best practices are provided in this document. The recommendations provided in this guideline are for bulk materials and do not necessarily apply to thin films. Details on material characterization of thin films can be found in published literature. [5], [6], [7], [8]

A1.2 Silicon Silicon is one of the most pervasively used material across the microelectronics industry. To enable consistency, some basic silicon material properties are provided in this section.

A1.2.1 Modulus The elastic modulus of a material is defined as the resistance of the material to applied force. It can also be defined as the slope of the material's stress-strain curve in the elastic deformation region [9]. How the modulus of a material is measured can significantly impact the properties derived. For some more common materials like silicon, the baseline properties shown in Table A1-1 can be used. In the table, the crystal directions [100] and [110] are industry-standard designations for the two common wafer types. The crystal directions may not be parallel, but be rotated by 45° to the chip edges. For both wafer types, “x” and “y” are in the plane of the wafer and “z” is in the through-the-thickness direction.

Table A1-1 Typical Material Properties of Silicon

Elastic Properties for [100] p-type Si wafers									
	-200	-100	0	20	100	200	300	400	600
Ex, Ey, Ez	1.32E+11	1.31E+11	1.30E+11	1.30E+11	1.29E+11	1.28E+11	1.27E+11	1.26E+11	1.23E+11
Gyz, Gxz, Gxy	8.04E+10	8.00E+10	7.96E+10	7.95E+10	7.91E+10	7.85E+10	7.79E+10	7.71E+10	7.54E+10
vyx, vxy, vxz, vyz, vzy, vzx	0.2794	0.2789	0.2785	0.2784	0.2782	0.2779	0.2777	0.2775	0.2773
Elastic Properties for [110] p-type Si wafers									
Temp (°C)	-200	-100	0	20	100	200	300	400	600
Ex	1.71E+11	1.70E+11	1.69E+11	1.69E+11	1.68E+11	1.67E+11	1.65E+11	1.64E+11	1.60E+11
Ey	1.71E+11	1.70E+11	1.69E+11	1.69E+11	1.68E+11	1.67E+11	1.65E+11	1.64E+11	1.60E+11
Ez	1.32E+11	1.31E+11	1.30E+11	1.30E+11	1.29E+11	1.28E+11	1.27E+11	1.26E+11	1.23E+11
Gyz	8.04E+10	8.00E+10	7.96E+10	7.95E+10	7.91E+10	7.85E+10	7.79E+10	7.71E+10	7.54E+10
Gxz	8.04E+10	8.00E+10	7.96E+10	7.95E+10	7.91E+10	7.85E+10	7.79E+10	7.71E+10	7.54E+10
Gxy	5.14E+10	5.12E+10	5.09E+10	5.09E+10	5.06E+10	5.02E+10	4.98E+10	4.93E+10	4.80E+10
vyx	0.0635	0.063	0.0626	0.0625	0.0621	0.0616	0.0611	0.0605	0.0593
vxy	0.0635	0.063	0.0626	0.0625	0.0621	0.0616	0.0611	0.0605	0.0593
vxz	0.3631	0.3625	0.3619	0.3618	0.3614	0.3611	0.3609	0.3608	0.361
vyz	0.3631	0.3625	0.3619	0.3618	0.3614	0.3611	0.3609	0.3608	0.361
Vzy	0.2794	0.2789	0.2785	0.2784	0.2782	0.2779	0.2777	0.2775	0.2773
Vzx	0.2794	0.2789	0.2785	0.2784	0.2782	0.2779	0.2777	0.2775	0.2773

Elastic properties for p-type silicon are calculated for [100] and [110] wafers from the temperature- dependent stiffness coefficients [10]:

$$C_{11} = 165.6 \cdot (1 - \Delta T \cdot 73.25 \times 10^{-6} - \Delta T^2 \cdot 49.26 \times 10^{-9}) \text{ (GPa)}$$

$$C_{12} = 63.9 \cdot (1 - \Delta T \cdot 73.25 \times 10^{-6} - \Delta T^2 \cdot 49.26 \times 10^{-9}) \text{ (GPa)}$$

$$C_{44} = 79.5 \cdot (1 - \Delta T \cdot 73.25 \times 10^{-6} - \Delta T^2 \cdot 49.26 \times 10^{-9}) \text{ (GPa)}$$

where ΔT is the temperature of interest (°C) minus the reference temperature, 25 °C.