Numerical Modeling of a System Level Failure Evaluation of Populated Flexible PCBs Subjected to Twisting/Tension Loading

Luciano Arruda, Jairo Quintero

Instituto Nokia de Tecnologia
Rodovia Torquato Tapajós, 7200 – Colônia Terra Nova,
CEP: 69054-415 – Manaus - Brazil
Phone: +55 92 21261925/fax: +55 92 21261062,
e-mail: jairo.quintero@indt.org.br

Abstract: Flexible Printed Circuit Boards (fPCBs) are being used more extensively in portable electronic devices due to their reduced thickness and ability to bend and adapt to various shapes. These characteristics have a significant role in miniaturization and therefore fPCBs are going to be prevalent in the future of electronic devices. The failure analysis due mechanical loads induced by stretching, twisting and bending due user environment is a key point of product reliability.

The ABAQUS computational package was employed to build a 3D FEA (Finite Element Analysis) considering all the BGA components welded on a flexible polyimide board as well as its solder joint. The critical point of this analysis is the different stiffness of polyimide and metals, and rigid electronics components attached to the flexible board during a dynamic loading history. The ABAQUS has been important to understanding the stress-strain distributions of the full fPCB to the final product design. The polyimide material strain-stress curve used as a flexible board was obtained experimentally as a plastic hardening material model. The most critical strain points on system level analysis have been verified near holes; solder joints, connections in flex-rigid parts, and in the corners of electronic components. The results obtained from simulation model should provide useful information for the scientific community and manufactures of electronic products, especially due to the useful ABAQUS as a FEM tools.

1. Introduction

Current market is driving portable electronic devices to become smaller and to have multiple functionalities. Some manufacturers have even started to embed portable electronic devices inside clothes. The addition of multiple functionalities such as cameras and multimedia players in small packages, and the ability to be shaped inside clothes often come through the use of Flexible Printed Circuit Boards (fPCBs), as they allow the use of mechanisms (slide, flip, rotate) and are easier to be used as non flat shapes (Coombs, 2001).To be able to be applied in mass production products, fPCBs need to be reliable regarding usage conditions. Some of the main causes of failure in electronic devices are thermal, moisture and mechanical loadings. Frequently, the thermal loading is applied to electronic devices using cycling loads (Arruda, 2010) in electronics industry and is a coupled analysis with mechanical field. The mechanical loads are frequently impact due to drops, twisting, swelling (Zhang, 2006) and bending during transportation and field use by consumers.

Particularly, this study will be discussing the twisting and bend analysis to quantify the failure of the components of printed circuits boards. There are two ways of failure analysis frequently used: 1) laboratory testing beyond products samples or 2) numerical simulation with a virtual model to make a predictable its physical behavior. A common tool to verify the reliability of electronic components in early stages of design is the use of the Finite Element Method (FEM). Nowadays, it is common the manufacturer makes laboratorial tests and numerical simulation in a earlier stage of product design. The FEA is an efficiency numerical technique before product manufacturing due its capacity to evaluate a lot of product design conceptions before its production considering time effort and economic expenditure..In the other hand FEA is applied too indentify physical behavior of the product design that not had been able to capture in lab experimentation or develop its correlations (Arruda, 2010). This work is based in work of product conception to shooting it in the market employing ABAQUS package (ABAQUS, 2010).

Since fPCBs have a different behavior from traditional rigid FR4 PCBs, this work aims at proposing a test vehicle and a simulation model for fPCBs. The 3D sample will undergo twisting and bend and results will be from simulation models. The structure of fPCBs (different number of layers) contains rigid parts FR4 board and flexes parts with polyimide material. Non linear geometric effects (stress stiffening) and material characteristics on bend and twisting simulation will be considered due a thin film in geometry and polyimide has a viscoelasticity material behavior. Over time, changes in the polymer due to physical aging will have profound effects on the viscoelastic compliance of the material, and hence will affect its long-term durability (Nicholson, 2002). The linear model is the simpler one, but the polyimide as substrate material used in the flexible board of a product is a thermosetting viscoelastic material. In this study will be used a hardening representation captured in a laboratorial experiments. The viscoelastic approach will be developing in future work. In this time, the hardening approach seems enough to the first concept design. It idea was captured in other works with laboratorial experiments with good correlations (Arruda, 2008 and 2009)

So, it will be discussed a preliminary step considering the main goal to obtain the strain level for the product design employing FEA technique. The finite element method here will be applied in order to improve the reliability analysis techniques to evaluate the failure of electronic boards made with polyimide and FR4 substrates.

2. Materials Data

The prototype was proposed in order to evaluate the reliability of flex-rigid interconnection. It consisted of a rigid PWB segmented into three parts which were connected by flexible parts (PCBs). The PCB layout is shown in Figure 1. Table 1 shows all material properties considered for the computational modeling of FR4 material. The FR4 linear anisotropic material approach shows adequate to the level of loading by manufacturer design.

Viscoelastic properties and material nonlinearities were not considered in this first approach for polyimide based flex part. It is important to note that the main goal of this approach using FEA is to understand how effective could be the nonlinear model in a hardening approach as a failure predictor, as well as to detect the critical regions.

Table 1. Material Properties

Material	Elastic Modulus (MPA)	Poisson Ratio	Coefficient of Thermal Expansion ppm/°C
FR4	E_x= 25,000 E_y= 25,000 E_z= 11,000	ν_{xy}= 0.11 ν_{xz}= 0.39 ν_{yz}= 0.39	α_x=17E-6 α_y=17E-6 α_z=60E-6
Copper	E_x= 117,000	ν= 0.30	α_x=17.7E-6
Polyimide	E_x= 2300	ν= 0.34	α_x=50.0E-6

Tensile tests were performed on in order to verify if material properties applied on first simulations are coherent with the real material properties in the level of loading acquired for a flex part. The figure 1 shows the test results in a INSTRON equipment.

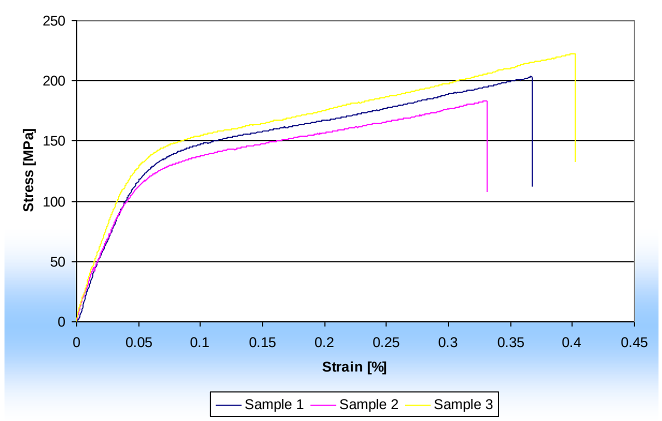


Figure 1 – Tensile test stress performed on thin film of Polyimide Material.

3. Finite Element Modeling

3.1. Finite Element Modeling

The simulation model is shown in figure 2 in a 3D solid element mesh. The rigid-flex board was modeled in ABAQUS using the solid element type of 3D solid element for each layer. It is important to observe that this model is a full 3D mesh where each layer was meshed with this 3D solid element. The flexible printed circuits board (fPCB) attached with several electronics components. The major difficulties of product design are the transitions of the polyimide material in a fPCB part and the rigid part represented by PWB and electronics components materials. It is possible to note the differences between Young modulus of these materials beyond table 1 (FR4) and the figure 1 (tension test of polyimide).

Using ABAQUS package is possible reduce analysis time and modeling if employ the contact analysis areas considered in the FE Model as showed in figure 3. In this way all electronics components was attached in its respective area by tie command in ABAQUS. It was considered the simpler contact properties called tie.

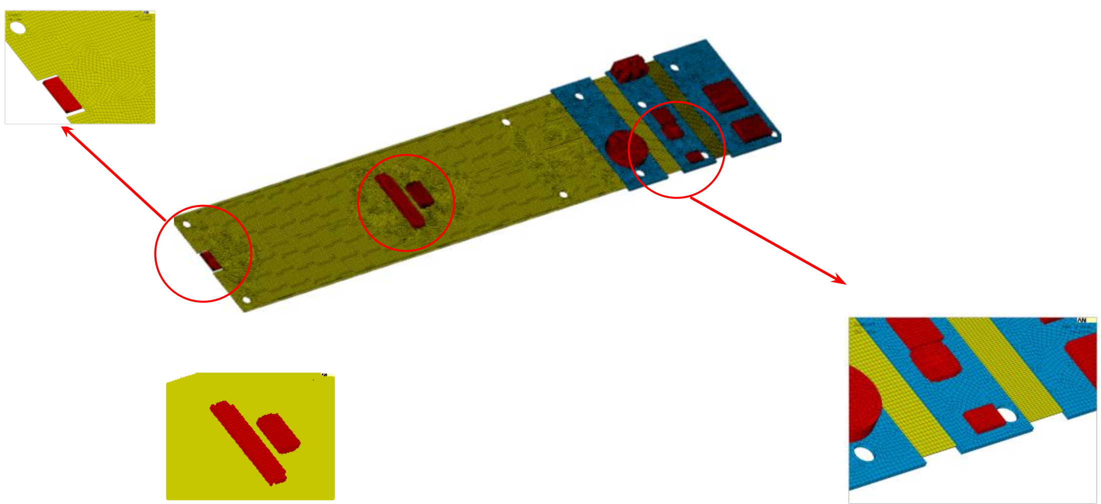


Figure 2 – Global and Detail's mesh used in a flex-PCB model.

3.2. Finite Element Summary

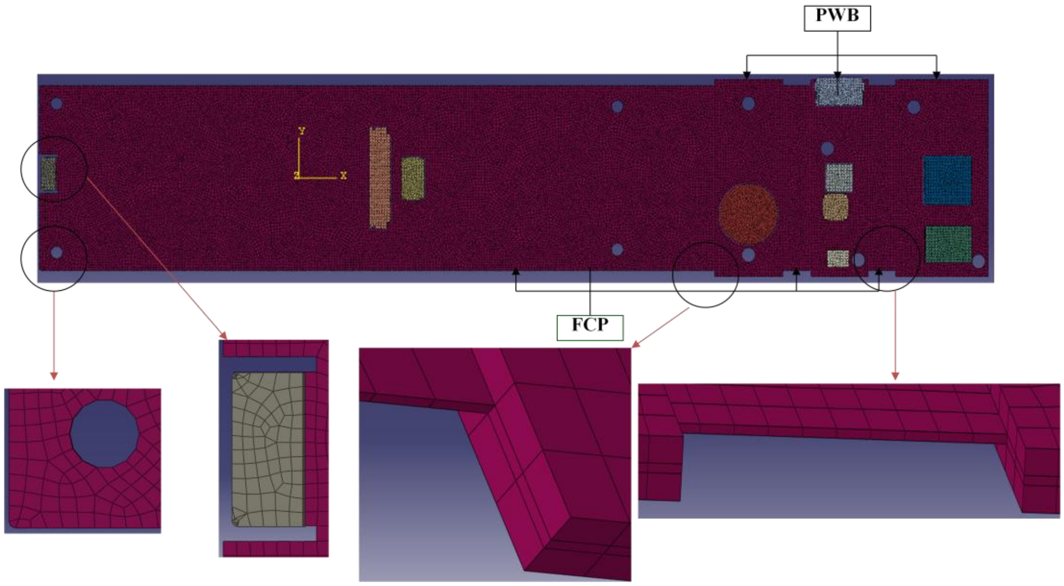
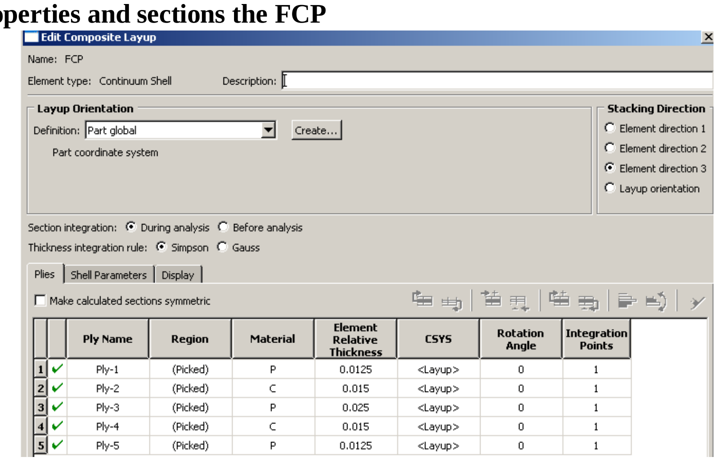


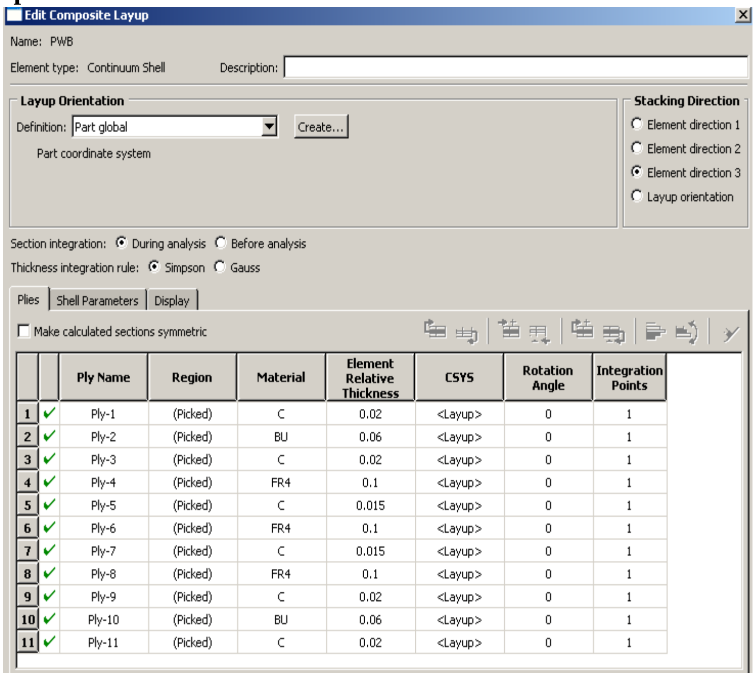
Figure 3 – Contact areas considered in the FE Model.

Material Properties and sections the FCP



	Ply Name	Region	Material	Element Relative Thickness	CSYS	Rotation Angle	Integration Points
1	Ply-1	(Picked)	P	0.0125	<Layup>	0	1
2	Ply-2	(Picked)	C	0.015	<Layup>	0	1
3	Ply-3	(Picked)	P	0.025	<Layup>	0	1
4	Ply-4	(Picked)	C	0.015	<Layup>	0	1
5	Ply-5	(Picked)	P	0.0125	<Layup>	0	1

Material Properties and sections the PWB



	Ply Name	Region	Material	Element Relative Thickness	CSYS	Rotation Angle	Integration Points
1	Ply-1	(Picked)	C	0.02	<Layup>	0	1
2	Ply-2	(Picked)	BU	0.06	<Layup>	0	1
3	Ply-3	(Picked)	C	0.02	<Layup>	0	1
4	Ply-4	(Picked)	FR4	0.1	<Layup>	0	1
5	Ply-5	(Picked)	C	0.015	<Layup>	0	1
6	Ply-6	(Picked)	FR4	0.1	<Layup>	0	1
7	Ply-7	(Picked)	C	0.015	<Layup>	0	1
8	Ply-8	(Picked)	FR4	0.1	<Layup>	0	1
9	Ply-9	(Picked)	C	0.02	<Layup>	0	1
10	Ply-10	(Picked)	BU	0.06	<Layup>	0	1
11	Ply-11	(Picked)	C	0.02	<Layup>	0	1

Table 2 – Model Summary

Options	Employed
Problem Dimensionality	3-D
Degrees of Freedom	Ux, Uy and Uz
Analysis Type	Static (Steady-State)
Equation Solver Option	Direct Sparse
Newton-Raphson Option	Program Chosen
Globally Assembled Matrix	symmetric
Number of Element	41,350
Number of Nodes	69530
Solution Time	27.80 sec

Table 3 – Model Summary

Options	Employed
Problem Dimensionality	3-D
Degrees of Freedom	Ux, Uy and Uz
Analysis Type	Static (Steady-State)
Equation Solver Option	Direct Sparse
Newton-Raphson Option	Program Chosen
Globally Assembled Matrix	symmetric
Number of Element	41,350
Number of Nodes	69530
Solution Time	26.40 sec

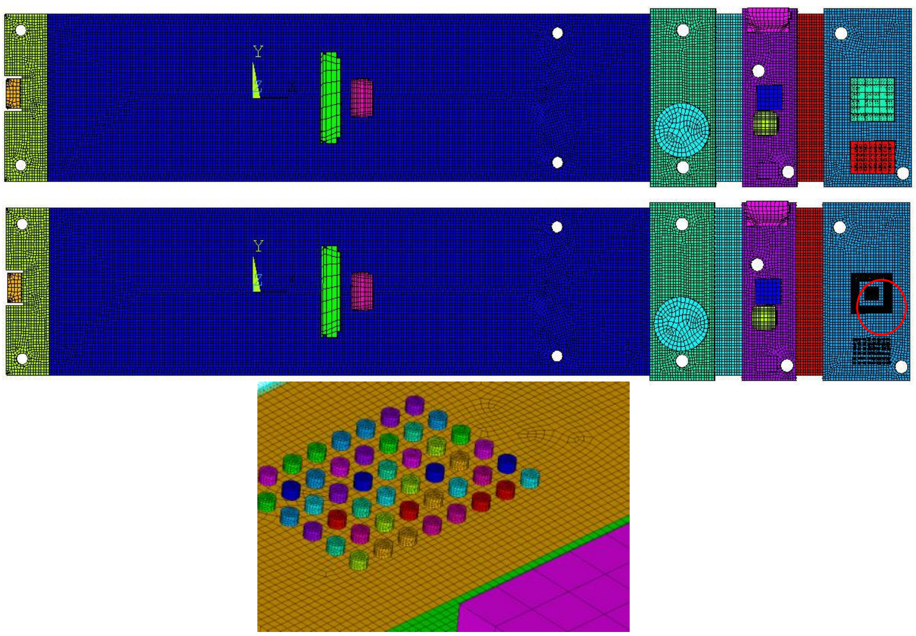
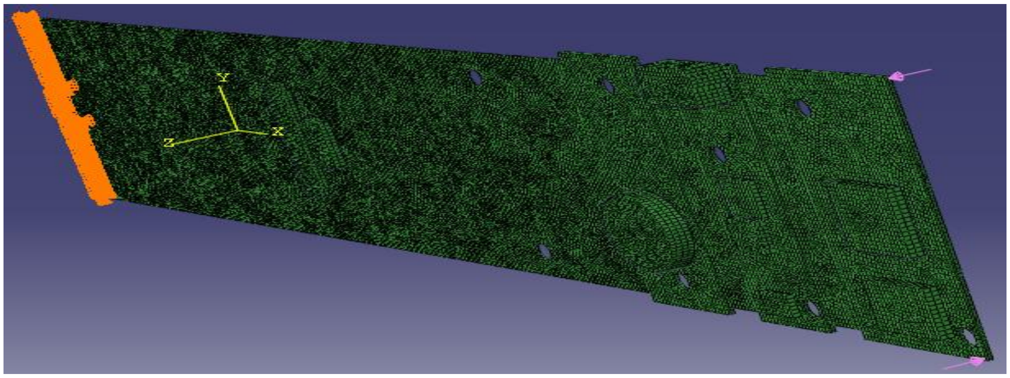


Figure 4 – Shows the BGA components attached to the board by solder joints considered in a full 3-D geometry model.

The electronic components were glued to the flexible substrate by contact element. The 3D solder joints geometry was used only in the BGA components. This is the approximation that was chosen in this preliminary stage of the product design. In this way it was possible to study the strains and stresses between the flexible parts and attached components in order to choose the best attachment. Additionally the contact elements were employed to glue the flexible parts to the rigid parts. The position of the contact areas can be seen in Figure 3.

4. Results

4.1 – Twisting



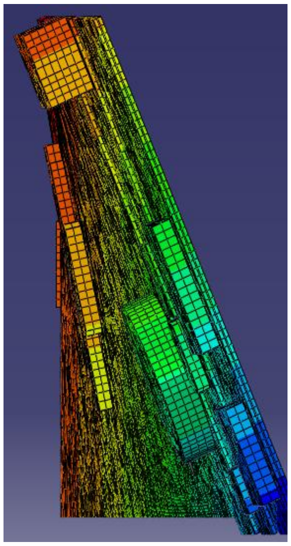


Figure 5 – Twist displacement will be gradually applied up to 45°.

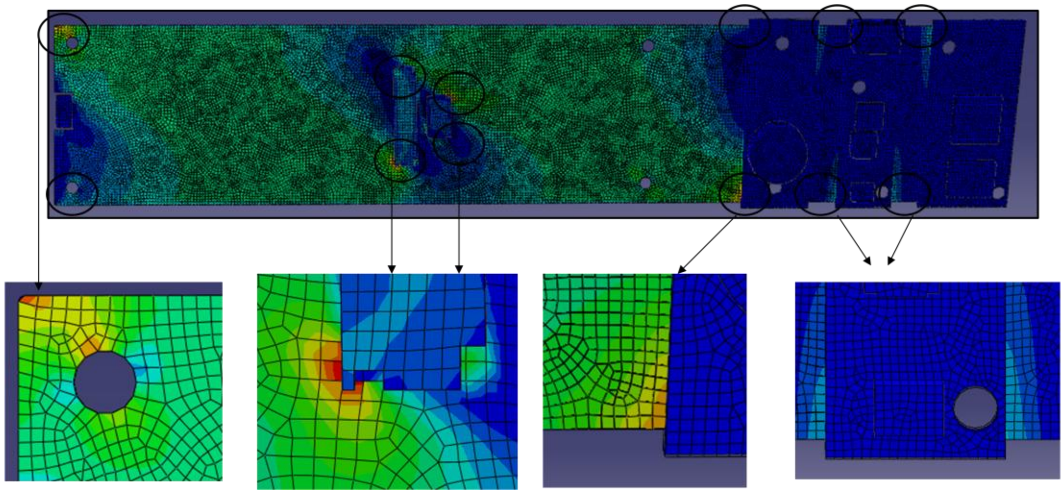


Figure 6 – Resuls of Twisting Load with 45^o Rotation

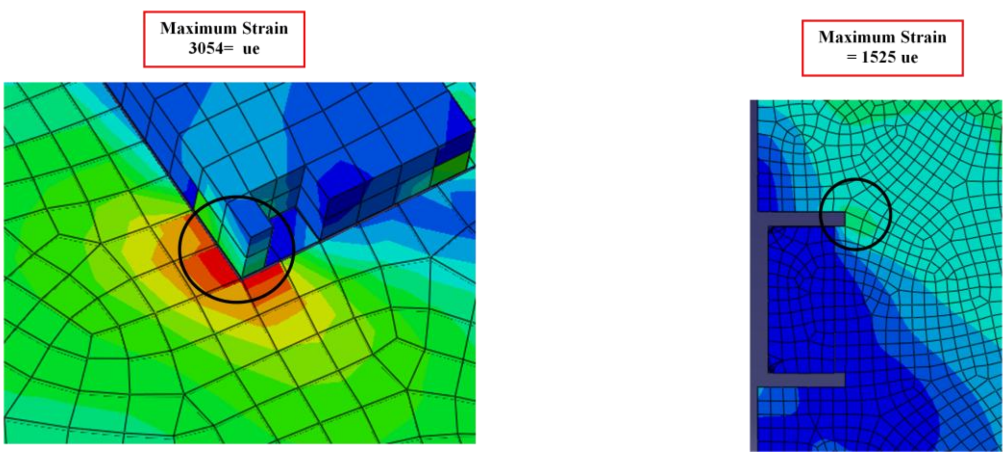


Figure 7 – Results of Twisting Load with 25^o Rotation.

4.2 – Bending

Table 3 - Model Summary

Options	Employed
Problem Dimensionality	3-D
Degrees of Freedom	Ux, Uy and Uz
Analysis Type	Static (Steady-State)
Equation Solver Option	Direct Sparse
Newton-Raphson Option	Program Chosen
Globally Assembled Matrix	symmetric
Number of Element	41,350
Number of Nodes	69530
Solution Time	27.80 sec

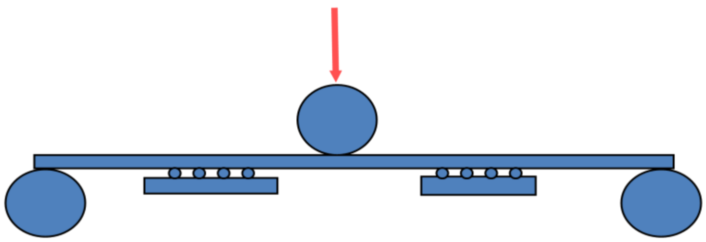


Figure 8 – Three Point Model

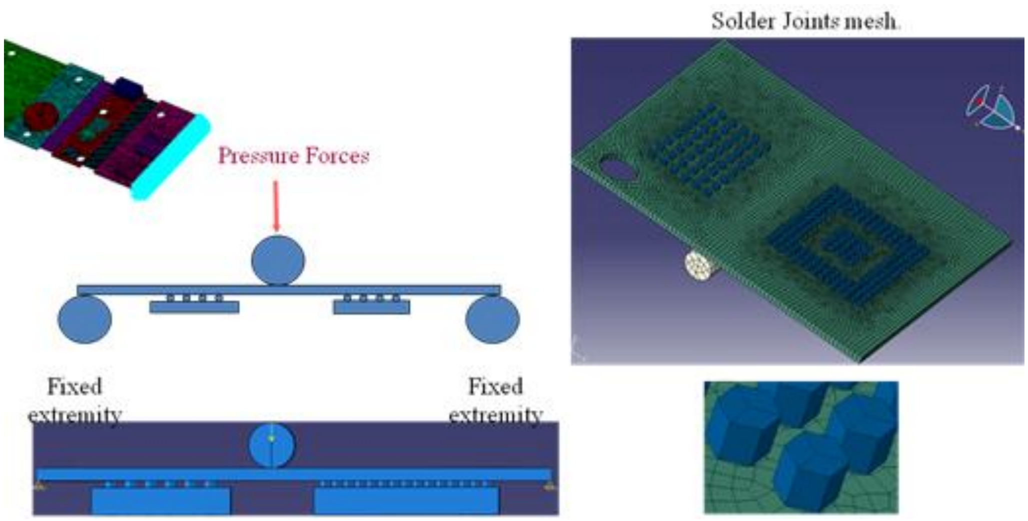


Figure 9 – FE Mesh and Applied Boundary Conditions. The loading was increased until reached 3000ue.

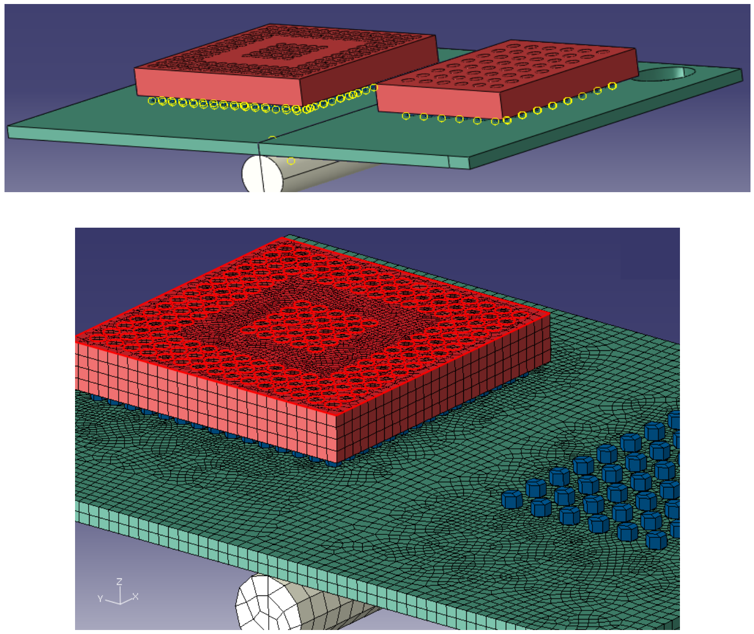


Figure 10 – FE Model. A: Solder Joint attached by contact region on PWB part. B: Contacts Pairs

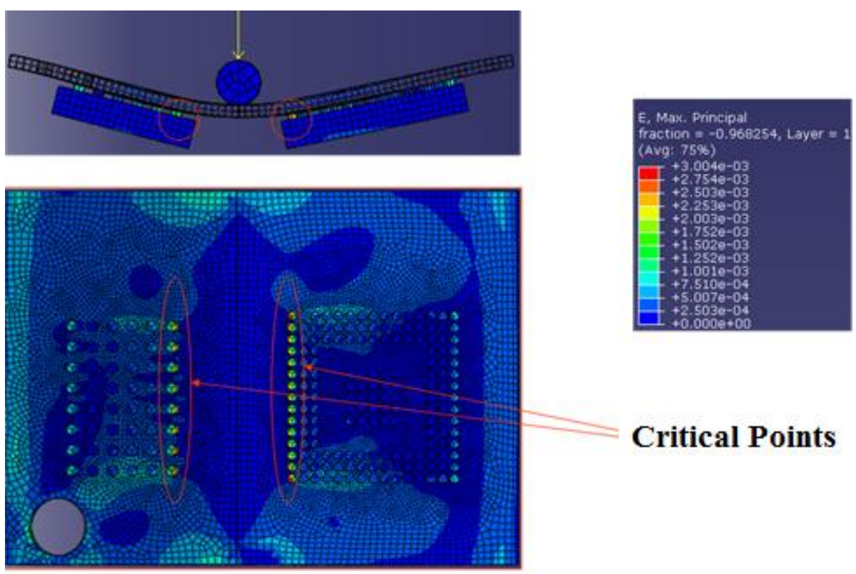


Figure 11 – Critical Points Maximum value (3004 ue)

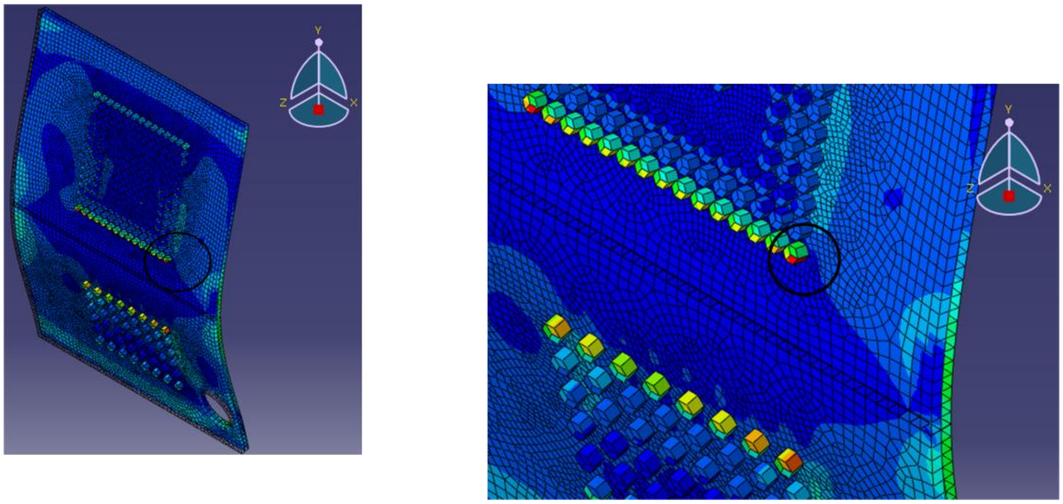


Figure 12 – Von Mises strain distribuition on posterior view of PWB part. Maximum value (3004 ue)

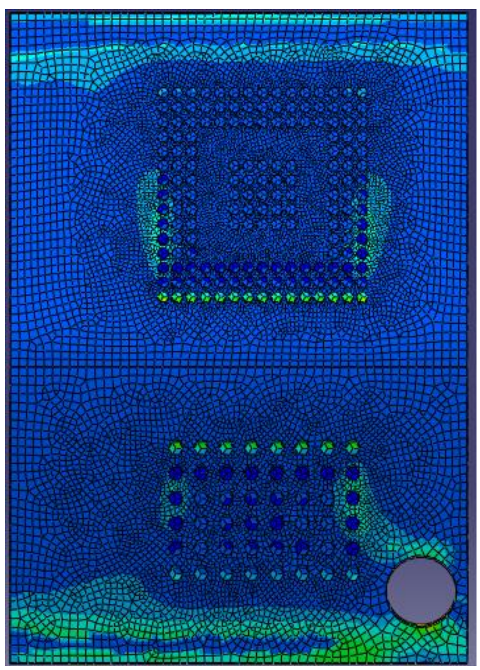


Figure 13 – Von Mises stress (MPa) distribuition on posterior view of PWB part. The red circle shows the 125.5MPA

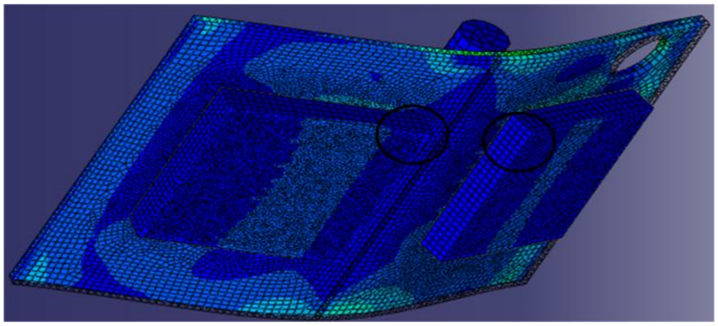


Figure14 – Von Mises strain distribution on front view of PWB part.

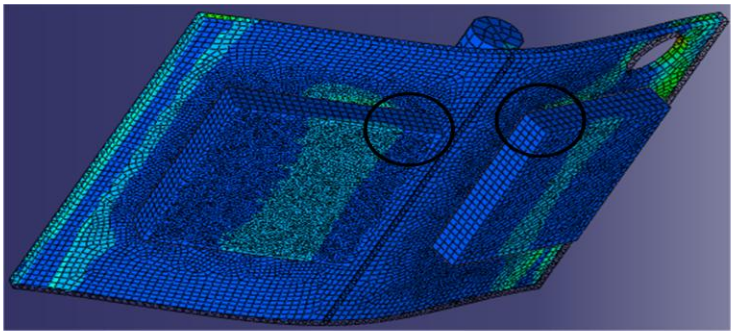


Figure 15 – Von Mises stress (MPa) distributions on Front view of PWB part.

5. Conclusion

The failure analysis shows that the probability of microcracks would be in the connection between the rigid and flexible boards and components and flexible parts. The large number of failures is probably related to the design of the board and the differences between the Young modulus values of materials. Thus, both factors (design and mismatch materials) could contribute to the occurrence of high stresses in the connections.

The proposed simulation model is able to identify the critical regions. Thus, this FEM approach may be considered to be successful when applied in early stages of design. Enhanced FEM simulation models considering non-linear material properties and the fracture model would be the next steps to provide a better model for the evaluation of product design.

6. Acknowledgment

The authors would like to thank José Gaston for supervising the program. Thanks and appreciation also extended to Antti Salo from NRC Beijing as well as all the INdT and NRC members that provided support for this work directly and indirectly.

7. References

1. ABAQUS, Electronics Manual, Version 10.0, USA, 2010.
2. Arruda, L., " On the Simulation of Flexible Circuit Boards" In: EMPC2007 - 30th Anniversary Conference of IMAPS, 2007, Oulu. EMPC2007 - European Microelectronics and Packaging Conference & Exhibition in Oulu, Finland June 17-20, 2007. IEEE, 2007.
3. Arruda, L. , Bonadiman, R., Costa, J., Reinikainen, T. "Cracking Phenomena on Flexible-Rigid Interfaces in PCBs under Thermo Cycling Loading". Circuit World. , v.35, p.18 - 22, 2009.
4. Arruda, L., Chen,Q. and Quintero, J., " Failure Evaluation of Flexible-Rigid PCBs by Thermo-Mechanical Simulation" In: 2009 International Conference on Electronic Packaging Technology & High Density Packaging(ICEPT-HDP), 2009, Beijing. 2009 International Conference on Electronic Packaging Technology & High Density Packaging(ICEPT-HDP). Beijing: Keyun BI (CEPS) Jian CAI (Tsingha University), 2009. p.1201 – 1205
5. Arruda, L., Bonadiman, R., Costa, J. and Freitas, G. G. "Experimental and Numerical Analyses of Flexible PCBs under Various Loading Conditions" In: EUROSIME 2009 - 2009 International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Micro-Electronics and Micro-Systems, 2009, Delft. 2009 International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Micro-Electronics and Micro-Systems. Piscataway, NJ 08855 USA: IEEE, 2009.
6. Banevicius, P., Gydas, J. "Analysis of Stress Relaxation Process in Polymers and High and Low Temperatures" *Materials Science.*, Vol. 10. No.2, pp. 237-243 (2004).
7. Coombs' Jr, Clyde F., Coombs' "Printed Circuits Handbook", McGraw Hill (2001), pp. 56.3 – 56.6.
8. Nicholson, L.M. et all, "The Combined Influence of Molecular Weight and Temperature on the Aging and Viscoelastic Response of a Glassy Thermoplastic Polyimide". *NASA /TM - 2000-210312. Technical Memorandum*, September, 2002. Hampton, Virginia, USA. pp. 1-2.
9. Suhir, E., Arruda, L. "The Coordinate Function In The Problem Of The Nonlinear Dynamic Response Of An Elongated Printed Circuit Board (PCB) To A Drop Impact Applied to its Supports". EPJ. Applied Physics. , v.48, p.20505 - , 2009.
10. Suhir, E., Arruda, L. "Duffing Oscillator: Could an Impact Load of Finite Duration be Substituted with an Instantaneous Impulse?". JSME International Journal. Series A, Solid Mechanics and Material Engineering (Cessou em 2006). , v.4, p.1381 - 1397, 2010.
11. Ward, I.M., Sweeney, J. "An Introduction to the Mechanical Properties of Solid Polymers", 2nd Edition, John Wiley & Sons, (2004), England.
12. Zhang, G. Q.; van Driel, W.D.; and X.J. Fan – "Mechanics of Microelectronics – Solid Mechanics and Its Applications". Springer, 2006, Netherlands.