SOLDER JOINTS DESIGN OPTIMIZATION OF QFN PACKAGES FOR DIFFERENT BOARDS BY INVESTIGATING SOLDER JOINT RELIABILITY

By

RISHIKESH TENDULKAR

Presented to the Faculty of the Graduate School of

The University of Texas at Arlington in Partial Fulfillment

of the Requirements for the Degree, of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

May 2017

ACKNOWLEDGEMENTS

I would like to thank Dr.Dereje Agonafer for giving me an opportunity to work in his lab at EMNSPC which really shaped my career. I thank him for his valuable guidance and support throughout this work. He has always been encouraging and motivating. He believed in me and boosted my confidence.

I would like to thank my lab mates Unique, Mugdha, Aniruddha, Navya, Pavan and Abel, working along you all has been one of the best experiences of my Master's Degree. Special thanks to Sally Thompson, who supported and helped me and my team.

I am thankful to all my friends, especially the ones back home in India, for being through thick and thin, for believing in my abilities and being there for me in every possible way.

Last but not the least; I would like to thank my father for supporting me during the entire duration of this work and my mother, who has been a pillar of support for me and who made me what I am today.

May 27, 2017

ABSTRACT

SOLDER JOINTS DESIGN OPTIMIZATION OF QFN PACKAGES FOR DIFFERENT BOARDS BY INVESTIGATING SOLDER JOINT RELIABILITY

Rishikesh Tendulkar
The University of Texas at Arlington, 2017

Supervising Professor : Dr. Dereje Agonafer

Quad Flat No-lead package (QFN) is one of the most cutting-edge technologies emerged in the market, exhibiting high performance and efficiency with unparalleled cost effectiveness. QFN, a leadless package, is an ideal choice for applications where size, weight thermal and electrical characteristics are critical, particularly in mobile and handheld devices. Applications like automotive, defense and high current circuits require the package to be mounted on thick printed circuit boards (PCB).

The motivation of this work is to understand the effect of design variations of solder joints affecting reliability and optimize the solder package to improve reliability for application on different PCBs. Initially, The material properties of the PCBs were determined using Thermal Mechanical Analyzer (TMA), Dynamic Mechanical Analyzer (DMA). The PCBs were then cross sectioned to find the layer composition and Copper content in the boards. Different designs of solder joints were made using ANSYS SpaceClaim. The quarter models of boards with the different solder designs were then subjected to Accelerated Thermal Cycling Tests, to study stresses generated on the solder joints and variation in Plastic work. A comparative study was performed on obtained results for different solder designs and on different PCB boards to propose the optimum solder design to reduce solder damage and increased life in QFN assemblies.

Table of Contents

LIST OF ILLUSTRATIONS

LIST OF TABLES

Chapter 1

INTRODUCTION

1.1 Quad Flat No-Lead (QFN) Packages

The QFN or a Quad Flat No-Lead package is a thermally enhanced standard size IC package designed to eliminate the use of bulky heat sinks and slugs. QFN is a leadless package where the electrical contact to the printed circuit board (PCB) is made through soldering of the lands underneath the package body rather than the traditional leads formed along the perimeter [1]. This package can be easily mounted using standard PCB assembly techniques and can be removed and replaced using standard repair procedures. The QFN package is designed such that the thermal pad (or lead frame die pad) is exposed to the bottom of the IC. This configuration provides an extremely low resistance path resulting in efficient conduction of heat between the die and the exterior of the package (see Figure 1-1).

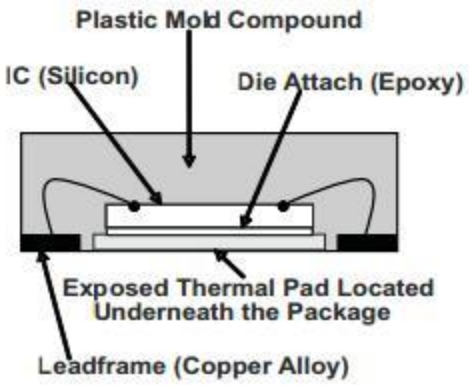


Figure 1-1 QFN PACKAGE CROSS SECTION

Due to its superior thermal and electrical characteristics, this device package has gained popularity in the industry during the last couple of years. Because of its compact size, QFN package is an ideal choice for handheld portable applications and where package performance is required. The QFN packages for this work were provided by Texas Instruments.

THICK PRINTED CIRCUIT BOARDS

A printed circuit board (PCB), typically consists of alternate layers of copper and non-conductive laminate made up of glass fiber and resin (FR4). PCBs are available in different types based on its application E.g. Single sided (one copper and one laminate layer), double sided (two copper layers and laminate layers), multi-layered boards. Multi layered boards are mainly used for high component density applications like defense and automotive. A thick printed circuit boards (PCB) consists of large number of Cu layers (>8) and large board thickness (>3 mm). Cross-section of a typical thick PCB with 16 copper layers, is shown in the figure.

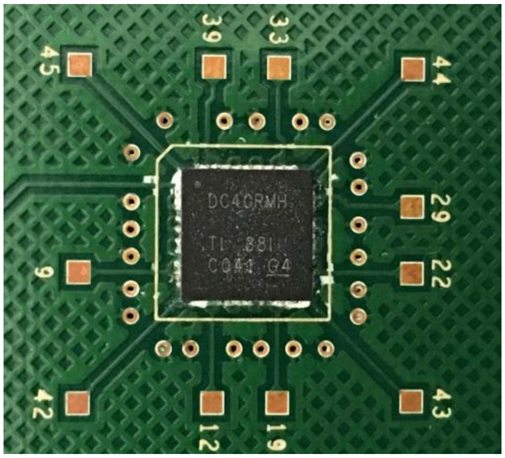


Figure 1-2 PCB Under Consideration

BASIC COMPONENTS OF AN ELECTRONIC PACKAGE

Components typically used in an integrated circuit are as following:

1. Substrate:

A substrate could be a solid substance onto which another substance is applied and it adheres to that. It is additionally known as a wafer. It is a foundation onto which elements like diodes, resistors, capacitors, etc. are mounted.[3]

Figure 1-3 Substrate

2. Die:

It is a block of semi-conducting material onto which a printed circuit is fabricated; such a material is called a die.

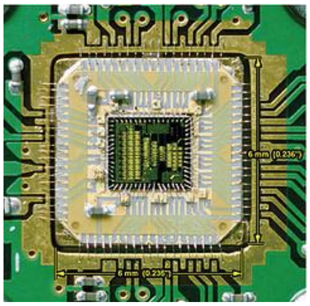


Figure 1-4 Die

3. Copper Layer:

The area on surface of the printed circuit board filled with Copper. Its primary function is to make the connections onto the PCB. A Copper pour also provides a ground plate and creates an electric net. It flows around the tracks and pads on the board, connecting all circuits.

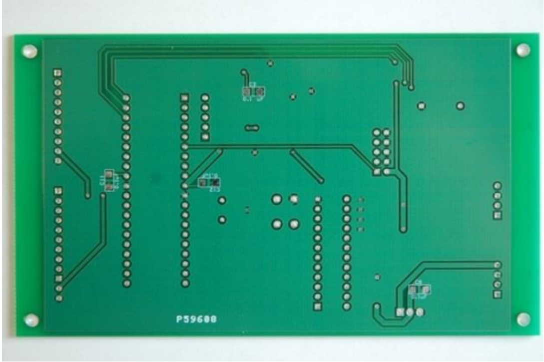


Figure 1-5 Copper Layer

4. Thermal Pad:

Thermal pad is a thermal relief pad which allows heat dissipation during operation of the circuit. It is connected to the copper pour by ‘thermal connection’ making it easier to solder and establish electric connections.

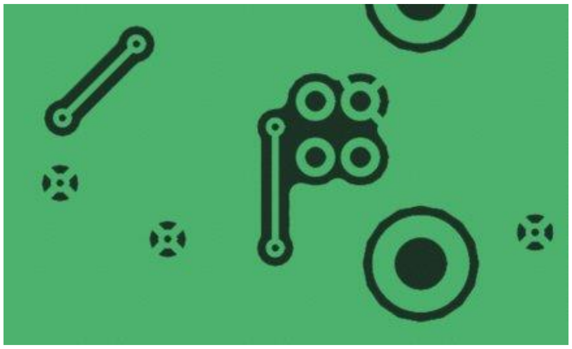


Figure 1-6 Thermal Pad

5. Lead Frame

Lead frame is a metal framework that connects the internal parts of the IC to the external circuit. It carries the signal from die to the other components of the PCB. The die is glued to the lead frame and the die pads are attached to the leads with the help of bond wires.

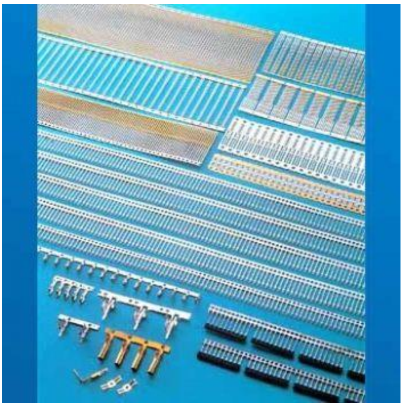
.


Figure 1-7 Lead Frame

6. Solder Mask

It is a thin layer of polymer which covers the Copper traces of PCB and further protects it all, against oxidation and prevents formation of short circuit because of mixing of individual solder joints called ‘Solder Bridge’.[3]

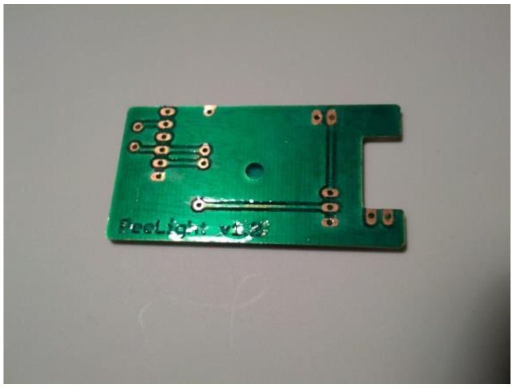


Figure 1-7 Solder Mask

BOARD LEVEL RELIABILITY (BLR)

Reliability can be expressed as a ability of a system or component to perform its required functions under stated conditions for a specified period. To quantify reliability, “ability” is usually interpreted as a “probability”. By virtue this definition, all the products will ultimately always fail. A possibility of zero failure during a certain amount of time is physically impossible, even for an integrated circuit [2].

There have been many precursors[8] to describe reliability and one of the most widely used is the failure rate. If a plot of failure rate versus time is depicted, a curve in the shape of a bathtub cross-section is obtained as shown in Figure. Therefore, it’s widely referred to as a bathtub curve.

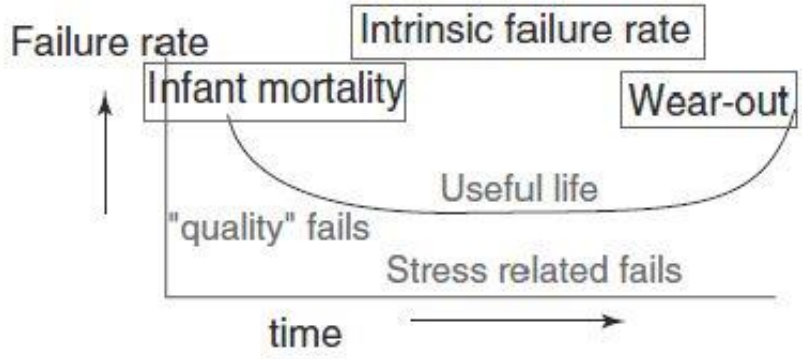


Figure 1-8 Bath-tub Curve

There are three phases of time which can be observed in a bathtub curve; infant mortality, intrinsic failure and wear out. Infant mortality or early failure is the period of time when the product fails due to defects in the fabrication or assembly of the product. The failure region in the middle has nearly a constant rate of failure since the poorly manufactured parts and defects are already screened out and eliminated and the majority of the population left are robust products where failures only occur randomly. In the end, as the product ages, chemical, mechanical, or electrical stresses begin to weaken the product’s performance and the product eventually fails. This region on the bathtub curve is called the wear-out region[4].

USE CONDITION	THERMAL EXCURSION (C)
Consumer electronics	0 to 60
Telecommunications	-40 to 85
Commercial Aircrafts	-55 to 95
Military Aircrafts	-55 to 125
Space	-40 to 85
Automotive – Passenger	-55 to 65
Automotive – Under the hood	-55 to 160

Table 1-1 Thermal environments for electronic products

The reliability tests are either focused on package level or board level. Package level or 1st level reliability tests are dedicated to the robustness of the package component materials and design to withstand extreme environmental conditions and does not consider the interconnects when it is mounted on board. Whereas for the board level or 2nd level reliability tests, stresses are examined on the solder joint of the surface mount package when mounted on boards.

Chapter 2

MATERIAL CHARACTERIZATION

To determine the fatigue correlation parameters from finite element analysis, we need actual material properties to use as inputs for each material. The important properties required for characterization of material properties are:

- Coefficient of Thermal Expansion (CTE)
- Young's Modulus.

The EMNSPC Reliability lab located in ELB 215 at The University of Texas at Arlignton has the machines required for performing the experiments to characterize the PCBs;

- Thermo Mechanical Analyzer (TMA)
- Dynamic Mechanical Analyzer (DMA)

The TMA was used to find the Coefficient of Thermal Expansion (CTE) in all the direction on x, y and z plane. The DMA is used to obtain the out of plane Young's Modulus.

We shall now define the Coefficient of Thermal Expansion and Young's Modulus.

Coefficient of Thermal Expansion (CTE):

The change in length or volume of a material for a unit change in temperature is defined as The Coefficient of Thermal Expansion (CTE). The overall coefficient is the linear thermal expansion per degree Fahrenheit or Celsius. It is calculated by the change in length divided by the quantity of length at room temperature, multiplied by the change in temperature.[4]

Generally it is given by-

—

Where,

α - Coefficient of Thermal Expansion (ppm/°C)
ϵ - Strain (mm/mm)
ΔT - Temperature Difference (°C)

Young’s Modulus (E):

Young’s Modulus is an elastic property of a material which can be defined as the ratio of stress to strain.

–

ϵ - Strain (mm/mm)
E - Young’s Modulus (N/mm^2)
σ - Stress (N/mm^2)

TMA measurements record changes caused by changes in the free volume of a. Experimentally, a TMA consists of an analytical train that allows precise measurements of position and can be calibrated against known standards. The sample is coupled to a temperature system with a furnace, heat sink and a temperature measuring device (Thermocouple). Fixtures to hold the sample during the test are commonly made of quartz due to its low CTE. The fixtures can be of expansion, three-point bending, parallel rate and penetration tests. TMA data determines the Coefficient of Thermal Expansion (CTE) and from the same set of data, Glass Transition Temperature (Tg) can be calculated. For an anisotropic material, the CTE will be different depending on which direction it has been measured.

Figure 2-1

For obtaining the CTE, TMA SS6000 was used and the sample of PCB was placed in the holder with a quartz probe. The orientation of the sample was changed after every reading to obtain CTE in all the 3 directions.

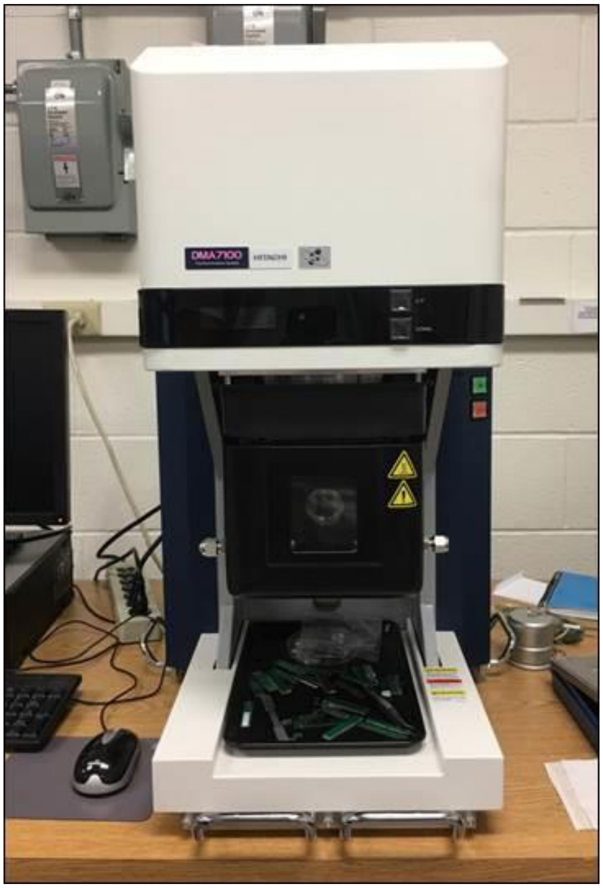


Figure 2-2

The DMA is generally used to find out loss and storage modulus, at the room temperature at 0 hertz. But, the Young's Modulus is equal to the storage modulus as the storage modulus is the elastic response of the sample and the loss modulus is the viscous response of the sample. Since the temperature applied is not beyond the room temperature, the viscous response can be neglected. Viscous response will be prominent after the Glass Transition Temperature (T_g).[14][15]

The values of Young’s Modulus and CTE obtained by the experiments are in table below. Coeffcient of Thermal expansion of the die , die attach, lead frame, were obtained from literature provided by Texas Instruments.for the provided boards.

Material	**CTE (ppm/°C)**	**E (GPa)**
Die	2.61	131
Die Attach	64	11.8
Lead Frame	17	129
Epoxy Mold	10.3	3
Epoxy die pad	17	129
PCB (62 mil)	α(x,y)=15.5 α(z)=39.4	15.37

Table 2-1 Material Characterization Results

Chapter 3

METHODOLOGY

RESEARCH OBJECTIVES AND PROPOSED STUDIES FOR SOLDER DESIGN

In the 1st study of this work: Constant Solder Volume Analysis

Study of keeping the solder volume constant and varying the solder angle with respect to the lead anchor was performed to gain better understanding of effect of solder angle when solder material bulk is does not vary. Different designs were made, keeping the area of triangle formed by solder as constant, models in SpaceClaim were made for the following dimensions.

In the 2nd study of this work: Angle Variation with varying solder volume

From the first study is was established that plastic work is lower when solder material is more on solder mask compared to lead anchor pin. Keeping the height of solder constant. Since, the solder cannot rise above lead anchor pin, which becomes a design constrain. The solder angle was increased and results were analyzed.

In the 3rd study of this work: Comparison of different solder profiles

The solder profile was varied, the same solder height and width were kept constant as per the dimensions in the standard model of solder package. Previous[8][9] studies implied that, increased solder material results in decreased plastic work. But, a parametric study was not performed. Hence, for comparative study, a concave, convex profile of varying radius of curvature was compared to the standard solder profile.

In the 4th study of this work: Effect of fillet on lead anchor side

Designs with fillets of solder introduced on lead anchor pin side of the solder joint were made on SpaceClaim and simulated, using imported load boundary conditions. The fillet was introduced along the z-axis and along x-axis of the solder package.

FINTIE ELEMENT MODELLING

Introduction to Finite Element Method

The Finite Element Method is a computational technique used to get a solution to boundary value problem in engineering. FEM is employed in almost all industries that could be imagined. Common application of FEA are:

- Aerospace/Mechanical/Civil/Automobile Engineering
- Structural Analysis (Static/Dynamic/Linear/Non-Linear)
- Thermal/Fluid Flow
- Electromagnetic Study
- Biomechanics
- Geomechanics
- Biomedical Engineering
- Hydraulics
- Smart Structures
-

The Finite Element Method is a very powerful numerical technique devised for solving differential equations of initial and boundary value problems in complex geometric regions.[11] Often it is difficult to obtain a solution of complicated geometry, loading condition and material properties. So, FEA is computational techniques required to obtain the desired results. The FEM problems can be in any of 1D, 2D or 3D space. In FEA the whole continuum is divided into several smaller elements of simple geometric shapes. Each of the geometric shapes intersections forms several different nodes. The displacement of these nodes and conditions on each of these nodes is solved and the result of the entire structure is obtained in this manner.

$$\{F\} = [K]\{u\}$$

Where, $\{F\}$ = Nodal load/force vector

$[K]$ = Global stiffness matrix

$\{u\}$ = Nodal displacement

Structure’s stiffness (K) depends on its geometry and material properties. Load (F) value has to be provided by user. The only unknown is displacement (u). The way in general FEA works is, it creates the number of small elements with each containing few nodes. The manner normally FEA works is, it creates the number of tiny components with every containing few nodes.[4] There are equations referred to as shape function in software package, that tells software the way to vary displacement (u) across the component and average value of displacement is determined at nodes. Those stress and/or displacement values are accessible at nodes which explains that finer the mesh elements, more accurate the nodal values would be. So, there are certain steps that we need to follow during the modeling and simulation in any commercial code to reach approximately true solution, which would be explained. In this study, commercially available FEA tool, ANSYS Workbench v17.2 has been leveraged.

FEA Problem Solving Steps

These five steps need to be carefully followed to reach satisfactory solution to FEA problem:

1) Geometry and Material definition
2) Defining Connection between bodies
3) Meshing the model
4) Defining load and boundary condition
5) Understanding and verifying the results

ANSYS is a general purpose FEA tool which is commercially available and can be used for wide range of engineering application. Before we start using ANSYS for FEA modeling and simulation, there are certain set of questions which need to be answered based on observation and engineering judgment. Questions may be like what is the objective of analysis? How to model entire physical system? How much details should be incorporated in system? How refine mesh should be in entire system or part of the whole system? To answer such questions computational expense must be compared to the level of accuracy of the results that needed[12]. After that ANSYS can be employed to work in an efficient way after considering the following:

- Type of problem
- Time dependence
- Nonlinearity
- Modeling simplification

From observation and engineering judgment, analysis type has to be decided. In this study the analysis type is structural; to be specific out of different other structural problem focus in this study is on Static analysis. Non-linear material and geometrical properties such as plasticity, contact, and creep are available.

PLASTIC WORK

Plastic work, additionally referred to as strain hardening or cold operating, is that the strengthening of a metal by plastic deformation. This strengthening happens as a result of dislocation movements and dislocation generation among the crystal structure of the fabric. Several non-brittle metals with a fairly high temperature further as many polymers may strengthen during this process. Before work hardening, the lattice of the fabric exhibits regular, nearly defect-free pattern (almost no dislocations). But, because the material is hardened it becomes progressively saturated with new dislocations, and several dislocations prevent it from nucleating (a resistance to dislocation-formation develops). This resistance to dislocation-formation manifests itself as a resistance to plastic deformation; thence, the ascertained strengthening. Such deformation will increase the concentration of dislocations which can after kind low-angle grain boundaries close sub-grains. A material's work hardenability is often foreseen by analyzing a stress-strain curve, or studied in context by performing hardness tests before and after.

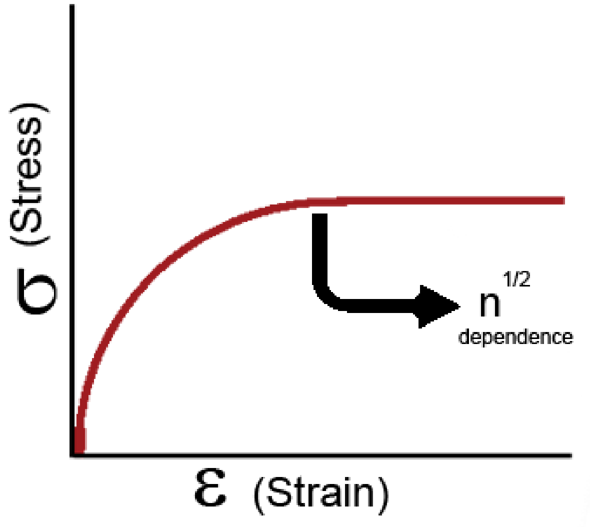


FIGURE 3-1 The yield stress of an ordered material has a half-root dependency on the number of dislocations present.

`The modified code used for solving for plastic work in the simulation is as following:`

```
!APDL SCRIPT TO CALCULATE PLASTIC WORK
/post1
allsel,all
!CALC AVG PLASTIC WORK FOR CYCLE1
set,4,last,1 !LOAD STEP
cmsel,s,solderball,elem !ELEMENT FOR VOL AVERGAING
etable,vo1table,volu
pretab,vo1table
etable,vse1table,nl,plwk !PLASTIC WORK
pretab,vse1table
smult,pw1table,vo1table,vse1table
ssum
*get,splwk,ssum,,item,pw1table
*get,svolu,ssum,,item,vo1table
pw1=splwk/svolu !AVERAGE PLASTIC WORK
!CALC AVG PLASTIC WORK FOR CYCLE2
set,8,last,1 !LOAD STEP
cmsel,s,solderball,elem
etable,vo2table,volu
pretab,vo2table
etable,vse2table,nl,plwk !PLASTIC WORK
pretab,vse2table
smult,pw2table,vo2table,vse2table
ssum
*get,splwk,ssum,,item,pw2table
*get,svolu,ssum,,item,vo2table
pw2=splwk/svolu !AVERAGE PLASTIC WORK
!CALC DELTA AVG PLASTIC WORK
pwa=pw2-pw1
!CALC AVG PLASTIC WORK FOR CYCLE3
set,12,last,1 !LOAD STEP
cmsel,s,solderball,elem
etable,vo3table,volu
pretab,vo3table
etable,vse3table,nl,plwk !PLASTIC WORK
pretab,vse3table
smult,pw3table,vo3table,vse3table
ssum
*get,splwk,ssum,,item,pw3table
*get,svolu,ssum,,item,vo3table
pw3=splwk/svolu !AVERAGE PLASTIC WORK
!CALC DELTA AVG PLASTIC WORK
pwb=pw3-pw2
my_pwb=pwb
my_pwa=pwa
my_pw1=pw1
my_pw2=pw2
my_pw3=pw3
```

We get the plastic work at the end of third cycle and the difference of plastic work between third and second cycles.[10]

MESH SENSITIVITY ANALYSIS

Finite element preprocessors have really evolved over the years, to the point where users with minimal training can create meshes that appear good. But, how to really know if the mesh is good enough for the analysis?[13] Meshes that are good enough, are the ones that produce results with an acceptable level of accuracy, if all other inputs to the model are accurate. Mesh density is a significant metric used to control accuracy. Assuming no singularities are present, a high-density mesh produces results with higher accuracy. However, if a mesh is too dense, it requires a large amount of computer memory and longer run times, especially for multi-iteration runs that are typical of nonlinear and transient analyses. One of the most fundamental and accurate method for evaluating mesh quality is to refine the mesh until a critical result, such as the maximum stress in a specific location converges that is, to the point that it doesn’t change significantly with each refinement. This forms the basis of mesh sensitive analysis. As, in this study different models of solders are compared and analyzed, mesh sensitive analysis is plays important part in ensuring that the results and variation in plastic work obtained between different models is down to change in geometry and not because of irregular mesh, or because of mesh dependent result variation.

No. of Elements	Plastic work (3-2)	Pw 3	Mesh Size
60	8.12E+05	3.40E+06	100
94	8.01E+05	3.33E+06	80
111	7.93E+05	3.24E+06	60
447	7.66E+05	3.12E+06	40
1985	7.24E+05	3.00E+06	20
4009	7.21E+05	3.64E+06	15

Table 3-1 Mesh Sensitivity Analysis

Analysis is first performed with certain number of elements and then with twice the elements. Then both the solutions are compared, if solutions are close enough then initial mesh configuration is considered to be adequate. If solutions are different than each other than more mesh refinement and subsequent comparison is done until the convergence is achieved. There are different types of mesh elements for 2D and 3D analysis in ANSYS which can be used based on application. For this study the mesh used were hex-dominant, combined with mesh sizing.

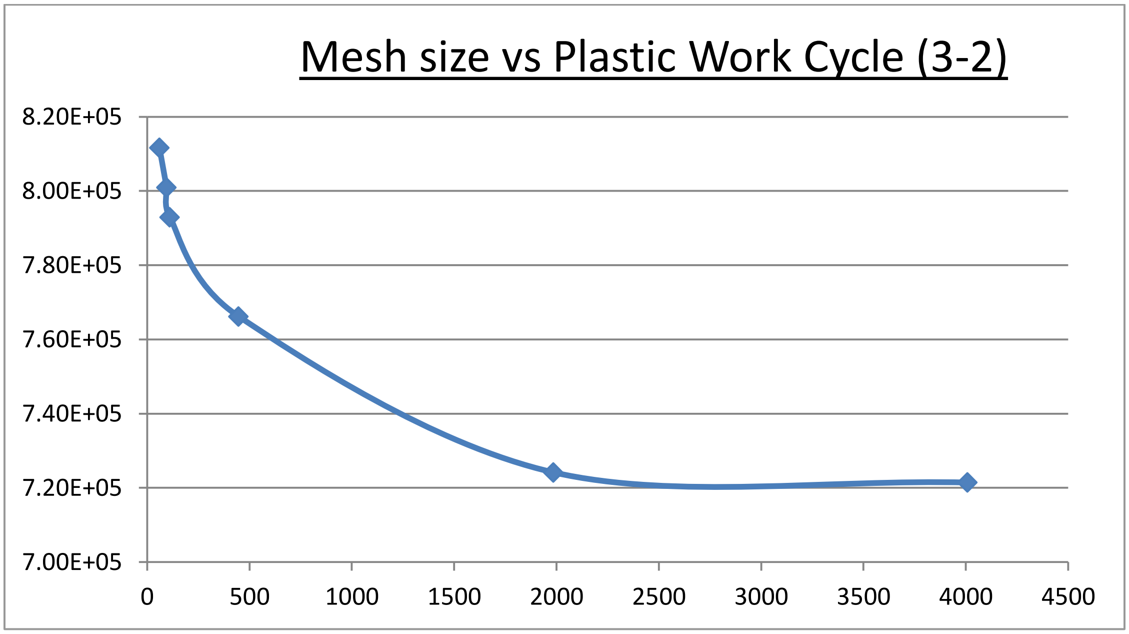


Figure 3-2 Mesh Size vs Plastic Work

The mesh size was reduced from 100 micrometers to 15 micrometers, finally selecting 20 micrometers as optimum mesh size for all the solder models, to obtain consistent, mesh independent results and a faster solving time and better memory requirement.

LOADS AND BOUNDARY CONDITIONS

In the simulations, a quarter model of the package was used. Due to the package being symmetric geometrically and since, the boundary conditions are also symmetric for the package. Quarter model can be used for better simulation times and saving the memory. In the quarter model of the package considered, the loads and boundary conditions which were applied to the global model can be seen in the figure. Symmetric boundary conditions were applied to the boundary planes of the quarter symmetry model. The center node was fixed to prevent rigid body motion.

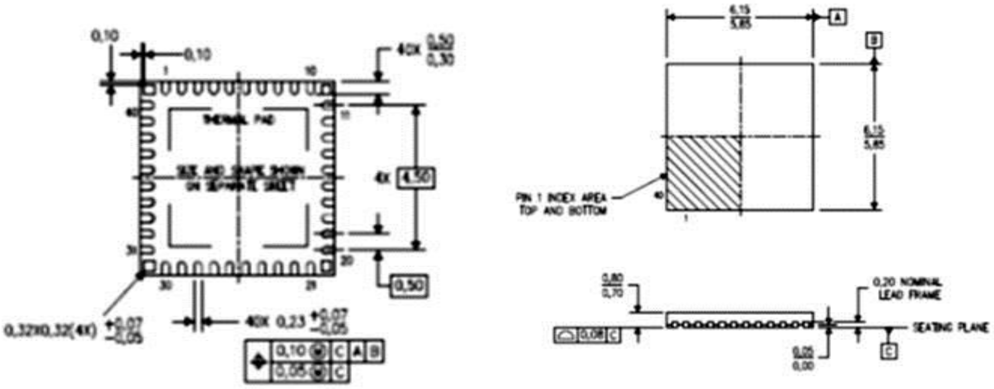


Figure 3-3 QFN package in 2D

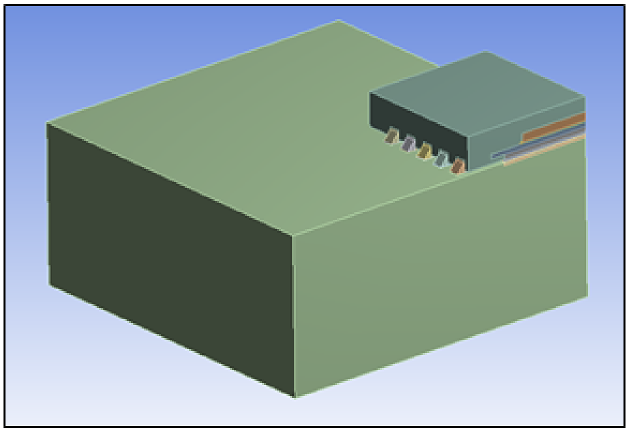
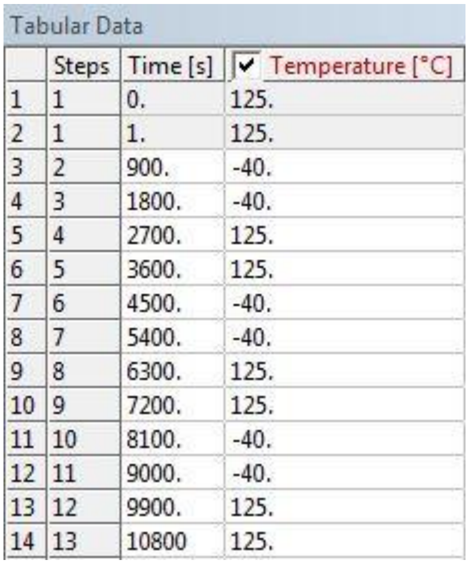


Figure 3-4 Quarter symmetry QFN Package

Accelerated Thermal Cycling

Thermal cycling load of from -40 degree centigrade to 125 degree centigrade were applied from 15 min ramp/dwell were applied on the model as shown in the figure. Simulations are run for three complete cycles since most of the solder joints reach a stable state after the end of the third cycle. The initial stress-free temperature was set to be maximum temperature in the thermal cycle.[4][5] Choosing a high dwell temperature as the stress-free temperature helps the system reach stabilized state much faster.



Tabular Data

	Steps	Time [s]	☑ Temperature [°C]
1	1	0.	125.
2	1	1.	125.
3	2	900.	-40.
4	3	1800.	-40.
5	4	2700.	125.
6	5	3600.	125.
7	6	4500.	-40.
8	7	5400.	-40.
9	8	6300.	125.
10	9	7200.	125.
11	10	8100.	-40.
12	11	9000.	-40.
13	12	9900.	125.
14	13	10800	125.

Figure 3-5 Tabular data of the thermal cycling temperatures and time

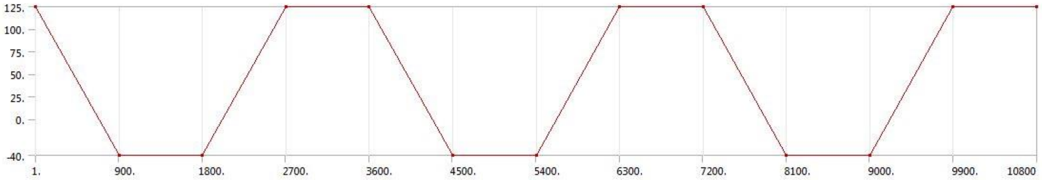


Figure 3-6 Graphs showing the accelerated thermal cycling in simulations

Chapter 4

MODELLING AND SIMULATION

Study I

In the first study, The design constraint considered was to check for constant volume of SAC. The Tin-Silver-Copper is widely used solder material. The solder height in the model of Texas Instruments model was 0.2 mm. The width was also around 0.2mm for the standard model. The solder forms a triangle between the lead solder anchor edge and the solder mask side. The triangle formed by the sides is of area half * 0.2 * 0.2 will become 0.02 sq.mm. The edge of the solder is 0.23 mm. The total solder volume become 0.0046 cu.mm. The design was made in a manner that the solder volume remains constant for the first study.

The possible dimensions for achieving the results were. By increasing height to 0.25mm and reducing the base width to 0.16mm. By these new dimensions we get the triangle area same as the original at 0.02 sq.mm. Maintaining the total SAC volume at 0.0046 cu.mm. The solder joint sub model was made in ANSYS SPACECLAIM and then imported to ANSYS 17.2 where cut load import boundary conditions in thermal conditions were used for simulation. The mesh sensitive analysis performed previously was employed and mesh edge side was maintained for all models.

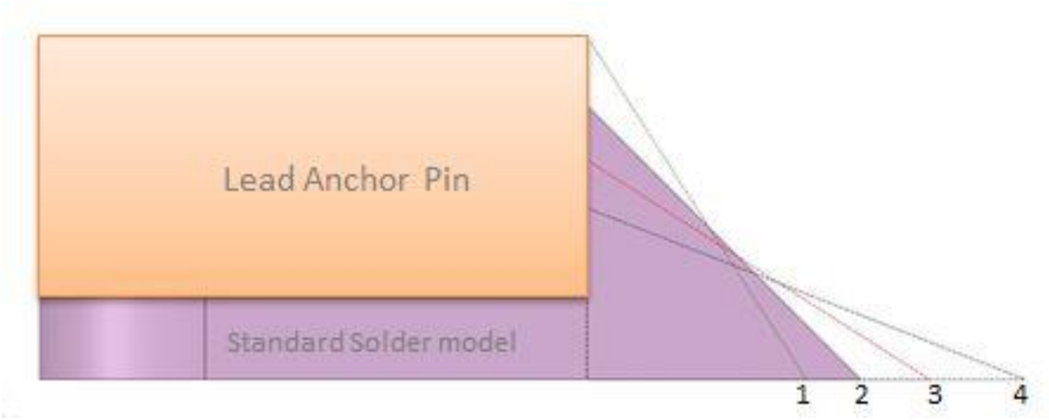


Figure 4-1 Profiles of different designs of solder

For the second model of constant solder volume study, the solder height was reduced from 0.2mm to 0.16mm and the solder base width was increased from 0.2 to 0.25mm. It could be seen as mirror image of the first design variation. The motivation was to study the change in effect of reduction of plastic work, when the solder volume is more on the lead solder anchor of the solder mask side, the total solder material will then be considered for the material of the solder joint in the solder model and the total solder. The plastic work comparison gives a better idea of where the presence of SAC has more significant effect on the plastic work. Since in the second design variation the solder volume is the same as first. The model is made on SpaceClaim and then solved in workbench. The obtained delta plastic work is tabulated and compared.

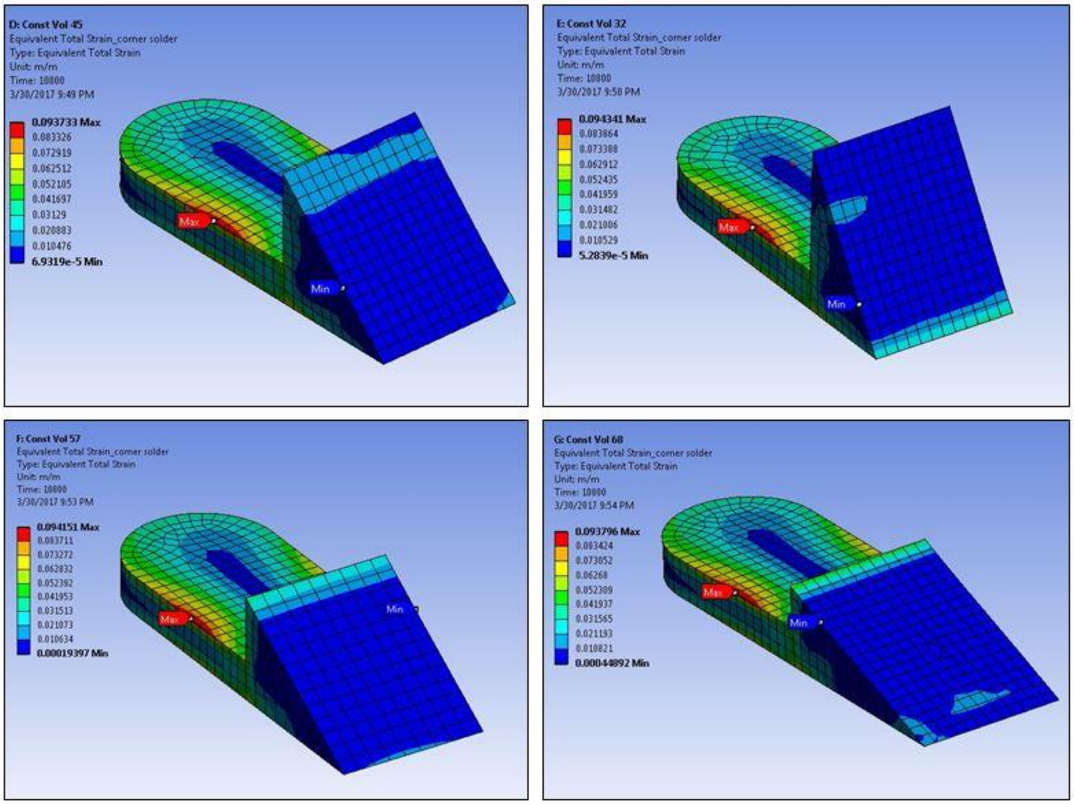


Figure 4-2 Profiles and Strain Contours

For the third model the solder height is reduced to 0.125mm and the base width increased to 0.32mm for purpose of understanding effect of reduced solder angle and how does it affect plastic work. The other design where the solder height is increased and width reduced is not performed as the designs constrain is considered for the maximum solder height. The solder height cannot be more than the lead anchor pin height. The third design dimensions of 0.125 * 0.32 also maintain the same 0.02 sq.mm. Allowing studying the effect of same volume of SAC of 0.0046 cu.mm. The results are than compared to that of the standard solder height and width and then tabulated and compared.

Analysis

By comparing all the obtained plastic work curves, it was seen that the plastic work is minimum when the solder angle is maintained around 45 degrees. When the solder on lead anchor pin side and the base is maintained to be of same dimension we see least amount of plastic work in the 3rd cycle. When the amount of solder is more on the lead anchor pin side than the base width the plastic work is seen to be the same at 7.35E+05 and 7.36E+05. It can be established that maintaining the solder angle at 45 degrees gives up minimum plastic work thereby, maximum life. The model from Texas instruments holds true and maintains the solder angle to near optimum and reduces the plastic work to minimum possible.

Design	Plastic Work (3-2)	PWork (3)	Max Strain
1	7.35E+05	3.00E+06	9.4341e-002 m/m
2	7.24E+05	3.00E+06	9.3733e-002 m/m
3	7.36E+05	2.99E+06	9.4151e-002 m/m
4	7.38E+05	2.99E+06	9.3796e-002 m/m

Table 4-1 Plastic work of solder models; first study

The solder angle vs Plastic work graph obtained, shows the slight reduction in plastic work at 45 degrees, where the solder height is equal to the solder base width. As, the angle increases, the plastic work also continues to increase.

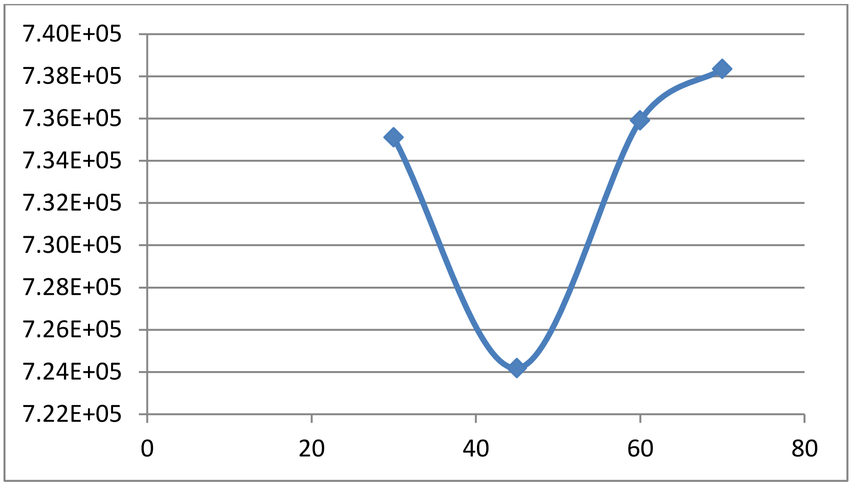


Figure 4-3 Solder angle vs Plastic work graph

Study II

In the second study, the solder angle formed between lead anchor pin side and the solder slope is varied and the solder height is maintained constant. The solder height is selected to be the same as the standard model and the solder height is maintained at 0.2mm. The design constrains being that the maximum solder height is the equal to the lead solder pin and the height being kept same as the one in the standard model to compare the results.

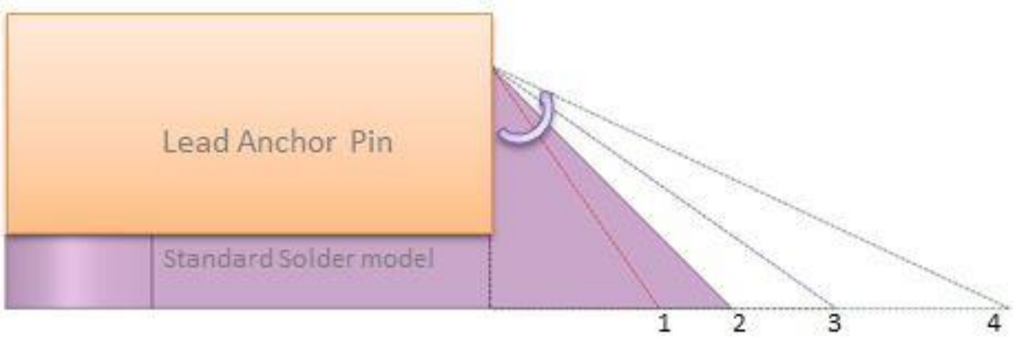


Figure 4-4 Solder profiles at varying angles

The angles considered are at interval of 10 degrees. The standard model has angle of 45. So, the angles considered are 35, 45, 55, 65. As the solder angle increases, the solder volume also increase with the increased area of the triangle formed by lead solder anchor pin, the solder mask and the solder slope. The 4 sub models were made in SpaceClaim and the models were then imported to workbench for performing accelerated thermal cycle tests under between -40 to 125 degrees in three cycles of 900 seconds for the analysis. The results were compared for delta plastic work between third and second cycle.

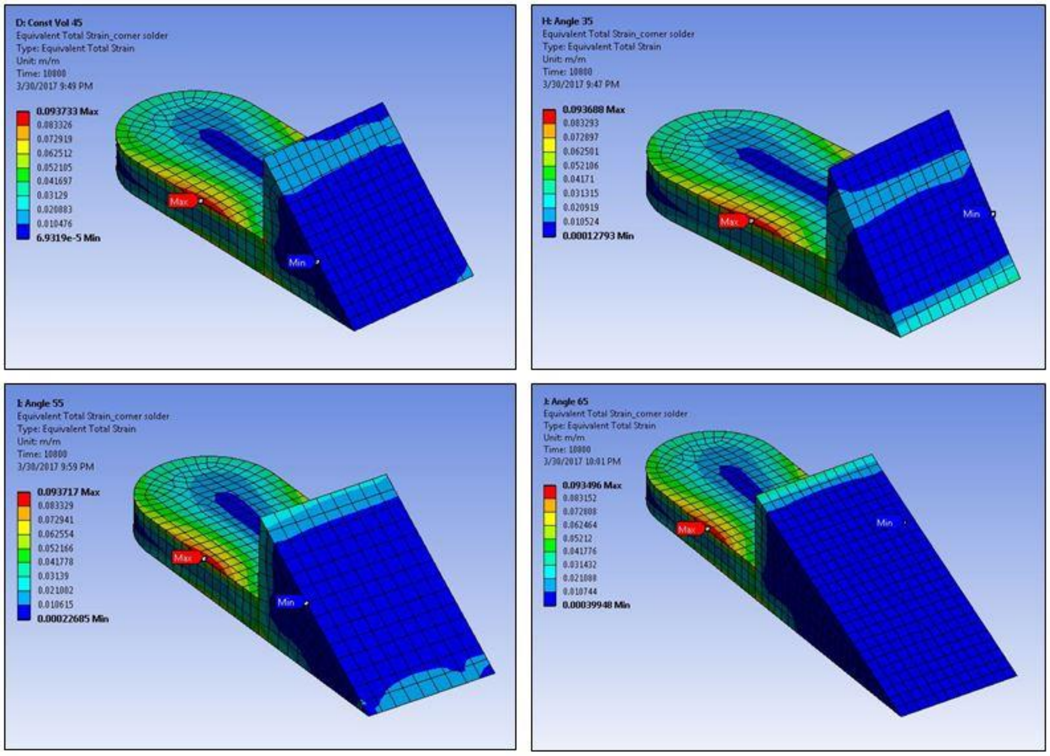


Figure 4-5 Profiles and Strain Contours

Analysis

The values were tabulated and compared to the original standard model. As previous studies suggested, the increase in the solder volume of SAC reduces plastic work. It is observed that as the angle between the solder profile and lead anchor increases, the volume of solder material also increases. The solder mass increasing in this study is when the solder height remains constant. The design constrain for the lead solder anchor side being the same height as the standard solder model.

θ	Plastic Work (3-2)	Pwork 3	Max Strain
35	8.11E+05	3.31E+06	9.3688e-002 m/m
45	7.24E+05	3.00E+06	9.3733e-002 m/m
55	6.56E+05	2.66E+06	9.3717e-002 m/m
65	5.67E+05	2.27E+06	9.3496e-002 m/m

Table 4-2 Plastic work vs Solder angle

The design constraint considered to increase the angle along the solder mask was maintained to a maximum of 65 degrees as the purpose of SAC is to solder the lead anchor pin onto the board and having more solder material does not serve the purpose of soldering the board. But, acts as redundant material on the board. Therefore, the angles were compared and we see a very significant dip in the plastic work from 8.11 to 5.67 (e+05). It can be established that when solder volume goes on increasing the plastic work reduces to the point of design constraint.

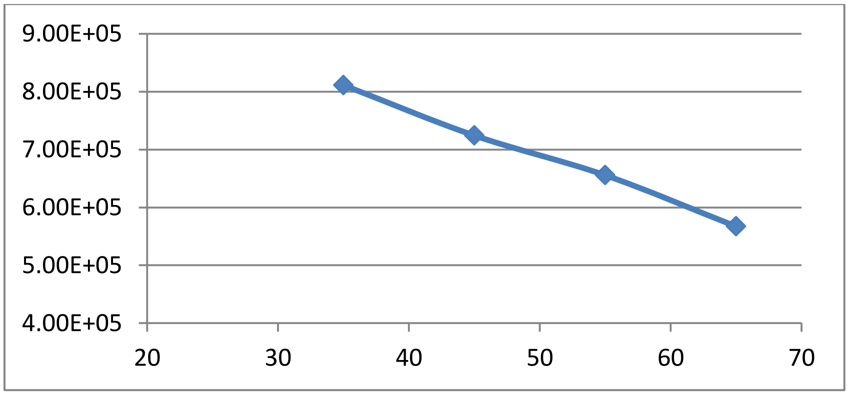


Figure 4-6 Plastic work vs solder angle

Study III

For third study, geometric variations are made on the profile of the solder. The solder height and solder base width is maintained same as the standard model. Bulbous and shrunk profiles are created on the solder profile, with radius of curvature varying from negative to positive values to study the effect to increasing solder mass of SAC on the solder joint reliability. Of the 4 models created for this study, first one is a shrunk profile with curvature of negative 25 micrometer arc. The second model is the original standard model being used throughout the work to form basis of comparative study. The third model created on SpaceClaim is a bulbous profile with radius of arc as the first model but in positive direction hence 25 micrometers. The fourth model, the arc is equal to the height of the lead anchor pin and is equal to the height of lead solder at lead anchor pin side, of 20 micrometers. The models created on SpaceClaim were then imported to ANSYS and simulations were performed with import cut boundary conditions The mesh size used was the same for all the models as the mesh size determined from mesh sensitive analysis.

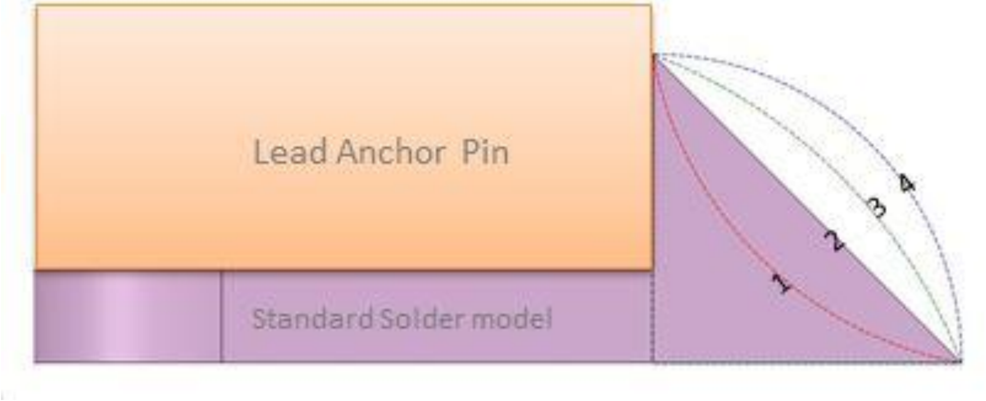


Figure 4-7 Profiles of varying solder mass

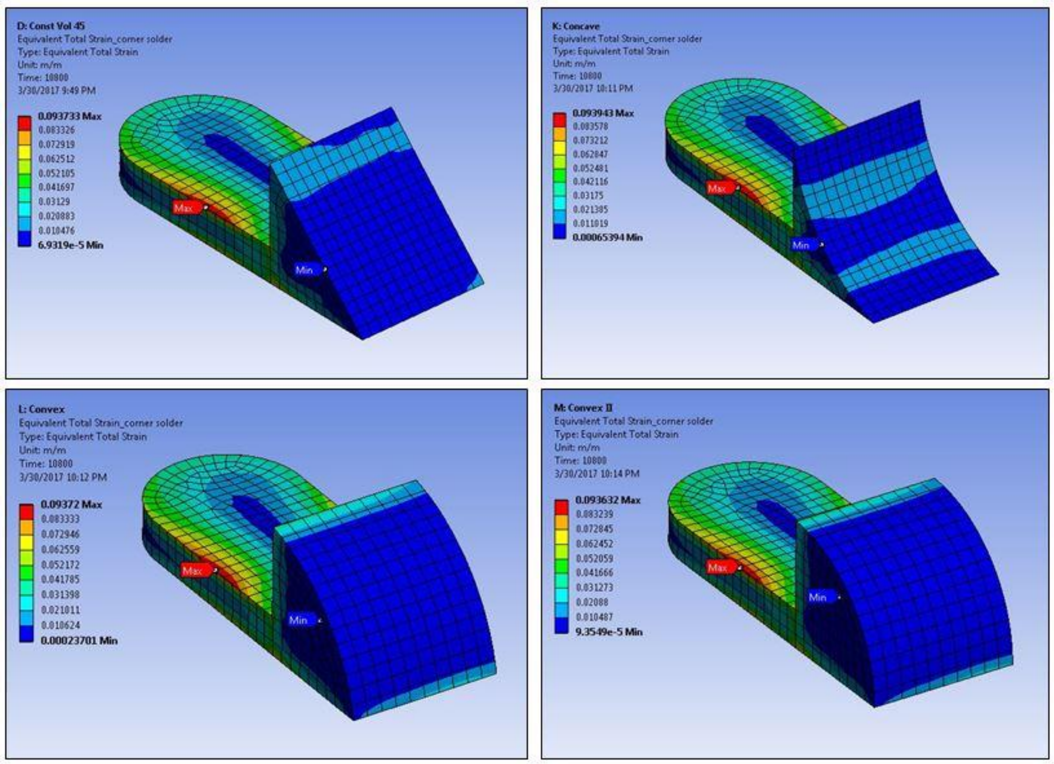


Figure 4-8 Profiles and Contours

Analysis

The data table with plastic work between third and second cycle and the plastic work at the end of third cycle was generated for all the models. A significant reduction in plastic work is observed as the solder mass keeps on increasing from shrunk to bulbous profile. The reduction in plastic work continues up until the solder profile’s curvature is becomes same as the height on the lead solder anchor side. Then, an upward trend in the curve is observed, it can be inferred that the reduction in plastic work due to added solder mass only continues up to a certain value and beyond it the SAC material has no effect on the plastic work generated by the solder joint.

Design	Plastic Work (3-2)	Plastic Work 3	Max Strain
Concave 25	8.66E+05	3.52E+06	9.3943e-002 m/m
Flat 45	7.24E+05	3.00E+06	9.3733e-002 m/m
Convex 25	6.20E+05	2.54E+06	9.372e-002 m/m
Convex 20	5.88E+05	2.41E+06	9.3632e-002 m/m

Table 4-3 Solder profiles and Plastic work

The graph plotted by the simulated data, shows the reduction in the plastic work with increased solder mass. It is also observed that increase in solder volume as the radius of bulbous mass reaches near height of solder, it begins to even out and become constant.

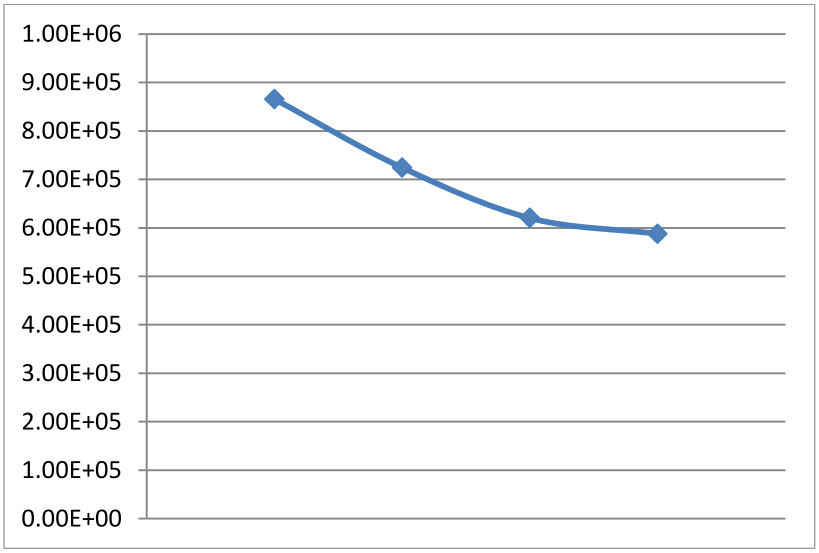


Figure 4-9 Graph of Plastic work vs solder mass

Study IV

For the fourth study, parametric analysis of effect of introducing fillet along x and y axis of the solder fillet was performed. Designs with fillets of solder introduced on lead anchor pin side of the solder joint were made on SpaceClaim and simulated in ANSYS workbench, using imported load boundary conditions. The motivation behind introduction of fillet was to try and increase the SAC material to get better reliability by performing simple design changes to lead anchor pin design.

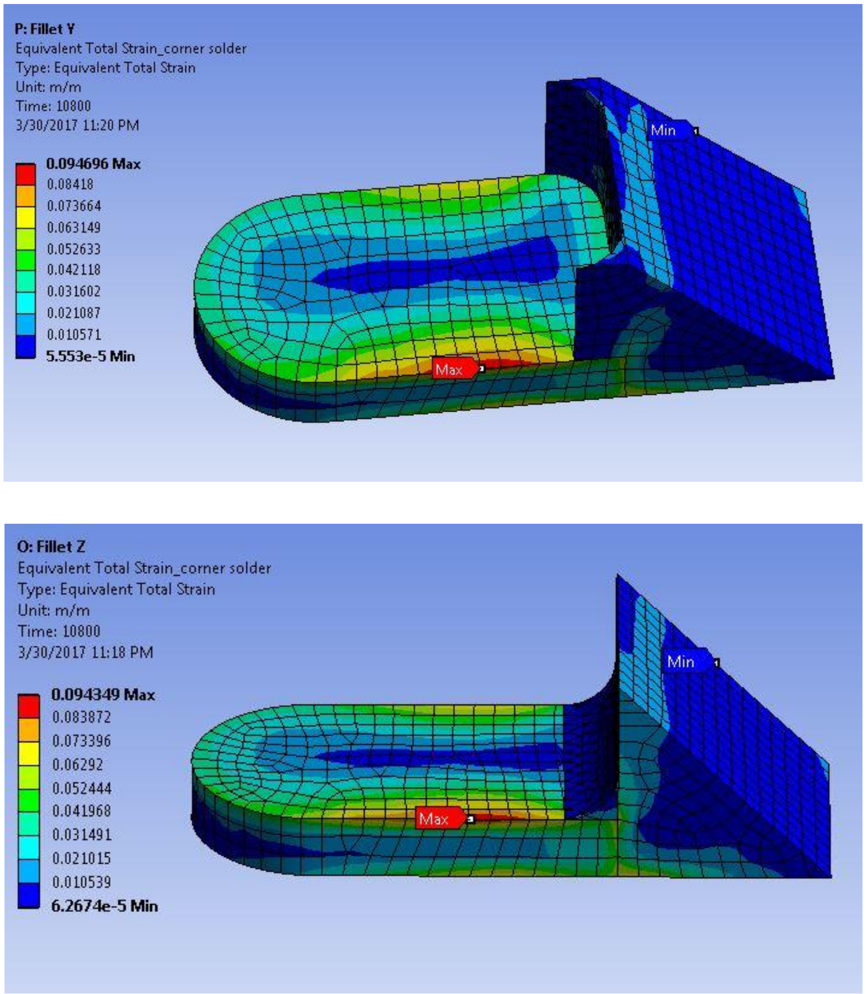


Figure 4-10 Solder fillets on x and y axis

Design	Plastic Work (3-2)	Pwork 3
Fillet (Z) 0.5	7.24E+05	2.95E+06
Fillet (Y) 0.5	7.30E+05	2.98E+06

Table 4-4 Solder Fillet and corresponding Plastic work

The results of introducing the fillet on x and y axis were tabulated. A design with solder fillet of increased fillet radius equal to the solder height was also made and simulated in workbench.

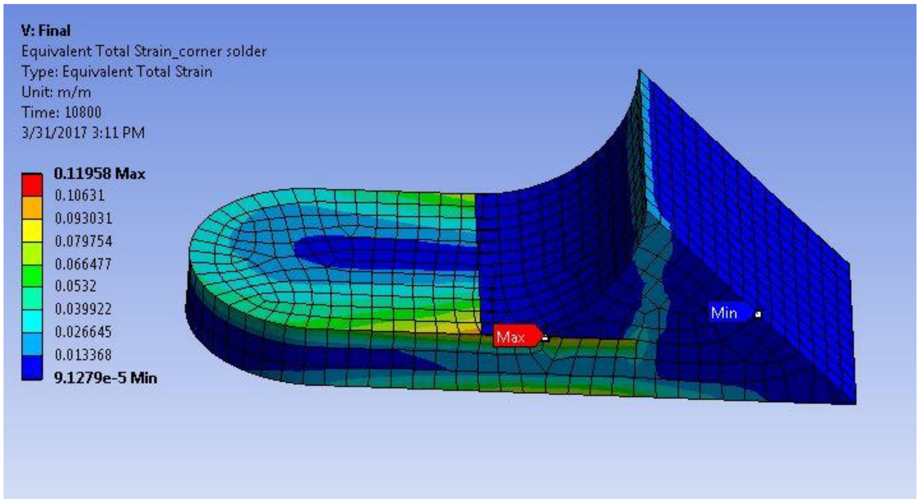


Figure 4-11 Solder Fillet

The design with solder fillet along the edge of the qfn package, as the QFN is a flipcihp and the chip in turned upside down before being soldered. The fillet along the edge of package will considered more plausible to manufacturing. It would require a much simpler change in existing manufacturing of the QFN and hence is a better design improvement choice.

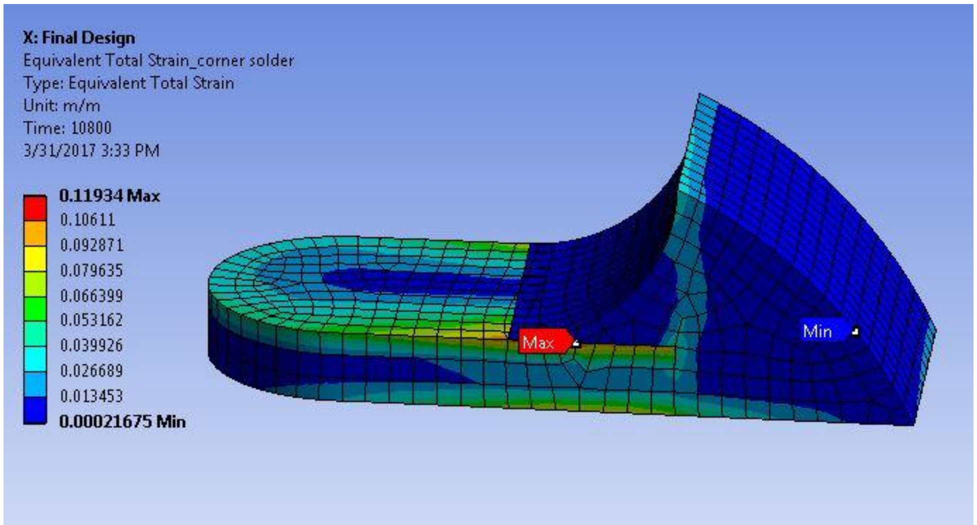
FINAL DESIGN

The final optimized design of the solder joint was made incorporating all the previous studies performed. A solder design with fillet along the edge was designed. The radius of the fillet was kept same as the height of the solder fillet on the lead anchor side of the package. The packages are soldered using hot air gun. If a correct angle of the solder gun is maintained and the solder angle is maintained near 45 degree, a bulbous fillet of SAC material is introduced. The proposed optimized solder fillet can be made.

Design Parameters (mm)	
Solder dimensions	0.2x0.23
Solder height	0.20
Fillet height	0.14
Fillet radius	0.14
Curve radius	0.20

Table 4-5 Optimized Design Parameters

The introduction of the fillet, would provide a cavity for the solder material SAC to fill in. Thereby, more solder Sn-Ag-Cu can be added, resulting in reduced plastic work and significantly improving the reliability.



X: Final Design
Equivalent Total Strain_corner solder
Type: Equivalent Total Strain
Unit: m/m
Time: 10800
3/31/2017 3:33 PM

Figure 5-1 Final Solder Design

The model was made in SpaceClaim. The solder was based on standard model. It has same height and base width as 0.2, 0.2 micrometers. What changes, is the fillet on lead anchor pin. A fillet of height and radius 0.14 micrometer. The radius is equal to height of solder above the solder base height 0.6. The angle between solder height and base is also 45 degrees. A bulbous mass is introduced on it, of 0.20 micrometer radius of arc. The depth of solder is 0.23 micrometers. The plastic work generated by this design was 6.22E+05. A reduction of 1.02E+05 or 14% is observed in plastic work. The optimized solder model helps in decreasing the plastic work, thereby increasing the reliability of the package.

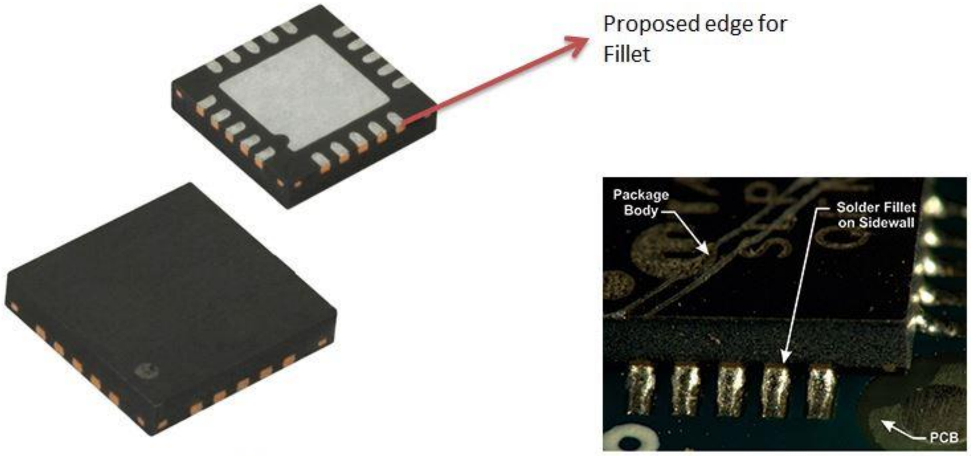


Image source : ieeexplore.org

Image source : Allegro Microsystems

Figure 5-2 Proposed Solder Edge

The figure above shows the proposed change on a QFN package. The edge of the package where the fillet needs to be introduced in order to carry out the design changes proposed in this work. Work can be done into looking at manufacturing processes for performing the necessary changes, on providing the necessary fillet on the redundant part of extra lead anchor pin and plastic mold which covers the package,

CONCLUSION

- Thick Printed Circuit Board was successfully characterized using Thermo-Mechanical Analyzer, to find Young’s Modulus and Coefficient of Thermal Expansion, which are later used in FEA analysis.
- Parametric analysis was performed to determine design parameters critically affecting QFN reliability on this board.
- Analysis of effect of solder angle on the plastic work was performed. Effect of change of volume by keeping solder height constant and varying the solder angle was also performed on the package.
- Study of maintaining correct solder angle when soldering the QFN packages to boards was performed.
- Final design change was proposed, altering the lead anchor pin by introducing a filler on the edge, 14% improvement in reliability was achieved by the solder design change.

FUTURE WORK

The future work, open for research to continue of this work:

- Similar study and Design optimization can be performed for different boards of different thickness.
- Multi Design Variable Optimization can be performed using ANSYS tools' using the design constrains discussed in thesis, as precursors for the work.
- Other parameters such as analyzing the effect of altering the layout of lead solders on a QFN package and study how it affects the plastic work.
- Varying the lead solder pins on x or y axis and altering the design of critical solder on the corners of QFN package can mitigate the package failure on PCBs. This is a very important aspect yet to be established and studied.

REFERENCES

[1] Y. B. Quek, "QFN Layout Guidelines," Texas Instruments, Dallas, 2006.

[2] G. Q. Zhang, Mechanics of Microelectronics, XIV ed., Springer, 2006.

[3] Sayalee Sabne "Impact of Layer Thickness of QFN Package mounted Printed Circuit Boards During Drop Testing", Ms-Thesis, The University of Texas at Arlington.

[4] Abhishek Deshpande "Comprehensive Design Analysis Of Thick Fr-4 QFN Assemblies For Enhanced Board Level Reliability" Ms-Thesis, The University of Texas at Arlington.

[5] A. Syed and W. Kang, "Board Level Assembly and Reliability Consideration for QFN Typer Packages," Chandler, AZ, 2003.

[6] T. Y. Tee, H. S. Ng, D. Yap and Z. Zhong, "Comprehensive board-level solder joint reliability modeling and testing of QFN and PowerQFN packages," *Microelectronics Reliability,* vol. 43, no. 8, 2003.

[7] Wu Zhaohua,"Optimal Design and Fatigue Life Prediction for QFN Solder Joints by BP Artificial Neural Networks and Genetic Algorithm", School of Mechanical & Electrical Engineering Guilin University of Electronic Technology.

[8] A R Nazmus Sakib "Enhancement of ThermoMechanical and Impact Reliability of Passive and Active Microelectronic Devices" PhD Dissertation, The University of Texas at Arlington.

[9] Simulation Model Development for Solder Joint Reliability for High Performance FBGA Assemblies, CALCE Electronic Products and Systems Center at University of Maryland, College Park.

[10] Robert Darveaux, "Effect of Joint Size and Pad Metallization on Solder Mechanical Properties", Amkor Technology, Inc.

[11] J. N. Reddy, An Introduction to the Finite Element Method, 3 ed., McGraw-Hill, 2005.

[12] T. Raman, "Assessment of the mechanical Integrity of Cu/Low-K dielectric in a Flip Chip package," UT Arlington, Arlington, 2012.

[13] "ANSYS Documentation, Release 15.0," 2014. [Online]. Available: http://www.ansys.com.

[14] Sumanth Krishnamurthy Experimental And Computational Board Level Reliability Assessment, The University Of Texas At Arlington Of Thick Board QFN Assemblies Under Power Cycling

[15] Basics Of Thermomechanical Analysis With TMA 4000, Kevin Menard, Bruce Casel, Perkinelmer, Inc., Shelton, Ct Usa

Biographical Information

Rishikesh Tendulkar was born in New Delhi, India. He received his Bachelor of Engineering degree from the University of Mumbai, India in 2014. He enrolled into Master of Science in Mechanical Engineering program at the University of Texas at Arlington in Fall 2015. He has been working in the Reliability Engineering Laboratory under Dr. Dereje Agonafer in Electronics, MEMS and Nano-electronics Systems Packaging Center. (EMNSPC). He has been working with QFN electronic packages and their thermal and mechanical reliability. He has also worked on DMA, TMA, and INSTRON machines for material characterization. He is proficient in ANSYS Workbench, ANSYS Icepack, Solidworks and Auto-Cad. He is involved in the live project collaborated with Texas Instruments. He graduated from The University of Texas at Arlington with Masters Degree in Mechanical Engineering in May 2017.