UNIVERSITI TEKNIKAL MALAYSIA MELAKA

FATIGUE LIFE SIMULATION OF SPOT WELD JOINT STRUCTURE BASED ON FINITE ELEMENT ANALYSIS

This report submitted in accordance with requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Engineering Technology (Manufacturing Engineering Technology) (Hons.) (Product Design)

by

MUHAMMAD FAIZ BIN CHE MUSA
B071310451

FACULTY OF ENGINEERING TECHNOLOGY
2016

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Fatigue Life Simulation on Spot Weld Joint Structure Based On Finite Element Analysis**

SESI PENGAJIAN: **2015/16 Semester 2**

Saya **MUHAMMAD FAIZ BIN CHE MUSA**

Mengaku membenarkan Laporan PSM ini disimpan di PerpustakaanUniversiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐ SULIT (Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☐ TIDAK TERHAD

Disahkan oleh:

_______________________ _______________________

AlamatTetap:

NO 2, JALAN MELOR 1/2

SG BULOH COUNTRY RESORT

47000 SG BULOH, SELANGOR

Cop Rasmi:

Tarikh: _______________________

DECLARATION

I hereby, declared this report entitled "Fatigue Life Simulation Of Spot Weld Joint Structure Based On Finite Element Analysis " is the results of my own research except as cited in references.

Signature : ………………………………………….

Author's Name : …………………………………………

Date : …………………………………………

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering Technology (Product Design) with Honours. The member of the supervisory is as follow:

………………………………

(Project Supervisor)

ABSTRAK

Projek ini bertujuan untuk menjalankan analisis unsur-unsur pada model tempat struktur kimpal untuk meramal dan menentukan hayat lesu dengan menggunakan perisian Solidworks. Bahan yang digunakan adalah besi tergalvani dan aluminium. Dua plat besi Tergalvani dan aluminium akan dianalisis dalam perisian Solidworks bertindak sebagai dua plat yang digunakan pada tempat kimpalan rintangan. Kajian mengenai hayat lesu pada bahan yang digunakan akan dijalankan.

Bahan-bahan yang akan dianalisis mempunyai ketebalan yang berbeza. Beban ini akan digunakan untuk bahan bagi menentukan apakah had bahan boleh bertahan sebelum ianya lesu. Data daripada masa kitaran dan beban bahan yang akan dikumpulkan dan ianya akan dianalisis bagi menentukan kaedah yang berkesan untuk mengurangkan kelesuan bahan. Hayat lesu akan diteliti melalui analisis unsur-unsur yang terdiri daripada plot kontur dalam proses simulasi. Nilai yang menyebabkan kelesuan bahan boleh ditentukan daripada plot kontur.

Hasil ramalan akan dibandingkan dengan data eksperimen dalam mencari kaedah prestasi dan analisis yang terbaik. Daripada data yang telah dikumpulkan, ianya akan menunjukkan masa kitaran dan loading yang sesuai untuk bahan-bahan bagi mencegah kegagalan lesu di tempat kimpalan rintangan.

ABSTRACT

This project was aimed to run finite element analysis on spot welded joint model to predict and determine fatigue life for spot welded by using Solidworks software. The material that is using is the galvanized iron and aluminum. Two plates of galvanized iron and aluminum will be analyzed in the Solidworks software reacted as two plates that are using by spot welding. There will be studies about fatigue life on the material that are used.

The materials that will be analyzed have different thicknesses. The load will be applied to the material to determine what are the limitation of the material can hold before it fatigue. The data from cycle time and loading of the material will be collected and it will be analyzed which are the effective method to reduce the fatigue of the material. Fatigue life will be observed through the finite element analysis which consists of the contour plot in the simulation process. The value that causes the fatigue of the material can be determined from the contour plot.

The result of the prediction will be compared to an experimental data in finding the best performance and analysis method. From the data that have been collected, it will show that which cycle time and loading that are suitable for materials to prevent the fatigue failure in the spot welding.

DEDICATION

Firstly thank to Allah S. W. T for the opportunity to finish this project. I owe this project and my true happiness to my beloved parent. Since the day I started going to this university until today, they are very caring and supporting for me.

ACKNOWLEDGMENTS

This report teaches and provides me a basic knowledge of engineering and helps me to understanding more about Fatigue life studies and force direction. This report inevitably involves many helping hands. First of all, I am extremely grateful and thanks to my supervisor, Mr. Mohammad Khalid b. Wahid for all the guidance and critics given to me directly or indirectly, and also his friendly in time to teach and explain to me amicably. Once again thanks you for your idea, knowledge and guidance that make me more understand and can develop further thinking for this project.

Finally, I would like to thank to all my family and friends, thank you for all the spirit word, love and helping me during the whole period which gave me the strength I need in order to carry on and finally finish this report.

TABLE OF CONTENTS

CHAPTER 3: METHODOLOGY

CHAPTER 4: RESULT AND DISCUSSION

CHAPTER 5: CONCLUSION & FUTURE WORK

LIST OF TABLE

LIST OF FIGURES

CHAPTER 1

INTRODUCTION

1.1 Background of The Project

Fatigue life is the processes that are weakening the material when the load is applied repeatedly. It is occurred relying upon the cyclic loading. Cyclic stresses may be caused by fluctuating mechanical load. The crack of the material is known as fatigue failure. Fatigue test method involves testing specimens under various states of stress, usually in a combination of tension and bending. [Serope Kalpakjian et al. 2010]

In this project, the fatigue life will be determine through the Finite Element Analysis (FEA) by applied the load and determine the cycle time until the material are crack. The data will be collected from the simulation analysis and then, from the observation, the current of the spot welding are determine depending on the selecting of the material such as aluminum which has a much higher thermal conductivity and electrical conductivity. The material properties will be inserting into Solidworks software.

After that, the simulation of the software will be setting to the next stage which is the load are inserted. The result will be determine in the contour plot of the simulation that will be analyzes. From the result, the data will be collected and then, it will be compared to the experimental process to determine whether it can be made some improvement from the analysis or the experimental method.

1.2 Statement of Problem

There is a requirement to do the spot welding on the simulation by using the galvanized iron and aluminum. The uses of the material are limited and also not economical to do the experimental. From the experimental, the material has been testing to predict what the cycles times and load that is caused the fatigue failure in spot welding. The simulation on the software with help to analyzed the result similar to the actual force and stress. By using Solidworks software, it can predict the fatigue life with the accurate analysis in simulation on the material.

1.3 Objectives

The aim of the project is to predict and determine fatigue life for spot welded by using Solidworks software.

The main objectives are:

- Determine fatigue life for spot welded in simulation environment

1.4 Scope of Project

The main purpose of this project was to determine fatigue life on spot welded joint model on simulation and make a comparison to the experimental existing method. There is requirement to analyze the materials which are:

- Type of material that are used is Galvanized iron and aluminum
- The thickness of the material is the limit to 0.5 mm-1.0 mm.
- Solidworks software as analysis software

1.5 Significance of Project

The method that is using to predict the fatigue life by the simulation is way faster, easier and accurate analysis for the material before it take to the actual worked. The project will determine the fatigue life thus it helps in order to achieve the aim of the project to verify the fatigue failure that occurred in the selecting material of the spot welding.

1.6 Summary

Basically, this chapter describes how the project is going to be carried out. Hence, it points out each objective and scopes that the project is going to targets and focuses on. In addition to that, this chapter also shows the significance of the project, where it will tell what this project will benefit or contribute towards.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter will be review literature related to the finite element analysis, fatigue failure of the material in spot welding, principle of spot welding and properties of the iron and aluminum. Literature shows there are a lot of cause that occur the fatigue life in the spot welding. By using the experimental method, it is waste of materials used. The study will review before analysis method were using.

2.2 Fatigue Life

Basically, the testing on the spot weld material is included tensile-shear and cross tension. Fatigue occurs in the spot weld based on the different loading condition and also it intermediate-cycle fatigue life increased with increased base material strength. When the cycle fatigue life at the higher, the performance of the fatigue was found to be independent of strength.

In the spot welding, Fatigue performance was controlled by the primary variables which are nugget size, sheet thickness, and the corresponding joint stiffness. When the dynamic load is applied, the fatigue occurs when it is repeated cyclic stress from a maximum to a minimum. The example of the fatigue is the fracture piece of wire that had been bending in one direction and the other repeated for the number of cycle [R. W. Rathbun at al. 2003]

When the crack begins, it starts to propagate to the area that had form higher stress concentration and it also proceed to propagates until the remaining stress area becomes small and when it cannot support the load statically further, the sudden fracture occurred.

The fatigue failure slowly spreading a crack with the load that applied to the cycle of the stress, and it is known as the basic of the mechanism for the high-cycle fatigue failure. In order for a crack to propagate, it only occurred when the stress of the part in tension condition.

For the surface is hardened, it increases the fatigue strength. But in fact, it is ambiguous statement which is the result for the material is depend on the material that are chosen and type of heat treatment that applied on the material. In fatigue testing, a specimen is subjected to periodically varying constant amplitude stress [Abass at al. 2013]

To prevent the fatigue or extending the life of the fatigue, the constant of compressive surface stress need to be created in the outer layer of the specimen and it is known as residual stress. It is also help to reduce or eliminate the tensile stresses that occur during loading. In weld, it is consider as the stress that would exist in the body if the external load were removed and sometimes it is called as internal stresses. Apparently, it is caused by non-uniform temperature in welding.

2.3 Fatigue Life Parameter

Resistance spot welding is the most preferred method to join metal sheets. The design variables for spot-weld joints affecting their strengths are basically sheet thickness, spot weld nugget diameter, number of spot welds and the joint type as exemplified in tensile shear (TS), modified tensile shear. [Ertas at al. 2008]

2.4 S-N Fatigue Curve

S-N curve is the graph between stress and number of cycle plotted for fatigue loading. When the fatigue failure is consider, it means that the material is subjected to alternate tensile and compressive stresses at the same time. The result of the fatigue testing plotted on the graph is called as S-N curve. To develop the curve, a series of samples is tested to failure at various stress ranges.

The resulting lives are plotted versus the corresponding stress range. The S-N curve is the locus of these data points. In more thorough testing, multiple samples are tested at each stress range. Common practice is to plot the S-N curve through the mean value at each stress range. [Robert Stone, 2008]

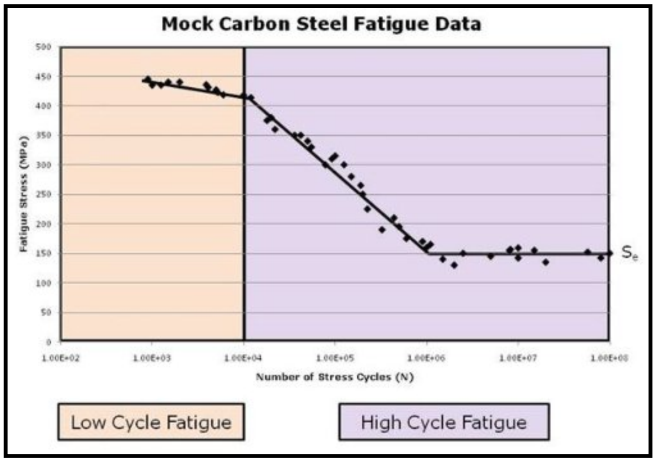

Figure 2.0: S-N curve

(Source: <http://www.solidworks.com>05/01/16)

The stress range and its associated mean stress are determined from the load history shown in the graph. The load history is "filled with rain." After the stress range and mean have been determined, the "rain" is drained from the lowest point. The range and mean for each remaining portion of trapped "rain" are then determined. From the results, Miner's rule can be applied, and the fatigue life can be calculated. [http://www.solidworks.com]

2.5 Finite Element Analysis (FEA)

Finite element analysis is the analysis based on the computerized method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis will determine whether a product will break, wear out, or work the way it was designed. The finite element method is comprised of three major phases:

1. Pre-processing
2. Solution
3. Post-processing

2.5.1 Pre-Processing

In Pre-processing, the subject geometry is divide into sub-domains which is, it is builds up by finite element mesh for mathematical analysis, applies material properties and boundary conditions. In pre-processing, the finite element mesh subdivides the geometry into elements, which are found are nodes. The nodes, which are really just point locations in space, are generally located at the element corners and maybe near each mid side. For three-dimensional (3D) thin shell analysis, it is basically a 2D element and may be slightly different in order to comfort to the 3D surface. For the spot weld, nodes that are created will be mesh to get the element size for the model.

Developing the mesh is usually the most time consuming task in FEA. In the past, node locations were keyed in manually to approximate the geometry. The more modern approach is to develop the mesh directly on the CAD geometry, which will be (1) wireframe, with points and curves representing edges, (2) surfaced, with surfaces defining boundaries, or (3) solid, defining where the material is. Solid geometry is preferred, but often a surfacing package can create a complex blend that a solids package will not handle. [http://www.autodesk.com/solutions/]

As far as geometric detail, an underlying rule of FEA is to "model what is there", and yet simplifying assumptions simply must be applied to avoid huge models. Analyst experience is of the essence. The geometry is meshed with a mapping algorithm or an automatic free meshing algorithm.

The first maps a rectangular grid onto a geometric region, which must therefore have the correct number of sides. Mapped meshes can use the accurate and cheap solid linear brick 3D element, but can be very time consuming, if not impossible, to apply to complex geometries.

Free meshing automatically subdivides meshing regions into elements, with the advantages of fast meshing, easy mesh size transitioning (for a denser mesh in regions of large gradient), and adaptive capabilities. Disadvantages include generation of huge models, generation of distorted elements, and, in 3D, and the use of the rather expensive solid parabolic tetrahedral element.

It is always important to check elemental distortion prior to solution. A badly distorted element will cause a matrix singularity, killing the solution. A less distorted element may solve, but can deliver very poor answers. Acceptable levels of distortion are dependent upon the solver being used. [http://www.finiteelement.com/feawhite3.html]

2.5.2 Solution/Solver

Solution is the programs that are used to solve the matrix equation from the model and solve the equation to the major quantities. The solvers that are usually used in the solving method are by using the ACUSOLVE software in the hypermesh which is, it can solve the equation for the numerical equation of fatigue failure for the spot welding.

2.5.3 Post-Processing

The post-processing is in which the analyst check the legitimate solution of the model and examines the values of major quantities such as displacements and stresses, and derives and examines additional quantities such as specialized stresses and error indicators.

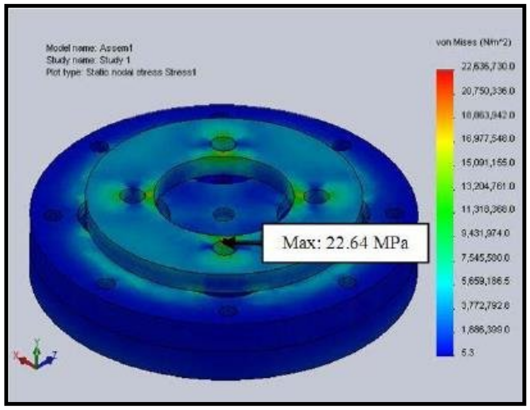

Figure 2.1: The result of stress analysis on the coupling (Rayothee al et. , 2015)

The FEA analysis can be performed to shown the stress-strain which determined the critical area and safety level at various region in the component. The element behavior is usually known under the support and load system. After the model was designing by using Hyperwork software, the finite element analysis is used. In order to study the stress-strain distribution, the model was designed as actual size and can be use for further experiment. [Pitchaya Rayothee at al. 2014]