University of Southern Queensland

FACULTY OF HEALTH ENGINEERING AND SCIENCES

Finite Element Analysis of Fillet Welded Joint

A dissertation submitted by

Rafiqul Islam Mohammed

In fulfilment of the requirements of

ENG4111/ENG4112 Research Project

Towards the degree of

Bachelor of Engineering (Mechanical)

Submitted: October 2015

Abstract

T-joint fillet welding is the most common welding in engineering applications. Transport vehicles, marine ships, mobile plant equipment are few examples where fillet welding are used extensively. Analysis of welded structures are still remains a challenge for the designer to produce desired output results. In welding process rapid heating and cooling introduced residual stress and geometrical deformations. Heat effected zone play pivotal role in determining the strength of a welded joint which changes the properties of parent material and reduce the strength after welding operation. There are many case which structures are continuously under cyclic loading when the fatigue life of the welded joints are a major design consideration

The aim of this project is to analyse the normal stress and fatigue life of fillet welded joints using computer modelling and experiments. Finite element based tool ANSYS Workbench 15.0 was been used to analyse the normal stress and the fatigue life under cyclic loading. Computer model of the joint developed using three different types of material which was parent metal, heat affected zone metal and weld metal. Experimental tests were carried out at USQ laboratory on double side welded T-joints. Grade 250 Structural steel was used to prepare specimen and gas metal arc welding (GMAW) process applied to welding the joints.

The ultimate purpose of the project has been achieved with developing techniques of the finite element analysis of fillet welded joint. The experimental investigation validate the performance of the FEA analysis results were found 1.2% error on tensile test. The experiment yield stress was found 263.4 MPa and simulation yield stress at the same location appears 266.7 MPa. In order to calculate fatigue life of welded joint used iterative process to define stress at one million cycle. The analysis found 274 MPa stress and 7740 cycle fatigue life applying yield load. After reduced load at 12kN and found the fatigue life one million cycle where shows 88 MPa stress which is 35% of yield stress. So that designer can consider 35% of yield strength when design structure for fluctuating and repeated loading conditions.

University of Southern Queensland

FACULTY OF HEALTH ENGINEERING AND SCIENCE

ENG4111/ENG4112 Research Project

Limitation of Use

Certification

I certify that the ideas, designs and experimental work, results, analysis and conclusions set out in this dissertation are entirely my own effort, except where otherwise indicated and acknowledged.

I further certify that the work is original and has not been previously submitted for assessment in any other course or institution, except where specifically stated.

Rafiqul Mohammed

Signature

16/09/2105

Date

Acknowledgements

This project was carried out under the principal supervisor Mr. Chris Snook. I would like to offer my sincere appreciation to Mr. Chris for his support and guidance throughout the completion of this dissertation. I am very grateful for his supports and innovative ideas which make this project in reality.

I would like to thank my friends and family for their support and assistance throughout this project, whether deliberate or not. I like would also to thank Dr. Ray Malpress for his generous time for useful tips to using ANSYS workbench.

Rafiqul Mohammed

University of Southern Queensland

October 2015

Table of Contents

Table of Figures

List of Table

Chapter 1 – Introduction

1.1 Overview

The aim of this chapter is to give an insight on the background of fillet welding, purpose of the study, and research objectives of the project. An understanding on the welding behaviour under real world situation is presented. This information will be relevant throughout the dissertation and underlining the comparison of experiment test results with simulation results.

1.2 Background

Welding is the most commonly used process for permanent joining of machine parts and structures. Welding is a fabrication process which joins materials (metals) or thermoplastics, by causing union (A. Thirugnanam 2014). In the joining process of welding application uses heat and/or pressure, with or without the addition of filler material. Various auxiliary materials, e.g. shielding gases, flux or pastes, may be used to make the process possible or to make it easier. The energy required for welding is supplied from outside sources.

Fillet welding is the process of joining two pieces of metal together whether perpendicular or in angle between 80-100 degrees (AWS 2010). This welds are commonly referred to as Tee joints which is perpendicular to each other's or lap joints which are overlap one another and welded at the edges. Figure 1-1 is the example of fillet weld.

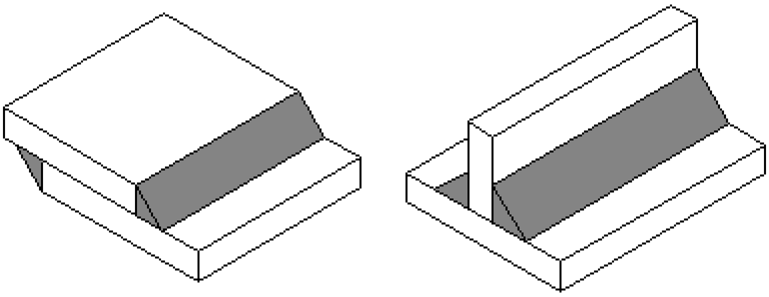


Figure 1-1: fillet welding diagram (*Welded Connections*)

Due to the influence of the welding residual stress, residual plastic deformation, heat affected zone and stress concentration effect the fatigue life of welded components is far lower than the parent metal (Yang, Zou & Deng 2015).

Fatigue life is most common word in engineering design, it simply means life of structure under repeated or fluctuating loading. Fatigue failure of the welded structures remains the most common type of failure (Richards 1969). Repeated or fluctuating load disturbs the material strengths and it causes the initiation of cracks. Neither metals nor the welds joining them are as smooth as they look, they have pits, grooves and cracks (Hicks 1987). The pits, grooves and cracks in a metal under load causes high strains over very small areas. Fluctuating loads will create small tears or fractures which increases in length with each application of the load.

In general, welded joints are more susceptible to fatigue cracking in comparison with bolted joints (A. Thirugnanam 2014). Residual stress and stress concentrations due to weld geometry induce micro-cracks that's are often accelerate fatigue damage. Failure caused by fatigue can be minor or catastrophic. Fatigue is a failure under repeated or otherwise varying load which never reaches a level sufficient to cause failure in single application. The consequences of failure are often very costly and it is estimated that 80-90% of all structural failures are caused by fatigue (*Leap Australia* 2014). Another statics shows 70-90% of the welded structures invalidation accidents in the past several years were caused by fatigue failure (Yang, Zou & Deng 2015). Leap Australia shows an example in Figure 1-2 the damage of fatigue failure. This is a structural failure due to fatigue.

Figure 1-2: Structure failed due to fatigue (Leap Australia 2014)

Fatigue cracking on bridge structures are quite often occurred where always induced dynamic loads. The crack started from weld defects at the intersection between the filleted welds connecting to the longitudinal stiffeners to the girder web and the butt welds made for transverse splices in the longitudinal stiffener. Fatigue damage involving web crack on a girder, as well as in flange to web weld caused by the radial stress in Figure 1-3.

Figure 1-3: Fatigue cracking in a bridge girder from weld defects (Haghani 2012)

The damage was caused by the fatigue cracking on the structure was found and reported that the connection between flanges to the web was made by fillet welding (Haghani 2012). The cracks on the girder web of the bridge can grow with radial stress component as seen in Figure 1-4

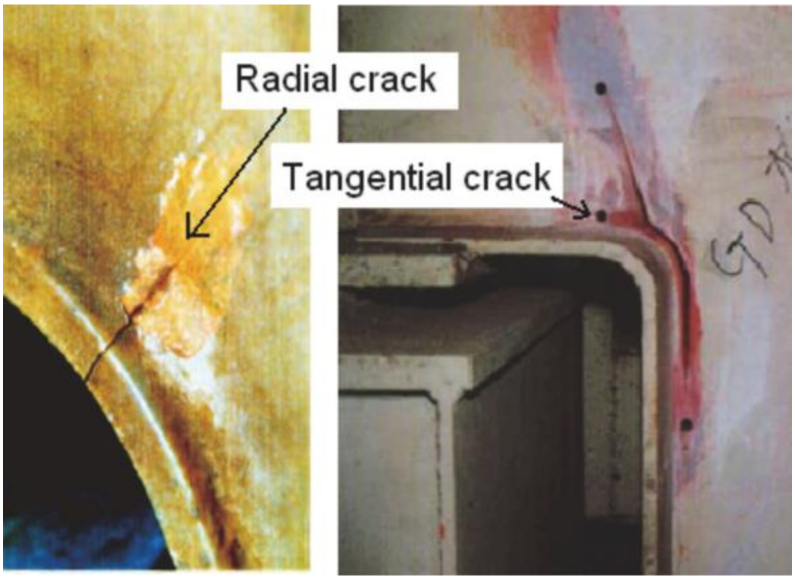


Figure 1-4: Example of radial and tangential cracks (Haghani 2012)

As the stories of building getting higher and the spans of bridge are getting longer, there are greater demands for high performance joints capability. T-joint fillet welds are extensively employed in various engineering applications. The welded joints has residual stress which interrupt the life of structures.

Processes for the fatigue analysis of metallic structures are now well defined, both in the stress-life or strain-life regimes (Aygul 2012). It has been recognized for some considerable time that the design of fabricated welded structures can be heavily dependent on the fatigue life of welded regions. Finite Element tools are often used to analyse complex welded joints. Figure 1-5 shows an analysis of a multi-sided weld joint using FEA. The optimization of such designs is therefore dependant on having good predictions of the fatigue life at the weld.

The fatigue life of welded metallic structures is dependent on the increased levels of stress found at the weld toe, root or throat due to weld geometry and the reduction in material properties with the heat affected zone. In addition to this, the amount of fusion of the weld into the parent metal and whether there is hot or cold fusion at a weld end all add to the inherent variability of welded structures. In short, the prediction of a robust fatigue life at a weld is extremely difficult.

This project was carried out FEA analysis with an overview of the work in the area of weld fatigue. Its includes industrial real world applications of current weld life prediction and modelling methods which gave an insight into the decisions to be taken to achieve robust results.

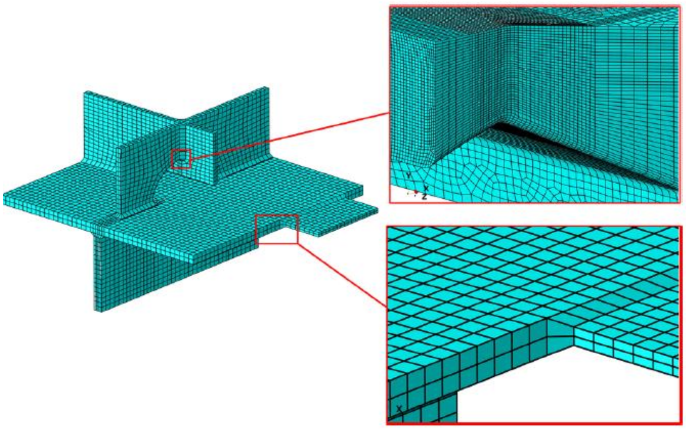


Figure 1-5: Analysis of fillet weld using FEA (Aygul 2012)

The way finite element analysis obtains the temperatures, residual stresses, and the load application, or other desired unknown parameters in the finite element model are by minimizing an energy functional. An energy functional consists of all the energies associated with the particular finite element model. Based on the law of conservation of energy, the finite element energy functional must equal zero. The finite element method obtains the correct solution for any finite element model by minimizing the energy functional. The minimum of the functional is found by setting the derivative of the functional with respect to the unknown grid point potential for zero. Thus, the basic equation for finite element analysis is $\frac{\partial F}{\partial P} = 0$

Where F is the energy functional and p is the unknown grid point potential in mechanics, the potential displacement to be calculated. This is based on the principle of virtual work, which states that if a particle is under equilibrium (Chang & Lee 2009), under a set of a system of forces, then for any displacement, the virtual work is zero. Each finite element will have its own unique energy functional.

1.3 Project topic

The topic of the project is to develop and evaluate techniques to analyse the strength and stiffness of welded joints. The analysis of welding with the software to validate simulation method compare with experiment testing of fillet welded T-joints. The aim of the project is to develop techniques for simplifying finite element analysis such as stress analysis and dynamic properties of structures under vibration excitation on non-complete joint penetrated (Fillet) welding. The analysis allow us to understand and compare experiment test result and software analysis result.

1.4 Scope of the research

In welding process used electric current and produce arc to melt the consumable electrode and the work piece to be welded. Electric current flow through high resistance air gap generates an intense arc with temperature running from 3000^{o} C – 6000^{o} C

Residual stresses caused by welding can have various kinds of influence on the welded structure, e.g. increasing the susceptibility of a weld to fatigue damage, stress crossing cracking and fracture. Moreover, residual stresses developed in T-joint fillet welds made of steels are probably different from those of full penetrated welds in magnitude. Residual stresses are unavoidable, and the effects on welded structures are cannot be disregarded. Therefore, it is very important to clarify the characteristics of residual stresses in T-joint fillet welds in the structures. Welded steel joints always susceptible to fatigue damage when subjected to repetitive loading. Fatigue failure may occur even under modest in-service stresses. Furthermore, fatigue lives exhibit considerable scatter even under constant amplitude loading in controlled laboratory conditions. This phenomenon makes statistical methods indispensable and fatigue life has to be predicted at given probability levels of failure for a given welded detail under defined environment and loading conditions.

The driving factor for conducting this research project is need for a more confidently using of finite element analysis tool to predict life of the welded structures. This is the time for engineers to design more sustainable, safe and desirable component for the society and finite element analysis tool is more convenient and very less time required to solve the most complex structures.

The comparison of physical test with finite element analysis are not much available which can refer to predict the life of structures. Finite element analysis (FEA) is a computer based method of simulating/analysing the behaviour of engineering structures and components under variety of conditions.

Fillet weld is most common welding in the engineering field and this weld used almost everywhere in building structures. Finite element analysis of simple fillet welding compare with physical testing will allow us to analyse more complex structures with more accurately with minimum error.

1.5 Project objectives

- The primary objective of this project is to develop techniques of finite element analysis to define normal stress and fatigue strength of fillet welded joint.
- Compare experimental test results with finite element analysis results of the investigated joint
- Characterise the strength of fillet welded joints in comparison with parent metals.

1.6 Expected outcomes and significance of the study

The research outcome will help designer to estimate strength of welded structural component under dynamic load. Welding is the common engineering join method which used for various structures. The major construction projects in all around the worlds are using cranes, earth moving equipment, those also always under dynamic load, the marine ships another heavy welded structure which are also subjected to dynamic loads at all the time. The body of transportation equipment are highly impacted by dynamic load which is focused in this analysis. Fatigue test and analysis of a T joint with fillet welding in a typical connection is presented in this study. Relevant finite element analysis of the joint is also established. The project will help to characterise the behaviour of fillet welded joints.

This study is expected to provide with an in-depth understanding of the behaviour of fillet welded joint under cyclic loading. The comparison of the test results with the finite element analysis will also give an indication about the assessment capability of the analysis tools for this type of joint. A simple and useful graphic representation in the form of chart, which will aid the readers to actualise the effect of cyclic loading on the joint is also expected.

1.7 Dissertation outline

The dissertation is arranged in the following sequence of the chapters:

- Chapter 1 introduces the study and states the objectives of the study followed by the expected outcomes from the study.
- Chapter 2: Literature Review contains the recap of the available literature. Potential gap in the studies is identified and reason for this study is highlighted.
- Chapter 3: Methodology and Project planning composed of the details of the methodology of the tests and simulations to be carried out in this investigation.
- Chapter 4: Physical Investigation provides with the test properties of the fillet weld joints in the laboratory.
- Chapter 5: Finite Element Analysis and simulation with ANSYS program on the fillet welded joints for comparison with the physical testing.
- Chapter 6: Discussion of Outcomes narrates analysis of the results and findings in terms of the parameters involved.
- Chapter 7: Conclusion gives a summary of the results and scope of further study and outlined.

Chapter 2 – Literature Review

2.1 General

The following chapter reviews literature relevant to the welding process and welding analysis using finite element methods. To calculate strength of welded structure and to find behaviour under cyclic load, it is very important to do analysis according to real life problems, so that analysis results can be used to predict the life of structures. After welding of structures there are many factor involve defining parameter in the finite element analysis. In this this report will go through and underline the associating maters with the welding. Material behaviour will influence by the heat effected zone, residual stress, types of welding, types of joint and different types of stress applications also included in the report. Fatigue analysis will take place to determine life of structure under cyclic load.

2.2 Welding process

Welding is the process of permanently joining two or more materials together, usually metals, by heat or pressure or both. When heated, the material reaches molten state and maybe joined together with or without additional filler materials being added (*Welding fundamentals and processes* 2011).

Welding is an expedient by which metals may be joined by increasing the temperature of the work pieces to their fusion point and allowing the molten metal formed to flow together and solidify (Morris 1955). There are many different types of welding process available now a days. The common welding processes are manual metal arc (MMA) welding, gas metal arc (GMA) welding, tungsten inert gas (TIG) welding, and submerged arc (SA) welding.

All of those processes used electric arc to melt electrode and parent metals. Due to high productivity GMA welding is widely used in the industry for general fabrications. In GMA welding process electrode is melted and molten metal is transferred to the work piece. The transfer of molten metal from the electrode to the work piece can be divided in to three modes (*Welding : theory and practice* 1990). Olson D L. et al. found increasing current increases weld penetration and weld size thereby distortion and residual stress is more which is affect to the welded structures.

Heat induced by welding has metallurgical effect on the metals it caused grain growth found observing microstructural feature (Dodge et al. 2014). Based on observations in the number of study found, a coarse-grained HAZ, martensite grain in the welded region but post weld heat treatment can be developed the HAZ materials. Normally very high percentage of welding structures used without heat treatment however, most of the structures induce stress in the joint which called residual stress.

2.3 Brief History of FEA

The finite element method is a numerical procedure that can be applied to obtain solution to many different type of engineering problem. The modern finite element method can be traced back to early 1900s. However, Courant has been credited with being first person to develop the finite element method in paper published in early 1940s (Moaveni 2007). The next significant step in the utilisation of finite element methods was taken by Boeing in 1950s after that 1960 Clough made the term finite element popular.

ANSYS is a comprehensive general purpose finite element computer program which can use for structural mechanics, fluid dynamics, electromagnetics, system and multiphasic analysis and simulations this program was released in 1971 for first time. ANSYS has more than 100,000 lines of code and it capable of performing static, dynamic, heat transfer, fluid flow etc. ANSYS has been leading FEA program over 35 years.

2.4 Effect of residual stress

Due to varying temperature distribution of welding process, thermal stress is generated. It is known that thermal stress leads to residual stress. At higher temperature when metal melted as liquid stage and solidified very sort of time, it has change grain structure of surrounding of weld metal which causes shrinkage and introduce residual stress. The welding residual stress has an effect on deformation of structures, concentration of structure, fatigue fracture etc.

The investigation done by Seok et al. (Seok, Suh & Park 1999) for the residual stress of H-type beam in Figure 2-1: H-type beam on butt welding made of high tensile steel by finite aliment method, they found the case of longitudinal component of the residual stress.

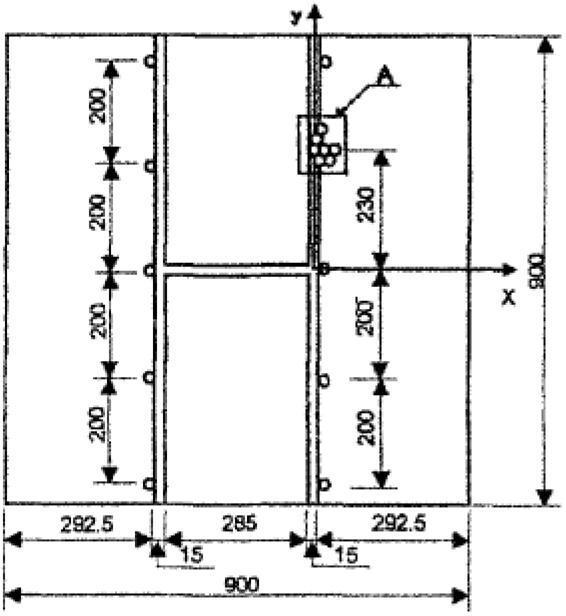


Figure 2-1: H-type beam(Seok, Suh & Park 1999)

The tensile residual stress increases from the centreline of weld bead to 5mm in the transverse direction and decreases to converge to zero after the location of 5mm off. That is the maximum tensile residual stress occurs in HAZ (heat effected zone), which is the location of 5mm apart from the centre line of weld bead, and magnitude is 62.5% of yield stress shown in Figure 2-2

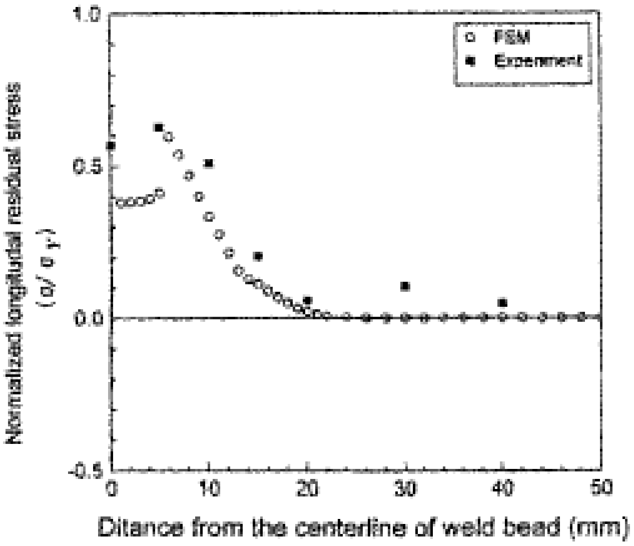


Figure 2-2 Longitudinal residual stress vs location from the centreline of weld bead (Seok, Suh & Park 1999)

There are many different stress levels in the welded joint, in the Figure 2-3 as following;

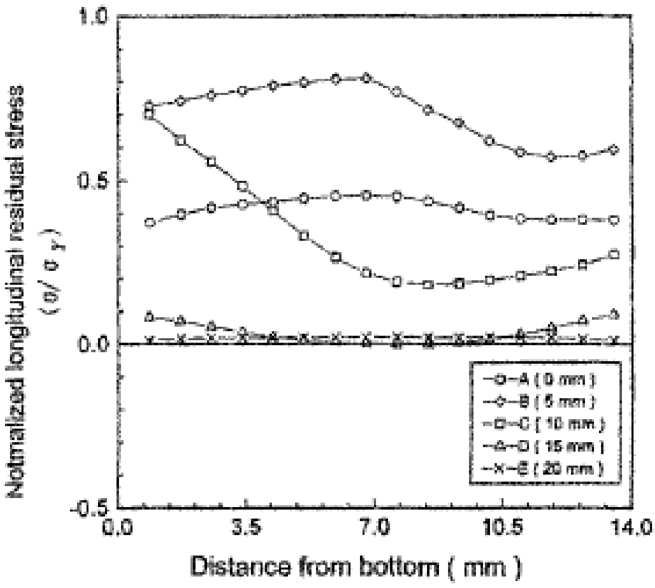


Figure 2-3:Longitudinal residual stress from the bottom on weld metal and parent metal.(Seok, Suh & Park 1999)

The location A and B are corresponding to the weldment and HAZ respectively in addition C, D and E are for base metal. In figure 5 it is observed that residual stress distribution of B on HAZ is higher than the A weldment.

T-joint fillet welding is extensively using in manufacturing industry such as shipbuilding and bridge constructions. During welding localized heating and rapid cooling causes tensile residual stress which develops close to the toe of the welding. Finite element analysis done by (Teng et al. 2001) on T-joint fillet welded joint. In order to analyse the Tee section Mr. Teng consider symmetric and used mesh refinement in the Figure 2-4.

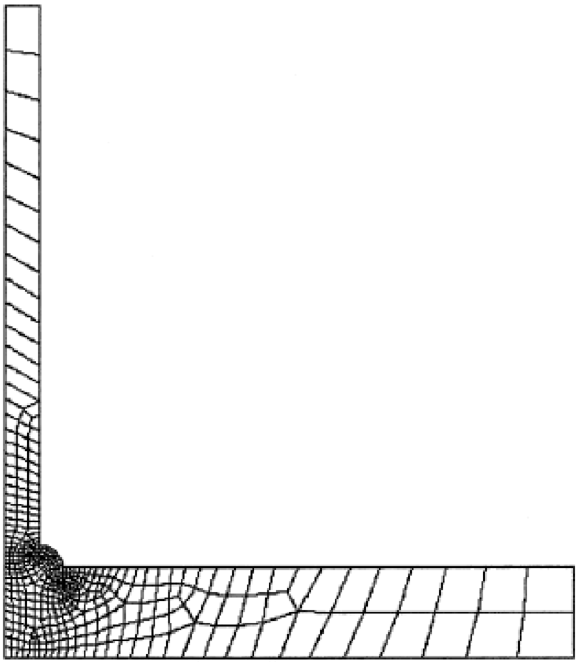


Figure 2-4: Finite element meshes of the T-joint fillet weld(Teng et al. 2001)

A stress acting on the direction of the weld bead is known as transverse residual stress, Figure 2-5 represents distribution of residual stress along X direction. A very large tensile residual stress is produced at the toe of the welding where stress shows 25 MPa (Teng et al. 2001).

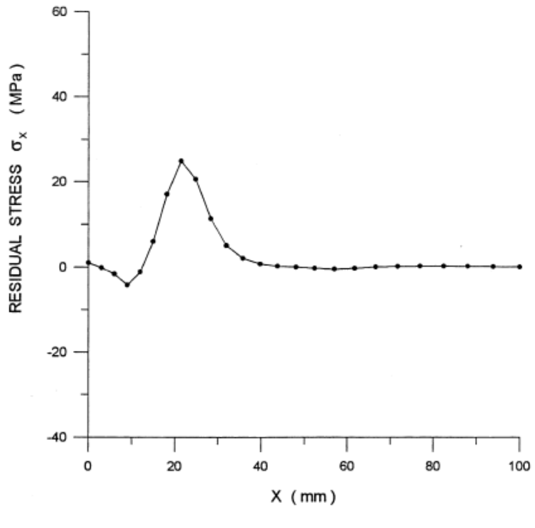


Figure 2-5: Transverse residual stress distribution along the X direction (Teng et al. 2001)

A very large tensile residual stress is produced at the surface of the base plate near the fillet weld toes. The value of the residual stress near the weld toe is 25 MPa and decreases to zero as the distance from the weld toes increases. A stress acting parallel to the direction of the weld bead is called longitudinal stress denoted at Figure 2-6 depicts the distributions of the residual stress along the X-directions. The residual stress value found is 110 MPa approaching the yield stress of the material.

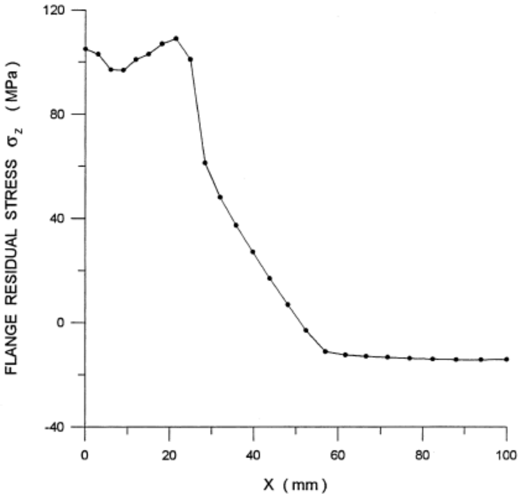


Figure 2-6: Longitudinal residual stress distribution along the X-direction

In the welding process, different weldment thicknesses required different weld penetration depths to controls the heat input to investigate the effect of different weld penetration depths on residual stress and distortions. In Figure 2-7 it can be seen that residual stress of fillet weld is higher than the full penetration joint.

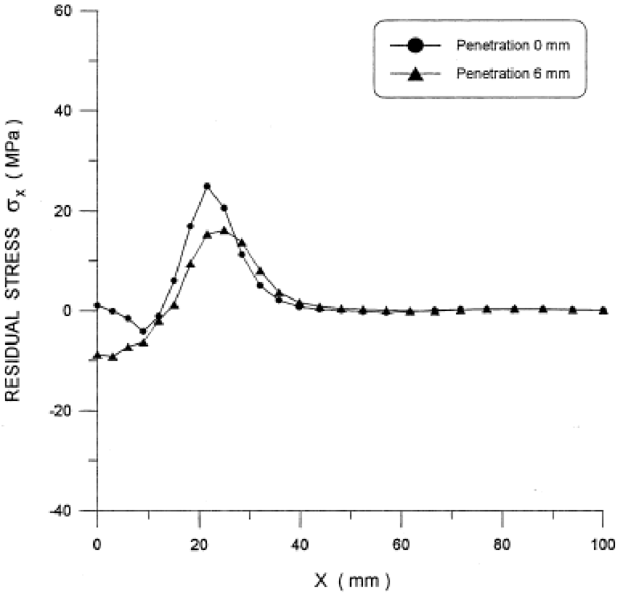


Figure 2-7: Transverse residual stress distribution for different penetration depths

The following research investigated both an experimental and finite element simulations for welding distortion in a T-joint fillet welding. There are many research and investigation done on the above topics. In principle, a finite element (FE) simulation of the welding process consist of two main parts: thermal analysis and mechanical stress analysis (Perić et al. 2014). In thermal analysis, the temperature field is determined as a function of time for each integration point. This temperature time-history is used as an input into the thermal stress analysis. Herein, the thermal solution can be sequentially or fully coupled with the mechanical solution of the structure. As presented in the literature, the use of three dimensional (3D) models is required for accurate prediction of post-weld deformation and residual stress distribution. Peric et al. 2014 done experiment according to Figure 2-8

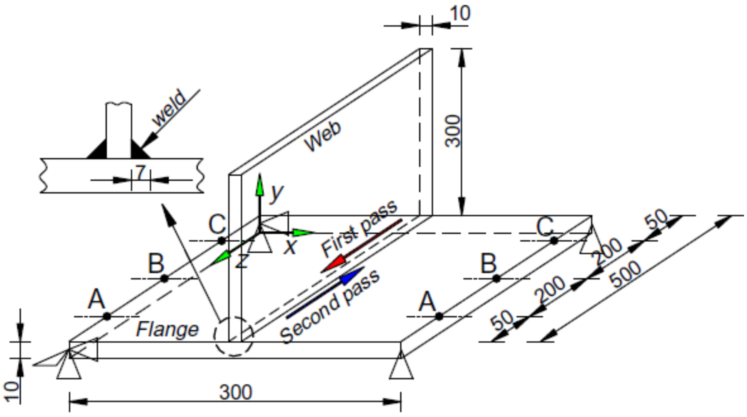


Figure 2-8: Geometry of T joint (Perić et al. 2014)

The welding experiments are conducted and the measured temperatures and displacements are compared with the results obtained by the numerical analysis by (Perić et al. 2014). It is shown that the numerical results agree well with the experimental results in Figure 2-9.

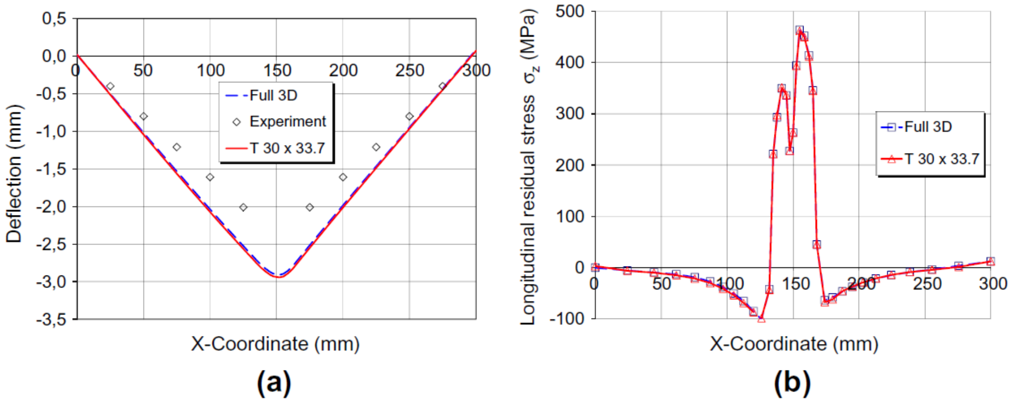


Figure 2-9: : Deflection and residual stress of fillet welding at direction along the line C-C, from figure 10 (Perić et al. 2014)

To control residual stress in the welded joint clamping method is given significant different result from unclamped joint. Three different clamping condition has been analysed by Padma Kumari.T and Venkata Sairam.S the condition (i) is unclamped (ii) is clamped but released immediate after welding and (iii) clamped and released after cold down at ambient temperature. Among the three cases considered, the third case yields the minimum transverse residual stresses at the upper surface of the plate while it is reversed with regard to the bottom surface. Since the plate is unclamped after cooling down to ambient temperature, it increases the longitudinal stresses at the bottom surface due to the self-weight of the plate as it acts as a fixture to the bottom surface found (PadmaKumari 2013). Hence the third case is the most preferable condition in regard to the residual stresses in Figure 2-10.

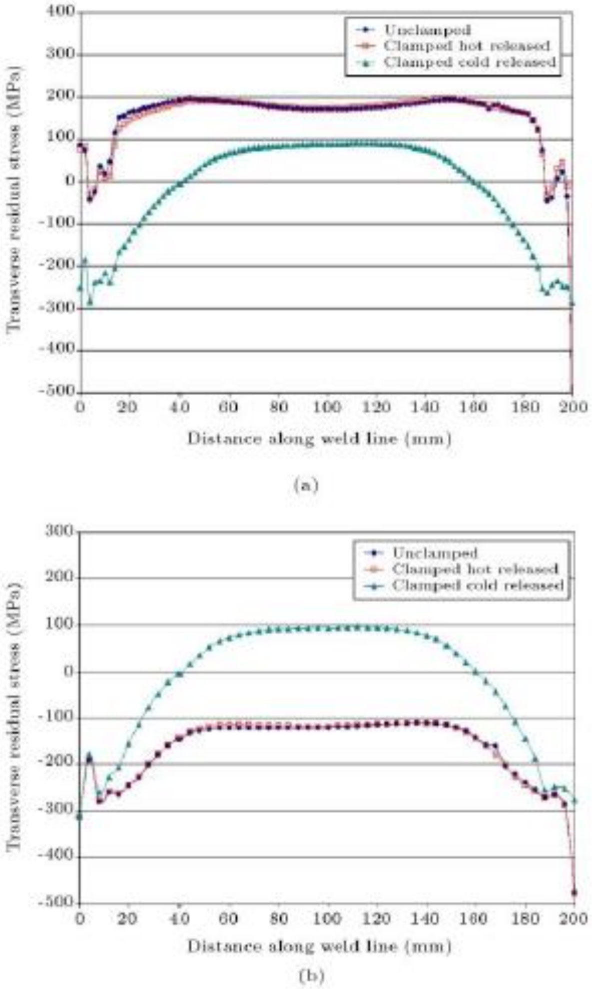


Figure 2-10: Transverse stress distribution at the top surface (a) and the bottom surface (b) (PadmaKumari 2013)

To predict residual stress of T-joint fillet welding (Kyong-Ho Chang & Chin-Hyung Lee) described in Figure 2-11 thermal analysis and mechanical stress analysis where they found stress are higher on the weld toe area and its get lower while distance from centre is increased.

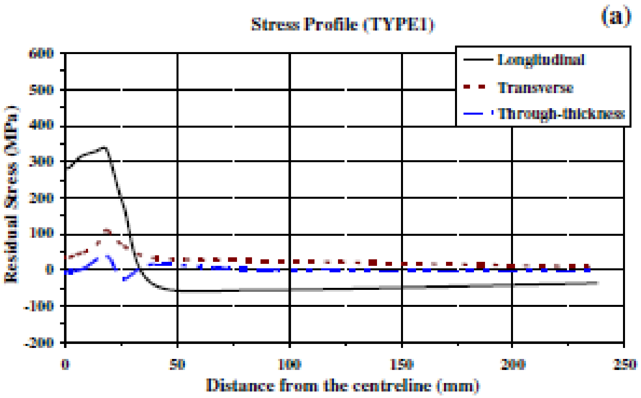


Figure 2-11: Residual stress profile at cross section of flange (Chang & Lee 2009).

2.5 Effect of welding sequence

Influence of welding sequences on the distribution of residual stress and distortion generated when welding flat bar as a stiffener on to the steel plate. Gannon et al. 2010 found In the case of longitudinal residual stresses, welding sequence did not have a significant influence on the distribution pattern of the stress; however it did affect the peak values. In the Figure 2-12 welding Sequences (**a)** and (**d)** caused the notably higher compressive residual stresses in the plate than sequences **(b)** and **(c)**. In the stiffener, the maximum compressive residual stress due to sequence **(b)** was approximately 3.5 times greater than the next lowest value. The distribution and peak values of residual stress were similar to measured values and predictions used in literature. Maximum tensile residual stresses equal to the material yield strength were predicted in the vicinity of the weld and maximum compressive residual stresses from _97 MPa to _58 MPa in the plate and stiffener respectively (Gannon et al. 2010).

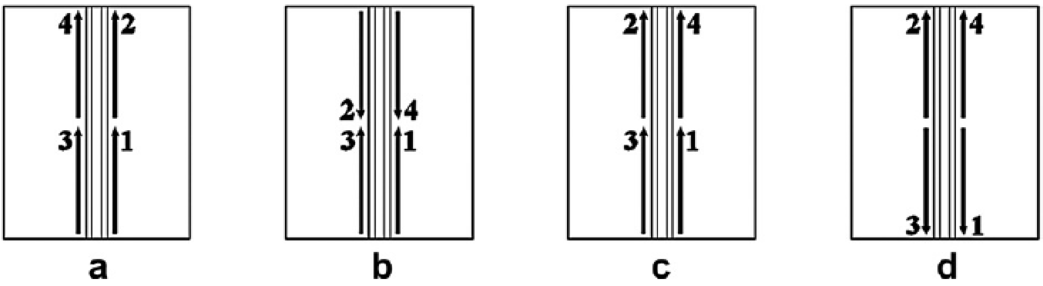


Figure 2-12: Welding sequences (Gannon et al. 2010)

In the case of welding-induced distortion, welding sequence B resulted in the largest out-of-plane deflection of the plate which may result in a reduced plate effectiveness. Sequence B resulted in the largest out-of-plane deformation of the plate along the axis of the stiffener. These predicted distortions were of lower magnitude than typical values suggested in literature. Welding sequences A and B resulted in the largest lateral deflections of the stiffener. Sequences C and D resulted in the least distortion of both plate and stiffener for the geometry considered. Considering both residual stress and distortion as a result of welding, welding sequence D is identified as the preferred welding sequence with the lowest welding-induced residual stress and distortion in Figure 2-13.

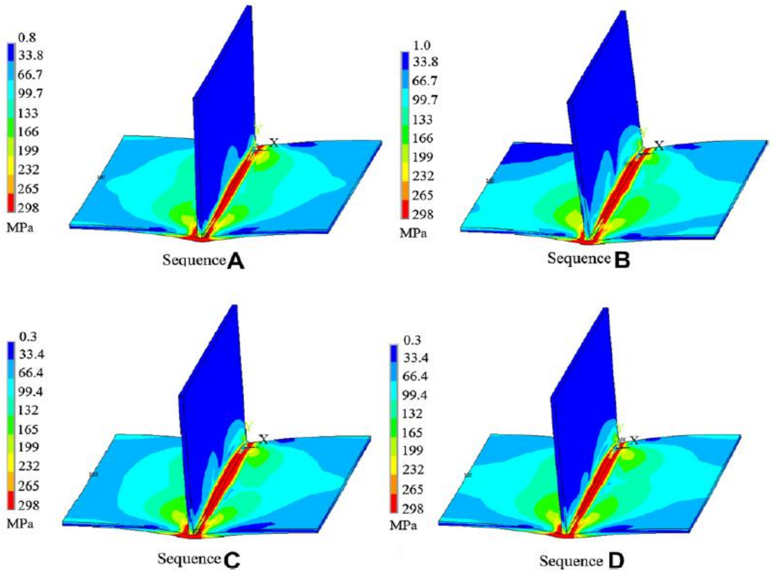


Figure 2-13: Deformation shapes with von misses stress contours (Gannon et al. 2010)

Overall, the finite element model including both thermal and mechanical procedures has provided simulation results in reasonably good agreement with the experimental measurements. The variations observed between the numerical and experimental results were considered to be within the acceptable limits (Gannon et al. 2010).

Fillet welded joint has very common angular distortion, restricted stiffeners welding and free restrained stiffeners welding are given different angular distortion and residual stress see Figure 2-14. Type of weld, joint preparation, thickness of plate, size of weld, joint restraint, heat input as well as welding sequence are effect of residual stress on welded joint.

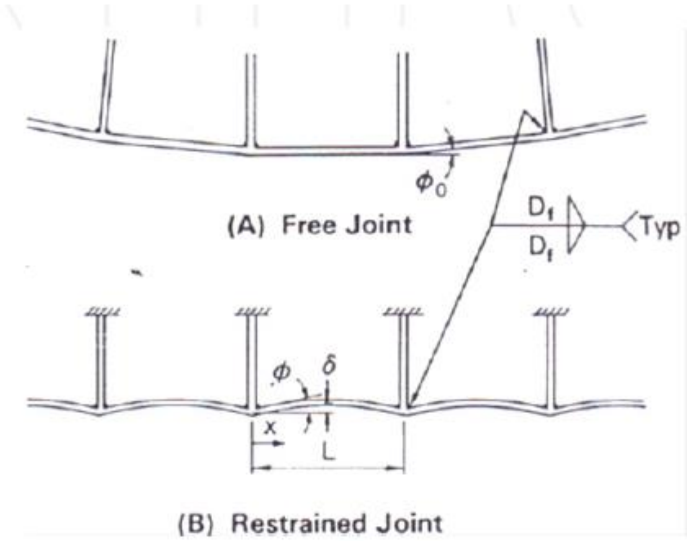


Figure 2-14: Angular change in T-fillet joint (A) free restrained (B) Restricted

(Syahroni & Hidayat 2012) consider in his study and the numerical investigation according to the following sequence in Figure 2-15 welding sequences considered were the one direction welding (WS-1), the contrary direction welding (WS-2), the welding from centre of one side (WS-3), and the welding from centres of two sides (WS-4).

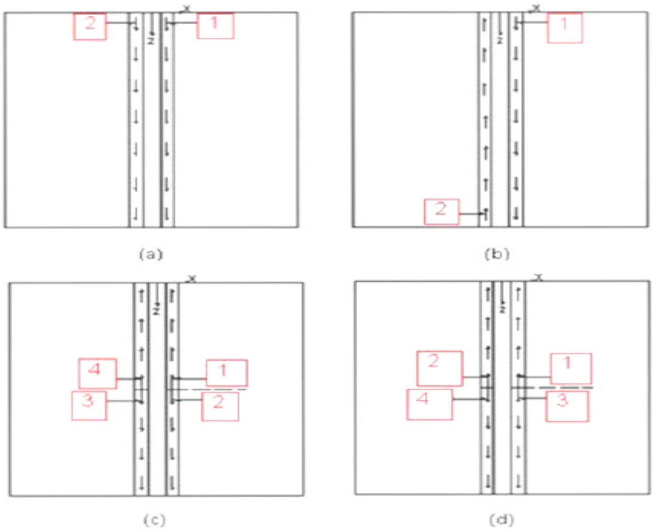


Figure 2-15: Variation of welding sequence, (a) WS-1, (b) WS-2, (c) WS-3, (d) WS-4 (Syahroni & Hidayat 2012).

According to above preparation finite element procedure was employed to simulate the thermos mechanical response of welding problem Syahroni & Hidayat 2012 found results on the problem considered are presented in this section. The finite element simulation for all the variation of welding was completed in 45 load-steps (LS). During the number of load-steps, the welding process took for 40 load-steps, while the cooling one took for the rest of the LS. For the presentation of welding simulation, the results of the LS which respectively represent the conditions of the peak temperature and the beginning of cooling processes were taken and plotted. Note that the temperature went down towards the room temperature after the LS of 41. Accordingly, the longitudinal and transverse residual stresses and the distortions occurred due to the welding sequences. Peak temperature is always varying during welding in different sequencing found in the Figure 2-16

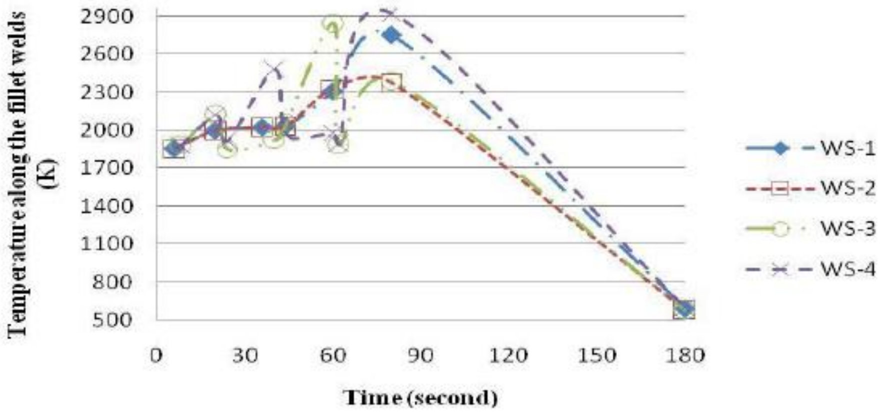


Figure 2-16: Peak temperature for each welding sequence (Syahroni & Hidayat 2012).

Figure 2-17 describes the transverse residual stress distribution along the fillet weld for each WS. The maximum values of longitudinal and transverse stresses as well as von Mises stress for each welding sequence were summarized in Table 2. The ratio between the longitudinal and the transverse residual stress values for the problem considered varies from 1. 06 to 1. 22 (Syahroni & Hidayat 2012).

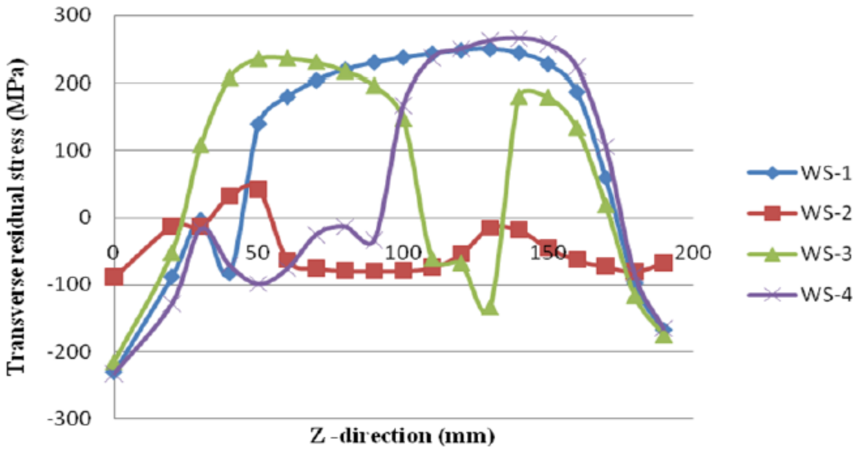


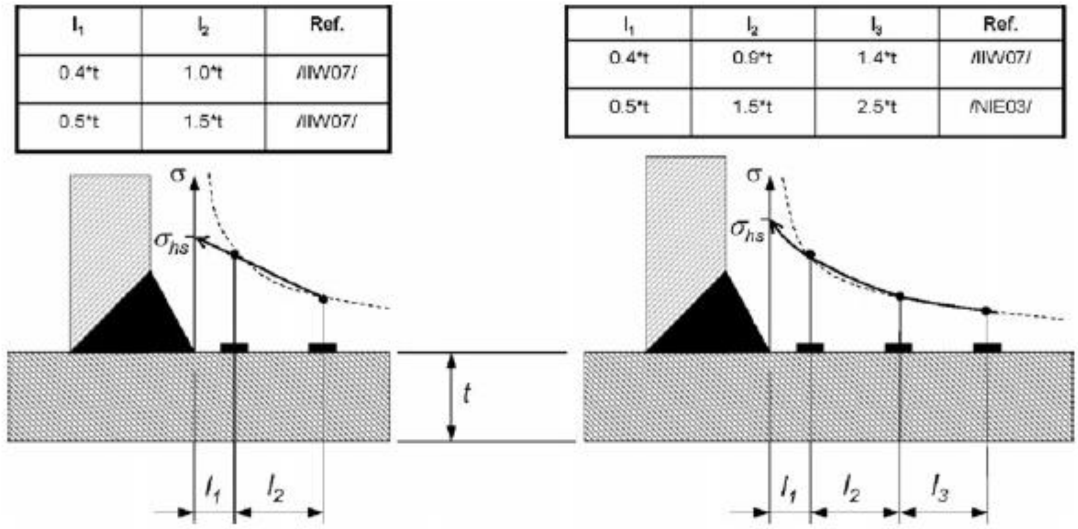
Figure 2-17: Distribution of transverse residual stress along the fillet weld for each welding sequence (Syahroni & Hidayat 2012)

It can be clearly observed that the distributions of transverse residual stresses produced by WS-3 and WS-4 and WS-1 and WS-2, respectively, are in consistent nature with respect to the welding sequences.

2.6 Fatigue life calculation

Welded steel joints are vulnerable to fatigue damage when subjected to repetitive loading. Fatigue failure may occur even under modest in-service stresses. Furthermore, fatigue lives exhibit considerable scatter even under constant amplitude loading in controlled laboratory conditions (Lassen, Darcis & Recho 2006). Fatigue strength of weld seams can be evaluate according to nominal stress or the notch stress approach. Determination of the stress of welding seams have to be generated a concept of compatible way (Rother & Rudolph 2011) that means correct stress need to be assigned on finite element analysis.

Characteristic of structures after welding with the method of hot spot stress Rother et el. found in the Figure 2-18 where distance can be calculated based on plate thickness.



l_1	l_2	Ref.
0.4*t	1.0*t	/IIW07/
0.5*t	1.5*t	/IIW07/

l_1	l_2	l_3	Ref.
0.4*t	0.9*t	1.4*t	/IIW07/
0.5*t	1.5*t	2.5*t	/NIE03/

Figure 2-18: Linear (Left) Quadratic (Right) extrapolation based on stress derived from strain gage measurement (Rother & Rudolph 2011)

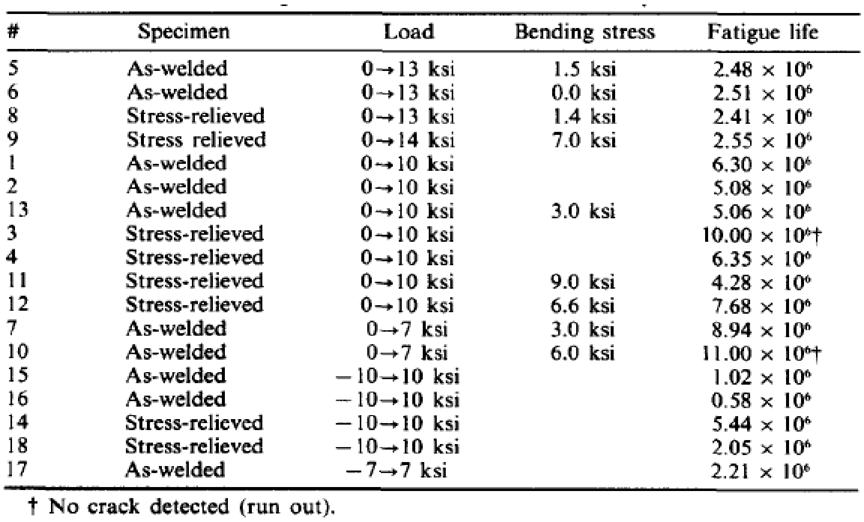
Rother & Rudolph done seven test specimens according notch stress method to predict fatigue life they found the following results in Table 2-1

Table 2-1:Computed structural notch stress including derived fatigue life (Rother & Rudolph 2011)

	CAB-method		Haibach method		Notch stress concept R1MS		Test
Position	σ_{CAB} (MPa)	N_{CAB} (cycles)	$\sigma_{\mathrm{Haibach}}$ (MPa)	N_{Haibach} (cycles)	σ_{R1MS} (MPa)	N_{R1MS} (cycles)	N_{Test} (cycles)
I	122	0.8×10^6	118	0.9×10^6	302	0.8×10^6	$1 \ldots 2 \times 10^6$
II	–	–	94	1.8×10^6	218	$>2 \times 10^6$	No cracks before 2×10^6

(Shen & Clayton 1996) have done testing before stress relief and after stress relief, Table 2-2 shows the fatigue test result under pulsed-tension there was no consistence effect of residual effect on fatigue strength. With tension-compression cyclic loads, stress relieved specimens exhibited a superior fatigue life.

Table 2-2: Fatigue test result of fillet welded A515 Steel specimen (Shen & Clayton 1996)



#	Specimen	Load	Bending stress	Fatigue life
5	As-welded	0→13 ksi	1.5 ksi	2.48×10^6
6	As-welded	0→13 ksi	0.0 ksi	2.51×10^6
8	Stress-relieved	0→13 ksi	1.4 ksi	2.41×10^6
9	Stress relieved	0→14 ksi	7.0 ksi	2.55×10^6
1	As-welded	0→10 ksi		6.30×10^6
2	As-welded	0→10 ksi		5.08×10^6
13	As-welded	0→10 ksi	3.0 ksi	5.06×10^6
3	Stress-relieved	0→10 ksi		10.00×10^6†
4	Stress-relieved	0→10 ksi		6.35×10^6
11	Stress-relieved	0→10 ksi	9.0 ksi	4.28×10^6
12	Stress-relieved	0→10 ksi	6.6 ksi	7.68×10^6
7	As-welded	0→7 ksi	3.0 ksi	8.94×10^6
10	As-welded	0→7 ksi	6.0 ksi	11.00×10^6†
15	As-welded	−10→10 ksi		1.02×10^6
16	As-welded	−10→10 ksi		0.58×10^6
14	Stress-relieved	−10→10 ksi		5.44×10^6
18	Stress-relieved	−10→10 ksi		2.05×10^6
17	As-welded	−7→7 ksi		2.21×10^6

† No crack detected (run out).

The most recent design code in this regard is the ASME Boiler and Pressure Vessel Code, Section VIII-2 (2007) (Joshi & Price 2009). Annex 3.F in Part III gives equations for the number of allowable design cycles for non-welded and welded joint fatigue curves based on empirical constants. In the following eq. code states the formula for N, the number of design cycles for welded joint fatigue curve, as

$$N = \frac{f_I}{f_E}\left(\frac{f_{MT}C}{\Delta S_{range}}\right)^{\frac{1}{h}}$$

f = Fatigue improvement factor, when the structure has been burr grinded, TIG dressed, or hammer peened,

f_E = Environmental modification factor, which is typically a function of the fluid environment, loading frequency, temperature, and material variables such as grain size and chemical composition, for structures operating in environments other than ambient air,

f_{MT} = Temperature adjustment factor, which is required for materials other than carbon steel and/or for temperatures above 21 ^{o}C (70 ^{o}F),

ΔS_{range} = Structural stress range.

Residual stresses are unavoidably generated in the component after welding (Zhang et al. 2013). To investigate the effect of residual stress Zhang et al. used 16 inch pipe, the result from the centre-hole measurements suggested that there are no clear effect of residual stress, however, he found high tensile residual stress on the surface of the weld root where welding started and stopped.

2.7 Fatigue life improvement

In many cases, the fatigue life can be improved employing good detail design of welding profile. However, there are two main improvement method addresses by (Kirkhope et al. 1999) which is weld geometry modification methods and residual stress methods. Following Figure 2-19 is the layout of the fatigue improvement methods.

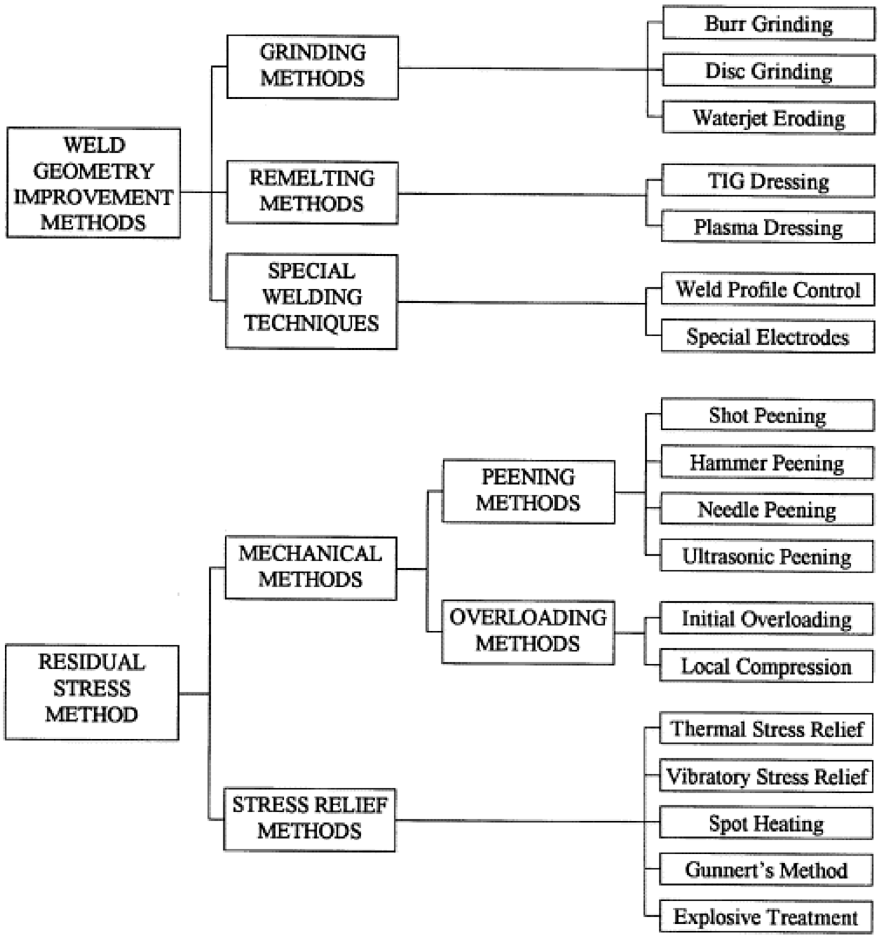


Figure 2-19: Classification of some weld improvement methods(Kirkhope et al. 1999)

The Welding Institute recommended that after welding fatigue life can be improved with some technic, such as Hammer peening, Machining, Shot peening, Plasma dressing and Disc grinding locally burr machining seen in the Figure 2-20

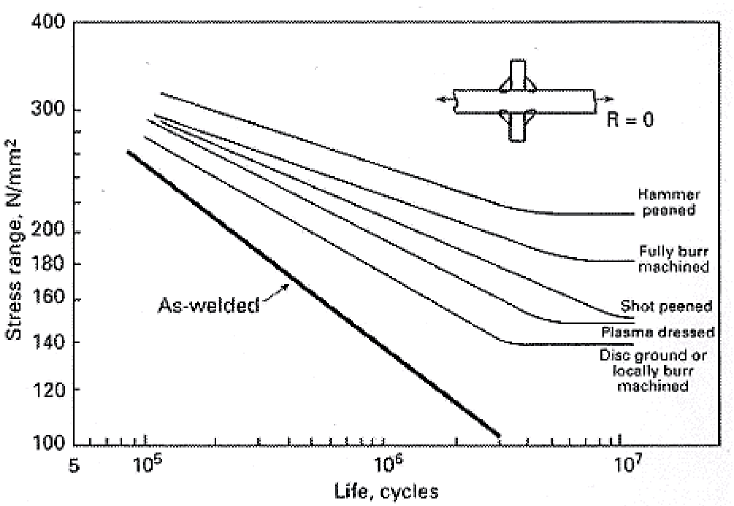


Figure 2-20: Typical improvement in fatigue strength of mild steel fillet weld resulting from selected weld toe improvement technic (Maddox 2002)

The offshore technology report provides information of selection design process for specially oil platforms. In the Figure 2-21 from OT report produce basic design curve for welded tubular and plate in the air.

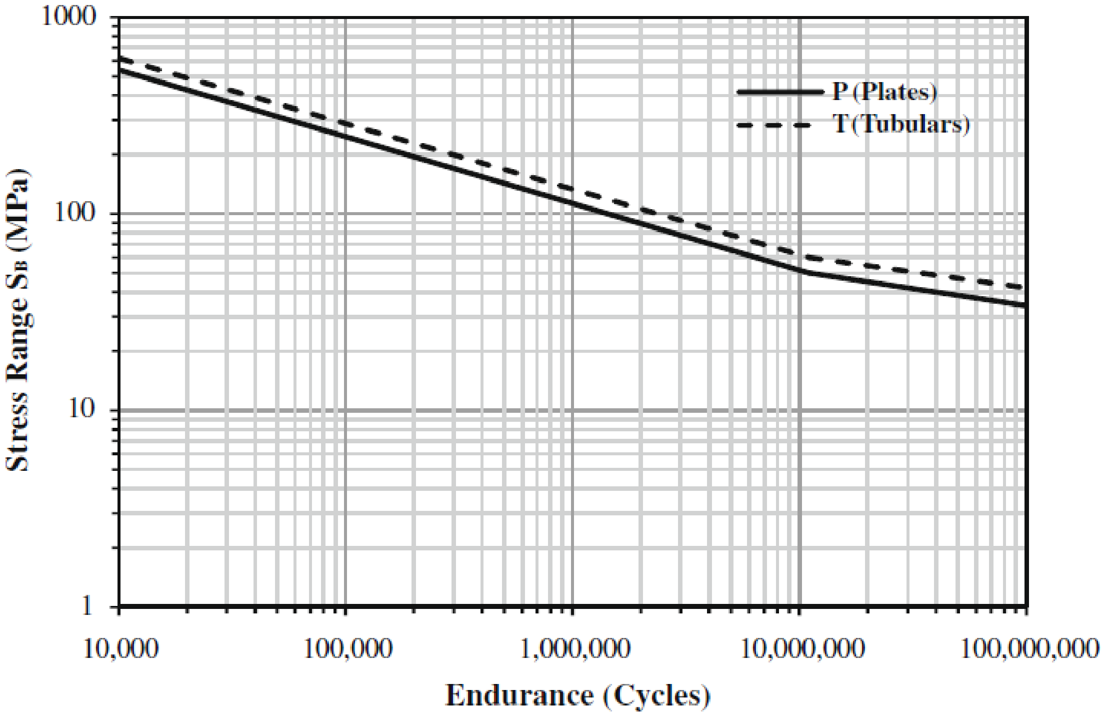


Figure 2-21: Basic design curve for welded tubular plates in the air (Joshi & Price 2009)

2.8 Computational challenges in welding simulation

Welding analysis is an important topic in engineering research and it is widely employed in the fabrication due to their advantage of improved structures performance, cost savings and easy implementation. However, welding application cause undesirable permanent distortion and residual stress in the material (Deng, Liang & Murakawa 2007). Welding has about 26 imperfections e.g. cracks, porosity, worm hole, inclusions, lack of penetration, lack of fusion, lack of fit, undercut, excessive weld overfill, insufficient weld throat, root overfill, misalignment, weld sag, incomplete root, cold lap, arc strike, sputter etc. (Hobbacher 2009). With all this imperfection simulation parameter will be in challenge. Material modelling is, together with the uncertain net heat input, one of the major problems in welding simulation (Lindgren 2001)

2.9 Thermal and mechanical finite element analysis

Finite element analysis (FEA) has been used widely by many researchers Bibby et al. (1992), Goldak et al. (1991), Zhang et al. (2006). Wikander et al. (1996), Breiguine et al.(1992), Gundersen et al. (1997), Lindgren L-E et al.(1988)], to perform welding simulations and to predict residual stresses in different types of welded joints and materials. Prediction is very difficult due to the complex variations of temperature, thermal contraction and expansion, and variation of material properties with time and space (PadmaKumari 2013).

Over the last few years, the technology of laser direct metal deposition (LDMD) has gained increasing attention in the industry for the rapid manufacture, repair, and modification of metallic components especially those involving high cost material such as superalloys. The technology is used in various industrial disciplines for its demonstrated possibility in the production of parts with complex internal structures that could not be achieved by machining. Furthermore, its associated lower material wastage ratios in low volume manufacturing applications is among those attributes leading to its high level of utilization in the aerospace industry.

The prediction of residual stress in laser deposited Waspaloy parts was found that experimental result and simulation result has some relation. While the results of the study indicated that modelling the deposition process with slight overestimation about 20% in the width of the deposited wall has no significant effect on predicted residual stress, the effect (A. M. Kamara 2011), which an underestimation in the width of the deposited wall could have on predicted residual stress, is also an interesting issue to be investigated.

(Brickstad & Josefson 1998) simulate the residual stresses due to welding using ABAQUS to perform finite element analysis, their analysis consist of two main parts the thermal and the structural. The analysis is two-dimensional and axisymmetric. The thermal analysis models the heat input from the welding torch into the weld elements causing the weld to melt. Heat losses allow the weld region to solidify. The temperature reliefs obtained from this part of the analysis are used in the sequential, structural analysis to derive the stresses generated as the material heats up and cools down again. The behaviour of the material involves non linearity and therefor residual stresses remain in the welded joint after cooling.

The mechanical effect of the material due to heat source, during analysis need to consider the following effect in **Figure 2-22** two dimensional model of heat source. This model gave a more realistic model with better heat distribution in the melted zone.

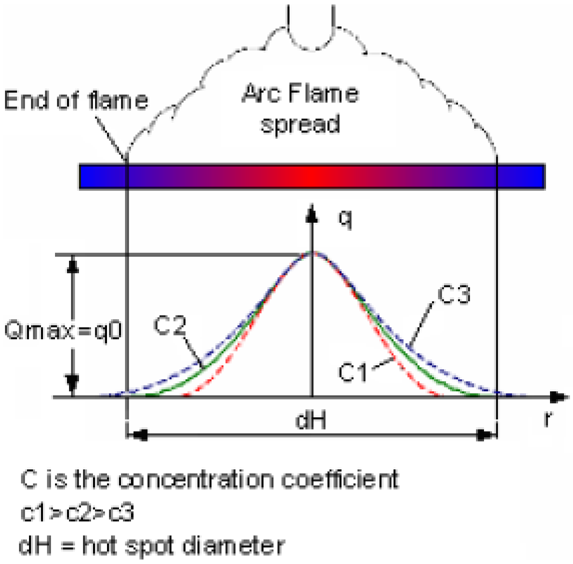


Figure 2-22: Heat distribution of melted zone(Rabih Kamal Kassab1 2012)

2.10 Heat affected zone

Heat affected zone is the most critical location in welded joint where the most residual stress develop due to metal grain coarsened. Heat affected zone is very hard to predict heat effected zone property (Wang et al. 2007) however, the region of HAZ partially ferrite and austenite at peak temperature (Woolling & Carrouge 2002). Figure 2-23 shows the grain growth in the ferrite phase.

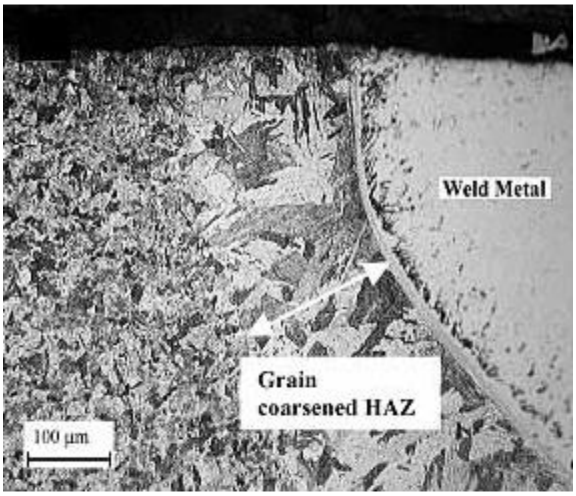


Figure 2-23: High temperature HAZ showing grain coarsened region (Woolling & Carrouge 2002)

The next to the weld is assigned strength properties slightly lower than the values given in the design codes in order to trigger strain localization. The width of this zone should be approximately equal to the plate thickness and discretised with one or several elements. The remaining part of the HAZ is assigned the strength properties given in the design code, and discretised with several elements. The accuracy of this procedure remains to be validated. After welding heat affected zone are completely change geometry of grain structure as Figure 2-24, due to the affect material strength has been change.

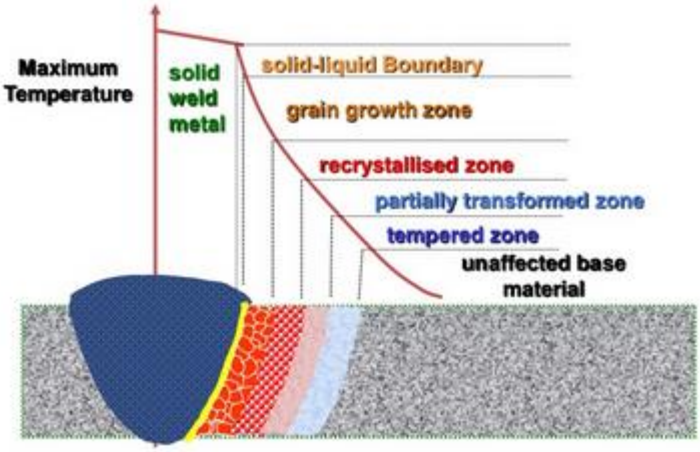


Figure 2-24: Heat affected zone after welding (Welding Inspection Cswip 2010)

2.11 Welding deformation

The deformation of welded structure in Figure 2-25 result from non-uniform expansion and contraction of weld and the surrounding base material due to heating and cooling cycle during welding process. Welding deformation play an important role in sealing capabilities and service life of welded joints says (Abid & Siddique 2005). There are lots of numerical and analytical model has been available for butt welding process. However, only very limited literature describing welding deformation of fillet welds are available (Deng, Liang & Murakawa 2007). Deng et al. found that the flange thickness of T-joint has influence on welding deformation and the simulated result demonstrates that the temperature gradient through thickness is a main factor that strongly governs the generation of angular distortion in fillet-welded joint.

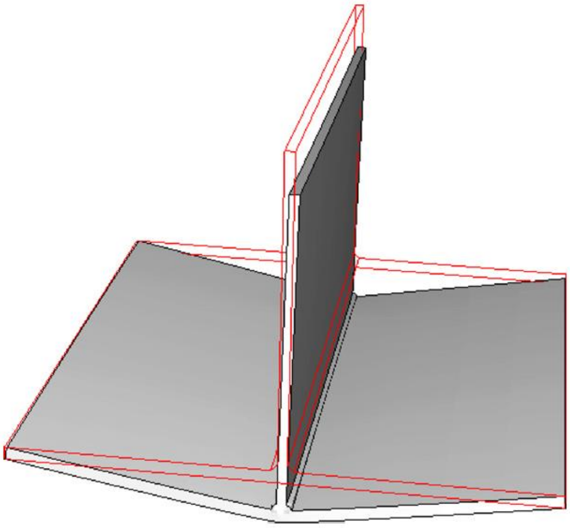


Figure 2-25: deformation after welding (Deng, Liang & Murakawa 2007)

The heat source used in fillet welding joint in Figure 2-26

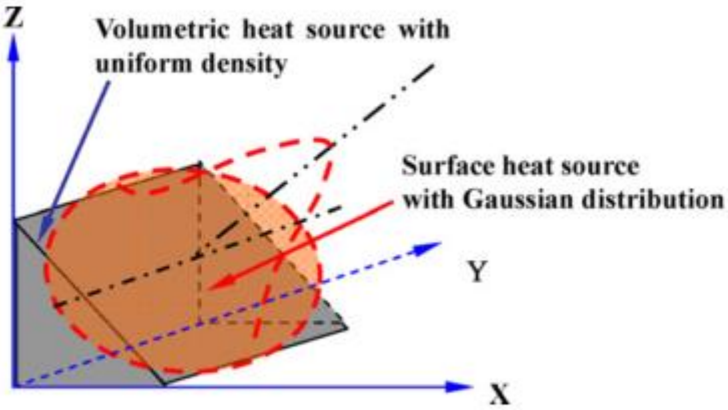


Figure 2-26: Combined heat source in the fillet welding joint (Deng, Liang & Murakawa 2007)

Distortion after welding in Figure 2-27 shows 500mm of welding can angle difference .022 rad.

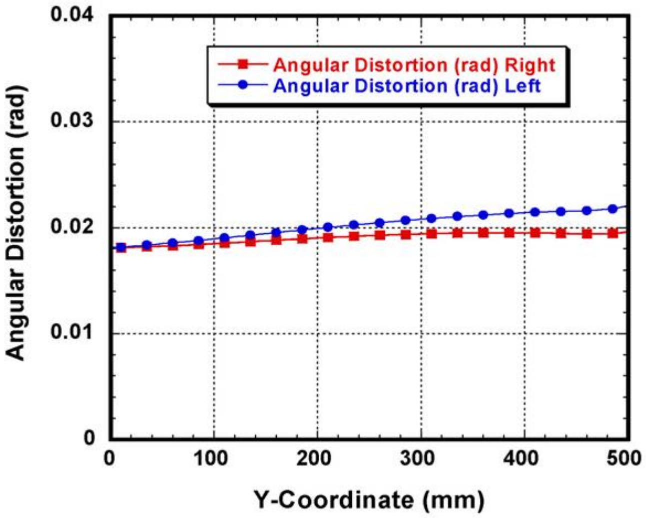


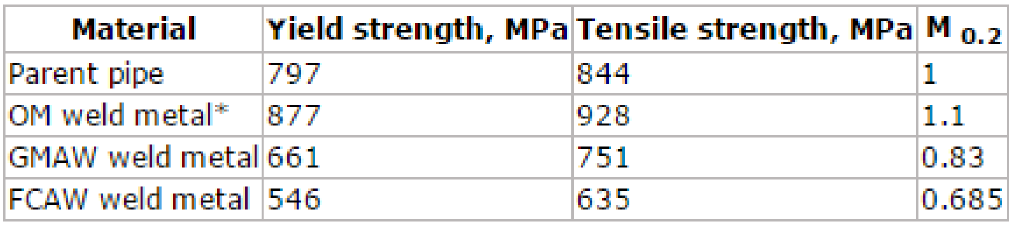
Figure 2-27: Distribution of angular distortion along welding line (Deng, Liang & Murakawa 2007)

The distortion of welded structure also be subject to welding sequence as well as number of pass of welding. Analysis of a welding thermo-mechanical response due to welding is the important factor. Welding distortions depend on the geometry, welding condition as well as the material properties (WANG Rui 2008). Sulaiman et al. 2011 has done investigation of simulation and experimental distortion of welded T-joint, where they found 20.9 % error of the result (Sulaiman et al. 2011) which is reasonable agreement.

2.12 Weld metal property

Weld metal properties are not exactly same as a parent metals, it has different strength and different grain structures. Pisarski et al. investigated on a grade X100 pipe, they found the tensile property of the weld metal after different type of weld process used which shown in Table 2-3 and fi can see that weld metal strength are lower than the parent metal.

Table 2-3: Tensile properties of grade X100 pipe (Pisarski, Tkach & Quintana 2004)



Material	Yield strength, MPa	Tensile strength, MPa	$M_{0.2}$
Parent pipe	797	844	1
OM weld metal*	877	928	1.1
GMAW weld metal	661	751	0.83
FCAW weld metal	546	635	0.685

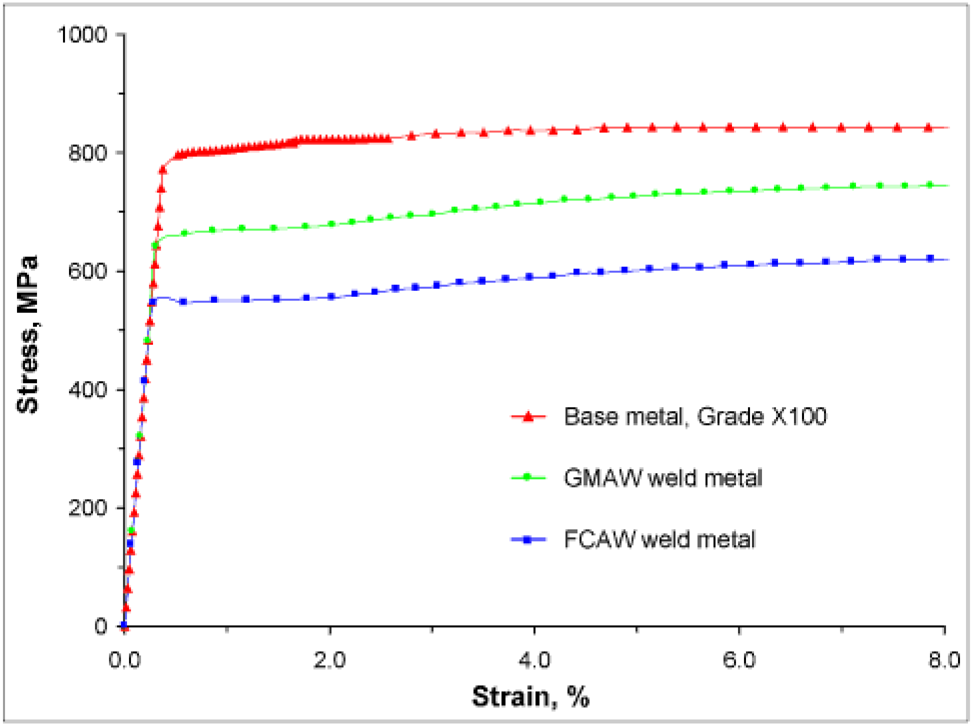


Figure 2-28: Experimental stress strain carves of parent pipe and GMAW, FCAW weld metal (Pisarski, Tkach & Quintana 2004)

2.13 Visualise stress and deformation

Founded in 1828, Bureau Veritas is a global leader in testing, inspection and certification. In the marine industry, Bureau Veritas as an official certification body (SIEMENS 2015). Bureau Veritas provides numerical stress analysis services based on the Finite Element Method. FEA calculates and visualises stresses and deformations resulting from applied loads in Figure 2-29. With stresses accurately calculated, actual safety margins over material strength can be determined. This approach to design is recognised within the design standards as being an appropriate alternative to manual calculation. Complex geometries may be accurately assessed and time dependant load histories may be applied. Loads may include force, restraint, pressure, temperature, gravity and dynamic loads.

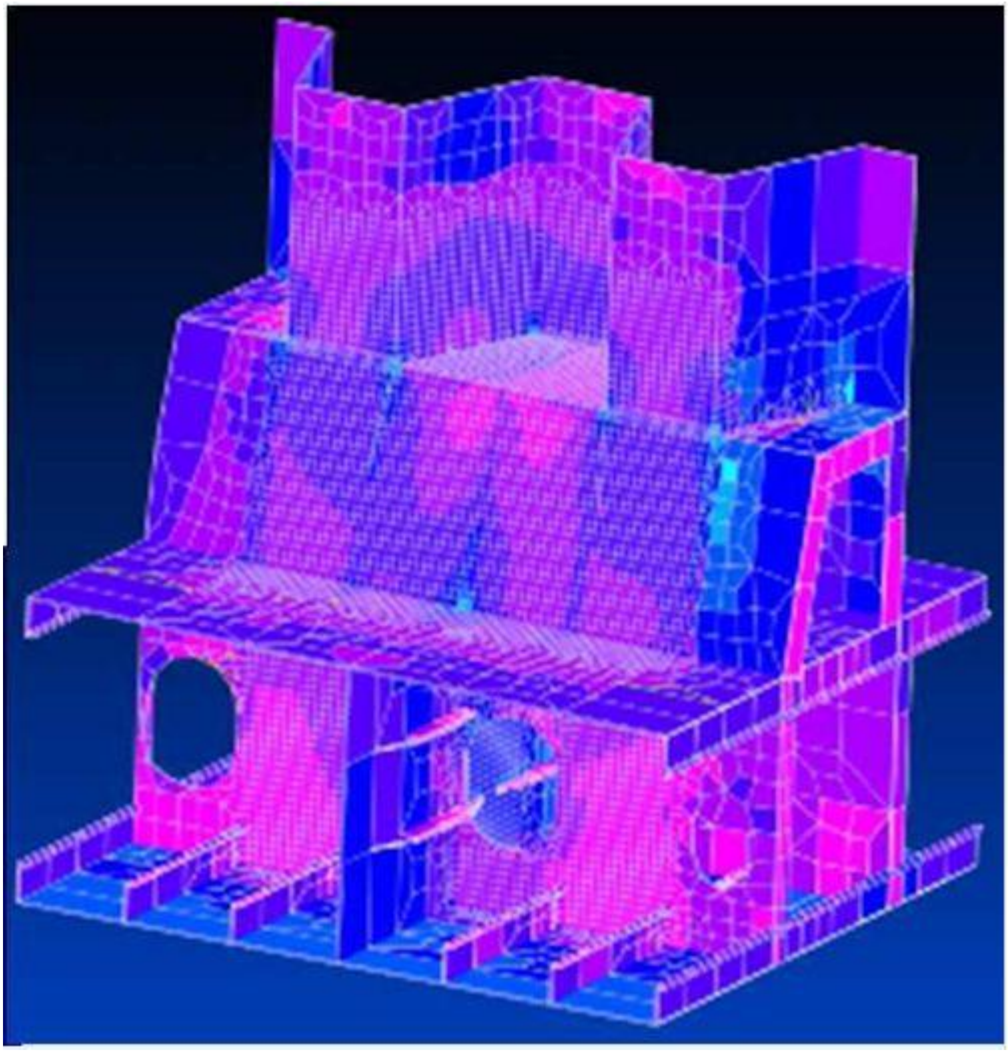


Figure 2-29: Bureau Veritas use FE map to help designs conform to regulations (SIEMENS 2015)

2.14 Adaptive mesh technique

(Qingyu et al. 2002) have developed an adaptive mesh technique applied in the three-dimensional numerical simulation of the welding process on the basis of the commercial software MARC. The adaptive mesh technique generates a dense mesh and makes it move simultaneously with the heat source. Any part of the mesh away from the heat source is much coarser, significantly saving CPU time. The calculation time comparison shows that the adaptive mesh technique can reduce the CPU time by almost one-third.

In traditional finite element analysis, as the number of element increases the accuracy of solution will be improves. Mesh density will be chosen when the result difference will be very less from one elements size to another elements size. Example of the

Figure 2-30 shown, where, a 2D bracket model is constrained at its top end and subjected to a shear load at the edge on the lower right. This generates a peak stress in the fillet, as shown. The curve shows that as the mesh density increases, the peak stress in the fillet increases. Ultimately, increasing the mesh density further produces only minor increases in peak stress. In this case, an increase from 1134 elements per unit area to 4483 elements per unit area yields only a 1.5% increase in stress (Hale 2014).

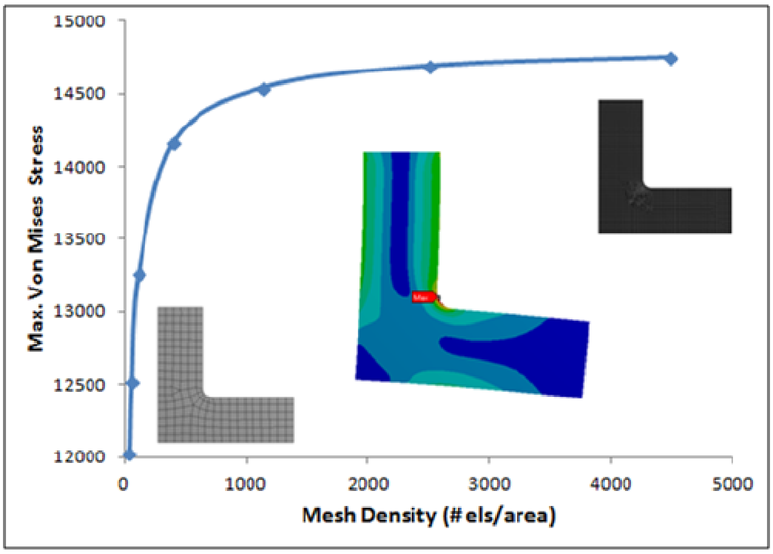


Figure 2-30: Stress sensitivity to mesh density (Hale 2014)

The difference results between coarse mesh and finer mesh found from above investigation in Figure 2-31

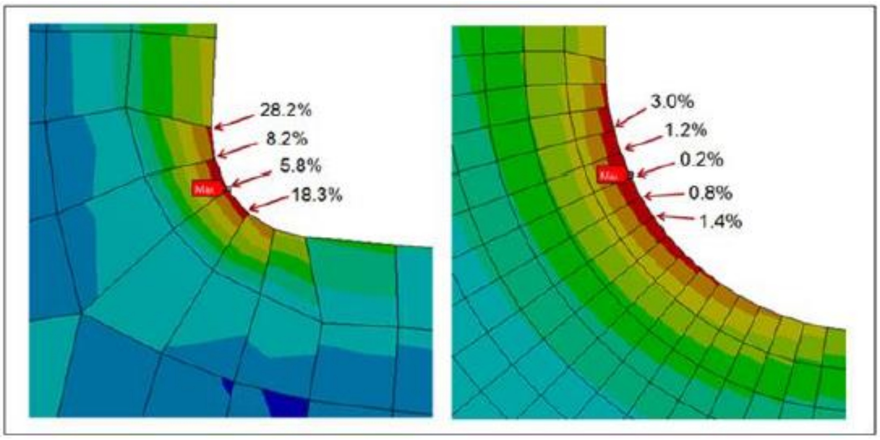


Figure 2-31: Relative difference in stresses at shared nodes for coarse mesh (L) and finer mesh (R) (Hale 2014)

2.15 Further Research

Fillet weld joints are one of the mostly used welding joints in the industry. Current body of literatures suggests considerable effect of thermal and cyclic loading on fillet weld joints. Numerous studies are dedicated to characterise the behaviour of these types of joints. Complexity of dimensions and physical parameters possess a challenge for the researchers. However, there are lack of study on fillet weld joints comparing experiment test and finite element analysis in light of nominal stress method and hot spot method. This study exerts an effort to actualise this behaviour to further enrich this domain of knowledge. This research will allow us to justify uses of finite element tools for this kind of applications.

Chapter 3 – Methodology and Project planning

3.1 Introduction

This chapter details the specimens used as well as the testing methods and processes used in both simulation and physical investigation of fatigue life of T-joint fillet welded structure. In the simulation define parameter of thermal and mechanical behaviour of the joints. In the physical test will be used maximum load to disturb the object and compare with simulation result. Also covered in this chapter is a guideline into the finite element analysis methods and procedures used to computationally investigate/correlate the grating properties and data. The design process is to over view of the project research that carried out to complete the finite element analysis and physical investigation.

3.2 Methodology

This study comprised of experimental investigations and computer modelling of fillet welding. The double sided T-section of structural steel welded specimen were prepared for tensile test. Computer modelling used the finite element based commercial package of ANSYS Static Structural.

3.2.1 Experimental Investigation

The preparation of experimental specimens and the procedure of experiment is discussed in this section.

3.2.1.1 Experiment preparation

The experiment made from standard structural steel grade 250, it has 250MPa yield stress. Three pieces of plate cut from full plate with automatic gas cutting and the size chosen for the base plate one is 400mm X 150mm and other two 400mm X 150mm. All the edges of plate has been grinded to remove any kind of slug or rust. Three plate now join together using tack weld and before welding fitted three back-up plate to control deformations. Welding has been carried out using gas metal arc welding (GMAW). In the following Figure 3-1 can see the shape of specimen which is created using AutoCAD.

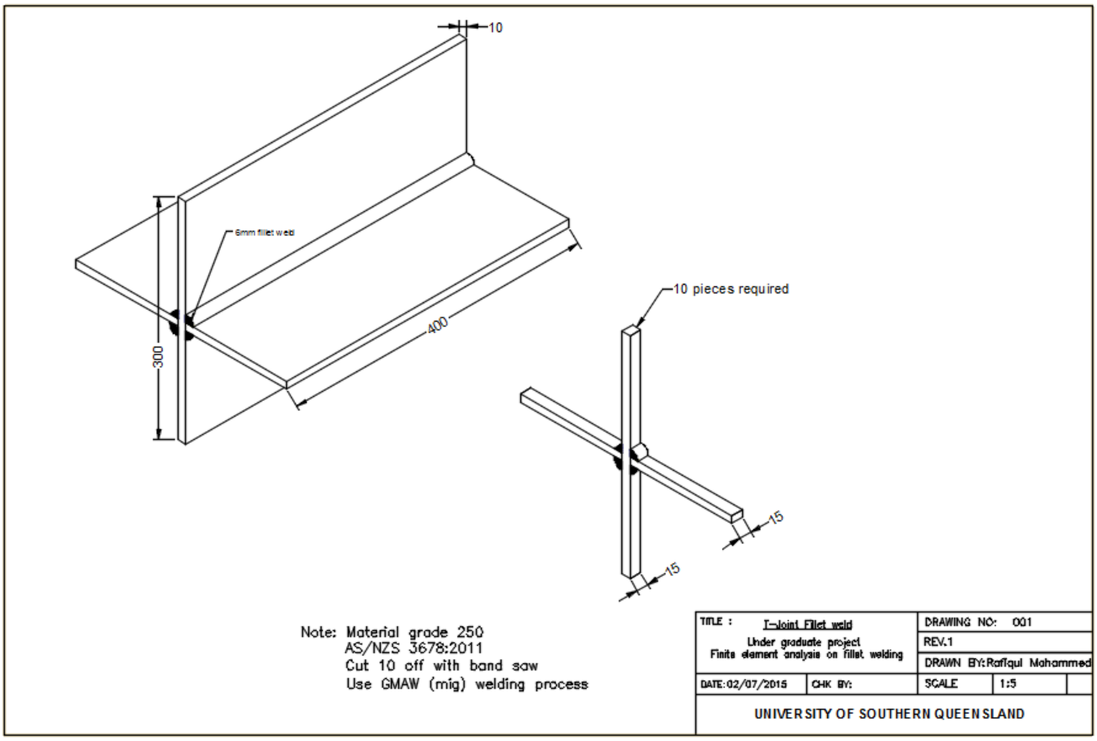


Figure 3-1: Geometry of test sample

After welding completed, the object cut-off small sections (15mm) with bend saw and grind all sharp edges.

3.2.1.2 Procedure

Tensile test is the fundamental test in the material science where we can find yield stress and ultimate stress. The test has been carried out to find yield stress of the welded joint where mostly heat passes through during welding. The load will be applied on two axis, one is on the base metal which is continue plate, this will allow us to find stress on the heat affected zone Another loading axis will be normal to the welded metal this will us to find welding strength or bonding strength. Details of applying load on the two different directions can see Figure 3-2

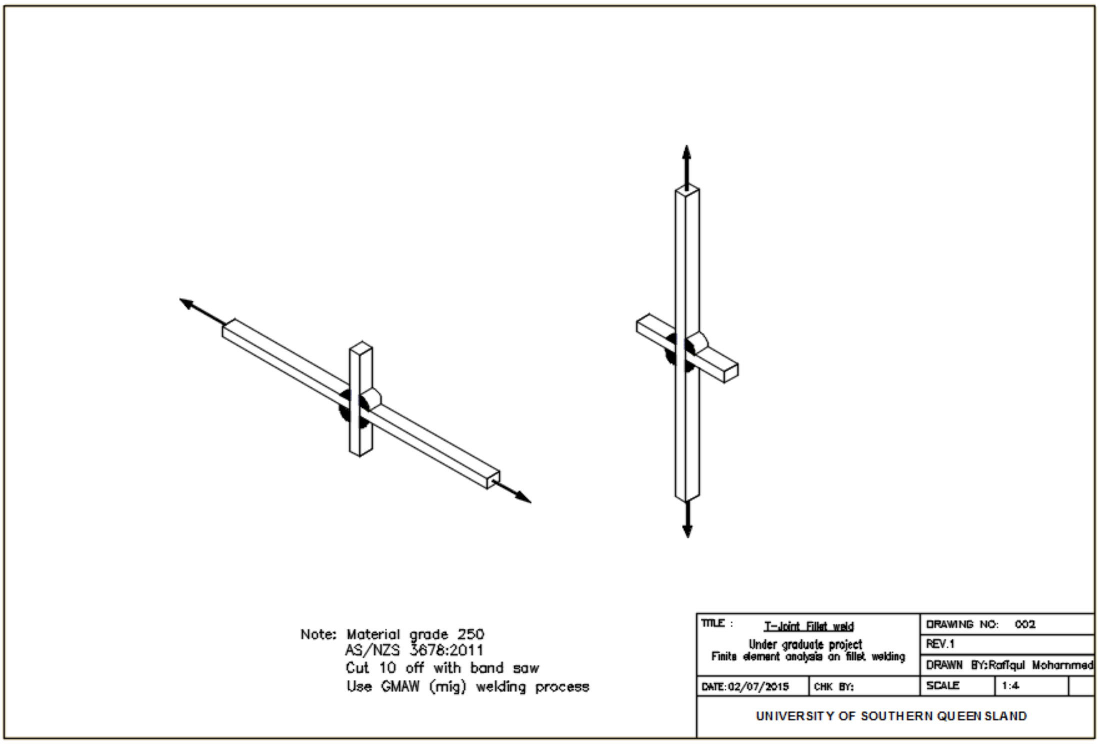


Figure 3-2: Direction of load applied

Following procedure taken to consider for the tensile test:

- Six specimens where chosen and marked 1-6 for the records and identifications
- Record down cross section area of individual pieces, where 6 pieces are not same sizes
- Visual inspection conducted to ensure that, the specimens did not have any notching or cracks
- Clean up machine parts and clamping area then started the machine
- Wait for warm up and set graph paper in to the barrel for load deflection graph and make sure pen is working condition
- Before loading the specimen make sure safety glasses is worn
- Load specimen and set dial indicator to zero position then apply load, slowly increase the load and wait until the necking started and increase the load to break the specimen
- Switch down to idle position and take off graph paper and rest the machine for next test.

3.2.2 Finite Element analysis

The section presents finite element model development, creating fillet weld, methodology of mesh element size development and model set-up. Meshing element size define with various number of elements and graphical presentation also made.

3.2.2.1 Model development

Model development is the crucial part of the analysis, if the model parameter is not match with the experiment then desire results will not match with the simulation results. The dimension of the model is taken 15mm width and 10mm thick of double sided T-joint. The model separated in three main parts which is body, heat effected zone and weld material. Reason to do that its can apply different type of material property. Figure 3-3 drawn in PTC Creo parametric and for analysis Creo model will be exported to ANSYS for the simulation.

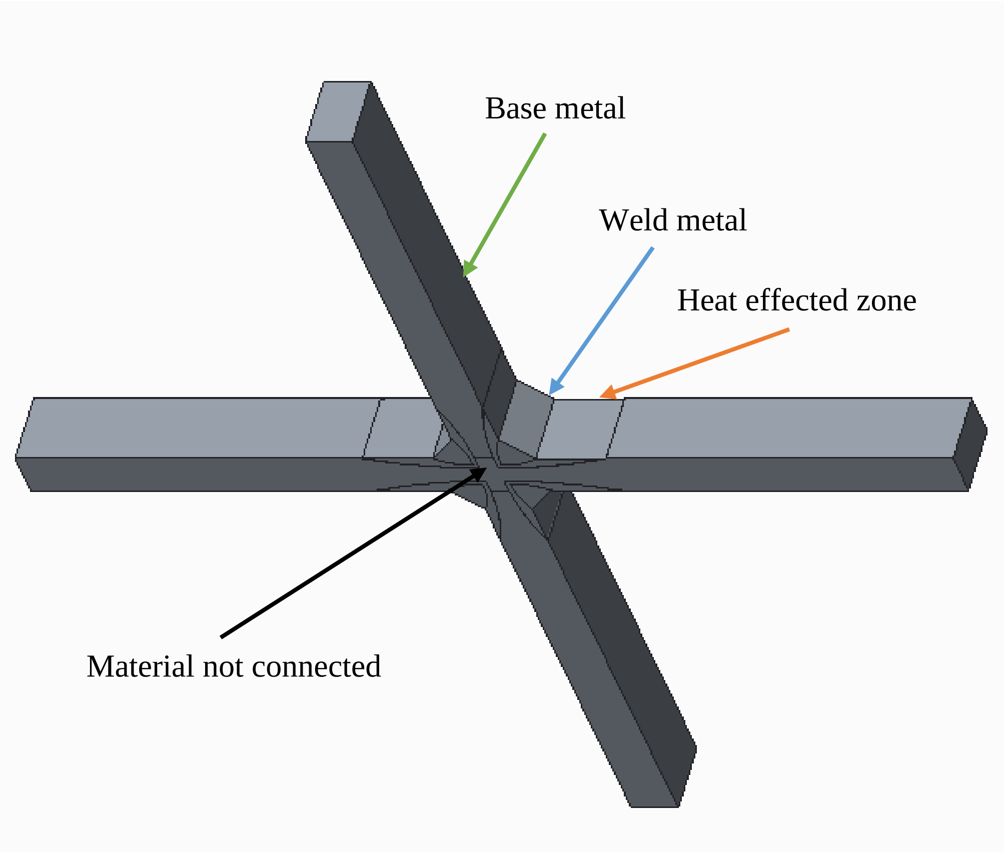


Figure 3-3: basic model of welded joint

The following Figure 3-4 is the exploded view of the model where can find the different parts of the model.

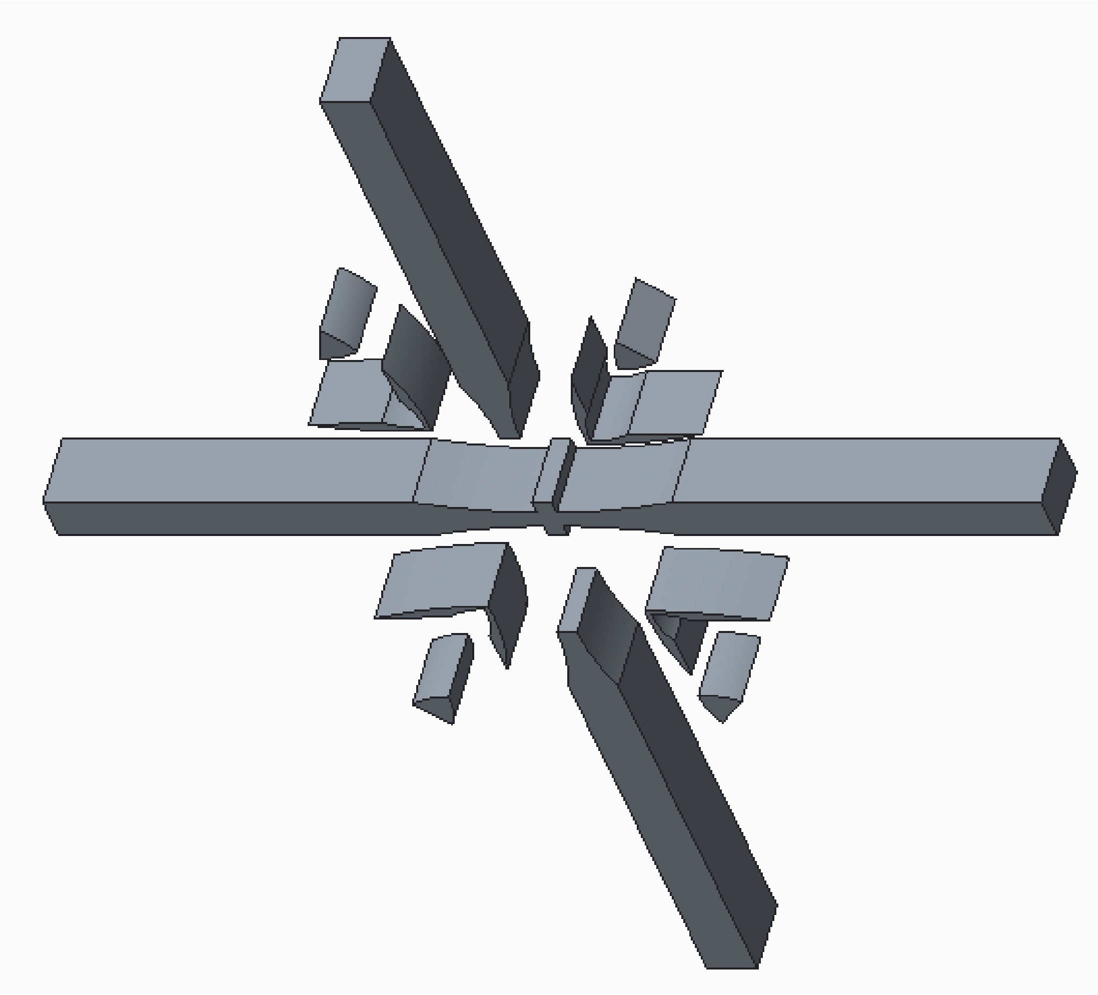


Figure 3-4: Exploded view of the model

3.2.2.2 Creating fillet weld

In order to create fillet welding in the model, three pieces of plate will not connected together with one another. The connected area will be the welded material only. In this model base plate and welded metal will be different parts. In the connection tab from model tree can find contact region and suppressed plate connection to disconnect the surface. After applying load Figure 3-5 can see that there is no connection between plates except welding.

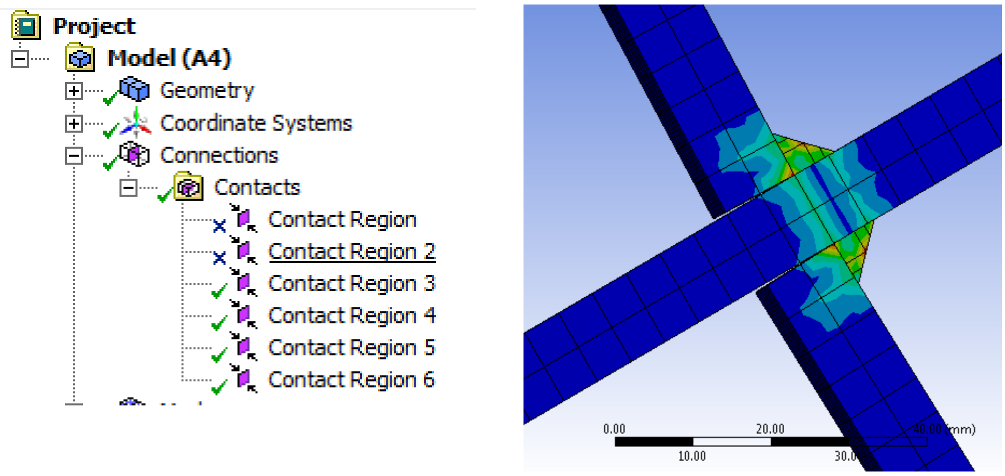
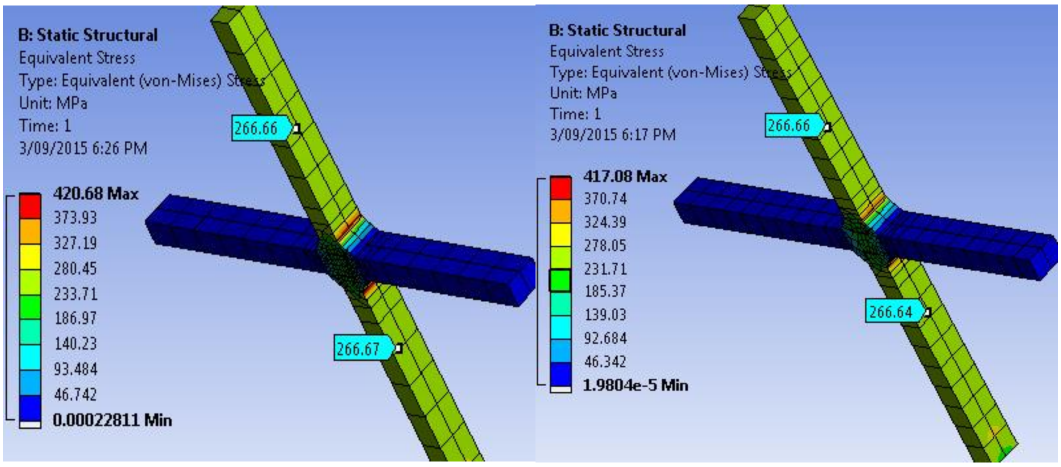


Figure 3-5 Proof of fillet welding

3.2.2.3 Meshing

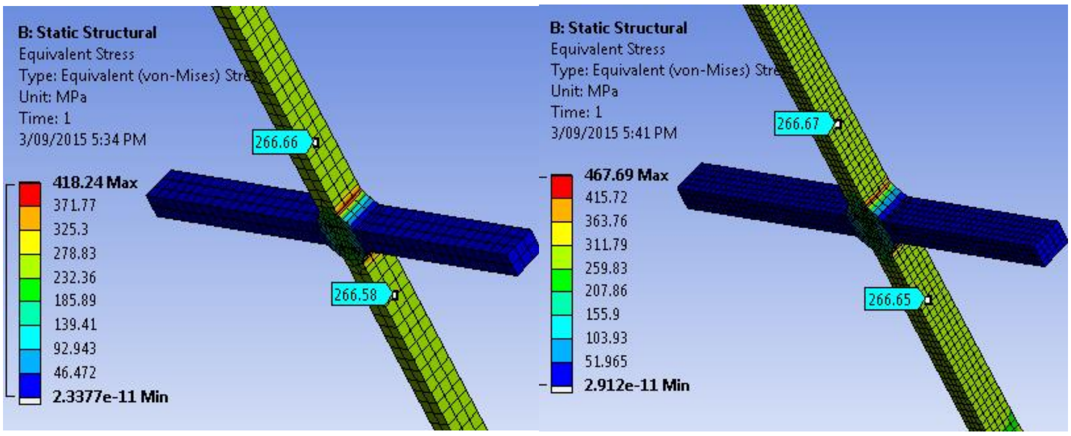
In traditional finite element analysis we know that, the number of element increases the accuracy of solution will be improved also it is not necessary to put element size very small. Smaller element size will takes longer time to analysis and some time it is almost impossible to run the model.

In order to find optimum element size I have chosen different element size and analyse the model, where the stress are not changing dramatically assume that is the optimum size. Following Figure 3-6 and Figure 3-7 can see how different element size vary the stress with same applying load.



Default mesh size

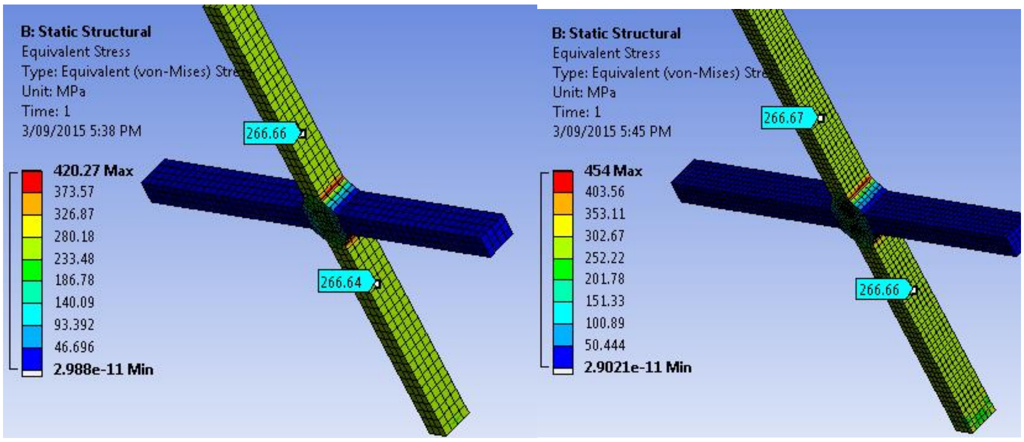
Mesh size 10mm



Mesh size 5mm

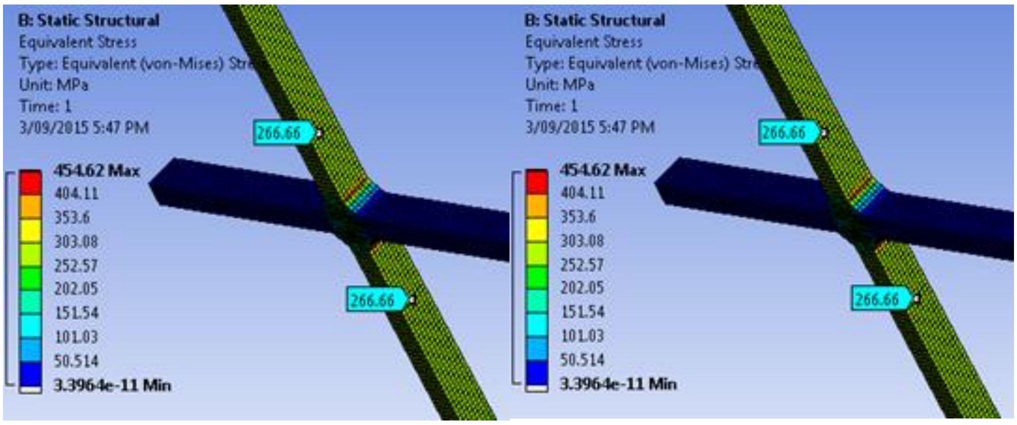
Mesh size 4mm

Figure 3-6: Mesh development



Mesh size 3mm

Mesh size 2.5mm



Mesh size 2mm

Mesh size 1.5mm

Figure 3-7: Mesh development

The stress concentration on red colour location is very high, its dose not display actual results because the area is very small compare with cross section and its only 0.5 mm. The grinding operation can be removed sharp concentration areas. In the Figure 3-8 shows the stresses are changing due to different size of element.

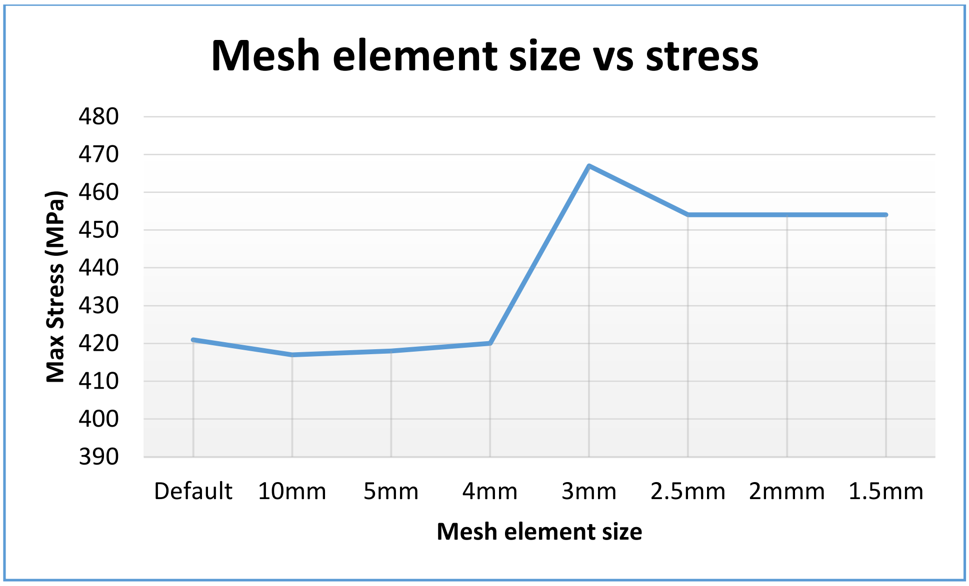


Figure 3-8: Mesh element size Vs stress

Above figure can identify that element size between 2.5mm to 1.5mm where the stress haven’t been changes. There for the adequate mesh element size can be considered 2.5mm to 1.5mm.

3.2.2.4 Model set-up

There are many different parameter has been set-up for the model in the Figure 3-9 is the model tree of the model set-up.

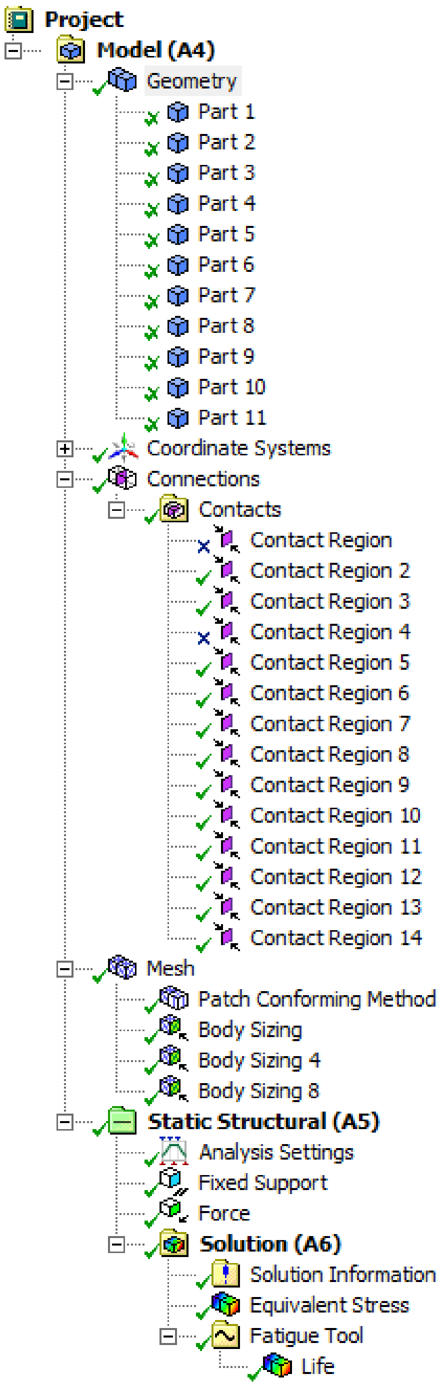


Figure 3-9: Model set-up tree

There are 11 different parts in the model and its sets different material property. Base metal used structural steel which tensile yield strength 250 MPa, for heat affected zone used 350 MPa due to rapid heating and cooling process, and for the weld metal strength used 450 MPa (*Austmig ES6*) which provided Austmig ES6 material datasheet. Contact region sets two suppress to make disconnect surface between the plates. In the mesh set-up tab, all mesh is patch conforming method and different body has sets different element size.

Chapter 4 – Finite Element Simulation

4.1 Introduction

In this chapter included basic simulation on a double sided T-joint fillet welding using ANSYS workbench. The property of base metal chosen structural steel material of 250 MPa, for heat affected zone used 350 MPa yield strength and for weld material used 450 MPa yield strength. Result set-up for stress chosen equivalent stress and fatigue life. Adequate mesh applied to the model.

4.2 Background

Finite element analysis (FEA) is the modelling of products and systems in a virtual environment, for the purpose of finding and solving potential (or existing) structural or performance issues (SIEMENS 2015). FEA is the practical application of the finite element method (FEM), which is used by engineers and scientist to mathematically model and numerically solve very complex structural, fluid, and multi physics problems.

A finite element model comprises a system of points, called “nodes”, which form the shape of the design. Connected to these nodes are the finite elements themselves which form the finite element mesh and contain the material and structural properties of the model, defining how it will react to certain conditions. The density of the finite element mesh may vary throughout the material, depending on the anticipated change in stress levels of a particular area. Regions that experience high changes in stress usually require a higher mesh density than those that experience little or no stress variation. Points of interest may include fracture points of previously tested material, fillets, corners, complex detail, and high-stress areas.

FE models can be created using one-dimensional (1D beam), two-dimensional (2D shell) or three-dimensional (3D solid) elements. By using beams and shells instead of solid elements, a representative model can be created using fewer nodes without compromising accuracy. FEA is originally developed for solving solid mechanics problem (Qi 2006). The Finite Element Analysis is offers a mean to find the approximation, this is not an exact solution. The general procedure of FEA in Figure 4-1 shows how does it works:

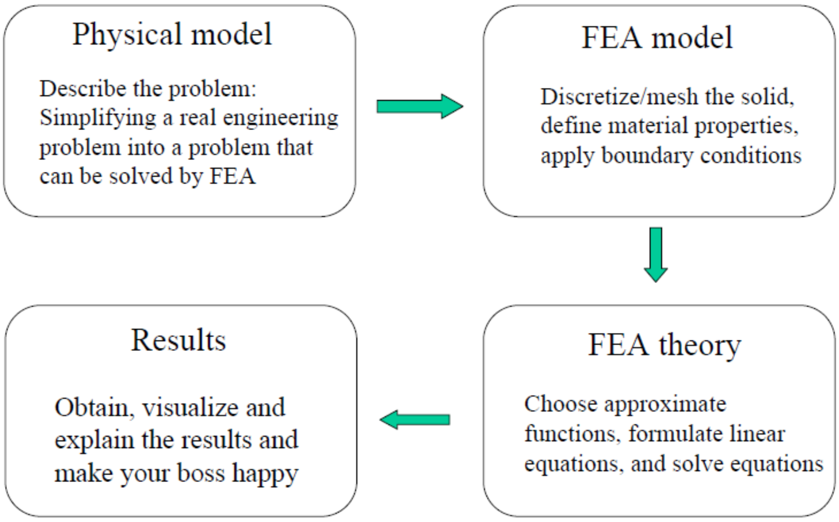


Figure 4-1: General procedure of finite element analysis (Hobbacher 2009)

In the past, there are many different way welding joints considered and analysed with different types of simulation tools such as ANSYS, ABAQUS, and SOLIDWORKS etc. Using those tools and analysed normal stress, residual stress, fatigue life calculations and deformations etc. where mostly found after welding of steel strength of metal specially fatigue life are less than original metal. The welding institute tested welding joints applying axial force and they found the design of welded joint has a dominant effect on fatigue life. Figure 4-2 can see the S/N curves for welded and un-welded specimens where welded carbon steel has less fatigue strength than un-welded steel.

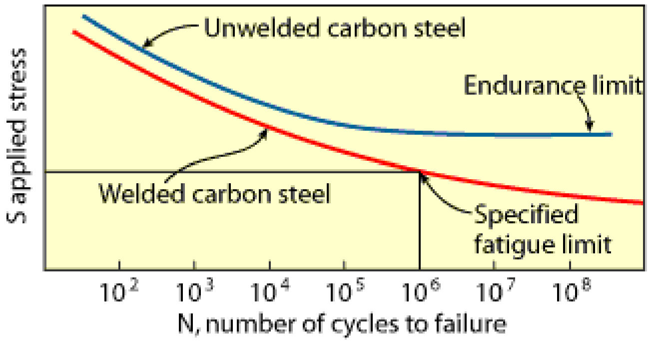


Figure 4-2 : S/N curves for welded and un-welded carbon steel (TWI Fatigue testinng 2015)

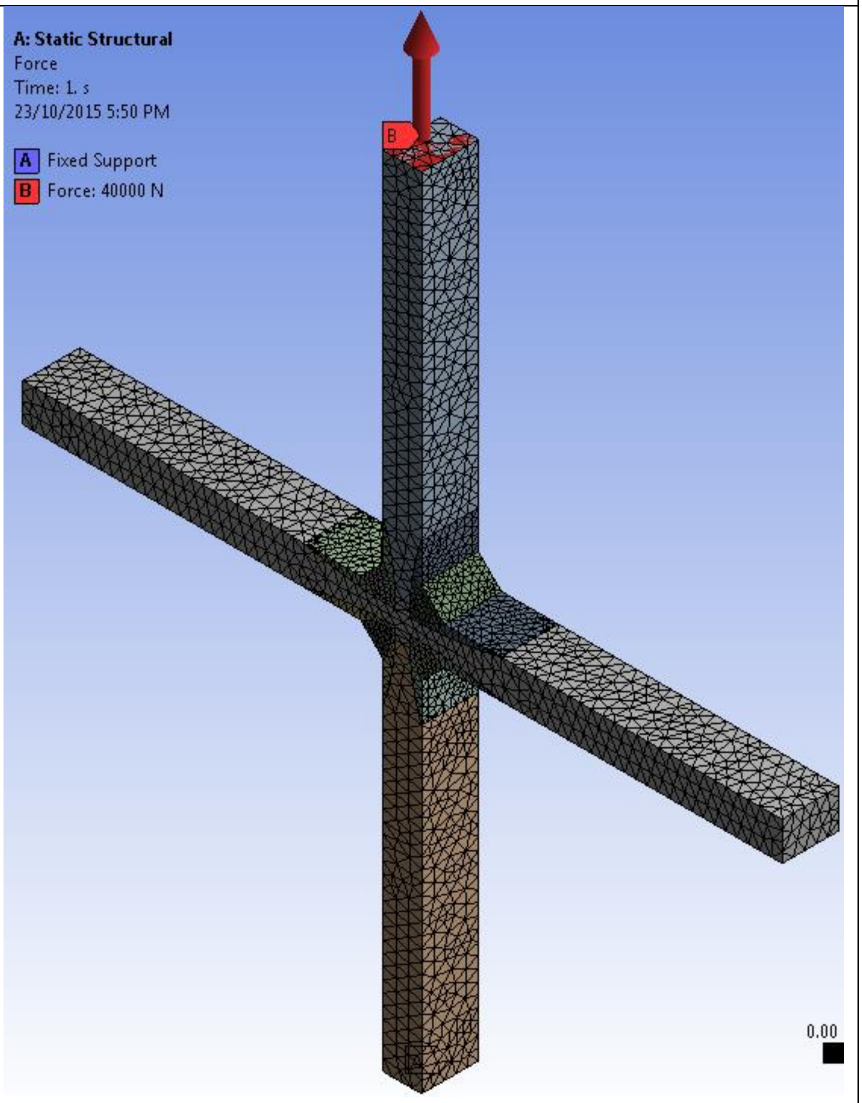
4.3 ANSYS simulation

In order to analyse the model with ANSYS the nominal stress and fatigue life calculations are rely on the ASME Boiler and Pressure Vessel Code, Section III (and Section VIII, Division 2) for guidelines on range counting, simplified elastic-plastic adaptations, and cumulative fatigue summation by Miner's rule. There are few number of method available in fatigue assessment of welded joint;

- Nominal stress method
- Hot spot method
- Fracture mechanic analysis
- Effected notch stress method

Simulation details as following

Project name	Finite element analysis of fillet welding
FEA tools & version	ANSYS 15.0.7 Release
Analysis type	Static structural
Material data	• Structural steel • HAZ material • Weld material
Unit system	Metric (mm, kg, N, mV, mA)
Length on X direction	200mm
Length on Y direction	210mm
Length on Z direction	15mm
Applied force	40000N
Connections	Total 14 connections, region 1 and 3 suppressed
Mesh	Method applied Tetrahedrons, element size 3mm for base metal, 2mm for heat affected zone and 1.5mm for welded material.
Nodes	50230
Elements	29694

Results	Maximum principal stress and Fatigue life
Loaded model	 Figure 4-3 Model with force direction and fixed support

4.3.1 Stress analysis

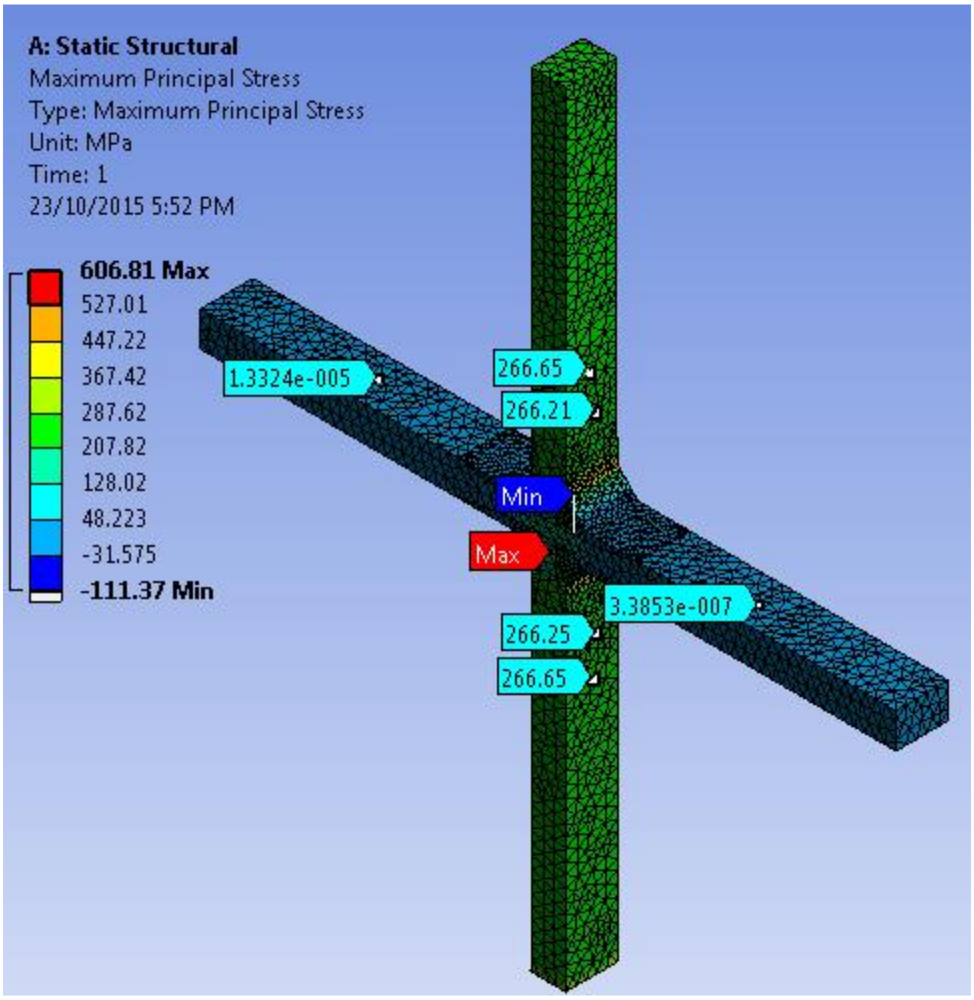


Figure 4-4: Stress distribution on the fillet welded specimen

Static structural analysis has been carried out applying maximum principal stress as shown above in Figure 4-4. Maximum stress was found at 606.8 MPa with a appled yield load of 40 kN. However, the stress in the vertical axis was 266.65 MPa which demonstrate that those regions were not affected by welding.

The maximum stress was concentrated at the toe of the welding which can identify very small area (red coloured in Figure 4-5). Approximately 1.5 mm away from the maximum stress point, the stress are significantly less than the maximum stress which was below of 260 MPa. The gap in the figure demonstrate that some part of the material are not bonded in the fillet welding.

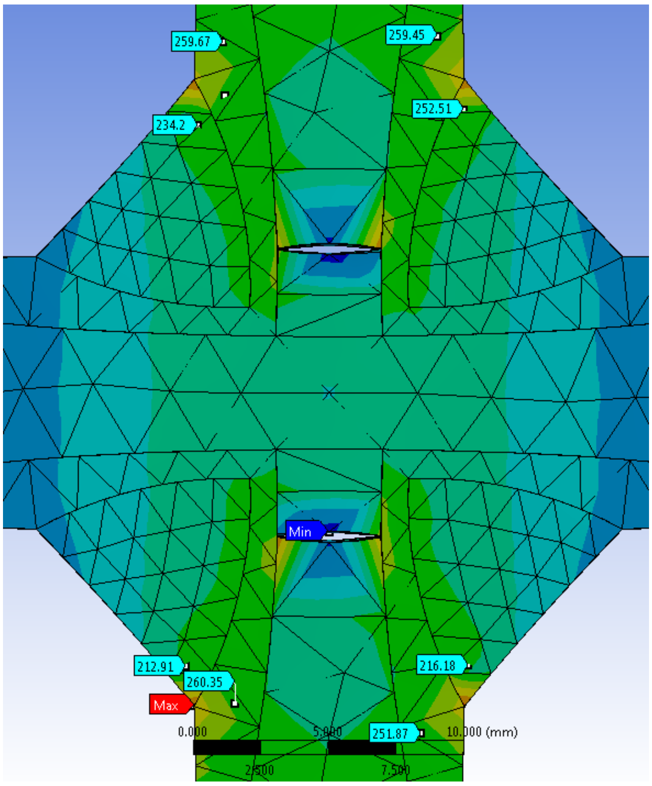


Figure 4-5: Stress distribution in the welded region

Above Figure 4-5 shows stress distribution in zoomed view. Stress appears 250-260 MPa which is very close to the yield stress. However, maximum stress 606 MPa distributed only 0.5 mm areas.

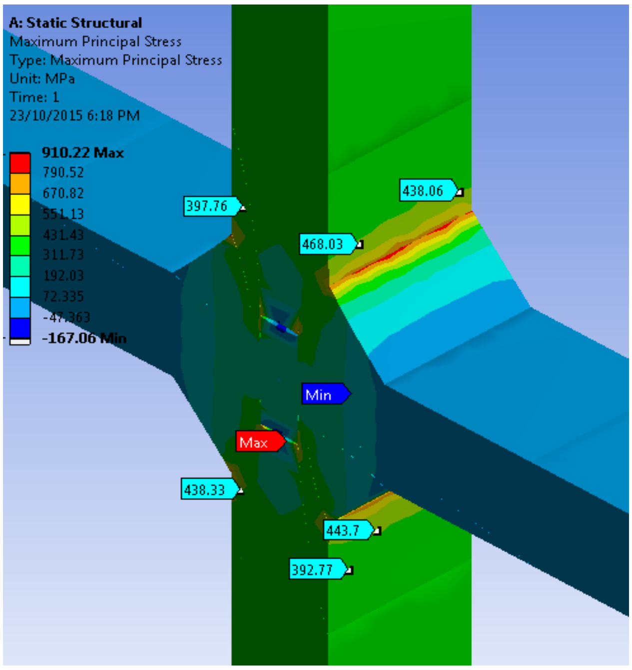


Figure 4-6: Stress distribution with 60kN load

Figure 4-6 (above) applied 60kN load, where found maximum 910 MPa stress at the concentrated area. However, the average stress found in the randomly selected location close to the concentrated area (no. 1-6 is from lower to upper value) as follows:

Table 4-1: Calculation of stress with 60kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	393	444	438	468	398	438	**430**

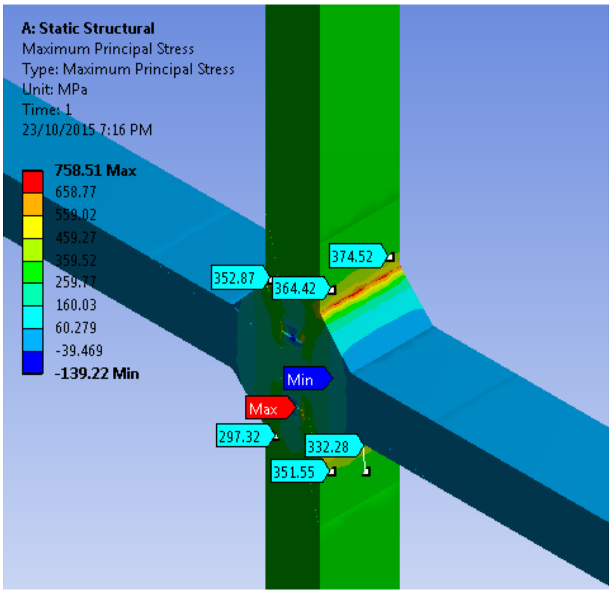


Figure 4-7: Stress distribution with 50kN load

Figure 4-7 (above) applied 50kN load and maximum stress found 758 MPa at the concentrated area. However, the average stress found as follows:

Table 4-2: Calculation of stress with 50kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	351	332	297	364	352	374	**345**

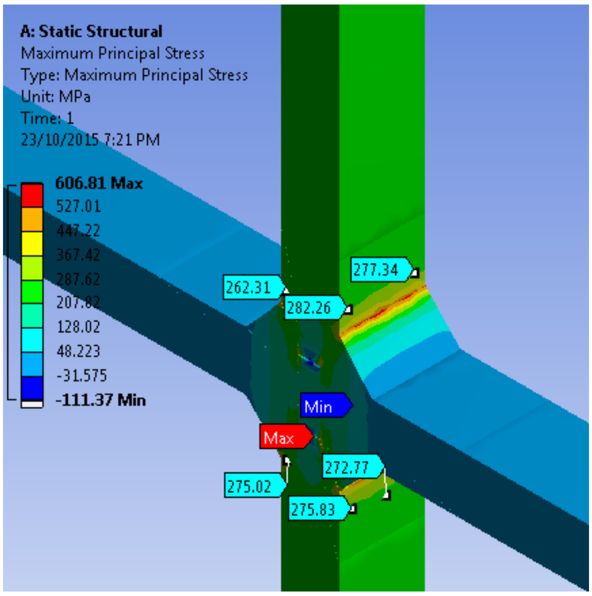


Figure 4-8: Stress distribution with 40kN load applied

Figure 4-8 (above) applied 40kN load where found 606 MPa stress at the concentrated area on red coloured marked. 40kN load was found in the experiment as its yield load. However, the average stress in surrounding found as follows:

Table 4-3: Average stress of 40kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	276	275	273	282	262	277	**274**

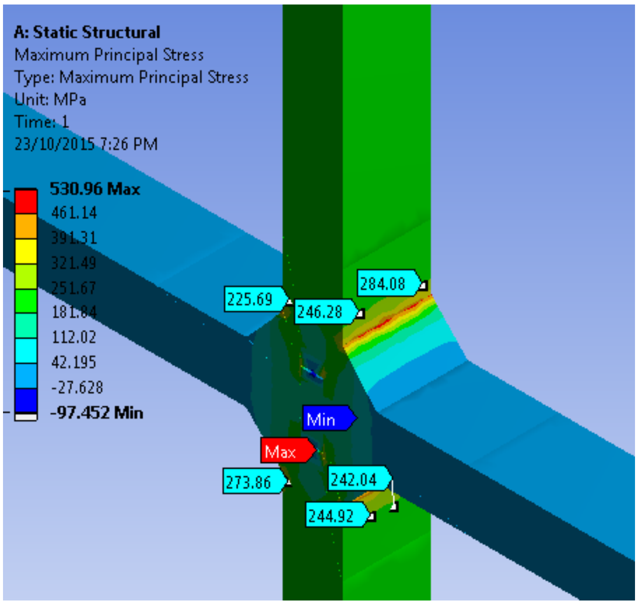


Figure 4-9: Stress distribution with 35kN load applied

Now load reduction interval reduced by 5kN for better result. Figure 4-9: (above) applied load is 35kN, where maximum 530 MPa stress found at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-4: Average stress of 35kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	245	242	273	246	225	284	**252**

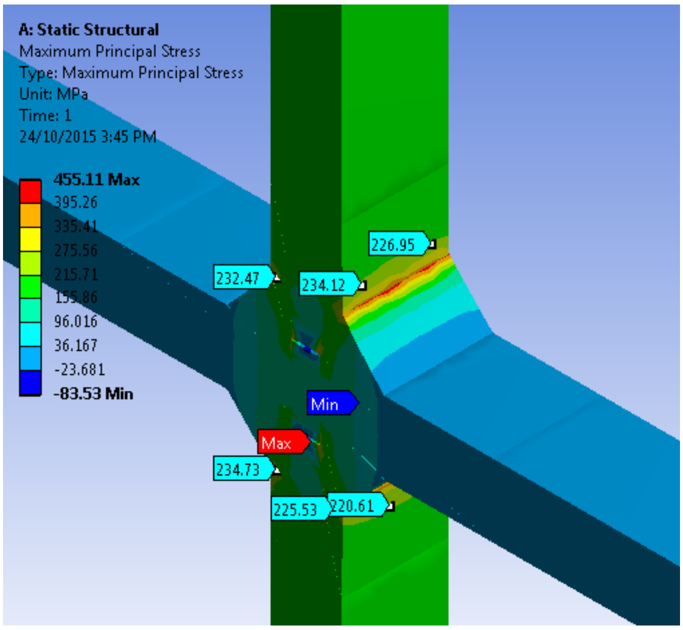


Figure 4-10: Stress distribution with 30kN load

Figure 4-10 (above) applied load is 30kN, maximum 430 MPa stress found at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-5: Average stress of 30kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	225	220	234	234	232	226	**235**

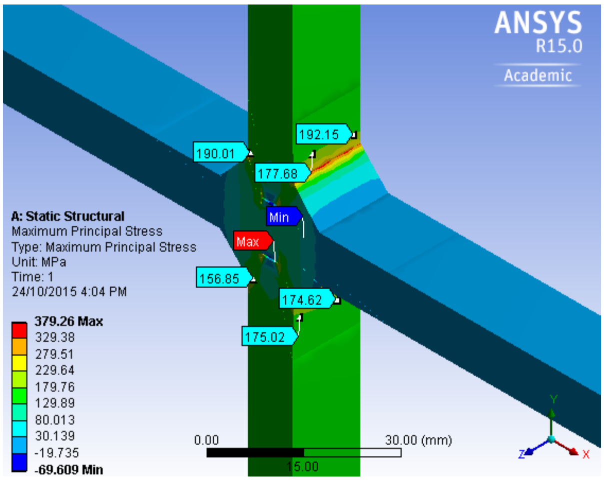


Figure 4-11: Stress distribution with 25kN load

Figure 4-11 found maximum stress 358 MPa with applied load 25kN. The maximum stress shows at the inner part of the welding where weld metal started fusion. Actual stress occurs surrounding to the red coloured marks. The average stress as follows:

Table 4-6: Average stress of 25kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	166	164	163	219	197	174	**180**

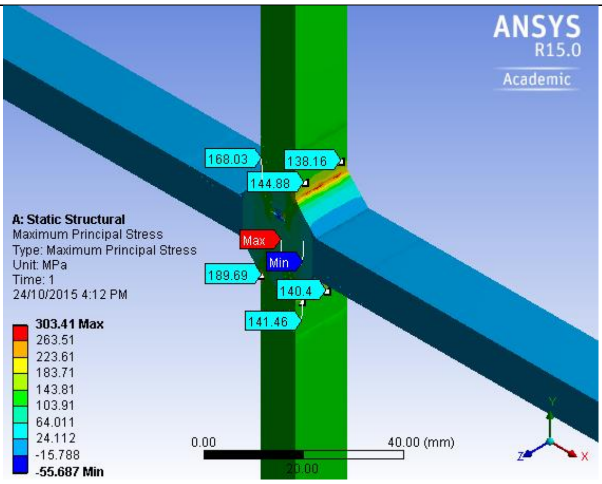
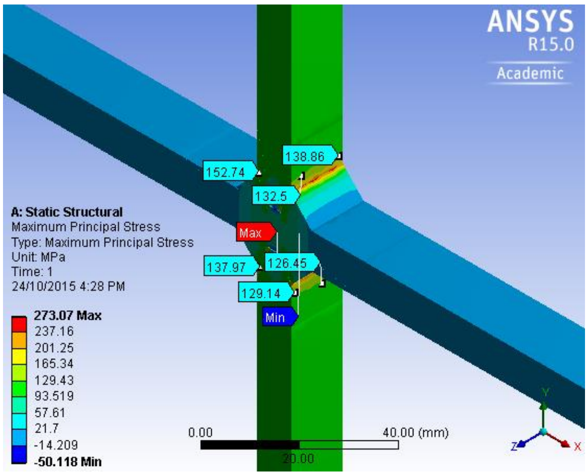


Figure 4-12: Stress distribution with 20kN load

Figure 4-12: (above) applied 20kN load, where found 303 MPa stress at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-7 Average stress of 20kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	141	140	189	144	138	168	**153**



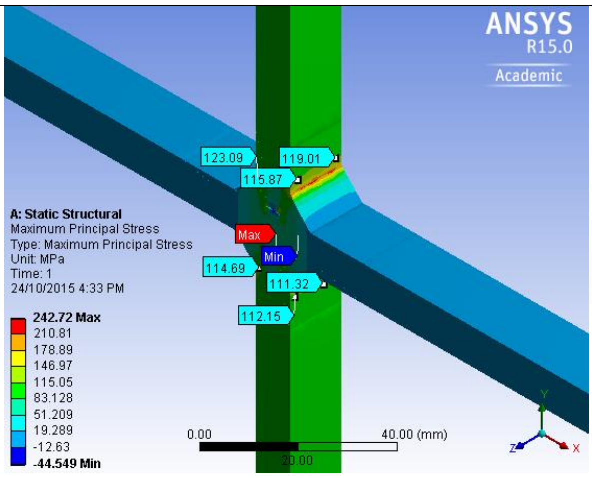
A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: MPa
Time: 1
24/10/2015 4:28 PM

Figure 4-13: Stress distribution with 18kN load

Figure 4-13: (above) applied load is 18kN, maximum 273 MPa stress found at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-8: Average stress of 18kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	129	126	138	132	152	139	**136**



A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: MPa
Time: 1
24/10/2015 4:33 PM

Figure 4-14: Stress distribution with 16kN load

Figure 4-14: found maximum stress 229 MPa with applied load 16kN. Now reduce load by 2kN only to get better results. The maximum stress shows at the inner part of the welding where weld metal started fusion. Actual stress occurs surrounding to the red colour marks. The average stress as follows:

Table 4-9: average stress of 16kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	112	111	114	115	123	119	**115**

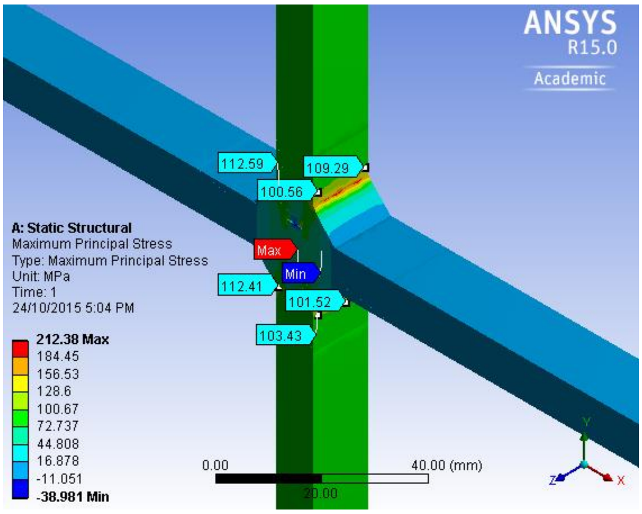


Figure 4-15: Stress distribution with 14kN load

Figure 4-15 (above) applied load is 14kN, maximum stress found 212 MPa at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-10: Average stress of 14kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	103	101	112	100	109	112	**106**

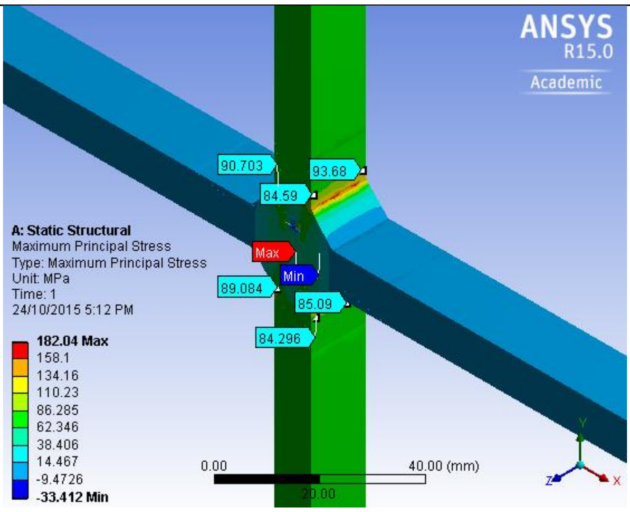


Figure 4-16: Stress distribution with 12kN load

Figure 4-16 (above) applied load is 14kN, maximum stress found 182 MPa at the concentrated area. However, the average stress in surrounding found as follows:

Table 4-11: Average stress of 12kN load

Location	1	2	3	4	5	6	Average
Stress (MPa)	84	85	89	85	94	91	**88**

4.3.2 Fatigue life analysis

In order to predict fatigue life of the structure, in this analysis chosen fatigue tool from solution tab and selected life. The loading type used fully reversed constant amplitude and stress life with Goodman mean stress theory. Finite element analysis starts with a structural simulation to calculate the how many repeated load can be applied without disruption of structures. Following figure shows the fatigue life in terms of cycle and result taken using probe selection on the model.

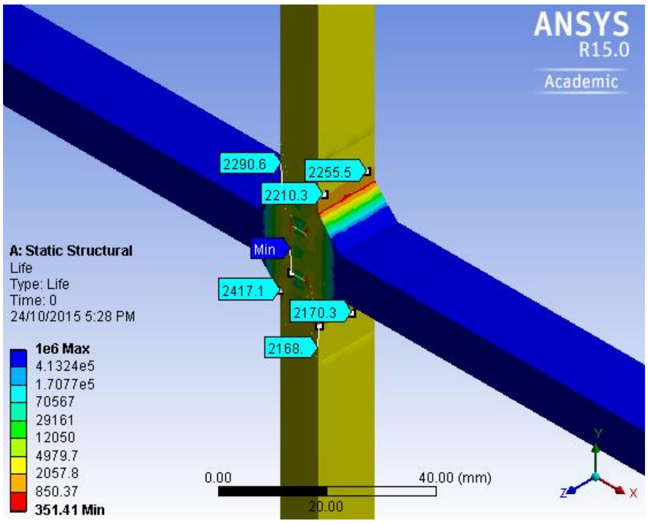


Figure 4-17 Fatigue life with 60kN load

Figure 4-17 (above) shows the calculated fatigue life, applying 60kN forces which is ultimate load. The red spotted line shows minimum 351 cycles of alternating load can be applied. Average number of cycles in Table 4-12 found in the surrounded area of red coloured line which is more realistic and area is more than the concentrated area.

Table 4-12: Calculation of fatigue life with 60kN load

Location	1	2	3	4	5	6	Average
No. of cycle	2713	2433	2578	2016	2344	2689	**2462**

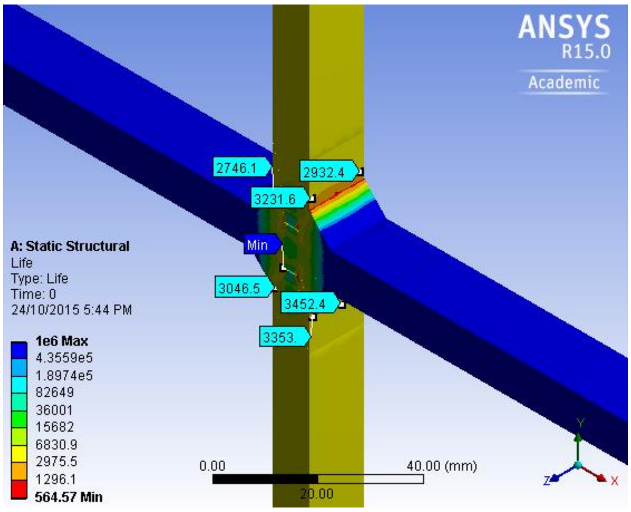


Figure 4-18 Fatigue life with 50kN load

Figure 4-18 (above) shows the calculated fatigue life from simulation at 50kN load which is ultimate load. Minimum 564 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-13: Calculation of fatigue life with 50kN

Location	1	2	3	4	5	6	Average
No. of cycles	3353	3452	3046	3231	2932	2746	**3127**

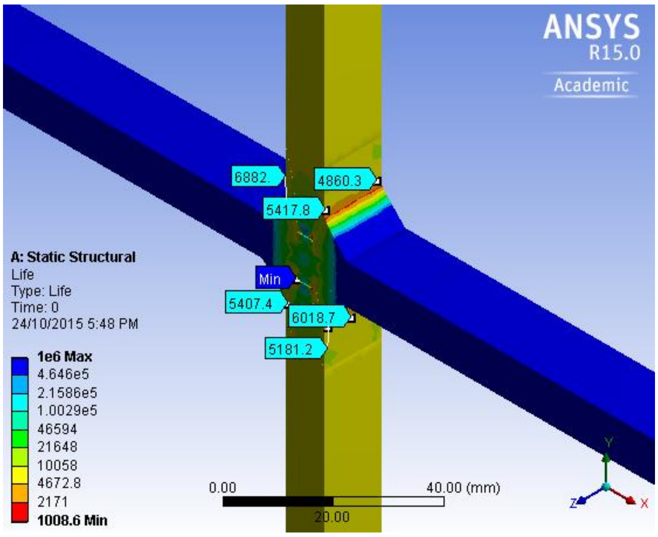


Figure 4-19 Fatigue life with 40kN load

Figure 4-19 (above) shows the calculated fatigue life from simulation at 40kN load which is ultimate load. Minimum 1008 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-14: Calculation of fatigue life with 40kN

Location	1	2	3	4	5	6	Average
No. of cycles	5181	6018	5407	5417	4860	6882	**5627**

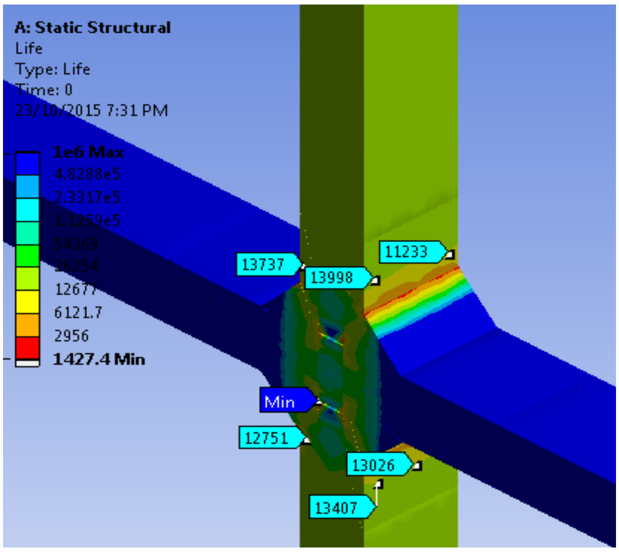


Figure 4-20 Fatigue life with 35kN load

Figure 4-20 (above) shows the calculated fatigue life from simulation at 35kN load which is ultimate load. Minimum 1427 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-15: Calculation of fatigue life with 35kN

Location	1	2	3	4	5	6	Average
No. of cycles	13407	13026	12751	13998	13737	11233	**13025**

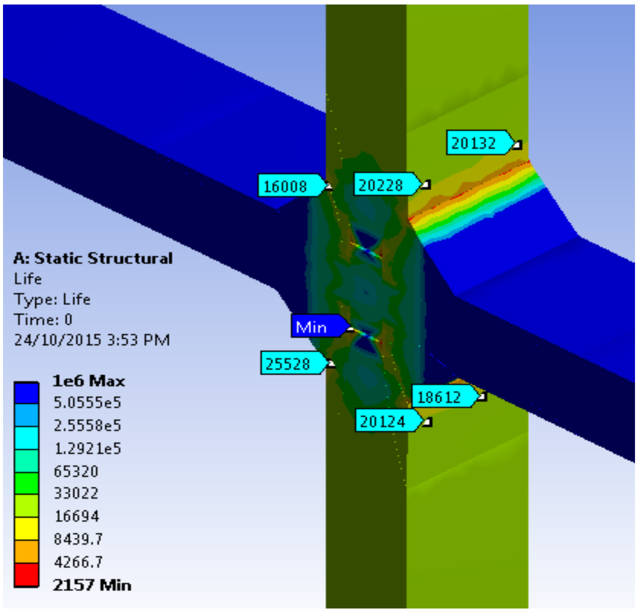


Figure 4-21 Fatigue life with 30kN load

Figure 4-21 (above) shows the calculated fatigue life from simulation at 30kN load which is ultimate load. Minimum 2157 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-16: Calculation of fatigue life with 30kN

Location	1	2	3	4	5	6	Average
No. of cycles	20124	18612	25528	16008	20228	20132	**20105**

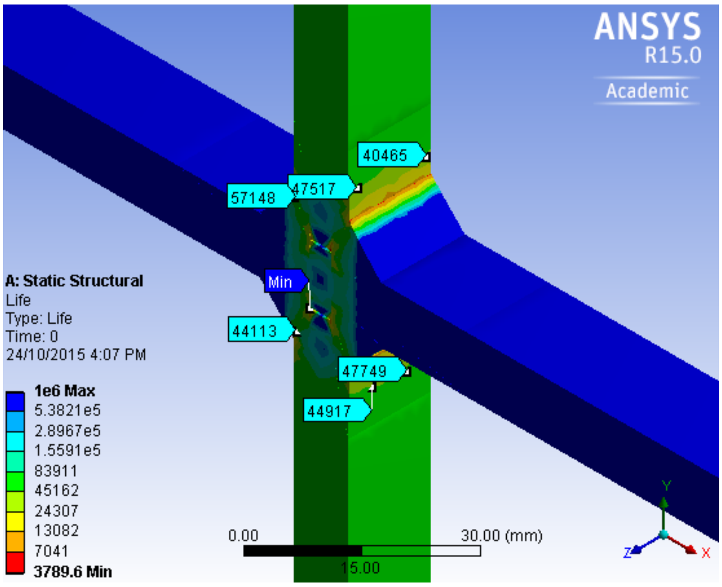


Figure 4-22 Fatigue life with 25kN load

Figure 4-22 (above) shows the calculated fatigue life from simulation at 25kN load which is ultimate load. Minimum 3789 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-17: Calculation of fatigue life with 25kN

Location	1	2	3	4	5	6	Average
No. of cycles	44917	47749	44113	57148	47517	40465	**46985**

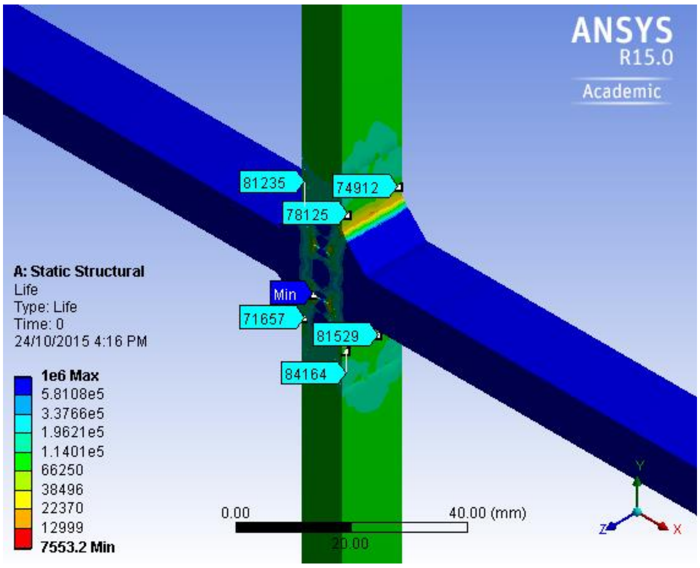


Figure 4-23 Fatigue life with 20kN load

Figure 4-23 (above) shows the calculated fatigue life from simulation at 20kN load which is ultimate load. Minimum 7553 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows;

Table 4-18: Calculation of fatigue life with 20kN

Location	1	2	3	4	5	6	Average
No. of cycles	84164	81529	71657	78125	81235	74912	**78603**

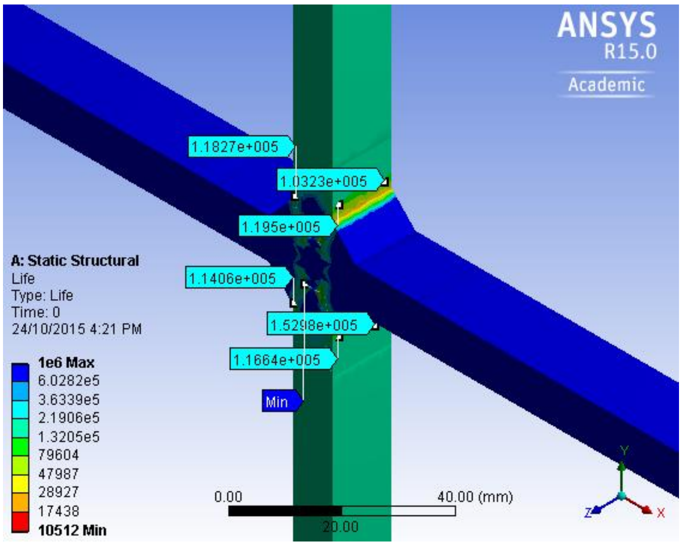


Figure 4-24 Fatigue life with 18kN load

Figure 4-24 (above) shows the calculated fatigue life from simulation at 18kN load which is ultimate load. Minimum 10512 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot as follows:

Table 4-19: Calculation of fatigue life with 18kN

Location	1	2	3	4	5	6	Average
No. of cycles	116640	152980	114060	119500	103230	118270	**120780**

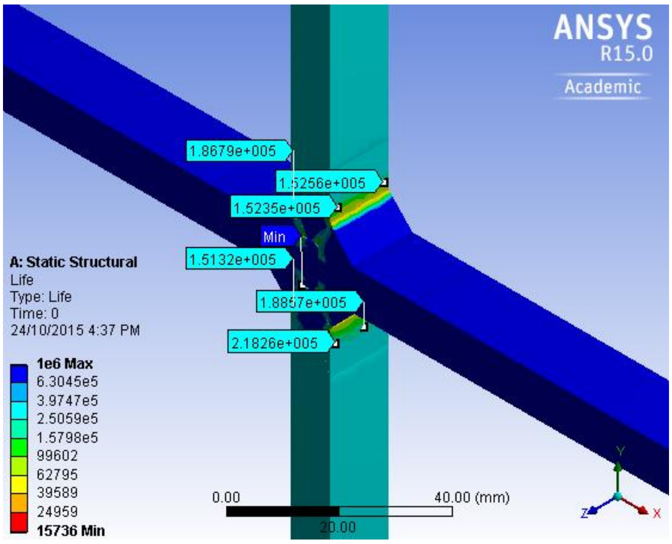


Figure 4-25 Fatigue life with 16kN load

Figure 4-25 (above) shows the calculated fatigue life from simulation at 16kN load which is ultimate load. Minimum 15736 cycles of alternating load can apply and its shows in red spot area only. Average cycles found in the surrounding of red spot is 174975 cycles.

Location	1	2	3	4	5	6	Average
No. of cycles	218260	188570	151320	152350	152560	186790	**174975**

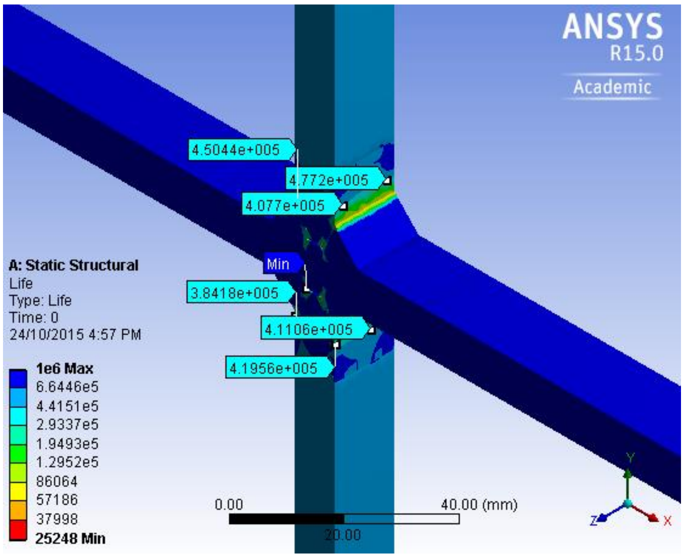


Figure 4-26 Fatigue life with 14kN load

Figure 4-26 (above) shows the calculated fatigue life from simulation at 14kN load which is ultimate load. Minimum 25248 cycles of alternating load can apply and its shows in red spot area only. Average cycle found in the surrounding of red spot is 425023 cycles.

Location	1	2	3	4	5	6	Average
No. of cycles	419560	411060	384180	407700	477200	450440	**425023**

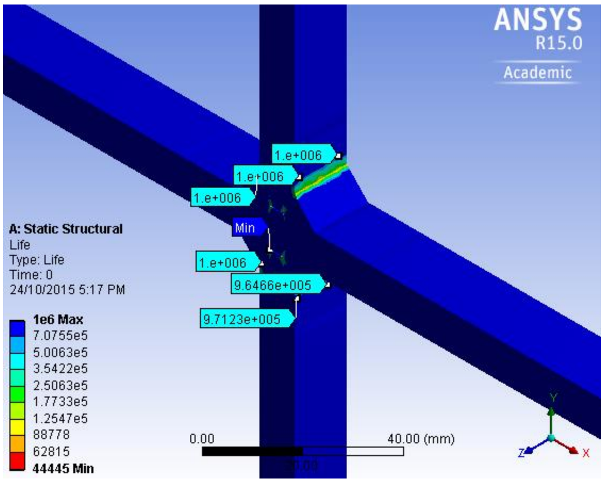


Figure 4-27 Fatigue life with 12kN load

Figure 4-27 (above) shows the calculated fatigue life from simulation at 12kN load which is ultimate load. Minimum 44445 cycles of alternating load can apply and its shows in red spot area only. Average cycle found in the surrounding of red spot is 1000000 cycles. The million cycle consider to be infinite life according to ASME.

Chapter 5 – Results and Discussion

5.1 Introduction

This chapter includes computer simulations results, experiment test results and discussions. Discussions carried out for how the results are interpreted and justify the methods that set up in the methodology. Limitations of the project was the critical part of the discussion.

5.2 FEA Simulation results

In order to find stress of the welded joint, FEA simulations set up for maximum principal stress. There are many different load combination has been used in the simulations. However, this project compare yield stress of the experiment yielding location. During simulation two different results has been deployed one is stress and other is fatigue life. Table 5-1 shows the stresses and fatigue life of welded joint using FEA tools (ANSYS).

Table 5-1: Results of stress and fatigue life of different load combination

Force (kN)	Maximum principal Stress (MPa)	Fatigue life No. of Cycle
60	430	2462
50	345	4299
40	274	7740
35	252	14258
30	235	21082
25	180	45426
20	153	90508
18	136	136166
16	115	274938
14	106	545923
12	88	1000000

The following Figure 5-1, S-N carve obtain from simulation data. In this graph shows 88 MPa stress can be consider for the fillet welded joints.

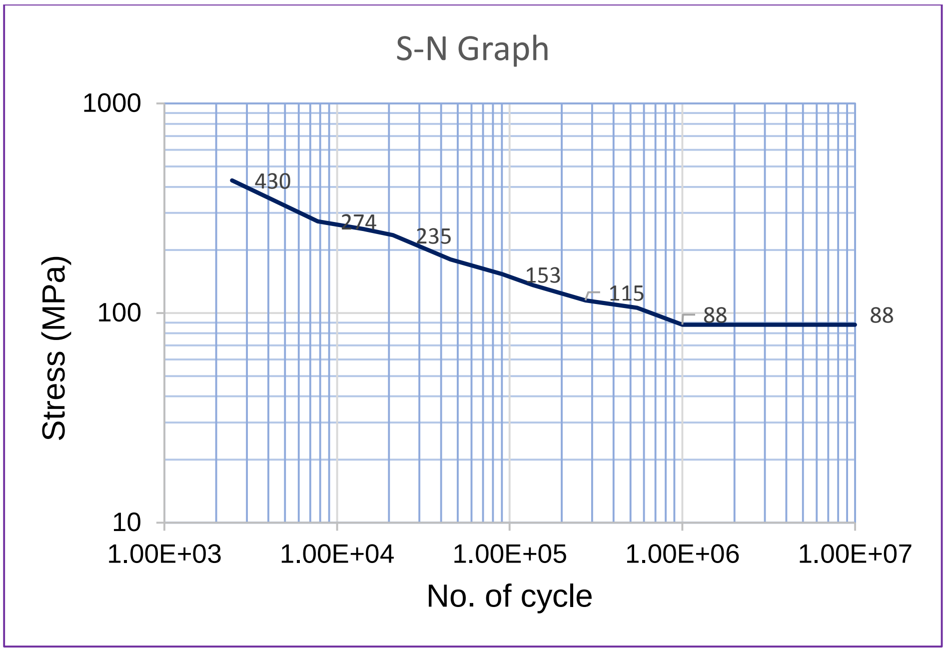


Figure 5-1: S-N carve of welded joint

5.3 Experiment results

Tensile test has been carried out in the experiment where disruption occurs in the base metal. The breaking point is far from the welding joint following Figure 5-2 and Figure 5-3 can see the break happen 60mm apart from welding joint.

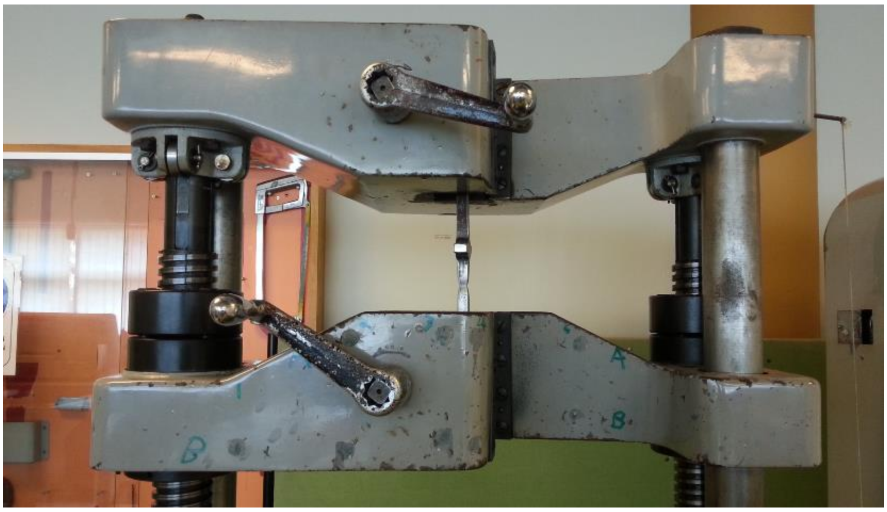


Figure 5-2: Motion of tensile test

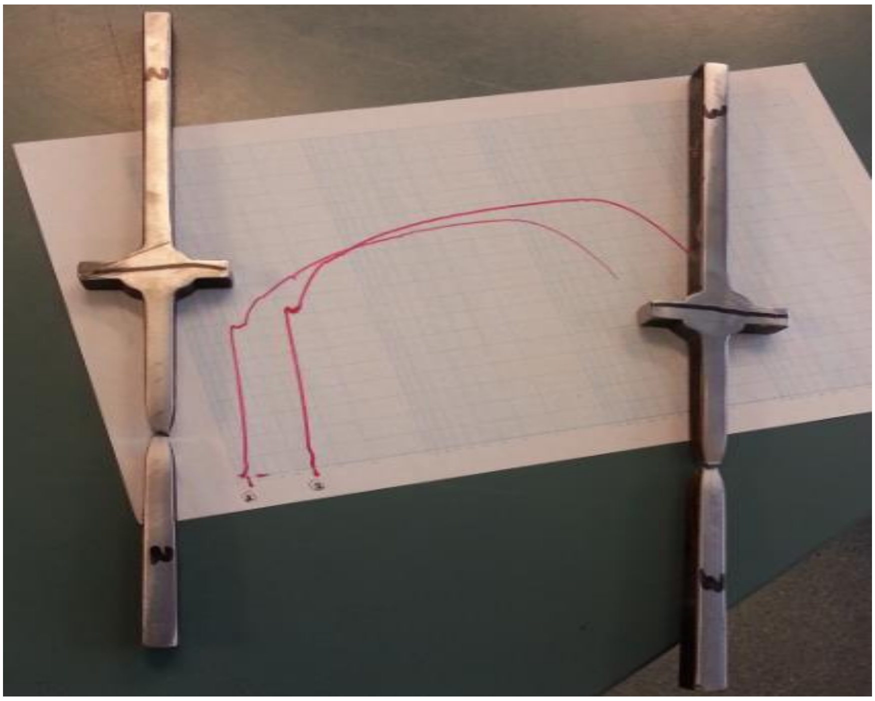


Figure 5-3: Brocken test pieces after tensile test

Above figure shows broken location and the reading from the tensile test machine. Five different specimen has been tested and the specimen was numbered as 2, 3, 4, 5 and 6.

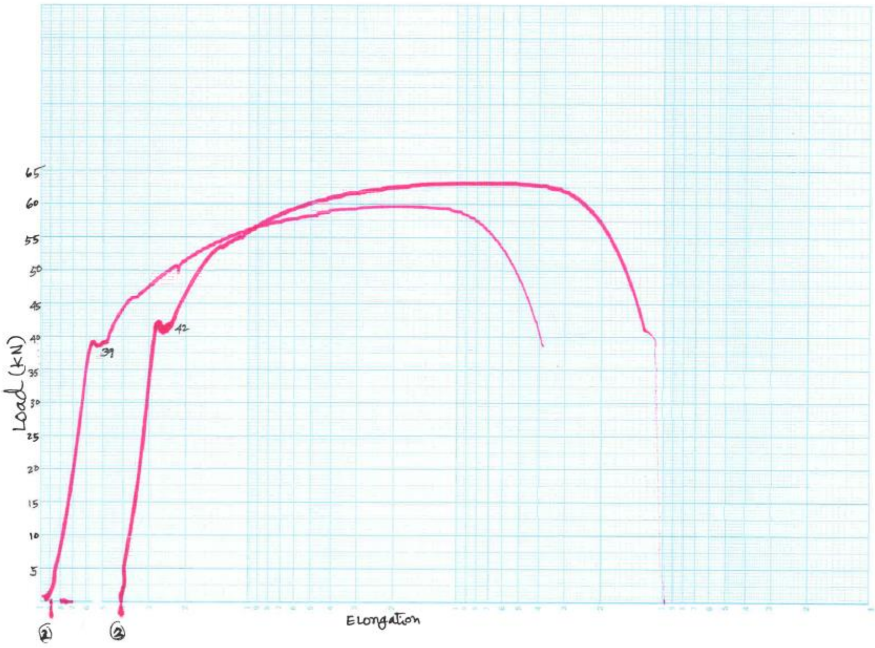


Figure 5-4: Load elongation graph of item 2 & 3

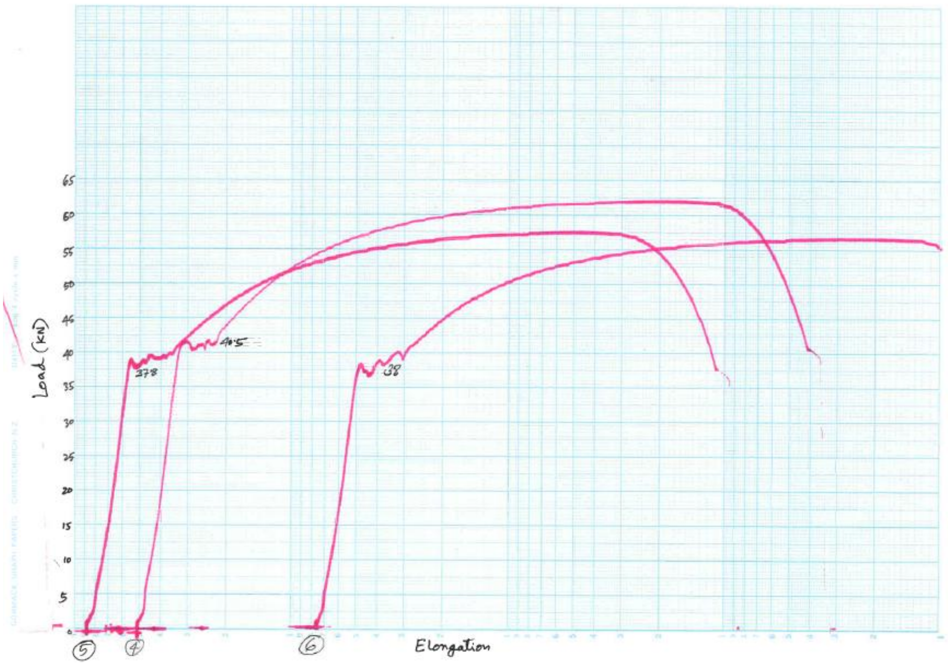


Figure 5-5: Load elongation graph of item 4,5 and 6

Total six number of item carried out tensile test. Item 1 has failed after loaded. Everyone has different size of width due to manual cutting process. Table 5-2 given all the test result and average yield stress found 263.4 MPa

Table 5-2: Test results of experiment

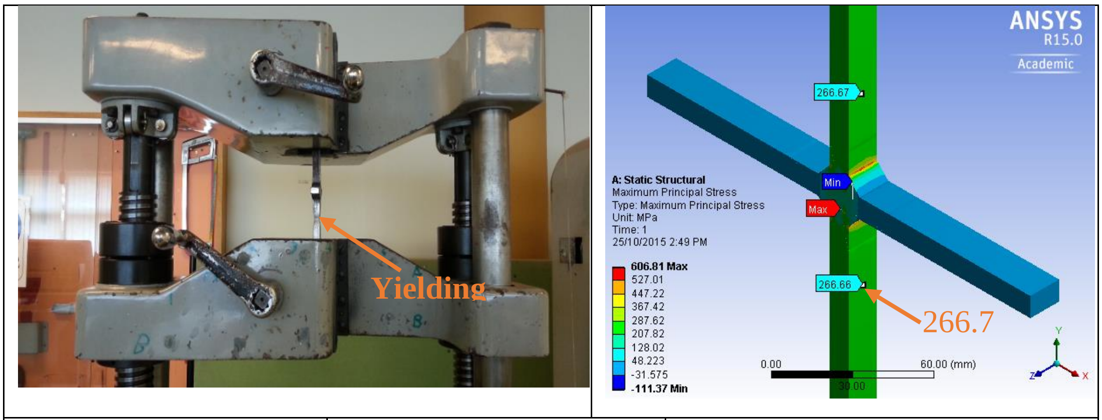
Item no	**Cross section area of experiment**	**Yield load (N)**	**Ultimate load (N)**	**Yield stress (MPa)**	**Ultimate stress (MPa)**
1	10x15.66 = 156.6mm^2	Fail	Fail	Fail	Fail
2	10x15 = 150mm^2	39000	59900	260	400
3	10x16 = 160mm^2	42000	63000	263	394
4	10x15.3 = 153mm^2	40500	61800	265	404
5	10x14.5= 145mm^2	37800	57900	261	399
6	10x14.2 = 142mm^2	38000	56400	268	397
Average stress		**40000**	**60000**	**263.4**	**399**

5.4 Discussions

Fatigue damage occurs when stress at a point changes over time. Fatigue life set up in the simulation as constant amplitude loading with fully reversed (+1 to -1). Fatigue life can be over the time (no. of day hour) or number of cycle, to develop SN carve we chose number of cycle.

The main aim of the project was to compare physical data of fatigue life with finite element analysis data. However, fatigue life test of the specimen could not be completed because of time and resource constraints from the technical support area due to their commitments to post graduate projects

In order to validate finite element analysis results with comparing experiment results we have completed tensile test. Figure 5-6 shows the experiment test piece yielding at 263.4 MPa. The necking started far from the welded area. In the model simulation found 266.7 MPa stress at the same location where specimen broken. The results different between experiment and simulation only 1.2%. Considering this minimum error of the results we are conducting simulations for different type of analysis such as fatigue life.



Applied force	Experiment yield stress	Simulation stress
40 kN	263.4 MPa	266.7 MPa

Figure 5-6: Comparison of experiment test with model simulation

Experiment yield stress taken average of five different specimen because the cross sectional area of the each specimen are not same due to manually prepared of the test pieces. Simulation stress taken at the same location where yield started although stress of the model are same everywhere except weld toe.

Weld toe has the maximum stress and minimum number of fatigue life cycles. However, the weld toe is the concentrated area Figure 5-7 shows the dimension of the area about 0.25 mm where, we can observe it is very smaller area compare to the cross sectional area. In real life situation molten weld metal toe will be a bit curve or after removed some material from toe using grinding operation will be removed most red coloured area.

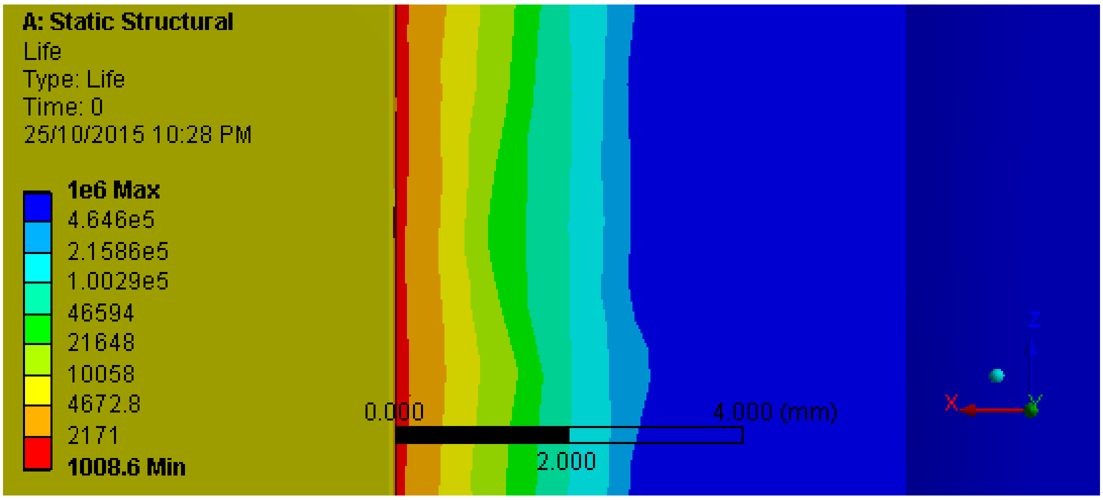


Figure 5-7: Zoomed view of stress concentrated area

There are eleven different load combination has been applied to predict one million cycle fatigue life. Started from ultimate stress load (60 kN) than gradually decrease the load to get 1 million cycles, the result shows at 88 MPa stress can be used for the design calculation of this kind of material.

Chapter 6 – Conclusions

6.1 Chapter overview

This final chapter summarises the achievement of project objectives, identifies future works and provide recommendations. Definitive project conclusion also find end of this chapter.

The objective of this project was to validate finite element analysis outcome compare with experimental investigation. Primary results has been achieved successfully with some lack of boundary condition input. Initial aims of experiment investigation was fatigue life of fillet welded joint have not been achieved within the research project time frame. However tensile test has been done to validate the FEA results.

6.2 Achievement of the project

The ultimate purpose of this project has been to perform finite element analysis of fillet welded joint and comparison with experimental investigation. In order to achieve the reasonable outcome study begin with introduction of the project then literature review, methodology and project planning, model simulation finally result discussion.

Chapter 1 was to understand background of the welded structures and how the structures fails during operations initiate cracks from the welded area. From background knowledge define scope of the research and set up expected outcome than outlined the dissertation.

Chapter 2 of this research has been successfully investigated a broad range of available literature related to the finite element analysis of welded structure and how welding operation affected to the materials and how can improved the life of welding joints.

Methodology and project planning developed in the chapter 3, where describe details drawings of specimen and model development. In the FEA model, mesh element size also define using iterative process then used boundary condition for analysis.

Chapter 4 detailed the finite element simulation and predicted principal stress and fatigue life of the model. The stress found for infinite life under cyclic load is 88 MPa which is 35% of yield stress.

Finally chapter 5 carry out discussion of the results from FEA analysis and experiment investigation. Appendices has been included end of this report and there are some boundary condition haven't been included to the model which will be discussed in the future work and recommendation topic.

6.3 Further work and recommendation

Although a significant amount of the original goals have been achieved as part of this research project, a substantial amount of future work to be completed for the more accurate results and comparison between experiment investigation and model simulation. Details recommendations of simulation and experimental investigation listed below:

- **Fatigue testing –** Experiment test has been carried out for this project was tensile test where observed yielding started on parent metal rather than weld toe area which means welded joint stronger than original material in terms of tensile force. To investigate fatigue life of the structure, Fatigue testing of experiment is more relevant to compare with FEA results.
- **Model simulation –** FEA model will be given result according to input data and boundary condition. In the model simulation at weld needed to induce residual stress.
- **Mesh element size –** Proper mesh element size is the key parameter to get better results. Mesh method need to be justified between Tetrahedron, Hex dominant, Sweep and Multizone which will be given more accuracy.

6.4 Conclusion

The ultimate purpose of the project has been achieved with developing techniques of the finite element analysis of fillet welded joint. The experimental investigation validate the performance of the FEA analysis results were found 1.2% error on tensile test. The experiment yield stress was found 263.4 MPa and simulation yield stress at the same location appears 266.7 MPa. In order to calculate fatigue life of welded joint used iterative process to define stress at one million cycle. The analysis found 274 MPa stress and 7740 cycle fatigue life applying yielding load. After reduced load at 12kN and found the fatigue life one million cycle where shows 88 MPa stress which is 35% of yield stress. So that designer can consider 35% of yield strength when design structure for fluctuating and repeated loading conditions.

List of References

A. M. Kamara, SMaLL 2011, 'A Numerical Investigation into Residual Stress Characteristics in Laser Deposited Multiple Layer Waspaloy Parts', *Manufacturing science and engineering*, viewed 22-05-15, <http://manufacturingscience.asmedigitalcollection.asme.org/article.aspx?articleid=1439228#FiniteElementModelingofResidualStress>.

A. Thirugnanam, MkaLR 2014, 'Analysis of Stress in Welded Joint in Bending And in Torsion Using ‘Ansys’', *Middle-East Journal of Scientific Research*, vol. 20, <http://www.idosi.org/mejsr/mejsr20(5)14/7.pdf>.

Abid, M & Siddique, M 2005, 'Finite-element simulation of tack welds in girth welding of a pipe-flange joint', *Acta Mechanica*, vol. 178, no. 1-2, pp. 53-64, <http://dx.doi.org/10.1007/s00707-005-0241-3>.

Austmig ES6, WIA, viewed 7/9/2015, <http://www.welding.com.au/filler-metals/view/austmig-es6>.

AWS 2010, *Structural Welding Code Steel*, American Welding Society, Miami,Florida.

Aygul, M 2012, 'Fatigue Analysis of Welded Structures Using the Finite Element Method ', Chalmers University of Technology, Sweden.

Brickstad, B & Josefson, BL 1998, 'A parametric study of residual stresses in multi-pass butt-welded stainless steel pipes', *International Journal of Pressure Vessels and Piping*, vol. 75, no. 1, pp. 11-25, <http://www.sciencedirect.com/science/article/pii/S0308016197001178>.

Chang, K-H & Lee, C-H 2009, 'Finite element analysis of the residual stresses in T-joint fillet welds made of similar and dissimilar steels', *The International Journal of Advanced Manufacturing Technology*, vol. 41, no. 3-4, pp. 250-8, <http://dx.doi.org/10.1007/s00170-008-1487-4>.

Deng, D, Liang, W & Murakawa, H 2007, 'Determination of welding deformation in fillet-welded joint by means of numerical simulation and comparison with experimental measurements', *Journal of Materials Processing Technology*, vol. 183, no. 2–3, pp. 219-25, <http://www.sciencedirect.com/science/article/pii/S0924013606008570>.

Dodge, MF, Dong, HB, Gittos, MF & Mobberley, T 2014, 'Fusion zone microstructure associated with embrittlement of subsea dissimilar joints', in vol. 5.

Gannon, L, Liu, Y, Pegg, N & Smith, M 2010, 'Effect of welding sequence on residual stress and distortion in flat-bar stiffened plates', *Marine Structures*, vol. 23, no. 3, pp. 385-404, <http://www.sciencedirect.com/science/article/pii/S0951833910000225>.

Haghani, R 2012, 'Fatigue-Prone Details in Steel Bridges', *Buildings (Basel)*, vol. 2, no. 4, pp. 456-76, <http://www.mdpi.com/2075-5309/2/4/456/htm>.

Hale, S 2014, *How do I know if my mesh is good enough*, CAE Associates, viewed 27/05/15, <https://caeai.com/blog/how-do-i-know-if-my-mesh-good-enough>.

Hicks, JG 1987, *Welded joint design*, 2nd ed. edn, BSP Professional Books, Oxford.

Hobbacher, AF 2009, 'The new IIW recommendations for fatigue assessment of welded joints and components – A comprehensive code recently updated', *International Journal of Fatigue*, vol. 31, no. 1, pp. 50-8, <http://www.sciencedirect.com/science/article/pii/S0142112308001072>.

Joshi, S & Price, JWH 2009, 'A comparative study on application of design codes for prediction of fatigue life of a mining dragline cluster', *Engineering Failure Analysis*, vol. 16, no. 5, pp. 1562-9, <http://www.sciencedirect.com/science/article/pii/S1350630708002446>.

Kirkhope, KJ, Bell, R, Caron, L, Basu, RI & Ma, KT 1999, 'Weld detail fatigue life improvement techniques. Part 1: review', *Marine Structures*, vol. 12, no. 6, pp. 447-74, <http://www.sciencedirect.com/science/article/pii/S0951833999000131>.

Lassen, BT, Darcis, P & Recho, N 2006, 'Fatigue Behavior of Welded Joints

Part 2: Physical modelling of the fatigue process', *Welding journal*, vol. 26, viewed 1/6/2015, <https://app.aws.org/wj/supplement/WJ_2006_01_s19.pdf>.

Leap Australia, 2014, LEAP Australia Pty Ltd, viewed 7-5-2015, <http://www.finiteelementanalysis.com.au/featured/what-is-fatigue/>.

Lindgren, L-E 2001, 'FINITE ELEMENT MODELING AND SIMULATION OF WELDING. PART 2: IMPROVED MATERIAL MODELING', *Journal of Thermal Stresses*, vol. 24, no. 3, pp. 195-231, viewed 2015/05/06, <http://dx.doi.org/10.1080/014957301300006380>.

Maddox, SJ 2002, *Fatigue strength of welded structures*, Second edn, The Welding Institure, Milton Keynes England.

Moaveni, S 2007, *Finite element analysis : theory and application with ANSYS*, 3rd ed. edn, Pearson Prentice Hall, Upper Saddle River NJ.

Morris, JL 1955, *Welding processes and procedures*, Prentice Hall, New York.

PadmaKumari, V 2013, 'Finite Element Analysis of EBW Welded Joint Using Sysweld', *International Journal of engineering technology and advance engineering*, vol. 3, no. 2, viewed 11-04-15, <http://www.ijetae.com/>.

Perić, M, Tonković, Z, Rodić, A, Surjak, M, Garašić, I, Boras, I & Švaić, S 2014, 'Numerical analysis and experimental investigation of welding residual stresses and distortions in a T-joint fillet weld', *Materials & Design*, vol. 53, no. 0, pp. 1052-63, <http://www.sciencedirect.com/science/article/pii/S0261306913007541>.

Pisarski, HG, Tkach, Y & Quintana, M 2004, 'Evaluation of weld metal strength mismatch in X100 pipeline girth welds', <http://www.twi-global.com/technical-knowledge/published-papers/evaluation-of-weld-metal-strength-mismatch-in-x100-pipeline-girth-welds/>.

Qi, DHJ 2006, *Finite Element Analysis*, viewed 24/04/2015, <http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf>.

Qingyu, S, Anli, L, Haiyan, Z & Aiping, W 2002, 'Development and application of the adaptive mesh technique in the three-dimensional numerical simulation of the welding process', *Journal of Materials Processing Technology*, vol. 121, no. 2–3, pp. 167-72, <http://www.sciencedirect.com/science/article/pii/S0924013601008305>.

Rabih Kamal Kassab1, HC, Ngan Van Lê1, Jacques Lanteigne2 and Marc Thomas1 2012, 'Experimental and Finite Element Analysis of a T-Joint Welding', *Journal of Mechanics Engineering and Automation*, viewed 3/5/2015, <http://www.davidpublishing.com/davidpublishing/upfile/9/7/2012/2012090702055701.pdf>.

Richards, KG 1969, *Fatigue strength of welded structures*, Welding Institute, Cambridge.

Rother, K & Rudolph, J 2011, 'Fatigue assessment of welded structures: practical aspects for stress analysis and fatigue assessment', *Fatigue & Fracture of Engineering Materials & Structures*, vol. 34, no. 3, pp. 177-204, <http://dx.doi.org/10.1111/j.1460-2695.2010.01506.x>.

Seok, C-S, Suh, M-W & Park, J-H 1999, 'Investigation of welding residual stress of high tensile steel by finite element method and experiment', *KSME International Journal*, vol. 13, no. 12, pp. 879-85, <http://dx.doi.org/10.1007/BF03184755>.

Shen, WY & Clayton, P 1996, 'Fatigue of fillet welded A515 steel', *Engineering Fracture Mechanics*, vol. 53, no. 6, pp. 1007-16, <http://www.sciencedirect.com/science/article/pii/0013794495001603>.

SIEMENS 2015, *Finite Element Analysis*, Siemens, viewed 12/04/15, <http://blog.industrysoftware.automation.siemens.com/blog/tag/fea/>.

Sulaiman, M, Manurung, Y, Haruman, E, Rahim, M, Redza, M, Lidam, R, Abas, S, Tham, G & Chau, C 2011, 'Simulation and experimental study on distortion of butt and T-joints using WELD PLANNER', *Journal of Mechanical Science and Technology*, vol. 25, no. 10, pp. 2641-6, <http://dx.doi.org/10.1007/s12206-011-0701-8>.

Syahroni, N & Hidayat, MIP 2012, *3D Finite Element Simulation of T-Joint Fillet Weld: Effect of Various Welding Sequences on the Residual Stresses and Distortions*.

Teng, T-L, Fung, C-P, Chang, P-H & Yang, W-C 2001, 'Analysis of residual stresses and distortions in T-joint fillet welds', *International Journal of Pressure Vessels and Piping*, vol. 78, no. 8, pp. 523-38.

TWI Fatigue testinng, 2015, viewed 15/09/15, <http://www.twi-global.com/technical-knowledge/job-knowledge/fatigue-testing-078/>.

WANG Rui, RS, SERIZAWA Hisashi, MURAKAWA Hidekazu, Jianxun ZHANG 2008, 'Numerical and Experimental Investigations on Welding Deformation', vol. 37, viewed 28/05/15, <http://www.jwri.osaka-u.ac.jp/publication/trans-jwri/pdf/371-14.pdf>.

Wang, T, Hopperstad, OS, Lademo, OG & Larsen, PK 2007, 'Finite element analysis of welded beam-to-column joints in aluminium alloy EN AW 6082 T6', *Finite Elements in Analysis and Design*, vol. 44, no. 1–2, pp. 1-16, <http://www.sciencedirect.com/science/article/pii/S0168874X07001011>.

Welded Connections, viewed 14/8/2015, <http://www.mitcalc.com/doc/welding/help/en/welding.htm>.

Welding : theory and practice, 1990, North-Holland, Amsterdam.

Welding fundamentals and processes, 2011, ASM International, 10/6/2015, <http://products.asminternational.org/hbk/index.jsp>.

Welding Inspection Cswip, 2010, viewed 2/6/15, <http://www.slideshare.net/guest831c1e/welding-inspection-cswip>.

Woolling, P & Carrouge, D 2002, 'Heat affected zone microstructures in supermartensitic stainless steels', <http://www.twi-global.com/technical-knowledge/published-papers/heat-affected-zone-microstructures-in-supermartensitic-stainless-steels-october-2002/>.

Worksafe, 2015, Government of western Australia, viewed 1/6/15, <http://www.commerce.wa.gov.au/worksafe/how-get-started>.

Yang, X, Zou, L & Deng, W 2015, 'Fatigue life prediction for welding components based on hybrid intelligent technique', *Materials Science and Engineering: A*, vol. 642, pp. 253-61, <http://www.sciencedirect.com/science/article/pii/S092150931530160X>.

Zhang, Y-H, Smith, S, Wei, L, Johnston, C & Stacey, A 2013, 'Measurement and modelling of residual stresses in offshore circumferential welds', in vol. 3.

List of Appendices

A - Project specification

University of Southern Queensland

FACULTY OF ENGINEERING AND SURVEYING

ENG4111/ENG4112 Research Project
PROJECT SPECIFICATION

FOR:	RAFIQUL ISLAM MOHAMMED
TOPIC:	FINITE ELEMENTS ANALYSIS OF FILLET WELDED JOINT
SUPERVISOR:	CHRIS SNOOK
PROJECT AIM:	The aim of the project is to develop techniques for simplifying finite element analysis such as stress analysis and dynamic properties of structures under vibration excitation on non-complete joint penetrated (Fillet) welding.
SPONSORSHIP:	University of Southern Queensland

PROGRAMME: **Issue A, 18th March 2015**

- Research the back ground information of finite element analysis of welded joint.
- Research on behaviour of welded joints under cyclic load.
- Learn how to use ANSYS to analyse fillet welding joints with specific parameter, including ANSYS fatigue analysis.
- Design and analyse the model structures with software (Creo/ANSYS) to compare between complete penetration joints and non-complete penetration joints.
- Fabricate the T-joint specimen in workshop.
- Complete physical testing of the specimen (using INSTRON).

As time permits:

- Develop techniques for simplifying finite element analysis of welding joints.

Examiner/Co-examiner______________________________________

B - Resource analysis

In order to complete the project various resources are required. This project divide in to two part, one is to prepare experiment and conduct physical test, another is to design and simulation with software using computer.

The resource that will required to complete the experiment:

- Access to metal fabrication workshop
- Grade 250 mild steel, size 400mmX300mmX10mm, one piece
- Grade 250 mild steel, size 400mmX150mmX10mm, two pieces
- Welding (MIG) machine and welding wire (ES609S)
- Band saw
- Angle grinder
- Personal protective equipment (PPE)
- Access to material testing lab (Z104.1)
- Test machine operator personnel

The resource that will be required to complete simulation:

- A standard computer capable to run simulation software and producing data
- Access to the internet, journals, articles, websites and newspaper to collect theoretical information and perform literature review
- Access to library data-base for previous work which were published
- Access to standards
- ANSYS software for design and simulations
- Access to ANSYS official tutorials to learn operating and useful commands
- Access to Z block computer laboratories

Critical resource:

Learning ANSYS program is the critical service in this project. In the simulation most important data input is to applying welding parameter on the geometry. Knowing this operation is the most critical.

Cost involves

Material cost: \$100, manpower cost: \$100 X 4 = \$400, overhead cost: \$100 there for total cos involves = \$600

C - Risk management

Risk assessment is a process for developing knowledge and understanding about hazards and risks so that good decisions can be taken about controlling them. This project required use of welding machine, argon mixture with CO_2 gas, high power cable and band saw in laboratory Z4. For testing use of 810 material testing system in a testing lab Z104.1.

To carry out a basic risk assessment, follow these four steps (*Worksafe* 2015);

- Gather information about each hazard that can identify.
- Workout likelihood of an accident or incident occurring and consider how many people are likely to involve.
- Assess the consequence of hazard for example, fatal, suffer major injuries, suffer minor injuries or negligible.
- Rate the risk.

Risk assessment for location Z4

Description of hazards	People at risk	Number at risk (1-5)	Parts of body	Risk level
Material falling off	Person who involve material handling	Up to 5	Feet, lower body	Low

Categories	Short term controls	Long term controls	Completion details
Material handling Welding on plate Cutting off welded plate Workshop PPE	Avoid bending and twisting of body	Use trolley or lifting equipment	Employer: USQ Faculty of Health Engineering and Science Prepared by : Rafiqul Mohammed Assented to by: Chris Snook Position: Discipline group leader Mechanical engineering Signature: Date:

Description of hazards	People at risk	Number at risk (1-5)	Parts of body	Risk level
Sharp edges of machine parts	Person who involve material cutting	Up to 5	Fingers, hands and feet.	Low

Categories	Short term controls	Long term controls	Completion details
Material handling Welding on plate Cutting off welded plate Workshop PPE	Do not keep fingers close to the machine blade PPE must worn	Fixed machine guard	Employer: USQ Faculty of Health Engineering and Science Prepared by : Rafiqul Mohammed Assented to by: Chris Snook Position: Discipline group leader Mechanical engineering Signature: Date:

<table>
<tr><th colspan="2">Description of hazards</th><th>People at risk</th><th>Number at risk (1-5)</th><th>Parts of body</th><th>Risk level</th></tr>
<tr><td colspan="2">Electric shock</td><td>Person who involve Welding</td><td>Up to 5</td><td>All</td><td>Medium</td></tr>
<tr><th>Categories</th><th colspan="2">Short term controls</th><th colspan="2">Long term controls</th><th>Completion details</th></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">PPE must worn, do not weld if place is wet</td><td colspan="2">Inspect welding machine and cable regularly to make sure no damage If damage during operation report to the supervisor</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
</table>

<table>
<tr><th colspan="2">Description of hazards</th><th>People at risk</th><th>Number at risk (1-5)</th><th>Parts of body</th><th>Risk level</th></tr>
<tr><td colspan="2">Welding fume and gases</td><td>Person who involve Welding</td><td>Up to 5</td><td>Lung, skin, kidneys</td><td>Low</td></tr>
<tr><th>Categories</th><th colspan="2">Short term controls</th><th colspan="2">Long term controls</th><th>Completion details</th></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">PPE must be worn and fume mask (welding mask) need to use during welding</td><td colspan="2">Local exhaust (ventilation)
Adequate air flow (ventilation) should be in the workplace</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
</table>

<table>
<tr><td colspan="2">Description of hazards</td><td>People at risk</td><td>Number at risk (1-5)</td><td>Parts of body</td><td>Risk level</td></tr>
<tr><td colspan="2">Radiations</td><td>Person who involve Welding and surrounding people</td><td>Up to 5</td><td>Eyes and skin</td><td>Low</td></tr>
<tr><td>Categories</td><td colspan="2">Short term controls</td><td colspan="2">Long term controls</td><td>Completion details</td></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">Skin protection, Eye protection Non-flammable clothing
PPE must be worn</td><td colspan="2">Welding bay covered with welding screens</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
<tr><td colspan="2">Description of hazards</td><td>People at risk</td><td>Number at risk (1-5)</td><td>Parts of body</td><td>Risk level</td></tr>
<tr><td colspan="2">Fire</td><td>All people, whoever in the workshop</td><td>Up to 5</td><td>All body</td><td>Low</td></tr>
<tr><td>Categories</td><td colspan="2">Short term controls</td><td colspan="2">Long term controls</td><td>Completion details</td></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">Use fire extinguisher and fire hoses
Checked welded area after half an hour</td><td colspan="2">Safe working practice, Flammable goods never allow in the welding bay
Inspect welding machine regularly before start work</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
</table>

<table>
<tr><td colspan="2">Description of hazards</td><td>People at risk</td><td>Number at risk (1-5)</td><td>Parts of body</td><td>Risk level</td></tr>
<tr><td colspan="2">Heat during welding</td><td>Person who involve Welding</td><td>Up to 5</td><td>Whole body</td><td>Low</td></tr>
<tr><td>Categories</td><td colspan="2">Short term controls</td><td colspan="2">Long term controls</td><td>Completion details</td></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">Use proper clothing’s, stay away from welding spark as much as possible</td><td colspan="2">Local exhaust and adequate air flow</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
</table>

<table>
<tr><td colspan="2">Description of hazards</td><td>People at risk</td><td>Number at risk (1-5)</td><td>Parts of body</td><td>Risk level</td></tr>
<tr><td colspan="2">Compressed gas cylinder explosion</td><td>Everybody in the workshop</td><td>Up to 5</td><td>All</td><td>Low</td></tr>
<tr><td>Categories</td><td colspan="2">Short term controls</td><td colspan="2">Long term controls</td><td>Completion details</td></tr>
<tr><td>Material handling
Welding on plate
Cutting off welded plate
Workshop PPE</td><td colspan="2">Positioned gas cylinder upright
Use proper valve to control the gases
ensure all gas cylinder are kept in the cool area and away from potential heat source</td><td colspan="2">Case around cylinders preventing access or accidental contact</td><td>Employer: USQ
Faculty of Health Engineering and Science
Prepared by : Rafiqul Mohammed
Assented to by: Chris Snook
Position: Discipline group leader Mechanical engineering
Signature:
Date:</td></tr>
</table>

Risk management chart for Z104.1 testing lab

Description of hazards	People at risk	Number at risk (1-5)	Parts of body	Risk level
Electric shock	Person undertaking the test	Up to 5	Whole body	Low

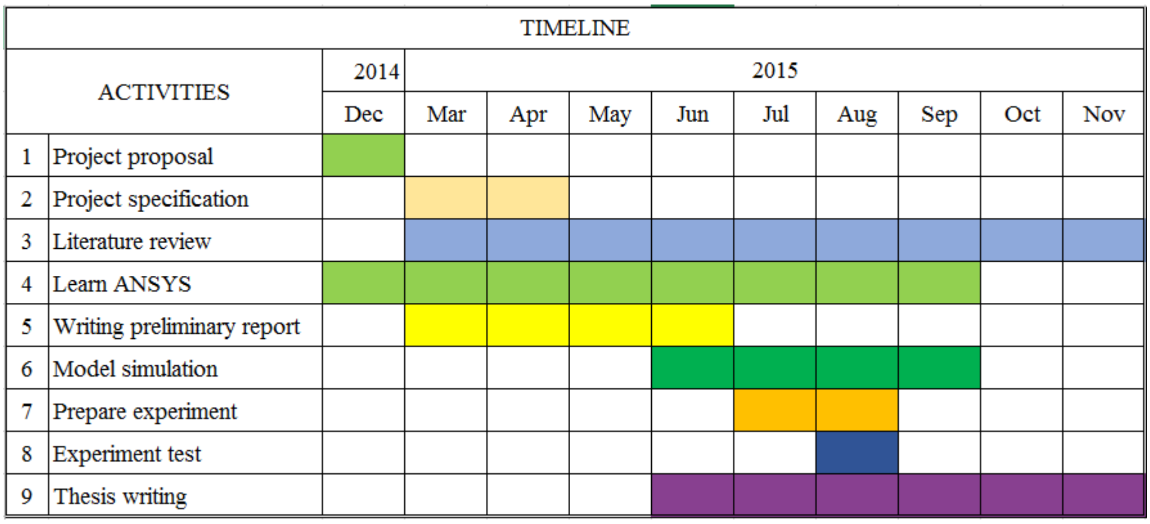
Categories	Short term controls	Long term controls	Completion details
Material handling Clamping test piece Operating test machine Workshop PPE	Check all the cables connected to the machine for inspection tags whether it is up to date	Operate machine according to specifications Do not over loaded Follow instructions	Employer: USQ Faculty of Health Engineering and Science Prepared by : Rafiqul Mohammed Assented to by: Chris Snook Position: Discipline group leader Mechanical engineering Signature: Date:

Description of hazards	People at risk	Number at risk (1-5)	Parts of body	Risk level
Flying metal chips	Person undertaking the test	Up to 5	Any parts of body	Low

Categories	Short term controls	Long term controls	Completion details
Material handling	PPE must be worn such as full covered shows, safety goggles and air plugs.	Use proper machine guards	Employer: USQ Faculty of Health Engineering and Science
Clamping test piece	Know the location of emergency switch	Machine up to date	Prepared by : Rafiqul Mohammed
Operating test machine	Do not touch specimen during loading		Assented to by: Chris Snook
Workshop PPE			Position: Discipline group leader Mechanical engineering
			Signature:
			Date:

D - Project timeline

Table 6-1: project timeline



TIMELINE											
ACTIVITIES		2014	2015								
		Dec	Mar	Apr	May	Jun	Jul	Aug	Sep	Oct	Nov
1	Project proposal	🟩									
2	Project specification		🟧	🟧							
3	Literature review		🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦	🟦
4	Learn ANSYS	🟩	🟩	🟩	🟩	🟩	🟩	🟩	🟩		
5	Writing preliminary report		🟨	🟨	🟨	🟨					
6	Model simulation					🟩	🟩	🟩	🟩		
7	Prepare experiment						🟧	🟧			
8	Experiment test							🟦			
9	Thesis writing					🟪	🟪	🟪	🟪	🟪	🟪

Project proposal: Completed

Project specification: Completed

Literature review: Partially completed, further need to review until completing the project.

Learn ANSYS program: Introduction has been completed with YouTube help. Found access of ANSYS software at USQ lab with help from supervisor (Mr. Chris Snook). Still unable to access official tutorial due to registration problem, this learning will be carried out until September, in the meantime some simulation will be progressed.

Writing preliminary report: First preliminary report submitted 4th June 2015, feedback received with lots of comments. Revision 2 of this report will submitted by 17th July 2015.

Model simulation: Model has been finalised at this stage now updating knowledge to applying welding parameter to the model.

Prepare experiment: Request has been sent to the workshop to fabrication of specimen and expected completion time will be end of July.

Experiment test: Once specimen done then booked for the test. The test will required one day to complete. According to timeline by August can be done.

Thesis writing: Thesis writing in progress, continue writing until the day of submission.