Defence Science Journal, Vol. 60, No. 1, January 2010, pp. 112-118

Strength Investigation of Thick Welded T-Joint using Finite Element Modelling

Aidy Ali*, Tun Chun Yung, Z.A. Zulkefli, Nuraini Abdul Aziz, B.B. Sahari, and M. Zadeh

University Putra Malaysia, Selangor, Malaysia

**E-mail: aidy@eng.upm.edu.my*

ABSTRACT

The paper discusses the computation of finite element modelling (FEM) of a thick welded joint as a high load transfer joint. The FEM utilises MSC PATRAN/NASTRAN software programs to predict and simulate the critical area of a welded joint. The elasticity limit for the specimen was determined and stress distribution was achieved in the joint to indicate critical parts of a welded T-joint and predict the critical locations for crack initiation in this kind of joint. Simulation and experimental results show good in agreement and the sources of some differences in these results are discussed.

Keywords: T-joints, finite element modelling, FEM, MSC PATRAN, Welded T-joints, MSC NASTRAN

1. INTRODUCTION

Nowadays, welding has become one of the popular joining methods among other mechanical joining techniques. Joining various types of materials in a permanent manner and making a homogenous environment between two parts of a joint to fasten them, is one of the advantages of welding as a joining technique. There are various types of weld joints: butt, corner, lap, and T-joints are the most common types of joints used in welding of metallic materials. These can be used with metal inert gas (MIG) and tungsten inert gas (TIG) welding[1]. T-joints are one of the most common types of joints used in mechanical and civil industries, especially in making a building structure. Several types of welding joints were studied-in analysing and finding characteristics by several researchers. Pinho da Cruz[2], *et al.* studied fatigue life prediction in some *Al* alloy lap joint weldments, and during two series of tests, the effects of post-weld heat treatment were investigated.

To analyse the bending, shear stress, and strain in some T-joints, some common methods such as analytical, experimental, numerical finite element method or boundary element method (FEM or BEM) are widely used by researchers. To better evaluate T-joint fillet welds, prediction of welding residual stresses are needed to provide a sensible safety margin for design against failure[3]. Some FEM analyses on welding joints were also carried out by Mackerle[4]. Recently, Teh and Rasmussen[5] carried out some laboratory tests of arc-welded T-joints between equal width rectangular hollow sections, and obtained an equation which allows for calculation of the strength of the weld. In other works, Balasubramanian and Guha[6], identified two types of cracking that normally cause a failure of a fillet welded joint–root cracking and toe cracking. The effect of welding processes on fatigue crack growth behaviours of load carrying cruciform joints has also been analysed.

To obtain the stress and strain distributions in T-joint structures and investigate the strength of the joint, 3-D FEM modelling can be used and it can be evaluated by carrying out an experimental test. Utilisation of new generation of computer software reduces time consuming long numerical procedures into a considerably shorter time. As was mentioned, T-joints are widely used in mechanical and civil structures such as ships and bridge structures. Also it is used in piping and supporting frames for pressure vessels. In most cases, these kinds of joints are subjected to bending forces which result in tensile and shear stresses in the joint. Finding stress and strain distribution in a T-joint will help to investigate the strength of the joint and predict unexpected failures.

2. SIMULATION

2.1 Creating Finite Element Model

In this research, the finite element software programs named MSC PATRAN, together with MSC NASTRAN were used for analysing the stress distribution on the joint, and elastic analysis was carried out. As an initial step, the geometry of the structure was modelled for a T-joint weld and mesh. Figure 1 demonstrates the schematic model which was created and meshed in the MSC PATRAN environment which are just pre-process and post-process softwares. To complete the problem the boundary conditions and loading properties were assigned. Boundary conditions for the joint were simulated as a constraint at the end of the pillar. Figure 2 shows how boundary conditions and loading properties are applied.

Due to the model not being united, a multipoint point constraint (MPC) boundary condition was chosen. The MPC is a linear equation relating displacement degrees

Received 19 February 2009, Revised 15 May 2009

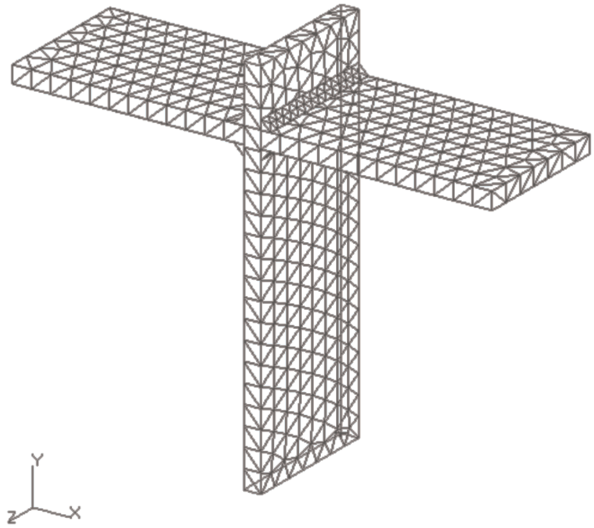

Figure 1. The model after meshing.

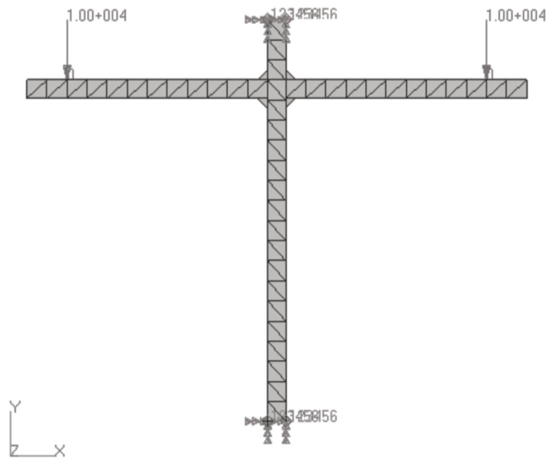

Figure 2. FEM model after applying the force and constraint.

Table 1. Mechanical properties of the specimen.

Modulus of elasticity, E (GPa)	Yield tensile strength, σ_y (MPa)	Poisson ratio, v	Density, ρ (kg/m^3)
200	200	0.3	7800

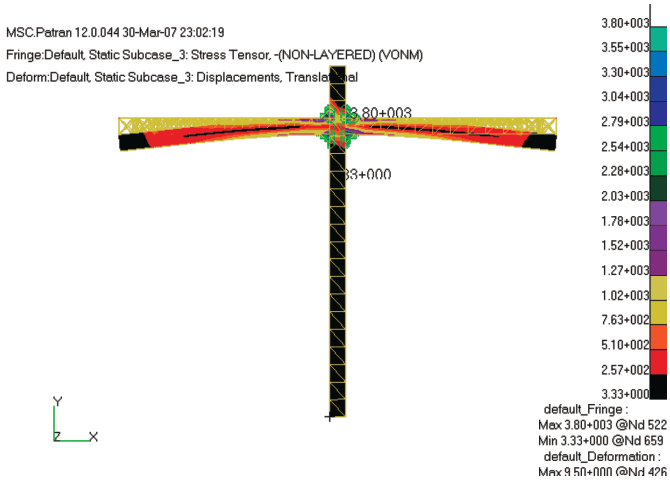

Figure 3. FEM result obtained using MSC NASTRAN.

of freedom. The relationship of this can be used to define ad hoc connectivity between the degrees of freedom of the connected solid. MPC is created between all connected surfaces, including weldment-flange and weldment-pillar. For the next step, the MSC PATRAN interface was used to input material properties, which was mild steel for our specimen. Mild steel was selected as the material for the specimen due to its popularity. The pillar and flange were made of the same material. The properties were input as mentioned in Table 1.

2.2 FEM Model Analysis

MSC PATRAN is just the pre-process and the post-process software. Once a finite-element model is completed, it can be submitted for structural analysis. MSC NASTRAN is used to solve and analyse the problems created in MSC PATRAN. Figure 3 shows the results obtained from the FEM software and the stress and strain distribution at the critical points are shown in the figure.

3. EXPERIMENTAL MODEL

3.1 Test Specimen

The test specimen was designed according to the model which was developed for FEM analysis. Dimensions are highlighted in a schematic model of the specimen (shown in Fig. 4). To utilise the results for comparing with FEM outcomes, mild steel was used to prepare the test specimen. To verify the material properties for the specimen, a tensile test was performed. The modulus of elasticity, E, and yield tensile strength, σ_y , were obtained and the results were identical to those which were used in the FEM analysis. For repeatability, three specimens were fabricated. Metal arc welding (GMAW) was chosen as welding method and was carried out according to ANSI/AWS B2.1:2005 standard[8]. The welding parameters are given in Table 2.

3.2 Test Procedure

The experimental work were carried out in the Strength Laboratory in UPM. The results obtained were as load-strain data and load-displacement data. Strain measurement at a selected point was obtained using strain gauge. Figure 5 shows the positionwhere the strain gauge was placed. The specimen and jigs were mounted on the universal testing machine with a maximum loading capacity 250 kN, as illustrated in Fig 6. The bending load was applied by the lower jig vertically from the bottom while the pillar was a fixed constraint. The application of the load was controlled manually to obtain the strain measurement for every increasing load level.

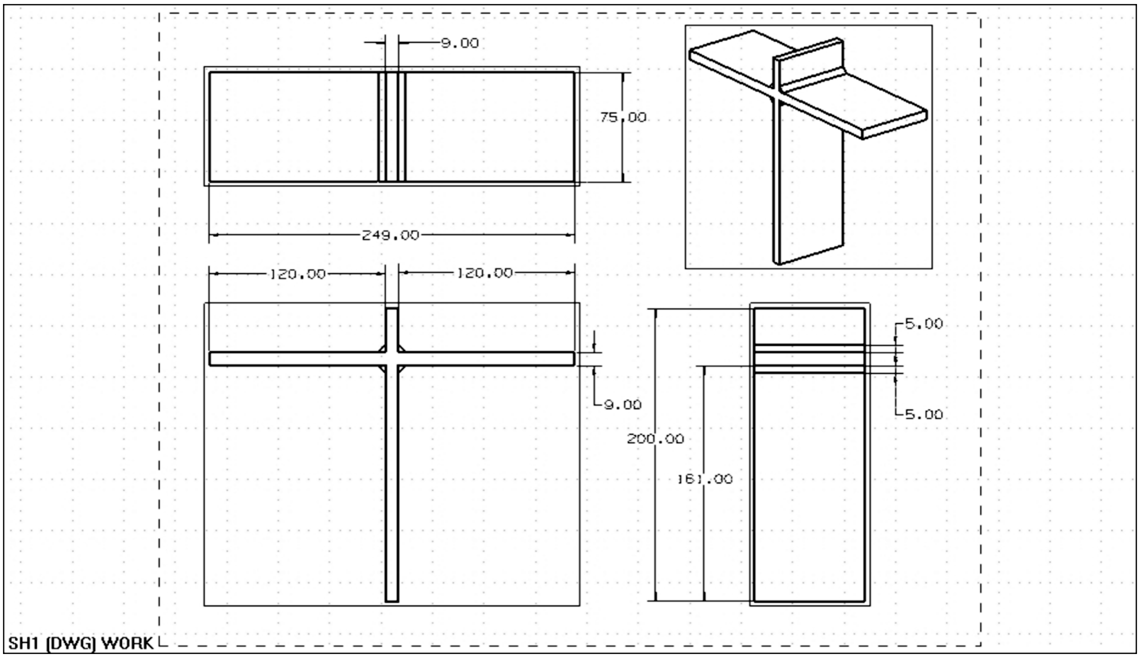

Figure 4. Dimension for the specimen in mm.

Table 2. GMAW parameters used for the experiment

Wire diameter (mm)	Current (Amps)	Wire feed rate (m/min)	Voltage (V)	Gas selection (%)
1.2	200	6.5	17	Argon 80–Co_2 20

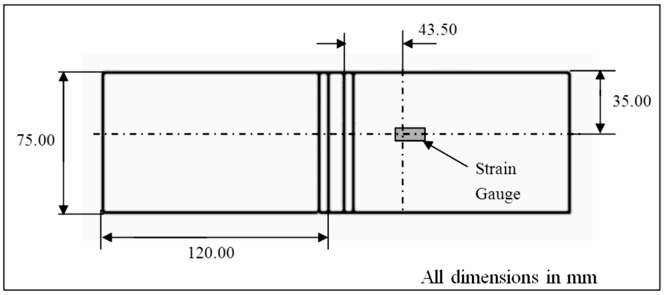

Figure 5. Top view of the strain gauge location.

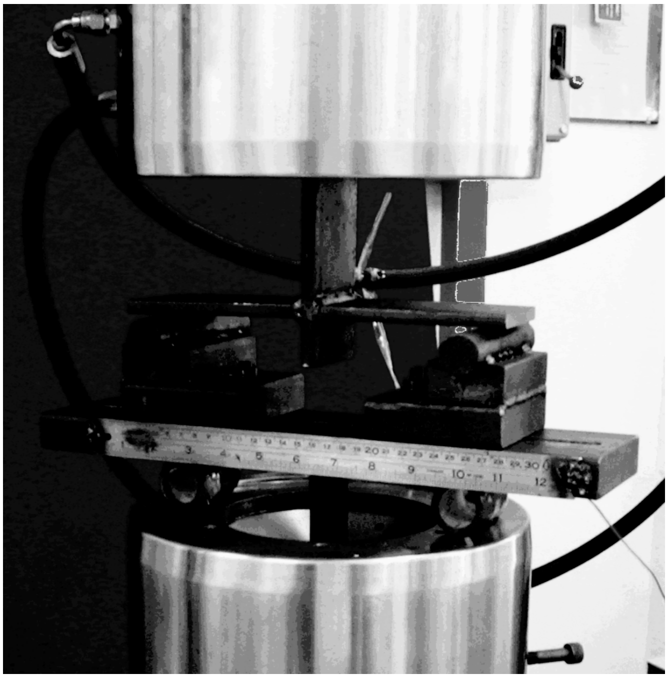

Figure 6. Experimental configuration.

4. RESULTS AND DISCUSSION

4.1 Surface Strain Calibration

Surface strain and joint displacement are used to compare FEM analysis and experimental results. After that, stress distribution of the weldment and the flange of the FEM were investigated. Moreover, the critical area of the model was pinpointed.

Experimental results show that all the points fall on to a straight line (Fig. 7). Three specimens were tested under different conditions but all the three specimens performed precisely the same. Line regression was done as it is strongly believed that the specimen behaves elastically. In the process of comparing with the experiment, a location of strain gauge placement in the testing specimen was selected as a reference node in the simulation part. Figure 8 demonstrates the result obtained from simulation together with the experimental results for comparison.

Results obtained from experimental and simulation tests show a slight disagreement. The only agreement between the simulation and experimental results is the specimen bending under elastic conditions. Under elastic conditions, the force is linearly proportional to the strain. As a standard, the common Young's modulus, E, for mild

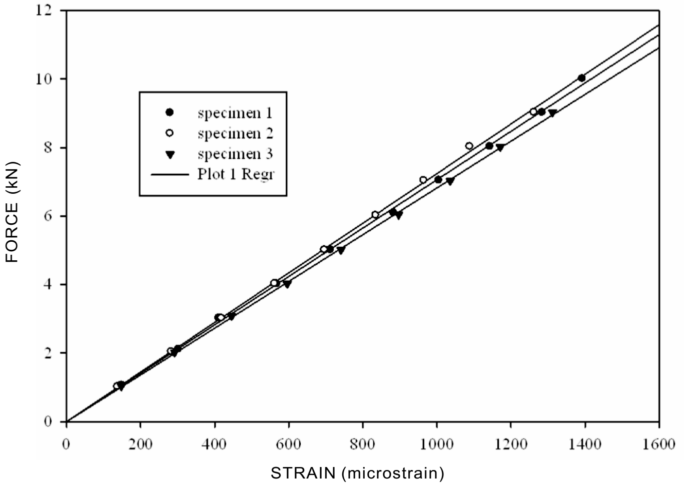

Figure 7. Comparison of the relation between applied force vs strain of three specimens.

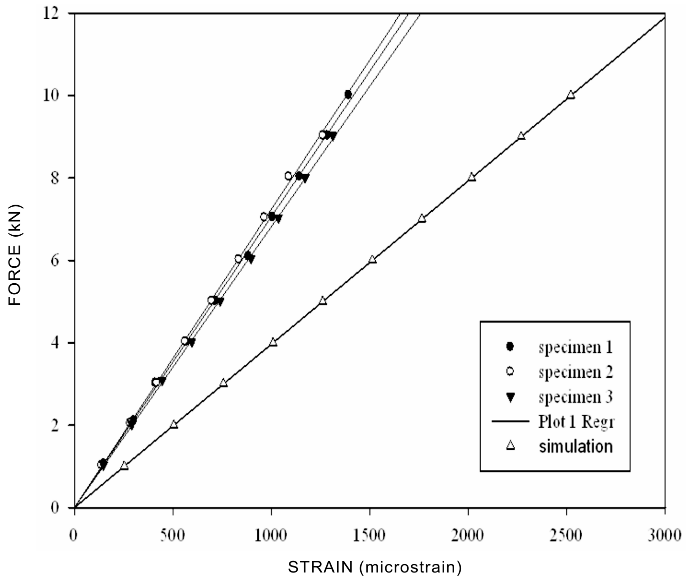

Figure 8. Result comparison for experimental and simulation work.

steel is 200 GPa, and this value was used in the simulation. But, an examination of the T-joint fillet weld shows that this was a gross oversimplification. There are a few factors that influence the value of *E*, such as the temperature history and residual stresses.

In the current research, the mechanical properties for a T-joint fillet weld are highly dependent on the temperature history (Teng *et a.,* 2001) and the exact value of *E* for the tested specimens was difficult to define as the specimens were prepared manually. Because a weld is locally heated by the welding heat source, its temperature distribution is not uniform and it changes as the welding progresses. The strain produced during this process is accompanied by plastic upsetting and, as a result, residual stresses remain after welding is complete. Residual stresses cause shrinkage and distortion to the specimen. However, the prediction of the FEM was still acceptable since the differences between simulation and experimental results are tiny (microstrain) when compared to the macro state.

4.2 Joint Deformation

The FEM is capable of being used to analyse elastic deformation. For high or super plasticity deformation, the software becomes in convergence, or mean unsolved. So, to investigate the strength of the welded joint, further increment loading until the plastic region in the experimental work is needed. Figures 9 and 10 exhibit the strength of the welded joint for a situation in which force were applied at two different points, 10 mm and 60 mm from weld root. It can be seen that the maximum elastic limit was 10.2 kN (42.7 MPa) and 17.5 kN (44.0 MPa) respectively for the situation 10 mm and 60 mm from weld root.

If the 3 per cent error, obtained by a comparison of these two cases, is omitted in calculations, it can be concluded that yield stress of the welded joint was around 44 MPa.

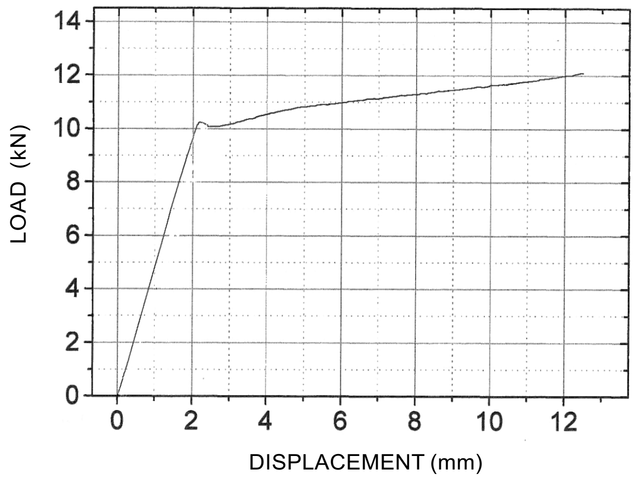

Figure 9. Strength of the welded joint in a situation which force was applied 10 mm from the weld root.

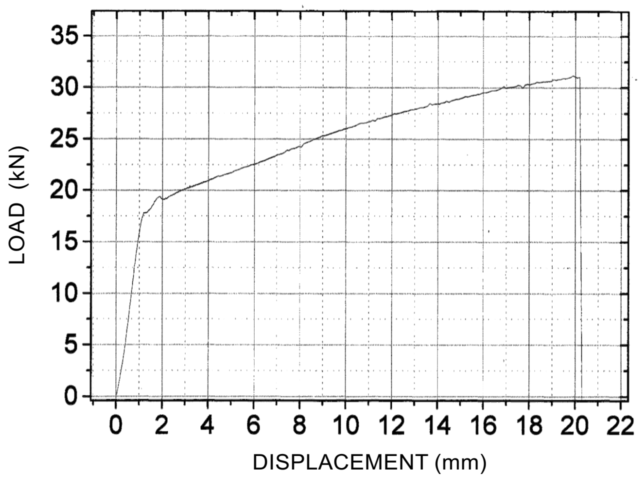

Figure 10. Strength of the welded joint in a situation in which force was applied 60 mm from the weld root.

Further increasing of the stress from 44 MPa will cause plasticity deformation. The comparison of the experimental result with the simulation work considers the elasticity region. So the experimental and FEM with an applied load of 10 kN was used for comparison since the behaviour was still elastic. The deformed shape of the FEM was compared with the actual deformation, as shown in Fig 11.

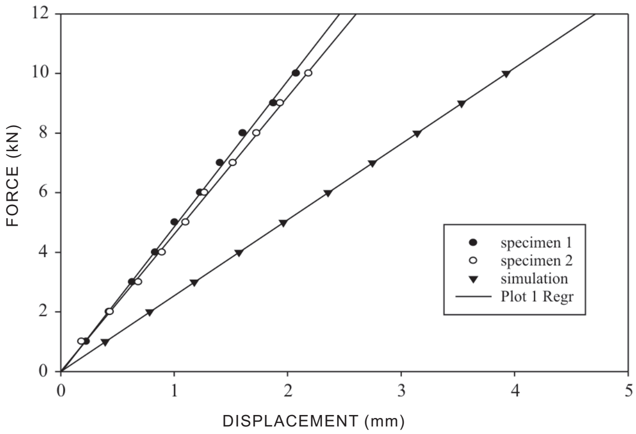

Figure 11. Comparison of the joint displacement for experimental and simulation work.

It is worth noting that both of the curves have different slopes. The similarity is that both performed with linear behaviour and were always true in the elastic region. Residual stress is the main reason that makes the results different. Residual stress causes distortion and shrinkage in the welded specimen. Due to differential thermal expansion and contraction of different parts of the welded assembly, angular distortions always exist in a T-joint fillet weld and vary for different specimens[7]. This simple example established confidence in the capabilities of the theoretical finite element computation to predict the welding joint.

4.3 Stress Analysis

Because weldment and flange are the most important parts in FEM analysis, these are analysed under this section. Stress analysis of the weldment will include the toe and root of the weld. Meanwhile, for the flange, the longitudinal stress and transverse stress were considered. Stress distribution in the weldment is shown in Fig. 12. To get a better view of the distribution of the stress in those two important parts of weldment (toe and root), can be interpreted from the diagrams as shown in Figs 3 and 4.

According to Figs 13 and 14, maximum stress in the weld toe is located at 1,250 MPa, and in weld root at 1,500 MPa. It clearly shows that the stress distribution of the weld root is more consistent relative to the weld toe.

Another important part in a T-joint fillet weld is the flange on which the load is applied and it is used to transfer the load to the joint. Longitudinal and transverse stress distributions are analysed in this part. Figures 15 and 16 demonstrate these stress distributions along the weld path.

Figure 12. Visual FEM result for stress distribution of weldment.

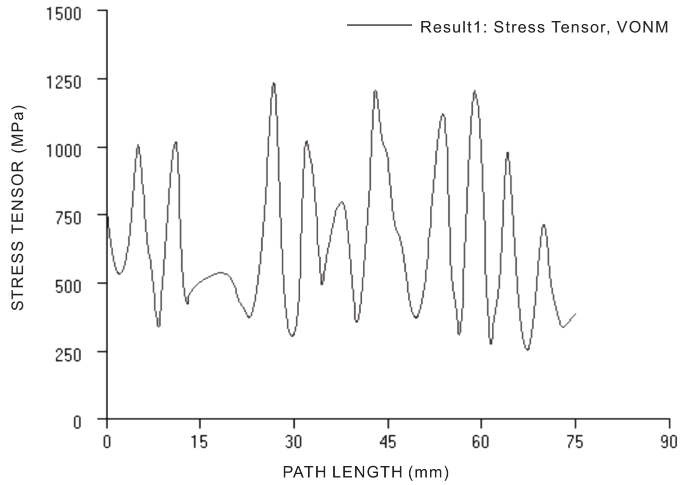

Figure 13. Stress distribution of a weld toe.

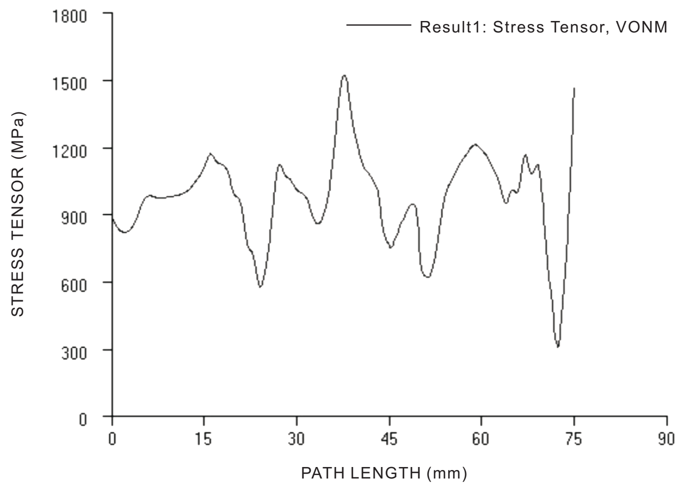

Figure 14. Stress distribution of a weld root.

It can be seen that the peak value of the longitudinal stress was 1,650 MPa and stresses were distributed more consistently if compared to Figs 13 and 14. On average, the specimen experiences a high tensor bending stress.

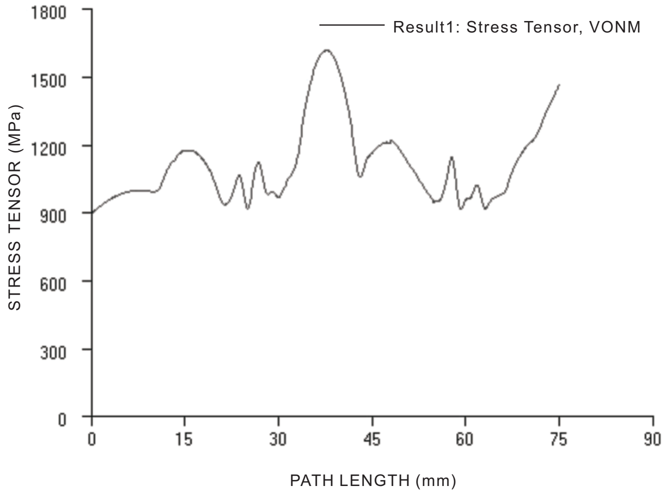

Figure 15. Longitudinal stress distribution of a flange.

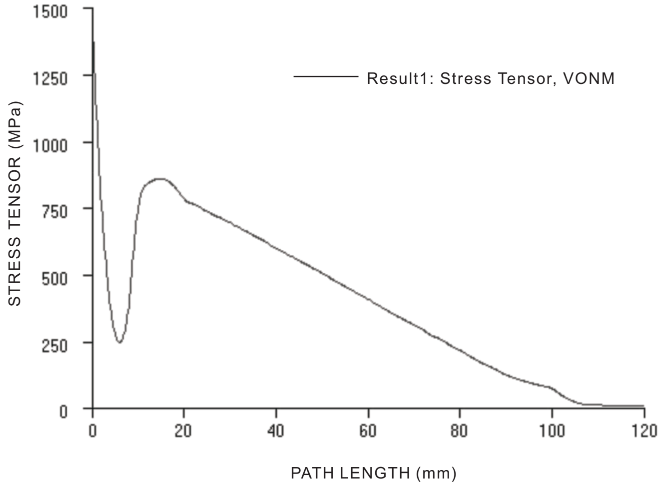

Figure 16. Transverse stress distribution of a flange.

Transverse stress starts from a high value (1,370 MPa) and becomes nearly zero at the end of the flange. This is correct from the theoretical view that the maximum stress point is located at the farthest distance from the applied load. It is expected that it will carry almost all the bending tensile stress. The overall prediction of the stress distribution using the FEM is excellent. It can be used to map the critical part for a specimen and thus estimate the useful life for it.

4.4 Critical Analysis

In the previous section the stresses of two main components were investigated. It was shown that the most critical part would be the connection of flange and weld root. According to Balasubramanian and Guha[6], the cracking that normally will cause failure of a fillet weld joint is root cracking or toe cracking. Furthermore, from the FEM model, it was clearly shown that the high stress is distributed along the longitudinal direction of the flange, as shown in Fig. 17, at the connection of the weld root and flange.

The critical point occurs at the centre of the connection

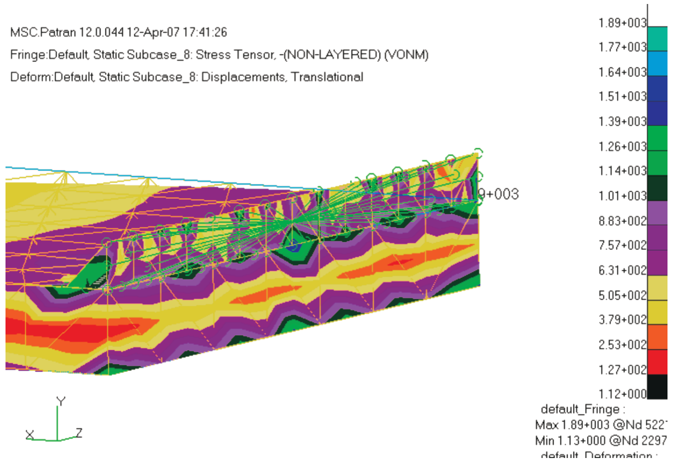

Figure 17. Visual FEM result for stress distribution in the connection of the weld root and flange.

and the stress condition may be regarded as a plane strain problem. Crack initiation will start from this point and propagate out along the connection. It is strongly believed that a failure would be here if there was further increase in the load.

5. CONCLUSIONS

The investigations were carried out on a T-joint fillet weld specimen. In this study, the FEM was used to predict the result that would be obtained from experimental work. The results demonstrate the usefulness of the FEM technique in the simulation and analysis of the behaviour of a T-joint fillet weld. From the surface strain calibration and joint deformation test, similar characteristics were demonstrated by the FEM analysis and experimental work. The elastic limit for this welded joint was 44 MPa. Even though there were some differences among the results, reasonable agreements were shown supported by the current research. Overall theoretical prediction of the strength and stress distribution of a T-joint fillet weld specimen is excellent.

From FEM simulation, the strength of a T-joint fillet weld was found to be highly dependent on the weld root. It was shown that this is a critical regime of crack initiation for a welded T-joint. The FEM can give a clear image about the critical point that may affect the total performance of a welded joint life. Therefore, prediction and modification can be made to suit the desired condition and requirement of the welded T-joint. The crack growth behaviour is influenced by the welding process utilised to fabricate the joint and the residual stress that accompanies it after heating. The results provide an illustrative example of how this approach can be used for future comparison and simulation.

REFERENCES

1. Inert gas welding, Engineering Fundamentals. Efunda Inc. Sunnyvale, CA, USA: 2006. http://www.efunda.com/processes/metal_processing/welding_inertgas.cfm?search_string=fillet%20weld. [Accessed on 3 October 2006].

2. Pinho da Cruz, J.A.M.; Costa, J.D.M.; Borrego, L.F.P.; & Ferreira, J.A.M. Fatigue life prediction in AlMgSi1 lap joint weldments. *Int. J. Fatigue*, 2000, **22**, 601-10.
3. Teng, T.L.; Fung, C.P.; Chang, P.H. & Yang, W.C. Analysis of residual stresses and distortion in t-joint fillet welds. *Int. J. Press. Vessel Pipe*, 2001, **78**, 523-38.
4. Mackerle, J. Finite element analysis and simulation of welding-an addendum: A bibliography. (1996-2001). *Model Simul. Mater. Sci. Eng.*, 2002, **10**, 295-318.
5. Teh, L.H. & Rasmussen, K.J.R. Strength of arc-welded T-joints between equal width cold-formed RHS. J. Constructional Steel Research., 2006, **63**, 571–79.
6. Balasubramanian, V. & Guha, B. Effect of welding processes on toe cracking behaviour of pressure vessel grade steel. *Eng. Failure Anal.*, 2003, **11**, 575-87.
7. ASTM International. Standard specification for general requirements for steel plates for pressure vessels, ASTM A20/A20M-07. 2007
8. American Welding Society, Specification for welding procedure and performance qualification (including 2006 Errata). ANSI/AWS B2.1:2005, 2004.
9. Kalpajian, S. & Schmid, S.R. Manufacturing Engineering and Technology", Ed. 4, Prentice Hall, Inc. Englewood Clifts, NJ, USA.2004, 826-28.