Numerical Fracture Analysis of Cryogenically Treated Alloy Steel Weldments

Rasool Mohideen. S*, Thamizhmanii. S, Muhammed Abdul Fatah. M.M and W. Najmuddin W. Saidin
International College of Automotive (ICAM)
DRB-HICOM Pekan Automotive Complex
Peramu Jaya Industrial Area
26607 Pekan, Pahang Darul Makmur, Malaysia

*rasmirasool@gmail.com

Abstract. Cryogenic treatment is being used commercially in the industries in the last two decades for improving the life of many engineering component such as bearings and cutting tools. Though their influence in improving the wear resistance of tool materials is well established, the effect of treatment on weldments is not much investigated. In the present work, a two dimensional finite element analysis was carried out on the compact tension specimen model for simulating the treatment process and to study the fracture behaviour. The weldments were modelled by thermo- mechanical coupled field analysis for simulating the temperature distribution in the model during weld pool cooling and introducing thermal stresses due to uneven contraction and cooling. The model was subjected to cryogenic treatment by adopting radiation effect. The fracture analysis was carried out using Rice’s J-Integral approach. The analysis produced a similar outcome of experimental results i.e. Increase in the fracture toughness of the specimen after cryogenic treatment in the heat affected zone of weldment.

1. Introduction
Welded Joints are generally in need of post weld heat treatment (PWHT) otherwise known as stress relieving to enhance mechanical properties, to decrease residual stresses and to attain dimensional stability [1]. Application of cryogenic treatment as a post weld treatment to weldments has not been used by many investigators even though it is expected to give improved fatigue performance in service [2]. Many investigators have tested the materials at sub zero temperatures such as at -180 °C [3]. But it has been found that very few information is available about treating the materials at cryogenic temperature and then testing them at room temperature to evaluate the mechanical behaviour of weldments [2]. The effect of cryogenic treatment on the mechanical behaviour of welded joints is needed to be studied in order to understand the mechanism of this process for many industrial applications.

In view of large variables involved in the welding process, creating adequately large experimental data base to understand and control welding process is expensive and time consuming if not impractical. An alternative is to simulate the weld process through set of mathematical equations representing the essential physical process of welding. Modelling and simulation capabilities of finite
element analysis and availability of high performance computing facilities made the analysis of welded structures easier.

Rice’s J integral Approach otherwise known as Elastic Plastic Fracture Toughness approach is widely used for the measurement of fracture toughness of engineering materials [4]. It is possible to obtain accurate results for homogeneous materials but for weldments a number of factors play a role during these testing bringing about a marked scatter in the fracture toughness values. The main influence came from weld residual stress, mismatching of mechanical properties between parent and weld metals and microstructural gradients. All of them take place simultaneously and their individual influence upon overall value is hard to discern [5]. Finite Element Analysis (FEA) is an useful tool to focus in just one of the factors to assess if it is playing an important or negligible role on the change of fracture toughness [6].

2. Modelling
Fracture Analysis of the weldment subjected to cryogenic treatment in this investigation is pursued by studying the elastic plastic fracture toughness of the weldment using J Integral approach. In order to simplify the investigation, a two dimensional symmetric model for the compact tension specimen (CTS) of the standard fracture toughness testing is developed and analysed using ANSYS FEM code. The specimen geometry for CTS (as per ASTM E 1820, 1999) in this investigation is shown in figure. 1 (all dimensions shown in the figure are in mm). The details of various aspects of modelling follow:

![Compact Tension Specimen Geometry](image)

Figure 1. Compact Tension Specimen Geometry

2.1. Material Modelling
Material under investigation is 1.25 Cr 0.5 Mo V steel (ASTM SA-387 Grade 11 Class I) of 10 mm thickness was chosen. It is widely used in the petroleum industry and in elevated temperature applications such as steam power generating equipment. The weldments were prepared using E 8018 Low Hydrogen electrodes. Chemical composition of the base material and the weld are detailed in Table 1.
Table 1. Chemical Composition of Base Metal and Weld Metal

| Composition       | C   | Mn  | Si  | P   | S   | Cr  | Mo  | V   | Al  | Ni  | Sn  | Cu  | N   |
|-------------------|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|-----|
| BM (ASTM SA-387)  | 0.162 | 0.614 | 0.612 | 0.009 | 0.009 | 1.36 | 0.606 | -   | -   | -   | -   | -   | -   |
| Grade 11 Class II |     |     |     |     |     |     |     |     |     |     |     |     |     |
| WM (E8018-B2L)    | 0.07 | 0.75 | 0.5 | 0.025 | 0.02 | 1.3 | 0.5 | 0.14 | 0.01 | 0.14 | 0.008 | 0.04 | 0.016 |

For the material modelling of FEA, Elasto-Plastic material property was incorporated in the parent metal by considering the multiple isotropic hardening option (MISO) in the ANSYS code. Table 2 gives the details of mechanical properties adopted and the stress strain relationship introduced in the property table is shown in figure 2.

Table 2. Mechanical Properties of Parent Metal

| Property                      | Value   |
|-------------------------------|---------|
| Modulus of elasticity X-Direction | 2.1e+01 N/m² |
| Thermal expansion coefficient in X-Direction | 1.1e-005 |
| Minor Poisson's ratio along Z-Plane | 0.33 |

Figure 2. Stress Strain Relation Ship for the Chosen Material
2.2. Finite Element Structural Modelling
A 2D structural eight noded element was used to model the Compac Tensile specimen (CTS) geometry. The 8-noded element has two degrees of freedom at each node: translations in the nodal x and y directions. This element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

2.3. Weld Modelling
Weldment in this analysis was modeled and analyzed by thermo mechanical coupled field analysis. Thermal model of weldments was created with an eight nodded 2D thermal element. This element has one degree of freedom, i.e., temperature, at each node. These 8-node elements have compatible temperature shapes and are well suited to model curved boundaries. The 8-node thermal element is applicable to a 2-D, steady-state or transient thermal analysis and is similar in geometry to that of the structural element chosen.

The molten weld pool has temperature dependent conductivity and enthalpy. The enthalpy table fed as material property for the thermal model captures the latent heat as the weld pool solidifies. The thermal properties of parent metal are given in Table 3.

| Table 3. Thermal Properties of Parent Metal |
|-------------------------------------------|
| Density | 7800 kg/m³ |
| Specific Heat | 560 J/kg k |
| Thermal conductivity in X-Direction | 40 W/m k |

3. Finite Element Analysis
The FEA in this investigation consists of thermo- mechanical analysis of CTS geometry to acquire the thermal loads in to the structural model during welding process and Fracture analysis of the geometry using the path dependent J-integral.

3.1. Thermo – Mechanical Coupled Field Analysis
In the Present work, a sequential thermo-mechanical analysis was adapted, where the nodal temperatures from thermal analysis were applied as the "body force" loads in the subsequent structural analysis. The Flow chart exhibited in figure 3 shows the process of thermo-mechanical coupled field analysis.

3.2. Boundary Conditions for Thermal Model
The temperature distribution during the solidification of weld pool was modeled as a thermal physics environment. The thermal model was subjected to convective heat transfer as the heat dissipates from weld pool to parent metal. Convection boundary conditions with initial temperature loads at weld pool and base metal were applied to the model. The boundary conditions for thermal model are shown in figure 4.

3.3. Boundary Conditions for Structural Physics in the Coupled Field Analysis
The thermal model was solved and its results were fed in to the structural model as thermal stresses. The CT specimen with the thermal stresses of welding was modelled as another structural physics environment. The structural model was then analysed for fracture toughness with fracture load similar
Thermo- Structural Coupled Field Finite Element Analysis

Thermal Modelling
- Plane 77 element
- Temperature dependent material property

Structural Modelling
- Plane 82 element
- Elastic and plastic material properties

Applying Boundary Conditions
- Temperature at weld pool
- Convection

Transient Thermal Analysis

Solution

Radiation Analysis
- Surf151 surface effect element

Heat Input
- Radiation node at 873 k

Cryo Input
- Radiation node at 77 k

Boundary Conditions
- Symmetric boundary conditions
- Temperature From Thermal analysis

Static Analysis

Residual Stress Build up

Restart Analysis
With additional thermal loading

Fracture loading

Transient Analysis

Solution

Post Processing

Figure 3. Flow chart of Thermo – Mechanical FEA
to base metal fracture analysis. Figure 5 shows the structural model with temperature distribution induced from thermal analysis.

3.4 Boundary conditions for Cryogenic Treatment
The Cryogenic Treatment was simulated in the thermal model using surface effect radiation element giving the treatment temperature at the extra node and uniform temperature at the specimen model. The model with extra node of radiation is shown in figure 6.

The treatment was modelled in three load steps. In the first load step, the temperature was decreased gradually from room temperature to the Cryogenic temperature of 88 K. In the second step the soaking was done and the third load step would be return to room temperature. The Cryogenic Treatment cycle adapted is shown in figure 7.

3.5 Fracture Toughness Testing Simulation and Analysis
Solving a fracture mechanics problem in FEA involves performing a linear elastic or elastic-plastic analysis and then using specialized post processing commands or macros to calculate desired fracture parameters. The most important region in a fracture model is the region around the edge of the crack which is referred to as a crack tip in a 2-D model. The stresses and strains should be singular at the crack tip.

The Structural model is subjected to tensile load as in the typical fracture toughness testing. The boundary conditions are fracture analysis is shown in figure 5. Once the structural analysis is
completed, the general postprocessor would be used to calculate fracture parameters. Typical fracture parameters are stress intensity factor, the J-integral, and the energy release rate. As suggested by many investigators J integrals for different conditions are evaluated in the present work.

3.5.1 Rice’s J integral: It is defined as a path-independent line integral that measures the strength of the singular stresses and strains near a crack tip. The Equation below shows an expression for J in its 2-D form.

\[
J = \int_W dy - \int_{\Gamma} \left( t_x \frac{\partial u_x}{\partial x} + t_y \frac{\partial u_y}{\partial y} \right) ds
\]

Where \( \Gamma \) = any path surrounding the crack tip
\( W \) = strain energy density (strain energy per unit volume)
\( t_x \) = traction vector along x axis = \( \sigma_x n_x + \sigma_{xy} n_y \)
\( t_y \) = traction vector along y axis = \( \sigma_y n_y + \sigma_{yx} n_x \)
\( \sigma \) = component stress
\( u \) = displacement vector
\( s \) = distance along the path \( \gamma \)

It assumes that the crack lies in the global Cartesian X-Y plane, with X parallel to the crack. Fig.10(a) shows a typical J integral contour path surrounding the crack tip and Fig. 10 (b) shows the paths chosen around the crack tip for our model to calculate the J Integral. A macro or sub routine code is written for evaluating J integral.

![Figure 8(a). Typical J Integral](image1)

![Figure 8(b). J –Integral path in the actual Model](image2)

4. Results and Discussion
The Finite element analysis was carried out on the CT specimen model and the results obtained from various analysis such as Thermal, Structural, Thermo-Mechanical and Fracture Analysis are discussed

4.1. Thermal Analysis
The results of transient weld pool analysis for the chosen period are shown in figure 9. The figure shows temperature distributions across the weldment from weld centre through Heat Affected Zone
HAZ) to base metal regional. The cooling iso-therms show sudden temperature fall in weld region and no or less changes in the base metal region.

Figure 9. Temperature Distribution During weld pool cooling

4.2. Structural Analysis

The temperature obtained during weld pool cooling is given as the input for structural analysis and this developed residual stresses in the structural model. Figure 5 shows the inbuilt body temperatures from thermal analysis input and figure 10 shows the residual stress developed in the structural model. The residual stress in tensile in the weld region and compressive in the other regions which is same as a typical residual stress formation in any weldment.

Figure 10. Residual Stress Developed in the Structural Model
4.3. Fracture Analysis

The results of transient fracture analysis done on the CT model after thermal analysis simulating Cryogenic treatment is shown in terms stress intensity factors (SIF). The SIF from cryogenically treated structural model is compared with that of as welded model and base metal model. The values shown are SIF of different models in the various zones of weldments such as weld zone, HAF zone and base metal region. On comparison it can be observed that the stress intensity factor is higher in the weld region than the HAZ region and base metal. Hence they are likely to have less fracture toughness. Figure 11 to Figure 14 show various SIF obtained for different Models and conditions.

Figure 11. SIF (Base Metal)

Figure 12. SIF (As Welded Weld)

Figure 13. SIF (Cryo Weld)

Figure 14. SIF (Cryo-HAZ)
4.4. J Integral

After solving the structural model under each condition, the J integral calculation is done using the sub routine code and the values of J integral for various regions under different conditions are obtained as shown in table 3. As expected the weld region show poor fracture toughness than the heat affected zone in the cryogenically treated specimens when compared to as welded specimen. The FEA J integral values are also compared with the experimental values and they are quite similar in trend. Figure 18 shows the J Integral values of FEA simulation in comparison with the experimental analysis.

| Specimens | J Integral Values (N/mm) |
|-----------|------------------------|
| BM        | 612.9                  |
| AW        | 511.26                 |
| CW        | 539.54                 |
| AH        | 401.8                  |
| CH        | 650.8                  |

BM- Base Metal, AW- As welded Weld, CW- Cryo Weld, AH- As welded HAZ, CH- Cryo HAZ

5. Conclusion
The numerical fracture analysis of Alloy steel weldments after subjecting them to cryogenic treatment at 77K is done and the following concussions are arrived

- Finite Element Simulation of Cryogenic Processing of alloy steel weldments is carried out successfully using Thermo- Mechanical coupled field analysis
• Simulation of Fracture toughness testing using CT specimen to find the fracture toughness in
the various regions of weldments such as weld region, HAZ and parent metal region is done
• J Integral values are determined using Rice Path Independent method and the result from finite
element analysis is compared with that of experimental method.
• Finite element analysis produced similar values of J integral with respect to the various
regions of weldments subjected to different conditions

6. Acknowledgment

Authors acknowledge the financial support provided by International College of Automotive and those
who help to complete the work successfully.

7. References

[1] Bob Irving 1991 Why Heat Treatment Is Crucial In Weld Fabrication Welding Journal 41
[2] Johan Singh P., Guha B. and Achar D.R.G. 2003 Fatigue Life Improvement of AISI 304L
cruciform welded joints by Cryogenic treatment Engineering Failure Analysis. 10 1-12
[3] Shindo Y., Mano Y., Horiguchi K. and Sugo T. 2001Cryogenic Fracture Toughness
determination of structural alloy weldments by notch tensile measurement ad Finite element
analysis Journal of Engineering Materials and Technology Transaction of ASME
123(1) 45
[4] Rice J.R., Paris P.C. and Merkle J.G. 1973 Some Further Results of J-Integral Analysis and
Estimates ASTM STP 536 231-245
[5] Wang, Qiu-Cheng, Ke and Ying-Lin 2003 Relief of residual stresses in 7050 aluminum alloy
by deep cryogenic treatment Journal of Zhejiang University Engineering Science. 37 6 748-751
[6] Suarez Juan C., Murakawa Hidekazu and Ueda Yukio 1996 Effect of Residual Stresses on
Fracture toughness Testing. Part 1 FEM Analysis of CTOD specimens in multi pass
weldments of thick plates Transactions of JWRI 25. 91-99
[7] S. Rasool Mohideen, Ahmad Mujahid Ahmad Zaidi 2010 Investigation on Fracture
Behavior of Low Alloy Steel Weldments Subjected to Controlled Cryogenic
Treatment International Journal of Advances in Mechanics and Applications of
Industrial Materials 1 (1) 61