In-Service Welding Simulation of 28” Pipeline Using Finite Element Method

Sahrudin Tambunan¹, Ardiyansyah Yatim², Putu Wira Sanjaya¹ and Michael Simon P H¹

¹PT 3S Engineering, Bukit Rivaria, Depok, Indonesia.
²Mechanical Engineering, Universitas Indonesia, Depok, Indonesia.

E-mail: Sahrudin.tambunan@3S-eng.co.id, ardiyansyah@eng.ui.ac.id

Abstract. To maintain the integrity of the gas pipeline, the repairing of pipeline system with common method such as cut and replaces method by conducting hot tap and bypass or clamp method is conducted. Both methods require in-service welding in the process. During in-service welding, the process affects the structure strength. The structure then needs to be analysed during the welding process. There are two common problems associated with in-service welding. Firstly, the high gas flow in pipe causes the weld rapidly to cool due to the convective transfer of heat. The other problem is burn-through during in-service welding. Pressurized natural gas imposes a significant stress on the pipe wall, and since the pipe strength is decreased during welding, this cause failure in pipe wall. Burn-through occurs when the region around the weld pool has insufficient strength to withstand the internal gas pressure. This paper shows the finite element analysis procedure of In-Service Welding Analysis to avoid burn-through failure. A thermal-mechanical based Finite Element model had been conducted to assess the risk. The model is simulated using three-dimension (3D) mechanical, thermos-elastic-plastic-metallurgical finite element computational procedure. The temperature, stress and strain parameters of inner surface were used to assess the model. The temperature and strain check are below the allowable value while the stress check with some applicable method which are based on either stress or strain based.

1. Introduction

In-service welding is commonly used for the fabrication of structures ranging from, large to small and simple to complex especially when the system cannot be shut down during the repair process. Fusion welding is a significant engineering process because of the unparalleled advantages it has over other joining methods. However, welding process causes the material properties to change that result in weaker material. To capture this changes, finite element analysis need to be conducted.

Finite element method has been widely used to simulate the in-service welding process especially when the analytical models hit its limitations. Heat source distribution applied in the finite element should represent the actual process. A model of heat distribution was introduced to accommodate the real welding process.

Therefore, the analysis of in-service welding process needs to be presented to show that the parameters obtained from finite element are still within allowable value. The pipeline is shown in table 1 [1-4].
Table 1. Pipeline Data

| Parameter          | Unit | Value     |
|--------------------|------|-----------|
| Outside Diameter   | mm   | 711.2     |
| Wall Thickness     | mm   | 8.7       |
| Material           | -    | API 5L X65|
| Operating Pressure | Psi  | 950       |
| Design Pressure    | Psi  | 1151      |
| Operating Temperature | °F | 80       |
| Design Temperature | °F   | 150       |
| Flow Velocity      | m/s  | 5.5       |
| Content            | -    | Natural Gas|

2. Modelling and Assessment Method

2.1. Thermo-Mechanical Analysis
The in-service welding analysis was conducted with two type analysis, non-linear mechanical analysis and transient thermal analysis. Non-linear mechanical analysis was performed by applying pressure load. While the transient thermal analysis, was performed by applying thermal load from the welding process. Non-liner mechanical and transient analyses are performed simultaneously.

2.2. Thermal Boundary Condition
Radiation losses will be dominant at high temperature area and in the weld zone while convection losses has a major role in low temperature area which away from the weld pool. The combined convective and radiative heat transfer coefficient was taken into account with the following equation[5]:

\[ h = \begin{cases} 
0.0668T \left( \frac{W}{m^2 \cdot ^\circ C} \right) & \text{when } 0 \leq T \leq 500^\circ C \\
(0.231T - 82.1) \left( \frac{W}{m^2 \cdot ^\circ C} \right) & \text{when } T \geq 500^\circ C 
\end{cases} \]  

(1)

The equation was applied to the outer surface of the 3D model. While convection heat transfer coefficient at the inner side of the pipe-wall was determined by the equation below [5]:

\[ \frac{h_g}{k_g} = 0.023 \left( \frac{\rho_g \mu_g}{\rho_g} \right)^{0.8} \left( \frac{C_p \mu_g}{k_g} \right)^{0.4} \]  

(2)

Where, \( k_g, \rho_g \) and \( C_p \) are the thermal conductivity, gas density and specific heat respectively. \( \mu_g \) and \( v_g \) are the viscosity and gas velocity respectively. \( D \) is the inside pipe diameter.

2.3. Weld Heat Source Modelling
The double ellipsoidal heat source model was used in this analysis. It based on Gaussian distribution of power density (W/m³) that was introduced in [6] as shown in figure 1. This model is able to analyse the thermal history of shallow and deep penetration weld. It simulates welding action and capable of transporting heat well below the surface of the weld pool. This model combines two ellipsoidal sources that represent the actual experimental experience i.e. the temperature gradient in front of the source is much higher, compared to lower gradient in the rear of the heat source [7].
Figure 1. Double ellipsoidal heat source model

The power density distribution expressed in moving Cartesian coordinate system (x, y and z), is explained in the following equation.

Distribution inside the front quadrant:

\[ q_f(x, y, z, t) = \frac{6\sqrt{3}fQ}{abc\sqrt{\pi}} \exp\left\{-3\left(\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z+vt)^2}{c_f^2}\right)\right\} \]  \hspace{1cm} (3)

For the rear quadrant:

\[ q_r(x, y, z, t) = \frac{6\sqrt{3}fQ}{abc\sqrt{\pi}} \exp\left\{-3\left(\frac{x^2}{a^2} + \frac{y^2}{b^2} + \frac{(z+vt)^2}{c_r^2}\right)\right\} \]  \hspace{1cm} (4)

DFLUX, an Abaqus user subroutine [8], was used to apply the non-uniform distributed flux as function of time, position, temperature to the model during heat transfer analysis. Parameters below as shown in table 2 [5] to be used in double ellipsoidal power density distribution equation

| Parameter                        | Symbol | Unit | Value | Ref  |
|----------------------------------|--------|------|-------|------|
| Length of front ellipsoidal      | \(c_f\) | mm   | 12.9  | [10] |
| Length of rear ellipsoidal       | \(c_r\) | mm   | 10.3  | [10] |
| Depth of the heat source         | \(b\)  | mm   | 4     | [10] |
| Width of the heat source         | \(a\)  | mm   | 5     | [10] |
| Heat fraction (front)            | \(f_f\) | -    | 6     | [10] |
| Heat fraction (rear)             | \(f_r\) | -    | 8     | [10] |

2.4. Analysis Setup

The pipe was modelled as a half circle to reduce the number of elements that will reduce running time. The most critical case of in-service welding analysis is when the pipe is directly exposed to a heat source (welding), hence this analysis was focused in pipe section only. The analysis was done in the first pass sequence only because in this case all heat sources from welding will be absorbed by the pipe only.
While in other cases when the pipe and split tee are exposed to heat at the same time, it is not as critical as the first case because in that case, the pipe has shared the heat with a split tee.

2.5. **Assessment Criteria**

The analysis was carried out by using Finite Element software. The split tee model was assessed using the following step.

Temperature criteria were checked according to the API RP2201 – 1995 [9] recommendation. The critical temperature for welding process is 980°C (1250 K). If the temperature higher than recommendation then high risk of burn through will be happen.

Stress criteria were checked according to ASME B31.8 [10] and Vakili et.al. [7]. Based on the ASME B31.8, the combined stress (von Mises) shall not exceed the given value of 0.9xSMYS and the temperature of pipe shall not exceed 232°C. (Table 841.1.8-1). While Vakili et.al [7], in preventing burn-through effective stress along the main pipe wall should be checked against the yield stress at the same temperature.

Strain criteria were checked according to ASME B31.8 or ASME BPVC VIII Division 2 [10]. Based on the ASME B31.8, the maximum permitted strain is limited to 2%. While ASME BPVC VIII Division 2, the calculated equivalent plastic strain (PEEQ) from FE results should be lower than the minimum of plastic strain at tensile strength and limiting triaxial strain ($\varepsilon_L$).

Displacement criteria were checked by measure the radial deformation. The maximum allowable radial deformation is 1 mm.

First assessment step is temperature check. If temperature check is pass then it will be continued with stress or strain or displacement check. Until one of the parameters is ok then the structure will within the allowable value.

3. **Finite Element Results**

The analysis was evaluated based on the design criteria in section 2.5. Those parameters are obtained from element that located on inner surface pipe and centre line of weld pass as shown in the following figure 3. History of temperature, stress, and strain at inner surface of pipe during welding are given in figure 4, figure 5 and figure 6 respectively.
Figure 3. Location of evaluated line – Inner surface

Figure 4. Temperature history at the evaluated line – Inner surface

Figure 5. Combined stress history at the evaluated line – Inner surface
Figure 6. Strain history at the evaluated line – Inner surface

The colour variation of the curve in the figure above shows the state of the inner surface node of the pipe that is passed by the heat source right on the top surface. Based on the figure above, maximum temperature, maximum combined stress and strain during welding in the evaluated area are 425°C, 300 MPa and 0.08% respectively as shown in the following table 3.

| Design Criteria                          | Unit | Value | Allowable | Status |
|------------------------------------------|------|-------|-----------|--------|
| Temperature Check                        | °C   | 425   | 980       | PASS   |
| Stress Check (Vakili et.al [7])          | MPa  | 300   | 268       | FAIL   |
| Stress Check (ASME B31.8 [1])           | MPa  | 295   | 240       | FAIL   |
| Strain Check (ASME B31.8 para 833.5 [10-12]) | %    | 0.08  | 2         | PASS   |

The value of the parameters temperature check and strain check are below the allowable value. While the stress parameters are above the allowable value. Although stress parameters indicate that the value are outside of the allowable value but since temperature and strain parameter pass the assessment then the structure are within the allowable value[13-16].

4. Discussion and conclusions

Thermal-mechanical based Finite Element (FE) model had been utilized to assess the risk of burn-through during the welding of pressurized gas pipelines. In this analysis, the pipe sleeve will be assessed based on temperature, stress and/or strain parameter. The welding of Split Tee was simulated using three-dimension (3D) mechanical, thermos-mechanical and metallurgical finite element model.

The FEM pre-processing, calculation have been carried out by finite element software. The thermos-elastic-plastic-metallurgical finite element computational procedure was performed to analyse the welding temperature and the welding residual stress in pipeline which has been welded when the pipeline still in operating condition (in-service welding).

The analysis has shown that assessment parameters obtained from in-service welding simulation with internal pressure of 950 psig are within allowable value.
References

[1] 28-in Pipeline Repair Application Data Sheet.
[2] Stopple Plus Tee 28x28 in 600 lb RF.
[3] ASME BPVC: Materials, 2015.
[4] API 5L 2015 Specification of Line Pipe.
[5] ABID et.al; 2009 3D Thermal Finite Element Analysis of Single Pass Girth Welded Low Carbon Steel Pipe-Flange Joints, Turkish J. Eng. Env. Sci.
[6] Brickstad B.B.I, “Josefson 1998 A Parametric Study of Residual Stresses in Multiphase but Welded Stainless Steel Pipes Journal of Pressure Vessel 75, pp 11-25.
[7] Farid Vakili-Tahami Finite Element Analysis of the In-Service Welding of T Joint Pipe Connections.
[8] Simulia Abaqus version 6.13.
[9] API RP 2201. 1995 Safe Hot Tapping Practices in the Petroleum & Petrochemical Industries.
[10] ASME Section VIII Div.2. 2017ASME BPVC: Rules for Construction Pressure Bessel – Alternative Rules.
[11] AWSW A5.1 2004 Specification for Carbon Steel Electrodes for Shielded Metal Arc Welding.
[12] Welding Procedure Specification Pipe Sleeve API 5L X65.
[13] In-Service Welding Analysis Report of Pipe Sleeve Design – Pipeline Repair Option A.
[14] Parmar,R.S 2002 Welding Engineering and Technology (Delhi: Khanna Publisher).
[15] Sabapathy, P.N. 2002 Predicting Weld Cooling Rates and The Onset of Failure during In-Service Welding (Adelaide: The University of Adelaide).
[16] Karayan, A.I., Hersuni, A., Adisty D., Yatim, A. Failure analysis of seawater inlet pipe. Journal of Failure Analysis and Prevention Volume 11, Issue 5, October 2011, Pages 481-486