Temperature Prediction in Hot Tapping Process for High Pressure Pipeline

Muhammad Aizadt Rosman, William Pao
Universiti Teknologi PETRONAS, Bandar Seri Iskandar 32610, Perak, Malaysia
Corresponding author: William.pao@utp.edu.my

Abstract: Welding onto an in-service pipeline during hot tapping process are extremely risky and require thorough preparation and precautions, where the probability of occurrence of occupational hazards and injuries are high. Since the increase in surrounding temperature inside the working system are unpredictable, the internal pipe wall temperature and the process fluid temperature inside the pipe are unknown during operation. Therefore, this project is conducted to i) predict inner pipe wall temperature and ii) to estimate temperature of the process fluid inside the pipeline during in-service welding and iii) to construct relevant methodology and reference chart in predicting internal fluid temperature by using existing information from wall temperature chart provided by PTS 31.38.60.10 "Hot-Tapping on Pipelines, Piping and Equipment". These objectives will be achieved by conducting thermal Computational Fluid Dynamics (CFD) simulations on 2D heat transfer model on a pipe's cross-sectional plane by using ANSYS Mechanical APDL. The thermal CFD simulations for this project are divided into two phases. In the first phase of this project, CFD simulations are conducted without the introduction of process fluid. The data obtained will be used to compare with the information gathered from existing PTS chart for the purpose of CFD model validation. Then, the second phase of the project is conducted by introducing process fluid inside the validated 2D model. All of the temperature data obtained will be used to construct a new temperature prediction chart for hot-tapping process which include the process fluid parameter. At the end of this project, it is expected that the proposed temperature prediction chart will be able to assist field engineers and operators in estimating the temperature of critical locations inside the working system and establishing a safe permissible temperature range to safely conduct in-service welding to avoid explosion and burn through risk.

Keywords: ***hot-tapping, in-service, heat source modelling, finite element analysis, burn-through, auto ignition temperature***

1. INTRODUCTION

Welding onto in-service pipeline in hot tapping process are usually required in order to facilitate repair and modifications to an existing pipeline, such as to install a branch connection to the main pipeline in a process plant without having shutdown. However, welding onto in-service pipeline is a high risk activity where strict precautions and engineering considerations must be taken at all times during performing the activity. The most common risks encountered are leaking and explosion by weld burn-through, chemical reaction due to system instability and increased susceptibility of pipe wall material to hydrogen induced cracking that can result in loss of structure's strength.

The risk of hydrogen induced cracking at the weld points arise when the cooling rate and heat loss from molten weld pool is accelerated due to high flow rate of process fluid flowing inside pipeline. As a result, the weld heat are removed away from the pipe wall material in short amount of time, resulting in fast weld cooling rate. Apart from fluid's flow rate, this phenomena are dependent on various fluid flow characteristics such as type of fluid, density, viscosity, velocity, hydrostatic and dynamic pressure, thermal conductivity, heat transfer coefficients and more. Pipe's geometrical, physical, mechanical, and metallurgical properties also contribute to this phenomena. Moreover, fast weld cooling rate will decrease the HAZ toughness at the weld points, which in turns reduce the pipe's mechanical strength. This situation might lead to localized rupture at weld points due to high internal pressure acting along the radial direction of a pipeline. Another common issue is the risk of burn-through, where depth of penetration of molten weld pool become more significant as the temperature of the weld pool is raised. Here, the thermochemical interaction between the fluids with the increasing temperature of the inner surface of the pipe (i.e. due to fluid's flammability) can be violent, and might lead to sudden explosion in the pipe.

According to Sabapathy et al. (2005), one the factors that make in-service welding difficult is the characteristics of internal flow of process fluid (gas or liquid) inside the pipeline which create large heat losses across the pipe wall during welding. This results in fast weld cooling rates where the welds will most likely to have hard heat affected zones (HAZ) and decreased in HAZ toughness through formation of hardened areas. This situation will increase their possibility and susceptibility to hydrogen induced cracking (HIC). The second problem addressed by Sabapathy et al. (2005) is the loss of mechanical strength due to high temperature rise during welding. The loss of mechanical strength will create the possibilities of localized rupture in the pipe wall structure due to high internal process fluid pressure. The third factor is the violent interaction between the fluid and the inner pipe wall surface in high temperature environment which can lead to explosion as addressed in API 577 "Welding Inspection and Metallurgy".

Current studies by EWI/BMI (Edison Welding Institute and Battelle Memorial Institute) provides a numerical 2D finite difference model to simulate welding and hot tapping onto in-service pipelines that allows the prediction of inner pipe wall surface temperature and the weld cooling time for given set of boundary conditions. This model evaluates the risk of penetration (burn-through) and limits the risk of hydrogen cracking in the HAZ region during welding process. Meanwhile, Goldak et al. (1992) and Sabapathy et al. (2005) used a 3D finite element model to calculate the thermal fields for circumferential fillet welds of direct branching. Their research finding shows that the shape and the weld bead size have strong influence on the calculated depth of weld penetration and temperature profile around the weld pool.

However, there is a lack of study in terms of prediction of process fluid temperature inside a pipeline during in-service welding. Therefore, the main purposes of this project are to predict the inner surface temperature of pipe wall during welding as well as the temperature of the process fluid inside the pipeline for given set of boundary conditions. In order to achieve the target, this paper will construct relevant methodology and a reference chart to predict the fluid temperature by using existing information from wall temperature chart provided by PTS 31.38.60.10 "Hot-Tapping on Pipelines, Piping and Equipment".

2. HOT TAPPING RISK ANALYSIS

2.1 Hot Tapping Risks

When dealing with welding onto in-service pipeline for hot tapping process, there are three common risks: leaking and explosion via burn-through, thermochemical reaction inside pipeline due to chemical instability at high temperatures and hydrogen induced cracking (HIC) at the weld locations. According to Sabapathy (2005), hydrogen induced cracking (HIC) in high strength steels are particularly caused by the flow of fluid (liquid or gas) inside the pipelines which tend to cause a large heat loss in the pipe wall, resulting in fast weld cooling. During fast weld cooling of molten weld pool area at the heat affected zone (HAZ) of the base metal, the metallurgical and chemical properties in that areas are altered. This cause material's sensitization, cracking and reduction of material's resistance towards corrosion. Toughness of HAZ will decrease through formation of hard microstructures and creep which are brittle and hard at the affected region (Lima, 2014). Metallurgical changes also can lead to the formation of nitrides at the HAZ, which can affect the weldability making the process of welding more difficult. The factors that influence the characteristics of HAZ at the weld locations include the properties of base material, properties of weld filler materials for non-autogenous welding processes, and the concentration of heat input during welding.

In most application, weld personnel and engineers will usually increase the heat input to reduce significant heat loss into the flowing process fluid in order to deal with high cooling rate issues during in-service welding. However, it should be noted that this action can cause loss of mechanical strength in pipe's material due to significant temperature rise in the system. As a consequent, high local stresses in or near the HAZ are formed during welding. This situation will directly induce localized rupture of the pipe wall due to high radial forces coming from internal fluid pressure when the weld heat input is increased (Lima, 2014). The depth of weld penetration and risk of pipe wall burn-through are also increased. However, the research conducted by Tahami and Asl (2009) stated that localized rupture may occur even with partial penetration from welding by considering the effect of internal pressure added with existing thermal stresses in the pipe wall. Therefore, in practical application as addressed in API 2201 "Procedures for Welding or Hot Tapping on Equipment in Service'', welding process for pipe wall with thickness below 5mm are strictly restricted to avoid risk of burn-through due to high heat input generated from welding processes. Based on the study conducted by Lima (2005), he limits the weld temperature to be 980°C for low hydrogen electrodes and 760°C for cellulosic electrodes to avoid the risk of penetration.

Hydrogen induced cracking which occurs when ambient hydrogen permeate into the pipe wall during welding at high temperatures can be diffused from the welded area by conducting post weld heat treatment (PWHT). This process is known as post heating where the material needs to be heated to a certain temperature for a number of hours and gradually cooled, depends on the thickness and properties of the materials involved. Usually, materials with higher carbon content or carbon equivalent (CE) are more likely required to undergo PWHT. This is very important consideration since the high-strength low-alloy steel (HSLA) material for most pipe application involved for hot tapping process have significant amount of CE to suit for weldability. Post weld heat treatment should be conducted immediately after the weld process is completed rather than allowing the weld to eventually cool. This will prevent significant heat loss into surrounding environment, especially into the process fluid. At the same time, this heat treatment serves to relieve the residual stress formed due to increase in temperature during welding, out of the system.

In addition to the risks of welding onto in-service pipelines, Lima (2014) considered a third factor which is the interaction between the process fluid and the temperature on the inner surface of the pipe when the temperature of the system is raised significantly during welding process. He furthermore addressed that internal explosion might occur due to instable thermochemical reaction as most process product involved in hot tapping for oil and gas applications are flammable and has low flash point. This findings are consistent with the statement addressed in API 577 "Welding Inspection and Metallurgy".

2.2 Existing Methodologies Developed in Hot Tapping Analysis

To cope with the hot tapping related risks, numerical simulation of welding service has been demonstrated by the work of EWI/BMI (Edison Welding Institute/Battelle Memorial Institute) to predict the inner surface temperature of the pipe and the cooling time for the molten weld to solidify ($\Delta T_{800\text{-}500}$), for given set of welding parameters, pipe geometry and properties, along with the coefficient of heat transfer by convection which is obtained empirically as the function of process fluid flowing inside the pipeline. This is a 2D model finite difference which simulates the welding gloves of welding activity conducted during hot tapping. Welding processes are considered as thermal-mechanical-metallurgical coupled processes. Therefore, the most important boundary condition in BMI/EWI model is the heat source modelling.

During welding, the heat input melts both filler materials added to the base metals creating a molten pool area. Compared to the traditional punctual and linear welding heat source assumption, Lima (2014) modelled the heat source from the welding activity by using Gaussian Heat Source Distribution which is more realistic and accurate to be implemented practically. Gaussian heat source distribution is a model to describe heat generated distribution over a surface.

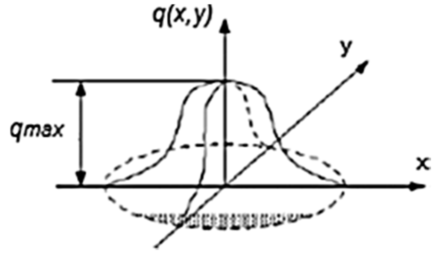

Fig. 1 Gaussian heat source distribution model

Research conducted by Goldak et al. and Sabapathy et al. in 1992, found that the shape and the weld bead size have a strong influence on the calculated depth of penetration and temperature profile around the molten weld pool. They have used a 3D finite element model to calculate the thermal fields for circumferential filet welds of direct branching. According to Sabapathy et al. (1992), the use of empirical relations between the welding parameters and the size and shape of the weld bead is an appropriate way to define the geometry of the weld pool and the coordinate of the heat sources. Sabapathy et al. and Goldak et al. also develop an equation to characterize the heat distribution by non-autogenous welding sources which is the Double Ellipsoidal Heat Source (DEHS) model which defines the heat flow Q (kJ/mm^3). The model is further described by Equation (1) and Figure 2.

$$q(x,y,\xi,t) = \frac{6f\sqrt{3Q}}{abc\pi\sqrt{\pi}} e^{\frac{-3x^2}{a^2}} e^{\frac{-3y^2}{b^2}} e^{\frac{-3z^2}{c^2}} \tag{1}$$

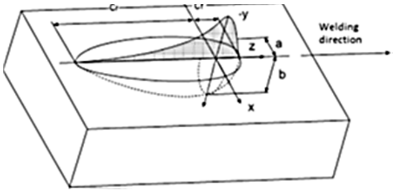

Fig. 2 Double ellipsoidal heat source (DEHS) model developed by Goldak and Akhlagi

This equation considers the factors of amount of heat input, pipe's thickness, weld speed, voltage, arc efficiency, type of welding processes to estimate the size and shape (geometries) of molten weld pool in terms of width, depth, and length which are represented by symbols: a, b and c respectively. This allow the analysis on burn-through prediction to avoid the risk of penetration in welding onto in-service pipelines.

3. RESEARCH METHODOLOGY

3.1 Prediction of Inner Pipe Wall Temperature and Depth of Penetration by Using PTS Chart

For the purpose of numerical calculations and simulations, the pipe material selected is API 5L Grade B Carbon Steel Pipe which is commonly used in pipelines application. The selection of pipe geometries (nominal pipe size, outside diameter and wall thickness) are done by referring to ASME/ANSI B36.10 "Welded and Seamless Steel Pipe" on Schedule 30.

Pipe size of 12 inch are selected for the purpose of numerical calculations and simulations. The geometric parameters for the pipe selected are:

- Nominal diameter : 12 inch or 300 mm
- Outer Diameter : 12.75 inch or 323.8 mm
- Pipe Wall Thickness : 0.330 inch or 8.38 mm

In this calculation, the annual corrosion rate for the pipeline is not applied, where it is assumed that the pipe material is new and experience no corrosion. Therefore, it is assumed that the actual pipe wall thickness is the same as the pipe wall thickness listed in the catalogue which is 8.38 mm.

In order to estimate the inner pipe wall temperature during welding, heat input from welding must first be identified. The formulation of heat input from welding is defined by Equation (2).

$$\mathrm{HI} = \mathrm{K}\left(\frac{\mathrm{V.A}}{\mathrm{S}}\right) \tag{2}$$

The heat input from welding calculation only consider the value obtained from first pass welding where GTAW (Gas Tungsten Arc Welding) or TIG (Tungsten Inert Gas Welding) process is selected. GTAW process is selected for first pass weld since this process allow precise control of welding variables especially the amount of heat input generated in order to avoid possibility of total penetration through the pipe wall or risk of burn through. The parameters selected for the calculation of heat input are listed as follows:

- Amperage, A = 180 A
- Voltage , V = 15 V
- Welding travel speed, S = 100 mm/min
- Net factor, K = 0.57 for fillet welds

Therefore, heat input generated from welding:

$$\mathrm{HI} = \mathrm{K}\left(\frac{\mathrm{V.A}}{\mathrm{S}}\right)$$

$$\mathrm{HI} = 0.57\left(\frac{15 \times 180}{1.667}\right)$$

$$\mathrm{HI} = 923.26\ \mathrm{J/mm}$$

Then, the value of heat input obtained will be used to estimate the depth of weld penetration and maximum inner pipe wall temperature by using Appendix 2 "Welding Temperature of Pipe Wall – Initial Pipe Wall temperature of 25°C" from PTS 31.38.60.10 "Hot Tapping on Pipelines, Piping and Equipment".

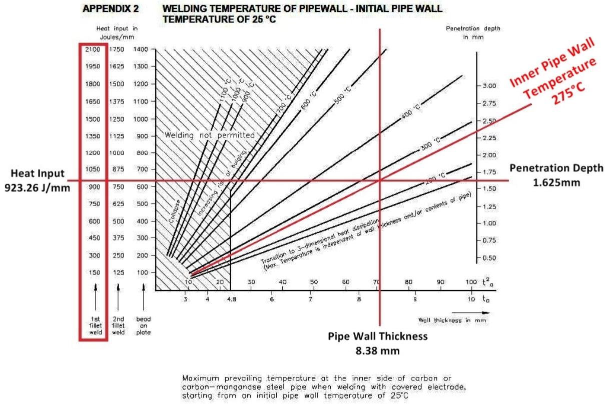

Fig. 3 Estimation of Inner Pipe Wall Temperature Using PTS Chart

From Figure 3, the inner pipe wall temperature during welding is estimated to be 275°C while depth of penetration is 1.625 mm.

3.2 Maximum Allowable Internal Pressure inside Pipeline during Welding

The calculation of maximum allowable internal pressure inside pipeline during welding is given by the equation below:

$$\mathrm{P} = \frac{2\mathrm{S{\cdot}t{\cdot}F{\cdot}E{\cdot}T}}{\mathrm{D}} \tag{3}$$

3.3 ANSYS 2D finite Element Model Development

In order to predict inner pipe wall temperature and process fluid temperature inside a pipeline during welding, 2D finite element model are developed and simulated under specified boundary conditions by using ANSYS Mechanical Workbench software under Steady State Thermal analysis option. The simulations are divided into two stages as mentioned in the previous project methodology section.

The first stage of ANSYS simulations are done to compare and validate the value of inner pipe wall temperature obtained from ANSYS simulations with the value obtained from the former method of using temperature prediction chart provided by PTS. Once these values have been compared and validated with each other, the methodology of prediction inner pipe wall temperature by using defined ANSYS simulation model is proven to be accurate. This will help to obtain accurate results for next stage of ANSYS simulations: to predict inner fluid temperature inside the pipeline during welding processes where a process fluid is introduced into the validated ANSYS 2D finite element model. The 2D finite element model is constructed based on the cross-sectional plane of a pipeline. It is assumed that the heat transfer mechanism through the pipe wall material occurs via pure conduction and heat transfer via free convection in the process fluid and to ambient atmosphere. Figure 4 shows the visualization of problem as a 2D heat transfer finite element model.

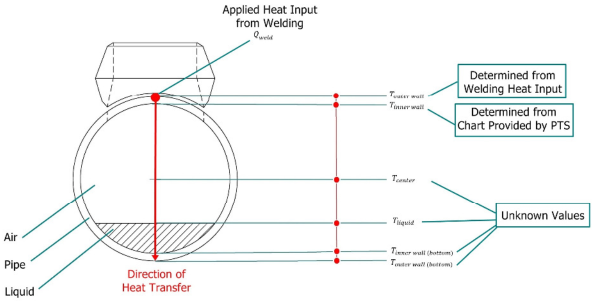

Fig. 4 Visualization of problem as 2D heat transfer finite element model

As indicated in Figure 4, the point of interest of $T_{inner\ wall}$ (inner pipe wall temperature) is located directly beneath the location of $T_{outer\ wall}$ (outer pipe wall temperature). Previously, the value of $T_{inner\ wall}$ has been obtained via Appendix 2 "Welding Temperature of Pipe Wall – Initial Pipe Wall temperature of 25°C". The value of $T_{inner\ wall}$ obtained is 275°C while depth of penetration is 1.625 mm for 12 inch API 5L Grade B Carbon Steel Pipe with thickness of 8.28mm. These values will be used to validate against the value of $T_{inner\ wall}$ obtained via ANSYS simulations. In order to find $T_{inner\ wall}$ by using ANSYS simulations, pipe geometries and process boundary conditions must be properly defined. Figure 5 shows the defined pipe wall geometries.

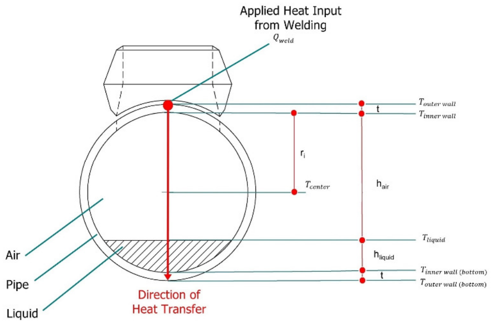

Fig. 5 Defining the geometries of the 2D heat transfer finite element model

3.4 ANSYS Simulation Boundary Conditions and Preprocessor Settings

After defining 2D finite element model geometries, ANSYS simulations are prepared and simulations boundary conditions are required to be defined. Setting Onshore Gas Terminal (OGT) under PETRONAS Carigali Sdn. Bhd. (PCSB) as a reference, natural gas condensate as process liquid will be used in ANSYS simulations. The physical properties of natural gas condensate processed in OGT are specified in the table below.

Table 1 Physical properties of natural gas condensate in OGT

Natural Gas Condensate Properties	
Specific Gravity, γ	0.7
Density, ρ	700 kg/m^3
Viscosity, μ	0.5 cP
Convection Coefficient, h	397.481 W/m^{2}K
Temperature	25°C

Throughout the simulations, the pipeline model is assumed to be filled with 50% liquid. The process fluid is static and heat transfer mechanism within it are assumed to be via free convection. The physical properties of API 5L Grade B Carbon Steel material are shown in Table 2.

Table 2 Physical properties of API 5L Grade B Carbon Steel

Physical Pipe Properties and Pipe Geometries	
Material	Carbon Steel (API 5L Grade-B: SCH 30)
Thermal Conductivity, k	54 W/m.°C
Specific Heat Capacity, c_p	502.4 J/kg. °C
Density, ρ	7850 kg/m^3
Nominal Diameter, D_n	300.0 mm or 12.00 in
Outer Diameter, D_o	323.8 mm or 12.72 in
Pipe Wall Thickness, t	8.38 mm or 0.330 in
Initial Pipe Wall Temperature, $T_{initial}$	25°C

This model also take the account of presence of ambient air surrounding the inner and outer side of the pipe wall. The half remaining portion of fluid that is not filled with process fluid inside the pipeline model are assumed to be filled with air. Air is static.

Table 3 Convection properties of air

Air Convection Heat Transfer Properties	
Initial Air Temperature	25°C
Convection Heat Transfer Coefficient, h	20W/m^2. °C
Heat Input	923.26 J/mm or 1539 J/s

After the boundary conditions are defined in preprocessor section of ANSYS Mechanical Workbench, the model is created, meshed and the directions of heat transfer (conduction and convection) are defined as indicated in the figure below. Here, the heat input from welding is set to be localized at one node as indicated. This model doesn't consider the process fluid introduction. Then, the temperature data from simulations will be tested and validated against the temperature data obtained from PTS chart to check for model validity.

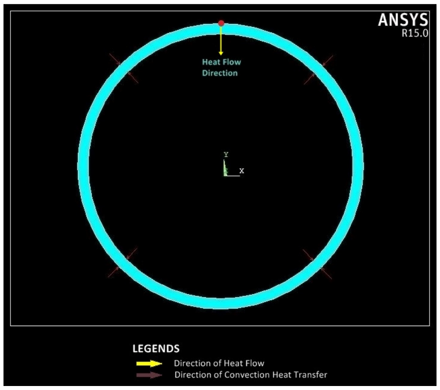

Fig. 6 Developed ANSYS 2D finite element model (without fluid introduction)

4. FINITE ELEMENT ANALYSIS

4.1 ANSYS Simulation Stage I: Prediction of Inner Pipe Wall Temperature

The result of the simulation are extracted and represented in the form of temperature profile.

Fig. 7 Temperature profile showing maximum pipe wall temperature and inner pipe wall temperature

Inner pipe wall temperature, $T_{\text{inner wall}}$ (located at node 7) obtained by using ANSYS simulation is 98.318°C while the value obtained by using PTS chart is approximately 275°C. There is a significant difference between those two numerical values. This suggest that the modelling of heat source for this simulation model is not accurate. The heat source visualization as localized and concentrated at one node is not practical and irrelevant to be used in the simulation. Therefore, the solution are improved by defining more accurate heat source distribution model. This finding related to the study that was conducted by Lima and Santos (2016) in which they concluded that modeling of heat source must be accurate to obtain valid results of computational models from experimental models.

Heat source modelling are corrected by defining heat source or heat flow along a path as close as to the real welding conditions to get better results. This is because, in the real conditions, the heat source will travel along certain path as the weld rod travels. The improved heat source model is visualized in Figure 8.

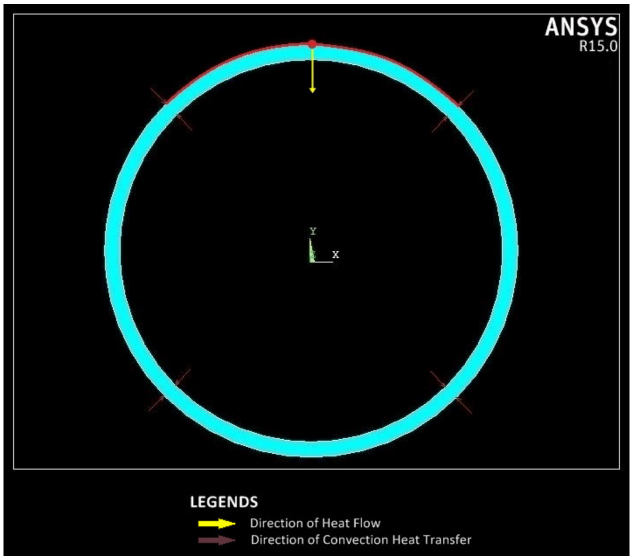

Fig. 8 Improved heat source modelling on ANSYS 2D model

With the new heat source modelling definition, a number of ANSYS simulations are conducted with variation of pipe size and pipe thickness by specifying constant welding heat input; 932.22 J/mm. The data obtained are compared with the temperature data obtained from PTS chart as in Table 4.

Table 4 Comparison of pipe wall temperature from PTS Chart with ANSYS simulation for heat input of 923.22 J/mm

Pipe Size (in)	Outer Diameter, D_o		Pipe Wall Thickness, t		Estimated Pipe Wall Temp. from PTS chart, (°C)	Estimated Pipe Wall Temp. from ANSYS, (°C)
	in	mm	in	mm		
8	8.625	219.075	0.277	7.036	400.00	394.24
10	10.750	273.050	0.307	7.798	325.00	325.16
12	12.750	323.850	0.330	8.382	275.00	275.04
14	14.000	355.600	0.375	9.525	180.00	180.18

The new model indicated that the results are improved since that there are less variation in terms of estimated pipe wall temperature value from ANSYS simulations with the data obtained from PTS chart. Thus, it can be concluded that the modelling of heat source distribution for the 2D finite element of model of pipeline by specifying heat flow along a weld path are valid and accurate. The simulations are conducted further with more variations of heat input. The results are shown in Table 5.

Table 5 Inner pipe wall temperatures from ANSYS simulations for different weld heat input values

Pipe Size (in)	Pipe Wall Thickness, t (mm)	$T_{\text{innerwall}}$ (°C)				
		HI: 800 J/mm	HI: 923.2 J/mm	HI: 1000 J/mm	HI: 1100 J/mm	HI: 1200 J/mm
8	7.03	344.6	394.2	445.1	480.4	523.6
10	7.79	280.7	325.1	360.3	399.0	445.8
12	8.38	230.9	275.0	310.0	345.5	390.1
14	9.52	136.7	180.1	207.2	242.5	277.8

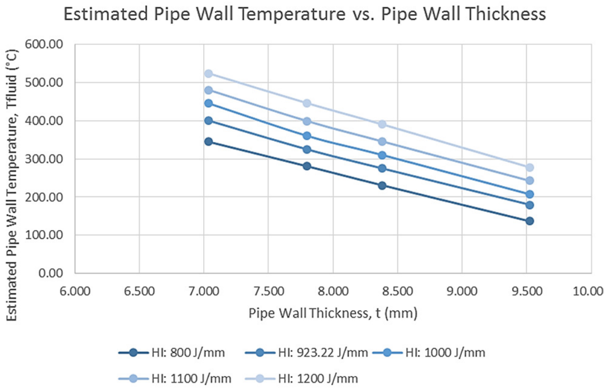

Fig. 9 Graph of estimated pipe wall temperature vs. pipe wall thickness

Figure 9 illustrates the results from performed simulations. With increased pipe wall thickness for large pipe sizes, estimated pipe wall temperature decreases with every weld heat input supplied to the system. The straight line curves with negative slopes indicates that estimated pipe wall temperature are proportional to the pipe wall thickness. The simulations are proceeded to the next stage, with introduction of process liquid.

4.2 ANSYS Simulation Stage II: Prediction of Process Fluid Temperature

The result of simulations with the introduction of process fluid are represented in the form of temperature profile below.

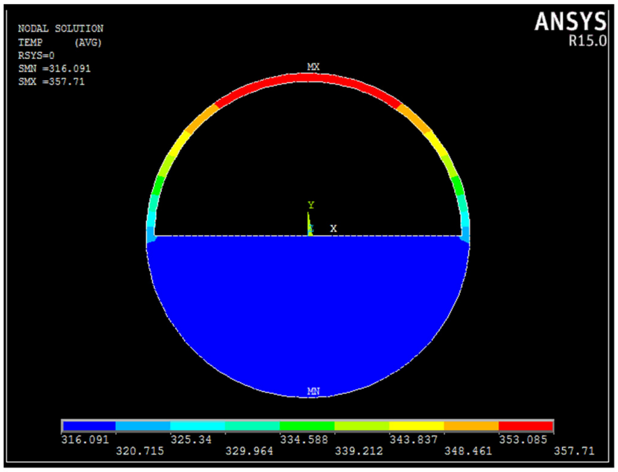

Fig. 10 Temperature profile with the introduction of process fluid inside the system

Figure 10 shows temperature distribution profile for the developed 2D finite element model with the introduction of process fluid filled at 50% level. The nominal pipe size is 14 in with 0.375 in pipe thickness and heat input supplied is 800 J/mm. It is observed that the maximum process fluid temperature (T_{maxoil} = 45°C) is located on the fluid surface that is in direct contact with pipe wall structure as shown in Figure 11.

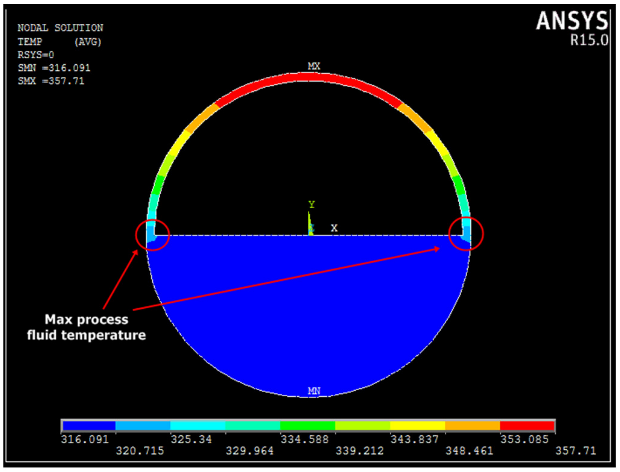

Fig. 11 Locations of maximum process fluid temperature

The results for this stage of ANSYS simulations are presented in the Table 6 with variation of tested heat input values for specified pipe sizes. All simulations are performed for 50% filled process liquid at static condition.

Table 6 Estimated process fluid temperatures from ANSYS simulations for different weld heat input

Pipe Size (in)	Pipe Wall Thickness, t (mm)	T_{maxoil} (°C)				
		HI: 800 J/mm	HI: 923.2 J/mm	HI: 1000 J/mm	HI: 1100 J/mm	HI: 1200 J/mm
8	7.036	92.41	103.87	109.37	117.84	126.32
10	7.798	67.62	74.22	78.36	83.71	89.08
12	8.382	53.23	59.90	64.05	69.46	74.87
14	9.525	45.00	50.23	53.48	57.70	61.93

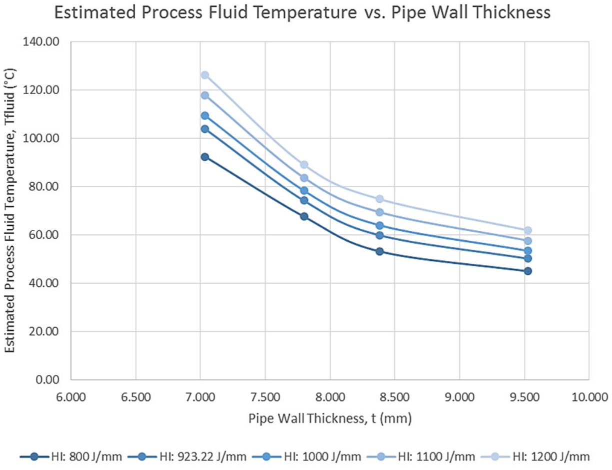

Fig. 12 Graph of estimated process fluid temperature vs. pipe wall thickness

From Figure 12, it is observed that process fluid temperature decreases with decrease in pipe wall thickness. At supplied heat input of 1200 J/mm for 8 mm pipe wall, the process fluid temperature can rise up to around 126 °C during welding in hot-tapping at defined conditions. In real hot tapping application, process fluid temperature inside pipeline at same conditions as specified in the simulation's boundary conditions can be estimated by referring to this graph. The input parameter that are required are the existing pipe wall thickness and supplied weld heat input to get the process fluid temperature. Then, the estimated process fluid temperature will be compared with the auto ignition temperature of the process fluid to determine the safe allowable working temperature during welding. For the process fluid used in this project simulations, the auto ignition temperature for natural gas condensate is approximately 232°C.

In practice, the estimated fluid temperature must be always kept lower below the auto ignition temperature of the process fluid to avoid risk of explosion since that temperature can cause the process fluid to spontaneously ignite, even without the presence of any external ignition sources such as flame or spark. In addition, oxygen concentration inside the pipeline also need to be checked and considered since auto ignition temperature of a flammable liquid will decreases as oxygen concentration increases.

5. CONCLUSION

The data from the simulations have provide basis on how to estimate the pipe wall temperature and process fluid temperature during welding in hot tapping process. The first part of ANSYS simulations conclude that the modelling and definition of heat source distribution for simulated finite element model must be accurate in order to validate the temperature results against temperature data obtained via PTS chart method. As for the second part of ANSYS simulations, the estimated process fluid temperature value need to be compared to the auto ignition temperature of tested process fluid to establish the knowledge on the safe working temperature during welding. At any conditions, the process fluid temperature should not exceed this value to avoid spontaneous combustion of flammable process liquid inside pipeline.

However, the results and data obtained in this project are still needed to be validated with reliable methodologies instead of comparing with existing PTS chart as proposed in this project, such as experimental laboratory data from acknowledged researches. The reason is that there might be difference in test conditions and boundary conditions between the project simulations with the methods developed by PTS team in developing the referred temperature prediction chart. Eventhough the obtained project data might be close to the compared data from PTS chart, there is no guarantee that the developed model is valid since the difference in terms of specified boundary conditions for both methodologies might results in model inaccuracy or even produce completely false simulation data.

In future works, the obtained project results can be improvised by providing more wide range of data such as providing the information for extended range of supplied heat input and temperature changes for specific process liquid level inside pipeline. In addition to temperature information, the depth of weld penetration information can be added to the final chart to enable the weld operator to investigate the burn-through risk during welding in real engineering application.

The accuracy of process fluid temperature estimation can be improved by considering the remaining unfilled region (empty region) inside a pipeline section to be filled with process liquid vapor with specific saturation level as a part of analysis in the finite element model simulation. The presence of process liquid vapor that possesses specific value of heat transfer coefficient will affect the heat transfer rate, thus affecting the final estimated fluid temperature as the heat travels from outer pipe wall surface to the process fluid. This will give a close approximation to the real situation of welding in hot tapping process.

REFERENCES

[1] Lima, O. L., Santos, A. B., "Mathematical Approaching and Experimental Assembly to Evaluate the Risks of In-Service Welding in Hot Tapping," J. of Pressure Vessel Tech., 138 (16). pp. 1-11.

[2] Sabapathy, P. N., Wahab, M. A., and Painter, M. J., 2005, "The Onset of Pipewall Failure During 'In-Service' Welding of Gas Pipelines," J. Mater. Process. Technol., 168(3), pp. 414-422.

[3] Tahami, V. F., and Asl, M. H., 2009, "A Two-Dimensional Thermochemical Analysis of Burn-Through and In-Service Welding of Pressurized Canals," J. Appl. Sci., 9(4), pp. 615-626.

[4] Eager, T. W., and Tsai, N. S., 1983, "Temperature Fields Produced by Traveling Distributed Heat Source,'' Weld. J., 62 (12), pp. 346-355.

[5] Talbot, E., Hubert, P., Dube, M., and Yosefpour, A., 2011, "Optimization of Thermoplastic Composites Resistance Welding Parameters Based on Transient Heat Transfer Finite Element Modeling," J. Thermoplastic Composite Materials.

[6] Atwa, H. A. E. A., 1993, "Study of the Effects of Defects on the Fatigue Behavior and the Fracture Toughness for Low Carbon Steel (API 5L Grade B) Gas Transmission Pipelines", Engineering Fracture Mechanics, 44(6), pp. 921-935.

[7] Gomez, R., Munoz, D., Vera, R., and Escobar, J. A., 2004, "Structural Model for Stress Evaluation of Pre-stressed Concrete Pipes of the Cutzamala System," Pipeline Divisions Specialty Congress.

[8] Morales, F. R., Scott, A.D., Perez, M. R., Diaz, E. M. C., and Morejon J. A. P., 2010, "Calorific Energy Required During In-service Welding of Pipelines for Petroleum Transport," Welding International, 24(9), pp. 655-664.

[9] Krutz, G. W., and Sergerlind, L. J., 1978, "Finite Element Analysis of Welded Structures," Weld. J. Res. Suppl., 57, pp. 211-216.

[10] Bang, I. W., Son, Y. P., Oh, K. H., Kim, Y. P., and Kim, W. S., 2002, "Numerical Simulation of Sleeve Repair Welding of In-Service Gas Pipelines," Weld. J., 81(12), pp. 273-282.

[11] Law, M., Kirstein, O., Luzin, V., 2012, "Effect of Residual Stress on the Integrity of a Branch Connection," International Journal of Pressure Vessels and Piping, 96-97 (12), pp. 24-29.