FINITE ELEMENT MODELING OF THICK WALLED VESSELS UNDERGOING LOCALIZED POST WELD HEAT TREATMENT

ZUBAIRI FARIHAN BIN ROFFIE

MECHANICAL ENGINEERING
UNIVERSITI TEKNOLOGI PETRONAS
SEPTEMBER 2012

Finite Element Modeling of Thick Walled Vessels Undergoing Localized Post Weld Heat Treatment

by

ZUBAIRI FARIHAN BIN ROFFIE

Dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

SEPTEMBER 2012

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan.

CERTIFICATION OF APPROVAL

Of Research Project

Finite Element Modeling of Thick Walled Vessels Undergoing Localized Post Weld Heat Treatment

By

Zubairi Farihan bin Roffie

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

____
(DR. MOKHTAR BIN AWANG)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
SEPTEMBER 2012

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

ZUBAIRI FARIHAN BIN ROFFIE

Mechanical Engineering Department,
Universiti Teknologi PETRONAS.

ABSTRACT

Usage of thick walled vessels is important for equipments operating under high temperature and pressure. Post weld heat treatment (PWHT) has been used for stress relief purpose and to improve mechanical properties at welded joints of these equipments during fabrication or repair. Cracks and fatigue failures are the main problems when localized PWHT were performed at weld joints near thick structures because sufficient since ideal heat transfer and proper temperature gradients cannot be achieved. This project studied the effect of these geometrical constraints on thermal distribution and stress level when localized PWHT is applied. Finite element analysis (FEM) using ANSYS™ were performed on 2D models of pressure vessel having different adjacent structures and under several PWHT conditions. Simulation results show significant increase in radial thermal gradients up to 60°C with the presence of thick adjacent structures near the weldment. Thermal stress also increases with the addition of those structures, mainly on the tubesheet side. Increase in heating band width gives better thermal distribution for local PWHT compare to decreasing the rate of heating and cooling.

ACKNOWLEDGEMENT

The author would like to express his gratitude and appreciation to Universiti Teknologi Petronas management for providing necessary software to complete the project. Besides, appreciation goes to supervisor Dr. Mokhtar Awang for his guidance, attention and helping throughout the duration of the project and his continuous support since beginning until the end of the project. Also not to forget to Petronas Chemical Fertiliser (Kedah) for providing necessary documents and information regarding the project. Lastly, the author wishes to thank all friends and families for their endless support and motivation to make this project a success.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF EQUATIONS

CHAPTER 1

INTRODUCTION

1.1 Project Background

In manufacturing world, there are many key fabrication techniques used for metal and steel structures such as machining, cutting, bending, forging and welding. In recent times, welding has become an essential metal joining technique for industrial applications. In petrochemical, oil refineries and nuclear fields, usage of thick-walled vessels for reactors, heat exchangers and column sections are important for service in elevated temperature and pressure. Manufacturing process for thick-walled vessels followed special procedures especially in welding and heat treatment process to ensure a good resistance and properties of the joints [1]. There are major setbacks of welding two thick steel structures, which is the existence of distortions and high amount of residual stress that can significantly impair the performance and reliability of the welded structures [2].

If not treated properly, this matter can lead to premature failures for the equipments during service under high temperature and pressure in term of service-induced crack initiation and stress induced corrosion [3]. Failures for the critical equipments are a major concern and such events can lead to catastrophic consequences and the occurrence cannot be tolerated. During fabrication or repair of pipes or safety-critical equipments, Post-weld Heat Treatment (PWHT) has become a staple choice to reduce the impact of residual stress and also to improve mechanical properties of weldment sections [4-5].

Residual stress is defined as any stress that is present within a body when all external loads were removed [6]. In steel manufacturing, residual stress exists after welding process as the results of non-uniform temperature distribution and plastic deformations [6]. PWHT process can be performed to reduce this residual stress by heating the structure in a furnace or locally at the specific weld location using heating blankets [6].

In recent years, it has been reported several cases of cracking failures for pressure vessels at petrochemical plants around the world [7][8]. The cracks locations were found at closing welds of thick-walled vessels. From investigation, one major factor contributing to the cracks is the use of localized PWHT to the welded joint that was not sufficient to reduce the level of residual stress [8]. It has been claimed that the use of localized PWHT at thick-walled welded joint with geometrical constraint (ie. next to large structure such as nozzles) introduce high thermal stresses at the area thermal gradient reaching unacceptable level within the wall's thickness [8].

This research describes the effect of performing localized PWHT at structures with geometrical and shape variations in term of residual stress distribution and heat transfer behavior. Several models of pressure vessels will be designed with different adjacent structures and the environment and condition during the localized PWHT will be simulated according to the standards using Finite Element Modeling to see the effects on thermal-mechanical behavior of the structures.

1.2 Problem Statement

Post-weld Heat Treatment has been used extensively to reduce residual stress and improve properties of critical welded joint to avoid failures during operation. There were cases of cracking failure of vessels which pointed out that localized PWHT for thick-walled vessels were not sufficient to reduce residual stress because of the existence of thermal gradients [8] and geometrical constraints which hinder ideal heat transfer during PWHT.

1.3 Objectives

The main purpose of this research is using finite element modeling (FEM) to simulate post-weld heat treatment PWHT process on various pressure vessel designs via ANSYS software. From the simulation, the following objective will be studied:

1. The effects of localized PWHT on temperature distribution and residual stress values of thick-walled structures with geometrical constraints under different conditions.

1.4 Scope of Study

This project will involve the usage of ANSYS v.14 software that is renowned for performing finite element modeling (FEM) to simulate the heat treatment process. For that purpose, transient thermal analysis for 2D models of the vessels' cross sectional area will be used.

In this project, the types of materials studied will be low alloy steel (2.25Cr 1Mo). For the cracking failure that occurred, the joint type is V-shape closure butt weld, with localized PWHT performed after the welding. Studied properties from the simulation will be conduction heat transfer and thermal stresses.

CHAPTER 2

LITERATURE REVIEW

2.1 Literature Review

Since long, industries such as oil & gas, petrochemical, energy and nuclear has incorporated the usage of safety-critical equipments and parts like pipes, vessels, heaters and other equipments which operated under high thermal and mechanical loads for extended time periods [4]. When consequences involving catastrophic effect from failures of the said equipment cannot be tolerated, special measures has to be taken in term of regular inspection during service and from advanced and regulated fabrication techniques set by various standards.

Welding has become one of the most important processes in constructions of various steel structures in many engineering fields. One common problem with welding process is the stability and the dimensional tolerance of the finished products [9]. The instability of welded joints comes in mean of residual stresses which became a concern because it affected the fracture toughness and fatigue behavior [9]. Residual stress and distortion in welded structures are related to the solidification shrinkage of the weld metal, non-uniform thermal expansion and contraction of the base metal, and internal/external constraints of the welded structures [9].

Since 1930, researchers has attempted to understand the mechanism of residual stress and the way to measure it using various techniques using predictive methodology, parametric experiments, and empirical formulations [9]. Because of welding residual stress has negative effect in term of fatigue life and strength of the joints, several mitigation methods were developed. Some of the methods are weld sequencing, weld parameter definition, precambering, prebending and post weld heat treatment to name a few [6]. In recent years there has been many researches made in order to find the relationship between heat treatment and its effectiveness in reducing weld distortion and residual stress. In predicting and measuring residual stress of welded joints, two most used approaches from the scientists are by experimental method and by modeling technique. The experimental method of evaluating residual stress is from destructive or non-destructive approach Ohms et al. (2006) have used the neutron diffraction method to assess a bead weld of a thin plate specimen. In the experiment, they also compare residual stresses value from two types of heat treatment, PWHT and APWT (Advanced Post Weld Treatment) which is a localized thermal shock based approach [4]. In their experiment, the neutron diffraction testing has been performed using a large Combined Powder and Stress Diffractometer.

Besides the usage of experimental method, there were also researchers who incorporated the usage of both simulation and experimental method in measuring residual stress of welded joints. Cho et al. (2004) has investigated residual stress for post weld heat treatment of multi-pass weld using finite element analysis and deep hole drilling method at the test specimen [10]. For the analysis, a simplified 2D model of V-type and K-type butt weld was designed using ANSYS to simulate the welding process and the resulting residual stress. After the model has been simulated with PWHT, results shows significant reduction in residual stress and the values are consistent with experimental data after the actual specimen being tested via hole drilling method.

Hao Lu et al. (1999) has studied different types of multiple welding passes at their effects on residual stress after localized post-weld heat treatment were applied [11]. From their research, it showed that residual stress at outer surface decreases with increment of heating band width. On the other hand, inner surface residual stress can be effectively removed with narrow band width. However, their study did dot test welded structures with different geometrical constraints for final residual stress values.

Besides PWHT, APWT technique has also been studied in its ability to reduce residual stress in repair weld of a steam header by Mirzaee-Sisan et al. (2005). In their project, residual stress value after APWT was predicted using Finite Element method and the results were compared with Deep Hole Drilling (DHD) method [12]. Results from both methods showed significant reduction in the level of residual stress. From the studied literatures, it is clear that many modeling projects focus more on welding simulation, where a very accurate finite element model of welding process has been designed in order to correctly measure the resultant residual stress from the weld. To successfully model a welding process, it involves many complex mathematical models and parameters such as welding heat transfer, elastic and thermal strains model, temperature, material shrinkage and many more [13].

There were also papers focusing on failures and repair works from cracking problem in thick-walled pressure vessels. Research by Firth et al. investigated cracking failure in a synthesis gas heat exchanger. It revealed that the cracks originated from high amount of residual stress combined with high thermal and mechanical loads during operations [13]. From past data, the vessel was welded and PWHT was performed according to the codes per stated in ASME VIII div 1. However, since the localized PWHT for the weld was performed at thick wall near large structures (tubesheet & nozzle), it was suspected that the process failed to achieve stress relieving purpose by the existence of thermal gradient from high temperature variation between inner and outer wall of the vessel. This can trigger creep reaction and introduce high residual stress on the inside. The paper concluded that localized PWHT should not be applied next to large structures that can act as heat sink that can hinder ideal heat penetration through the wall’s thickness [13].

Since there were not much researches being made to correlate geometrical constraints and effectiveness of PWHT process, this project is ideal to be done in order to verify the claim. This project is also an effort to study more on safety and reliability of critical equipments which are the main concern for many industries today. By using Finite Element Modeling method, parameters involved during localized PWHT will be used to simulate the ideal conditions. Since modeling of both welding and the heat treatment requires very complex calculations and time-consuming, this project will focus on simulating the localized PWHT to examine the thermal distribution and level of thermal stress developed with those different geometries of the vessels.

2.2 Theory

2.2.1 Post Weld Heat Treatment (PWHT)

PWHT is a heat treatment that is carried out after welding process to improve properties of the material and the weldment joint. PWHT is required for certain situation for the equipment, according to the materials, operating conditions, and properties. The two most common procedures in PWHT are post heating and stress relieving [5]. The need to perform PWHT is determined by international codes and standards and the application. The goal of having PWHT is to increase the material resistance to brittle fracture and relieving residual stress [5]. Other objective of performing PWHT is hardness reduction and increasing strength.

ASME Sec. VIII Div I have stated the requirement of performing PWHT when fabricated carbon or low alloy steel vessels contain lethal substance either liquid or gases [14]. Post heating is used to minimize the potential of hydrogen induced cracking or high temperature hydrogen attack at the equipment [5]. Post heating process is not necessary for most applications, but it needed to be done when the equipment operates at high hydrogen content like synthesis gas heat exchanger.

Stress relieving is used to reduce residual stress that present in a structure as a result of its manufacturing process. Residual stress is the remaining stress inside the structure of a material after the original stress (thermal, external force) has been removed [5]. In standard PWHT process, there are divided into three parts, uniformly heating the material to the desired holding temperature, heating the part for a holding period, and uniform cooling. There are several guidelines in doing PWHT for pressure vessels stated by ASME Sec. VIII, such as:

1. Heating the whole vessel in an enclosed furnace.
2. Heating the circumferential joints using a soak band that extends around the entire circumference. The portion outside the soak band shall be protected so that the temperature gradient is not harmful.

Stress relieving and properties changes of the steel happen during PWHT due to phase transformations and structural changes. The thermal energy received by the metal allows for grain boundaries sliding and removal of metallurgical defects like dislocations, vacancies and slip planes. To achieve sufficient stress reduction, reaching the re-crystallization temperature after the heating period has far greater impact than the holding time itself [20]. So it is important to carefully control the heating and cooling rates while performing PWHT.

PWHT procedures and requirements are controlled by many standards such as ASME Section VIII, API 582 and API 934A. Basically, the PWHT procedures and factors, such as heating rate, holding time, temperature and cooling rates are determined from types of materials, design of the equipment, and types of PWHT used. One basic rule for PWHT for steel structures is to heat the component at temperatures between 600 – 700 ^{o}C below the lower transformation temperature for 1 hour/inch for the weld’s thickness [5-6]. Figure 1 shows a typical heating and cooling profile for PWHT.

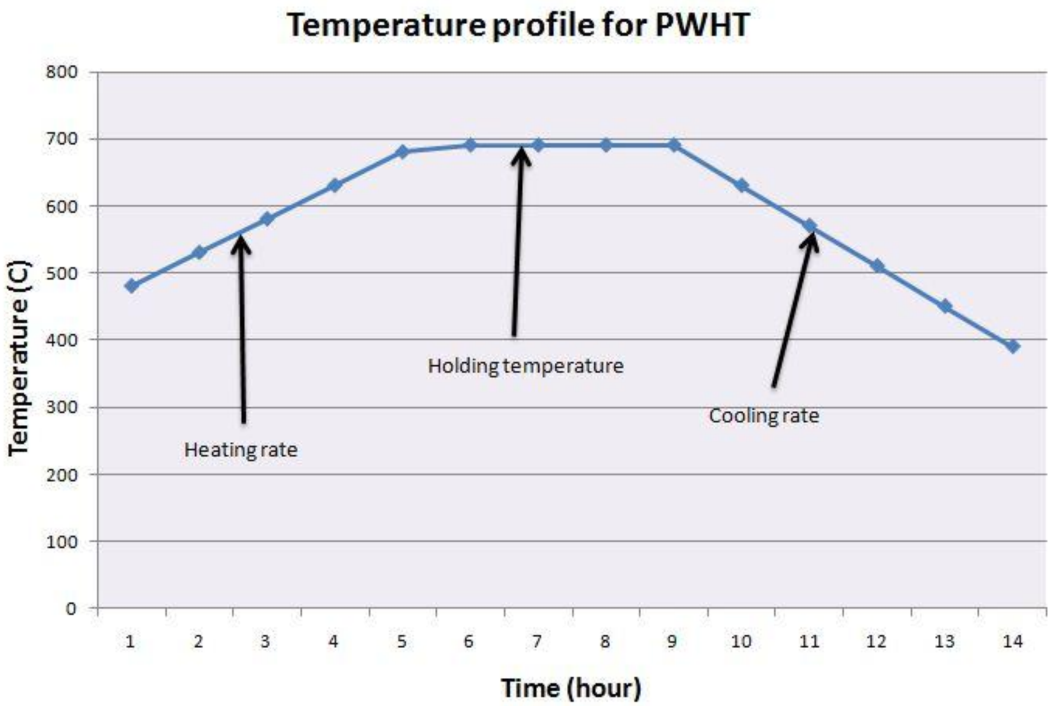


Figure 1: A typical heating and cooling profile of PWHT process

2.2.2 Governing Equations for Thermal and Structural FEM

The governing differential equation for transient heat conduction in solid is given by:

$$-\left(\frac{\partial q_x}{\partial x}+\frac{\partial q_y}{\partial y}+\frac{\partial q_z}{\partial z}\right)+Q=\rho c\frac{\partial T}{\partial t} \qquad (1)$$

Where

q_x, q_y, q_z: heat flow rates in x, y, and z-directions

ρ: density of the material,

c: specific heat capacity,

T: unknown temperature,

Q: energy generated in the material per unit volume and time.

According to Fourier’s Law, heat transfer rates depend on thermal conductivity coefficients and temperature gradients:

$$\begin{Bmatrix} q_x \\ q_y \\ q_z \end{Bmatrix} = -\begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{21} & k_{22} & k_{23} \\ k_{31} & k_{32} & k_{33} \end{bmatrix} \begin{Bmatrix} \frac{\partial T}{\partial x} \\ \frac{\partial T}{\partial y} \\ \frac{\partial T}{\partial z} \end{Bmatrix} \tag{2}$$

Where [k] is the thermal conductivity matrix.

Heat loss due to convection and radiation is given by:

$$q_n = h(T_a - T_s) + \sigma_s \varepsilon \left(T_a{}^4 - T_s{}^4\right) \tag{3}$$

Where

h: convection surface heat transfer coefficient

T_s: surface temperature

T_a: ambient temperature

σ_s: Stefan–Boltzmann constant

ε: emissivity of the surface

Strain-Stress Relationships (Hooke’s law for an isotropic medium) [19]:

$$\varepsilon_r = \frac{1}{E}[\sigma_r - v(\sigma_\theta + \sigma_z)] \tag{4}$$

$$\varepsilon_\theta = \frac{1}{E}[\sigma_\theta - v(\sigma_r + \sigma_z)] \tag{5}$$

$$\varepsilon_Z = \frac{1}{E}[\sigma_Z - v(\sigma_r + \sigma_\theta)] \tag{6}$$

$$\tau_{r\theta} = G\gamma_{r\theta}, \tau_{rZ} = G\gamma_{rZ}, \tau_{\theta Z} = G\gamma_{\theta Z} \tag{7}$$

Stress-strain Relationships [19]:

$$\sigma_r = \frac{E}{(v+1)(2v-1)}[(v-1)\varepsilon_r - v(\varepsilon_\theta + \varepsilon_z)] \tag{8}$$

$$\sigma_\theta = \frac{E}{(v+1)(2v-1)}[(v-1)\varepsilon_\theta - v(\varepsilon_r + \varepsilon_z)] \tag{9}$$

$$\sigma_Z = \frac{E}{(v+1)(2v-1)}[(v-1)\varepsilon_Z - v(\varepsilon_r + \varepsilon_\theta)] \tag{10}$$

Difficulty in controlling temperature gradients often being an issue when localized PWHT is performed which can results in high thermal gradients between the internal and external walls of the vessel, resulting in high stress from the material expansion as shown in Figure 2 below.

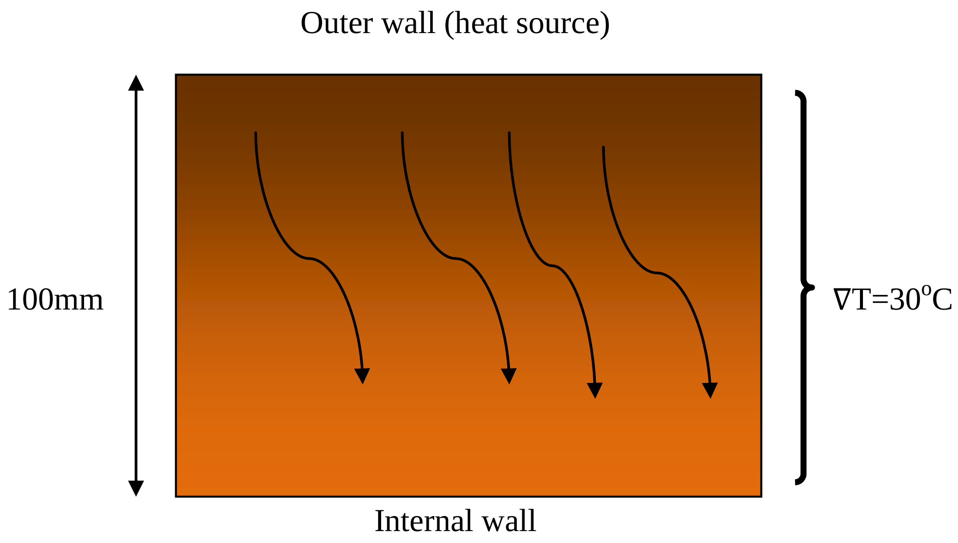


Figure 2: Thermal gradients during heat treatment for thick wall structure

From the above diagram, introduction of high temperature difference between inner and outer wall can trigger creep reaction. For example, 100mm thick 2.25Cr 1Mo steel will start to creep at 700^{o}C with internal stress at 100Mpa when there is 30^{o}C temperature difference across the wall [8]. It is given by the thermal stress equation of:

$$\sigma = E\alpha\Delta T \tag{11}$$

Where

E : Elastic Modulus

α: Coefficient of thermal expansion

ΔT: Temperature difference

From the above situation, any errors during heating or cooling phase of localized PWHT can lead to many negative effects on the toughness of the structure and its reliability during future operation.

CHAPTER 3

METHODOLOGY

3.1 Overview

This project will only focus on modeling and simulation and not comparing with experimental values. So it is important to gather as many data as possible to be used as input in the simulation process later. For this purpose, the data from actual equipment that has failed from the PWHT errors has been chosen, courtesy of Petronas Fertilizer Kedah (PFK) Sdn. Bhd.. Modeling and simulation will be carried out using ANSYS v14. The flowchart of the project works is shown in Figure 3 below.

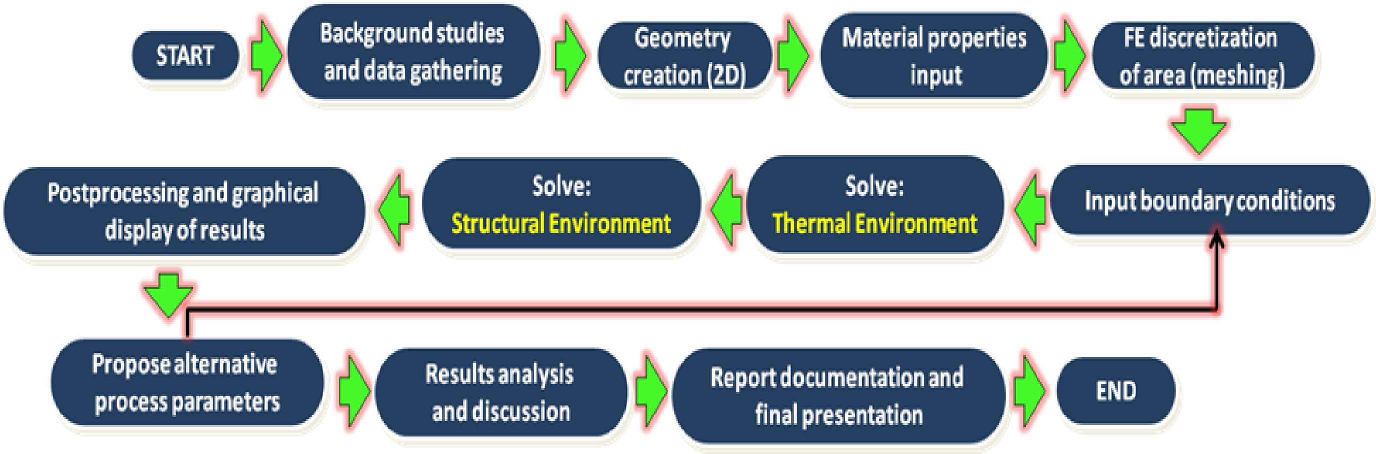


Figure 3: Flowchart of the whole project work

3.2 Simulation Tool

In the area of computer simulation and modeling there are many softwares capable of perfoming and solving variables using FEM. For this project, ANSYS v14 Mechanical APDL is chosen for its capability in modeling and solving various engineering problems including structural analysis and heat transfer. This project will incorporate coupled transient thermal and structural analysis based on the given condition to predict the amount of residual stress after the heating and cooling process of the heat treatment.

3.3 Simulating Localized PWHT using ANSYS

Modeling and solving the heat treatment condition using ANSYS 14 requires 3 of the following steps [17]:

1) Preprocessing

   Preprocessing is the beginning step towards solving any FEM problems. At this stage, it is required to define keypoints, constructing lines and defining areas or volume to create the desired. Meshing the model into smaller elements will create nodes that will be solved individually later on. In this stage also suitable element type need to be chosen and material properties will be keyed in.

2) Solution

   This step is where we specify the type of analysis to be carried out whether it is in steady state, transient or other. It is also important to put all the necessary boundary conditions, constraints, and loads such as the displacement, temperature, pressure or force on the preferred locations on the model.

3) Postprocessing

   This is the final stage of the analysis after the solution for the problem has been obtained. In postprocessing, graphical display and result list of the variables such as nodal displacement, temperature plot, and stress contour can be seen.

3.3.1 Design and Operation Principle of the Vessel

The type of pressure vessel to be designed for the analysis is given in Figure 4 below. The vessel is a shell-and-tube heat exchanger. The equipment is used to transfer heat between two different fluids at different temperature within two separate parts of the vessel, the channel side and the shell side. Gas from the channel side enters the shell side in a bundle of tubes which is separated by a tubesheet.

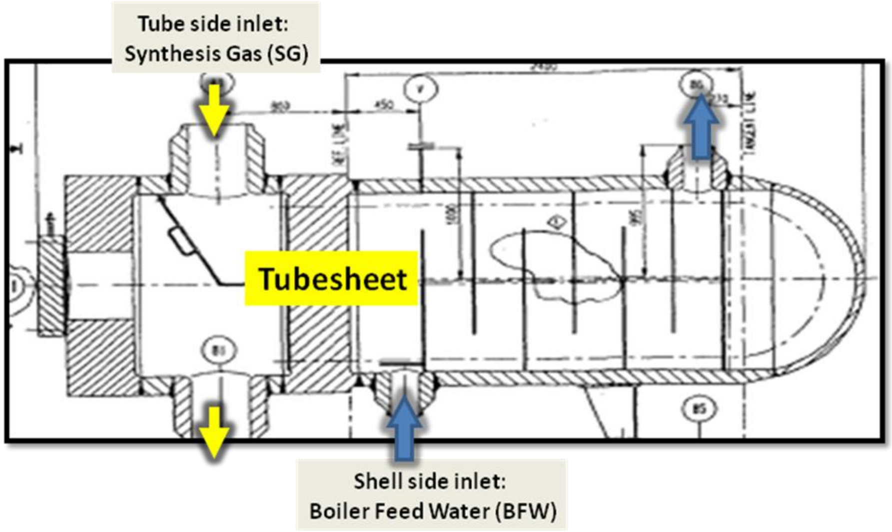


Figure 4: Operating principle of shell-and-tube heat exchanger

3.3.2 Geometry and Model Creation

Because of complexity of producing mesh and long computational time for 3D models, along with its relatively large vessel size, only 2D models will be considered for the heat treatment simulation process. The dimension of the original vessel model is shown below in Figure 5.

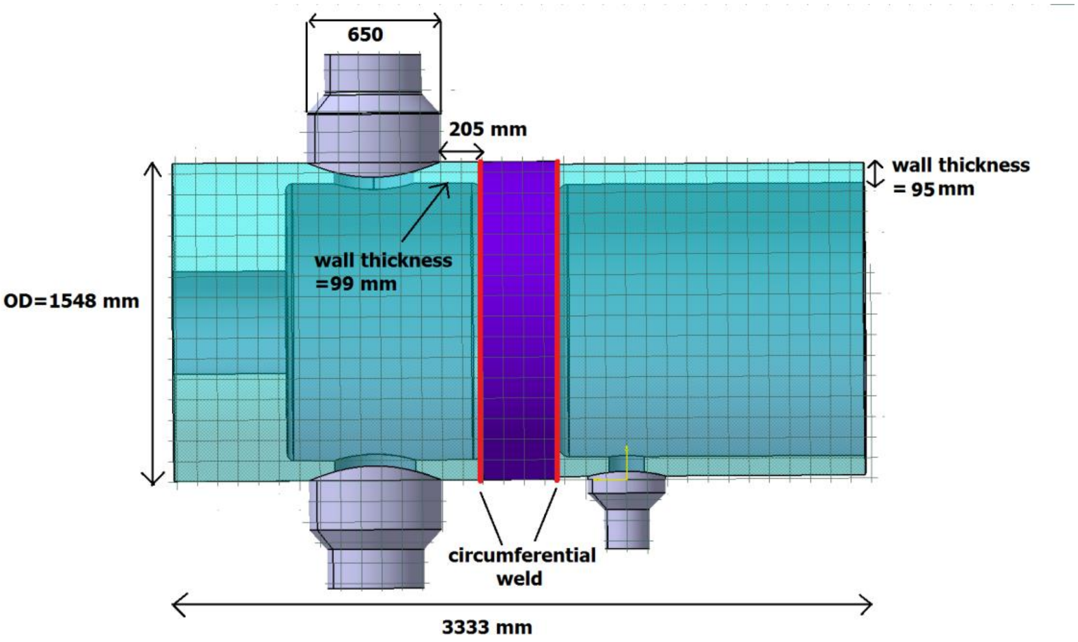
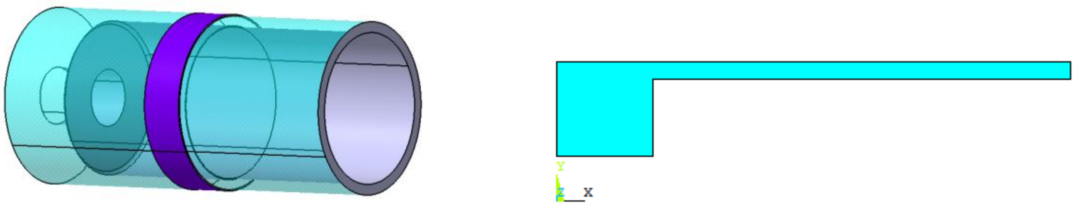


Figure 5: Side view of the vessel with important dimensions and the location of circumferential weld.

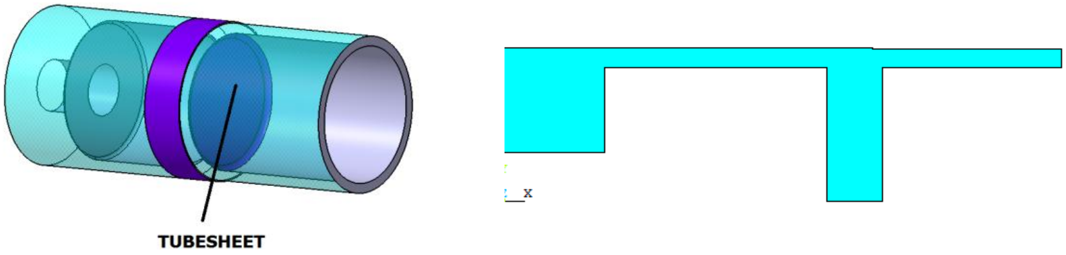
Because of the axy-symmetric nature of the vessel's shape (apart from the shell side inlet and outlet nozzles), the 2D models will only consist of top portion of the vessels' cross sectional area. To prove the effect of adjacent thick structures to influence the sufficiency of localized PWHT at the weld between channel and the tubesheet, 3 model variations are designed as followed:

1) Cylindrical structure without any adjacent component.
2) Cylindrical vessel with internal tube sheet on the side of the weld joint.
3) Cylindrical vessel with tube sheet and nozzles



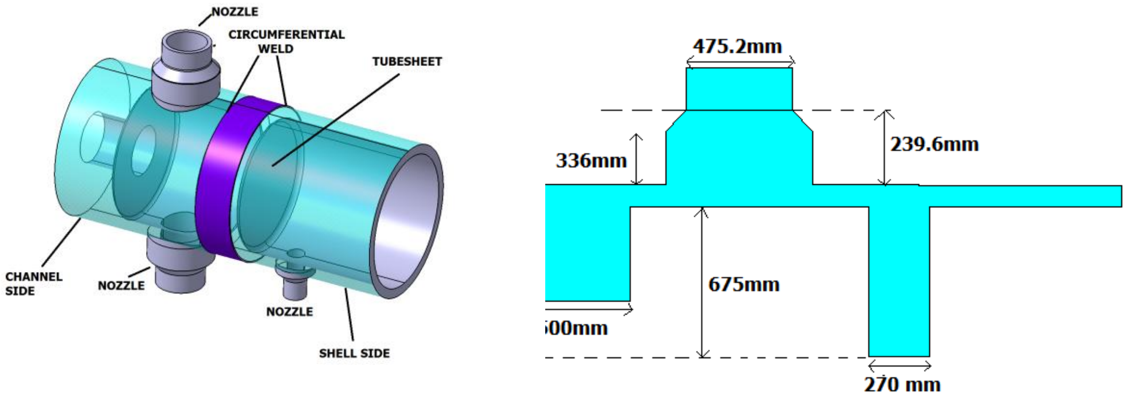
(a) *(b)*

Figure 6 (a) and (b): Model (1) with no additional structures near the weld area (purple area) and its 2D representation



(a)

Figure 7 (a) and (b): Model (2) with internal tubesheet



(a) *(b)*

Figure 8 (a) and (b): Model (3) with additional nozzles, two on channel side and one the shell side

3.3.3 Element Type and Material Properties

Element Type

There are 2 element types used for this analysis. Because the ANSYS solver cannot solve thermal and structural degree of freedom (DOF) and its variables simultaneously, the models need to be solved separately with two element types given in ANSYS Tutorial Guide below [15]:

1) PLANE 55

Plane55 is used for 2D thermal solid analysis. The element has four nodes with a single degree of freedom, temperature; at each node .The element is applicable to a 2-D, steady-state or transient thermal analysis. The element can also compensate for mass transport heat flow from a constant velocity field. The load for this element can be temperature, heat generated and heat flux. The main output data from this element will be nodal temperature and thermal gradients. The element geometry for plane55 is shown in Figure 9 below.

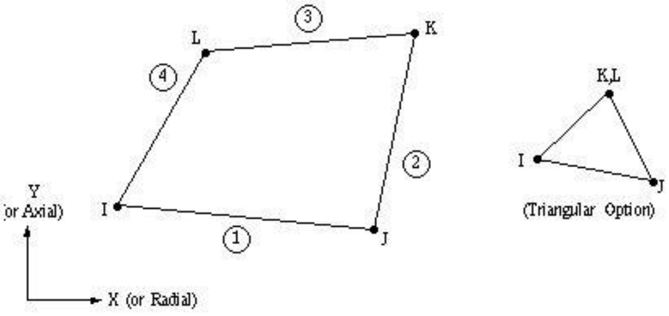


Figure 9: Plane 55 geometry

2) PLANE 183

Plane183 is a higher order 2-D, 8-node or 6-node element. PLANE183 has quadratic displacement behavior and is well suited to modeling irregular meshes. This element type is used for 2D structural analysis to obtain solutions for plane deformation and thermal stresses from the thermal result of the previous analysis. It has 8 nodes or 6 nodes having two DOF at each node: translations in the nodal x and y directions. The element may be used as a plane element (plane stress, plane strain and generalized plane strain) or as an axisymmetric element Figure 10 shows Plane 183 geometry.

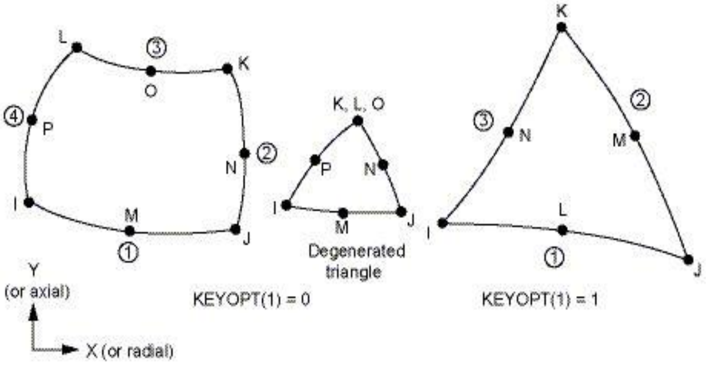


Figure 10: Plane 183 geometry

Material Properties

Material used for the vessel construction is SA336Gr.F22 CL.3 (2.25Cr 1Mo low alloy steel). It is used for all components that have been designed for the analysis. The weldment area that will be applied with PWHT is the channel to tubesheet. Since the weldment section is not simulated earlier, it is assumed to be as one element with the other part of the model made up of the said alloy steel. Also, the weld does not give difference in material behavior at its location and its subsequence effect after the analysis is ran. All the required properties are assumed to be linear throughout the analysis except for elastic modulus which is non linear as shown in Table 1 and 2.

Table 1: Material properties of SA336 low alloy steel

2.25Cr 1Mo Low Alloy Steel Properties				
Density (kg/m^3)	Specific Heat (J/kg^{o}K)	Thermal Conductivity (W/m^o K)	Coeff. of Thermal Expansion (10^{-6} $^oC^{-1}$)	Poisson’s ratio
7850	420	35	15	0.3

Table 2: Nonlinear properties of Elastic Modulus at elevated temperature [18]

Elastic Modulus (Gpa) at Elevated Temperature			
200 ^{o}C	400 ^{o}C	500 ^{o}C	600 ^{o}C
190	185	175	160

3.3.4 Meshing

Mesh generation is one of the most critical aspects of engineering simulation. In simulating this heat treatment process, the surface or volume needs to be divided into smaller subdomain or cells in order for their partial differential equations to be solved individually to get the solution. This process is called mesh generation, and the sizing and orientation of mesh must be considered to optimize simulation time and at the same time get the most accurate results. Sample of meshed areas on the model are shown in Figure 11.

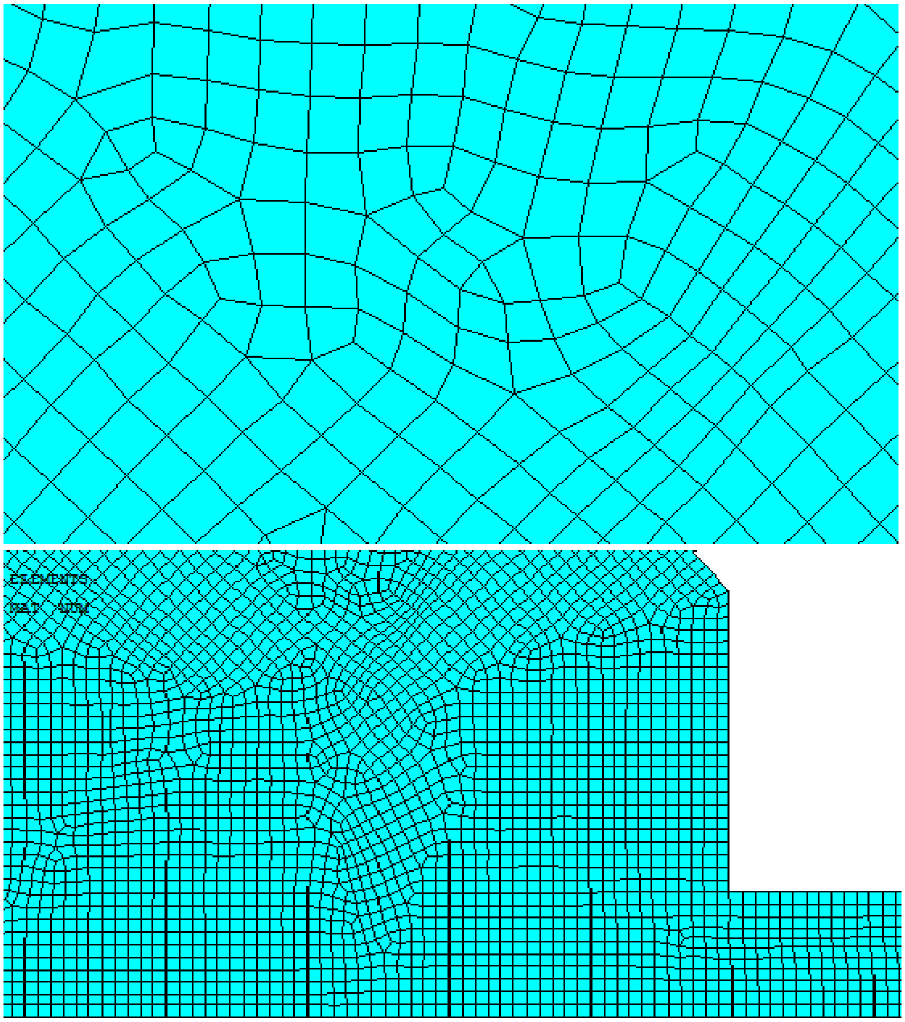


Figure 11: Area of the model after meshing.

3.3.5 Important Assumptions and Boundary Conditions

Under the solution options, all the necessary boundary conditions and loads will be applied to the models to replicate the localized PWHT process to get the most accurate results from the simulation. Several important assumptions in thermal and structural made before applying the loads include:

- Localized PWHT is for channel to tubesheet weld.
- Heat application is in form of temperature rise from outside to inside to imitate wrapped heating band around the vessel.

- No heat lost through convection and radiation during the heating and cooling sequence.
- Zero initial stress before the simulation.
- No translational displacement of the model during analysis.
- All boundaries are constrained to zero displacement to prevent severe expansion and thermal deformation.

Zero stress is assumed for the model because of the difficulty in predicting the values and distribution of weld residual stress from the welding process prior to PWHT and can only be determined from weld simulation itself. Because of the scope of this project is to study the thermal behavior and stress amount from localized thermal input itself, the stress relieving effect from microstructural phase transformation by the heating will not be taken into account.

Thermal Boundary Conditions

There 2 two of heating and cooling profiles tested for the models, with one condition was the actual profile used as stated in the manufacturer’s record as shown in Figure 12 below:

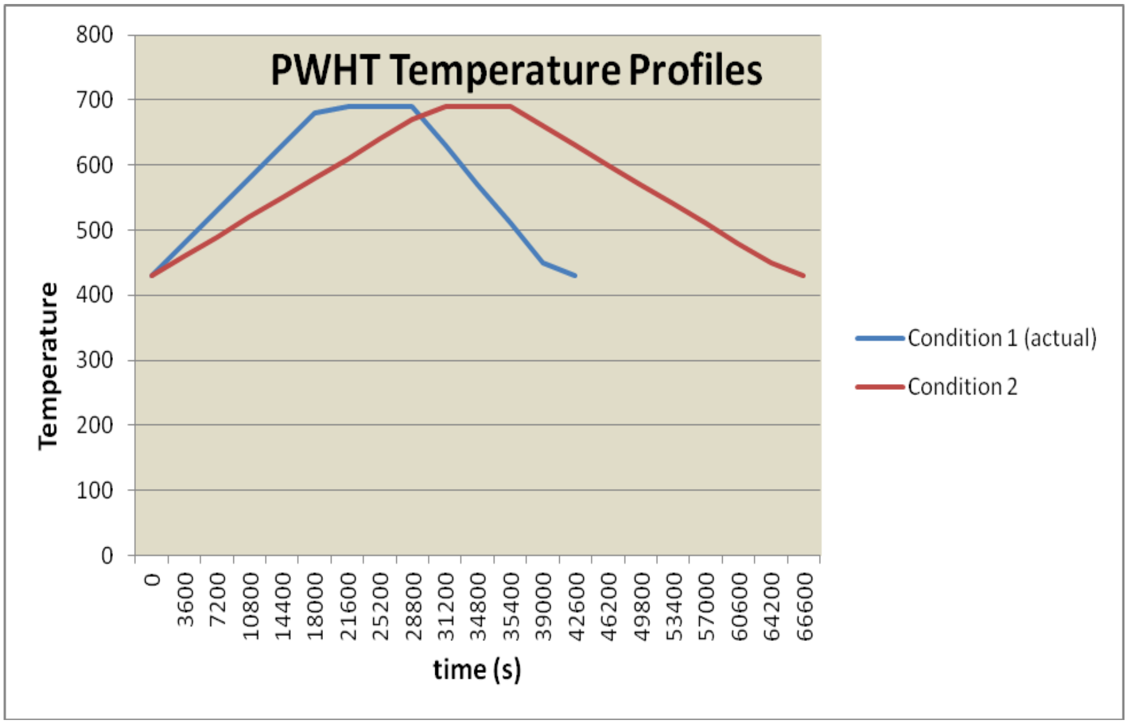


Figure 12: Temperature profiles for PWHT at 2 different rates

Beside the alteration with the heating and cooling rate from the actual condition, another parameter that will be used is the width of the heating band to see its effect on thermal distribution and stress for the structure. The original parameter is based on ASME requirement given by the equation below and will be used with the two different temperature profiles given above.

$$W = 2t \qquad (12)$$

Where:

W: width of heating band

t: Wall thickness

=99mm x 2

=198mm

Another way to calculate the recommended heating band width based on other standards is given in the following equation [21]:

$$W = 2.5(Rt_s)^{1/2} \qquad (13)$$

Where:
R: Vessel radius
t_s: Wall thickness

$=2.5(675x99)^{1/2}$
$=646mm$

The above width is said to extend from each direction of the edge of the weld. So the total width of the band is 1292mm. For Model 3 with attached nozzle, the heating band could not be extended to the required length because of the nozzle, so the width on the direction to the nozzle is until to the edge of the nozzle attachment. This width is used with the actual PWHT temperature profile. All the 3 localized PWHT conditions are summarized in Table 3:

Table 3: Thermal conditions to be simulated using ANSYS

	Initial Temperature (^oC)	Heating Band Width (mm)	Treatment Time (h)	Heating Rate (^oC/hour)	Holding Time (hour)	Holding Temperature (^oC)	Cooling Rate (^oC/hour)
Condition 1 (actual)	430	198	10.7	50	1h 10 mins	690	60
Condition 2		1292	10.7	50			60
Condition 3		198	18.5	30			30

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Thermal Analysis

Temperature plot throughout the PWHT process for all models and conditions are obtained in the ANSYS postprocessing option after the solution for the environment is complete. The temperature plot is taken at weld centre on the inner surface of the vessel and compared with the heating band temperature at outer wall as shown in Figure 13-15.

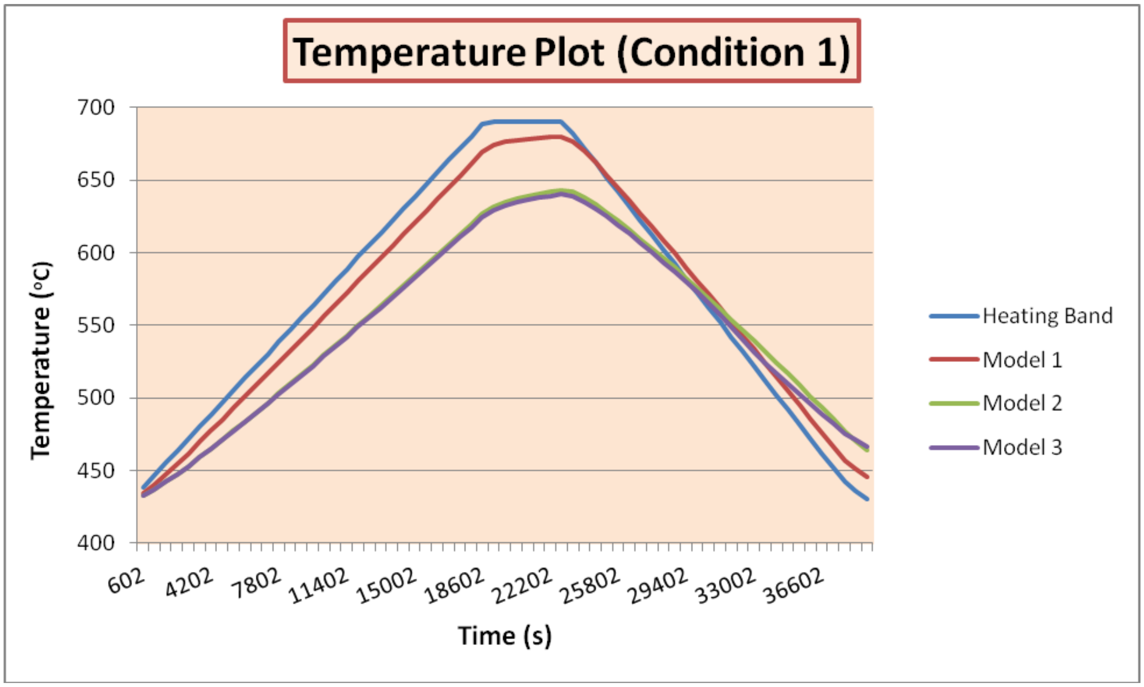


Figure 13: Temperature plot for PWHT Condition 1

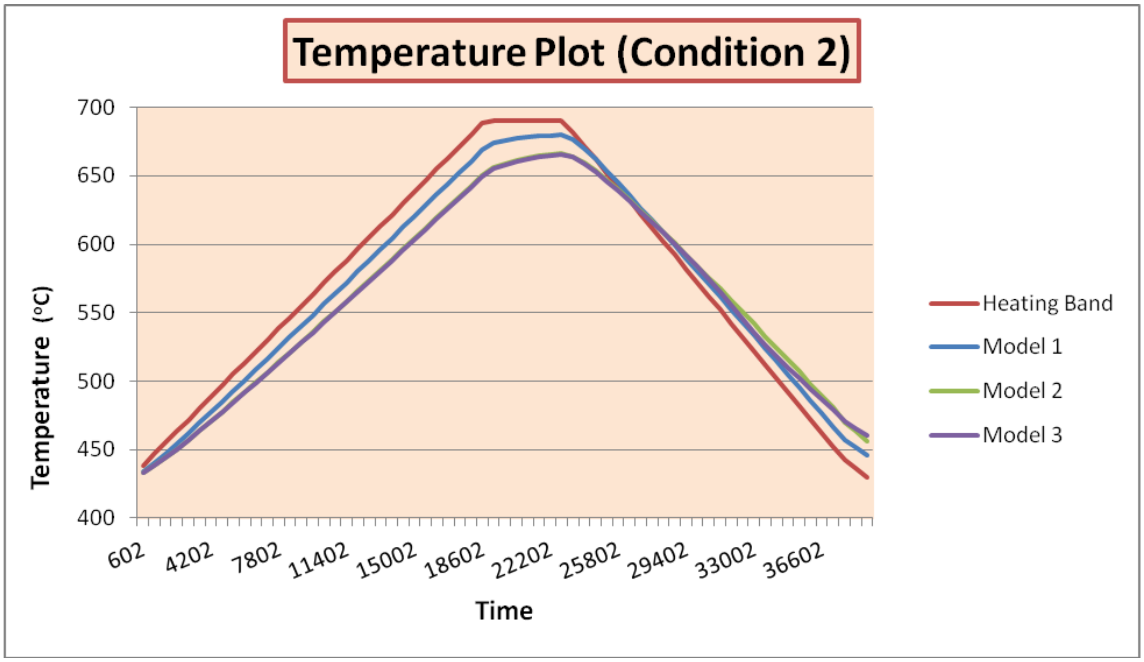


Figure 14: Temperature plot for PWHT Condition 2

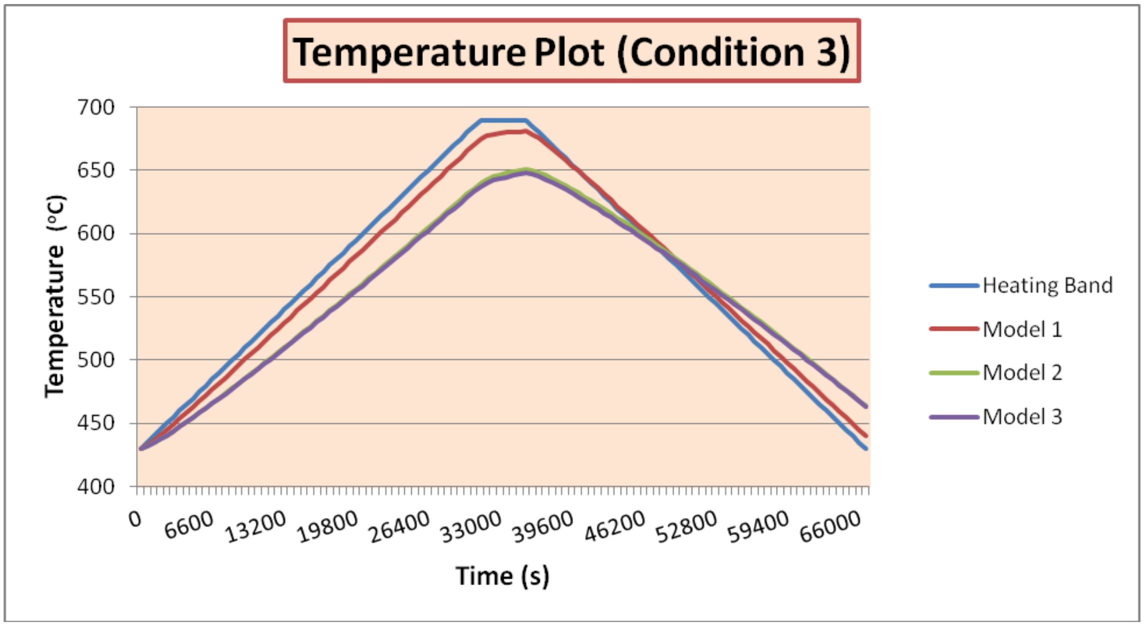


Figure 15: Temperature plot for PWHT Condition 3

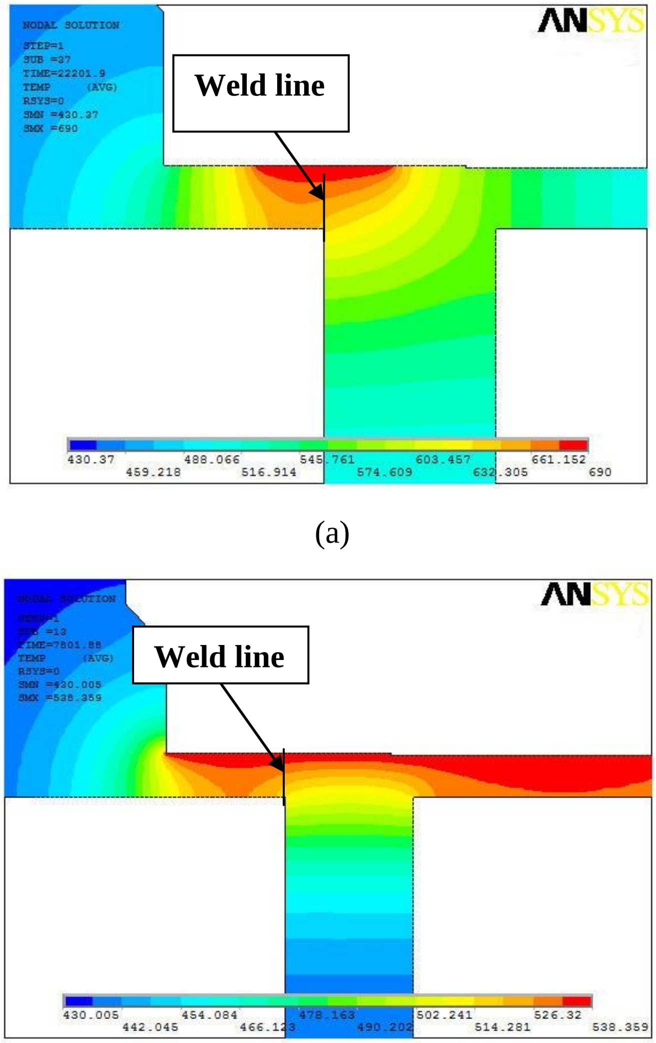
From the tabulated graphs above, it shows that under any conditions, Model 2 and Model 3 has considerable amount of thermal gradients at the inner surface compare to the heating band on the outer surface as high as 64 ^{o}C as seen in Table 4 below. Model 1 that has no adjacent structures near the weld area has temperature plot close to the heating band, indicating that the thermal distribution across the wall's thickness is acceptable. It can be seen that the presence of additional structures near the weld area has hindered ideal through-thickness heat penetration and ideal holding temperature cannot be reached. As being mentioned by Khaleel Ahmad et al. [20], it is important for the weld region and the heat affected zone (HAZ) to reach recrystallization temperature so that stress relieving effect can takes place.

Table 4: Highest temperature achieved average temperature difference at inner wall for the simulated conditions.

	Highest temperature at inner surface (^oC)		Biggest temperature difference (^oC)	Average temperature difference (^oC)
Condition 1 (actual)	Model 1	680	19.4	11.94
	Model 2	643	61.8	32.41
	Model 3	640	64	32.49
Condition 2 (extended heating band)	Model 1	680	19.4	11.94
	Model 2	666	37.7	21.88
	Model 3	665	38.5	21.57
Condition 3 (decreased heating/cooling rate)	Model 1	681	15	8.12
	Model 2	657	50	25.54
	Model 3	647	52	26.3

From the table, for the Condition 1 at Model 1 which resembles the PWHT process for the real equipment, the highest temperature achieved in inner surface at weld centre is only at 640 ^oC, well below the target temperature at 690 ^oC. In Condition 2 and 3, slight improvement in thermal distribution can be observed, even though the extended heating band width gives more adequate through-thickness heat penetration than altering the heating and cooling rates of the process.

From the thermal distribution results, the presence of tubesheet is the main factor contributing to the large thermal gradient and gives larger effect compare to the attached nozzle on the left. Since the location of the closure well that is so close to the thick tubesheet, it has prevented the desired conduction to the root of the weld as well at the nearing HAZ and because of its superior area compare to the wall's thickness, the tubesheet are tend to be the hat outlet away from the inner surface the "heat sink" effect is seen as suggested by D.M. Firth et al. [8]. Even with the extended width of heating band applied for Condition 2, the effect is still seen as shown in Figure 16 below.



(a)

(b)

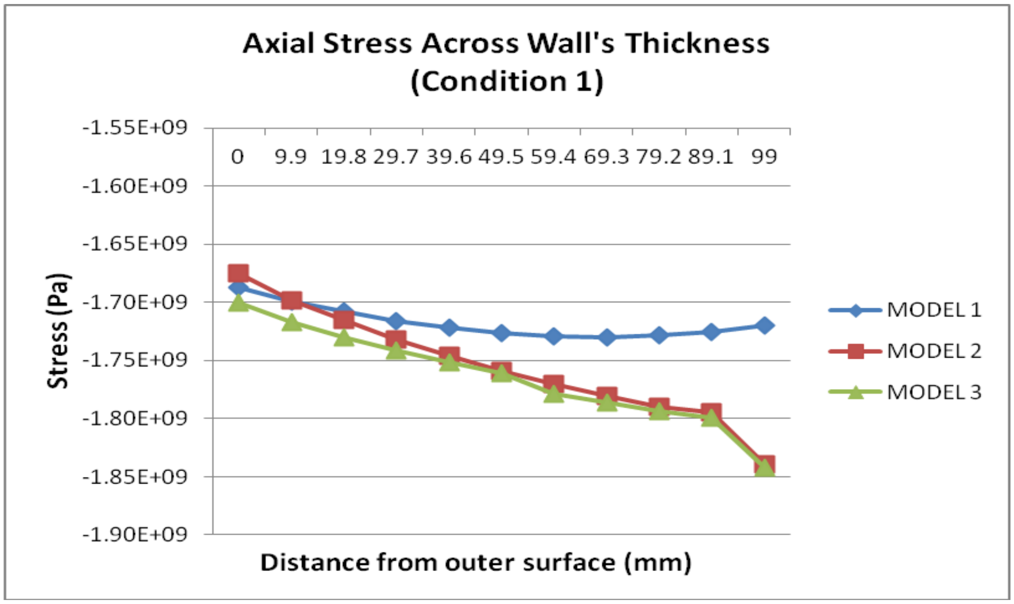
Figure 16 (a) and (b): Nodal temperature plot solution for Condition 1 and 2, showing lower temperature near the tubesheet region.

From the above figure, it can be seen the difference in maximum temperature penetration at the vessel on the right side of tubesheet and the area constrained between the nozzle and tubesheet. Furthermore, the width of heating band used for the process also plays important to the thermal distribution across the weld thickness. Since the target area of PWHT not only targets the weld region but also the HAZ, the heating band width must be large enough to ensure minimum required temperature extends through the wall's thickness in holding period and to prevent bending moment and shear stresses as stated by McEnerney et al.[21].

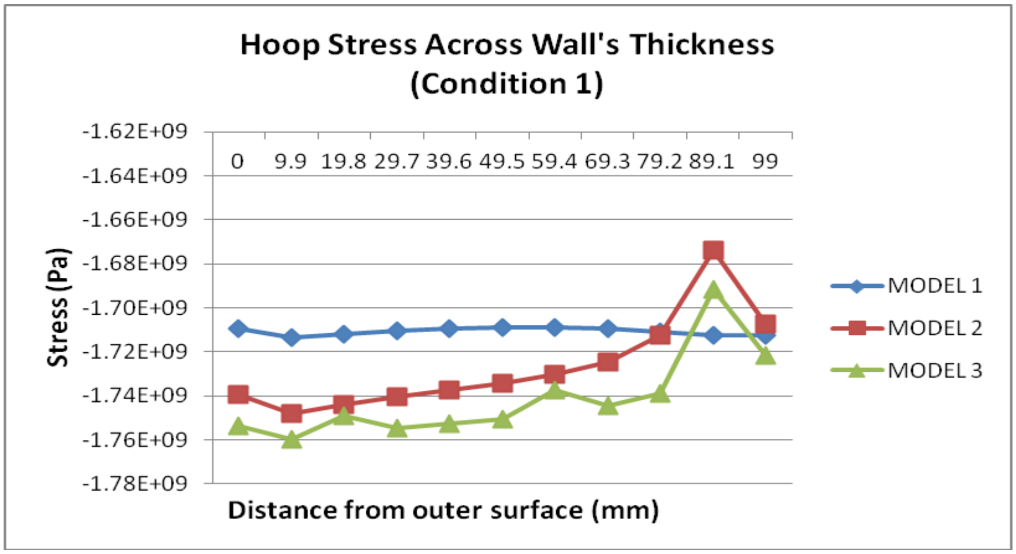
4.2 Structural Analysis

4.2.1 Nodal Stress Plot

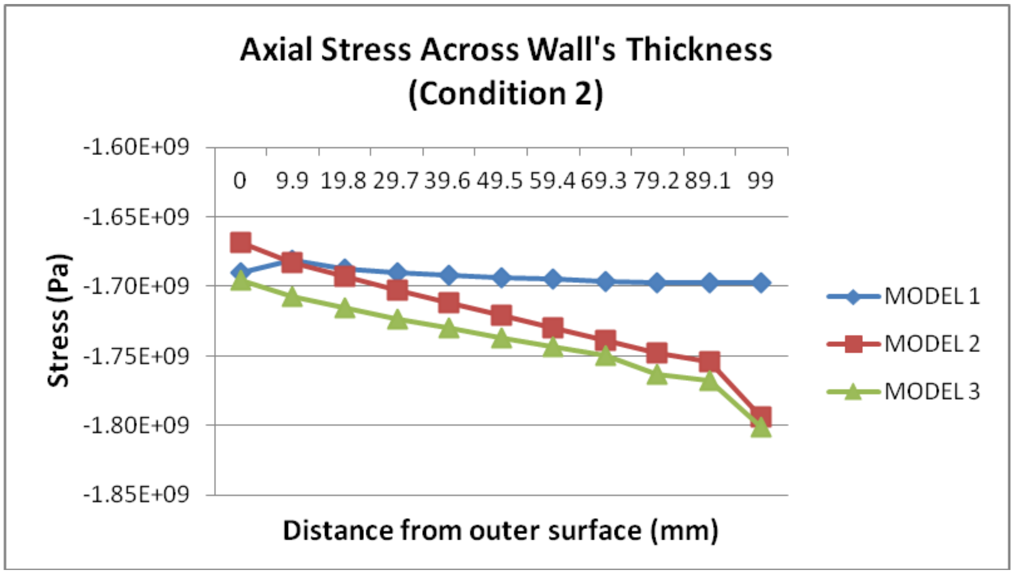
The structural analysis results obtained are from the thermal results from the previous thermal analysis. The graphs shown in Figure 17 below are axial and hoop stress taken across the wall's thickness at the weld centre at the end of cooling period.



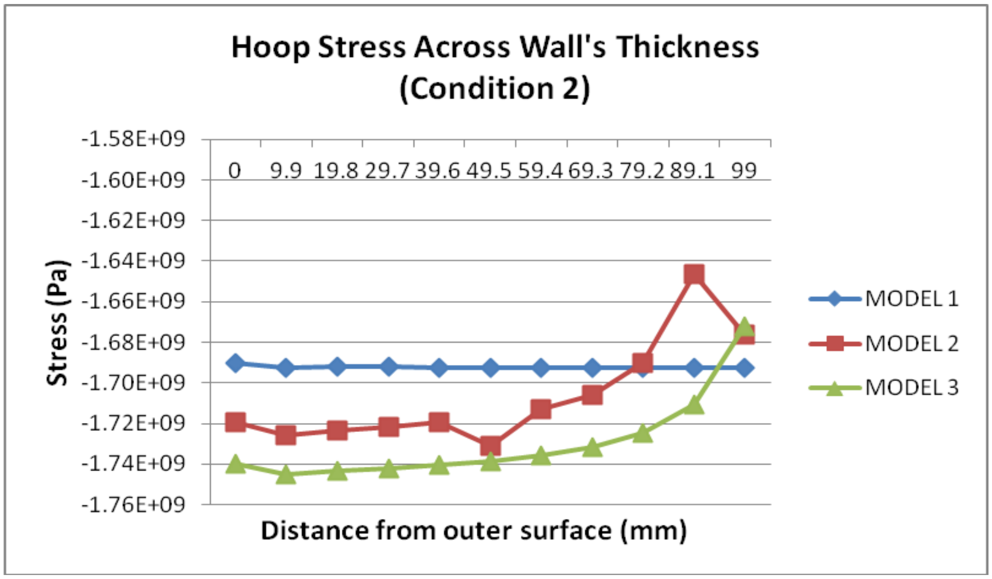
(a)


Hoop Stress Across Wall's Thickness
(Condition 1)
-1.62E+09
-1.64E+09
-1.66E+09
-1.68E+09
-1.70E+09
-1.72E+09
-1.74E+09
-1.76E+09
-1.78E+09
Stress (Pa)
0
9.9
19.8
29.7
39.6
49.5
59.4
69.3
79.2
89.1
99
Distance from outer surface (mm)
MODEL 1
MODEL 2
MODEL 3

(b)

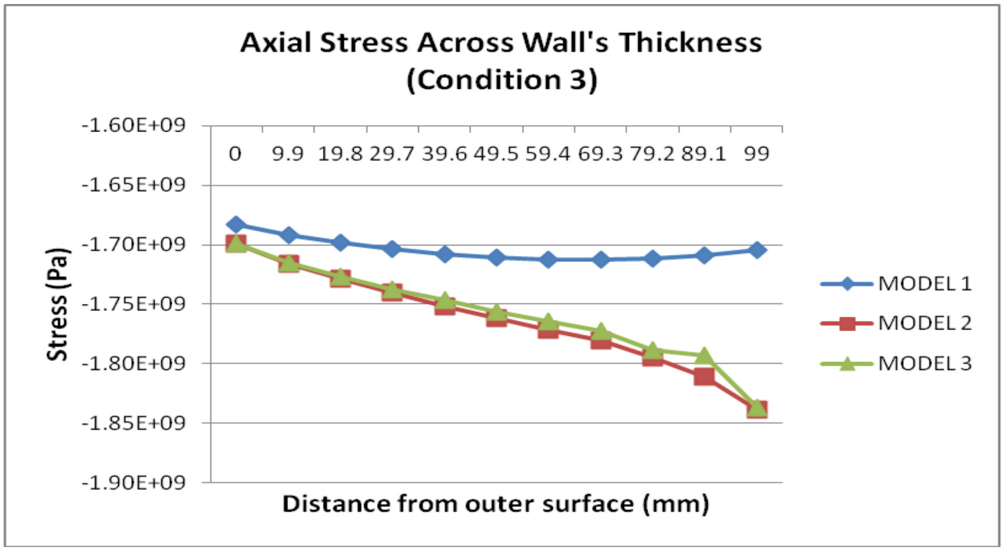

Axial Stress Across Wall's Thickness
(Condition 2)
-1.60E+09
-1.65E+09
-1.70E+09
-1.75E+09
-1.80E+09
-1.85E+09
Stress (Pa)
0
9.9
19.8
29.7
39.6
49.5
59.4
69.3
79.2
89.1
99
Distance from outer surface (mm)
MODEL 1
MODEL 2
MODEL 3

(c)



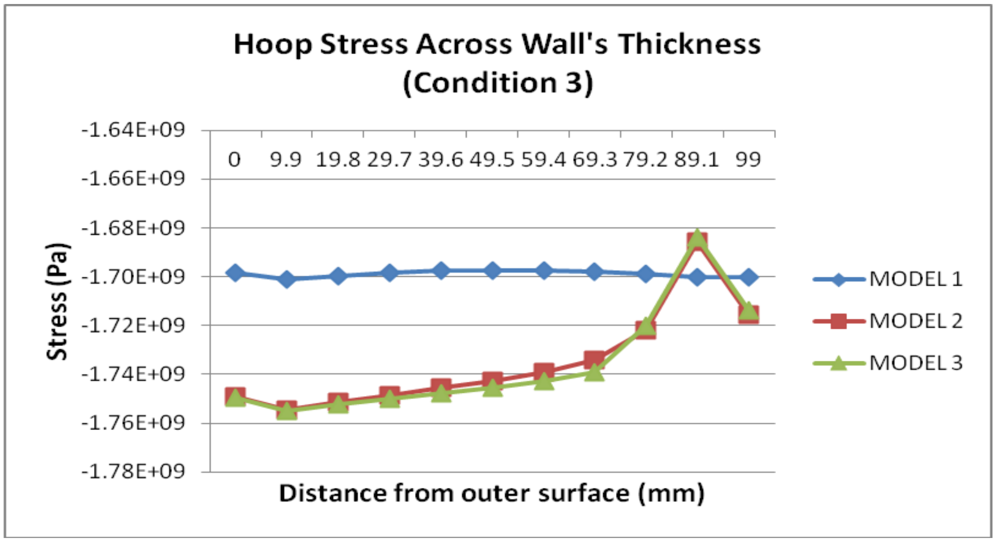
Hoop Stress Across Wall's Thickness (Condition 2)
-1.58E+09
-1.60E+09
-1.62E+09
-1.64E+09
-1.66E+09
-1.68E+09
-1.70E+09
-1.72E+09
-1.74E+09
-1.76E+09
Stress (Pa)
0
9.9
19.8
29.7
39.6
49.5
59.4
69.3
79.2
89.1
99
Distance from outer surface (mm)
MODEL 1
MODEL 2
MODEL 3

(d)



Axial Stress Across Wall's Thickness (Condition 3)
-1.60E+09
-1.65E+09
-1.70E+09
-1.75E+09
-1.80E+09
-1.85E+09
-1.90E+09
Stress (Pa)
0
9.9
19.8
29.7
39.6
49.5
59.4
69.3
79.2
89.1
99
Distance from outer surface (mm)
MODEL 1
MODEL 2
MODEL 3

(e)



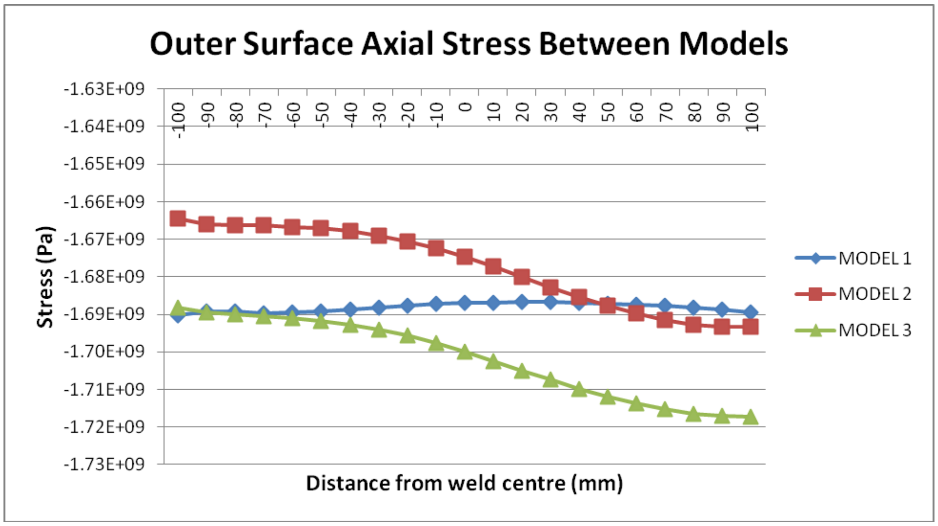
(f)

Figure 17 (a)-(f): Axial and hoop stress plot against wall thickness for all conditions

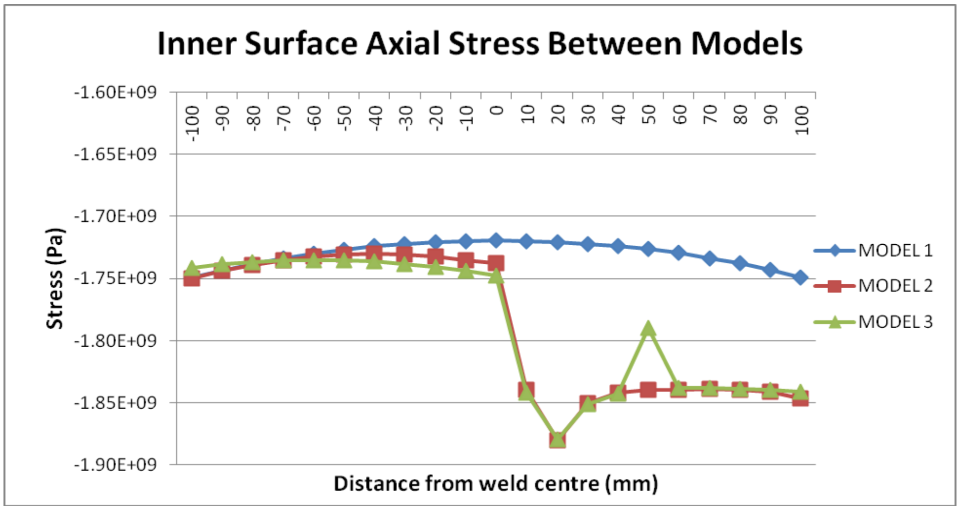
By comparing stress plot between the models, the models with additional structures have given higher stress values at all conditions, confirming the insufficiencies of the localized PWHT performed at the models. This happened from the high thermal gradients that lead to high thermal stresses throughout the PWHT process; even the values were taken after cooling process ended [8]. Axial stress for the models in all conditions is higher near the inner surface because of the expansion of the tubesheet that is in fact constrained during the heat treatment, while the hoop stress values are higher near the top surface because of the heat input from the heating band is from outside to inside.

For Condition 1, average axial stress increase 2.7% from Model 1 to Model 3, while average hoop stress increases 1.79%.

Comparing surface stress can also be useful in determining the effect could the adjacent tubesheet and nozzle give to the level of induced stress after the localized PWHT process. Figure 18 (a) and (b) show outer and inner surface axial stress with the weld being at centre. It can be observed that stress on outer surface between models vary between 1.66 GPa to 1.71 GPa. For inner surface, stress value increases at the tubesheet side for Model 2 and Model 3 from the resulting high thermal gradient, which induce higher amount of stress at the end of PWHT process. Highest stress value is located 20 mm to the right of the weld centre with 1.88 GPa, 9% increase from the Model 1without tubesheet and nozzle.



(a)



(b)

Figure 18 (a) and (b): Outer and inner axial stress plot, centered at the weldment

Having a heating band with critical length can be useful to reduce the induced stress from localized PWHT. Hao Lu et. al suggested that residual stress level reduces with increase of heating band width [11]. It can be observed in Figure 19 below that stress level through the wall's thickness is lower in Condition 2 having wide heating band width. By having greater area with uniform heating and cooling, high axial thermal gradient can be prevented.

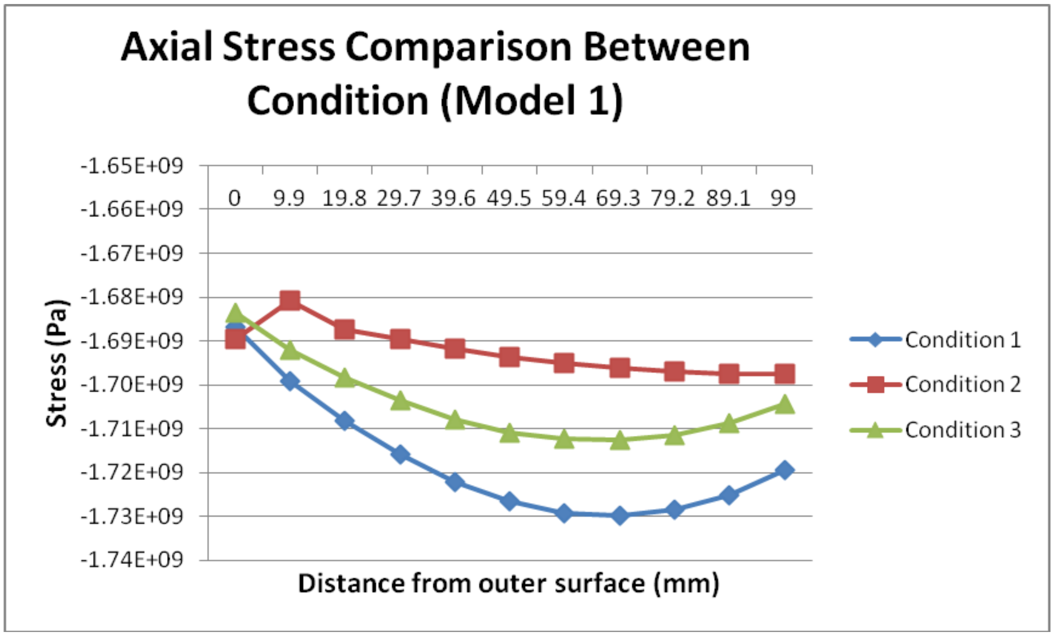
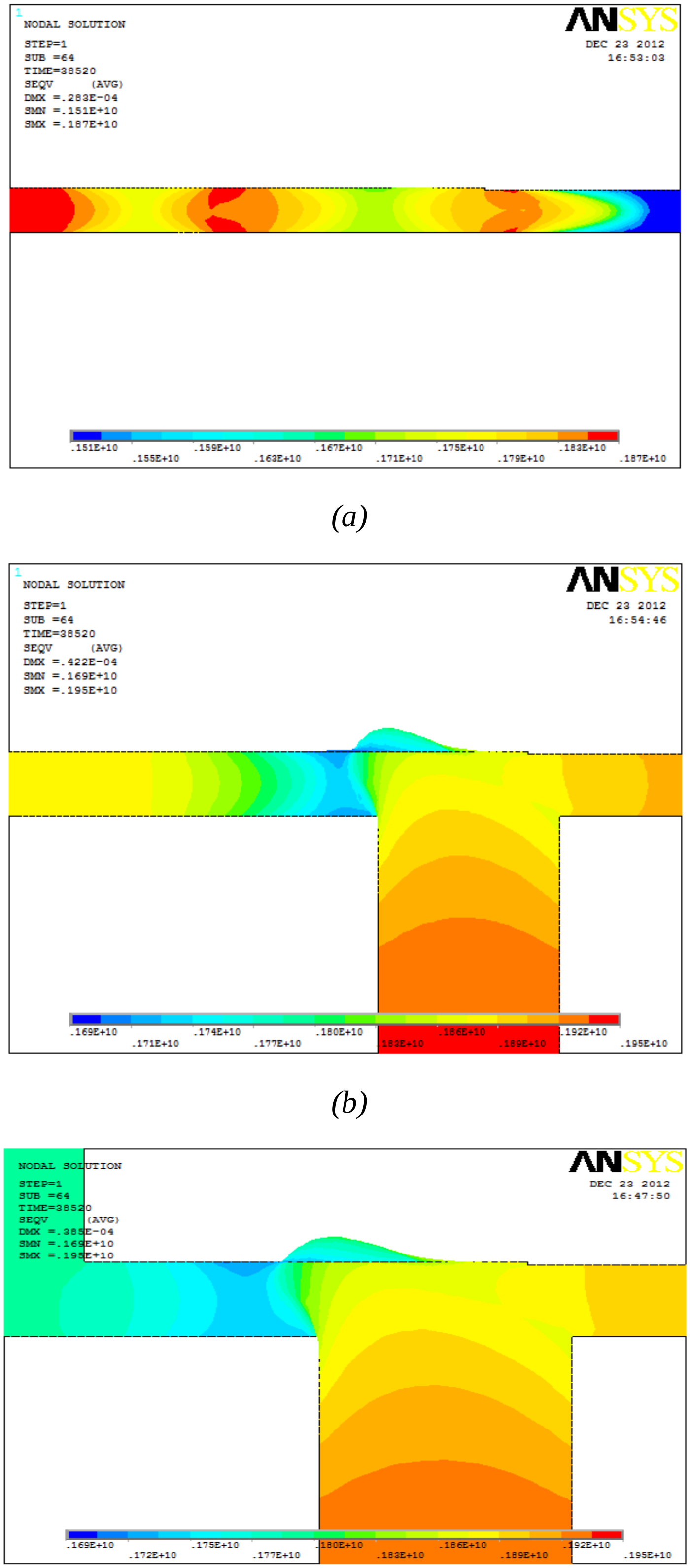


Figure 19: Through thickness axial stress between conditions

4.2.2 Von Mises Stress Plot

With Von Mises stress plot, the area within the vessel's models with high stress concentration can be observed. Higher Von Mises stress can indicate high possibility of yielding to happen when it reaches critical value, especially to ductile material. From Figure 20 (a), (b), and (c), higher stress intensity can be seen on the right side of the weld when the tubesheet is present. Stress intensity also higher at the nozzle area.



(a)

(b)

(c)

Figure 20 (a), (b), and (c): Von Mises stress plot. From top to bottom, Model 1, 2 and 3

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 Conclusion

It can be concluded that for component having thick adjacent structures give much effects to the sufficiency of localized PWHT. By having the tubesheet and nozzle, the thermal gradients increase up to 61^{o}C. Because of its large area, heat sink effect can be seen during the simulation which disrupts the proper heat transfer throughout the target area. The process also failed to reach the target temperature during the process, only manage to reach 640^{o}C. From structural analysis, stress value increases with the addition of the tubesheet and nozzle, with the higher stress coming from the tubesheet side. For simulation under different conditions, it can be observed that having wider heating band around the circumferential weld can be advantageous in covering more area with uniform heat transfer, while altering the heating and cooling rates could not give great improvement in term of thermal distribution because of the presence of nozzle and tubesheet.

5.2 Recommendations

In term fabrication process, it is recommended to uses furnace PWHT whenever possible when the weld area are close to thick structures to have uniform heat transfer from all directions to have better stress reduction effect [8]. If the localized PWHT have to be done, heating band width of $5(Rt)^{1/2}$ is recommended for thick walled vessels.

For future works, 3D FEM analysis of localized PWHT can be done with more detail and accurate parameters. To get more accurate results, welding process prior to the heat treatment can be perform to include the initial residual stress value to see the effect of thick structures on the stress relieving capability of local PWHT.

REFERENCES

[1] Phillipe Gilles, Ludovic Nouet, *Residual Stress Numerical Simulation of Two Dissimilar Metal Weld Junctions*, Residual Stress and Its Effects on Fatigue and Fracture, 2006, p. 3

[2] Zhili Feng, *Processes and Mechanisms of Welding Residual Stress and Distortion*,Woodhead Publishing in Materials, 2005

[3] D.E. Katsareas, *Finite Element Simulation of Welding in Pipes: A Sensitivity Analysis*, Residual Stress and Its Effects on Fatigue and Fracture, 2006, p. 15

[4] R.C. Wimpory, *Evaluation of Novel Post-weld Heat Treatment in Ferritic Steel Repair Welds Based on Neutron Diffraction*, Residual Stress and Its Effects on Fatigue and Fracture, 2006, p. 55

[5] R. Scott Funderburk, *Postweld Heat Treatment*, Welding Innovation Vol. XV, No. 2, 1998

[6] F.W. Burst, D.S. Kim, *Mitigating Welding Residual Stress and Distortion*, Processes and Mechanisms of Welding Residual Stress and Distortion, 2005

[7] *BFW Preheater E-18-02-A Crack at Channel Head to Tubesheet Weldment*, RCA documents, Petronas Fertilizer Kedah, 2010.

[8] D.M. Firth, D. Keen, C. Jones, A. Karstensen, *Cracking and Repair of Closing Welds in 2.25Cr 1Mo Steel Vessels Operating in High Temperature Synthesis Gas*, AMMONIA TECHNICAL MANUAL, 2005

[9] C.L. Tsai, D.S. Kim, Understanding Residual Stress and Distortion in Welds: An Overview, Processes and Mechanisms of Welding Residual Stress and Distortion, 2005

[10] J.R. Cho, B.Y. Lee, Y.H. Moon, and C.J. Van Tyne, *Investigation of Residual Stress and Post-weld Heat Treatment of Multi-pass Welds by Finite Element Method and Experiments*, Journal of Materials Processing Technology, Vol. 155–156, 2004, p. 1690

[11] Hao Lu, Jianhua Wang, Hidekazu Murakawa, *Mechanical Behaviour in Local Post Weld Heat Treatment,* Trans. JWRI Vol. 28. 1999, p. 55-59

[12] A. Mirzaee-Sisan, A.J. Fookes, C.E. Truman, D.J. Smith, T.B. Brown, T.A. Dauda, *Residual Stress Measurement in a Repair Welded Header in the As-welded Condition And After Advanced Post Weld Treatment*, International Journal of Pressure Vessels and Piping Vol 84, 2007, P. 265

[13] *ASME Section VIII, Rules for Constructions of Pressure Vessels Div I and II*, 2010 Edition

[14] P. Dechaumphai, *Finite Element Method: Fundamentals and Applications*, Alpha Science International Ltd., 2010

[15] *ANSYS v.14 Tutorial Guide*, ANSYS Inc., 2012.

[16] Crystal Cooper, *An Introduction to Thermal Stress, http://www.brighthub.com/engineering/mechanical/articles/23567.aspx*, last accessed on July 4th,2012

[17] T. Stolarski, Y. Nakasone, S. Yoshimito, *Engineering Analysis with ANSYS Software*, Elsevier Butterworth-Heinemann, 2006

[18] Bruno A. Latella, Samuel R. Humphries, *Young's Modulus of a 2.25Cr–1Mo Steel at Elevated Temperature*, Scripta Materialia V.51, 2004, p635–639

[19] Ya Tao Wu, *Residual Stress Study at the Dissimilar Metal Butt Joint due to the Weld Overlay Repair on a Nuclear Power Plant Nozzle Using ANSYS*, Rensselaer Polytechnic Institute, 2012

[20] Khaleel Ahmed, J. Krishnan, *Post-Weld Heat Treatment – Case Studies*, International Symposium on Thermal Spray, 2002

[21] Joseph W. McEnerney, Pingsha Dong, *Recommended Practices for Local Heating of Welds in Pressure Vessels*, WRC bulletin V.452, Welding Research Council, 2000.

APPENDICES

APPENDIX I

Gantt chart of activities for FYP 1

	May		**June**				**July**				**August**			
Week Number / Activities	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
Topic selection/confirmation	🟥	🟥												
Preliminary research studies on FEM and PWHT			🟥	🟥	🟥	🟥								
Extended proposal submission						🟥								
Proposal defense presentation								🟥	🟥					
Familiarization with FEM and ANSYS software							🟥	🟥						
Data gathering for design and modeling								🟥	🟥	🟥				
Design of various pressure vessels models											🟥	🟥		
Submission of interim draft report													🟥	
Submission of interim report														🟥

APPENDIX II

Gantt chart of activities for FYP 2

	September		October				November				December			
Week Number / Activities	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
Meshing and boundary conditions input	🟥													
Thermal analysis		🟥	🟥	🟥										
Progress report submission				🟥										
Structural analysis					🟥	🟥	🟥							
Results analysis								🟥	🟥	🟥				
Poster presentation											🟥			
Submission of project dissertation (soft bound)												🟥		
Technical paper submission												🟥		
Oral Presentation													🟥	
Submission of project dissertation (hard bound)														🟥

APPENDIX III

Key Milestones Table for FYP 1 and FYP 2

Milestones / Date of completion (week)	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
Topic selection		25/5												
Proposal defense									20/7					
Familiarization with FEM and ANSYS software									20/7					
Pressure vessel model designs with different geometries											3/8			

Milestones / Date of completion (week)	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**
Thermal anlysis				12/10										
Structural analysis								8/11						
Poster Presentation											3/12			
Oral Presentation														26/12
Sumission of Dissertation(hard bound)														28/12