NIPES

Journal of Energy Technology and Environment

Journal homepage: www.nipesjournals.org.ng

Design and Analysis of a Butt Welded Pressure Vessel Reactor

Eyere Emegbetere*, Boye ThankGod, Benjamin U. Oreko, Alexander Akene, Peter A. Oghenekowho, Bright A. Edward

Mechanical Engineering, Federal University of Petroleum Resources, Effurun, Nigeria.

**Corresponding Author- Email: emagbetere.eyere@fupre.edu.ng*

Article Info

Article history:
Received 8 May 2021
Revised 14 May 2021
Accepted 27May 2021
Available online 12 June 2021

Keywords:
Pressure vessel, Welding joints, Butt weld, Modelling, FEA, ANSYS

https://doi.org/10.37933/nipes.e/3.2.2021.8

https://nipesjournals.org.ng

Abstract

This article focused on the analysis of butt welded joints of a carbon steel pressure vessels reactor under certain conditions to determine a safe operating condition of the system. ANSYS was used to simulate and evaluate the butt welded joints. The welds were analyzed under load, thermal and fluid flow conditions so as to determine the optimum condition under which the welds and system may deform or become compromised. Results showed that the values of induced stress and strain obtained under a thermal load of 160MPa were 610.10MPa and 0.0069, respectively. In addition, the maximum principal stress and strain-induced on the butt welded joints under 160 MPa load condition were 1.096x10^5 MPa and 0.445, respectively. It was observed that the deformation on the system due to residual stresses developed around the heat affected zone of the butt weld would not cause the system to fail.

1. Introduction

Welding is a joining technique that has been around for centuries and it involves the use of heat or/and pressure with or without the use of filler rod to join pieces of metals together [1]. Most welding operations can either be automated or done by hand and since the 19th centuries, different welding processes have been developed and each has its own requirements, applications, advantages and disadvantages as compared to the other. In welding, as the parts cool to room temperature and become whole, an effective and strong bond is formed. However, for bonding to occur, an intensely localized heat input is required thus creating internal stress in the workpiece which causes deformation to the structure. It also generates residual stress in the heat affected zone (HAZ) of the structure. These undesirable conditions so developed are evident in most welded joints and if not properly analyzed and prevented can lead to weldment strength concerns. In reactor pressure vessels, fluid flow, thermal and load conditions can impact on the welded area. Therefore, there is the need to ascertain an operational safe working condition of the welds in other to avoid failures and subsequent accidents that may occur. In recent years, estimating the degree of welding deformation and the effects of the welding conditions can be achieved by simulation. The Finite Element Method (FEM), which is also known as the finite element analysis, is a numerical tool that can be applied to provide information problems associated with welded components [2-4]. [5] studied welded pipe induced by heating and was able to predict the welding residual stress and deformation associated with the workpiece with the use of FEM. [6] significantly reduced deformation by applying a cooling source on T-butt welding components. A study on laser welding in a two dimensional (2D) position with 13 mm thick high-strength steel to reduce welding defects on the flat position was conducted by [7]. The simulation of the fluid dynamics showed that welds can be enhanced if an appropriate balance can be achieved between the recoil pressure from the metal vapour, the surface tension and the hydrostatic pressure. Similarly, [8] reduced workpiece deformation and residual stress distribution by applying an outside heat source to the welding tube after extensive studying on pipe welding. [9] considered thermophysical material properties and by using the Goldak model investigated residual stress and temperature distribution in submerged welding pipe steels, thereby simulating the thermal cycles and welding residual stress in test steel. This study is aimed at analyzing residual stress and deformation developed in butt welded joints under fluid flow, thermal and load conditions in a reactor pressure vessel during operation. ANSYS, which is a finite element analysis software tool, was used to evaluate the effects of these conditions on the welds in the pressure vessel reactor.

2.0 Methodology

Most atmospheric and low-pressure tanks are designed to operate at pressures between $3.45x10^3$MPa (0.5 psig) and 0.1034MPa (15 psig) whereas, pressure vessels, on the other hand, are designed to operate between 15psig and 3000psig [10]. For this reason, carbon steel and low alloy steel are commonly used materials for construction of pressure vessel due to their high tensile strength. However, stainless steel 300 series can equally be used for designing pressure vessels with temperatures up to815.6 ^oC (1500^{o}F). It should be noted also that the minimum wall thickness of the welded metal plates subject to pressure should not be less than 2.4 mm [3]. Therefore, for the purpose of this research (butt welded reactor pressure vessel), a 5 mm thick carbon steel material whose tensile strength and yield strengths are 485MPa and 260MPa was selected for analysis. The designed vessel was subjected to a 160 MPa load and a pressure of 0.527MPa (76.45psig). ANSYS, a finite element analysis software was employed in this work. This software can be utilized in the analysis of fluid flow, non-linear, static structural, thermal, explicit and implicit dynamics, electric and electromagnetic field, etc. For efficiency, accuracy and reliability of the results obtained from the analysis, the following procedures would be adopted to analyze the welded joint of the pressure vessel.

2.1 Geometry importation

Autodesk inventor was used to generating the geometry of the proposed reactor with the welded joint. The parameters of the already fabricated reactor (pressure, tensile strength, wall thickness, length, etc.) were used to create the geometry and imported onto the ANSYS software for further analysis. Table 1 shows the parameter used for developing the reactor, while Figure 1, shows the geometry of the pressure vessel reactor developed.

Table 1: Parameters for the creation of the reactor pressure vessel

NOTATION	US Customary Unit	SI unit
P	73.46 psi	0.507 MPa
ID	157.48 in.	4m
R	78.74 in.	2m
Wall thickness	0.1975	5mm
Length, L or Height H	948.25 in.	25m

S	23,300 psi	160.7 MPa
E	1	1
Corrosion Allowance	0.079 in	2mm
Factor of safety	3.5	3.5
Tensile Strength	70,300 psi	485 MPa
Yield Strength	37,700 psi	260 MPa

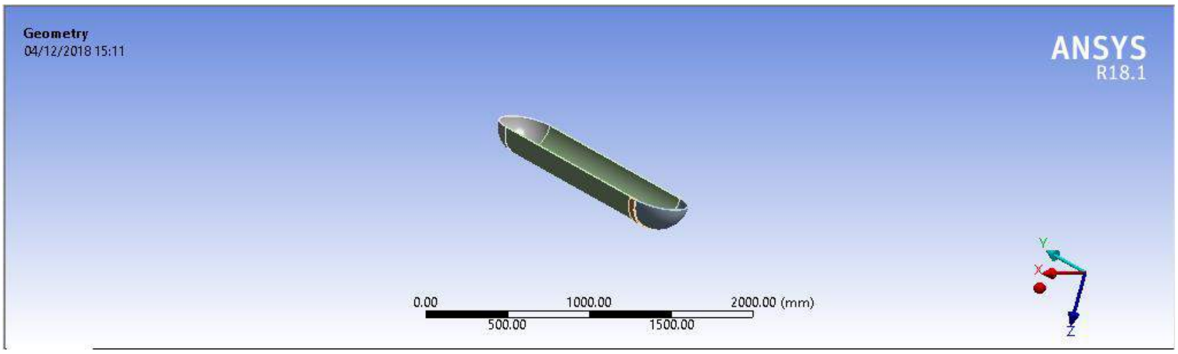

Figure 1: Geometry of the pressure vessel reactor with a Butt welded joints

2.2 Meshing

It involves dividing the component (pressure vessel) into finite number of smaller elements for the purpose of creating finite elements analysis. Meshing tends to hold the entire material element closely together in order to optimize the results obtained during analysis and in this design; a total of 1616 rectangular meshes was used as it offers better result for analysis since all elements are closely packed together, and further mesh refinement did not affect the result significantly. As shown in Figure 2, the weld was meshed together to produce a higher accuracy during the solution process. The meshing process generally involves;

i. Assignment of attributes to the created geometry
ii. Specification of mesh controls on the identified geometry
iii. Meshing
iv.

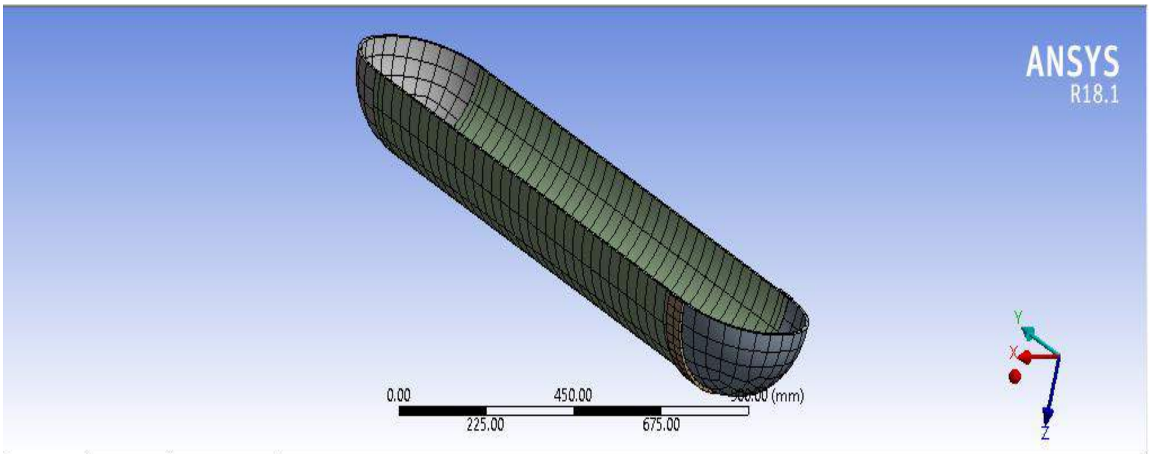

Figure 2: Meshing of the weld

2.3 Application of boundary condition and load

In order to simulate the system to obtain results as compared to the pre-existing pressure reactor, the force and maximum stress that the entire system will be subjected to during operating conditions are inputted into the design for analysis. In this case, a load of approximately 160.7 MPa was considered. The maximum principal stress and elastic strain analysis were however, carried out under the loading condition.

2.4 Thermal analysis

The thermal analysis involves subjecting the whole reactor to a certain operating temperature in order to monitor the behaviour of the welds joint and stimulate a safe operating temperature range. In performing this analysis, a temperature of 343^0C was adopted with respect to a maximum working temperature of the vessel reactor applied and simulate the response of the welds. The corresponding values for the maximum principal stress and elastic strain behaviour of the material were observed.

2.5 Flow analysis

Flow analysis was also carried out in order to determine the behaviour of the weld joint when subjected to the actions of fluid movement in and out of the system in other to determine whether or not the properties of a moving fluid within and out of the reactor would produce a corresponding effect on the weld.

3. Results and Discussion

3.1 Results

Based on the analysis conducted, Figures 3, 4, 5, 6, and 7 show the results of deformation and the principal stresses developed in the butt welds under loading, thermal and fluid flow conditions. The distribution of stresses is maximum around the butt welds whereas the deformation is least around this region; this is in line with results obtained from previous studies [11, 12]. Also, minimal stress levels were observed around the body of vessel, thus, confirming the operational safety of the system for the designed condition.

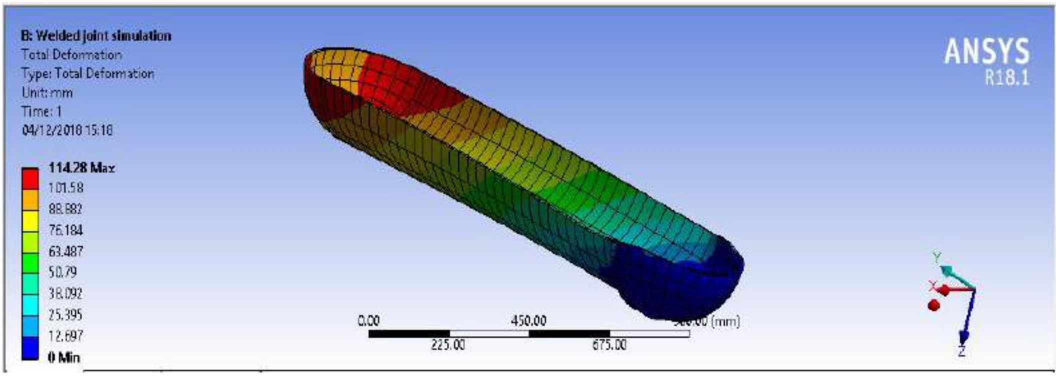

Figure 3: Total deformation of the butt welded joint at 160.7 MPa

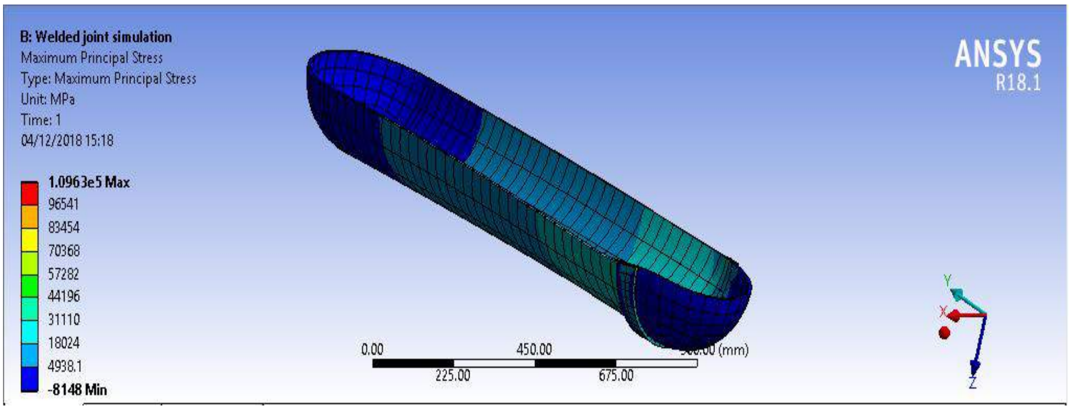

Figure 4: Maximum principal stress of butt welded joint at 160.65MPa

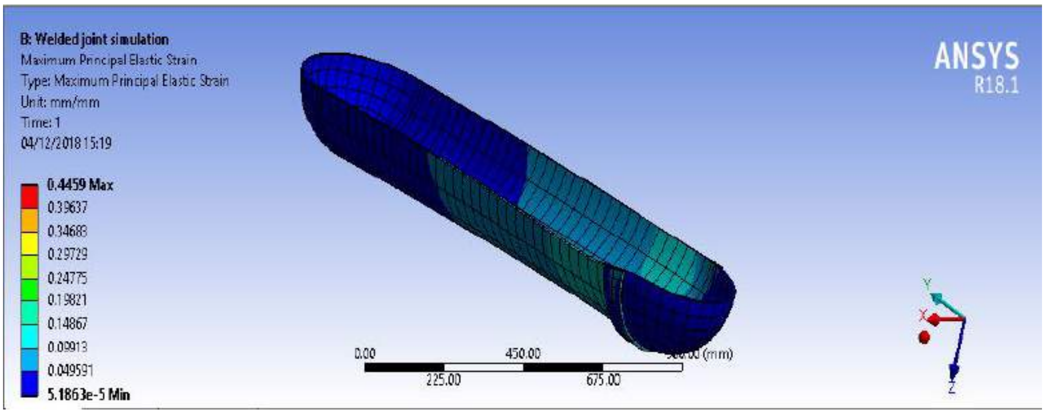

Figure 5: Maximum principal strain of butt welded joint at 160.65MPa

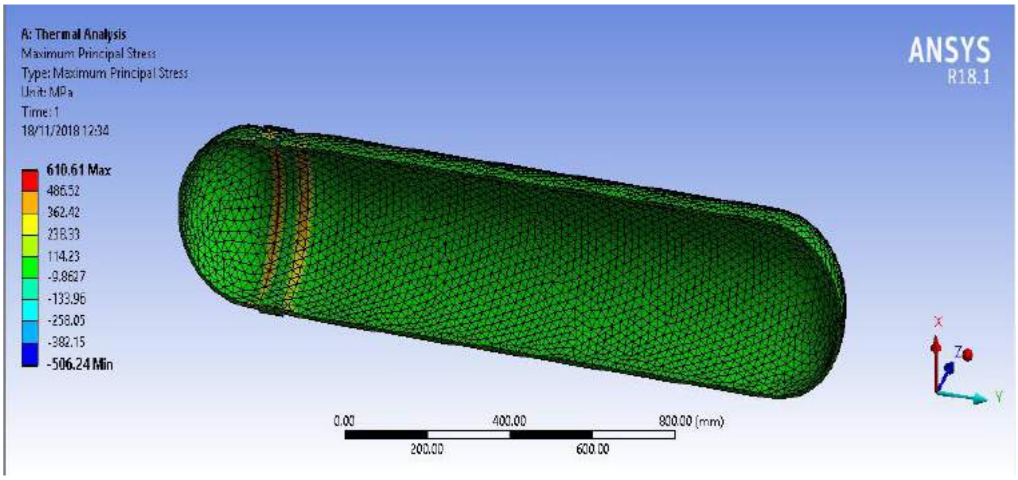

Figure 6: Maximum principal stress in the butt welded joint at 343^0C

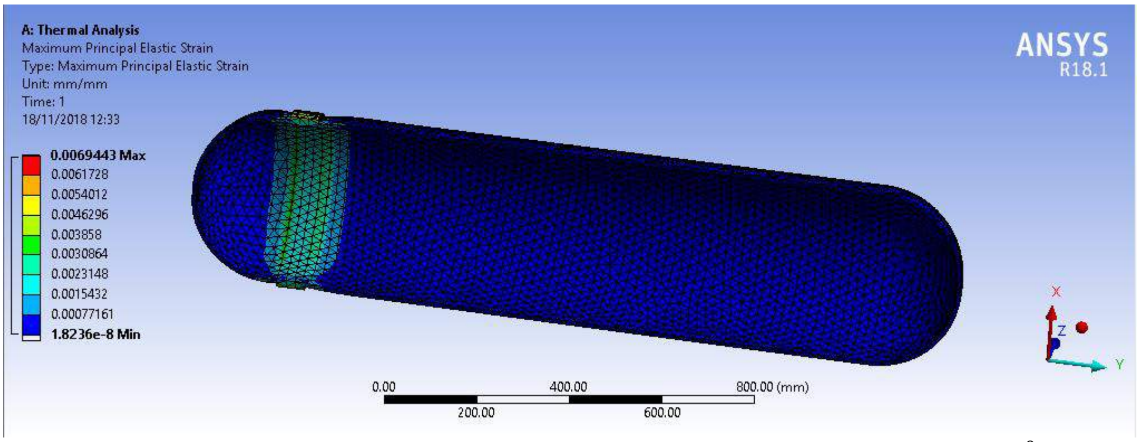

Figure 7: Maximum principal elastic strain of the butt welded joint at 343^0C

3.2 Discussion

Figure 3 shows the deformation that occurred during the simulation process of the pressure vessel reactor using ANSYS. The blue to amber colouration (0 – 101.58mm) indicates that the elastic deformation within which the butt welded reactor can safely operate if a load not greater than 160MPa is applied. Whereas the red colouration (114.28mm) indicates the point the material properties of the heat affected zone will become weak and thus affect the welds which could lead to failure of the system. In other words, when the butt weld is allowed to deform to such an extent that it tends toward the red colouration, the material will undergo plastic deformation which will, in turn, cause the material to yield and subsequently fail.

Figure 4 and 5 show the maximum principal stress and strain developed when the reactor is subjected to a load of 160MPa. The results obtained indicate that the induced stress ($1.096x10^5$ MPa) on the butt weld and the material, in general, cannot create a yielding effect around the heat affected zone (HAZ) of the weld as the developed stress is too little to cause any significant effect. The principal stresses in Figure 4 would cause a corresponding strain in the material however, the maximum strain so developed as simulated (0.4459) is too small to cause any significant effect on the materials integrity.

Figure 6 and 7 are the results obtained for the principal stress and strain developed when the reactor is subjected to an operating temperature of 343^0C. From the simulated results, under thermal condition, the maximum principal stress and strain so developed are 610.61MPa and 0.0069443 respectively. This, therefore, indicates a high level of induced stress on the weld and a low level of plastic deformation (strain) on the weldment area. However, the induces stress and strain computed is not large enough to cause any deformation to the weld and overall material owing to the high tensile strength of the material in use.

4.0 Conclusion

The ANSYS simulation of butt welded joint of a pressure vessel reactor indicate a safe operating conditions of the vessel, provided that the conditions are within the recommended safe region of the induced stresses, strains and load levels established. Simulated results obtained show that the values of induced stress and strain obtained under a thermal load of 160MPa were 610.10MPa and 0.0069, respectively. In addition, the maximum principal stress and strain-induced on the butt welded joints under 160 MPa load condition were $1.096x10^5$ MPa and 0.445, respectively. It was observed that the deformation on the system due to residual stresses developed around the heat affected zone of the butt weld would not easily cause the system to fail. This work however, can further be improve upon by developing a comperative approach in determining the most optimal and reliable approach in the application of predictive analysis of butt welds behaviour in pressure vessel reactors.

References

[1] Singh S.; and Gupa N. (2016) Analysis of Hardness in Metal Inert Gas Welding of Two Dissimilar Metals, Mild Steel and Stainless Steel. Journal of Mechanical Engineering and Civil Engineering 13 (3) 94-113

[2] David H. V., Plant J.; and Williams L. K. (2004) Basic Concept of Finite Element Analysis. Foundation of Finite Element Analysis.1-16

[3] Peters T. C.; Jordan, M. F.; and Frederick, V. (2004) Nondestructive Testing of Pressure Vessels, NUREG/CR-0909, U.S. Nuclear Regulatory Commission, Washington, DC.

[4] Dragi Stamenković and Ivana Vasović, (2009) Finite Element Analysis of Residual Stress in Butt Welding Two Similar Plates, Scientific Technical Review, Vol.LIX, No.1 Belgrade, Serbian

[5] Deng, D.; and Kiyoshima, S. (2010) FEM Prediction of Welding Residual Stresses in a SUS304 Girth-Welded Pipe with Emphasis on Stress Distribution Near Weld Start/End Location. Computer Material Science. 50, 612–621.

[6] Mochizuki, M.; Yamasaki, H.; Okano, S.; and Toyoda, M. (2013) Distortion Behaviour of Fillet T-Joint during In-Process Control Welding by Additional Cooling. Weld. World, 50, 46–50.

[7] Guo, W.; Liu, Q.; Francis, J. A.; Crowther, D.; Thompson, A.; Liu, Z.; and Li, L. (2015) Comparison of Laser Welds in Thick Section S700 High-Strength Steel Manufactured in Flat (1G) and Horizontal (2G) Positions. Manufacturing Technology., 64, 197–200.

[8] Ando, Y.; Yagawa, G.; and Hayase, Y. (1982) Evaluation of Induction Heating Stress Improvement (IHSI) Treatment Applied to Nuclear Primary Piping. International Journal Pressure Vessel. Pipe. 10, 399–406

[9] Nezamdost, M. R.; Nekouie Esfahani, M. R.; Hashemi, S. H.; and Mirbozorgi, S. A. (2016) Investigation of Temperature and Residual Stresses Field of Submerged ArcWelding by Finite Element Method and Experiments. International Journal. Advanced Manufacturing Technology., 87, 615–624.

[10] Perry, R. H; and Green D. W. (1997) Perry's Chemical Engineering Handbook. 7th Edition, McGraw-Hill, New York.

[11] Velaga, S. K.; Rajput, G.; Murugan, S.; Ravisankar, A.; Venugopal, S. (2015) Comparison of weld characteristics between longitudinal seam and circumferential butt weld joints of cylindrical components. Journal of Manufacturing Processes, 18, 1-11

[12] Krejsa, M.; Brovzosky, J.; Mikolasek, D; Parenica, P.; Koubova, L.; Materna, A. (2017) Numerical Modelling of Fillet and Butt Welds in Steel Structural Elements with verification using Experiments. Procedia Engineering, 190, 318-325