Science & Technology Development Journal – Engineering and Technology, 4(SI2):SI12-SI19

Open Access Full Text Article

Research Article

Simulation of thermal conduction and residual stress formulation upon a welded structure

Thien Tich Truong[1,2,*], **Tuan Hoang Lai**[1,2]

ABSTRACT

So far, welding has long been proved its diverse application in a large number of industries and manufactures. As versatile as welding can be, there have been a wide range of papers nationally and internationally published in scientific journals which granted an insight into various aspects of the welding procedure. Moreover, thanks to the advancement of computer softwares over the last couple decades, approaches and processes of carrying out welding simulation to determine weld temperature as well as post–welding residual stress have been publicly announced via a plethora of scientific papers on journals with high credibility. By excelling in the simulation, it is feasible to predict and modify welding parameters, for example heat source and material properties, before practical implementation, or even suggest proper post-weld heat treatment which is commonly utilized to eliminate structural defects after welding process is finished. Nonetheless, it is a plain truth that those papers whose scope were just limited to perform the inspection of temperature and residual stress. How significant those related results will impact the welded structure when it is put under operating conditions was hardly predicted and evaluated. That is the main objective of this paper. In this paper, a welding process taking place on a thin-walled vessel which was previously introduced in a published paper shall be re-simulated to validate the appropriateness of input data. Then, the structure is to be cooled until its temperature roughly equal to that of ambient one. After that, the welding results, namely post-welding temperature and residual stress, shall be imported into the strength testing simulation of the surveyed structure. Overall, with the equivalent stress accounting for 78% of the material's yield stress in the strength testing problem, the thin-walled vessel is still guaranteed for normal operation.

Key words: Welding, residual stress, transient welding, welding simulation, FEA

[1]*Department of Engineering Mechanics, Faculty of Applied Sciences, Ho Chi Minh City University of Technology, 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam.*

[2]*Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Vietnam.*

Correspondence

Thien Tich Truong, Department of Engineering Mechanics, Faculty of Applied Sciences, Ho Chi Minh City University of Technology, 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam.

Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Vietnam.

Email: tttruong@hcmut.edu.vn

History

- Received: 30-7-2021
- Accepted: 08-10-2021
- Published: 06-11-2021

DOI : 10.32508/stdjet.v4iSI2.884

INTRODUCTION

Theoretically, welding is a fabrication technique in which weld material shall be melted in order to be merged into the base material. The purpose of this technique is to join two or more separate objects or to fix damaged structures. By simulating the welding process, it is likely to estimate the proper heat source as well as anticipate residual stress so that effective solutions can be taken to resolve defects within the structure after welding. In this paper, a simulation of bonding two vessels by welding method shall be conducted through ANSYS Workbench to determine the temperature during the welding period, the residual stress, and parameters in the structure's strength testing.

As a matter of fact, an array of papers that were published in scientific journals have granted an insight into welding simulation and welding–related subjects. For example, the paper "Three-dimensional Finite Element Analysis for Estimation of The Weld Residual Stress in The Dissimilar Butt Weld Piping" (Kyoungsoo Lee, 2012) presented the welding simulation among two different metals to evaluate residual stress results through ABAQUS software. In 2018, author Hong Thanh Nguyen and his partners have shown their paper at the national scientific conference organized by the University of Thai Nguyen. The paper has introduced the simulation method to predict residual stress and distortion after welding by SYSWELD software. Those studies have resolved the temperature and post–welding residual stress as well as distortion. However, whether the structure can sustain operating conditions with the residual stress retained in it was not proposed. That is why this paper is carried out.

In detail, the model used in this paper is referred to Ref.[1] in which two vessels (as shown in. Figure 1 shall be bonded together. The type of welding applied in this study is Gas Tungsten Arc Welding (GTAW) and parameters relating to the welding torch are presented in Section 3 below. Moreover, prior to welding, a chamfer of 45° is generated at one end of each vessel, which forms the V-groove. The results of temperature shall be recorded during the circumferential

Cite this article : Truong T T, Lai T H. **Simulation of thermal conduction and residual stress formulation upon a welded structure**. *Sci. Tech. Dev. J. – Engineering and Technology;* 4(SI2):SI12-SI19.

Figure 1: Vessel Welding

traveling of the welding torch. Residual stress is determined shortly afterward. Then, the residual stress shall be applied in the strength test as a typical load.

The process of analyzing welding results carried over time requires huge quantity of calculations and time. Therefore, certain limitations are confronted on account of the restriction of software and hardware. A careful study of papers relating to welding which have been published on scientific journals reveals that it is necessary to simplify the transient analysis of welding so as to grant more memory space for hardware and reduce response time as well as calculations for software. The simplification of the problem can be conducted by an array of means (e.g., applying symmetrical conditions in terms of geometry, removing substructures and replacing them with equivalent constraints, and so forth). Nonetheless, those means must closely describe the practical conditions of the evaluated structure to ensure the precision of outcomes.

In addition, welding is also a nonlinear problem whose material properties change constantly during the surveyed period. Hence, correct material – property – versus – temperature input is another contributing factor to reliable results. However, hardly can these parameters be figured out for a specific material without measurement from experiments. It is the obstacle that occurred when Ref.[1] does not provide sufficiently the material properties used in the analysis. Thus, Ref.[2], together with other sources, were also referred to give out enough data to be inserted into FEA program.

The commercial FEA software in general and ANSYS Workbench in particular, are those applying the Finite Element Method theories in their solvers. Therefore, it is an advantage to obtain certain knowledge of this method prior to utilizing the software. In this paper, the finite element analysis for welding results shall be carried out in accordance with the procedure clearly illustrated in the flow chart in Figure 3 of Ref.[3].

BASIS OF THEORIES

Thermal Theories

The domain of heat distribution during the traveling of weld torch is determined in accordance with Fourier's heat flux equation as follows:

$$q = -k\nabla T \tag{1}$$

where k is the thermal conductivity ($\mathrm{Wm^{-1}K^{-1}}$).

In addition, each heat source is specified by its shape or in other words by its distribution; therefore, the equation depicting heat transfer is illustrated differently. In this paper, the widely well-known heat distribution Double Ellipsoidal equation which was first introduced by John Goldak and his partners in 1984 (Ref.[4]), is employed. This model of heat source provides fine power and density distribution in the weld pool as well as in the Heat Affected Zone (HAZ).

$$q = \frac{6\sqrt{3}Q\eta f}{\pi\sqrt{\pi}abc} e^{-3\left(\frac{x^2}{a^2}+\frac{y^2}{b^2}+\frac{z^2}{c^2}\right)} \tag{2}$$

where,

Q : total heat input (W), $Q = UI$;

U : welding voltage (V);

I : welding current (A);

f : fraction of heat;

η : welding efficiency;

a : length of heat source (m);

b : width of heat source (m);

c : length of heat source (m).

Besides, on all surfaces of the model, the heat loss due to convection and radiation shall be considered and presented in Equations (3) - (5) below:

$$q_{loss} = q_{convection} + q_{radiation} \tag{3}$$

$$q_{loss} = h_{total} \times A \times (T - T_{amb}) \tag{4}$$

$$h_{total} = \left[h + \varepsilon \times \sigma \times (T + T_{amb}) \times \left(T^2 + T_{amb}^2\right)\right] \tag{5}$$

where,

A : area of the exposed surface ($\mathrm{m^2}$);

T : current temperature (K);

T_{amb} : ambient temperature (K);

h : convection coefficient ($\mathrm{Wm^{-2}K^{-1}}$);

ε : emissivity;

σ : Stefan – Boltzmann constant (5.67 x 10-8 $\mathrm{Wm^{-1}K^4}$).

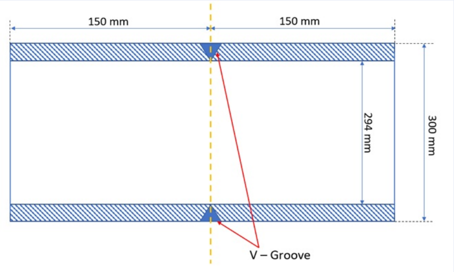

Figure 2: Dimensions of Vessel Segments

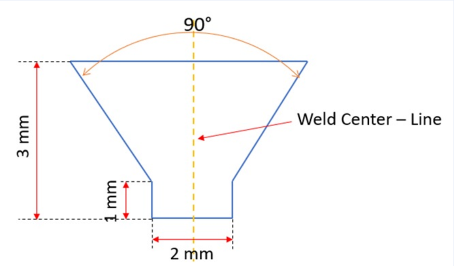

Figure 3: V – Groove Dimensions

Structural Theory

As for the welding problem, the behavior of the material is nonlinear, with sharp distortion. Thus, the isotropic hardening model is often applied for materials withstanding typically large strain. This model is illustrated by yield surfaces which are concentric; however, the size of the initial yield surface is smaller than subsequent ones.

Also, the post-welding residual stress is determined by the Yield Criterion of von Mises (as per Ref. [1]).

$$\sigma = \sqrt{\frac{1}{2}(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2} \qquad (6)$$

where

σ_1, σ_2, σ_3 are principal stresses respective to three axis of coordinate system (Pa).

Method for Research Implement

On account of analyzing the welding problem, ANSYS Workbench shall be employed, in which a specific type of element must be stated prior to solving. In the program, a combination of two modules, namely Transient Thermal and Static Structural, was utilized for this problem.

It should be noted that in the Transient Thermal module, the element type SOLID70 which is a linear eight-node element is employed. Each node of this element possesses only one degree of freedom. Meanwhile, the SOLID185 element is used in the Static Structural module. Both SOLID70 and SOLID185 share their geometry as linear eight-node element. Nonetheless, the only difference between SOLID70 and SOLID185 is that each node of SOLID185 obtains three degrees of freedom.

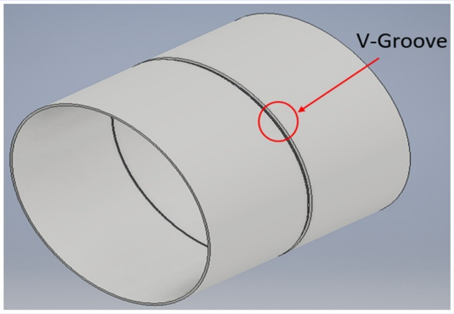

Figure 4: The Complete 3D Model

WELDING SIMULATION

Model Description

In this simulation, two separate segments of the vessel shall be bonded by welding one end of one segment to another. Each of them possesses an outside radius of 150 mm with a thickness of 3 mm and the length of 150 mm (Figure 2). Moreover, the dimensions of the V-groove are illustrated in Figure 3 below. Also, the 3D geometry of the model is shown in Figure 4. However, as aforementioned in Section 1, to cut down on the amount of calculation time and make it possible for the response capacity of both software and hardware, just a quarter of the complete model was analyzed (Figure 5).

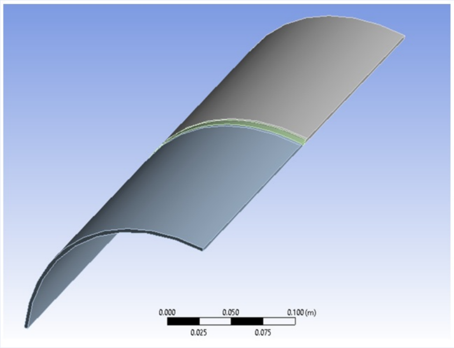

Figure 5: 3D Geometry of a Quarter Model

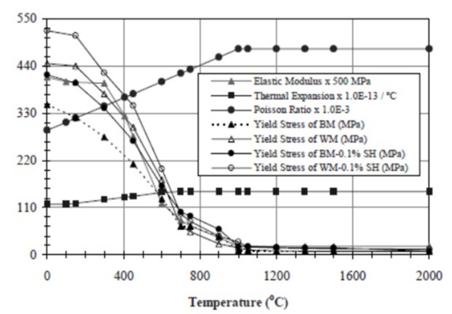

Figure 6: Mechanical Properties of Base Metal and Weld Metal

Material properties

There are two types of material used in this simulation as regards Base Metal (BM) and Weld Metal (WM – ASTM AH36 Alloy Steel). Their properties are taken from Ref. [1] as well as Ref. [2] and shown in Figure 6 and Figure 7, correspondingly.

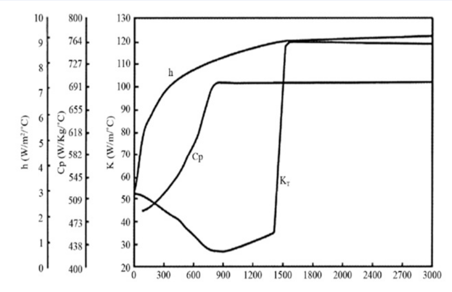

Figure 7: Thermal Properties of Weld Metal

Loads

Boundary Conditions

A quarter of the complete vessel is evaluated; thus, it is necessary to apply symmetrical conditions for the simplified model where appropriate. Besides, at the end of one vessel segment, a displacement constraint about x-, y- and z-axis is set. Ambient temperature is taken as 300K.

Loads

Loads in the welding problem are thermal loads including heat flux, convection, and radiation. On the exposed faces of the model, the condition of convection, with convection coefficient $h = 8\ \mathrm{Wm^{-2}K^{-1}}$, as well as radiation, with the emissivity $\varepsilon = 0.51$, is applied. Meanwhile, the model of the moving Double Ellipsoidal heat source model is employed through an APDL code. The parameters of this heat source are presented in Table 1 below.

Furthermore, the complete model takes roughly 314 seconds to finish the welding process, with load step is 0.87 seconds. Also, it is an assumption that the velocity of the welding torch during the whole process is constant.

The strength test is conducted through two load steps. In this simulation, four types of load are applied, namely internal pressure, end cap pressure, moment, and residual stress from the welding process. In detail, the internal pressure of 3.8 MPa is applied for all the inner faces of the vessel in two load steps. Next, an end cap pressure of 176 MPa is set on the other end of the vessel that is not covered by the displacement constraint as mentioned above in both steps. Also, at this end, a moment of 7850 N.mm which rotates about the axial axis of the structure shall be applied in step two only. Finally, the syntax "INISTATE" which is introduced in Ref. [5] shall be employed to import residual stress from the welding process into the structure's strength testing simulation.

Meshing

During the welding process, the areas where the welding torch passes as well as its neighboring surfaces (or so-called Heat Affected Zones - HAZs) usually witness considerable fluctuations in terms of the evaluated results. Thus, the mesh quality in these zones shall be particularly much finer than the others to achieve result convergence.

In this simulation, a distance of 10 mm measured from the weld center – line on both vessel segments is assumed to be the HAZs. Within the weld zone and HAZs, the element size shall be 1 mm. The mesh gets coarser as it reaches toward both ends of two segments (Figure 8). In addition, a comparison between the mesh quality of Ref. [1] and that of this paper is also shown in Table 2 below. It should be noted that just a quarter of the complete model is evaluated; therefore, the number of nodes and elements in this paper is just about one-fourth of that in Ref. [1].

RESULTS AND DISCUSSIONS

The results of temperature and residual stress in this study shall be compared to those in Ref. [1] respectively. Firstly, the temperatures at section 45^o from weld start at 39.27s, 117.81s and 274.89s are illustrated in Figure 9, Figure 10, and Figure 11, correspondingly.

Then, the temperature at the end of the surveyed period is imported into the structural analysis to determine the residual stress which is indicated in Figure 12. The residual stress is inspected at cross section

Table 1: Parameters of heat source

Parameter	Unit	Value
Length of heat source, a	m	0.02
Width of heat source, b	m	0.01
Depth of heat source, c	m	0.003
Fraction of heat, f	-	1.25
Welding voltage, U	Volt	12.5
Welding Current, I	Ampere	200
Welding efficiency, η	-	0.8
Welding speed	mm/s	3

Table 2: Comparison of mesh quality

Description	Ref. [1]	This paper
Total Number of Nodes	71040	18182
Total Number of Elements	54720	14022

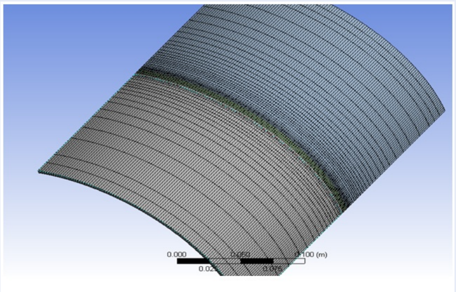

Figure 8: Mesh of The Model

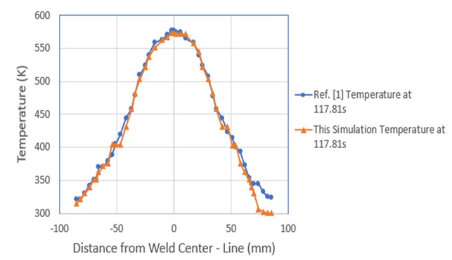

Figure 10: Temperature at 117.81s

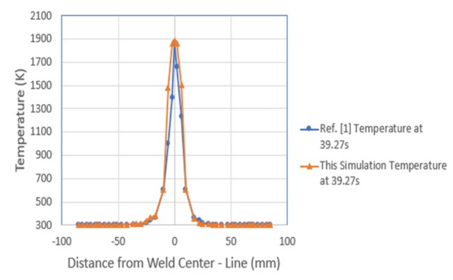

Figure 9: Temperature at 39.27s

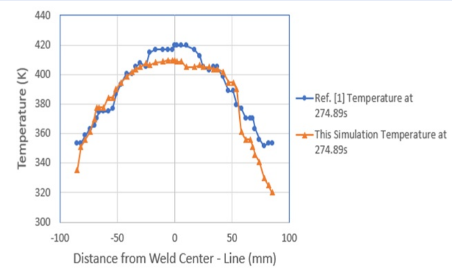

Figure 11: Temperature at 274.89s

90^o from the weld start.

With the aforementioned conditions of loading, the von Mises equivalent stress and displacement results of the vessel structure are shown in Figure 13 and Figure 14, correspondingly.

As is shown in Figure 9, in the time step of 39.27s, the highest difference regarding temperature among both studies occurring at the position of -6 mm is 12.35%. Likewise, Figure 10 reveals the temperature versus distance from weld – centerline at 117.81s received from both studies. In this time step, the highest difference regarding temperature taking place at the

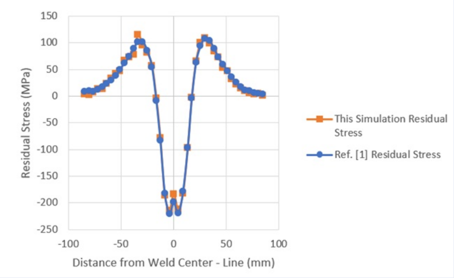

Figure 12: Residual Stress at Section 90^o Axial

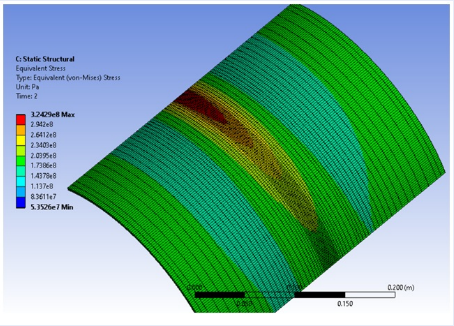

Figure 13: Equivalent Stress of von Mises

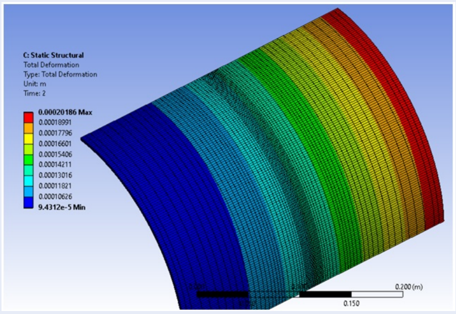

Figure 14: Total Displacement of The Structure

position of 82 mm is 7.92%. Moreover, it is apparent from Figure 11 that at 274.89s, the highest difference regarding temperature happening at the position of 85 mm is 9.41%.

In the case of structural outcomes, the residual stress versus the distance from weld – centerline is also recorded constantly at cross-section 90° from the weld start point. At the weld – center line, the residual stress extracted from Ref.[1] is 198.38 MPa, whereas that from this simulation is 183.03 MPa. The difference in terms of residual stress at weld – centerline is 7.74%. In addition, at the position of -33 mm from the weld – centerline, the residual stress in this simulation is the most significantly different from Ref.[1], 101.18 MPa compared to 115.82 MPa correspondingly (difference by 14.47%).

CONCLUSION

This paper has introduced a procedure for predicting the strength of a structure after it experienced a welding process. It can be concluded from stated results that:

- Though there are certain differences from Ref.[1] in the results of weld temperature and residual stress, the distribution domain of these parameters of both studies illustrated in Figure 9 – Figure 12 was relatively comparable to each other.
- With the results of post-welding strength testing problem, this paper has shown the degree of completion of the simulated structure. Based on this result, proper solutions, typically post-welding heat treatment, can be proposed prior to practical experiment or manufacture. By excelling in this procedure, the lifespan of the structure shall be extended.

In a near future, we hope to improve this discrepancy. Besides, the post-welding heat treatment is a common solution in practice to minimize the amount of residual stress as well as strengthen the whole structure after welding processing. However, this type of simulation requires an enormous figure of software calculations. In other words, it takes massive hardware memory to carry out the simulation thoroughly. That is indeed a challenge that we would like to take on soon.

ACKNOWLEDGMENT

This research is funded by Ho Chi Minh City University of Technology – VNU-HCM, under grant number SVCQ-2020-KHUD-43. We acknowledge the support of time and facilities from Ho Chi Minh City University of Technology (HCMUT), VNU-HCM for this study.

NOMENCLATURES

FEA: Finite Element Analysis
GTAW: Gas Tungsten Arc Welding
HAZ: Heat Affected Zone
BM: Base Metal
WM: Weld Metal
ASTM: American Society for Testing and Materials
APDL: Ansys Parametric Design Language

CONFLICT OF INTEREST

Group of authors declare that this manuscript is original, has not been published before and there is no conflict of interest in publishing the paper.

AUTHORS' CONTRIBUTION

Thien Tich Truong is the supervisor, contributes ideas for the proposed method and also takes part in the work of gathering data and checking the numerical results

Tuan Hoang Lai works as the chief developer of the method and the manuscript editor.

REFERENCES

1. Dar NU, et al. Analysis of weld - induced residual stresses and distortions in thin - walled cylinders. Journal of Mechanical Science and Technology. 2009;23:1118 –1131. Available from: https://doi.org/10.1007/s12206-008-1012-6.
2. Guimaraes PB, et al. Determination of Residual Stresses, Numerically Obtained in ASTM AH36 Steel Welded by TIG Process, Material Sciences and Applications. 2013;4:268 –274. Available from: https://doi.org/10.4236/msa.2013.44033.
3. Mondal AK, et al. Experimental and FE Analysis of Submerged Arc Weld Induced Residual Stress and Angular Deformation of Single- and Double-Sided Fillet Welded Joint. International Journal of Steel Structures. 2017;17. Available from: https://doi.org/10.1007/s13296-016-0093-9.
4. Goldak J, et al. A New Finite Element Model for Welding Heat Sources. 1984;15B:305. Available from: https://doi.org/10.1007/BF02667333.
5. ANSYS Workbench Help, version 18.1;.

Tạp chí Phát triển Khoa học và Công nghệ – Kĩ thuật và Công nghệ, 4(SI2):SI12-SI19

Bài nghiên cứu

Mô phỏng sự truyền nhiệt và hình thành ứng suất dư của kết cấu hàn

Trương Tích Thiện[1,2,*], Lại Hoàng Tuấn[1,2]

TÓM TẮT

Từ lâu, hàn đã được chứng minh được sư đa dạng của nó trong việc ứng dụng vào nhiều ngành công nghiệp và sản xuất. Với đặc điểm đa dụng mà phương pháp hàn mang lại, đã có rất nhiều bài báo trong nước lẫn quốc tế được xuất bản trên các tạp chí khoa học nhằm mang lại một cái nhìn chuyên sâu hơn về nhiều khía cạnh khác nhau của quá trình hàn. Hơn nữa, nhờ sự phát triển của các phần mềm máy tính trong vài thập kỷ qua, các phương pháp tiếp cận và quy trình thực hiện mô phỏng hàn để xác định nhiệt độ mối hàn cũng như ứng suất dư sau hàn đã được giới thiệu rộng rãi thông qua một loạt các bài báo khoa học đăng trên các tạp chí có uy tín. Thông qua mô phỏng, các thông số hàn, ví dụ như nguồn nhiệt và đặc tính vật liệu, có thể được dự đoán trước cũng như có thể được điều chỉnh, trước khi thực hiện hàn thực tế, hoặc thậm chí là đề xuất biện pháp nhiệt luyện thích hợp nhằm loại bỏ các khuyết tật của kết cấu sau khi kết thúc quá trình hàn. Tuy nhiên, trong thực tế, đa phần các bài báo liên quan đến vấn đề hàn đã được công bố thường chỉ giới hạn ở việc thực hiện kiểm tra nhiệt độ và ứng suất dư. Nhưng liệu các kết quả của bài toán hàn sẽ ảnh hưởng như thế nào đến kết cấu hàn khi nó được đưa vào hoạt động vận hành thì hầu như không được phân tích và đánh giá. Đó cũng là mục tiêu chính của bài báo này. Trong bài báo này, quy trình hàn diễn ra trên bồn có thành mỏng mà đã được công bố trước đó trong một bài báo sẽ được mô phỏng lại để đảm bảo tính đúng đắn của mô hình được mô phỏng so với bài báo tham khảo. Tiếp theo, kết cấu sẽ được làm nguội đến nhiệt độ của môi trường xung quanh. Sau đó, kết quả hàn, cụ thể là nhiệt độ sau khi hàn và ứng suất dư, sẽ được đưa vào mô phỏng kiểm tra độ bền của kết cấu được để xem liệu ứng suất có ảnh hưởng đến kết cấu bồn khi vận hành. Nhìn chung, với ứng suất tương đương bằng 78% ứng suất chảy của vật liệu trong bài toán thử độ bền, kết cấu bồn thành mỏng vẫn đảm bảo hoạt động bình thường.

Từ khoá: Hàn, ứng suất dư, quá trình hàn, mô phỏng hàn, phần tử hữu hạn

[1] *Bộ môn Cơ Kỹ Thuật, Khoa Khoa học Ứng Dụng, Trường Đại học Bách Khoa Thành phố Hồ Chí Minh, Việt Nam*

[2] *Đại học Quốc Gia Thành phố Hồ Chí Minh, Việt Nam*

Liên hệ

Trương Tích Thiện, Bộ môn Cơ Kỹ Thuật, Khoa Khoa học Ứng Dụng, Trường Đại học Bách Khoa Thành phố Hồ Chí Minh, Việt Nam

Đại học Quốc Gia Thành phố Hồ Chí Minh, Việt Nam

Email: tttruong@hcmut.edu.vn

Lịch sử

- Ngày nhận: 30-7-2021
- Ngày chấp nhận: 08-10-2021
- Ngày đăng: 06-11-2021

DOI : 10.32508/stdjet.v4iSI2.884

VNU-HCM Press

Trích dẫn bài báo này: Thiện T T, Tuấn L H. **Mô phỏng sự truyền nhiệt và hình thành ứng suất dư của kết cấu hàn.** *Sci. Tech. Dev. J. - Eng. Tech.;* 4(SI2):SI12-SI19.