International Journal of Mechanical and Sustainability Engineering Technology

Journal homepage:
https://uniexpertsacademy.com/index.php/IJMSET/index
ISSN: 3083-8363

Mesh Convergence Study of A106 Grade B Carbon Steel Pipe on Different Type of Ceramic Pads in Post-Weld Heat Treatment (PWHT)

Nurul Shahadah Roslan[1], Eliza M.Yusup[1,*], Iman Fitri Ismail[1], Muzzahhir Mohmad Hadzir[2], Mohamad Nur Hidayat Mat[3], Shahzulreza Saiful[4].

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia.
[2] Petronas Technical Services Sdn. Bhd., Kompleks Bersepadu Pengerang (PIC), 81600 Pengerang, Johor, Malaysia.
[3] Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia.
[4] Mechanical Section, Engineering Department, Group Technical Solutions, Project Delivery & Technology (PD&T), Petroliam Nasional Berhad (PETRONAS), Level 15, PETRONAS Tower 3, Kuala Lumpur City Centre, 50088 Kuala Lumpur, Malaysia.

ARTICLE INFO

Article history:
Received 29 November 2024
Received in revised form 03 December 2024
Accepted 06 December 2024
Available online 29 December 2024

Keywords:
Mesh convergence study; joining pipe; standard pad; thick wall pad; long strip pad; short strip pad; thermal profiles; thermal expansion.

ABSTRACT

A mesh convergence study in finite element analysis was conducted using various mesh sizes to determine if the result varied. This study aims to obtain accurate data before comparing it with analytical results. A joining pipe with different types of ceramic pads, such as standard pad, thick wall pad, long strip pad, and short strip pad was modeled using Solidworks software. The thermal profiles and thermal expansion of the pipe were analyzed using various converged mesh sizes and mesh modes. A temperature of 650 ℃ was applied to the ceramic pad, as it acts as an insulator that spreads heat across the pipe. It was found that mesh number 10 (size: 0.05) was selected for the standard pad, and mesh number 6 (size: 0.13) for the thick wall pad, and the data had already converged, making the mesh size accurate and reliable with relative errors of 0.019% and 2.41%, respectively. For the long strip pad and short strip pad, mesh number 13 (size: 0.02), and mesh number 9 (size: 0.06) were selected as the most optimal meshes, providing a converged solution with minimal relative errors of 0.06% and 0.02%, respectively.

1. Introduction

PWHT is a process that is applied after welding and involves heating a welded material to a temperature below its critical transformation temperature and holding it for a predetermined period of time [1]. This process can be classified into three forms: annealing, normalizing, and quenching [2].

i. Annealing – used to soften the metal for forming or machining
ii. Normalizing – used to provide uniformity in particle size
iii. Quenching – used to harden iron-based metal

* *Corresponding author.*
E-mail address: elizay@uthm.edu.my

https://doi.org/10.37934/IJMSET.2.1.2132

Metal pipes and tubes are widely used in different fields of engineering such as automotive, chemical, nuclear, and petroleum industries. Thus, having a thorough understanding of their mechanical properties is crucial for both manufacturing and application. Additionally, it is crucial to have accurate models of material behavior including deformation, anisotropy, and failure for reliable numerical modelling of forming process [3].

Carbon steel (CS) alloy, grade A106 grade B seamless pipe is typically utilized in power plants, boilers, petrochemical plants, oil and gas refineries and ships to transport fluid and gases that are subjected to high pressures and temperatures. In petroleum refineries, steel pipelines are used to transport gas, oil and their byproducts from their production area to the local market or for export [4].

A numerical Finite Element Analysis (FEA) model was created using the ANSYS/Mechanical FEA software [5] to study the edge effect of a legacy host pipe with a circumferential crack on the PIP repair system. A mesh convergence study was conducted on the PIP system to compare the accuracy of the numerical results with the analytical solution [6].

A team of researchers led by Prayoga *et al.,* [7] conducted a study to determine the appropriate mesh for further research on ocean thermal energy conversion (OTEC). They carried out a mesh convergence study using a step buckle in ABAQUS software. The study found that as the number of elements increased, the critical moment decreased, and the results obtained became more convergent.

In this study, a mesh convergence study will be explained thoroughly by using ANSYS Transient Thermal. Mesh convergence study was conducted to obtain the accurate data before comparing it with analytical result. CS alloy grade A106 grade B was designed by using Solidworks software before being imported to the ANSYS Transient Thermal. The temperature distribution can be obtained from the ANSYS Transient Thermal, while the thermal expansion can be obtained from ANSYS Transient Structural.

2. Methodology

Simulation is the process of replicating a real-world system, process, or phenomenon through a model or computer program. It involves creating a simplified representation of the actual system and running simulations to understand its performance, behavior, or outcomes under various conditions. Simulations are used to test hypotheses, predict outcomes, optimize processes, and gain insights into complex systems that may be difficult or costly to study directly [8].

In this study, the simulations were carried out at Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, through registered Ansys software R1 2023. Ansys Transient Thermal was used to determine the thermal profiles and Ansys Transient Structural was used to determine the thermal expansion of the pipe. The Ansys software R1 2023 simulated the thermal profiles at the welding area of the pipe where PWHT process began. The design stage of the pipe used in this research was completed before the simulation work began. The pipe was designed by using Solidworks software before being imported to Ansys software R1 2023.

For instance, simulation of PWHT on CS pipe material A106 Grade B is conducted by using Ansys Transient Thermal to identify the thermal profiles of the pipe. The geometry model of the CS pipe and ceramic pad (CP) was designed using Solidworks software. The 95% alumina CP will function as an insulator to heat up the CS pipe. Next, the geometry model was imported into Ansys Transient Thermal for simulation investigation. All of the data needed before beginning the simulation. Table 1 shows the mechanical properties of CS pipe material A106 Grade B. Table 2 shows the mechanical properties of 95% alumina CP.

Table 1
The mechanical properties of CS pipe A106 grade B

Property	Value
Density (kg m^{-3})	7850
Young Modulus (GPa)	210
Poisson's Ratio	0.3
Bulk Modulus (Pa)	1.75×10^{11}
Shear Modulus (Pa)	8.0769×10^{10}
Tensile Ultimate Strength (Pa)	415
Isotropic Thermal Conductivity ($Wm^{-1}C^{-1}$)	24.3
Specific Heat Constant Pressure, C_p ($Jkg^{-1}C^{-1}$)	450

Table 2
The mechanical properties of 95% Alumina CP

Property	Value
Density (kg m^{-3})	3650
Young Modulus (GPa)	3.2×10^{-7}
Poisson's Ratio	0.23
Bulk Modulus (Pa)	197.53
Shear Modulus (Pa)	130.08
Isotropic Thermal Conductivity ($Wm^{-1}C^{-1}$)	27
Specific Heat Constant Pressure, C_p ($Jkg^{-1}C^{-1}$)	780

2.1 Geometry Model of A106 Grade B

The pipe size notation is furnished in Figure 1, where L is the length of the pipe, t is the thickness of the pipe, O_D the outside diameter, and I_D the inside diameter. The pipe size notation describes the shape and size of each geometry part in the FE model. The parameters of geometry model, temperature, and ambient temperature are presented in Table 3. There are four types of CP used in this study such as standard pad, thick wall pad, long strip pad, and short strip pad. All of the pads are used to wrap the welding area and spread the heat across the pipe. Table 4 shows the parameters of CP that are used in this study.

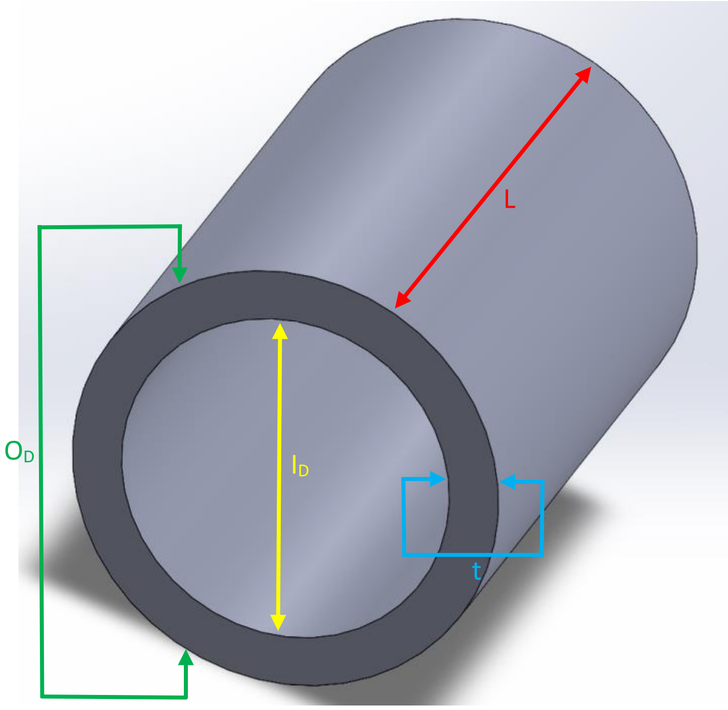

Fig. 1 Pipe size notation

Table 3
The parameters of A106 grade B CS pipe

Object	Joining Pipe
Outer Diameter, O_D (mm)	450
Inner Diameter, I_D (mm)	411.9
Thickness, t (mm)	19.05
Length, l (mm)	3000
Temperature, T (℃)	650
Ambient Temperature (℃)	34

Table 4
The parameters of the CP

	Width (mm)	Length (mm)	Thickness (mm)
Standard Pad	305	165	15
Thick Wall Pad	150	330	15
Long Strip Pad	610	85	15
Short Strip Pad	255	85	15

2.2 Transient Thermal Analysis in ANSYS

According to ANSYS Innovation Course [9], transient thermal analysis is used to determine temperatures, heat flux, and directional heat flux over time. This method is different from steady-state thermal analysis as it considers the significance of time, making it more similar to the explicit method in structural FEA. The analysis of the CS pipe was done to identify the temperature distribution of the A106 grade B CS pipe after being heated at temperature of 650 ℃. The simulation can determine the temperature reduction and thermal expansion along the CS pipe. A mesh convergence study was carried out to ensure that element sizes are adequate enough to yield an FEA solution that is accurate to the desired level [10].

The geometry model of A106 grade B CS pipe was designed as shown in Figure 2. The pipe was designed using Solidworks software according to the parameters provided by the industry. CP wrapped the weld area at the joint between elbow pipe and straight pipe. While the ceramic fiber blanket was used to wrap the CP to avoid excessive heat release during the heating period.

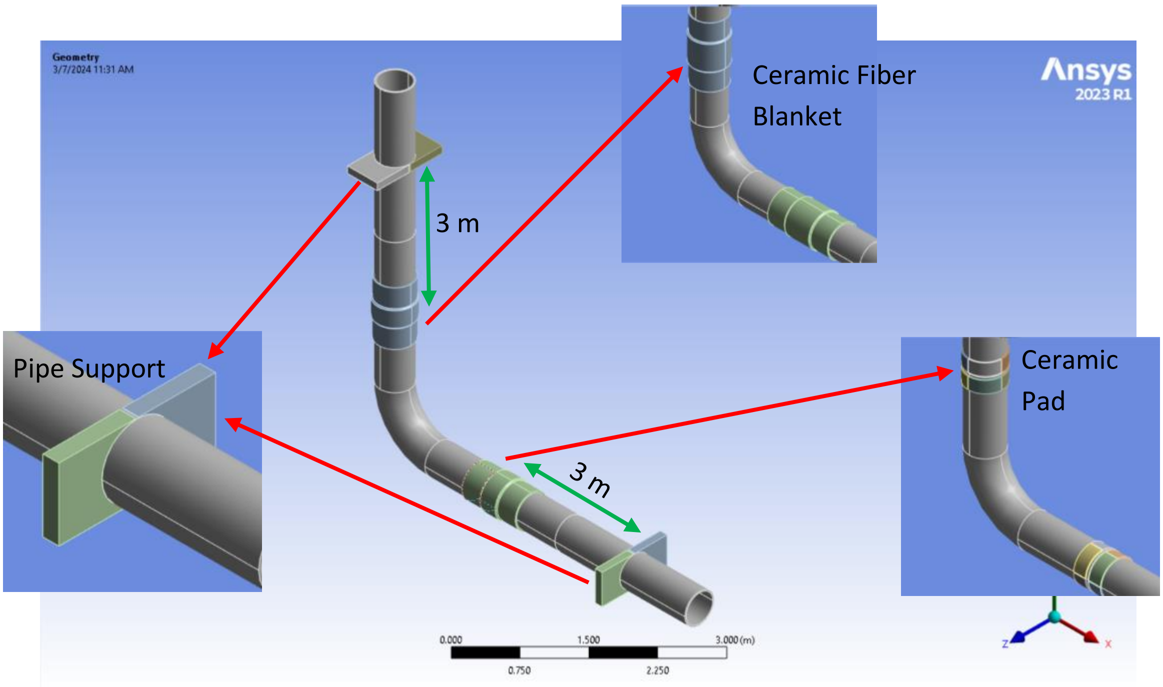

Fig. 2 The geometry model of A106 grade B CS pipe

2.3 The steps of mesh convergence

The Finite Element Method (FEM) is a numerical technique that is widely used in engineering to solve a variety problem. The FEM divides complex geometries into small pieces called elements, using a mesh process. The common points shared by multiple elements are called nodes. There are many types of elements used in the FEM, including 1D element like linear elements (2 nodes), quadratic elements (3 nodes), and cubic elements (4 nodes). 2D elements include rectangular elements (4 nodes), quadratic quadrilateral elements (8 nodes), linear triangular elements (3 nodes), and quadratic triangular elements (6 nodes). 3D elements include 4-node tetrahedral elements, 10-node tetrahedral elements, 8-node brick elements, and 20-node brick elements [11]. In this study, the quadratic elements are selected because it retains mid-side nodes on elements created in the part or body. This means that all parts of the body will have mid-side nodes.

According to Asim Rashid *et al.* [10], mesh convergence study is performed to make sure that element sizes are sufficient enough such that the solution obtained using finite element analysis is accurate to a desired level. There are two steps to create the mesh, either create the nodes and elements manually or use the program's built-in meshing capabilities to mesh the solid model geometry [12]. In this subtopic, the steps to specify the attributes of the elements to be generated and how to set the mesh controlled will be discussed in this subtopic.

Firstly, elements attributes such as element type, material type, and real constant are assigned to the solid model geometry. Next, set the mesh controls. Iteration method is used to identify the best element size to obtain accurate data. These mesh controls dictate the element shape to use, the size of the elements, the type of mesh to generate, etc [12]. Once the tasks have been filled, the model can be generated. Repeat the same process by using iteration method until relative error is less than 5%. The steps of the iteration methods will be shown below. Table 5 shows the number of nodes and elements of standard pad. Table 6 shows the number of nodes and elements of thick wall pad. Table 7 shows the number of nodes and elements of long strip pad. And Table 8 shows the number of nodes and elements of short strip pad. All numbers of nodes and elements shown below are already convergent.

Here are the steps of the iteration methods to obtain the accurate data:

i. Start iteration with number of element size of 0.05. Use the formula in Eq. (1) to get the next value of element size:

$$\frac{Number\ of\ sizes}{h_m} > 1.3 \tag{1}$$

where number of sizes = element size.

ii. Generate the mesh after filling up the required data. Write down the value of elements and nodes
iii. Calculate the number of element size. Make sure the value of r > 30%
iv. Take the value of the temperature according to the appropriate range
v. Compute the value of relative error based on the formula in Eq. (2):

$$Relative\ error = \left[\frac{\emptyset_1 - \emptyset_0}{\emptyset_0}\right] \times 100\% \tag{2}$$

where $\emptyset_1$ = estimated value, and $\emptyset_0$ = actual value

vi. Repeat the same process to procure the accurate data

Table 5
The number of nodes and element of standard pad

	Nodes	Elements
Coarse	188241	98792
Medium	209846	102896
Fine	483223	265206

Table 6
The number of nodes and element of thick wall pad

	Nodes	Elements
Coarse	14512	50396
Medium	18580	57492
Fine	27094	79531

Table 7
The number of nodes and element of long strip pad

	Nodes	Elements
Coarse	635340	313296
Medium	1278601	678650
Fine	2201724	1197555

Table 8
The number of nodes and element of short strip pad

	Nodes	Elements
Coarse	156646	85645
Medium	81313	40458
Fine	283929	167978

2.4 Boundary Condition (BC)

Boundary condition plays a crucial role in determining how heat is transferred at the boundaries of a system over time. They define the temperature, heat flux, convection, or radiation at the surfaces or edges of the model. Accurately simulating transient thermal behaviour requires the correct specification of boundary conditions since they govern heat exchange between the system and its surroundings, affecting the temperature distribution and thermal responses during the simulation period [13]. In this study, the boundary condition refers only to the temperature, which is set at 650 °C to heat the pipe. Figure 3 illustrates the boundary condition (temperature) of the pipe.

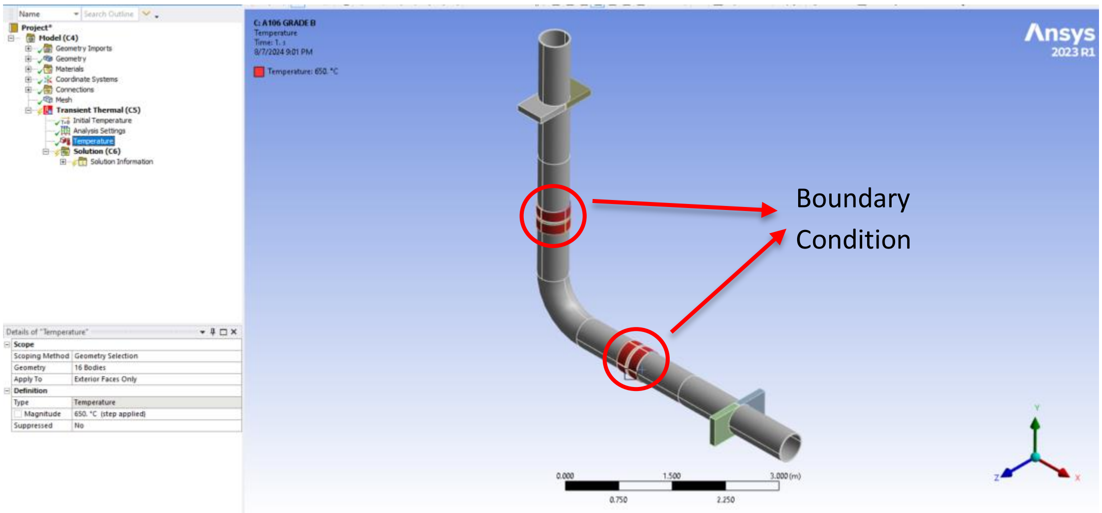

Fig. 3 The boundary condition of the pipe

3. Results

In general, finer meshes produce more accurate results. As the number of lines increases, so does the number of nodes. A comprehensive method is to conduct a mesh convergence study to ensure an adequate mesh which is thoroughly discussed in this section [14]. In this study, temperature is applied at the ceramic pad that is shown in Figure 3. The temperature applied is 650 ℃ at four different type of ceramic pads which are standard pad, thick wall pad, long strip pad, and short strip pad.

When conducting a mesh convergence study, it is necessary to assess the results obtained from simulations run on different mesh sizes. The objective is to determine the minimum number of solutions needed to achieve accurate results. If increasing the number of mesh elements leads to an improved solution, then it can infer that a finer mesh is necessary for accurate results. However, if the solution does not show any significant improvement, it can be concluded that the current mesh is adequate and further refinement is not required. While opting for a finer mesh may increase the accuracy of the results, it may also lead to a longer computation time. Hence, it is essential to strike a balance between accuracy and computational efficiency while choosing the number of solutions [14]. Figure 4 shows the graph of mesh convergence study for standard pad.

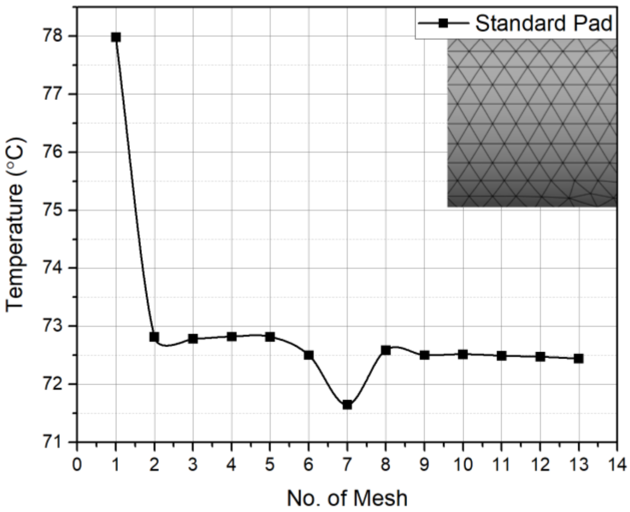

Fig. 4 The graph of mesh convergence study for standard pad

A comprehensive mesh convergence study was conducted to determine the optimal mesh size for thermal simulation accuracy. Initial simulations with a mesh size of 0.5 yielded a temperature of 77.987 ℃ serving as a reference point without a specified relative error. Through systematic mesh refinement, the solution demonstrated progressive convergence with significant improvements observed at mesh size of 0.29 (72.748 ℃, 0.045%) and 0.17 (72.818 ℃, 0.0027% error). This study identified an optimal mesh size 0.05, which achieved an excellent balance between computational efficiency and accuracy, producing a temperature of 72.517 ℃ with a minimal relative error of 0.019%. Further refinement to mesh size of 0.04 and 0.03 yielded comparable result (72.491 ℃ and 72.473 ℃, respectively) with similarly low relative error (0.036% and 0.025%) confirming solution stability. This systematic approach to mesh optimization ensures reliable thermal analysis results while maintaining computational feasibility, particularly crucial for PWHT applications in metallic pipe system. Figure 5 shows the graph of mesh convergence study for thick wall pad.

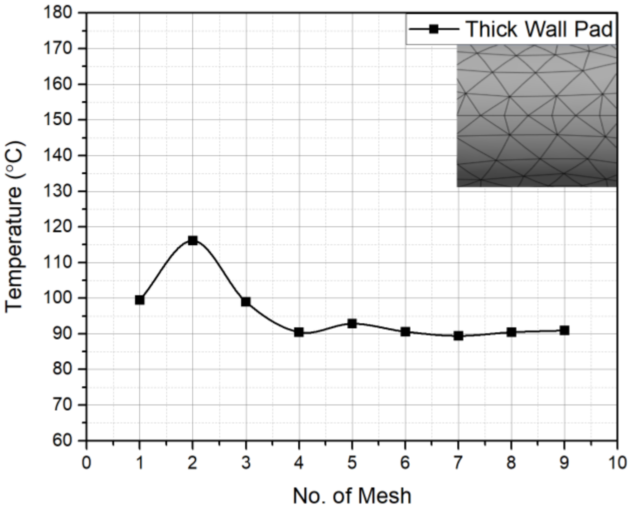

Fig. 5 The graph of mesh convergence study for thick wall pad

In Figure 5, a graph of mesh convergence study for thick wall pad is displayed. Initial simulations with coarse mesh sizes (0.5 to 0.22) exhibited significant temperature instability with values ranging from 99.48 ℃ to 116.14 ℃ and high relative errors (up to 16.75%). A notable improvement in stability occurred at mesh size of 0.17 where temperature stabilized around 92.83 ℃ with a substantially reduced relative error of 2.63%. The study identified an optimal mesh size of 0.13 corresponding to 79531 elements which produced a temperature of 90.59 ℃ with a relative error of 2.41%. This configuration represented the ideal balance between computational efficiency and solution accuracy. Further mesh refinement to size of 0.08 and 0.06 yielded temperature of 90.42 ℃ and 90.44 ℃, respectively, with the latter showing a minimum relative error of 0.022%, confirming solution convergence. Figure 6 shows the graph of mesh convergence study for long strip pad.

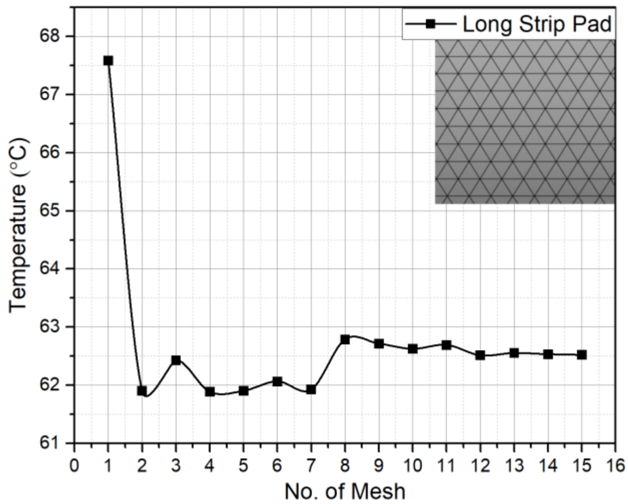

Fig. 6 The graph of mesh convergence study for long strip pad

In Figure 6, initial simulations with coarse mesh sizes (0.5 to 0.22) demonstrated significant temperature variations with values ranging from 67.588 ℃ to 61.901 ℃, indicating solution instability. A notable improvement in result consistency occurred at mesh size of 0.17 where temperature stabilized between 61.902 ℃ and 62.065 ℃. Progressive mesh refinement revealed steady convergence of the solution. At mesh size of 0.08, the temperature reached 62.786 ℃ with a relative error of 1.39%, while further refinement to 0.05 yielded 62.626 ℃ with a reduced error of 0.14%. The study identified an optimal mesh size of 0.02 which achieved excellent accuracy with a minimal relative error of 0.03%. Subsequent refinement to mesh size between 0.015 and 0.012 produced consistent temperatures ranging from 62.514 ℃ to 62.532 ℃, confirming solution convergence. The finest mesh size of 0.012 yielded a temperature of 62.522 ℃ with a 0.02% relative error validating the stability of the solution. Figure 7 shows the graph of mesh convergence study for short strip pad.

Lastly, Figure 7 shows the graph of mesh convergence study for short strip pad. Initial simulation with a coarse mesh (size: 0.05, 13183 nodes, and 5813 elements) produced a temperature of 67.806 ℃. Refinement to 0.38 (167286 nodes, 93011 elements) resulted in a significant temperature reduction to 62.077 ℃ with 8.45% relative error, demonstrating the substantial impact of initial mesh refinement. Further optimization to 0.29 (246806 nodes, 151553 elements) yielded 61.704 ℃ with markedly improved 0.60% relative error. The study identified an optimal mesh size of 0.06 (283793 nodes, 167995 elements) which achieved a temperature of 63.025 ℃ with a minimal relative error of 0.02%. While further refinement to 0.03 (467013 nodes, 258374 elements) yielded a temperature of 62.965 ℃ with 0.09% error. This optimization demonstrates that mesh size of 0.06 provides the ideal balance between simulation accuracy and computational efficiency for PWHT analysis of short strip pad configurations.

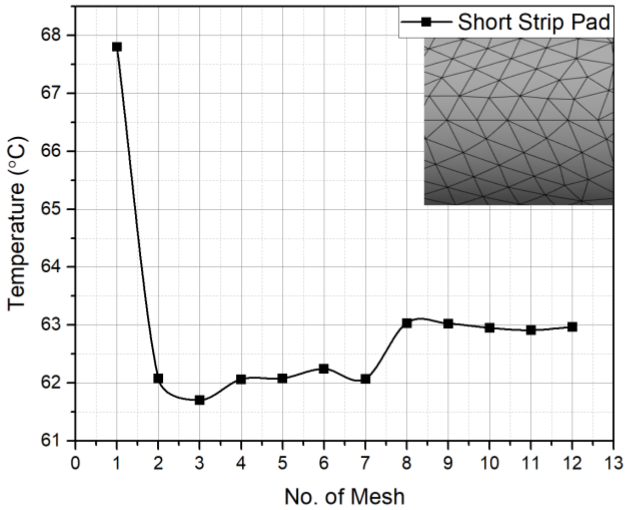

Fig. 7 The graph of mesh convergence study for short strip pad

After examining Figures 4, 5, 6, and 7, it can be concluded that the data become more converged after several meshes. When the area size increases, the mesh convergence study takes longer time to penetrate [15]. Despite the meshing taking a longer time, the simulation produced accurate data, making it a reliable technique to be used in natural environments.

4. Conclusions

In conclusion, it is imperative for researchers to perform mesh convergence studies to determine the optimal mesh size for creating the FEA model since an incorrect meshing size can lead to significant variations in the output. A mesh convergence study verifies that the FEA model has converged into a solution. The final results of the simulation are more accurate and reliable when conducting the mesh convergence study, as it helps obtain the correct data at the end. Performing the simulation allows for completing tasks that are not feasible in the real world, thereby saving both time and cost.

Acknowledgment
The authors would like to thank Universiti Tun Hussein Onn Malaysia (UTHM) for supporting this research through a Postgraduate Research Grant (Vot. No. Q603). Also, thanks to the Faculty of Mechanical and Manufacturing Engineering (FKMP) for providing the facility. The appreciation is also dedicated to Petroliam Nasional Berhad (PETRONAS) for the guidance of expertise in the project.

References

[1] K. Sotoodeh, "Corrosion study and material selection for cryogenic valves in an LNG plant," in *Cryogenic Valves for Liquefied Natural Gas Plants*, Elsevier, 2022, pp. 175–211. doi: 10.1016/B978-0-323-99584-9.00001-X.
[2] M. Stewart, *Surface Production Operations*, vol. 3. Elsevier, 2016. doi: 10.1016/C2009-0-20127-9.
[3] A. Khalfallah, Z. Ktari, C. Leitão, and J. V. Fernandes, "New Mandrel Design for Ring Hoop Tensile Testing," *Exp Tech*, vol. 45, no. 6, pp. 769–787, Dec. 2021, doi: 10.1007/s40799-021-00462-4.
[4] J. Thermal and S. Technol, *Corrosion: Fundamentals, Testing, and Protection*. ASM International, 2003. doi: 10.31399/asm.hb.v13a.9781627081825.

[5] Z. Zhang, "Application of finite element analysis in structural analysis and computer simulation," *Applied Mathematics and Nonlinear Sciences*, vol. 0, no. 0, May 2023, doi: 10.2478/amns.2023.1.00273.

[6] C. M. T. Tien *et al.*, "Effects of the legacy pipe ends on the behaviour of pipe-in-pipe repair systems under internal pressure," *Eng Fail Anal*, vol. 144, p. 106957, Feb. 2023, doi: 10.1016/j.engfailanal.2022.106957.

[7] P. W. Adie *et al.*, "Non-linear assessment of cold water pipe (CWP) on the ocean thermal energy conversion (OTEC) installation under bending load," *Procedia Structural Integrity*, vol. 47, pp. 142–149, 2023, doi: 10.1016/j.prostr.2023.07.005.

[8] X. Chen and Y. Liu, *Finite Element Modeling and Simulation with ANSYS Workbench* , 2nd ed. CRC Press, 2019.

[9] Ansys Innovation Courses, "Intro to Transient Thermal Analysis - Lesson 1," Ansys Innovation Courses .

[10] A. Rashid, "Mesh Convergence: The modern way," LinkedIn.

[11] S. Z. Jia, J. S. Gunasekera, and J. Glancey, "Computer integrated sustainable manufacturing," in *Sustainable Manufacturing Processes*, Elsevier, 2023, pp. 313–334. doi: 10.1016/B978-0-323-99990-8.00001-1.

[12] A. Rashid, "How to estimate mesh convergence just by looking at stress contour plot," LinkedIn.

[13] G. P. Gujarathi and Y.-S. Ma, "Parametric CAD/CAE integration using a common data model," *J Manuf Syst*, vol. 30, no. 3, pp. 118–132, 2011.

[14] H. Azahari, S. Saiful, F. Mansor, M. N. H. Mat, and E. M.Yusup, "A Mesh Convergence Study for 2D Axisymmetric Pipe Wall Thickness," pp. 1–14, 2023.

[15] C. Devals, T. Vu, Y. Zhang, J. Dompierre, and F. Gubalt, "Mesh convergence study for hydraulic turbine draft-tube," *IOP Conf. Series: Earth and Environmental Science*, pp. 1–10, 2016.

[16] M. Ahmad, K. A. Ismail, and F. Mat, "Convergence of Finite Element Model for Crushing of a Conical Thin-walled Tube," *Procedia Eng*, vol. 53, pp. 586–593, 2013, doi: 10.1016/j.proeng.2013.02.075.

[17] H. Patil and P. Jeyakarthikeyan, "Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS," *IOP Conf. Series: Materials Science and Engineering*, pp. 1–11, 2018.