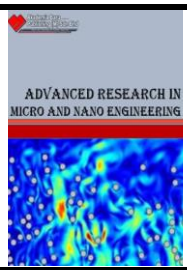
Penerbit Akademia Baru

Journal of Advanced Research in Micro and Nano Engineering

Journal homepage: https://www.akademiabaru.com/submit/index.php/armne/index
ISSN: 2756-8210



A Mesh Convergence Study for 2D Axisymmetric Pipe Wall Thickness

Eliza M. Yusup[1,*], Haqeemy Azahari[2], Shahzulreza Saiful[3], Faisal Mansor[4], Mohamad Nur Hidayat Mat[5], Balasem Abdulameer Jabbar[6]

1 Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia
2 Manufacturing Division, Pengerang Refining Company Sdn. Bhd., Kompleks Bersepadu Pengerang, 81600 Pengerang, Johor, Malaysia
3 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
4 Planning & Corporate Strategy, Group Technical Solutions, Project Delivery & Technology (PD&T), Petroliam Nasional Berhad (PETRONAS), Level 54, PETRONAS Tower 2, Kuala Lumpur City Centre, 50088 Kuala Lumpur, Malaysia
5 Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
6 Engineering Technical College-Najaf, Al-Furat Al-Awsat Technical University, 3200 Najaf, Iraq

ARTICLE INFO

Article history:
Received 3 June 2024
Received in revised form 11 July 2024
Accepted 19 August 2024
Available online 30 September 2024

Keywords:
Numerical; equivalent stress; mesh convergence study; 2D axisymmetric

ABSTRACT

A meshing size study is essential for any simulation, as it affects the accuracy of the results by approximating the real-world geometry. This paper presents a detailed mesh convergence study for a 2D axisymmetric pipe model. The model incorporates a one-quarter cross-section due to the axisymmetric nature of the pipe. Four pipe wall thicknesses were investigated: 3.40 mm, 7.11 mm, 10.97 mm, and 18.26 mm. The results revealed a significant advantage for simulations involving thin pipes. They achieved convergence, a state with stable solution values, by utilizing a less dense mesh, leading to reduced computational time. This trend was exemplified by the fact that a thinner pipe with thickness, t = 3.40 mm has converged with only 8 mesh elements, whereas a significantly thicker pipe, t = 18.26 mm necessitated 14 elements for convergence. This suggests that more complex geometries, with intricate details, may require a larger and denser mesh for convergence, leading to increased computational time. In conclusion, the mesh convergence study confirms that the finite element analysis (FEA) model has achieved a converged solution.

1. Introduction

In the realm of engineering, pipings are ubiquitous components playing crucial roles in diverse applications like transporting fluids, withstanding pressure, and providing structural support. Accurately predicting their behaviour is paramount to ensure safety and functionality in these systems. Finite Element Analysis (FEA) has emerged as a powerful tool for analysing piping behaviour [1,2]. It allows the user to discretize the pipe geometry into smaller elements, enabling detailed stress and strain calculations. However, the size of these elements, referred to as mesh size, significantly impacts the accuracy and efficiency of FEA simulations.

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

https://doi.org/10.37934/armne.23.1.7790

This study delves deeper into this challenge by focusing on a specific aspect – the impact of pipe wall thickness on optimal mesh size. The employment of a 2D axisymmetric pipe model, which leverages the symmetry of the pipe to analyse just a quarter section, thereby reducing computational cost [3]. By varying the pipe wall thickness and performing a mesh convergence study, the aim is to establish a relationship between these factors. This relationship will ultimately help users determine the optimal mesh size for their specific pipe simulations. Achieving a balance between accuracy and computational efficiency is crucial, and this study aims to provide valuable insights for users working with pipes of varying wall thicknesses.

The design and construction of this piping system adhere to the requirements of ASME B31.3 for safe and reliable operation. To ensure structural integrity, the minimum wall thickness of these piping system pipes was determined according to ASME B31.3, considering factors like design pressure, material properties, and corrosion allowance. It is important to note that the minimum wall thickness specified by ASME codes is a requirement for safe and reliable operation. To determine the exact wall thickness for a specific piping system, it is necessary to consult the relevant ASME codes and perform a detailed engineering analysis [1]. The simulation was conducted to investigate the maximum stress the pipe could withstand during hydrostatic pressure prior to its operation.

Meshing is an integral part of simulation work in any Finite Element Analysis (FEA) method [2]. Whether the simulation results were independent of the underlying mesh or not, a few simulations with identical parameters were run a few times with different mesh resolutions to check if the results changed [3]. The repetitive simulation is known as the Mesh Convergence Study in Static Structural Analysis. For other studies, previously, the 3D hybrid mesh was used for the solid element for all wing components and models where it shows that the mesh independent study (for static structural analysis) was achieved with an optimized grid at 116,796 elements, as shown in Figure 1 [4]. The meshing process includes several important decisions such as choosing the types of elements, element size, and element orientation. The process should be carefully optimised to ensure accurate and efficient simulations.

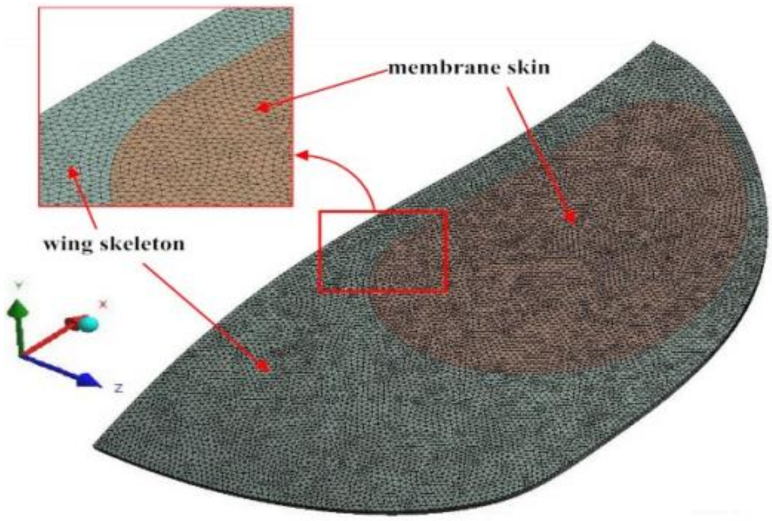


Fig. 1. The optimized mesh for solid component of wash out TM wing (half-wing view) [4]

Meanwhile, in other people's work, overset meshes, unlike other mesh methods, do not need mesh deformation or remeshing, as is done in this research. The numerical solution for this class of problems using overset meshes involves creating two meshes: one fixed background mesh and one moving mesh fitted to the moving body [5]. The domain connectivity is obtained through proper interpolation in the overlapping areas. As shown in Figure 2, these two overlapping meshes are used.

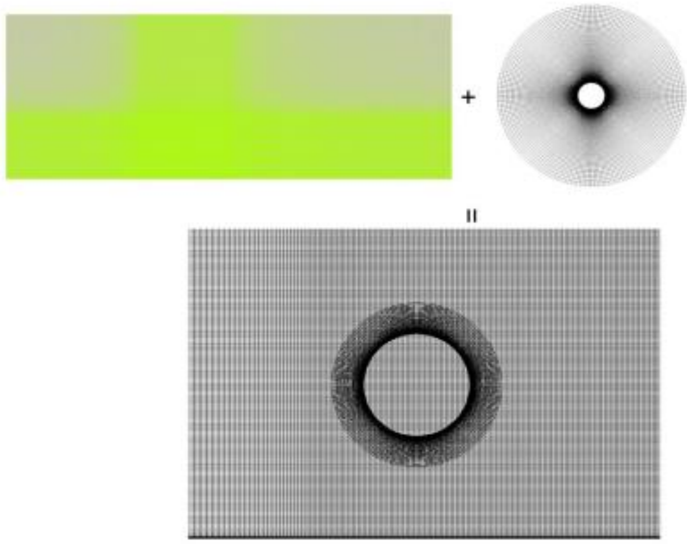


Fig. 2. Schematic diagram of overset mesh: one fixed background mesh and one moving mesh fitted to the moving body [5]

In Computational Fluid Dynamics (CFD) simulation, Darmawan, Raynaldo, and Halim conducted a comprehensive study in 2022, delving into the realm of thruster design for Remotely Operated Vehicles (ROVs) by employing a systematic approach, which involved the application of both general mesh and unstructured mesh arrangements. Their primary objective was to ascertain the optimal thrust parameters. By utilizing these varied mesh configurations, they meticulously examined and compared the fluid dynamics within the thruster system. The findings, as presented in their research, offered valuable insights into the efficiency and performance of different thruster designs, crucial for enhancing the manoeuvrability and operational capabilities of ROVs [6]. For a visual representation of their analyses and results, refer to Figure 3, which provides illuminating illustrations depicting the flow patterns and pressure distributions within the examined thruster configurations.

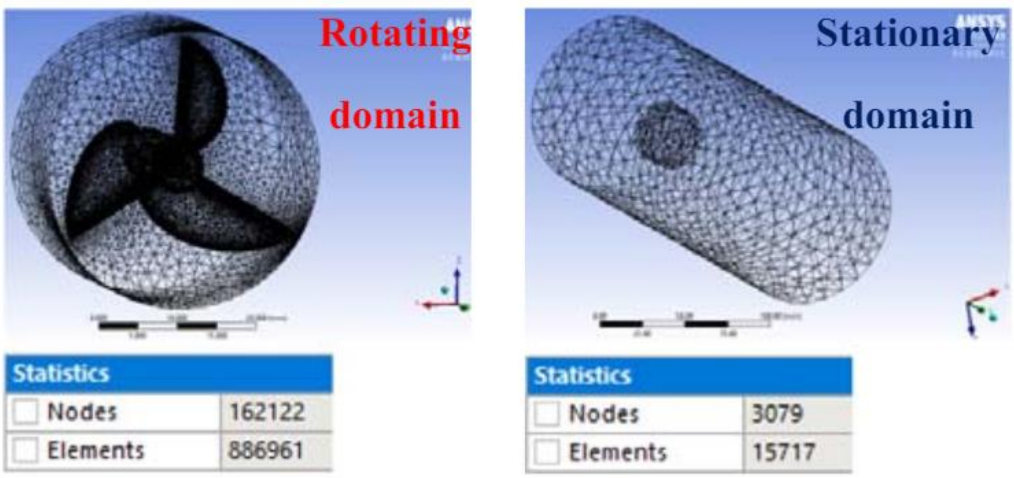


Fig. 3. Computational mesh of part P3-2020 [6]

In this study, the axisymmetric 2D refers to a type of two-dimensional (2D) analysis where the geometry of the model is symmetrical around an axis. In this type of analysis, only one plane (circumferential) is considered, and the results are obtained by assuming rotational symmetry around the axis of the symmetry. An axisymmetric 2D analysis is commonly used in the analysis of cylindrical geometries, such as pipes, pressure vessels, and nozzles [7,8]. This analysis simplifies the geometry and reduces the number of degrees of freedom, making it easier and more efficient to perform the analysis. The results of axisymmetric 2D analysis can provide valuable insights into the behaviour and performance of cylindrical structures, including stresses, strains and deformations. However, it should be noted that the results may not be fully representative of the actual three-dimensional (3D) behaviour of the structure, and in some cases, a 3D analysis may be required [9].

In previous work, which used the design modeler in ANSYS FLUENT, the assembly was grouped into two zones, namely, the dynamic zone and the static zone as shown in Figure 4 [10]. This simulation employed two distinct meshing techniques. In the static zone, where geometry is simpler, a multizone approach was used. This method combines hexahedral elements (hexa-core mesh) for efficient computation and tetrahedral elements (free mesh) for flexibility. For the dynamic zone, featuring the complex train geometry, a patch conforming mesh was chosen to ensure a good fit. To optimize the number of cells used and achieve efficiency, an adaptive sizing technique was applied throughout the mesh.

The same study also employed two approaches to achieve optimal mesh quality. The first method involved a mesh independence test, evaluating three different mesh densities: 9.6 million, 14 million, and 16.5 million elements. This test aimed to identify the mesh density that yielded consistent results. In contrast, the second method focused on minimizing skewness, a measure of mesh element distortion. Since skewness can significantly impact simulation accuracy, reducing it was crucial.

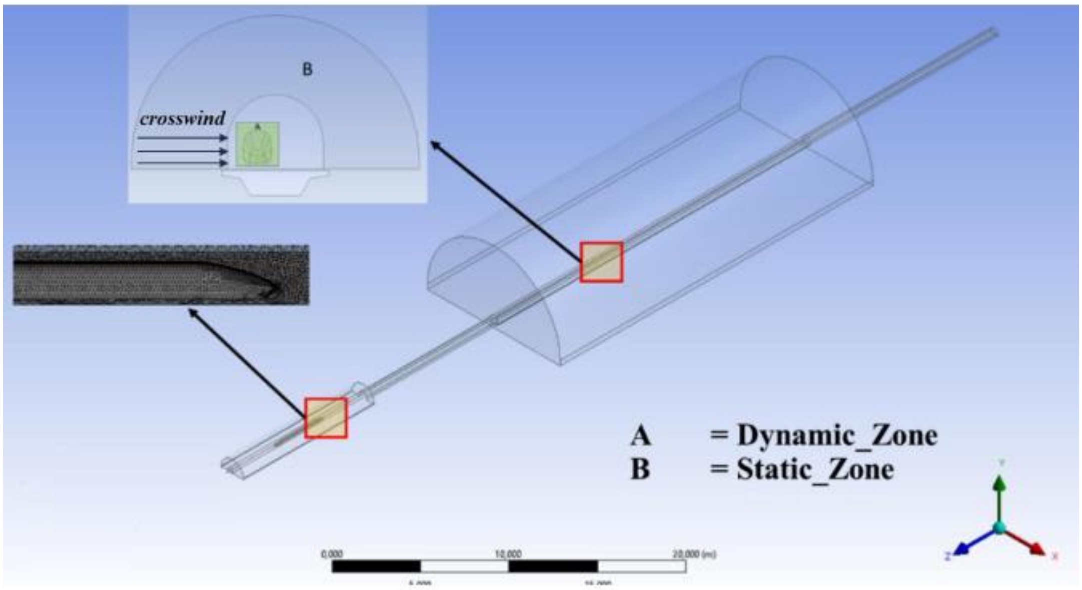


Fig. 4. Schematic of simulation model [10]

2. Methodology

Simulations were carried out at the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Malaysia, through registered Ansys software R1 2022. The simulation's domain used for this research was Static Structural, which determined the displacements, stresses, strains, and forces in structures or components caused by static loads of the pipe [11]. In this study, the Ansys Workbench R12022 Student Edition simulated the stress test at a certain thickness of the pipe. The simulation was then analysed to obtain results. Prior to the simulation work, the design stage of the pipe used in this research was done. The pipe design was based on the existing size widely used in the Oil and Gas industry. In this study, geometry was designed in the Ansys Modeler software and can directly open in Ansys Workbench without converting any file format. A 6 inches pipe size with an outside diameter of 168.3 mm was chosen for this simulation because the size is commonly used in industry [12]. Since the pipe is axisymmetric, a 2D design quarter of the pipe size is the preferred design for this simulation. Ko and his co-workers used the same approach where they also used a quarter of the domain since the pipe was in symmetry of typical modelling [10]. Hence, it can save more time for the simulation to complete.

2.1 Parameters

The chosen parameters were their thicknesses; *t* = 3.40 mm, *t* =7.11 mm, *t* = 10.97 mm, and *t* = 18.26 mm, as various pipe sizes according to ASME B36.19 provided by industry [13]. The thickness of the pipe can be determined by referring to the pipe's schedule number. The thickness was extracted from Schedule 10S, 40S, 80S, and 160 from the ASME B36.19 [13]. Figure 4 shows an example of a 6 inches 2D pipe quarter design. Table 1 shows the example of schedule number and thickness even though the pipe size is the same.

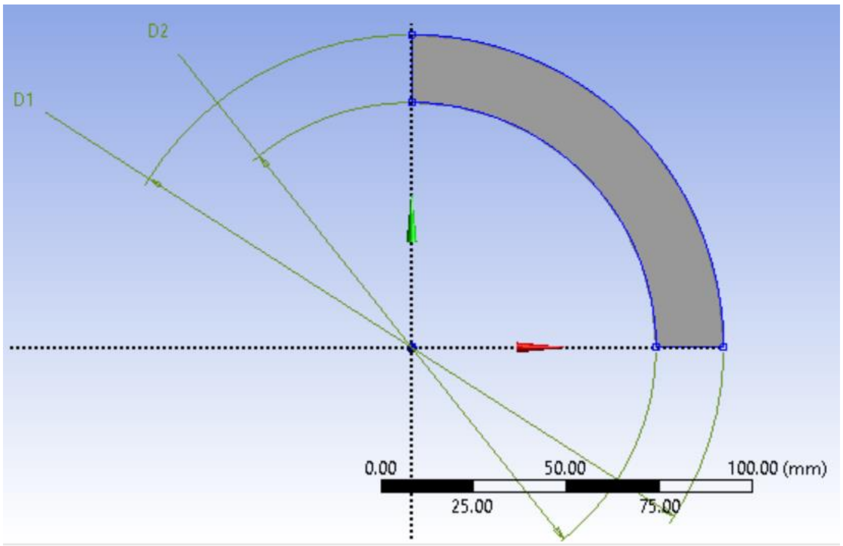


Fig. 4. Two-dimensional design quarter for the pipe of 6-inch

Table 1
6-inch pipe with various schedule numbers and thickness

Pipe size (inch)	OD (mm)	Schedule number, SCH	Thickness, t (mm)
6	168.3	10S	3.40
		40S	7.11
		80S	10.97
		160	18.26

2.2 Boundary Conditions

In Ansys, displacement and fixed support are the boundary conditions that can be applied to a finite element model to define how the structure is supported. A displacement boundary condition is applied to a specific degree of freedom of a node in the model, and it defines the prescribed displacement of that degree of freedom. This type of boundary condition is used to simulate a load that is applied directly to the structure, such as a force or a displacement load. On the other hand, a fixed support boundary condition is applied to all degrees of freedom of a node in the model, and it defines that the node is fully restrained and cannot move in any direction. This boundary condition is used to simulate a support that restrains the structure from moving in any direction, such as a roller support or a pinned support. When defining boundary conditions in Ansys, it is important to choose the correct type of boundary condition that corresponds to the type of support or load that is being modelled [2,14-17]. The choice of boundary conditions can significantly affect the results of the analysis, so it is important to carefully consider the type of boundary conditions that are being applied to the model. In a research paper back in the year 2022, the researchers defined their boundary conditions and simplified them in one table as shown herein in Table 2 [18].

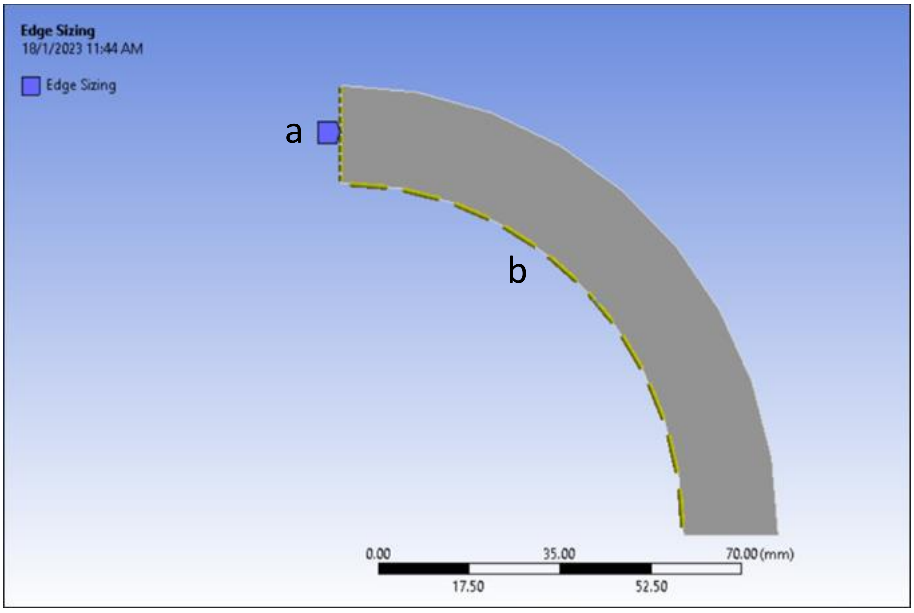
Table 2
Boundary conditions [18]

Description	Values
Inlet velocity of the air	18 m/s
Heat transfer coefficient	3570 M/m2
Pressure outlet of the air	1 atm
Inlet temperature of the air	30 °C
Ambient temperature	20 °C
Viscous model	k-epsilon model

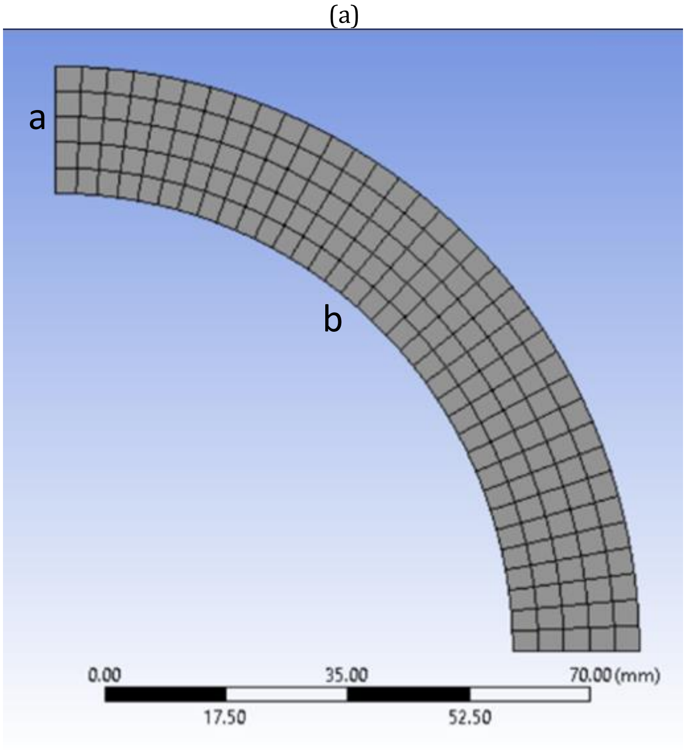
A weak spring is a boundary condition that can be used to model a spring-like support that has a limited ability to support loads. A weak spring boundary condition can be implemented by using a non-linear spring element, such as a non-linear elastic spring or a nonlinear dashpot element. The non-linear spring element can be defined using a force-displacement relationship that describes how the spring force changes as the displacement changes. This relationship can be defined using a mathematical equation or by specifying a table of force-displacement data. In Ansys, the non-linear spring element can be used to model a variety of spring-like supports, including supports with a limited stiffness, supports with a limited deformation capacity, and supports with hysteretic behaviour. By using a non-linear spring element, it is possible to model the behaviour of a structure more accurately under load, especially when the support is expected to deform significantly under load.

When defining a weak spring boundary condition in Ansys, it is important to choose the appropriate type of non-linear spring element and to carefully specify the force-displacement relationship to accurately represent the behaviour of the spring-like support. The results of the analysis can be affected by the choice of spring element and the definition of the force-displacement relationship; hence, it is important to carefully consider these factors when using a weak spring boundary condition in Ansys.

A mesh convergence study is a numerical analysis technique used to evaluate the effects of mesh refinement on the accuracy of a computational simulation. The study involves running the simulation multiple times using finer meshes and comparing results to determine if the solution has converged into a final result as the results obtained are constant. The objective of the study is to identify the minimum mesh size required to achieve a specified level of accuracy. This method was carried out by increasing the number of divisions along each edge. Adjusting the size of the edge by one increment ensured that the difference between results was as steady as possible as shown in Figure 5. Hence, quadrilateral mesh, as shown in Figure 5, was used for the initial analysis. Quadrilateral mesh is often used in pipe or axisymmetric 2D simulations because it provides a more efficient and accurate way to model the geometry of the pipe or axisymmetric structure.



(a)



(b)

Fig. 5. (a) No mesh, and (b) Edge sizing with quadrilateral mesh

Determining the optimal mesh size in simulations is an iterative process. It typically begins with a coarse mesh, generated by the default settings in Ansys software. This coarse mesh is then used to obtain an initial stress measurement. The process continues by generating a medium mesh and comparing the resulting stress measurement with the coarse mesh value. If the difference between the two values, also known as relative error, exceeds an acceptable tolerance, a further refinement of the mesh is necessary. This process of refining the mesh and comparing stress measurements continues until a desired level of accuracy is achieved.

Quadrilateral mesh is a type of finite element mesh used in computational simulations, such as finite element analysis (FEA), computational fluid dynamics (CFD), and other numerical methods. A quadrilateral mesh is composed of 4-sided elements, where each element has four nodes or vertices. These elements can be either regular or irregular in shape, and they can be made up of straight or curved sides, depending on the geometry of the domain being modelled. The quadrilateral mesh is often preferred over other types of meshes, such as triangular or tetrahedral meshes because it provides a more structured and regular grid, which can be beneficial for many types of simulations.

Quadrilateral meshes are often used in simulations involving two-dimensional (2D) planar geometries, and they can also be used in three-dimensional (3D) simulations when a structured mesh is needed. In a pipe or axisymmetric 2D simulation, the geometry is symmetric around an axis, and therefore, the mesh can be generated using a quarter or half symmetry approach, which reduces the size of the mesh and speeds up the simulation time. Quadrilateral meshing provides a more regular and structured mesh, which can help reduce errors in the simulation results and increase the accuracy of the solution. Additionally, quadrilateral elements are more computationally efficient than other element types, such as triangles or tetrahedra, because they have fewer nodes, which reduces the overall size of the finite element model and reduces the computational resources required to solve the problem. In summary, quadrilateral mesh is used in pipe or axisymmetric 2D Ansys simulations because it provides a more efficient, accurate, and computationally efficient way to model the geometry of the structure and obtain reliable simulation results.

In Static Structural, meshing is an important step in creating a finite element model for analysis. The mesh is generated in Ansys Workbench, Static Structural. Static Structural provides various element types, such as linear, quadratic, and cubic elements. The element type can be selected based on the complexity of the model and the accuracy required. Static Structural provides several tools for setting up meshing parameters, such as the "Mesh" tab in the tool bar. The Mesh Method option is used to control the way the mesh is generated for the geometry.

It provides various methods to generate the mesh, such as "Quadrilateral Dominant", “Triangle”, and "Multizone Quad/Tri”. Multizone Quad/Tri is chosen because it is a feature in Ansys that allows users to partition a complex geometry into smaller subdomains or zones to analyse each subdomain independently. This approach can help in reducing the computational cost and time required for solving large and complex geometries. Multizone Quad/Tri is selected, and the next step is to generate the mesh. This can be done by selecting the "Generate Mesh" option in the "Mesh" tab. The Ansys Static Structural will generate a mesh based on the settings and controls provided. Symmetry boundary conditions were applied on the symmetric edges of the sector. A set of data was collected for each mesh node and the resulting stress. As the number of nodes increased, the time required for the simulation's solution increased. Consequently, the finer the mesh, the more precise the results were. Table 3 shows the number of edge sizing with the number of nodes and stress results.

Table 3
Number of edge sizing, nodes, and stress

Solution	a x b	Number of nodes	Equivalent stress, MPa
1	1 x 35	258	73.689
2	2 x 35	413	73.709
3	3 x 35	568	73.705
4	4 x 35	723	73.707
5	5 x 35	878	73.707
6	6 x 35	1033	73.707
7	7 x 35	1188	73.707
8	8 x 35	1343	73.708
9	9 x 35	1498	73.708
10	10 x 35	1653	73.708
11	11 x 35	1808	73.708
12	12 x 35	1963	73.708

3. Results and Discussion

Generally, a finer mesh leads to more accurate results as it increases the number of elements which consist of lines and nodes that represent the geometry. To ensure an adequate mesh and reliable results, a mesh convergence study is crucial. This study, discussed in detail later, will determine the optimal mesh size for this analysis. In the study, a pressure of 5 MPa was applied at edge "b" of the pipe as referred to in Figure 5. This standardized pressure allows for a direct comparison of the number of solutions required for convergence across different pipe thicknesses. Even with the same pressure applied, it is expected to see variations in stress results due to the differences in pipe wall thickness.

Comparing mesh convergence study results with different numbers of solutions involves evaluating the results of simulations run on different outcomes and comparing them. The goal is to determine the minimum number of solutions required to obtain an accurate solution. If increasing the number of mesh elements results in an improved solution, the results indicate that a finer mesh is required for an accurate solution. If the solution does not improve significantly, then it can be concluded that the current mesh is sufficient and further refinement is unnecessary. It is important to consider that a finer mesh might increase computation time, therefore a trade-off between accuracy and computational efficiency must be considered when choosing the number of solutions.

The mesh convergence study for a pipe thickness of t = 3.40 mm is illustrated in Figure 6. Solution number 4 is selected as the optimal mesh size because the equivalent stress, σ stabilizes at a constant value of 111.86 MPa between solution numbers 4 and 12. This indicates that further mesh refinement (increasing the number of nodes and elements) beyond solution number 4 has minimal impact on the stress results. In contrast, the graph shows a significant variation in stress values from solutions 1 to 3. This initial increase in stress results is likely due to the insufficient mesh density in these solutions, which cannot accurately capture the stress distribution within the pipe. This trend aligns with previous findings, where the initial stages of mesh refinement often exhibit instability and generate significant errors [17].

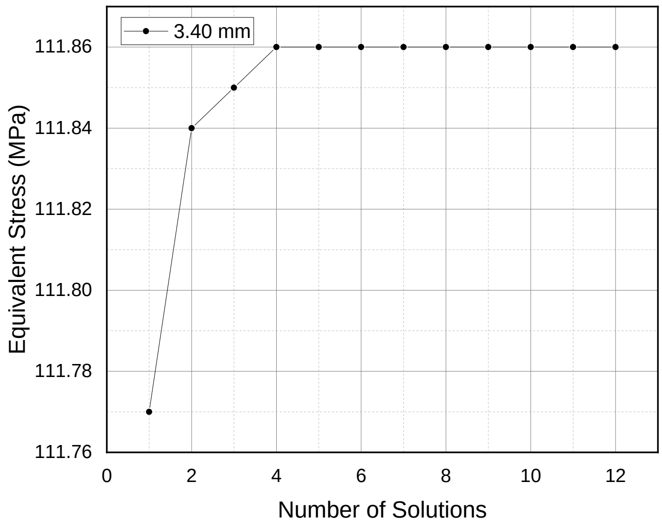


Fig. 6. Graph convergence study for *t* = 3.40mm

Figure 7 illustrates the convergence study graph specifically for thickness *t* = 7.11 mm. Analysing the graph reveals a pivotal observation: among the solutions plotted, solution number 15 stands out as the optimal choice for this particular thickness, demonstrating consistent and reliable outcomes persisting through solution number 20. Furthermore, the comparative analysis between thicknesses elucidates a significant disparity in the number of solutions obtained. Notably, for *t* = 7.11 mm, the abundance of viable solutions contrasts starkly with the limited instances found for *t* = 3.40 mm. This discrepancy can be attributed to the inherent variation in pipe thickness, which inherently impacts the convergence and resultant solutions. It is worth noting that despite the uniformity in applied pressure at 5 MPa across these different thicknesses, the divergent outcomes underscore the influential role of varying thicknesses in shaping the obtained results. This further emphasizes the need for meticulous consideration of all parameters, especially material dimensions, in predicting and interpreting solution behaviour within the studied system [18].

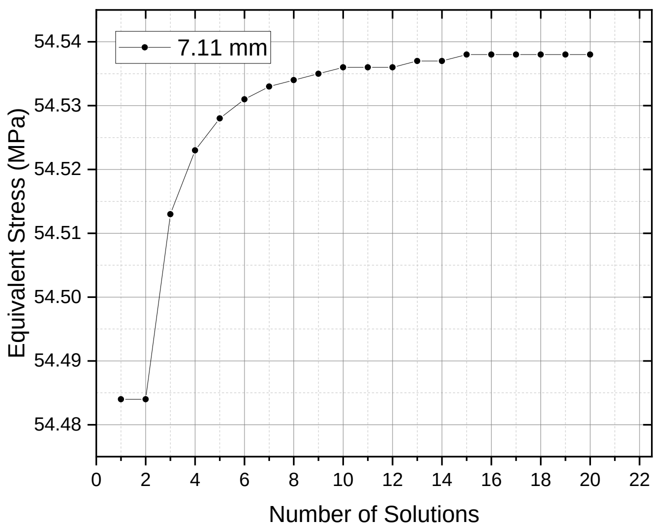


Fig. 7. Graph of convergence study for *t* = 7.11 mm

Upon careful analysis of the graph depicted in Figure 8, it becomes evident that for a thickness of *t* = 10.97 mm, the selection of solution number 18 emerges as the optimal choice owing to its sustained consistency across subsequent solutions until number 24. This intriguing observation remains consistent despite the considerable disparity in the meshing size range between solutions 18 and 24, characterized notably by a substantial variance in the number of nodes. Despite the anticipated divergence in results due to varying mesh complexities, solution number 18 remarkably showcases a remarkable stability in its outcomes, maintaining identical trends and patterns observed in solution 24. This congruity, despite the differences in meshing, highlights the robustness and reliability of solution number 18 within this range. The coherence exhibited by solution number 18 through solution number 24 signifies a significant degree of convergence in results, reaffirming the accuracy and resilience of this solution amidst altering meshing complexities. This unexpected uniformity reaffirms the efficacy and robustness of solution number 18 as a dependable choice for the given thickness, underscoring its consistency and reliability in producing accurate simulation outcomes across the varied meshing sizes considered.

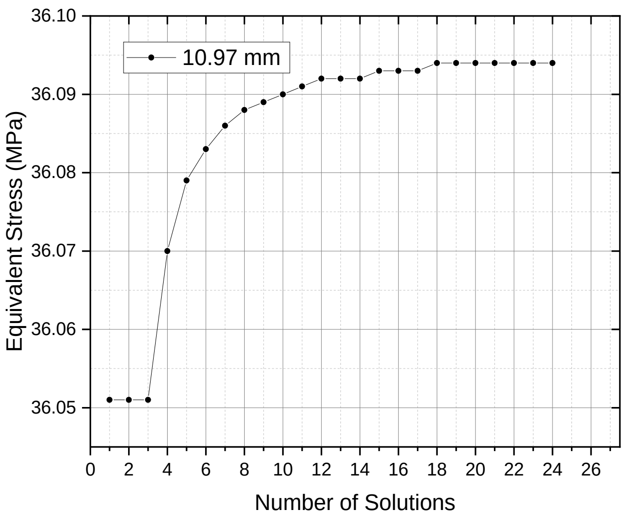


Fig. 8. Graph of convergence study for *t* = 10.97 mm

Furthermore, the discernible pattern depicted in Figure 9 regarding the correlation between thickness and the number of viable solutions is of paramount significance. The increasing trend in feasible solutions with rising thickness elucidates a pivotal aspect of the experimental outcomes, shedding light on the influence of this specific parameter on the range of available options. This nuanced understanding not only emphasizes the importance of precise thickness determination but also underscores the potential for optimizing solutions by strategically manipulating this parameter. Moreover, the conspicuous deviation in trends witnessed in the prior graphs highlights the unique impact of thickness variations, establishing it as a crucial factor in the decision-making process for selecting an ideal solution. Therefore, the prominence of solution number 18 within this range underscores its reliability and consistency amidst this dynamic relationship, making it a standout choice for further exploration and potential implementation in the given context.

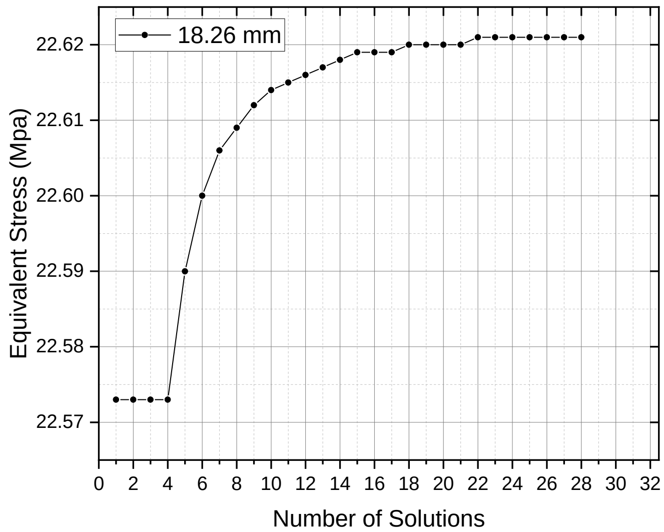


Fig. 9. Graph of convergence study for t = 18.26 mm

Upon conducting an in-depth mesh convergence study utilizing data from Figure 6-9, the consistent trend observed reveals a significant pattern. Notably, as the expansive area size increases, the rate of mesh convergence displays a gradual slowdown, aligning with previous research findings [18-20]. This observation underscores the crucial aspect of the comprehensive duration required for such a study—an essential investment owing to its direct correlation with the precision and reliability of the ensuing simulation outcomes [21,22]. The reliability of these simulations not only substantiates their accuracy but also renders them compelling for real-world implementation scenarios. Consequently, leveraging the dependable data obtained from these simulations empowers the formulation of meticulously designed pipeline structures even before their physical installation. The robustness and promise showcased by the simulation results ensure that the proposed pipeline designs are not just theoretically sound but also practically feasible. This comprehensive approach establishes a solid foundation for confident decision-making in pipeline infrastructure, paving the way for optimal performance and durability in real-world environments.

4. Conclusions

The mesh convergence study conducted in this simulation confirms that the FEA model has achieved a converged solution. This ensures the model accurately captures the system's behavior while minimizing computational time. The results demonstrate that thicker pipes require more mesh elements to converge, as observed by the longer solution times for t = 18.26 mm where stability was recorded at solution 18 compared to t = 3.40 mm with stability at solution 4. Performing a mesh convergence study allows us to identify the optimal mesh size, balancing accuracy with computational efficiency. This approach guarantees reliable and cost-effective simulation results.

While SolidWorks is a popular software for design tasks, Ansys offers dedicated tools for advanced simulations like mesh convergence studies. These specialized tools in Ansys can potentially lead to higher accuracy and reduced errors compared to using more general design software like SolidWorks for simulation purposes.

Acknowledgement
The authors would like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support through this research work. The appreciation also dedicated

to Petroliam Nasional Berhad (PETRONAS) for the guidance of expertise in the project. This research was not funded by any grant.

References

[1] "ASME Code for Pressure Piping, B31," n.d. https://doi.org/10.1115/B31

[2] Wang, Dong, Britta Bienen, Majid Nazem, Yinghui Tian, Jingbin Zheng, Tim Pucker, and Mark F. Randolph. "Large deformation finite element analyses in geotechnical engineering." *Computers and geotechnics* 65 (2015): 104-114. https://doi.org/10.1016/j.compgeo.2014.12.005

[3] Burkhart, Timothy A., David M. Andrews, and Cynthia E. Dunning. "Finite element modeling mesh quality, energy balance and validation methods: A review with recommendations associated with the modeling of bone tissue." *Journal of biomechanics* 46, no. 9 (2013): 1477-1488. https://doi.org/10.1016/j.jbiomech.2013.03.022

[4] Ismail, N. I., M. Asyraf Tasin, Hazim Sharudin, M. Hisyam Basri, S. Che Mat, H. Yusoff, and R. E. M. Nasir. "Computational aerodynamic investigations on wash out twist morphing mav wings." (2022): 1090-1102. https://doi.org/10.5109/6625721

[5] Li, Ruan, Mohamed Karma, and Changhong Hu. "Two-Dimensional VIV Simulation of a Cylinder Close to a Wall with High Reynolds Number by Overset Mesh." (2023): 219-229. https://doi.org/10.5109/6781072

[6] Darmawan, Steven, Kevin Raynaldo, and Agus Halim. "Investigation of thruster design to obtain the optimum thrust for rov (remotely operated vehicle) using cfd." (2022): 115-125. https://doi.org/10.5109/4774224

[7] Al-Gahtani, Husain J., and Faisal M. Mukhtar. "Simplified formulation of stress concentration factors for spherical pressure vessel–cylindrical nozzle juncture." *Journal of Pressure Vessel Technology* 138, no. 3 (2016): 031201. https://doi.org/10.1115/1.4032112

[8] Xu, J., Z. L. Zhang, E. Østby, B. Nyhus, and D. B. Sun. "Constraint effect on the ductile crack growth resistance of circumferentially cracked pipes." *Engineering Fracture Mechanics* 77, no. 4 (2010): 671-684. https://doi.org/10.1016/j.engfracmech.2009.11.005

[9] Loseille, Adrien, Alain Dervieux, and Frédéric Alauzet. "Fully anisotropic goal-oriented mesh adaptation for 3D steady Euler equations." *Journal of computational physics* 229, no. 8 (2010): 2866-2897. https://doi.org/10.1016/j.jcp.2009.12.021

[10] Harinaldi, Harinaldi, and Farhan T. Pratama. "Transient Analysis on the Crosswind Effect to the Aerodynamics of High-speed Train Travelled on the Bridge Between Two Tunnels at Jakarta-Bandung Track." *CFD Letters* 16, no. 10 (2024): 64-80. https://doi.org/10.37934/cfdl.16.10.6480

[11] Zhyriakov, Dmytro, Oleksandr Grebenikov, Andrig Humennyi, and Dmytro Konyshev. "Design of High Fatigue Life Joints of Fuselage Structures Considering Fracture Mechanics." In *Conference on Integrated Computer Technologies in Mechanical Engineering–Synergetic Engineering*, pp. 159-173. Cham: Springer Nature Switzerland, 2022. https://doi.org/10.1007/978-3-031-36201-9_14

[12] Ko, Junyoung, Sangseom Jeong, and Joon Kyu Lee. "Large deformation FE analysis of driven steel pipe piles with soil plugging." *Computers and Geotechnics* 71 (2016): 82-97. https://doi.org/10.1016/j.compgeo.2015.08.005

[13] Berger, William, June Ling, Michael Merker, Mark Sheehan, and David Wizda. "ASME Standards and Certification: Now and Beyond." *Mechanical Engineering* 131, no. 06 (2009): 22-26. https://doi.org/10.1115/1.2009-JUN-2

[14] Agrawal, Vikash Kumar, and H. P. Khairnar. "Experimental & analytical investigation for optimization of disc brake heat dissipation using cfd." (2022): 1076-1089. https://doi.org/10.5109/6625720

[15] Devals, C., T. C. Vu, Y. Zhang, Julien Dompierre, and François Guibault. "Mesh convergence study for hydraulic turbine draft-tube." In *IOP Conference Series: Earth and Environmental Science*, vol. 49, no. 8, p. 082021. IOP Publishing, 2016. https://doi.org/10.1088/1755-1315/49/8/082021

[16] Patil, Hemesh, and P. V. Jeyakarthikeyan. "Mesh convergence study and estimation of discretization error of hub in clutch disc with integration of ANSYS." In *IOP conference series: materials science and engineering*, vol. 402, p. 012065. IOP Publishing, 2018. https://doi.org/10.1088/1757-899X/402/1/012065

[17] Salifu, Smith, Dawood Desai, Festus Fameso, Olugbenga Ogunbiyi, Samson Jeje, and Azeez Rominiyi. "Thermo-mechanical analysis of bolted X20 steam pipe-flange assembly." *Materials Today: Proceedings* 38 (2021): 842-849. https://doi.org/10.1016/j.matpr.2020.04.882

[18] Burcet, Martí, Beñat Oliveira, Juan Carlos Afonso, and Sergio Zlotnik. "A face-centred finite volume approach for coupled transport phenomena and fluid flow." *Applied Mathematical Modelling* 125 (2024): 293-312. https://doi.org/10.1016/j.apm.2023.08.031

[19] Andisso, Fellek Sabir, and Gemechis File Duressa. "Graded mesh B-spline collocation method for two parameters singularly perturbed boundary value problems." *MethodsX* 11 (2023): 102336. https://doi.org/10.1016/j.mex.2023.102336

[20] Engelmann, Linus, Josef Hasslberger, Seung-Jin Baik, Markus Klein, and Andreas Kempf. "Direct numerical simulation of an unsteady wall-bounded turbulent flow configuration for the assessment of large-eddy simulation models." *Scientific Reports* 13, no. 1 (2023): 11202. https://doi.org/10.1038/s41598-023-37740-7

[21] Zadeh, Saman Naghib, Matin Komeili, and Marius Paraschivoiu. "Mesh convergence study for 2-D straight-blade vertical axis wind turbine simulations and estimation for 3-D simulations." *Transactions of the Canadian Society for Mechanical Engineering* 38, no. 4 (2014): 487-504. https://doi.org/10.1139/tcsme-2014-0032

[22] Valeš, Jan, and Zdeněk Kala. "Mesh convergence study of solid FE model for buckling analysis." In *AIP Conference Proceedings*, vol. 1978, no. 1. AIP Publishing, 2018. https://doi.org/10.1063/1.5043796