STM JOURNALS

Journal of
Offshore Structure and Technology

ISSN: 2349-8986
Volume 13, Issue 2, 2026
May–August
DOI (Journal): 10.37591/JoOST

https://journals.stmjournals.com/joost

Research JoOST

Grid Sensitivity and Its Impact on the Hydrodynamics of a Submerged Vessel

Sahul Muhique[1], Kalyan Kalita[2], Kalyan Das[3,]*

Abstract

Accurate prediction of hydrodynamic characteristics is essential for the design and performance assessment of underwater vehicles and offshore structures. Computational fluid dynamics (CFD) has emerged as an effective tool for analyzing fluid flow behavior around submerged bodies, enabling detailed evaluation of pressure distribution, drag forces, and flow patterns. In this study, the effect of mesh refinement on the hydrodynamic analysis of a DRDC-STR submarine nose section was investigated using ANSYS Fluent. The geometry employed in the numerical simulations was based on the benchmark model proposed by Mackay, which is widely used for validating submarine flow analyses. A series of mesh profiles with varying element densities were generated to examine the influence of grid resolution on simulation accuracy and computational efficiency. The simulations were conducted using a pressure-based solver under steady-state flow conditions. Hydrodynamic parameters, including pressure distribution over the submarine nose section and drag coefficient values, were evaluated for each mesh configuration. Particular attention was given to identifying mesh-independent solutions that provide reliable results without excessive computational cost. The numerical results demonstrate that mesh refinement significantly affects the prediction of hydrodynamic forces and surface pressure characteristics. As the mesh density increased, the computed results converged toward stable values, indicating improved numerical accuracy. An error analysis was performed to validate the simulation outcomes and assess the consistency of the numerical predictions. The maximum numerical variance between successive mesh refinements was found to be less than 5%, which falls within accepted engineering design tolerances. The findings confirm that an appropriately refined mesh can enhance the accuracy of CFD simulations while maintaining computational efficiency. The study provides valuable guidance for future hydrodynamic investigations of submarine hull forms and other submerged marine structures using ANSYS Fluent.

Keywords: ANSYS Fluent, coefficient of drag, standardized DRDC-STR submarine geometry, patch-conforming tetrahedron method, pressure drag, viscous drag

INTRODUCTION

Mesh refinement is arguably the most critical step in the pre-processing phase, as it is the bridge between a theoretical geometry and a reliable numerical solution. If the mesh is inadequate, even the most advanced physics models produce vague results [1].

In this study, the computational mesh was optimized through an iterative sensitivity analysis to produce grid-independent solutions while maintaining reasonable simulation times. By progressively refining the mesh and monitoring the result stability, we identified the ideal coefficient of drag. The nose section served as the primary test case, where three mesh configurations were compared at a velocity of 50 m/s to verify convergence [2, 3].

***Author for Correspondence**
Kalyan Das
E-mail: kkdas1971@gmail.com

[1]Student, Department of Mechanical Engineering, Assam Engineering College, Jalukbari, Kamrup Metro, Assam, India
[2]Professor, Department of Mechanical Engineering, Assam Engineering College, Jalukbari, Kamrup Metro, Assam, India

Received Date: June 05, 2026
Accepted Date: June 30, 2026
Published Date: July 06, 2026

Citation: Sahul Muhique, Kalyan Kalita, Kalyan Das. Grid Sensitivity and Its Impact on the Hydrodynamics of a Submerged Vessel. Journal of Offshore Structure and Technology. 2026; 13(2): 1–8p.

MESH REFINEMENT AND SENSITIVITY ANALYSIS

Mesh refinement is one of the most important stages in the pre-processing phase of computational fluid dynamics (CFD) simulations, as it establishes the connection between the physical geometry and numerical solutions. The quality of the computational mesh directly influences the accuracy, stability, and reliability of simulation results. An inadequate mesh may lead to significant numerical errors and inaccurate predictions, regardless of the sophistication of the governing equations or the turbulence models employed. Therefore, careful mesh generation and optimization are essential to ensure that the numerical model accurately represents physical flow phenomena.

In this study, the computational mesh was optimized through an iterative mesh sensitivity analysis aimed at achieving grid-independent solutions while maintaining acceptable computational costs. The mesh refinement process involved generating multiple mesh configurations with progressively increasing element densities and evaluating their effects on key hydrodynamic parameters. Particular attention was given to the drag coefficient and pressure distribution, as these parameters are critical indicators of submarine hydrodynamic performance [4–6].

The DRDC-STR submarine nose section was selected as the primary test geometry for this sensitivity study. Three different mesh configurations (coarse, medium, and fine) were developed and analyzed under identical flow conditions corresponding to a free-stream velocity of 50 m/s. The numerical results obtained from each mesh were compared to assess the convergence behavior and determine the extent to which the mesh density affected the predicted hydrodynamic characteristics [7].

The analysis revealed that the variation in the drag coefficient decreased significantly with mesh refinement, indicating convergence toward a stable solution. Beyond a certain mesh density, further refinement produced only marginal changes in the calculated results while substantially increasing the computational time. Consequently, an optimal mesh configuration was identified that provided an appropriate balance between computational efficiency and numerical accuracy, ensuring reliable hydrodynamic predictions for subsequent simulations [8].

Modelling: A standardized DRDC-STR submarine geometry (hull-sail-tail configuration) reproduced from Mackay (1).

The simulations in this report utilized the DRDC generic submarine geometry in a hull-sail-tail (HST) configuration (Figure 1). Characterized by an l/d ratio of 8.75 and a Riegels Type D2 nose of a hull, which is modelled at a full-scale length of 30 m, this study examines the impact of strong currents on submerged vehicles by applying fundamental marine engineering hydrodynamic principles. The resulting data offers a framework for the design optimization of maritime components [9, 10].

MESHING RESULTS AND DISCUSSIONS

To optimize the simulation, a mesh sensitivity analysis was performed to balance the accuracy and computational efficiency. Grid independence was achieved by iteratively refining the grid until the results reached a stable plateau. The refinement focused on the submarine's nose section by comparing three mesh profiles at a velocity of 50 m/s. Post-convergence, the resulting Cd and Pressure Drag Force were compared to the theoretical values, and a percentage error analysis was conducted to verify the model's reliability.

The submarine hull was evaluated using three distinct meshing configurations, as described in Table 1, to determine the optimal balance between the resolution and computational effort.

- *Type A (Baseline):* Utilized system default settings with a relevance of +100 and a fine relevance center.
- *Type B (Local refinement):* Incorporation of vertex sizing control using a Sphere of Influence (1 m radius) with an element size of 0.01 m. This configuration utilizes the patch-conforming tetrahedron method.

- *Type C (high resolution):* Built upon Type B by adding an edge size of 0.001 m in addition to the 1 m Sphere of Influence (0.01 m element size), maintaining the patch-conforming tetrahedron method for volume meshing (Table 1).

Table 1 details a progressive mesh refinement strategy across three distinct tetrahedral configurations used to optimize the hydrodynamic accuracy of the simulation. Type A serves as the baseline, relying entirely on a high global grid density with a relevance of +100 but without any localized geometric controls. Type B introduces regional volumetric refinement by implementing a Sphere of Influence with a 1 m radius and a 0.01 m element size, although it sacrifices overall density by dropping the global relevance to 0. Type C represents the most comprehensive and fully refined setup; it integrates the high global density of Type A (+100 relevance), preserves the regional volumetric control of Type B (0.01 m element size within the Sphere), and layers on micro-level precision by adding a vertex sizing element size of 0.001 m at critical geometric boundaries (Figures 2–4).

Boundary Conditions

- A boundary condition was set at the inlet, with the inlet velocity profile set to 50 m/s.
- Density of water: 998.2 kg/m^3
- Temperature of water (operating temperature): 298.15 K
- Dynamic viscosity of water: 0.001003 N-s/m^2
- Density of the material of construction of the submarine: 2719 kg/m^3
- The equation used to calculate the theoretical value of drag force is given by

$$F_D = \frac{1}{2} \times C_D \times \rho \times A \times v^2$$

- The simulated values of the coefficient of drag were calculated by Fluent using the following formula:

$$c_d = \frac{1}{\rho v^2 A} \int_S dA(p - p_0)(\hat{n} \cdot \hat{\imath}) + \frac{1}{\rho v^2 A} \int_S dA(\hat{t} \cdot i) T_w$$

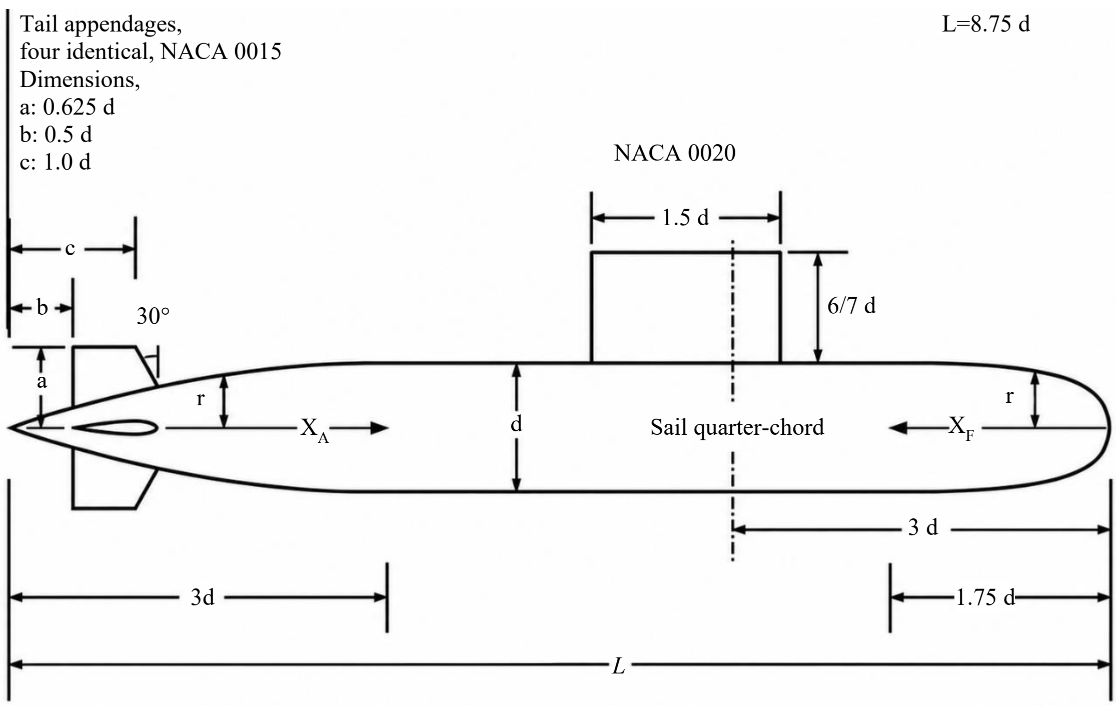

Figure 1. A standardized DRDC-STR submarine geometry.

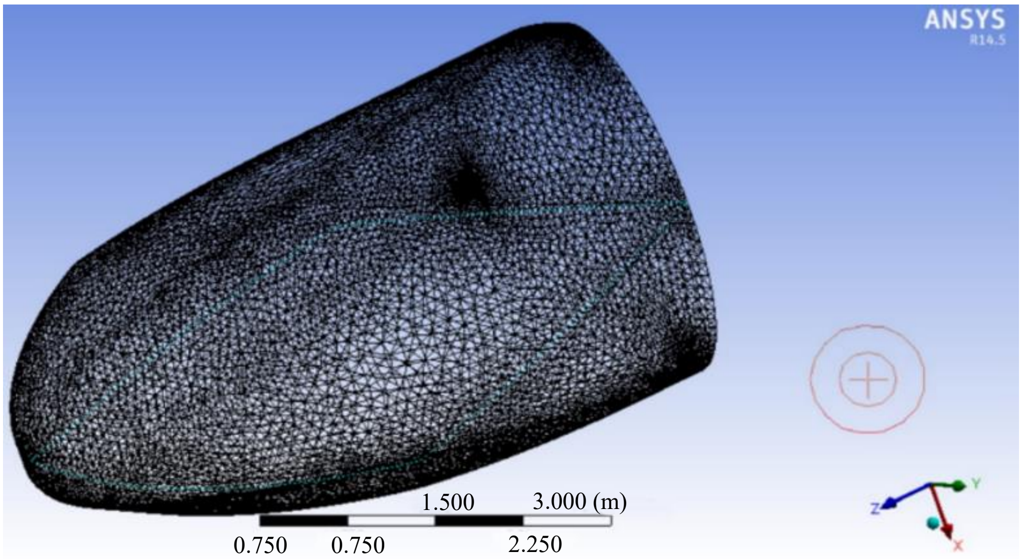

Figure 2. TYPE A meshing.

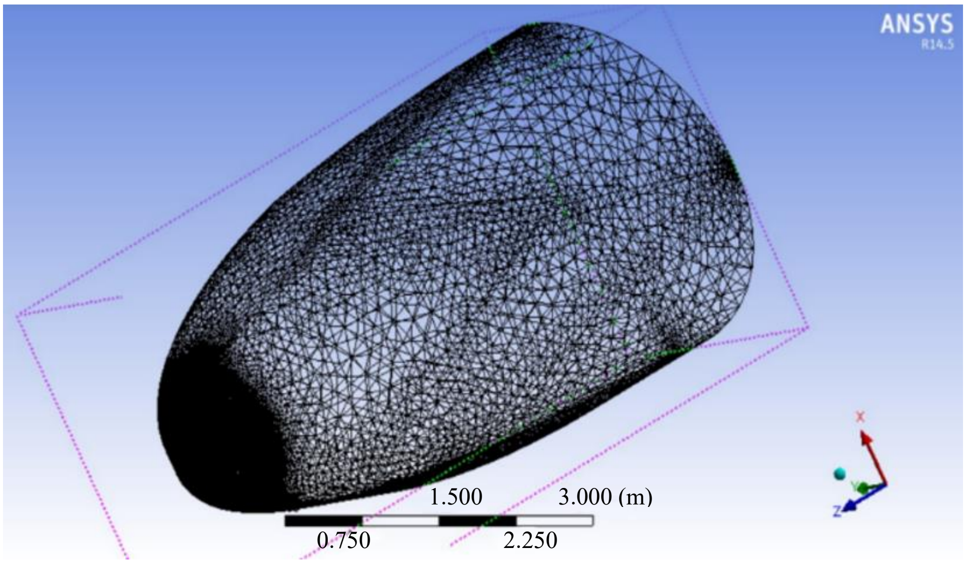

Figure 3. Type B meshing.

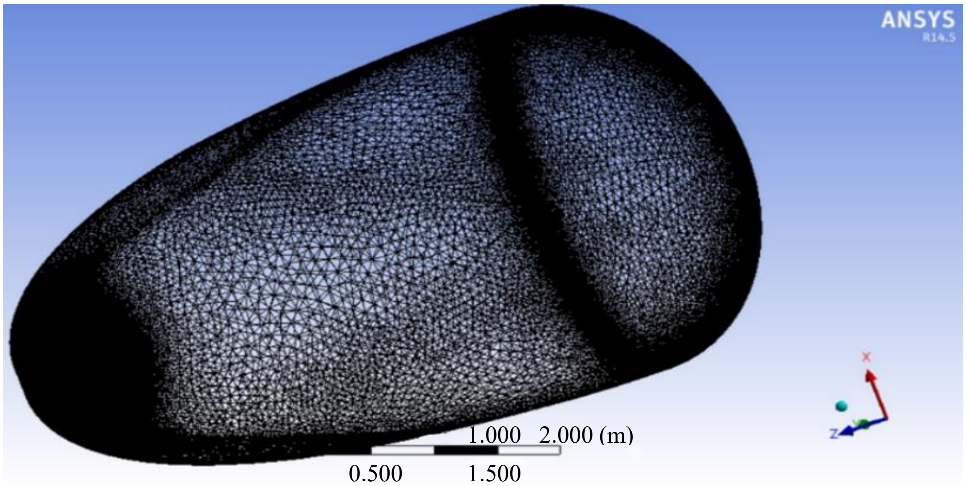

Figure 4. Type C meshing.

Table 1. Types of meshing.

Type	Sphere of influence		Relevance	Vertex sizing	Element shape
	Radius	***Element size***		***Element size***	
A	-	-	+100	-	Tetrahedron
B	1 m	0.01 m	0	-	Tetrahedron
C	1 m	0.01 m	+100	0.001 m	Tetrahedron

Table 2. Error analysis of the three different types of mesh.

Mesh profile	C_d	Pressure drag along z-axis (N)	Theoretical value (N)	Error (%)
Type A	0.0433	3295876.3	3129277.144	5.32
Type B	0.0402	2849007.4	2905241.136	1.97
Type C	0.0417	3040675.6	3013645.656	0.89

The above equation represents the calculation of the total drag coefficient (c_d) of a body (e.g., submarine hull or airfoil) moving through a fluid. It breaks the total drag force into its two fundamental physical components: pressure drag (form drag) and viscous drag (skin friction drag).

In mathematical terms, the equation integrates the local forces over the entire wetted surface area (S) of the body and normalizes them against the dynamic pressure and reference area.

ANALYSIS AND DISCUSSION

The flow behavior was assessed using the pressure and velocity distributions across the three candidate meshes. The convergence of the Cd and drag force values was monitored during the simulation. By extracting the mode of the Cd data once the solution was stabilized, the drag force was calculated analytically for verification. The discrepancy between the software's direct output and the analytical calculation was quantified as a percentage error. This comparative study allowed for the selection of an optimal mesh profile that balances numerical precision and computational efficiency.

RESULT

By selectively meshing and iterating the parts of the submarine hull, the pressure drag force and coefficient of drag at the nose section at 50 m/s were determined, as shown in Table 2. From this, it can be concluded that the best results with the least error were obtained from the Type C Meshing Profile, which is the most refined meshing profile.

Based on Table 2, the error analysis of the three different types of meshes provides a quantitative validation of the numerical simulation by comparing the computed drag forces with theoretical benchmarks. Across the three mesh configurations, the calculated drag coefficient (c_d) ranged between 0.0402 and 0.0433, yet the resulting forces revealed distinct accuracy gaps. TYPE A over-predicted the pressure drag along the z-axis at 3,295,876.3 N, yielding the highest deviation with a 5.32% error relative to its theoretical baseline of 3,129,277.144 N.

Conversely, Type B under-predicted the pressure drag at 2,849,007.4 N, though it improved accuracy by narrowing the deviation to a 1.97% error against its theoretical value of 2,905,241.136 N. Demonstrating the highest precision, Type C computed a pressure drag of 3,040,675.6 N, which closely aligned with its corresponding theoretical target of 3,013,645.656 N to achieve a negligible 0.89% error. This progressive reduction in the error percentage directly validates that the advanced multilevel localized grid controls integrated into the Type C mesh profile successfully minimized numerical dissipation and delivered the most mathematically reliable hydrodynamic solution (Figures 5–7).

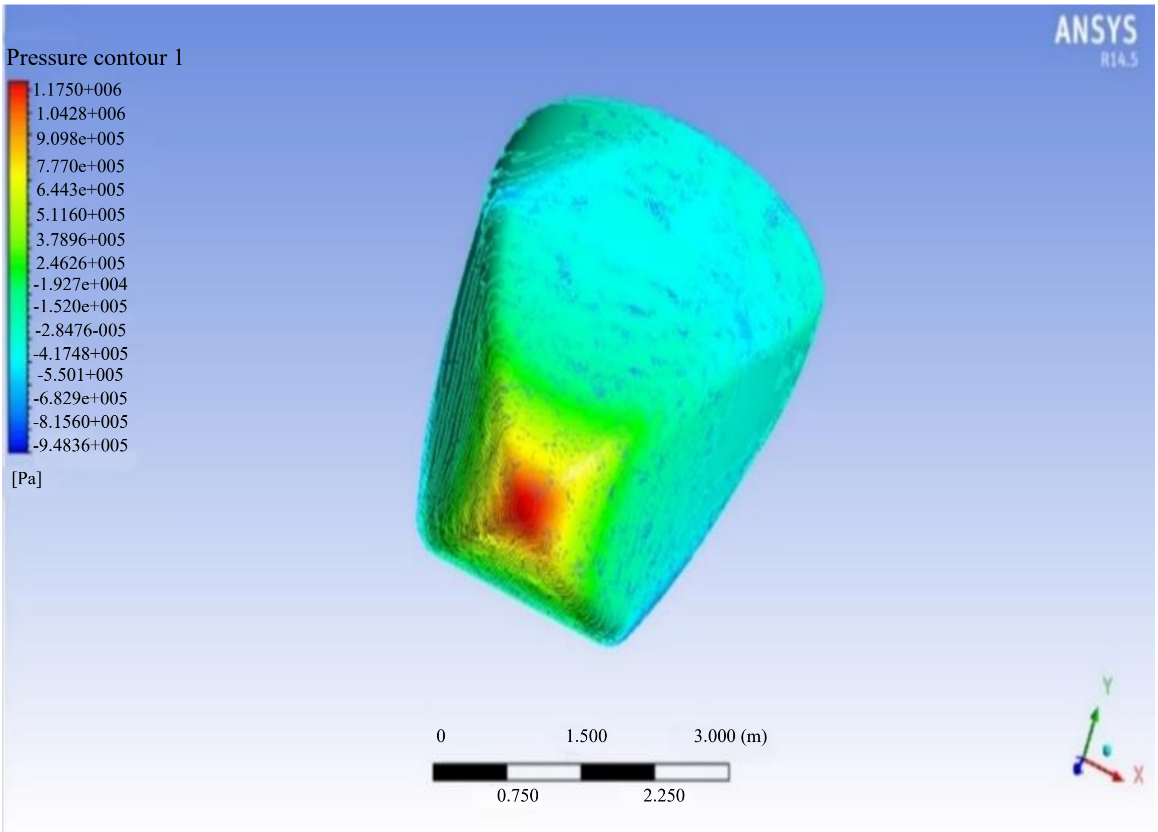

Figure 5. Pressure contour over the nose at a velocity of 50 m/s with Type A meshing.

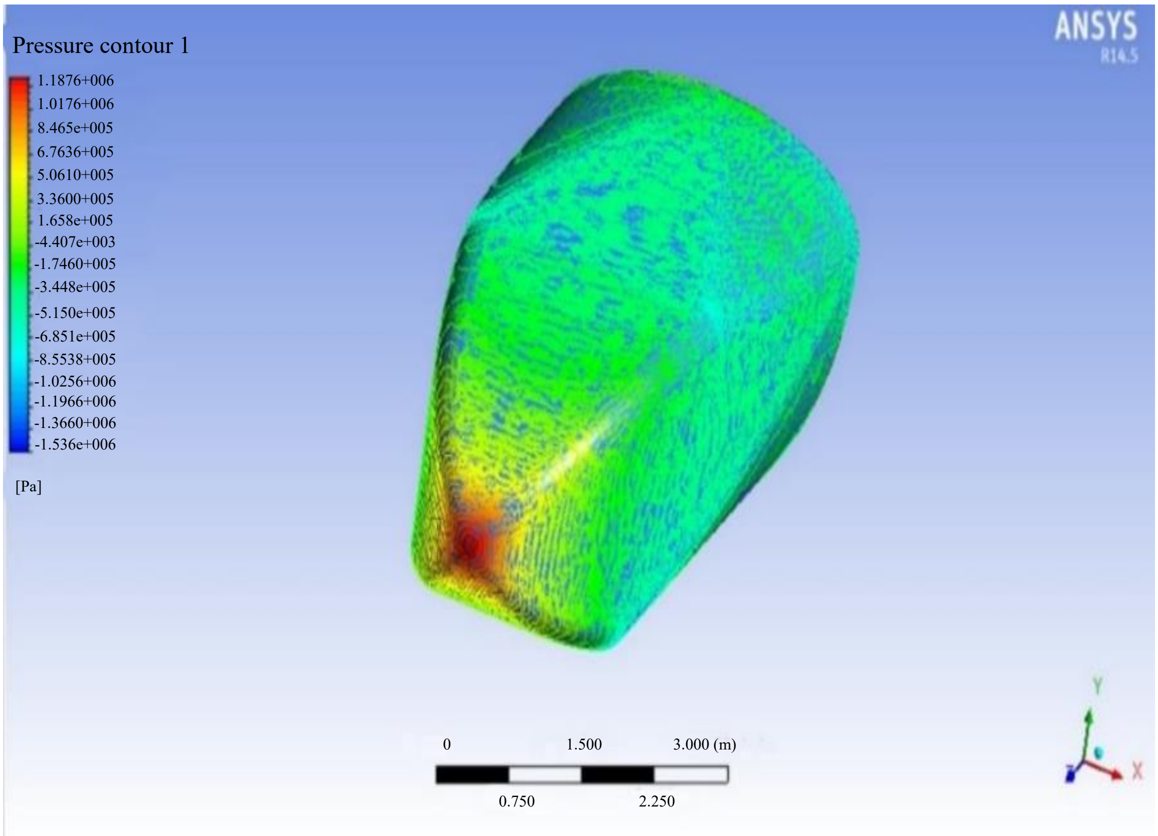

Figure 6. Pressure contour over the nose at a velocity of 50 m/s with Type B meshing.

Journal of Offshore Structure and Technology
Volume 13, Issue 2
ISSN: 2349-8986

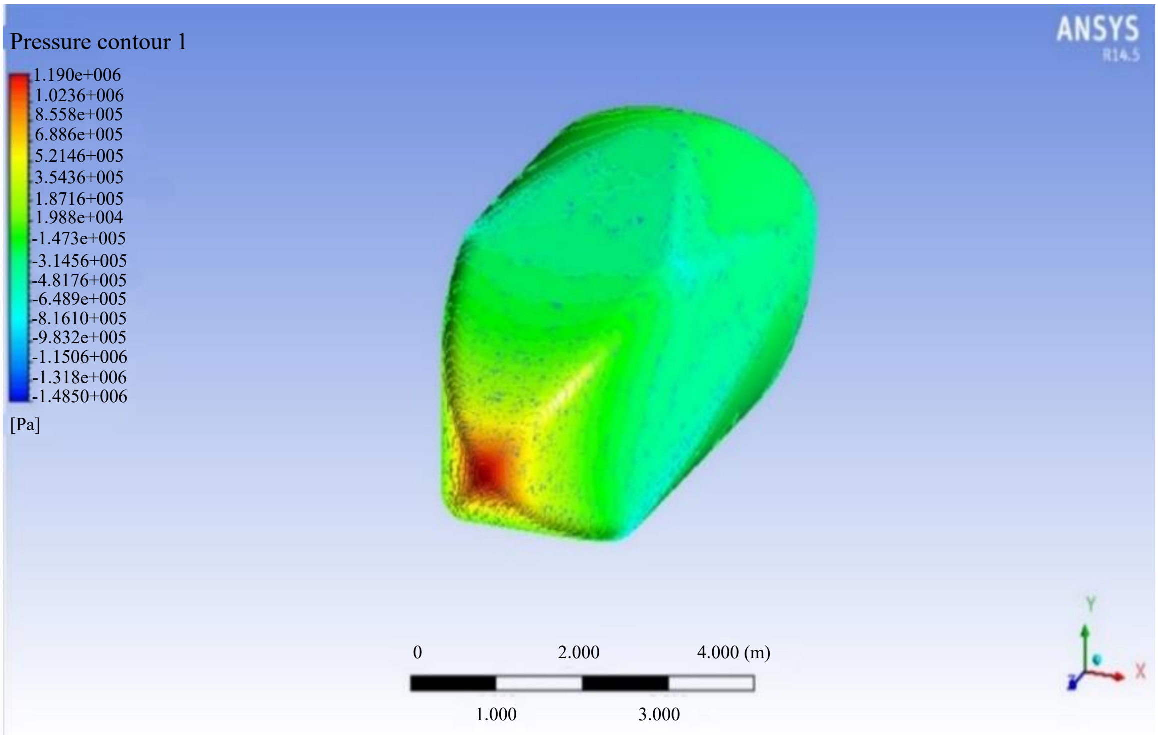

Figure 7. Pressure contour over nose at velocity of 50 m/s with Type C meshing.

CONCLUSION

This study evaluated the critical role of mesh sensitivity in the hydrodynamic analysis of a standard submarine hull using ANSYS Fluent, alongside a comparative study of various NACA airfoils. A DRDC-STR model was simulated to determine the drag coefficient (Cd) and resultant drag force across the nose and hull using three distinct mesh profiles. The numerical results were validated against the theoretical values to quantify the relative error for each configuration.

Although the Type A mesh utilized a specified element size, it resulted in a 5.32% error. The Type B configuration improved accuracy by reducing the error to 1.97%. The Type C mesh yielded the highest precision, with a negligible error of 0.89%. The superior performance of the Type C profile is attributed to the integration of global relevance and Local Vertex Sizing. While "Relevance" dictates the overall mesh density across the entire geometry, "vertex sizing" provides localized refinement at critical geometric points, ensuring the most accurate capture of flow physics.

Although the error margins between the mesh types were small, they represented a high degree of risk in the context of deep-sea engineering. A slight inaccuracy in the force prediction can result in a design that fails under the extreme pressures of the actual environment. This underscores the necessity of mesh refinement; precision in the pre-processing stage is vital to prevent structural failure and ensure the reliability of the submarine geometry.

In summary, this study demonstrates that although minor variations in mesh density may appear statistically subtle, they are paramount in high-stakes hydrodynamic modelling. The progression from a 5.32% error in the Type A mesh to a negligible 0.89% error in the Type C configuration highlights the efficacy of combining global relevance with targeted Local Vertex Sizing within ANSYS Fluent. By meticulously refining critical geometric regions, the Type C mesh successfully captured the complex flow physics across the DRDC-STR submarine nose with high precision. Ultimately, these findings underscore that rigorous mesh sensitivity analysis is not merely a routine validation exercise but a critical pre-processing mandate in deep-sea engineering, where even a slight underestimation of drag forces can mean the difference between structural integrity and catastrophic failure.

REFERENCES

1. Baker C. Estimating Drag Forces on Submarine Hulls. Defence R&D Canada-Atlantic CR. 2004;2004-2125.
2. Mackay M. The standard submarine model: a survey of static hydrodynamic experiments and semi-empirical predictions. Defence R&D Canada-Atlantic TR. 2003;2003-2079.
3. Kukner A. Investigation of flow distribution around a submarine. J Nav Sci Eng. 2006;12(2):1-26.
4. Molland AF, Turncock SR. Wind Tunnel Investigation of the Influence of Propeller Loading on Ship Rudder Performance. Ship Science Report No. 46. [place unknown]: Ship Science; 1991.
5. Larsson L, Stern F, Visonneau M. Numerical Ship Hydrodynamics: An Assessment of the Gothenburg 2010 Workshop. Dordrecht: Springer; 2014.
6. Han KJ. Numerical Optimization of Hull/Propeller/Rudder Configurations. Gothenburg, Sweden: Department of Shipping and Marine Technology, Chalmers University of Technology; 2008.
7. Kawabuchi M, Kubota M, Ishikawa S. Development of technology to estimate the flow field around ship hull considering wave making and propeller rotating effects. Mitsubishi Heavy Industries Technical Review. 2016;53(2):53-57.
8. International Towing Tank Conference. Practical Guidelines for Ship CFD Applications. Proceedings of the 26th International Towing Tank Conference; 2011; Rio de Janeiro, Brazil. International Towing Tank Conference; 2011.
9. Ferziger JH, Perić M. Computational Methods for Fluid Dynamics. 3rd ed. Berlin: Springer; 2002. doi:10.1007/978-3-642-56026-2.
10. Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics: The Finite Volume Method. 2nd ed. Harlow, UK: Pearson Education; 2007.

