Secure Systems Design and Technology Development. Veliko Tarnovo, Bulgaria
Proceedings of the International Scientific Conference, 2026. Volume II, 335-338

Pitot Tube Simulation: A Key Tool for Real Engineering Applications in Marine Engineering and Mechatronics

Cvetelina Velkova
Department of Mechatronics
Naval Academy N. Vaptsarov
Varna, Bulgaria
cvvelkova@nvna.eu

Abstract— **The Pitot tube is a classical and widely used instrument for measuring fluid flow velocity, with applications across various engineering fields, including marine engineering systems and modern mechatronic applications. This paper presents a numerical simulation of a Pitot tube performed using ANSYS Fluent, aiming to demonstrate its operating principle, measurement accuracy, and significance as both an educational and research tool. The adopted approach enables observation of velocity and pressure distribution around the tube, providing a deeper understanding of fluid flow phenomena and their relation to real-world measurements. Within the simulation framework, different flow regimes are analyzed, along with the influence of geometric parameters and the Reynolds number on velocity determination accuracy. The results clearly illustrate the regions of high and low pressure that define the tube's readings, confirming the reliability of the method for engineering applications.**

The proposed methodology has strong educational value, as it allows students from the "Ship Machines and Mechanisms" and "Mechatronics" programs to visualize real fluid flow processes that are often difficult to observe experimentally. Through simulation, they acquire skills in CFD software operation, numerical data interpretation, and the practical application of theoretical knowledge. Thus, the Pitot tube becomes not only a measuring device but also a key educational instrument, bridging fluid mechanics theory with real engineering practice and promoting analytical thinking among future engineers.

Keywords— CFD simulation, Fluid Mechanics, Mechatronics, Marine engineering, Pitot's tube

I. INTRODUCTION

In modern engineering practice, the accurate measurement of fluid velocity is a critical component for the efficiency and safety of transport systems. Despite the advent of advanced technologies such as Laser Doppler Anemometry (LDA) and ultrasonic methods, the classical Pitot tube remains an indispensable tool due to its reliability, simplicity, and lack of moving parts. The present paper, titled *"Pitot Tube Simulation: A Key Tool for Real Engineering Applications in Marine Engineering and Mechatronics,"* examines the integration of this classical measuring instrument with Computational Fluid Dynamics (CFD) methods and contemporary mechatronic signal processing systems.

The fundamental basis for the operation of the Pitot tube relies on classical fluid mechanics. As noted by F. M. White (2015) in his work *"Fluid Mechanics"* [1], the relationship between static, dynamic, and total pressure, described by Bernoulli's equation, serves as the theoretical foundation for converting pressure differentials into flow velocity. However, theoretical idealizations often diverge from real-world conditions. P. Robert et al. (1984) in *"Fundamentals of Temperature, Pressure, and Flow Measurements"* [2] emphasizes that factors such as fluid compressibility, nozzle geometry, and turbulence can introduce significant errors. Therefore, to ensure industrial applicability, strict adherence to standardization is imperative. The geometric parameters and calibration methods used in this study are aligned with the international standard ISO 3966:2020 *"Measurement of fluid flow in closed conduits — Velocity area method using Pitot static tubes"* [3], which defines permissible deviations and correction coefficients.

The specificity of marine engineering imposes additional requirements on these sensors. The application of Pitot tubes for measuring ship speed (speed logs) is complicated by the hydrodynamic interaction with the hull. A.F. Molland et al., (2011), in their analysis *"Ship*

Online ISSN 3044-7771
https://doi.org/10.68302/std2026.vol2.61

Resistance and Propulsion" [4], explain in detail the formation of the boundary layer around the hull. If the sensor is positioned within the boundary layer zone, the measured speed will be lower than the actual speed, necessitating precise positioning. This issue is also regulated in the procedures of the International Towing Tank Conference (ITTC Recommended Procedures and Guidelines: Speed and Distance Measurement [5]), which serve as a benchmark for validating ship performance characteristics.

In the era of Industry 4.0, the Pitot tube cannot be viewed merely as a mechanical device, but as part of a complex mechatronic system. Converting differential pressure into an electrical signal requires highly sensitive transducers. J. Fraden (2016) in *"Handbook of Modern Sensors"* [6] discusses contemporary piezoresistive and capacitive sensors, which are critical to the accuracy of the mechatronic module. The significance of this integration becomes even more apparent in autonomous underwater vehicles, where navigation relies entirely on sensor data. J. C. Kinsey et al., (2006), in their survey *"A Survey of Underwater Vehicle Navigation" [7]*, point out that precise speed-over-water measurement is key to control algorithms for underwater robots, thereby expanding the application field of the Pitot tube beyond conventional shipping.

Since physical experiments in towing tanks are expensive and labor-intensive, computer simulations (CFD) have become a primary tool for design and optimization. J. D. Anderson (1995) in *"Computational Fluid Dynamics: The Basics with Applications"* [8] provides the mathematical framework for solving fluid motion equations. For simulating the flow around the Pitot tube, the present work employs the Finite Volume Method (FVM), detailed by H. K. Versteeg et al., (2007) in *"An Introduction to Computational Fluid Dynamics"* [9]. The selection of an appropriate turbulence model (such as k-epsilon or k-omega) is crucial for correctly predicting pressure distribution.

The aim of this paper is to unify these theoretical, normative, and computational aspects. Through a simulation model based on the Navier-Stokes equations and validated against the ISO 3966 standard, we analyze the behavior of the Pitot tube under conditions characteristic of the marine environment and mechatronic applications, thereby closing the loop between theory, virtual prototyping, and real-world operation.

II. General Regulations

The fundamental basis for the operation of the classical Pitot tube relies on the fundamental laws of fluid mechanics, and more specifically on Bernoulli's equation. This equation serves as the primary theoretical foundation for converting the measured pressure differential between total (dynamic) and static pressure into actual fluid flow velocity. However, as noted by White (2015) in his work, theoretical mathematical idealizations often diverge from real-world operating conditions. According to Benedict (1984), a number of physical factors, including fluid compressibility, specific nozzle geometry, and the presence of turbulence, can introduce significant errors and deviations in the final measurements.

To compensate for these inaccuracies and to ensure the reliable industrial applicability of the method, strict adherence to established regulatory standards is necessary. Within the framework of this study, all geometric parameters of the model and calibration methodologies are precisely aligned with the international standard ISO 3966:2025, which clearly defines permissible design deviations and required correction coefficients.

The specificity of modern marine engineering imposes even stricter additional requirements on these measuring sensors, especially when Pitot tubes are integrated as hydrodynamic speed logs for recording ship speed. Their practical application is seriously complicated by the continuous hydrodynamic interaction with the vessel's hull. In their analysis, Molland, Turnock, and Hudson (2011) examine in detail the formation of a viscous boundary layer around the ship's hull. If the measuring sensor is positioned within this specific zone, the recorded speed will be significantly lower than the actual speed. Therefore, extremely precise and careful positioning is required, which is regulated in detail in the recommended procedures of the International Towing Tank Conference (ITTC).

From a modern mechatronics perspective, in the era of Industry 4.0, the Pitot tube cannot be viewed in isolation as a simple mechanical device, but rather as a key component of a complex system. The reliable conversion of the recorded differential pressure into an electrical signal requires the integration of highly sensitive piezoresistive or modern capacitive transducers, as detailed by Fraden [6]. This deep sensor integration is of critical importance for autonomous underwater vehicles (AUVs). In these vehicles, navigation algorithms and control systems depend entirely on precise speed-over-water measurement, which expands the application field of the Pitot tube far beyond conventional shipping, as pointed out by Kinsey, Eustice, and Whitcomb [7].

III. Figures, Tables and Equations

Since physical experiments in towing tanks are often extremely expensive, time-consuming, and labor-intensive, computer simulations using Computational Fluid Dynamics (CFD) have established themselves as a primary and indispensable tool for design, analysis, and optimization in modern engineering. In the present work, the ANSYS Fluent software package is used to conduct the numerical simulation of the Pitot tube. To analyze the fluid behavior, the study applies the Finite Volume Method (FVM) to solve the complex Navier-Stokes differential equations. This method, detailed by Versteeg and Malalasekera [8], allows for precise discretization of the computational domain, ensuring strict adherence to the laws of mass and momentum conservation in every control cell of the grid. The mathematical framework underlying these calculations is based on the fundamental principles laid out by Anderson [9].

Numerical Settings and Parameters:

To achieve maximum reliability and stability of the results, the following key parameters of the simulation model have been defined:

- Viscous Model**:** A laminar flow model was selected for the purposes of simulation. This decision is directly based on the calculated number of Reynolds relative to the specific diameter of the tube. Under these geometric and velocity conditions, the fluid flow maintains its stratified character without highly pronounced turbulent pulsations, making the laminar model not only physically adequate but also computationally efficient.
- Pseudo-Transient Settings: To accelerate the convergence of the numerical solution, a pseudo-transient approach was utilized. The length scale was initially set to correspond to the external diameter of the Pitot tube, and subsequently refined and reduced to the diameter of the internal tubes during the calculation process. This allows for a smooth adaptation of the model to the local velocity and pressure gradients.

Initialization: Full Multigrid (FMG) initialization was applied. This technique is widely used in aerodynamics because it initially calculates the solution on coarser computational grids before interpolating it to the fine base grid. This approach provides a significantly better initial solution and drastically reduces the overall computational costs and the time required to reach a steady state in the simulation.

IV. Other Recommendations

This section presents and analyzes the data obtained from the CFD simulation, with each result named separately for clarity.

Fig. 1. Contours of pressure.

Fig. 1 shows that these contours show the effect of the orientation of the ports on the pressure measurement. The pressure increases significantly in the central tube, where the fluid stops completely (stagnation). This measured pressure corresponds to the total pressure of the fluid. In the side ports, the pressure is lower and corresponds to the static pressure of the surrounding environment.

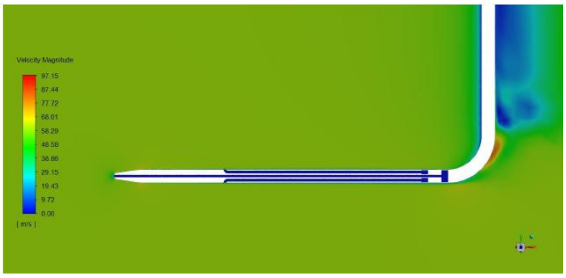

Fig. 2. Contours of velocity magnitude.

The visualization in Fig. 2 shows that the air velocity inside the Pitot tubes is zero. In the central channel, the air cannot move forward and becomes "trapped" while the flow passes parallel to the side static pressure ports without entering them.

Area-Weighted Average Static Pressure	(pascal)
static-pressure-port	65779.227
total-pressure-port	67472.61

Fig. 3. Area-weighted average static pressure.

Using the reporting function in Ansys Fluent, the average value of the static pressure on the ports is calculated, Fig. 3. The image shows the calculated value of 65779.227 Pa, which serves to accurately determine the differences between the total and static pressure.

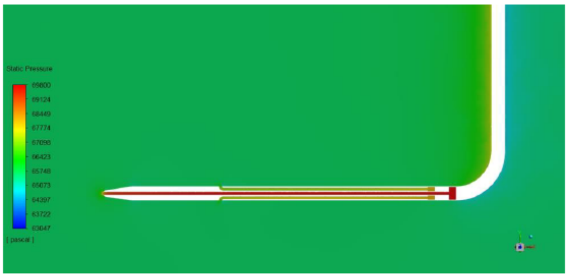

Fig. 4. Pressure distribution after FMG initialization.

Velocity distribution, Full Multigrid Initialization (FMG) initialization is used in aerodynamics to obtain a better initial value of the solution, Fig. 4. It calculates the solution on coarser grids and interpolates it to the base grid, which reduces the overall computational cost of the simulation.

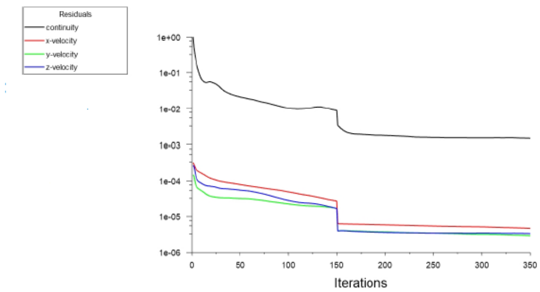

Fig. 5. Residuals.

Fig. 5 tracks the convergence of the solution. It can be seen how the errors for the velocity components (x, y, z) and the continuity equation drop sharply and stabilize, which is an indication of a correctly calculated and stable simulation.

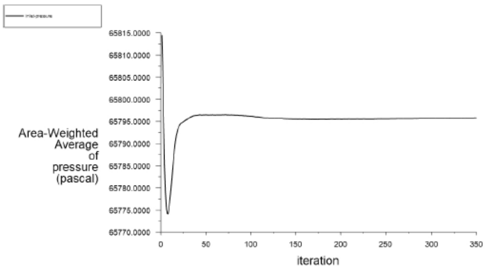

Fig. 6. Plot of inlet static pressure.

The graph in Fig. 6 shows how the static inlet pressure changes and stabilizes over the course of the iterations. After initial fluctuations during FMG initialization (around the 100th iteration), the value levels off, confirming that a steady state has been reached.

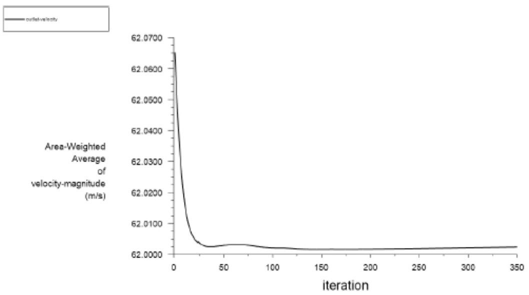

Fig. 7. Plot of outlet velocity magnitude.

Similar to pressure, the outlet velocity is monitored to ensure that mass balance is achieved, Fig. 7. The stabilization of the curve around the 150th iteration indicates that the flow through the outlet boundary is constant.

Applications in Marine Engineering and Mechatronics

The simulation results have direct applications in designing ship control systems. Understanding high and low-pressure zones around the sensor allows engineers to avoid errors caused by hydrodynamic interaction with the hull. In mechatronics, these data are critical for calibrating digital pressure sensors used in underwater robotics, where navigation depends on precise speed data.

ACKNOWLEDGMENTS

This report/article was financed by the Ministry of Education and Science under the National Science Program "Security and Defense", carried out in implementation of the National Strategy for the Development of Scientific Research 2017-2030 and adopted by Decision of the Council of Ministers No. 731 of October 21, 2021. The material reflects only the author's opinion and the Ministry of Education and Science is not responsible for the content.

REFERENCES

[1] F.M. White, „Fluid Mechanics “, (2015), [Online]. Available: https://www.academia.edu/8669730/Frank_M_White_Fluid_Mechanics S. M. Hemmington, Soft Science. Saskatoon: University of Saskatchewan Press, 1997.

[2] P. Robert, P.E. Benedict, Fundamentals of Temperature, Pressure, and Flow Measurements, (1984), [Online]. Available: https://www.scribd.com/document/391722697/Fundamentals-of-Temperature-Pressure-and-Flow-Measurements-Robert-P-BenedictR. Hayes, G. Pisano, D. Upton, and S. Wheelwright, Operations, Strategy, and Technology: Pursuing the Competitive Edge. Hoboken, NJ: Wiley, 2005.

[3] ISO 3966:2025– „Measurement of fluid flow in closed conduits — Velocity area method using Pitot static tubes “, [Online]. Available: https://www.iso.org/standard/86000.html

[4] A.F. Molland, S.R. Turnock, and D.A. Hudson, „Ship Resistance and Propulsion “, 2011.

[5] International Towing Tank Conference (ITTC) – „Recommended Procedures and Guidelines: Speed and Distance Measurement “

[6] J. Fraden, „Handbook of Modern Sensors, Physics, Designs and Application “, 2016.

[7] J. C. Kinsey, R.M. Eustics, L.L. Whitcomt, "A Survey of Underwater Vehicle Navigation: Recent Advances and New Challenges", 2006

[8] H. K. Versteeg and W. Malalasekera, „An Introduction to Computational Fluid Dynamics “, 2007

[9] J.D. Anderson, Computational Fluid Dynamics: The Basics with Applications, 1995.