Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294

Volume 3, No 2, December 2024 [42-50]

Analysis of Aircraft Hydraulic Filter Flow: Computational Fluid Dynamics Simulation Using SimFlow 4.0

Muhammad Aiman Rahimi Adli[1*]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia

Received 9 August 2024, Revised 2 September 2024, Accepted 7 September 2024

ABSTRACT

Hydraulic systems are critical to the performance and reliability of various aircraft and machine operations, such as landing gear, brake systems, and control surfaces. Understanding the flow properties of hydraulic circuits is essential to optimize these systems. This study aims to analyze an aircraft hydraulic filter's performance using SimFlow and Computational Fluid Dynamics (CFD), focusing on mesh resolution and inlet velocity influence to predict pressure and velocity accurately. A comprehensive three-dimensional filter model is developed, and meshing is conducted at different resolutions. The flow is then modeled using the k-ω SST turbulence model within the Reynolds-Averaged Navier-Stokes (RANS) framework, considering fully turbulent, incompressible, and steady-state flow conditions. The findings are expected to show that finer mesh resolutions yield more precise predictions of pressure drops and flow distributions within the filter. As mesh density increases, the variance in maximum pressure and velocity values is anticipated to decrease, leading to more consistent simulation outcomes. This research provides insights into optimal meshing strategies for accurate CFD analysis of hydraulic filters, emphasizing the importance of careful mesh selection in achieving reliable simulation results. The results have practical implications for designing and optimizing more efficient hydraulic systems. Future work should focus on attaining mesh independence, simulating transient flows, and cross-validating the findings with experimental data.

Keywords: Simulation and modeling, SimFlow, Aircraft Hydraulic Filter, Computational fluid dynamics.

1. INTRODUCTION

The study of hydraulic systems in aircraft and machinery is essential for achieving optimal performance and ensuring long-term reliability. These systems play a critical role in numerous critical operations, including the actuation of control surfaces, the deployment and retraction of landing gear [1], the operation of braking systems [2], and the powering of various other equipment components [3]. Given their integral role in maintaining the safety and efficiency of aircraft, a deep understanding of hydraulic systems is necessary. Hydraulic circuits, which form the backbone of these systems, rely on fluid dynamics principles to operate effectively. The precise movement and control of hydraulic fluid within these circuits enable mechanical components' smooth and responsive operation [4]. Any deviation or inefficiency in the flow characteristics can lead to significant performance issues, potentially compromising the safety and functionality of the entire system. Therefore, a thorough understanding of the flow properties within hydraulic circuits is critical for optimizing system performance and preventing potential failures. Besides, technological advances and the increasing complexity of aircraft systems demand that engineers and researchers continuously refine their knowledge of hydraulic

*Corresponding author: aiman89.adli@gmail.com

systems. This includes exploring new materials [5], improving the design of hydraulic components [6], and employing advanced computational tools to simulate and predict fluid behavior under various operating conditions [7]. Through such efforts, the aviation industry can continue to enhance the dependability and efficiency of hydraulic systems, ensuring they meet the rigorous demands of modern aircraft operations.

Various aspects of hydraulic systems, such as pressure drops, flow distribution, and filter efficiency, have been thoroughly examined in prior studies. According to Zhang, W., and Zhang, Y. (2016) [8], their study provided detailed insights into how different mesh resolutions and turbulence models affect the predicted pressure drops and flow distributions. It found that finer meshes and advanced turbulence models resulted in more precise and reliable predictions, which are crucial for designing efficient hydraulic filters. A study by Luo, X., and Chen, Q. (2019) [9] stated the impact of various design parameters, such as filter geometry and placement, on the performance of hydraulic systems. It was found that small changes in these parameters could lead to significant improvements in system performance. Plus, the study demonstrated that CFD could be effectively used to optimize filter configurations to achieve minimal pressure loss and efficient flow distribution. By simulating various design configurations, the researchers identified design modifications that significantly improved filter performance. Korkmaz et al. [10] investigated the flow and pressure drop in hydraulic filters through both experimental and numerical methods to ensure the removal of contaminants from hydraulic systems and the protection of sensitive components like pumps, motors, and actuators. Experiments were conducted using eight different Reynolds numbers, ranging from 1250 to 2350, under constant viscosity conditions. Pressure measurements were taken between the filter's inlet and outlet, and the numerical analysis, validated by experimental results, provided a detailed examination of the flow. They found that the fluid does not pass homogeneously through all surface areas of the filter element, with pressure drops attributed to the Dean vortex formation at the filter outlet.

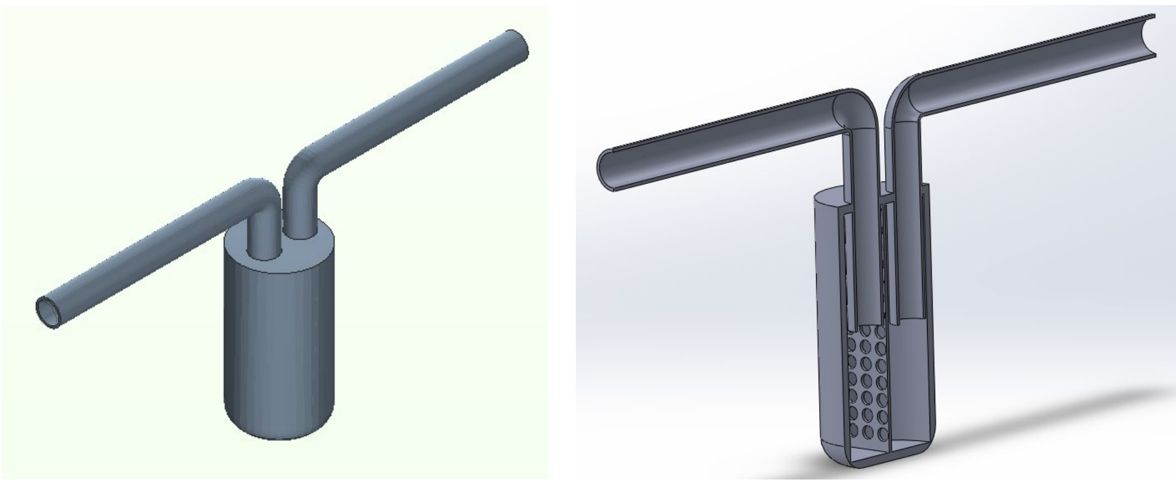
Konstantinov et al. [11] developed a methodology for calculating flow rate characteristics and determining screen filters' absolute and nominal filter ratings using ANSYS CFX. The methodology involves a three-stage numerical simulation process: simulating liquid flow at the micro-level in the filter’s mesh, simulating liquid flow in a single corrugation of the filter element, and simulating fluid flow in the entire filter at the macro level. Verification was conducted by comparing the simulation results with experimental data from the “Diagnostics and identification of hydraulic systems” stand in USATU and “Gidravlika,” showing a less than 2% divergence. This validated the methodology for designing new filter constructions. Ni et al. [12] focused on the impact of fluid pressure pulsations on the performance and durability of aircraft engines and control systems, particularly those caused by positive displacement pumps under high pressure. Using CFD analysis, the research examined how flow-induced pressure waves and their coupling with structural components contribute to mechanical fatigue, noise, and cavitation in vane pumps within the aircraft engine control system. The findings indicated that severe localized responses at the system's resonant frequency can lead to failures in the hydraulic control system. Si [13] studied a dynamic pressure simulation analysis of the return line in a civil aircraft's hydraulic system, focusing on the effects of critical factors on the pressure peaks under various scenarios. The study found that opening the safety valve in the event of a hydraulic pump failure results in no significant pressure shock in the downstream return pipeline. In scenarios involving the instantaneous reversing of a single user, increased user speed leads to higher pressure peaks, with users inside the wing experiencing more significant peaks than those outside, given the same motion conditions. The simultaneous reversing of multiple users significantly raises the dynamic pressure peak in the return line compared to a single user’s action. These findings emphasized the need to consider multi-user interactions when designing the return line pressure for civil aircraft, thereby aiding in selecting and designing the hydraulic system return pipeline.

This project uses SimFlow and Computational Fluid Dynamics [14] to analyze the flow characteristics of an aircraft hydraulic filter. The primary goal is to assess how mesh resolution affects the filter's ability to anticipate pressure and velocity accurately. Using the k-ω SST turbulence model and the Reynolds-Averaged Navier-Stokes (RANS) technique, the study considers thoroughly turbulent, incompressible, and steady-state flow conditions. It is anticipated that this work will offer significant insights into the best meshing techniques for precise CFD simulations of hydraulic filters, helping to develop and improve hydraulic systems with higher levels of efficiency.

2. MATERIAL AND METHODS

The simulation of the aircraft hydraulic filter was conducted using SimFlow 4.0 [15], focusing on a detailed 3D model representing the typical geometry and features of such a filter. The filter model's dimensions are 150mm in height and 80mm in diameter. A three-dimensional model of the filter is shown in Figure 1(a), and Figure 1(a) depicts the cross-sectional view of the aircraft hydraulic filter. The 3D model was created in SolidWorks and then exported into STL file format. Then, the STL format was imported into the SimFlow 4.0 software for pre-processing. For the meshing, it was performed using an unstructured mesh. The mesh division is set for independent parameters at (50, 50, 10), (70, 70, 20), and (100, 100, 50). The hex meshing step and the box meshing method were applied, as shown in Figure 2. The boundary conditions were defined on the box meshing. After the meshing, the fluid domain of the hydraulic filter was created (Figure 3(a), and the boundary conditions were applied to the fluid domain Figure 3(b). After that, quality checks were carried out to ensure the reliability of the mesh, with skewness values maintained and aspect ratios controlled, as summarized in Table 1.

The simulation employed a turbulent flow model, explicitly utilizing the Reynolds-Averaged Navier-Stokes (RANS) approach [16]. The turbulence model used was k-ω SST. Boundary conditions were applied as follows: a velocity inlet at 5m/s and 7m/s as a second independent parameter, a pressure outlet at a fixed value, and no-slip conditions on the walls. Figure 3(b) illustrates the boundary conditions. For solver discretization, SIMPLE was selected, and several assumptions were made for this simulation. The flow was considered steady, incompressible, and fully turbulent.



(a) (b)

Figure 1: (a) 3D model of aircraft hydraulic filter and (b) cross-section view in SolidWorks.

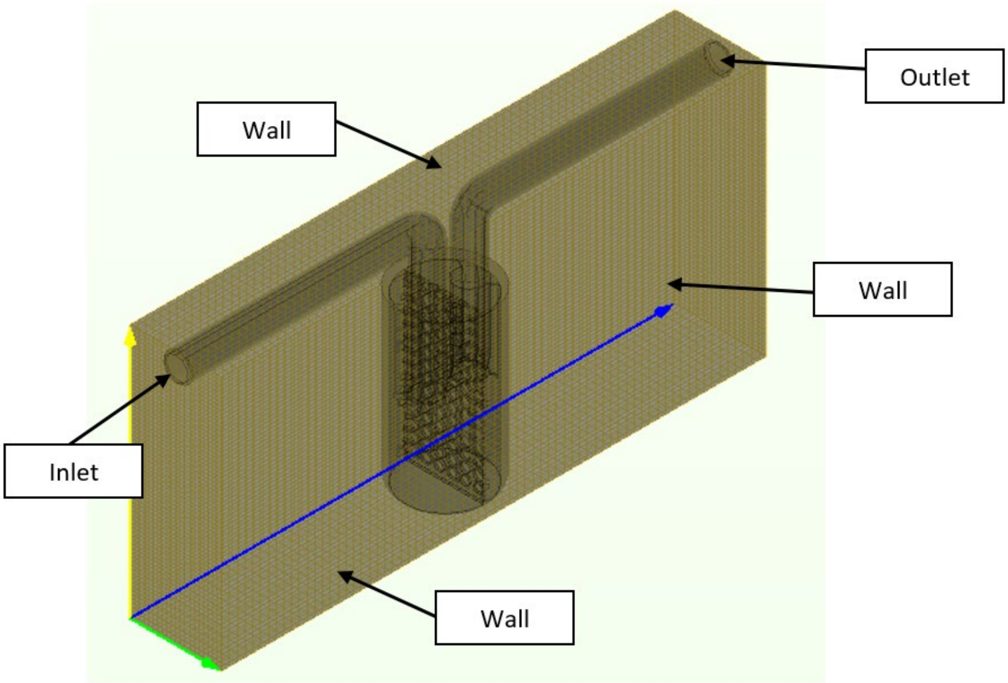


Figure 2: Hex meshing and boundary conditions defined prior to meshing of the aircraft hydraulic filter.

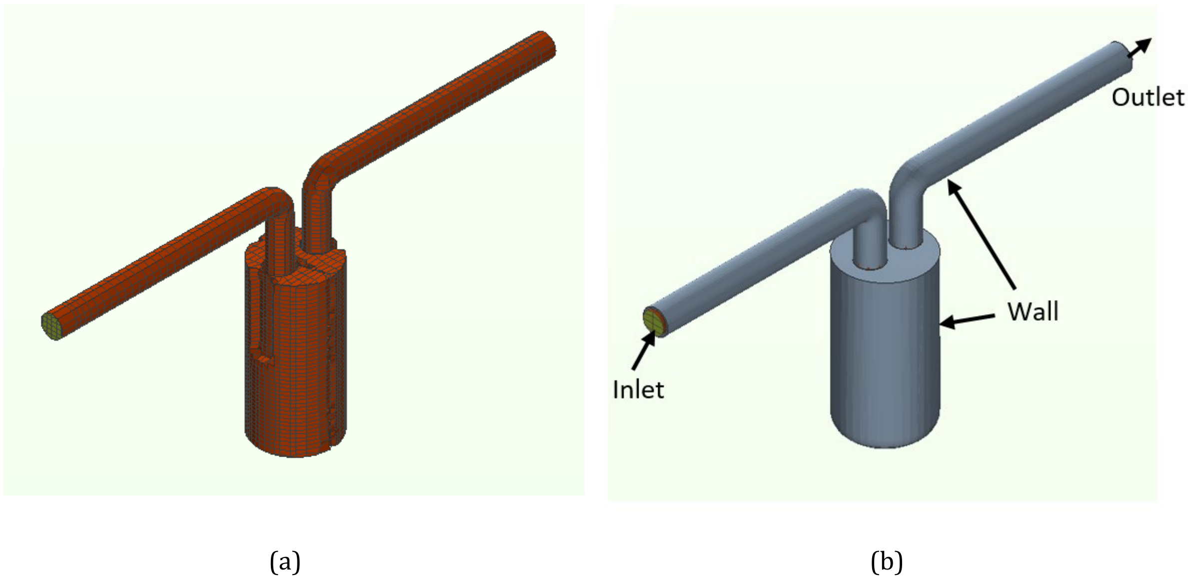


Figure 3: (a) Hex-dominant meshed model and (b) boundary conditions.

Table 1: Meshing information of different mesh divisions.

Parameter	Mesh Division (50,50,10)	Mesh Division (70,70,20)	Mesh Division (100,100,50)
Cells	2090	7837	39340
Nodes	3003	10132	46020
Max aspect ratio	15.46406	15.46406	51.9662
Element type	Hex-dominant	Hex-dominant	Hex-dominant
Max skewness	3.542169	3.542169	15.69821

3. RESULTS AND DISCUSSION

The simulation results for the aircraft hydraulic filter were analysed using various graphs, tables, and contour plots, which provided thorough insights into the filter's performance at different mesh divisions and intake velocities. The simulation results show that the maximum pressure and velocity values vary dramatically among mesh divisions (Tables 2 and 3). For example, with a 5 m/s input velocity, the highest pressure measured for the mesh division (50, 50, 10) was 67.817 Pa, whereas (70, 70, 20) and (100, 100, 50) had pressures of 47.222 Pa and 49.748 Pa, respectively. At a 7 m/s input velocity, the highest pressures observed were 124.43 Pa, 92.405 Pa, and 97.558 Pa for the corresponding mesh divisions. For the calculation of the percentage deviation, we compared the coarsest and finest meshes (50, 50, 10 vs. 100, 100, 50). At 5 m/s, the greatest pressure deviation was roughly 26.67%, whereas at 7 m/s it was around 21.60%. These discrepancies demonstrate the sensitivity of pressure measurements to mesh resolution, emphasizing the significance of mesh quality in CFD simulations. The observed phenomena are due to mesh refinement's impact on simulation accuracy. Finer meshes capture flow features more accurately, reducing numerical errors and providing more reliable results. The discrepancies in maximum pressure and velocity values across mesh resolutions are due to numerical accuracy and convergence principles. The k-ω SST turbulence model is sensitive to mesh quality.

Figures 4 and 5 illustrate the maximum pressure and velocity inside a hydraulic filter at different mesh divisions and inlet velocities. At an inlet velocity of 5 m/s, the lower mesh division (50,50,10) exhibits the highest maximum pressure of 67.817 Pa and a velocity of 7.4153 m/s. Increasing the mesh division to (70,70,20) and (100,100,50) significantly reduces the pressure to 47.222 Pa and 49.748 Pa, respectively, while the velocity remains relatively constant. When the inlet velocity is increased to 7 m/s, the maximum pressure and velocity increase across all mesh configurations. The (50,50,10) mesh shows the highest pressure of 124.43 Pa and a velocity of 10.38 m/s. The finer meshes (70,70,20) and (100,100,50) have lower pressures of 92.405 Pa and 97.558 Pa, respectively, but similar velocities around 10.38 m/s. These results suggest that finer mesh divisions provide better flow characteristics in the hydraulic filter, especially at higher inlet velocities. The data highlights the importance of optimizing mesh size for specific flow rate requirements to ensure the efficient operation of the hydraulic filter.

Table 2: Maximum pressure and velocity inside the hydraulic filter at inlet velocity 5 m/s.

Mesh Division	Max Pressure (Pa)	Max Velocity (m/s)
(50,50,10)	67.817	7.4153
(70,70,20)	47.222	6.9384
(100,100,50)	49.748	7.4243

Table 3: Maximum pressure and velocity inside the hydraulic filter at inlet velocity 7 m/s.

Mesh Division	Max Pressure (Pa)	Max Velocity (m/s)
(50,50,10)	124.43	10.38
(70,70,20)	92.405	9.7555
(100,100,50)	97.558	10.384

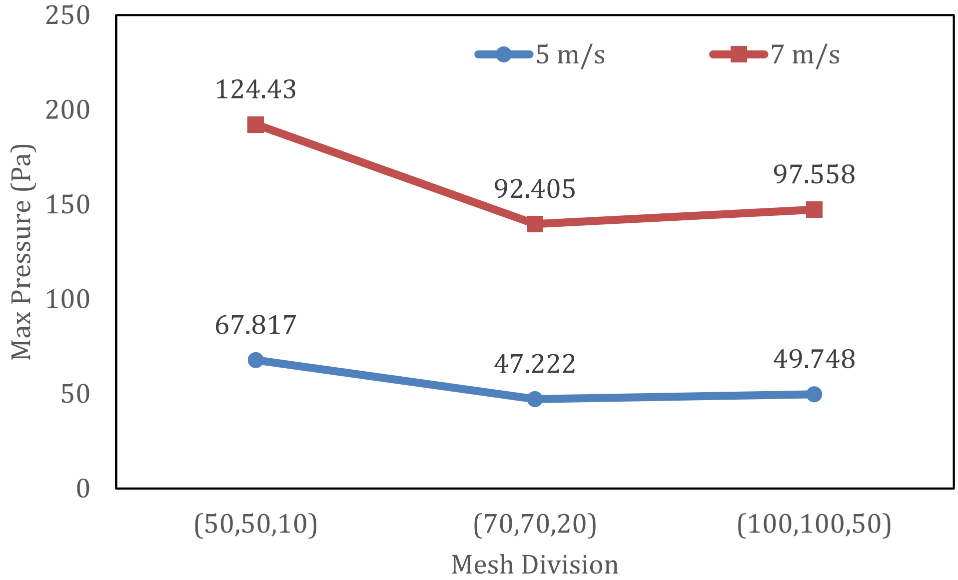


Figure 4: Maximum Pressure for different mesh divisions at 5 and 7 m/s inlet velocity.

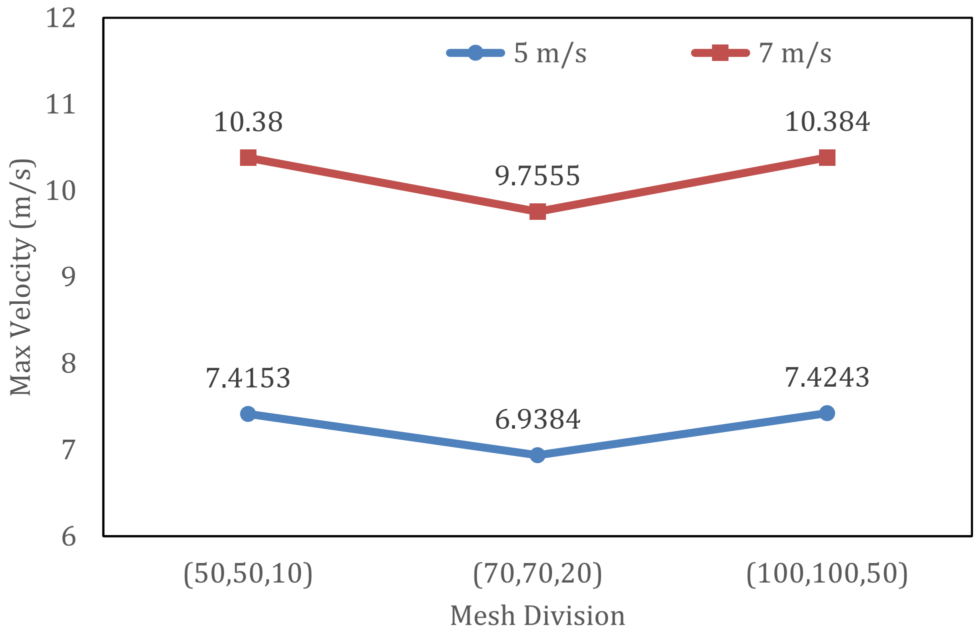


Figure 5: Maximum velocity for different mesh divisions at 5 and 7 m/s inlet velocity.

The pressure and velocity contours of the hydraulic filter were analyzed using different mesh resolutions to understand the impact on simulation accuracy. The contours were examined at flow velocities of 5 m/s and 7 m/s across three mesh divisions: low (50, 50, 10), medium (70, 70, 20), and high (100, 100, 50). The pressure contour (Figure 6) reveals how pressure varies within the hydraulic filter under different mesh resolutions. The pressure distribution in the low mesh division appears less refined, indicating potential inaccuracies in capturing localized pressure changes. As the mesh resolution increases to medium and high, the pressure contour becomes more detailed, showing a smoother and more continuous pressure distribution. This refinement suggests that higher mesh resolutions provide a more accurate representation of the pressure fields within the filter, especially at higher flow velocities.

Similarly, the velocity contour demonstrates the flow characteristics within the filter at various mesh resolutions. In the low mesh division, the velocity distribution is coarse, potentially missing critical flow dynamics, such as eddies or regions of stagnation. The velocity contour becomes more precise with medium and high mesh resolutions, highlighting detailed flow patterns and areas of higher velocity gradients. This increased resolution allows for a better understanding of the flow behavior within the filter. It enhances the ability to predict performance under different operational conditions. Comparing the pressure and velocity contours across the different mesh resolutions shows that higher mesh resolutions significantly improve the accuracy and detail of the simulations. The high-resolution mesh captures finer details in pressure and velocity fields, leading to more reliable performance predictions of the hydraulic filter. This comparison underlines the importance of selecting an appropriate mesh resolution to ensure accurate computational fluid dynamics (CFD) analysis, especially for applications where precision in pressure and velocity predictions is critical.

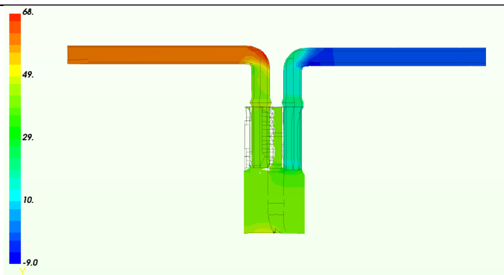
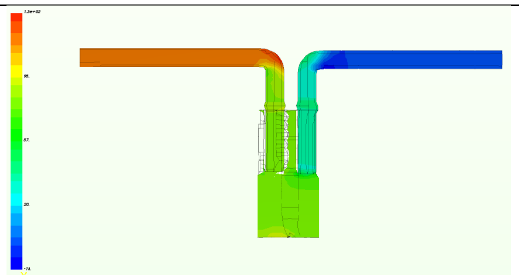
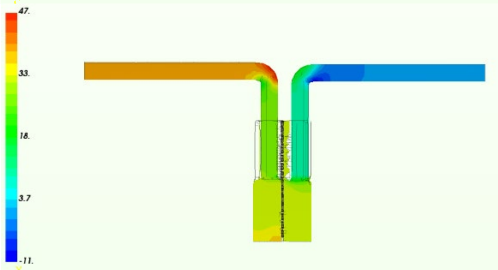
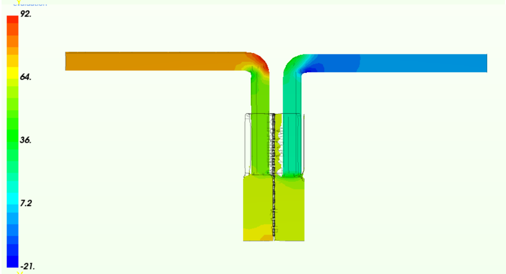
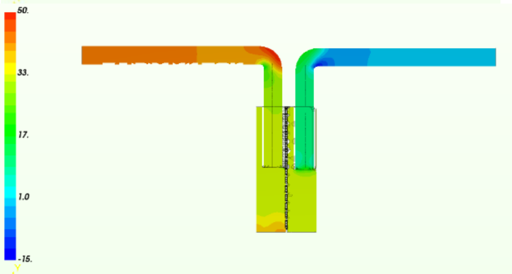
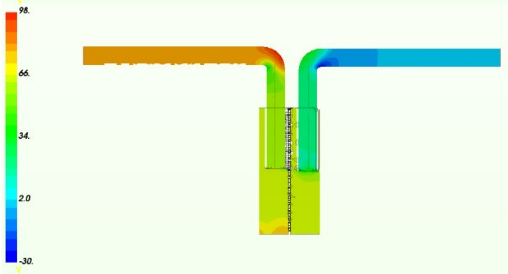
Mesh Division	5 m/s	7 m/s
(50,50, 10)		
(70,70, 20)		
(100,100, 50)		

Figure 6: Pressure contour inside the hydraulic filter.

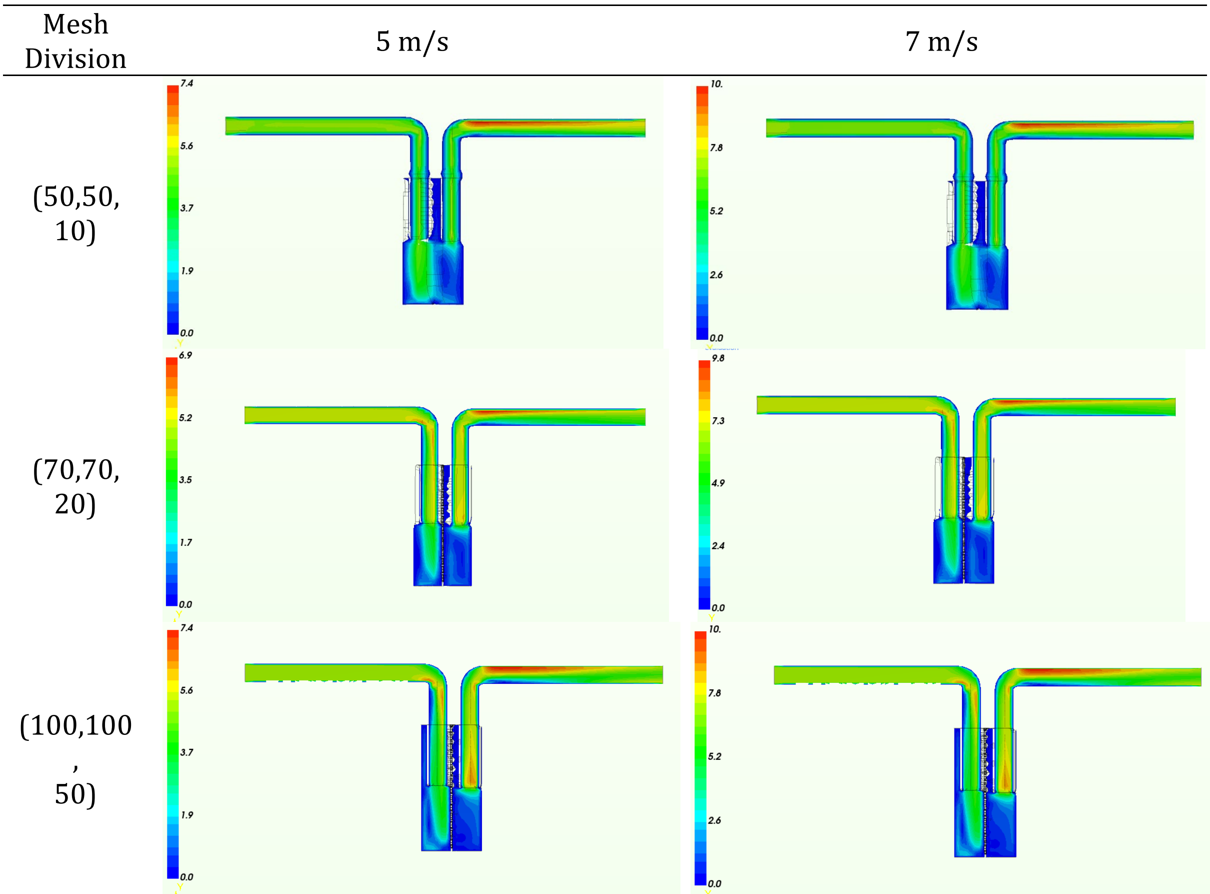


Figure 7: Velocity contour inside the hydraulic filter.

4. CONCLUSION

This study comprehensively analyzed an aircraft hydraulic filter's flow characteristics and performance using Computational Fluid Dynamics (CFD) with SimFlow 4.0. By constructing a detailed 3D model of the filter, meshing it at various resolutions, and employing the Reynolds-Averaged Navier-Stokes (RANS) method with the k-ω SST turbulence model, the research provided critical insights into the effects of mesh resolution on simulation accuracy. The findings revealed that mesh resolution significantly impacts the maximum pressure and velocity values within the filter. Specifically, the maximum pressure recorded was 67.817 Pa for the coarsest mesh, decreasing to 47.222 Pa and 49.748 Pa for finer meshes. The maximum velocity values ranged from 7.4243 m/s at a 5 m/s inlet velocity to 10.384 m/s at a 7 m/s inlet velocity for the finest mesh. The study highlighted that finer meshes yielded more accurate and detailed flow patterns, with the percentage deviation in maximum pressure between the coarsest and finest mesh at 5 m/s being approximately 26.67% and about 21.60%. These results highlight the importance of meticulous mesh selection to achieve precise simulation outcomes in hydraulic filter analysis.

ACKNOWLEDGEMENTS

The author sincerely thanks the Faculty of Mechanical Engineering & Technology for their invaluable support and resources throughout this project.

REFERENCES

[1] Gao, S., Fan, S., & Wang, W. The dynamic behavior and nonlinear characteristics of aircraft landing gear retraction mechanism considering atmospheric corrosion. Applied Mathematical Modelling, vol 135, (2024) pp. 438-456.
[2] Jiao, Z., Zhang, H., Shang, Y., Liu, X., & Wu, S. A power-by-wire aircraft brake system based on high-speed on-off valves. Aerospace Science and Technology, vol 106, (2020) p. 106177.
[3] Shengrong, G. U. O., Jinhua, C. H. E. N., Yueliang, L. U., Yan, W. A. N. G., & Hongkang, D. O. N. G. Hydraulic piston pump in civil aircraft: Current status, future directions and critical technologies. Chinese Journal of Aeronautics, vol 33, issue 1 (2020) pp. 16-30.
[4] Xu, B., & Cheng, M. Motion control of multi-actuator hydraulic systems for mobile machineries: Recent advancements and future trends. Frontiers of Mechanical Engineering, vol 13, (2018) pp. 151-166.
[5] Parveez, B., Kittur, M. I., Badruddin, I. A., Kamangar, S., Hussien, M., & Umarfarooq, M. A. Scientific advancements in composite materials for aircraft applications: a review. Polymers, vol 14, issue (22) (2022) p. 5007.
[6] Zhang, R. C., Yu, X., Hu, Y. L., Zang, H. J., & Shu, W. Active control of hydraulic oil contamination to extend the service life of aviation hydraulic system. The International Journal of Advanced Manufacturing Technology, vol 96, (2018) pp. 1693-1704.
[7] van Heerden, A. S., Judt, D. M., Jafari, S., Lawson, C. P., Nikolaidis, T., & Bosak, D. Aircraft thermal management: Practices, technology, system architectures, future challenges, and opportunities. Progress in Aerospace Sciences, vol 128, (2022) p. 100767.
[8] Zhang, W., & Zhang, Y. Numerical Simulation and Analysis of Hydraulic Filter Performance Using CFD. Journal of Fluids Engineering, vol 138, issue 12 (2016) p. 121101.
[9] Luo, X., & Chen, Q. Impact of Filter Design on the Performance of Hydraulic Systems: A CFD Approach. Journal of Hydraulic Research, vol 57, issue 2 (2019) pp. 225-235.
[10] Korkmaz, Y. S., Kibar, A., & Yigit, K. S. Experimental and numerical investigation of fluid flow in hydraulic filters. Journal of Applied Fluid Mechanics, vol 15, issue 2 (2022) pp. 363-371.
[11] Konstantinov, S. Y., Tselischev, D. V., Tselischev, V. A., & Tuk, D. E. Numerical Simulation of Hydraulic Screen Filter. In 2022 International Conference on Dynamics and Vibroacoustics of Machines (DVM) (2022) pp. 1-8.
[12] Ni, W. W., Bartholme, D., & Cass, M. The CFD analysis of pressure pulsation in the aircraft engine and control systems lubrication pump. SAE International Journal of Aerospace, vol 6, issue 2084 (2013) pp. 49-55.
[13] Si, W. Dynamic pressure analysis of civil aircraft hydraulic system return pipeline. In CSAA/IET International Conference on Aircraft Utility Systems (AUS 2022), vol. 2022, (2022). pp. 523-527.
[14] Sharizal Abdul Aziz, Mohd, Mohd Zulkifly Abdullah, and Chu Yee Khor. "Influence of PTH offset angle in wave soldering with thermal-coupling method." Soldering & Surface Mount Technology vol 26, issue 3 (2014) pp. 97-109.
[15] Tan, K. M. Computational Fluid Dynamics Analysis on the Road Bike Using Different Flow Models under Extreme Inlet Velocity. Advanced and Sustainable Technologies (ASET), vol 3, issue (1) (2024) pp. 62-70.
[16] Li, H., & Sansalone, J. Benchmarking Reynolds-averaged Navier–Stokes turbulence models for water clarification systems. Journal of Environmental Engineering, vol 147, issue 9 (2021) p. 04021031.

Conflict of interest statement: The author declares no conflict of interest.

Author contributions statement: Conceptualization; Methodology; Software; Formal Analysis; Investigation; Writing & Editing, Muhammad Aiman Rahimi Adli.