International Journal Of Engineering Research And Development
e- ISSN: 2278-067X, p-ISSN: 2278-800X, www.ijerd.com
Volume 21, Issue 1 (January, 2025), PP 251-254

Computational Fluid Dynamics Investigation of Flow Dynamics Within a Heat Exchanger Pipe

[1]Ravi Karan Tripathi,[2]Jayasheel I Harti,[3]Sahadeva G N,[4]Anjan Babu V A
[1]*Department of Mechanical Engineering, EPCET, Bengaluru*
[2]*Assoc. Professor, Department of Mechanical Engineering, EPCET, Bengaluru*
[3]*Professor, Department of Mechanical Engineering, EPCET, Bengaluru*
[4]*Assoc. Professor, Department of Mechanical Engineering, EPCET, Bengaluru*

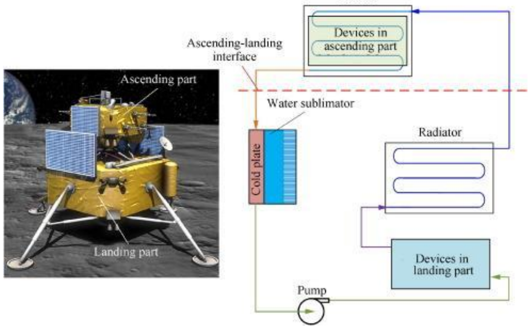
Abstract: - *The utilization of heat exchangers spans across various industries that includes automotive sector, power generation sector, refrigeration, and air conditioning systems. This study focuses on enhancing the performance of heat exchangers, specifically those constructed with aluminum (Al) and copper (Cu) materials, aiming to improve heat transfer rates while addressing pressure differentials, velocity distributions, and temperature profiles. The research employs the technique of Computational Fluid dynamics, utilizing a two-phase circular cross-section geometry developed in Gambit software. The K-Epsilon model within Fluent software facilitates the CFD calculations. The simulation allows for the determination of key parameters such as pressure, temperature, heat transfer rates, and velocity distributions along the pipe. The study considers water as the fluid flowing inside the pipe, with air serving as the coolant outside. A comparison was carried out between copper and aluminum materials for pipe construction, focusing on their performance and suitability. The study utilized numerical simulations to predict the behavior and characteristics of both materials under various conditions. To ensure the reliability of these numerical results, a thorough validation process was undertaken. This involved simulated data comparison with experimental results sourced from relevant literature. The validation process was meticulous, involving the extraction and examination of experimental data to serve as a benchmark for the numerical predictions. This experiment is not only highlighting the robustness of the numerical models used but also underscores the reliability of the conclusions taken from this analysis. Consequently, the study provides a comprehensive and validated circumstance of the performance differences between copper and aluminum materials in pipe construction.*

Date of Submission: 13-01-2025 Date of acceptance: 27-01-2025

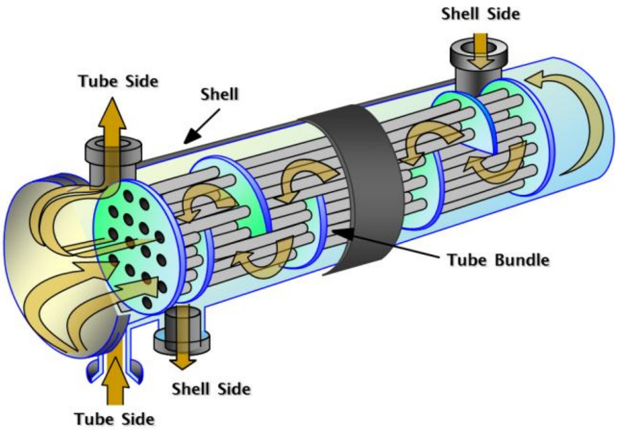
I. INTRODUCTION

Cooling systems play a pivotal role in maintaining the operational efficiency and longevity of various machines and equipment. Among these systems, radiators, condensers, and evaporators are integral components, each contributing to the overall thermal management of the machinery they serve. Of particular importance is the radiator, which serves the critical function of dissipating heat generated by engines.

In the context of automotive engines, the radiator assumes a vital role in maintaining optimal operating temperatures. As the engine operates, it generates significant amounts of heat, which if not effectively dissipated, can lead to overheating and mechanical failures. The radiator serves as the primary heat exchanger, that transfers thermal energy from the engine coolant to its surrounding environment through the method of convection and radiation.

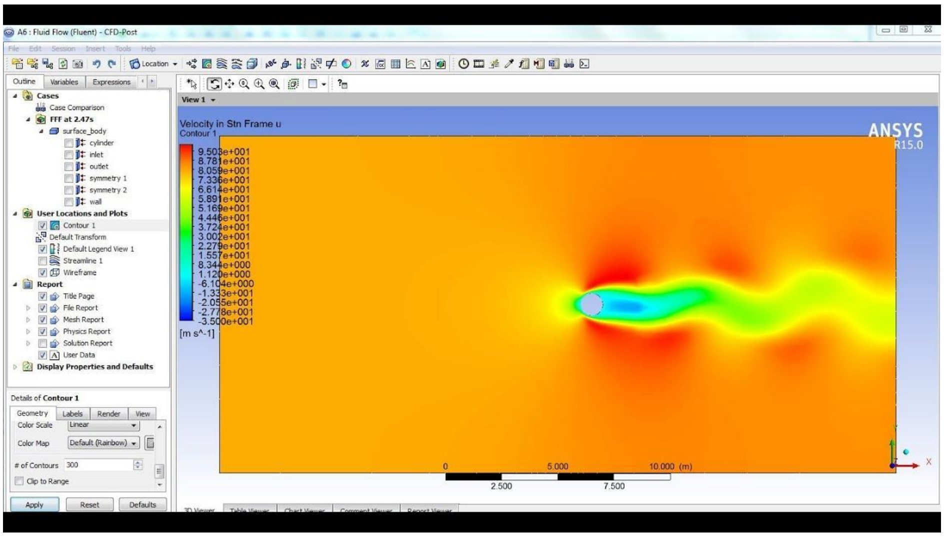


The efficient working of the radiator directly impacts the performance of the engine. Efficient cooling ensures that the engine operates within its designated temperature range, optimizing combustion efficiency and overall power output. Conversely, inadequate cooling can lead to engine overheating, reduced performance, and potentially catastrophic mechanical failures such as piston seizure or cylinder head warping. To maintain engines and machines in proper operating conditions, cooling systems utilize heat 2exchangers with water as a cooling medium to dissipate heat generated through the machinery. Heat exchangers play a crucial role in to the process by utilizing a fluid medium, typically a coolant, to decrease temperatures and prevent machine parts from melting. The coolant absorbs heat from the machinery and circulates through the heat exchanger and this is where heat transfer occurs through air or water flow over its surfaces.



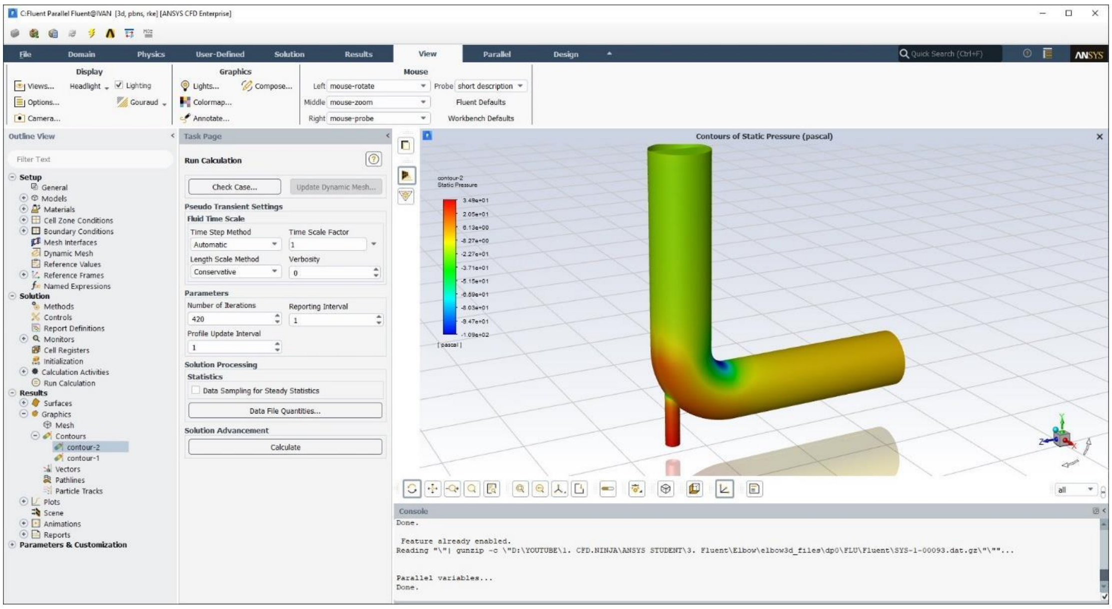
II. METHODOLOGY

The research undertaken in this study leveraged FLUENT, a computational fluid dynamics (CFD) software package, to tackle the complex balance equations governing fluid flow and heat transfer. The approach employed the control volume method, which involves discretizing the domain into finite control volumes and solving the governing equations within each volume. By doing so, the intricate partial differential equations describing fluid flow and heat transfer phenomena were transformed into simpler algebraic equations, which could then be efficiently solved computationally.

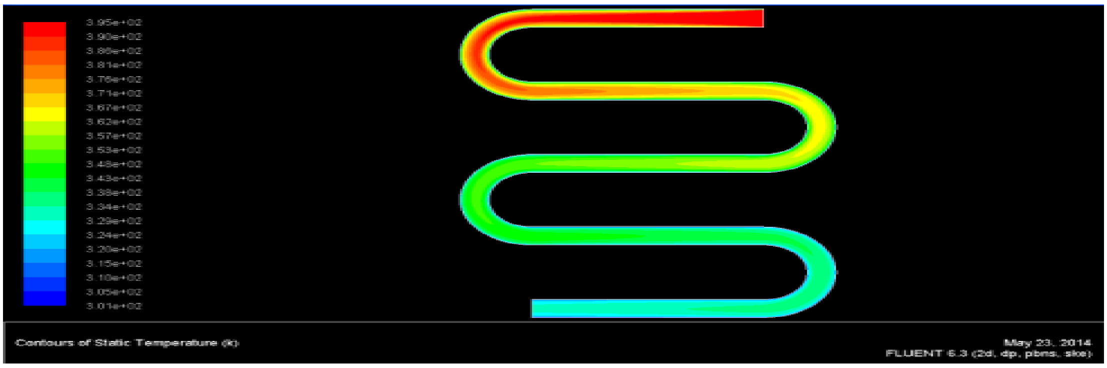


- To accurately represent the geometry of the system under investigation, initial modeling and meshing were performed using GAMBIT software. This involved creating a geometric representation of the system, refining it as needed to capture relevant details, and generating a computational mesh to discretize the domain into smaller elements. The mesh refinement process, often employing techniques such as successive ratio, ensured that the mesh resolution was appropriate for capturing the intricacies of fluid flow and heat transfer within the system.

Phase

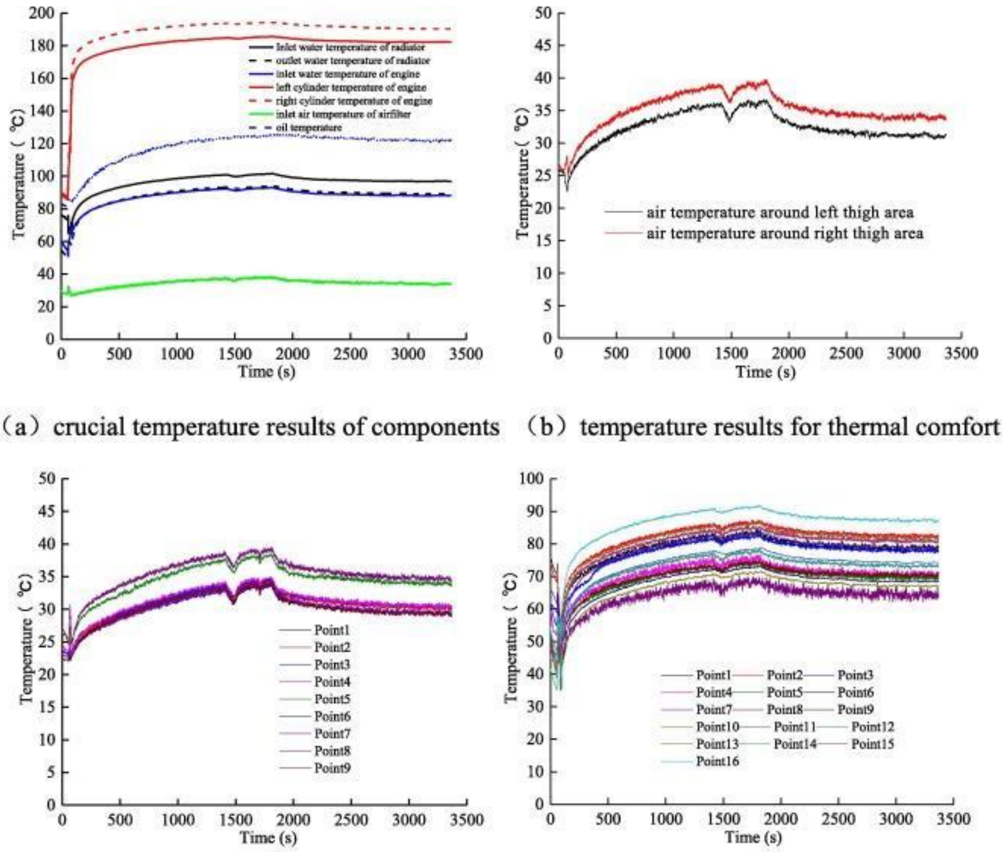


III. RESULTS



IV. CONCLUSION

Numerical simulations were conducted to analyze the fluid flowing through the radiator/heat exchanger, utilizing water as one of the working fluids and employing steady and unsteady flow models within the k-epsilon turbulence scheme. This approach enabled a comprehensive analyzing the fluid dynamics within the heat exchanger. Key observations focused on to the pressure, temperature, and velocity contours throughout the flow process within these heat exchangers. These contours provided valuable insights into the performance characteristics and efficiency of the radiator.



(a) crucial temperature results of components (b) temperature results for thermal comfort

(c) air temperature of radiator at inlet side (d) air temperature of radiator at outlet side

REFERENCES

[1] Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.

[2] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of shell and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering, china.

[3] Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in shell and tube heat exchangers for staggered tube arrangement", source book on Int.J. Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.

[4] Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of Shell and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (2008), pp. 548-552.

[5] Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD".

[6] Colebrook C.F.,"Turbulent Flow in Pipes with Particular reference to the Transition Region between the Smooth and Rough Pipe Laws", J. Inst. of Civil Engrs.N° 11,(1939), pp.133-156.