WWW.IJRPR.COM

International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

CFD Simulation of Waste Heat Recovery Unit System: A Case Study of Engine Exhaust

Ravishanakar Rawat*[1], Manjit Kumar Mishra[2]

[1,2,] Mechanical Engineering Department, APEX Institute of Technology, Jaipur
E-mail address: jec.pinkjaipur@gmail.com

A B S T R A C T

In the current study, the utilization of waste heat generated from a diesel engine was investigated using CFD modeling techniques. Waste heat recovery has become crucial in industrial applications due to the ongoing energy scarcity. Globally, researchers are employing both simulation and experimental techniques to conserve waste energy in thermal devices. This study focused on optimizing heat recovery from diesel engines using different fin configurations, leveraging CFD simulation methods and design of experiments for optimization. Key findings identified the parameters that enhance heat recovery through signal-to-noise ratio analysis and regression modeling. However, the study was limited to using water as the heat transfer fluid (HTF) due to funding constraints, despite the availability of more effective HTFs. The significance of this research lies in its contribution to energy conservation in an era of depleting conventional fuels. Engines, widely used in electricity generation and transportation, necessitate efficient waste heat recovery systems. Therefore, developing devices to harness this waste heat is imperative, driving ongoing research in this field.

Keywords: waste heat recovery, CFD, Fins, Heat Ex-changer, Signal to Noise ratio analysis, Regression Modeling

Introduction

In recent years, stringent emission norms have been imposed on NOx, CO, and particulate emissions from engines. Diesel engines, known for their high efficiency in energy conversion, are heavily relied upon for numerous heavy-duty vehicles, industrial equipment, marine engines, and various construction machinery. Their preference stems from low fuel consumption and reduced CO emissions. However, the emissions from these engines limit their usage, and while various researchers have worked to reduce these emissions, NOx and soot emissions from compression ignition (C.I.) engines remain high. To meet pollution guidelines, it is crucial to reduce the amounts of NOx and soot in exhaust gases. NOx emissions contribute to the formation of acid rain and photochemical smog, damaging structures, vehicles, and natural formations. Soot particles, on the other hand, penetrate the lungs, carrying mutagenic toxic hydrocarbons on their surfaces, causing severe health issues. These emissions not only affect the environment but also pose significant risks to human health, making their reduction a primary focus of ongoing research.

Investigations have demonstrated that precise air-fuel ratio (AFR) control can effectively reduce emissions of harmful exhaust gases such as carbon dioxide, nitrogen oxides, and unburnt hydrocarbons. However, evaluating AFR control systems on actual engines is both time-consuming and expensive. A computer-based simulation facility can significantly accelerate the development of new AFR control systems. Engine simulation models are not only useful tools for engine manufacturing but also essential for developing and analyzing engine control systems. A wide range of AFR control strategies, including advanced proportional-integral (PI), fuzzy logic, neural networks, adaptive, and predictive controls, can be effectively and conveniently evaluated using engine simulation facilities.

This paper addresses a crucial gap in the research: while various techniques have been developed to control exhaust emissions, fewer methods exist for managing waste heat generated during engine operation. Diesel engines produce both waste heat and exhaust gases, and this study focuses on waste heat recovery. A simple diesel engine was selected for this investigation, and computational fluid dynamics (CFD) simulations were conducted to explore the effects of different design configurations, such as varying fin designs and boundary conditions, on heat recovery. The optimization of these configurations aims to enhance the efficiency of waste heat recovery systems, thereby contributing to both energy conservation and emission reduction.

The emphasis on waste heat recovery is particularly relevant in the context of energy scarcity and the decreasing availability of conventional fuels. Diesel engines are widely used in electricity generation and transportation, where efficient waste heat recovery systems can make a significant impact. By harnessing waste heat, it is possible to improve the overall efficiency of diesel engines, reduce fuel consumption, and lower emissions, addressing both environmental and energy challenges. This study employs CFD modeling techniques to investigate the potential of different fin configurations in optimizing heat recovery from diesel engines. The findings from this research can inform the development of more effective waste heat recovery systems, contributing to sustainable industrial practices and environmental protection.

Literature Review

The literature review focuses on the various aspects of Waste Heat Recovery Systems (WHRS) to enhance efficiency and reduce emissions. Key points include the shape of WHRS units, the fluid flow range, experimental versus simulation methods, and response parameters. P. Liu et al. (2016) investigated a dual-flow WHRS with obstacles on a metallic safety plate, finding that the central region yielded optimal efficiency. The study highlighted that obstacles could enhance performance, suggesting further improvements for commercial use. M. Saeedinia et al. (2012) conducted numerical studies on surfaces with roughness, creating 60-degree angles on protective plates. The research showed strong agreement between experimental and numerical results, optimizing rib configurations for better performance. FaJiang, Cao Wei Wu et al. (2012) developed cost-effective WHRS for Bangladesh's winter conditions, utilizing local materials and comparing experimental data with theoretical estimations. Their work underscored the practicality of WHRS in specific regional contexts. Pranab Kumar Pal and Sujoy K Saha (2014) explored thermal coatings on solar panel collectors, using a transient numerical model for Jeddah, Saudi Arabia. The study concluded that nickel-tin based coatings provided optimal thermal performance.

Sagnik Pal et al. (2015) examined WHRS with mesh-type packing beds in single and double-pass configurations, finding that increased air mass flow rates enhanced thermal efficiency within a specific range. V. Singh et al. (2016) used genetic algorithms (GA) to optimize basic WHRS, considering variables like Reynolds number and tilt angle. Their findings indicated that GA could significantly improve thermal efficiency. O. Sadeghi et al. (2016) studied W-shaped ribs under single pass flow conditions, developing correlations for heat transfer and pressure drop. Their research showed that simulated roughness increased thermal performance. T.O. Oni and M.C. Paul (2016) investigated the impact of double-pass and external-recycle procedures on WHRS performance, demonstrating that external means to increase liquid flow rates significantly improved heat transfer efficiency. V. Gnielinski (1976) compared dual-pass level and V-ridged plate WHRS, finding that V-creased plates were 11-14% more efficient than flat plates. H.A. Mohammed et al. (2013) examined curved rib roughness, showing that rib-roughened paths improved thermal performance compared to smooth paths. W.H. Azmi et al. (2013) assessed three types of counter-flowing WHRS with different fin configurations, concluding that more fins increased thermal efficiency, with the highest efficiency observed at 85.9%.

L.S. Sundar et al. (2012) studied combined ribs and delta-winglet vortex generators (DWs) in WHRS, finding that larger DW attack angles resulted in higher heat generation. M.M.K. Bhuiya et al. (2012) compared double-pass finned and V-rippled plate WHRS, showing that V-rippled plates were more efficient. P. Sivashanmigam and S. Suresh (2012) used a particle swarm optimization algorithm to improve flat plate WHRS, achieving satisfactory adaptability and speed. Finally, A. Tohidi et al. (2012) presented a mathematical model for a single-pass flat-plate solar air collector, showing that improved designs outperformed traditional solar air heaters in thermal efficiency. This comprehensive review highlights the various innovative approaches and methodologies employed in optimizing WHRS for enhanced efficiency and emission reduction. The primary aim of this study is to develop a waste heat recovery device to capture and utilize the thermal energy generated by a diesel engine. The device is constructed using locally available materials, such as circular pipes and rods, to ensure cost-effectiveness. Due to limited funding, Computational Fluid Dynamics (CFD) is employed for the majority of the analysis. The study utilizes the signal-to-noise ratio to identify key input parameters that enhance waste heat utilization from the diesel engine.

CFD Modeling

For the CFD simulation, a single-cylinder direct ignition engine is selected. The study includes experimental validation to demonstrate the software's capabilities. Technical details of the diesel engine are provided in Table 1. The setup is integrated with a 5 kW electrical generator and a load bank, which can be adjusted as needed. The experimental validation in this study is conducted under 0% load conditions, allowing for a comprehensive comparison with the CFD simulation results. The focus on utilizing locally sourced materials and CFD modeling not only addresses the cost constraints but also ensures the practical applicability of the waste heat recovery device. This approach demonstrates how innovative techniques can be applied to optimize energy efficiency in diesel engines, contributing to broader efforts in energy conservation and emission reduction. The integration of experimental validation further strengthens the reliability of the simulation results, providing a robust foundation for future research and development in this area.

The genuine picture of test motor is available in figure 1, as found in figure the motor is water cooled and introduced at school studio. Bu-rate is introduced to gauge the fuel utilization during information recording. In chamber pressure estimation is finished utilizing piezoelectric transducer and information lumberjack. The approval is show great understanding however the top for investigation and recreation is contrast from one another and the explanation is that in useful the test rig is working from past certain years and that is the motivation to change the information from its maker evaluated information. What's more CFD is demonstrated utilizing maker information. In any case, in general the approval is very much contrasted and one another. The blunder among test and reproduced information is approx equivalent to 15% most extreme at top which is adequate for recreation work investigation.

Figure 1 Experimental Setup for present study

CFD Modeling Steps

CFD modeling is most important step for better simulation for waste heat recovery unit. In present study Ansys Fluent software is used for CFD modeling. In CFD three basic steps are required to complete the simulation and gather results. These three steps are following:

Pre-Processing

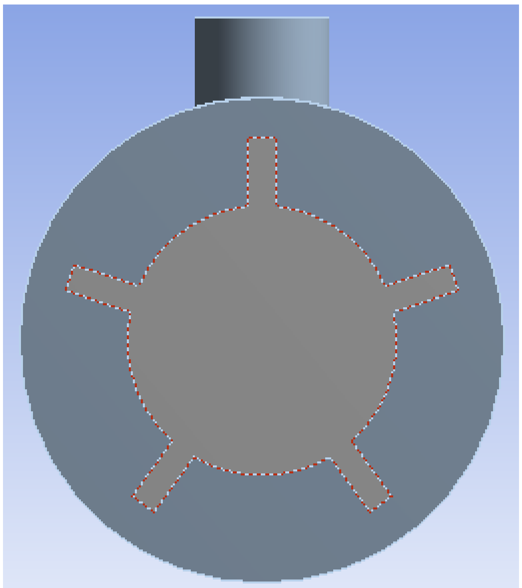
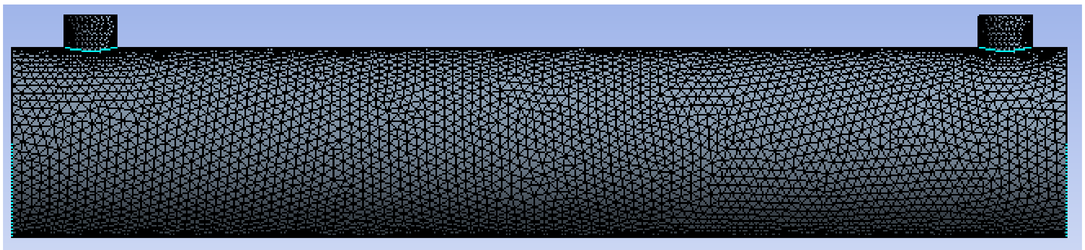
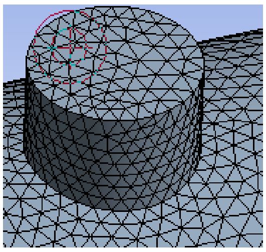
Solver

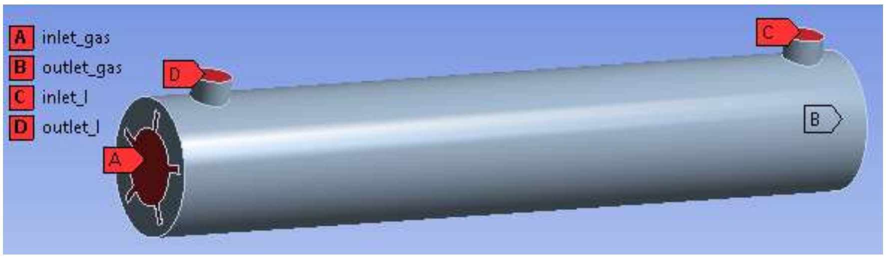
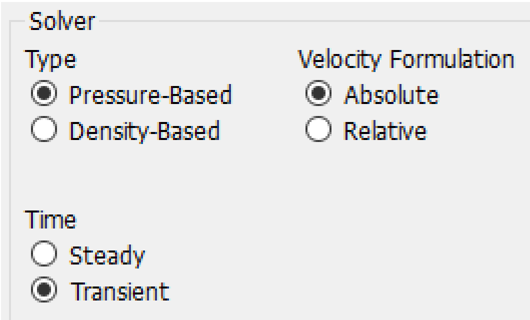
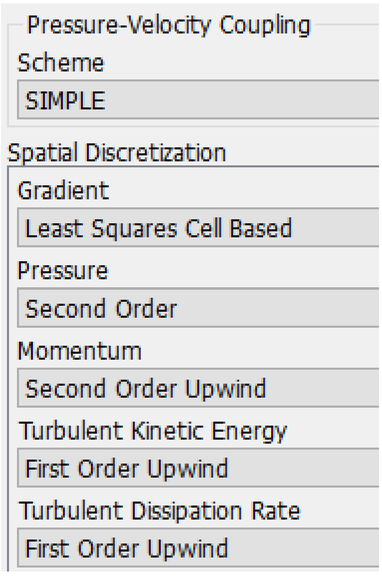
Post Processing

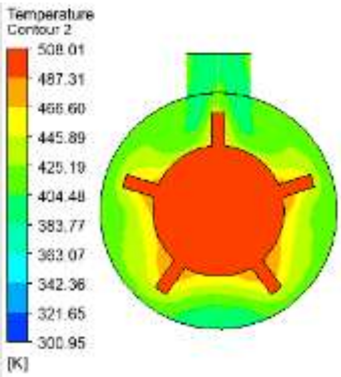
All three steps required to complete the CFD simulation all required steps are list out in table 1, which is present here.

Table 1 CFD Modeling Steps

Step-I: Geometry Modeling (Ansys DM)


Step-II: Meshing (Ansys ICEM CFD)



Step-III: Boundary Selection (Ansys ICEM CFD)

Step-IV: CFD General Settings

Step-V: Governing Equations
Energy Equation-Yes
Continuity Equation-Yes
Momentum Equation-Yes
Step-VI: Solution Methods

Step-VII: Fluid
Flue Gas and Water
Step-VIII: Results



Assumptions Required for CFD Simulation

In CFD some assumptions are required to complete the modeling in better way, in present study some assumptions are set by Researcher which are following

Flue gas is treated as in-compressible gas for Simulation

Water is treated as in-compressible

SIMPLE algorithim is selected

2-Eq turbulence model is selected

Structural limitations of the Device is neglected

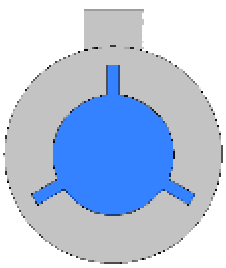
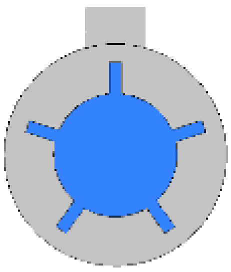
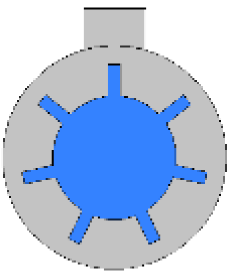
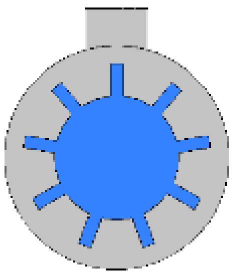
Radiation effect is neglected

Perfect insulation at outer walls are assumed

Results and Discussion

In present study four designs of the special type of Waste Heat Recovery Unit (WHRU) is selected. In these designs the inner pipe of the WHRU is having extended hollow fins which improve the heat transfer surface area. All four designs are list out in table 2.

Table 2 Four designs selected for CFD simulation of WHRU

Design-I (3 Fin WHRU)	Design-II (5 Fin WHRU)
	
Design-I (7 Fin WHRU)	Design-II (9 Fin WHRU)
	

Total length of the WHRU is 1000 mm having 180 mm outer diameter and 150 mm inner diameters. CFD visual analysis is performed for all four designs. Contours are made at different location of the WHRU for axial directions and show in figure 4.

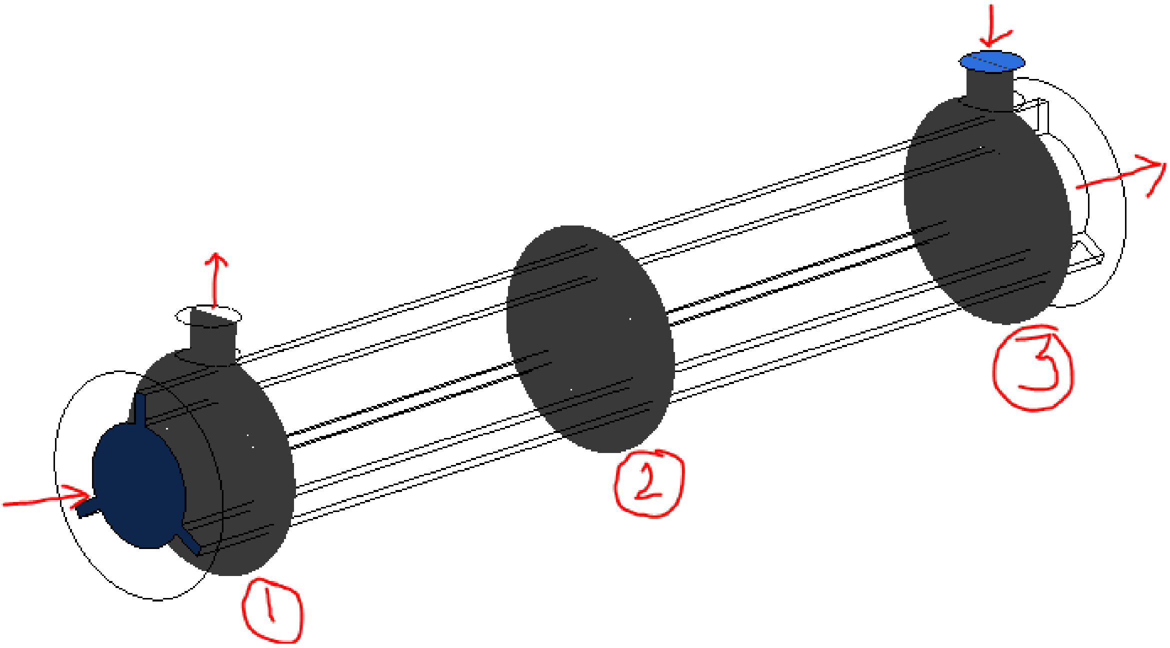
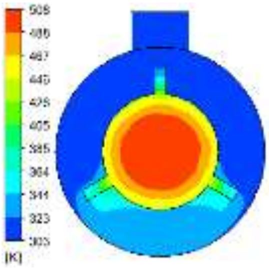
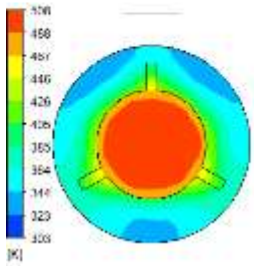
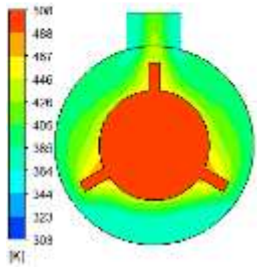
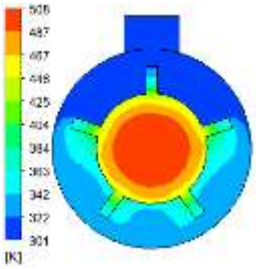
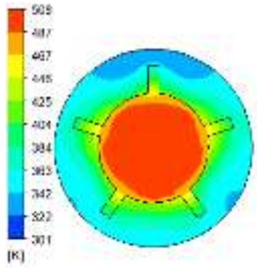
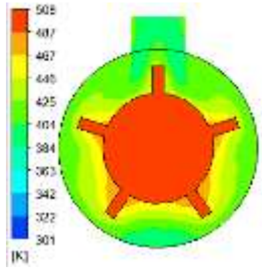


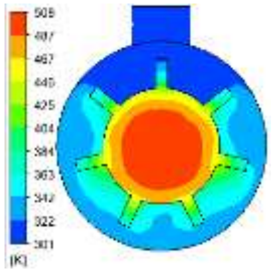
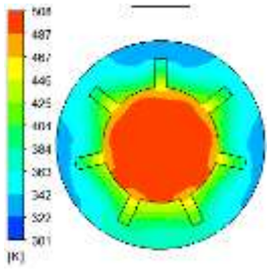
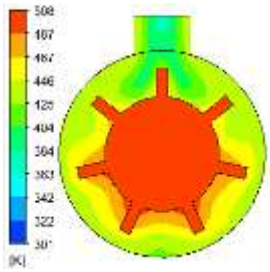
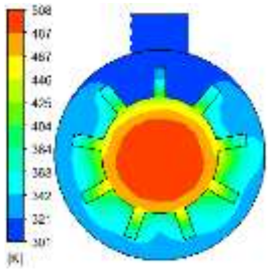
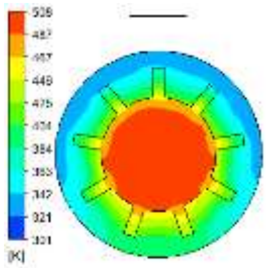
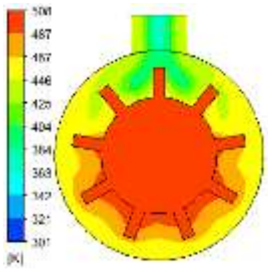
Figure 2 Three location where contour is making for analysis

Thermal Comparison

In thermal comparison analysis all four designs are compare with each other on the basis of temperature in WHRU. All results are list out in table 4. In table 4 inlet is represent the water side inlet cross section and outlet represent inlet section of the Flue gas or outlet section of the water side. Middle is the center cross section of the WHRU unit as shown in figure 2. In this section temperature range for all cases are treated same. Flue gas inlet temperature is 508 K having 0.5 m/s velocity and water has 303 K inlet temperature having 0.5 m/s velocity boundary conditions.

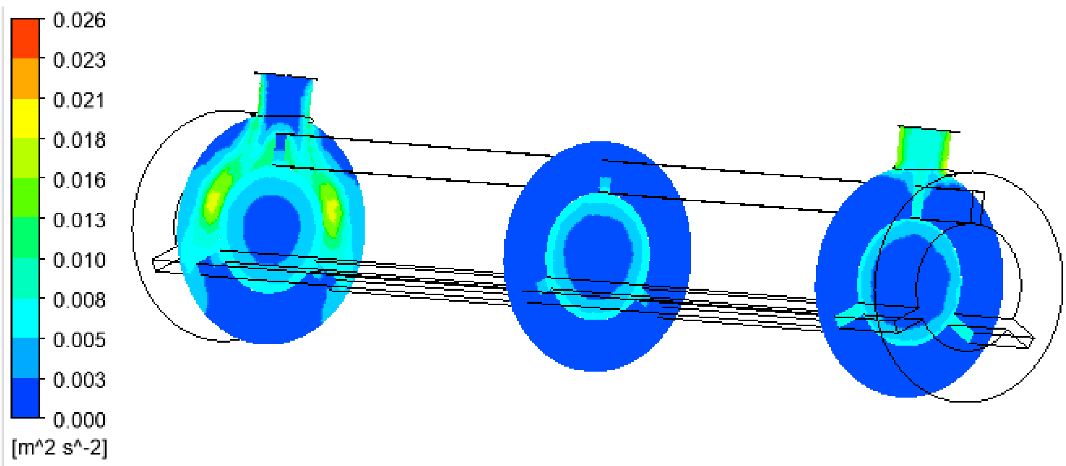
Table 3 Temperature Contours at different Location of WHRU

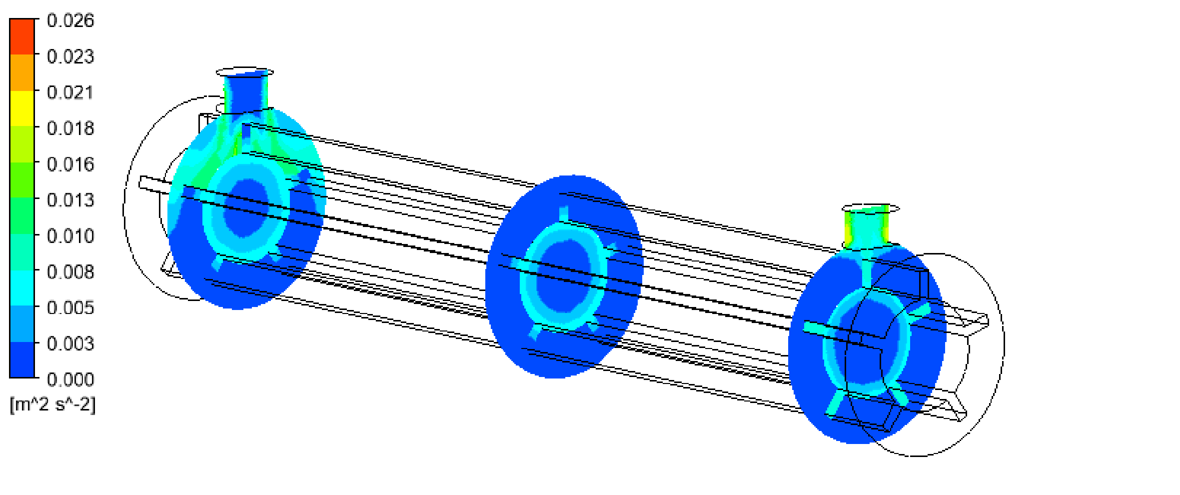
Design-I (3 Fin WHRU)		
Inlet	Middle	Outlet
		
Design-II (5 Fin WHRU)		
Inlet	Middle	Outlet
		

Design-III (7 Fin WHRU)		
Inlet	Middle	Outlet
		
Design-IV (9 Fin WHRU)		
Inlet	Middle	Outlet
		

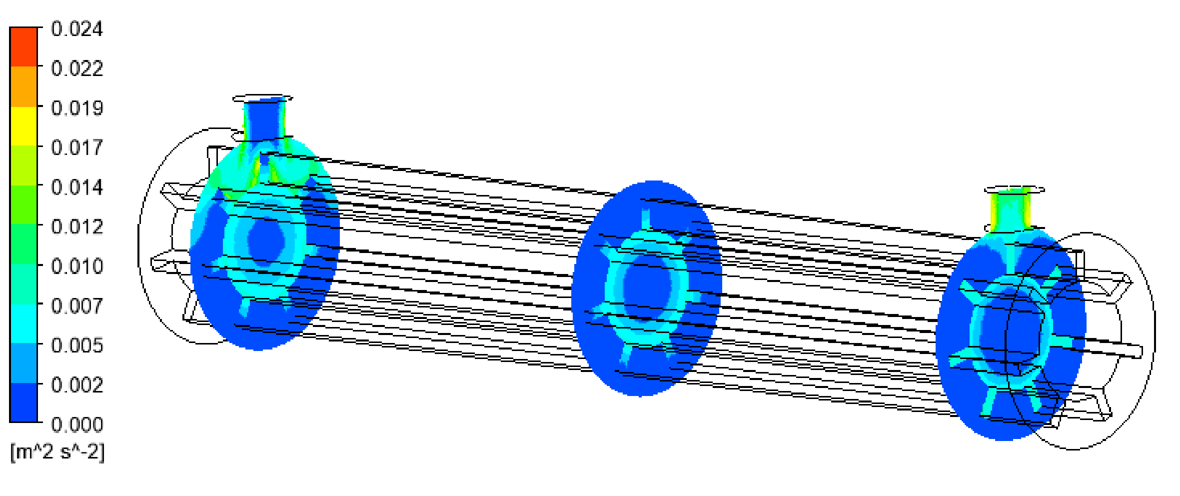
Turbulence Comparison (Turbulent Ki Energy)

Table 4 Turbulent Ki Energy Contours at different Location of WHRU

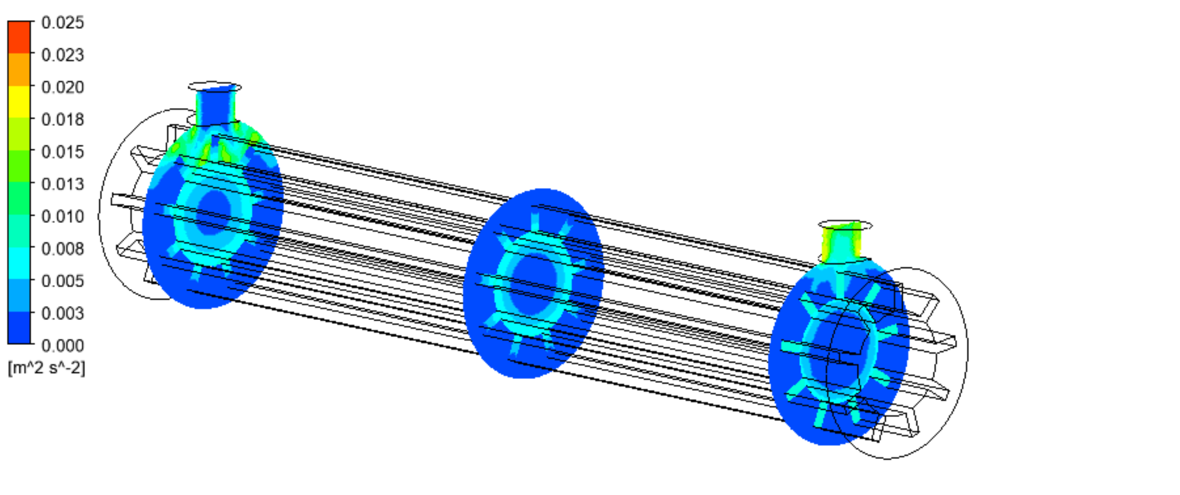
Design-I (3 Fin WHRU)

Design-II (5 Fin WHRU)



Design-III (7 Fin WHRU)



Design-IV (9 Fin WHRU)



As seen in table 3, temperature contours for all four designs are list out. In table 4 temperature contours show the difference among all fins configurations, the same analysis is show in table 4 for turbulent kinematic energy contours.

Conclusion

This study successfully demonstrates the potential for utilizing waste heat generated by diesel engines through the development of a waste heat recovery device using locally available materials. By employing Computational Fluid Dynamics (CFD) for simulation and analysis, the study identifies key optimization parameters to enhance heat recovery. The results show that integrating obstacles in a dual-flow waste heat recovery system can significantly improve thermal efficiency, aligning with findings from previous research. Despite the limitations of using water as the heat transfer fluid due to funding constraints, the study highlights the effectiveness of CFD modeling in optimizing the design of waste heat recovery systems. The experimental validation under 0% load conditions further confirms the reliability of the simulation results. This research underscores the importance of energy conservation, especially in the context of declining conventional fuel resources, and provides a foundation for future advancements in waste heat recovery technologies. The successful recovery and utilization of waste heat from diesel engines can contribute to reducing environmental pollution and improving energy efficiency in industrial applications.

References

[01]. Bui Hung Thang, Cao Van Quang, Van Trong Nghia, Phan Ngoc Hong, Nguyen Van Chuc, Ngo Thi Thanh Tam, Le Dinh Quang, Dao Duc Khang, Phan Hong Khoi, Phan Ngoc Minh, Thermal dissipation efficiency in a micro-processor using carbon nanotubes based composite, in: International Workshop on Advanced Material for New and Renewable Energy, Jakarta, 9–11 June 2009, pp. 138–143.

[02]. Mohan, R.. (2011). Experimental and CFD analysis of WHRS with base plate for CPU cooling. Journal of Mechanical Science and Technology. 25. 2003-2012. 10.1007/s12206-011-0531-8.

[03]. Chen, H. T., Tseng, H. C., Jhu, S. W., & Chang, J. R. (2017). Numerical and experimental study of mixed convection heat transfer and fluid flow characteristics of plate-fin WHRS. International Journal of Heat and Mass Transfer, 111, 1050-1062. https://doi.org/10.1016/j.ijheatmasstransfer.2017.04.065

[04]. Bin Li, Sora Jeon, Chan Byon,Investigation of natural convection heat transfer around a WHRS with a perforated ring,International Journal of Heat and Mass Transfer, Volume 97,2016,Pages 705-711

[05]. Park, Kuen & Kim, Hyun Jung & Kim, Dong-Kwon. (2014). Experimental Study on Natural Convection From Vertical Cylinders With Branched Fins. Experimental Thermal and Fluid Science. 54. 10.1016/j.expthermflusci.2014.01.018.

[06]. Singh, Pankaj & Patil, Anil. (2015). Experimental investigation of heat transfer enhancement through embossed fin heat sink under natural convection. Experimental Thermal and Fluid Science. 61. 10.1016/j.expthermflusci.2014.10.011.

[07]. Guei-Jang Huang, Shwin-Chung Wong, Chun-Pei Lin, Enhancement of natural convection heat transfer from horizontal rectangular fin arrays with perforations in fin base, International Journal of Thermal Sciences, Volume 84, 2014, Pages 164-174

[08]. S. Siddiqa, M.A. Hossain, Suvash C. Saha, The effect of thermal radiation on the natural convection boundary layer flow over a wavy horizontal surface, International Journal of Thermal Sciences, Volume 84, 2014, Pages 143-150

[09]. Amer Al-Damook, FhadSaleh&Alkasmoul 2018, 'Heat transfer and airflow characteristics enhancement of compact plate-pin fins heat sinks', Propulsion and Power research, vol. 7, no. 2, pp. 138-146.

[10]. Nguyen, Manh & Bui, Thang & Trinh, Pham & Hong, Phan & Nguyen, Tuan & Chuc, Nguyen & Le, Dinh & Hong Khoi, Phan & Minh, Phan. (2016). Thermo-mechanical properties of carbon nanotubes and applications in thermal management. Advances in Natural Sciences: Nanoscience and Nanotechnology. 7. 025017. 10.1088/2043-6262/7/2/025017.

[11]. Bui, Thang & Hong, Phan & Trinh, Pham & Chuc, Nguyen & Tam, Ngo & Hong Khoi, Phan & Minh, Phan. (2010). Simulation of thermal dissipation in a µ-processor using carbon nanotubes based composite. Computational Materials Science. 49. 10.1016/j.commatsci.2010.01.027.

[12]. ErginBayraka, Ali BahadırOlcayb& Mustafa FazılSerincanc 2018, 'Numerical investigation of the effects of geometric structure of microchannel heat sink on flow characteristics and heat transfer performance', International Journal of Thermal Sciences, https://doi.org/10.1016/j.ijthermalsci.2018.08.030

[13]. E. C. Silva, Á. M. Sampaio, and A. J. Pontes, "Evaluation of Active WHRS Design under Forced Convection—Effect of Geometric and Boundary Parameters," Materials, vol. 14, no. 8, p. 2041, Apr. 2021.

[14]. Kwak, DB., Kwak, HP., Noh, JH. et al. Optimization of the WHRS with a concentric cylinder and triangular fins installed on a circular base. J Mech Sci Technol 32, 505–512 (2018). https://doi.org/10.1007/s12206-017-1252-4

[15]. Yue-Tzu, Yang &Huan-Sen, Peng 2009, 'Investigation of planted pin fins for heat transfer enhancement in plate fin heat sink', Microelectronics Reliability, vol. 49, no. 2, pp. 163-169.