Academic Guru Publication House

Technological Advancements : Research & Reviews
Volume 1
Year: 2021

AG PH Books

To analyze the Temperature distribution of Helical Fin Profile of engine through CFD investigation

Saurabh Bachakaiyan[1*]

[1]*Research Scholar, Department of Mechanical Engineering, Sagar Institute of Research and Technology, Bhopal*

Abstract

In order to convert the non-circular ducts into circular pipes, hydraulic diameter is used. This phrase, which refers to a circular tube, may be used to compute a variety of different things. The area of the cross-section and the perimeter of the wetted cross-section are represented by A and P in the equation below. 2D design is changed to 3D design for this study, and CFD results have been compared. Within the fins, water as well as ethylene glycol was also circulated, and also the effects were examined. As a result, the current study contributes to revealing and expanding the potential of helical fins with ethylene glycol in improving total heat exchanger performance while also lowering material costs for design engineers as well as manufactures

Keywords: CFD, internal combustion engine, FLUENT Software, helical fin.

1. INTRODUCTION

After the fuel has burnt up, there is additional heat created by the friction between the moving elements of the engines. Only around 30% to 35% of the energy released is actually used for labour. The remainder (65% to 70%) of the engine was removed to prevent the pieces from melting. Limited big vehicles employ water-cooling methods and virtually all two-bike engines use air-cooled engines, which have advantages such as reduced weight and a smaller space need, making them the sole choice

* ISBN No. 978-81-953278-8-1

for this purpose. While it's important to keep the heat generated during combustion in an internal combustion engine at a high level in order to improve its thermal efficiency, removing some of that heat is necessary in order to protect engine components from thermal damage. Fins, which are externally stretched surfaces, are used to enhance heat transmission in an air-cooled engine's cylinders. For IC engines, fins are a common method of cooling. The outer extended surfaces of the engine cylinder fins are designed to evacuate heat from the area. The quantity of heat transferred by an item is determined by its conduction, convection, and radiation properties.

1.1. Fins in engine

Conduction heat transmissions from inside and convection dissipation into an ambient at T are both supported by a temperature gradient in the x direction. Conduction and convection are the primary modes of energy transmission between a solid and its environment. [1]

$$Q = A_s(T_s - T_\infty)$$

Where h= convection heat transfer coefficient

A_s= Heat transfer area of the surface

There are only two ways to improve the “heat transfer coefficient” when the temperatures TS and T are fixed due to design considerations: (i) to increase the surface area, A ; (ii) to enhance the convection coefficient h. To improve heat transfer rates in situations when boosting h is either feasible or cost-effective, such as when building a pump or fan or replacing an existing fan with a bigger one, increasing the surface area may be used. [2]

1.2. Material used in fins

To achieve maximum heat transfer rates, the best material to choose is one with the highest thermal conductivity and sufficient thickness. Thermal resistance, corrosion resistance, as well as material weight all seems to be critical considerations, particularly at high temperatures. [3]

Aluminum alloy is the most often utilised material for fin production. Aluminum alloys 6063 and 7068 are now being used to replace the current aluminium alloys.

Table 1 Chemical composition of AL alloy

Element	**Weight %**
C	0.29
Mn	0.9
P	0.03
S	0.04
Si	0.15-0.30
Mo	0.45-0.60
Ni	0.05

1.3. Objectives of the Study

Both liquid and air cooled engines may be used to dissipate heat. Although liquid cooled engines are more complicated and need more area and control, air cooled engines are more often used in smaller vehicles because of these difficulties. The following are the goals of this research:

- CFD result will be validated with the selected base paper.
- The 2d design will be converted in 3d and the CFD result of both will be brought under comparison.
- The 3d design will be modified and fins design will be converted into spiral shape
- Water will be passed through the spiral fins.
- Water and ethylene glycol will be circulated inside the fins and both the result will be compared.

2. LITERATURE REVIEW

(Zargoushi et al., 2020) [4] ANSYS FLUENT is used in this work to construct a CFD model of a plate-fin heat exchanger in a gas refining firm to better understand transport phenomena, particularly phase change. The computational geometry takes into account the cold box's flow channels, ducts, and passages. As a result of an excessive rise in the number of computational grids in the numerical domain, the porous media approach is introduced in the computational domain. Analysis of chemical species in a cold box is done using a combination of CFD and flash calculations (FC). "Local thermal equilibrium (LTE-non mass), local thermal equilibrium (LTE) and local thermal non equilibrium (LTNE)" between the porous media and fluid flow with mass transfer are examined in the modelling of an industrial cold box in this research.

(Santosa et al., 2019) [5] CO_2 gas coolers' total heat transfer coefficients were studied using experiment and CFD simulations (CFD). For the total heat transfer coefficient, the CFD model predicted accuracy with a maximum error of 9 percent compared to CFD predictions. Experimental and computer modelling evidence suggests that optimizing the gas cooler circuit design may boost performance by up to 20% when comparing the two different types of gas coolers. 8 percent more heat is transferred through the gas cooler's fins when the first and second rows of tubes are sliced horizontally.

(Zhang et al., 2019) [6] The finned-tube CO_2 gas cooler is a critical component of a refrigeration system and so must be extensively examined. Computational Fluid Dynamics (CFD) models and simulations are used to forecast and analyse the CO_2 and air fluid velocity fields, temperature profiles, and heat transfer characteristics under various operating situations. Conventional empirical correlations can't reliably estimate the local heat transfer coefficients of both air and refrigerant. CFD modelling can. Based on a CFD model, this research examines various operational parameters on the heat transfer coefficients and temperature profiles of working fluids in a finned-tube CO_2 gas cooler.

(Effendi et al., 2018) [7] “Natural convection around hollow hybrid fin heat sinks” is the focus of this investigation. Hollow pin fins are concatenated with radially-placed plate fins in a staggered array in the HHFHS. A commercial CFD software tool was utilised to create 3-D computational thermal models, which were then correlated to forecast Nusselt numbers in the vicinity of HHFHSs.

(Lindqvist & Næss, 2018) [8] In this paper, a computational fluid dynamics model of helically wrapped fin tube bundles is presented and shown for its capacity to forecast thermal-hydraulic performance. For four distinct fin tube designs, two with plain fins and two with serrated fins, a consistent validation against experimental data is provided. With a maximum RMS error of 13.8 percent and 14.4 percent, the predicted heat transfer and pressure drop measurements are within or very near to the experimental uncertainty. When three fin efficiency models are evaluated using their predicted temperature distribution, it is shown that correction equations might be significantly incorrect for tall plain fins.

(Effendi & Kim, 2017) [9] Computational and experimental studies study the impact of orientation on the thermal performance of hybrid fin heat sinks (HFHSs) under natural convection. As an example of an HFHS, a hollow and solid hybrid fin heat sink is compared to a pin fin heat sink of the same size (PFHS). In the “hollow hybrid fin heat sink”, perforations in the area of the fin bases are found in the staggered array of hollow pin fins merged with radially-oriented plate heat sinks. The solid hybrid fin heat sink is made up of a staggered array of solid pin fins and extruded radially-oriented plate fins. Using CFD models of the “HHFHS, the SHFHS, and the PFHS”, we study the thermal performance implications of orientations ranging from zero to 180 degrees.

(Liu et al., 2017) [10] To construct a plate-fin heat exchanger for the hydraulic retarder, a theoretical optimization was performed. In order to increase the original heat exchanger's performance, CFD simulation and multi-objective optimization were used in conjunction. Multi-objective optimization was used to the optimizations of the friction factor f and Colburn factor j because of the competing goals. The form of the heat exchanger was optimized using the second generation “Non-Dominated Sorting Genetic Algorithm (NSGA-II)”.

(S. Singh et al., 2017) [11] Thermal energy system design management is an important aspect of establishing fundamental designs that fulfil large-scale user demand under certain operational parameters. Thermal energy systems using fin and tube heat exchangers are becoming more popular, thanks in part to the increasing attention being paid to their design and development. A cost-effective operation may be achieved with enough performance data from various fin designs. It is the goal of this study to investigate three different fin patterns, namely rectangular, polynomial and sinusoidal, in a fin and tube heat exchanger. Fin and tube heat exchanger models with varied fin patterns are constructed to explore the fin pattern's effect on heat transfer and pressure loss data.

3. RESEARCH METHODOLOGY

3.1. Steps of working

- Design and modeling in CAD software according to the selected base paper.
- Further converting the CAD File in .step format for importing it in ANSYS Fluent work bench.
- Designating the name choice to the various parts.
- Meshing is used to carry out the simulation procedure.
- Creating appropriate boundary conditions based on the chosen base paper.
- Assigning the qualities of the material
- Creating the ideal environment for the CFD analysis technique.
- After the simulation work is completed, evaluating the outcomes.

3.2. Material properties

Table 2 Thermal properties of water

Properties	**Value**
Thermal conductivity($Wm^{-1}K^{-1}$)	0.6
Density (Kg/m^3)	998.2
Specific heat(J/KgK)	4182

Table 3 Thermal properties of ethylene glycol

Properties	**Value**
Thermal conductivity($Wm^{-1}K^{-1}$)	0.252
Density (Kg/m^3)	1111.4
Specific heat(J/KgK)	2415

Table 4 Thermal properties of aluminum

Properties	**Value**
Thermal conductivity($Wm^{-1}K^{-1}$)	202.4
Density (Kg/m^3)	2719
Specific heat(J/KgK)	871

Table 5 Number of cases

Case-1	Validation 2d design
Case-2	3D design
Case-3	Spiral fins, solid
Case-4	Spiral hollow fins with water flowing through it
Case-5	Spiral hollow fins with ethylene glycol flowing through it

3.3. Model design

The fin has a rectangular shape with a length of 26 mm, a width of 65 mm, and a thickness of 3 mm.

Table 6 Dimensional specifications of IC engine

Engine Specifications	**Dimensions(m)**
outer diameter	0.062
bore diameter	0.050
thickness	0.006
No of fins	8
Length of fins	0.026
thickness of the fins	0.003
Pitch length b/w fins	0.004

3.4. Meshing

"Computer-aided engineering (CAE)" is a simulation method in which meshing is a key component. The precision, convergence, and speed of the solution are all affected by the mesh. As a consequence, the time it takes to produce a mesh model is often a significant percentage of the time it takes to get CAE results back. It follows that improved meshing tools and automated meshing methods lead to a better solution.

3.5. Boundary condition

- "Constant and homogeneous convective heat transfer coefficient over the entire fin surface (Taken as 30 W/m^2 K)"
- Expansion process is selected for analysis
- Engine inside temperature is taken as 1387.34 °C.
- Steady state condition is selected
- Radiation from the fin surface is negligible
- Ambient temperature is 27°C.
- For hollow fins cases velocity inlet condition is taken with 0.5 m/s velocity.
- Outlet is taken as constant pressure outlet.

4. RESULTS AND DISCUSSION

Calculated results are used to estimate heat transfer coefficients for different configurations based on Newton's cooling law using numerical temperature field computations Initial conditions, such as input temperatures and mass flow rates of the working fluids, are assumed to be the same for all simulations, regardless of fin spacing. Afterwards, the results are studied and evaluated in terms of the process phases.

4.1. Case-1 Validation 2d design

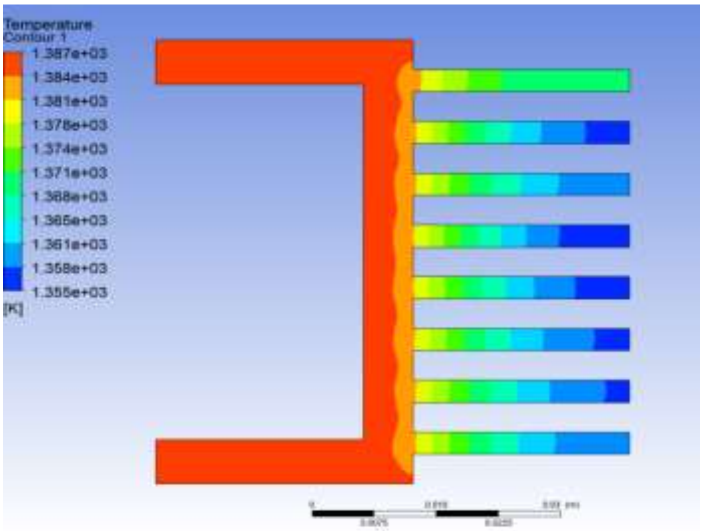

Figure 1: Temperature variation at surface

Above figure shows variation of temperature in engine because of fins. And the maximum and minimum temperature is 1387 and 1355 K respectively.

3D design

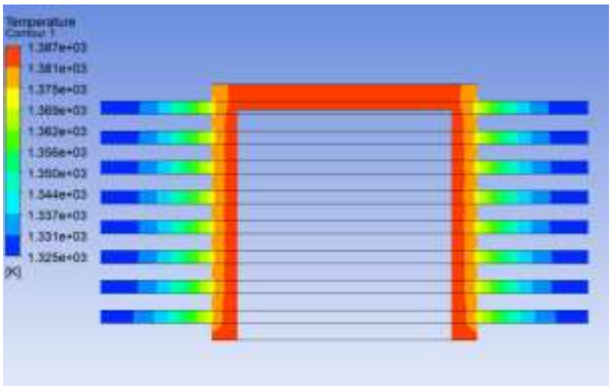

Figure 2: temperature variation at mid plane

4.2. Spiral fins, solid

Shape of fins are modified in spiral shape and the variation in temperature is shown in below figure. Here maximum and minimum temperature is found 1387 and 1318 K respectively. As it shows the lower temperature is less as compare to the above design results which shows spiral shape is better in heat transfer rate.

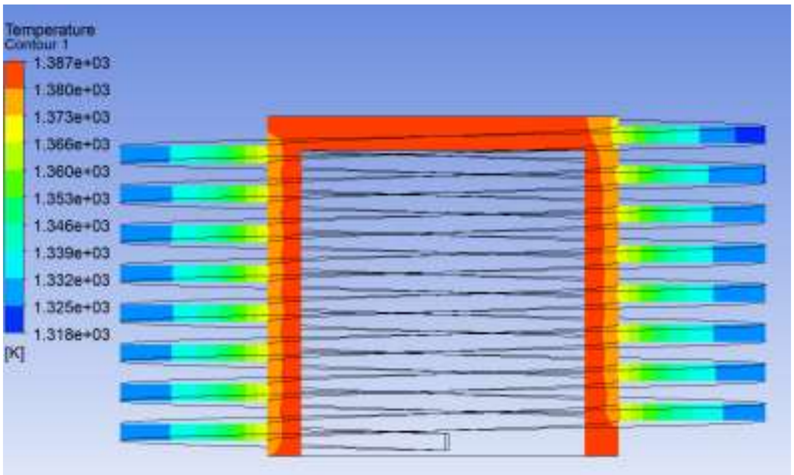

Figure 3: Temperature variation at mid plane

4.3. Spiral hollow fins with water flowing through it

For better cooling of engine through fins, fins are modified in spiral shape and water is flowed through them, in which water can absorb heat. Below figure shows variation in temperature in case of hollow fins with water flowing through it, and the maximum and minimum temperature is found 1387 and 301.6 K respectively.

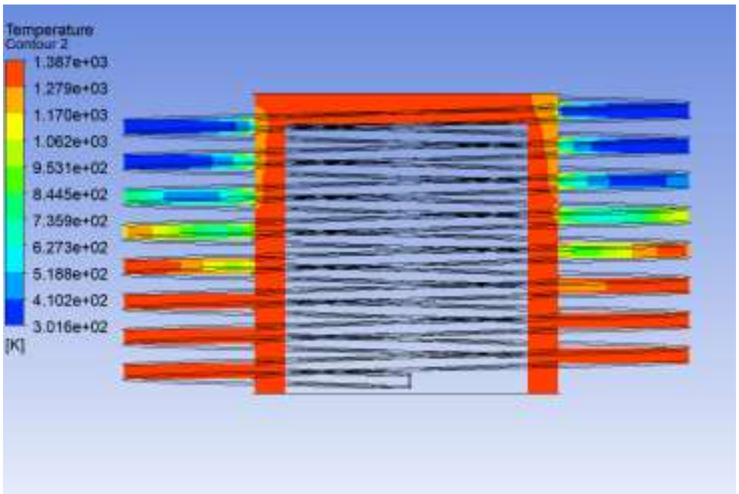

Figure 4: temperature variation at mid plane

Figure 5 shows velocity of flow of water through fins. Water is flowing from upward to downwards direction so the heat absorption rate of water will be high at the top of engine. As it shows water is flowing linearly with almost constant velocity.

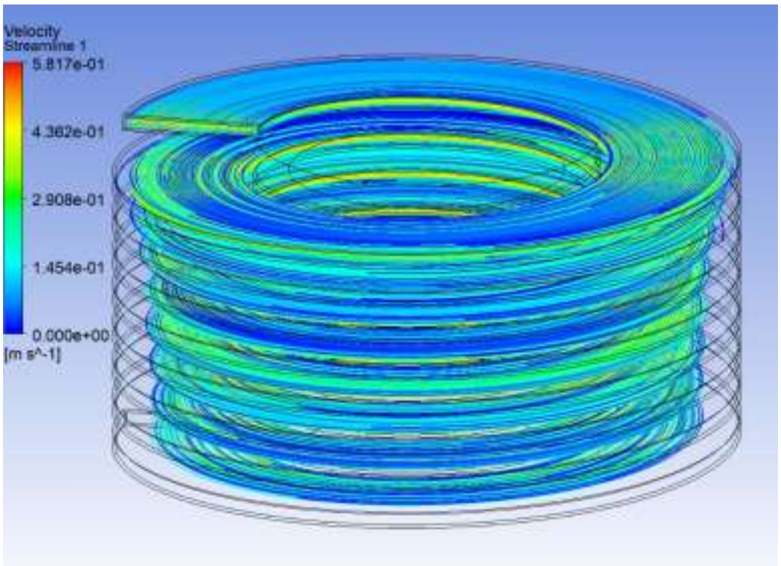

Figure 5: velocity flow of water

4.4. Spiral hollow fins with ethylene glycol flowing through it

Figure 6 shows variation in temperature at mid plane of engine having hollow fins with flowing ethylene glycol inside them. Maximum and minimum temperature is found 1387 and 308.2 K respectively.

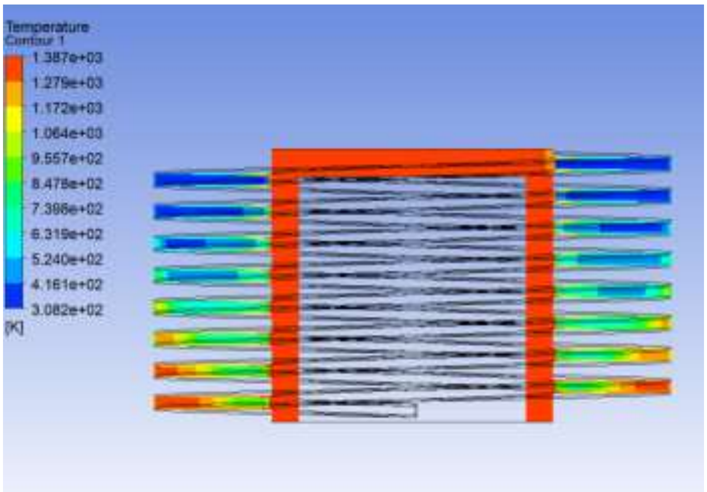

Figure 6: temperature variation at mid plane

Figure 7 shows velocity of flow of water through fins. Ethylene glycol is flowing from upward to downwards direction so the heat absorption rate of ethylene glycol will be high at the top of engine. As it shows water is flowing linearly with almost constant velocity.

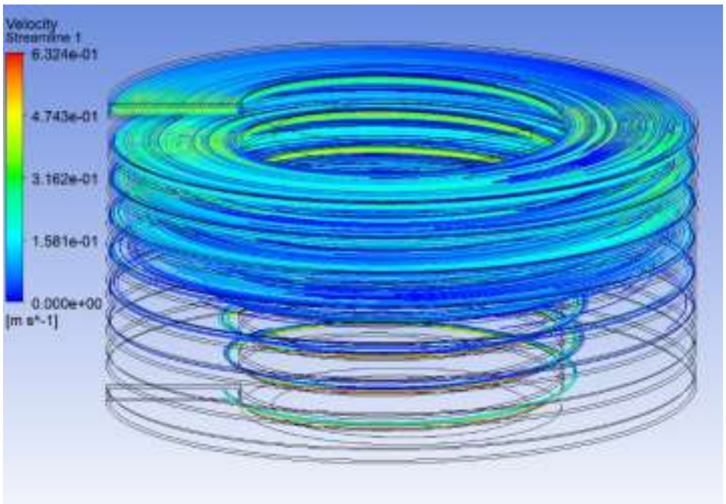

Figure 7: velocity flow of ethylene glycol

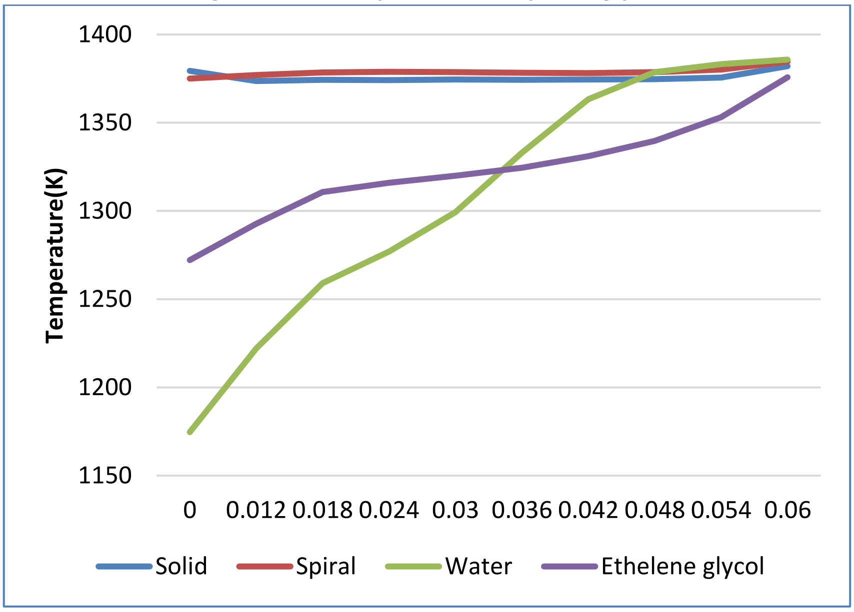

Figure 8: Line graph in all cases with respect to temperature

Above figure shows graph of temperature with respect to vertical length of engine. A line is selected near the fins inside the engine and the variation of temperature is noted down in all the cases at same location which can be seen in above figure. Fins having hollow design with flowing of fluid perform better as compare to solid fins. And ethylene glycol keeps the temperature of engine much lower as compare to all other cases.

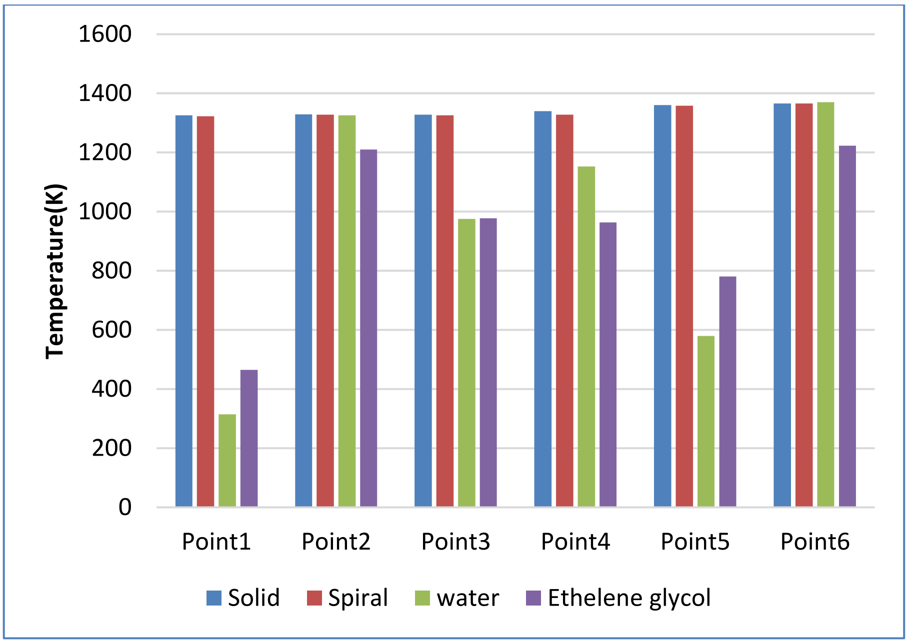

Figure 10: velocity flow of ethylene glycol

6 points are selected inside the fins and the temperature is noted at these points in all 4 cases which can be seen in above mentioned figure. As the figure shows, fins having solid design show almost same results at all the points whether it's simple solid or spiral solid. But fins having hollow shape with flow of fluid through them keep the temperature cool. Lowest temperature is observed in case of hollow fins having ethylene glycol flowing through it.

5. CONCLUSION

The space, maintenance, and high cost of a liquid-cooled engine are all taken into account in this analysis. The results of water and ethylene glycol are compared in the present study. After comparing the results, it was observed that water and ethylene glycol shows better results in comparison with solid and spiral geometry. After selecting the six different points, it can be concluded that water and ethylene glycol are showing better results and among them, ethylene glycol absorbs more heat and shows excellent results.

6. References

ADDIN Mendeley Bibliography CSL_BIBLIOGRAPHY [1] V. Singh, O. Abdelaziz, V. Aute, and R. Radermacher, "Simulation of air-to-refrigerant fin-and-tube heat exchanger with CFD-based air propagation," Int. J. Refrig., vol. 34, no. 8, pp. 1883–1897, 2011, doi: 10.1016/j.ijrefrig.2011.07.007.

[2] L. Sheik Ismail, C. Ranganayakulu, and R. K. Shah, “Numerical study of flow patterns of compact plate-fin heat exchangers and generation of design data for offset and wavy fins,” Int. J. Heat Mass Transf., vol. 52, no. 17–18, pp. 3972–3983, 2009, doi: 10.1016/j.ijheatmasstransfer.2009.03.026.

[3] O. Tonomura, S. Tanaka, M. Noda, M. Kano, S. Hasebe, and I. Hashimoto, “CFD-based optimal design of manifold in plate-fin microdevices,” Chem. Eng. J., vol. 101, no. 1–3, pp. 397–402, 2004, doi: 10.1016/j.cej.2003.10.022.

[4] A. Zargoushi, F. Talebi, and S. H. Hosseini, “CFD modeling of industrial cold box with plate-fin heat exchanger: Focusing on phase change phenomenon,” Int. J. Heat Mass Transf., vol. 147, no. xxxx, p. 118936, 2020, doi: 10.1016/j.ijheatmasstransfer.2019.118936.

[5] I. M. C. Santosa, K. M. Tsamos, B. L. Gowreesunker, and S. A. Tassou, “Experimental and CFD investigation of overall heat transfer coefficient of finned tube CO2 gas coolers,” Energy Procedia, vol. 161, no. 2018, pp. 300–308, 2019, doi: 10.1016/j.egypro.2019.02.096.

[6] X. Zhang, Y. Ge, J. Sun, L. Li, and S. A. Tassou, “CFD modelling of finned-tube CO2 gas cooler for refrigeration systems,” Energy Procedia, vol. 161, pp. 275–282, 2019, doi: 10.1016/j.egypro.2019.02.092.

[7] N. S. Effendi, S. S. G. R. Putra, and K. J. Kim, “Prediction methods for natural convection around hollow hybrid fin heat sinks,” Int. J. Therm. Sci., vol. 126, no. July 2017, pp. 272–280, 2018, doi: 10.1016/j.ijthermalsci.2018.01.002.

[8] K. Lindqvist and E. Næss, “A validated CFD model of plain and serrated fin-tube bundles,” Appl. Therm. Eng., vol. 143, no. June, pp. 72–79, 2018, doi: 10.1016/j.applthermaleng.2018.07.060.

[9] N. S. Effendi and K. J. Kim, “Orientation effects on natural convective performance of hybrid fin heat sinks,” Appl. Therm. Eng., vol. 123, no. May, pp. 527–536, 2017, doi: 10.1016/j.applthermaleng.2017.05.134.

[10] C. Liu, W. Bu, and D. Xu, “Multi-objective shape optimization of a plate-fin heat exchanger using CFD and multi-objective genetic algorithm,” Int. J. Heat Mass Transf., vol. 111, pp. 65–82, 2017, doi: 10.1016/j.ijheatmasstransfer.2017.03.066.

[11] S. Singh, K. Sørensen, and T. Condra, “Investigation of material efficient fin patterns for cost-effective operation of fin and tube heat exchanger,” Appl. Therm. Eng., vol. 126, pp. 903–914, 2017, doi: 10.1016/j.applthermaleng.2017.08.010.