International Journal on
"Technical and Physical Problems of Engineering"
(IJTPE)
Published by International Organization of IOTPE

ISSN 2077-3528
IJTPE Journal
www.iotpe.com
ijtpe@iotpe.com

March 2023 Issue 54 Volume 15 Number 1 Pages 248-254

COMPUTATIONAL SIMULATION OF HEAT TRANSFER THROUGH FINS OF DIFFERENT SHAPES IN AN AIR-COOLED INTERNAL COMBUSTION ENGINE

A. Dorri [1] **M. Alcani** [1] **K. Dhoska** [2] **M. Bako** [1]

1. Department of Energy, Polytechnic University of Tirana, Tirana, Albania
adorri@fim.edu.al, malcani@fim.edu.al, marjol.bako@fim.edu.al
2. Department of Production and Management, Polytechnic University of Tirana, Tirana, Albania, kdhoska@fim.edu.al

Abstract- The Engine is the most important component in a moving vehicle, and it is subject to high temperature and thermal stresses. The purpose of this paper is to simulate the thermal behavior of the airflow through the fins of a lightweight motorcycle engine. To this aim, three different geometrical forms of cooling fins (rectangular, step shape and trapezoid shape) were considered during simulation. Since the engine was air-cooled, air velocity is one of the main factors that count for the heat transfer process. For this, numerical simulations were performed considering four different air velocities. Simulations were carried out through Computational Fluid Dynamics (CFD) software. Geometry of fins was built in SolidWorks and air flow simulation is performed in Ansys FLUENT software. CFD code solves the conservation equation for mass, momentum, and energy. The main objective is to investigate the influence of the geometry of the fins on the improvement of heat transfer rate, under various air velocities. Results show different behaviors for all three fins under study.

Keywords: CFD, Modeling, Heat Transfer, Engine, Energy.

1. INTRODUCTION

Nowadays, the main challenges facing designers and manufacturers of internal combustion engines are related to pollution control and reducing fuel consumption [1, 2]. In addition to these, there has been a significant improvement in overall efficiency, noise reduction, and the cost of their manufacture and maintenance [3]. The engines supply power to the vehicle, produced by fuel combustion over each cycle. Almost 30% of the energy goes for mechanical work and the rest is turned to heat energy. The engine cooling system is designed to remove excessive heat, provide the best thermal condition and ensure the engine's normal performance. The cooling system removes a large part of the heat, the rest is removed from the lubrication system and the environment [4]. Depending on the coolant being used, the engines use either liquid or air cooling. Normally, in a liquid-based system, water is used as a coolant, while in the air-cooling system, ambient air is used as a coolant. The accumulation of thermal energy may lead to an increase in the thermal stresses of the various engine parts, which accelerates their wear and tear. The accumulation of too much or too little heat inside the engine has a direct effect on the combustion process, significantly aggravating it. Addressing these issues requires in-depth knowledge of engine theory, construction and design from designers, related to engine manufacturing and maintenance [5]. Nowadays, major attention is paid for the use of computers in the conception and design of the engines. In recent decades, Computational Fluid Dynamics has been extensively applied to a broad variety of complex physical issues. CFD is the use of computer methods for the solution of fluid engineering systems, including math and physical modeling and numerical methods (FEM, FDM, grid generation, etc.). CFD methods are developing quickly these days and are used in many fields of engineering and life. The main reason why CFD is rapidly growing, are the advantages that it provides compared to experiments [6].

Performance of various devices and machinery, like engines, besides others are based on the heat transfer and cooling process [7]. In our case, cooling is realized through cooling fins that allow air to remove heat from the motor. The heat transfer rate (HTR) is a function of the velocity of the motorbike, the shape of the fins and the surrounding temperature [8, 9]. The low heat transfer rate from the fins is the major issue with the air-cooling system. [10]. In our work, we used CFD tools to simulate heat transfer for various forms of cooling fins and analyzed the impact of the fin shape on the HTR. We tested three different cooling fins shape: rectangular shape, step shape and trapezoid shape. The main objective is to investigate the influence of the geometry of the fins on the improvement of heat transfer rate, under various air velocities.

2. NUMERICAL SIMULATION

Work for the CFD simulation is organized according to the scheme in Figure 1. Geometry and mesh were generated first in the pre-processor. Next steps were the simulations through CFD software and the last were results and analysis presentation in post-processor environment.

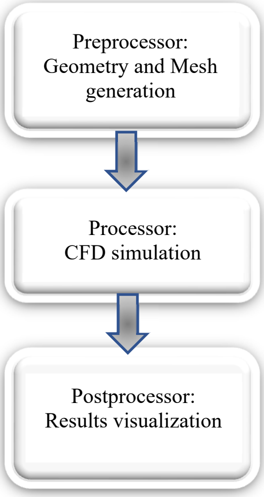


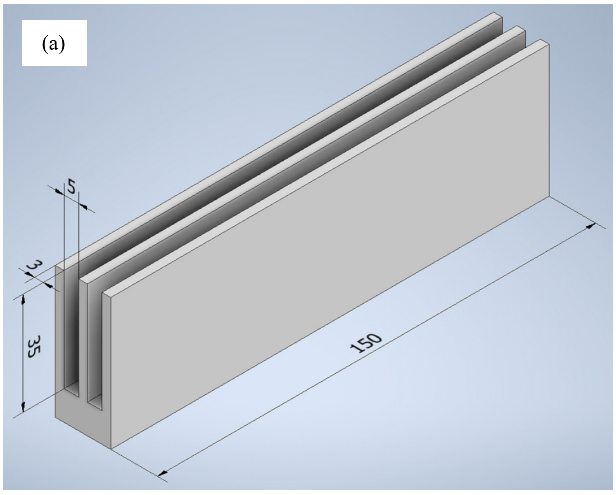
Figure 1. Scheme of CFD simulation performed

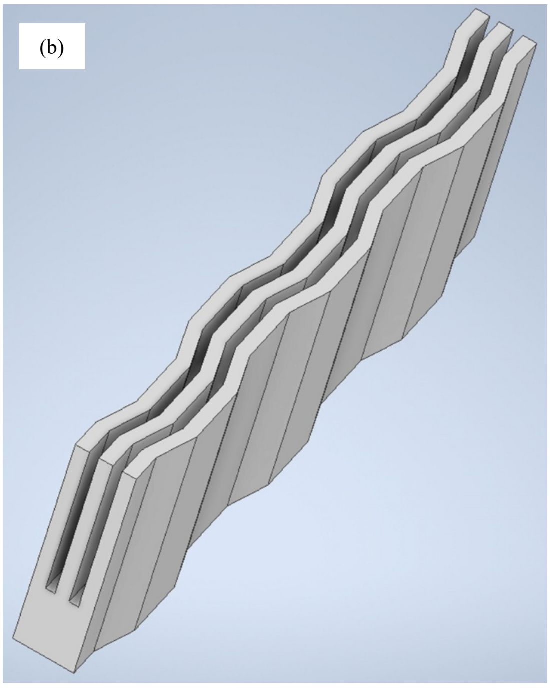
2.1. Physical Modeling

Cooling fins taken in consideration were of three types of shape: rectangular, step shape and trapezoid, as in Figure 1. The geometry of three fins was created in SolidWorks, computational mesh was generated in Meshing (preprocessor), and simulation are realized in ANSYS FLUENT (processor) as Figure 2.

The computation domain has a large rectangular volume, that include the finned body in its central part, as shown in Figure 3. It was finned-oriented, and proper boundary conditions have been applied at the ends of the domain to preserve continuity. Parameters of fins are specified in Table 1.

Creating the mesh of the model is a very important step when working with a CFD software because of this, depends the convergence of the problem. Fine grids have been generated near the fins, to obtain better results for the thermal boundary layer, and far from the surface of the fin, the grid is coarse for a quick solution. The domain mesh has been made by tetrahedron elements. In order to see meshing sensitivity, we created four meshes with different resolution. After running simulations for these three meshes, we compared the results for the pressure and velocity. As a result, we choose the third (finer) mesh.





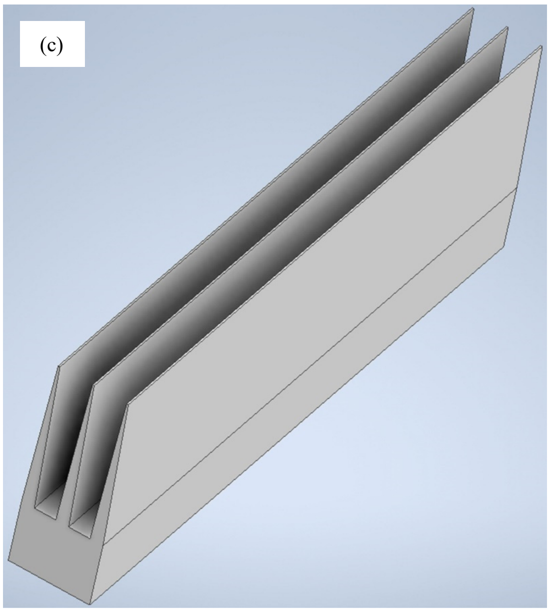


Figure 2. Geometry of three types of fins used for simulation: (a) Rectangular, (b) Step shape, (c) Trapezoid

Table 1. Parameters of fins

Parameter	Values (in mm)		
	Rectangular	Step Shape	Trapezoid
Length	150	150	150
Thickness	3	3	3-2
Width	35	35	35

2.2. Mathematical Modeling

In this work, the basis of CFD simulations and analysis are presented by the equation of conservation of mass, impulse, and energy. CFD code solves these equations by the finite volume method in the fluid domain [11]. For compactness, the transport equations are written in unit vector notation, as Equations (1), (2) and (3).

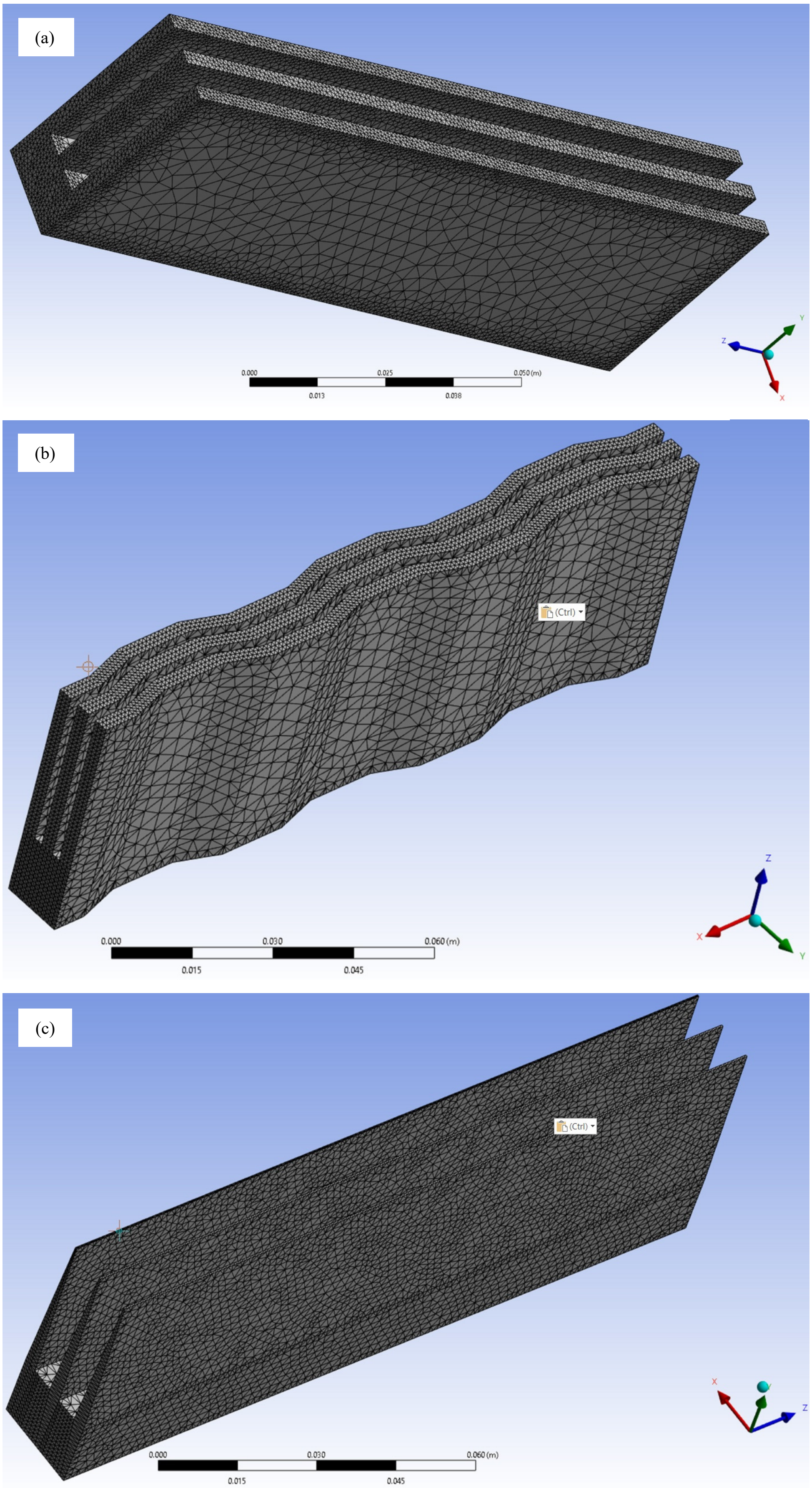


Figure 3. Generated mesh for three types of fins used for simulation: (a) Rectangular, (b) Step shape, (c) Trapezoid

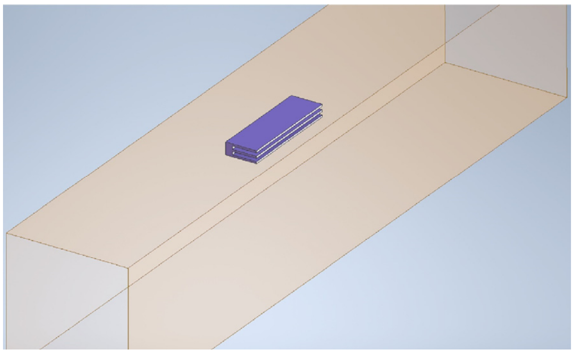


Figure 4. Fluid domain

- Conservation of mass:

$$\nabla \cdot (\rho v) = 0 \qquad (1)$$

where, ρ, v are air density and velocity vector, respectively.

- Conservation of impulse:

$$\nabla(\rho v v) = -\nabla p + \nabla \cdot (\mu \nabla v) \qquad (2)$$

where, p is the mean pressure and μ is the effective viscosity, representing both laminar and turbulent viscosity.

- Conservation of energy:

$$\nabla(\rho v c_p t) = \nabla \cdot (\lambda \nabla t) + \dot{q} \qquad (3)$$

where, c_p represents the specific heat and $\dot{q}$ is the internal heat generation.

In the process of averaging Equation (2), a new term appears, referred to as Reynolds stresses $(\overline{\rho \overrightarrow{u''} \overrightarrow{u''}})$, and depicts the effects of turbulence. Its presence prevents closure of the system of the equations. To this aim serves the turbulence models, which solve determination of Reynolds stress by using turbulent viscosity hypothesis. There are different turbulence models, but in our case, we selected *k-ε* Realizable. This model contains a newly alternate formula for the turbulent viscosity, and for the dissipation rate the transportation equation has been modified. It also satisfies some mathematical limitations on the Reynolds stresses. A special attention was given to the boundary conditions, which are given in Table 2.

We repeated the simulation for four different air velocity inlets: 20 km/h; 40 km/h; 60 km/h; 80 km/h. Other conditions remained the same.

Table 2. Boundary conditions

Zone Name	Conditions
Combustion Chamber	$T = 1600$ K
Inlet	$v = 20$ km/h
	$T = 300$ K
Outlet	Gauge pressure = 0 Pa
Walls	stationary walls
	no slip

3. RESULTS AND DISCUSSIONS

All simulations done in this work for solving equations are produced from the numeric method "segregate solver"; interpolation scheme "First Order Upwind" is used for the discretization of convective terms. Algorithm "SIMPLE" was considered for the pressure-velocity coupling. For the turbulence modeling, as previously mentioned, "*k-ε* Realizable" is used together with Enhancement Wall Treatment (EWT).

Since heat transfer from engine body to environment is the main purpose of cooling fins, our results are focused on Heat Transfer Coefficient (*HTC*) of fins. Also, air movement through fins play an important role in heat transfer. For this we have also analyzed pressure losses. Figures 5, 7 and 9 represents *HTC* for three fins, while Figures 6, 8 and 10 represents pressure losses.

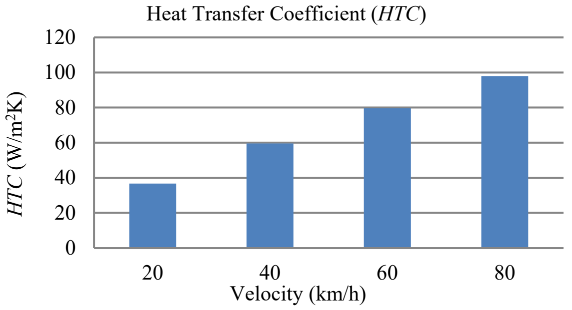


Figure 5. Heat transfer coefficient for rectangular fins

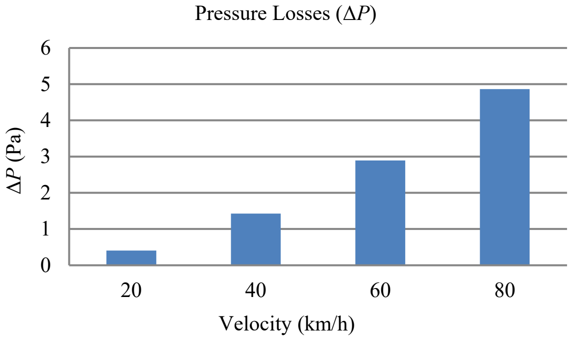


Figure 6. Pressure losses for rectangular fins

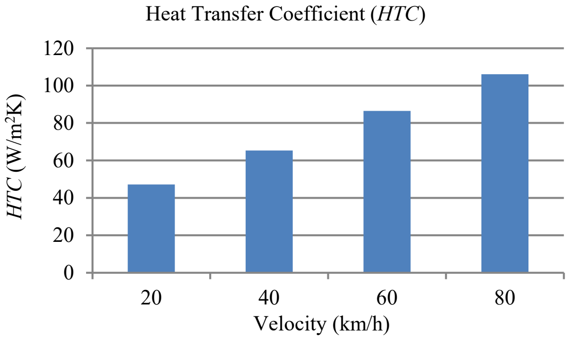


Figure 7. Heat transfer coefficient for step shape fins

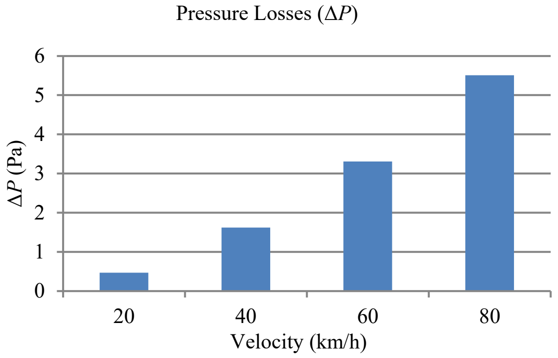


Figure 8. Pressure losses for step shape fins

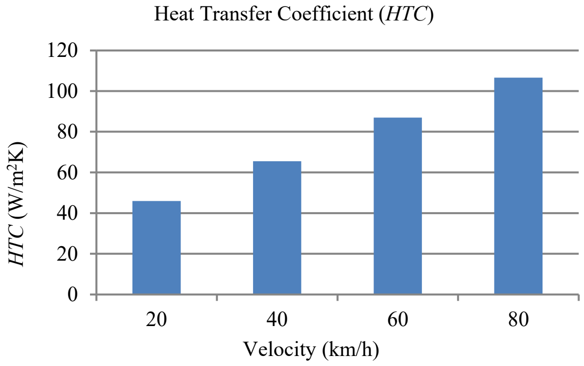


Figure 9. Heat transfer coefficient for trapezoid shape fins

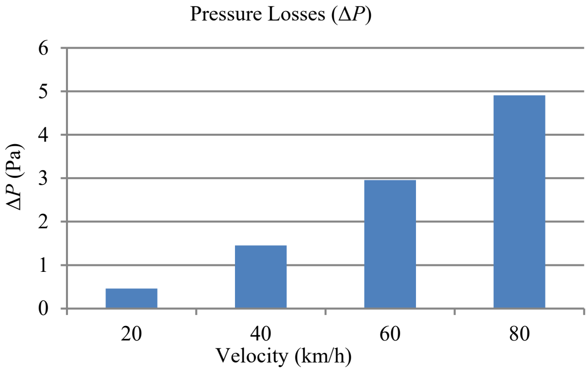


Figure 10. Pressure losses for trapezoid shape fins

Analyzing results of *HTC* for all three fin shapes (Figures 5, 7 and 9) we can observe that it depends strongly on-air velocity. *HTC* increase with velocity increase and this relation is almost linear. While analyzing results for pressure losses (Figures 6, 8 and 10) we can figure out that again losses increasing with velocity increase. But in this case the rate of increasing is bigger compared to *HTC*. We can see that dependence of pressure losses on velocity is almost parabolic.

To have a better comprehension of the influence of fins shape on *HTC* and pressure losses, results for all three shapes are overlapped and compared (Figures 11 and 12).

As it is shown in Figure 11, Heat Transfer Coefficients for step shape fins and trapezoid shape fins are almost the same. In low air velocity *HTC* of trapezoid fins is smaller than *HTC* of step shape fins.

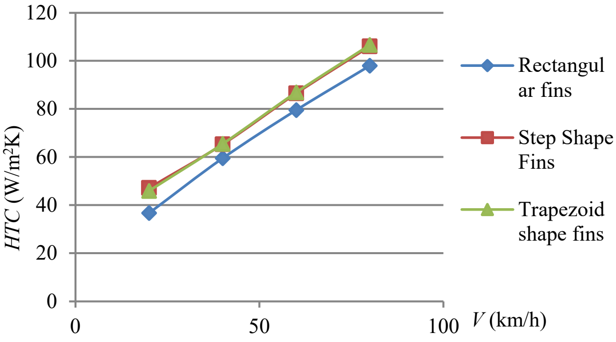


Figure 11. Comparison of HTC for three models of fins

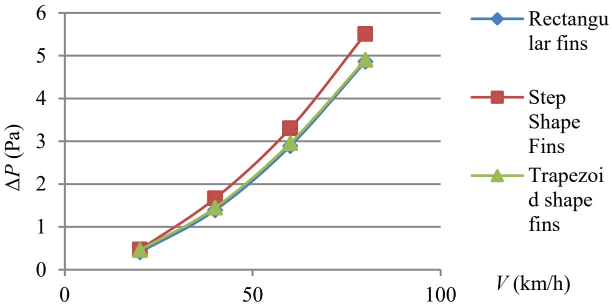


Figure 12. Comparison of pressure losses for three models of fins

But, considering that the motorbike tends to run in the medium speed range, the *HTC* of the two shapes is almost the same. So is difficult to decide which of these shapes is better. The difference is obvious between them and rectangular fins. Step shape fins and trapezoid shape fins have the advantage that they dissipate more heat in the same time and same conditions against rectangular fins. This is so important for thermal stresses of engine. In this way engine will work in better conditions. However, rectangular fins have a lower cost than the other two types because they are easier to produce. That is the reason that rectangular fins are found in a wide range of today motorcycle's engine. But if we take into consideration just the working conditions of the engine and the thermal stresses of it, this isn't an easy task to decide which fins should be recommended. To do that, designers need to simulate more influencing factors and data to analyses.

While analyzing Figure 12, pressure losses are almost the same for rectangular fins and trapezoid fins. They are greater for the step shape fins. So, this shape of fins has a disadvantage against two others because it creates greater resistance for the air flow, so it is more difficult for the air to flow through the fins and this could worsen heat exchange between engine and surrounding ambient. With this analysis we recommend the trapezoid shape fins. They have the advantage of a greater *HTC* against rectangular fins. This has a good effect at engine working conditions and smaller thermal stresses for the material of engine body. Another advantage is the lower pressure losses than step shape fins. So will be easier for the air to flow thru the fins and this will intensify heat exchange between engine and ambient. Also, trapezoid fins have an advantage the cost of production which is lower than step shape fins. Farther work can be done with other geometrical shapes.

4. CONCLUSIONS

In this paper we have analyzed three different geometrical shapes of cooling fins in a motorbike engine. It is known that heat transfer from inside engine to the surrounding ambient is affected from many factors such as air velocity, geometrical shape, dimensions of fins, etc. In our case, we have kept the same dimensions for the fins and fluid domain and changed air velocity through fins. To this aim CFD methods was proved to be really useful and faster than experiment. By using them, we can produce quick and accurate results for a broad range of operational conditions.

In simulations that are made in this study, are shown that heat transfer increases when air velocity increase for every fin shape. However, the rate of increase was different for each of the fins. Also, pressure losses increase with velocity increase but with a greater rate than *HTC*. The purpose was to identify the fin shape with greater heat transfer rate. Analyzing results was observed that trapezoid shape fins have advantages compared to others shapes (rectangular and step shape).

As the airflow pattern is determined by the physical geometry of the fins, further simulations could be carried out for different fin lengths, widths, and thicknesses. In that way could be found the optimum of these dimensions. Also, simulations could be done for others operating conditions.

NOMENCLATURES

1. Acronyms

CFD	Computational Fluid Dynamics
EWT	Enhancement Wall Treatment
FEM	Finite Elements Methods
FDM	Finite Difference Methods
HTC	Heat Transfer Coefficient

2. Symbols / Parameters

c_p: The specific heat
p: The mean pressure
$\dot{q}$: The internal heat generation
t: The temperature
v: The velocity
ρ: The air density
μ: The effective viscosity

REFERENCES

[1] R.D. Reitz, H. Ogawa, R. Payri, et al., "IJER Editorial: The Future of the Internal Combustion Engine", International Journal of Engine Research, Vol. 21, No. 1, pp. 3-10, 2020.

[2] K. Shala, A. Dorri, "Analysis of the Motor Vehicle Fleet as a Way to Reduce Air Pollution in the Republic of Kosovo", International Journal of Innovative Technology and Interdisciplinary Sciences, Vol. 4, No. 2, pp. 685-690, May 2021.

[3] J.B. Heywood, "Internal Combustion Engine Fundamentals", McGraw-Hill, The 2nd ed., pp. 1-6, New York, USA, 2018.

[4] A.T. Kirkpatrick, "Internal Combustion Engines: Applied Thermosciences", John Wiley and Sons, 4th Edition, pp. 319-320, New Jersey, USA, 2020.

[5] A. Meghdir, M. Bouhafs, "Effect of a Radiative Intermediate wall in an Annular Cylindrical Space", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 52, Vol. 14, No. 3, pp. 53-60, September 2022.

[6] Vijayashree, V. Ganesan, "Application of CFD for Analysis and Design of IC Engines", D.K. Srivastava, A.K. Agarwal, A. Datta, R.K. Maurya, (eds.) "Advances in Internal Combustion Engine Research - Energy, Environment and Sustainability", Springer, pp. 251-306, Singapore, 2018.

[7] S.K. Sharma, V. Sharma, "Maximising the Heat Transfer through Fins Using CFD as a Tool", International Journal of Recent advances in Mechanical Engineering (IJMECH), Vol. 2, No. 3, pp. 13-28, August 2013.

[8] A.A. Mohsin, S.M. Kherde, "Heat Transfer Analysis by CFD Simulation for Different Shapes of Fins", International Journal for Research in Applied Science and Engineering Technology (IJRASET), Vol. 3, Issue 2, pp. 106-110, February 2015.

[9] H. Yaagoubi, H. Abouchadi, M. Taha Janan, "3D Mathematical Thermal Analyses of PA12 Powder Bed on the SLS Process by Two Numerical Methods", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 51, Vol. 14, No. 2, pp. 74-83, June 2022.

[10] K. Sathishkumar, K. Vignesh, N. Ugesh, P.B. Sanjeevaprasath, S. Balamurugan, "Computational Analysis of Heat Transfer through Fins with Different Types of Notches", International Journal of Advanced Engineering Research and Science (IJAERS), Vol. 4, Issue 2, pp. 175-183, February 2017.

[11] ANSYS Fluent 18.0 User Guide, ANSYS, Inc, Canonsburg, USA, January 2017.

BIOGRAPHIES

Name: **Altin**
Surname: **Dorri**
Birthday: 03.12.1972
Birth Place: Kelcyre, Albania
Bachelor: Mechanical Engineer, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 1996
Master: Mechanical Engineer, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 1996
Doctorate: Energy, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 2010
The Last Scientific Position: Assoc. Prof., Department of Energy, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, Since 2014
Research Interests: CFD Applied in Fluid Machinery, Energy Efficiency and Conversion, Environmental Issues
Scientific Publications: 50 Papers, 2 Books, 7 Projects, 3 Theses
Scientific Memberships: Albanian Mechanical Engineer Association

Name: **Majlinda**
Surname: **Alcani**
Birthday: 10.06.1964
Birth Place: Librazhd, Albania
Bachelor: Mechanical Engineering, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 1988

Master: Mechanical Engineering, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 1988
Doctorate: Energy, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 2010
The Last Scientific Position: Assoc. Prof., Department of Energy, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, Since 2015
Research Interests: Waste Treatment, Energy Efficiency and Conversion, Environmental Issues
Scientific Publications: 30 Papers, 4 Books, 6 Projects, 3 Theses
Scientific Memberships: Albanian Mechanical Engineer Association

Name: **Klodian**
Surname: **Dhoska**
Birthday: 25.05.1983
Birth Place: Librazhd, Albania
Bachelor: Mechanical Engineering, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 2007
Master: Measurements Science, Faculty of Science and Technology, University of Tartu, Estonia, 2010
Doctorate: Mechatronics, Faculty of Mechanical Engineering, Tallinn University of Technology, Estonia, 2016
The Last Scientific Position: Assoc. Prof., Department of Production and Management, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, Since 2022
Research Interests: Physics and Technical Physics and Energy Conversion, Mechatronics
Scientific Publications: 100 Papers, 1 Books, 10 Projects, 3 Theses
Scientific Memberships: Albanian Mechanical Engineer Association

Name: **Marjol**
Surname: **Bako**
Birthday: 10.06.1983
Birth Place: Korce, Albania
Bachelor: Mechanical Engineering, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 2017
Master: Mechanical Engineering, Faculty of Mechanical Engineering, Polytechnic University of Tirana, Tirana, Albania, 2019
Research Interests: HVAC Plant Design, Oil and Gas Plants Design
Scientific Publications: 2 Theses
Scientific Memberships: Albanian Mechanical Engineer Association