A Numerical Analysis of the Air-Cooling System of a Spark Ignition Aeronautical Engine

Maria Faruoli^{1,*}, Annarita Viggiano¹, Paolo Caso² and Vinicio Magi^{1,3}

¹ School of Engineering, University of Basilicata, Potenza, 85100, Italy

² Costruzioni Motori Diesel CMD S.p.A., San Nicola La Strada (CE), 81020, Italy

³ Department of Mechanical Engineering, San Diego State University, CA 92182, USA

Abstract. It is well known that spark ignition internal combustion engines for aeronautical applications operate within a specific temperature range to avoid structural damages, detonations and loss of efficiency of the combustion process. An accurate assessment of the cooling system performance is a crucial aspect in order to guarantee broad operating conditions of the engine. In this framework, the use of a Conjugate Heat Transfer method is a proper choice, since it allows to estimate both the heat fluxes between the engine walls and the cooling air and the temperature distribution along the outer wall surfaces of the engine, and to perform parametric analyses by varying the engine operating conditions. In this work, the air-cooling system of a 4-cylinder spark ignition engine, designed by CMD Engine Company for aeronautical applications, is analysed in order to evaluate the amount of the air mass flow rate to guarantee the heat transfer under full load operating conditions. A preliminary validation of the model is performed by comparing the results with available experimental data. A parametric study is also performed to assess the influence of the controlling parameters on the cooling system efficiency. This study is carried out by varying the inlet air mass flow rate from 1.0 kg/s to 1.5 kg/s and the temperature of the inner wall surfaces of the engine combustion chambers from 390 K to 430 K.

1 Introduction

An increase of the load conditions of Internal Combustion Engines (ICEs) causes an increase of the engine wall temperatures. High temperature of the engine walls leads to several problems as, for instance, the deterioration of the cylinder lubricating oil and the seizure of the engine components [1]. The temperature of the combustion chamber liner on the gas-side surface should be generally kept below about 180°C, in order to preserve the thermal stability of the lubricating oil and to reduce the thermal stresses [2]. Analogously, the temperature of the outer wall surfaces of the engine must be limited in order to avoid fatigue cracking and deformation, and it must be constrained based on the thermal properties of wall material (e.g. 400° C for cast iron, 300° C for aluminium alloys).

* Corresponding author: maria.faruoli@unibas.it

The engine cooling system specifications are generally assessed by means of an analysis of the heat energy balance of the engine. For instance, it has been proven that the heat-transfer rate to coolant under full load conditions of a spark ignition engine is about 60-70 % of the engine brake power [3]. The amount of transferred heat can be evaluated by employing semi-empirical relations, such as Annand's correlation, which provide the wall heat fluxes in the combustion chamber during the working cycle [4].

ICEs are generally classified in two main categories based on the cooling strategy: water-cooled and air-cooled engines. The water-cooled system employs cooling liquid that flows through the water-jackets surrounding the engine cylinders. The cooling fluid transfers the heat from the cylinders to a heat exchanger, named radiator [3]. On the other hand, the air-cooled engines are generally characterized by outer finned surfaces, which are located around the cylinders and the engine head. The cooling fluid is ambient air and the efficiency of the system is related to both the size and distribution of the fins, as well as to the flow velocity and turbulence level [5]. These parameters influence the mean convective heat transfer, which accounts for the performance of the cooling system.

The main advantages of using air as a coolant are a) the weight of the engine is lower, b) the steady-state wall temperature is reached sooner and c) it is more economically convenient with respect to a liquid-cooled system with the same brake power [3]. The inner wall surfaces of the cylinders are characterized by higher temperatures than those of water-cooled engines, hence a higher thermal efficiency can be obtained within the thermal limits required to avoid the deterioration of lubricating oil and knocking effect [6]. On the other side, the liquid cooling system allows both higher volumetric efficiency, due to lower working temperatures, and lower noise.

This work focuses on a detailed analysis of an air-cooled Spark Ignition (SI) engine. This type of engine is extensively studied in the literature by means of several numerical techniques. Numerical models are valuable tools to evaluate the influence of different parameters on the heat transfer process of ICEs, under several working conditions. The numerical approach saves resources and time during the design process since some experimental setup can be bypassed. Due to the complexity of the heat transfer phenomenon, the analyses are often limited to a single cylinder with one or more fins. The efficiency of the air-cooling system can be computed by modifying either the geometry of the fins [5] or the ambient flow conditions, in terms of velocity and/or temperature [6,7]. In the work of Nitnaware *et al.* [5] and Sagar *et al.* [8], the fluid and solid structure temperatures are separately computed. On the other hand, Brusiani *et al.* [9] and Mishra *et al.* [7] carried out computations of fluid and solid simultaneously, by employing a Conjugate Heat Transfer (CHT) method. This leads to a most accurate evaluation of the interaction between the cooling flow and the engine structural components, by including all the involved phenomena. However, only simplified engine geometries are considered in the scientific literature as regards air-cooling systems. Besides, the CHT method has been employed by Fontanesi *et al.* [10] to study a liquid-cooled ICE and by Berni *et al.* [11] to carry out in-cylinder simulations.

This work aims to model the air-cooling system of a spark ignition engine designed by CMD Engine company for aeronautical applications, referred to as CMD22 [12]. Different operating parameters are considered by varying the air mass flow rate and the inner wall cylinder temperature, in order to select the parameters that guarantee the required heat transfer of the engine under full load conditions. A Conjugate Heat Transfer model is developed to simulate the thermo-fluid dynamic behaviour of the cooling flow and of the engine solid structure. In the first part of the work, the study is limited to the analysis of the air flow pattern, to get a first estimate of the air mass flow rate to transfer the required amount of heat. At this stage, fixed temperatures are set for the engine outer wall surfaces, by considering feasible values for different engine parts. In the second part of the work, a

CHT model that solves both fluid flow and solid regions is developed, by setting the inner wall temperatures of the combustion chambers as a boundary condition. A calibration of the model is carried out by employing measured wall temperatures at specific locations along the fins. Finally, a parametric analysis is performed to study the influence of both the air flow rate and the inner wall temperature on the transferred thermal power. This work is organized as follows: first, the specifications of the CMD22 engine are given with an energy balance analysis to provide an estimation of the amount of thermal power to be transferred; then, the CFD model is discussed, and results obtained by assuming selected temperatures of the engine outer walls are presented. Finally, the numerical model that simulates both fluid flow pattern and temperature distribution of the solid region is described, and results obtained by considering the inner combustion chambers surface temperature as a boundary condition are shown. The work ends with the most relevant concluding remarks.

2 CMD22 Engine

The CMD22 engine is a 4-stroke spark ignition internal combustion engine, with a boxer 4 cylinders configuration and with a total displacement of 2200 cm³ [12]. This engine is employed for aeronautical applications, such as ultralight aircrafts. Its maximum power is 87 kW during take-off, whereas the engine supplies 78 kW under maximum continuous power conditions at 4850 rpm. The cooling system must guarantee that the temperature on the outer surfaces of the engine is not higher than 443 K, in order to avoid deformations of the engine structure.

The engine is cooled by an air-oil cooling system. As regards the air-cooled part, the engine is enclosed in a metallic hood to address the air flow. The air mass flow rate is controlled by a fan. The cooling system is based on the use of finned surfaces, as shown in Figure 1, to enhance the heat transfer by increasing the heat exchange surface. It is well known that the parameters that mostly influence the cooling performance are the fin sizes, the velocity and the turbulence of the cooling flow air, as well as its temperature [13].

Generally, a spark ignition engine cooling system must transfer a thermal power of about 60-70 % of the brake power. By increasing the engine performance, the cooling system should guarantee higher values of the transferred heat.

For the engine under examination, it is required that about 20 kW thermal power must be transferred for each bank at 4850 rpm regime. Due to the symmetric structure of the engine, only one bank, i.e. right bank, has been considered. The numerical domain is bounded by considering the metallic hood walls as boundaries. The cylinder on the left side of Figure 1 is identified as Cylinder 3, whereas the one on the right side is referred to as Cylinder 1.

The study aims to evaluate the efficiency of the cooling system in terms of heat transferred and temperature distribution on the engine outer walls.

Fig. 1. Right bank of CMD22 engine enclosed in a metallic hood.

3 CFD Model

The first part of the work focuses on the simulation of the cooling flow pattern by using a CFD solver. The temperature distribution along the outer wall surfaces of the engine is set as a boundary condition and the influence of the air mass flow rate on the efficiency of the heat transfer is assessed.

The fluid numerical domain is the region between the metallic hood and the engine outer wall surfaces. The model accounts for both aerodynamic and thermal properties of air, which is considered as a compressible fluid. The ideal gas law is used to compute density, whereas Sutherland's law [14] is employed to determine the kinematic viscosity, and the specific heat is defined by a polynomial dependence on temperature.

The mass flow rate varies in the range 0.2 - 0.8 kg/s, and the Reynolds number at the inlet section changes from 50'000 to 200'000. Hence, the flow is fully turbulent. The Reynolds-Averaged Navier-Stokes (RANS) equations are solved under steady-state conditions. The continuity, momentum and energy equations read:

$$\nabla \cdot (\rho \mathbf{u}) = 0, \quad (1)$$

$$\nabla \cdot (\rho \mathbf{u} \mathbf{u}) - \nabla \cdot ((\mu + \mu_t) \nabla \mathbf{u}) + \nabla p = 0, \quad (2)$$

$$\nabla \cdot (\mathbf{u}(\rho E + p)) - \nabla \cdot ((\lambda + \lambda_t) \nabla T) = 0. \quad (3)$$

In the above equations ρ is the gas density; $\mathbf{u}$ is the gas velocity; μ and μ_t are the molecular dynamic viscosity and the turbulent dynamic viscosity, respectively; p is the pressure; E is the total energy; λ and λ_t are the gas thermal conductivity and turbulent thermal conductivity, respectively; T is the gas temperature. The k- ϵ model proposed by Launder and Spalding [15] with standard wall functions is employed for all cases to close the system of governing equations.

3.1 Computational Grid

The right bank of the engine consists of two side-by-side cylinders. The metallic hood provides a rotation by 90° of the air cooling flow pattern. The inlet (blue arrow) and outlet (red arrow) sections are shown in Figure 1. The engine is closer to the outlet section of the hood. In order to avoid any influence of the outlet boundary conditions on the flow structure, the computational domain is extended downstream of the outlet section as shown in Figure 2a. In the figure, the boundary conditions are also shown: three outlet sections are identified with red arrows, the back side and the remaining side boundaries are defined as walls, whereas on the front side the symmetry boundary condition is imposed due to the left bank of the engine.

An extremely high grid resolution is required to accurately compute every details of the flow pattern due to the complexity of the engine geometry. The grid is mainly composed by hexahedral element and a minimum number of four numerical cells are required along the fins height and between two consecutive fins, to provide an accurate solution. Since a flow recirculation is expected at the outlet section of the hood, a higher resolution is required in that region with respect to the further downstream region. The same high level of refinement is required along the hood, where a complex flow pattern is also expected. The smallest numerical cells are located near the walls, to correctly represent the details of the geometry. The grid is composed by about 7 million of numerical cells. An overview of the mesh with some details are shown in Figure 2b.

Fig. 2. Modified computational domain (a) and numerical grid (b).

3.2 Boundary conditions

Several engine working conditions have been considered in order to evaluate the efficiency of the cooling system. The air mass flow rate varies from 0.2 to 0.8 kg/s, which corresponds to the range supplied by the fan.

As already mentioned, the flow is under fully turbulent regime, thus the inlet turbulent kinetic energy (k) and the turbulent dissipation rate (ϵ) are set by considering a turbulence intensity equal to 5 % and a turbulent length scale of 16.8 mm, which is about 7% of the inlet characteristic length [16]. Standard wall functions are used in order to model the turbulent boundary layer, with a $y^+ > 30$ close to the walls. The pressure at the outlet sections is set equal to 1 bar.

As regards the fluid temperature, the air entering into the domain has a temperature of 288 K, whereas three temperature zones are selected for the engine outer walls, with a temperature stratification from cold (top) to warm (bottom) region as shown in Figure 3. Despite the temperature stratification procedure represents a simple approximation, it allows a preliminary estimation of the amount of transferred heat as a function of the cooling mass flow rate. The temperature of the three zones are selected based on typical temperatures of the inner wall surfaces of the cylinders and on the maximum allowed temperature of the engine outer wall surfaces, i.e. 443 K.

Two different sets of temperatures are employed in the simulations, as given in Table 1. The hood walls are considered adiabatic, since no significant heat transfer occurs along such wall surfaces.

Fig. 3. Temperature boundary conditions with zone ID.

Table 1. Temperatures along the engine outer wall surfaces.

	Case 1	Case 2
T1	350 K	370 K
T2	380 K	400 K
T3	410 K	430 K

3.3 Results

Pressure loss and velocity distribution are evaluated to assess the fluid dynamic features. The outflow air temperature, the total heat exchanged between fluid and the engine walls, and the fluid temperature distribution are also evaluated as well. The pressure drop is computed as:

$$\Delta p = p_{in} - p_{out}, \tag{4}$$

where p_{in} is the area-averaged pressure at the inlet section and p_{out} is the area-averaged pressure computed at the hood outlet section.

The transferred thermal power $\dot{Q}$ is evaluated as:

$$\dot{Q} = \dot{m} c_p (T_{out} - T_{in}), \tag{5}$$

where $\dot{m}$ is the air mass flow rate at the inlet section, c_p is the air specific heat at constant pressure, and T_{in} and T_{out} are the mass-averaged temperatures computed at the inlet section and at the hood outlet section, respectively.

Fig. 4. Sections used for the analysis of the flow field: the grey section is defined as Plane A and the green section as Plane B.

Two sections are considered, as shown in Figure 4, named Plane A and B. The velocity field distributions on the two planes, together with the corresponding streamlines, are shown in Figure 5. On Plane B, the highest velocity is observed in the region between the two cylinders heads, whereas on Plane A an increase of the fluid velocity is observed between two consecutive fins. Due to the engine intake plenum, a recirculation zone is generated, as shown in Plane A, whereas in the upper part of Plane B no recirculation is observed. On both planes, the presence of stagnation regions downstream the engine cylinders is observed.

Pressure loss is computed for different inlet mass flow rates and is given in Figure 5c. As expected, a quadratic dependence of the pressure loss (Δp given in Pa) with mass flow rate ($\dot{m}$ given in kg/s) is found:

$$\Delta p = 2088.1\dot{m}^2 + 130.12\dot{m} - 6.0695. \tag{6}$$

Fig. 5. Coloured streamlines on Plane A (a) and Plane B (b) and pressure loss vs inlet mass flow rate (c).

Temperature distribution contour plots (on both planes A and B) are shown in Figure 6 for the first set of temperatures of Table 2. Similar trends are obtained with the second set of temperatures. On Plane A, a higher temperature region in the shear downstream the intake plenum is observed, due to the larger temperature difference between the plenum walls and the fluid. In the bottom region, i.e. the region where the engine outer wall temperatures is set to T3, the highest air temperatures are observed on both planes A and B, since the air is heated by flowing through the engine cylinders. Maximum temperatures are reached in the recirculation zones. In Figures 7a and 7b, the mass-averaged temperature at the outlet section and the transferred thermal power are given, respectively, as a function of the air mass flow rate for the two sets of temperature of Table 2. The figures show that the transferred thermal power increases with the air mass flow rate, even if the target cooling requirement, i.e. 20 kW, is not reached. This result suggests that a higher air mass flow rate is needed. The fluid temperature distribution presents a similar behaviour for both temperature sets, even if the mass-averaged temperature at the outlet section and the

transferred thermal power show noticeable differences as shown in Figure 7. Indeed, for Case 2, both temperature and thermal power are larger than Case 1.

This analysis shows that, with fixed temperatures along the engine outer walls, the selected mass flow rates are not adequate to guarantee enough transferred heat to the cooling system. Hence, in the next section, a CHT model will be employed to provide more accurate simulations of the thermal behaviour of the engine.

Fig. 6. Temperature distribution contours plots on Plane A (a) and Plane B (b)

Fig. 7. Mass-Averaged temperature at the outlet section (a) and transferred thermal power (b) vs air mass flow rate.

4 CHT Model

The Conjugate Heat Transfer strategy allows the evaluation of the heat transfer through the interface between the fluid and the solid regions. The model solves two sets of equations for both the fluid and the solid. As regards the fluid, Eqs. (1-3) are solved, with the k - ϵ model for the closure of the Navier-Stokes system. For the solid region, the thermal equation is solved:

$$\nabla(\lambda_s \nabla T) + S_h = 0, \quad (6)$$

where λ_s is the solid thermal conductivity and S_h represents any heat source within the solid region.

The interface is defined by considering *the coupled condition*. Indeed, the solver computes the heat transfer fluxes directly from the solution of adjacent cells. *The coupled condition* also assures that there is no mass transfer through the interface. The thermal properties of the solid material are needed to correctly represent its behaviour. Based on this strategy, only the temperature of the inner wall surfaces of the combustion chambers is set as a boundary condition. Specifically, this temperature is considered as a parameter to analyse the cooling system efficiency.

4.1 Computational Grid

In the mesh generation process, a perfect correspondence of the contact surfaces of fluid and solid cells is required to satisfy *the coupled condition* and to accurately solve the heat fluxes through the interface. Because of an unnecessary level of details of the initial engine CAD geometry, a new CAD geometry is generated, by removing some small details, as the smallest fillets, and by partially modifying the intake plenum. In Figure 8a, the new geometry is shown. The downstream region of the fluid domain is kept even for this analysis. The solid region has four empty zones, i.e. the two combustion chambers and the zone containing the valves and the shaft.

An unstructured mesh is generated for both fluid and solid regions. This strategy leads to an increase of the total number of grid elements with respect to the previous CFD analysis, with about 26 million cells in the fluid domain, and 9 million cells in the solid domain. Details of the grid are reported in Figure 8b for the fluid domain, and in Figure 8c for the solid domain.

Fig. 8. New CAD geometry (a) and computational grids for the fluid (b) and the solid (c) regions.

4.2 Boundary Conditions

As regards the fluid domain, the boundary conditions are the same as the previous CFD analysis. Indeed, the mass flow rate is set as the inlet boundary condition, ranging from 1

kg/s to 1.5 kg/s, since in the previous study it has been observed that the selected air mass flow rates were not sufficient to transfer the required thermal power. The pressure at the outlet section is equal to 1 bar. As regards the turbulent boundary conditions, the turbulent length scale is the same as before, whereas the turbulence intensity is set equal to 20 % of the bulk velocity at the inlet. The inlet air temperature is set to 288 K.

As regards the solid domain, only thermal boundary conditions are needed. The temperature for the combustion chamber internal walls ranges from 390 K to 430 K, which is a plausible range for the temperature in order to avoid the degradation of the oil and the deformation of the metallic components [1]. The thermal conductivity of the aluminium alloy is 202 W/m K. In the valves region, the wall temperature is 20 K lower than the temperature of the inner combustion chamber walls. All the other walls are considered as adiabatic.

The contact surfaces between fluid and solid domains are defined as interfaces with *the coupled condition*.

4.3 Results

First, a comparison between the previous CFD analysis and the CHT model is carried out and shown in Figure 9. Plane A is the same as that of Figure 4. The comparison is done by showing the flow streamlines, to assess if the simplified geometry provides similar results in terms of flow pattern. The figure shows some differences, even if the main flow structures, like the recirculation zones, are provided for both cases and are in the same region with approximately the same size. Major differences are mainly due to the less geometrical details of the new numerical domain, and, for some extent, to the different grid resolution of the bottom region.

Fig. 9. Coloured streamlines on Plane A for CFD (a) and CHT (b) simulations.

As regards the fluid and solid temperatures, Figure 10 shows the temperature distributions on Planes A and B with an air mass flow rate of 0.8 kg/s and two temperatures of the combustion chamber walls, $T_{ch,walls}$, equal to 410 K and 430 K. Figures 10 (a) and (b) show that, for both cylinders, the upper finned surface has a relatively low temperature, due to the cooling air that flows initially through this zone. However, a warmer air flows through Cylinder 1, because the air has already swept over the intake plenum. On Plane B, a higher temperature is observed downstream both cylinders for the case with 430 K due to higher thermal gradient.

Fig. 10. Temperature distribution contours plots on Plane A for $T_{ch,walls}=410\text{ K}$ (a) and $T_{ch,walls}=430\text{ K}$ (b) and on Plane B for $T_{ch,walls}=410\text{ K}$ (c) and $T_{ch,walls}=430\text{ K}$ (d).

The temperature distribution along the outer wall surfaces of the engine is shown in Figure 11a. The figure shows a more realistic temperature distribution from the crankcase to the head of the engine. Measurements of outer surfaces temperature have been performed by means of thermocouples located as in Figure 11b. A comparison of the experimental data with the numerical results is given in Table 2. The table shows that with a wall chambers temperature of 410 K, the numerical results are comparable to the measured temperatures.

Fig. 11. Temperature distribution along the engine outer wall surfaces for $T_{ch,walls}=430\text{ K}$ (a) and location of thermocouples probe for Cylinder 1 and Cylinder 3 (b).

Table 2. Comparison between experimental and numerical data on the outer surfaces of the engine.

	Experimental	Numerical - $T_{ch,walls}=410\text{ K}$	Numerical - $T_{ch,walls}=430\text{ K}$
Cylinder 1	86 °C	85.0 °C	97.9 °C
Cylinder 3	90 °C	88.5 °C	102.1 °C

Finally, a parametric analysis is performed to assess the efficiency of the cooling system as a function of the mass flow rate and of the combustion chamber wall temperatures. The results are summarized in Figure 12, where the air mass-averaged outlet temperature and the thermal power are given vs the air mass flow rate. The figure shows that, for all $T_{w, ch}$, a decrease of the mass-averaged outlet temperature occurs as the mass flow rate increases, whereas the thermal power increases. Specifically, the 20 KW target thermal power is reached with an inner wall temperature of 430 K, whereas for lower inner wall temperatures a further increase of the mass flow rate is required for each bank.

Fig. 12. Mass-Averaged temperature at the outlet section (a) and transferred thermal power (b) as a function of the mass flow rate.

5 Concluding Remarks

It is well known that spark ignition internal combustion engines for aeronautical applications operate within a specific temperature range to avoid structural damages, detonations and loss of efficiency of the combustion process. An accurate assessment of the cooling system performance is a crucial aspect in order to guarantee the operating conditions of the engine.

In this work an air-cooled 4-stroke SI engine, named CMD22, with a boxer 4-cylinders configuration has been considered for aeronautical applications. For this engine, a thermal power of about 20 kW needs to be transferred to the cooling system for each bank.

The cooling system of this engine has been studied by using two different numerical strategies. The first strategy is limited to the analysis of the flow field, whereas the second strategy includes the evaluation of the temperature distribution in the solid region by means of a Conjugate Heat Transfer method. The first CFD simulations provide a preliminary evaluation of the efficiency of the cooling system by setting fixed wall temperatures along the engine outer wall surfaces. However, the results show that a more realistic temperature distribution of the outer wall surfaces is required. Then, a CHT strategy has been implemented.

CHT simulations have been performed, by considering a simplified geometry of the engine that provides a fluid flow pattern which is close to that of the starting geometry. Based on the CHT approach, the inner wall temperatures of the combustion chambers is set as a

boundary condition, leading to a more accurate evaluation of the outer engine surfaces temperature distribution. The analysis shows that, with an inner wall temperature of the combustion chambers of 410 K, a good comparison with the experimental data is obtained. A parametric analysis has also been carried out with the air mass flow rate ranging from 1 kg/s to 1.5 kg/s and with temperature of the inner walls ranging from 390 K to 430 K. The results show that with 430 K, the target transferred thermal power of 20 kW is reached within the selected range of the air mass flow rate, and an increase of the inlet mass flow rate is needed for lower inner wall temperatures.

References

1. J. B. Heywood, *Internal Combustion Engine Fundamentals (2nd edit)*, McGraw-Hill Education, (2018)
2. W. Pulkrabek, *Engineering fundamentals of the internal combustion engine*, Pearson Education (2004)
3. G. Ferrari, *Motori a combustione interna*, Società Editrice Esculapio (2019)
4. Thermodynamics, Fluid Mechanics Group, W. J. D. Annand, *Heat transfer in the cylinders of reciprocating internal combustion engines*, Proceedings of the Institution of Mechanical Engineers 177(1), 973-996 (1963)
5. P. T. Nitnaware, S. G. Prachi, *Design Optimization Of An Air Cooled Internal Combustion Engine Fin Using CFD*, Journal of Multidisciplinary Engineering Science and Technology (2015)
6. P. Agarwal, M. Shrikhande, P. Srinivasan, *Heat transfer simulation by CFD from fins of an air cooled motorcycle engine under varying climatic conditions*, Proceedings of the World Congress on Engineering, **3** (2011)
7. A. Mishra, S. Nawal, R. Thundil Karuppa, *Heat transfer augmentation of air cooled internal combustion engine using fins through numerical techniques*, Research Journal of Engineering Sciences, **1**(2), 32-40 (2012)
8. P. Sagar, P. Teotia, A. D. Sahlot, H. C. Thakur, *Heat transfer analysis and optimization of engine fins of varying geometry*, Materials Today: Proceedings, **4**(8), 8558-8564 (2017)
9. F. Brusiani et al., *Definition of a CFD methodology to evaluate the cylinder temperature distribution in two-stroke air cooled engines*, Energy Procedia, **81**, 765-774 (2015)
10. S. Fontanesi, M. Giacomini, *Multiphase CFD-CHT optimization of the cooling jacket and FEM analysis of the engine head of a V6 diesel engine*, Appl. Thermal Eng., **52**(215), 293-303 (2013)
11. F. Berni, G. Cicalese, S. Fontanesi, *A modified thermal wall function for the estimation of gas-to-wall heat fluxes in CFD in-cylinder simulations of high performance spark-ignition engines*, Appl. Thermal Engineering, **115**, 1045-1062 (2017)
12. CMD22 - Engine for ultralight aircrafts, URL: <http://cmdavio.com/cmd22-aircraft-engine/>, (2020)
13. A.S. Sorathiya, P. H. Hiren, P. P. Rathod, *An Effect of Different Parameters of Fins on Heat Transfer of IC Engine-Review Study*, IOSR Journal of Mechanical and Civil Engineering, **11**, 63-71 (2014)
14. W. Sutherland, *The viscosity of gases and molecular force*, Philosophical Magazine, S. 5, **36**, 507-531 (1893)
15. D.C. Wilcox, *Turbulence Modeling for CFD, Second edition*, Anaheim: DCW Industries (1998)
16. H. K. Versteeg, W. Malalasekera, *An introduction to computational fluid dynamics: the finite volume method*, Pearson education (2007)