Heat Transfers in the Water Jacket of Heavy Duty Diesel Engine and its Analysis

Aishwarya Kadam[1], Bhushan Kamble[2]

[1]M. Tech. student, Department of Mechanical Engineering, KIT'S Collage of Engineering, Kolhapur, India
[2]Assistant Professor, Department of Mechanical Engineering, KIT'S Collage of Engineering, Kolhapur, India

---***---

Abstract - *A water jacket having a coolant flow crossing the space over the cylinder head is provided for an internal combustion engine of an automotive system. The internal combustion engine is equipped with a cylinder and a cylinder head. The water jacket includes a lower water jacket having side passages surrounding the cylinder and connected together by a plurality of branches disposed over the cylinder head, so as to create the coolant flow crossing the space over the cylinder head.*

***Key Words*: diesel engine, cooling system, temperature, heat transfer, cooling jacket.**

1. INTRODUCTION

Heavy duty diesel engines (CI) are used in a wide range of applications such as power generators, navy propulsion and rail traction applications, each of which uses different design of cooling systems. For this reason, a fundamental understanding of the processes considered in designing a cooling system for diesel engines is a necessary prerequisite for optimal cooling based on dimensions, type of components and thermal capacity. One of the most important parts in the cooling system of diesel engines is the water jacket installed around the liner and inside the cylinder head for the coolant fluid to flow inside it. The ability to predict the amount of heat transfer between the fluid and the walls of the suction system, the exhaust system and the cooling system is very momentous for the designer engineer, because the heat transfer of CI engines is important in many ways:

- Protecting the materials used in sensitive parts of the engine against melting or deformation
- Increasing the engine's useful power by reducing energy leakage inside the combustion chamber
- Reducing contamination by reducing engine warm-up time
- Improving lubrication performance and reducing the hammering phenomenon.

2. GEOMETRY PRODUCTION AND COMPUTATIONAL MESH PREPARATION

The targeted engine in this study is a heavy duty 16-cylinder turbocharged diesel engine equipped with median cooler the diameter-to-length ratio of the pathway is 165 mm/195mm and the ratio of the nominal power/nominal speed is 2000 kW/1800 rpm. The cylinder head of this engine is ductile cast iron. The three- dimensional model of the cooling pathway around the liner and inside the cylinder head can be seen in Fig. 1(a-c). Each row of the engine consists of eight cylinders cooled in series. Coolant fluid is divided into two parts after circulating around each liner. Part of it goes to the next liner, and the other part enters the cooling pathway inside the cylinder head. To carry out the thermal analysis, the solid and liquid parts of one cylinder (including the cooling pathway around the liner and the cooling pathway inside the cylinder head) have been simultaneously entered to the ANSYS Meshing software and meshed using the unstructured meshing method. Furthermore, in order to reduce the number of meshes and meshing errors, geometry correction and removal of additional lines have been done. The number of the generated meshes in the software is about 2,500,000, mainly consisting of hexagonal cells, as well as a small number of pyramidal cells and prismatic cells. The meshed model of this geometry is shown in Fig. 1(d-e). In addition, it is worth noting that in order to increase the quality of the computational mesh, the mesh dimensions in a range of geometries such as regions with extreme temperature gradients, surfaces in contact with each other, and surfaces that have a common interface with fluid analysis have been fined, and thus, the precision of the calculations has increased. On the other hand, by expanding mesh dimensions in regions that do not have much effect on the results of the analysis, the computational time can be reduced.

3. BOUNDARY CONDITIONS

Boundary conditions are divided into two categories; fluid flow boundary conditions and thermal boundary conditions. To analyze fluid flow, the flow characteristics must be defined in the inlet and outlet channels of the water jacket. In the inlet channel, the mass flow boundary condition is used, and the outlet pressure boundary condition is used in the outlet flow. The static pressure of the fluid in the outlet is 0.4 bar. Furthermore, the cooling fluid is considered to be in a general combination of water and ethylene glycol with a volume ratio of 50% water and 50% ethylene glycol, where the physical and thermal properties of this combination are considered at the engine operating temperature (i.e. 363.8 K). (See Table 1). It should be noted that given the critical condition of the cylinder head operation at maximum engine power, all values have been calculated at engine speed of 1800 (rpm) and full- load operating conditions. Table 2 shows the boundary conditions of the wall of the cylinder head. In addition, for a better understanding, a view of the points defined in geometry as boundary conditions is shown in Fig. 1(b).

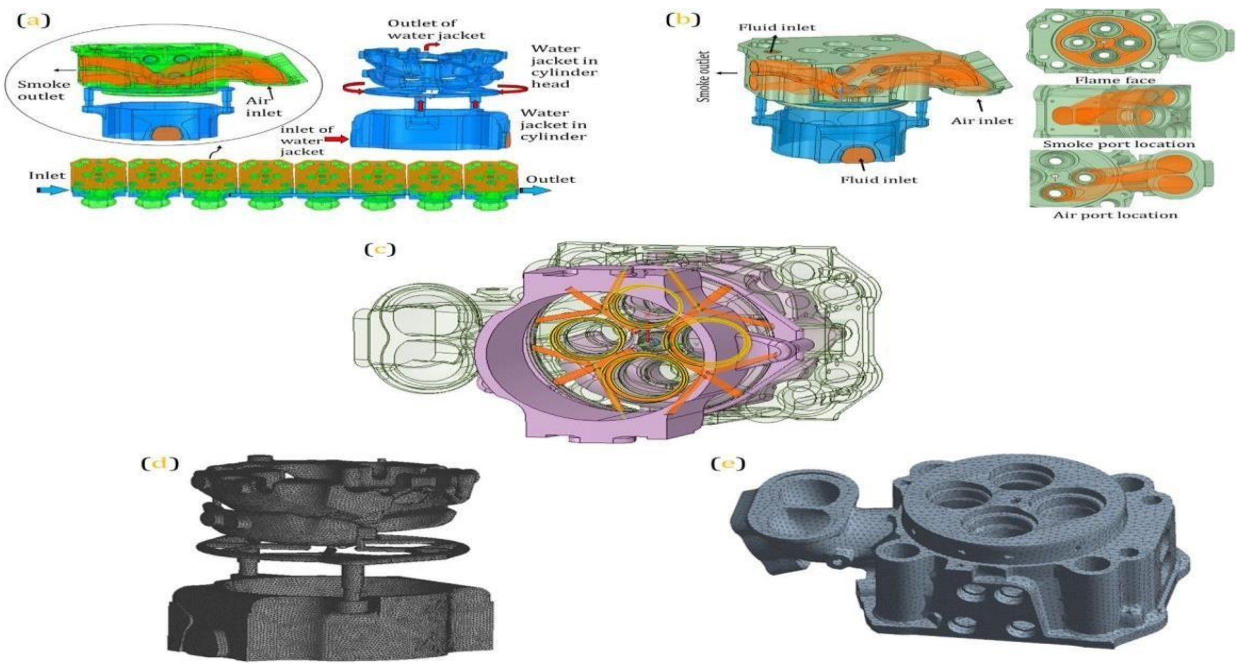

Fig. 1. Overall view of the water jacket and cylinder head (a-c) ; meshing of the Geometry (d-e).

Table 1

Thermo physical properties of 50% water and 50% ethylene glycol at the engine operating temperature (i.e. 363.8 K)

Physical properties

Specific heat	3470 (j/kg K)
Density	1030 (kg/m3)
Heat conduction coefficient	0.44 (W/m K)

Table 2

Temperature and heat transfer coefficient in cylinder head walls

Region	Temperature (K)	Heat transfer coefficient (W/m2 k)
Flam face	1050	600–650
Smoke port	l 880	300–350
Air port location	380	100–110

Furthermore, in this simulation for determining the relationship between momentum, continuity and pressure equations, the solution starts with a simple method, and the repetition continues until the variations in the transient and steady solution reach the lowest possible value. At the end, the values of temperature distribution and heat flux in the cylinder head, velocity distribution, pressure distribution, and convection heat transfer coefficient in the water jacket are obtained.

4. RESULTS AND DISCUSSIONS

4.1 CFD SIMULATION Validation

One of the important issues in turbulent flow simulations is to ensure the presence of sufficient number of boundary layers to observe the gradient of velocity. Fluid dynamics mechanisms in the near-wall region are investigated with quasi-experimental equations in dependent variables. These equations are based on functions of a dimensionless distance perpendicular to the wall. In addition, considering the dependence between the determined thickness of the boundary layer and the value of y+ is important. The value of y+ depends on the geometry and the fluid flow variables, whose actual value is determined after the simulation. Fig. 2(a) shows the value of y+ within the acceptable range .The comparison of the results of CFD analysis with experimental data indicates the precision and accuracy of the present simulation.

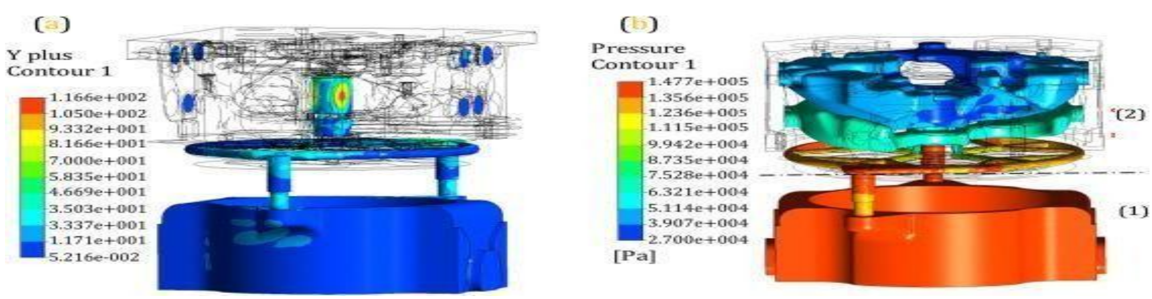

Fig. 2. View of the contour y+ (a) and distribution of pressure over the cooling jacket (b).

4.2 Pressure Distribution in the Cylinder Head Water Jacket

As noted above, the pressure variations in the convective heat transfer do not have much effect on the heat transfer rate. However, when the heat transfer of the boiling type occurs, fluid pressure can have a considerable effect on the heat transfer rate. Therefore, the effect of pressure should be considered at low velocities where boiling has more importance. Pressure distribution in the engine's cooling water jacket in the present problem with a mass flow of 0.65 (kg/s2) is shown in Fig. 2(b). In the first region, where the fluid is moving around the liner, the pressure drop is almost negligible due to the relatively large hydraulic diameter of the cooling path around the liner. In contrast, in the second region, where the coolant fluid is moving in the cylinder head, a relatively large pressure drop occurs in these regions due to the smaller hydraulic diameter of the cooling paths and the more complex fluid flow path. A notable point in the design of the present geometry is the relatively high pressure of the coolant fluid in the regions of exhaust valves and glow-plugs, which delays the film boiling and makes it stay in the nucleate boiling area.

4.3 Velocity Distribution in the Cylinder Head Water Jacket

The distribution of the velocity field and the flow lines in the cooling water jacket around the liner and inside the cylinder head is shown in Fig. 3(a-b). The highest velocity of the coolant fluid is 8.43 m/s, which occurs in the region around the valves and around the glow-plugs, and the lowest fluid velocity is about 0.3 m/s near the exhaust gas outlet. The high velocity of the coolant fluid in these sensitive thermal regions, where there is a risk of the occurrence of film boiling phenomenon, causes these regions to be protected from this phenomenon. In addition, the low velocity of the coolant fluid around the seat of the exhaust valve and around the exhaust pathway increases the sensitivity to the starting point of the film boiling due to the high thermal flux, so it is necessary to modify the design to improve the fluid motion.

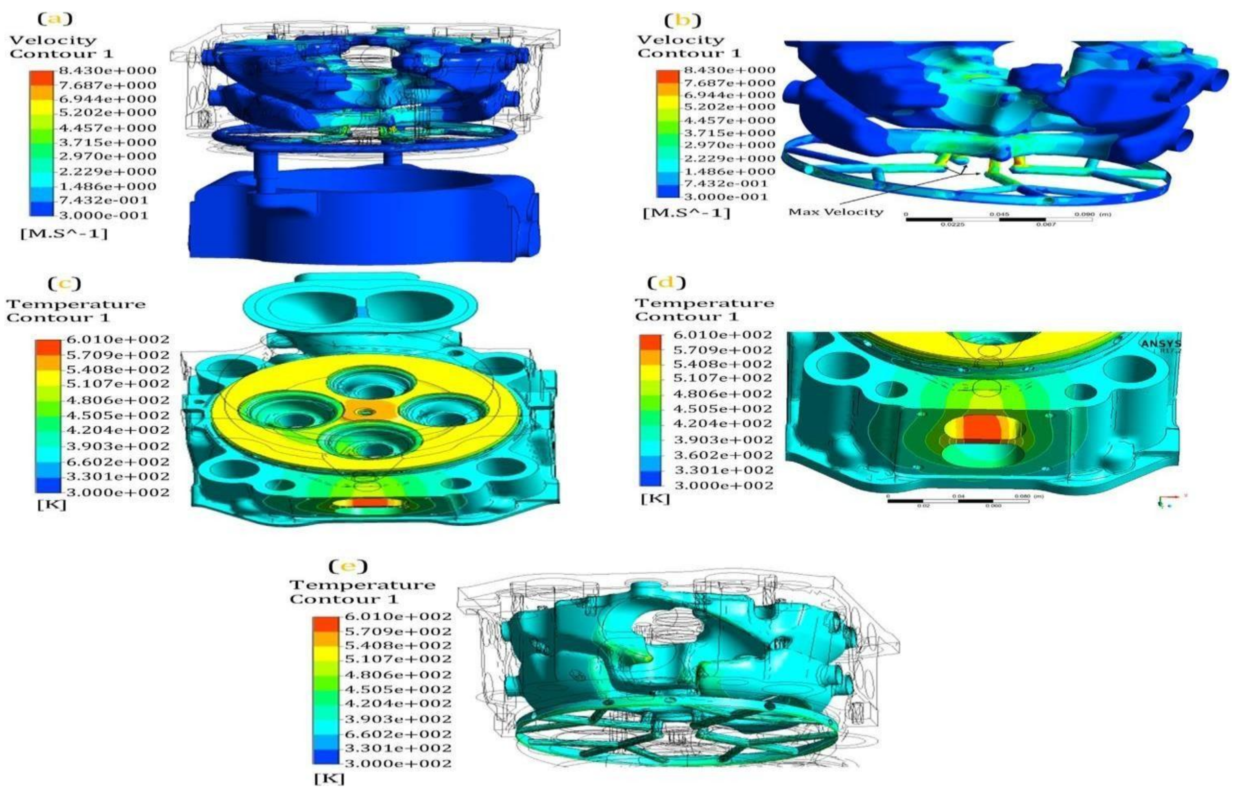

Fig. 3. View of the velocity (a-b) and temperature contours (c-e).

4.4 Temperature Distribution Is the Cylinder Head Water Jacket

Figs.3(c-e) to 4(a-b) show the results of the distribution of temperature and thermal flux in the cylinder head and the cooling water jacket. The results indicate that the cylinder head maximum temperature in the bridge area occurs between the exhaust valves. An increase in the temperature of the coolant fluid at the outlet of the exhaust pathway can also be seen in Fig. 4(c-e). This is due to the low velocity of the coolant fluid and the lack of design of the vortex path behind the exhaust valves. In addition, according to the simulation results, it is predicted that the coolant fluid temperature will increase by 12.5 degrees in the pathway from the cylinder inlet to the outlet of the cylinder head. The heat flux passing through the wall of the cylinder head without considering the boiling phenomenon is also depicted in Fig. 4(a-b). The comparison of this figure with the numerical simulation diagrams of the boiling phenomenon shows that the use of boiling phenomenon can significantly increase the heat flux. As shown in Fig. 4(a), the heat flux passing through the valves is larger than the other parts. Therefore, the lack of proper cooling in these regions can cause heat cycle fatigue at the level of the cylinder head flame. Therefore, this necessitates more precision in designing cooling water jacket and better use of the boiling phenomenon around this region. On the other hand, the thermal flux in the seat of the exhaust valve is significantly low, which indicates low heat transfer rate in these regions. The reason for this according to the above explanations is the lack of design of the vortex path for cooling in these sensitive thermal regions, which reduces the thermal flux and increases temperature at these surfaces (Fig. 4b). The strength point of this research is the integrated or simultaneous solution for the thermal analysis of the coolant fluid flow. That is, the thermal analysis of the structure of the cylinder head and the coolant fluid inside it has been carried out simultaneously.

4.5 Distribution of Heat Transfer Coefficient in the WaterJacket

The results of the heat transfer coefficient (HTC) for the cooling water jacket are shown in Fig. 4(c- d). The heat transfer coefficient has increased considerably in the points of the water jacket where the temperature is higher. In the present simulation, the maximum HTC is about 49,000 W/m2k, whose maximum values occur in the vicinity of the glow-plugs according to the temperature distribution and velocity distribution. Also, HTC is greatly reduced in the vicinity of the exhaust path due to the high temperature of the exhaust gases and the low velocity of the coolant fluid around the wall, which causes the accumulation of heat in these regions.

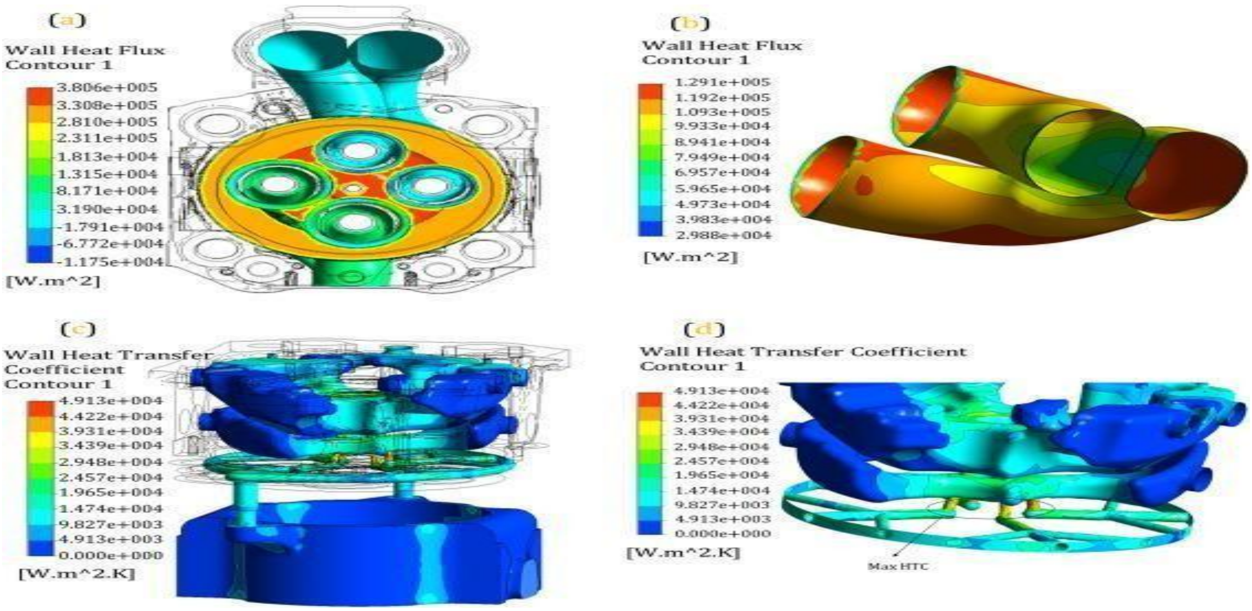

Fig. 4. View of the heat flux (a-b) and heat transfer coefficient (c-d)

5. CONCLUSIONS

In this paper, the distribution of the velocity field, pressure, and heat transfer coefficient (HTC) in the cooling water jacket of a diesel engine has been investigated through three-dimensional CFD method using ANSYS/Fluent software. The significant point is the simultaneous solution of the structure and fluid in this analysis. The results show that:

- Using nucleate boiling flow regime can increase the heat transfer coefficient, which results in dissipating more heat from the surface and provides a more uniform temperature distribution for theengine.
- The temperature at which the nucleate boiling will start increases by increasing water flow velocity.
- The effect of pressure on decreasing or increasing the heat transfer rate is very important when the heat transfer occurs through boiling. In this way, high pressure can delay the start offilmboiling.
- The low velocity of the coolant fluid and the lack of design of the vortex path behind the exhaust valves lead to an increase in the fluid temperature in the seat of the exhaust valve and around it, especially the narrow bridge between the two valves.
- The comparison of the thermal flux passing through the wall of the cylinder head with and without considering the boiling phenomenon shows that the occurrence of the boiling phenomenon can significantly increase the heat flux.

- In the thermal analysis of the water jacket, it was determined that the maximum value of the convective heat transfer coefficient is about 49,000 W/m2k, that its maximum value occurs in the vicinity of the glow-plug according to the temperature distribution.

6. REFERENCES

[1] H. Lee, Heat Transfer Predictions using the Chen Correlation on Sub cooled Flow Boiling in a Standard IC Engine, SAE Technical Paper 2009-01-1530, 2009.

[2] H. Lee, A. O'Neill, Forced convection and nucleate boiling on a small flat heater in a rectangular duct: experiments with two working fluids, a 50–50 ethylene glycol—water mixture, and water, Proc. Inst. Mech. Eng., Part D: J. Automob. Eng. 223 (2009) 203–219.

[3] L. You-chang, G. Xiao-hong, C. Dan, Research on cooling system for 4-cylinder diesel engine, SAE Technical 2007-01-2064, 2007.

[4] H. Punekar, S. Das, Numerical Simulation of Subcooled Nucleate Boiling in Cooling Jacket of IC Engine, SAE Technical Paper 2013-01-1651, 2013.

[5] V.A. Romanov, N.A. Khozeniuk, Experience of the Diesel Engine Cooling System Simulation, International Conference on Industrial Engineering, Procedia Engineering 150, 490–496, 2016.