International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

Design and Thermal Analysis of Cylinder Head in Diesel Engine

Md Shabuddin
M.Tech (Thermal engineering),
P.G. Scholar
Nishitha College of engineering and technology
shabu335@gmail.com

Rangdal Srikanth
Mtech(AMS), Asst professor
Nishitha College of engineering and technology

ABSTRACT

The aim of this study is to discover the characteristics of the cooling water in the cylinder head of a high-power diesel engine by using the Computational Fluid Dynamics (CFD). A 3-D geometric model of a cylinder head water jacket was previously constructed using Unigraphics software. A mesh was then imposed on the model using the preprocessor of Fluent software, GAMBIT. Three types of thermal boundaries are defined and the standard model is utilized to carry out numerical simulations with the CFD software Fluent. The pressure distribution, velocity field distribution, heat transfer coefficient distribution, and temperature distribution of the cylinder head water jacket are presented and analyzed. In addition, the impact flow rates have on the cooling system of a cylinder head is studied. The relevant figures show that enhancement in flow rate will increase the velocity and heat transfer coefficient of the coolant flow but in the meanwhile bring in higher pressure loss. The design of the cylinder head water jacket is evaluated to be satisfactory in terms of cooling effects. Simulation results are compared with the work done by others and limited experimental data, and it is pointed out that the disparity of heat transfer coefficients is mainly due to over simplification of the model and inappropriate use of the turbulence model or wall function heat transfer coefficient. Some suggestions on the types of boundary conditions and appropriate turbulence models are proposed, which include the definition of boundary conditions on different locations and the use of coupled simulation.

Key words: *cylinder head water jacket; coolant flow; numericalsimulation; CFD*

INTRODUCTION

Motivation and Significance of Study

Internal Combustion (IC) Engines play an important role in the modern industry. Great efforts are put to improve the power performance and fuel economy of IC

Engines. As the speed and loading capability of IC engines increase, the thermal and mechanical load increase greatly. Under normal working condition, the peak temperatures of burning gases inside the cylinder of IC Engines are of the order of 2500 K, consequently the temperatures of parts, including valves, cylinder head, and piston in contact with gases, rise rapidly due to a large amount of heat absorbed. The substantial heat fluxes and temperatures lead to thermal expansion and stresses, which further destroy the clearance fits between parts and escalate the distortions and fatigue cracking of parts. To prevent overheating, cooling must be provided for the heated surfaces. However, it should be noted that overcooling will also cause some serious problems, such as combustion roughness, lower overall efficiency and high emission pollution. To some extent, further improvements on the performance of an IC Engine depend on the effective resolution of heat transfer problems. Therefore, an optimal design of the cooling system is required to maintain trouble-free operation of engines, which must take into account all the considerations described above.

As one of the most complicated parts of an IC Engine, the cylinder head is directly exposed to high combustion pressures and temperatures. In addition, it needs to house intake and exhaust valve ports, the fuel injector and complex cooling passages [1]. Many compromises must be made in design to satisfy all these requirements. The complicated structure of a cylinder head leads to the difficulty in acquiring necessarily detailedinformation for design for conducting flow and heat transfer experiments. With improved computer performance and rapid development of Computational Fluid Dynamics (CFD), numerical simulation provides a tool for engineers to use in evaluating their design. With the assistance of numerical simulation techniques, some basic features, such as pressure and velocity distributions of the flow field, can be easily predicted. Coupled with Computer-Aided Design (CAD), the structure of a cooling system for an engine can be modified and optimized, in order to control the temperature at key zones, decrease the power loss, and improve the reliability of parts working at high temperatures.

GEOMETRIC AND MESH MODELS

Geometric model

Characteristics of the Solid Model

The geometric model of the cylinder head was created using the software Unigraphics (UG) modeling software, which is useful in component and surface modeling, virtual assembly, and in generating engineering drawings. The component studied in this project is one cylinder head of an eight-cylinder 1015 high-power diesel engine. Some of the characteristic parameters of the investigated engine are presented in the table below.

Table 2 Engine Characteristics

Engine Type	Diesel TC
Cylinders	8V
Bore	132mm
Stroke	145mm
Displacement	16L
Compression Ratio	17
Rated Power	440kW
Rated Engine Speed	2100r/min

The 1015 engine has a split structure, namely one cylinder head for one cylinder. The cooling system of this engine is distinguished by the manner in which coolant is distributed. The cylinder block water jackets are connected in series, while the cylinder head water jackets are connected in parallel. This design enables the coolant collected in cylinder blocks to enter each cylinder head separately through transfer channels separately, which is also the reason why only one cylinder head has been investigated.

Principles of Modeling

Based on experience gained from the repeated modeling and numerical simulation process, the following principles of modeling are recommended.

1. Embodiment of features of the structure and flow

A cylinder head consists of the firedeck, side plate, fuel injector protection sleeve and valve guide. It can be described as a box that has been trimmed by intake and exhaust ports and padded by intake and exhaust valves, and fuel injector. All these features create a rather irregular cylinder head structure with special cavities and bores, which complicate the process of building a water jacket model, since the shape and position of parts in cylinder head determine the shape of the water jacket. Moreover, the most complicated areas in the cylinder head usually happen to be key places where investigators put most emphases. In those locations, either the velocity and heat fields have large gradients, or advection is significant. Therefore, the geometric

International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

features of these parts should be modeled as accurately as possible.

2. Impact on mesh generation

Complexity of the solid models may result in potential difficulty in mesh generation, poor quality of the mesh model and unnecessary work. To simulate viscous effects near walls, it is usually needed to create a boundary layer, which requires some simplification in order to avoid unacceptable meshes due to severe curvature change of surfaces.

It can be concluded that building the solid model is a trade-off process between the above two principles. Appropriate simplification is needed on the premise that most geometric details have been retained.

The completed cylinder head solid model is displayed below.

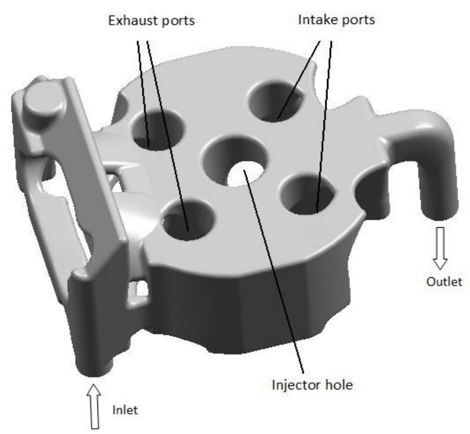
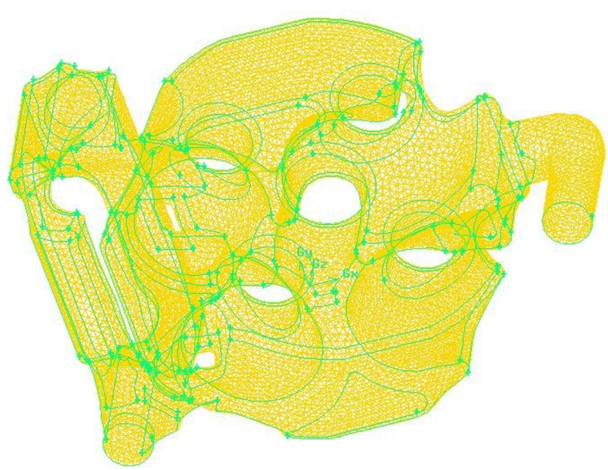


Figure 2 Geometric Model of Cylinder Head Water Jacket

Mesh Model

A high quality mesh model is one of the most important factors that determine the successful prediction of the behavior of an objective. The geometric model of cylinder head water jacket was imported to GAMBIT, a preprocessor of the CFD software Fluent. Tetrahedral volume meshing elements were chosen to generate the mesh model. The mesh model is shown below, which contains 130365 cells and 29150 nodes.



International Journal of Research
International Journal of Research
Available at https://edupediapublications.org/journals
p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

Figure 3 Mesh Model

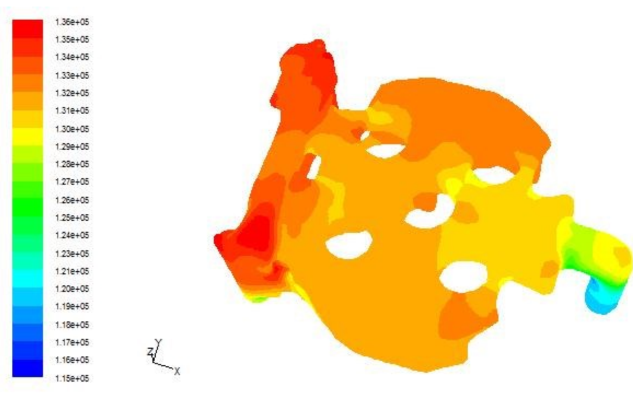
Using a 0-1 scale for mesh quality with 0 representing the best and 1 the poorest, more than 90 percent of the cells are of the scale less than 0.5, which implies a high quality of the mesh model.

RESULTS AND ANALYSIS

With the high quality mesh model, the solutions converge within half an hour after 400-600 iterations, depending on the cases studied. In this paper, the results of pressure field, velocity field and temperature field are presented and analyzed. More attention is paid to the heat transfer process. The influence of flow rates on those parameters mentioned above is discussed. Refer to Figure 2 for the definition of coordinates. The bottom of the cylinder head water jacket, where inlet and outlet are located, is referred to as the datum plane, and the center of the fuel injector bore on this plane is the origin. The positive z-axis is directed toward the top of cylinder head water jacket, the positive y-axis is directed toward the paper, and the positive x-axis is directed toward the outlet. Because the computation of the pressure and velocity fields is independent of the temperature, the simulation results of these two fields will only be presented once, for all three thermal boundary conditions. The results shown were calculated at the inlet flow rate of 1.0508 kg/s if they were not specified.

Simulation Results with Convection Thermal Boundary

1. Pressure Field Distribution



p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

Figure 4 Contour of Pressure Field Distribution (Pa)

It can be seen from the contour of pressure that the outlet displayed the lowest pressure, while the side plate presented the highest pressure, which indicated that the largest pressure drop occurs between the inlet and outlet. The interior of the water jacket does not have obvious pressure drop. The pressure loss between the inlet and outlet is calculated to be 8.4177 kPa, which is a good indicator of low resistance in flow and potential to enhance mechanical efficiency.

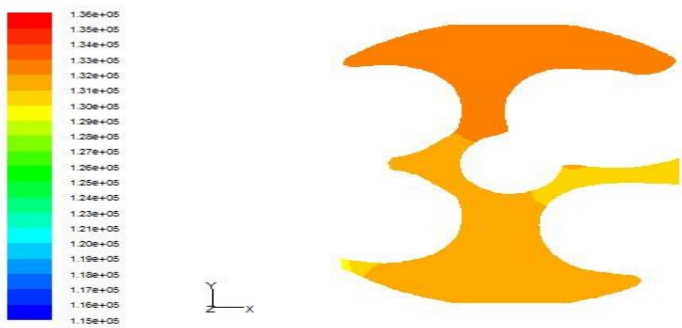
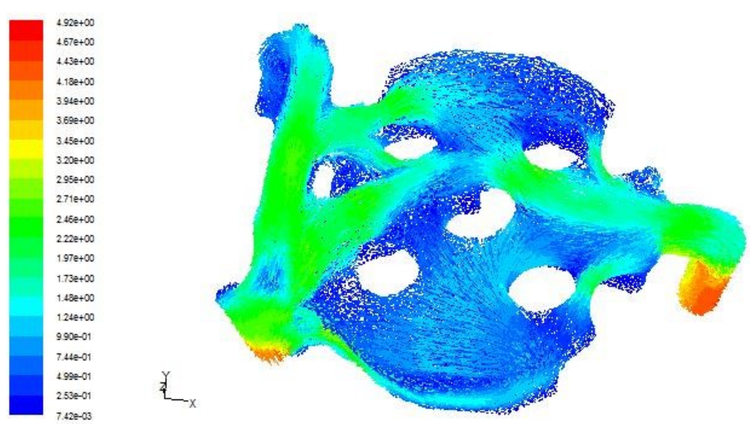


Figure 5 Pressures (Pa) on Nose Zones at z = 35 mm

Focusing on the nose zones, the narrow bridges between intake ports, exhaust ports, intake and exhaust ports, and two ports and the fuel injector, it is noticed from Figure 5 that the pressure varies between 129 kPa and 132 kPa, which is a small difference that reduces the possibility of bubble producing, and therefore the destruction of the cylinder head.

2. Velocity Field Distribution



p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

Figure 6 Contour of Velocity Field Distribution (m/s)

The average velocity magnitude over the whole flow field is 0.892 m/s, which meets the need of flow rate 0.5 m/s for cooling the engine. Some key zones were investigated particularly.

1) Exhaust port

The design for this cylinder head forced the flow to go around the exhaust port first, where most of the heat was output, then cooled down the fuel injector and finally the intake port. It is seen that the velocity of the flow around the exhaust port was higher than other locations, which ensures that the cylinder head is relatively evenly cooled. Because of the small inlet and outlet diameters, the velocity at these locations was largest.

2) Fuel injector

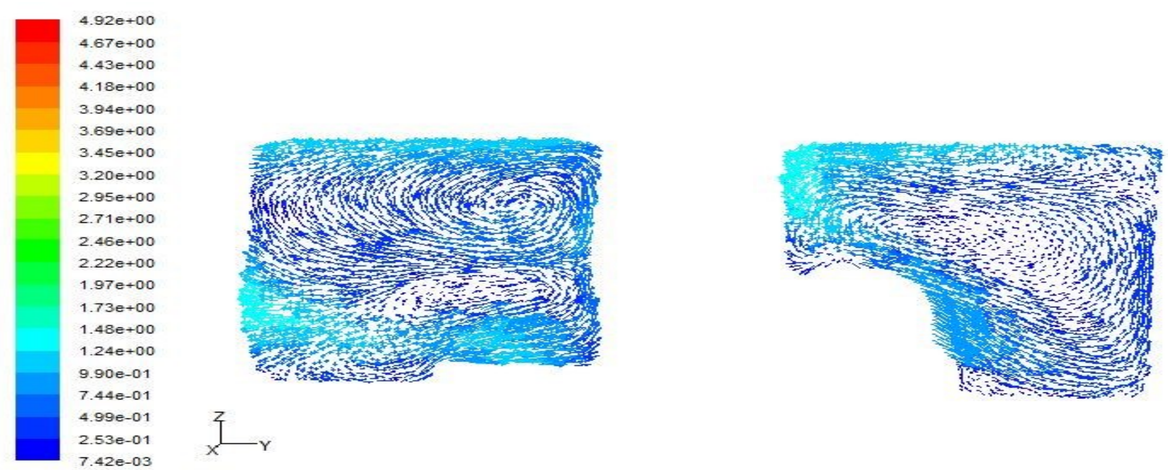
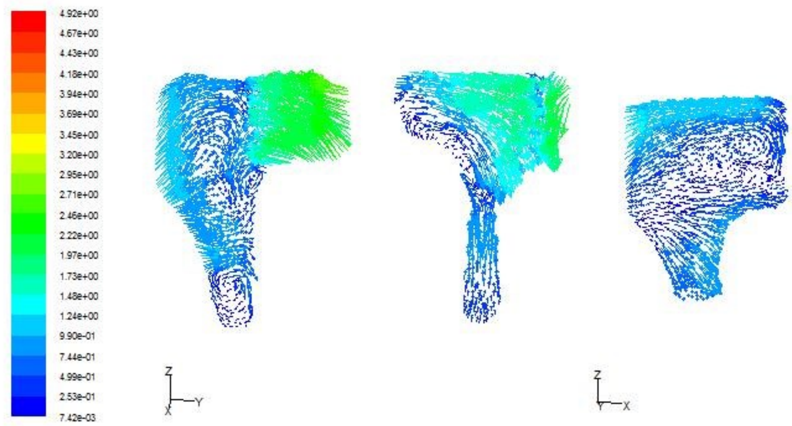


Figure 7 Velocity Field around the Fuel Injector (m/s)

The upper part of the fuel injector hole is narrower than the lower part. Such a design coordinates with the flow regularity that the flow comes from the under part of the hole and up to the top of the cylinder head, which ensures a relatively high velocity of the flow on the top. In this case, the velocity changed from 0.7m/s to 1.25m/s.

1) Nose zones



The higher flow rate in the narrow bridges between intake and exhaust valves ensured the cooling effects in the most problematic regions.

3. Heat Transfer Coefficient Distribution

The cooling effects can be reflected directly by the heat transfer coefficient (HTC). The HTC shown below is the surface HTC provided by Fluent.

The simulation results of the surface heat transfer coefficients are as follows:

1) Wall

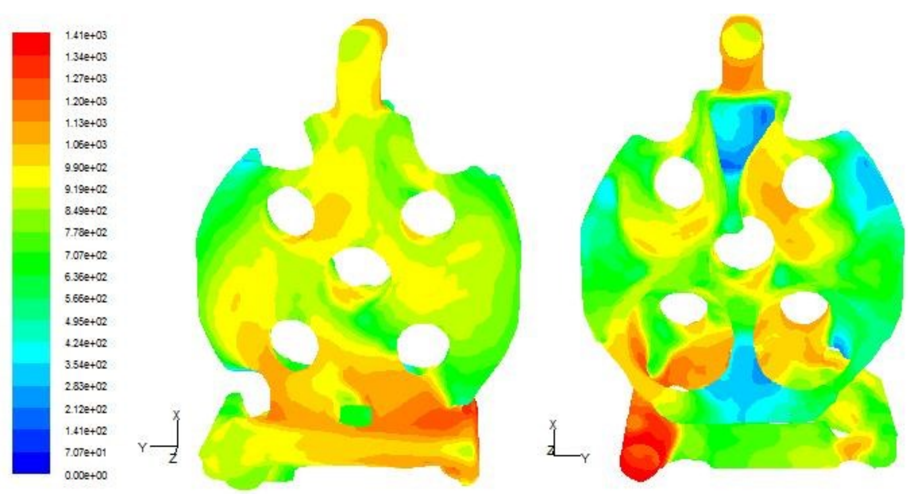


Figure 9 Contour of Surface HTC Distribution ($W/(m^2 \cdot K)$)

The area-averaged surface HTC is calculated to be 836.52 $W/(m^2 \cdot K)$.

2) Specific regions

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

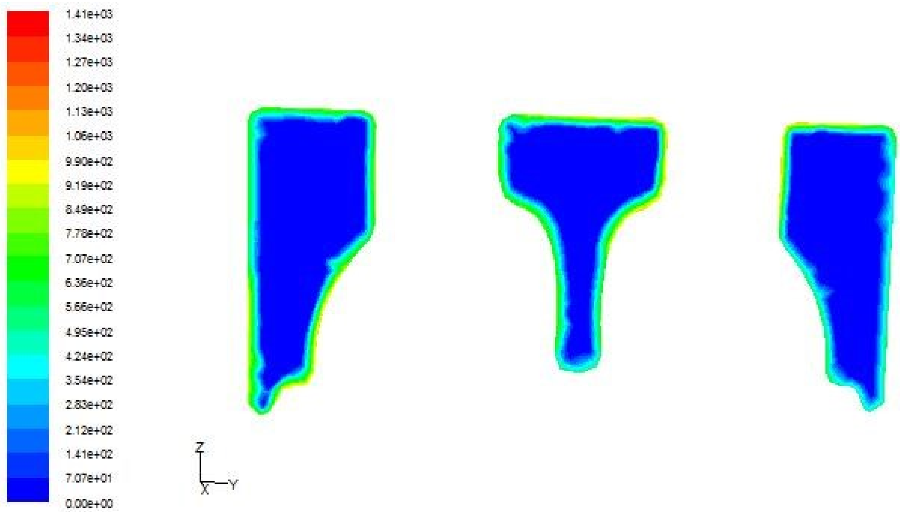


Figure 10 Surface HTC ($W/(m^2 \cdot K)$) on Nose Zones at x = -30 mm

Figure 10 displayed the surface heat transfer coefficient distribution of the narrow bridge regions at x =-30 mm. Data at different locations and on different height planes was collected and summarized in the table below.

Table 4 Surface HTC at Different Locations

HTC / Section	Exhaust valves	Intake and exhaust valves	Intake valves	Fuel injector
z=18mm	600-750	——	——	——
z=20mm	700-900	600-800	——	600-700
z=25mm	900-1000	800-900	700-800	700-800
z=30mm	900-1200	800-900	750-900	900-1100

The fire deck is located at the plane where z = 0.

4. Temperature Distribution

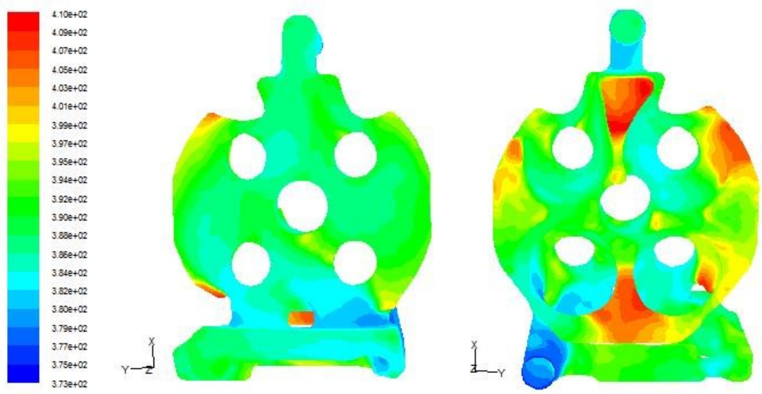


Figure 11 Contour of Temperature Distribution (K)

With the limitation of the boundary conditions, the temperature distribution was displayed in the range of 373K and 418K. The bottom contacted with the fire deck, was of extremely high temperature, which is reasonable in the sense of the structure characteristics and work principle of the cylinder head. The temperatures around the wall of the annular cavities, where the fresh air and exhaust passed, were ranging from 384K to 397K, which showed acceptable cooling effects of the cylinder head. By comparison of HTC and temperature distributions, it is evident that the regions with high heat transfer coefficient correspond to ones with low temperature, and vice versa. This also implies the inherent relations between the velocity and heat transfer coefficient, which will be discussed.

Simulation Results with Fixed Heat Flux Thermal Boundary

1. Heat Transfer Coefficient Distribution

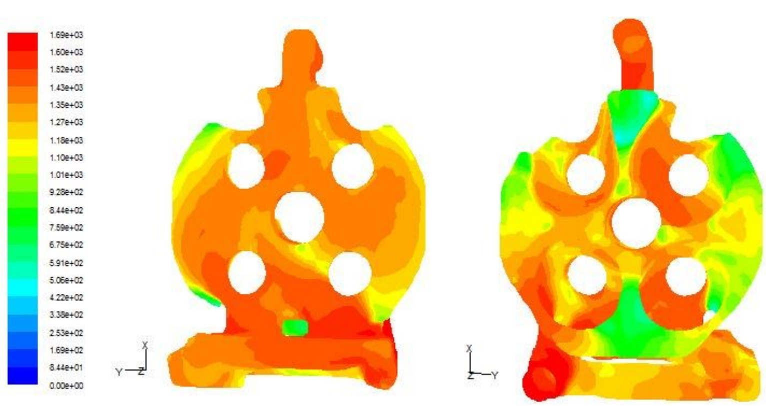


Figure 12 Contour of Surface HTC Distribution ($W/(m^2 \cdot K)$)

The area-averaged surface HTC at the wall is 1287.0704 $W/(m^2 \cdot K)$, indicating a relatively good cooling effect.

2. Temperature distribution
1) At the wall surface

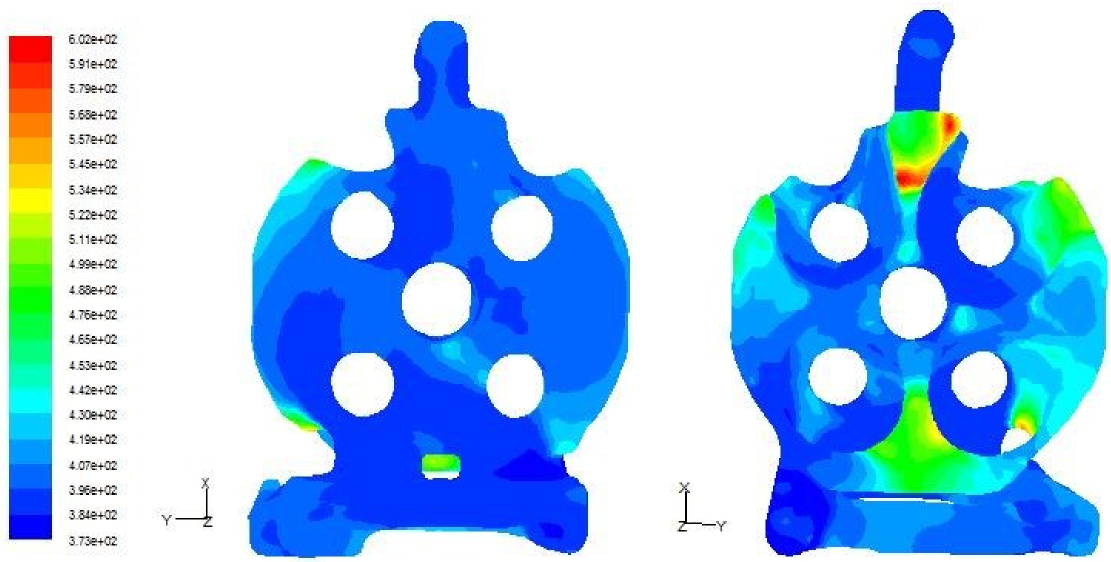


Figure 13 Contour of Temperature Distribution at Wall (K)

2) Sections orthogonal to z-axis

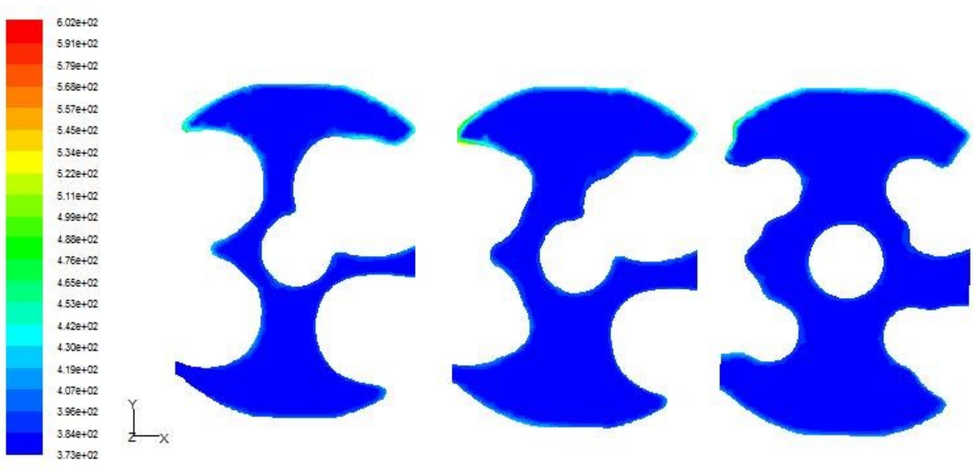


Figure 14 Contour of Temperature Distribution at z = 35mm , 45mm and 55mm (K)

The contour of the wall surface shows that the hottest part is located at the narrow bridge on the heating surface, but the magnitude exceeds the boiling point of glycol, which means either it is not realistic, or the design of the cylinder head will cause destruction due to film boiling.

Simulation Results with Fixed Temperature Thermal Boundary

Heat Transfer Coefficient Distribution

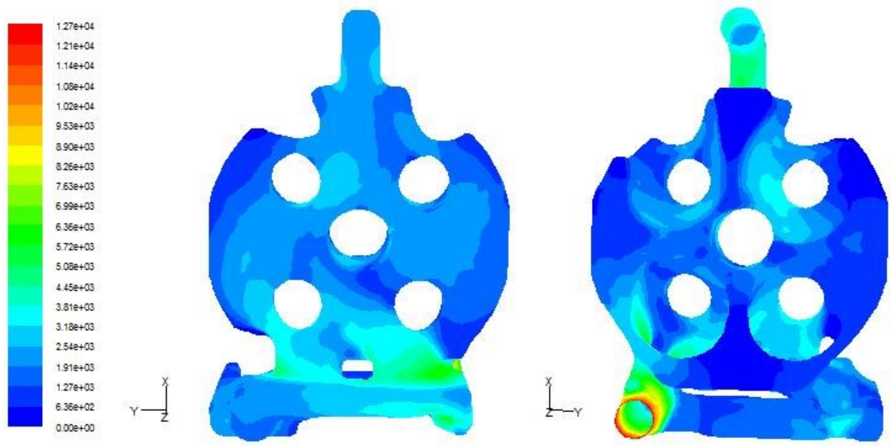


Figure 15 Contour of Surface HTC Distribution ($W/(m^2 \cdot K)$)

The area-averaged surface HTC at the wall is 2142.2268 $W / (m^2 \cdot K)$, which is greater than the results of convection and fixed heat flux thermal boundaries. Moreover, the area-averaged total surface heat flux reaches 277706.5 W / m^2 , which is almost twice the referenced [22] heat flux of 150 kW / m^2 . In a cylinder head of an engine, the temperature varies from point to point, which depends on the location of the heat source and the detailed structures of different regions. It is not practical to assume the temperature is constant over the wall surface, but the simulation conducted under the fixed temperature thermal boundary provided a profile of the heat transfer coefficient, which depicted the general distribution, but not the exact value of the heat transfer coefficient.

HTC Distribution Using Wall Function Formula

The wall function HTC is based on the semi-empirical formula for HTC (refer to Equation 2.31), in spite of the different thermal boundaries chosen. Therefore, the wall function HTCs in the previously described three types of wall boundary conditions give the same result, the contour of which is shown in Figure 16.

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

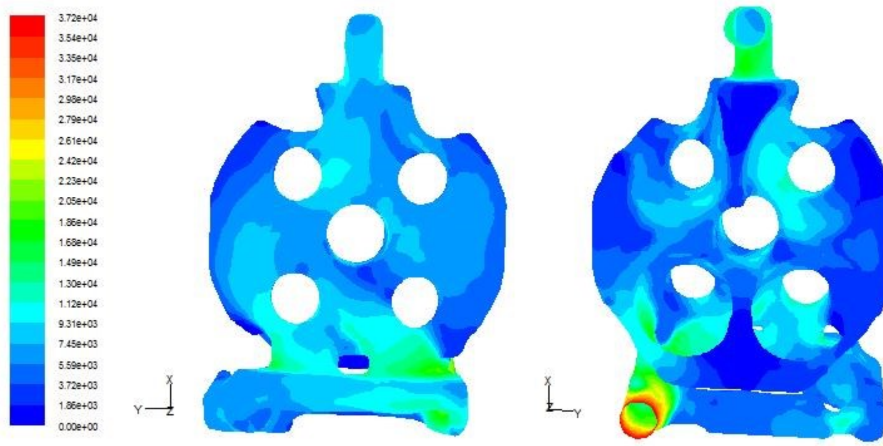


Figure 16 Contour of Wall Function HTC Distribution ($W/(m^2 \cdot K)$)

In addition, the surface HTC and wall function HTC distributions present similar profiles. It means that locations of high HTC and low HTC are almost the same in different contours, but the magnitudes of HTC at these locations are different. It can be expected that the wall function HTC will be more reliable when heat flux or temperature gradient is sufficiently large.

Effects of Flow Rates

As the engine has been strengthened, it's necessary to check the consequent effects on the cooling system. Four typical working conditions (refer to Table 3) were taken into account, as listed in figures below. The convection thermal boundary condition is the most reasonable one for numerical simulation, which gives not only the variations of the pressure field, velocity field, temperature field and heat transfer coefficient field at different zones, but also the magnitudes of these parameters at particular points, which are close to the experimental data. Therefore, the simulation results used below are obtained from this condition.

1. Pressure Loss

Pressure loss is computed by taking the difference of area-averaged pressures at the inlet and outlet.

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

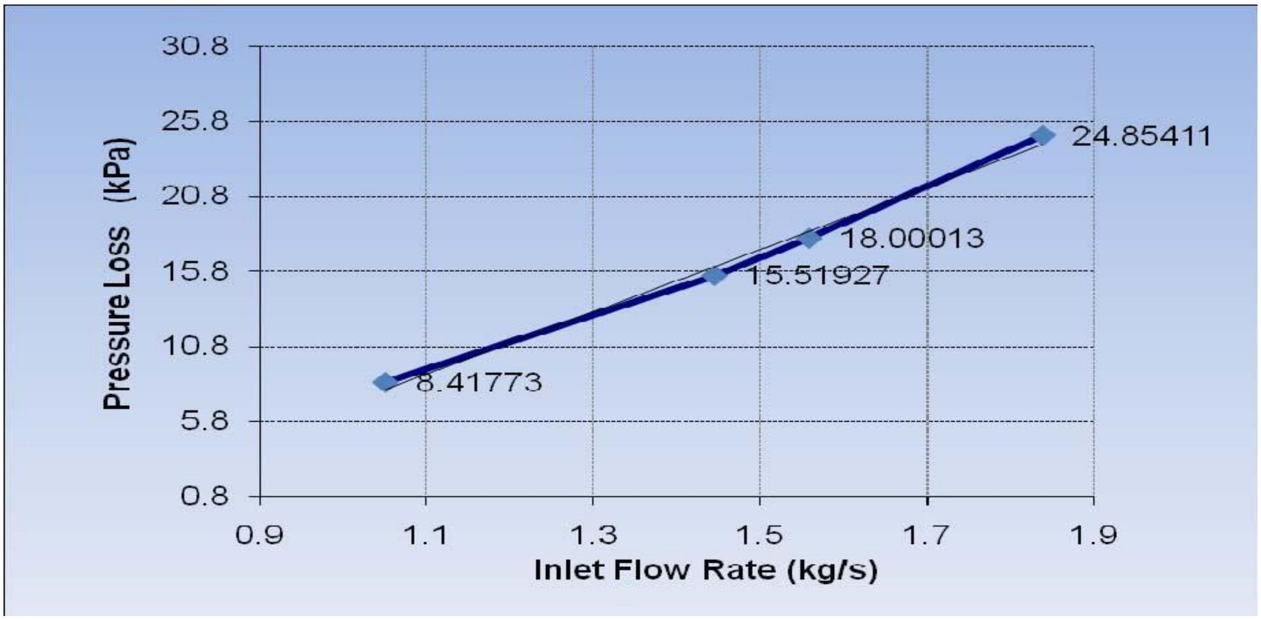


Figure 17 Pressure Losses at Different Flow Rates

2. Velocity

The velocity presented below is the volume-averged velocity calculted over the interier of the water jacket.

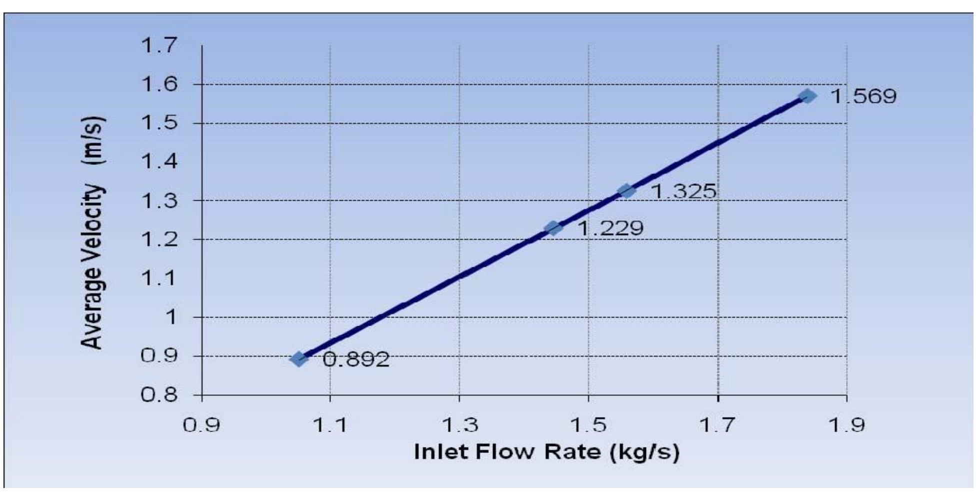


Figure 18 Average Velocity at Different Flow Rates

3. Surface HTC with Convection Boundary Condition

p-ISSN: 2348-6848
e-ISSN: 2348-795X
Volume 03 Issue 11
July 2016

The surface heat transfer coefficient is the area-averaged HTC at the surface of the water jacket.

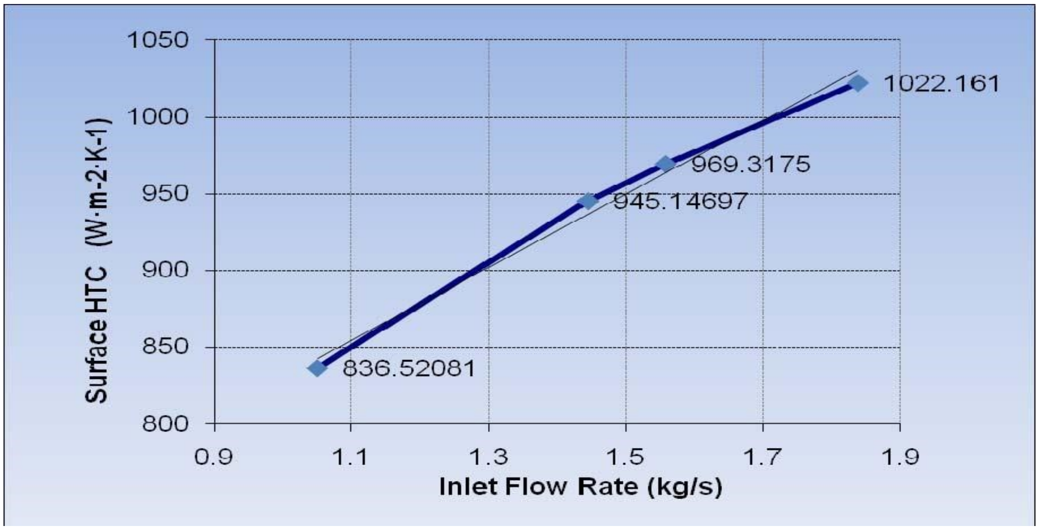


Figure 19 Surface HTC at Different Flow Rates

With the increase of rated power, the inlet flow rate increases, the velocity and heat transfer coefficient increase consequently. However, the pressure loss will increase too, which indicates the decrease of the torque efficiency. It is known that if the pressure loss exceeds 50 kPa , the resistance of the cooling system will be large, and loss of the power will be considered to be high.

CONCLUSIONS AND FUTURE WORK

Evaluation of Cooling Effects of the Cylinder Head

In general, the work of this cylinder head water jacket satisfied the demand of cooling effects, which can be analyzed in the following ways.

1. The coolant in cylinder head performed a good flow rate distribution, and relatively even pressure distribution.
2. The arrangement of the parts in cylinder head provided necessary forces to enhance the cooling in problematic zones, so that the intake and exhaust valves and the fuel injector will not be too hot to break down.

Evaluation of the Simulation

The flow field and heat field of the cylinder head water jacket were computed based on the boundary condition values, which were obtained from the experiments and experience. However, they should also be tested and verified by available information. The following table provides some information of the coolant-side heat transfer process, which was reported in the literature.

Table 5 Boundary Conditions Considered in the Mode

Boundary condition description	Link	Heat transfer coefficient [W m^{-2}K^{-1}]	Bulk temperature [K]
Insulated surfaces (negligible heat-transfer rate)	20	0 (adiabatic)	–
Free surfaces (contact with ambient air)	29	5	320
Cooling passages	30	3 000*	350
In-cylinder surfaces	21	450	1120
Intake-port surfaces	22	800	330
Exhaust-port surfaces	23	800	700

In spite of the different types of engines studied, the results should not be of way too big differences.

By the careful comparison with all available, but limited papers on the heat transfer analysis of a cylinder head, it can be seen that the simulation results of the pressure and flow rate distributions are relatively consistent with each other, while the results on heat transfer vary a lot.

According to the information given in the above table, this present study showed more accurate simulation results. Moreover, it should be noticed that many of the investigators obtained rather high heat transfer coefficients, even higher than 10^4 $W / (m^2 \cdot K)$, which deviated from the experimental data. The main causes of the disparity may lie on two aspects:

1. The geometric model and mesh model are over simplified or some of the important characteristics of the structure are ignored.
2. The chosen turbulence model is not appropriate for the studied model, and the wall function HTC is not an acceptable prediction for the HTC profile of the coolant flow given the boundary conditions described previously.

Future Work

It has been years that researchers put efforts on a realistic and reliable simulation of the cooling system of the cylinder head. Until now, there are a variety of results especially in heat transfer coefficient, some of which are not consistent with the experimental data. This study is another attempt to better describe the heat transfer process in a cylinder head. The results are closer to the limitedly available information obtained directly from the experiments, but they still need to be improved on the following

potential aspects.
1. Boundary conditions

This study already showed that the convection thermal boundary for the wall is the best way of simulating the heat transfer process, and the same value was considered for the whole wall surface. However, it is evident that different zones of the wall are exposed to different environment, resulting in different boundary conditions which should be set separately according to the locations of the zones.

To achieve this purpose, boundary layers should be employed to divide the model into different areas in Gambit, which will take effect when adding wall boundaries in Fluent.

2. Turbulence model and semi-empirical HTC formula

The classical two-equation $k-\varepsilon$ model has been widely used with its accuracy and universality. The simplicity is also an advantage of this model, which is based on and developed from the Reynolds' analogy. All the assumptions of the Reynolds' analogy made the problems simpler and easier to solve. Nevertheless, the $k-\varepsilon$ model can not reflect the Reynolds' relaxation effect along the stream line, so it's not the best simulation tool for more complex flow in engineering.

It is shown that the standard wall function described the distribution of heat field with relatively high heat transfer coefficients, which were inconsistent with the available experimental results. The semi-empirical heat transfer coefficient formula could be improved further, which needs to be investigated.

3. Coupled simulation

In this study, the entity is the water jacket of a cylinder head. If we can set up the solid cylinder head model as well, both structural and fluid simulations can be carried on to obtain a more satisfactory result. Finite Element Method will be applied in this kind of simulation.

4. Improvement of design

Once the cooling effects of the current model were figured out, it is significant to make some change in design to achieve a better cooling model, to carry on simulations and evaluate the new model, and to modify it again, which is a repeating process. For this cylinder head model, it is worth trying to add a water hole in order to strengthen the cooling effect around the fuel injector.

REFERENCES

[1]. M. Diviš, R. Tichánek, M. Španiel, Heat Transfer Analysis of a Diesel Engine Head. *Acta Polytechnica*, 2003 Vol. 43, No. 5 : 34-39.

[2]. Malcolm H.Sandford,Lan Postlemthwaite. Engine Coolant Flow Simulation-A Correlation Study. *SAE TECHNICAL PAPER SERIES*, 930068.

[3]. Mark D.Moechel. Computational Fluid Dynamic (CFD) Analysis of

a Six Cylinder Diesel Engine Cooling System with Experimental Correlations. *SAE TECHNICAL PAPER SERIES*, 941081.1994.

[4]. Franz J.Laimbock, Gerhard Meister et al. Application in Compact Engine Development. *SAE TECHNICAL PAPER SERIES*, 982016.

[5]. Toyoshige SHIBATA, Hideo MATSUI, Masao TSUBOUCHI et al. KATSURADA. Evaluation of CFD Tools Applied to Engine Coolant Flow Analysis, MITSUBISHI MOTORS TECHNICAL REVIEW 2004, (16):56-60.

[6]. Curtis M.Hill, Glenn D.Miller, Robert C. Gardner. 2005 Ford GT Powertrain-Supercharged Supercar. *SAE TECHNICAL PAPER SERIES*, 2004-0101252.

[7]. Xunjun Liu, Qun Chen, Ju Li, Kang Li, Haie Chen, Automotive Diesel Engine Water Jacket CFD Analysis[J]. *Transaction of CSICE*, 2003, Vol.21, No.2: 125-129.

[8]. Haie Chen, Fanchen Meng, Jianlong Song, Jun Li, CFD Analysis on Engine Cooling Jackets[J]. *Design Computation Research*, 1000-3703(2003).

[9]. Qun Chen, Xunjun Chen, CFD Analysis of Cooling Water Jacket of Vehicle Diesel Engine CA498 [J]. *Design Computation Research*, 1000-3703(2003)11-0008-04:8-11.

[10]. Numerical Simulation on Flow and Heat Transfer of Cooling System in a Six-Cylinder Diesel Engine [J]. *Transaction of CSICE*, 2004, Vol.22, No.6: 525-531.

[11]. Shengguan Qu, Wei Xia, Numerical Simulation and Experimental Research on Cooling Water Flow around Cylinder liner for Heavy Duty Diesel Engine [J].*Chinese Internal Combustion Engine Engineering*, 2004, Vol.25 No.4: 32-35.