A Review of computational fluid dynamics analysis of cylinder head water jacket of a diesel engine

Nitesh R. Patel[1], Devendra Patel[2], Darshan Gajdhar[3]

[1]P.G.student, Department of thermal engineering, Dr. Jjivraj Mehta institute of technology, , Anand
[2,3] Associate professor, Department of Mechanical engineering, Dr. jivraj Mehta institute of tech., Anand

---

Abstract – *To improve engine performance, engine heat transfer and cooling is always been a crucial area of interest. As the cylinder is one of the most complicated parts of IC engine, it is required to maintain uniform cooling. Computational fluid dynamics methods are used today to provide clear and more detailed data on temperature, pressure and flow variation. The main objective of the project is to carry out heat transfer as well as flow analysis of existing cooling jacket of 4 cylinder 4 stroke 4 valve medium duty diesel engine. The engine head is designed using CATIA V5 R20 modelling software. ANSYS FLUENT is used to carry out heat transfer and flow analysis and then investigate the factors affecting cooling performance to optimize the parameters and validate them using comparison of existing research papers' readings. Some suggestions are given to improve the cooling performance for future work by modifying the jacket design and mixture proportion.*

Key Words: computational fluid dynamics, diesel engine, cylinder head water jacket, heat transfer co-efficient, pressure and temperature contours, FLUENT.

1 INTRODUCTION

IC engine is an important prime mover used in various fields mainly in automotive and power generation. Great efforts are put to improve the power performance and fuel economy of IC engines. As the speed and loading capability increase, the thermal and mechanical load increase greatly. Under normal working condition, the peak temperature of burning gases inside the cylinder of IC engines are the order of 2500K, therefore the temperature 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 temperature lead to thermal expansion and stresses, which further destroy the clearance fits between parts and escalate the distortions and fatigue cracking of parts [2].

Most susceptible to overheating are the bottom of the cylinder head, the upper belt of the cylinder liner, the piston crown, the upper compression ring and the exhaust valve cap. The stresses from moving thermal field in these elements, due to the periodic nature of the work of the internal combustion engine, may give rise to fatigue crack propagation with specific mechanisms, more dangerous than crack propagation mechanisms in static thermal fields[3].

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]. The complicated structure of a cylinder head leads to the difficulty in acquiring necessarily detailed information 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.

1.1 REVIEWS OF CFD ANALYSIS AND ITS IMPORTANCE ON DIESEL ENGINE COOLING JACKET OF OTHER LITERATURE

The verification of engine cooling design requires expensive experimental test rigs setups and lengthy tests. It is essential to have the ability to predict accurately the rate of heat transfer in forced convection sub cooled boiling regime under various pressures and geometry conditions in an engine to reduce mainly the cost and the size of the engine together with an appropriate cooling system. The supplementary information could reduce or eliminate experiments by reliable numerical analyses. This is of relevant interest in advance engine design and analysis of subsystems such as cylinder head, liner, crank case and so on these days. Sub cooled boiling condition is common and noticeable phenomenon in many industrial applications as well as in engine cooling system because of its high heat transfer criteria.

The heat transfer and cooling system in IC engine has been studied extensively by a number of researchers with great contributions. These work aimed at developing predictive tools to help the design of cooling systems with good precision. Experimental work has been carried out on different rectangular or circular ducts to gather important data for model development and validation [1-4].

Other researchers have developed heat transfer algorithms and have successfully applied them. In the approach made by Rohsenow [5], he has successfully correlated the nucleate boiling caused by heat transfer and proposed a basic equation for saturated pool boiling. Bowring [6], Bergles and Rohsenow [7], correlated a subcooled boiling condition using the simple power law, which needs empirical constants to be derived by experiments. Mostinskii [8] introduced the effect of reduced pressure in correlation as a key factor to predict the onset of boiling. Chen [9] proposed a superposition models which is widely accepted and used today especially in engineering applications and engine design. His model is based on Froster/Zuber for a saturated fluid condition in vertical flows. Later, this was extended by Butterworth [10] to a subcooled regime. In the subcooled regime Chen distinguished two extra competing effects on the total wall heat flux, the enhanced convective transport due to bubble agitation, and the flow-induced suppression of the nucleate boiling. Bennett D.L. and Chen J.C. [11] modified Chen approach for calculating HTC using aqueous ethylene glycol for saturated condition in vertical tube. Kandlikar [12] and Shah [13], suggested correlations considering geometrical combination of early works. These model which do not consider superposition theory, formulate the effective heat transfer coefficient as a correction function due to nucleate boiling.

In CFD analysis of flow in complex geometries, it is hardly possible to define a reasonable bulk flow Reynolds number. H.Steiner et al. [14] developed a modified superposition model for computation of the specific wall heat transfer rate in subcooled boiling flow based on the Chen-type superposition idea. The suggested correlation for the flow-induced suppression depends only on local flow quantities which are in general the numerical solution of the liquid phase flow field. In this work, the vapor bubble lifts-off on a heated surface is referred to as the lift-off point and force balance equations are written for that time. An accurate method of heat transfer calculation is derived based on H.Steiner et al. model [14], where both flow and thermal fields are solved.

In this paper, a fast method is proposed to see the effects of boiling and conductive heat transfer concurrently. The paper presents an attempt to use a coupled analysis inclusive of boiling effect to solve for cylinder head, crankcase and its liner, also to investigate the engine cooling design assembly as a whole. A-two way coupling approach with temperature dependent material behavior has been used to obtain wall temperature and HTC. G.Aurtenetxe1 [15] used this approach by interchanging data between flow and thermal analysis.

1.2 MODELLING STRATEGY

The methodology is presented in this paper is based on CFD and validation is done by comparison of the simulation results with available experimental data and work done by others. In present study, 6-cylinder, 4-stroke, 4-valve medium duty diesel engine is selected.

The geometric model of the cylinder head was created using the software CATIA V5 R20 modeling software, which is useful in component and surface modeling, virtual assembly, and in generating engineering drawings. Some of the characteristic parameters of the investigated engine are presented in the table below.

Description	Specification
Engine Type	In Line 4 Cylinder 4 Stroke 4 Valve
No. Of Cylinders	4 Cylinders
Combustion Chamber	Pent roof + Curved Top Piston Type
Displacement	2.351L
Bore	86 mm
Stroke	105.1mm
Compression ratio	15.5:1
Rated Power	150kw at 3500rpm
peak torque	130N.m at 2500rpm

Table 1 Engine description

1.3 GEOMETRIC MODEL

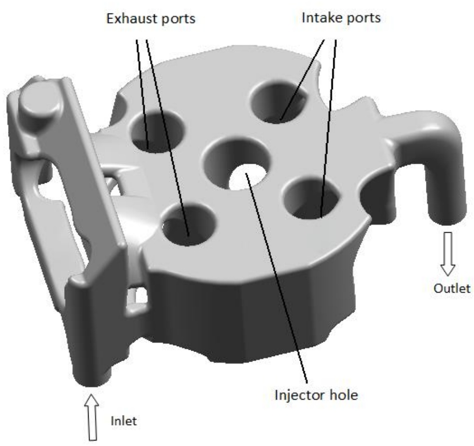

Figure 1geometric model of single cylinder head of diesel engine from internet.

2. FLUID PROPERTIES

CFD simulation will be carried out using the coolant mixture of water and Ethylene Glycol at a constant inlet temperature 373Kelvin. Fluid properties are listed below:

- Density : 1008.8 kg/m^3
- Specific gravity : 0.000675 kg/(m.s)
- Thermal conductivity : 0.39964 W/(m.K)
- Freezing point : 238K
- Boiling point of glycol : 470K (197 degree celsius)
- Boiling point of glycol and water : 401K

3. BOUNDARY CONDITION

1. Inlet Boundary
Type: Mass flow inlet: 3.75kg/s
Temperature: 373K
2. Outlet Boundary
Type: Pressure outlet
Pressure: 1.2 bar.
3. Wall Boundary

Three types of thermal conditions at the wall surface

1) Fixed heat flux
The value used for the heat flux is 150 kW
2) Fixed temperature
The value used for the temperature at the wall is 418K, which is the empirical data often used for the heating surface of the cylinder head.
3) Convection
The values used for heat transfer coefficient 3000·W/m^2K and free stream temperature 418K were taken from the literature.

4. RESULTS AND ANALYSIS

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.

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.

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.

The area-averaged surface HTC at the wall is 1287.070W/m^2.K, indicating a relatively good cooling effect.

5. CONCLUSION

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.

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.

6. REFERENCES

[1]. Amit V. Paratwar, and D.B. Hulwan, (Aug 2013), "Surface Temperature Prediction and Thermal Analysis of Cylinder Head in Diesel Engine", International Journal of Engineering Research and Applications (IJERA), pp.892-902

[2] Marcel Diviš, and Josef Božek (1998), "Heat Transfer Analysis of Diesel Engine Head", Research CenterCzech Technical University, Prague.

[3] P.N.Shrirao, and Dr. Rajeshkumar SambheI, (March2014), " Thermal Analysis of Mullite Coated Diesel Engine Cylinder Head Using 3-D Finite Element Method", International Journal of Engineering and Innovative Technology(IJEIT), Volume 3, Issue 9, p 244-248.

[4] J. Krishnaveni, G. Sowmya and U. Sudhakar, (Nov 2014), " Thermal Analysis Of Cylinder Head By Using Finite Element Analysis", International Journal of Mechanical Engineering (IJME) ISSN(P): 2319-2240, Vol. 3, Issue 6, p.p. 11-20.

[5] M. Fadaei, and A. Noorpoor, (2011), "New thermo-mechanical analysis of cylinder heads using a multi-field approach", ScientiaIranica, volume 18, Issue 1, pp 66–74.

[6] K. Ashok Reddy, and S. Satpagiri, (2006), "Heat Flux and Temperature Distribution Analysis of I C Engine Cylinder Head Using ANSYS", Dept of Mechanical Engineering, Guru Nanak Institutions Technical Campus, Ibrahimpatnam, Hyderabad.

[7]C.Donoselli, and C.Gorla, (1998), "Experimental evaluation of thermal characteristics of internal combustion engine cylinder liner", Journal of Metallurgical science and technology.

[8] Quingzhao Wang, (2003), "Numerical analysis of cooling effect of a cylinder head water jacket", Thesis of GIT.

[9] Dr. S. M. Khot, (2010), "Comparative cooling of engine cooling jacket models of diesel engine using CFD", Third Biennial national conference of Nascent Technologies, NCNTE .

[10] Pamela. M. norris, and Mardi.c. hastings, (1988), "An experimental investigation of liquid coolant heat transfer in diesel engine", SAE Technical paper no.891898 .

[11] K.shiling, and G. woschni, (1991), "Experimental investigation of instantaneous heat transfer in cylinder of high speed diesel engine", SAE Technical paper no.790833.

[12] Zhang Ping (2008), "Numerical simulation of coolant flow in diesel engine cooling water jacket", CAU, Beijing, 100083.

[13] john J. Truhan, and Peter J. Blaub, (2005), "A rig test to measure friction and wear of heavy duty diesel engine piston rings and cylinder liners using realistic lubricants", Tribology, International,Vol.38,PP211-218.

[14] Tutorial Manual Guide, "Combustion Engine ANSYS Mesh Tutorial".

[15] Gataowski, and J.A., (1990), "Evaluation of a Selectively Cooled Single Cylinder 0.5L Diesel Engine", SAE, Paper 900693, USA.

[16] Borman G., and Nishiwaki K., (1987), "Internal combustion engine heat transfer, Progress in Energy and Combustion Science".

[17] M.A. Hazrat, H.H. Masjuki, and S.C. Pang, (2003), "Steady state analysis of coolant temperature distribution in a spark ignition engine cooling jacket" Centre for Energy Sciences, Department of Mechanical Engineering, Faculty of Engineering, University of Malaya.

[18] T.H. Liou, W.W. Shabbir, Yang Z., and Zhu J., (1994), "A new k-epsilon eddy viscosity model for high reynolds number turbulent flows - model development and validation", NASA TM 106721.

[19] Helgi Fridriksson, and Bengt Sunden, (2011), "CFD investigation of heat transfer in a diesel engine with diesel and PPC combustion models", SAE Technical paper no.01-1838.

[20] Aditya Mulemane, and Ravindra Soman, (2007), "CFD based complete engine cooling jacket development and analysis", SAE Technical Paper no. 01-4129.

[21] Michael Vincent Jensen, (2012), "Heat Transfer in Large Two-Stroke Marine Diesel Engines" Doctoral Thesis, Technical University of Denmark.

[22] Carlos Adolfo, and Finol Parra, (1998), "Heat Transfer Investigations in a Modern Diesel Engine", Acta Polytechnica, Vol. 43 No. 5/2003.

[23] John B Heywood, (1998), "Thermal loading and temperature measurement in diesel engine component "Fundamentals of I.C engines.

[24] R.Tatschi, and B. Basara, (2006), "Advanced Turbulent Heat Transfer Modeling for IC –Engine Applications Using AVL FIRE", International Multidimensional Engine Modeling User's Group Meeting.

[25] JukkaTiainen, and IlariKallio, (2004), "Heat Transfer Study of a High Power Density Diesel Engine" SAE Technical Paper no. 01-2962.

[26] Malcolm H.Sandford, and, Lan Postlemthwaite, (2007), "Engine Coolant Flow Simulation-A Correlation Study", SAE Technical paper series, 930068.

.