Case Studies in Thermal Engineering 12 (2018) 497–509

Contents lists available at ScienceDirect

Case Studies in Thermal Engineering

journal homepage: www.elsevier.com/locate/csite

Numerical investigation of heat transfers in the water jacket of heavy duty diesel engine by considering boiling phenomenon

M. Gholinia[a], M. Pourfallah[a,*], H.R. Chamani[b]

[a] *Department of Mechanical Engineering, Mazandaran University of Science and Technology, Iran*
[b] *Department of Mechanical Engineering, Iran University of Science and Technology, Iran*

ARTICLE INFO

Keywords:
Heavy duty diesel engine
Cooling jacket
Heat transfer
Subcooled nucleate boiling
Numerical Simulation

ABSTRACT

The flow of the coolant fluid and its heat transfer directly affect the cooling performance, heat load of the hot parts and also the thermal efficiency of the diesel engine. The proper estimation of heat transfer and temperature distribution in a diesel engine is essential for investigating the thermal stresses and calculating its performance, which requires a precise simulation of the cooling water jacket. An efficient approach to study the cooling system is to simulate using Computational Fluid Dynamics (CFD) as a three-dimensional model by simultaneously solving the structure and fluid, which leads to accurate prediction of wall temperature and heat flux. In the present paper, the distribution of heat transfer coefficients (HTC) in the cooling jacket of a 16-cylinder heavy-duty diesel engine has been calculated using ANSYS/Fluent based on 3D-CFD method. Also, equations of subcooled boiling phenomenon have been solved based on methods of Chen and BDL, and the effects of fluid pressure, velocity, and temperature (At the time of the phenomenon of boiling) on the heat transfer of cooling jacket wall have been studied. The results show that the sensitive thermal region that is at risk is the region between the exhaust valve and around the glow-plug. This region if not properly cooled, will result in gas leakage from the combustion chamber, which will result in a decrease in engine power and torque.

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.

* Corresponding author.
E-mail address: m.pourfallah@ustmb.ac.ir (M. Pourfallah).

https://doi.org/10.1016/j.csite.2018.07.003
Received 5 May 2018; Received in revised form 17 June 2018; Accepted 8 July 2018
Available online 10 July 2018

Heat transfer from the wall of the cylinder head to the cooling fluid is a combination of convective heat transfer and nucleate boiling. Due to the passage of the operating fluid from the cooling water jacket, the bubbles formed from the hot surface of the wall are separated and enter a lower temperature region, and thus a large amount of energy on the hot wall of the cylinder head is transferred. By increasing the wall temperature, the number of bubbles formed at the surface also increases. The bubble formation regime gradually reaches a stage where the number of bubbles separated from the hot surface of the cylinder head is more than the number of bubbles formed. The time from the formation of the first bubble until this stage is called nucleate boiling. Nucleate boiling is beneficial in the hot regions of the cylinder head, i.e. the region between the exhaust valves and around the glow-plugs, since it transmits a high thermal flux for a low temperature change. It is worth noting that nucleate boiling consists of two regions; saturated boiling and subcooled boiling. Since the type of boiling phenomenon that occurs in the engine is of a kind of subcooled flow boiling, therefore the stated topics relate to the subcooled nucleate boiling. Many studies have been conducted to optimize the design of cooling water jacket of CI engines. One of the first activities in the field of boiling heat transfer is the work done by Norris et al. [1]. In a study, they tried to find pure convection regions and nucleate boiling in a 6-cylinder diesel engine by using the measurement of cylinder head temperature and finite element analysis. Another very effective research on the field of nucleate boiling for use in internal combustion engines is the work done in 2003 by Robinson et al. [2]. The results of Robinson et al. show that the effect of convective heat transfer is so high at high speeds that boiling does not actually occur. In the context of subcooled boiling modeling, Lee and O' Nill [3] calculated the temperature at the cylinder head of an internal combustion engine experimentally by implanting a thermocouple. Lee has done similar works in the same field as can be seen in references [4,5]. Furthermore, computation fluid dynamics (CFD) was investigated by You-chang and Xiao-hong [6] for studying the cooling system in a diesel engine in three dimensions for the water jacket and in one-dimensional simulation for the cooling system. Their results show that the cooling system's performance can be optimized by changing the speed, pressure and distribution of the heat transfer coefficient in the cooling jacket. Jafarabadi et al. [7] investigated the two-phase mixing of passing through the cooling water jacket of a 12-cylinder heavy duty diesel engine by simultaneous three-dimensional analysis of the structure and fluid. They showed that the boiling phenomenon occurs at hot points with a low flow of coolant, which results in an increase in the convective heat transfer coefficient. In the continuation of their studies in the field of cooling systems, Jafarabadi et al. [8] presented four designs for cooling passages and selected the proposed design based on the advantages and disadvantages of the presented designs. In addition, Punekar and Das [9] of ANSYS Company investigated the boiling phenomenon in the engine's cooling jacket by presenting a computational fluid dynamic (CFD) model. Their study was carried out in order to investigate the temperature distribution in two cases, taking into account the boiling phenomenon and without it. The results show a temperature decrease at critical points with consideration of the boiling phenomenon. Another very effective research on the diesel engine's cooling is the work by Romanov and Khozeniuk [10] in 2016. Their goal was to achieve a high quality engine cooling design and to obtain uniform cylinders at the same level of thermal stresses. The results of Romanov for the considered diesel engine showed that it is possible to control the cooling in a uniform path without changing the design of the chamber, crankshaft and cylinder head.

All these activities were performed with the aim to develop predictive tools to help design high-precision cooling systems. The aim of this paper is the numerical simulation of the subcooled boiling phenomenon in the cooling jacket of a 16-cylinder heavy duty diesel engine based on the two commonly used models of Chen and Boiling departure lift-off (BDL). In addition, the investigation of the indices of velocity, pressure and heat transfer coefficient (HTC) in the engine's cooling jacket has been done using a three-dimensional fluid simulation using Fluent software with proper mesh density, which guarantees the accuracy of the results and a logical time of the calculation. A noteworthy point in the present simulation is the simultaneous solution of the structure (cylinder head) - fluid.

2. Numerical simulation of boiling

For modeling the boiling phenomenon, all the mechanisms governing this phenomenon should be considered. However, finding the exact relations governing all these mechanisms, such as the volume of the formed bubble, the frequency of bubble separation, the diameter of the bubble separation and so on is very difficult. Therefore, a series of simpler assumptions is used to simulate the heat transfer of the boiling phenomenon. One of the patterns used to simulate boiling is the following pattern proposed on the basis of the assumptions of Rohsenow [11] on the sum of the values of forced convective heat transfer and nucleate boiling thermal flux.

$$q''_{total} = q''_{conv} + q''_{boil} = h_{conv}(T_w - T_f) + h_{boil}(T_w - T_s) \tag{1}$$

In the above equation, h_{conv} is the forced convective heat transfer coefficient, and h_{boil} is the nucleate boiling heat transfer coefficient. The most known relationships proposed for the forced convective heat transfer coefficient and the nucleate boiling heat transfer coefficient are the relationships proposed by Ditus Bulter and Foster-Zuber [12].

$$h_{conv} = k/D_h 0.023\,\mathrm{Re}^{0.8}\,.\,\mathrm{Pr}^{0.4} \tag{2}$$

$$h_{conv} = 0.00122\left[\frac{k_1^{0.79} C_{p,l}^{0.45} \rho_l^{0.49}}{\sigma^{0.5}\mu_1^{0.29}(h\rho_{l,g})^{0.24}}\right] \times (\Delta T_s)^{1.24}(\Delta p_s)^{0.75} \tag{3}$$

Several researchers used this model in a variety of ways, among which Chen's method and the BDL method were more accepted.