Experimental Investigation of Surface Roughness Effect on Flow Boiling in Internal Combustion Engine Water Jacket

A.Ghasemian[1],A.Kesharaz[2], H.Sotoudeh[3]

1. Ph. D. Student, 2.Associate Professor, 3.M. Sc. Student, K. N. Toosi University of Technology, Tehran, Iran

Keshavarz@kntu.ac.ir

Abstract

The subjects of heat transfer and cooling system are very important topics in the Internal Combustion Engines (ICE). In modern cooling systems, low weight, small size and high compactness are the critical designing criteria that requires heat transfer enhancement. Boiling phenomenon which is occurred in the water jacket of the ICE is one of the methods to increase heat transfer in the coolant system of an ICE. A research has been shown that parameters such as material, temperature, and roughness of the heated surface have direct effect on the rate of heat transfer in a boiling phenomenon. In this paper the potential of boiling phenomenon and the effect of the surface roughness on the amount of heat flux removed by the coolant flow in the engine water jacket is investigated experimentally. For this purpose the experiments was carried out in three different flow velocities and also three different surface roughnesses. Results show that the boiling and roughness of a hot surface will increase the heat removal significantly.

Keywords: *Internal Combustion Engine, heat transfer, flow boiling, roughness*

1. Introduction

The heat released in a combustion chamber of an engine is divided into three main parts. Only about one third of the input energy is converted into useful output power and the rest is wasted by means of the exhaust gases and cooling system. The main goal of the cooling system is to keep the engine components at proper temperature. Although the heat rejected to the coolant of an ICE varied with the type, load and speed of an engine, but in general, it is about 17 to 26 percent of the input energy in a SI engine and 16 to 35 percent in a CI engine [1].

Some regions of the cylinder head such as exhaust valve and valves bridges may experience heat fluxes as high as 10 MW/m2 during the combustion period [1]. This high heat flux in one hand and the industry demand for lower cooling system power and downsizing the engine in the other hand requires cooling system enhancement. Clough [2] proposed precision cooling system due to different heat fluxes regions along the water jacket. In the precision cooling the heat transfer coefficient is increased around high heat flux regions to achieve uniform temperature. In a water cooled engine, the heat is removed by forced convection through the water jacket. Therefore to increase the heat rejection from the chamber walls, the convection coefficient should be somehow enhanced. A method which causes a considerable increase in convection coefficient is boiling phenomenon [3, 4]. This motivates the researchers to study the nature of this phenomenon in ICE.

Boiling takes place at a solid-liquid interface in which the temperature of the solid surface is higher than saturation temperature of the liquid as shown in figure 1. When the fluid adjacent to the hot surface, has no motion, the process called pool boiling and otherwise it is called flow boiling. It also can be classified as saturated boiling and sub cooled boiling.

In the saturated boiling regime, the bulk temperature of the fluid is at the saturation temperature and in the sub cooled boiling regime the bulk temperature of the fluid is less than its saturation temperature. In the saturated boiling regime, the bulk temperature of the fluid is at the saturation temperature and in the sub cooled boiling regime the bulk temperature of the fluid is less than its saturation temperature. The boiling phenomenon which may

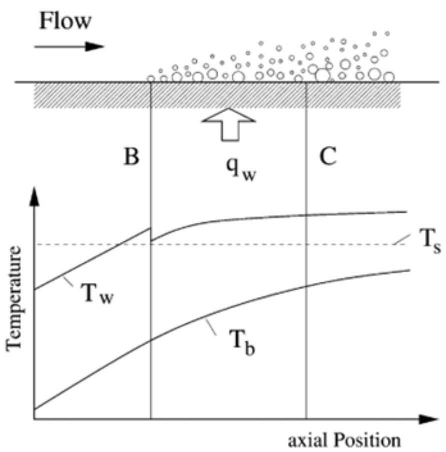

Fig1. Flow boiling on heated surface

occurs in some regions of an ICE water jacket is a sub cooled flow boiling regime. During the boiling phenomenon, due to the latent heat of the fluid, huge amount of energy is transferred from the hot solid surface. By increasing the wall temperature, the rate and number of bubbles creation increases and consequently the heat transfer coefficient increases.

At a specific point of the wall temperature- called CHF the velocity of bubble formation outpaces the velocity of the bubbles departure from the heated surface. In this stage, gradually a vapor layer covers the heated surface and the heat transfer coefficient decreases which must be avoided in the ICE cooling.

Finlay [5] is one of the pioneers in the modeling of sub cooled flow boiling in ICE. He simulated the boiling process in the water jacket experimentally and theoretically. Mixture of water and ethylene glycol is used as the coolant. Finlay showed that in high flow velocities, the forced convection is dominating mode of the heat transfer. Whereas, at low flow velocities strong nucleate boiling takes place.

The effect of surface roughness on boiling phenomenon was studied experimentally by Campbell et al [6]. A rectangular duct with changeable bottom surface that can simulate the engine coolant passage is used in their equipment. Using this equipment the effect of surface roughness, inlet pressure and temperature and also the fluid velocity on flow boiling is investigated. A 50-50 mixture of water and ethylene glycol as the coolant, 2.0 bar pressure and 90°C temperature for the coolant is used to provide the most resemblance to the engine water jacket condition. Robinson et al [7, 8] continued Campbell work theoretically. The sub cooled flow boiling was investigated experimentally and theoretically by Steiner [9] too. The surface roughness effect was neglected in this study. Another experimental and theoretical research was conducted on an ICE by Lee et al [10, 11] based on Chen’s model. Chen didn’t consider the effect of surface roughness on the flow boiling.

As it mentioned earlier, none of the above works considered completely the effect of surface roughness on the flow boiling heat flux. The purpose of this study is to show the effect of surface roughness on the amount of heat flux removal by flow boiling phenomenon.

2. Experimental Setup

To show the effect of flow boiling phenomenon on the heat removal from hot surfaces and also the surface roughness on the heat removal by this phenomenon an experimental study was conducted thoroughly. In this experiment the flow boiling behavior of both fluids of pure water and mixture of 50-50 water and ethylene glycol was investigated. The Schematic of experimental setup is shown in figure 2. The experimental apparatus is composed of a rectangular duct, test section, heater, variable pump, a reservoir, pre heater, condenser, flow meter, three pressure transducers, aluminum head, copper body, and few thermocouples. The test section (circular surface) with diameter of 15mm is located at the bottom of the rectangular channel and heated by heater as shown in figure 3. This test section is placed 90 cm from inlet of the duct to ensure the flow is hydrodynamically fully developed at this section. The test section surface temperature is evaluated by extrapolating of three measured temperatures along the tip of the test section as shown in figure 4. The aluminum head is mounted on the top of a copper body which is heated by a 1000W rod heater as shown in figure 4. To minimize the heat loss between the aluminum head and the copper body, special high conductivity oil is used at the conjunction. The copper body and aluminum head are well insulated during the test. K-type thermocouples with accuracy of 0.1 K are used in this experiment. To visualize the flow and the bubbles, the channel was fabricated with plexy glass panels. Part of the channel is shown in figure 5. Length of the channel is 140cm and its cross section area is 2×3cm2. To provide the most resemblance to the engine water jacket, the fluid pressure and temperature around the test section was set to be 1.4bar and 85°C respectively. A pre-heater with a controller was used to control the inlet temperature of the fluid in the reservoir. The pre heater was turned on and off by an on-off controller with respect to the temperature of the fluid in the reservoir. A condenser was used in the reservoir to provide constant pressure and also to condense any existing vapor. All Thermocouples and heater were connected to a data acquisition system (model: ADAM 5000/TCP) to record the experimental data. The flow rate was controlled by two bypass valves and measured by a Rota meter (model: GEC- Elliotte) with accuracy of 0.1 L/min. This Rota meter has been calibrated for pure water at a definite temperature and it must be recalibrated for different fluids with different densities. The following equation is used to calculate the flow rate for different fluids:

$$Q_2 = Q_1\sqrt{\frac{\rho_1(\rho_f-\rho_2)}{\rho_2(\rho_f-\rho_1)}} \tag{1}$$

Where Q2 and Q1 are the volumetric flow rate of new fluid and reference one.

All the setup was calibrated prior to the data collection. Each test was run few times to ensure its repeatability and their average values were used in each case.

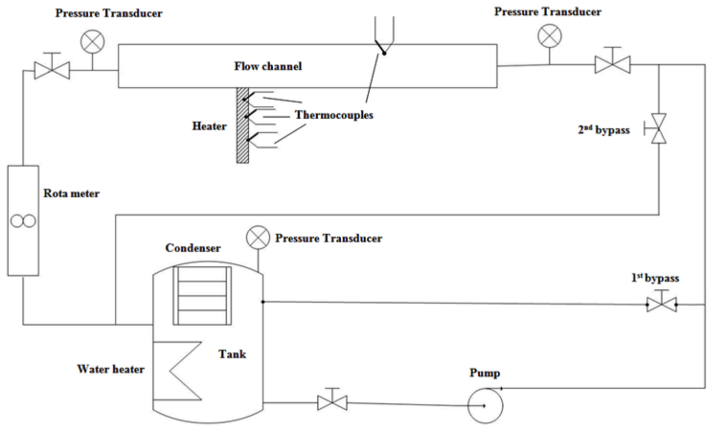

Fig2. Schematic of the experimental setup

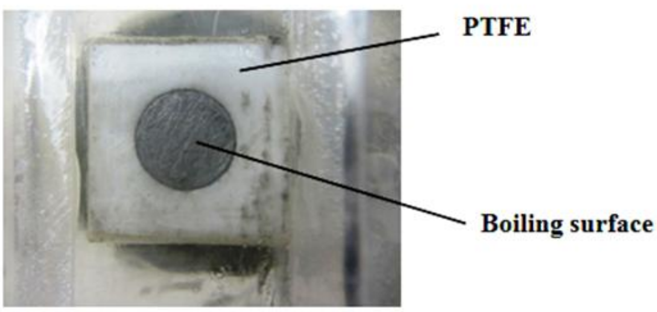

Fig3. Test section at the bottom of the channel

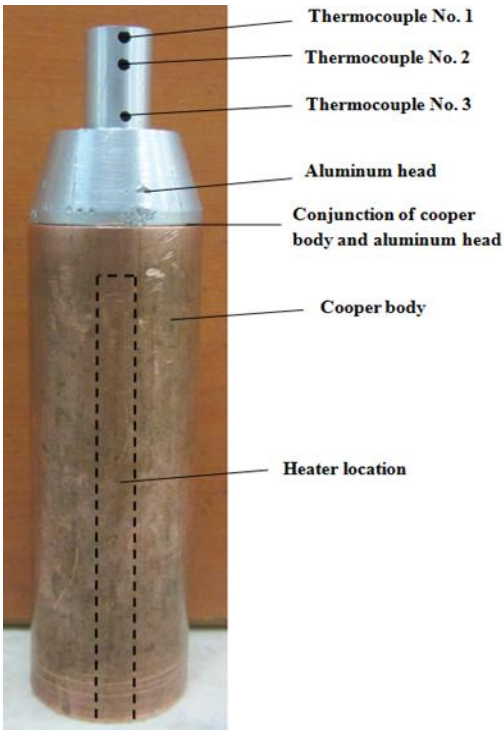

Fig4. Cooper body and aluminum head

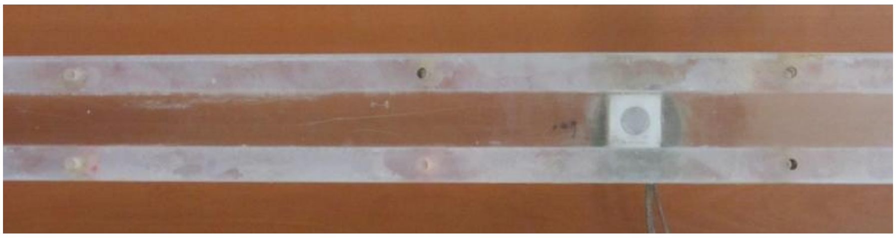

Fig5. Top view of a part of the channel

3. Error Analysis

In this section the uncertainty of the results is calculated. According to theory of error distribution by Taylor series, equation 2 is used to calculate the uncertainty of a multivariable parameter with 95% certainty [12].

$$U_{95} = \left[\sum_{i=1}^{J} \left(\frac{\partial r}{\partial X_i}\right)^2 U_i^2 \right]^{\frac{1}{2}} \quad (2)$$

In this equation U95 is the total error of a multivariable parameter r, Xi is an independent variable and Ui is its error and J is the number of variables. The accuracy of each measuring parameters are given in Table 1. These values are provided by the manufacturers.

In this study the error analysis are carried out for the two main measured parameters, namely the heat flux and temperature. The temperature is an independent variable that its error is given in table 1. But the heat flux is a dependent variable which depends to some variables as it shown in equation 3.

$$q'' = -k \frac{\Delta T}{\Delta x} \quad (3)$$

$$\frac{U_{q''}}{q''} = \sqrt{\left(\frac{U_{\Delta T}}{\Delta T}\right)^2 + \left(\frac{U_{\Delta x}}{\Delta x}\right)^2} \quad (4)$$

$$\frac{U_{q''}}{q''} = 2.7\% \qquad (5)$$

With respect to the table 1 and measured data for ΔT and Δx the calculation error of the measured heat flux is about 2.7%.

4. Results and Discussion

As mentioned earlier the effect of surface roughness on the heat flux removal by the flow boiling was investigated experimentally. For this purpose, two fluids of water and 50-50 water-Ethylene Glycol were used. Three different surface roughnesses of 0.65, 4.4 and 30 micrometer were considered on the test section s. The experiments were conducted at three different flow velocities of 0.5, 0.7 and 0.9 m/s. below pictures. Figures 6, 7 and 8 show the behavior of surface heat flux versus its temperature for water.

As mentioned earlier the effect of surface roughness on the heat flux removal by the flow boiling was investigated experimentally. For this purpose, two fluids of water and 50-50 water-Ethylene Glycol were used. Three different surface roughnesses of 0.65, 4.4 and 30 micrometer were considered on the test section s. The experiments were conducted at three different flow velocities of 0.5, 0.7 and 0.9 m/s. below pictures. Figures 6, 7 and 8 show the behavior of surface heat flux versus its temperature for water.

As shown in the figures the trend of the curves is similar to the water's except the boiling temperature point takes place around 120°C. Again this point is a little bit higher than the boiling point of 50-50 water-EG at 1.4 bar due to the flow velocity. In this case by increasing the surface roughness, the heat transfer coefficient enhances too. The potential of the boiling phenomenon for heat removal is also observed in these diagrams. This fact implies that the engine coolant heat removal increases when the boiling and roughness are considered.

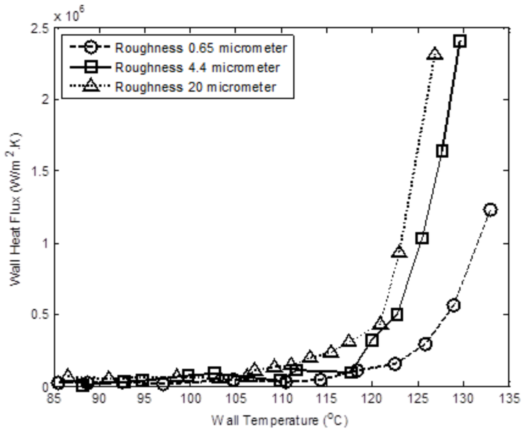

Fig6. Wall heat flux vs. temperature for water at the velocity of 0.5m/s

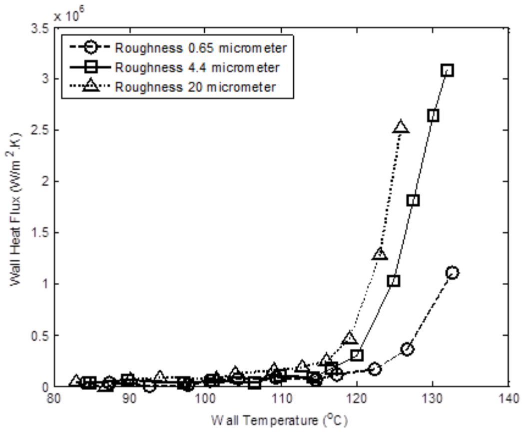

Fig7. Wall heat flux vs. temperature for water at the velocity of 0.7m/s

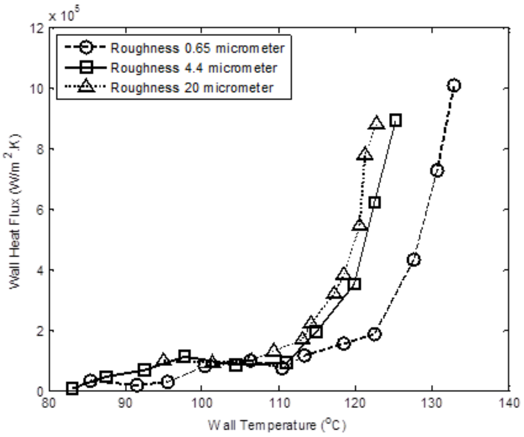

Fig8. Wall heat flux vs. temperature for water at the velocity of 0.9m/s

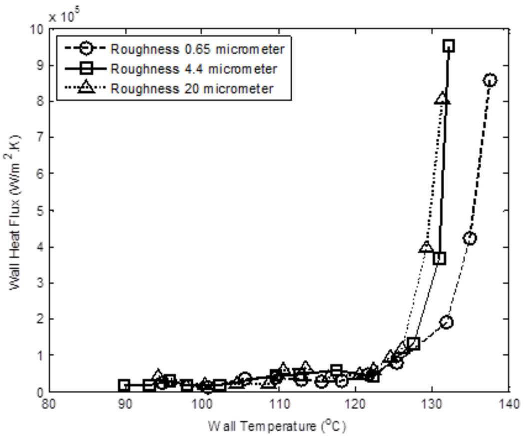

Fig9. Wall heat flux vs. temperature for Water-EG at the velocity of 0.5m/s

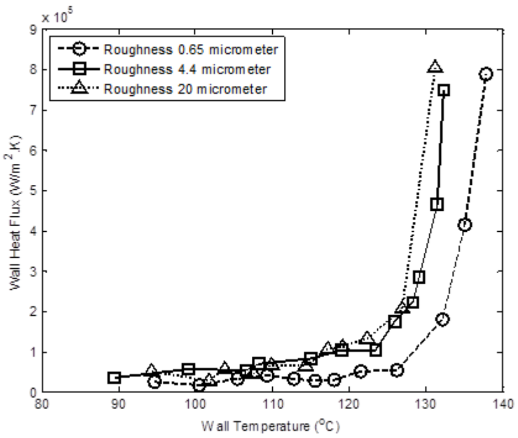

Fig10. Wall heat flux vs. temperature for 50-50 Water-EG at the velocity of 0.7m/s

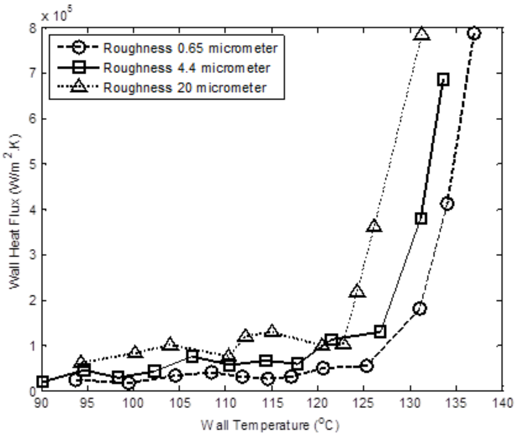

Fig11. Wall heat flux vs. temperature for 50-50 Water-EG at the velocity of 0.9m/s

5. Conclusion

In this study the impact of surface roughness and flow boiling phenomenon which occurs in the water jacket of an ICE was investigated experimentally. For this purpose an experimental test rig was set up. To provide more resemblance to the engine water jacket condition, a rectangular channel with a circular hot surface at the bottom was used. Three different surface roughness were considered for the hot surface along with three different flow velocities at pressure 1.4 bar. The obtained results showed that by increasing the surface roughness the heat transfer coefficient in both regions of pure convection and flow boiling are increased too. It can also be concluded that the occurrence of boiling in the flow causes significant increase in the heat transfer coefficient which leads to higher heat flux removal with a small surface temperature increment. Thus the engine water jacket hot surface areas such as regions around the spark plugs and exhaust ports can be cooled more efficiently when the potential of boiling phenomenon and surface roughness are considered.

Nomenclatures

T	Temperature
Q”	Heat flux
X	Distance
U	Error
K	Thermal conductivity
Q	Volume flow rate

References

[1]. Heywood, J.B., Internal Combustion Engine Fundamentals, Mc Graw-Hill, 1988.

[2]. Clough, M. J., (1993) Precision Cooling of a Four Valve per Cylinder Engine, SAE, 931123.

[3]. Bergman, T. L., Lavine, A. S Incropera, F. P., DeWitt, D. P., Fundamentals of Heat and Mass Transfer, 7th ed., John Wiley & Sons, 2011.

[4]. Ghiaasiaan, S. M., (2008) Two phase Flow, Boiling, and Condensation in Conventional and Miniature System, Cambridge University press.

[5]. Finlay C., and Gallacher G .R., (1988) The Application of Precision Cooling to the Cylinder Heated of a Small Automotive Petrol Engine, SAE, 880263.

[6]. Campbell, N. A. F, Hawley, J. G, Leathard, M. J, (1999) Nucleate Boiling Investigation and the Effect of Surface Roughness, SAE, 1999-01-0577.

[7]. Robinson, K., Hawly, J. G, Campbell, N. A. F, (2003) Experimental and Modeling Aspects of Flow Boiling Heat Transfer for Application to Internal Combustion Engines, Proceedings of the institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, , 217-877.

[8]. Robinson, K., (2001) IC Engine Coolant Heat Transfer Studies, Ph. D. thesis, University of Bath, UK.

[9]. Steiner, K. H., Kobor, A., Gebhard, L., (2005) A Wall Heat Transfer Model for Subcooled Boiling Flow, International Journal of Heat and Mass Transfer, vol. 48, pp. 4161-4173.

[10]. Lee, H. S, O'Neill, A. T., (2006) Comparison of Boiling Curves between a Standard S.I Engine and a Flow Loop for a Mixture of Ethylene Glycol and Water, SAE, 2006-01-1231.

[11]. Lee, H. S, (2009) Heat Transfer Predictions Using the Chen Correlation on Subcooled Flow Boiling in a Standard Engine, SAE 2009-01-1530.

[12]. Coleman, H. W., Steele, W.G., Experimentation, validation, and uncertainty analysis for engineers, Wiley, 2009.