

# Two Phase Flow Simulation for Subcooled Nucleate Boiling Heat Transfer Calculation in Water Jacket of Diesel Engine

**A. Mohammadi\***

Iran Khodro Powertrain Company  
Tehran, Tehran, Iran

**M. Yaghoubi**

Professor of Mechanical Engineering in Shiraz University  
Shiraz, Iran

---

Corresponding Author\*  
Received: Dec. 18, 2011  
Accepted in revised form: Feb. 14, 2011

## Abstract

Basic understanding of the process of coolant heat transfer inside an engine is an indispensable prerequisite to devise an infallible cooling strategy. Coolant flow and its heat transfer affect the cooling efficiency, thermal load of heated components, and thermal efficiency of a diesel engine. An efficient approach to studying cooling system for diesel engine is a 3D computational fluid dynamics (CFD) calculation for coolant jacket. Therefore, computer simulation can analyze and consequently optimize cooling system performance, including complex cooling jacket. In this paper a computational model for boiling heat transfer based on two-phase Mixture model flow is established. Furthermore, the phenomenon of nucleate boiling, its mathematical modeling, and its effect on heat transfer is discussed. Besides, the static, total pressure, velocity and stream lines of the flow field, heat flux, heat transfer coefficient and volume fraction of vapor distribution in the coolant jacket of a four-cylinder diesel engine is computed. Also, comparison between experimental equation (Pflaum/Mollenhauer) and two-phase Mixture model for boiling heat transfer coefficient is done and good agreement is seen. In conclusion, it is observed that at high operating temperatures, nucleate boiling occurs in regions around the exhaust ports. Numerical simulation of boiling heat transfer process of cooling water jacket and temperature field in the cylinder head of the diesel engine is compared with the data measured on the engine test bench. The calculated results indicate that this method can reflect the impact of boiling heat transfer on water jacket more accurately. Therefore, this method is beneficial to improve the computational precision in the temperature field computation of a cylinder head.

**Keywords:** Diesel Engine, Boiling, Water Jacket

## INTRODUCTION

With the development of the engine technology, engine power has been improved significantly and rising heat load is an inevitable corollary of increasing engine power. Consequently, there is more interest in the researches to optimize the design of cooling system. Modern cooling systems should be stable in the best heat state, even in different temperatures and operating conditions, which is not too cold or too hot and can integrate good drive and economy with good reliability. Under these circumstances, issues on control of cooling loss, prevention of knocking, thermal stress, etc. have come to be greatly addressed in the area of engine development to achieve low fuel consumption, high power, and light weight. In spite of that, it is difficult to investigate the complete behavior of cooling jackets experimentally due to complex geometries and flow paths. Hence, it is absolutely essential to employ numerical methods in designing the intricate cooling passages of an engine. One of the key phenomena of cooling system is phase change. As phase change is accompanied by high rate of heat transfer, high local heat transfer coefficient can be achieved in IC engine cooling passages through the onset of boiling when the coolant mass flow rate is relatively low. Effective use of boiling in engine coolant design can make the structure more compact. The boiling taking place in the engine cooling passage is considered as within the regime of subcooled nucleate boiling in global terms, since the bulk temperature of the fluid is normally controlled below the saturation point. However, locally saturated nucleate boiling, or even film boiling could occur, depending on the local wall heat flux and flow conditions. When boiling develops beyond the nucleate regime, overheating could occur which may result in remarkable damages to the engine. In order to control boiling over a wide range of operation conditions, detailed flow and heat transfer analysis are essential.

CFD has made rapid advances over the years and is now used as an effective tool in the analysis and visualization of fluid flows in complex systems including the engine cooling jackets. Along with visualizing the flow development in the jacket passages, CFD techniques are also applied to estimate temperature distribution over the entire engine block. It also helps to study and understand complex phenomena that commonly occur in cooling jackets, like cavitations and nucleate boiling. A large amount of time and effort is invested by engineers in trouble-shooting

and optimizing cooling jacket designs. CFD helps in the development of a cooling jacket by reducing the time and number of prototypes involved.

Gopinath, and Vijay [1] reviewed heat transfer and wall heat flux partitioning models and correlations that are applicable to subcooled forced flow nucleate boiling. Details of both empirical and mechanistic models have been proposed in their literature. A comparison of the experimental data with predictions from selected models is also included in it. Han, Wang, Filipi [2] presented a mathematical model to simulate boiling flows in industrial applications. Following the Eulerian multiphase flow, separate sets of mass, momentum, and energy conservation equations were solved for liquid and vapor phases, respectively. The interactions between the phases were computed by including relevant mass, momentum, heat exchanges and turbulent dispersion effects. Basu, Warriar and Vijay [3] worked on a mechanistic model for the wall heat flux during subcooled flow boiling. A fraction of this energy was utilized for vapor generation while the rest of the energy is utilized for sensible heating of the bulk liquid. The contribution of each of the mechanisms for heat transfer to the liquid with forced convection and transient conduction as well as, the energy transport associated with vapor generation has been quantified in terms of nucleation site densities, bubble departure and lift-off diameters, bubble release frequency and flow parameters like velocity, inlet subcooling, wall superheat, and fluid and surface properties including system pressure were discussed. Steiner, Kobar, Gebhard [4] proposed a model for computation of the specific wall heat transfer rate, a modified superposition model where the total heat flux is assumed to be additively composed of a forced convective and a nucleate boiling component and validated wall heat fluxes predicted by their model with experimental data. Watanabe, Kubo and Yomoda [5] used a combination of 1D and 3D fluid flow models. The internal flow in a component simulated in 3D and incorporated into a system, containing components represented in 1D and showed results for internal flow of water flow circuit of engine cooling system and discussed. Lee and O'Neill [6] found good agreement of their flow data with a standard IC engine data on subcooled flow boiling, leading to the assurance that the rectangular duct in the flow apparatus indeed well represents cooling passage in the standard IC engine. Urip and Yang [7] studied conjugate heat transfer in cylinders of an engine at full load with KI-

VA-3V code to simulate the transient heat transfer process between the gas and the solid phases and liquid coolant flow with FLUENT to obtain the temperature distribution on cylinder wall and cylinder head. Mulemane and Soman [8] simulated the cooling circuit of diesel engine based on 1D-3D coupling and predicted realistic flow rates through the cooling jacket and boundary condition for thermal analysis are obtained from in-cylinder simulations and the effect of nucleate boiling on heat transfer was discussed. You-chang, Xiao-hong and Dan [9] used CFD calculation to study cooling system of diesel engine, 3D for coolant jacket and 1D cooling system simulation. The results showed velocity, pressure and heat transfer coefficients distribution in the coolant jacket and could optimize cooling system performance. Lee [10] used experimental flow-loop data and suggested a correction factor on forced convection component of Chen correlation. Li, Liu, and Bai [11] used the concept of void fraction and homogeneous flow model for boiling heat transfer with hexahedral element of the flow field. The model based on the assumption that the boiling liquid field is a single phase flow, in which vapor and liquid are homogeneously mixed with void fraction being calculation based to reflect boiling heat transfer. Their result showed that heat transfer coefficient considering boiling factor is evidently different from that of pure heat convection, and the maximum deviation can be more than fifty percent. Ajotikar, Eggart, Miers [12] investigated nucleate boiling in the cooling passages of an IC engine cylinder head in a computational and experimental domain. Their experimental test rig consisted of a single combustion chamber section from a 5.4L V8 cylinder head. Water was used as the coolant. They demonstrated the phase change physics involved in the boiling in an engine cooling passage. Pressure variations in the coolant, heat flux data associated with the onset of nucleate boiling, and a comparison with existing boiling curves for water were shown.

The objective of the author's work is to develop a CFD solver for the study of engine coolant flow with boiling. The solver is required to be capable of predicting flow and heat transfer associated with nucleate boiling occurring in the engine cooling passages with acceptable accuracy. As the method is for engineering use, high computational efficiency is important. Considering the feature of subcooled nucleate boiling flow in the engine cooling passage, the approach based on the second category described above

has been adopted. The source of the void fraction equation has been constructed to model the thermodynamic mechanisms governing the vapor evaporation and condensation. The solver developed is incorporated into the commercial CFD code Fluent.

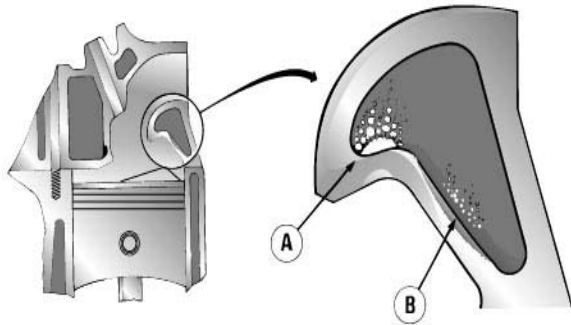
## Nomenclature

$c_{1\epsilon}, c_{2\epsilon}$	constants of the $\kappa$ - $\epsilon$ model
$\sigma_k, \sigma_\epsilon$	constants
$c_{\mu b}, c_\mu$	constants
$d_b$	bobble diameter
$\vec{F}$	body force
$\vec{g}$	specific gravity vector
$G$	production of turbulent kinetic energy of $\kappa$
$p$	coolant fluid pressure
$R$	surface roughness
$T$	fluid temperature
$T_w$	wall temperature
$T_b$	coolant Boiling temperature
$T_{coolant}$	coolant temperature
$\vec{U}_m$	mass-averaged velocity
$\vec{U}_{d,k}$	drift velocity of phase k
$a_k$	volume fraction of phase k
$\mu_m$	effective viscosity of mixture
$k_m$	effective conductivity of mixture
$k_t$	turbulent thermal conductivity
$\rho_k$	density of phase k
$\rho_m$	mixture density
$\kappa$	turbulence kinetic energy
$\epsilon$	dissipation rate of $\kappa$
$\mu_t$	turbulent viscosity

## Nucleate Boiling

As we discuss thermal issues in engine cooling jackets, an important aspect to be addressed is the occurrence of coolant vapors due to localized boiling of the coolant in very high temperature zones. This phenomenon is referred as Nucleate boiling. When heat stressed metal surfaces exceed the thermal capacity of the coolant material, the coolant begins to boil, forming a vapor layer at the surface. This vapor layer acts as insulation and prevents efficient heat transfer from the hot engine surfaces to the coolant

[8]. This causes localized over-heating and coolant vaporization. This is represented schematically in Fig. 1.



**Fig 1.** Schematic presentation of occurrence of nucleate boiling [9]

## CFD BOILING MODELS

The approaches used to model boiling-related two-phase flow with CFD are mainly the following:

1. Incompressible single phase flow simulation with a modified thermal boundary condition to account for the heat transfer enhancement as a result of boiling. The wall-temperature/heat-flux under the boiling condition is calculated using empirical correlations when the boiling criterion is satisfied. The major drawback of this single phase approach is that energy addition will translate directly into a rise in temperature rather than phase change. Thus, the calculated density, temperature and flow fields will be incorrect [13].

2. A homogenous flow model is used to represent both the liquid and vapor phases based on the assumption that vapor bubbles are small and are perfectly mixed with the liquid phase. The concentration of the vapor phase is described by an additional variable, the void fraction. Thus, the governing equations are still single phase but with an additional equation to describe the void fraction. The effect of density change due to the fluid phase change is fully considered, but the detailed interfacial dynamics between the two phases is not normally modeled. Until now the applications of this approach have been restricted to study cavitations. The source terms in the void fraction equation model the transfer between the two phases as driven by the pressure difference. Due to its simplicity and high computational efficiency, this method has potential to be extended for use in CFD engine coolant boiling analysis [13].

3. Liquid and vapor phases are solved separately us-

ing two sets of governing equations. Mass, momentum and energy transfer between the two phases are explicitly modeled. When the size of vapor bubbles is of the same order or smaller than the size of the computational mesh, the boundary between the two phases is not predicted, the interactions between phases are calculated based on the locally estimated bubble size and number. When the bubble size becomes larger than the size of the mesh, details of the phase boundary can be predicted with the help of an interface treatment. This method is so far the most promising in principle despite the fact that the accuracy of the prediction relies heavily on the quality of models while many of these models have not yet reached maturity. Compared with methods one and two, the increase of computational time for obtaining numerical solution in two phases with this method is significant [13].

4. Similar to method three in the respect that liquid and vapor phases are solved separately using two sets of governing equations. However, a very fine computational mesh is used which enables the detailed bubble dynamics to be simulated directly. Phase boundary and interactions between phases are part of the solution rather than modeled parameters. Currently the application of this method is still limited to the study of the behavior of a single bubble. Due to the high computational demands, this kind of approach offers no immediate potential for engineering applications [13].

## MATHEMATICAL DESCRIPTION

The mixture model is a simplified multiphase model that can be used to model multiphase flows where the phases move at different velocities. It can also be used to model homogeneous multiphase flows with very strong coupling and the phases moving at the same velocity or different velocity with the concept of slip velocity. The mixture model in boiling heat transfer can model two phases (liquid water and vapor) by solving the momentum, continuity, and energy equations for the mixture, the volume fraction equations for the secondary phases. The 3D fluid dynamics calculations were carried out using the Fluent software that coupled with UDF. The nucleate boiling model (method 3) was developed for the application in the Mixture multiphase model. Two phases are modeled; the primary phase is liquid and the secondary is vapor bubbles. The liquid is treated as the continuous phase and the vapor bubbles are treated as the dispersed phase. The flow of water is re-

garded as incompressible and turbulent. The same pressure is shared by the two phases. For the application described in this paper the  $\kappa$ - $\epsilon$  model has been used for its robustness and simplicity. The ensemble-averaged conservation equations for each phase are as follows:

**Continuity equation for mixture:**

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{U}_m) = 0 \tag{1}$$

$$\rho_m = \sum_{k=1}^2 \rho_k \alpha_k \tag{2}$$

$$\vec{U}_m = \frac{\sum_{k=1}^2 \rho_k \alpha_k \vec{U}_k}{\sum_{k=1}^2 \rho_k \alpha_k} \tag{3}$$

where that  $\rho_m$  is mixture density,  $\vec{U}_m$  is mass-averaged velocity and  $\alpha_k$  is volume fraction of phase k.

**Continuity equation for vapor (phase 2):**

$$\begin{aligned} \frac{\partial}{\partial t}(\rho_k \alpha_k) + \nabla \cdot (\alpha_k \rho_k \vec{U}_m) \\ = -\nabla \cdot (\alpha_k \rho_k \vec{U}_{d,k}) \\ + \sum_{i=1}^2 (\dot{m}_{ik} - \dot{m}_{ki}) \end{aligned} \tag{4}$$

**Momentum equation:**

$$\begin{aligned} \frac{\partial}{\partial t}(\rho_m \vec{U}_m) + \nabla \cdot (\rho_m \vec{U}_m \vec{U}_m - \mu_m (\nabla \vec{U}_m + (\nabla \vec{U}_m)^T)) \\ = -\nabla P + \rho_m \vec{g} + \vec{F} \\ + \nabla \cdot \left( \sum_{k=1}^2 \rho_k \alpha_k \vec{U}_{d,k} \vec{U}_{d,k} \right) \end{aligned} \tag{5}$$

$$\mu_m = \sum_{k=1}^2 \mu_k \alpha_k \tag{6}$$

$$\vec{U}_{d,k} = \vec{U}_k - \vec{U}_m \tag{7}$$

where  $\vec{F}$  is body force and  $\vec{U}_{d,k}$  is drift velocity of phase k.

**Energy equation:**

$$\frac{\partial}{\partial t} \left( \sum_{k=1}^2 \rho_k \alpha_k E_k \right) + \nabla \cdot \left( \sum_{k=1}^2 \alpha_k U_k (\rho_k E_k + P) \right) = \nabla \cdot (k_m \nabla T) + \dot{Q} \tag{8}$$

$$k_m = \sum_{k=1}^2 \alpha_k (k_k + k_t) \tag{9}$$

$$E_k = h_k - \frac{p}{\rho_k} + \frac{v_k^2}{2} \tag{10}$$

where, the subscripts k is phase denotations (k = 1, 2). Variables and  $\mu_m$  and  $k_m$  represent the effective viscosity and conductivity of mixture and  $k_t$  is turbulent thermal conductivity.  $\dot{Q}$  is volumetric heat source due change of phase form liquid to vapor.  $h_k$  is enthalpy of phase k. Since the density of the vapor phase is much lower than that of the liquid phase, it is assumed that the motion of the dispersed vapor bubbles follows the fluctuations in the continuous liquid phase. The standard  $\kappa$ - $\epsilon$  turbulence model could be used to calculate the influence to the whole flow field from the turbulence kinetic energy, which is given as:

$$\begin{aligned} \frac{\partial}{\partial t}(\rho_m \kappa) + \nabla \cdot (\rho_m \vec{U}_m \kappa) \\ = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_\kappa} \nabla \kappa \right) + G_{m,k} - \rho_m \epsilon \end{aligned} \tag{11}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho_m \epsilon) + \nabla \cdot (\rho_m \vec{U}_m \epsilon) \\ = \nabla \cdot \left( \frac{\mu_{t,m}}{\sigma_\epsilon} \nabla \epsilon \right) \\ + \frac{\epsilon}{\kappa} (c_{1\epsilon} G_{m,k} - c_{2\epsilon} \rho_m \epsilon) \end{aligned} \tag{12}$$

$$G_{m,k} = \mu_{t,m} (\nabla \vec{U}_m + (\nabla \vec{U}_m)^T) : \nabla \vec{U}_m \tag{13}$$

where  $\rho$  is the density of liquid,  $\kappa$  is the turbulence kinetic energy,  $\epsilon$  is the dissipation rate of  $\kappa$ ,  $\mu_{t,m}$  is the turbulent viscosity, p is pressure, G is the production of turbulent kinetic energy of  $\kappa$ ,  $c_{1\epsilon}$ ,  $c_{2\epsilon}$ ,  $\sigma_\kappa$  and  $\sigma_\epsilon$  are the constants of the  $\kappa$ - $\epsilon$  model. Due to existence of the vapor bubbles, the extra turbulence induced by the bubbles in the liquid phase is modeled according to the assumption of Sato [15] where the effective viscosity of the liquid phase is defined by:

$$\mu_{t,m} = \mu_1 + C_\mu \rho_m \frac{\kappa^2}{\epsilon} + C_{\mu b} d_b \alpha_v |\vec{U}_k - \vec{U}_l| \tag{14}$$

Where, the first term in the right-hand-side of Eq. 14 represents the molecular viscosity of the liquid phase, the second and the third terms are the shear-induced and bubble-induced turbulent viscosity, respectively. The coefficients are  $C_\mu = 0.09$  and  $C_{\mu b} = 0.6$ .

**CFD ANALYSIS FOR COOLANT JACKET**

The target engine here is a 4-cylinder turbocharged and intercooled diesel engine, its Bore/Stroke is 76mm/82.5mm, and Rated power /Rated speed is 90kW/4000rpm. The coolant jacket geometric model is made up of cylinder block and cylinder head water jacket. 3D model for coolant jacket can be seen in the Fig. 2. Cylinder head encounters very high temperatures due to direct exposure to hot

combustion gases. Efficient cooling of the exhaust ports and the exhaust valve seat regions is required. Efficient cooling reduces wear and improves parts durability.

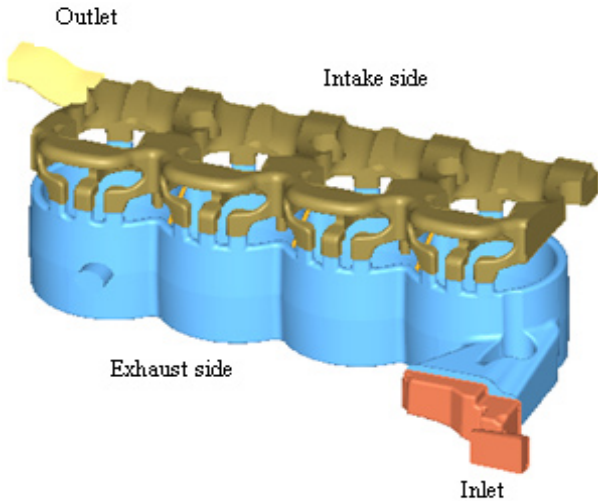


Fig 2. 3D model of coolant jacket

## GRID GENERATION

A water jacket of four-cylinder diesel engine is modeled three-dimensional. Prior to CFD simulation, computational mesh was generated for the engine using Hypermesh scheme. The geometry is completely modeled from outlet of water pump to outlets connected to water circuit line. In the analysis of cooling water, it is necessary to secure the accuracy for heat transfer coefficient which is provided as a boundary condition to structure analysis software. For that purpose, the computation accuracy for heat transfer coefficient is secured by refining boundary layer, so the total number of mesh point is approximately 826269. Fig. 3 shows the final computational tetrahedron mesh.

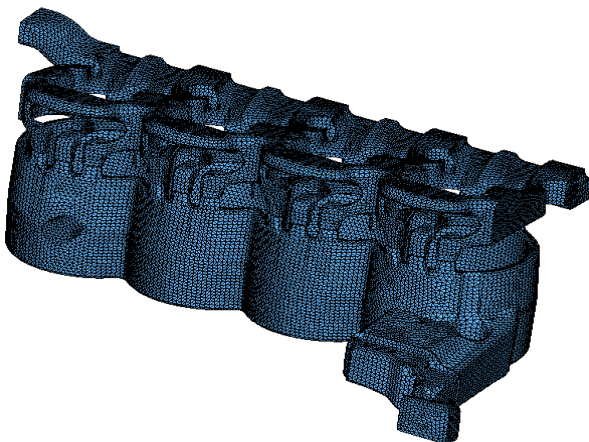


Fig 3. computational tetrahedron mesh

## BOUNDARY AND OPERATING CONDITION

The inlet boundary condition prescribes a constant velocity 4.588 m/s, constant flow rate with 130.0 liter/min at temperature of 378 K, that corresponds to the maximum revolution speed of 6000 rpm of water pump. The outlet of the coolant is zero gradient. The coolant is a mixture of water/glycol (50/50) with density 1030 kg/m<sup>3</sup>, thermal conductivity 0.44 W/m.K, specific heat 3470 J/kg.K and dynamic viscosity 0.0007 kg/m.s. The first guess of wall surface temperature at the coolant jacket surface is obtained from modified heat transfer coefficient defined by a non-linear formulation, considering nucleate boiling effects according to a law of Pflaum/Mollenhauer, which describes the increase of the HTC's in dependence on the wall temperature [15]:

$$HTC_{nucleation} = 10.6 (T_w - T_b)^{3.33} p^{0.7} R^{0.44} (T_w - T_{coolant})^{-1} \quad (7)$$

In which,  $T_w$  is Wall temperature,  $T_b$  is Coolant Boiling temperature that is approximately 130 °C for an ethylene glycol-water mixture at slight overpressure,  $T_{coolant}$  is coolant temperature and is assumed 105 °C,  $p$  is coolant system pressure in bar that is assumed 2.5 bar absolute and  $R$  is Surface roughness in  $\mu\text{m}$  that is assumed 100  $\mu\text{m}$ .

## NUMERICAL METHOD

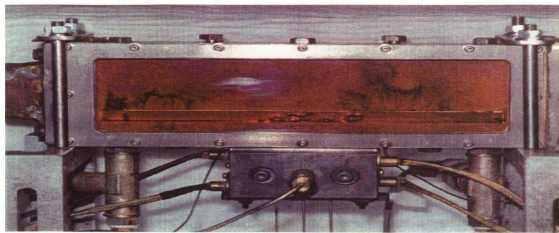
For the CFD calculations, Fluent was used with UDF for mass transfer between phases and source of energy for change of phase. Momentum, energy equation and turbulent flow model, standard  $k-\epsilon$  turbulent model were solved under the conditions of steady, incompressible, viscous fluid with the SIMPLE method. The accuracy of computation is second-order upwind for momentum and energy equations and first-order upwind for  $k-\epsilon$  turbulent model equations. All wall conditions are non-slip conditions under the logarithmic law-of-the-wall.

## VALIDATION

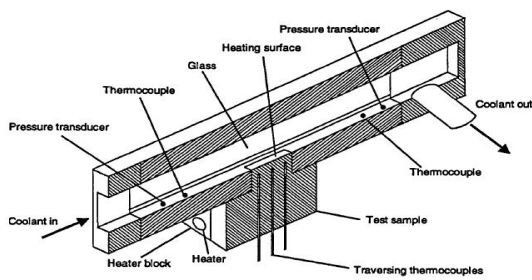
The performance of the model developed in this work has been assessed by validations against experimental data of two cases, first experimental data from literature in channel typed flows and second experimental data of real engine under boiling condition.

### VALIDATION CASE I

The experimental data from Robinson’s work [16] were used for validation of this approach. The variation of heat flux versus wall temperature under different operating pressures were simulated. The measured data were obtained from a cooling gallery simulator rig designed to simulate the condition of engine coolant flow with nucleate boiling. As shown in Fig. 4, the main flow channel has 0.241m long and a rectangular cross section of 0.016×0.01 m<sup>2</sup>. The heating surface is at the bottom of the flow channel, 0.01×0.05 m<sup>2</sup> located at 0.076 m downstream of the channel. The test sample was machined from billet which was cast from an aluminium alloy (AS10G) used for cylinder head castings. The coolant fluid used in the experiment was a mixture of (50-50) % ethylene glycol and water. Pressure is varied from 1 to 3 bar at a fixed bulk flow velocity 0.25 m/s and inlet temperature of 90 °C for liquid mixture.



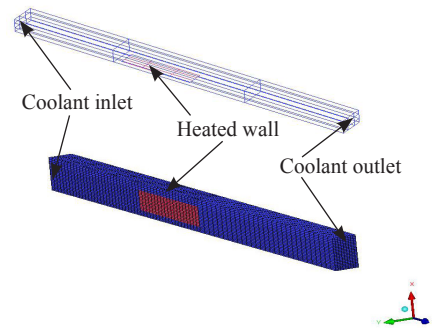
**Fig 4.** Close-Up View of the Test Duct of the Engine Cooling Gallery Simulator [16]



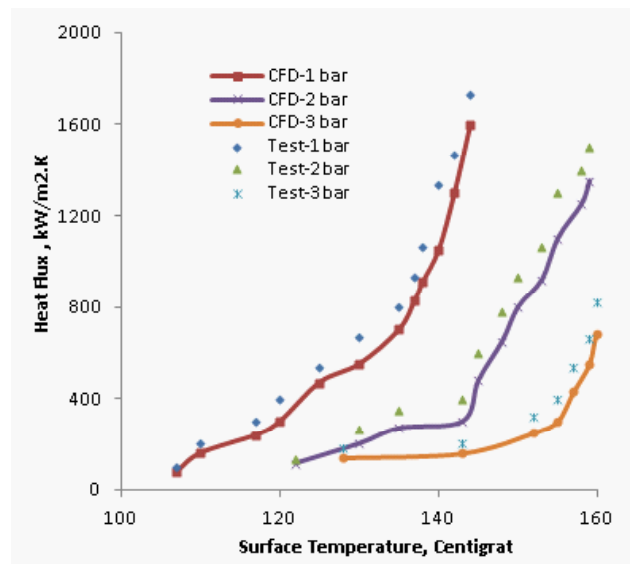
**Fig 5.** Cutaway Diagram of the Test Duct of the Engine Cooling Gallery Simulator

On the heated wall, a constant temperature condition was applied. The rest of the walls were assumed to be adiabatic. Photograph of test rig is seen in Fig. 4. A cutaway diagram of the test duct is shown in Figs. 5, 6. When the inlet and wall boundary conditions are given, the saturation temperature determines subcooling, and thereby onset

of the boiling. The saturated temperatures associated with the pressures of 1 bar, 2 bar and 3 bar are respectively, 108°C, 128°C and 142°C [16]. It is expected that before the wall temperature reaches to the saturation, the wall heat flux increases linearly with the wall temperature. After the wall temperature exceeds above the saturation, the heat flux increases rapidly as a result of nucleate boiling. This trend has been correctly reproduced by the CFD simulations. The linear relation between the wall heat flux and the wall temperature before the saturation point is seen more clearly in Fig. 7. A very good trend between CFD and experimental data is seen.



**Fig 6.** Computational mesh for engine cooling gallery

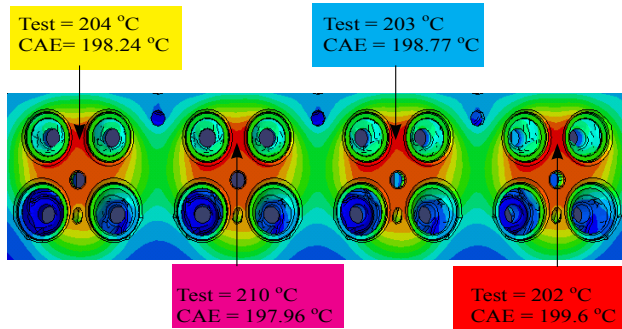


**Fig 7.** Comparisons numerical and measured heat flux versus wall temperature under different pressure and inlet flow velocity

**VALIDATION CASE II**

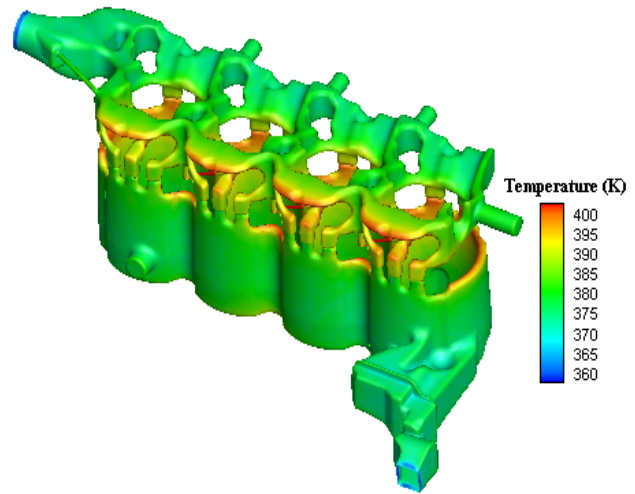
Fig. 8 shows the comparison between simulation and measurement temperature of test points on cylinder head. Four thermocouples were mounted in the cylinder head. The temperature of them was compared with numerical results. The calculated values are very close to the measurement data, the error of the simulated results is 7%. It is proved that the numerical simulation has sufficient precision to reflect the actual heat transfer of the flow in the cooling water jacket of the cylinder head accurately. The performance of the model developed in this work has been assessed by validations against experimental data of a real engine under boiling condition. Fig. 8 illustrates the distribution of temperature on the section where thermocouples were mounted, about 2 cm below the surface of cylinder head. Thermo-fluid analysis was compared with test results. The maximum temperature in the cylinder head is about 483 K, the maximum temperature on the water-jacket surface is 392 K. Near the side of the exhaust valve on the water-jacket surface, the temperature is higher because of the concentration of thermal load, the temperature of intake valve side is lower than the exhaust valve side. Maximum error between CAE and test is about 13°C in the second cylinder.

**RESULTS AND DISCUSSION**



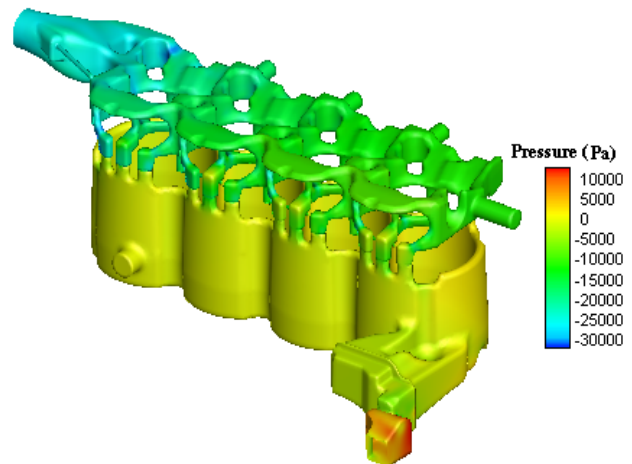
**Fig 8.** Comparison between simulation and measured temperature in the cylinder head

Fig. 9 shows temperature distribution on the wall of water jacket. The results have achieved from thermo fluid analysis. Maximum temperature is about 405 K in channels of water jacket, in the cylinder head, near the side of exhaust valves. Average surface temperature is about 380 K.



**Fig 9.** Temperature distribution of cylinder block coolant jacket

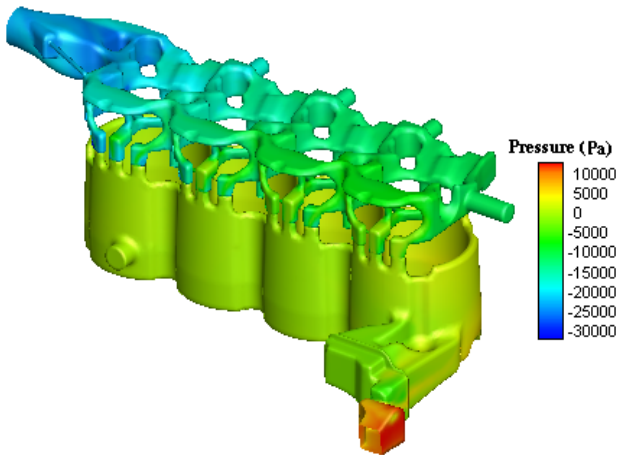
Static pressure is an indicator of the flow distribution in the cooling jacket. Pressure contours indicate the regions that contribute to increase in pressure drop. Also, lower pressure indicates regions which have the tendency to cavitation. Fig .10 shows static pressure plots. A lower pressure drop across the cooling jacket along with proper flow distribution is desirable for better performance. A lower pressure drop across the cooling jacket along with proper flow distribution is desirable for better performance.



**Fig 10.** Static pressure distribution in cooling jacket

Fig. 11 shows total pressure in water jacket. Maximum total pressure is in outlet of water pump or inlet of water jacket, losses cause decrease in total pressure in channels of water jacket. Velocity plots are good indicators of flow development and distribution. Velocity distribution vector of in coolant jacket in three section (y=0.12, y= 0.17, y=

0.23) are shown in Figs.12a,b,c.



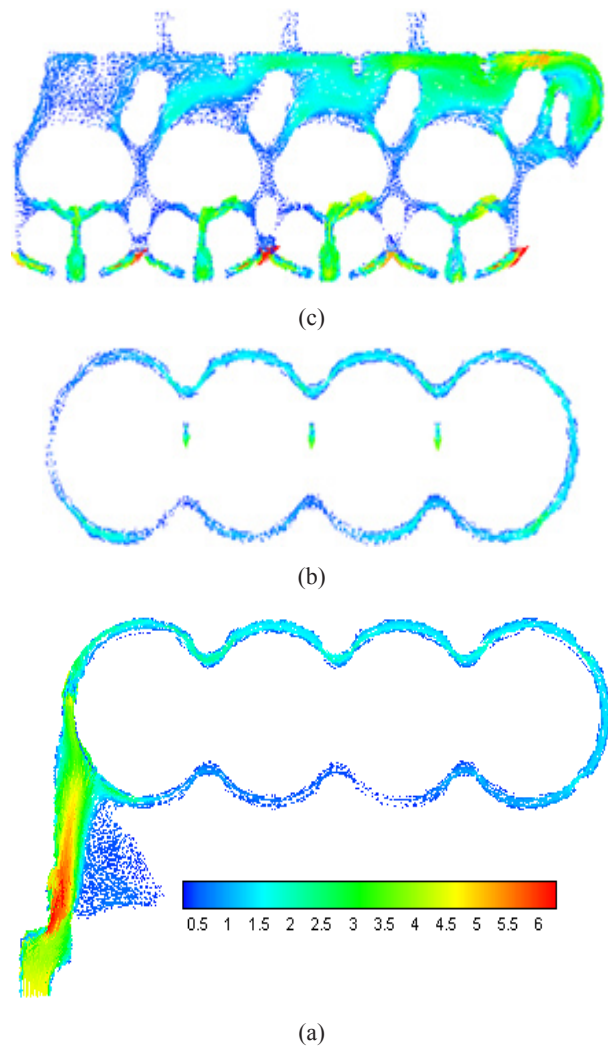
**Fig 11.** Total pressure distribution in coolant jacket

In Fig. 12a, velocity vector near the bottom of water jacket is seen. Maximum velocity about 6.55 m/s exist around the inlet of coolant jacket. Fig. 12b shows velocity vector near the middle of height of water jacket. Green channels in this figure are connections between coolant that passes around intake side valves to coolant flow near the exhaust valves for better cooling of exhaust system. Fig. 12c shows velocity in channels in cylinder head, maximum velocity is about 7 m/s. Mean velocity of the cylinder head coolant jacket is 2.33 m/s. Because of the high exhaust gas temperatures and the narrow bridge between the two exhaust valve seats, especially the area around the exhaust is one of the most endangered zones. Flow velocity around the exhaust channel is about 6.2 m/s and the mean velocity of the coolant jacket in the cylinder block is 1.47 m/s.

As it can be seen from the Fig. 12, the coolant of cylinder block coolant jacket is injected from the bottom of the fourth cylinder, and thus causes the water flow velocity in the first cylinder to be slower obviously and the velocity of water flow is getting slightly slower from the bottom of the fourth cylinder to the first cylinder, so does the velocity of water flow from the upper of the fourth cylinder to the first which makes the gas cooling equilibrium in each cylinder. As desired the flow is directed using the gasket holes so that the majority of it reaches the exhaust side which requires more cooling. The spots showing up in dark correspond to high coolant flow velocities as the coolant flows through the gasket holes. It is apparent from this figure that the flow velocity at the water inlet side is faster than that at

the outlet side and, therefore, cooling effect is higher at the inlet. It was also made apparent that comparing the upper and lower portions of head, the flow velocity at lower portion is generally faster.

Another effective way of visualizing the flow development is using flow path-lines. Flow path-lines help in visualizing flow development and verifying if all regions are supplied with coolant. These also help visualize occurrence of re circulation zones. Fig. 13 shows the flow path-lines through the cooling jacket.

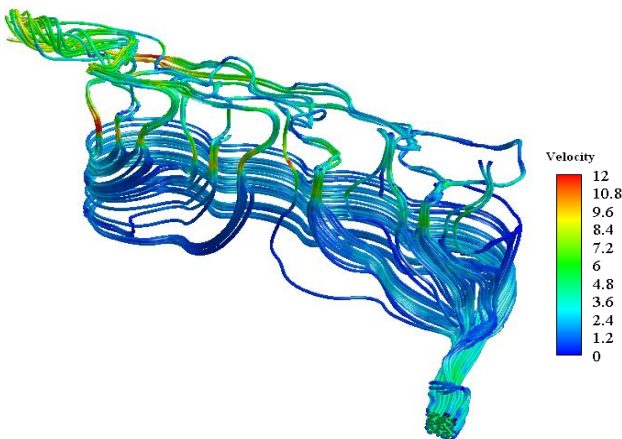


**Fig 12.** Velocity distribution in three sections of cylinder block coolant jacket

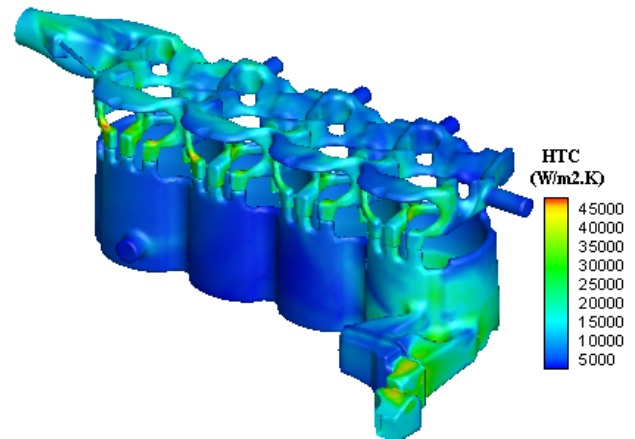
Fig. 14 shows the distribution of heat flux on the wall surface of water jacket. On the other hand, calculation result shows that heat transfer coefficient distribution trend of water flow for cylinder block and cylinder head accords with velocity field distribution, heat transfer coefficient

distribution of water flow for cylinder block and velocity distribution of coolant jacket is described in Fig. 15, from which we can see that the velocity of water flow in the side near the cylinder body is obviously faster. Heat flux is approximately constant, about  $250 \text{ kW/m}^2 \cdot \text{K}$  around the engine-cylinders. Maximum of heat flux in the First cylinder head of engine is seen about  $700 \text{ kW/m}^2 \cdot \text{K}$ , due to maximum difference between fluid temperature and cylinder-head temperature. Maximum heat flux occurs in the same location for four cylinders but, the value of heat flux decreases as coolant flow absorbs heat of engine. Minimum of heat flux occurs near the inlet and exhaust of water jacket as can be seen from Fig. 14. that illustrates the distribution of the coefficient of heat transfer on water-jacket surface with two phase boiling. When boiling is considered, the heat transfer coefficient between cooling water and water-jacket surface is decided by the velocity of the cooling water and the wall temperature.

The effect of heat transfer is greatly increased where the temperature reaches boiling on the surface of the water jacket, and the heat transfer coefficient of boiling heat transfer is in accord with the actual situation. As intended in its design, the heat transfer coefficient at the lower portion of exhaust port indicates a high value of  $47 \text{ kW/m}^2 \cdot \text{K}$ . The points of maximum HTC are shown in Fig. 15 with arrows that are in the third and fourth cylinder. In Fig. 16, HTC on the wall of coolant jacket according to a law of Pflaum/Mollenhauer [15] is seen. Two figures (15 and 16) show the same HTC distribution on the wall, but Pflaum/Mollenhauer correlation shows wrong HTC distribution in channels that connect coolant flow that is passed through cylinder around the intake valves side and exhaust valves. But in other locations both figures predict approximately the same value for HTC. Difference between two figures due to high value of vapor in these canals that affects HTC value and Pflaum/Mollenhauer cannot predict this phenomenon.



**Fig 13.** Path-lines showing flow development through the jacket and recirculation zones



**Fig 15.** Heat transfer coefficient distribution of coolant jacket from two phase flow calculation



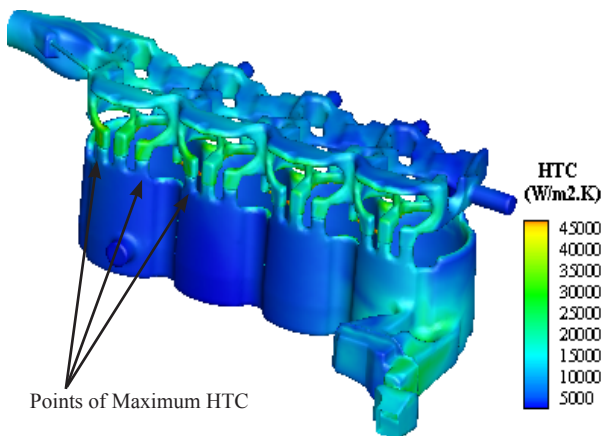
**Fig 14.** Heat flux distribution on cylinder block coolant jacket

Fig. 17 illustrates the predicted void fraction on a horizontal plane passing through the hot points on the cylinder heads. Up to 40% of vapor volume fraction is predicted at some locations. In cylinder inside of intake valves no appreciable vapor is seen due to low temperature of this side that is lower than saturation temperature. In exhaust side in the first cylinder vapor is formed and in the third and fourth cylinder about 25% vapor is seen. Near the outlet of water jacket maximum value of vapor volume fraction about 40% is formed. It shows that cooling jacket works well for engine cooling.

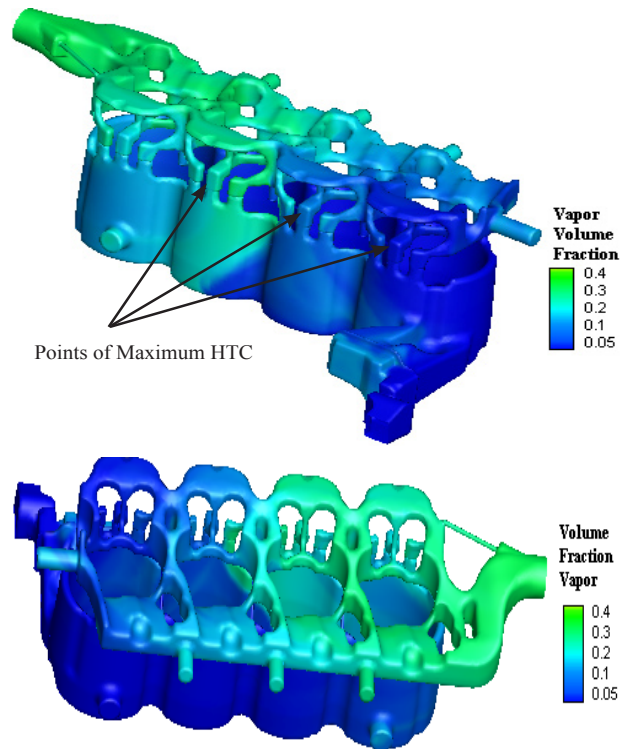
## CONCLUSIONS

In this paper CFD simulation of water jacket of diesel engine was done. The simulation predictions correlate well with the test results validating the developed procedure. The paper highlights the phenomenon of nucleate boiling and its prediction using CFD codes. Poor flow in regions corresponding to stagnation zones and adjacent to hot surfaces act as initiation sites for nucleate boiling. The following results are achieved:

- 1) Numerical simulation on temperature field of the cylinder head in this diesel engine was carried out with the Pflaum/Mollenhauer correlation that corrects the force-convection heat transfer coefficient for boiling effect. The calculated results are in good agreement with measured results. The engineering application of the approach has been demonstrated by the CFD analysis of an engine cooling passage flow involving local boiling.
- (2) With the study of static pressure, possibility of cavitation phenomena in water jacket was studied.
- (3) Velocity vectors in water jacket showed near the exhaust port, velocity magnitude is high enough to avoid pool boiling for mation.
- (4) Stream lines around cooling jacket showed good dispersion of water vapor in it.
- (5) The value range of the void fraction for the design of the engine water jacket is proposed and recommended, and the average void fraction values are between 0.05 and 0.4.



**Fig 16.** Heat transfer coefficient distribution of coolant jacket from PFLAUM/MOLLENHAUER equation [16]



**Fig 17.** Void fraction distribution in coolant jacket

It is shown that when boiling occurs, the heat transfer rate on the heated surface, the flow field and the temperature distribution are significantly modified. CFD and thermal analyses without modeling the boiling is insufficient.

## Acknowledgments

The authors would like to thank the directors of Iran Khodro Powertrain Company for permission to use experimental data in several sections and publish this paper.

## References

- [1] Gopinath, R., and Vijay, K., 2006, "Heat Transfer and Wall Heat Flux Partitioning During Subcooled Flow Nucleate Boiling-A Review" , J. Heat Transfer, 128 (12) , pp. 1243.
- [2] Han, J., Wang, D. M. , Filipi, D., 2005, "Numerical Simulation of Boiling Flows Using an Eulerian Multi-Fluid Model", Fluids Engineering Division Summer Meeting, June 19–23, Houston, Texas, USA, pp. 863-869.
- [3] Basu, N., Warriar, G. R., Vijay, K., 2005, "Wall Heat Flux Partitioning During Subcooled Flow Boiling: Part 1—Model Development", J. Heat Transfer, 127 ( 2) , pp. 131.
- [4] Steiner, H., Kobor, A., Gebhard, L., 2005, "A Wall Heat Transfer Model for Subcooled Boiling Flow", Int. J. of Heat and Mass Transfer, 48 (19), pp. 4161-4173.
- [5] Watanabe, N., Kubo, M., Yomoda, N., 2006, "A 1D-3D Integrating Numerical Simulation for Engine Cooling Problem", SAE 2006-01-1603.
- [6] Lee, H. S., O'Neill, A. T., 2006, "Comparison of Boiling Curves Between a Standard SI Engine and a Flow Loop for a Mixture of Ethylene Glycol and Water", SAE 2006-01-1231.
- [7] Urip, F., Yang, S. L., 2007, "An Efficient IC Engine Conjugate Heat Transfer Calculation for Cooling System Design", 2007-01-0147.
- [8] Mulemane, A., Soman, R., 2007, "CFD Based Complete Engine Cooling Jacket Development and Analysis", SAE 2007-01-4129.
- [9] You-chang, L., Xiao-hong, G., Dan, C., 2007, "Research on cooling system for 4-cylinder diesel engine", SAE 2007-01-2064.
- [10] Lee, H. S., 2009, "Heat Transfer Predictions using the Chen Correlation on Sub cooled Flow Boiling in a Standard IC Engine", SAE 2009-01-1530.
- [11] Li, G. X., Liu, Y., Bai, S., 2009, "A homogeneous flow model for boiling heat transfer calculation based on single phase flow" Energy Conversion and Management, Vol. 50, Issue. 7, pp. 1862-1868.
- [12] A jotikar, N., Eggart, B. J., Miers, S. A, 2010, "Nucleate Boiling Identification and Utilization for Improved Internal Combustion Engine Efficiency", ASME 2010 Internal Combustion Engine Division Fall Technical Conference (ICEF2010), pp. 949-958., September 12–15, San Antonio, Texas, USA
- [13] Bo, T., 2004, "CFD Homogeneous mixing flow modeling to simulate sub cooled nucleate boiling flow", SAE 2004-01-1512
- [14] Sato, Y., Sekoguchi, K., 1975, "Liquid velocity distribution in two-phase bubble flow", Int. J. Multiphase Flow 2 (1), pp. 79–95.
- [15] Pflaum, W., and Mollenhauer, K., 1977, "Wärmeübergang in der Verbrennungskraftmaschine" , Springer Verlag, .
- [16] Robinson, K , 2001, "IC Engine Coolant Heat Transfer Studies", PhD thesis, University of Bath, UK, .