International Journal of Mechanical and Materials Engineering (IJMME), Vol. 7 (2012), No. 3, 243-250.

STEADY STATE ANALYSIS OF COOLANT TEMPERATURE DISTRIBUTION IN A SPARK IGNITION ENGINE COOLING JACKET

M.A. Hazrat, H.H. Masjuki, M.A. Kalam, I.A. Badruddin, R. Ramli and S.C. Pang*
Centre for Energy Sciences, Department of Mechanical Engineering, Faculty of Engineering, University of Malaya, 50603 Kuala Lumpur, Malaysia
*Corresponding author's E-mail: alihazrat20@yahoo.com

Received 09 January 2013, Accepted 12 January 2013

ABSTRACT
A full scale SI engine has been imported in to the CFD tool to analyse the temperature distribution of coolant throughout the cooling channels. The segregated approach solver has been adopted to solve the energy equations along with the RANS two layer turbulence model to find out accordance with the available theoretical and published results. The input values are collected from complete vehicle test and from available documents, too. The main objective of the analysis was to observe the coolant temperature distribution inside the cooling jacket when the engine is turned off. The steady state simulation shows that though the average coolant outlet temperature is found within the acceptable limit of cooling system operation principle, there is a large temperature gradient in fluid thermal boundary layers within cross section and overall jacket path. The analysis demands that there should be some special arrangement of maintaining the fluid flow inside the cooling jacket even after the engine is turned off to avoid further loss of the engine body due to high temperature accumulation inside the cooling jacket and fluid in it. The lump capacity conduction equation for first sec shows that the wall temperature obtained through the energy equation is in accordance with it, too.

Keywords: SI engine heat transfer, Coolant temperature, Steady state heat transfer, CFD, Segregated approach.

NOMENCLATURE

Symbols

k	Thermal conductivity (W/m.K)
C	Specific heat capacity (J/kg.K)
Pr	Prandtl number
q	Heat flux (W/m^2)
u	Wall-cell velocity component parallel to the wall (m/s)
T	Temperature (K)
y	Normal distance from the wall (m)
x	Distance (m)
V	Volume (m^3)
S	Source energy
l_c	Characteristic length
v	Velocity vector
T	Viscous stress tensor
H	Total enthalpy (J/kg)
q	Heat flux vector
E	Total energy (W)
u^+	Dimensionless velocity
t^+	Dimensionless temperature

Greek symbols

μ	Viscosity
ρ	Density (kg/m^3)
τ	Shear stress (Pa)

Subscripts

c	Coolant
e	Effective
g	Gravity (m/s^2)
p	Constant pressure
s	Solid
t	Turbulent
w	Wall
ref	Reference level

1. INTRODUCTION
Internal combustion engine heat transfer analysis thus improving the removal of excess heat from the engine thermal system and leading to the improvement of engine thermal efficiency has been considered as one of the scientific challenges for decades. Various types of engine geometry and fuels, operation principle, high temperature and high speed of both the crank shaft as well as the pistons inside the cylinder, etc. are the challenging factors (Demuynck et al., 2009) in developing globally acceptable model for engine heat transfer, i.e. effective thermal management system. Characterizing the engine as well as the vehicle thermal system management, reduction of emissions and fuel demand per crank shaft rotation and load governance, etc. are analysed both in experimentally and theoretically (Torregrosa et al., 2008). Theoretical analysis of heat transfer from combustion chamber to the surrounding areas i.e. water jackets in both the cylinder blocks and cylinder heads require appropriate solution of continuity, momentum and energy equations (Borman and Nishiwaki, 1987). Solution of these equations is not easy to perform and therefore, various computational codes have been developing to perform such calculations. When the analytical results are found in accordance with the available experimental data the conceived numerical model is considered as a benchmark to improve the real model through computational analysis.

Liquid cooling system of an internal combustion engine is sometimes dealt as the channel flow system and the heat transfer coefficient for convective mechanism is calculated through those equations (Çengel, 2002; Dittus and Boelter,1930; Heywood, 1988; Ozisik, 1980;

Rohsenow and Griffith, 1955; Sieder and Tate, 1936). Moreover, the experimental investigations are performed through temperature measurement of the coolant in the jackets inserting various thermocouples in various points. In an actual sense, the heat transfer and temperature distribution is varied in every point of the cooling jacket. An extensive CFD model employing the energy equations for both fluid and solid parts can help providing with such results if proper boundary conditions can be used. The industrial researches on automotive cooling system development claims the CFD analysis as one of the effective and faster approaches and it can provide some key information which are not accessible through the experimental works like special issues: warm up, idling and key-off temperature conditions of the coolant in cooling jacket (Fontanesi et al., 2010; Jasak et al., 1999; Shibata et al., 2004).

2. PROBLEM STATEMENT

When an engine is keyed-off after a long time of full load travel for a few minutes like in traffic or in parking, the residual heat of the engine combustion chamber is conducted to the stagnant hot coolant inside the coolant jacket. Since modern internal combustion engines are designed to run under precision cooling mechanisms, coolant is already circulated through the jacket in higher temperature near to boiling point based on the working pressure. This excess heat can't be carried out to the radiator as the actuators are idle/ stopped as soon as the engine is in idle condition, thus resulting sudden temperature rise of the coolant and it leads to severe coolant spill out, too due to the high pressure generated from the produced vapours or boiling bubbles.

Piccione and Sergio (2010) have described this problem stating that, as the engine is shut downand coolant flow stops thus the engine is brought down in an idle condition for 5sec to 80sec, the head metal may be hotenough to vaporize a fraction of the coolant contained inthe cylinder head passages, causing the pressure within the coolingcircuit to rise above the threshold value of the radiatorcap pressure valve and, consequently, an important quantity of thecoolant to be expelled. A few researchers (Chastain, 2006; Allen et al., 2004; Ap and Tarquis, 2005; Chominsky and Dehart, 2010; Franchetta et al., 2006; Piccione and Sergio., 2010; Bova et al., 2004) have already noticed this phenomena and few resolutions are proposed by them too. Here in this article, an internal combustion engine model has been contrived to investigate the temperature distribution of the coolant inside the cooling jacket through CFD based steady state analysis. The problem is considered as conjugate heat transfer mechanism. Therefore, only a few input boundary conditions required to use the energy equations in computational codes. Steady state heat transfer and temperature distributions may help providing the insight of the coolant, coolant-solid interfaces and coolant side walls in a point to point basis. The steady state CFD simulation can guide the automotive design engineers as well as the manufacturers to find out required resolution to devise to resolve any harmful events due to heat transfer from the combustion chamber walls to the cooling jacket coolant after key-off.

3. MODELING THE PHYSICS

The researchers are generally inclined to avoid the computational intricacies of employing energy equations in modelling the heat transfer phenomenon of engines due to data insufficiency (Nijeweme et al., 2001; Nuutinen, 2008; Torregrosa et al., 2008); rather they easily use either the experimental correlations or empirical correlations obtained from dimensional analysis while modelling internal combustion engine cylinder heat transfer to the coolant. But here in this analysis, the energy equation is employed to observe the temperature distribution of the coolant in the water jacket.

Following the assumptions of incompressible fluid and simplified numerical modeling of actual Navier-Stokes models, the "Realizable k-ε Two-Layer turbulence" model by Shih et al.(1994) has been adopted for this simulation.Wolfstein(1969) model has been utilized here as "Two-Layer model formulation". This model can satisfactorily resolves the issues of normal stress constraints with consistency on physics defined by turbulent flows. The success of this equation over the other standard k-ε models is that it can well analyse the prediction of axisymmetric jet's spreading rate as well as planar jets; and it has wide spread validity on channel as well as boundary layer flows with same ability on separated flows, too. The realizable model computes the thermal conductivity of the material by $k_e = k + \frac{C_p \mu_t}{Pr_t}$, with a constant turbulent Pr value of 0.85. The wall heat flux is thus computed as,

$$q = \frac{\rho C_p \left(\frac{u}{u^+}\right)}{t^+}(T - T_w) \tag{1}$$

In a non-dimensional approach the wall laws are related to,

$$y^+ = \frac{yu'}{\nu} = \frac{y(\frac{u}{u^+})}{\nu} = \frac{yu}{\nu u^+} = \frac{y(\sqrt{\tau_w/\rho})}{\nu} \tag{2}$$

Here, y is the normal distance from the wall to wall cell-centroid, u is the wall-cell velocity component parallel to the wall, T is the fluid cell temperature near wall, T_w is the wall-boundary temperature, q is the heat flux, $u'$is the fluid reference average velocity, ν is the kinematic viscosity, τ_w is the wall shear stress.

The wall function also provides the heat transfer coefficient as follows:

$$h_{eff} = \frac{q}{(T_w - T_{ref})} = \frac{\frac{\rho C_p \left(\frac{u}{u^+}\right)}{t^+}(T - T_w)}{(T_w - T_{ref})} = \frac{(T_w - T)\rho C_p C_\mu^{1/4} k^{1/2}}{t^+(T_w - T_{ref})} \tag{3}$$

The turbulence model (RANS) is then mixed with the energy equations, continuity equations and momentum equations. The combined model is formulated and run in a solver's solutions process. Here, both the velocity and the pressure components are analysed separately in the segregated flow model solver (CD-Adapco, 2011; Trottenberg et al., 2001). The methodology is also known as uncoupled solution and the predictor-corrector approach is employed to link the momentum as well as the continuity equations to define the cases with continuity equations. Generally, the segregated approach is considered as co-located (where the pressure and velocity are considered to be in the same location, it is not staggered) but the Rhie-and-Chow (1983) type pressure-velocity coupling combined with a SIMPLE-type algorithm by Patankar (1980) are employed for advection velocities (where the cell-faces' velocities are utilized to compute the mass fluxes). This model is appropriate for incompressible fluid flows (constant density flows) and can compute the compressible fluids with low Mach number (less than 0.3) and few natural convection problems but this is not capable of capturing shocks in the high Mach number compressible fluid flows or high Raleigh-numbered fluid dynamic applications. A second order upwind convection scheme is used for the convective flux computation in the governing equation for better convergence.
The governing equations for continuity and momentum (in integral form) can be presented as follows respectively;

$$\frac{d}{dt}\int_V \rho x \, dV + \oint_A \rho(\mathbf{v}-\mathbf{v}_g).\, d\mathbf{a} = \int_V [S_u]\, dV \quad (4)$$

$$\frac{d}{dt}\int_V \rho x\mathbf{v}\, dV + \oint_A \rho\mathbf{v}(\times)(\mathbf{v}-\mathbf{v}_g).\, d\mathbf{a} = -\oint_A p\mathbf{I}.\, d\mathbf{a} + \oint_A \mathbf{T}.\, d\mathbf{a} + \int_V (f_g + f_u + f_\omega)\, dV \quad (5)$$

The transient as well as the convective flux terms are on the left side of the equation (5); whereas, the gradients due to pressure, viscous flux (due to viscous stress tensor, **T**) and the body forces (f) due to gravity (g), user defined (u) and other type of body forces like vorticity confinement (ω) respectively are in the right side.
The "Total Energy Equation" for fluid flow can be presented in integral form as follows:

$$\frac{d}{dt}\int_V \rho E\, dV + \oint_A [\rho H(\mathbf{v}-\mathbf{v}_g) + \mathbf{v}_g p].\, d\mathbf{a} = -\oint_A \mathbf{q}.\, d\mathbf{a} + \oint_A \mathbf{T}.\mathbf{v}\, d\mathbf{a} + \int_V \mathbf{f}.\mathbf{v}\, dV + \int_V S_u\, dV \quad (6)$$

Here, E is the total energy, H is total enthalpy, **q** is the heat flux vector, **T** is the viscous stress tensor, **v** is the velocity vector, $\mathbf{v}_g$ is the velocity vector for grids, **f** is body force vector; S_u is the energy source as user defined.
It is to be noted that the total energy equation for the solid model in the segregated solid energy model can be expressed as follows:

$$\frac{d}{dt}\int_V \rho_s C_{p,s} T_s\, dV = -\oint_A \mathbf{q}.\, d\mathbf{a} + \int_V S\, dV \quad (7)$$

Here, ρ_s is the solid density, $C_{p,s}$ is the solid's specific heat capacity, T_s is the solid's temperature, **q** is the heat flux vector and S is the energy source as defined.

3.1 Boundary conditions

Mass flow inlet: 1.3 kg/s (i.e. 78 L/min),
Pump capacity: 80 L/min.
Coolant inlet temperature: 355 K (i.e. 82 ^{0}C)
Coolant: Water (water has higher specific heat capacity than 50/50 EG-H_2O)
Initial pressure: 1.2 bar (gauge) (i.e. boiling point is >401 K)
Temperature of the combustion chamber walls, the cylinder head skirts and the exhaust manifolds: 600 K (i.e. 327 ^{0}C)
Materials: Cylinder block (Fe) and Cylinder head (Al)

3.2 Assumptions

The CFD simulations were performed in steady state conditions. The fluid was coolant (50/50 Ethylene glycol-H_20) and the simulations were performed considering the wall surfaces as smooth for the coolant flow. Several cases for combustion chamber wall temperature were considered to check the fluid flow heat transfer and temperature distribution. In all cases, it was considered that the heat transfer from the cooling jacket side walls to the flowing coolant is fully a forced convective mechanism. Since there is an improvement of BSFC (Brake Specific Fuel Consumption) due to maintaining a higher temperature of combustion chamber walls (about 4-6% improvement due to 80-100 ^{O}C more than the usual consideration of wall temperature (Rehman et al., 2010), the wall temperatures are mainly considered as of 573K, 600K and 773K (for high speed engines) ranges for the combustion side hot walls (Rakopoulos et al., 2008; Idroas et al., 2011).

The fluid (coolant: H_2O - Ethylene glycol) properties were considered to be as the incompressible fluid properties (i.e. constant properties). The corresponding boundary conditions are presented in the following sections of analysis.

4. RESULT AND ANALYSIS

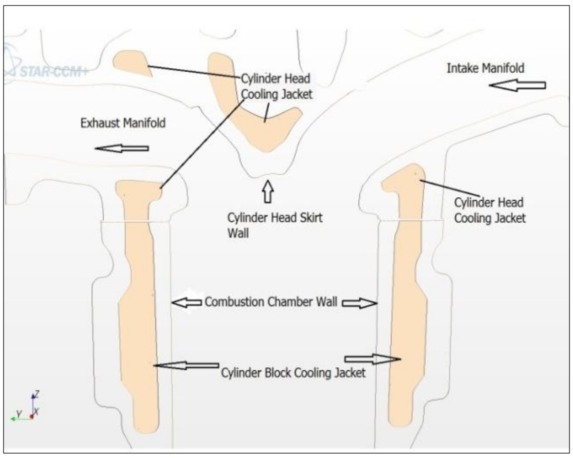

Figure 1 Cross sectional view of engine cooling jacket.

A schematic of the cooling jacket geometry is shown for internal combustion engine (Figure 1). The steady state simulation is performed with the value of the order 10^{-7} for the residual terms when it is stabilize by solving the required energy equations in the segregated approach. There were about 25 million volume cells in the solid parts and 4.6 million volume cells in the liquid continua of the engine's physical model. The result of coolant temperature distribution is presented here to observe their effect in those regions.

4.1 Temperature distribution

The flowing water carries the heat from the walls while flowing through the cooling jackets. The corresponding wall temperature and coolant temperature are shown in the "Figure 2" and "Figure 5".

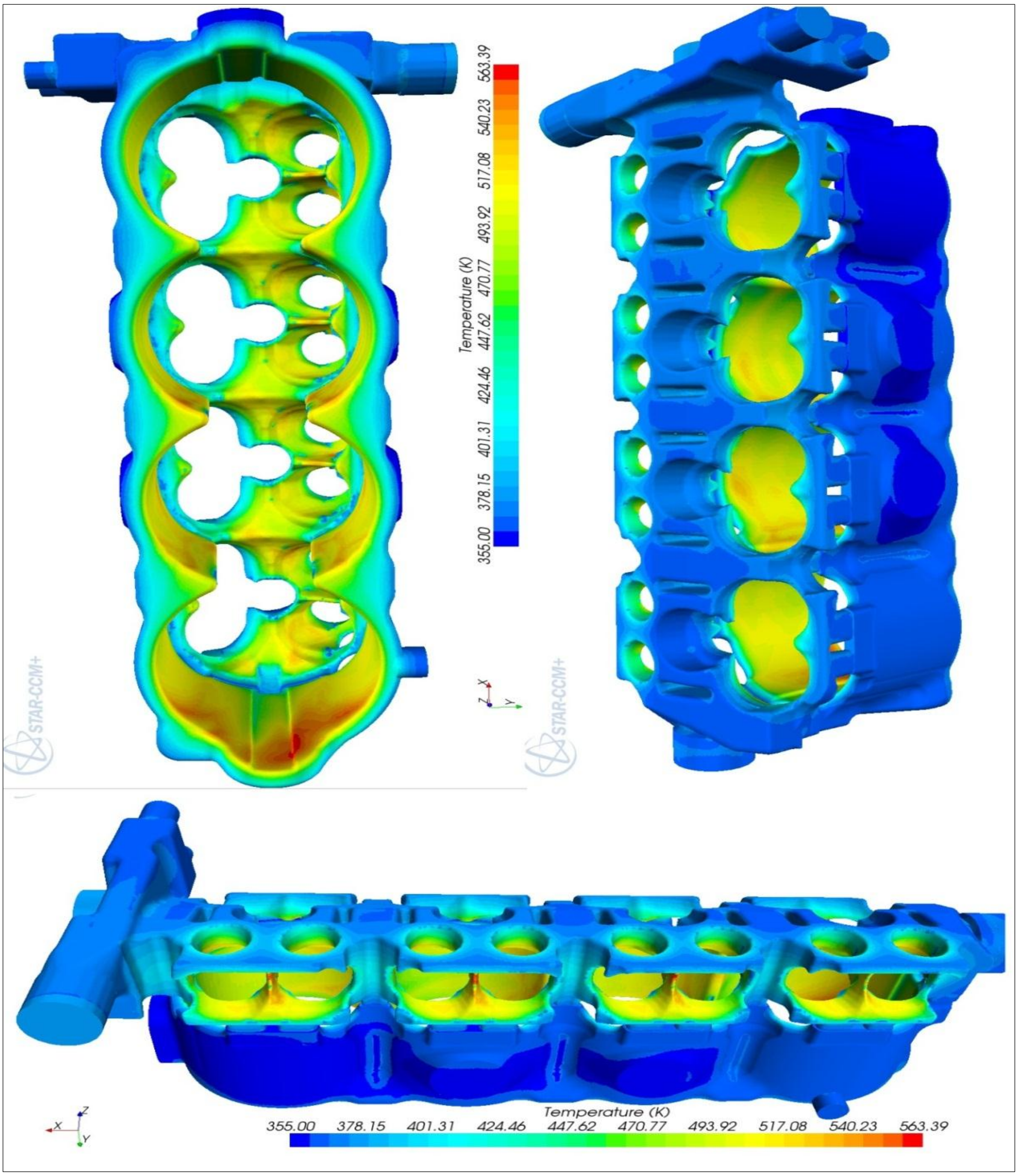

Figure 2 Temperature distribution of 1.3 kg/s coolant flow heat transfer.

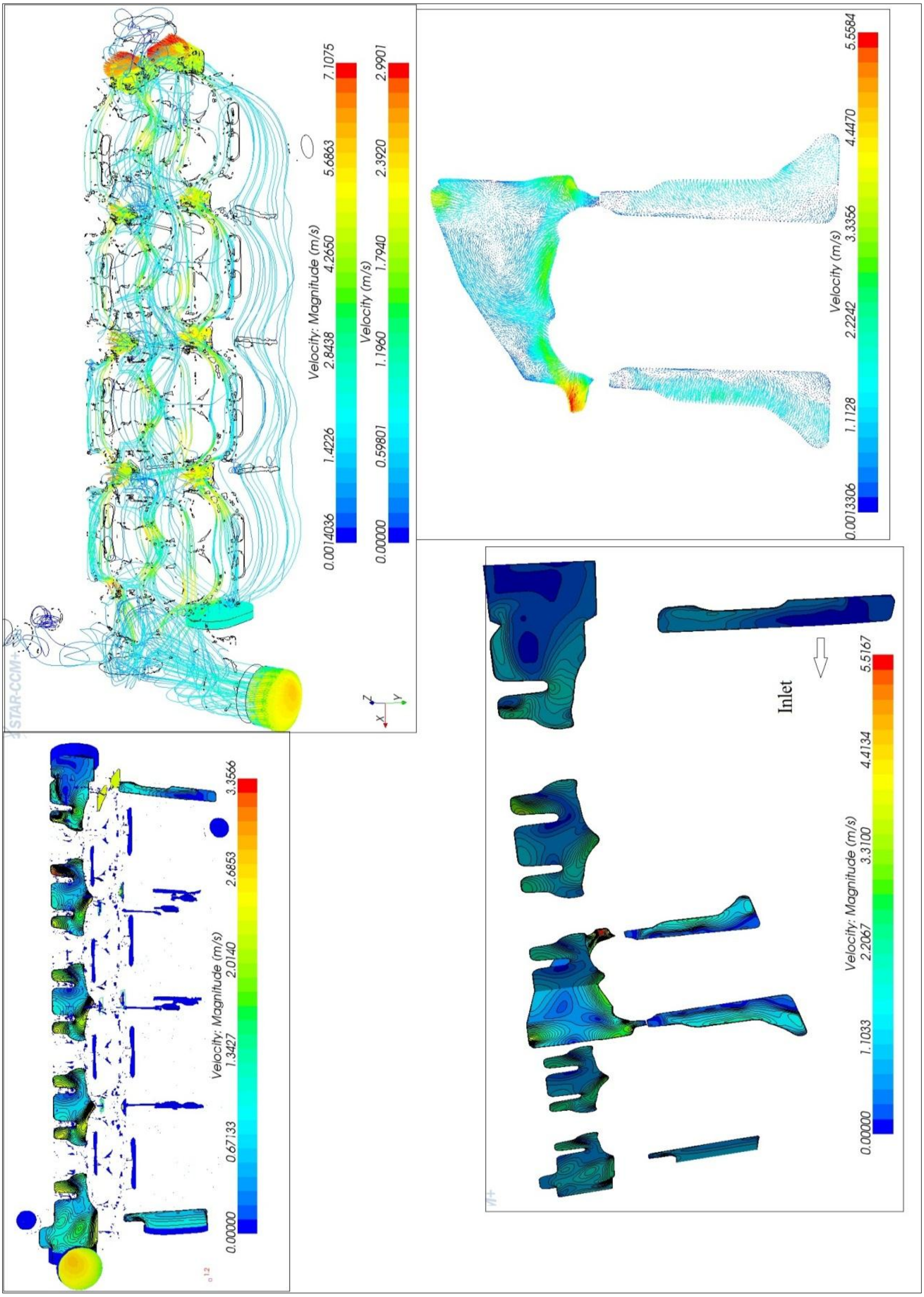

Figure 3 Velocity distribution of water (1.3 kg/s) in the cooling jacket.

From the analysis of fluid flow and heat transfer in an internal combustion engine where the water is used as a coolant shows a good temperature distribution; though high temperature boundary layers are found showing the boiling action (since the temperature is beyond boiling point of coolant used) in the thin layers of the fluid in contact with the hot side walls. Because of the simplified flow in the block jacket the heat transfer coefficient is showing almost a constant value in this section “Figure 4”. But the value found is a surface average quantity. In actual cases, the fluid near to the hot walls shows higher heat transfer coefficient for convective mechanism. The coolant surfaces which are in contact with the water jacket walls shows higher temperature ranging from 190 ^{O}C to 290 ^{O}C. This is pretty severe condition though engine cylinder head coolant outlet temperature is within the safe range of operation (Figure 5). As the fluid passes through the cooling jacket it gains heat and rises the temperature on various irregular geometric positions. Figure 2 also shows that the temperature of the coolant varies rapidly from the hot wall side to the other side of the channels. It is to be mentioned that after the high temperature fluid layer near the hot walls, the temperature of the coolant decreases within its cross section of the flow stream and that is completely in the safe zone.

Therefore, it is essential to manage a fluid flow through the cooling jacket even after the engine is shut down to avoid the further temperature rise and incipience of severe boiling in full channel. External flow energy should be obtained to manage this issue of fluid flow after the shutdown of engine.

4.2 Velocity distribution

The velocity distribution of the coolant in the channel is presented in the Figure 3. The simulated result demonstrates that the flow field is distributed in a gradual increase in velocity but in the lower mass flow rate. Because of low mass flow around the end part of the cylinder block it can’t take away the entire thermal load imposed on it. The velocity increment or the mass flow increment also increases the pressure losses in the obstacles and flow uniformity is disturbed.

5. THEORETICAL ANALYSIS OF THE HEAT TRANSFER IN THE COOLANT CHANNEL

Here the combustion wall temperature is considered as mentioned by Stone(1992). And the coolant mass flow rate is calculated as per the general energy equation, $\dot{m}_c C_{pc} \Delta T = \mathbf{Q}$. Where $\dot{m}_c$is the coolant mass flow rate (kg/s), C_{pc} is the specific heat capacity of coolant (3.7 kj/kg.K), ΔT is the temperature difference between engine coolant inlet and outlet, Qis the heat to be carried away by the coolant. The total amount of heat to be carried away for the conceived engine is obtained from authors’ published article Hazrat et al.(2012).

Now, as per the average heat transfer coefficient found in the Figure 4:

HTC for cylinder head jacket, h_{ch}= 3397 W/m^{2}K and for cylinder block side jacket, h_{cb}= 3192.56 W/m^{2}K

For both the cylinder block and head, the wall thickness (V_s/A_s) is considered as characteristic length, l_c = 6e-3m. Adopting the material properties for cylinder head and block, the following equation can be used to obtain theoretical value of wall temperature inside the cooling jacket hot side.

$$\frac{(T_w(t)-T_c)}{(T_{w.g}-T_c)} = \exp\left(-\frac{h_c A_s t}{\rho_s V_s C_s}\right), \tag{8}$$

Where $T_{w.g}$ is the wall temperature at gas side, T_c is the coolant inlet temperature, t is the reference time and in this case it is t=1

Therefore, from equation (8),

For block, $T_{w,Fe}$= 355+(600-355)*exp{-(3192.56)/(7870x448x6e-3}=565.68 K

And, for cylinder head, $T_{w,Al}$ = 355+(600-355)*exp{-(3397)/(2702x903x6e-3}=549.26 K

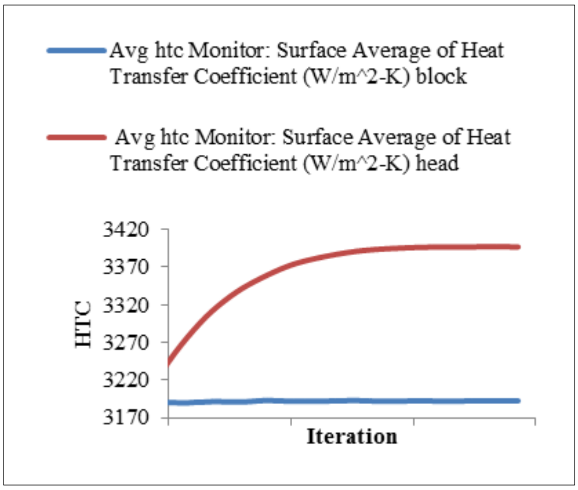

Figure 4 Heat transfer coefficient resulting from the water cooling of the engine.

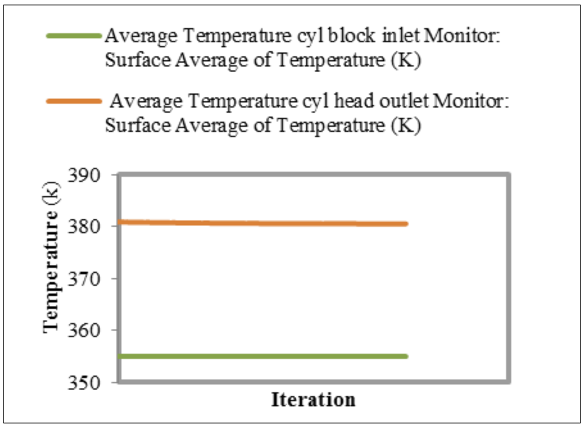

Figure 5 Coolant inlet and outlet temperature distribution.

The conduction heat transfer equation from lump capacitance also supports the higher temperature distributions in the coolant side wall. So the conceived CFD model can be considered as acceptable in some extent though Stone(1992) mentioned that the inner wall temperature should be 400K if the gas side wall temperature is 600 in the spark ignition engines. But the author did not mention the standard coolant flow rate to be maintained in case of predefined heat removal conditions.

6. CONCLUSION

The steady state heat transfer analysis of a commercial passenger vehicle SI engine has been obtained through out a complete high performance computing facility and standard computational code to run the segregated approach solvers to resolve the energy equations. The result shows that though engine coolant outlet temperature is within the safe region of the operation principle, there is still a requirement of coolant flow management inside the coolant channels more efficiently so that the coolant can consume more heat and the velocity pressure of the coolant should be managed through improved design of the complex cooling jackets. It has been shown that the solid wall thickness between the combustion chamber wall and the cooling jacket channel is also a factor for conduction heat transfer and temperature distribution. In standard cases (Stone, 1992), the wall thickness is considered as 10mm but here in the conceived model it was 6mm only and therefore, the coolant side wall temperature is also found as more higher than as mentioned in the existing documents. But a proper flow management of coolant through the cooling jacket can reduce the severity due to temperature increment in the coolant side. It may require dynamic analysis and the authors are recently trying to establish a full scale transient simulation for further analysis with a sufficient computational facility.

ACKNOWLEDGEMENT

The authors would like to acknowledge the Techno fund (TF001-2010) and Centre for Energy Sciences in University of Malaya for providing complete facility to conduct this research work.

REFERENCES

Allen, D., Hnatczuk, W., Lasecki, M. and Chalgren, R. 2004. Advanced thermal management for military application. 130100 - Air Condition, Heating, Lighting and Ventilating. Defense technical information center.

Ap, N. and Tarquis, M. 2005. Innovative engine cooling systems comparison. SAE Technical Paper, (2005-01-1378).

Borman, G. and Nishiwaki, K. 1987. Internal combustion engine heat transfer, Progress in Energy and Combustion Science 1 (13): 1-46.

Bova, S., Piccione, R., Durante, D. and Perrussio, M. 2004. Experimental analysis of the after-boiling phenomenon-in a small i.C.E. SAE Technical Paper, (2004-32-0091).

Cd-Adapco 2011. Star ccm+ user manual. 6.02.009 ed.

Çengel, Y.A. 2002. Heat transfer: A practical approach, Mcgraw-Hill (Tx).

Chastain, J.H., Jr. 2006. Internal combustion engine cooling strategies: Theory and test. Master of Science, Clemson University.

Chominsky, W. and Dehart, R.C. System and method for thermal management of engine during idle shutdown. United States patent application 12/448,540.

Demuynck, J., Raes, N., Zuliani, M., De Paepe, M., Sierens, R. and Verhelst, S. 2009. Local heat flux measurements in a hydrogen and methane spark ignition engine with a thermopile sensor, International Journal of Hydrogen Energy 34 (24): 9857-9868.

Dittus, P.W. and Boelter, L.M.K. 1930. Heat transfer in automobile radiators of the tubular type, University of California Publications in Engineering 2(13): 443-461.

Fontanesi, S., Cicalese, G. and Giacopini, M. 2010. Multiphase cfd-cht analysis and optimization of the cooling jacket in a v6 diesel engine, SAE Technical Paper: 2010-01-2096.

Franchetta, M., Bancroft, T.G. and Suen, K.O. 2006. Fast transient simulation of vehicle underhood in heat soak, SAE Technical Paper: 2006-01-1606.

Hazrat, M.A., Masjuki, H.H., Kalam, M.A., Pang, S.C., Memon, L.A. and Magami, I. 2012. Determination of time variant 1d-3d temperature and heat transfer distribution inside the cooling jacket of a si engine cooling system after key-off, SAE Technical Paper: 2012-01-0642.

Heywood, J.B. 1988. Internal combustion engine fundamentals, NY, McGraw-Hill, Inc.

Idroas, M.Y., Farid, N.A., Zainal, Z.A., Noriman, K. and Azman, M. 2011. Mechanical power assessment of an alpha V-type stirling engine converted diesel engine, International Journal of Mechanical and Materials Engineering 6 (2): 160-166.

Jasak, H., Luo, J., Kaluderčić, B., Gosman, A., Echtle, H., Liang, Z., Wirbeleit, F., Wierse, M., Rips, S., Werner, A., Fernström, G. and Karlsson, A. 1999. Rapid cfd simulation of internal combustion engines, SAE Technical Paper: 1999-01-1185.

Nijeweme, D.J.O., Kok, J.B., Stone, C.R. and Wyszynski, L. 2001. Unsteady in-cylinder heat transfer in a spark ignition engine: Experiments and modeling, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering 215: 747-760.

Nuutinen, M. 2008. Improved near wall treatment for ci engine cfd simulations. Ene-39.4052 Post Graduate Seminar on Heat and Mass Transfer. Helsinki University of Technology, Internal Combustion Engine Technology.

Ozisik, M.N. 1980. Heat conduction, New York. , John Wiley and Sons.

Patankar, S.V. 1980. Numerical heat transfer and fluid flow, New York, Hemisphere.

Piccione, R. and Sergio., B. 2010. Engine rapid shutdown: Experimental investigation on the cooling system transient response. ASME Journal of

Engineering for Gas Turbines and Power 132 (7): 078801.
Rakopoulos, C. D., Giakoumis, E.G. and Rakopoulos, D.C. 2008. Study of the short-term cylinder wall temperature oscillations during transient operation of a turbocharged diesel engine with various insulation schemes, International Journal of Engine Research 9 (3): 177-193.
Rehman, A., Sarviya, R.M., Dixit, S. and Pandey, R.K. 2010. The influence of coolant temperature on the performance of a four stroke S.I. Engine employing a dual circuit cooling system. Agricultural Engineering International: the CIGR Journal 12 (1): 84-90.
Rhie, C.M. and Chow, W.L. 1983. Numerical study of the turbulent flow past an airfoil with trailing edge separation, AIAA Journal 21: 1523–1532.
Rohsenow, W.M. and Griffith, P. 1955. Correlation of maximum heat flux data for boiling of saturated liquids, Technical report (Massachusetts Institute of Technology, Heat Transfer Laboratory).
Shibata, T., Matsui, H., Tsubouchi, M. and Katsurada, M. 2004. Evaluation of cfd tools applied to engine coolant flow analysis, Mitsubishi Motors Technical Review 16: 56-60.
Shih, T.H., Liou, W.W., Shabbir, A., 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.
Sieder, E.N. and Tate, G.E. 1936. Heat transfer and pressure drop of liquids in tubes, Industrial & Engineering Chemistry 28 (12): 1429-1435.
Stone, R. 1992. Introduction to internal combustion engines, Hampshire and London, MacMillan.
Torregrosa, A.J., Olmeda, P.C. and Romero, C.A. 2008. Revising engine heat transfer, Journal of Engineering Annals of Faculty of Engineering Hunedoara 6 (3): 245-265.
Trottenberg, U., Oosterlee, C.W., Sch¨Uller, A., Brandt, A., Oswald, P. and St¨Uben, K. 2001. Multigrid, San Diego, Academic Press.
Wolfstein, M. 1969. The velocity and temperature distribution in one-dimensional flow with turbulence augmentation and pressure gradient, International Journal of Heat Mass Transfer 12: 301-318.