**Amit V. Paratwar, D.B Hulwan / International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com**
**Vol. 3, Issue 4, Jul-Aug 2013, pp.892-902**

# Surface Temperature Prediction and Thermal Analysis of Cylinder Head in Diesel Engine

**Amit V. Paratwar[1], D.B Hulwan[2]**

[1]( *P.G student, Mechanical Engineering Department. Vishwakarma institute of Technology, Pune*
[2](*professor, Mechanical Engineering Department. Vishwakarma institute of Technology, Pune*

## ABSTRACT

**Engine heat transfer and cooling is always been a crucial area of interest for improvement of engine performance.CFD methods and tools used today provide clearer and more detailed data on temperature, flow and pressure variation. Aim and objective of the present study is to carry heat transfer as well as flow analysis of existing cooling jacket of 6-cylinder turbo after-cooled medium duty diesel engine and then investigate the factors affecting cooling performance to optimize the said parameters through steady state CFD analysis and validate them with experimental results.**

***Keywords*** – *heat transfer, cylinder head water jacket, coolant flow, numerical simulation, CFD*

## I. INTRODUCTION

IC engine is an important prime mover used in various fields mainly in automotive and power generation. In early days of IC engine development, power output and efficiency were the main focus of researchers. Heat in commercial diesel engine has to travel through engine components leads to increase its temperature. This percentage of heat loss through engine cooling varies according to various engine and cooling system parameters. In this study, for one single head of diesel engine, three-dimensional coolant flow inside cooling water jacket is simulated using 3D CFD software. Some estimating study is developed, including whether cooling water system capacity for cylinder head is well or not, the distribution of coolant to cylinder is even or not and so on. Finally, some advices are put forward, which can provide gist for optimizing the diesel engine..

## II. MODELING STRATEGY

The methodology presented in this paper is based on CFD and validation is done experimentation in which templug where used The CFD analyses constitute the first step of the proposed modeling strategy. Two different aspect are investigated in detail: i) the fluid-dynamic behavior of the cooling circuit is firstly analyzed and optimized aiming at improving the cooling efficiency ii) the point-wise fluid/solid heat transfer is then evaluated. In particular, benefits on the overall predicting capability brought in by the adoption of a proper phase-change model are highlighted by means of a preliminary comparison with a simplified model and by a subsequent validation of the methodology against experimental measurements of the temperature distribution within the engine head.

Thus in present study 6-cylinder, 4-stroke 4-valve, turbo after-cooled, high pressure diesel engine is selected. Initially geometrical layout will be studied in terms of flow and heat transfer analysis by using CFD as a tool. The parametric study of an IC engine cooling jacket will be carried out using experimentations and results will be used to comment on effectiveness of selected parameters and optimizing these parameters by using statistical methods.

## III. HEAT TRANSFER MODELING

### 2.1 Engine heat transfer

The heat transfer coefficient depends on the engine geometry, such as the exposed cylinder area and bore, and the piston speed. Due to the complex gas flow in the cylinder, it varies with location in the cylinder and in time with changing piston position. The value of the heat transfer coefficient is found from a Nusselt number - Reynolds number type correlation, A number of empirical equations exist for hg in the cylinder at any instant; perhaps the most commonly used being that by Woschni.[8]

There are three types of heat transfer coefficients used in engines heat transfer. The peak values of the instantaneous and local coefficients can be many times higher than the averaged values.

Table 1 Types of heat transfer coefficient[5]

| | Heat transfer model | Description |
|---|---|---|
| h(x,t) | Averaged over time & space | Overall steady state energy balance calculations |
| h(x,t) | Instantaneous, space average | Heat transfer versus crank angle |
| h(x,t) | Instantaneous, local | Local calculations of thermal stress |

For heat transfer versus crank angle calculation the second type of heat transfer coefficient from above table.is selected i.e. Instantaneous, space average- heat transfer versus crank angle. Because the peak values of the instantaneous and local coefficients can be many times higher than the averaged values. It gives actual condition during the engine cycle with respect to crank angle or space.

Woschni model is preferred over other model because of the following reasons [13]

1. Universally applicable equation for the instantaneous heat transfer coefficient in the internal combustion engine.
2. Calculates the amount of heat transferred to and from the charge.
3. It is the most commonly used heat transfer model and applied to all cylinder elements.

Assumptions:

1. On all surfaces of the cylinder heat flow coefficient and velocity of charge is uniform.
2. This model accounts the increase in gas velocity in the cylinder during combustion.
3. Woschni proposed that the average gas velocity should be proportional to the mean piston speed.
4. Air is the working medium.
5. Heat transfer by conduction through the walls is one-dimensional

### 2.2 Calculation for heat transfer model

#### 2.2.1 Slider crank model

The geometric parameters of the piston cylinder are represented in Figure 5.4 .Top dead center refers to the position of the crank shaft at a crank angle of $0^o$. This position is otherwise known as the clearance volume, $V_c$. At bottom dead center the crank angle is at $180^o$. In this position the cylinder volume is at its maximum, $V_1$. The difference between the maximum and minimum volume, $V_1$-$V_o$, is defined as the displacement volume, $V_d$. The displacement volume can also be represented as a function of the bore and stroke[8].

$$V_d = \frac{\pi}{4} b^2 s$$

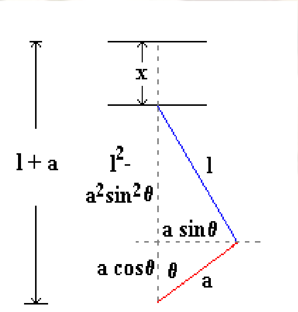



**Figure 1.** Piston cylinder geometry[8]

The final form of the Slider-Crank model is given as a non-dimensional relationship

$$\frac{V}{V_d} = \frac{1}{r-1} + \frac{1}{2}\left[1 + R - \cos\theta - (R^2 - \sin^2\theta)^{1/2}\right]$$

The surface area of the cylinder has to be evaluated. The combustion chamber surface area is given by:

$$A = A_{ch} + A_p + \pi bx = \frac{\pi}{4} b^2 + \frac{\pi}{a} b^2 + \pi bx$$

Calculation of heat transfer coefficient [5]

$$h_w = 3.26.B^{-0.2}.p^{0.8}.T_g^{-0.55}.\left\{c_1.S_p + c_2\left(\frac{V_d.T_r}{P_r.V_r}\right)(p - p_m)\right\}^{0.8}$$

$$-180 \le \theta \le \theta_0 \rightarrow c_1 = 2.28\,, c_2 = 0$$

$$\theta_b \le \theta \le 180 \rightarrow c_1 = 2.28\,, c_2 = 3.24 \times 10^{-3}$$

Where,

P = Combustion Pressure, Ta = Combustion temperature $P_{motor}$ = Motored pressure, $T_r$ = Reference temperature at start of combustion, $P_r$ = Reference pressure at start of combustion, Vr = Reference volume at start of combustion

Calculation of combustion temperature $T_g$:

The gas properties in the correlation equation are evaluated at the instantaneous average cylinder temperature determined from the ideal gas law [5]:

$$T_g = \frac{P\ M}{\rho\ R}$$

Calculation of density

Density of air at intake conditions is calculated by using ideal gas law [5]

$$PV = mRT$$

$$P = \frac{RT}{\rho M} \qquad \therefore\ \rho = \frac{PM}{RT}$$

Cylinder head flame face temperature:

The mathematical expression for the calculation of instantaneous cylinder head temperature without ceramic insulation coating through this model is [19]

$$Q_h = \frac{T_g - T_c}{rh1}$$

The actual combustion pressure with respect to crank angle in the engine cylinder during working cycle, It is also input in model to determine all the above parameters.

#### 2.2.2 Heat transfer model output:

Crank angle Vs bulk gas temperature

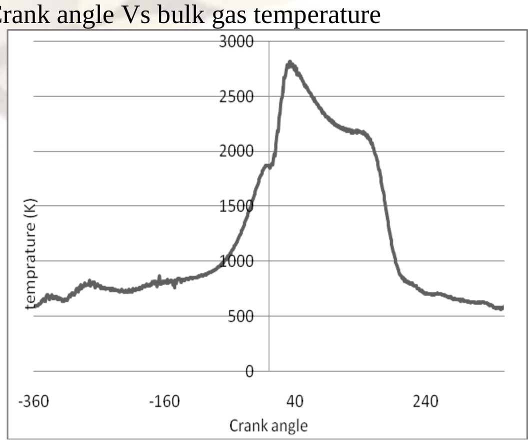


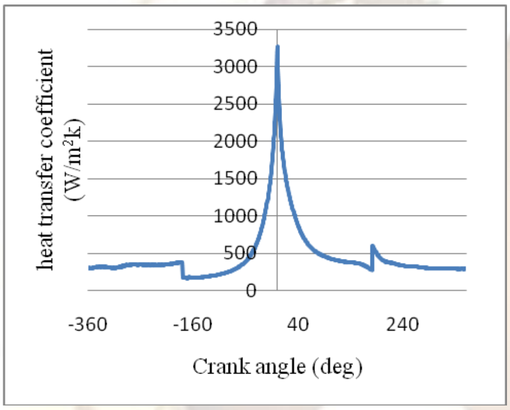

**Figure 2.** crank angle vs bulk gas temperature

Fig .Provides the theoretical values of mean gas temperature with respect to change in crank angle. During suction stroke the mean temperature of air is about constant value and in compression stroke it increases with crank angle. The peak value is reached at about 20° after TDC which is about 2700 k and the average temperature of T-ϴ is about 2780 k:

**Figure 3.** Crank angle vs. heat transfer coefficient

Fig. represents the theoretical values of the gas side heat transfer coefficient with crank angle. It reaches peak value of 3000 $W/m^2K$ at about 15° after TDC. The graph is obtained from heat transfer coefficient calculated from Woschnis correlation in which average heat transfer coefficient is found to be 600 $W/m^2k$ and maximum value is 3587 $W/m^2k$.

## IV. CFD ANALYSIS AND MODEL SETUP




### 3.1 Three-Dimension Model of Cooling Water Jacket.

The geometric model of the cylinder head was created using the software 3D modeling software, which is useful in component and surface modeling, virtual assembly, and in generating engineering drawings The component studied is one cylinder head of 6 cylinder in line high power diesel engine. Some of the characteristics properties of the invested engine are presented in the table below.

The engine has a split structure, namely one cylinder head for one cylinder. The cooling system of this engine is distinguished by the manner in which coolant is distributed. The cylinder block water jackets are connected in series, while the cylinder head water jackets are connected in parallel. This design enables the coolant collected in cylinder blocks to enter each cylinder head separately through transfer channels separately, which is also the reason why only one cylinder head has been investigated

The completed cylinder head solid model is displayed below

.

**Figure 4** geometric model of single cylinder head.

Where in the above figure the inlet is the inlet for coolant in the cooling jacket of cylinder head from the pump inlet of coolant circuit and the outlet goes into the crankcase cooling jacket with the help of outlet hole.

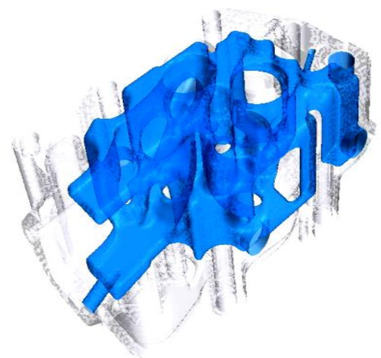


**Figure 5** Extracted cooling jacket from single head

1.2 Flow and thermal boundary conditions for cylinder head & water jacket

In the numerical simulations, the coolant flow in coolant cavities of a cylinder head was assumed to be 3D steady state, incompressible turbulent flow, and the viscosity in the near wall region was taken into account, while boiling of the coolant and roughness of walls were also considered. It was already discussed that model and solvers selection plays an important role in any CFD analysis.

Boiling model and gravity model were also imparted to account for the possibilities of effect of boiling and buoyancy. For boiling model boiling temperature, latent heat, vapor density and surface tension were provided as input. For gravity model vector value of gravitational acceleration was provided depending on actual position of engine in working condition Three dimensional, Steady, Liquid, Segregated flow, Constant density, Turbulent, Reynolds average Navier-strokes, K-epsilon turbulence, Realizable K-epsilon two layer, Two layer all y+ wall treatment

For solid head Region Other than three dimensional and steady state models, constant density model used as it was assumed that properties of block material will remain constant. As heat input needs to be given, „Segregated Solid Energy‟ model was selected. „Cell Quality Remediation‟ model was used to obtain better results at poor quality meshes if any. Table shows solid block properties provided

Table 2 initial conditions

| Parts | Temperature (K) | Heat transfer coefficient($W/m^2k$) |
|---|---|---|
| Deck face | 1200.0 | 625 |
| Exhaust port | 1000 | 580 |
| Inlet port | 313 | 100 |
| Exhaust valve guide | 313 | 160 |
| inlet valve guide | 313 | 100 |
| Exhaust valve seat | 1100 | 600 |
| inlet valve seat | 313 | 600 |
| Solid head. | 313 | 100 |

3. COOLING JACKET:
1. Inlet
*Mass flow inlet*: 3.75 Kg/sec
*Total temperature*: 72˚C
2. Outlet
*Pressure outlet*: gauge
*Static temperature*: 77˚C
3. Cooling jacket
*Wall roughness factor*: 1mm
4. Wall boundary (Fixed temperature type )
The value used for the temperature at the wall surface is 303 k, which is the empirical data often used for the heating surface of the cylinder head. *K*

All CFD simulations were carried out using the coolant mixture of water (50%) and glycol (50%) at a constant inlet temperature 373 Kelvin. The fluid properties are listed below.
Density: 1008.8 3 kg/$m^3$
Specific heat capacity: 3592.3 J/Kg-k
Dynamic viscosity: 0.000675 Kg/ms
Thermal conductivity: 0.39964 W/mk
Boiling point of glycol and water: 401*K* (128) *C*

## V. SIMULATION RESULTS

### 4.1 Flow simulations

### 4.1.1 flow field distribution of cylinder head water jacket

The average velocity magnitude over the whole flow field is 0.892 m/s, which meets the need of flow rate 0.5 m/s for cooling the engine. Some key zones were investigated particularly such as exhaust port, fuel injector, and nose zone

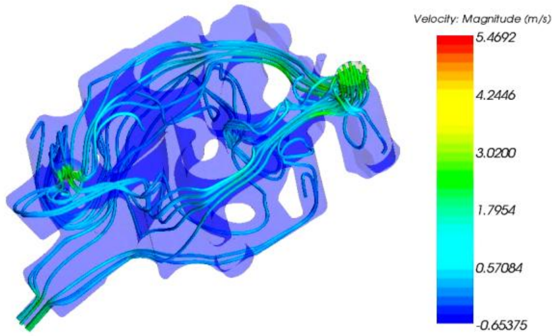



**Figure 6** Contour of velocity streamline Distribution (m/s)

The design for this cylinder head forced the flow to go around the exhaust port where most of the heat was output, then cooled down the fuel injector and finally the intake port. It is seen that the velocity of the flow around the exhaust port was stagnant, thus it is necessary to increase the coolant flow rate in this section of head.

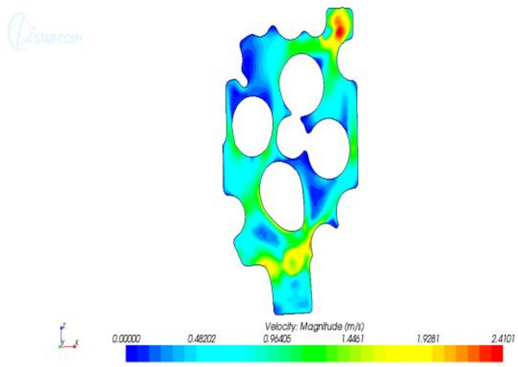



**Figure 7** Contour of velocity distribution at cylinder head bottom section (m/s)

### 4.1.2 Pressure distribution of cylinder head water jacket

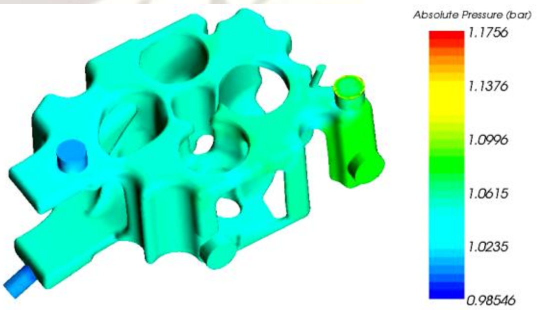



**Figure 8** Contour of Pressure Field Distribution (bar)

It can be seen from the contour of pressure that the outlet displayed the lowest pressure, while the region near inlet and at nose area presented the highest pressure, which indicated that the largest pressure drop occurs between the inlet and outlet.

The pressure loss between the inlet and outlet is calculated to be 0.029 bar which is a good indicator of low resistance in flow and potential to enhance efficiency.

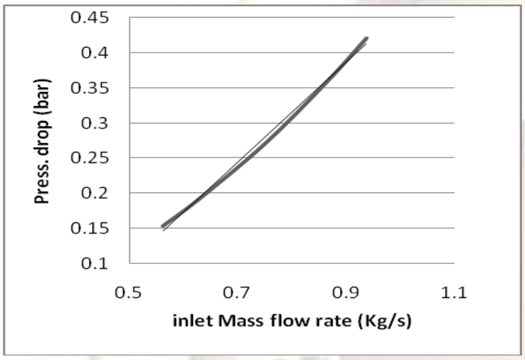
### 4.1.3 Effects of Flow Rates on flow simulation

As the engine has been strengthened, it's necessary to check the consequent effects on the cooling system. Four typical working conditions were taken into account, by varying its flow rate and the results are being presented below.

1. Pressure Loss

Pressure loss is computed by taking the difference of area-averaged pressures at the inlet and outlet for different mass flow rate

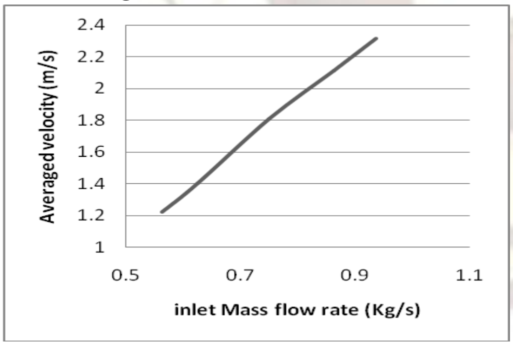
**Figure 9** pressure losses at different flow rates




2. Velocity

The velocity presented below is the volume-averaged velocity calculated over the interior of the water jacket

**Figure 10** average velocities at different flow rates




Enhancement in flow rate will strengthen the cooling effects to some extent, but in the meanwhile bring in higher pressure loss of the flow. The variation of heat transfer coefficient has close relationship with that of the velocity. It is seen from the velocity contour and the surface heat transfer coefficient contour that higher velocity leads to higher heat transfer coefficient. From the semi-empirical formula of the heat transfer coefficient, this can be proved which in turns induces pressure drop within cooling jacket

### 4.2 Thermal simulations

4.2.1 Cylinder head

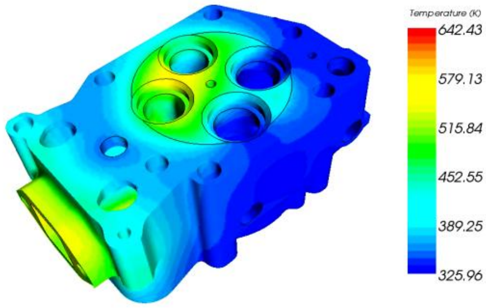



**Figure 11** contour of temperature distribution across head

The above figure clearly shows the maximum temperature is found in the region of exhaust port and the region near the exhaust bridge on the deck face. Thus more attention is given for the pressure and velocity contours of the cooling jacket as there is the possibility of nucleate boiling of coolant within this region

4.2.2 Deck face

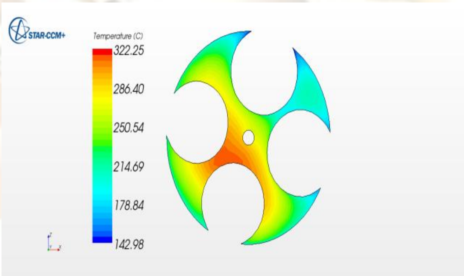



**Figure 12** Contour of temperature Distribution across deck face

Deck face is the only part on cylinder head which is exposed to burnt gases within the combustion chamber and is the part where maximum temperature is supposed to be reached in the cylinder head .Deck face accommodates both the exhaust and inlet seats their respective ports and the injector. Maximum temperature is is found at the exhaust valve bridge, i.e. 558 K.where heat transfer coefficient is around 2000 $W/m^2k$

4.2.3 Exhaust port

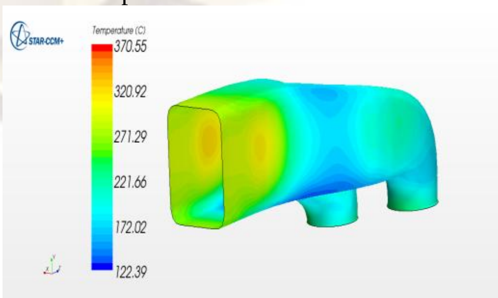



**Figure 13** contour of temperature distribution across exhaust port

4.2.4 Cooling jacket

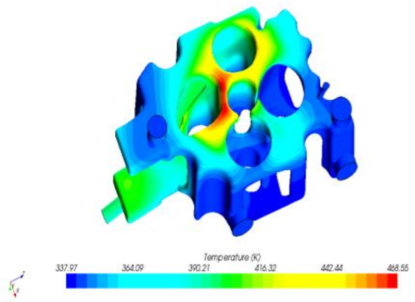



**Figure 14** contour of temperature distribution across cooling jacket

The temperature distribution was displayed in the range of 373K and 418K. The bottom contacted with the fire deck, was of extremely high temperature, which is reasonable in the sense of the structure characteristics and work principle of the cylinder head. The temperatures around the wall of the annular cavities, where the fresh air and exhaust passed, were ranging from 384K to 397K, which showed acceptable cooling effects of the cylinder head By comparison of HTC and temperature distributions, it is evident that the regions with high heat transfer coefficient correspond to ones with low temperature, and vice versa also the value of wall boiling heat flux is maximum at the location of maximum temperature in cooling jacket Here it must be understood that red region indicates no boiling whereas blue region indicates heavy boiling. Boiling was proved to be good in cooling jacket but up to a considerable limit. Nucleate boiling was very initial stage of boiling. Nucleate boiling increases heat absorption from boiling surface in the form of latent heat required for phase change of liquid. Boiling usually starts above magnitude of 900,000.

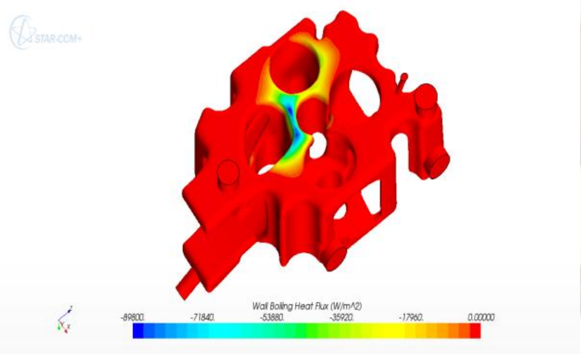



**Figure 15** wall boiling heat flux on cooling jacket

## VI. EXPERIMENTAL METHODOLOGY

In diesel engine certain component temperature can be critical for durability and function. The components considered are cylinder head combustion chamber, cylinder bore, piston, valves. Due to the different requirement of measuring systems or technique it is necessary to carry out different procedures appropriate to the component under investigation.

Common techniques used
1. Thermocouples or temperature sensor
2. RTD (Resistance Temperature Detector)
3. Temperature gun.
4. Templug

in present work due to complication of geometry, drilling the holes for thermocouples is risky for cooling jacket .The use of templug is done to measure the temperature at critical zones in the cylinder head.

### 5.1Templug

Templug were developed in the early 1970‟ s in a joint effort by TEI & Shell ResearchTemplug is a temperature sensitive steel set-screw. It is used for determining the maximum temperature in locations that are difficult to instrument using conventional thermocouple type techniques. They function on the relatively simple principle of thermal tempering of hardened steel. The components made up of nickel alloys, aluminum ,composites, ceramics can be tested with this technique. Templug set-screws are available in two sizes, which are M3 and M1.6

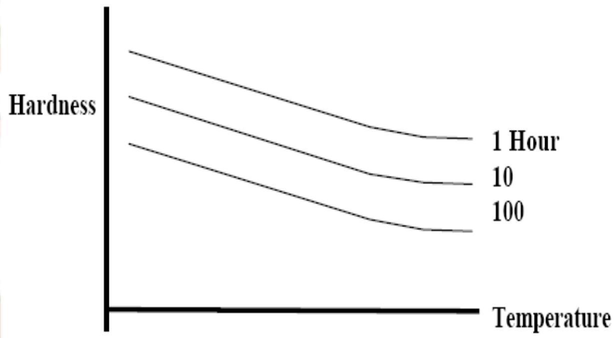



**Figure 16** Hardness vs temperature for various exposure periods [16]

By drilling and tapping, a Templug can be installed directly into the location at which temperature needs to be measured. No other instrumentation necessary. Master calibration curves of the type shown below were developed by 'Shell' for special heat treated alloy steels and are the basis for determining the maximum temperature [16].

From the above curves, the maximum temperature is determined based on the time of exposure (known from experimental conditions) and the measured hardness.

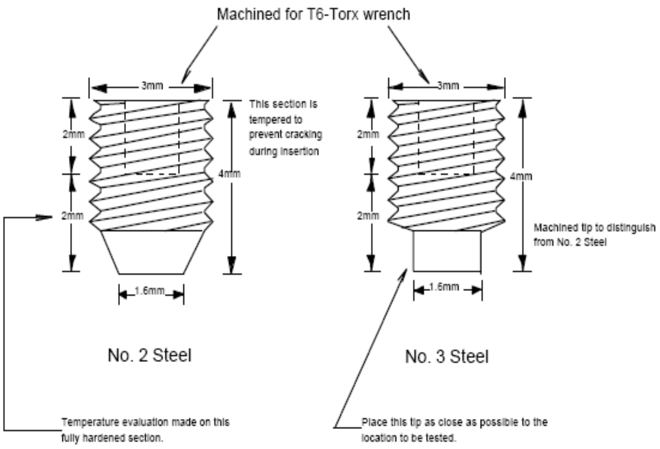



**Figure 17** standard m3 x 0.5 templug [16]

The ideal temperature/time cycle for determining maximum temperature would be where the Templug is heated rapidly to a uniform maximum temperature, maintained at this temperature followed by rapid cooling. This cycle simulates conditions that are used for generating the master calibration curves and consequently, should minimize error. The ideal cycle has been shown

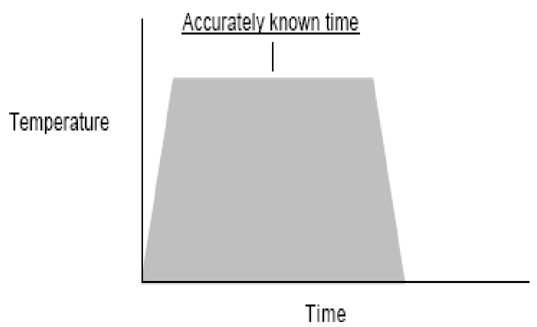



**Figure 18** Ideal Temperature/Time Cycle [16]

### 5.2 Location of templug in cylinder head

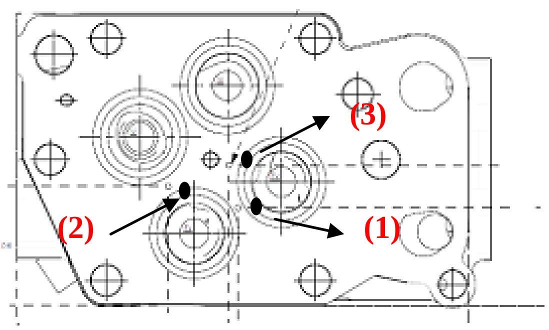



**Figure 19** Location of templug in cylinder head bottom

Locations are,
1-Between two exhaust ports
2-Between inlet and exhaust port
3-Between exhaust port and fuel injector

The above figure shows location of the templug in the cylinder head. The drawing is prepared in the Auto-CAD 2008. By focusing on literature survey and CFD analysis done on the engine we finalize the maximum temperature location in the cylinder head as shown in figure by red spot.

### 5.3 Experimentation results

The above specified engine is tested for the temperature measurement in the cylinder head using templug. Lube oil pump is used to supply pressurized oil to various parts in the engine. The power given to lube oil pump from the main crankshaft power trough gearing To carry out the experimentation using templug there is need to follow the specified test cycle as explained in the templug user guide and templug manual , thus by keeping the load on the engine constant and increasing the load to full load value within certain time steps , running the engine on full load for two hours so that the templug employed may be exposed to maximum temperature after two hours gradually decreasing the load by the same time steps employed earlier to zero load .the test cycle has its own significance because the accuracy of the results is depended upon the specific test cycle

**Table 3** variation of load with time

| Step | Time (min) | Speed | Load |
|---|---|---|---|
| 1 | 5 | 100 % | 10 % |
| 2 | 10 | 100 % | 20 % |
| 3 | 5 | 100 % | 40 % |
| 4 | 90 | 100 % | 97 % |
| 5 | 2 | 100 % | 10 % |

### 5.4 model validation

Convergence criteria checked, residual appears to be stable, average temperatures on various faces and averaged HTC on liner jacket interface were constant for more than 3000 iterations, outlet temperature and pressure drop across cooling jacket were also comes to constant value. So it can be concluded that result was converged. But convergence doesn't mean accuracy; it needs to be justified by available experimental results

Table shows experimental and simulation temperature at various locations with error. The temperature rise across circuit was also show in table which might be reduced if can able to account heat added due to lubricating oil cooling

**Table 4** Experimental and simulated temperature at various locations with error

| Location | Region | Sim | Expt | Error % |
|---|---|---|---|---|
| 1 | Deck Face(between. Exhaust valves) | 322 | 357 | 12 |
| 2 | Between Inlet & Exhaust Valves | 263 | 263 | 0 |
| 3 | Between exhaust & Injector | 280 | 252 | 11 |

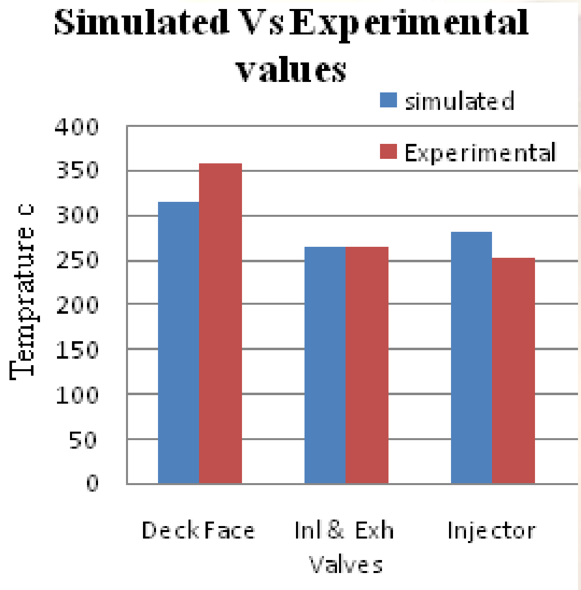



**Figure** 20 comparison of Experimental and simulated temperature at various locations.

Experimental and simulated temperatures at various locations on cylinder head, exhaust pore and liner have been compared in above graph with the percentage of errors induced in calculation. the temperature raise across circuit which might reduced if it can able to account heat added due to lubricating oil cooling, further the simulation work in present case is done by assuming model of constant density i.e. the density of bulk gas temperature is given at its mean temperature without considering the variation at respective crank angle, which in turn can also reduce the errors ,another reason for temperature variation is the roughness factor considered for cooling jacket the induces the turbulence within the flow , and governs the heat carrying capacity of the coolant , the exact measurement of roughness factor of cooling jacket of present engine was not possible hence the value is taken as 1 mm based on literature study and experience from company professionals. The calculated value of heat transfer coefficient which is used in initial condition for determining the temperature distribution may vary, the variation can be explained by the choice of significant parameters and assumptions while evolving the empirical correlations

## VII. Optimization of cooling jacket

Based on the results of the computational fluid dynamic predictions and calculating the Static pressures and flow visualization results, a redesign of the cylinder head cooling jacket occurred. The purpose of the cooling jacket redesign was to provide for a uniform positive pressure gradient from the inlet of coolant to outlet and to minimize the axial pressure gradients at various locations in cylinder head. The stagnancy in various critical zones of cooling jacket lead to modification in cooling passage of block and head cooling passage by providing extra blockages i.e. diverting the flow and adding some additional passages in the jacket.
The CFD model was modified by selectively removing and adding elements to the baseline model which was further analyzed for its results

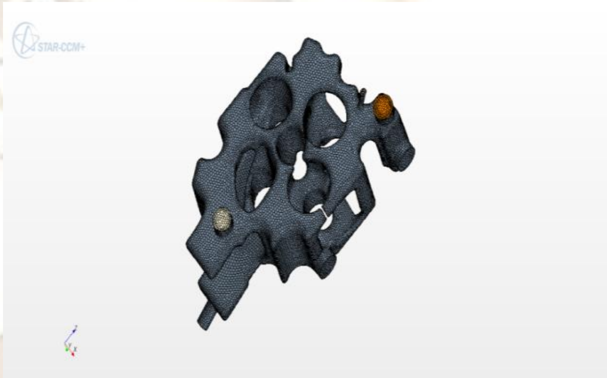


**Figure 21** Cooling jacket in Modified model

### 6.1 Flow and thermal simulations results after modification:

**1.Flow across head bottom section**

The modifications in the head and block cooling jacket lead to various changes in pressure and velocity schemes within the cooling jacket and are discussed in the following sections.

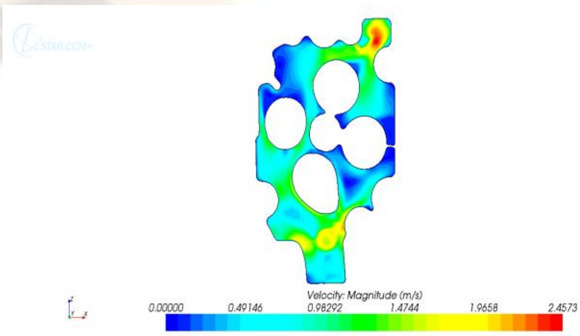



**Figure 22** Velocity contour at head bottom section for Modified model

Before modification Stagnancy of flow is observed at the valve bridges were velocity is approx. 0.1 m/s Restriction of flow across the inlet manifold forces the coolant to flow through exhaust valve bridges is observed. Thus the flow of coolant leads to uniform cooling and can decrease the value of maximum temperature across the exhaust valves. Pressure drop across the cooling jacket is obtained to be 0.04943 bar, before modification it was 0.04738 bar .Thus improved flow rate across liners increases the pressure drop across cooling jacket. The modified model thus shows stagnancy at the restricted area, thus it can be compensated as the temperature in the stagnant zones is not higher and the chances of boiling in such stagnant zone are minimum.

**2. Temprature across deck face**

Deck face is the area of maximum temperature in cylinder head, where nucleate boiling of coolant can occur and totally depends on its maximum temperature. Area adjacent to Exhaust port bridge and fuel injector shows maximum value of temperature i.e. Valve bridge area The value of maximum temperature has been decreased by 6 K , thus cooling rate has been improved due to the modification in cooling jacket.

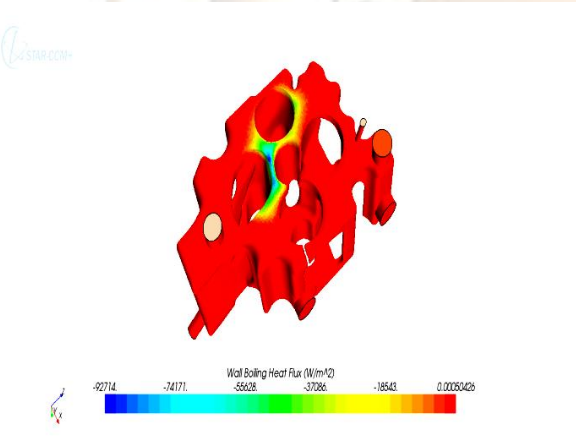



**Figure 23** Temperature contour at Deck face for Modified model

**3. Exhaust port**

The design for this cylinder head forced the flow to go around the exhaust port first, where most of the heat was output, then cooled down the fuel injector and finally the intake port. It is seen that the temperature around the exhaust port was higher than other locations, which ensures that the cylinder head should be relatively evenly cooled at these locations. The maximum temperature reached in the exhaust port is about 690 K

The maximum temperature has been reduced by 7 k as the coolant was forced to flow through the exhaust port area

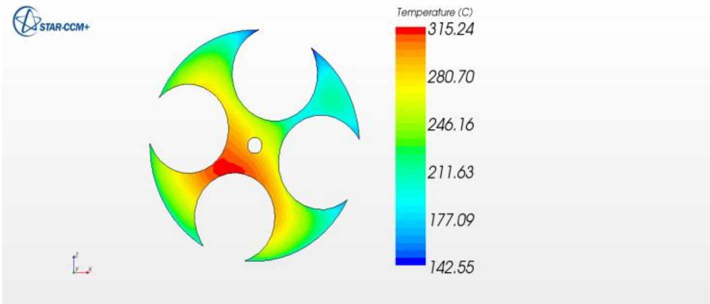



**Figure 24** Temperature contour at Exhaust port for Modified model

Thus after modification the temperature distribution across the deck faces and exhaust port is studied because the maximum temperature in cylinder head has reached in these areas and the results shows an effective temperature drop in these regions..

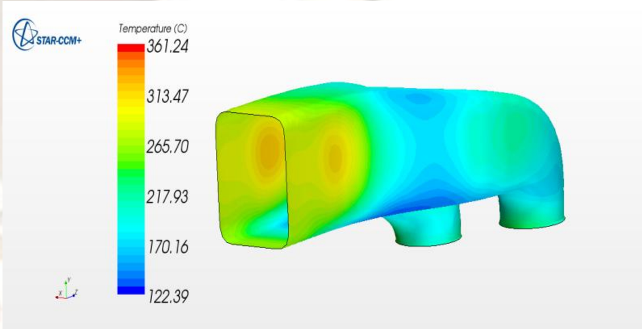



**Figure 25** Wall boiling heat flux on cooling jacket

The initialization of bubble formation can also be reduced which in turns reduces the chances of nucleate boiling and this may increase the cooling capacity of the cooling jacket and avoid the hot spot that causes nucleate boiling or thermal cracking of critical zones within the engine

**Table 6.4** Comparison of parameters before & after modification at various locations

| Parameter | Before modification | After modification |
|---|---|---|
| Pressure in jacket (bar) | 0.04738 | 0.04943 |
| Temperature on deck face(K) | 322 | 315 |
| Temperature on exhaust (K)port | 370 | 362 |

The CFD analysis of modified cooling jacket is carried in order to study the effect of variation of geometry on flow and thermal parameters of engine .From above analysis it can be conclude that model after modification has advantage of improved velocity in head jacket at exhaust port due to source of flow directly coming from sideways instead of flowing from inlet port which has less temperature compared to previous one. The pressure drop is increased by very small value which does not have significant variation on other parameters; the increased pressure drop is mainly because of the resistance offered to flow by restrictions in cooling jacket. Hot spot temperature is reduced after modification mainly because the coolant is forced to flow through exhaust side. The possibilities of vapor formation in cylinder jacket after modification is reduced to certain extent due to temperature drop at hot spots this further avoids chances of nucleate boiling of coolant at these spots.

Finally it can summarized from above discussion that flow path of coolant across jacket significantly affects the heat transfer analysis and maximum temperature value of engine components

## VIII. CONCLUSION

An heat transfer model is developed is developed for modeling thermal behavior of diesel engine cylinder heat transfer which illustrates effects of various parameters affecting the heat transfer .it also deals with study of temperature distribution in various bodies of diesel engines. In general the work of this cylinder head water jacket satisfied the demands of cooling effects as there is a good flow rate distribution and even pressure rate distribution, the flow field and heat field of engine water jacket where computed based on boundary conditions values obtained from heat transfer model developed .the simulated values obtained are further tested and verified by experimentation. Based on the above work the following points can be concluded.

In case of flow analysis the enhancement in flow rate will strengthen the cooling effect to certain extent but in mean while bring higher pressure loss of flow, further it is can be stated that higher velocity leads to higher heat transfer coefficient.

Steady state heat transfer analysis of cylinder head and liner of engine was done using 3D CFD software &. Model was verified using measured temperatures. The computed values correspond to the experimental data. The results of computation confirmed the necessity of incorporation of possible occurrence of local boiling and associated steep changes in values of heat-transfer fluxes.

Possible zones of nucleate boiling in head and block jacket are being determined; where maximum wall boiling heat flux is obtained at exhaust Valve Bridge and top land of liners. Nucleate boiling shows heat flux magnitude up to 900,000. But in present case there does not exist area showing flux magnitude greater than 900,000 so possibility of film boiling was not present.

The modification in cooling jacket in head and block provided necessary forces to enhance the cooling in problematic zones so that intake, exhaust valves and fuel injector will not be too hot to break down. The simulation results show that temperature values have been dropped by 7-10 K in the critical zones.

## REFRENCES

[1] Marcel Davis, Radek Tichanek, Mirsolav spaniel, **"**Study of heat transfer analysis of diesel engine head" Acta Polytechnica Vol. 43 No. 5/2003 [2001]

[2] M.Fadaei, H.Vafadar, A.Noorpoor, "Thermo-mechanical analysis of cylinder head using multifield approach" Scientia Iranica B (2011) 18 (1), 66–74 [2010]

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

[4] Quingzhao Wang, "Numerical analysis of cooling effect of a cylinder head water jacket "Thesis of GIT" [2005]

[5] Mirsolav Spaniel, Radek Tichanek, "Heat Transfer Investigations in a Modern Diesel Engine" Acta Polytechnica Vol. 43 No. 5/2003 [1998]

[6] E.urip, S.L yang. Arici, "Conjugate Heat Transfer for Internal Combustion Engine Application Using KIVA code"[1998]

[7] R.T Green, J.Jambunathan, "Heat transfer through diesel engine cylinder liner "Applied energy 14 (1983) 175-196 [1999]

[8] John B Heywood, "Thermal loading and temperature measurement in diesel engine component "Fundamentals of I.C engines [1988]

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

[10] Toyoshige Shibata, Hideo Matsui, Masao Tsubouchi, "Evaluation of CFD tools applied to engines coolant flow analysis through cylinder head"Mitshibushi motors,Technical review no.16 [2004]

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

[12] Zhang Ping and etal "Numerical simulation of coolant flow in diesel engine cooling water jacket**"**CAU,Beiging ,100083 [2008].

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

[14] Hai-bo Chen et al," Development of numerical simulation on flow and heat transfer in ic engine cooling application, Chinese journal of mechanical engineering No.10 [2008]
**[15]** H.zao and N.collings, "The cylinder head temperature measurement by thermal imaging technique" SAE Technical paper no.912404 [1991]
[16] Templug user information guide, Revision 3.0,[ April 2004]
[17] "Engines web pages", Mechanical engineering department colorado state university, Fort collins. 2006
[18] Hee Jun Park "Development of an in-cylinder heat transfer model with variable density effects on thermal boundary layers" Doctoral Thesis, Mechanical Engineering university of the Michigan, 2009
[19] Rakopoulos C.D., Rakopoulos D.C., Mavropoulos G.C., Giakoumis E.G. "Experimental and theoretical study of the short term response temperature transients in the cylinder walls a diesel engine at various operating conditions" Applied thermal engineering 24 (2004) 679-702
[20] Ghojel Jamil, Honnery Damon "Heat release model for the combustion of diesel oil emulsions in DI diesel engines" Applied thermal engineering 25 (2005) 2072-2085
[21] Tutorials on CFD by 'CD Adapco'
[22] Thermocouple theory 'Pyromation'.