

Theoretical Consideration to Improve Engine Cooling and Application of Coupling 3D Combustion Simulations with Heat Transfer in Water Jacket and Components

Jun XIN*
Yoshio MAEDA**

Stephen SHIH*
Michio KAWAMOTO**

Edwin ITANO*
Yoshio YAMAMOTO*

ABSTRACT

要 旨

Improving engine cycle thermal efficiency is an effective way to increase engine torque and to reduce fuel consumption simultaneously. However, the extent of the improvement is limited by engine knock, which is more evident at low engine speeds when combustion flame propagation is relatively slow. In this paper, computational fluid dynamics (CFD) tools were applied to evaluate the potential for improvement of V6 engine, and effectiveness of various design concepts of engine water jacket by judging surface heat transfer coefficient under isothermal condition. Theoretical considerations were given to optimize the cooling efficiency. Consequentially, engine performance was carried out in base engine and a prototype engine, improvement of engine output and fuel economy was confirmed. Furthermore, a new CFD methodology was developed in which iterative coupling approach was taken to include combustion chamber, water jacket and solid components. Component temperatures were predicted explicitly and compared with measurement results.

トルク増大と同時に燃料消費率を向上するには図示熱効率向上が有効である。しかしながら、その量は特に火炎伝播が相対的に遅い低速領域においてノッキングにより制限されてしまう。本論文では計算流体力学(CFD)を V6 エンジンに適用し、熱効率向上の可能性について検討を行った。特にいくつかのウォータージャケットの設計コンセプトについて等水温下でのウォータージャケット表面の熱伝達係数を計算してその有効性の評価を行った。冷却効率を最適化すべく理論的検討を加えた結果、ベースエンジンに比較して本検討による試作エンジンは出力、燃料消費率共に向上することが確認された。本検討中に用いた新しい CFD の手法は、混合気の燃焼ならびに燃焼ガスの筒内流動による熱伝達への影響をシミュレート可能とするため、燃焼室やウォータージャケット形状のソリッドモデルと融合されたものであり、その有効性を各部温度の予測と実測の比較により検証を行った。

1. Introduction

As the US light duty truck market (SUV, SUT, etc.) continues to grow, it is of great importance to increase low engine speed torque for better vehicle dynamic performance. Improving engine gas exchange process is a critical area to accomplish the goal. Higher volumetric efficiency usually demands anti-knock measures to take full advantages of improved gas exchange. At the same time, it is imperative to reduce vehicle fuel consumption to curb global warming. Improving engine cycle thermal efficiency is an effective way to achieve higher engine output and lower fuel consumption simultaneously. However, the extent of the improvement is limited by engine knock. Improving the overall level of knock resistance and uniformity of all cylinders of a multi-

cylinder engine is necessary to increase thermal efficiency.

In the next section, theoretical considerations were presented to provide uniform and effective cooling for multi-cylinder engines. Several design concepts were proposed, and their effects on cooling performance were analyzed, and design parameters were assessed and optimized by means of CFD simulations. After design concepts were optimized with respect to heat transfer coefficient from isothermal CFD simulations, one design was tested, and main test results were presented.

Then, in order to quantitatively evaluate effects of design concepts on combustion chamber wall temperature, and subsequently on volumetric efficiency and combustion, a newly developed methodology was described that couples in-cylinder flow and combustion

* Honda R&D Americas, Inc.

** Honda R&D Co., Ltd.

with convection and conduction in water jacket and solid components such as cylinder head and block. Procedures and calculation results from the applications were depicted. The major advantage over the isothermal water jacket analysis is that interactions among combustion, component temperatures and coolant flow pattern of water jacket can be simulated directly.

2. Basic Considerations and Design Concepts of Engine Water Jacket

To address constantly increase of engine output density and environmental concerns, water jacket of a multi-cylinder spark ignition engine needs to meet four functional requirements. First of all, water jacket provides adequate cooling capability to engine components in order to keep component temperatures below maximum levels. Secondly, it helps reduce temperature difference among cylinders so that all cylinders operate at optimal conditions. Thirdly, it is desired to arrange flow pattern so that temperature gradient in both circumferential and cylinder axial directions is reduced. At last, water jacket has low thermal capacity so that engine warm-up can be shortened.

Nowadays, three-dimensional CFD calculations have been routinely applied in engine water jacket design and optimization^{(1), (2)}. The addition of conjugate heat transfer capability in most commercial CFD packages enables designers to evaluate water jacket and component temperature together. For the purpose of analyzing cooling efficiency and uniformity of water jacket among individual cylinders, isothermal coolant flow analysis is adequate to provide reliable indication with respect to relative cooling differences among cylinders as well as among water jacket designs. Therefore, in this paper, isothermal calculations were applied to quickly evaluate coolant flow pattern and optimize design parameters. After an optimized design was identified, coupling analysis followed so that wall temperatures and component temperatures could be determined to judge the benefit on knock toughness and durability concern.

In the coupling analysis, in-cylinder flow and combustion calculations were coupled with flow and heat transfer calculations in water jacket, as well as heat conduction in components, which was described in details in the next section.

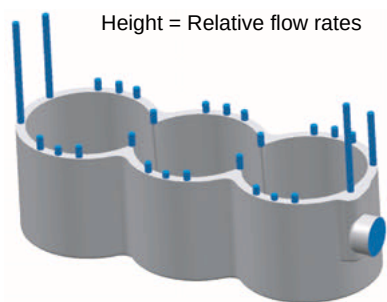


Fig. 1 Flow rate distribution through gasket holes of base design

A general CFD package, STAR-CD was used in isothermal fluid calculations. For isothermal flow analysis, only governing equations of continuity, momentum and $k-\epsilon$ turbulence model were solved. For simplicity, one bank of a V6 engine water jacket was modeled. A flow rate imposed on the inlet boundary was corresponding to engine operating condition of 3 000 rpm. Fig. 1 illustrates coolant mass flow rates through each gasket hole of a baseline V6 engine. The height of bars represents normalized flow rate through individual gasket holes. In the baseline, coolant flows through engine block and cylinder head in a parallel fashion. Due to that design concept, it is necessary to have two large gasket holes on the opposite end from the water pump inlet to maintain certain level of coolant velocity in the top portion of liners. It is apparent that most coolant enters head from front and rear end holes. In the base engine design, about 35% coolant flow enters the cylinder head from the two rear gasket holes. This 35% flow is for block cooling only, and not utilized for cylinder head cooling. The combustion chamber of the first cylinder gets about 30% coolant flow rate.

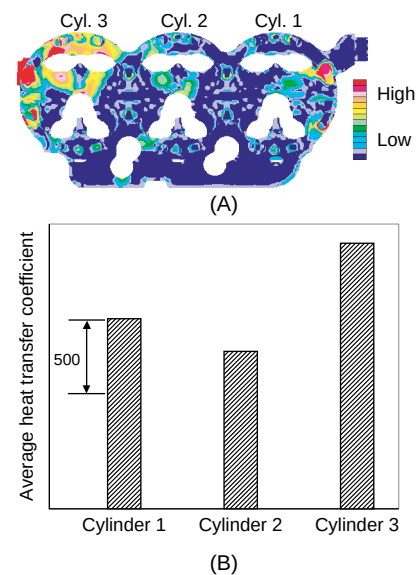


Fig. 2 Distribution of heat transfer coefficient

Heat transfer coefficient (HTC) distribution on the water jacket surface is shown in Fig. 2A, heat transfer coefficient is a good indicator to evaluate local heat removal capability. Ideally, each cylinder in a multi-cylinder engine has similar cooling capability, and heat removal capability matches the thermal loading on the combustion chamber from the burnt gas. Fig. 2A indicates that the base design does not achieve relatively spatially uniform HTC distribution, the middle cylinder has the lowest cooling, and the third cylinder has the highest HTC, because coolant flow rate is the highest there as flows through gasket holes merge. The CFD results indicate that there is some room for even better performance, and such improvement will be necessary whenever there is substantial increase of engine volumetric efficiency and/or compression ratio.

Fig. 2B presents area-average HTC value on the head water jacket surface of each cylinder. The HTC of the middle cylinder is 25% lower than the cylinder 3. From the CFD simulation results of the baseline V6 engine, it is evident that both fuel economy and low speed torque increase can be achieved simultaneously by improving uniformity of engine cooling capability among individual cylinders and increasing overall cooling capability to meet the demand for higher low speed torque density. It should be mentioned that the baseline V6 engine's water jacket was adequately designed to provide necessary cooling capacity to maintain component temperatures and knock toughness according to the strict design standard, and was confirmed by engine and vehicle tests. Besides, the number of parts, manufacturability and commonality of parts should be considered to provide the best value to the customer. For instance, in the baseline engine design, the head gaskets for the front and rear banks are the same.

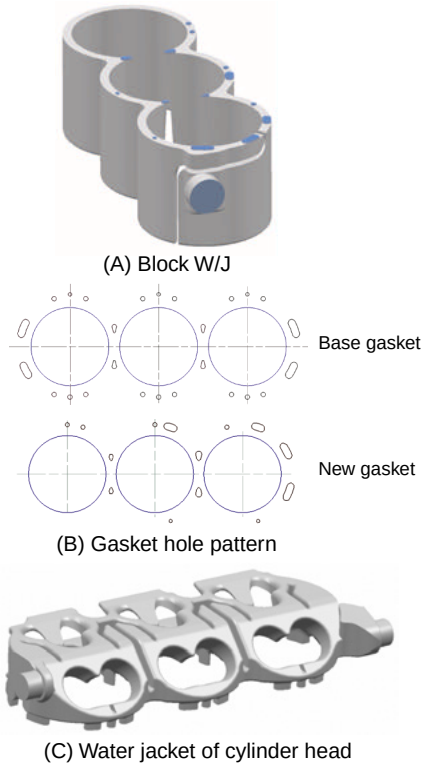


Fig. 3 Modifications to engine components for an optimized water jacket design

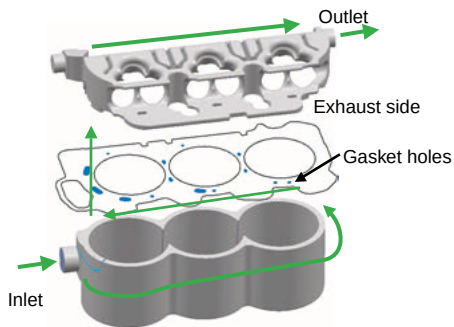


Fig. 4 Overall coolant flow path in the new water jacket

In the new design, a serial cooling concept was proposed⁽³⁾. The first measure was taken to apply a block partition so that most of coolant flow was to flow through entire cylinder block from the exhaust side to the intake side before entering water jacket in cylinder head, as shown in Fig. 3A. To accommodate this design idea, the hole pattern of head gasket was required to change, as illustrated in Fig. 3B, two holes at the rear end in the base V6 engine gasket were eliminated. Because coolant flow rate in the engine block and head was expected to increase significantly, and top liner regions were expected to be properly cooled. The total area of gasket holes on the intake side was reduced to ensure more coolant to flow through all combustion domes in the cylinder head. To further increase HTC level on the combustion domes, ribs were added to the top surface of water jacket in the new cylinder head as illustrated in Fig. 3C.

Fig. 4 illustrates coolant flow path in a water jacket after adopting features described in Fig. 3. The comparison of HTC distribution of the base and new cylinder head water jackets is given in Fig. 5A, the shown surface is viewed from the combustion chamber. It is obvious that the HTC of new design is much higher everywhere comparing to the base engine design, the spatial HTC distribution becomes more uniform. Area-averaged HTC values for individual cylinders are shown in Fig. 5B, the HTC of the middle cylinder increases by almost 75%. With respect to the trend, the HTC values gradually increase from the upper stream to the down stream cylinders as coolant temperature rises. The HTC distribution of engine block is compared in Fig. 5C between the base V6 engine and a new V6 engine. Due to high coolant flow rate flowing through the entire

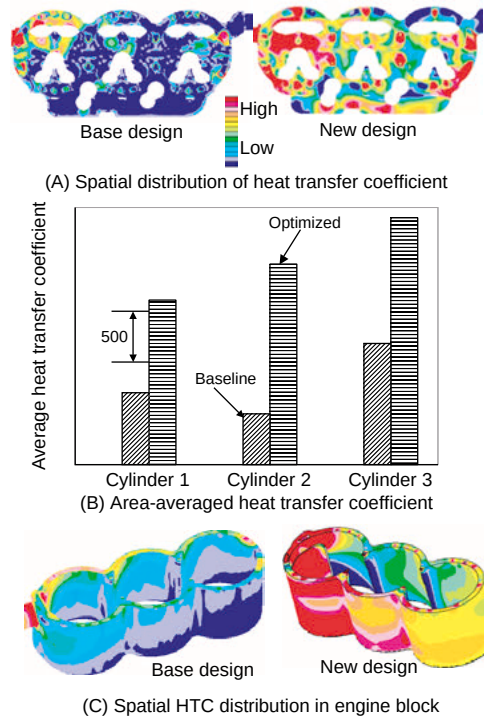


Fig. 5 Comparison of spatial HTC between the baseline and an optimized design

block portion of the new water jacket, HTC values on the top liner region are also increased considerably, as shown in Fig. 5C.

Then, the base and new V6 engines were tested with regular gasoline to evaluate effects of the above design changes on engine power and fuel economy. The knock limited ignition timing vs. volumetric efficiency is shown in Fig. 6. Each line represents knock characteristics of one cylinder. It is apparent that there is a wide variation of knock limited ignition timings among 6 cylinders. For 4 000 rpm, it is almost 6.5 degrees of crank angle. With the new water jacket design, the scatter of knock limited ignition timings becomes narrower in an engine speed range from 3 000 rpm to 5 000 rpm, as shown in Fig. 6, due to improved cooling efficiency uniformity of individual cylinders. Furthermore, it is also evident that lower limit of ignition timing of the 6 cylinders is raised within this speed range, and is advanced by 2.5, 3.5, and 3.5 degrees, respectively for 3 000 rpm, 4 000 rpm and 5 000 rpm. However, the upper limit of ignition timing is not advanced as much as the lower limit. Other test results are presented in a separate paper⁽³⁾.

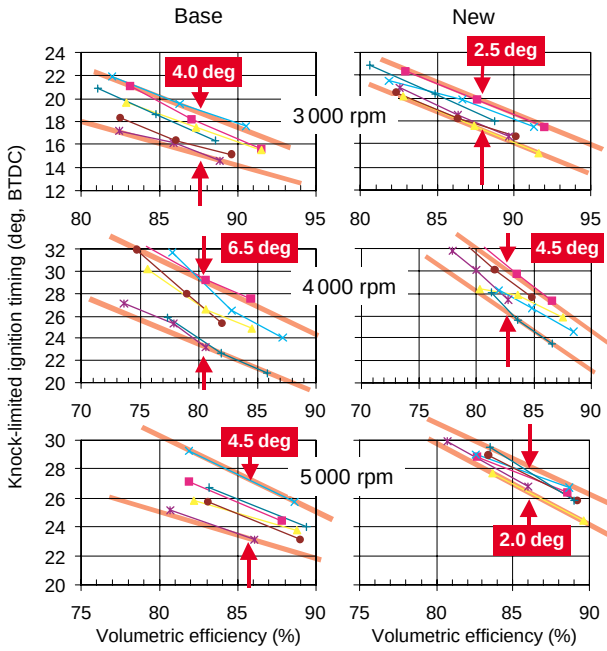


Fig. 6 Effects of water jacket design on knock limited ignition timings of individual cylinders

3. Combustion Simulations Coupling with Water Jacket

In the preceding section, isothermal CFD simulations were performed to optimize water jacket design with respect to cooling efficiency and uniformity of heat transfer coefficient on surfaces. However, isothermal fluid CFD simulations predict neither effects of water jacket on component temperature nor its effects on in-cylinder flow and combustion. Such quantitative information is crucial to predict part durability and knock

toughness. Therefore, a simulation methodology was developed to couple in-cylinder flow and combustion with coolant flow and heat conduction in components.

3.1. Description of In-cylinder Flow and Combustion Modeling

The modified KIVA-3V code⁽⁴⁾ was used to simulate full cycle of 720 crank angle degrees for a 4-stroke engine which includes intake, compression, ignition and combustion, and exhaust processes. In order to obtain certain level of prediction accuracy of component temperatures, realistic heat flux boundary condition is essential. Whether flame propagation and combustion rate can be predicted reasonably is a fundamental requirement. Enormous efforts to develop more accurate sub-models for physical and chemical phenomena have been made to make 3D simulations predictive, comprehensive reviews can be found in literature⁽⁵⁾. To be complete, some critical models are described here.

3.1.1. Ignition Model

Implemented ignition model was similar to DPIK (Discrete Particle Ignition Kernel) model proposed by Fan and Reitz⁽⁶⁾ with some small modifications to accommodate applications to sequential ignition process⁽⁷⁾. In DPIK ignition model, flame kernel is represented by mass-less particles. The locations of these particles define the outer surface of flame kernels, the number of particles within a computational grid cell represents flame surface area which is related to reaction rates. The particle velocity components are calculated as:

$$u_{p,i} = \left[\frac{T_{ad,i}}{T_{l,i}} S_{L,i} + \sqrt{\frac{2k_i}{3}} \right] \cos(\theta) \cos(\phi) \quad (1)$$

$$v_{p,i} = \left[\frac{T_{ad,i}}{T_{l,i}} S_{L,i} + \sqrt{\frac{2k_i}{3}} \right] \cos(\theta) \sin(\phi) \quad (2)$$

$$w_{p,i} = \left[\frac{T_{ad,i}}{T_{l,i}} S_{L,i} + \sqrt{\frac{2k_i}{3}} \right] \sin(\theta) \quad (3)$$

where, $T_{ad,i}$ is the adiabatic flame temperature, and subscript 'i' means *i*th spark plug. For instance, $T_{l,i}$ is the local gas temperature at ignition timing corresponding to spark plug 'i' for multiple spark plug applications.

$$T_{l,i} = T_{k1,i} \left(\frac{P_{k2,i}}{P_{k1,i}} \right)^{1-\gamma} \quad (4)$$

where $T_{k1,i}$ and $P_{k1,i}$ are the temperature and pressure of associated ignition cell at corresponding ignition timing, $P_{k2,i}$ is instantaneous pressure inside a kernel region.

The parameter k is the turbulence kinetic energy, and S_L is laminar flame speed which is calculated as follows

$$S_{L,i} = [s_1 - s_2(\Phi_i - s_3)^2] \left(\frac{T_{l,i}}{298} \right)^{\alpha_i - \alpha_s(\Phi_i - 1)} \times \left(\frac{P_{k2,i}}{1.013 \times 10^5} \right)^{-\beta_i + \beta_s(\Phi_i - 1)} (1 - 2.1R_i) \quad (5)$$

The instantaneous flame kernel diameter is calculated as:

$$D_{\text{kernel},i} = \frac{2}{N_i} \sum_{n=1}^{N_i} |\vec{r}_{n,i} - \vec{r}_{p,i}| \quad (6)$$

where, N_i is the number of active flame kernel particles (particles that collide with walls are eliminated from the count), $\vec{r}_{p,i}$ is the center location of an initial ignition kernel, and $\vec{r}_{n,i}$ is the location of a discrete kernel particle. If a kernel diameter reaches the order of integral length scale, chemical reactions in cells swept by kernel particles from the corresponding spark plug are modeled with the characteristic time scale combustion model, while other kernel is still growing, and kernel growth and reaction rates in a volume under the influence of that spark plug are still modeled with DPIK model.

The criterion to switch from DPIK ignition model to normal combustion model is kernel diameter, based on

$$D_{\text{kernel},i} \geq C_k \bar{l}_i \quad (7)$$

the parameter $\bar{l}_i$ is averaged turbulence length scale which is defined as

$$\bar{l}_i = 0.16 \left[\sum_{j=1}^{ncell} k_j / ncell \right]^{1.5} / (\sum_j \varepsilon_j / ncell) \quad (8)$$

3.1.2. Combustion Model

The characteristic-time scale combustion model⁽⁸⁾ is used, and reaction rates are calculated by

$$\frac{\partial \rho_s}{\partial t} = -\frac{\rho_s - \rho_s^*}{\tau_c} \quad (9)$$

where ρ_s is the density of species s , and ρ_s^* is the local and instantaneous thermodynamic equilibrium value of the density of species s . The characteristic conversion time is considered to be a combination of a turbulent-mixing time and local laminar-kinetics time, that is:

$$\tau_c = \tau_t + \tau_l \quad (10)$$

Here is assumed that time scale for all species is identical, the effect of different time scales on species is reported in the early publication⁽⁹⁾. The turbulent mixing time scale is determined with

If

$$\frac{c_{m2}(Y_p - Y_p^*)}{c_{m3}(Y_F - Y_F^* + Y_{O2} - Y_{O2}^*)} \geq 1 \quad (11)$$

or if

$$\frac{c_{m2}(Y_p - Y_p^*)}{c_{m3}(Y_F - Y_F^* + Y_{O2} - Y_{O2}^*)} < 1$$

$$\tau_t = c_{m3} \frac{k}{\mathcal{E}} \cdot \frac{(Y_F - Y_F^* + Y_{O2} - Y_{O2}^*)}{(Y_n - Y_n^*)} \quad (12)$$

where $c_{m3} = 0.09$, and $c_{m2} = 0.6c_{m3}$

Laminar flame time scale is determined by the following⁽¹⁰⁾,

$$\tau_l = 1.916 \times 10^{-12} \cdot T \cdot \left(\frac{P_0}{P} \right)^{[1+2(-0.16+0.22(\phi-1))]} \cdot e^{(1-0.08|\phi-1.15|) \times 3 \times 10^4 / (1.987T)} / (1-2.1R)^2 \quad (13)$$

where R is residual gas fraction, and $P_0 = 1\text{atm}$.

3.1.3. Model of Wall Function

To resolve near wall transport, improved wall function in KIVA-3V is used⁽⁴⁾. The wall heat flux is determined by,

$$\dot{q} = \frac{\rho v_l c_p F(T - T_w)}{\text{Pr} y} \quad (14)$$

where

$$F = \frac{R' \cdot \text{Pr}_l / \text{Pr}_t}{1/\kappa \ln R' + B + 11.05(\text{Pr}_l / \text{Pr}_t - 1)} \quad (15)$$

if $R' > 11.05$, or $F = 1$ if $R' \leq 11.05$.

And

$$R' = \frac{c_\mu^{1/4} K^{1/2} y}{v_l} \quad (16)$$

where ν_l is laminar kinematic viscosity, y is distance from the wall, κ is Karman's constant. Pr_l and Pr_t are laminar Prandtl and turbulent Prandtl numbers, respectively. K is turbulence kinetic energy.

3.2. Cycle Averaged Heat Flux Calculations

In-cylinder phenomena and heat transfer within components are of different time scale, it has been shown that component temperature distribution is not affected by transient processes in a combustion chamber except within an extremely thin layer. It is reasonable to couple in-cylinder analysis with component and water jacket calculations on cycle-averaged basis. The schematic of the coupling process⁽⁷⁾ is shown in Fig. 7.

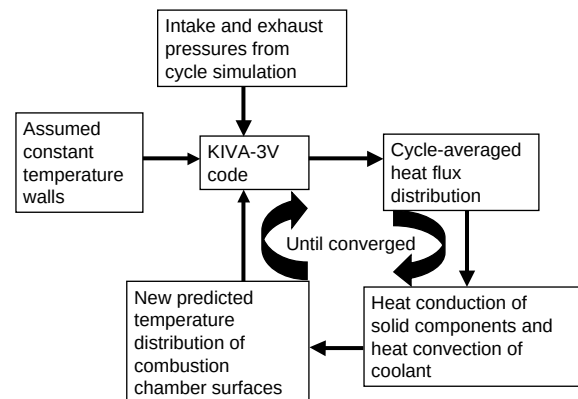


Fig. 7 Illustration of iterative coupling procedures

Cycle-averaged local heat flux is calculated as:

$$\dot{q}_{i,j} = \sum_{n=1}^{n_{720}} \frac{\dot{q}_{i,j}^n \cdot \Delta t^n}{\frac{120}{N_s}} \quad (17)$$

where N_e is engine speed in rev/min, and the subscripts i and j are indexes of cells on 3D surface domain. Heat flux on liner attributes to three sources. One is directly from in-cylinder gas on the portion of liner surface that is exposed to gas at any instant. The second is gaseous heat transferred from piston through rings to the liner. And the last is from friction-converted heat from rings and skirt. In the current investigation, the last one was omitted. At the very bottom portion, the liner could either receive heat from or dissipate heat to crankcase flow. In this study, the interest is on the top liner portion, and it is believed that the bottom portion does not have meaningful impact on the top portion with respect to temperature distribution. Therefore, a guessed value is used for that region. As to heat flux on cylinder liner portion that is swept by the piston ring, it is calculated by the following formulations:

$$\dot{q}_{i,j,liner} = (\dot{q}_{i,j,liner})_{gas}, z \leq h \quad (18)$$

$$\dot{q}_{i,j,liner} = (\dot{q}_{i,j,liner})_{gas} + \zeta \dot{Q}_p / (\pi DS), z > h \quad (19)$$

where $\dot{Q}_p$ is the total heat flux of piston top surface from gas, and ζ is the percentage of piston heat flux transferred to the liner through piston rings, in this study it is assumed 70% of heat that piston receives is transferred to the liner. The meanings of other symbols are explained in Fig. 8.

Heat flux on valve seats is calculated based on the heat balance of valves. Cycle-averaged heat flux on valve heads is distributed into three: part of it goes through valve seats, and some goes through valve stems and some to gas. The computational domain for the in-cylinder flow and combustion calculations is shown in Fig. 9, time-dependent pressure conditions from one dimensional simulations are applied to inlet and outlet.

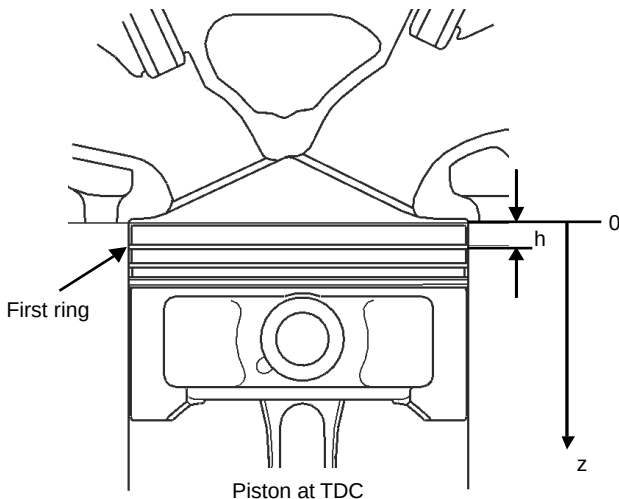


Fig. 8 Zone definitions on cylinder liner surface to calculate heat flux

3.3. Solid Components and Cooling Jacket

Computational domain for heat conduction analysis consists of cylinder head, head gasket, cylinder block and cylinder liners. For the applications in this work,

only one bank of a V6 engine is considered assuming symmetry of two banks. The CFD domain for heat convection in cooling passage is also a one-bank water jacket, the solid and water jacket domains are shown in Fig. 9. A general computational fluid dynamics code VECTIS was used for solid components and water jacket, time-step resolved coupling for both liquid/solid and solid/solid interfaces is available in the package. Standard k-e turbulence model is employed for water jacket flow calculation, phase change in water jacket is not considered, which could result in higher predicted temperatures than actual in some regions where phase change is expected to be important.

Valve seats are not directly modeled in this investigation, instead that heat flux from the valve heads is imposed on the head surfaces which valve seats are exposed to.

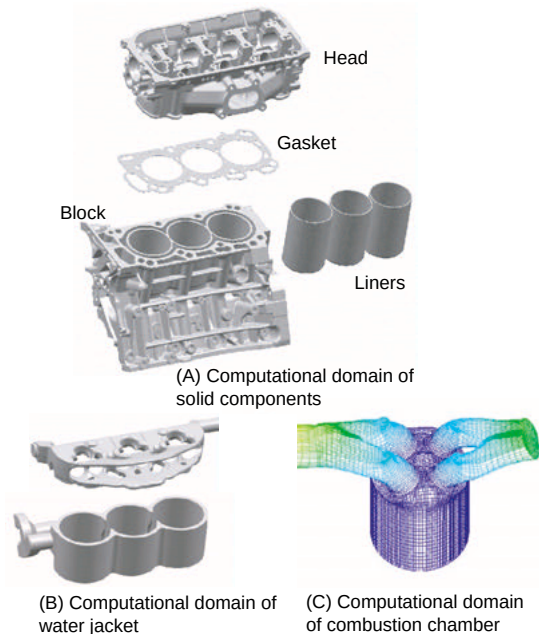


Fig. 9 Computational domain for conjugate heat transfer analysis

3.4. Applications of the Methodology

A critical engine operating condition (6 000 rpm, equivalent A/F ratio 0.759) was chosen to illustrate the methodology and its applications. To begin with, constant wall temperatures of 410K, 460K, 470K, 415K, and 580K were applied to cylinder liner, head, piston, intake valves and exhaust valves, respectively. A full cycle of 720 crank angle degrees was simulated with KIVA code. Cycle-averaged heat flux distribution on cylinder head and liner surfaces is shown in Fig. 10. It is apparent that surface heat flux is highly non-uniform even though the initial surface temperatures were assumed to be uniform. Area between exhaust valves has the highest heat flux.

Consequently, the predicted spatial heat flux results from 3D combustion simulations were applied to conjugate heat transfer analysis with VECTIS as

illustrated in Fig. 7. Fig. 11 shows calculated surface temperature distributions on combustion chamber and liner surfaces. It should be emphasized that spatial temperature distributions are not uniform due to spatially varied heat flux from combustion chamber, non-uniform wall thickness, accessibility of coolant flow and cooling efficiency. Furthermore, the results also demonstrated large wall temperature variation among cylinders.

It is known that local heat flux on combustion chamber surface depends on instantaneous heat transfer coefficient, local gas temperature and local wall temperature. As indicated in Fig. 11, wall temperature varies both along cylinder axis and along tangential direction, which is expected to have significant effect on magnitude and direction of instantaneous local heat flux. Furthermore, wall temperature changes will affect thermodynamic properties of gaseous media and trapped mass. These variations will change heat transfer coefficient and gas temperature. Therefore, uniform wall temperature assumption in 3D combustion calculations cannot produce satisfactory heat flux information necessary to predict overall engine component temperature levels. With this consideration, it is necessary to iterate between combustion calculations and

conjugate heat transfer analysis until wall temperatures on combustion chamber surfaces converge. Therefore, calculated wall temperature distribution is mapped to KIVA combustion chamber model. Fig. 12 shows temperature distribution on chamber surface for one cylinder. New temperature conditions for KIVA combustion calculations were correctly mapped. Then, new in-cylinder flow calculation with the new wall temperature distribution was performed, and new heat flux results on combustion chamber surface were obtained. Typically, the results are converged after 3 iterations⁽⁷⁾.

To validate the robustness of the tool and the accuracy, the developed tool was applied to a new V6 engine. Two different water jacket design concepts named D1 and D2, respectively are evaluated with the proposed CAE tool. The design D1 is the baseline design. In the design D2, a vertical insert is placed in the water jacket of the engine block, and head gasket is modified accordingly to achieve a desired coolant flow pattern. Overall flow path of these two design concepts is illustrated in Fig. 13.

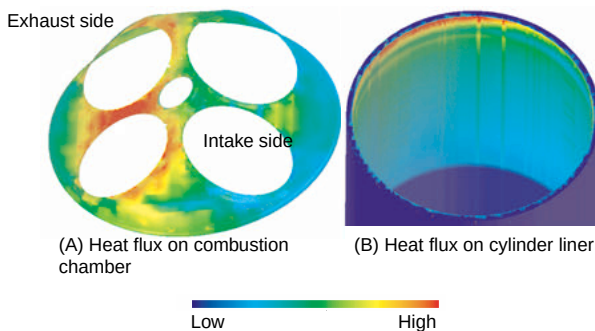


Fig. 10 Calculated spatial heat flux mapped on corresponding component surfaces

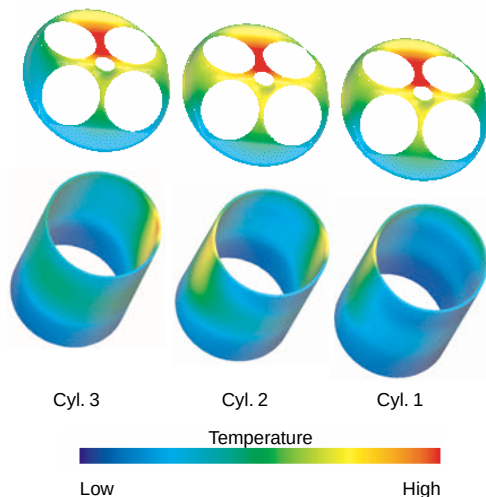


Fig. 11 Calculated spatial temperature distribution

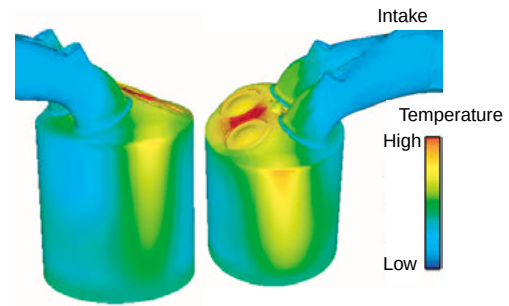


Fig. 12 Mapped temperature distribution on combustion chamber

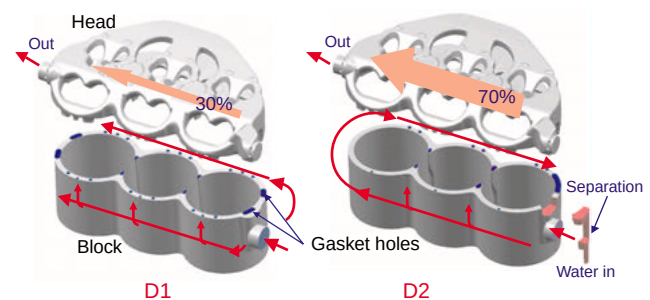


Fig. 13 Coolant flow path of water jacket designs

The comparison of measured and calculated component temperature difference between D1 and D2 designs is shown in Fig. 14 at engine speed of 6 000 rpm. The results show reasonable agreement between experimental and numerical results, and good correlation is confirmed even though there is discrepancy in terms of the magnitude of temperature reduction. Unlike at other three locations, the

calculate temperature reduction at L3 is lower than the measured. It is not clear what specific factors result in this discrepancy. Due to the fact that no experimental data was used to set up the model, such reasonable agreement can be achieved with the methodology, critical information can be obtained. It makes the methodology a valuable tool to new engine development. Fig. 15 shows the measured temperature at the location L1 across the entire engine operating speed range. Again, as predicted, the design concept D2 has lower temperature values than the design concept D1.

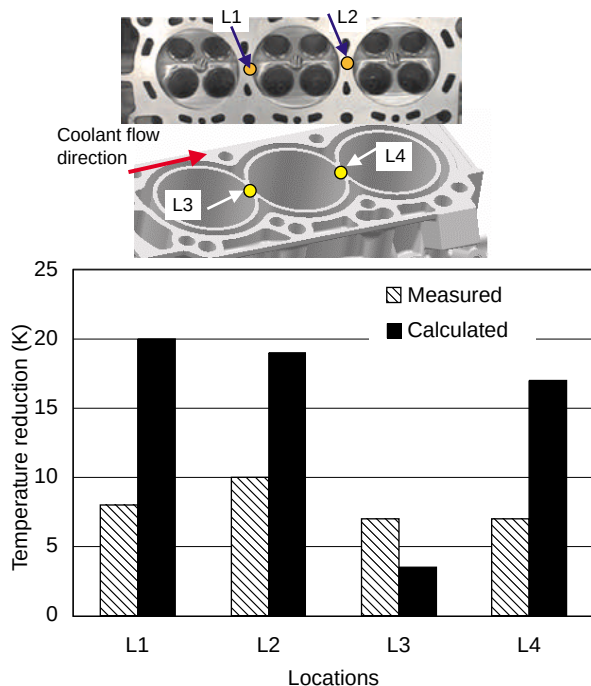


Fig. 14 Comparison of measured and calculated temperature difference

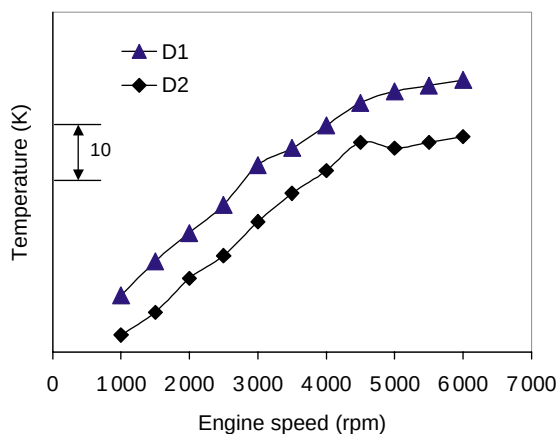


Fig. 15 Measured temperatures at the location L4 for D1 and D2 designs

4. Conclusions

In this paper, a Honda V6 engine was analyzed by using three-dimensional CFD calculations to evaluate the potential for improvement without significantly altering the current mass production process in order to improve engine power and fuel economy simultaneously. In the first part, isothermal CFD simulations were performed to effectively evaluate and optimize design parameters of the engine water jacket with respect to heat transfer coefficient on water jacket surface. In the second part, the methodology of coupling in-cylinder flow and combustion simulations with heat transfer calculations in solid components and cooling passage was developed to account for interactions among them, and therefore, to provide more complete information to evaluate design concepts in the early stage of engine development. The conclusions can be summarized as follows:

- (1) For the purpose of evaluating effects of design changes of engine water jackets on cooling efficiency and uniformity, isothermal 3D flow analysis is effective and adequate. Design optimization can be achieved within relative short period.
- (2) In the new optimized design, the cooling uniformity of water jacket among 6 cylinders was significantly improved, and the overall level of cooling efficiency in both cylinder head and block was increased.
- (3) To quantify improvement of knock toughness and to address durability concerns, a coupling methodology was developed to predict combustion chamber temperatures. For this purpose, a coupling methodology was developed.
- (4) The predictions from the coupling methodology were validated by the measurement results. There is good correlation between predicted temperature reduction and measured temperature reduction.

5. Acknowledgment

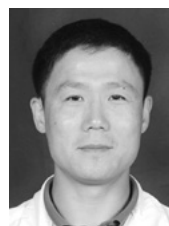
The authors would like to thank Prof. Rolf Reitz at the University of Wisconsin-Madison for productive research collaboration and his insights.

6. References

- (1) P. Norris, W. Wepfer, K. Hoag, and D. Courtine-White: Experimental and Analytical Studies of Cylinder Head Cooling, SAE 931122.
- (2) M. Moeckel: Computational Fluid Dynamic (CFD) Analysis of a Six Cylinder Diesel Engine Cooling System with Experimental Correlation, SAE 941081.
- (3) S. Shih, E. Itano, M. Kawamoto, Y. Maeda, J. Xin: Engine Knock Toughness Improvement Through Water Jacket Optimization, SAE 2003-01-3259.
- (4) A. Amsden: KIVA-3V, A Block-Structured KIVA Program for Engines with Vertical or Canted Valves, Los Alamos Report LA-13313-MS, July, 1997.
- (5) R. D. Reitz, and C. J. Rutland: Development and Testing of Diesel Engine CFD Models, *Prog. Energy Combust. Sci.* Vol. 21, 1995.

- (6) L. Fan and R. D. Reitz: 2000 Spray and Combustion Modeling in Gasoline Direct Injection Engines, ICLASS 2000 issue of *Atomization and Spray*.
- (7) J. Xin, S. Shih, E. Itano, and Y. Maeda: Integration of 3D Combustion Simulations and Conjugate Heat Transfer Analysis to Quantitatively Evaluate Component Temperatures, SAE 2003-01-3128
- (8) J. Abraham, F. V. Bracco, and R. Reitz: Comparisons of Computed and Measured Premixed Charge Engine Combustion, *Combustion and Flame*, Vol. 60, pp. 309-322, 1985.
- (9) J. Xin, D. Montgomery, Z. Han, and R. Reitz: Multidimensional Modeling of Combustion for a Six-Mode Emissions Test Cycle on a DI Diesel Engine, *J. Engineering for Gas Turbines and Power*, Vol. 119, pp. 683-691, 1997.
- (10) T. Kuo, and R. Reitz: Three Dimensional Computations of Combustion in Premixed-Charge and Direct Injected Two-Stroke Engines, SAE 920425.

■ 著 者 ■



J u n X I N



Stephen SHIH



Edwin ITANO



Yoshio MAEDA



Michio KAWAMOTO



Yoshio YAMAMOTO