

A Study of Combustion Technology for a High Compression Ratio Engine: The Influence of Combustion Chamber Wall Temperature on Knocking

Yoshihiro Imaoka, Kiyotaka Shouji, Takao Inoue, and Toru Noda
Nissan Motor Co., Ltd.

ABSTRACT

Technologies for improving the fuel economy of gasoline engines have been vigorously developed in recent years for the purpose of reducing CO₂ emissions. Increasing the compression ratio is an example of a technology for improving the thermal efficiency of gasoline engines. A significant issue of a high compression ratio engine for improving fuel economy and low-end torque is prevention of knocking under a low engine speed. Knocking is caused by autoignition of the air-fuel mixture in the cylinder and seems to be largely affected by heat transfer from the intake port and combustion chamber walls. In this study, the influence of heat transfer from the walls of each part was analyzed by the following three approaches using computational fluid dynamics (CFD) and experiments conducted with a multi-cooling engine system. First, the temperature rise of the air-fuel mixture by heat transfer from each part was analyzed. Heat transfer from the intake port and cylinder head was found to be higher than that from other parts due to the high flow velocity during the intake stroke. Therefore, lowering the temperature of the intake port and cylinder head would be effective for cooling the air-fuel mixture. Next, the influence of hot spots, produced by sharp edges and conventionally said to be a cause of knocking, was analyzed by using laser-induced phosphorescence thermography of the piston surface under a low engine speed. The results showed that the wall temperature of the piston surface immediately after combustion was 30 K higher than the average temperature, although at the time of ignition in the next cycle, the wall temperature was the same as the average temperature and hot spots at sharp edges were not observed. Therefore, sharp edges do not affect knocking at low engine speeds. Thirdly, the effect of squish on knocking was analyzed by CFD and engine experiments. It was found that the thermal boundary layer accounted for a large portion of the squish zone at top dead center. Unburned air-fuel mixture near the squish zone increased heat transfer from the squish zone by reverse squish flow, thereby reducing the temperature of the unburned air-fuel mixture near the squish zone. This insight can be incorporated in the combustion chamber design and used effectively for preventing knocking. Significant parameters for combustion design were made clear by elucidating these three mechanisms.

CITATION: Imaoka, Y., Shouji, K., Inoue, T., and Noda, T., "A Study of Combustion Technology for a High Compression Ratio Engine: The Influence of Combustion Chamber Wall Temperature on Knocking," *SAE Int. J. Engines* 9(2):2016, doi:10.4271/2016-01-0703.

INTRODUCTION

The development of technologies for improving the fuel economy of automotive gasoline engines, such as techniques for raising the compression ratio, has been actively promoted in recent years with the aim of reducing carbon dioxide (CO₂) emissions. One issue for a high compression ratio engine is that torque and fuel economy are reduced by knocking that occurs at low engine speeds. Presumably, knocking that accompanies autoignition in the cylinder of such an engine is significantly influenced by the intake port wall and combustion chamber wall temperatures. Therefore, heat transfer from wall surfaces, including the combustion chamber, was analyzed in detail using the following three approaches.

First, the rise of the in-cylinder temperature due to heat transfer from each part of the combustion chamber was analyzed using computational fluid dynamics (CFD). The study of in-cylinder wall temperature by using CFD has been carried out in a previous paper

(10). Additionally, the effect of reducing the temperature of each part on mitigating knocking was investigated experimentally using a test engine fitted with a multi-cooling system.

Second, an investigation was made of the influence of hot spots produced by sharp edges and traditionally said to have a large effect on knocking. That was done by using laser-induced phosphorescence thermography to obtain the temperature distribution on the piston crown surface, including sharp edges, in each stroke of one cycle during firing operation under low speed and high load conditions.

Third, the effect of squish on knocking was analyzed. The squish flow effect has investigated in a previous paper (9). A thermal boundary layer with a low temperature close to the wall temperature accounts for a large portion of the squish area. The effect of the low-temperature unburned air-fuel mixture in the squish zone on knocking was investigated.

TEST APPARATUS AND METHOD

The applicability of the test engine fitted with the multi-cooling system was first verified in order to conduct the three analyses mentioned above. The specifications of the test engine used in this study are given in Table 1. A 2-liter 4-cylinder direct injection naturally aspirated engine was used in the experiments. The multicylinder engine and a single-cylinder engine having the same specifications were used to evaluate the effect of the wall temperature of each part of the combustion chamber.

Figure 1 is a schematic of the optically accessible engine used in the laser-induced phosphorescence thermography measurement system. Visualization experiments were conducted by installing a glass window in the normal position of one of the exhaust valves to facilitate laser irradiation and combustion visualization. This system was used to investigate the temperature distribution on the piston crown surface, including sharp edges, in each stroke during firing operation under low speed and high load conditions. Table 2 shows the experimental conditions used in the visualization experiments. Using a pulsed Nd:YAG laser and a high-speed camera, the piston crown temperature was measured separately in the intake, compression, expansion and exhaust strokes by the lifetime-based method (7).

The computational grid used in the CFD calculations is shown in Figure 2. The grid was configured with a computational mesh of 0.5 to 3.0 mm. CFD calculations were performed to investigate the effect of heat transfer from each wall on the in-cylinder temperature and the effect on temperature differences in the end-gas region during flame propagation. Calculations were performed with STAR-CD, a general-purpose CFD analysis code. The re-normalization group (RNG) k- ϵ model(3) was used as the turbulence model. The implicit scheme was employed as the time propagation method in the discretization procedure. The Monotone Advection and Reconstruction Scheme (MARS) was used to discretize the convection term, and the Pressure-Implicit with Splitting of Operators (PISO) algorithm was used as the solution for velocity and pressure. The physical models used in this study are listed in Table 3. A time-scale interaction (TI) combustion model(5) was used in the flame propagation calculation. The Angelberger wall function was used in the heat transfer model.

Table 1. Engine specifications

Engine Type	Inline 4
Displacement cc	1997.2
Bore x Stroke	84 x 90.1
Compression Ratio	13.0:1
Fuel Supply System	Direct Injection (side)
Injector Type	Multihole
Number of Holes	6
Max. Fuel pressure (MPa)	20
RON	90
Aspiration	NA
Valve Train	Int. & Exh. CVTC

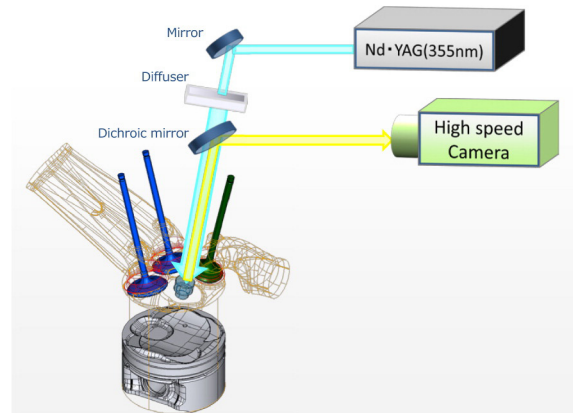


Fig. 1. Laser-Induced Phosphorescence Thermography measurement system

Table 2. Experimental Conditions

Engine condition	
Engine speed	1200rpm
Load	WOT
Fuel	iso-octane
Measurement timings	-270, -90, 90, 270 deg. ATDC
Binder type	Aron Ceramic-C (90%wt.)
Piston surface coating	
Phosphor	La2O2S:Eu(10%wt.)
Thickness	100 μ m
Excitation laser	
Source	Pulsed Nd:YAG
Power	100 mJ
Wavelength	355 nm
Camera	
Shutter speed	19 μ s
Flame rate	50000 fps

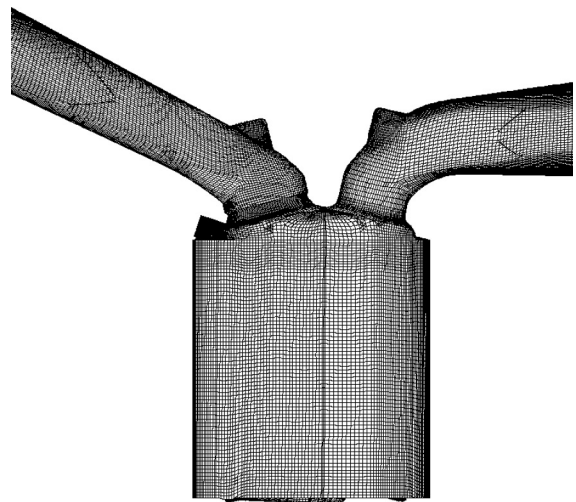


Fig. 2. CFD computational grid

Table 3. Configuration of simulation

Turbulence model	RNG k- ϵ model ^[3]
Wall heat transfer	Angelberger wall function ^[4]
Combustion	TI combustion model ^[5]
Burnt gas dissociation	Post chemistry model ^[6]

RESULTS AND DISCUSSION

Heat Transfer from Each Part of Combustion Chamber

As the first step, the rise in the in-cylinder temperature due to heat transfer from each part of the combustion chamber was analyzed. Using CFD, a comparison was made of the amount of heat transferred from each wall to the intake air-fuel mixture in the cylinder from the intake stroke to the compression stroke. Figure 3 shows a time history of the total heat transfer from each wall to the in-cylinder air-fuel mixture. It is seen that heat transfer from each wall increases during the intake stroke and decreases in the compression stroke when the in-cylinder air-fuel mixture temperature and the wall temperature invert. Total heat transfer sharply decreases near the compression top dead center in the latter half of the compression stroke because the cylinder head wall in particular accounts for a large portion of the contact area with the air-fuel mixture in the cylinder. However, the total heat transfer from the cylinder head ancillary parts to the in-cylinder air-fuel mixture accounts for around 70% (within the red frame) of the total even at 30 deg. BTDC near the compression top dead center, which represents a markedly large share. In terms of the individual parts, it is seen that the intake port accounts for around 40% of the total, which is an exceptionally large share compared with the other parts.

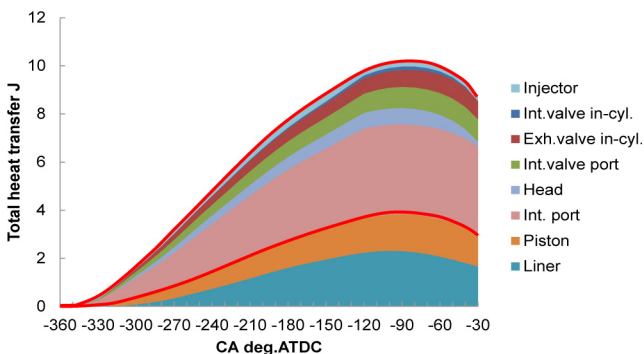


Fig. 3. Total heat transfer from each wall to in-cylinder air each CA ($N_e=1200\text{rpm}$, $\text{IMEP}=700\text{kPa}$, $T_{\text{int}}=25\text{ deg. C}$)

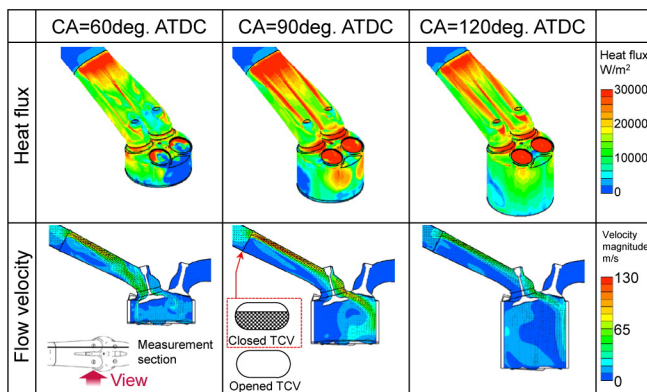


Fig. 4. Heat flux and intake flow velocity magnitude under intake stroke ($N_e=1200\text{rpm}$, $\text{IMEP}=700\text{kPa}$, TCV close)

Figure 4 shows the heat flux and flow velocity vector and its magnitude during the intake stroke with the TCV closed. Heat flux from cylinder head parts such as the intake port, intake valve seat, exhaust valve umbrella table and others shows a very high distribution. In the intake stroke, very high flow velocities are seen in the intake air passage formed by the intake port and near the cylinder head close to the inlet into the cylinder. Presumably, that is the reason why heat transfer from the cylinder head accounts for such a large portion of the total.

It was inferred from the foregoing results that reducing the cylinder head temperature, especially the intake port wall temperature, would be effective in reducing the in-cylinder temperature. The effect of reducing the wall temperature of each part was analyzed and identified by conducting experiments using the multi-cooling engine system shown in Figure 5. A three-section water jacket was applied to the intake side, center and exhaust side of the cylinder head, and a two-section water jacket was applied to the upper and lower parts of the cylinder liner. This cooling system allowed the coolant temperature in each water jacket section to be controlled individually during the experiments. The wall temperatures at various locations in the cylinder and in the intake port were measured simultaneously, and the effect of lowering each wall temperature on the timing of trace knock was made clear.

Figure 6 compares the trace knock timing and the indicated specific fuel consumption (ISFC) when the coolant temperature in each section of the water jackets was reduced. The results show that the trace knock timing was advanced the most when all the coolant temperatures were reduced, which had a large effect on mitigating knocking relative to the reduction of the combustion chamber wall temperature. All the coolant temperatures are plotted almost on the same line, which suggests that the combustion state in relation to each temperature was the same for the most part. Consequently, ISFC decreased as the trace knock timing advanced to an earlier crank angle. In terms of the individual parts, the results show that reducing the coolant temperature on the intake side of the head in particular had a large effect on mitigating knocking.

Figure 7 shows the temperature distribution of the intake port wall for each coolant temperature. When the coolant temperature in all sections of the water jackets was 86 deg. C, the intake port wall temperature was 80-85 deg. C until the upstream area owing to heat transfer. The intake port wall temperature was approximately 40-55 deg. C when only the coolant temperature in the intake side section of the cylinder head water jacket was set at 40 deg. C. When the coolant temperature in all sections of the water jackets was set at 40 deg. C, the intake port wall temperature was 40-45 deg. C. These results indicate that the coolant temperature in the cylinder head water jacket, especially that on the intake side, has a large effect on the intake port wall temperature attributed to heat transfer. Consequently, the effect on knock mitigation obtained by lowering the coolant temperature in the water jacket on the intake side of the cylinder head presumably results from reducing the intake port wall temperature.