2019-01-0154 Published 02 Apr 2019

A Transient, 3-Dimensional Multiphase CFD/Heat Transfer and Experimental Study of Oil Jet Cooled Engine Pistons

Sujan Dhar and Raghu Godavarthi Simerics Inc.

Ashwini Mishra and Sanjeev Bedekar Cummins India, Ltd.

Raj Ranganathan Simerics Inc.

Citation: Dhar, S., Godavarthi, R., Mishra, A., Bedekar, S. et al., "A Transient, 3-Dimensional Multiphase CFD/Heat Transfer and Experimental Study of Oil Jet Cooled Engine Pistons," SAE Technical Paper 2019-01-0154, 2019, doi:10.4271/2019-01-0154.

Abstract

This paper reports on a novel three-dimensional computational fluid dynamics (CFD) and heat transfer coupled methodology for analyzing piston cooling using oil jets. The method primarily consists of models of the fluid and the solid domains that are thermally coupled to one another. One of the models is a crank angle transient, three-dimensional, multiphase, volume of fluid (VOF) CFD model of the fluid behind the reciprocating piston consisting of the piston jet and crankcase gases. This model is coupled to a piston solid model. The piston motion and heat transfer from the piston to the liner are rigorously accounted for. The combustion heat flux on the piston surface was an input to the current analysis as a boundary condition. All simulations were performed using the commercial CFD software Simerics MP+.

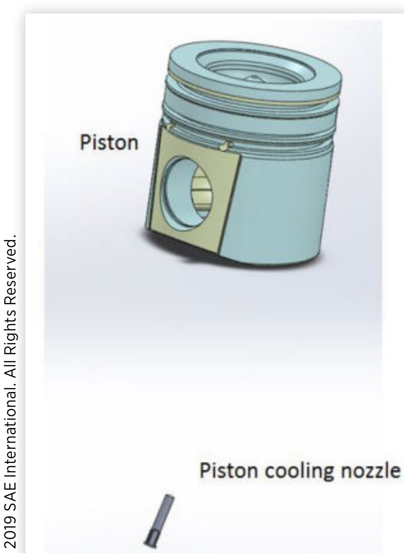
The developed method is applied to three DI Diesel engine pistons, one piston without a cooling gallery and two pistons with cooling galleries. Jet orientations aligned and non-aligned with piston motion directions were both simulated. All temperature predictions from the simulation were validated against experimental measurements. The simulation methodology was demonstrated to have the capability to consistently predict the temperature inside the pistons to within < 15C. In addition, it was also demonstrated that the methodology is general enough to be applied to any piston design including gallery cooled designs with straight and/or tilted jets. As oil jet cooled pistons become increasingly common due to higher efficiency engine development, this piston cooling simulation methodology can be now used as a predictive and valuable tool in the tool box of the analyst while guiding the design and development of the engine.

Introduction

The combustion process in the internal combustion engine leads to heat fluxes on the chamber walls. While the cylinder head and liner/block are cooled directly by the engine coolant, the cooling of the piston by the coolant is an indirect process. First the heat is transferred from the piston to the liner/block through (fluid) heat conduction paths at the rings, lands and the skirt and from there is transported to the engine coolant through metal conduction. Under high combustion heat fluxes, while the coolant can be applied aggressively to maintain the block and head temperatures, it becomes relatively more difficult to maintain safe piston temperatures. While, the motivation for engine downsizing, turbocharging and optimizing the combustion process is to improve fuel economy, the side effect is one of greater heat fluxes on the chamber walls. Piston cooling jets using engine lubrication oil are then a frequently used option to meet piston temperature requirements.

A typical setup used for a jet cooled piston is shown in the Figure 1. The piston cooling nozzle (PCN) is typically a part of the lubrication system, and the engine oil is directed

FIGURE 1 Geometry of the piston and the piston cooling jet nozzle



by the jet to the under-crown area of the piston. Several pistons also have a gallery where the oil can collect and aid in the cooling of the piston.

Simulation of the piston cooling process has several computational challenges. Firstly, the under-crown area of the piston has two fluid phases – the gaseous phase comprising the air (and crankcase gases) and a liquid phase for the engine oil released by the PCN. In addition, the jet hits the under-crown area of the piston and aids in the cooling of the piston. Therefore, heat transfer effects must be rigorously modeled in addition to a multiphase model which can resolve the interface between the two phases accurately.

Due to the complexity of the physics involved, detailed models of the piston cooling phenomena are not abundant in literature. Several papers that have attempted to model the complete engine thermal management system have frequently assumed a heat transfer coefficient; however the results reported by such approaches vary widely and do not often have a common basis for the assumptions made [1, 2, 3, 4, 5, 6]. With increasing fidelity of computational fluid dynamics (CFD) codes and the speed of computers a few recent works have attempted CFD modeling of the piston cooling phenomena. Of these several have large simplifications in the geometry of the problem such as using an axisymmetric geometry assumption in [7, 8, 9, 10, 11]. While the theoretical insights gained from such models are very useful, they are some way off from being useful to predict the piston temperatures in a real engine and to help the designers to optimize the design of the cooling jets and the piston galleries without relying heavily on experimental testing.

Recently, papers by Wang et al. [12] have presented a 3D CFD multiphase, transient model of a gallery cooled piston where the entire gallery was modeled. While the paper reported several details about the heat transfer within the fluid simulation, the prediction of the piston temperatures were beyond the scope of the paper and the model only focused on the fluid part of the simulation. A more comprehensive work has been presented by Pan et al. [13], where piston temperatures were calculated using 2D and 3D CFD and FEA simulations. However, even in this case the fluid geometry was simplified to consider the gallery portion of the fluid model only, leaving out the piston under-crown area. This approach has the serious simplification that it assumes that the jet enters the piston throughout the entire cycle, which need not be true at higher piston speeds and also for when the cooling jet is not aligned with the piston motion direction.

The current work presents a complete 3D transient, multiphase CFD model of piston cooling created within the commercial CFD program Simerics MP+ (previously known as PumpLinx). The model includes both the fluid and the solid parts of the domain and the aim of the model was to predict the temperatures of the piston in order to provide designers of the cooling jets and piston under-crown regions with enough insight to make design decisions.

In the following sections the details of the methodology to create this model will be provided for three piston and PCN assemblies from Cummins, Inc. along with relevant details of the mathematical model and algorithm used. One of the main achievements of this methodology developed was that a consistent meshing and modeling strategy was used for all three pistons so that the method developed could be used as a truly predictive tool. The 3 pistons are listed below and are names according to the following convention (N/G for with/without gallery and T/S for with tilted/straight cooling jet):

1. Piston NT: No cooling gallery. Piston is cooled by a tilted jet i.e. one that is not aligned with the piston motion direction.
2. Piston GT: Has a cooling gallery. Piston is cooled by a tilted jet.
3. Piston GS: Has a cooling gallery. Piston is cooled by a straight jet.

Finally, the results from this model will be discussed and the temperature predictions will be validated by experimental temperature measurements.

Mathematical Model

The approach used in the current study solves conservation equations of mass, momentum, and energy of a compressible fluid using a finite volume approach. Those conservation laws can be written in integral representation as

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho d\Omega+\int_{\sigma}\rho\left(\boldsymbol{v}-\boldsymbol{v}_{\sigma}\right)\cdot\boldsymbol{n}d\sigma=0 \tag{1}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho\boldsymbol{v}d\Omega+\int_{\sigma}\rho\left(\left(\boldsymbol{v}-\boldsymbol{v}_{\sigma}\right)\cdot\boldsymbol{n}\right)\boldsymbol{v}d\sigma= \int_{\sigma}\tilde{\tau}\cdot\boldsymbol{n}d\sigma-\int_{\sigma}p\boldsymbol{n}d\sigma+\int_{\Omega}\boldsymbol{f}d\Omega \tag{2}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho Ed\Omega+\int_{\sigma}\rho\left(\left(\boldsymbol{v}-\boldsymbol{v}_{\sigma}\right)\cdot\boldsymbol{n}\right)Ed\sigma= \int_{\sigma}k\nabla T\cdot\boldsymbol{n}d\sigma-\int_{\sigma}p\boldsymbol{v}\cdot\boldsymbol{n}d\sigma+\int_{\sigma}\left(\boldsymbol{v}\cdot\tilde{\tau}\right)\cdot\boldsymbol{n}d\sigma+\int_{\Omega}\boldsymbol{f}\cdot\boldsymbol{v}d\Omega \tag{3}$$

For solution within the solid model, only the conduction part of Eq. 3 was solved. The standard $k-\varepsilon$ two-equation model with wall function is used to account for turbulence,

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho kd\Omega+\int_{\sigma}\rho\left(\left(\boldsymbol{v}-\boldsymbol{v}_{\sigma}\right)\cdot\boldsymbol{n}\right)kd\sigma= \int_{\sigma}\left(\mu+\frac{\mu_t}{\sigma_k}\right)\left(\nabla k\cdot\boldsymbol{n}\right)d\sigma+\int_{\Omega}\left(G_t-\rho\epsilon\right)d\Omega \tag{4}$$

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho\varepsilon d\Omega+\int_{\sigma}\rho\left(\left(\boldsymbol{v}-\boldsymbol{v}_{\sigma}\right)\cdot\boldsymbol{n}\right)\varepsilon\, d\sigma= \int_{\sigma}\left(\mu+\frac{\mu_t}{\sigma_{\varepsilon}}\right)\left(\nabla\epsilon\cdot\boldsymbol{n}\right)d\sigma+\int_{\Omega}\left(c_1G_t\frac{\varepsilon}{k}-c_2\rho\frac{\varepsilon^2}{k}\right)d\Omega \tag{5}$$

Together with equation of state, where properties are functions of temperature and pressure, to form a closed system:

$$\rho=f\left(p,T\right) \tag{6}$$

In the solver, each of the fluid properties will be a function of local pressure and temperature, and can be prescribed as an analytical formula or in a table format.

VOF Model for Multiphase

VOF models are widely used in simulation of two phase flow [14, 15]. VOF solves a set of scalar transport equations representing the fraction of the volume each fluid component occupies in every computational cell. The transport equation of the volume fraction for each fluid component can be written as:

$$\frac{\partial}{\partial t}\int_{\Omega(t)}\rho_i F_i d\Omega + \int_{\sigma}\rho_i\left(v - v_{\sigma}\right)\cdot \boldsymbol{n} F_i d\sigma = 0 \tag{7}$$

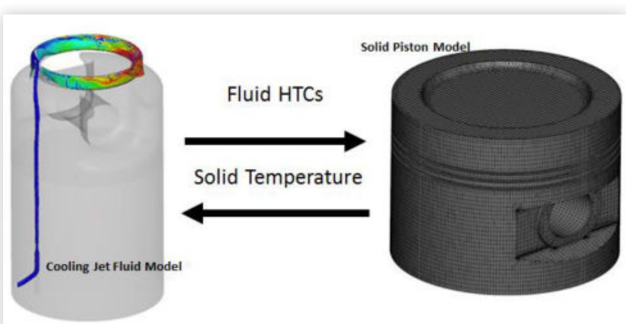
Where F_i is the volume fraction of the *ith* fluid component, and ρ_i is the local density of *ith* fluid component. The weighted mixture density of the fluid in equation (1) to (5) are then calculated as:

$$\rho = \sum \rho_i F_i \tag{8}$$

Both implicit and explicit methods can be used to solve the equation. High Resolution Interface Capturing (HRIC) scheme can be used for the convective term in the transport equation. The VOF model within used in the current approach has also been validated extensively for multiphase flow without including the energy equation [15, 16] and for multiphase flow including the energy equation [17].

In the present piston cooling model the solid and fluid domain solutions are coupled together. This coupling was performed taking into account the large difference in the time-scales of the thermal problem for the fluid and the thermal problem for the solid. While the fluid temperatures can be expected to reach stable values within a few cycles the piston temperature is expected to take of the order of minutes due to the high thermal inertia of the solid piston. In addition, the higher thermal inertia of the piston also means that the piston temperature change is negligible during one complete revolution of the crankshaft (which would take ~0.06 seconds for an engine running at 1000 RPM). Due to this difference in the time scales a loosely coupled algorithm was used wherein the cycle averaged heat transfer coefficients and reference temperatures are mapped from the fluid to the solid and the piston surface temperatures are mapped back on the fluid model as shown in Figure 2. In such an approach the fluid simulation is transient but the solid simulation can be steady state. Therefore the final result from such an approach will be a periodic flow and thermal solution for the fluid and the final steady temperatures and heat fluxes for the solid.

FIGURE 2 Coupled Heat Transfer Model of Piston Cooling



The coupled simulation is continued until the temperatures and fluxes for both the solid and fluid have stabilized.

Model Attributes and Mesh Generation

The fluid volume of the entire piston cooling jet fluid domain is extracted from the solid CAD of the pump using Boolean operations common in any CAD package and imported. A view of the entire computational domain and the mesh for Piston NT is shown in Figure 3. Mesh generation was performed using automated Binary Tree and Template meshers to create a good quality hex-dominant mesh of the fluid and the solid domain.

Local mesh refinements are created to capture the jet interface accurately and also to resolve the wall heat transfer process (Figure 4). The jet direction for the NT and GT pistons

FIGURE 3 Meshes of A) Fluid domain and B) Solid domain.

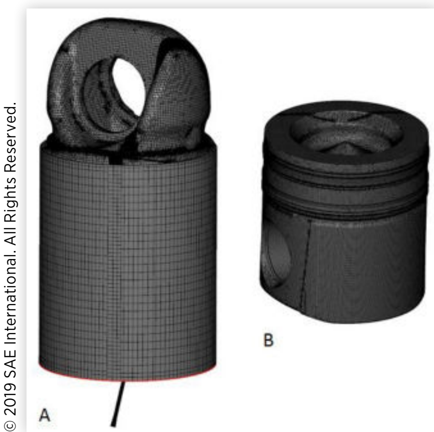


FIGURE 4 Section through the fluid mesh. Inset shows the zoomed in view of the mesh refinement areas.

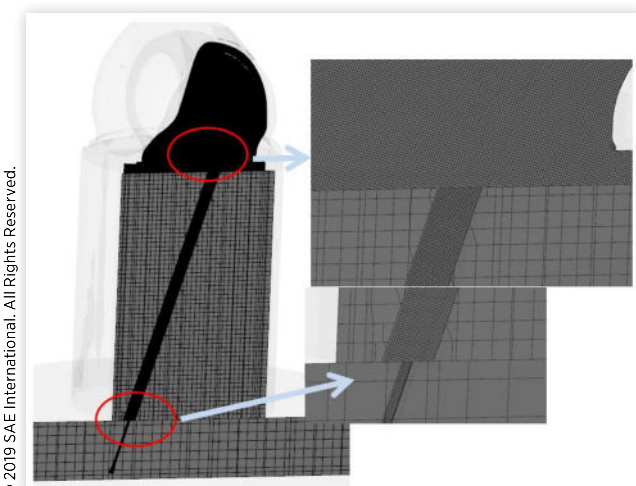
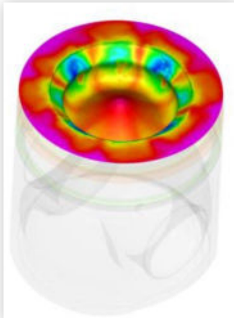


FIGURE 5 Mapped combustion heat flux field on the piston bowl boundary.



were not aligned with the direction of the oscillatory motion of the piston and the piston oscillatory motion was incorporated into the simulation using mesh motion/deformation. In this way, the exact path of the tilted jet flow is reproduced and the areas of the piston that the jet impinges upon are accurately captured. Engine oil properties for 15W-40 oil were used and density, viscosity, conductivity and specific heat were a function of temperature. Piston material was Aluminum alloy and the piston rings were gray cast iron - again the specific heat and conductivity were f (T).

For all three pistons, on the piston bowl boundary the cycle averaged heat flux from a separate combustion analysis is imposed as a mapped heat flux field as shown in Figure 5. In addition, (equivalent) convection boundary conditions are specified on the outer surfaces of the piston, primarily the piston rings and the skirt which exchange heat with the liner. The reference temperature for the boundary condition was set using a dwell time averaging technique on the liner temperatures.

In the following sections the results from the coupled piston cooling simulation and comparisons with experiment will be presented in detail.

Results

All results discussed in the present section are from transient 3D CFD piston cooling simulations with both fluid and solid domains simulated in a coupled manner. In order to respect the confidentiality requirements, all reported quantities have been normalized using reference quantities (marked by the subscript *Ref*). However, the normalization has been done in a consistent manner such that the results are still meaningful from a relative standpoint.

Results for the three pistons will be presented for the following operating conditions.

The fluid jet VOF solution viewed as an isosurface of 0.5% oil is shown for 4 different positions of Piston NT in Figure 6 alongside the heat flux to the oil. It can be seen that when the piston is close to bottom dead center the jet impinges on a larger area of the piston crown leading to much higher heat fluxes to the oil. In addition, the heat transfer to the oil is higher when the piston is moving from TDC to BDC.

FIGURE 6 Heat Flux to Oil (left) and VOF Isosurface of Oil (right) for different positions of the piston for the NT piston

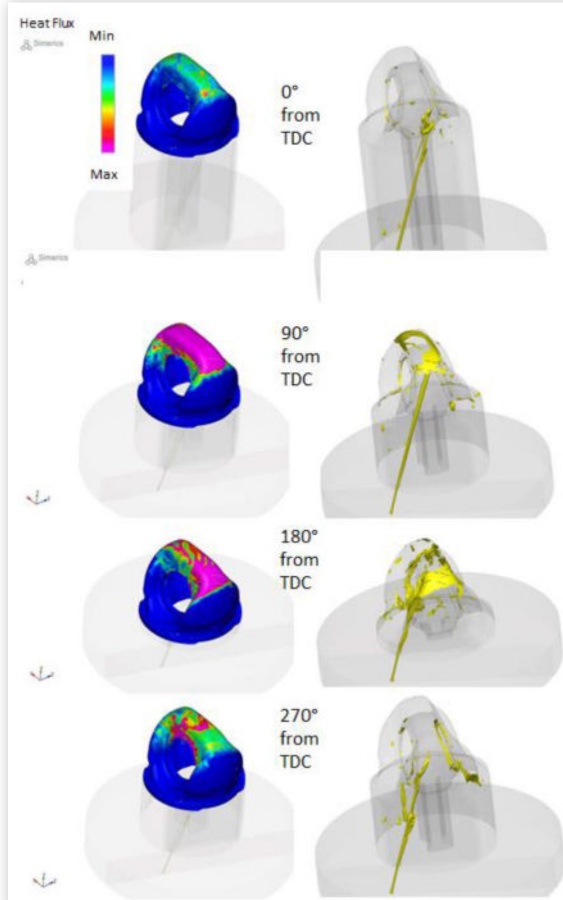


TABLE 1 Operating conditions for piston cooling simulations

Piston ID	Engine Speed [RPM]	Oil Flow Rate (Q/Q_{Ref})	Oil Temperature (T/T_{Ref})
NT	2500	1	1
GT	2300	0.75	1.1
GS	3500	0.825	1.01

In order to verify that the solution was mesh independent two separate meshes of 0.7 and 0.9 million cells were compared for the wetted area on the piston crown region. The results from the two finest meshes used are shown in Figure 7. Since it was observed that the change in the wetted area was minimal between the two meshes the 0.7 million cell mesh was used for the simulation.

The piston temperature predicted by the coupled fluid solid simulation is shown in Figure 8.

The meshing and modeling approaches described for the NT piston above was then used for a further two pistons, both of them gallery cooled. Due to the additional mesh within the gallery the mesh size for these cases were in the region of 1.7 million cells. The fluid solution for the GT piston at TDC and BDC alongside the heat flux distribution was shown in Figure 9. As can be seen, due to the tilted orientation of the oil jet nozzle the jet does not enter the gallery directly at TDC

FIGURE 7 Wetted area of the piston crown region for mesh sizes of 0.7 and 0.9 million.

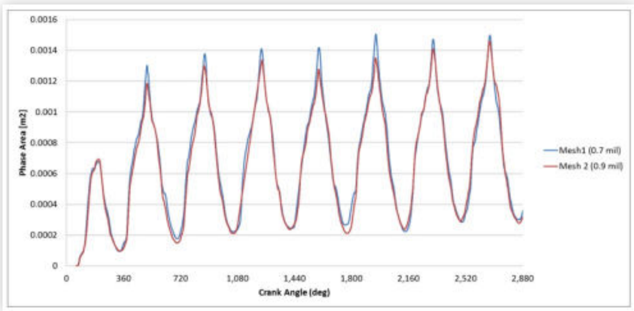


FIGURE 8 Temperature predictions shown on the NT piston surface (left) and through a section cut within the piston (right). The temperature shown has been normalized using a reference temperature (T_{Ref})

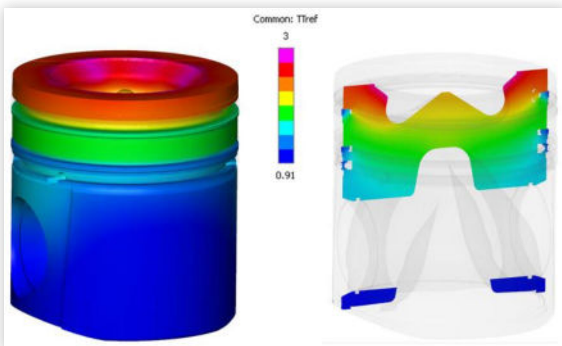
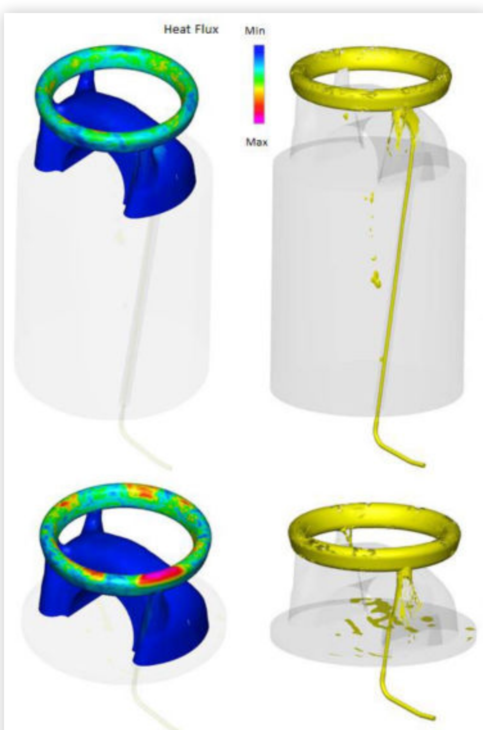


FIGURE 9 Heat Flux to Oil (left) and VOF Isosurface of Oil (right) at piston TDC and BDC positions for the GT piston



or at BDC, but does enter the gallery while the piston is moving from TDC to BDC. Special design features on the piston when the piston is at BDC enable it to direct a large part of the flow into the gallery. It can also be seen that the heat flux from the gallery plays a dominant role in transferring heat from the piston to the oil.

The temperature predictions for the GT piston are shown in Figure 10.

Fluid results for the GS piston are shown in Figure 11. Since this piston had a straight jet, the jet is directed toward the gallery entrance at both TDC and BDC – therefore only a view at piston TDC is shown. Again it can be seen that a majority of the heat removed is from the gallery.

The temperature predictions for the GS piston are shown in Figure 12.

The differences in the geometry of the pistons meant that for the NT piston, the piston under-crown heat transfer was of the greatest importance, the other two gallery cooled pistons were dominated by the heat transfer caused by oil sloshing within the gallery. The gallery effect is particularly dominant in the GS piston, since the oil jet never directly hits

FIGURE 10 Temperature predictions shown on the GT piston surface (left) and through a section cut within the piston (right). The temperature shown has been normalized using a reference temperature (T_{Ref})

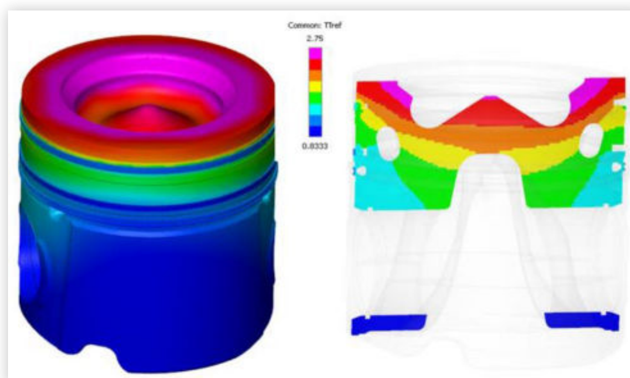


FIGURE 11 Heat Flux to Oil (left) and VOF Isosurface of Oil (right) at piston TDC position for the GS piston

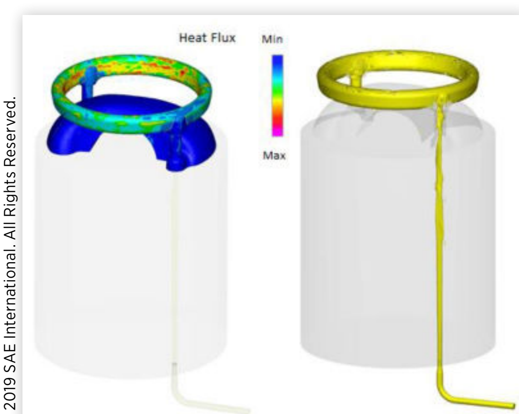


FIGURE 12 Temperature predictions shown in the GT piston surface (left) and through a section cut within the piston (right). The temperature shown has been normalized using a reference temperature (T_{Ref})

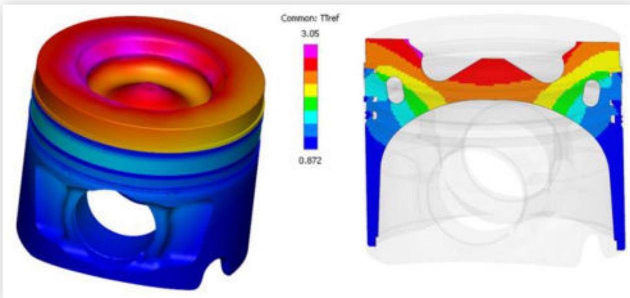
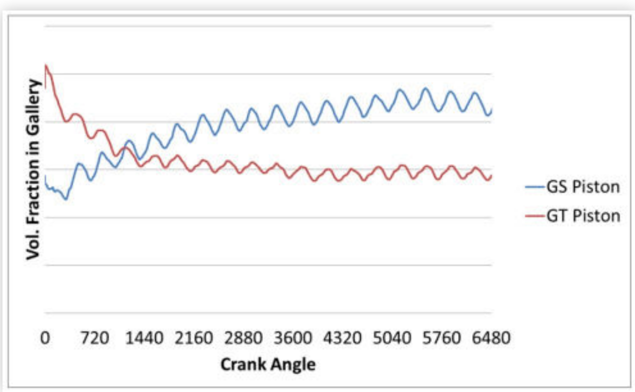


FIGURE 13 Convergence of the volume inside the piston cooling gallery for the GS and GT pistons from initial guess to the final periodically varying value.



the piston under-crown region, whereas for the GT piston (as seen in Fig. 9) the jet directly impinges on the piston when the piston is at TDC, leading to a higher heat flux from the under-crown region.

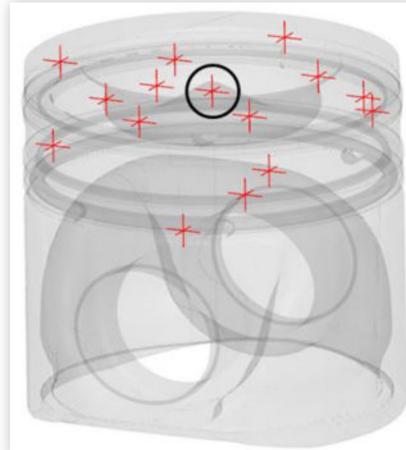
One of the main quantities which determine the convergence of the simulation to stable periodic results for piston cases with cooling gallery is the volume fraction of oil inside the gallery. This typically takes <20 crank revolutions. The complete history starting from initial guess of the volume of oil inside the gallery for the GT and GS pistons are shown in Figure 13.

The simulation time for 1 crank revolution was around 90 minutes per revolution using 48 cores (Intel Xeon 2.20GHz), thus the total simulation time to achieve piston temperature predictions was typically within 24 hours.

Validation and Experimental Comparison

For the piston and the operating conditions described in the sections above, the internal temperature of the piston were measured at 15 locations inside the piston using temperature sensors. The measurement points were well distributed throughout the piston, with measurements near lands and rings as well as near the crown and the bowl of the piston. The measurement points for the NT piston are shown in Figure 15. The distribution of measurement points is similar for the GT and GS pistons.

FIGURE 15 Measurement point locations for the NT piston. The point in the piston dome is highlighted using a circle.



The comparison between the simulation results and the experimental measurements for all 3 pistons are shown in Figure 16. The measurement point on the piston dome is highlighted using a circle in Figures 15 and 16.

As can be seen, the internal temperature predictions of the piston from the simulation method match very well with the experimental data. The piston cooling simulation results are seen to capture the temperature trends as well as the absolute level of temperature within the piston with excellent accuracy. In particular, the temperature at the piston dome is widely considered to be the most challenging to predict accurately and the good prediction of the temperature on the piston dome is highlighted in Figures 15 and 16. The predictions at several points are within 1 C of the experimental measurements, and the RMS error calculated for the simulation results are < 15 C for all 3 pistons.

Simulation Study

The validated piston cooling model was then used to study the effect of changing the temperature of the oil jet. The simulation at 2500 RPM was performed using oil inlet temperature of 5C higher than the original oil inlet temperature. The temperature change at the 15 points inside the piston solid for the change in oil inlet temperature is shown in Figure 18. In general the effect is of the order of 1C change in piston temperatures for 5C change in oil jet inlet temperatures. This is expected since the NT piston does not have a gallery and due this the oil carries away a relatively smaller percent of the total combustion heat flux.

An additional simulation was also performed for an engine speed of 1300 RPM. In general the temperatures observed on

FIGURE 16 Comparison of Piston temperatures from Simulation and Experimental Measurements for A) NT, B) GT and C) GS pistons. The points in the piston dome are highlighted using a circle.

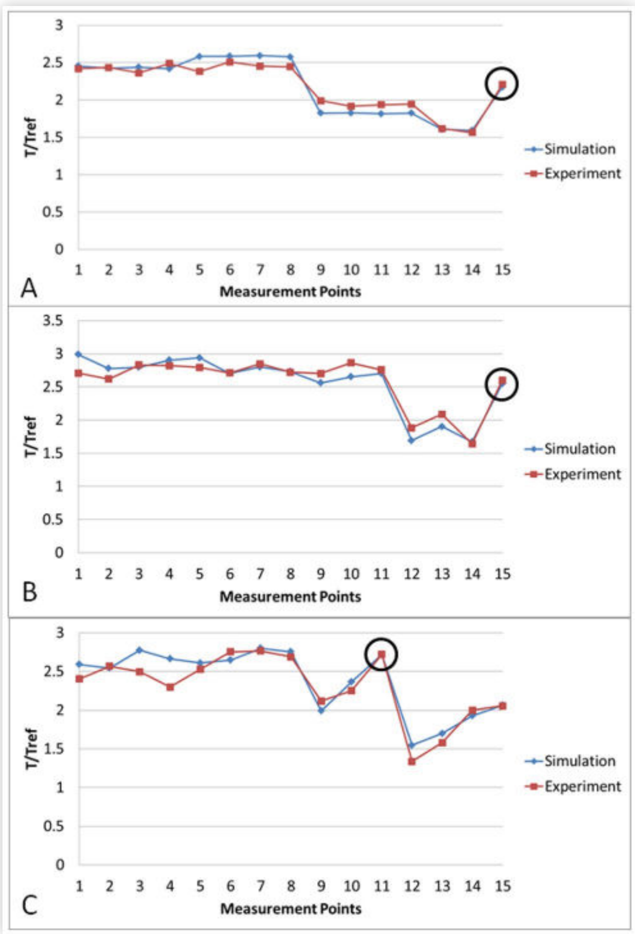
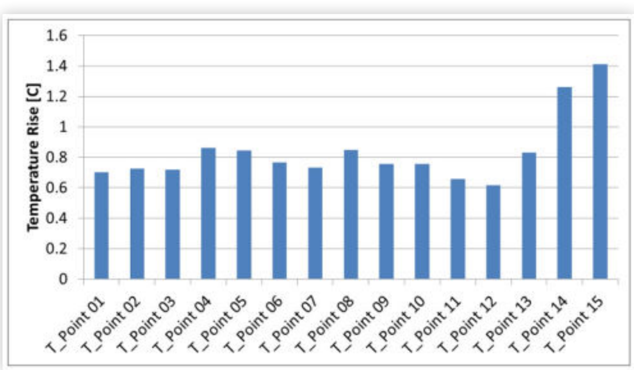
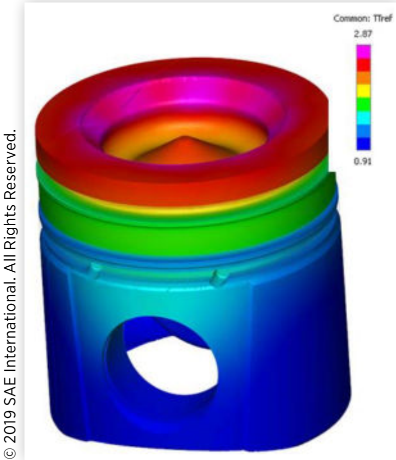


FIGURE 18 Change in the temperature at different points inside the piston when oil jet temperature is changed from 110C to 115C



and inside the piston were lower than those observed at 2500 RPM as shown in Figure 19. This change in temperature was due to the lower heat flux from combustion and also because of the lower relative speed between the piston and the cooling jet due to which the cooling effect of the jet is lower.

FIGURE 19 Piston temperature from simulation at 1300 RPM



Conclusions

In the present work a piston cooling simulation methodology was described where a transient multiphase VOF fluid simulation was coupled with a solid conduction simulation model to achieve prediction of piston temperature for oil jet cooled pistons. The methodology has been demonstrated for 3 pistons which have significant differences in geometry: one with no gallery and tilted jet, one with gallery and tilted jet and one with gallery and straight jet. This simulation project had been conducted with a view to have a truly predictive tool that could be used for piston cooling analysis and with this in mind, meshing and modeling practices have been developed and these practices were consistently used for the three pistons presented in this work.

All three piston simulation results were compared with experimental measurements and the agreement between simulation results and measurement were observed to be excellent. The simulation methodology was demonstrated to have the capability to consistently predict the temperature inside the pistons to within < 15C. In addition, it was also demonstrated that the methodology is general enough to be applied to any piston design including gallery cooled designs with straight and/or tilted jets. As oil jet cooled pistons become increasingly common due to higher efficiency engine development, this piston cooling simulation methodology can be now used as a predictive tool to analyze the designs of piston cooling galleries, piston under-crown channels and various nozzle positions/orientations and has the potential to drive the design of oil jet cooled pistons.

References

1. Bush, J. and London, A., "Design Data for "Cocktail Shaker" Cooled Pistons and Valves," SAE Technical Paper 650727, 1965, doi:10.4271/650727.
2. French, C., "Piston Cooling," SAE Technical Paper 720024, 1972, doi:10.4271/720024.

3. Nozawa, Y., Noda, T., Yamada, T., Takeuchi, Y., and Akimoto, K., "Development of Techniques for Improving Piston Cooling Performance (First Report): Measurement of Heat Absorption Characteristics by Engine Oil in Cooling Channel," *JSAE* 5:78-85, 2005.
4. Torregrosa, A.J., Broatch, A., Olmeda, P., and Martı, J., "A Contribution to Film Coefficient Estimation in Piston Cooling Galleries," *Experimental Thermal and Fluid Science* 34(2):142-151, 2010.
5. Esfahanian, V., Javaheri, A., and Ghaffarpour, M., "Thermal Analysis of an SI Esngine Piston Using Different Combustion Boundary Condition Treatments," *Applied Thermal Engineering* 26(2-3):277-287, 2006.
6. Kajiwara, H., Fujioka, Y., and Negishi, H., "Prediction of Temperatures on Pistons with Cooling Gallery in Diesel Engines Using CFD Tool," SAE Technical Paper 2003-01-0986, 2003, doi:10.4271/2003-01-0986.
7. Agarwal, A.K. and Varghese, M.B., "Numerical Investigations of Piston Cooling Using Oil Jet in Heavy Duty Diesel Engines," *International Journal of Engine Research* 7(5):411-421, 2006.
8. Varghese, M. and Agarwal, A., "Numerical Investigations of Piston Cooling Using Oil Jet," SAE Technical Paper 2004-28-0061, 2004, doi:10.4271/2004-28-0061.
9. Stevens, J. and Webb, B.W., "Local Heat Transfer Coefficients under an Axisymmetric, Single-Phase Liquid Jet," *Journal of Heat Transfer* 113(1):71-78, 1991.
10. Cicalese, G., Berni, F., and Fontanesi, S., "Integrated In-Cylinder / CHT Methodology for the Simulation of the Engine Thermal Field: An Application to High Performance Turbocharged DISI Engines," *SAE Int. J. Engines* 9(1):601-617, 2016.
11. Yi, Y., Reddy, M., Jarrett, M., Shyu, P. et al., "CFD Modeling of the Multiphase Flow and Heat Transfer for Piston Gallery Cooling System," SAE Technical Paper 2007-01-4128, 2007, doi:10.4271/2007-01-4128.
12. Wang, P., Liang, R., Yu, Y., Zhang, J. et al., "The Flow and Heat Transfer Characteristics of Engine Oil inside the Piston Cooling Gallery," *Applied Thermal Engineering* 115:620-629, 2017.
13. Pan, J., Nigro, R., and Matsuo, E., "3-D Modeling of Heat Transfer in Diesel Engine Piston Cooling Galleries," SAE Technical Paper 2005-01-1644, 2005, doi:10.4271/2005-01-1644.
12. Ubbink, O., "Numerical Prediction of Two Fluid Systems with Sharp Interfaces," 1997.
13. Hirt, C.W. and Nichols, B.D., "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries," *Journal of computational physics* 39(1):201-225, 1981.
14. Ding, H., Jiang, Y., Wu, H., and Wang, J., "Two Phase Flow Simulation of Water Ring Vacuum Pump Using VOF Model," in *ASME/JSME/KSME 2015 Joint Fluids Engineering Conference*, 2015, V001T33A019-V001T33A019. American Society of Mechanical Engineers.
15. Kucinschi, B.R. and Shieh, T.-H., "Estimation of Oil Supply Time during Engine Start-Up at Very Low Temperatures," *SAE International Journal of Fuels and Lubricants* 9(2):363-369, 2016.
16. Ding, H., and Jiang, Y., "CFD Simulation of an Oil Flooded Scroll Compressor Using VOF Approach," 2016.

Contact Information

Sujan Dhar
Simerics, Inc.
1750112th Ave. NE, Ste C250
Bellevue, WA 98108
(425) 502 9978

Raj Ranganathan
Simerics, Inc.
1750112th Ave. NE, Ste C250
Bellevue, WA 98108
(425) 502 9978

Definitions/Abbreviations

CFD - Computational fluid dynamics

VOF - Volume of Fluid

PCN - Piston Cooling Nozzle

NT - No cooling gallery, tilted jet

GT - With cooling gallery, tilted jet

GS - With cooling gallery, tilted jet

CAD - Computer Aided Design

TDC - Top Dead Center

BDC - Bottom Dead Center

ISSN 0148-7191