2019-01-0095

Validation and analysis of heat losses prediction using Conjugate Heat Transfer simulation for an internal combustion engine

Author, co-author (Alberto Broatch, Xandra Margot, Jorge Garcia-Tiscar, and Johan Escalona)

Affiliation (CMT- Universitat Politècnica de València)

Abstract

New technologies are required to improve engine thermal efficiency. For this it is necessary to use all the tools available nowadays, in particular computational tools, which allow testing the viability of different solutions at reduced cost. In addition, numerical simulations often provide more complete and precise information than experimental tests. Such is the case for the study of the heat transfer through the walls of an engine. Conjugate Heat Transfer (CHT) simulations permit precise calculations of the heat transfer rate from gas to walls throughout the whole engine cycle, and thus it is possible to know such details as the instantaneous heat losses and wall temperature distribution on the walls, which no experiment can give. Nevertheless, it is important to validate CHT calculations, either with some experimental measurements or with some other reliable tool, such as 0D-1D modelling known to work well.

The proposed work is based on the CHT simulation of the heat transfer to the walls of an engine piston during an entire cycle to determine the parameters that permit obtaining good results. This will be ascertained by comparison with the results of a lumped model previously validated for many applications. Another objective of this work is also to determine if it is significant to take into account the spatial and temporal variations of the wall temperature for the prediction of the heat losses during the engine cycle, as generally a mean and constant wall temperature (isothermal walls) is assumed for CFD combustion calculations.

Introduction

While current trends clearly focus on the development of electrified vehicles, it is highly unlikely that internal combustion engines will completely disappear in the coming years. For this reason it is important to explore new technologies that will allow improving the thermal efficiency of internal combustion engines, while reducing further the pollutant emissions. New challenges have to be tackled when downsizing engines for instance, linked to more extreme conditions within the combustion chamber that will affect the combustion process [1], and thus the engine efficiency.

In-depth understanding of the combustion process is essential to improve engine efficiency and reduce emissions. This knowledge is essential to optimize engine performance, especially in the early stages of design when the components such as the cylinder head, combustion chamber or piston may still be modified. Traditionally, engine prototypes are built and experimentally tested on dedicated benches. However, this more traditional approach is costly and the information obtained is not sufficient to bring insight into the formation of soot and other emissions. Hence, computer models have been developed throughout the years [2] to address the very complex chemistry and thermodynamics involved in the combustion process. These allow predicting emissions at different operation conditions without having to perform many expensive experimental tests. The problem is that predictions are not yet accurate enough, so that more effort is required to improve and optimize these models [3, 4].

However, computational simulations are still very useful, especially at pre-design stage, when it can be checked if some modification in the combustion chamber may affect the overall performance of the engine. Also, they provide additional information that it is very difficult or simply not possible to obtain with experimental measurements. For instance, while it is important to localize hot spots in the combustion chamber of premixed air-fuel gasoline engines, because they may cause knock [5], it is experimentally very difficult to use any kind of device, such as thermo-couples, to measure gas temperature. Another example: to calculate the heat losses through the combustion chamber walls, it is necessary to measure the temperature of the walls exposed to the gas, which is complicated and intrusive [6, 7]. Some experimental techniques exist, such as the Laser Induced Phosphorescence [8, 9], but this yields very limited information for just one point of the surface. Hence, heat fluxes through the combustion chamber walls are mostly estimated with 1D and lumped models [10, 11].

One way to improve the performance and fuel efficiency of an internal combustion engine (ICE) is to reduce the heat losses through the walls of the combustion chamber [12-16]. By adequately insulating the walls of the engine, it may be possible to reduce the temperature difference between the wall and the gas (the so-called temperature swing) and thus the heat losses. In fact, past investigations centered on this issue [17] but the insulation material used at the time was ceramic, and due to its poor characteristics (low heat conduction, but high heat capacity), the idea did not prosper. Nonetheless, in some recent studies, new insulating materials of low thermal conductivity and low heat capacity have been applied on the piston surface of Diesel engines [18-20] and the authors reported some heat losses gains. However, in order to achieve a better thermal efficiency, the total heat fluxes throughout the complete engine cycle have to be considered. This requires detailed knowledge of the engine thermal behavior, taking into account the heat fluxes through the various systems, such as intake and exhaust ports, coolant circuit, and lubricant oil subsystem. Approaches based on validated lumped models provide a reasonable compromise between computational cost and solution accuracy [10, 11], but cannot give detailed information such as temperature distribution on the wall surfaces. Computational Fluid Dynamics flow simulations on the other hand allow predicting the temporal and spatial distribution of all the gas variables, including temperature within the combustion chamber, but the heat fluxes through the walls are more or less imposed, rather than calculated, since the wall temperatures are generally fixed as boundary conditions and considered as isothermal [21, 22].

In order to calculate the heat losses in the engine, it is therefore necessary to take into account the changes in wall temperature during the whole cycle. For this, a decoupled iterative approach may be used, as in [23], whereby the heat fluxes and gas properties obtained from a CFD calculation with constant temperature wall boundary conditions are used to calculate the heat conduction in the solid regions. Then the wall temperatures are updated for the following CFD simulation, and the procedure is repeated until the wall temperature converges. This approach is not very efficient, as it requires the exchange of information between the two different codes used.

Based on this idea, the Conjugate Heat Transfer (CHT) method is a coupled approach that solves simultaneously the fluid and the solid regions in a unique computational iterative process first developed and validated by [24] and further by [25], who used it for transient calculations of heat conduction through the piston and cylinder head walls of a Diesel engine. With the CHT method it is therefore possible to obtain the local and instantaneous heat fluxes through the walls throughout the engine cycle, as well as the temporal evolution and spatial distribution of temperature on any surface exposed to the gas. It is also possible to take into account the thermal stresses to optimize the cooling system and the solid materials [26]. Broatch et al. [22] used CHT to calculate the heat losses through the walls of an SI engine and proposed an optimized CHT calculation based on the use of the rate of heat release (RoHR) as source term for the energy equation.

The purpose of this paper is to use a fully coupled CFD-CHT calculation to analyze the heat transfer through the walls of an SI single-cylinder engine and validate the results against a one-dimensional model, since no experimental results are available. The CHT is applied to the piston only, as these calculations are computationally expensive. Indeed, the solid regions converge at a much slower rate than the fluid regions, and several engine cycles have to be performed to guarantee the convergence of the thermodynamics properties. In addition, the heat fluxes to the various walls of the engine are compared with the ones calculated with a simple CFD simulation with fixed temperature boundary conditions. The paper is divided in 4 main sections. The first deals with the numerical set-up, including the delicate issue of the piston boundary conditions, and the second with the methodology followed for the validation. In the third section results of the validation for the CFD part and then a full analysis of heat fluxes and temperature distribution on the piston wall are presented. Finally, the conclusions section summarizes the main findings of this study.

Numerical setup

CHT calculation is a very useful tool to determine the temporal evolution and spatial distribution of temperature on the solid regions in an internal combustion engine. However, the results obtained need to be validated with experimental data. Since measuring the temperatures on the solid components of the engine is highly difficult as well as costly from the economical point of view, one way to validate the CHT simulation results is to use 0D-1D models available in the literature and previously validated against experimental data. It is clear that 0D-1D models are based on assumptions that allow simplifying the problem, in particular that the heat flux follows a one-dimensional way. This not necessarily true in a real case like the one in this study, but it is known that the main heat transfer does occur in a preferential direction. Therefore, though differences may exist between the 0D-1D models and the 3D CHT model, results are nonetheless expected to show similar tendencies and temperature levels.

Geometry and engine specifications

The computational domain is formed by the combustion chamber with the intake and exhaust manifolds, including the solid piston in the domain. The full engine geometry provided by IFP Energies Nouvelles, with the parts integrating the solid and the fluid domain is shown in Fig. 1.

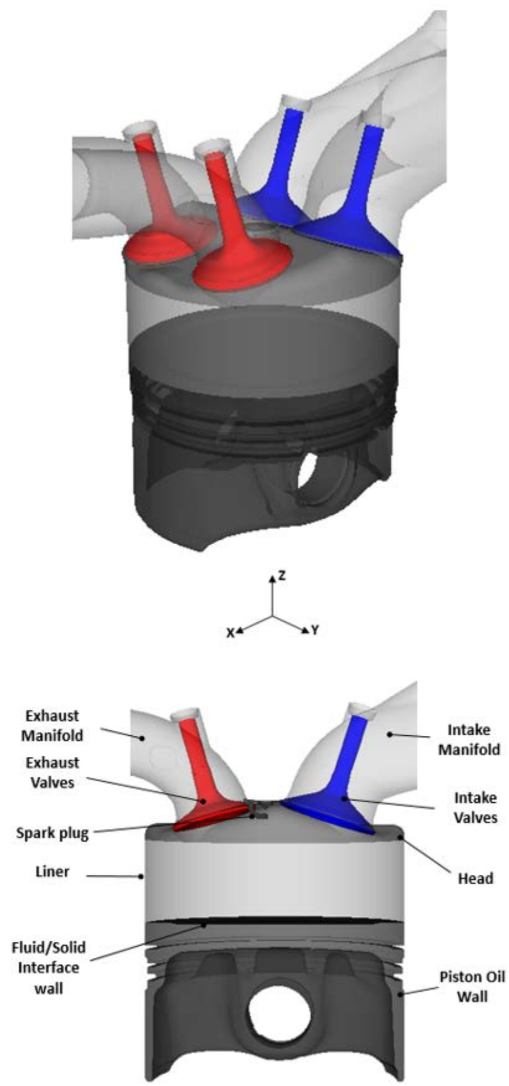


Figure 1. Calculation domain with the engine parts.

The main features of the premixed gasoline engine used for the CHT calculations are presented in Table 1. Some characteristics cannot be presented here for confidential reasons.

Table 1. Engine specifications

Number of cylinders	1
Stroke	93 mm
Bore	75 mm
Connecting Rod	150 mm
Compression ratio	14:1
Number of Valves	2 intake and 2 exhaust
Engine speed [rpm]	2000
IMEP [bar]	4
Ignition timing [cad aTDC]	-27.5

Mesh generation

The commercial code CONVERGE was used for the CHT calculations. CONVERGE generates the mesh at run-time, trimming the cells at the intersecting surfaces, after which the intersection information (surface areas, normal vectors) is reduced before storing for each cell. This process is done at the start of the simulation and whenever the mesh has to be refined.

The computational domain has a cell size of 2 mm for the base grid. In order to refine the mesh for better accuracy of the results, the base grid size is reduced in specific regions where the physical processes are more complicated. The full engine mesh with the solid and fluid domain is displayed in Fig. 2.

Near the combustion chamber walls, 2 levels of grid refinement are added in order to model the boundary layer properly, as this is critical for the heat transfer. In addition, for the cases where the full combustion process is calculated, the region near the spark plug is refined at the start of the ignition to improve the accuracy (see Fig. 2). This refinement is only activated during the ignition process (-27.5 to -13.5 cad aTDC).

The initial mesh has between 1 227 630 cells at bottom dead center (BDC) and 329 021 cells at top dead center (TDC), including both the fluid and solid domains. In addition, an adaptive mesh refinement (AMR) algorithm is used to refine the mesh in zones of the combustion chamber where the velocity and temperature gradients are highest. This is advantageous compared to a statically refined mesh, as it allows reducing the computational cost. In addition, a strategy that consists in relaxing the Courant number from EVO to IVC [27] is used to optimize the calculation time.

Models and boundary conditions

The combustion process is simulated using the SAGE combustion model, which models detailed chemical kinetics. This model calculates the reaction rates for each elementary reaction while the CONVERGE CFD solver solves the transport equations [28]

The RNG (renormalization group) k-ε model is used for turbulence, combined with the O'Rourke and Amsden heat transfer wall model [29, 30]. This model was chosen on the basis of a preliminary study made with various other models available in CONVERGE, where results were compared to those obtained with a lumped model [10, 11].

For the O'Rourke and Amsden model, the wall heat transfer is given by

$$k\frac{dT}{dx_i} = \frac{a_m c_p F\,(T_f - T_w)}{Pr_m y} n_i \tag{1}$$

$$F = \begin{cases} 1.0 & y^+ < 11.05 \\ \dfrac{\frac{y^+ Pr_m}{Pr_t}}{\frac{1}{\kappa}\ln y^+ + B + 11.05\left(\frac{Pr_m}{Pr_t} - 1\right)} & y^+ > 11.05 \end{cases} \tag{2}$$

$$y^+ = \frac{\rho u_t y}{\mu_m} \tag{3}$$

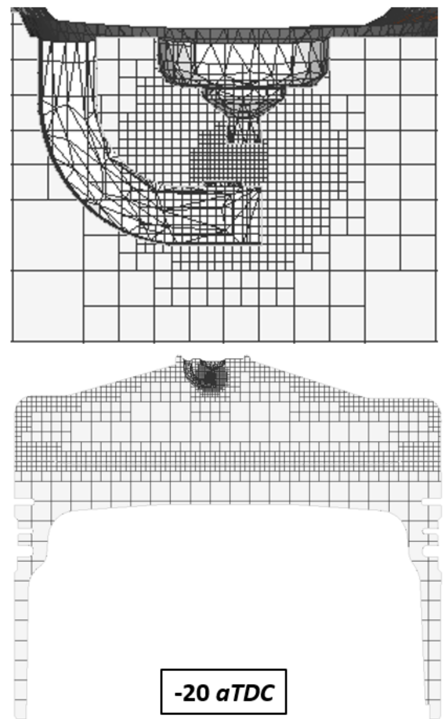


Figure 2. Mesh generation at spark timing.

k is the molecular conductivity, κ is the Von Karman constant (0.4187) and B is a law-of-the wall constant; in this case the value is 5.5. Pr_m is the molecular Prandtl number, Pr_t is the turbulent Prandtl number, T_f is the fluid temperature, T_w is the wall temperature and u_t is the shear speed.

Only the temperature of the solid surfaces of the piston is calculated with the CHT approach. All other gas-exposed surfaces (liner, valves, cylinder head) are considered as isothermal during the whole engine cycle. The temperature boundary conditions on these walls are calculated with a lumped model [10, 11, 31-33] and iterated until the pressure trace obtained from CFD is tuned with the experimental data.

In this investigation the piston wall temperature is calculated with the conjugate heat transfer model of CONVERGE [28]. For this, the required boundary conditions at the inner surfaces in contact with the gas are the heat capacitance, the thermal conductivity, and the density of the wall material (aluminum). The thickness of the material is directly given by the meshed geometry of the wall.

In order to simulate more realistic conditions for the outer walls of the piston, the solid domain needs to be divided into various regions, so that different values of the boundary conditions are applied to the walls in contact with the gas, the piston segments and the cooling oil, as indicated in Figure 3. The boundary conditions for the piston crown are calculated from the cycle averaged gas temperature and heat transfer coefficient taken near the liner. For the piston oil cavity, piston skirt and piston rings values for the boundary conditions were obtained with correlations and information available in the literature, as summarized in Table 2.

To accelerate the convergence of the CHT calculations, which are very time-consuming, the final wall temperatures of the piston obtained from the lumped model are used as initialization values for the solid region. Initializing the region with a random temperature could increase the number of engine cycles needed.

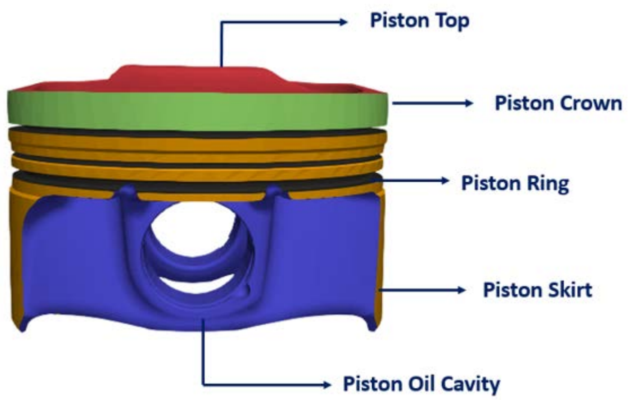


Figure 3. Walls considered for the solid region.

Table 2. Engine specifications

Piston Top	Contact with combustion gas
Piston Crown	Contact with combustion gas
Piston Ring	Piston segments [35]
Piston Skirt	Contact with oil [38]
Piston Oil Cavity	Contact with cooling oil [39]

Taking into account that it takes much longer to converge the solid region than the fluid region, the super-cycling approach of CONVERGE [35] has been used to solve the conjugate heat transfer problem. This method solves both fluid and solid phases using the transient solver. During the first engine cycle it stores periodically the heat transfer coefficient and the near wall temperature at the fluid/solid interface cells in stages at defined time-step intervals. After 720 CAD (stages number*degree interval), CONVERGE averages the heat transfer coefficient and the temperatures of the solid cells. It then uses the new wall temperatures to solve the fluid and solid with the transient solver. The code freezes the fluid solver at the super-cycling time step interval and solves the steady-state solid temperature. This is updated to recalculate the solid and fluid with the transient solver. The process is repeated until the solid temperature has converged. This strategy reduces the number of engine cycles needed to reach convergence of the solid temperature [1]. For this study multiple cycles were run to calculate the evolution of the temperature on the solid surface of the piston, with a time step interval of 4 CAD to store the heat transfer variables.

Results and discussion

Since CHT calculations are highly time-consuming, only 7 cycles of the engine were run. As can be seen in Fig. 4 the experimental data available for this operating point shows a high cyclic dispersion. The traces are the result of 300 cycles performed with the experimental engine. While to model this cyclical dispersion more cycles should be simulated, the purpose of this work is to assess the heat exchange with the solid region. As there were no significant cycle to cycle differences in the solid wall temperature after the sixth cycle, the simulation was stopped. In Fig. 4 the experimental traces are compared with the simulated traces. It should be noted that with the combustion settings used in the simulation the pressure trace is underestimated with respect to the averaged cycle.

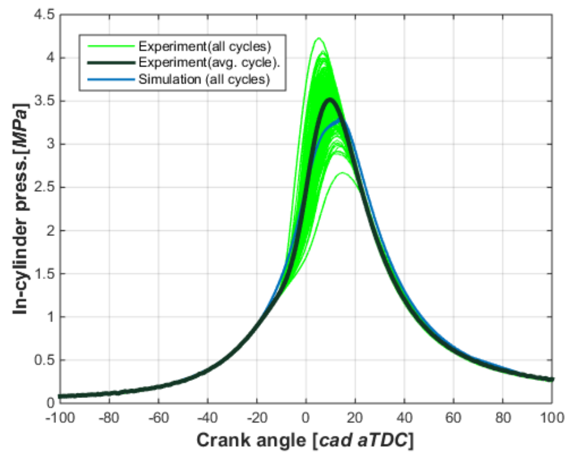


Figure 4. In-cylinder pressure.

The calibration of a gasoline engine model and the adjusting of the boundary conditions in the solid region becomes the study case in a multivariable problem. In order to better adjust model, it becomes necessary to carry out a multifactorial test with a sweep of the different variables involved and determine which ones are more relevant for the simulation. However, considering that the numerical pressure trace is within the experimental dispersion, and that further adjustment is computationally very time-consuming it was decided to do the CHT calculation with this combustion setting.

Fig. 5 shows the temperature distribution on the engine piston. As expected, the surfaces exposed to coolant oil are colder while those in contact with the combustion gas are hotter. It can also be seen that even though the heat transfer happens mainly in a unidimensional axial direction, the radial heat transfer has some effect on the thermal gradient.

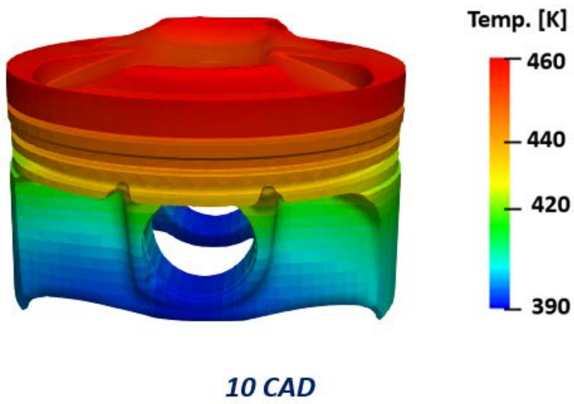


Figure 5. Temperature distribution on engine piston.

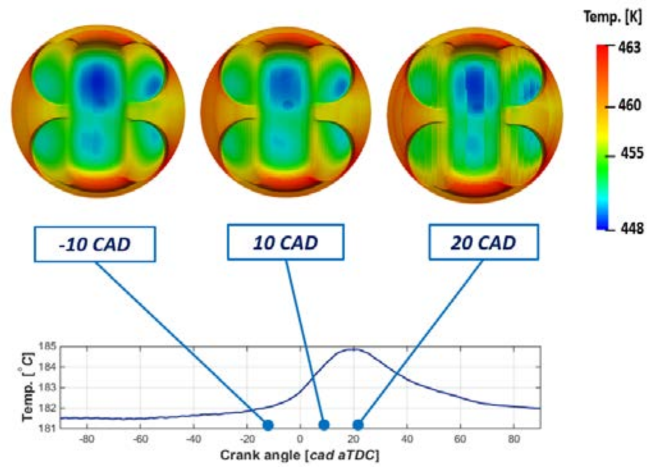


Figure 6. Temporal temperature evolution on the combustion gas exposed surface on the piston engine.

Though Fig. 5 shows a quasi-uniform distribution of temperature on the combustion gas-exposed surface of the piston, it is necessary to analyze this wall with more detail.

In Fig. 6 the temporal evolution of the wall temperature during the combustion is displayed. It seems that the lower part of the wall (on Fig. 6) is hotter due to the fact that the gas temperature near the wall is hotter in this region of the piston. This is seen in Fig. 7, which shows the gas temperature on the cell closest to the wall. The scale in Fig. 6 was reduced in order to better appreciate the temperature distribution on the surface. There is around 15 K of thermal gradient across the surface and the difference between the hotter side and the colder one is approximately 1 K. The coldest regions are at the center of the piston since cooling oil convection was considered under the surface. The periphery of the piston is hotter due to the boundary condition on the piston crown, which considers that it is in contact with the combustion gas. Therefore, the radial heat transfer is affecting the temperature distribution on the piston.

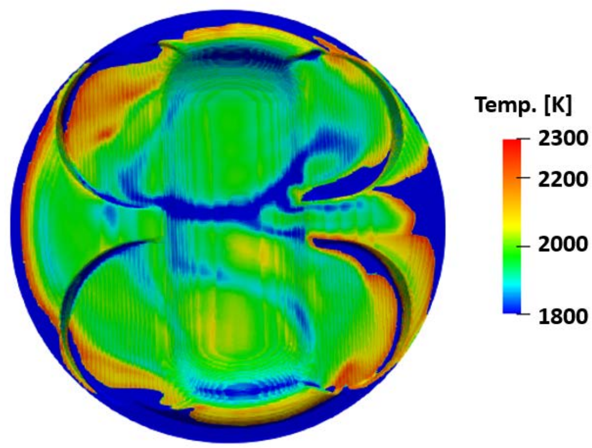
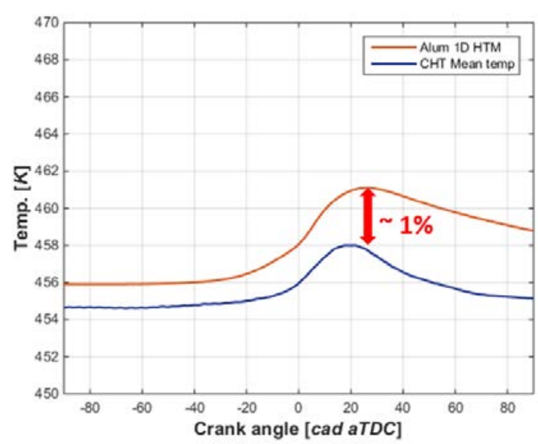


Figure 7. Near wall temperature on the combustion gas exposed surface on the piston engine.

Since no experimental measurements of the wall temperature are available, the results of the CHT simulation are compared with the in-house well-validated 1D model results. Figure 8 shows the comparison between the mean wall temperature on the gas-exposed surface of the piston with the mean temperature obtained with the 1D model. Not only does the temperature evolution obtained with the CHT follow a similar tendency to that of the 1D model, but it yields in addition very similar values of temperature, with a maximum difference of around 1 %. This confirms that the results achieved with CHT are reliable.



The temporal temperature evolution for different spatially located points is displayed in Fig. 9, where it can be appreciated that indeed, the hottest spots are located in the peripheral region.

The advantage with the CHT calculation is that the total heat transferred through the combustion chamber walls can be determined. As is shown in Fig. 10 there are significant differences between a reference CFD simulation with isothermal walls and the CHT calculation presented in this investigation.

Figure 8. Mean wall temperature on the piston surface.

While the percentages of total heat transfer through piston, liner and head are similar for both approaches, some clear differences can be observed. These values are in agreement with the available literature [35]: most of the heat is transferred to the head and to the piston. Regarding the CHT calculation, 33.5% is transferred to the piston, 26.7% to the liner and 39.8% to the head. The CFD equivalent calculation for this operation point (with fixed wall temperatures) on the other hand, shows that the percentages are 34.7%, 27.7% and 37.6 % to the piston, liner and head respectively. All in all, the relative difference in heat transfer prediction between the two approaches can reach up to 15%. For many applications, these differences are non-negligible, and thus the CHT approach demonstrated in this investigation should be preferred.

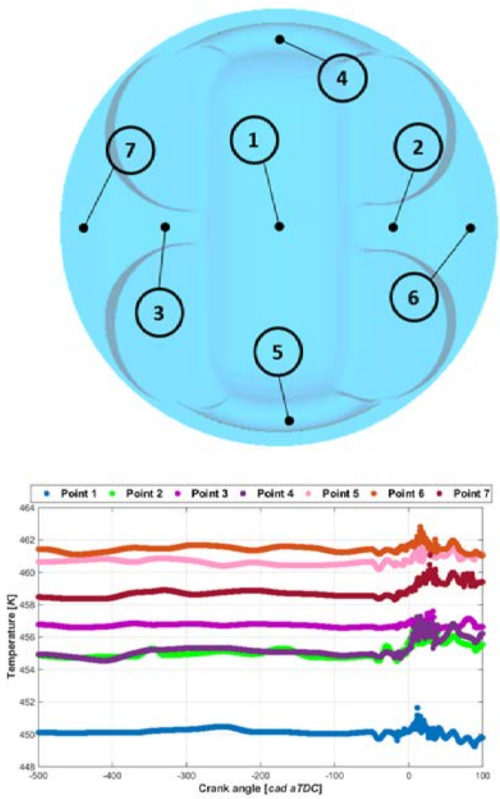


Figure 9. Temperature evolution of monitor on the solid piston surface.

Conclusions

This study presents the results of a CHT simulation applied to 4-stroke SI engine and the analysis of the heat transfer through the combustion chamber walls , as well as the wall temperature distribution of the solid region during the whole engine cycle. Due to computing cost considerations, the conjugate heat transfer model has been used only for the piston wall.

The validation has been carried out by comparing the mean wall temperature on the combustion gas exposed surface of the piston with the mean temperature obtained with a well-validated 1D heat transfer

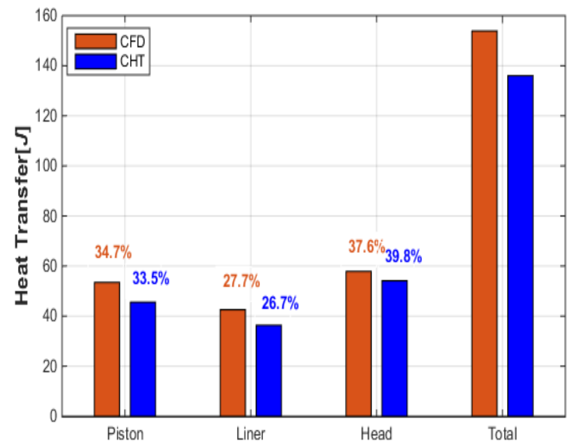


Figure 10. Total heat transferred to the combustion chamber walls.

model. Though both models are obviously very different in nature, the CHT calculated temporal evolution of the piston wall temperature follows a very similar tendency to that obtained with the 1D model. Furthermore, the average temperature level throughout the engine cycle shows only 1% maximum difference between both models.

One of the advantages of using the CHT simulation is that it provides very useful information concerning the local heat transfer through the walls of the solid parts, as well as the temporal evolution and spatial distribution of the surfaces temperature. Therefore, a model with well calibrated settings is able to determine properly the hot spot on the walls and how the surrounding environment impacts on the heat transfer.

In summary, the CHT calculation yield very good results in terms of heat transfer prediction during the whole engine cycle. However it is important to remark that it is required to set suitable boundary conditions on the solid regions, since they have a high impact on the temperature distribution in the solid parts of the engine.

Also, clearly there is some radial heat transfer that is never taken into account in 0D-1D models, nor in combustion CFD calculations, and this affects the surface temperature distribution. This in turn has an impact on the combustion process, that is not seen with isothermal CFD calculations.

Acknowledgments

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 724084.

The authors wish to thank IFPEN for their permission to use their single cylinder engine geometry and pressure results.

The authors want to express their gratitude to CONVERGENT SCIENCE Inc. and Convergent Science GmbH for their kind support for performing the CFD-CHT calculations using CONVERGE software.

References

1. M. Leguille, F. Ravet, J. Le Moine, E. Pomraning, K. Richards, P. K. Senecal, Coupled fluid-solid simulation for the prediction of gas-exposed surface temperature distribution in a SI engine, SAE Technical Paper 2011-24-0132.
2. A. Mohammadi, M. Yaghoubi, Estimation of instantaneous local heat transfer coefficient in spark-ignition engines, International Journal of Thermal Sciences 49 (7) (2010) 1309 – 1317.
3. A. Babajimopoulos, D. N. Assanis, D. L. Flowers, S. M. Aceves, R. P. Hessel, A fully coupled computational fluid dynamics and multi-zone model with detailed chemical kinetics for the simulation of premixed charge compression ignition engines, International Journal of Engine Research 6 (5) (2005) 497–512.
4. M. Fischer, X. Jiang, Numerical optimisation for model evaluation in combustion kinetics, Applied Energy 156 (2015) 793 – 803.
5. J. Xin, S. Shih, E. Itano, Y. Maeda, Integration of 3D combustion simulations and conjugate heat transfer analysis to quantitatively evaluate component temperatures, SAE Technical Paper 2003- 01-3128.
6. O. Iqbal, K. Arora, M. Sanka, Thermal map of an IC engine via conjugate heat transfer: Validation and test data correlation, SAE International Journal of Engines 7 (1) (2014) 366–374.
7. S. Lee, C. Bae, Design of a heat exchanger to reduce the exhaust temperature in a spark-ignition engine, International Journal of Thermal Sciences 47 (4) (2008) 468 – 478.
8. J. Kashdan, G. Bruneaux. Laser-Induced phosphorenscence of combustion chamber surface temperature on a single-cylinder Diesel engine. SAE Technical paper 2011-01-2049. JSAE20119103.
9. C. Knappe, M. Algotsson, P. Andersson, M. Richter, M. Tunér, B. Johansson, M. Aldén, Thickness dependent variations in surface phosphor thermometry during transient combustion in an HCCI engine, Combustion and Flame 160 (8) (2013) 1466 – 1475.
10. A. J. Torregrosa, P. Olmeda, B. Degraeuwe, M. Reyes, A concise wall temperature model for di Diésel engines, Applied Thermal Engineering 26 (11-12) (2006) 1320–1327.
11. A. J. Torregrosa, P. Olmeda, J. Martín, C. Romero, A Tool for Predicting the Thermal Performance of a Diésel Engine, Heat Transfer Engineering 32 (10) (2011) 891–904.
12. P. Kundu, R. Scarcelli, S. Som, A. Ickes, Y. Wang, J. Kiedaisch, M. Rajkumar, Modeling heat loss through pistons and effect of thermal boundary coatings in Diésel engine simulations using a conjugate heat transfer model, SAE Technical Paper 12016-01-2235.
13. P. K. Senecal, E. Pomraning, J. Anders, M. Weber, C. Gehrke, C. Polonowski, C. Mueller, Predictions of transient flame lift-off length with comparison to single-cylinder optical engine experiments, Journal of Engineering for Gas Turbines and Power 136 (11) (2014) 111505.
14. S. Som, D. Longman, S. Aithal, R. Bair, M. García, S. Quan, K. J. Richards, P. K. Senecal, T. Shethaji, M. Weber, A numerical investigation on scalability and grid convergence of internal combustion engine simulations, SAE Technical Paper 2013-01-1095.
15. Y. Pei, R. Shan, S. Som, T. Lu, D. Longman, M. J. Davis, Global sensitivity analysis of a Diésel engine simulation with multi-target functions, SAE Technical Paper 2014-01-1117.
16. P. Andruskiewicz, P. Najt, R. Durrett, S. Biesboer, T. Schaedler, R. Payri, Analysis of the effects of wall temperature swing on reciprocating internal combustion engine processes, International Journal of Engine Research 19 (4) (2018) 461–473.
17. G. Woschni, W. Spindler, K. Kolesa, Heat insulation of combustion chamber walls—a measure to decrease the fuel consumption of IC engines?, Tech. rep., SAE Technical Paper (1987).
18. H. Kosaka, Y. Wakisaka, Y. Nomura, Y. Hotta, M. Koike, K. Nakakita, A. Kawaguchi, Concept of “temperature swing heat insulation” in combustion chamber walls, and appropriate thermo-physical properties for heat insulation coat, SAE International Journal of Engines 6 (1) (2013) 142–149.
19. K. Fukui, Y. Wakisaka, K. Nishikawa, Y. Hattori, H. Kosaka, A. Kawaguchi, Development of instantaneous temperature measurement technique for combustion chamber surface and verification of temperature swing concept, in: SAE 2016 World Congress and Exhibition, SAE International, 2016.
20. F. Hartmann, S. Buhl, C. Hasse, P. Krost, M. Henke, W. Hübner, Erschließung von wirkungsgradpotentialen durch reduzierung der wärmeverluste mittels innovativer kolbenbeschichtungen, 16TH Conference,The Working Process of the Internal Combustion Engines. Graz, September 2017.
21. A. Broatch, P. Olmeda, X. Margot, J. Gomez-Soriano, Numerical simulations for evaluating the impact of advanced insulation coatings on H2 additivated gasoline lean combustion in a turbocharged spark-ignited engine. Applied Thermal Engineering 148 (2019) 674-683.
22. A. Broatch, P. Olmeda, X. Margot, J. Escalona, New approach to study the heat transfer in internal combustion engines by 3D modelling, International Journal of Thermal Sciences 138 (2018) 405-415.
23. J.F. Wiedenhoefer, R.D. Reitz, A multidimensional radiation model for Diésel engine simulation with comparison to experiment, Numerical Heat Transfer Part A 44 (7) (2003) 665–682.
24. E. Urip, K. H. Liew, S. L. Yang, Modeling IC engine conjugate heat transfer using the KIVA code, Numerical Heat Transfer, Part A: Applications 52 (1) (2007) 1–23.
25. Y. Li, S.-C. Kong, Coupling conjugate heat transfer with in-cylinder combustion modeling for engine simulation, International Journal of Heat and Mass Transfer 54 (11) (2011) 2467–2478.
26. M. M. Patil, A. Pise, N. Gokhale, Simulation of conjugate heat transfer (CHT) between engine head and cooling medium of Diésel engine, SAE Technical Paper 2015-01-1662.
27. A. Broatch, X. Margot, R. Novella, J. Gomez-Soriano, Impact of the injector design on the combustion noise of gasoline partially premixed combustion in a 2-stroke engine, Applied Thermal Engineering 119 (2017) 530–540.
28. Convergent science inc., CONVERGE 2.2 Theory Manual.
29. P. O’Rourke, A. A. Amsden, A particle numerical model for wall film dynamics in port-injected engines, SAE Technical Paper 961961.
30. A. Amsden, KIVA-3V: A Block-structured KIVA Program for Engines with Vertical or Canted Valves, Los Alamos, National Laboratory, 1997.
31. A. J. Torregrosa, A. Broatch, P. Olmeda, J. Martín, A contribution to film coefficient estimation in piston cooling galleries, Experimental Thermal and Fluid Science 34 (2) (2010) 142–151.
32. P. Olmeda, V. Dolz, F. Arnau, M. Reyes-Belmonte, Determination of heat flows inside turbochargers by means of a one dimensional lumped model, Mathematical and Computer Modelling 57 (7-8) (2013) 1847–1852.

33. A. Broatch, P. Olmeda, A. García, J. Salvador-Iborra, A. Warey, Impact of swirl on in-cylinder heat transfer in a light-duty Diésel engine, Energy 119 (2017) 1010–1023.
34. C. D. Rakopoulos, E. G. Giakoumis, D. C. Rakopoulos, Study of the short-term cylinder wall temperature oscillations during transient operation of a turbo-charged Diésel engine with various insulation schemes, International Journal of Engine Research 9 (3) (2008) 177–193.
35. A. Kikusato, K. Terahata, K. Jin, Y. Daisho, A Numerical Simulation Study on Improving the Thermal Efficiency of a Spark Ignited Engine---Part 2: Predicting Instantaneous Combustion Chamber Wall Temperatures, Heat Losses and Knock---, SAE International Journal of Engines 7(1) (2014) 87-95.