

# Integrated In-Cylinder / CHT Methodology for the Simulation of the Engine Thermal Field: An Application to High Performance Turbocharged DISI Engines

Giuseppe Cicalese  
R&D CFD SRL

Fabio Berni and Stefano Fontanesi  
Universita degli Studi di Modena

## ABSTRACT

New SI engine generations are characterized by a simultaneous reduction of the engine displacement and an increase of the brake power; such targets are achieved through the adoption of several techniques such as turbocharging, direct fuel injection, variable valve timing and variable port lengths. This design approach, called “downsizing”, leads to a marked increase in the thermal loads acting on the engine components, in particular on those facing the combustion chamber. Hence, an accurate evaluation of the thermal field is of primary importance in order to avoid mechanical failures. Moreover, the correct evaluation of the temperature distribution improves the prediction of pointwise abnormal combustion onset.

The paper proposes an evolution of the CFD methodology previously developed by the authors for the prediction of the engine thermal field, which is applied to two different high performance turbocharged DISI engines: the methodology employs both in-cylinder 3D-CFD combustion simulations and CHT (Conjugate Heat Transfer) simulations of the whole engine, inclusive of both the solid components and the coolant circuit. In-cylinder analyses are used as thermal boundary conditions for the CHT simulations, which are in turn a fundamental benchmark to evaluate the accuracy of the combustion heat flux estimation by means of a combination of global engine thermal survey and local temperature measurements.

A preliminary evaluation of some consolidated heat transfer models is carried out to evaluate the accuracy of the predicted gas-to-wall heat fluxes. Then, a modified heat transfer model is proposed, critically motivated and applied to the specific engine conditions under investigations. The proposed model strongly improves the predictive capability of the combined in-cylinder/CHT methodology in terms of both global thermal balance and pointwise temperature distribution for both the investigated engines.

**CITATION:** 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):2016, doi:10.4271/2016-01-0578.

## INTRODUCTION

Recent developments in internal combustion engines are strictly guided by fuel consumption reduction and the simultaneous increase of specific brake power.

These targets are achieved by means of different techniques, among which turbocharging, engine downsizing and down-speeding, complex fuel injection strategies, variable valve timing, variable port length [1,2]. These approaches are often jointly adopted and, for more complex configurations, hybridization is also introduced.

If, on one hand, this development pathway is shared between compression and spark ignition engines, on the other hand SI engines are those undergoing the most relevant evolution. For such engines, the increase of the specific power by means of downsizing and

turbocharging has to face with the occurrence of abnormal combustions, which can strongly limit the engine performance [3]. Moreover, the raise of the thermal loads acting on the engine components can reduce their mechanical strength and can deeply reduce the engine reliability because of thermo-mechanical failures. [4]

A correct estimate of the gas-to-wall thermal loads is then fundamental since the early design stages, where CFD and FE tools can be proficiently used to reduce time- and cost-to market. To increase the reliability of such numerical tools, it is of primary importance to accurately define the combustion heat transfer to the engine components. This can be derived from either dedicated experiments or from CFD-1D/3D simulations.

From an experimental point of view, it is not easy to properly isolate the thermal power acting on the components facing the combustion chamber. A direct evaluation of the instantaneous heat flux usually requires the adoption of complex, expensive and intrusive sensors [5,6] which cannot be easily used, for example, to evaluate the behavior of moving parts such as the piston and the valves. Furthermore, such measurements can be performed only in limited portions of the components, and the resulting total thermal power due to combustion has to be extrapolated in both space and time. Indirect methods, instead, are widely used in the automotive industry. They consist of a thermal load calculation by means of a thermal survey of the engine: during the experimental tests, all the thermal contributions are identified (heat to lubricant, heat to coolant, brake power, etc.) and this procedure allows to quantify the power globally acting on the engine components. During such tests, thermocouples can also be placed in the most critical engine locations in order to evaluate pointwise temperature distributions.

Once the heat amount is estimated, it can be used in CFD-CHT and FE tools to calculate the thermal field of the engine: it is split among each component (for example head, liner and piston) thanks to a combination of experimental evidences and user experience. The calculation can be assumed to be successfully performed if the resulting temperatures well match the experimental ones. It should be clear that such an approach relies on several approximations, among which heat flux subdivision between all the components and thermal load distribution, which can be either constant in space or distributed on the basis of physical assumptions and experimental evidences.

Assumptions and uncertainties can be reduced by using 3D-CFD tools to calculate the spatial and temporal evolution of the combustion process: this allows to estimate the amount and pointwise distribution of the thermal loads acting on the engine components. A more accurate representation of the engine thermal field is expected, which may be used for example to evaluate design criticalities leading to abnormal combustions or to thermo-mechanical failures.

The integration between 3D-CFD and FE tools aiming at well capturing the engine thermal field under actual operating conditions has been deeply investigated by the authors and several applications to currently made engines show the benefits of an interaction between the in-cylinder simulation framework and the CHT (i.e. Conjugate Heat Transfer) simulation framework [7,8,9,10]. In such interaction, a fundamental role is played by the capability of the in-cylinder model to accurately estimate the wall heat flux due to combustion. This in turn strongly depends on the formulation of the thermal wall function.

Many research groups tried to develop numerical models able to capture the actual heat flux through the combustion chamber boundaries, among which important contributions can be found in [11,12,13,14]. Two major classes of engine simulation models are available: phenomenological models and the CFD ones. In the former, semi-empirical formulations are used for the heat transfer evaluation. In the latter, a more detailed analysis of the heat transfer mechanisms is possible. Regardless the approach, at the present moment it is nearly impossible to identify a unique thermal wall

function formulation suitable for all the engine applications because of the very wide range of engine revving speeds, in-cylinder turbulence levels, engine IMEPs (and therefore loads).

This paper aims at proposing a valid approach for current production high performance DISI engines to be used in in-cylinder simulations in order to properly compute (in both magnitude and distribution) high-load heat fluxes acting on the engine components. Once the thermal load is correctly estimated, the CFD-CHT simulations are capable to well describe the engine thermal field. In the paper, such predictive capability is assessed by means of the comparison against both global engine thermal surveys and experimental temperature measurements for two different engines.

## ENGINE OVERVIEW

The investigated engines A and B are currently produced direct injection spark ignition (DISI) turbocharged units, V-shaped 8 cylinder and in line 4 cylinder respectively, and their main characteristics are reported in tables 1 and 2. The first engine is analyzed at two different operating conditions: 5000 rpm WOT (peak torque point) and 7000 rpm WOT (peak power point). The second is investigated just at peak power condition, 5200rpm WOT.

Table 1. Main features of the Engine A

|                      |           |
|----------------------|-----------|
| B/S                  | 1.05      |
| Operating Condition1 |           |
| Revving Speed        | 7000 rpm  |
| BMEP                 | 18.33 bar |
| Operating Condition2 |           |
| Revving Speed        | 5000 rpm  |
| BMEP                 | 24.36 bar |

Table 2. Main features of the Engine B

|                     |           |
|---------------------|-----------|
| B/S                 | 0.93      |
| Operating Condition |           |
| Revving Speed       | 5200 rpm  |
| BMEP                | 22.96 bar |

## CFD-CHT SIMULATIONS FOR ENGINE THERMAL FIELD CALCULATION

### General Guidelines

Internal combustion engine thermal field depends on several cross-linked factors: the point-wise temperature distribution is the result of the heat removal effectiveness of the coolant circuit, the total amount of wall heat flux and its spatial distribution over the engine components. Furthermore, the cooling effect due to the lubricating circuit and to the environment has to be taken into account in addition with the heat due to the frictions between the moving parts. In order to include all these phenomena and to well represent the engine thermal field, the authors developed an integrated in-cylinder / CHT approach [15] that is briefly recalled hereafter.

A preliminary simulation of combustion is performed by imposing a uniform wall temperature on the boundaries facing the combustion chamber. The resulting wall heat fluxes are then averaged over 720 °CA to obtain a cycle average point-wise thermal load distribution without losing important information about the most heated portions of the computational domain.

The heat fluxes are then applied as thermal boundary conditions in a CHT model where the solid components (head, block, gasket, valves, valve seats, valve guides, liners) are simulated together with the coolant circuit in order to properly evaluate both the engine temperature field and the heat removed by the coolant. The adoption of cycle averaged heat fluxes relies on the hypothesis that the thermal inertia of the solid components facing the combustion chamber is too high to be sensible to cycle-resolved thermal loads.

The wall temperatures resulting from this framework are then mapped on the boundaries facing the combustion chamber. The incylinder simulation is then repeated with a point-wise temperature distribution, which deeply influences the heat exchanged through the walls. Again, the obtained wall heat fluxes are time-averaged and they are used to calculate the engine temperature field by means of the CHT model. This data exchange between the different frameworks is required to be performed at least twice in order to meet the convergence in terms of temperatures calculated by the CHT model.

CHT simulations are fundamental to consider the mutual effect between the coolant circuit and the solid regions. The thermal field of the engine components depends on the cooling effectiveness and on the thermal load; the three elements are strictly linked to each other: once the cooling system operating conditions are defined and the thermal load is imposed, the numerical thermal field has to match the experimental one.

### CHT Model and Numerical Setup

The computational domain includes the coolant circuit and all the surrounding solid components. Moreover, the valves, the valve seats and the valve guides are added to well capture the heat transfer between the different elements. Each solid component is characterized by its own thermal conductivity and a contact resistance is applied between adjacent region to evaluate temperature discontinuities. A schematic representation of the computational domain is represented in [Figure 1](#).

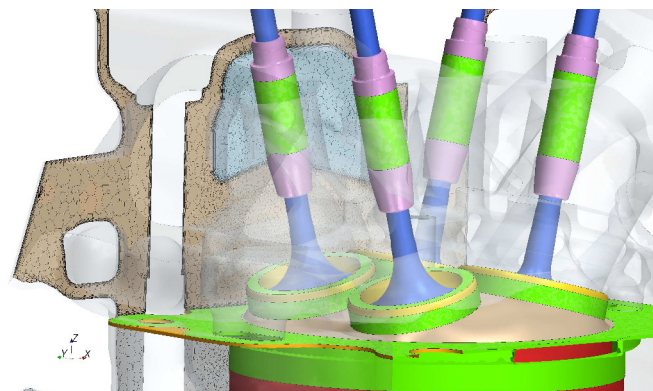


Figure 1. A representation of the computational domain

Pistons are not included in the computational domain, but only the heat exchanged with the cylinder liners is considered, as it will be better explained later in the paper.

Because of the large extent of the computational domain, it is fundamental to limit the number of cells and to speed up the calculation: a polyhedral core mesh and a multi-layer prismatic mesh adjacent to the walls are used to discretize the fluid region, and this approach is suitable for the adopted turbulence model.

On the solid side a multi-layer approach at the walls is chosen to better represent the temperature gradients.

The boundary conditions are time-invariant, i.e. the heat fluxes are not cycle-resolved because of the thermal inertia of the solid components. Furthermore, the moving elements are placed in a fixed position (i.e. valves) or not explicitly included in the computational domain (i.e. pistons). However, a transient solver is needed due to the adoption of a multiphase boiling model to take into account the phase transition in the coolant circuit and the related local increase in heat transfer.

The boundary conditions are derived both from numerical and experimental data: coolant mass flow rates and temperatures to the different inlets/outlets reproduce the actual test-bench operating condition. The effects of the lubricating circuit and of the external environment are represented with a pair of properly defined heat transfer coefficient and reference temperature.

The heat fluxes applied on the boundaries facing the combustion chambers and on the intake and exhaust ports come from in-cylinder simulations, whose setup will be accurately described in the following paragraphs. The thermal loads are point-wise distributions of a cycle-average combustion simulation and they are directly applied to the combustion dome and to the intake and exhaust ports. In order to take into account the revolution of the valves, the heat fluxes are separately space averaged over the faces and the stems. The valves are placed in the closed position and the heat transfer to the seats accounts for the lack of contact during a portion of the engine cycle by means of a contact resistance.

Since the piston is not included in the domain, the point-wise heat flux distribution on the liner is increased adding the contribution of the heat transfer from the piston skirt and rings and the quote of the friction acting on the cylinder due to the piston motion. Such contributions are separately analyzed as described in previous works of the authors [16].

A time-averaged heat flux distribution is imposed on the piston crown. Frictions due to the skirt and to the rings are evaluated based on the in-cylinder pressure trace, and they are split between the piston and the liner. The quote acting on the piston is added to the heat exchanged through the rings, which is mimicked by a pair of thermal resistance and reference temperature derived from [17] and calculated as a resistance network, as depicted in [Figure 2](#) and [Figure 3](#).

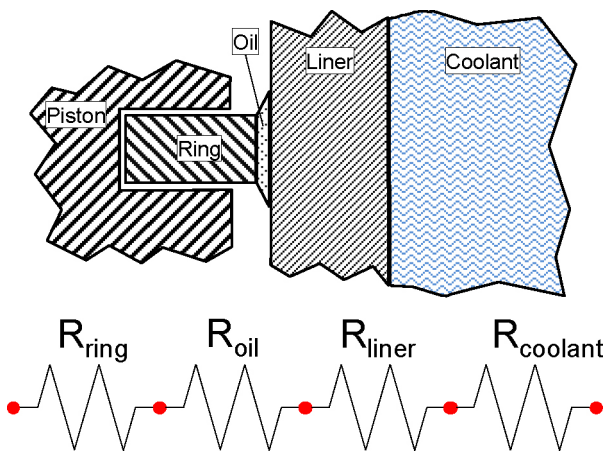


Figure 2. Scheme of the resistance network between the piston groove and the coolant circuit

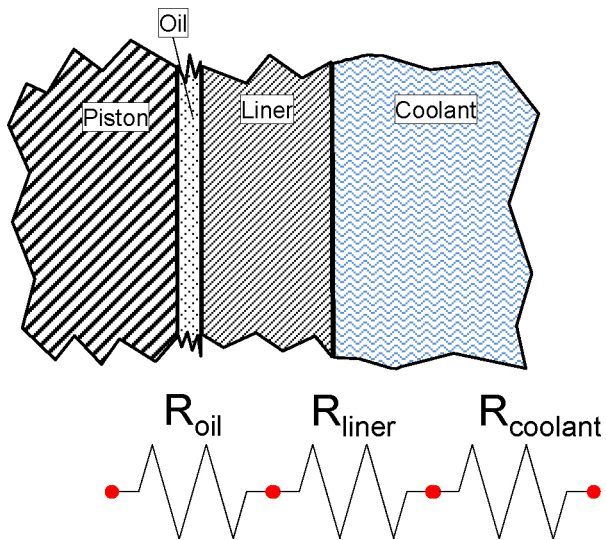


Figure 3. Scheme of the resistance network between the piston skirt and the coolant circuit

The effects on the piston underside are represented with a properly calculated pair of heat transfer coefficient and reference temperature. In particular, to account for the heat removal due to the oil jets acting on the piston underskirt, a heat transfer coefficient distribution similar to that proposed in [18] is applied. In particular, it is modified to consider the relative velocity between the piston and the oil jet, which strongly influences the removed heat amount. Furthermore, the motion of the stagnation point going from TDC to BDC is taken into account. A typical heat transfer coefficient distribution is shown in Figure 4.

A schematic representation of the piston boundary conditions is reported in Figure 5, where all the effects are simultaneously taken into account.

The resulting heat fluxes exiting from the rings and from the skirt are then added to the friction portion and to the combustion loads acting on the cylinder liner. Each contribution due to a complete engine cycle is applied only on the cylinder liner portion affected by it, as depicted in Figure 6, where all the thermal loads acting on the cylinder liner are reported.

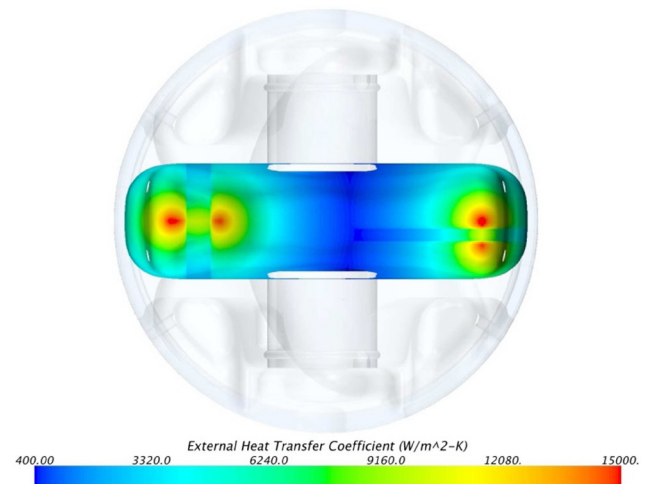


Figure 4. Heat transfer distribution on the underskirt portion of the piston

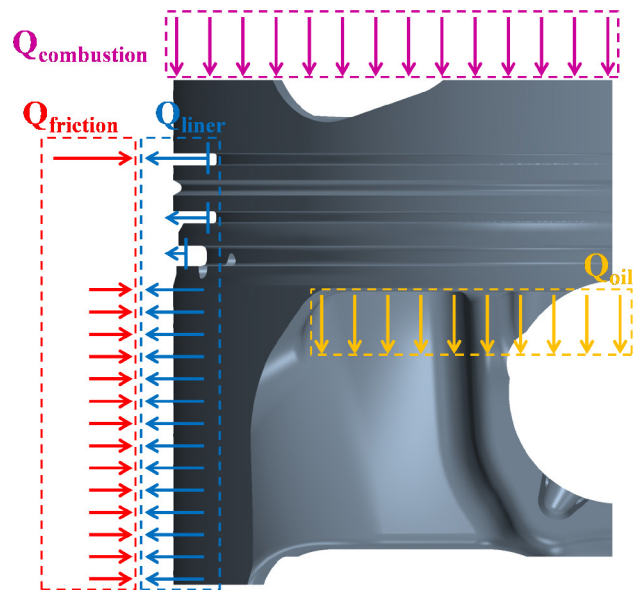


Figure 5. Piston thermal boundary conditions

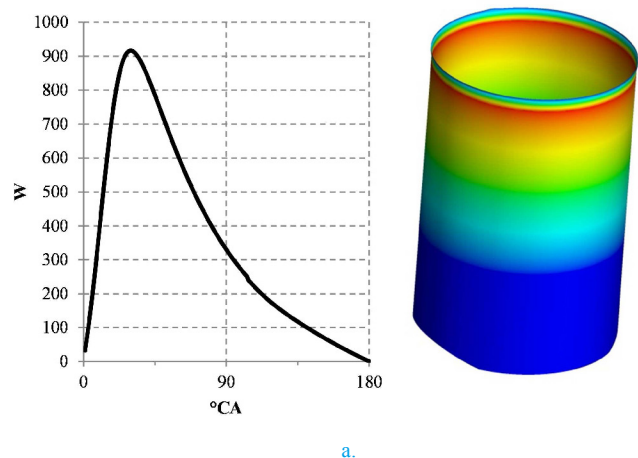


Figure 6. a) Heat flux due to the piston rings friction; b) heat flux due to the piston skirt friction; c) heat transfer from the piston to the liner

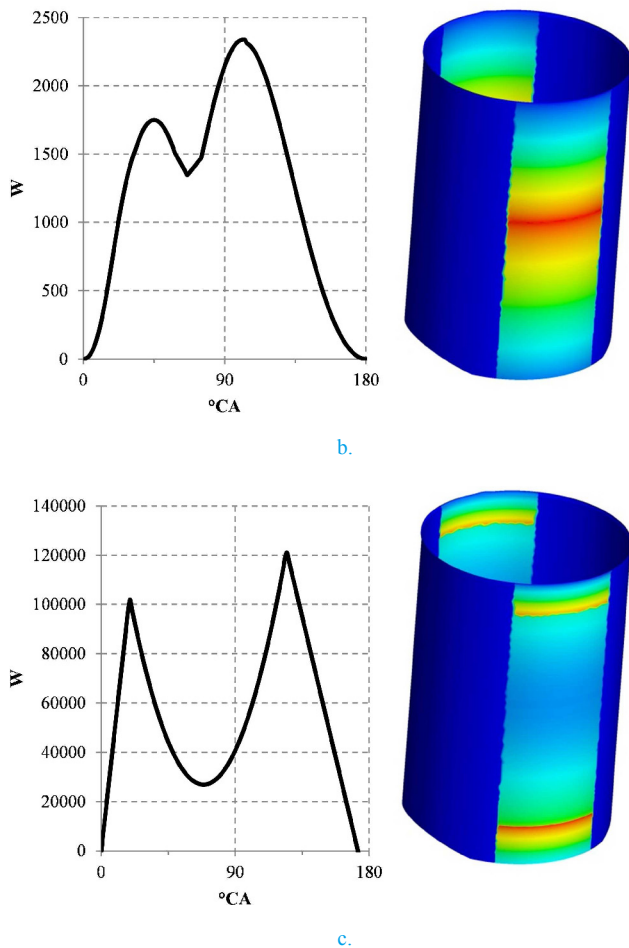


Figure 6. (cont.) a) Heat flux due to the piston rings friction; b) heat flux due to the piston skirt friction; c) heat transfer from the piston to the liner

The resulting thermal load is the sum of all the aforementioned contributions and the cycle-averaged combustion heat fluxes, as schematically depicted in Figure 7.

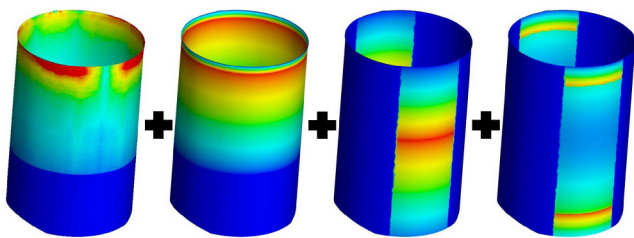


Figure 7. Thermal loads on the cylinder liner. From the left to the right side are reported the heat fluxes due to: the combustion, the piston rings, the piston skirt, the conduction from the piston.

### Methodology Limitations

The simulation results for both Engine A and Engine B are compared with experimental temperatures, which are measured by means of several thermocouples placed in the most interesting locations of the engines such as between the exhaust ports, in the peripheral portions of the combustion dome and along the liners in different angular positions, aiming at well capture the engine thermal status.

Calculated temperatures versus experimental ones are depicted in Figures 8, 9, 10 and 11 for all the analyzed operations. The simulated engine thermal field is always over-predicted for both engine head and engine block, and this is due to the overestimation of the wall heat fluxes calculated by the in-cylinder simulations.

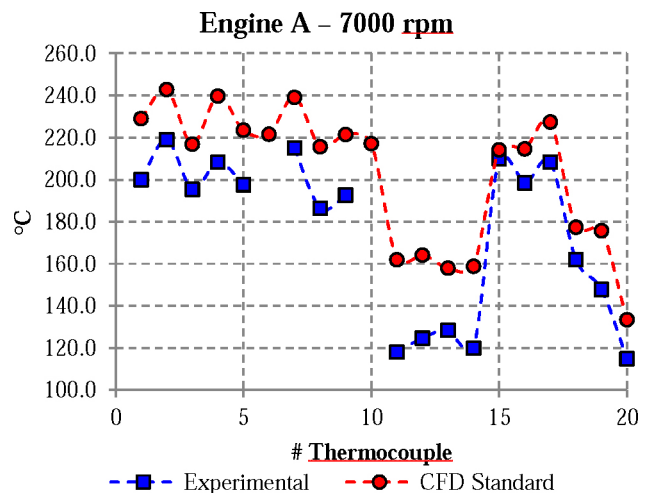


Figure 8. Experimental vs numerical temperatures for engine A @ 7000 rpm

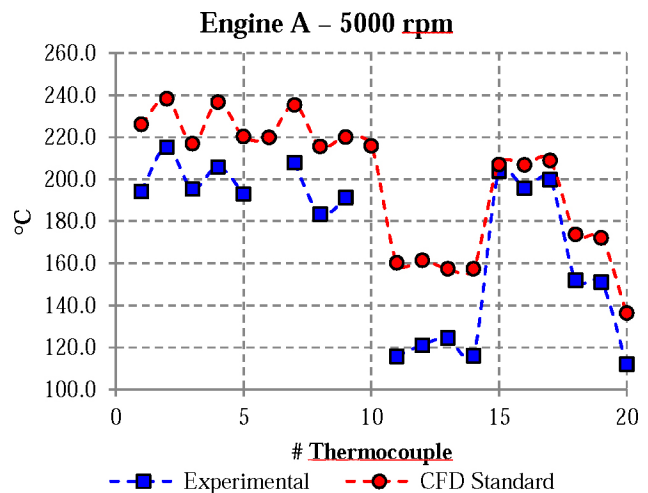


Figure 9. Experimental vs numerical temperatures for engine A @ 5000 rpm

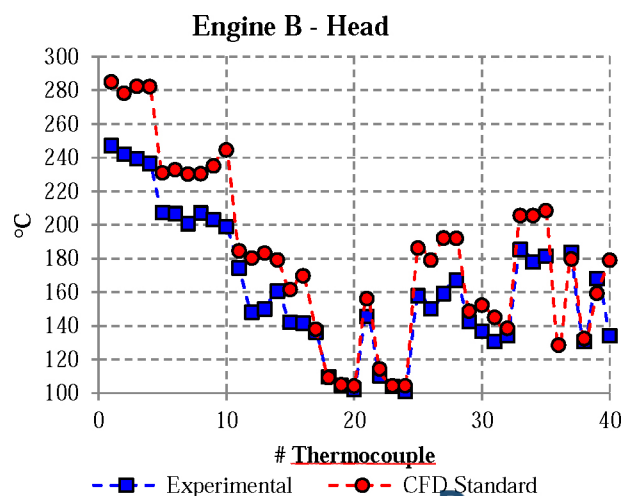


Figure 10. Experimental vs numerical temperatures for engine head B

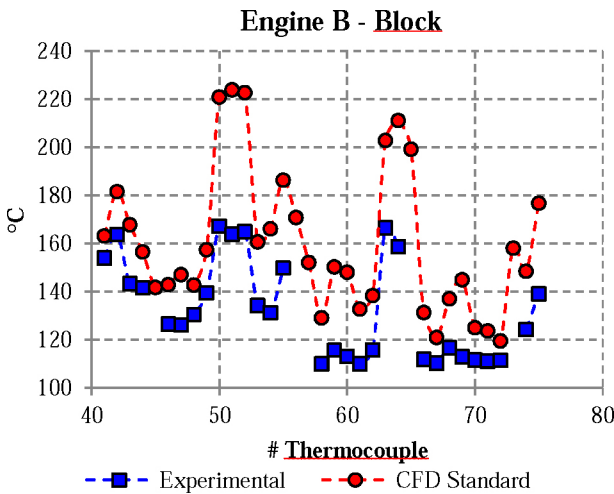


Figure 11. Experimental vs numerical temperatures for engine block B

Furthermore, this critical behavior is confirmed by the comparison between the calculated coolant temperature rise and the experimental one. Starting from these evidences the need to critically analyze the thermal wall function formulations adopted for the in-cylinder simulations emerges; in fact, they are the main suspect for the wall heat flux over-prediction. To identify the most suitable thermal law of the wall, the experimental data deriving from the available engine thermal surveys are used, as explained in the following paragraph.

## ENGINE THERMAL SURVEY

From a general point of view, an engine at a fixed operation can be analyzed as a steady-state system where energy enters and exits. The chemical energy stored into the fuel is converted into thermal energy (which in turn heats up the lubricating and the coolant circuits and increases the exhaust gas enthalpy) and into indicated work.

The most important role in terms of heat removal is played by the coolant circuit, whose layout for the analyzed engines is schematically reported in Figure 12.

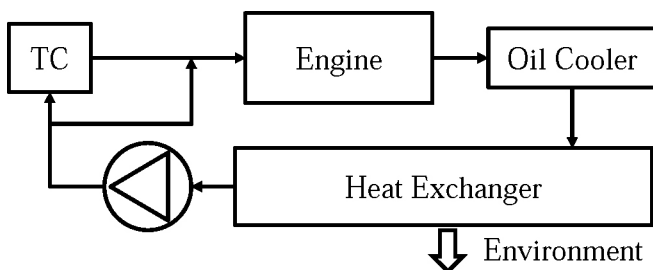


Figure 12. Coolant circuit simplified layout

The coolant surrounds the turbocharger, then enters the engine and finally cools down the lubricating circuit. All the thermal power is then restituted to the environment by means of a heat exchanger.

It is possible to estimate the heat transfer pertaining to each portion of the circuit by placing ad-hoc thermocouples at the inlets and outlets of each element.

In the CHT computational domain just the portion of the coolant circuit represented by the block “engine” in Figure 12 is included. Thanks to the many data which are usually collected during the experimental tests it is possible to correctly identify the heat removed by the coolant circuit when it passes through the engine galleries. This amount is almost totally due to the wall heat flux related to the combustion, so it can be used to identify the correct thermal wall function able to evaluate the thermal loads acting on the components facing the combustion chamber.

From the thermal balance reported in Figure 13 it is possible to reconstruct the thermal load acting on the walls due to combustion, as explained in (1):

$$\dot{Q}_{comb} = \dot{Q}_{coolant} + \dot{Q}_{env} + \dot{Q}_{lubr} - \dot{Q}_{fric} - \dot{Q}_{surr} \quad (1)$$

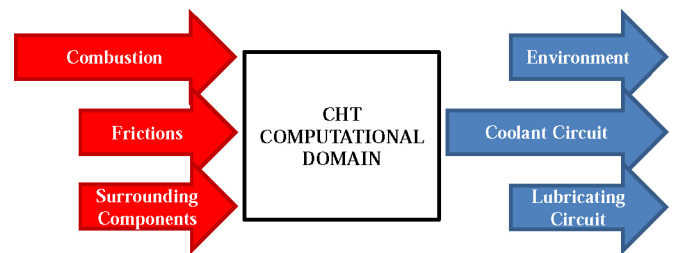


Figure 13. Engine thermal balance

Only the term  $\dot{Q}_{coolant}$  reported in the right side of (1) is strongly affected by a variation of  $\dot{Q}_{comb}$  because the coolant circuit surrounds the combustion chamber and removes the largest amount provided to the walls by the combustion. The other terms in (1) can be assumed to be almost constant and can be evaluated from the operating conditions (i.e. the frictions and the heat transfer from surrounding components) or from preliminary CHT simulations (the contribution of the environment and of the lubricating circuit).

The resulting target wall heat fluxes derived from the integration between the data deriving from the thermal survey of each operation and the data deriving from preliminary CHT calculations are reported in Table 3.

Table 3. Engine thermal balance

|                   | Coolant Circuit | Lubricating Circuit | Environment | Surrounding components | Frictions | Combustion Target | Cylinder Target |
|-------------------|-----------------|---------------------|-------------|------------------------|-----------|-------------------|-----------------|
|                   | kW              | kW                  | kW          | kW                     | kW        | kW                | kW              |
| Engine A 7000 rpm | 114.36          | 1.20                | 1.49        | 4.00                   | 4.38      | 108.67            | 13.58           |
| Engine A 5000 rpm | 85.68           | 1.00                | 1.00        | 2.00                   | 3.25      | 82.43             | 10.30           |
| Engine B          | 80.20           | 0.85                | 2.00        | 43.90                  | 2.68      | 36.47             | 9.12            |