

Thermal Map of an IC Engine via Conjugate Heat Transfer: Validation and Test Data Correlation

Owais Iqbal and Kunal Arora
Chrysler Group LLC

Manyam Sanka
ESI North America Inc.

ABSTRACT

Accurate numerical prediction of an engine thermal map at a wide range of engine operating conditions can help tune engine performance parameters at an early development stage. This study documents the correlation of an engine thermal simulation using the conjugate heat transfer (CHT) methodology with thermocouple data from an engine operating in a dynamometer and a vehicle drive cell. Three different operating conditions are matched with the simulation data. Temperatures predicted by simulation at specific sections, both at the intake and the exhaust sides of the engine are compared with the measured temperatures in the same location on the operating engine.

CITATION: Iqbal, O., Arora, K., and Sanka, M., "Thermal Map of an IC Engine via Conjugate Heat Transfer: Validation and Test Data Correlation," *SAE Int. J. Engines* 7(1):2014, doi:10.4271/2014-01-1180.

INTRODUCTION

Conjugate heat transfer (CHT) is a highly time efficient and cost effective simulation technique of generating the detailed thermal map of an IC engine. Because of accurate temperature prediction, CHT has become not only an integral part of the product development process but also a diagnostic tool for any post-production engineering issues related to the engines. CHT computational model of an IC engine includes the solid part of the engine partial assembly (the engine block, cylinder head, valves, guides and seats) and the fluid filled cavities (coolant jacket). Historically, engine thermal analysis is performed at peak power, wide-open throttle (WOT) operating condition corresponding to the worst-case scenario for the metal and coolant temperatures in the engine. But, now the engine thermal analysis activities are expanding into scenarios where engine efficiency and performance parameters could be improved upon at a wide range of engine operating conditions. In this regard, a comprehensive correlation of numerically generated engine thermal map via CHT simulation with test data is desirable for a range of engine operating conditions.

In-cylinder combustion is the primary heat (and mechanical power) source in an IC engine. Because of the nature of the thermodynamic cycle that an IC engine operates on, once the mechanical power is extracted, excess heat needs to be rejected in a timely manner to avoid unwanted thermal loading, which may result in durability or sealing issues. Typically, the desired rate of heat rejection is achieved via a blanket of coolant around the combustion chamber and surrounding

region. This blanket, known as the coolant jacket, is topologically the most complex path of the coolant loop. The optimization of the engine coolant distribution within the coolant jacket is a complete study in itself. However, for an optimum coolant distribution within coolant jacket, it is a pre-requisite to obtain as accurate a temperature distribution as possible. In this regard, there are essentially two ways to predict the distribution of heat flux generated and the resulting temperature distribution as a result of combustion: One method is an extensive instrumentation of an engine to physically collect the data; the other method is to use computational science to predict the heat flux numerically.

Literature regarding temperature data obtained via testing or experimentation for an IC engine is scarce mostly because of the applied nature of the study. Also, proprietary concerns associated with the underlying industry, makes it even more difficult to find documented methodologies. Since the focus of the current study is primarily computational, therefore the literature review is limited to studies documenting the numerical conjugate heat transfer.

Early efforts in the engine heat transfer research are focused on the development of effective convective heat transfer coefficients [1, 2, 3]. The correlations developed in those studies are only applicable to simplified engine cycle simulation, and those correlations result in increased errors when applied to multi-dimensional engine simulations. Later advances, fluid-only combustion simulations are carried out

and the resulting wall heat flux was then used for the heat conduction simulation for a specific engine component. In this approach, typically, the cycle averaged heat flux from combustion simulation was used. This was termed 'the decoupled approach'. Kleemann et. al. [4] and several other studies [5, 6, 7, 8] demonstrated correlation between the predicted wall temperatures and test data using the decoupled approach.

More recently, there have been studies using the Conjugate Heat Transfer (CHT) approach aimed at predicting the thermal field within an engine by solving fluid and solid computational domains simultaneously. The temperature at the fluid-solid interface is computed through thermal coupling of two domains within the same code. Despite the inherent differences in the orders of magnitude in time scales associated with heat transfer in the fluids and the solids, these studies are carried out with in-cylinder gases and engine solid walls. Egel Urip [9] documented the coupled approach of evaluating heat transfer between combustion gases and solids within the same code. It was concluded that the best way to transfer combustion load between in-cylinder gases and solids is by using the convective conditions, i.e., heat transfer coefficients (HTC) coupled with near wall gas temperatures (T_{gas}) rather than the heat flux itself.

Another set of studies consists of conjugate heat transfer analysis between the solid parts of an IC engine and the engine coolant. These studies are limited in the literature. In this approach the heat source, whether obtained from test data, 1-D, or 3-D combustion simulation, is first averaged over the engine cycle before applying into the CHT model. This methodology allows steady state simulation of conjugate heat transfer, hence reducing the simulation time while generating detailed thermal field in an engine. There is a recent study reported by Stefano et al. [10] which used the heat source from both 1-D and 3-D combustion code in conjugate heat transfer simulation for a high performance engine. Cycle averaged heat flux is applied to the boundaries exposed to combustion and exhaust gases. Based on the results reported in the comprehensive study, it was clearly demonstrated that conjugate heat transfer analysis helped generate temperature distribution predictions which correlated well with the test results. Correlation included coolant temperature rise, pressure drop across the engine and point measurement of temperature at a few locations in the cylinder head.

The current study undertakes the task of generating the temperature distribution in the engine block and cylinder head via conjugate heat transfer analysis at a few representative engine load points. The technique involved conjugate heat transfer analysis where the momentum equation was solved for coolant flow in the coolant cavities, and the energy equation was solved for fluid and solid domains together. With the recent advancements in computational hardware and software, heat loads for an IC engine is obtained from a multi-dimensional in-cylinder flow and combustion simulation with detailed chemical kinetics using SAGE solver [11]. The convective heat transfer coefficients (heat load) are computed in such a way

that the effective heat transfer at a particular location is equal to the integrated heat fluxes at that location over the complete engine cycle. Initial combustion simulation is carried out using a set of uniform wall temperatures for various boundaries of the computational domain. Time averaged HTCs and T_{gas} values are then mapped to the CHT model. The first iteration of in-cylinder flow and combustion simulation used assumed constant wall temperatures. For the second iteration, the wall temperatures for liners, combustion domes etc. based on CHT calculations are mapped for in-cylinder flow and combustion simulation. Based on the updated wall temperatures, updated cycle averaged convective conditions are mapped back to CHT calculations. This iterative process is continued till a converged wall temperature and cycle averaged convective conditions are obtained between two simulations.

OBJECTIVES

The objective of the current study is to document the systematic validation of CHT simulation against test data at representative operating conditions of an IC engine. Coolant inlet temperatures, ambient conditions and oil temperatures are used as input to CHT simulation. The resulting thermal field of the engine are validated and compared against the temperature data measured at the discrete probe locations.

MODEL PREPARATION

It is important to mention that the objective is to generate the cycle-averaged thermal field in all the main components of the engine. Therefore, the computational model (for CHT) is built mainly with non-moving components. These include three major components, namely; engine block, cylinder head and the head gasket between them. Both the engine block and the cylinder head have coolant cavities included in the model. Intake and exhaust valves (along with valve-seats and valve-guides) are the only moving components included in the model. Both valves are modeled at their closed position to establish the additional heat conduction route from the valves to the cylinder head. It has been concluded by a separate in-house study (not reported here) that heat conduction between the engine block and the cylinder head is mainly through the head gasket, but, is limited to a small region surrounding the combustion chamber. This region is named 'the fire ring' for the analysis. The head gasket is therefore simplified to a small region surrounding the combustion chamber and the coolant transfer holes (about 10 % of the total block deck surface area). [Figure 1](#) shows half of the computational domain. Note that the cylinder head includes valves, valve-seats and valve-guides.

Engine block, cylinder head and head gasket are all meshed individually in StarCCM+ using polyhedral cells. Cell sizes used in the study are established via separate grid dependency study. The finer prism layers are created in the fluid cavities to accurately capture the near wall turbulence characteristics using the two-layer all y^+ treatment. This is a hybrid approach that seeks to recover the flow and turbulence characteristics in

the limit of very fine and very coarse meshes. After assembly of all three sub-domains, final cell count for the engine model typically adds up to 8 million. The polyhedral mesh and the prism layers in the fluid cavities are shown in figure 2. Appropriate material properties are assigned to each of the regions involved in the simulation. Materials include various different solids and 50% dilute solution of ethylene glycol which is used as coolant. Coolant inlet and outlet boundaries are assigned with a mass flow rate (corresponding to specific engine load point) and pressure, respectively. No boundary condition is assigned at the fluid-solid interface as temperatures are computed through thermal coupling of two domains within the same code.

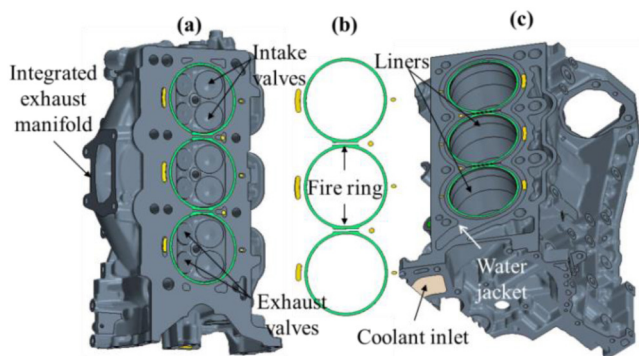


Figure 1. Half of the computational domain; (a) Cylinder head, (b) Head gasket, and (c) Engine block

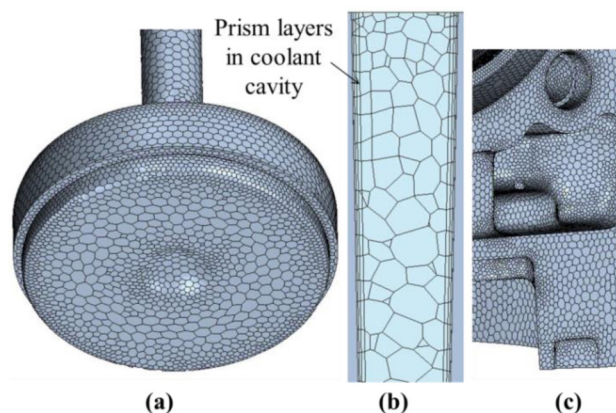


Figure 2. Polyhedral mesh in solid and coolant cavities (a) Exhaust valve, (b) Coolant cavity, (c) Part of engine block

Solid domain boundaries exposed to the ambient temperature and engine oil are assigned the appropriate convective boundary conditions (obtained from test data). These boundary conditions along with the coolant inlet and the outlet conditions changes for each engine load point studied. However, material properties of the solids remain the same for the test cases studied. Figure 3 shows a list of those boundaries of the computational domain. Note that heat conduction through the interface between two solid components is modeled using contact resistance. The magnitude of the contact resistance between non-moving parts is usually lower than the resistance

offered by the two contacts which are dynamic in nature. Contact resistance values (or magnitude) are calibrated based on best practice.

ASSUMPTIONS

The following assumptions are made in this simulation:

- Material properties are considered isotropic.
- In order to reduce the mesh count, part of the gasket is modeled along with the block and cylinder head.
- Pseudo steady state Rohsenow boiling model available in StarCCM+ [12] is used.
- Contact resistance between moving and non-moving parts is considered constant.
- Valves are kept in the closed position to produce worst case scenario.
- Heat load in the form of the convective heat transfer coefficients and the associated gas temperatures are obtained from the standalone 3-D combustion simulation for each engine load point studied.
- Fasteners, bearings, pistons, and connecting rods are not modeled.

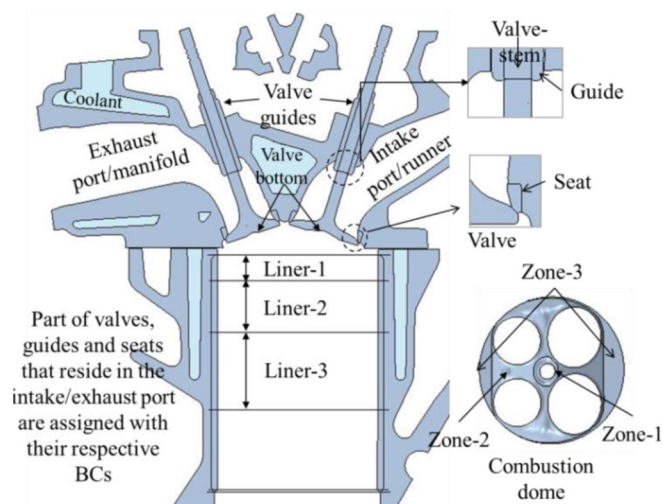


Figure 3. Section-cut through engine block and cylinder head showing boundaries of solid domain where convective heat transfer coefficients have been applied.

COMPUTATIONAL MODEL

The computational model is constructed as described in the previous section. In order to set up the model for conjugate heat transfer analysis, boundary conditions are required. This section describes how the convective heat transfer coefficients are obtained via 3D combustion simulation.

Figure 3 shows a section cut of a single cylinder model used to perform the detailed combustion analysis. The combustion simulation domain is extended at both inlet (intake side) and outlet (exhaust side) to avoid solution divergence due to reverse flow. The input condition at intake runner and exhaust