

Critical Aspects on the Use of Thermal Wall Functions in CFD In-Cylinder Simulations of Spark-Ignition Engines

Fabio Berni

Universita' di Modena e Reggio Emilia

Stefano Fontanesi

Universita degli Studi di Modena

Giuseppe Cicalese and Alessandro D'Adamo

Universita' di Modena e Reggio Emilia

ABSTRACT

CFD and FE tools are intensively adopted by engine manufacturers in order to prevent thermo-mechanical failures reducing time- and cost-to market. The capability to predict correctly the physical factors leading to damages is hence essential for their application in the industrial practice. This is even more important for last generation SI engines, where the more and more stringent need to lower fuel consumption and pollutant emissions is pushing designers to reduce engine displacement in favor of higher specific power, usually obtained by means of turbocharging. This brings to a new generation of SI engines characterized by higher and higher adiabatic efficiency and thermo-mechanical loads.

A recent research highlighted the different behavior of the thermal boundary layer of such engines operated at high revving speeds and high loads if compared to the same engines operated at low loads and revving speeds or even engines with a lower specific power. This means that CFD heat transfer models proposed and validated in the past decades on these last may not be predictive when applied to high specific power engines. This is why an alternative formulation was proposed in a previous work, for the estimation of the heat transfer in in-cylinder CFD simulations of high performance turbocharged SI engines.

Nevertheless, for both the proposed alternative thermal wall function and the other ones available in literature, there are essential limitations due to the dimensionless distance y^+ . In fact, even if heat transfer models provide a further formulation for low y^+ (viscous sub-layer), industrial practice seldom makes use of turbulence models enabling the integration up to the wall and low Reynolds approaches are even less used because of computational costs. Therefore in the present paper the authors aim to analyze critically the use of thermal wall functions along with high Reynolds turbulence models for the prediction of heat transfer in CFD in-cylinder simulations for different values of y^+ .

CITATION: Berni, F., Fontanesi, S., Cicalese, G., and D'Adamo, A., "Critical Aspects on the Use of Thermal Wall Functions in CFD In-Cylinder Simulations of Spark-Ignition Engines," *SAE Int. J. Commer. Veh.* 10(2):2017, doi:10.4271/2017-01-0569.

INTRODUCTION

SI engine design of the last two decades was characterized by a massive use of CAE tools such as CFD and FE in order to reduce the huge costs of experimental development and testing. Simultaneously, increasingly severe environmental regulations forced engine manufacturers to a continuous effort towards lower fuel consumption [1] and pollutant emissions. To this aim, one of the most popular choice was engine down-sizing and down-speeding [2]. However, such technique brings to an increase of the specific power which has to be limited because of the risk of abnormal combustion onset and thermo-mechanical failures. On one hand, knock can be suppressed by means of the reduction of the engine spark advance (thus lowering the engine performance) and, more recently, by the additional

injection of water and/or methanol [3,4]. On the other hand, investigations carried out in [5, 6, 7] show that in order to prevent thermo-mechanical damages, 3D-CFD in-cylinder simulations and CHT analyses can be adopted, coupled with FE tools, to evaluate the thermal field and subsequent thermo-mechanical stresses acting on the engine, which in turn may drive design improvements with shorter times and lower costs than the experimental tests. A correct estimation of an engine thermal field depends tightly on the gas-to-wall heat fluxes which represent, hence, a fundamental boundary condition for CHT models. Furthermore, heat transfer is a major player for the overall engine efficiency, exhaust emissions, charge mixing, flame front development and knock onset prediction [8,9]. In CFD in-cylinder simulations.

Gas-to-wall thermal loads can be assessed from either dedicated experiments or 1D/3D-CFD simulations. As for the first, the thermal power acting on the components facing the combustion chamber cannot be experimentally measured without efforts: a direct evaluation of instantaneous heat flux usually requires the adoption of complex, expensive and intrusive sensors [10,11] and measurements can be performed only in limited portions of the components. As a consequence, the resulting total thermal power due to combustion has to be extrapolated in both space and time. Therefore, engine manufacturers prefer to rely on indirect methods such as the heat transfer estimation by means of a thermal survey of the engine. Test bench experiments allow to measure the global amount of thermal power acting on the engine components and, once estimated, it is split among each part (e.g. head, liner and piston) thanks to a combination of experimental evidence and user experience. Finally they are used in CFD-CHT and FE tools to calculate the engine thermal field. Unfortunately, such kind of approach relies on several approximations, e.g. the global heat transfer split between all the components and its spatial distribution on each of them, which can be either constant in space or variable based on physical assumptions and experimental evidences. In order to reduce such uncertainties, 1D/3D-CFD tools may replace experimental measurements. In particular many 1D/3D heat transfer models have been proposed aiming to estimate reliably the heat transfer through the combustion chamber walls. Two major groups of heat transfer models are identified by Rakopoulos in [12]: the phenomenological and the 3D-CFD ones. As for the former, even if the proposed empirical formulations aim to be universally applicable, they should expect to be valid only if new constants are adopted when engine design varies, due to the relative simplicity of such formulations, as admitted by Woschni in [13] commenting his correlation. As for the latter, the proposed formulations try to investigate deeply the physical mechanisms of heat transfer, providing more reliable estimations if compared to the phenomenological one. This is the reason why the 3D-CFD approach currently represents the most adopted solution.

Researchers have proposed formulations for years, aiming at capturing the actual heat flux through the combustion chamber boundaries. Despite the huge number of 3D-CFD heat transfer models that can be found in literature, none seems to be suitable for all kinds of engines (CI and SI) and operating conditions (low and high revving speeds, partial and full loads). In fact in [7], the authors proved that heat transfer models widely available in CFD commercial codes behave correctly when applied to the well-known GM pancake engine test case, while they manifest evident shortages when used for highly-charged / highly-downsized spark ignition engines, operated at high revving speed. Overestimations of the wall heat fluxes were pointed out thanks to experimental engine thermal surveys and temperature measurements on a set of current production engines. Thus, in order to overcome such deficiencies, an alternative heat transfer model was proposed and motivated (based on the temperature gradients characterizing the boundary layer of current production engines rather than the pancake engine), whose heat fluxes allowed to match both the experimental engine thermal survey and local temperature measurements for all the investigated engines and operating conditions. Therefore, the proposed heat transfer model was demonstrated to be a valid tool to be used in in-cylinder

simulations of current production turbocharged SI engines operated at medium-to-high loads and revving speeds, in order to obtain reliable convective heat fluxes.

Such model as well as all the others analyzed in [7] offers the chance of modeling convective heat transfer in a RANS framework by means of a wall function, in order to limit the computational effort without loss of accuracy. As Nuutinen highlights in [14], in commercial CFD software there are two kinds of approaches: in the first one the near-wall profiles of velocity and temperature are spatially resolved thanks to a refined computational grid along with a low-Reynolds number or even a two-layer turbulence models. In the second one a wall function with a high-Reynolds number turbulence model is used to capture accurately the flow and the heat transfer in the proximity of the walls, avoiding the huge number of near-wall cells proper of the previous approach, thus limiting the computational cost. This is the reason why, despite the high resolution of a low-Reynolds approach which should provide the most accurate results, the adoption of an analytical wall function is still the most common solution in the industrial practice.

However, as pointed out by Kays and Crawford in [15] about isothermal or quasi-isothermal flows, the validity of the wall function is limited to a small range of y^+ values. This is commonly accepted also for the non-isothermal ones. In other words, restricting attention to the thermal law of the wall, the dimensionless distance from the wall should belong to an interval ranging from ~ 30 to ~ 300 . In fact the thermal law of the wall proposed in [15] is proved to reproduce faithfully the experimental dimensionless temperature profile only in such range. Two important remarks should be made here. Firstly the proposed limits are commonly accepted but often lower values for the upper one are accounted for (e.g. $150 \div 200$). Moreover, as highlighted in [15], the upper limit varies with the Reynolds number (based on enthalpy thickness). Secondly, heat transfer models commonly available in commercial CFD software provide a further dimensionless temperature profile (the exact boundary condition) to cover the y^+ range below 30, which is useful to model heat transfer by means of a low Reynolds approach. In fact, if on one hand this treatment requires the first centroid of the near-wall grid to be at y^+ lower than 3, on the other hand it requires a formulation of the thermal law of the wall valid for such dimensionless distance.

Although the heat transfer models are provided with a thermal law of the wall covering a wide y^+ domain, the dimensionless distance should fall within its prescribed range, based on the adopted wall treatment (either low-Reynolds or high-Reynolds), throughout the simulation. For example, if a high Reynolds approach is adopted, y^+ should assume values higher or equal to 30 during the calculation as requested by the high-Re number turbulence model, which is not able to account for the wall effects appearing in the viscous sub-layer ($y^+ < 30$). Unfortunately, as for 3D-CFD in-cylinder simulations, y^+ values can be seldom maintained within the range previously discussed. In particular, for part-load and low revving speed operating conditions, y^+ is usually lower than 30 during the entire engine cycle, unless the near-wall cell centroid is placed at a huge distance from the wall. This is possible only for simple domains such as the GM pancake engine analyzed in the next paragraph. As for high load and high revving speed operations, y^+ is within the range only for limited

portions of the engine cycle which depend, in turn, on the considered boundary. In fact, while flame deck, liner and piston provide sufficiently high values of the dimensionless distance only during the second half of the intake stroke, compression and the first half of the power one, the exhaust port wall exhibits y^+ values over the lower limit only during the exhaust stroke.

The aim of the present paper is the analysis of the heat transfer for different operating conditions. In particular the authors underline the importance of having y^+ values respecting the requirements of the adopted turbulence model. The paper consists of two sections. In the first one, the GM pancake engine test case is deeply investigated, as a representative of the part-load and low revving speed operating points, in order to evaluate the heat transfer prediction in case of y^+ values lower than the turbulence model requirement. In the second part, high load and high revving speed operating conditions are analyzed. In particular, the heat transfer prediction on a current production turbocharged SI engine operated at such conditions is investigated. Finally, numerical heat fluxes are proved to fail compared to experimental evidence despite the correct range of y^+ , if heat transfer models usually available in commercial codes are adopted. The use of the heat transfer model proposed by the authors in [7] allows obtaining reliable heat fluxes according to experimental measurements.

PART-LOAD / LOW REVVING SPEED OPERATING CONDITIONS

Engine Overview

In this section the General Motors pancake engine is considered as a representative of part-load and low revving speed operating conditions. Several heat transfer models were validated against this test case, considering also the Angelberger one which is used for the proposed simulations. The engine is a four stroke, V-8 (but modified to operate with one cylinder), spark-ignition and fueled with a premixed air-propane mixture. It is characterized by a disk-shaped combustion chamber with a centrally mounted spark plug and its main geometrical features are reported in Table 1. Experimental tests conducted by Alkidas and Meyers [10,11] provide point-wise wall heat flux measurements for both fired and motored operating conditions, which can be used to compare numerical results. In particular, data are available at four probes located on the cylinder head (here referred to as HT1 to HT4) and on the liner (referred to as HT5). The radial distances of probes HT1, HT2, HT3 and HT4 from the spark plug are 18.7, 27.5, 37.3 and 46.3 mm, respectively. On the liner, HT5 is located 6.3 mm below the head, as visible in Figure 1.

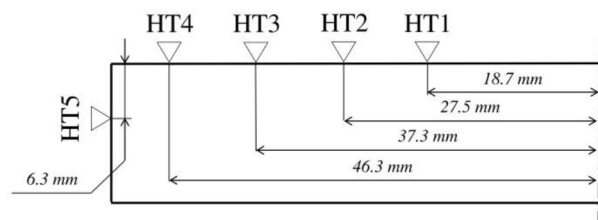


Figure 1. Probe location on pancake engine head and liner

Similarly to many studies available in literature [12, 16, 17, 18, 19], the gas exchange is not modelled: simulations start at 117°CA bFTDC (corresponding to IVC) and end at 80°CA aFTDC, as no information are available regarding the open-valve portion of the cycle.

Table 1. Geometrical features.

<i>Pancake Engine</i>	
Bore	105 [mm]
Stroke	95.25 [mm]
Connecting Rod Length	158 [mm]
Compression Ratio	8.56

Numerical Set-Up

A customized version of STAR-CD v4.22 licensed by Siemens is used to carry out the simulations which are run in a RANS framework. Different turbulence models are adopted based on the wall treatment: for a high-Reynolds approach, which requires wall functions to model near-wall velocity and temperature profiles, the $k-\epsilon$ RNG turbulence model is adopted in a modified version able to account for gas compressibility, since it is particularly suitable for the application to internal combustion engines [20]. As for the low-Reynolds wall treatment, which expects an integration of the equations up to the wall in order to solve both the velocity and the temperature profiles, the $k-\omega$ SST turbulence model is preferred. In [21], Menter observes the severe limitations of the ϵ -equation in the near-wall zone. In such region, compared to ϵ , the adoption of an equation for the turbulent frequency (ω) has significant advantages: reduction of numerical stiffness, more accurate estimation of the turbulent length scale in adverse pressure gradient flows (leading to improved wall shear stress and heat transfer prediction) and a lower near-wall resolution requested. Moreover, in order to avoid the main deficiency of the standard $k-\omega$ model, i.e. the strong sensitivity of the solution to free stream values for ω outside the boundary layer [22], the SST [23] model is adopted. This is a combination of the $k-\omega$ near the wall and the $k-\epsilon$ away from the wall.

Given the axisymmetric geometry, the computational grid consists of a thin 1° wedge provided with periodic boundary conditions, to reduce the computational effort, as visible in Figure 2.

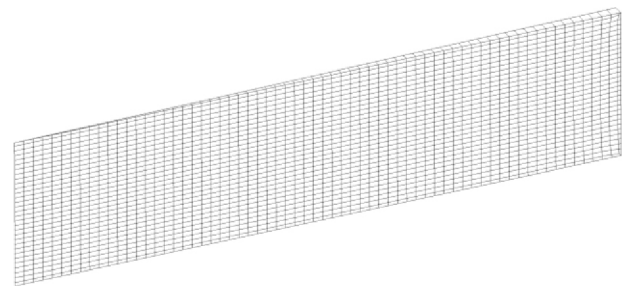


Figure 2. Pancake engine computational domain, corresponding to a 1° wedge

This is necessary mostly for Low-Reynolds near-wall treatment cases where the fine near-wall grid requires very brief time-steps (down to 1e-7÷1e-6 s) in order to maintain the Courant number in the order of 1e-2÷1e-1. The total number of fluid cells is about 4000÷7000 (based on the wall treatment) at 117°CA bFTDC and 2000÷4500 at FTDC.

ECFM-3Z [24] is used as combustion model, as adopted widely in previous publications by the authors [7]. The Angelberger heat transfer model available in STAR-CD package and in many commercial codes is used [16]. Gas properties such as specific heat and thermal conductivity are polynomial function of temperature, while molecular viscosity varies with temperature following Sutherland's law.

Operating Conditions

Information dealing with the operating points is reported in Table 2 as well as initial and boundary conditions.

Table 2. Operating parameters and initial and boundary conditions.

Parameters		
	Fired	Motored
Engine Speed	1500 [rpm]	1500 [rpm]
Equivalence Ratio	0.87	-
Volumetric Efficiency	40%	40%
Spark Timing	27 CA bFTDC	-
Initial Conditions		
TKE (k)	84 [m^2/s^2]	84 [m^2/s^2]
Length scale (L)	0.93 [cm]	0.93 [cm]
Pressure	0.821 [bar]	0.843 [bar]
Temperature	449 [K]	453 [K]
Axial Velocity w [z-direction]	Decreasing linearly from the piston velocity (at the surface of the latter) to 0 (at the cylinder head)	Decreasing linearly from the piston velocity (at the surface of the latter) to 0 (at the cylinder head)
Radial Velocity u, v [x,y-directions]	Swirl Ratio=1	Swirl Ratio=1
Mass Composition	Y _{C₃H₈} = 4.6 % Y _{O₂} = 18.76 % Y _{N₂} = 73.70 % Y _{CO₂} = 1.89 % Y _{H₂O} = 1.04 %	Air
Boundary Conditions		
Wall Temperatures	T _{Head} = 445 K T _{Liner} = 405 K T _{Piston} = 445 K	T _{Head} = 380 K T _{Liner} = 380 K T _{Piston} = 380 K

As for the fired case, the reported values are widely discussed in literature [16, 17, 19]. Data for the motored operating point are less available. In particular, the initialization of turbulent kinetic energy (k), length scale (L) and velocity field are supposed by the authors to be equal to the fired case. This hypothesis is acceptable considering that the engine speed as well as the volumetric efficiency do not change. As for the initial temperature, the reported value is found in [12] and it is similar to that of the fired case. Initial pressure is calculated by the authors based on initial temperature, volumetric efficiency and mixture composition, which is now consisting of pure air.

Results

A matrix of cases reported in Table 3 is proposed for both the fired operating condition and the motored one (except for Case 1, which is performed only for the motored condition). High-Reynolds wall treatment cases are simulated by means of various grids characterized by different near-wall layer sizes, in order to cover a wide range of y^+ values. As for the low-Reynolds approach, two different grids are used: a finer one with 16 layers, a ratio between them of 1.3 and a thin 0.0025 mm height first layer, and a coarser one with 12 layers, a ratio between them of 1.2 and a thin 0.01 mm height first layer.

Table 3. Simulations run for both the fired and motored operating condition.

Case	Turb. Model	Wall Treatment	Near-Wall Grid
1	k-ε RNG	High-Reynolds	h=0.01mm 1 Layer
2	k-ε RNG	High-Reynolds	h=0.05mm 1 Layer
3	k-ε RNG	High-Reynolds	h=0.15mm 1 Layer
4	k-ε RNG	High-Reynolds	h=0.3mm 1 Layer
5	k-ε RNG	High-Reynolds	h=1mm 1 Layer
6	k-ε RNG	High-Reynolds	h=1.5mm 1 Layer
7	k-ω SST	Low-Reynolds	h=0.01mm 12 Layer
8	k-ω SST	Low-Reynolds	h=0.0025mm 16 Layer

As for the fired operating condition, first the combustion model (ECFM-3Z) as well as the ignition one are tuned, thanks to the available experimental pressure trace. The comparison between numerical CFD and experimental data is reported in Figure 3. Also the in-cylinder pressure for the motored case is reported. Even if experimental data are not available for such operating condition, the reported curve appears very close to the numerical one proposed by Reitz in [18].

Numerical and experimental data are compared in terms of heat transfer in Figure 4. Numerical heat fluxes are evaluated at the same position where the experimental sensors HT1 to HT5 (mentioned above) are located. Moreover, because of its relevance for the simulation of both flow and thermal fields, y^+ is monitored at the same locations and reported in Figure 5.

Numerical and experimental heat fluxes are compared also for the motored case in Figure 6, with the exception that for such operating condition experimental data are available just for the probes located on the cylinder head. Even for the motored case, y^+ is monitored and shown in Figure 7.

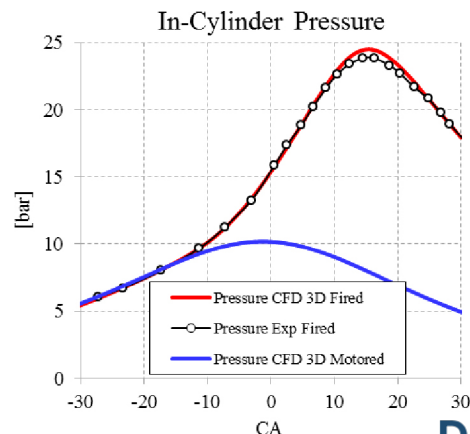


Figure 3. Comparison between numerical and experimental in-cylinder pressures for fired and motored cases. 0° CA is the BDC.

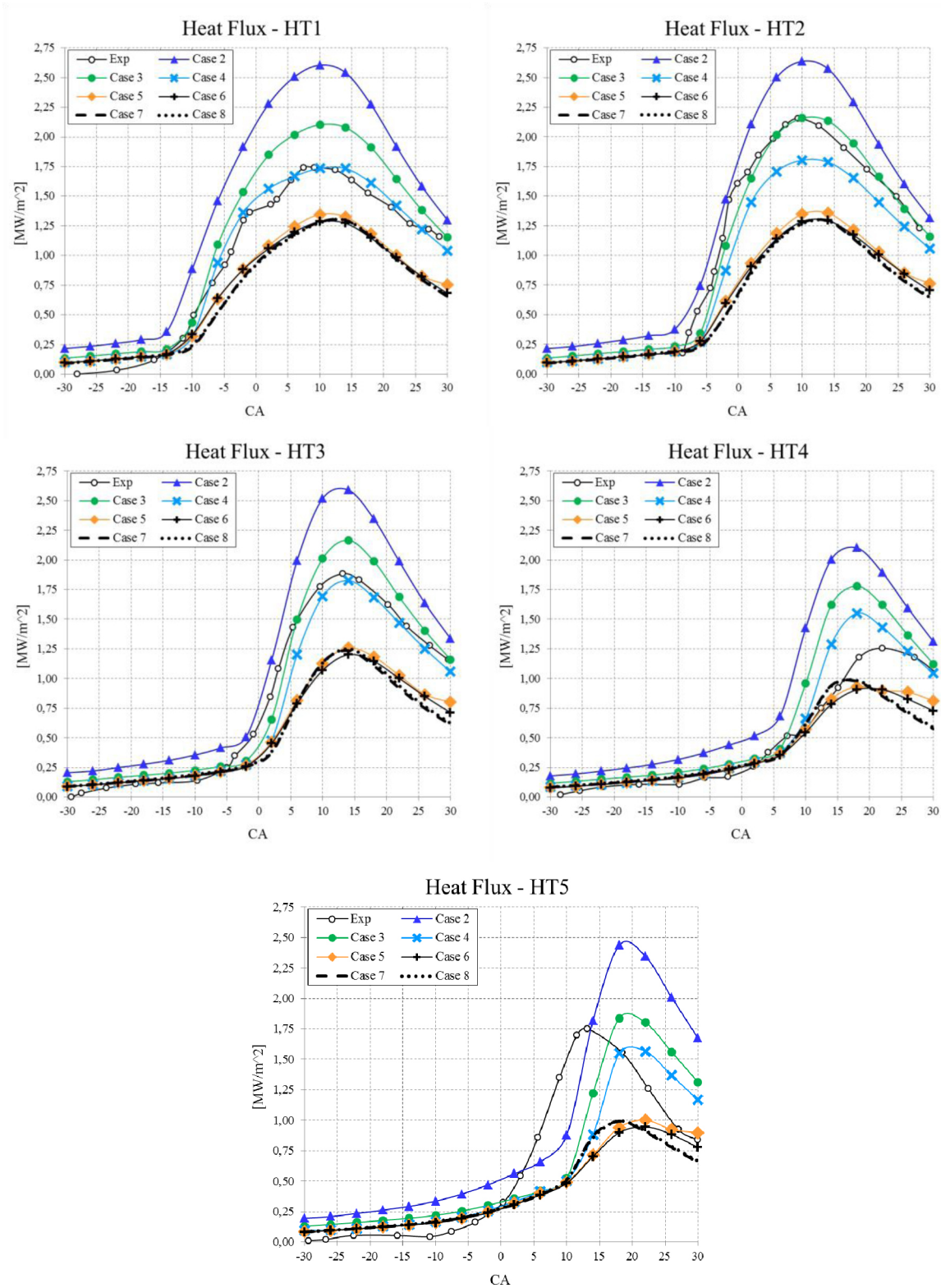


Figure 4. Numerical and experimental heat fluxes at HT1 to HT5 sensors for the fired operating condition.