CFD-FE SIMULATION FOR THE OPTIMIZATION OF A CYLINDER PRESSURE SENSOR APPLICATION IN A LARGE GAS ENGINE

Kober M.(1)*, Schubert-Zallinger C.(2), Wimmer A.(2), Milošević I.(3)
*Author for correspondence
(1)LEC GmbH, Graz, Austria
(2)Graz University of Technology, Austria
(3)Montanuniversität Leoben, Austria
E-mail: martin.kober(at)lec.tugraz.at

ABSTRACT

This paper reports on the development of a methodology for designing an on-board pressure sensor and its application to a large bore gas engine cylinder head. A simulation combining CFD and FE calculations is set up and supporting measurements are carried out on a single cylinder engine to gain data for calibration of the model and validation of the simulation results. In addition to the crank angle resolved pressure data for the combustion chamber, the measurements include component temperatures in the cylinder head, on-board sensor and the piston, as well as the heat fluxes into the fire deck and the sensor.

Based on this experimental data, various submodels of the CFD calculations are investigated, including heat transfer wall models, the nucleate boiling module for the cooling jacket, and the influence of mesh refinements. Special attention has been given to the impact of the combustion model adjustment on the resulting heat transfer. In the FE model, contact conditions between various components, for example valves and valve seat rings, are investigated thoroughly.

INTRODUCTION

Modern lean gas burn engines typically operate in a very narrow operating window between knocking and misfiring. Establishing cylinder pressure regulated control for this type of engine is a promising approach that guarantees the highest achievable efficiency and simultaneously fulfills stringent emission regulations. The on-board pressure sensor providing the data for the engine control must be not only nearly as accurate as a laboratory sensor but also significantly more durable.

To achieve the longest service life possible, it is important not only how the sensor is designed but also how it is integrated into the cylinder head. Along with the position of the sensor in the combustion chamber (location, offside installation), the type of connection (e.g., shieldings, screw connections) and the areas adjacent to the cylinder head (cooling jacket) are also critical. This type of optimization work can be conducted with a simulation model as long as the model correctly reproduces actual trends in changes in the areas under investigation. Confidence in model predictability increases when the model provides calculation results that largely correspond to measurement data for several configurations. Moreover, a well calibrated simulation model can provide reliable boundary conditions for subsequent calculation steps that would otherwise be difficult to generate from measurements alone. For example, the temperature distribution in the sensor and the area around it must be as accurate as possible in order to obtain a good estimate of the service life of the sensor membrane.

This paper will provide a report of the development of such a simulation model. It will describe parts of the model that contribute to a general improvement in the quality of the model as well as parts that still limit its performance.

NOMENCLATURE

CA	[deg CA]	Crank Angle
HTC	[W/m²K]	Heat Transfer Coefficient
RoHR	[J/deg CA]	Rate of Heat Release
TKE	[m²/s²]	Turbulent Kinetic Energy
y+	[-]	Dimensionless Wall Distance
CFD		Computational Fluid Dynamics
ECFM		Extended Coherent Flame Model
FE		Finite Element Analysis
FTDC		Firing Top Dead Centre
LEC		Large Engines Competence Center
SCE		Single Cylinder Engine

MODEL SETUP

The entire simulation model consists of three main parts: CFD combustion chamber calculation, FE component calculation and CFD cooling jacket calculation. While CFD calculations were conducted using AVL FIRE v2014.2, the FE model was created in Abaqus/CAE 6.14-3.

The CFD model on the gas side includes the combustion chamber as well as the intake and exhaust ports of a cylinder. The boundary conditions such as boost pressure, back pressure, charge air temperature, air mass and fuel mass were calculated at different load points on a single cylinder test engine at the LEC. The rate of heat release was computed from the measured cylinder pressure curve using 0D engine cycle analysis and provides a basis of comparison for combustion calibration in CFD calculation. The ECFM [1] was used to model combustion. Calculations are made over an entire cycle of 720°CA starting at the opening of the exhaust valve. The time increment varies according to the current cycle of 0.1°CA to

1°CA; the model has about 2.6 million cells at top dead centre. To link it to the FE software, the fluid temperatures near the wall and the heat transfer coefficients of all the walls with which the fluid has contact are recorded. In the mapping process, this data is averaged over the time of one cycle and provided to the steady state FE calculation as spatially dependent boundary conditions [2].

The FE model contains the cylinder head and the parts installed directly into it such as the valves, the prechamber unit and the on-board sensor. Neighboring parts such as the valve train or the exhaust pipe are not included and must be replaced by appropriate thermal boundary conditions. Because of the relatively large number of components, over 100 component contacts had to be modeled. Temperature dependent material parameters - heat conductivity in particular - were able to be used for a good share of the materials installed. The entire FE model consists of around 800,000 elements.

FE calculation provides boundary conditions in the form of spatially dependent wall temperatures for both CFD calculations (combustion chamber and cooling jacket), which are processed in a Python script and read by a FIRE mapping module. In contrast to combustion chamber calculation, the CFD cooling jacket calculation is drawn up in a steady state. The boiling module [3] of the CFD software is activated for this application. As a boundary condition for the FE calculation, a transfer coefficient from boiling effects is displayed in this calculation along with the fluid temperature near the wall and the convective heat transfer coefficient. This transfer coefficient is added to the convective share in a post-processing step. The CFD and FE calculations are now conducted iteratively until changes in the exchanged boundary conditions are negligible. Several runs are necessary between the cooling jacket calculation and the component calculation because boiling is strongly dependent on the spatial wall temperature. However, it results in a strong cooling off of this region. The number of required iterations can be reduced by working with a damping function in order to process the boundary conditions from the cooling jacket calculation.

CFD CALCULATION MESH

One general challenge with CFD calculations is the influence of the calculation mesh on the model and the results. This section assesses the influence of the mesh on the calculated heat transfer to the components. To this end, the sizes of the cells in the entire volume of the main combustion chamber were varied during the combustion phase.

The length of the edge of a combustion chamber cell is 1.125mm with the coarse mesh while it is only around 0.5625mm with the fine mesh. The meshes were generated with an automatic mesher in the CFD software. Although the intake and outlet ports are not affected by this measure, the number of cells more than doubles at top dead center. Accompanying the finer resolution of the cells, the time increment was also reduced during the combustion phase. The influence of cell size becomes apparent in the calibration of the combustion model. To reproduce the process of energy release through combustion in a similar manner [Figure 1], stretching factors with the fine mesh were set significantly lower than with the coarse mesh [Figure 2].

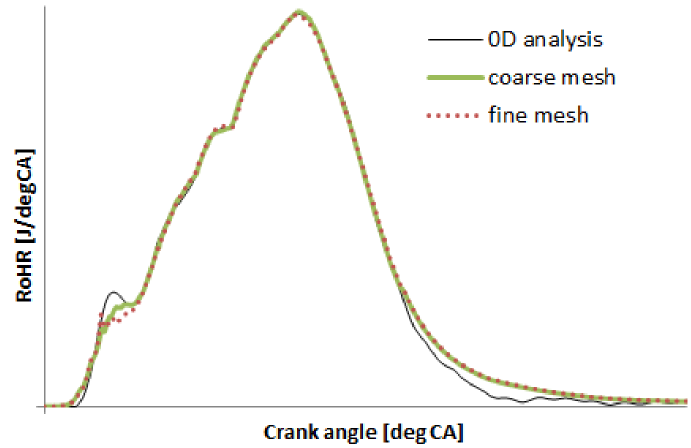

Figure 1 Rate of heat release: two calculations with different mesh sizes

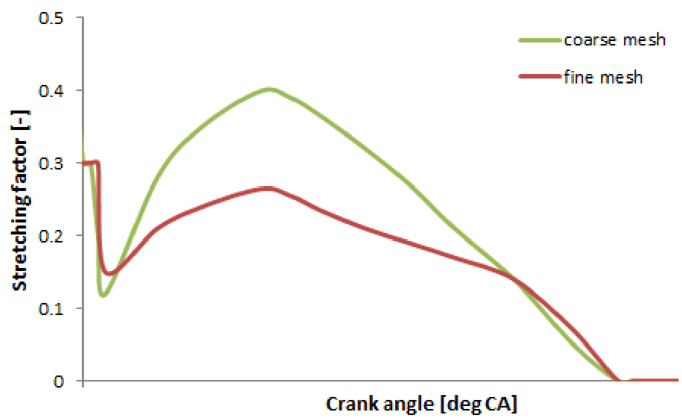

Figure 2 Stretching factors for ECFM calibration during combustion

Despite this clearly different model calibration, energy release and thus the cylinder pressure curves [Figure 3] and the average cylinder temperature [Figure 4] can be reproduced analogously to the measurement or 0D and 1D analysis.

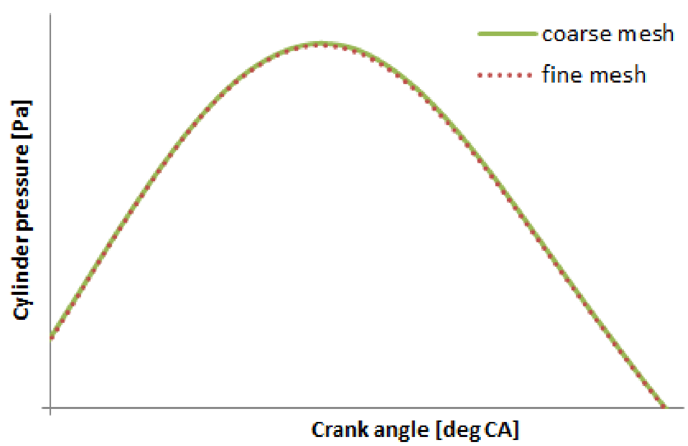

Figure 3 Cylinder pressure curves

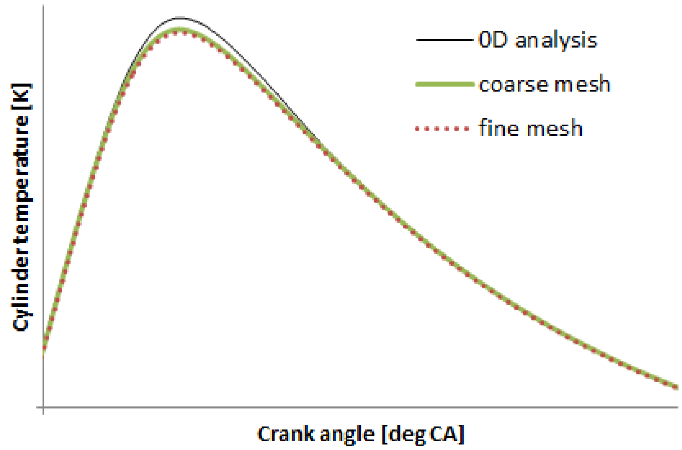

Figure 4 Cylinder temperature

If the heat transfer results are considered, however, the heat losses to the cylinder liner calculated with the finer mesh are significantly greater than with the coarser mesh [Figure 5]. The differences in terms of the piston and fire deck are such that with the fine mesh, heat flow to the piston is somewhat higher and heat flow to the fire deck is somewhat lower. In both calculations, the heat transfer coefficients do not differ considerably in any of the three areas [Figure 6]. This suggests that the wall heat model used with both meshes with their respective y+ areas works just as well. The values for y+ during the combustion phase are about 400 with the coarse mesh and about 220 with the fine mesh.

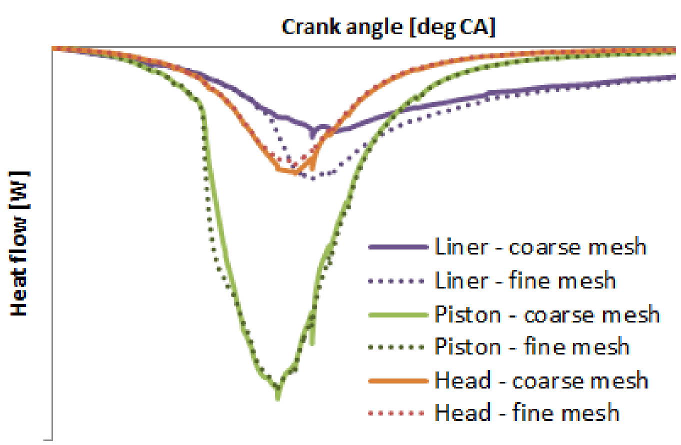

Figure 5 Convection heat rate

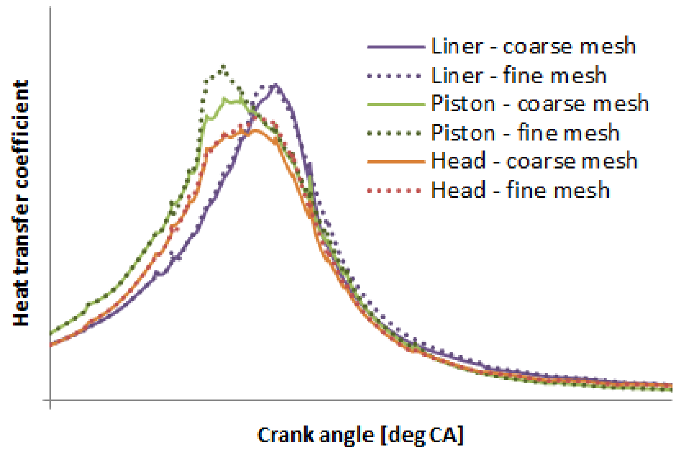

Figure 6 Mean convection heat transfer coefficient

The different heat transfer to the cylinder liner in the two calculations must be explained by the temperature distribution in the cylinder. In Figure 7 it can be seen that the flame torches have a more stretched out form at the start of combustion with the fine mesh. As a consequence, the flame front reaches the cylinder wall significantly earlier with the fine mesh, resulting in high heat transfer to the cylinder liner. With the coarse mesh, flows which do not move vertically to the edges of the cells experience greater cross-diffusion, which is reflected in the formation of rounder and slower flame torches.

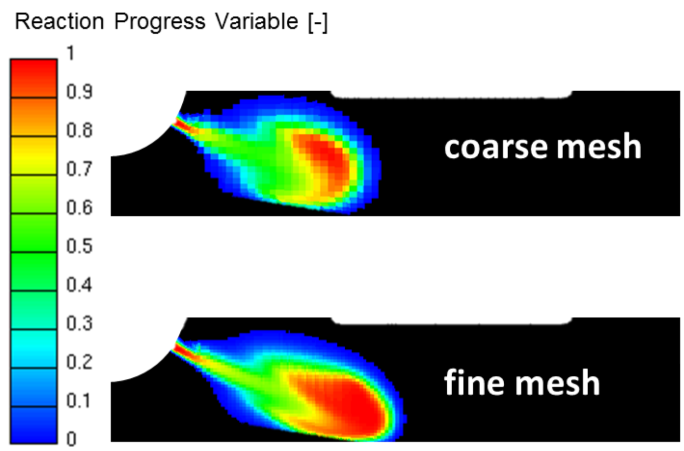

Figure 7 Reaction progress in the cylinder at the start of combustion

If these CFD results are used as the boundary conditions for FE calculation, an approximately 1-3°C lower component temperature is obtained near the fire deck with the fine CFD mesh. Only on the outer periphery of the fire deck, i.e. where the on-board sensor is installed, does a nearly 5°C lower temperature occur [Figure 8].

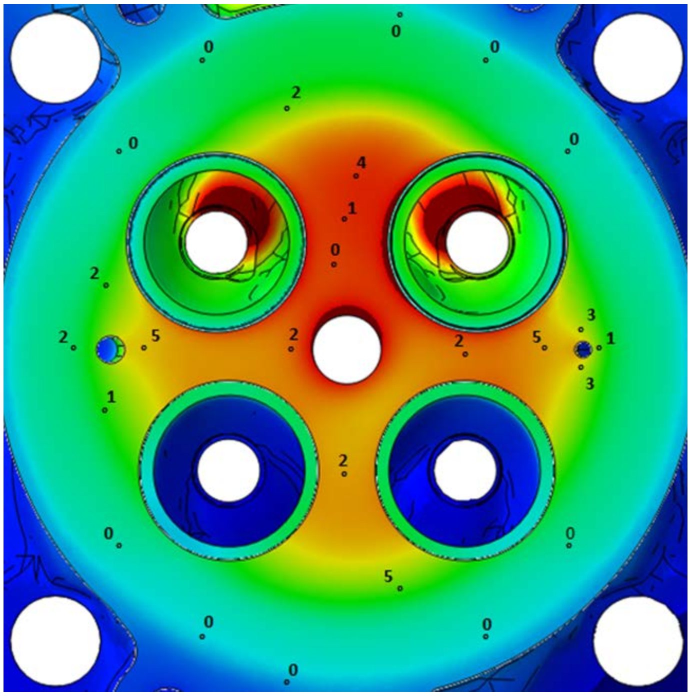

Figure 8 Simulated component temperatures 3.5mm below the fire deck surface. Boundary conditions obtained from the CFD calculation with the coarse mesh. The plotted numbers indicate the difference at these locations to the simulation with the fine mesh.

This can also be understood on the basis of the different flame propagation. With the fine mesh, combustion proceeds along the piston in the lower part of the cylinder and reaches the area of the upper and outer fire deck relatively late [Figure 8].

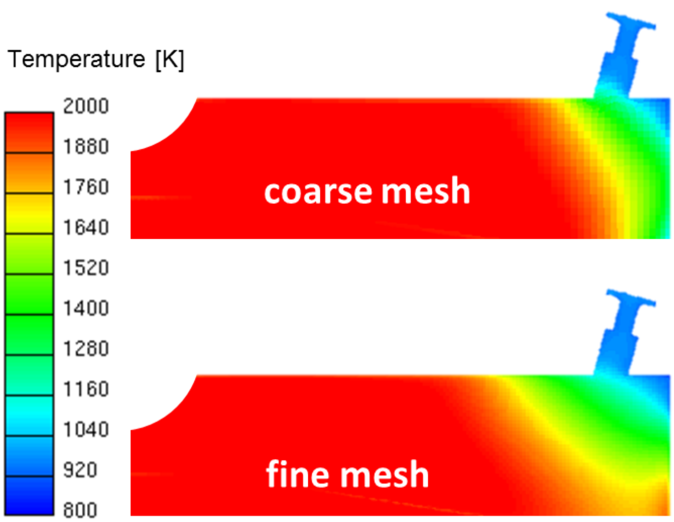

Figure 9 Temperature at the end of combustion

TURBULENCE AND WALL HEAT MODEL

As the model was being set up, different approaches to specific areas of the CFD model were also investigated. The approach recommended by the software manufacturer consists of the *k-zeta-f* turbulence model in combination with *Hybrid Wall Treatment* [4] to resolve the velocity in the area near the wall. The turbulence model k-zeta-f promises greater calculation stability and the hybrid approach for the boundary layer should be able to cover an additional y+ area, above all in the buffer layer of the boundary layer. One common alternative is to use the *k-epsilon* turbulence model along with the *standard wall function*. This second approach is of interest especially for comparisons with other CFD software suites since it is implemented in many codes. According to the developer of the CFD software, cross-linked models, for example k-epsilon in combination with the hybrid wall treatment, are not calibrated with each other and thus should be avoided. It appears that when different turbulence models are used, calibration of combustion requires somewhat different values but the results for pressure and temperature in the cylinder do not differ significantly. For heat transfer, great differences do not appear between the two models in most areas, but where there are larger flow velocities, for example at the valve seats, the k-epsilon model calculates higher heat transfer, while in areas with low flow velocities (e.g., by the sensor), it predicts lower heat transfer [Figure 11].

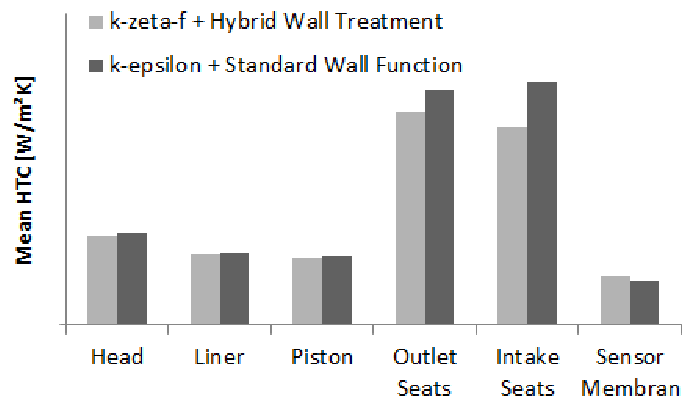

Figure 10 Heat transfer coefficient averaged over one cycle with different turbulence models

With CFD calculations of the coolant side, the k-epsilon model yields a slightly higher mass flow rate and thus greater velocities and a higher level of turbulence. This is reflected in greater convective heat transfer. At the same time, however, heat transfer caused by the effects of boiling diminishes since at higher flow velocities, bubble formation is suppressed [Figure 12].

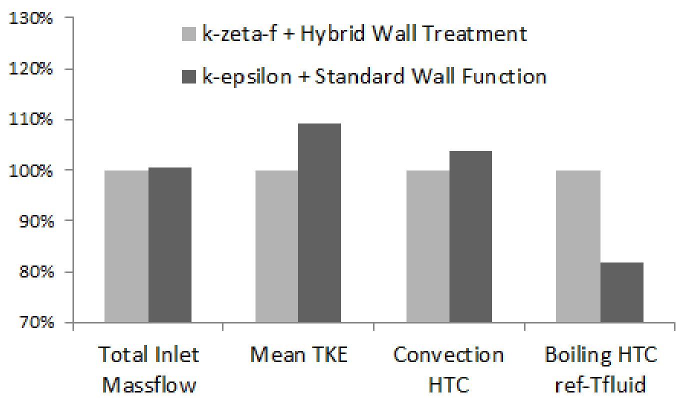

Figure 11 Comparison of two cooling jacket calculations with different turbulence models

The choice of wall heat model has a much more direct influence on the calculated heat transfer. Both the *standard wall function* and the *Han-Reitz model* are available in Fire. The latter takes into account the compressibility of the working gas. In the current version of the software, this behavior should also be considered in the basic hybrid approach. Heat transfer in the cylinder is about one third higher with the Han-Reitz model than with the standard wall function. The component temperatures computed with these boundary conditions are about 50°C too high in comparison with the measurements. It appears that when a CFD software suite is applied, it is critical to calibrate the submodels in the software with each other.

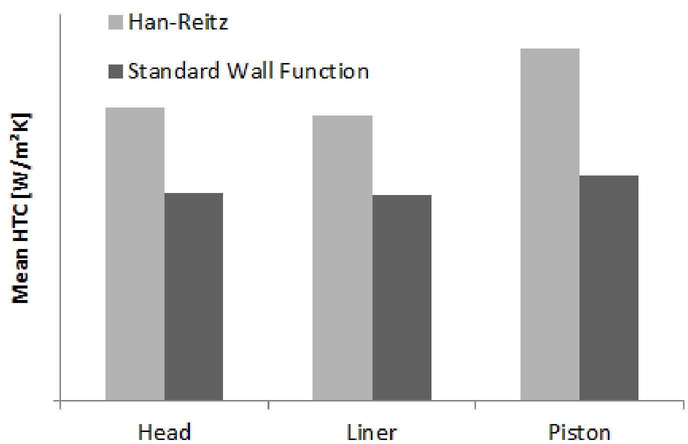

Figure 12 Heat transfer coefficient averaged over the combustion phase with two heat transfer wall models

NUCLEATE BOILING

The formation of bubbles in the coolant can be influenced by varying the coolant pressure, the flow temperature or the mass flow rate. A high speed camera and an endoscope in the cooling jacket were used on the SCE at the LEC to prove that at certain operating conditions, nucleate boiling actually occurs in the cooling jacket, for example when the coolant flow rate is lower than the standard value. The exact locations of boiling, however, could not be identified. According to cooling jacket simulation, bubbles only form locally at the hottest spots. These are the areas close to the exhaust flange and at the fire deck between the outlet valve seats. To assess the impact of boiling on the temperature of the cylinder head components, the simulation results must be compared with the measured component temperatures. Depending on the operating point, the effects of boiling can increase the transfer of heat energy to the coolant by up to 10%. If the effects of nucleate boiling are disregarded, the simulated temperature in the components is up to 20°C higher right at the hottest spots.

CONTACT MODELING

It is generally difficult to calculate heat transfer at the points of contact between different components. It is dependent not only on the heat conductivity, density and surface hardness of the materials of the contacts but also on their contact pressure and the condition of the surface [5]. With many contact pairs, for example press fits, it appeared that it was possible to approximate ideal contact conditions [6]. With complicated contact pairs such as screw connections, the heat transfer coefficients can be adjusted using data obtained from temperature measurements.

The modeling of the contacts between the valve disks and their valve seats is particularly important for the temperature distribution on the fire deck. The FE model was generated in a simplified form as a steady state model where the valves constantly remain closed. In the actual engine, however, the valves are intermittently open, which necessitates specific modeling of the valve seat contacts.

In CFD calculation, the thermal boundary conditions for the valve seats are also recorded during the gas exchange phase and averaged over the valve opening time. Thus it is possible to apply these boundary conditions to the FE model as if the valves remain open during the complete working cycle and the valve seats are influenced by the flowing charge the entire time. Temperature distributions calculated in this manner can then be compared with those that arise when the valves are closed during the entire cycle and contact conditions are ideal [Figure 14].

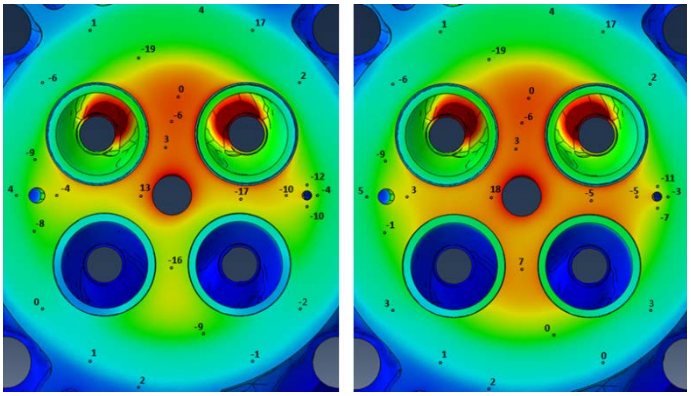

Figure 13 Temperature distribution at the fire deck with constantly open intake valves (left) and with the ideal contact conditions of constantly closed valves (right)

It appears that when the valves constantly remain open, the area around the exhaust valves is approximately 5-6°C cooler and the area around the intake valves is 5-23°C cooler. The cooling effect of the cold charge flowing in is a great deal stronger than that of the hot charge flowing out. With knowledge of valve timing, the average temperature distribution in these areas can be estimated over the entire cycle and the contacts of the valve seats at the intake and outlet ports modeled accordingly. The difference in temperature indicated above was multiplied by the share of opening times in the entire cycle time and then subtracted from the temperature distribution with ideal valve contacts. The resulting temperature distribution was used as the target value to adjust the valve contacts.

The valve contacts could naturally be calibrated from the results of the temperature measurements, but the difference between simulation and measurement can involve many causes of errors, all of which would then be attributed to the contact modeling. Even if the modeling approach described here contains great simplifications, for example the ignoring of heat released to the fluid at the seat surfaces of the valve and the valve ring during the opening times, it has proved itself in practice with different boundary conditions.

LOAD POINT VARIATION

It has been seen how great the sensitivity of the ECFM combustion model is to diverse changes in the modeling of the problem. Be it changes in turbulence modeling, in the mesh of the combustion chamber volume or simply in the fineness of the mesh on the surface of the CFD model, a fairly large number of adjustments to the settings of the ECFM parameters had to be made in order to obtain the desired energy conversion rate. How robust are the statements about temperature distribution in the cylinder head if they were computed using the CFD-FE model but the boundary conditions (geometry, load) have changed? To answer this question, the CFD-FE model calibrated at one full load point was applied using boundary conditions from an overload point. At first, the ECFM settings of the full load point were retained. In the next step, these parameters were adjusted in order to reproduce the rate of heat release obtained from a measurement at the SCE test bed in CFD calculation. Figure 15 shows the considerable differences in the rate of heat release between both of these simulations.

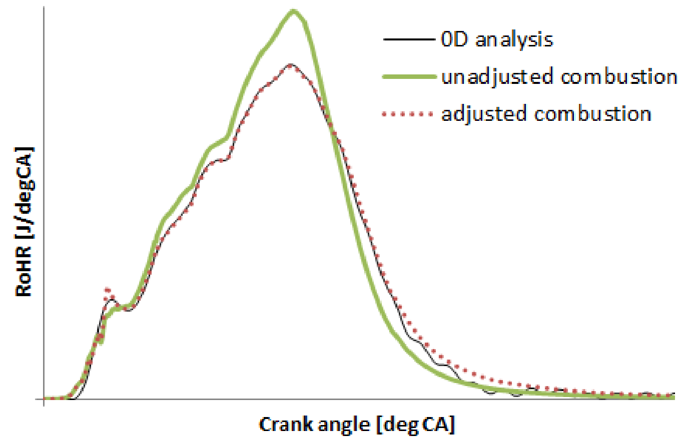

Figure 14 Rate of heat release in adjusted and unadjusted overload simulations

These differences, however, have a relatively minor impact on the calculated component temperatures. Figure 16 plots the differences in temperature of both the unadjusted calculation and the adjusted calculation. Even in the area of the fire deck, it is only around 3°C on average. Since overall energy conversion is essentially determined by fuel energy input, the heat transfer averaged over 720°CA is also roughly the same in both calculations. Moreover, it can be assumed that when the simulation model is actually applied, certain empirical values are often available for the rate of heat release and cylinder pressure of an engine, so in many cases the CFD simulation can be better adjusted in advance.

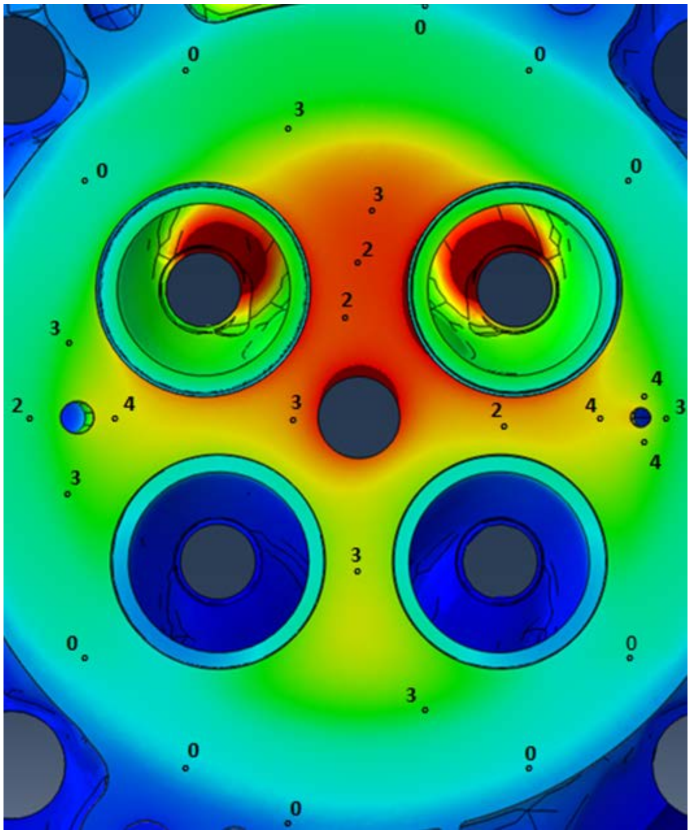

Figure 15 Comparison of the difference in component temperature between pre-simulation and simulation in which the combustion model is calibrated according to measurements

In comparison with the measured component temperatures, agreement in the range of +/- 10°C was obtained at most measuring points with the full load point as well as the overload point.

TRENDS AND RESULTS

The accuracy of CFD simulation increases as the spatial and temporal resolution becomes finer. In cases in which the goal of simulation is to obtain averaged values of the results, for example the temperature boundary conditions averaged over time, good predictions can also be made with coarser resolution. The details of the fine resolution, however, justify any additional calculation efforts.

When the simulation model is set up, consideration must be shown for how well the individual submodels are implemented in the software and above all how well these submodels are calibrated with each other.

In simulations that aim to provide the most accurate calculation of the component temperature in an engine, heat transfer from boiling should not be ignored unless the occurrence of boiling can be completely ruled out.

CONCLUSION

With the calibrated CFD-FE simulation model of the cylinder head and its built-in components, it is possible not only to optimize the pressure sensor's application to the cylinder head but also to predict the impact of different installation positions. This integrated model can also be used as the starting point for further investigations, for example variations in the geometry of the combustion chamber or the cooling jacket.

ACKNOWLEDGMENTS

The authors would like to acknowledge the financial support of the "COMET - Competence Centres for Excellent Technologies Programme" of the Austrian Federal Ministry for Transport, Innovation and Technology (BMVIT), the Austrian Federal Ministry of Science, Research and Economy (BMWFW) and the Provinces of Styria, Tyrol and Vienna for the K1-Centre LEC EvoLET. The COMET Programme is managed by the Austrian Research Promotion Agency (FFG).

REFERENCES

[1] AVL List GmbH, AVL FIRE Version 2014, Combustion Module, Document No. 08.0205.2014, Graz 2014, pp. 3-14ff

[2] AVL List GmbH, AVL FIRE Version 2014, Coupling Module CAE, Document No. 08.0207.2014, Graz 2014, pp. 4-1ff

[3] Steiner H., Kobor A., Gebhard L., A wall heat transfer model for subcooled boiling flow, Proceedings of The ASME – ZSZS International Thermal Science Seminar II, Bled 2004

[4] Popovac M., Hanjalic K., Compound Wall Treatment for RANS Computation of Complex Turbulent Flows and Heat Transfer, Springer Science + Business Media B.V. 2007

[5] Fieberg Ch., Kontaktwärmeübergang unter hohen Druck- und Temperaturrandbedingungen, *SV Sierke Verlag,1. Auflage* 2008

[6] Awender J., Methodik zur numerischen Bestimmung der Temperaturverteilung in Motorbauteilen durch gekoppelten Einsatz von FE- und 3D-CFD-Simulation, *Diplomarbeit,* November 2012