Proceedings of the 1st Global Power and Propulsion Forum
GPPF 2017
Jan 16-18, 2014, Zurich, Switzerland
www.pps.global

GPPF-2017-0045

MULTI-PHYSICS SIMULATIONS OF AN AERO ENGINE COMBUSTOR WITH OPENFOAM

Marco Konle
MTU Aero Engines AG
Marco.Konle@mtu.de
Munich, Germany

Ludovic de Guillebon
MTU Aero Engines AG
Munich, Germany

Francois Cottier
MTU Aero Engines AG
Francois.Cottier@mtu.de
Munich, Germany

ABSTRACT

The accurate prediction of wall temperatures of an aero engine combustion chamber is a challenging task. Different physical aspects as the flow field, the combustion, the radiation, and the heat flux have to be captured in a high quality. Due to this detailed level, such multi-physics simulations are of high computational demand. To reduce the computational costs, a multi-physics solver in the open source framework of OpenFOAM was set up by the authors. Besides a very detailed validation against measurement data of an academic test case, the application of the solver to an aero engine combustor of high technical relevance will be presented in this paper. The compromise of rather simple models and computational demands was chosen quite well. A comparison of calculated combustor liner temperatures with results of a thermal paint analysis reveals a very good agreement. Potential improvements for the future are identified, however, already with these results the applicability of the multi-physics solver for supporting combustor engineering is successfully presented.

INTRODUCTION

The regulations for emissions for future aero engines requires a significant improvement of the technologies in these engines. A significant contributor is the combustor of the engine. Numerical tools as CFD help to improve the efficiency of the combustion system. However, the high demand of accurate predictions for a wide range of operations needs a high accuracy of the combustor modelling. Not only an efficient design of the different components has to be delivered but also the predictions of emissions, stress and temperature loads on the components, behaviour at nominal as well as off-design operation have to be provided. For this purpose, the numerical modelling has to be validated with measurement data.

To predict accurately the thermal loads on combustor walls, the numerical modelling of such components has to cover a lot of physical aspects. Besides the precise prediction of the reacting flow field, the gas radiation including the significant contribution of soot particles as well as the heat transfer onto the combustor have to be covered.

Due to the high demand of computational power for the described modelling depth, MTU used the open source CFD code OpenFOAM and empowered the solver to capture all these aspects. The open access to the source code of OpenFOAM allowed the authors to implement and adapt all required models to their specific requirements. Additionally, since the usage of this tool is not limited by numbers of licences, an extensive range of test cases and model modifications could be investigated.

The accuracy of the different models was checked step by step with experimental data of an academic combustor investigated by DLR Stuttgart. This combustor specified and analysed in the EU funded project FIRST provided data for the isothermal as well as reacting flow field, temperature distribution, and soot concentration [1]. With this set of data, the combustion model was adapted and the newly implemented soot model was calibrated.

To proof the applicability of this solver also to real aero engine configurations, the combustor of the EU funded project TECC was calculated. This combustor configuration is of high technical relevance and provides a lot of data, especially the thermal load on the structure via thermal paint results [2].

In the following section, the different numerical models will be presented. The detailed validation will follow step by step for the different physical aspects. Finally, the application of the validated solver to the TECC combustor will be presented and discussed.

NUMERICAL MODELS

The following models were chosen for the simulation of a combustor capturing all physical aspects of interest. Due to constraints in computational time, quite simple models were used. With the possibility of calibrating the models, however, the approach has a very good quality in its predictions with an acceptable computational demand.

Main Equations

The mass, momentum, species, and enthalpy conservation equations below are solved to predict the flow behaviour and are solved within the Unsteady RANS turbulence approach according Menter [3], the k-ω-SST model.

$$\frac{\partial \bar{\rho}}{\partial t} + \nabla.\left(\bar{\rho}\,\tilde{u}_i\right) = 0 . \tag{1}$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\tilde{u}_i\right) + \nabla.\left(\bar{\rho}\tilde{u}_i \otimes \tilde{u}_i\right) = -\nabla \bar{p} + \bar{p} g_i + \nabla.\left(\tau_{ij} - T_{ij}\right). \tag{2}$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\tilde{Y}_k\right) + \nabla.\left(\bar{\rho}\tilde{Y}_k\,\tilde{u}_i\right) = \nabla.\left(\left(\frac{\bar{\mu}}{Sc} + \frac{\mu_t}{Sc_t}\right)\nabla \tilde{Y}_k\right) + \bar{\varpi}_k . \tag{3}$$

$$\frac{\partial}{\partial t}\left(\bar{\rho}\,\tilde{h}\right) + \nabla.\left(\bar{\rho}\,\tilde{h}\,\tilde{u}_i\right) = \nabla.\left(\left(\frac{\bar{\mu}}{\Pr} + \frac{\mu_t}{\Pr_t}\right)\nabla \tilde{h}\right) + \bar{\varpi}_T + \bar{\varpi}_{rad} . \tag{4}$$

where the subscript I and j represent the x_i and x_j direction component, k the species and t the turbulent part. u_i is the velocity, ρ the density, p the pressure, g_i the gravity, Y_k the mass fraction of species k, and h the enthalpy. μ and μ_t are the molecular and turbulent viscosity, Sc and Sc_t the laminar and turbulent Schmidt number, Pr and Pr_t the laminar and turbulent Prandtl number. τ_{ij} and T_{ij} symbolize the viscous and the Reynold stresses. Moreover, ω_k, ω_T, and ω_{rad} are source terms for the equations. The production and consumption of reactive species due to combustion is described by ω_k and heat release due to combustion by ω_T. ω_{rad} represents the radiation effects.

The solver used by the authors is intended to calculate flow fields with Mach numbers below 0.3. However, the density changes caused by the combustion required to implement all further models into the OpenFOAM solver rhoPimpleFoam.

Combustion modelling

A simple 1-step chemistry model is used to describe the chemical reaction. The nitrogen is assumed to be an inert species and, therefore, the fuel reacts only with the oxygen:

$$C_2H_4 + 3O_2 \rightarrow 2CO_2 + 2H_2O . \tag{5}$$

The combustion is solved using the Partially Stirred Reactor model (PaSR) and, with it, the empirical Arrhenius law (Kärrholm, [4]). The PaSR model is a finite rate chemistry method which assumes that the real flame is much thinner than any computational cell. Thus, each cell is divided into a reacting part, in which all present species are homogeneously mixed and react together, and a non-reacting part. The reactive volume fraction κ, which is a multiplying factor for ω_k and ω_T in Eq. oben and oben, results of the turbulence-chemistry interactions:

$$\kappa = \frac{\tau_c}{\tau_c + \tau_{mix}} . \tag{6}$$

$$\tau_{mix} = \sqrt{\tau_\eta \tau_T} = \sqrt{\frac{\mu + \mu_t}{\rho \varepsilon_t}} . \tag{7}$$

where τ_c represents the chemical timescale and τ_{mix} the turbulent timescale. τ_η and τ_T are the Kolmogorov and Taylor timescale, respectively, and C_{mix} a model constant. This constant has to be adequately chosen depending on the flame regime, i.e. the interaction of turbulence and chemistry, the combustor is operating in. For the understanding of flame regimes, the reader may check e.g. [5]. The calibration of C_{mix} was done by the authors using a wide range of combustor designs investigated in the past, as e.g. the TIMECOP-AE design [6], the TECC design [2], and the FIRST design [1]. All these designs are swirl stabilized combustors with high turbulence and operate in the regime of real aero engine combustors. As result of these studies, the constant C_{mix} was set to 5. The turbulent Schmidt number Sc_t for the thermal diffusion was calibrated via measured temperature profiles at the exit of these combustors as well.

Radiation modelling

High temperatures in combustors lead to significant radiative heat transfer. To cover the contribution of the hot walls as well as the gaseous radiation to the thermal loads on the combustor, this phenomenon has to be taken into account. To predict radiation, two aspects has to be considered: the spatial and the spectral radiation. In OpenFOAM, the fvDOM (finite volume Discrete Ordinate Method) solves a simplified form (Eq.8) of the radiative transfer equation (Viskanta and Mengüç, [7]) for a finite number of discrete solid angles (16 rays in the present study) and returns the ω_{rad} source term.

$$\nabla.\left(I\,s_i\right) + \alpha \Omega I = \frac{\Omega}{\pi}\left(\alpha \sigma T^4 + \frac{E}{4}\right). \tag{8}$$

with I the radiation intensity, s_i the x_i-component of the direction vector, Ω the solid angle considered, T the temperature, σ the Stefan-Boltzmann constant. E is the emission contribution, an offset which is set to $0 W.m^{-3}$ for this study.

The absorption coefficient of gas, α, is calculated with the spectral model. In this study, the grey gas model is used and takes into account the absorption/emission of the two species CO_2 H_2O [8]. Due to the significance of soot radiation, the contribution of soot to the gaseous radiation were added to the standard implementation.

Soot modelling

The soot modelling couples back into the simulation results via its impact on the radiation and, thus, on the temperature field inside the combustor and at the liner. The formation of soot itself is again affected by the temperature, thus, the soot modelling has to be solved on the fly and not in a post-processing step.

For the modelling of soot formation in the combustor, the authors have chosen the two-equation model of Magnussen/Tesner [9] and implemented the equations into OpenFOAM. This semi-empirical soot model calculates in a first step the specific concentration of radical nuclei and in a second step the formation of soot out of these nuclei. The oxidation of nuclei and soot particles (i.e. destruction of soot) is modelled by scaling the reaction rate of the combustion model to the soot concentration. The impact of turbulence on the mean reaction rate according the Eddy Dissipation Concept (Magnussen [9]) limits the soot formation. Besides that, the limitation to a maximum soot level is implemented (Kleiveland, [10]). Both limiters are reasonable. However, as the validation section will show, still the soot modelling based on such a simple two-equation model needs improvement in the future.

Conjugate Heat Transfer

The calculation of the thermal load on the combustor walls is performed via a Conjugate Heat Transfer (CHT) solution. Again starting with the CHT solver implemented in OpenFOAM, the authors implemented the models described above into this solver. Additionally, to increase the performance of the solver and to reduce the computational time significantly, the solver was adapted to allow the calculation of the wall heat flux in the solid only via a frozen-fluid-option. This option allows the decoupling of solving the solid temperature response and the fluid flow by freezing the fluid solution and just solving the heat transfer in the solid to a quasi steady state solution. Since the response of the solid domain is significantly lower, the time step is increased by a factor of 100. Thus, the steady state solution for the walls corresponding to the frozen fluid solution is achieved within a reasonable computational time. By unfreezing again the fluid and solving CHT, the impact of the solids' temperatures on the fluid is captured again. In case, the thermal loads on the walls are not significantly fluctuating over time, this process reaches the steady state solution of the combustor (i.e. of fluid and solid) after few loops. Figure 1 shows schematically this process.

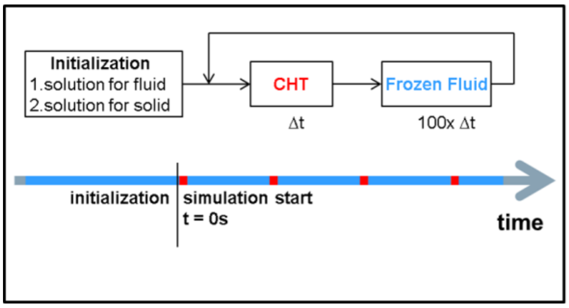

Figure 1: Scheme of the frozen-fluid process used to speed up the CHT simulation

The calculation of the wall temperatures is here not only required for the analysis of thermal loads on the structure, but it impacts also the results of the models mentioned upfront: the correct wall temperatures are mandatory for the evaluation of radiative heat transfer between gas and walls.

The next section presents the step-by-step validation of these models with data of academic and semi-technical test cases.

VALIDATION

The main vehicle for the validation of the models described upfront is the so-called FIRST combustor, designed and investigated by DLR Stuttgart. Thanks to the intensive effort of the DLR to investigate the combustor design in detail, validation data of the flow field (isothermal and reacting), the temperature distribution, and the soot concentration are available. More details about the combustor, the applied measurement techniques, and also numerical investigations performed by DLR Stuttgart can be found in [1] and [11].

Combustor Configuration

Figure 2 shows a scheme of the combustor. The combustor is composed of two parts which are the plenum and the combustion chamber. Its burning area measures 120mm in height and has a square section of 68×68mm^2. The plenum feds with three concentric flows: An inner inflow (diameter 12.3mm) and an outer ring inflow (diameters 14.4mm and 19.8mm) provides air at 293K. These two flows enter the chamber through two swirlers of 8 channels for the inner air and 12 for the outer air. Ethylene (C_2H_4) passes through 60 straight channels (0.5×0.4mm^2). A cooling air flow is injected through 4 oxidation tubes. They allow to refresh the hot gazes and to mix the rest of fuel with air, in order to burn all of the fuel before exiting the chamber through the outlet. More details about the combustor configuration are given by Geigle et al.[1].

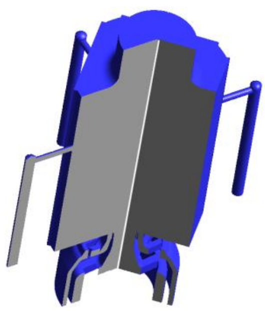

Figure 2: FIRST Combustion chamber geometry

Experimental Data

Table 1 shows the set of boundary conditions used for the model validation. The operation point represents a rich condition, the secondary air supply feds the air for a complete burn out. For the initialisation of the simulation, a starting temperature of 1100K is set at the chamber walls, before activating the CHT calculations.

For this operation point, DLR Stuttgart provided data of PIV measurements of the velocity field, LII measurements of the local soot concentration, and CARS measurements of local temperatures [1].

Inflow	Mass flow [$g.s^{-1}$]	Temperature [K]
Inner air	3.03	293
Outer air	7.08	293
Cooling air	4.04	293
Fuel	0.83	297
Chamber walls	/	1100
Other walls	/	Adiabatic

Table 1: Boundary conditions

Combustor Grid

The domain has been discretized in tetrahedral cells and includes about 3 million elements. The cell edges size, specified on each surface, varies from 1 to 2mm for the chamber and is about 0.8mm at the top of the 4 cooling pipes and 0.2mm on the fuel channels. Inside the chamber, the cells edges have a maximum length of 2mm. Special attention was paid to keep the minimum cell Jacobian as large as possible. Figure 3 shows the grid density through two cut planes (the upper plane shows the secondary air supply line).

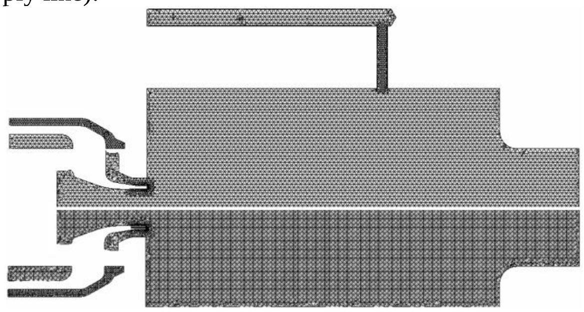

Figure 3: Computational domain discretized

Results

Starting with the flow field validation, figure 4 shows a time-averaged velocity field for the simulated operation point (table 1). At the heights x=12mm and x=18mm, the simulation results for the axial velocity profiles are compared to the measurements (figure 5 and 6). As one can see, there is a good agreement especially in regards to the maximum velocities and their locations. Thus, the flow field of this swirl stabilized burner is well calculated.

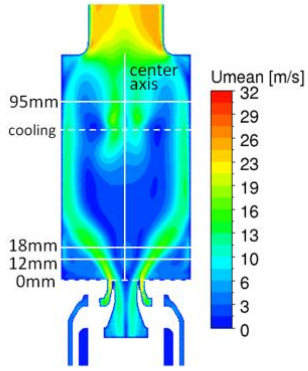

Figure 4: Time-averaged magnitude velocity field

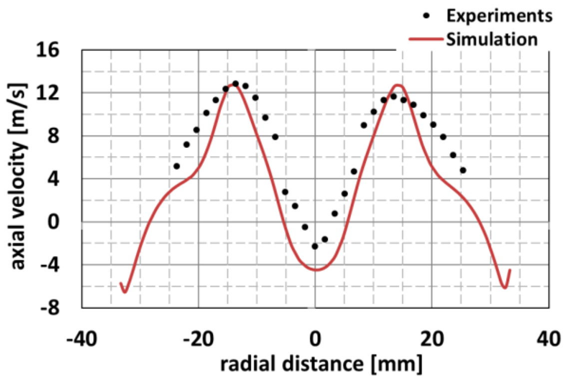

Figure 5: axial velocity profile at x=12mm

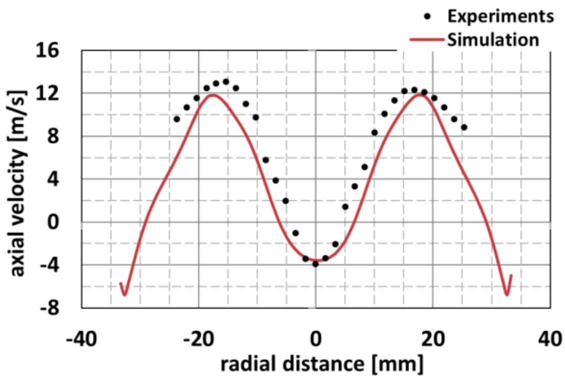

Figure 6: axial velocity profile at x=18mm

Figure 7 shows the velocity profile at x=95mm. This axial position is quite challenging because of the interaction of the secondary air flow entering the combustor via the four additional supply tubes. The simulation fits quite well to the measurements, the partially entrainment of the additional air into the recirculation zone is well captured.

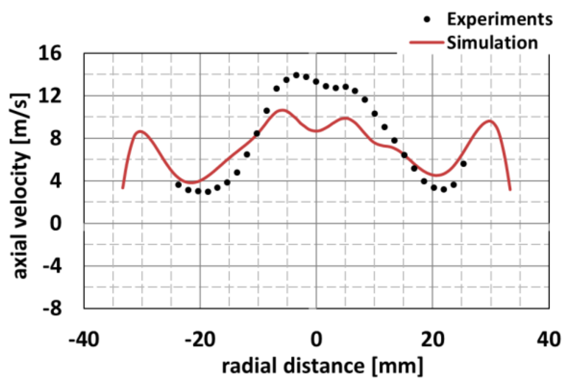

Figure 7: axial velocity profile at x=95mm, i.e. above the secondary air supply

Further on, figure 8 shows a time averaged temperature field for the reacting case. The flame is stabilized at the recirculation zone and is located slightly above the burner exit. The comparison of the radial temperature profile at x=12mm (figure 9) and on the centre line of the combustor (figure 10) reveals a good agreement. The local shape fits quite well to the local CARS measurements taking the mean temperature (Tmean), the temperature of the highest probability (Tmp), and the measurement scatter (bars in figures) into account. More details about the measurements and their accuracy can be found in [1].

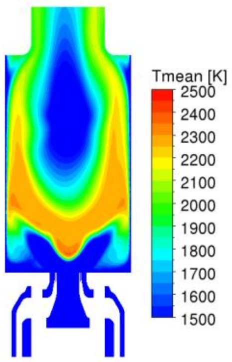

Figure 8: Time-averaged temperature distribution

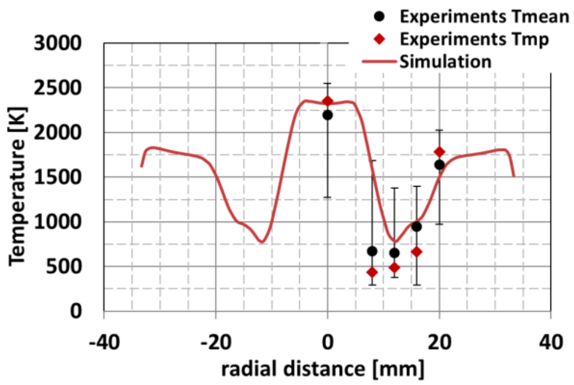

Figure 9: radial temperature profile at x=12mm

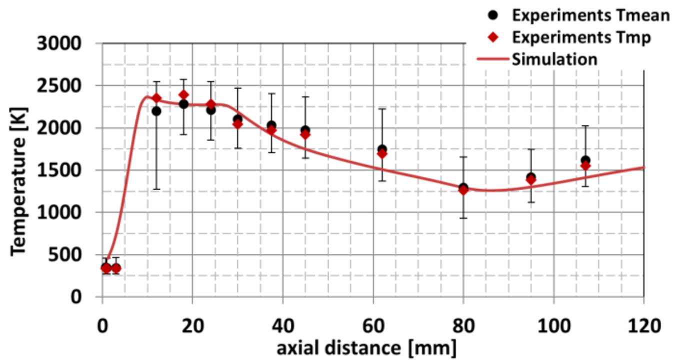

Figure 10: temperature along the centre line

Finally, the calculation of the local soot concentration has to be validated. Figure 11 shows the time averaged soot mass fraction measured at the DLR with LII. The black isolines overlaid visualize the numerical results. As one can see, the shape of the soot concentration and, thus, the location of soot formation are qualitatively well met. However, comparing the exact numbers of local soot volume fraction, again at x=12mm (figure 12) and on the centreline (figure 13), the concentration is overpredicted by a factor of ~3. Thus, the soot modelling is still an issue for the authors and also topic for future work. The necessity of including soot formation to a multi-physics simulation of an combustor, however, will be obvious in the next section.

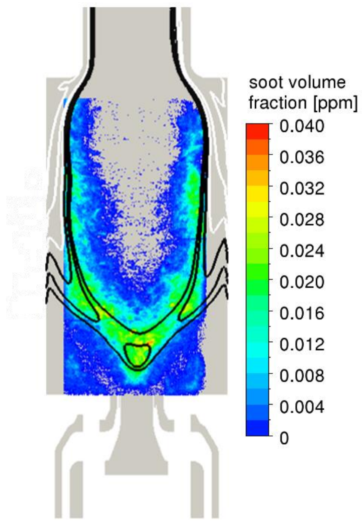

Figure 11: experimental soot distribution with overlaid isolines

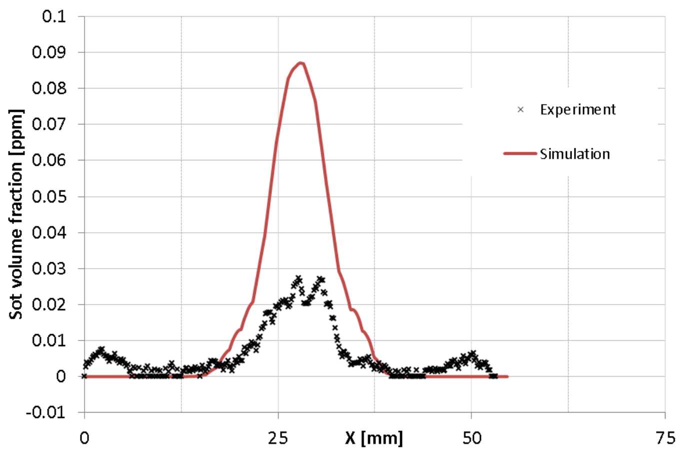

Figure 12: soot concentration at x=12mm

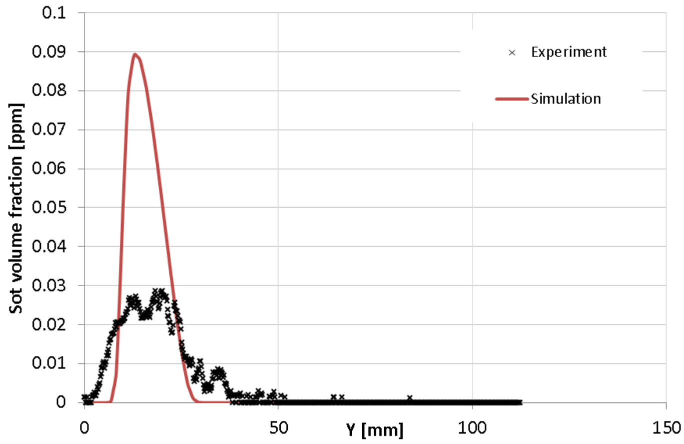

Figure 13: soot concentration along the centre line

Summarized, the newly implemented solver matched quite well the overall behaviour of the combustor.

APPLICATION TO AN AERO ENGINE

After this detailed validation of the aerodynamics, the combustion, and the soot formation, the outstanding aspects for a multi-physics simulation of an aero engine are the radiation and the thermal loads on the structure. For this purpose, another burner configuration was used. This burner was investigated in detail in a EU-funded project called TECC [2]. The configuration is a double swirl combustor with a staged air supply and was operated at engine like conditions, i.e. elevated pressures, preheated air supply and Jet-A fuel. Besides its high relevance from a technical point of view, the temperatures of the combustor walls were captured via thermal paint. Figure 14 shows a scheme of the combustor (l.h.s.) and the solid domain of the CHT combustor model (r.h.s.). The liner marked in red is the section analysed with thermal paint and, thus, used for wall temperature validation.

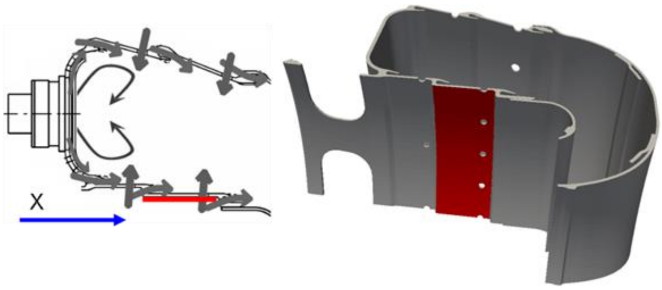

Figure 14: Scheme and solid domain of the TECC combustor

For this combustor liner, figure 15 presents the results of the thermal paint analysis (a), the numerical result of a CHT simulation neglecting soot formation and radiation (b), and the result of a full multi-physics simulation covering also the soot formation and radiation (c).

The measurements show three different temperature levels on the liner. Local fluctuations in cooling air supply driven by e.g. special flow pattern or design tolerances can lead to a non-uniform temperature distribution as in region 1.

Figure 15 b) shows that the CHT simulation of the combustor without the radiative heat load onto the structure does not even match the temperature levels. The underprediction of local wall temperatures is in the order of ~100K and, thus, of high inaccuracy. A structural analysis based on such thermal loads definitely would suffer due to that inaccuracy.

Figure 15 c) presents the result of a CHT simulation including the soot and radiation models. The time-averaged temperature distribution shows temperature levels as measured. Thus, the radiative heat flux closes the gap of ~100K. The local shape is also well met. Local fluctuations due to the transient character of a combustor (highly turbulent flows, precessing flame, 3D effects of design) are also included in the time-averaged result. The temperature scatter is in the same range as observed in the measurements.

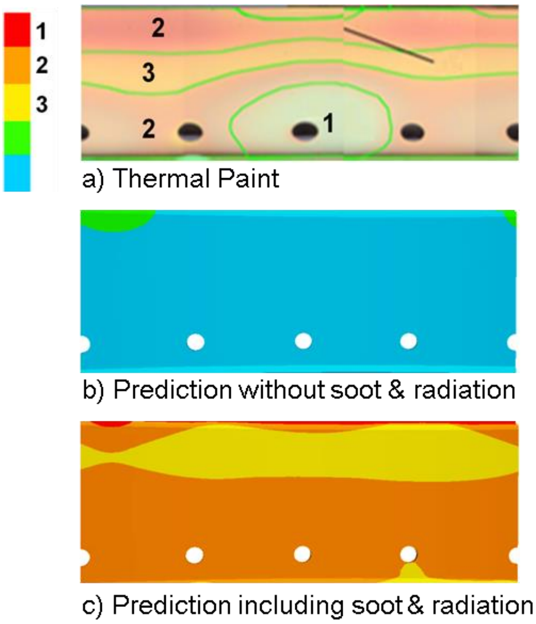

Figure 15: Combustor wall temperatures for the TECC burner

The comparison of the CHT result at the combustor wall segment with and without soot formation and radiation (fig. 15 b and c) reveals how important it is to take the soot formation and the soot contribution to the radiative heat load into account. On the other hand, the quite good agreement of the wall prediction with the thermal paint result (figure 15 a and c) shows that the overestimation of the soot formation (factor ~3 as shown in figures 12 and 13) is less severe than the negligence of soot formation at all.

CONCLUSIONS

The paper summarized the detailed validation of a multi-physics solver in OpenFOAM for combustion simulations of aero engines. The step-by-step validation of the flow field calculation, the combustion, and the soot modelling showed a very good prediction quality despite of the quite simple models used here. The simulation results of the TECC burner, which is a configuration of high technical relevance, is very encouraging. The successful prediction of wall temperatures on the combustor liners allows the support of engineering in regards to thermal load assessment as well as the analysis of e.g. off-design operation or concessions of design deviations.

Currently, the authors extend the validation with additional combustor designs and test more advanced modelling approaches for turbulence and combustion. The significant reduction of computational costs by choosing the open source tool OpenFOAM instead of commercial codes allow in the future also the incorporation of more detailed models as e.g. the soot model of Eberle et al. [11].

ACKNOWLEDGMENTS

The authors gratefully acknowledge the support with FIRST measurement data by K.P Geigle and his colleagues of the DLR Stuttgart.

REFERENCES

[1] Geigle K. P., Hadef R., and Meier W., “Soot formation and flame characterization of an aero-engine model combustor burning ethylene at elevated pressure”, Journal of Engineering for Gas Turbines and Power, Vol.136, 2014, pp. 021505-1-021505-7.

[2] Komarek, T., “TECC-AE : Final Report on the impact of main flow regarding cooling film efficiency”, 2012.

[3] Menter F.R., “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA-Journal, 32(8), 1994, pp.269-289.

[4] Kärrholm F., “Numerical modeling of diesel spray injection, turbulence interaction and combustion”, PhD Thesis, Chalmers University of Technology, Göteborg, 2008.

[5] Peters N., “Turbulent Combustion”, Cambridge Monographs on Mechanics, Cambridge University Press, 2000.

[6] Meier U., Heinze J., Freitag S., Hassa Ch., “Spray and Flame Structure of a Generic Injector at Aeroengine Conditions”, Proceedings of ASME Turbo Expo 2011: Power for Land, Sea and Air, GT2011-45282, Vancouver, 2011.

[7] Viskanta R., Mengüç M.P., “Radiation heat transfer in combustion systems”, Progress in Energy and Combustion Science, 13(2):97-160, 1987.

[8] Barlow R.S., Karpetis A. N., Frank J. H., Chen J.-Y., “Scalar profiles and NO formation in laminar opposed-flow partially premixed methane/air flames”, Combustion and Flame, 127:2102–2118, 2001.

[9] Magnussen B.F., “Modeling of NOx and Soot Formation by the Eddy Dissipation Concept”, International Flame Research Foundation First Topic Oriented Technical Meeting, Amsterdam, 1989.

[10] Kleiveland R.N., “Modeling of soot formation and oxidation in turbulent diffusion flames”, PhD Thesis, Norwegian University of Science and Technology, Trondheim, 2005.

[11] Eberle C., Gerlinger P., Geigle K.P., and Aigner, M., “Soot Predictions in an Aero-Engine Model Combustor at Elevated Pressure Using URANS and Finite-Rate Chemistry”, 50th AIAA/ASME/SAE/ASEE Joint Propulsion Conference, 2014.