Assumed PDF Modeling in Rocket Combustor Simulations

Markus Lempke, Peter Gerlinger and Manfred Aigner University of Stuttgart, Institute of Combustion Technology Pfaffenwaldring 38-40, 70569 Stuttgart

Abstract

In order to account for the strong interaction between turbulence and chemistry a multi-variate assumed PDF (Probability Density Function) approach is used to simulate a model rocket combustor with finiterate chemistry. The reported test case is the PennState preburner combustor with a single shear coaxial injector. Experimental data for the wall heat flux is available for this configuration. Unsteady RANS (Reynolds-averaged Navier-Stokes) simulation results with and without the assumed PDF approach are analyzed and compared with the experimental data. Both calculations show good a agreement with the experimental wall heat flux data. Significant changes due to the utilization of the assumed PDF approach can be observed in the radicals, e.g. the OH mass fraction distribution.

1. Introduction

Due to the high costs for ground testing it is inevitable to rely as much as possible on CFD (Computational Fluid Dynamics) simulations in the design process of rocket injectors and combustors. However, even the capabilities of high-fidelity approaches like LES (Large Eddy Simulation) have to be improved in order to be accurate enough, to have a quantitative influence on the injector design process [1]. Direct Numerical Simulation (DNS) approaches would yield much better results, but are computationally still more expensive and therefore restricted to the simulation of single injector near field phenomena assuming non-reacting flows or using simple combustion models. Ideally, however, a CFD tool should be able to simulate complete combustor geometries with up to several hundred injector heads including the nozzle in an acceptable timespan and at acceptable cost. Accordingly, RANS (Reynolds-averaged Navier-Stokes) methods are still the design tools of choice in industrial applications where mostly time-averaged quantities are of interest. However, recent work has shown the limitations of RANS methods when it comes to predicting the wall heat flux in a model rocket combustor with shear coaxial injectors [1, 2, 3].

Unsteady RANS simulations, on the other hand, have demonstrated the capability to yield quite useful results for this kind of combustor [2, 3]. However, they are per se not able to reproduce the strong non-linearity of the chemical source terms arising from turbulence-chemistry interaction. Instead they usually treat chemistry in a "laminar" way. The use of PDF (Probability Density Function) methods is a promising approach to tackle this problem. It appears to be the natural choice to treat statistical fluctuations in the flowfield with probabilistic methods. The two major groups of PDF methods are the *assumed PDF* approaches and the *PDF transport equation* methods.

The more sophisticated approach, that doesn't predefine the shape of the PDF, is the solution of a PDF transport equation. It has the advantage of treating convection and finite-rate nonlinear chemistry exactly, if the PDF comprises all relevant variables. Those variables are the velocity vector for the convection and the thermochemical variables (h and Y_i) for the source term due to chemical reactions. This approach is called joint velocity-scalar PDF. However, there remain closure problems due to the fluctuating pressure gradient and due to molecular transport [4] which have to be modeled. Furthermore transported PDF approaches require extensive computational resources.

In assumed PDF methods the shape of the PDF is mathematically predefined. Thus the main focus in these methods is to find functions that are as simple as possible from a mathematical point of view but flexible enough to be able to approximate realistic PDF shapes observed in experiments. Due to the comparatively low computational costs, assumed PDF methods are widely used and have proven the capability to produce useful results over a wide variety of combustion conditions [5, 6].

This paper aims to show the feasibility of the assumed PDF approach under rocket combustion conditions while only moderately increasing the computational cost.

ID 513 - MARKUS LEMPKE

2. Numerical Code

For all simulations in the present paper the scientific TASCOM3D (Turbulent All Speed Combustion Multigrid 3D) code is used. The code solves the conservation equations for mass, momentum, energy and species as well as equations for turbulence closure. Several two-equation turbulence models are implemented in the code. However, the presented results are obtained with the k- ω -model proposed by Wilcox [7]. When the assumed PDF module is activated, two additional equations for the temperature variance and the variance of the sum of the species mass fractions have to be solved. The complete set of equations in three-dimensional conservative form can be written as

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial (\mathbf{F} - \mathbf{F}_{\nu})}{\partial x} + \frac{\partial (\mathbf{G} - \mathbf{G}_{\nu})}{\partial y} + \frac{\partial (\mathbf{H} - \mathbf{H}_{\nu})}{\partial z} = \mathbf{S}, \qquad (1)$$

with the vector of conservative variables

$$\mathbf{Q} = \left[\bar{\rho}, \bar{\rho}\tilde{u}, \bar{\rho}\tilde{v}, \bar{\rho}\tilde{w}, \bar{\rho}\tilde{E}, \bar{\rho}k, \bar{\rho}\omega, \bar{\rho}\sigma_T, \bar{\rho}\sigma_Y, \bar{\rho}\mathbf{Y}\right]^T .$$
⁽²⁾

The variables in the variable vector \mathbf{Q} are the density $\bar{\rho}$ (averaged), the velocity components (Favre averaged) \tilde{u} , \tilde{v} and \tilde{w} , the total specific energy \tilde{E} , the turbulence variables k and $\omega = \epsilon/k$ (where k is the kinetic energy and ϵ the dissipation rate of k), the variance of the temperature σ_T and the variance of the sum of the species mass fractions σ_Y . The vector \mathbf{Y} contains the species mass fractions \tilde{Y}_i for $i = 1, 2, ..., N_k - 1$ and N_k is the total number of gaseous species. Since the set of equations includes the conservation of mass, $N_k - 1$ species transport equations have to be solved, only. In Eq. (1) the vectors \mathbf{F} , \mathbf{G} and \mathbf{H} specify the inviscid fluxes and \mathbf{F}_v , \mathbf{G}_v and \mathbf{H}_v the viscous fluxes in x-, y- and z- direction, respectively.

The momentary chemical production rate of species *i* is defined by

$$S_{Y_{i}} = M_{i} \sum_{r=1}^{N_{r}} \left[\left(v_{i,r}^{''} - v_{i,r}^{'} \right) \left(k_{fr} \prod_{l=1}^{N_{k}} c_{l}^{v_{i,r}^{'}} - k_{br} \prod_{l=1}^{N_{k}} c_{l}^{v_{i,r}^{'}} \right) \right] , \qquad (3)$$

where k_{f_r} and k_{b_r} are the forward and backward rate constants of reaction r, M_i the molecular weight and c_i the concentration of species *i*. Finally $v'_{i,r}$ and $v''_{i,r}$ denote the stoichiometric coefficients of species *i* in reaction *r*. When chemistry is treated in a laminar way, mean values of the species concentration and temperature are used to calculate the reaction rates in Eq. (3). PDF approaches on the other hand incorporate the turbulent fluctuations of these quantities. Thus, the averaged chemical source term is obtained by integrating the instantaneous source terms using a PDF according to

$$\bar{S}_{Y_i} = \langle S_{Y_i} \rangle = \int_{\hat{T}} \int_{\hat{Y}_1} \dots \int_{\hat{Y}_{N_k}} S_{Y_i} P(\hat{T}, \hat{Y}_1, \dots, \hat{Y}_{N_k}) \, \mathrm{d}\,\hat{T} \, \mathrm{d}\,\hat{Y}_1 \dots \, \mathrm{d}\,\hat{Y}_{N_k} \quad .$$
(4)

Presuming statistical independence between species and temperature fluctuations the PDF can be replaced by the product of a temperature and a species PDF, P_T and P_Y , respectively. The assumed PDF approach uses mathematically predefined probability density functions for the temperature and the species mass fraction. A widely used approach for the temperature PDF P_T is the clipped Gaussian distribution

$$P_T(\hat{T}) = \frac{1}{\sqrt{2\pi\sigma_T}} \exp\left(-\frac{\left(\hat{T} - \tilde{T}\right)^2}{2\sigma_T}\right) \left(\theta(\hat{T} - T_{min}) - \theta(\hat{T} - T_{max})\right) + A_l \delta(\hat{T} - T_{min}) + A_r \delta(\hat{T} - T_{max}) \quad .$$
(5)

The constants A_l and A_r correspond to the clipped parts and determine the magnitude of the delta pulses in order to satisfy the normalization condition.

For the species fluctuations Girimaji [8] proposed the multi-variate β -PDF

$$P_{Y}\left(\hat{Y}_{1},...,\hat{Y}_{N_{k}}\right) = \frac{\Gamma\left(\sum_{m=1}^{N_{k}}\beta_{m}\right)}{\prod_{m=1}^{N_{k}}\Gamma\left(\beta_{m}\right)} \left[\delta\left(1-\sum_{m=1}^{N_{k}}\hat{Y}_{m}\right)\prod_{m=1}^{N_{k}}\hat{Y}_{m}^{\beta_{m}-1}\right]$$
(6)

where

$$\sigma_Y = \sum_{m=1}^{N_k} \widetilde{Y_m'^2} \tag{7}$$

is the sum of the species mass fractions and

$$\beta_m = \tilde{Y}_m B, \quad B = \left[\sum_{m=1}^{N_k} \tilde{Y}_m \left(1 - \tilde{Y}_m\right)\right] / \sigma_Y - 1 \tag{8}$$

are model parameters. The delta function in Eq. 6 is to fulfill the normalization condition of the PDF. Though the neglection of the full matrix of variances and covariances means a loss of information, the consideration of only the trace of the matrix, leads to a significant reduction of computing time. When dealing with detailed chemistry the number of components N_k becomes quite large so that the solution of $n = (m-1)^2/2+m$ transport equations ($m = N_k-1$) becomes impossible for practical simulations.

In TASCOM3D chemical reactions are treated fully coupled with the fluid motion in the finite-rate chemistry approach, the reaction rate k_r is obtained from the Arrhenius law

$$k_r = A_r T^{n_r} \exp\left(\frac{-E_r}{R_m T}\right) \tag{9}$$

with the temperature T and the gas constant R_m . The parameters A_r , n_r and E_r are available from a reaction mechanism which has to be chosen. As for the simulations presented in this paper, the hydrogen oxidation scheme of Ó Conaire et al. [9] with 19 reactions and 8 species is used, because it is well validated, even for the high pressure regime.

Equation (1) is solved on structured multi-block meshes via an implicit Lower-Upper Symmetric Gauss-Seidel (LU-SGS) [10] finite-volume algorithm. Due to recent work [11] the spatial discretization is of up to fifth order, whereas the temporal resolution is of up to third order. The code has been parallelized through MPI (Message Passing Interface) and is optimized to run on array processor architectures. *TASCOM3D* has been validated by a variety of combustion simulations ranging from subsonic [5] to supersonic [6] flows.

3. Test Case

Within the framework of the program "Focused Validation Data for Full Flow Staged Combustion (FFSC) Injectors" funded by NASA a GO2/GH2 single element combustor (below called PennState preburner combustor) was examined at the Cryogenic Combustion Laboratory at the Pennsylvania State University. The specific interest in FFSC cycle engines arises from their simplicity and increased thrust to weight ratio, relative to other closed cycle engines. The experiments reported by Marshall et al. [12] were designed to characterize the chamber wall heat transfer. Therefore the circular combustion chamber was equipped with a large amount of Gordon type heat flux gauges and coaxial thermocouples. The axial positions of the temperature and heat flux measurements are illustrated in Fig. 1. The specific configuration was simulated as test case RCM1 in the 3rd International Workshop on Rocket Combustion Modeling [13].

The chamber diameter is 38.1 mm and its length 285.75 mm. Two upstream preburners produce oxidizer-rich and fuel-rich gases, respectively. The oxidizer-rich gas is fed to the combustion chamber through the inner tube of the coaxial injector with a diameter of 5.26 mm and is recessed 0.43 mm with respect to the combustion chamber face plane. The annular fuel feed has an inner diameter of 6.3 mm and an outer diameter of 7.49 mm. The operating conditions of the PennState preburner combustor are summarized in Tab. 1. It has to be noted that the GO₂ and GH₂ mass flow rates to the preburners were measured. Together with the measured preburner and combustion chamber pressures the properties downstream of the preburners were calculated using the NASA chemical equilibrium code CEA. More details can be found in Ref. [13]. The PennState preburner combustor test case has recently been of great interest for model validation and was simulated using a variety of modeling approaches [1, 2].

The simulations are performed as unsteady 2D axisymmetric calculations on the computational domain illustrated in Fig. 1 with approximately 215 000 volumes. Wherever possible, the measured temperatures are set as boundary conditions. The injector post tip wall is assumed to be isothermal with a temperature of 755 K. For the combustion chamber wall a temperature distribution corresponding to a least squares fit of the measured data points is set as boundary condition. The nozzle is water-cooled and has a temperature of 511 K. All other walls are assumed to be adiabatic. The inflow conditions correspond to the values in Tab. 1. Unfortunately no experimental data is available for turbulent quantities so that estimations had to be used.



Figure 1: Computational domain, measured wall temperatures and measured wall heat fluxes for the PennState preburner combustor.

		Oxidizer Preburner	Fuel Preburner
pressure	MPa	5.85	6.50
temperature of products	Κ	700	811
O ₂ mass flow in products	kg/s	$8.55 \cdot 10^{-2}$	-
H ₂ mass flow in products	kg/s	-	$1.33 \cdot 10^{-2}$
H ₂ O mass flow in products	kg/s	$4.98 \cdot 10^{-3}$	$1.98 \cdot 10^{-2}$
		Main Chamber	
pressure	MPa	5.42	

Table 1: Operating conditions of the PennState preburner combustor test case.

All calculations were performed on the CRAY XE6 supercomputer at the High Performance Computing Center Stuttgart (HLRS). It is composed of 64-bit AMD Opteron 6100 Series octocore processors at 2 GHz. Each computation was distributed over 256 MPI jobs and performed with a constant timestep of 10 ns. The spatial discretization was 5th order while the time integration was 3rd order accurate. From the bulk velocity the chamber flow-through time can be estimated to be 8.3 ms [1]. To make sure that disturbances from the initial solution are washed out of the computational domain preceding calculations were performed over approximately two flow-through cycles. Subsequently data averaging was carried out over 25 ms, corresponding to approximately three flow-through cycles.

4. Results

Figure 2(a) shows contour plots of the averaged OH mass fraction for the calculation without (top) and with (middle) the assumed PDF approach. The graph at the bottom shows an instantaneous hydroxyl distribution of the assumed PDF calculation. Note that all instantaneous contour plots in this section were extracted at the same simulation time. From the instantaneous contour plot it can be clearly observed that the flame is anchored at the oxidizer post tip, although it appears to be lifted in the averaged graphs. A close-up look at the near injector region in the averaged OH distribution, however, shows a region of high OH mass fraction right at the oxidizer post tip, indicating the flame base. The adjacent region with low hydroxyl mass fraction is due to strong corrugation of the flame just downstream the base, that can be observed in the instantaneous contour plot. In the calculation with laminar chemistry a region with high OH mass fraction stretches from approximately 80 mm to 200 mm downstream the injector. Here averaged OH mass fraction values reach around 5%. When turbulence-chemistry interaction is considered in the simulation the highest hydroxyl mass fractions in the same region are more than 20% lower and hardly reach a value of 4%. Furthermore the OH distribution is significantly broader towards the rear end of the combustion chamber. At the flame base that is anchored at the oxidizer post tip, however, both simulations compute averaged OH mass fraction of nearly 10%. It has to be noted that the maximum instantaneous values of the hydroxyl mass fraction are also in this order of magnitude throughout the combustion chamber although the colour legend in Fig. 2(a) is limited to a lower value.

Lempke, Gerlinger & Aigner. ASSUMED PDF MODELING IN ROCKET COMBUSTOR SIMULATIONS



(a) Averaged OH mass fraction contour with laminar chemistry (top) and assumed PDF approach (middle) and instantaneous OH distribution with assumed PDF approach at an arbitrary time step.



(b) Averaged temperature contour with laminar chemistry (top) and assumed PDF approach (middle) and instantaneous temperature distribution with assumed PDF approach at an arbitrary time step.

Figure 2: Hydroxyl and temperature contours of simulations with and without assumed PDF approach.

ID 513 - MARKUS LEMPKE



(b) Averaged (top) and instantaneous (bottom) contour plot of $\sqrt{\sigma_Y}$.

Figure 3: Contour plots of the modeling variables of the assumed PDF approach.

Although the differences in the OH contour plots between the calculation with laminar and turbulent chemical source term are apparent, the differences in the temperature distributions are more subtle. The graphs in Fig. 2(b) are arranged in the same way as those in Fig. 2(a). Besides the observation that hot product gases are carried further upstream by the main recirculation zone in the assumed PDF calculation there is hardly any difference between the two averaged contour plots. In both calculations the highest average temperatures are in the region of 3500 K. Moreover, due to the only moderate differences in the heat release in the combustion chamber the flow field is more or less unaffected by utilizing the assumed PDF approach in the presented test case. In both simulations two antipodal recirculation zones form in the upstream corner region of the combustion chamber. The main recirculation zone stretches from the injector face plane towards the combustion chamber wall at 150 mm. It encloses the smaller counterrotating recirculation zone that forms in the corner and coincides with laminar and turbulent chemical source term it can be concluded that the combustion in this configuration of the PennState preburner combustor is quite close to equilibrium conditions. The instantaneous contour plot at the bottom again shows how strongly corrugated the flame is in the near injector region and how those structures merge and form larger patterns further downstream.

The fundamental principle of the assumed PDF approach is the solution of transport equations for the temperature and the sum of the species variances as introduced in Eq. (5) and Eq. (6). From the perspective of physics, however, rather than the variance, the standard deviation is more tangible. Thus actually the square roots $\sqrt{\sigma_T}$ and $\sqrt{\sigma_Y}$ are plotted in Fig. 3(a) and Fig. 3(b), respectively.



Figure 4: Comparison between the measured wall heat flux and the simulations without (green) and with (red) assumed PDF modeling.

Figure 3(a) shows the standard deviation of temperature computed by the assumed PDF approach. The top graph represents the averaged distribution whereas an instantaneous distribution can be seen in the bottom graph. The highest average values of up to 800 K occur in the near injector region just downstream the oxidizer post tip. Overall the region of elevated values between 300 K and 600 K reaches approximately 180 mm into the combustion chamber. But also in the vicinity of the combustion chamber walls values of more than 200 K can be reached. However, it has to be mentioned, that the standard deviation decays in the boundary layer and vanishes directly at the wall. Also in the upstream corners a streak with standard deviations of around 200 K that separates the two recirculation zones can be observed. Comparing the instantaneous graph with this of Fig. 2(b) it can be observed that the highest standard deviations coincide with regions of high temperature gradients. In this regions the instantaneous values of $\sqrt{\sigma_T}$ can reach 1000 K and more and they can occur even up until the middle of the combustor.

The overall shape of the $\sqrt{\sigma_Y}$ contour that is shown in Fig. 3(b) shows a lot of similarity to the distribution of $\sqrt{\sigma_T}$. However, here high values appear not only at the flame base but also in a region 100 to 160 mm downstream the injector near the combustor symmetry axis. Furthermore the region of elevated standard deviation values reaches further downstream into the combustor. On the other hand no rise in $\sqrt{\sigma_Y}$ can be detected near the combustion chamber wall or in the recirculation zones. Maximum averaged and instantaneous values of $\sqrt{\sigma_Y}$ are approximately 0.3 and 0.6, respectively.

As was mentioned earlier experimental data is available for the heat flux to the combustion chamber wall at discrete axial positions. In Fig. 4 this data is compared to wall heat flux data extracted from the simulations. In a previous paper [3] simulations with laminar chemistry have been presented for the same test case. Due to a faulty temperature boundary condition in the older simulation the wall heat fluxes differ from the ones shown in this paper. It can be stated that both simulations show quite good agreement with the experiment as far as the general shape of the distribution and the position of the maximum are concerned. Especially the axial position of the initial steep rise is captured very accurately. The maximum heatflux is only slightly overpredicted by approximately 15-20 %. At 100 mm downstream of the injector, however, the decline of the wall heat flux is not pronounced enough in the simulations so that the heat flux in this region is overpredicted with an almost constant offset of approximately 3 MW/m². Due to the fact that the utilization of the assumed PDF approach has no major impact on the heat release and the flow field in the combustor there is also no major difference in the wall heat flux between the two simulations. Only between 30 and 70 mm downstream the injector a slight deviation of up to 1 MW/m² can be observed between the two calculations. The higher wall heat flux predicted by the simulation with assumed PDF approach is the result of the aforementioned stronger upstream transport of hot product gases.

Unfortunately no experimental data is available for the standard deviation of the wall heat flux. Therefore the

ID 513 - MARKUS LEMPKE

simulation data for this quantity will also not be presented here. Yet it should be mentioned that these results show only minor differences between the two simulations as well. In both cases the maximum standard deviation of the wall heat flux is approximately 3.5 MW/m² and coincides with the position of the maximum average wall heat flux at around 60 mm downstream the injector.

5. Conclusions and Outlook

A multi-variate assumed PDF approach has been successfully used for the unsteady RANS simulation with finite-rate chemistry of the PennState preburner combustor. The rise in computational cost compared to the calculations with laminar chemistry has found to be moderate (approx. 40-50%). The presented simulations are able to produce a good representation of the wall heat flux measured in the experiment. Merely in the downstream region larger deviations can be observed. Yet, for the design of the cooling system of the combustors, knowledge about the position and the magnitude of the maximum wall heat flux are most important. This information is reproduced quite accurately by both simulations, with a slight overprediction of the maximum by less than 20%. Bearing in mind the difficulty of performing wall heat flux measurements this constitutes a feasible value for engineering purposes.

The utilization of the PDF approach leads to observable changes in the distribution of the radical distributions. This has been shown in Fig. 2(a) for the hydroxyl molecule but is also true for atomic hydrogen and oxygen. The heat release and flow field, however, are more or less unaffected by these changes. Therefore the differences in the computed wall heat fluxes are also marginal. Only in the region of maximum wall heat fluxes a small deviation between the two simulations can be observed. The higher heat flux in the simulation considering turbulence-chemistry interaction was found to result from a stronger upstream transport of hot product gases by the recirculation zone. The fact that the assumed PDF approach only yields small changes indicates that the presented combustor configuration seemingly operates close to equilibrium conditions. A meaningful analysis of this aspect, however, is only possible with detailed knowledge of the actual probability density functions of the temperature and species mass fractions throughout the combustion chamber. Such information can only be obtained from extensive experiments with optical access to the combustion chamber or a numerical simulation incorporating a transported PDF approach. Currently there is work under way to pursuit such a numerical investigation.

It goes without saying that also the axisymmetric calculation, that was chosen due to the huge benefit in computation time, constitutes a strong simplification of the problem. For a more detailed analysis of the PennState Preburner combustor a full 3D simulation is necessary and currently under preparation.

Acknowledgments

This ongoing investigation is funded by the German Research Foundation (DFG) within the framework of the research program *SFB-TRR 40*. The support is greatly appreciated.

The simulations are performed at the High Performance Computing Center Stuttgart (HLRS). The authors wish to thank for the computing resources and the technical support.

References

- Tucker, K., Menon, S., Merkle, C., Oefelein, J. C., and Yang, V., "Validation of High-Fidelity CFD Simulations for Rocket Injector Design," 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, AIAA 2008-5226, 2008.
- [2] Lian, C. and Merkle, C. L., "Contrast between Steady and Time-Averaged Unsteady Combustion Simulations," 48th AIAA Aerospace Sciences Meeting, AIAA 2010-371, 2010.
- [3] Lempke, M., Gerlinger, P., Rachner, M., and Aigner, M., "Steady and Unsteady RANS Simulations of Cryogenic Rocket Combustors," 49th AIAA Aerospace Sciences Meeting, AIAA 2011-101, 2011.
- [4] Pope, S. B., "PDF Methods for Turbulent Reactive Flows," Progress in Energy Combustion Science, Vol. 11, 1985, pp. 119–192.
- [5] Gerlinger, P., "Investigations of an Assumed PDF Approach for Finite-Rate-Chemistry," *Combustion Science and Technology*, Vol. 175, No. 5, 2003, pp. 841–872.

Lempke, Gerlinger & Aigner. ASSUMED PDF MODELING IN ROCKET COMBUSTOR SIMULATIONS

- [6] Gerlinger, P., Stoll, P., Kindler, M., Schneider, F., and Aigner, M., "Numerical Investigations of Mixing and Combustion Enhancement in Supersonic Combustors by Strut Induced Streamwise Vorticity," *Aerospace Science* and Technology, Vol. 12, No. 2, 2008, pp. 159–168.
- [7] Wilcox, D. C., "Formulation of the k-ω Turbulence Model Revisited," AIAA Journal, Vol. 46, 2008.
- [8] Girimaji, S. S., "A Simple Recipe for Modeling Reacting-Rates in Flows with Turbulent Combustion," AIAA 1991-1792, 1991.
- [9] M.Ó Conaire, Curran, H. J., Simmie, J. M., Pitz, W. J., and Westbrook, C. K., "A Comprehensive Modeling Study of Hydrogen Oxidation," *International Journal of Chemical Kinetics*, Vol. 36, No. 11, 2004, pp. 603–622.
- [10] Gerlinger, P., Möbus, H., and Brüggemann, D., "An Implicit Multigrid Method for Turbulent Combustion," *Journal of Computational Physics*, Vol. 167, No. 2, 2001, pp. 247–276.
- [11] Gerlinger, P., "High-Order Multi-Dimensional Limiting for Turbulent Flows and Combustion," 49th AIAA Aerospace Sciences Meeting, AIAA 2011-296, 2011.
- [12] Marshall, W., Pal, S., Woodward, R., and Santoro, R., "Benchmark Wall Heat Flux Data for a GO₂/GH₂ Single Element Combustor," *41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*, AIAA 2005-3572, 2005.
- [13] Pal, S., Marshall, W., Woodward, R., and Santoro, R., "Wall Heat Flux Measurements in a Uni-Element GO₂/GH₂ Shear Coaxial Injector," *Proceedings of the 3rd International Workshop on Rocket Combustion Modeling*, Vernon, France, 2006.