Influence of Boundary Conditions on Sub-Millimeter Combustion

Shaurya Prakash,*†§ Roald Akberov,‡ Damena Agonafer,‡ Adrian D. Armijo,‡ and Mark A. Shannon†

Department of Mechanical and Aerospace Engineering, Rutgers, The State University of New Jersey, Piscataway, New Jersey 08854, USA and Department of Mechanical Science and Engineering, University of Illinois at Urbana—Champaign, Urbana, Illinois 61801, USA

Received January 16, 2009. Revised Manuscript Received May 4, 2009

Growing interest in small-scale, portable energy systems such as fuel cells has necessitated the development of small-scale fuel processing or reforming systems. Many fuel reforming systems require reliable heat sources as in some cases temperatures in excess of 600 °C may be required. Sub-millimeter combustors can provide such a heat source; however, a broader set of design rules are needed for constructing systematically engineered heat sources. In this article, experimental observations and computational fluid dynamics modeling results are presented for stable and steady confined flame structures within an alumina sub-millimeter combustor. Influence of inlet flow and thermal boundary conditions are evaluated through a parametric study. The inlet flow rates and relative gas composition, the thermal boundary conditions that include thermal conductivity of the walls, convection of heat to and from the walls, and radiation of heat energy through the walls all determine the position, structure, and temperature of the reacting fluid and combustor walls. The model shows the importance of radiative heat transfer in the formation of the steady-state flame structures within the microcombustor.

Introduction

Rapid growth in the use of miniature systems for a variety of applications has prompted the development of small-scale, portable, and highly durable power sources to attempt to supply both high power and energy densities. The primary power source in most miniature or microscale systems are batteries but there has also been extensive development of microfuel cells1,2 as a possible alternative to batteries. Combustion-based power systems combined with opportunities for integration with fuel cells, photovoltaic systems, and other thermal energy conversion systems present another attractive alternative due to even higher energy densities of combusting hydrocarbon fuels over that of direct fuel cells and batteries.3-7 Toward the goal of developing small-scale and portable fuel cells, there has also been significant interest in developing fuel reformers8-10 for generation of clean, high purity fuel (e.g., H2) for these energy-generation devices. Most fuel reformers require high temperatures (∼900 °C for some systems) making combustion-based systems very attractive heat sources.11,12 In addition, development of microscale combustion-based heat sources provides an avenue to supply high-quality heat for high-temperature microchemical reactors, engines, thrusters, and heaters.13-22 This article describes the influence of thermal and flow boundary conditions on flames within sub-millimeter combustors with the goal of providing data toward...
better engineering of microcombustors for the variety of energy and power applications mentioned above.

In the past, studies have shown the difficulties in sustaining homogeneous combustion using premixed flames in confined spaces with a large surface-area-to-volume ratio (>10^3 m⁻¹). To overcome some of the difficulties due to structural failure, non-premixed or catalytic combustion has been used with laminar flows and minimal influence of turbulent mixing as Reynold’s numbers are often below 100. In addition, the critical role played by surfaces in reducing or preventing quenching of essential radicals has also been discussed. Studies have evaluated the role of heat transfer, scale vortex chambers.

Microengineering micro stagnation-point flow combustor. 3235–3242.

scale vortex chambers. 3235–3242.

oxygen in microscale tubes. 3235–3242.

limit premixed flames in microchannels. 3235–3242.

Combustion Institute 2005 Proceedings of the

catalytic combustion in heat-recirculating burners. 3550


Other studies have evaluated the role of heat transfer, scale vortex chambers.

AIChE J.

metallic observations of methane-oxygen diffusion flame structure in a sub-


(51) Prakash, S.; Miesse, C. M.; Masel, R. I.; Shannon, M. A. Flame structure variations in sub-millimeter combustors due to heat transfer through combustion wall. Proceedings of the Fourth Joint Meeting of the U.S. Sections of the Combustion Institute, 2005.


(54) Dave, H. Further experiments on the combustion of explosive mixtures confined by wire-gauze, with some observations on flame. Philosophical Transactions of the Royal Society of London 1816, 105, 115–119.


(57) Tyndal, J.; Sound; D.; Appleton & Company: New York, 1897.
Al₂O₃ combustors. The purpose of this paper is to evaluate the influence of specific flow and thermal boundary conditions, as described later, on the steady-state flame structure observed in sub-millimeter combustion through experimental observations and computational fluid dynamics (CFD) modeling. It is hoped that by relating experimental observations to existing theoretical models and identifying boundary conditions that directly influence microcombustion, this article will provide a first step to future researchers toward developing better and more efficient microcombustors.

Experimental Section

The materials, surface preparation, and fabrication procedures for the Al₂O₃ non-premixed sub-millimeter or microcombustors have been described in detail previously. Briefly, two alumina sheets were machined with a diamond-coated saw to yield a Y-shaped microcombustor with a nominal width of 10 mm and length of 35 mm. The combustion occurs within a 5 mm wide by 0.75 mm deep channel. The overall thickness of the alumina wall is 1 mm. External wall temperatures are recorded by using type R thermocouples (TCs, Omega Engineering Inc., Stamford, CT). Combustion gases are high-purity methane (CH₄) and oxygen (O₂), which are controlled by M100B MKS mass flow controllers and flow through attached inlets. The entire assembly is packed in between two approximately 1 in. thick insulation layers of fibrous alumina. For certain experiments, as described later, a slot is cut out in the microcombustor to install a sapphire window to permit flame structure imaging. To evaluate the effect of changing combustor wall thermal conductivity, a metal plate (Mo coated with Pt) is secured to the external wall over the reaction channel for a combustor wall thermal conductivity, a metal plate (Mo coated with Pt) is secured to the external wall over the reaction channel for a few experiments. The metal plate is attached using a high-temperature alumina adhesive. The exhaust and unburned gases flow freely into the ambient air. The steady-state flame structure is visualized by still-frame imaging. The still frames are captured by a Canon EOS Mark II digital camera with an attached infrared filter.

Computational Model

A 3D computational fluid dynamics (CFD) model was developed to study the role of inlet flow conditions and thermal boundary conditions on the steady-state flame structure and provide a framework for validating the observed experimental trends. The numerical investigation is done in Fluent version 6.2, 59 with the governing equations given in the user’s guide. The model solves coupled species, momentum, diffusion, and energy equations. All thermo-physical properties except the thermal conductivity of the walls are taken to be temperature dependent. All physical constants and thermochemical data are obtained from the Fluent database, which relies on a NIST database for source information. The main assumptions used for the model are: (i) The flow remains laminar during the entire combustion process, as this has been established through previous experimental studies; (ii) the gas-phase chemistry is infinitely fast and controlled by the rate of mixing of CH₄ with O₂; (iii) single-step reactions occur with rate constants following Arrhenius type dependence; and (iv) surface reactions do not affect the steady-state flame structure, as the combustor is made from alumina and it has been shown that the surfaces are relatively inert and many surface reactions can be neglected to the first order. In addition, because of a lack of accurate models and thermochemical data for specific surface reactions on traditionally inert surfaces, no detailed predictions can be made. These assumptions have been chosen carefully to simplify the modeling efforts as the primary purpose of the model presented here is to complement and validate the trends observed in the experimental results and not to establish a theoretical foundation for the observed flame structure. To accurately model effects of the combustion process and to describe all of the observations, detailed surface reactions would need to be included to capture second-order and higher effects, which is beyond the scope of this article.

The governing equations used for the model are described next. The description for each of the terms can be found in the nomenclature list. Eqs 1 and 2 are the conservation of species and energy respectively, such that

\[
\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \vec{v} Y_i) = -\nabla \cdot J_i + R_i 
\]

(1)

\[
\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho \vec{v} h) = \frac{\partial}{\partial x_j} \left( \frac{\partial T}{\partial x_j} - \frac{\partial}{\partial x_j} \sum F_i J_i + S_h \right) 
\]

(2)

The specific heat at constant pressure is taken as a piece-wise-polynomial function of temperature as determined in the Fluent database. Transport of species is modeled using multicomponent diffusion based on the Stefan–Maxwell equations to obtain the generalized Fick’s law of diffusion and is given by

\[
\mathbf{J}_i = -\sum_{j=1}^{N} \rho D_{ij} \nabla Y_j - D_{rj} \nabla T 
\]

(3)

where \(D_{rj}\) is computed through the use of the Chapman–Enskog formula (eq 4) derived from the kinetic theory, such that

\[
D_{rj} = 0.0188 \left[ \frac{R^3}{M_{w,j}} + \frac{1}{M_{w,j}} \right]^{1/2} \frac{\rho_{db} \Omega_i^{2} \kappa_{ij}}{p_{bs} \Omega_i^{2} \kappa_{ij}} 
\]

(4)

and where \(D_{rj}\) is a function of chemical composition and temperature and is expressed by the empirical relation

\[
D_{rj} = -2.59 \times 10^{-7} T^{0.650} \left[ \frac{M_{w,j}^{0.511} Y_i}{\sum_{j=1}^{N} M_{w,j}^{0.511} Y_j} \right] - Y_j \left[ \sum_{j=1}^{N} M_{w,j}^{0.489} X_j \right] 
\]

(5)

The velocity field is determined by using the continuity and momentum equations, given by

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0
\]

(6)

and

\[
\frac{\partial \rho \vec{v}}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\frac{\partial p}{\partial x_i} + \nabla \left[ \frac{\mu}{\partial x_j} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial \varepsilon}{\partial x_i} \right) \right] 
\]

(7)

The mixture viscosity is determined using the kinetic theory of gases,

\[
\mu = \sum_{j} X_{\phi j} \mu_{\phi j}
\]

(8)

where \(\phi_{ij}\) and \(\mu_{ij}\) are given by

\[
\phi_{ij} = \left[ 1 + \left( \frac{\mu_{ij}}{\mu_{\phi j}} \right)^{1/2} \right]^{1/2} \left[ 8 + \left( \frac{\mu_{ij}}{\mu_{\phi j}} \right)^{1/2} \right]^{-1/2}
\]

(8.1)
The energy balance in eq 2 also accounts for the radiative heat transfer through the volumetric source term, $S_r$, as defined in the Fluent user manuals through the radiative transfer equation (RTE) and the weighted sum of gray gases model (WSSGM) model$^{60}$ used to calculate the absorption coefficient. The RTE incorporated is

$$\mu_j = 2.67 \times 10^{-6} \sqrt{\frac{M_{w,j} F}{\sigma_s^2 \Omega_{\mu,j}}} \quad (8.2)$$

The gas mixture thermal conductivity is determined from the thermal conductivity of each of the individual gases in the mixture using the ideal gas mixing law

$$\kappa = \sum_j \frac{X \mu_j}{\sum_j X j \phi_{ij}} \quad (8.3)$$

The energy balance in eq 2 also accounts for the radiative heat transfer through the volumetric source term, $S_r$, as defined in the Fluent user manuals through the radiative transfer equation (RTE) and the weighted sum of gray gases model (WSSGM) model$^{60}$ used to calculate the absorption coefficient. The RTE incorporated is

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = an\frac{\alpha^3}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{\pi} I(\vec{r}, \vec{s}')\Phi(\vec{s} - \vec{s}') d\Omega' \quad (9)$$

The energy balance in eq 2 also accounts for the radiative heat transfer through the volumetric source term, $S_r$, as defined in the Fluent user manuals through the radiative transfer equation (RTE) and the weighted sum of gray gases model (WSSGM) model$^{60}$ used to calculate the absorption coefficient. The RTE incorporated is

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = an\frac{\alpha^3}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{\pi} I(\vec{r}, \vec{s}')\Phi(\vec{s} - \vec{s}') d\Omega' \quad (9)$$

The energy balance in eq 2 also accounts for the radiative heat transfer through the volumetric source term, $S_r$, as defined in the Fluent user manuals through the radiative transfer equation (RTE) and the weighted sum of gray gases model (WSSGM) model$^{60}$ used to calculate the absorption coefficient. The RTE incorporated is

$$\frac{dI(\vec{r}, \vec{s})}{ds} + (a + \sigma_s)I(\vec{r}, \vec{s}) = an\frac{\alpha^3}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{\pi} I(\vec{r}, \vec{s}')\Phi(\vec{s} - \vec{s}') d\Omega' \quad (9)$$

Of the radiation models available in Fluent, the PN model has the broadest applicability for the work in this article. The equation used for solving the radiative flux was the PN model, which is the less complex case of the more general PN models. The equations are developed by multiplying the RTE (eq 9) by various direction cosines to yield two moment equations, which can be integrated to yield the Milne–Eddington equations.$^{61}$ The procedure is based on expansion of the local radiation intensity into an orthogonal series of spherical harmonics. The series is then truncated after a selected number of $N$ terms.$^{61}$ The model takes into account optical thicknesses in the range of optically thick (Roseland model) and thin mediums. The model also takes into account local emission and incoming and outgoing scattering from the gas mixture. The equation assumes the system to be in local thermodynamic equilibrium, which means that absorption in a small differential volume within the system is equal to the emission. For the CFD study reported, incoming and outgoing scattering was not considered and, as a consequence, anisotropic scattering can be neglected. The absorption coefficient of each species could be evaluated by calculating the WSGGM.$^{60}$ This method is implemented by fitting the absorption coefficient (eq. 9) and the emissivity factor in eq.

$$e = \sum_{i=0}^{l} \epsilon a(T)[1 - e^{-k_e}] \quad (10)$$

and radiative flux is related to the incident radiation by

$$\epsilon = \int_{\Omega} \frac{1}{3} \left[ 1 - \epsilon \right] - \epsilon \right] \nabla G \quad (11.1)$$

However, radiation intensity is also needed, which is related to the radiation flux by

$$\epsilon = \int_{\Omega} \frac{1}{3} \left[ 1 - \epsilon \right] - \epsilon \right] \nabla G \quad (11.1)$$

as described previously for radiating gas flow.$^{62}$ To obtain the temperature, radiative intensity, and the radiation flux, the RTE, the energy equation, and eq 11 need to be solved simultaneously.$^{59,61,63}$ The governing equations, mesh size, time-step, activation energy for flame ignition, and modeling methodology is similar to the CFD approach reported recently by some other researchers.$^{44,64,65}$ suggesting that the approach used for developing a model to validate experimental observations is based on acceptable methods already published.

**Results and Discussion**

One of the ways to better understand the microcombustion process is to observe and characterize the parameters that affect flame structure. The observed steady-state flame structure and subsequent heat output of the microcombustor, which is important for many applications, is strongly affected by the


Figure 2. CFD model results for the central plane temperature in the fluid within the microcombustor for varying relative inlet flow rates. Total flow rate for all cases is 300 sccm. (A) 100 sccm of CH₄ and 200 sccm of O₂, (B) equal CH₄ and O₂ flow rates, (C) 200 sccm of CH₄ and 100 sccm of O₂, and (D) equal CH₄ and O₂ flow rates but the external walls except at the gas inlets and outlets are modeled as adiabatic. The boundary conditions at the inlet and outlet are the same as those in cases (A)–(C). In cases (A)–(C), the combustor loses heat to the environment by convection (natural convection coefficient assumed to be 10 W/m²-K) and by radiation to the surroundings at 300 K.

Sub-Millimeter Combustion Energy & Fuels, Vol. 23, 2009 3553


have a significant temperature (500 °C or greater). Therefore, as the combustion process proceeds and heat generation occurs, the incoming gases undergo preheating. Experimentally, the heating of the gas inlet combustor legs was a problem in establishing secure gas inlet connections, which would often melt and cause gas leaks. However, the problem was resolved by using high-temperature ceramic tape to secure the connections. The preheating of incoming gases can be important as heat loss in microcombustion is often an area of concern.3 Part D of Figure 3 shows that the temperature distribution on the outside combustor wall is more uniform if the lateral walls are adiabatic. Once again, this result points to the value of thermal management in microscale combustion systems, which has been the subject of significant debate in the research community.3,68 Furthermore, in terms of developing design rules for building microcombustors the uniformity of external wall temperatures can be a significant consideration. It should be noted that, for the temperature predictions reported here, the radiation models are extremely important. In all models when radiation was ignored, the results generated by Fluent were not physically realistic and also did not match experimental observations at all. Therefore, as design rules for microcombustors are developed, heat loss through radiation will also be an important consideration. It can be seen from part A of Figure 2 and part A of Figure 3 that for the local fuel-oxidizer conditions, which are stoichiometric at the inlet, the fluid temperature and the wall temperature are higher than the other cases. In fact, near the anchor point, the CFD model predicts temperatures close to the adiabatic flame temperature (∼3000 K for CH4−O2 mixtures) in the gas phase as seen in part A of Figure 2. These results also correlate to the experimental observations (part B of Figure 1) where it was noticed that the local stoichiometric case at the inlet produces higher recorded wall temperatures than other inlet flow conditions. One more point to note about the CFD results presented in Figure 3 is that the highest wall temperature exists at the exit to the combustor similar to the experimental observations where the highest external wall temperature was recorded near the exit to the microcombustor. In addition, the general trends for temperature predicted here are in agreement with normal scale combustion69 and previously reported experimental data for microcombustors.30 The shape of the steady-state flame structure was also investigated as a function of relative inlet flow rates with parts A and B of Figure 4 showing still-frame images for two possible steady-state configurations of the flame structure. It was

Figure 3. CFD model results showing the temperature contours for the external walls of the microcombustor for varying relative inlet flow rates. Total flow rate for all cases is 300 sccm. (A) Equal CH4 and O2 flow rates, (B) 100 sccm of CH4 and 200 sccm of O2, (C) 200 sccm of CH4 and 100 sccm of O2, and (D) equal CH4 and O2 flow rates but the external walls except at the gas inlets and outlets are modeled as adiabatic. The boundary conditions at the inlet and outlet are the same as in cases (A)–(C). In cases (A)–(C), the combustor loses heat to the environment by convection (natural convection coefficient assumed to be 10 W/m·K) and by radiation to the surroundings at 300 K.

Figure 4. Figure shows the effect of mixing near the inlet to the combustion channel as the fuel and oxidizer come together. Even for equal flow rates of fuel and oxidizer, a small deviation from the center line is observed at the inlet. This is predicted by the streamlines in (C). Also, notice the bend in the flame structure toward the oxygen side in (B). Similar flame bending is observed in the CFD models also (parts (A) and (D) of Figure 2). A highly pronounced bend at the inlet occurs for asymmetric flow rates (part (A) of Figure 1).


observed (Figure 1 and Figure 4) experimentally that the flame deviates from the centerline location if the fuel or oxidizer have a higher relative flow rate. This deviation is understandable as a higher volumetric flow rate implies a higher mass and volume occupied by gas with the higher inlet flow rate. To minimize volumetric changes and flow shearing effect, several experiments were conducted at equal volumetric flow rates (150 sccm each) and hence equal flow velocity (∼0.67 m/s) of fuel and oxidizer. However, it was observed that even with equal flow rates a small deviation from the centerline occurs and is also predicted by the streamlines in the CFD model. The edge-like flame anchored at the inlet is also observed to bend toward the oxygen side as the temperature increases (part B of Figure 4), and is also predicted by the Fluent calculations (parts B and C of Figure 2). This flame bending can be understood based on the local fuel–oxidizer ratios. In each of the cases where flame bending is observed, the combustor is being run under fuel-rich conditions at the inlet. To sustain to the gas-phase reactions, the flame bends toward the oxygen side. Furthermore, the diffusion coefficient of methane is higher than that for oxygen, which shifts the reaction toward the oxygen side of the flow. Manipulating the relative inlet mixture composition, in principle, can provide another operational control parameter for the design and operation of microcombustors.

Figure 5 depicts the temperature at various y locations as a function of the lateral direction (x). The inlet to the combustion chamber is at y = −34 mm and the exit to the combustion chamber is at y = 0 mm. Following the temperature contour plots for the confined fluid in Figure 2 and the parametric plot in Figure 5, it can be observed that the highest temperature occurs at the inlet to the combustion chamber within the fluid. It can also be observed that the temperature distribution is symmetric with respect to the x axis. However, as the flow moves further downstream (toward y = 0) the temperature distribution is observed to be no longer symmetric about the center line (x = 0) with the temperature being higher on the oxygen side. The higher temperature on the O₂ side can be understood based on the relative mixing of the fuel and oxidizer and the reactions within the flow occurring off the center line, as discussed above. Figure 6 presents line plots for the contour plots of the microcombustor walls described in Figure 3. It can be observed that, for the representative lines plotted in Figure 6 for the center of the channel in the z direction (z = 1 mm) on the methane side of the combustion chamber, the wall temperatures rise steadily in the y direction as flow goes from the inlet to the outlet of the microcombustor due to the continuous heat generation resulting from the combustion process. The gradient between the inside wall, in contact with the reacting flow, and the outside wall is on the order of about ∼20 K. Furthermore, it is to be noted that the gradient at the inlet of the combustion chamber reaches an almost 50 K change within a distance of less than 1 mm in the direction of bulk flow. This temperature gradient decreases by 10 times near the exit (∼10 K over the last 2 mm) as the flow moves from the inlet toward the exhaust with the external diffusion flame extinguished. One caveat with respect to the CFD temperature gradient data discussed above is that the model fails to capture the discontinuous flame structure and instead predicts a continuous flame confined within the microcombustor. The formation of the discontinuous flame structure and presence of flame cells continues to remain an open question, despite several discussions in literature related to instabilities in non-premixed flames.27,30,45,53,70 In addition, the effect of surface reactions has also been neglected in the present study. As a consequence, the CFD model predicts temperature gradients that are lower than the experimentally observed temperature gradients. The under-prediction of temperature gradients is due the fact that there are no localized zones with high or low temperature that would arise due to a discontinuous flame structure, which implies that for the CFD model a relatively uniform temperature distribution exists and hence the predicted temperature gradients are lower than those measured experimentally.

Thermal boundary conditions were also investigated to determine their importance on the confined flames. The thermal conductivity of the external combustor wall was altered by attaching a metal plate (Mo coated with Pt, to prevent oxidation of the Mo). It was expected that the metal plate would assist in spreading the heat more uniformly along the length of the combustion chamber.

(70) Prakash, S.; Armijo, A. D.; Masel, R. I.; Shannon, M. A. In Unsteady flames in microcombustion, ASME International Mechanical Engineering Congress and Exposition, Chicago, IL, November 5–10, 2006; Chicago, IL, 2006.
The modeling results validate the global temperature trends for measured external wall temperatures; however, the CFD model fails to capture the intricate details of the flame structure. For example, experimental data shows that by attaching a metal plate to the outside wall temperature gradients along the wall can be reduced. The same is observed in the CFD model reported here where temperature gradients along the wall reduce from >100 °C/cm to on the order of 50 °C/cm. In addition, inclusion of radiative heat transfer is important toward accurate modeling of thermal conditions for the microcombustor as it has been shown in this article that ignoring radiation leads to unphysical results with predicted flame and wall temperatures deviating significantly from experimental results. The inlet flow conditions influence the shape and location of the flame within the microcombustor. This article shows that by carefully engineering the various boundary conditions such as the wall thermal conductivity, wall emissivity, and inlet flow conditions, the external wall temperatures and flame structure can be manipulated for possibly developing more efficient microcombustion systems with controllable, repeatable, and uniform heat output. It is hoped that this work will provide a platform for developing design rules toward better engineered microcombustors for varied applications in small-scale and portable power systems.

Acknowledgment. This work was supported by the Department of Defense Multidisciplinary University Research Initiative (MURI) program administered by the Army Research Office under grant DAAD19-01-1-0582. The authors thank Prof. Richard Masel and Dr. Craig Miesse for assistance during various phases of this work. The authors thank the Rutgers University Academic Excellence Fund (AEF) for financial support.

Nomenclature

Symbol, Description

Y, local mass fraction of species.

ρ, mixture density.

\( \vec{u} \), mixture velocity field.

J, diffusive flux of species.

R, net rate of production of a chemical species.

Table 1. Thermal Properties of Wall Materials

<table>
<thead>
<tr>
<th>Material</th>
<th>Thermal Conductivity (W/m·K)</th>
<th>Emissivity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Alumina</td>
<td>24</td>
<td>0.45</td>
</tr>
<tr>
<td>Stainless steel 316</td>
<td>21.5</td>
<td>0.4</td>
</tr>
<tr>
<td>Tantalum (Ta)</td>
<td>57.5</td>
<td>0.2</td>
</tr>
<tr>
<td>Molybdenum (Mo)</td>
<td>138</td>
<td>0.2</td>
</tr>
<tr>
<td>Tungsten</td>
<td>173</td>
<td>0.3</td>
</tr>
</tbody>
</table>

The last boundary condition evaluated was the effect of a heat source at the exit as heat loss in microcombustion often is considered a major hurdle toward developing systems with reproducible heat output.\(^{3,30,35-37,51}\) Toward the goal of evaluating the effect of an external heat source, the boundary condition at the outlet is changed so that the temperature of air at the combustor exit is 3000 K, close to the adiabatic flame temperature of methane—oxygen combustion. Experimentally, this case can be considered as the presence of an external diffusion flame at the exit (part A of Figure 1). One caveat with respect to the model conditions must be presented here. The boundary condition modeled is for thermal effects only: the species transport and interactions between an external flame and the confined flame are neglected in the CFD model. The CFD model predicts that the temperature of fluid in the central plane can increase to greater than 2500 K and the external wall temperature also reaches a maximum value of close to 1500 K. The high temperatures can be visually verified from part B of Figure 1, as observed from the glowing alumina walls (minimum temperature for alumina to visibly glow ~600 °C). In previous studies, wall temperatures of approximately 1200 °C have been reported.\(^{23}\)

Conclusions

The combustor channel. Indeed, the thermal energy was spread more uniformly and the temperature distribution with large gradients (>100 °C/cm) along the length of the combustor become negligible (~10 °C/cm or less).\(^{35}\) To validate the experimental data and better understand the role of attaching an external metal plate to the combustor wall, a CFD parametric study was conducted by varying the type of metal plate used. Four different metals that can sustain high temperatures were chosen: molybdenum (Mo), stainless steel 316 (SS 316), tantalum (Ta), and tungsten (W). Stainless steel has the lowest thermal conductivity, \( k = 21.5 \text{ W/m·K} \) (emissivity 0.4) and tungsten the highest thermal conductivity, \( k = 173 \text{ W/m·K} \) (emissivity 0.3) among the four materials chosen (Figure 7). Ta and Mo lie in between with respect to the thermal conductivity. The material properties used in the model are tabulated in Table 1. It can be seen from Figure 7 that the material with the highest emissivity (stainless steel 316, Table 1) leads to the highest heat loss due to radiation from the wall to the surroundings at room temperature and hence causes the external combustor walls to be the coolest by comparison. The stainless steel—alumina composite wall has a maximum temperature ~880 K, whereas the walls with lowest emissivity metals attached (Ta, Mo) reached a maximum temperature on the order of 950 K. It should also be noted that the CFD code predicts that by attaching metallic layers to the outside wall the temperature gradients along the wall can be reduced to about 50 °C/cm from >100 °C/cm in agreement with the assertion that metal plates on the outside walls assist in spreading the heat more uniformly along the combustor wall. However, it must be once again pointed out here that whereas the CFD model validates the global trend in experimental data for lower temperature gradients along the external wall, the model underpredicts the actual temperature gradient in contrast to the experimental data. This underprediction is attributed to the differences in the experimentally observed and model-predicted flame structures, as discussed above.

Figure 7. Predicted temperature profiles along the external wall for the heated exit boundary condition. The combustor channel is modeled as 40 mm length. Equal flow rates of CH\(_4\) and O\(_2\) are considered with a total flow rate of 300 sccm. The majority portion of the combustor channel is noted to have a uniform temperature distribution. The data presented is for a parametric study of various wall materials.

The high temperatures can be visually verified from part B of Figure 1, as observed from the glowing alumina walls (minimum temperature for alumina to visibly glow ~600 °C). In previous studies, wall temperatures of approximately 1200 °C have been reported.\(^{23}\)
\( h \), enthalpy of a chemical species.
\( u \), fluid velocity.
\( \kappa \), thermal conductivity of the mixture.
\( T \), absolute temperature.
\( S_h \), volumetric heat source defined by the user (e.g., radiation source term).
\( N \), total number of gas-phase chemical species.
\( D_{ij} \), binary mass diffusion coefficient.
\( \nabla Y \), gradient of the mass fraction of species.
\( D_T \), thermal diffusion coefficient.
\( \nabla T \), temperature gradient.
\( M_w \), molecular weight of species.
\( \Omega_D \), diffusion collision integral.
\( p_{abs} \), absolute pressure.
\( \sigma_{ij} \), the arithmetic mean of the diameters of molecules of species \( i \) and \( j \).
\( X \), mole fraction of species.
\( p \), mixture pressure field.
\( \mu \), dynamic viscosity of the mixture.
\( \delta_{ip} \), Kronecker delta.
\( \Omega_{ij} \), Lennard–Jones parameter (Fluent database).
\( \sigma \), Lennard–Jones parameter (Fluent database).
\( R \), Arrhenius molar rate of creation/destruction of a chemical species.
\( I \), radiation intensity.
\( \vec{r} \), position vector.
\( \vec{s} \), direction vector.
\( a \), absorption coefficient.
\( \sigma_s \), scattering coefficient.
\( x_n \), component of Cartesian coordinate system.
\( n \), refractive index.
\( \sigma \), Stefan–Boltzmann constant.
\( \vec{s} \), path length vector.
\( \vec{s}' \), scattering direction vector.
\( \Omega' \), solid angle.
\( \epsilon \), emissivity.
\( a_{ei} \), emissivity weighting factors for the \( i \)th gray gas.
\( k_i \), absorption coefficient for the \( i \)th gray gas.
\( P \), sum of partial pressures of absorbing gases.
\( S \), path length.
\( q_r \), radiative flux.
\( C \), linear anisotropic phase function coefficient.
\( G \), incident radiation.
\( k \), thermal conductivity of combustor wall (alumina or a composite with metal pieces).
Subscript \( i \), species \( i \) but if used in derivates with space coordinate, \( i \) denotes coordinate basis.
Subscript \( j \) or \( j' \), species \( j \) or \( j' \) but if used in derivates with space coordinate, denotes coordinate basis.

EF900040Q