Characteristics of H₂/air annular jet flames using multiple shear coaxial injectors

Wataru MIYAGI*, Takahiro MIKI**, Tsuneyoshi MATSUOKA* and Susumu NODA*
*Department of Mechanical Engineering, Toyohashi University of Technology
1-1 Hibarigaoka, Tempaku, Toyohashi, Aichi 441-8580, Japan
E-mail: noda@me.tut.ac.jp
**IHI Compressor and Machinery Co. Ltd.
3934 Inatomi, Tatsuno, Kami, Ina, Nagano 399-0492, Japan
Received 27 March 2014

Abstract

The heat transfer characteristics for the process tubes of a cracking furnace and the combustion chamber wall of a liquid oxygen/gas hydrogen rocket engine, both of which use multiple nozzles, are very important in terms of performance and safety. In order to determine the heat transfer characteristics of the combustion chamber wall, the present study investigates both experimentally and numerically the combustion characteristics of H₂/air annular jet flames using multiple shear coaxial nozzles in a small combustion chamber under normal conditions. Three-dimensional simulations are performed in order to clarify the flame-flame interaction. The standard \( k-\varepsilon \) model is used as the turbulence model, and so the evaluation of the model for multiple nozzles is also an objective of the present study. Each flame appears as an independent flame until amalgamation, at which point the temperature increases. Further downstream, the high-temperature regions once again merge and form a large flame. At this point, the flame becomes squeezed and the temperature distribution spreads rapidly in the radial direction downstream. The wall heat flux is strongly influenced by the flow characteristics. Heat transport is weak in the near field. The turbulent heat transport downstream is dominant where turbulence is developed, and thus the wall heat flux is increased. An increase in Reynolds number based on the airflow \( Re_{air} \) shifts the peak position of the wall heat flux upstream because the turbulent heat transport is enhanced. The increase in the recess shifts the amalgamation position upstream and shortens the flame length. Spreading of the flame is also suppressed. The temperature decreases downstream. The increase in the recess leads to a reduction in EINOx. Under the present experimental conditions, the numerical method reproduces the combustion characteristics with a high degree of certainty.

Key words: Multiple nozzles, Annular jet flame, Flame characteristics, Wall heat transfer, \( k-\varepsilon \) model

1. Introduction

Multiple nozzles are widely used in combustion burners in industry, primarily because they enable uniform heating and increase the reaction density, which are very important for process furnaces. In cracking furnaces, a uniform heat flux to reaction tubes is essential for achieving a high-quality reaction process. Increasing the reaction density can reduce the volume of the furnace (Fleifil, et al., 2004, Fleifil, et al., 2006). Another example of the application of multiple nozzles is a liquid oxygen/gas hydrogen rocket engine, which possesses numerous annular injectors. The expander bleed cycle (EBC) has attracted attention in the development of the next-generation rocket engines in Japan because of its simple structure and high safety characteristics. In the EBC, the propellant feed pump is driven by the expanded gas used to cool the combustion chamber. Efficient heat absorption is required in order to develop a large-thrust EBC, and therefore, predictions about the heat flux and a proper understanding of the effects on the combustion chamber wall are very important (Atsumi, et al., 2011). Knowledge about the wall heat flux is important for determining the useful life of the rocket engine.

Highly accurate prediction of the heat flux to the reaction tube or the combustion chamber wall is a challenging
issue. It is especially important in the development of rocket engines, because failures of prediction may lead to fatal results, when the numerical data are used in the design. Furthermore, liquid rocket engines consist of multiple shear coaxial injectors, which enhance mixing through the shear velocity difference. A number of studies have been performed on single-injector combustion (Oefelein and Yang, 1998, Oefelein, 2005, Oefelein, 2006, Zong, et al., 2004, Zong and Yang, 2006). Daimon et al. (2011) conducted a numerical simulation for H$_2$/O$_2$ combustion using a single-element coaxial injector. They reported that the wall heat flux reaches its maximum value at the position where the recirculation vortex attaches to the wall. However, few studies have examined multiple-nozzle combustion chambers. Masquelet et al. (2009) numerically investigated unsteady combustion inside a small-scale liquid rocket engine. However, for simplicity, they used an axisymmetric geometry to simulate multiple injectors aligned in a circle. This geometry ignores flame-flame interactions.

The present study focused on the heat transfer characteristics of the combustion chamber wall. Experimental and numerical investigations were carried out to determine the combustion characteristics of H$_2$/air annular jet flames using multiple shear coaxial nozzles in a small combustion chamber under normal conditions. Three-dimensional simulations were used in order to clarify flame-flame interactions. The standard $k$-$\varepsilon$ model was used as the turbulence model, and the evaluation of the model for the case of multiple nozzles was also an objective of this study.

2. Experimental setup and conditions

A schematic diagram of the combustion chamber used in the present study is shown in Fig. 1(a). It is made from carbon steel and has an inner diameter of 122 mm and a wall thickness of 15 mm, with the length from the nozzle face to the outlet being 480 mm. This chamber was replaced with a Pyrex glass tube when flame images were captured. Five shear coaxial nozzles are located in the center of the combustion chamber, as shown in Fig. 1(b). One nozzle is located at the center of the chamber and is surrounded by four other nozzles, which are aligned along a circle having a diameter of 32 mm. Figure 1(c) shows a shear coaxial nozzle consisting of an air post with an inner diameter of 5 mm and a rim thickness of 0.5 mm, and an H$_2$ annulus with an inner diameter of 8 mm and a rim thickness of 0.5 mm. The nozzle has a recessed structure with the air injection exit being inset $R = 5$ mm or 10 mm from the level of the H$_2$ injection port to enhance mixing.

In the present study, the flames were photographed in order to determine the comprehensive flame characteristics in the combustion chamber. In addition, the temperature distribution within the combustion chamber was measured and the heat flux through the chamber wall was determined. The flame images were captured using a digital camera (D-90, Nikon) with an exposure time of 1 s, an F value of 9, and a sensitivity of ISO 2000. The camera was set at a height of 200 mm from the bottom of the chamber and a distance of 600 mm from its center. The temperature distributions were measured using an R-type thermocouple with a diameter of 0.1 mm, which was inserted into the combustion chamber.
through the outlet. The heat flux was calculated based on the temperature difference between the inner and outer surfaces of the chamber wall measured using another R-type thermocouple.

The OH radical chemiluminescence and water vapor vibration luminescence were also visualized using intensified charge-coupled device (ICCD) cameras, Hamamatsu Photonics C5909 and C7265, respectively. In order to visualize the OH radical chemiluminescence, which reflects the reaction region, a band-pass filter with a center wavelength of 307.5 nm with a half-width of 10 nm was positioned in front of the lens of the C5909 camera. The C7265 camera had a sensitivity-wavelength range of 400 to 900 nm, which enables the water vapor vibration luminescence to be visualized, while eliminating the OH radical chemiluminescence. The water vapor vibration luminescence reflects thermal radiation. Note that the images were integrated along the optical path, i.e., they are not planar.

NOx concentration measurements were performed at the exit of the chamber using an NOx-O2 gas analyzer (SHIMAZU, NOA-700). A sampling probe with an inner diameter of 2 mm was cooled by water in order to freeze the reaction and withstand the high temperatures.

Table 1 lists the experimental conditions. In this experiment, the oxidizer and fuel were air and hydrogen, respectively. In order to investigate the influence of turbulence in the combustion chamber, the Reynolds number based on the airflow, $Re_{air}$, and the recess depth, $R$, were varied from 2,500 to 3,500 and from 5 mm to 10 mm, respectively. The overall equivalence ratio $\phi$ was fixed at 1.0. The effect of $\phi$ on the NOx emission was also investigated by the increase in the airflow rate with $R$ fixed at 5 mm. The multiple- and single-nozzle cases were compared with $R$ fixed at 5 mm.

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>Air velocity $U_{air}$ [m/s]</th>
<th>Fuel velocity $U_{H2}$ [m/s]</th>
<th>Recess $R$ [mm]</th>
<th>Global equivalence ratio $\phi$</th>
<th>Multiple nozzles</th>
<th>Single nozzle</th>
</tr>
</thead>
<tbody>
<tr>
<td>2,500</td>
<td>7.9</td>
<td>2.9</td>
<td>5</td>
<td>1</td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>3,500</td>
<td>11</td>
<td>4</td>
<td>5, 10</td>
<td>1</td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>4,000</td>
<td>12.5</td>
<td>4</td>
<td>5</td>
<td>0.9</td>
<td></td>
<td>*</td>
</tr>
<tr>
<td>4,500</td>
<td>14.1</td>
<td>4</td>
<td>5</td>
<td>0.8</td>
<td></td>
<td>*</td>
</tr>
</tbody>
</table>

3. Simulation method

Numerical methods used to simulate turbulent combustion involve several models for turbulent flow and reaction. This means that each method has an evaluation range for the combustion characteristics in terms of geometry, initial velocity, and species. If a numerical method is used to design the combustor, identification of this range is essential. Tucker et al. (2008) compared the results of computational fluid dynamics (CFD), based on Reynolds-averaged Navier-Stokes and large-eddy simulations techniques, with experimental results for a rocket injector. The present study uses the standard $k-\varepsilon$ model as the turbulent model and the partial stirred reactor model (Golovitchev, 2001, Somarathne et al., 2013) as the turbulence-reaction interaction model for hydrogen/air annular jet flames in a small chamber with multiple shear coaxial nozzles. This analysis is based on three-dimensional simulation of a 90° azimuthal section of the combustion chamber. In order to avoid the problem of numerical viscosity, advection terms are discretized with a second-order central derivative scheme. As mentioned earlier, evaluation of this numerical model for multiple nozzles is also an objective of this study.

The following are the governing equations with standard notations, in the order of the continuity equation, the momentum equation, the sensible enthalpy equation, and the mass-conservation equation for each species:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \quad ,
\]

\[
\frac{\partial (\rho \mathbf{U})}{\partial t} + \nabla \cdot (\rho \mathbf{U} \mathbf{U}) = -\nabla p + \nabla \cdot (\mathbf{r}_{eff}) + \rho \mathbf{g} \quad ,
\]

\[
\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho \mathbf{U} h) = \nabla \cdot (\rho \alpha_{eff} \mathbf{U} h) + \frac{D\rho}{Dt} + \omega \rho \quad ,
\]

\[
\frac{\partial \rho \bar{Y}}{\partial t} + \nabla \cdot (\rho \bar{U} \bar{Y}) = \nabla \cdot (\rho D_{\text{eff}} \nabla \bar{Y}) + \frac{\partial \bar{Y}}{\partial \bar{Y}}
\]

where a line over a variable denotes the mean value and a tilde means the density weighted mean. Jachimowski’s seven-step, seven-species kinetic model (1988) is used as the H\textsubscript{2}/air reaction model. The simulation code used was reactingFoam from OpenFOAM (2004). The CHEMKIN database was used to calculate the transport coefficients and variables of thermodynamics (Kee, et al., 1993). The computational domain corresponds to the experimental equipment shown in Fig. 1 and periodic boundary conditions are assumed to be applicable at 90° intervals in the azimuthal direction. The number of cells in the simulation was set to one million, and these cells were refined, especially near the nozzles and flames. The minimum mesh size was 0.3 mm, based on the grid independency of numerical result. The temperatures of the combustion chamber wall obtained from experiments were used as the boundary conditions for the wall temperature in the simulation. On the other hand, the \(k-\varepsilon\) model is based on the isotropic turbulence assumption. The assumption is not applicable for the vicinity of reaction zone, because the velocity acceleration owing to reaction occurs only in the normal direction to the flame surface and generates anisotropic flow field. However, the numerical error based on the anisotropic flow in the limited region is considered as being not large for simple jet diffusion flames (Lockwood and Naguib, 1975).

4. Results and discussion

4.1 Temperature and flow fields

Figure 2 compares the flame characteristics for single and multiple nozzles for \(Re_{\text{air}} = 3,500\) and \(\phi = 1.0\) in terms of (a) photographs of the flames, (b) numerical temperature distributions, (c) OH radical emission outlines, and (d) numerical turbulent kinetic energy maps in the near field. The photographs were taken from the direction of \(\theta = 45°\). Thus, the surrounding flames for the multiple nozzles case reflect the flames of both the front and back sides. In the photographs, the blue light-emitting region corresponds to the approximately the OH radical emission region, whereas the red light-emitting region is associated with vibrational excitation of water vapor. The other data are given at \(\theta = 0°\). The OH radical emission outlines are given from averaged images of one hundred images. Then, the outlines were determined on the basis of a brightness of 17 on bitmap images of OH radical emission as threshold. The threshold was determined by try and error so as to reproduce the outline of OH radical emission image. The calculations reproduce the appearances of both the single and multiple flames with a high degree of certainty. For the single nozzle, the emission picture reflecting the reaction region is open at the flame tip. The temperature at the top is decreased, and the tip is also open, as shown in Fig. 2(b). The center flame for the multiple nozzles is strengthened through thermal support from the surrounding flames and becomes longer. The temperature of the center flame is also

![Fig. 2 Comparison of flame characteristics between single and multiple nozzles (\(Re_{\text{air}} = 3,500, R = 5 mm, \phi = 1.0\)).](image-url)
increased at the amalgamation position. However, the OH radical emission outline exhibits only a slight change in the flame length, as shown in Fig. 2(c). This should indicate that the reaction of the center flame for the multiple nozzles is strengthened, but is completed around the amalgamation position. The turbulent kinetic energy for the single flame is larger in the near field than that for multiple flames, as shown in Fig. 2(d). This may decrease the flame length due to the enhancement of the mixing. On the other hand, the turbulent kinetic energy of the center flame for the multiple nozzles is decreased owing to the decrease in the velocity difference between the center flame and the surrounding flames. This is also a reason that the center flame becomes longer. Details of characteristics of the multiple flames are given below.

Figure 3 shows the measurement and simulation results for the mean temperature distribution in the chamber for \( Re_{\text{air}} = 3,500 \), \( R = 5 \text{ mm} \), and \( \phi = 1.0 \). The experimental and numerical data appear to be in good agreement with each other. At an axial distance of \( z = 80 \text{ mm} \), three temperature peaks appear. This shows that each non-premixed flame front is formed independently at the cross section. The feature disappears downstream of \( z = 140 \text{ mm} \) because of amalgamation of the flames. The peak temperature due to the amalgamation is slightly higher in the numerical result. However, the distribution reflects the real temperature distribution with a fair degree of precision. Figure 4 shows (a) a photograph of the flames, (b) a numerical mean temperature map at \( \theta = 0^\circ \), and (c) numerical mean temperature maps at three cross sections for \( Re_{\text{air}} = 3,500 \), \( R = 5 \text{ mm} \), and \( \phi = 1.0 \). The photograph was taken from the direction of \( \theta = 45^\circ \). The experimental and simulated values for the distance to the amalgamation of the flames are in good agreement. In the upstream region, each flame is independent until amalgamation occurs. At the amalgamation point, the emission intensity and the temperature both increase. Further downstream, the high-temperature regions again merge to form a large flame. At this point, the flame becomes squeezed and the temperature distribution spreads rapidly in the radial direction downstream. This phenomenon resembles the transition.
to a turbulent mixture of a jet, even though these flames are in turbulence at the nozzle exit. The phenomenon downstream is not confirmed in the photograph, which reflects only thermal radiation from the water vapor. However, the simulation sufficiently reproduces the features that appear in the photograph of the flame.

Figure 5 shows numerically determined mean axial velocity maps (a) at $\theta = 0^\circ$ and (b) at three cross sections, and (c) the turbulent kinetic energy and the experimental heat flux for $Re_{air} = 3,500$, $R = 5$ mm, and $\phi = 1.0$. The arrows in the velocity maps show only the flow directions, not the magnitudes. The arrows in Fig. 5(a) indicate the streamlines of the recirculation vortices. The potential core of the center jet is longer than the cores of the surrounding jets. This may be caused by a suppression of mixing and could be attributable to a decrease in the velocity difference between the center jet and surrounding gases. Another reason could be the mixing of the surrounding jets with the recirculation vortices, as shown in Fig. 5(a). The mixing results in shortening the length of the potential cores. The tips of the surrounding jets are bent toward the center jet slightly upstream of the vortex cores. This position corresponds to where the temperature distribution in Fig. 4(b) contracts. The heat flux has a peak between $z = 200$ and 260 mm, which corresponds to the wall attachment position for the recirculation vortex, as shown in Fig. 5(a). Therefore, the location of the recirculation vortex is an important factor that characterizes the wall heat flux distribution. Furthermore, Fig. 5(c) shows that the curves for the turbulent kinetic energy distribution and the heat flux distribution have similar characteristics. Therefore, the flow field also has a strong effect on the wall heat flux.

4.2 Effect of Reynolds number on the flame and heat transfer characteristics

Figure 6 shows the flow fields and the temperature fields for $Re_{air} = 2,500$ and 3,500 at $R = 5$ mm and $\phi = 1.0$. Figures 6(a), 6(b), and 6(c) show the mean axial velocity maps with the streamlines of the recirculation zone, the turbulent kinetic energy maps, and the temperature maps, respectively. Although the airflow rate was increased, no significant change occurred in terms of the jet width and the size of the recirculation vortex. For both cases, in the near field, the turbulence weakens, despite the recessed burner. The numerical data are obtained with a second-order central derivative scheme for advection terms, not upwind derivative scheme, in order to avoid the numerical viscosity. Thus the effect of artificial viscosity is not involved in the data. Therefore the decrease in turbulent kinetic energy in the near field should reflect the laminarization caused by the increase in laminar viscosity through rapid temperature increase and the decrease in turbulence viscosity owing to decrease in density. In the downstream region, the turbulent kinetic energy increases in the shear layer between the jet and surrounding fluid, specifically, for $Re_{air} = 3,500$. Thus, circulation of the vortex should increase with $Re_{air}$. The comparison between the two temperature maps indicates that the increase in $Re_{air}$ widens the temperature distribution at the amalgamation position of the center and surrounding
flames and decreases the temperature downstream, where the turbulent intensity is increased. Therefore, based on the above considerations, the effect of turbulent heat transport in the near field is small, and a little heat is transported in
the radial direction. On the other hand, in the downstream region, the heat is strongly transported in the radial direction because of the increase in turbulence. The increase in the Reynolds number enhances the heat transfer in the radial direction, in the downstream, owing to the increase in turbulence.

Figure 7 compares the wall heat flux and the turbulent kinetic energy for different $Re_{air}$ values at $R = 5$ mm and $\phi = 1.0$. Here, the heat flux is obtained experimentally, and the turbulent kinetic energy is calculated. For $Re_{air} = 2,500$, the wall heat flux gradually increases until $z = 260$ mm and then decreases in the downstream region. In the case of $Re_{air} = 3,500$, the heat flux exhibits a similar trend to that shown for $Re_{air} = 2,500$. However, the peak position of the wall heat flux is shifted upstream. Here, the values of heat flux for $Re_{air} = 3,500$ are larger than those for $Re_{air} = 2,500$ because of the increase in the fuel flow rate under $\phi = 1.0$. The turbulent kinetic energy increases with $Re_{air}$, and the peak location shifts downstream, in contrast to the heat flux. The increase in the turbulence should diffuse the heat earlier and transport the heat to the wall. This is indicated in Fig. 6(c) and is more clearly verified from the images of the water vapor vibration luminescence.

Figure 8 compares (a) the water vapor images and shows (b) the outlines of averaged water vapor images of one hundred images, for $Re_{air} = 2,500$ and 3,500 at $R = 5$ mm and $\phi = 1.0$. The images were taken from the direction of $0^\circ$. For $Re_{air} = 3,500$, the length of the water vapor jet is increased, and the fluctuation in the radial direction is increased.

**Fig. 9 Flame characteristics with respect to the recess ($Re_{air} = 3,500$, $\phi = 1.0$).**
strengthened. Thus, the outline of the water vapor jet spreads earlier and touches the wall upstream, as shown in Fig. 8(b). This is why the peak position of the wall heat flux is shifted upstream with increasing $Re_{\text{air}}$, as described above.

### 4.3. Effect of the recess depth on the flame characteristics

Figure 9 shows the effect of the recess on the flame characteristics for $Re_{\text{air}} = 3,500$ and $\phi = 1.0$, in terms of (a) photographs of the flames, (b) temperature distributions, (c) OH radical emission outlines, (d) axial velocity maps in the near field, and (e) turbulent kinetic energy maps with the streamlines of the recirculation zone. Figures 9(a) and 9(b) show that an increase in the recess decreases the flame length. Moreover, Fig. 9(b) shows that an increase in the recess shifts the amalgamation position upstream and decreases the maximum temperature, although this is not clear in the maps. The downstream temperature for $R = 10$ mm is clearly lower than that for $R = 5$ mm. The spread of the flame for $R = 10$ mm is also suppressed. The outline of the OH radical emission also shows that the flame length decreases as the recess increases. The axial velocity maps show that the flame holding position shifts slightly upstream for $R = 10$ mm, and the axial velocity is strongly increased near the exit of the fuel nozzle. This reflects the occurrence of reaction in the nozzle. Increasing the recess leads to enhanced fuel-air mixing and to the reaction in the nozzle. Furthermore, the outer nozzle directs axially gas flows accelerated by the thermal expansion and, as a result, suppresses spreading of the flame at the exit of the nozzle. The increase in the axial velocity leads to the increase in the turbulent kinetic energy downstream, as shown in Fig. 9(e).

Figure 10 shows the EINOx characteristics for different recesses. The EINOx decreases with increasing recess, and generally increases with the overall equivalence ratio. This is caused by the decrease in the flame temperature, as shown in Fig. 9(b).

![Fig. 10 EINOx characteristics with respect to the recess.](image)

**5. Conclusions**

In the present study, in order to clarify the combustion and heat transfer characteristics of $\text{H}_2$/air annular jet flames using multiple shear coaxial nozzles, we compared the results of numerical simulations and experiments using a small cylindrical combustion chamber and obtained the following conclusions.

Each flame is independent until amalgamation occurs. At the amalgamation point, the temperature increases. Further downstream, the high temperature regions once again merge and form a large flame. At this point, the flame becomes squeezed, and the temperature distribution spreads rapidly in the radial direction downstream. This phenomenon resembles the transition to a turbulent mixture of a jet, even though the flames are already in turbulence at the nozzle exit.

The flow characteristics are important for heat transport in the combustion chamber, and the wall heat flux is also strongly influenced by the flow characteristics in the combustion chamber. In the near field, the heat transport is weak because the turbulence is suppressed due to the laminarization phenomenon, despite the recessed nature of the burner. Turbulent heat transport is dominant downstream where turbulence is developed, and thus, the wall heat flux is...
increased. The peak position of the wall heat flux corresponds to the wall attachment position for the recirculation vortex. Under these conditions, an increase in $Re_{air}$ does not change the size of the recirculation vortex in the combustion chamber, despite the increase in the kinetic energy of the turbulence. As a result, the peak position of the wall heat flux is shifted upstream because turbulent heat transport is enhanced.

An increase in the recess shifts the amalgamation position upstream and shortens the flame length. Spreading of the flame is also suppressed. The temperature decreases downstream. The flame holding position inside the nozzle shifts slightly upstream with increasing recess depth. The axial velocity is strongly increased near the exit of the fuel nozzle. The increase in the recess leads to an EINOx reduction.

The center flame of the multiple flames is strengthened through the thermal support from the surrounding flames and becomes longer than the single flame. The temperature of the center flame is increased at the amalgamation position. The turbulent kinetic energy near the center flame is weakened because of the decrease in the velocity difference at the flame boundary.

Under the experimental conditions of the present study, the proposed numerical method based on the standard $k$-$\varepsilon$ model is applicable to $\text{H}_2$/air annular jet flames using multiple shear coaxial nozzles with a fair degree of precision.

**References**


