Numerical simulation of vortex structures in a two-dimensional vertical bluff-body burner for two different hydrocarbon fuels

H. Kawahara\textsuperscript{1} & T. Nishimura\textsuperscript{2}
\textsuperscript{1}Oshima National College of Maritime Technology, Yamaguchi, Japan
\textsuperscript{2}Department of Mechanical Engineering, Yamaguchi University, Yamaguchi, Japan

Abstract

The present article treats with numerical simulation of vortical structures in the combustion field in a two-dimensional bluff-body burner. The numerical results support the flow visualization results for C\textsubscript{3}H\textsubscript{8} combustion by Chin & Tankin. We identify the fuel effect on the dynamic vortical structures inside the flame for C\textsubscript{3}H\textsubscript{8} and CH\textsubscript{4} fuels. The thermal buoyancy is found not to be responsible for the flow instability inside the flame in the presence of a large co-flowing air stream although there exists thermally induced vortex outside the flame. 

Keywords: numerical simulation, vortical structure, bluff-body burner, fuel effect, thermal buoyancy

1 Introduction

Investigation of unsteady diffusion flames are important for understanding combustion phenomena in practical systems and developing theories of turbulent combustion process \cite{1-4}. Recently, direct numerical simulations of jet diffusion flames have provides experimental support and given additional insight into vortex/flame interactions. Yamashita et al. \cite{5} conducted a numerical study of the transition from laminar to turbulent flow in a two-dimensional planar methane jet diffusion flame in a small co-flowing air stream, using a flame sheet model of one-step irreversible reaction. Their results clearly show that the flow structures inside the flame interact with the flame. Roquemore & Katta \cite{6} simulated axisymmetric transitional diffusion flames of hydrogen, methane and propane fuels. It was revealed that viscosity, volumetric expansion and body
force due to buoyancy are responsible for the slower growth and the longer coherence length for the shear-layer vortices inside the flame.

While, non-premixed flame stabilized by a bluff-body combustor, such as occurs when a central fuel jet issues into a surround annular air jet, is often employed industrially. Such bluff-body combustors provide good flame stabilization as well as easy control of combustion. The stabilization mechanism is controlled by the interaction among combustible mixtures, air flow and hot combustion products. Exchange of species, momentum and energy becomes more intensive due to the existence of unsteady recirculating structures behind bluff-body. Therefore, a completed understanding of flow field in this region is very important for the study of flame stabilization. For axially symmetric combustors, there have been several experimental studies [7-9]. They indicated that vortical and thermal structures strongly depend on annular air stream. Furthermore, 2-dimensional bluff-body geometry has been studied to understand essential features of recirculating structures behind bluff-body. Mori et al. [10] found experimentally that two coherent vortical structures are generated inside and outside the flame, respectively, when external acoustic excitation is applied. Chin & Tankin [11,12] examined vortical shedding process under cold or combusting flow conditions, using flow visualization technique. Especially, for combusting flow of propane fuel [12], three flow zones are identified from the vortical shedding process, i.e., pre-penetration zone, penetration-transition zone and penetration zone, depending on central fuel stream. In the pre-penetration zone, oscillatory flame bulge is observed due to the behavior of recirculating structures consisting of inner and outer vortices behind bluff-body, in contrast to jet diffusion flames [6]. In the penetration-transitional zone, a strong interaction between inner and outer vortices behind bluff-body leads to flickering flame. In the penetration zone, the recirculating structure becomes steady and coherent vortical structures are observed even in the downstream region inside the flame. However, no detailed measurements such as velocity, temperature and concentration species of reacting field were reported, and also the mechanism of unsteady recirculating structures were not explained, which motivate the present investigation. We perform unsteady numerical simulation of combustion for propane and methane fuels on a two-dimensional bluff-body burner of Chin & Tankin [12].

2 Numerical simulation

The present computational domain is shown in Fig. 1. The flow is time-dependent and two-dimensional. A Cartesian coordinate system is taken such that \(x\) is the streamwise direction, \(y\) is the transverse direction and the origin is the center of the fuel slot exit plane. \(u\) and \(v\) represent the streamwise and transverse velocity, respectively, and \(z\) is the coupling function. The boundary conditions for \(u\), \(v\) and \(z\) for the combusting flow are described in the figure. The conservation equations for mass and momentum are given in the following forms.
Density is related to pressure \( p \), temperature \( T \) and mass fraction \( Y_i \) through the following state equation

\[
p = \rho R T \sum_{i=1}^{4} \left( \frac{Y_i}{m_i} \right)
\]

where \( m_i \) is the molecular weight of species \( i \) and \( R \) is the universal gas constant. The problem is further simplified by adopting the following coupling function \( z \), i.e., Shavb-Zeldvich formulation:

\[
z = \frac{Y_i - Y_{i,\infty}}{Y_{i,0} - Y_{i,\infty}} = \frac{Y - Y_{\infty}}{Y_{0} - Y_{\infty}} = \frac{h - h_{\infty}}{h_{0} - h_{\infty}}
\]

where \( Y \) is given by

\[
Y = \left( \frac{Y_F}{m_F \nu_F} \right) - \left( \frac{Y_O}{m_O \nu_O} \right)
\]

and \( h \) is the sum of the thermal and chemical enthalpies. Suffix \( o \) denotes the value for the injected fuel, while \( \infty \) denotes the value for the co-flowing air stream. The energy conservation equation and the mass conservation equations for each species reduce to the following equation for \( z \):

\[
\rho \left( \frac{\partial z}{\partial t} + u \frac{\partial z}{\partial x} + v \frac{\partial z}{\partial y} \right) = \frac{\partial}{\partial x} \left( \rho D \frac{\partial z}{\partial x} \right) + \frac{\partial}{\partial y} \left( \rho D \frac{\partial z}{\partial y} \right)
\]

The flame surface is located at the position where \( z \) becomes equal to \( z_f \), given by the equation:

\[
z_f = \left( \frac{Y_{O,\infty}}{j} \right) / \left( 1 + \left( \frac{Y_{O,\infty}}{j} \right) \right)
\]

where \( j = m_o \nu_o/m_F \nu_F \), i.e., mass of oxygen required to burn unit mass of fuel to attain complete combustion. The flame surface divides the whole region into two regions, i.e., fuel region inside the flame and air region outside the flame. The mass fractions and temperature in the respective regions can be written as follows for the when \( Y_{I,o} = 0 \)

Fuel region (\( 1 > z > z_f \))

\[
Y_i = Y_{i,\infty} (1 - z) \quad T = T_{\infty} + (T_o - T_{\infty}) z + (q_o/m_F \nu_F c_p) j^t Y_{O,\infty} (1 - z)
\]

Air region (\( z_f > z > 0 \))

\[
T = T_{\infty} + (T_o - T_{\infty}) z + (q_o/m_F \nu_F c_p) z
\]

In the present study, we shall adopt the following assumptions.
The mixture undergoes an overall one-step irreversible reaction. The reaction rate is infinitely fast and the reaction is concentrated within infinitesimally thin zone in the surface. The mixture behaves like an ideal gas. The Soret and Dufour effects, as well as the pressure diffusion, can be neglected. Specific heat at constant pressure of the mixture is constant. The Lewis number is equal to unity. Viscosity coefficient and diffusion coefficient depend on temperature. In the energy equation viscous dissipation can be neglected.

The numerical calculation was performed by the finite-difference method. The scheme adopted here was the SIMPLE method [5]. The time differential and convective terms in the governing equations are treated with the implicit Euler scheme and QUICK scheme, respectively. The computational domain is 0<x<14d and -5.5d<y<5.5d. The grid numbers are 91×257 non-uniform grid systems. The Reynolds numbers for fuel and air are defined by

\[ Re_{\text{fuel}} = u_f \cdot d / \nu_f \]
\[ Re_{\text{air}} = 2 \cdot u_a \cdot d / \nu_a \]

The Reynolds numbers are included with in the experimental range by Chin & Tankin [12].

1. Inlet boundary conditions
   fuel slot (A) :
   \[ u = u_f(y) \ (T = T_0, \ Yi = Yi_0) \]
   \[ v = 0 \]
   \[ z = 1 \]

   air slot (C) :
   \[ u = u_{\infty}(y) \ (T = T_{\infty}, \ Yi = Yi_{\infty}) \]
   \[ v = 0 \]
   \[ z = 1 \]

   Bluff-Body surface (B and D)
   \[ u = 0, \ v = 0, \ \partial z / \partial y = 0 \]

2. Outlet boundary conditions(F)
   \[ \partial u / \partial x = \partial v / \partial x = \partial z / \partial x = 0 \]

3. Side boundary conditions
   Et and Eb
   \[ (\partial u / \partial y)_g = (\partial v / \partial y)_n \]
   \[ (\partial z / \partial y)_g = (\partial z / \partial y)_n \]

Figure 1: Analytical domain and boundary conditions.
3 Result and discussion

3.1 Comparison with experiments by Chin & Tankin

Chin & Tankin [12] presented the typical structures of the flame and its vortical structures in the 2-dimensional bluff-body combustor of a C$_3$H$_8$ fuel system, using a flow visualization technique. Figure 2 shows photographs under two different flow conditions. One is an instantaneous picture in the pre-penetration region. There is a recirculating zone behind bluff-body consisting of inner and outer vortices, which are oscillating in the lateral direction. The other is a picture in the penetration region. The inner and outer vortices tend to keep steady state rather than unsteady state, and vortical shedding is observed downstream.

We simulated the combusting fields under the same experimental flow conditions. The former case is shown in Fig. 3 representing numerical streakline patterns at five moments during one oscillation cycle. The numerical streaklines agree well with the experimental ones (see Fig. 2). The flame is greatly deflected by the vortical motion inside the flame. The time histories of streamwise velocities indicate asymmetric flow, leading to a good fluid mixing in the recirculating zone behind bluff-body. The latter case is shown in Fig. 4. The vortex dynamics between numerical and experimental results are satisfactory. We can see that there are outerward rotating vortex and inward rotating vortex behind the recirculating zone. These vortices originate from the inner and outer vortices, respectively. Time variations of two velocities as also shown in Fig. 4 are in phase, indicating symmetric flow. Vortical shedding frequency is an important parameter in characterizing the instability of a flow field. Figure 5 shows the relationship between vortex shedding frequency and fuel Reynolds number, including experimental and numerical data. The agreement of these data is satisfactory. Thus, the validation of the present numerical simulation is confirmed.

![Figure 2: Visualization photographs in combusting flow by Chin & Tankin.](image-url)
Figure 3: Numerical results at $Re_{fuel}=92$ and $Re_{air}=515$ (i) time sequence of streaklines during one oscillation cycle, (ii) time histories of streamwise velocity.

Figure 4: Numerical results at $Re_{fuel}=360$ and $Re_{air}=515$ (i) time sequence of streaklines during one oscillation, (ii) cycle time histories of streamwise velocity.
3.2 Vortical structures for C₃H₈ and CH₄ fuels

In order to examine the effect of fuels on the vortical structure, we use two different hydrocarbon fuels, i.e., C₃H₈ and CH₄. The two fuels are different in density, viscosity and heat release etc. Especially, the density of C₃H₈ is larger than that of air, but the density of CH₄ is lower. The computations are conducted under the following flow conditions. The fuel Reynolds number ranges from 10 to 250 under a fixed air Reynolds number of 400.

Figure 6 shows several vortical structures for two fuels. The upper and lower parts in the figure present C₃H₈ and CH₄ results, respectively. At even a low fuel Reynolds number of 50, the flow field for C₃H₈ fuel is not stable and a flame bulge structure is observed downstream. The structure is similar to that known in axisymmetric diffusion jet flames [6], but is not due to thermal buoyancy outside flame. The reason for this is due to oscillation of the recirculating zone behind bluff-body. That is, the outer vortex above the inner vortex is oscillating in the recirculating zone, but not shown here. On the other hand, the flame bulge is not seen for CH₄ fuel. At a Reynolds number of 120, the vortical shedding and flame flickering are observed, which are induced by a strong interaction between inner and outer vortices for C₃H₈ fuel. A similar trend is seen for CH₄ fuel, but the magnitude is small. When the Reynolds number is increased furthermore, the flow changes from asymmetric to symmetric and the vortex street is seen. At a Reynolds number of 220, the inner vortex is shedding, but the downstream vortex rolls inward which is the same direction of rotation as the outer vortex. The reversal phenomenon is observed even in axisymmetric diffusion propane jets [6]. While, for CH₄ fuel, this phenomenon is never seen, although vortex shedding occurs. It should be noted that although the thermally induced vortex outside the flame is observed for any case, thermal buoyancy has little effect on the vortical structures inside the flame in the presence of a large co-flowing air stream [15].

![Figure 5: Comparison of numerical and experimental vortical shedding frequencies for different fuel Reynolds numbers.](image-url)
We examine the dynamic behavior of the vortices inside the flame, from the streamwise velocity fluctuations. Figure 7 shows the relationship between the amplitude of velocity and fuel Reynolds number for two fuels. For C$_3$H$_8$ fuel, the amplitude has a large value in the Reynolds number range of 50 to 180, where the outer vortex is detached and is shedding. While, for CH$_4$ fuel, the amplitude has a large value at the boundary between detached and attached vortices. The vortical shedding frequency results are shown in Fig. 8. The trend is similar for both fuels, in contrast to the amplitude result of Fig. 7. The frequency is slightly decreased with increasing the Reynolds number and has a minimum value, where the detached vortex is changed into the attached one. After that, the frequency is increased with the Reynolds number. It is striking that the frequency relationship is quite different between combusting and cold flows [14]. Thus, remarkable differences in the vortex structure for two fuels are identified.
Figure 7: Streamwise velocity amplitude vs. Re_{fuel}, (a) \( \text{C}_3\text{H}_8 \) and (b) CH\(_4\).

Figure 8: Vortical shedding frequency vs. Re_{fuel} for two different fuels.

4. Conclusions

We studied numerically vortical structures for combusting flow from a 2-dimensional vertical bluff-body burner. The following conclusive remarks have been drawn.

1. The present numerical simulation predicts well the flow visualization results of dynamic vortical structures obtained by Chin & Tankin.
2. The fuel effects on the vortical structure are identified in the pre-penetration, penetration-transition and penetration regions.
3. Thermal buoyancy is not dominant in the presence of a large co-flowing air stream although there exists a thermally induced vortex outside the flame.
This work was supported in part by a Grant-in-Aid for R&D from the Chugoku Technology Promotion Center of Japan.

References