

A Numerical and Experimental Assessment of Flameholder Effectiveness

Mohammad Shoeybi¹

Cascade Technologies, Inc., Palo Alto, CA, 94303

Frank Ham

Center for Turbulence Research, Stanford University, Stanford, CA, 94305

Heinz Pitsch

Institut für Technische Verbrennung, RWTH Aachen, Germany

Hung Le

Cascade Technologies, Inc., Palo Alto, CA, 94303

and

Barry Kiel and Balu Sekar

Air Force Research Laboratory, WPAFB, OH, 45433

An understanding of the effects of geometry and combustion processes on the flame stability is the first step required to design advanced, efficient, and stable combustion systems. Large-Eddy Simulation (LES) offers tremendous advantages for turbulent combustion simulations because the scalar mixing process and the dissipation rates, which are very important for flame stabilization, can be predicted with a high accuracy by LES. Moreover, accurate prediction of the numerical models is established based on their validation with experimental data. In this paper, a numerical and experimental assessment of flameholders in a combustion system will be presented. Both reacting and non-reacting cases are considered. The bluff body geometry developed by Ref 1 is used for validation. The experimental measurements are being performed at the AFRL test rig. An unstructured low-dissipative finite-volume LES solver is used to perform the numerical simulations. The computational domain is discretized using an unstructured body-fitted grid, which allows easy control over the grid resolution. A flamelet progress-variable (FPV) model is used to represent the combustion kinetics. The unresolved turbulent scales are modeled using dynamic Smagorinsky sub-grid closure. The operating conditions and test section geometry of the AFRL rig are used in the simulation. We have used six 1/8 inches-in-diameter fuel injection nozzles. To ensure consistency and to study the grid sensitivity, simulations are performed with two different grid levels. One grid contains pure hexahedral elements while the second grid takes advantage of unstructured mesh technology and is locally refined near the regions of actions. The flame stability is observed in both simulations. The simulation results will be validated with experimental measurements.

I. Nomenclature

FPV = Flamelet/progress variable
M = Momentum flux ratio

¹ Address all correspondence to this author; email: shoeybi@cascadetechnologies.com.

II. Introduction

AFTERBURNERS increase the thrust of a gas turbine engine by burning additional fuel with hot engine exhaust^{2,3}. State-of-the-art aircraft technology demands a substantial increase in this thrust, e.g. 50% increase in approach velocity, which in turn requires new and advanced designs. Maintaining flame stability over a wide range of operating conditions represents a major design challenge in combustion system design. An understanding of the effects of geometry and combustion processes on the flame stability is required to design advanced, efficient, and stable combustion systems.

The inflow mixture to the afterburner usually is at high-speed and high-temperature⁴. Because of the high speeds, flame stabilization is typically achieved by recirculating flows in the wake of bluff bodies. Bluff Bodies (flame holders) provide different features important for flame stability. As the presence of the flame holder modifies the velocity field, a recirculation zone (RZ) is generated aft of the flame-holder base. This recirculation zone provides sufficient residence time and chemical heat release needed to ignite the fresh gas mixture. In addition, the shear layer generated around the RZ provides a turbulent mechanism to mix fresh fuel and air with the combustion products from the recirculation zone^{5,6}. Finally, as the flame is stabilized past the flame holder base, combustion products at the end of the recirculation zone are entrained into the RZ to provide a self-ignition mechanism.

Key factors affecting flame stability are mixing of fuel and air, vitiation level of combustion products entering the combustion system, and flame holder vortex shedding. A comprehensive review of the subject and a collection of data for bluff-body-stabilized flames in combustion systems recently was provided by Ref. 7. Ref. 4 investigated the effect of the flame holder geometry on combustion efficiency. They reported a strong correlation between fuel to air ratio at extinction and the inflow parameters of pressure, temperature, and velocity. More recently, two experimental and numerical studies^{8,9} discussed flame extinction behind a set of bluff-body-stabilized flames. Ref. 1 also investigated the effect of geometry on improvement of the lean blowout limit of bluffbody stabilized methane flames. They showed that the presence of local cavities improves the blowout limits.

Several computational studies of bluff-body flame stabilization have been reported. Ref. 10, for instance, performed an LES study of a bluff-body-stabilized premixed flame. Ref. 11 studied a bluff-body configuration similar to the current work. They highlighted the effect of suppressing the von Karman shedding on the flame blowout characteristics. Refs. 12 and 13 used different numerical approaches in FLUENT to predict the Damk\''ohler number in a V-gutter-stabilized flame.

All these experimental and computational studies have contributed to the understanding of bluff-body supported flame stabilization; however, there still is a need for fundamental studies and flame holder testing. In this paper, a numerical and experimental assessment of a flameholder in a combustion system is presented. In Section III, we describe the experimental setup. Section V discussed numerical framework, and Section V numerical results are presented.

III. Laboratory Scale Flameholder

Recently, Ref. 1 developed new flame stabilizers for afterburner systems that lead to an improved static flame stability. They experimentally studied newly designed bluff body bases with various prototype base geometries showed that local cavity-based base geometries result in an improved flame stability. In this work, we base our studies on their bluff-body geometry.

A. Geometry

The bluff body geometry is shown in Figure 1-a. The basic geometry (solid lines in Figure 1-a) consists of a 4.5 inches long, 1 inch thick, and 6 inches wide rectangular body with a 1/2 inch in radius rounded nose. Six nozzles (three on each side) which are located 1/2 inch upstream of the base of the bluff body with 1 inch spacing produce two sets of opposing jets in the crossflow. Each of these nozzles has a 1/8 inches diameter. Furthermore, the dotted volume in Figure 1-a can be filled with various physical inserts as studied by

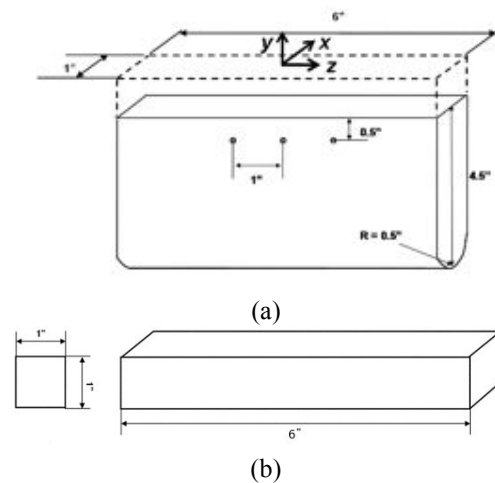


Figure 1. Bluff body geometry and the added block.

Ref. 1. Looking forward, the use of simulation allows for the analysis of the different flow structures induced by the different bluff body shapes and their impact on flame stability. Numerical simulation can be further used to optimize the bluffbody shape. As the first step of simulating such cases, we start with a base case. We replace the dotted section in Figure 1-a solid block extension with one inch square cross section (see Figure 1-b). This block modifies the geometry by increasing the height of the bluff body to 5.5 inches. The injection holes are unchanged and therefore, the reference case modifies the setup by moving the fuel injection jets in the upstream direction by 1 inch.

B. Experimental Setup

In parallel to the simulations, the experiments are going to be acquired at Air Force Research Laboratory (AFRL). The rig layout is shown Figure 2. At AFRL, the baseline flame holder, which is shown in Figure 1, is machined and placed in the 6"x5" test section. The rig is 43.5 inches long. Hot wire measurements including mean, rms and spectra will be performed at 5" upstream of the flame holder as well as at 3" downstream of the bluff body. The measurements at 5" will be used to impose inflow mean and turbulence to the simulation. At the downstream locations, data will be collected from the simulation and will be compared with the experimental measurements to examine the performance of computations.

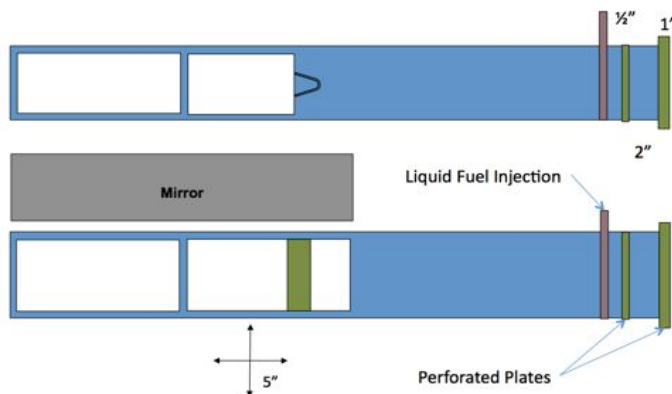


Figure 2. AFRL rig layout. Courtesy of AFRL.

IV. Numerical Framework

All the simulations here are performed using a fully unstructured multi-physics LES codes called CharLES. It solves both low-Mach number variable-density and fully compressible LES equations. The key feature of the code is the energy conserving unstructured collocated finite volume discretization of Ref. 14 and its subsequent improvements by Ref. 15. To model combustion, we use the flamelet/progress variable (FPV) approach developed by Ref. 16 is based on the flamelet concept¹⁷. This approach assumes that the chemical time scales are short enough that chemical reactions occur in thin layers around stoichiometric conditions¹⁸. The flamelet approach relates the species mass fractions and the energy to the mixture fraction through the flamelet equations.

V. Computational Setup

For the numerical simulation, we use the same configuration as the experiment. The inflow is placed 5 inches upstream of the leading edge. At this specific location measurements of turbulent intensities are going to be acquired and therefore, therefore, the intensities from the experiment can be used to generate inflow turbulence for the simulation.

Vitiated air at 700°K with the rate 0.34Kg/s enters the computational domain. As the first test case, no inflow turbulence is used. At the inflow, the mixture fraction is set to zero. Gaseous propane at 300°K is injected from the six fuel nozzles. The mixture fractions at the nozzle exits are unity. The jet velocity is adjusted such that the momentum flux ratio (M) is around 2. This relatively high value is chosen on purpose because at this high M , there are chances that the fuel injected into

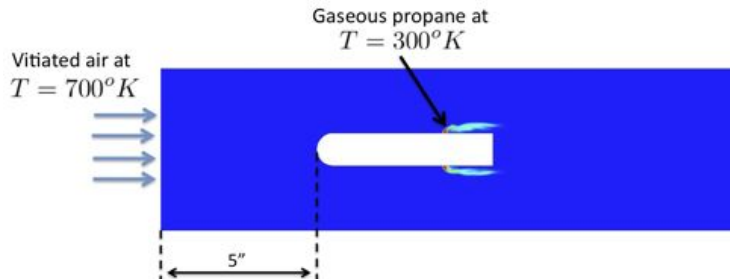


Figure 3. Schematic of the computational setup. The domain extends 27.5 inches downstream of the the trailing edge.

the cross flow does not mix with the air in the body's wake and therefore, the flame blows off. A schematic of the simulation setup is given in Figure 3.

Two different grid levels with 7 and 21 million elements (denoted by medium and fine grids) are used to examine the grid sensitivity. The medium grid contains pure hexahedral mesh elements. The resolution is distributed such that the grid is finer near the bluffbody walls and the injection nozzles. The fine grid takes advantage of Cascade's unstructured local refinement capability. In this grid, a homogeneous refinement is performed in the cells only close to the bluff body where a lot of action happens. Cascade's unstructured mesh technology with local refinement enables us to refine the grid only in the regions of action. This capability can be further leveraged towards more complex geometries. Figure 4 shows a spanwise cut ($z=0$ plane) and a vertical cut at $y=0.51$ " (right above the injection nozzles) of the the two grids. Further through the report we will discuss the importance of the refinement near the injection jets.

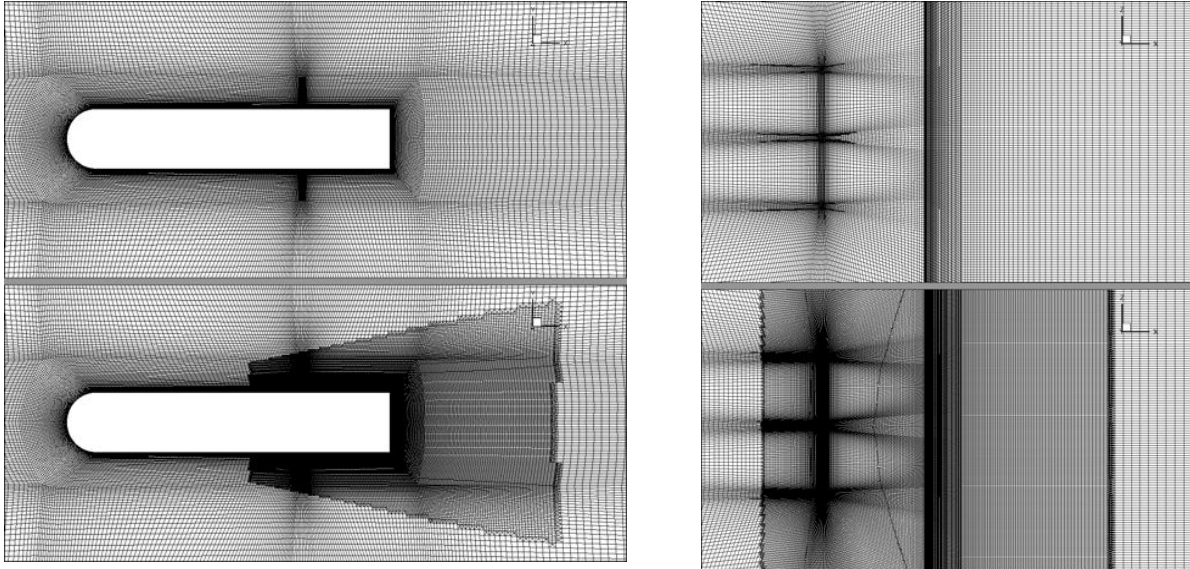
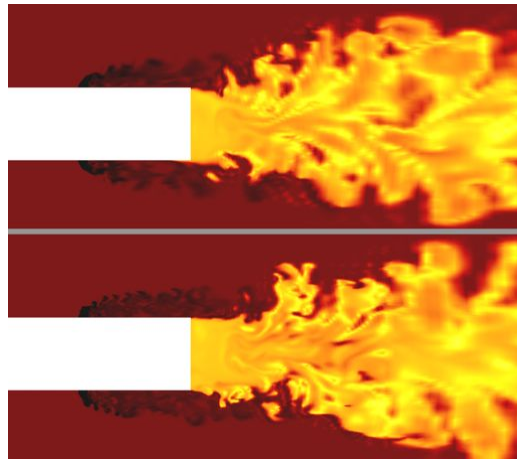


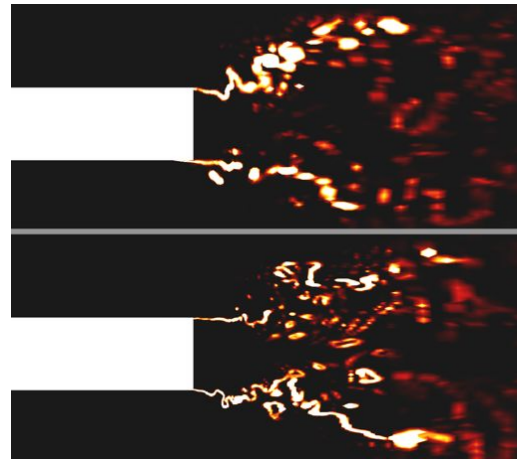
Figure 4. Two different grid levels used in the simulation. Top figures: medium grid; bottom figures: locally refined grid. Left figure: $z=0$ plane cut (middle plane; right figures: $y=0.51$ " plane (right above the fuel injection plane). Cascade's unstructured mesh technology is used to perform simulation on locally refined grids.

VI. Results

Figure 5 and Figure 6 compare the results from the two grids. Snapshots of both temperature contours and source term in the progress variable equation are shown in these figures. Two different planes at $z=0$ (mid plane in the spanwise direction) and $y=0.51$ " (plane right above the injection nozzles) are considered. Propane mixes with the vitiated air and it burns in the wake of the bluff body. The flame is stable although the momentum ratio is high and the overall equivalence ratio is low. The simulation with locally refined grid shows more structures and the flame in that simulation is thinner (clear in the progress variable source term contours). Due to three dimensional effects, some high temperature regions is observed near the test section walls. Overall, the flame behavior is similar in the two simulations, confirming that the flame stability is not grid sensitive, however more structures and thinner flames are observed in the locally refined grid. These details are important when sensitive quantities such as energy spectra and performance of the system are considered.

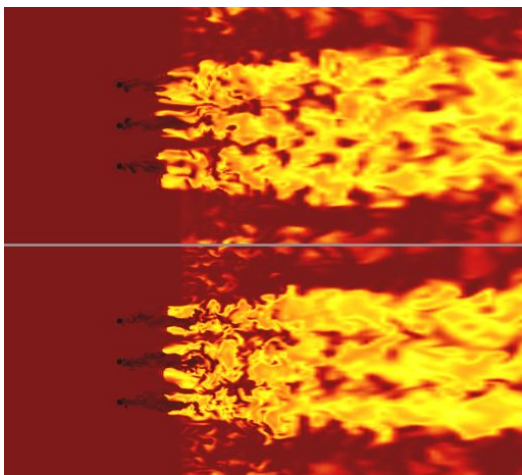


(a) Temperature contours

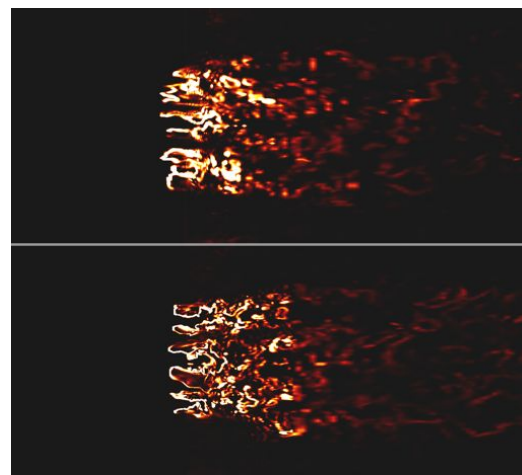


(b) Contour of reaction source terms for the progress variable

Figure 5. Results comparisons for $z=0$ plane cut; top and bottom figures are from the medium and locally refined grid simulations, respectively. Temperature contours are from 300°K to 3200°K and progress variable source term contours are from 0 to 100.



(a) Temperature contours



(b) Contour of reaction source terms for the progress variable

Figure 6. Results comparisons for $y=0.51''$ plane cut; top and bottom figures are from the medium and locally refined grid simulations, respectively. Temperature contours are from 300°K to 3200°K and progress variable source term contours are from 0 to 100.

To demonstrate the importance of the refinement near the regions of action, Figure 7 shows mixture fraction contours in the near fuel injection region for the two simulations. The jet instabilities and the interaction with the upstream turbulent boundary layer is much better represented with the locally refined grid. These small scales will represent the mixing and as a result the details of the flame/turbulence interaction in the wake of the bluff body with a better accuracy. We believe that with our unstructured mesh capabilities, we will be able to resolve the important features of the flow and flames in complex geometries, while keeping the computational cost tractable.



Figure 7. Comparison of mixture fraction for the two simulations with medium and locally refined grids. Contours are from 0 to 1. Top: medium size grid; bottom: locally refined grid. It is clear that the structures are more resolved in the locally refined grid.

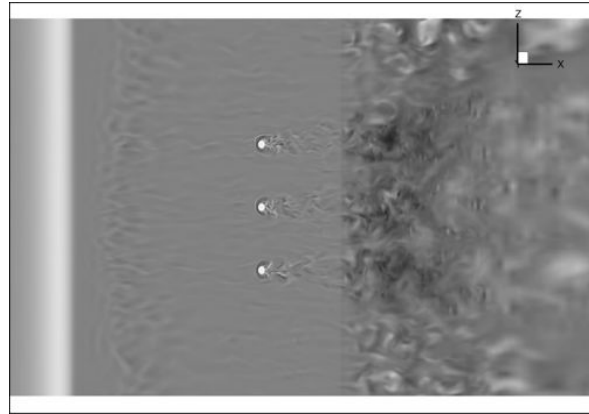


Figure 8. Wall normal velocity at the $y=0.51$ " plane. The fuel jets act as a solid body.

Figure 8 shows the wall normal velocity from the locally refined grid at the $y=0.51$ " plane. Flow undergoes a transition from laminar to turbulent right after the leading edge. The boundary layer becomes turbulent and it interacts with the jets. Another interesting observation is that the jets react as bluff body. This is due to the high momentum ratio (2) used in the simulation. The upstream turbulent boundary layer hit the jets and vortex shedding is observed in the wake.

Figure 9 shows temperature contours from the locally refined simulation at different y planes. At the centerline plane ($y=0$) the temperature seems to be homogeneous while at the other two planes, there are three clear jets that burn and then mix further downstream of the bluff body. At the $y=0.51$ " plane, reactions occur closer to the injection nozzles than that of plane $y=0.69$ ". One interesting observation is that at the centerline plane ($y=0$), hot regions are observed near the test section walls in the region close to the bluff body trailing edge. We believe this is due to the recirculation region in the wake of the bluff body and the fact that flow is three dimensional and the velocities are not high enough to push the hot gas outside of that region.

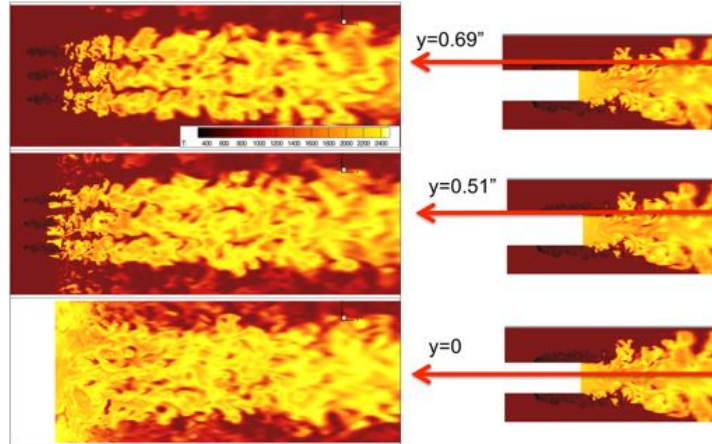


Figure 9. Temperature contours from the locally refined simulation at different y planes.

A. Sensitivity Analysis

To study the effect of inflow and fuel jet velocities on the flame stability, we have performed a numerical test. The injection nozzle size and the momentum flux ratio are kept the same. However, the vitiated air mass flow rate is doubled. To keep the momentum flux ratio identical, the injection mass flow rate is also doubled. To keep the CFL number the same, the time step size of the simulation is divided by half. All the other parameters are kept the same. The simulation is conducted up to a point that the solution is statistically stationary. Snapshots of temperature field at $z=0$ and $y=0.69$ " planes are shown for both the original simulation and the simulation with doubled velocities. It is

clear that with doubled velocities, the reaction zones are pushed back and the flame forms further downstream of the bluff body compared to the original simulation. Although the momentum flux ratio is the same for both cases, the flame in the case with smaller velocities is more attached to the bluff body. Nevertheless, the flame in both cases is stable confirming the suitability of the current setup.

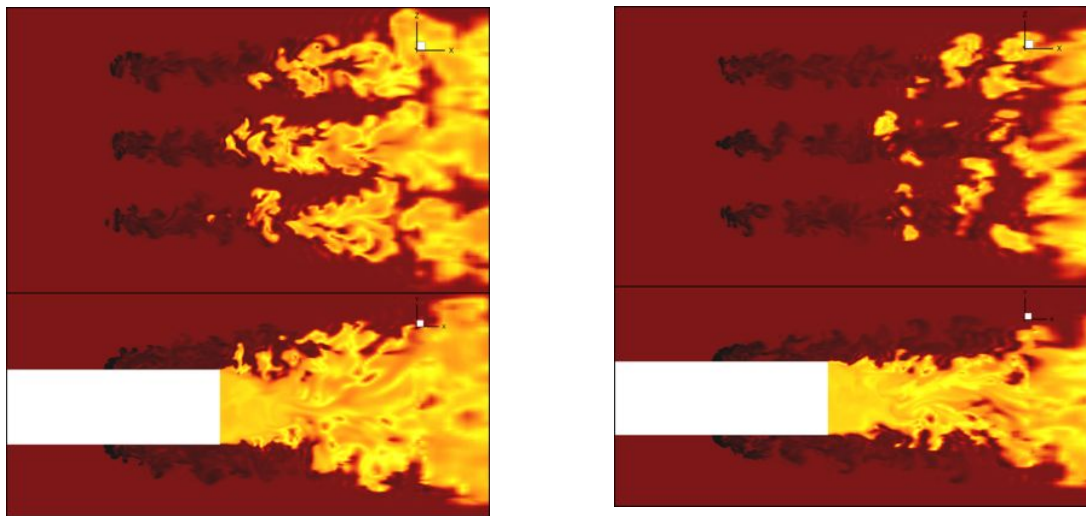


Figure 10. Temperature contours from the locally refined grid. Left figure: original test case from Section VI; right figures: the case with the same momentum flux ratio ~ 2 but with doubled inlet and fuel jet velocities.

VII. Validation

Experimental data are currently being acquired. The data for both reacting and non-reacting cases will be compared with the simulation results.

VIII. Acknowledgements

The work is sponsored by Air Force SBIR Program under the contract FA8650-10-M-2073.

IX. References

- ¹Kim, W., Do, H., and Mungal, G., 2010. "The improvement of blowout limit in partially/fully premixed flames with geometrically modified bluffbody bases."
- ²Lefebvre, A. H., 1998. *Gas Turbine Combustion*. Taylor and Francis, Philadelphia.
- ³Claire, S., 2008. *Gas Turbines*. Elsevier, San Francisco.
- ⁴King, C. R., and Nakanishi, S., 1957. "Effects of some configuration changes on afterburner combustion performance." *NACA Report, NACA-RM-E57C01*.
- ⁵Zukowski, E. E., and Marble, F. E., 1955. "The role of wake transition in the process of flame stabilization on bluffbodies." In *AGARD Combustion Researches and Reviews*. Butterworth Scientific Publishers, London, pp. 167–180.
- ⁶Longwell, J. P., Forest, E. E., and Weiss, M. A., 1953. "Flame stability in bluff-body recirculation zones." *Journal of Industrial and Engineering Chemistry*, **45**, pp. 1629–1633.
- ⁷Shanbhogue, J. J., Husain, S., and Lieuwen, T., 2009. "Lean blowoff of bluff body stabilized flames: Scaling and dynamics." *Prog. Energ. Comb. Sci.*, **35**, pp. 98–120.
- ⁸Kiel, B., Garwick, K., Gord, J. R., Miller, J., Lynch, A., Hill, R., and Phillips, S., 2007. "A detailed investigation of bluff-body stabilized flames." In *45th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, AIAA 2007-168.
- ⁹Kiel, B., Garwick, K., Lynch, A., Gord, J. R., and Meyer, T., 2006. "Non-reacting and combusting flow investigation of bluff-bodies in cross flow." In *42nd AIAA/ASME/SAE/ASEE Joint Propulsion Conference 9-12 July, 2006*, Sacramento, AIAA 2006-5234.

¹⁰Porumbel, I., and Menon, S., 2006. "Large eddy simulation of bluff body stabilized premixed flame." In *44th AIAA Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, AIAA 2006 0152.

¹¹Khosla, S., Leach, T. T., and Smith, C. E. "Flame stabilization and role of von Karman vortex shedding behind bluff-body flameholders". In *43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference 8-11 July, 2007, Cincinnati, OH, AIAA 2007-5653*.

¹²Knaus, D. A., Magari, P. J., Hill, R. W., Phillips, S. D., and Kiel, B. V., 2008. "Predicting augmentor static stability using local Karman number." In *46th AIAA Sciences Meeting and Exhibit, 7 - 10 January 2008, Reno, NV, AIAA 2008-1027*.

¹³Roach, J. M., Fisher, T. C., Frankel, S. H., Sekar, B., and Kiel, B. V., 2008. "Cfd predictions of Karman number fields for reduced order modeling of v-gutter flame stability." In *46th AIAA Sciences Meeting and Exhibit, 7 - 10 January 2008, Reno, NV, AIAA 2008-0509*.

¹⁴Mahesh, K., Constantinescu, G., and Moin, P., 2004. "A numerical method for large-eddy simulation in complex geometries."

¹⁵Ham, F. E., and Iaccarino, G., 2004. "Energy conservation in collocated discretization schemes on unstructured meshes."

¹⁶Pierce, C. D., and Moin, P., 2004. "Progress-variable approach for large-eddy simulation of non-premixed turbulent combustion." *Journal of Fluid Mechanics*, **504**, pp. 73-97.

¹⁷Peters, N., 1984. "Laminar diffusion flamelet models in non-premixed turbulent combustion." *Progress of Energy and Combustion Science*, **10**, pp. 319-339.

¹⁸Pitsch, H., 2006. "Large eddy simulation of turbulent combustion." *Annual Review of Fluid Mechanics*, **38**, pp. 453.