REPORT DOCUMENTATION PAGE

Form Approved OMB No. 074-0188

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	1999	Proceedings

4. TITLE AND SUBTITLE
Flow Field Considerations for Counter Flow Burners

5. FUNDING NUMBERS
N/A

6. AUTHOR(S)
M.P. Davis, J.W. Fleming, B.A. Williams, H.D. Ladouceur

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Combustion Dynamics Section, Code
6185, Chemistry Division, Naval Research
Laboratory, Washington, DC 20375-5342

8. PERFORMING ORGANIZATION REPORT NUMBER
N/A

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
SERDP
901 North Stuart St. Suite 303
Arlington, VA 22203

10. SPONSORING / MONITORING AGENCY REPORT NUMBER
N/A

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION / AVAILABILITY STATEMENT	12b. DISTRIBUTION CODE
Approved for public release: distribution is unlimited.	A

13. ABSTRACT ***(Maximum 200 Words)***
Many computational combustion tools (e.g. Sandia OPPDIF code) of opposed jet, counter flow diffusion flames apply a simplified potential flow model for the fluid flow portion of the problem. The resulting solution (for no flame) yields a stagnation region between the two burners and an axial velocity component that is independent of radius. J.C. Rolon recognized the need for experimental evidence to provide justification for a potential flow assumption. His results indicate that for a range of velocities and for two common burner designs (straight tubes with screens and dual converging nozzles) there are large discrepancies between the assumed flat radical profile and experimental measurements.

14. SUBJECT TERMS	15. NUMBER OF PAGES
SERDP, SERDP Collection, counter flow burner, combustion	4
	16. PRICE CODE
	N/A

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
unclass	unclass	unclass	UL

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. Z39-18
298-102

Davis, M.P., Fleming, J.W., Williams, B.A., and Ladouceur, H.D., "Flow Field Considerations for Counter Flow Burners," *Proceedings of the Fall Technical Meeting of the Eastern States Section of the Combustion Institute,* pp. 200-203, 1999. PP-1059

Flow Field Considerations for Counter Flow Burners

M.P. Davis[1], J.W. Fleming, B.A. Williams, H.D. Ladouceur[2]
Combustion Dynamics Section, Code 6185, Chemistry Division
Naval Research Laboratory, Washington, DC 20375-5342 USA

[1]Graduate Student, WPI
[2]Corresponding author: Code 6112, douglad@ccf.nrl.navy.mil

INTRODUCTION

Many computational combustion tools (e.g. Sandia OPPDIF code [1]) of opposed jet, counter flow diffusion flames apply a simplified potential flow model for the fluid flow portion of the problem. The resulting solution (for no flame) yields a stagnation region between the two burners and an axial velocity component that is independent of radius. Rolon [2] recognized the need for experimental evidence to provide justification for a potential flow assumption. His results indicate that for a range of velocities and for two common burner designs (straight tubes with screens and dual converging nozzles) there are large discrepancies between the assumed flat radial profile and experimental measurements. This paper investigates the flow field that exists between the nozzles of various counter flow burners through the use of a commercially available CFD package. Better understanding of this flow regime will lead to improved burner designs that more closely approximate the one dimensional treatment used in the combustion models.

The potential flow model ignores viscous effects, resulting in a failure to predict boundary layer development along the tube walls. A no-slip boundary condition at the tube walls causes fluid adjacent to the walls to be retarded, thereby increasing the velocity of the fluid along the centerline (in order to satisfy mass continuity). This results in a much higher centerline velocity (for the same flow rate) than that predicted by the potential flow model. A simple potential flow model was examined by neglecting viscous effects and solving the reduced Navier-Stokes equations. The solution is that of Laplace's equation for the velocity potential, which was evaluated for a geometry describing two 1 cm diameter tubes separated by 1 cm using a finite element package (FLEXPDE) [3]. This potential flow model consistently under predicts the observed flow velocity. This is reasonable since the acceleration of the fluid core within the tubes by viscous boundary layer growth is neglected.

In the case of two opposed flows, the situation is more complicated. Flow in both tubes will develop boundary layers resulting in the center core being accelerated. If the separation distance between the two tubes is small, the velocity profile at the tube exits will be greatly affected by the presence of the other tube due to the higher pressure stagnation region formed by the two impinging streams. The shape and length of the burner tubes, the tube separation distance, and the inlet flow velocities all play important roles in determining the shape of the radial velocity profile at the tube exits.

The opposed flow problem was studied using PHOENICS [4], a widely utilized CFD software. The geometry and flow conditions can be easily modified, and parametric studies were run to investigate different burner configurations over the entire desired range of strain rates. PHOENICS is used to better understand the fluid mechanics involved, as well as being a useful design tool to determine optimal burner geometries that produce flow fields more appropriate for the one-dimensional models.

COMPUTATIONAL SETUP

The CFD calculations were performed using PHOENICS version 1.6, running on an SGI workstation. PHOENICS uses a finite volume approach to provide solutions of differential equations having transient, convection, diffusion, and source terms. For this application, the appropriate equations are the incompressible Navier-Stokes equations. The user is able to specify geometry, boundary conditions, fluid properties, as well as many other options that control the solution procedure, output, etc. Taking advantage of the axisymmetric nature of the problem, the solution was specified in a single plane and then rotated to produce the desired 3-D burner. Boundary conditions were specified along each of the four edges of the solution domain as well as along the tube walls. At each of the tube inlets, a uniform velocity was set, and no-slip conditions were specified for each tube surface. For the far field radial boundary, a pressure boundary condition was utilized to set the pressure to 1 atmosphere.

RESULTS

Calculations were performed for a single, 50 cm long, 1 cm diameter straight tube flowing N_2 in order to assess the predictive capability of the code. For comparison, experimental measurements of the radial profile of the axial velocity were made on a single burner tube having the same dimensions as were used in the PHOENICS calculation. Data were collected at a height of 1 mm above the tube exit plane and a N_2 flow rate of 2.1 SLPM using an LDV system (QSP, Inc.) seeded with 0.3 μm diameter alumina particles entrained into the N_2 flow. Figure 1 plots both the measured and PHOENICS calculated axial velocities as a function of tube radius. Excellent agreement is observed between the measured and calculated velocities.

Calculations were carried out to investigate the effects of tube length on the shape of the radial velocity profiles. Figure 2 shows radial profiles of the axial velocity for N_2 flowing at an inlet velocity of 40 cm/s for tubes of lengths 3, 5 and 10 cm. The profiles show the viscous effects that predominate along the tube walls, lowering the local velocity, giving rise to a boundary layer, and accelerating the core fluid. For longer tube lengths, the boundary layer grows until a parabolic velocity profile results, after which the velocity profile ceases to change with increasing tube length (fully developed flow). The 10 cm tube demonstrates this, as the flat portion of the profile near the centerline has nearly vanished. For the 3 cm long tube, the velocity profile within 2 mm of the tube centerline is flat, closely approximating the one dimensional assumption made in the flame models. Thus, the degree of flatness in the radial profile can be varied by the length of the tube.. The optimal tube length depends on several factors including gas properties, temperature, and most importantly flow rate.

Converging nozzles are also used to generate flat radial profiles. However, the flat radial profile for an unopposed nozzle is affected by the presence of an opposing nozzle. Figure 3 presents the axial velocity across the nozzle exit for flow between two opposed converging nozzles. The contour of the nozzles is that specified by Rolon [2] and is described by an exponential curve. The opposed flows are presented for five strain rates, *a*, defined as the maximum axial velocity gradient in the streamwise direction. For higher flow velocities, a higher pressure area develops along the centerline at the stagnation plane, causing an axial velocity depression transverse to the flow. This results in the "M" or "W" shaped profiles seen in Figure 3. Such profiles were observed experimentally by Rolon [2] at similar strain values. This same nozzle produces an extremely flat profile in isolation.

The same phenomenon that causes the dip in the profile for the opposed converging nozzles will tend to flatten out the curvature from viscous effects seen in an unopposed straight tube. Thus, in order to achieve a flat profile near the flame zone, the exit radial profile for the isolated burner must lie somewhere between that of a fully developed parabolic flow and that of a nozzle designed to yield a completely flat profile. Computational determination of the optimum nozzle shape is an inverse problem, one that is not easily solved. Calculations with various contours suggest that a cubic or fourth order curve does a better job at producing flatter radial profiles for opposed flow conditions. Calculations are continuing to address this issue as well as to predict the results of Figure 2 for opposed tubes.

The choice of the optimum counter flow burner design depends on several parameters including choice of fuel and oxidizer, strain rate range of interest and whether one will be using powders or liquid aerosols in either flow. Straight tubes with movable flow straighteners are much easier to build than a converging nozzle, although flow straighteners complicate the introduction of aerosols. The structures imposed on the profile due to individual screen openings must also be considered. However, the possibility of producing flatter velocity profiles over a larger range of strain rates by varying the lengths of straight tubes will have clear advantages.

CONCLUSIONS

It has been shown that CFD can provide valuable insight into the flow physics of counter flow diffusion flame burners. Although no combustion is modeled here, many CFD software packages have the ability to tackle both the flow geometry and the chemistry (e.g., PHOENICS 3.1).

These results can serve as a starting point for those interested in designing an opposed flow burner that provides a one dimensional flow field more amenable to one dimensional flame models. For a given burner design, the radial velocity profiles will be very different depending on flow velocity, tube separation, length, and burner shape. It is possible to obtain a relatively flat radial velocity profile although this can only strictly be accomplished at a specific flow velocity and tube separation. Either straight tubes, converging nozzles, or variable length sections of straight tubes have the potential of being able to extend the range of strain that could be studied while maintaining some aspect of one dimensionality.

ACKNOWLEDGEMENTS

This research is sponsored by the Department of Defense's Next Generation Fire Suppression Technology Program funded by the DoD Strategic Environmental Research and Development Program. The authors would like to thank Will Gorton and Mike Kozma for their assistance..

REFERENCES

[1]. OPPDIF: A Fortran Program for Computing Opposed-Flow Diffusion Flames", SANDIA Report SAND96-8243, UC-1409, May 1997.

[2]. "Counter Jet Stagnation Flows", J.C. Rolon, D. Veynante, and J.P. Martin, *Exp. in Fluids*, **11**, 313-324 (1991).

[3]. FLEXPDE: Finite Element Analysis for Partial Differential Equations, SPDE, Inc., Fremont, CA (1995)

[4]. PHOENICS, Version 1.6, Concentration Heat and Momentum, Limited (CHAM), Bakery House, London, (1992).

Figure 1: Axial Velocity vs. Radial Position for a Single 50 cm Tube

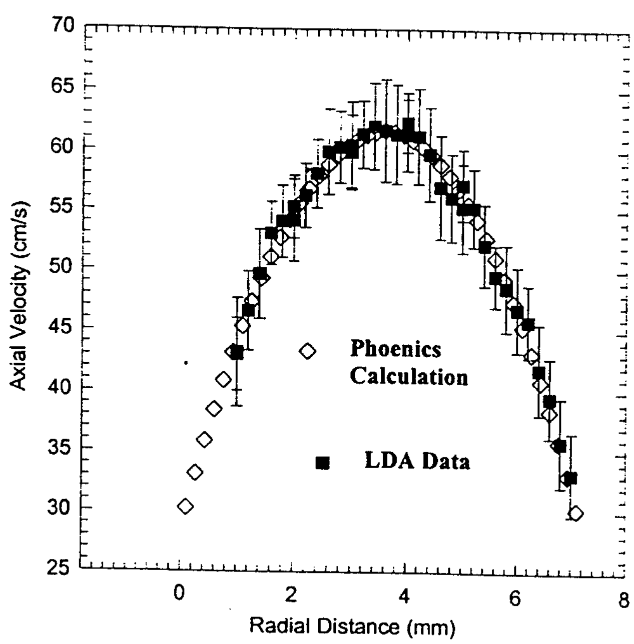

Figure 2: Phoenics Calculations for a Single Tube of Axial Velocity vs Radius for Varying Tube Lengths

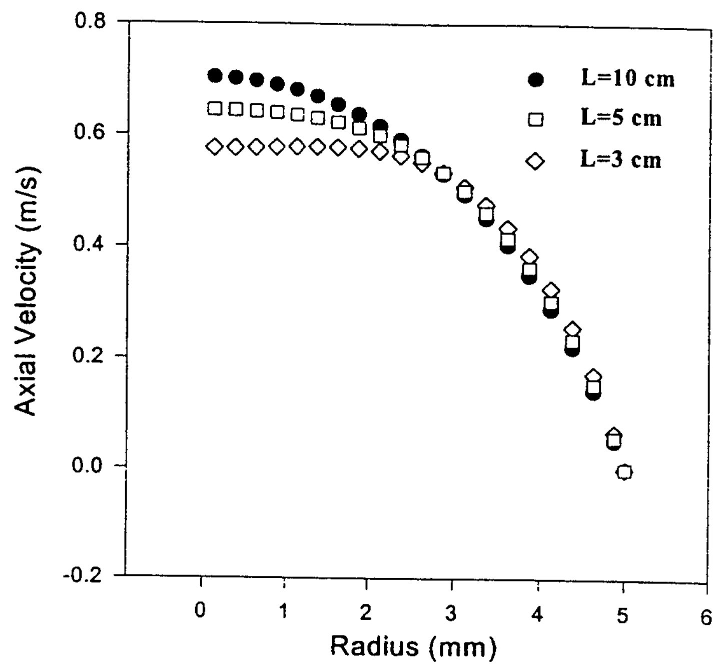

Figure 3: Normalized Axial Velocity at Tube Exit vs Radial Position for Specified Strain Rates, *a* For Two Opposed Converging Nozzles