Combustion in a Transonic Flow with Large Axial and Transverse Pressure Gradients

Jinsheng Cai*, Olgu Icoz†, Feng Liu‡ and William A. Sirignano§

Department of Mechanical and Aerospace Engineering
University of California, Irvine, CA 92697-3975

Abstract

A computational code has been developed for solving the full compressible Navier-Stokes equations with multiple species and chemical reactions. A k-omega turbulence model is employed. Fuel is injected at the entrance to a curved turbine passage amidst the main flow of heated air and combustion products exhausting from the main combustor. The code is used to study the flame structure in a transonic flow under the influence of large axial and transverse pressure gradients typical of conditions in a turbine passage. It is found that the transverse pressure gradient does not directly affect the steady-state flame structure significantly. However, the velocity nonuniformity due to the transverse pressure gradient thickens the flame in the low speed region and increases the combustion rate globally compared to the situation in a straight channel. Computational results will be presented on the ignition and flame structures under conditions as determined by inflow conditions, curvature of the channel, and variation of cross-sectional area. Comparisons with previous results for laminar flows and for algebraic turbulent models will be made.

1 Introduction

Designers of jet engines are attempting to increase the thrust-to-weight ratio and to widen the range of engine operation. Since the flow in a turbine passage is accelerating and power is extracted from the flow, it is possible to add heat without increasing the flow temperature beyond the turbine blade material limit. Sirignano and Liu [1, 2] show by thermodynamic analysis that the thrust of aircraft turbojet and turbofan engines can be increased significantly with little increase in fuel consumption by intentionally burning fuel in the turbine stages. For the ground-based gas turbine, benefits have been shown to occur in power/weight and efficiencies [1]. A mixing and exothermic chemical reaction in the accelerating flow through the turbine passage offers, therefore, an opportunity for a major technological improvement. The gas turbine engine is not the only potential application for this technology. The reduction in peak temperatures due to acceleration results in the promise of reduced pollutant formation and reduced heat transfer losses in many other combustion applications.

In order to provide insight into the fundamental behavior of multidimensional flows with mixing and chemical reaction in the presence of strong pressure gradients that support a transonic flow, Sirignano and Kim [3] obtained similarity solutions for laminar, two-dimensional, mixing, reacting and nonreacting layers with a pressure gradient that accelerates the flow in the direction of the primary stream. Fang, Liu and Sirignano [4] extended that study to mixing layers with arbitrary pressure gradients by using a finite difference method for the boudary layer equations. The influence of pressure gradient, initial temperature, initial pressure, initial velocity, and transport properties

*Visiting Research Specialist, Associate Professor, Northwestern Polytechnical University, Xi'an, China. E-mail: jcai@yoda.eng.uci.edu

†Graduate Student Researcher. E-mail: oicoz@uci.edu

‡Associate Professor. E-mail: fliu@uci.edu

§Professor. E-mail: sirignan@uci.edu

were studied. These solutions offer insight into the effect of flow acceleration on the flame structure in the mixing layer. However, they are restricted to only laminar flows and do not account for the transverse pressure gradient that are typical in a turbine blade passage.

Mehring, Liu, and Sirignano [5] extend the laminar boundary layer calculations of Fang, Liu, and Sirignano [4] to include turbulence by using a two equation $k - \omega$ turbulence model. The present authors developed a finite-volume method for solving the two-dimensional full compressible Navier-Stokes equations with chemical reaction without the assumption of a thin mixing layer [6]. However, only the Baldwin-Lomax algebraic turbulence model was used. In this paper, we extend our computations to include a more general two-equation $k - \omega$ turbulence model as was done in the boundary layer computations [5]. The method is used to compute transonic turbulent mixing flows in a curved duct to examine the ignition and combustion processes in a general transonic accelerating mixing layer.

2 Governing Equations and the Computational Method

The flow within a turbine blade row is at high speed and often transonic. There are large gradients of pressure, density and velocity in the flow field. One needs to investigate the mutual interaction between the combustion processes and the flow to understand both the combustion and determine the aerodynamic performance of the turbine blade rows. For this purpose, a model that is capable of handling transonic flow and chemical reaction is needed.

A finite-volume method with multigrid for accurately calculating the flow through turbomachinery blade rows based on the above system of equations was developed by Liu and Zheng [7, 8]. A flux-difference splitting scheme based on Roe splitting has also been developed [9]. Although there has been much work on computation of reactive flows, there are practically none on high-speed reacting flows in turbomachines. In this research we extend our CFD code for turbomachinery flows to include a one-step chemical reaction models.

The two-dimensional, Favre-averaged Navier-Stokes equations for N species are written in integral form over a fixed control volume Ω as

$$\frac{\partial}{\partial t}\iint_{\Omega}\theta(x,y)\mathbf{w}\,d\Omega + \oint\theta(x,y)(\mathbf{f}\,dS_x + \mathbf{g}\,dS_y)$$
$$= \oint\theta(x,y)(\mathbf{f}_\mu\,dS_x + \mathbf{g}_\mu\,dS_y) + \iint_{\Omega}\theta(x,y)\mathbf{s}\,d\Omega \qquad (1)$$

where the vector $\mathbf{w}$ is formed by the conservative variables for mass, momentum, energy, the turbulence kinetic energy k, and the turbulence specific dissipation rate ω. $\mathbf{f}$ and $\mathbf{g}$ are the convective flux vectors; $\mathbf{f}_\mu$ and $\mathbf{g}_\mu$ are the diffusive flux vectors; and $\mathbf{s}$ is a volume source term. $\theta(x, y)$ is the streamtube thickness function, which is equivalently the channel height in the third dimension in order to vary the cross-sectional area of a channel with constant width so that a streamwise pressure gradient may be imposed on the flow. Definitions of other variables in the above equations will be given in the final paper.

Methane (CH_4) is used for the current computations although the method is not restricted to only one type of fuel. The combustion process is described by the same one-step overall chemical reaction used in Ref. 4 as: $CH_4 + 2O_2 + 7.52N_2 \longrightarrow CO_2 + 2H_2O + 7.52N_2$

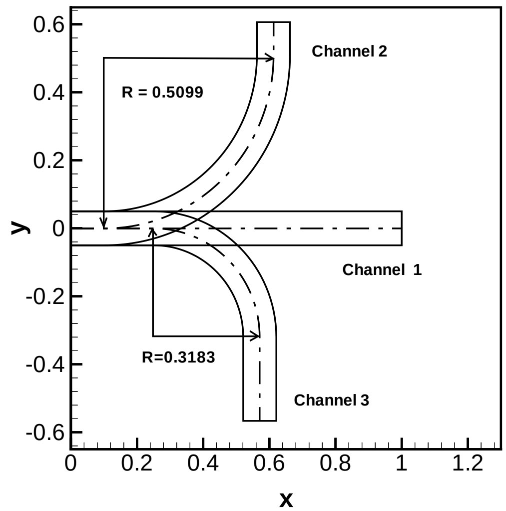

Figure 1: Schematic of two curved channel and a straight channel.

3 Computational Results

Results based on the algebraic model are summarized here. Detailed computational results and analysis based on the two equation turbulence model will be presented in the final paper.

3.1 Mixing Layer in a Curved Duct

In order to simulate the flow conditions in a typical HP turbine passage, a curved channel with fuel/air mixing is considered. Three different cases shown in Figure 1 are studied and compared to each other. In all three cases, the top half of the channel is injected with hot air at the inlet, while the bottom half is injected with fuel. Channel 1 is the base straight channel case. Channel 2 curves upward with a radius of $R = 0.5099$. Channel 3 curves downward with a radius of $R = 0.3183$. The streamwise pressure gradients are kept the same for all three cases by appropriately adjusting the streamtube thickness function $\theta(s)$. All other conditions are kept the same as the base case. Inviscid boundary conditions are used on the channel walls since the focus of the study is the effect of transverse pressure gradient on the flame. There is no transverse pressure gradient in the straight channel. The flows in Channel 2 and Channel 3 show significant transverse pressure gradient due to flow turning. Channel 3 has a stronger transverse pressure gradient due to its smaller radius of curvature.

Velocity, temperature, and density profiles at different streamwise distances will be shown in the final paper. Despite the large differences in the magnitude and direction of the transverse pressure gradients, the solutons near the center line appear to be essentially unchanged. There are, however, large differences towards either side of the channel walls since there are large differences in the pressure, density, and velocity values in those regions due to the transverse pressure gradients.

3.2 Fuel Injected in the Middle of a Curved Channel

Consider the situation where fuel is injected at the inlet between 20% below and 20% above the centerline of the three curved channels discussed in the above section. Figures 2 shows the temperature distributions in the direction normal to the centerline (shown as y in the figure) at $s = 7\ cm$,

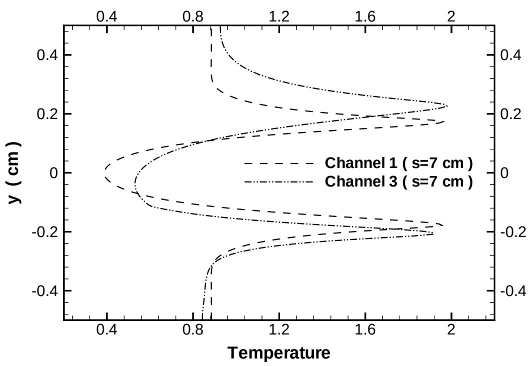

Figure 2: Temperature distribution in Channels 1 and 3 with fuel injected in the middle at $s = 7\ cm$.

where s is the arclength along the centerline of the channel. Only results for Channels 1 and 3 are shown. There are two peaks along the y direction which correspond to the two flames where air and fuel are in contact.

Compared to the flames in the straight channel the lower flame (closer to the inner wall) for Channel 3 is thinner while the upper flame is thicker than those for Channel 1. The two flames in the curved channel spread wider than those in the straight channel, indicating that the curved wall enhances combustion in a global sense.

3.3 Fuel Injection into a Curved Channel Typical of a Turbine Blade Passage

Figure 3 shows the geometric configuration and the computed temperature distribution in a curved channel typical of a turbine blade passage. The Mach number at the inlet of the duct is close to 0.1. The flow accelerates inside the duct from subsonic to supersonic. The Mach number at the exit of the duct is 1.4. There are large pressure gradients along the flow direction and also normal to the flow direction. Fuel is injected in the middle of this channel at the inlet. Figure 3 plots the temperature contours in the channel. The two flames at the interfaces between fuel and air are clearly seen.

4 Conclusions

Ignition and combustion processes in a transonic accelerating mixing flow in a curved duct are studied by a finite-volume method for the full elliptic Navier-Stokes equations and a two-equation k-ω turbulence model. Computations show that the velocity nonuniformity due to the transverse pressure gradient thickens the flame in the low speed region and appear to increase the combustion globally compared to the situation in a channel without transverse pressure gradient. Detailed analysis on the ignition and flame structures under conditions as determined by inflow conditions, curvature of the channel, and variation of cross-sectional area will be presented and conclusions drawn in the final paper.

References

[1] Sirignano, W. A. and Liu, F., "Performance Increases for Gas-Turbine Engines Through Combustion Inside the Turbine," *Journal of Propulsion and Power*, Vol. 15, No. 1, Jan. 1999,

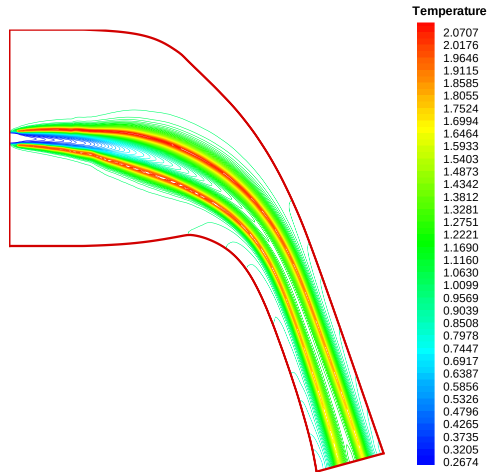

Figure 3: Temperature contours in a channel typical of a turbine blade passage.

pp. 111–118.

[2] Liu, F. and Sirignano, W. A., "Turbojet and Turbofan Engine Performance Increases through Turbine Burners," to appear in *Journal of Propulsion and Power*, May 2001, Also AIAA Paper 00-0741.

[3] Sirignano, W. A. and Kim, I., "Diffusion Flame in a Two-dimensional, Accelerating Mixing Layer," *Physics of Fluids*, Vol. 9, No. 9, 1997, pp. 2617–2630.

[4] Fang, X., Liu, F., and Sirignano, W. A., "Ignition and Flame Studies for an Accelerating Transonic Mixing Layer," AIAA Paper 00–0437, Jan. 2000, under revision for Journal of Propulsion and Power.

[5] Mehring, C., Liu, F., and Sirignano, W. A., "Ignition and Flame Studies for an Accelerating Transonic Turbulent Mixing Layer," AIAA Paper 01–0190, Jan. 2001.

[6] Cai, J., Icoz, O., Liu, F., and Sirignano, W. A., "Ignition and Flame Studies for Turbulent Transonic Mixing in a Curved Duct Flow," AIAA Paper 01–0189, Jan. 2001.

[7] Liu, F. and Zheng, X., "A Staggered Finite Volume Scheme for Solving Cascade Flow with a $k-\omega$ Turbulence Model," *AIAA Journal*, Vol. 32, No. 8, 1994.

[8] Liu, F. and Zheng, X., "A Strongly-Coupled Time-Marching Method for Solving the Navier-Stokes and k-ω Turbulence Model Equations with Multigrid," *J. of Computational Physics*, Vol. 128, 1996.

[9] Zheng, X. and Liu, F., "Numerical Solution of Navier-Stokes Equations and k-ω Turbulence Model Equations Using a Staggered Upwind Method," *AIAA Journal*, Vol. 33, No. 6, 1995.