Computing High Speed Mixing Flows - Two-equation vs. Algebraic Models

R. Krishnamurthy
D'Anthony M. Woods
S. Chandra

NASA Center for Aerospace Research
North Carolina A&T State University
Greensboro, NC 27411

Abstract This study considers analysis of data obtained from tests with a generic combustor model at the pulse facility T5 located in the Graduate Aeronautical Laboratories at California Institute of Technology (GALCIT). The analysis was performed using the numerical code GASP, version 2.2. Comparisons were made between the flow predictions using Baldwin-Lomax and k-epsilon turbulence models. Computed predictions of mixing efficiency, wall pressure signatures, eddy viscosity distributions, etc. obtained from both the turbulence models were similar. Computed wall pressure distributions agreed with those from experiments. Based on the results obtained, use of Baldwin - Lomax turbulence model can be considered to be sufficient for the conditions considered, as it yields adequate predictions with significant savings in computational cost.

1. Introduction

In recent years there has been a resurgence in research related to air-breathing propulsion systems. The primary impetus for this was the National Aerospace Plane(NASP) Program which had as its goal the development of a vehicle capable of transatmospheric flight operating in the hypervelocity regime. Predictions from previous tests performed on scramjets operating in the supersonic range may not be relevant for the hypersonic case. It is for this reason that tests have been performed at T5 (Hornung [1992]), the newest of the free-piston driven, reflected shock tunnels, located at the Graduate Aeronautical Laboratories at California Institute of Technology (GALCIT). T5's capabilities allow flow simulations with free stream velocities up to 6 km/s. A separate combustion-driven shock tunnel has been designed to supply "hot" hydrogen. Similar to existing propulsion systems developed and used by both military and civilian organizations, NASP will employ a propulsion system that uses its fuel as a coolant also. The effect of serving as a coolant on the fuel is an increase in enthalpy. Thus, the availability of hot hydrogen in T5 allows for a more realistic simulation. Also, the high pressure condition (43.9 kPa) that is obtainable for the test gas is representative of actual flight-capable propulsive systems.

Analysis of recent data obtained by Belanger [1993], from tests of a generic combustor model (Fig. 1) at T5 is the subject of the present investigation. The tests were conducted in a rectangular combustor model with the dimensions of 711 mm (length) by 50.8 mm (width) and 25.4 mm (height). Tests were performed with both cold and heated hydrogen injected at an angle of 15 degrees into a mainstream of either air or nitrogen. The analysis employs the use of an existing numerical code and computed solutions to investigate performance characteristics such as mixing. The numerical code, GASP (General Aerodynamic Simulation Package) 2.2 developed by Walters et al. [1993] is used. Comparisons with computational results are made with the measured pressure signatures on the upper and lower (injection) walls of the combustor.

Mixing of fuel and air is primarily governed by the turbulent interactions in the flow field. Thus the approach taken in modeling the turbulent interactions affects the computed results. It is therefore proposed that two different turbulence models be used to assess the effect of the choice of the turbulence model on the computations. Initially, Baldwin-Lomax (algebraic) model will be used. Comparisons will then be made with results from the two-equation k-epsilon model. The interaction between the injectant and the main stream results in rather elevated levels of shear stress near the injector. An algebraic model's predictions are usually considered to be unreliable in such situations. A two-equation model, on the other hand, is the simplest complete turbulence model and is generally expected to yield reliable results albeit at greater cost.

The primary objective of using an injector is to deliver fuel so that a desired level of mixing of fuel and air can be achieved over the length of the combustor. At high mach numbers, rates of mixing are inherently lower. Thus, a knowledge of the mixing rates is of extreme importance for vehicles traveling in the hypersonic range. The performance of the injector will be evaluated by computing the mixing efficiency. This parameter could not be measured in the experiment. Therefore, the computed value is the only available source

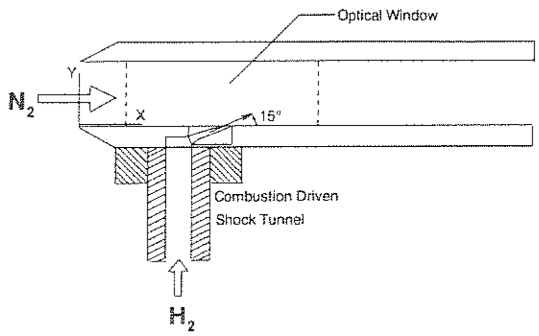

Fig.1 A schematic of the combustor model

2. Governing Equations and Boundary Conditions

The governing equations include three-dimensional, viscous, Parabolized Navier Stokes (PNS) equations and the corresponding conservation equations of energy and species and are given in Anderson et al. [1984] and in Woods [1997]. Since the injectant is injected at a shallow angle (15 degrees) to the main stream, there is no recirculation near the injector. This is clearly evident in the resonantly enhanced interferograms obtained by Belanger [1993]. Thus, the parabolized form of the Navier-Stokes equations which neglect all streamwise gradients except for the one involving pressure, would be appropriate to model this flow. The advantage with this system of equations is that streamwise marching is permitted. This study considered only those tests where hydrogen was injected into a nitrogen main stream. Thus no chemical reactions had to be accounted for.

The inlet flow is supersonic, so the velocities, static temperature, static pressure, and species concentration are all fixed quantities derived from corresponding entries in Woods [1997]. The no-slip condition is applied to all the walls of the combustor. Because of the inherent symmetry of the actual combustor, only one half is simulated. The vertical plane, located at z = 0.0 m that passes through the lower wall of the combustor was taken to be plane of symmetry. The solid walls are held constant at 300K, to reflect the small test times (1-2 ms) encountered in T5. On the injection wall (lower wall), over the injection region, values of density, temperature, etc. were specified based on the information in Woods [1997].

3. Grids

Computational simulation is performed with a two-dimensional grid as well as with a three-dimensional grid. There are 230 grid points in the x-direction (principal flow direction), 72 in the y-direction (direction along the normal to the plane of the lower wall) and 44 grid points in the z-direction. In the x-direction, the grid is clustered at the inlet to capture leading edge shocks. In the injection region the grid is uniformly spaced to capture the large flow variations. The grid is also clustered near all the walls to capture the associated boundary layers. An exponential formula was used to cluster the grids.

4. Numerical Code

The General Aerodynamic Simulation Package (GASP) is used to analyze the test data from T5. The code GASP solves the full Reynolds-averaged, compressible form of the Navier-Stokes, energy and species conservation equations. GASP can be run in explicit, implicit, space marching or elliptic modes. The governing equations are discretized using a finite volume approach and can be solved for one, two, and three dimensional models. GASP also solves subsets of the Navier-Stokes equations including the Parabolized Navier-Stokes equations.The code GASP has numerous physical modeling capabilities and contains various algorithms for handling both inviscid and viscous fluxes.

For the inviscid fluxes, a full flux algorithm with the Vigneron technique was used in the x-direction (marching direction). In the y- and z-directions, Roe's flux difference splitting method with Harten correction was used from the inlet to the beginning of the injector. The more robust, Van Leer's vector splitting method was employed from this point forward till the end of the combustor. The viscous portions of the flow field in both y-direction and z-direction were discretized using central differences. No viscous terms were included in the marching direction. Limiting algorithms were used to reduce oscillations in the solution. Van Albeda's limiter was used in the marching direction, and Spekreijse-Venkat limiter was used in both y and z-directions. A two species, hydrogen and nitrogen, chemistry model was added to GASP for this computation. Both hydrogen and nitrogen were taken to be ideal gases.

As stated earlier, a principal objective of this study was to consider two different turbulence models - the algebraic Baldwin-Lomax model and the two-equation k-epsilon model. The two-equation model used in this study is the low-Reynolds number k-epsilon model of Chien [1982]which reduces to the "standard" k-epsilon model of Jones and Launder [1972] away from the wall.

5. Results and Discussion

Based on the numerical procedure outlined previously, the governing equations were solved on a CRAY T-90 platform, using the code GASP. Selected results are presented and discussed here.

All computations were performed with the two-species, nitrogen and hydrogen, chemistry model. Both species were taken to be ideal gases. In order to conserve computational resources, only mixing runs were considered. These flows involved injection of hydrogen into a nitrogen main stream and thus no chemical reactions had to be accounted for. Previous studies (Belanger [1993]) indicate that the added heat release from combustion of hydrogen with air does not affect the dynamics of the flow significantly. Solutions were obtained with both the Baldwin-Lomax and k-epsilon turbulence models.

Comparisons of computed pressure signatures on the upper (non-injection) and lower (injection) walls of the combustor model are made with the Baldwin-Lomax and k-epsilon models are shown in figure 2. The pressure signature obtained with the Baldwin-Lomax model shows good agreement with experimental results. Likewise, the results obtained with the k-epsilon solution also show good agreement with the experimental results. The simulations differ slightly in the predicted value of the first peak in the wall pressure for the upper wall.The first peak in the wall pressure on the opposite wall results from the incidence of the primary bow shock. Other peaks in the pressure distribution on both the walls arise from subsequent reflections of the primary bow shock.

An important goal of this study is to analyze the extent of mixing between the nitrogen and hydrogen streams. Mixing is affected to a large extent by the turbulent interactions. Comparisons of eddy viscosity values, from the two turbulence models, provide a reasonable quantitative measure of turbulent interactions in the flow. Eddy viscosity, ee, is a function of all three spatial coordinates - x, y, and z. To show the variation of eddy viscosity, it is plotted as a function of y for different x and z locations, in figures (3a-3c). For each x, two locations of z are shown. These locations correspond to the centerline (z = 0.0 m) and near the far wall (z = 0.023637 m) respectively. The x locations correspond to upstream of the injection region (x = 0.105651 m), the plane preceding the injection region (x = 0.165874 m), the plane midway within the injection region (x = 0.177358 m), first plane downstream of the injection region (x = 0.188843 m), further downstream of the injection region (x = 0.420846 m), and near the exit (x = 0.6548 m). The region over which the fuel is injected in the computational grid, extends from x = 0.166232 m to x = 0.187766 m, and from z = 0.0 m to z = 0.002012 m, with y = 0.0 m.

Upstream of the injection region, eddy viscosity distribution is symmetric with respect to the y-axis (figure 3a). Also to be noted is the relatively elevated levels of eddy viscosity near the wall (z= 0.023637m) when compared with its value near the centerline (z=0). This can be attributed to the corner flow where two boundary layers meet, resulting in enhanced turbulent interactions. At locations downstream of injector, figures (3b-3c), as expected, no symmetry about the y-axis is observed. Near the injector a large increase in eddy viscosity is seen. The loss of symmetry and the higher eddy viscosity values are attributed to the rapid mixing of the injected fuel and the mainstream nitrogen. At the first plane downstream of the injection region (figure 3b), the Baldwin-Lomax model predicts rapidly varying eddy viscosity distribution. The predictions of the k-epsilon model are better behaved. Further downstream (figure 3c), at the centerline and near the far wall, the Baldwin-Lomax model predicts higher values of eddy viscosity. Near the exit, (figure 3c), the solutions obtained by the two turbulence models are similar and a good amount of mixing seems to occur near the far wall.

Next, we will examine a parameter that quantifies mixing, i.e. mixing efficiency. The mixing efficiency is expressed as

$$\eta_m = \frac{\int_{A_c} \rho u Y_R \, dA_c}{\dot{m}_R}$$

Mixing efficiency is a number between zero and one and is defined as the fraction of the least available reactant that can undergo complete reaction without further mixing. In the definition given above, mixing efficiency is expressed as the ratio of two mass flow rates. Y_R is the mass fraction of the least available reactant. In flow regions that are locally fuel lean, $Y_R = Y_{fuel}$ and in regions that are fuel rich, $Y_R = f(1 - Y_{fuel})$, where f is the stoichiometric fuel-air mass ratio. The product of ρ u Y_R gives the local mass flux rate of the least available reactant. Thus the numerator is the actual local mass flow rate of the least available reactant expressed in terms of equivalent fuel flow rate. The denominator, is the maximum possible mass flow rate of the least available reactant expressed in terms of equivalent mass flow rate of the fuel. Thus, if the overall fuel-air ratio, ϕ, is greater than one, $\dot{m}_R = f\dot{m}_{oxygen}$. If ϕ is less than one, then $\dot{m}_R = \dot{m}_{fuel}$. The mixing efficiency results are shown in fig. 4. Both the turbulence models yield similar results. The Baldwin-Lomax model shows generally higher values. However, at the exit of the combustor, both the turbulence models give identical results. These computations of mixing efficiency are consistent with cross-stream distributions of fuel-mass fractions as well as the behavior of eddy viscosity (figs. 3a - 3c)

The results from the eddy viscosity predictions, discussed earlier, are supported by the computed distribution of the mixing efficiency shown in figure 4. Within the injection region (at the x = 0.177358 m) the mixing efficiency is relatively small and a slight difference is observed between the predictions from the two turbulence models. Similar results were found at this location for eddy viscosity (figure 3b). Further downstream (x = 0.420846 m location) the predicted mixing efficiency obtained from the Baldwin-Lomax solution is greater than that from the k-epsilon model. This is in agreement with results from the eddy viscosity predictions, (figure

3c). Near the exit (x = 0.654800 m) both the eddy viscosity predictions (figure 3c), and mixing efficiency predictions (figure 4), as computed with these two different turbulence models are almost identical.

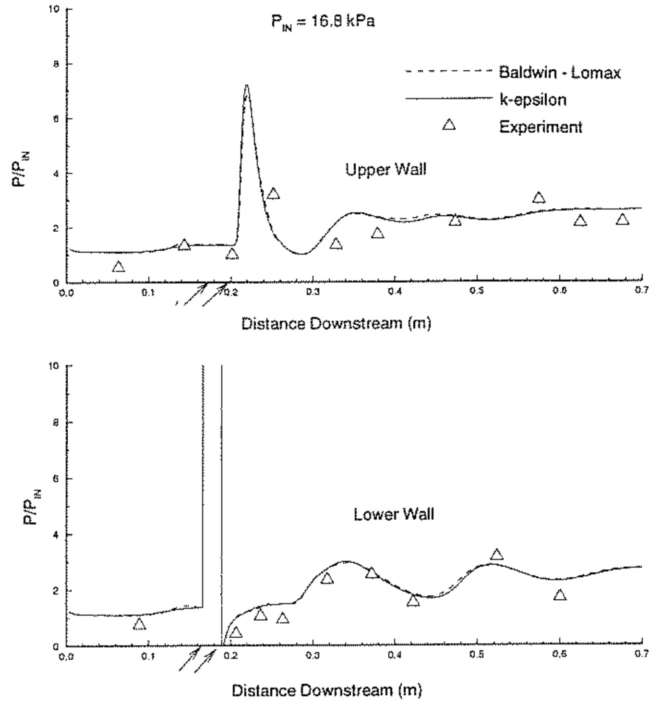

Fig. 2 A comparison of computed and measured pressure distribution

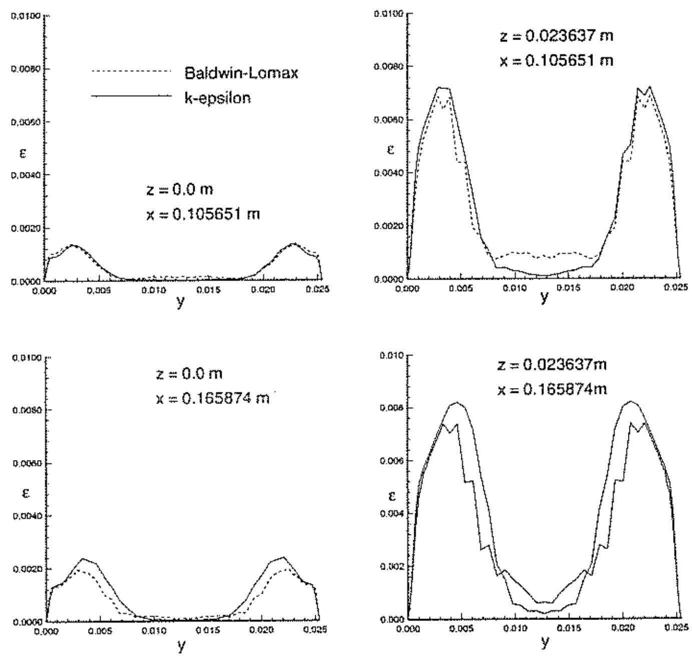

Fig. 3a Comparison of computed eddy viscosity distribution upstream of injector

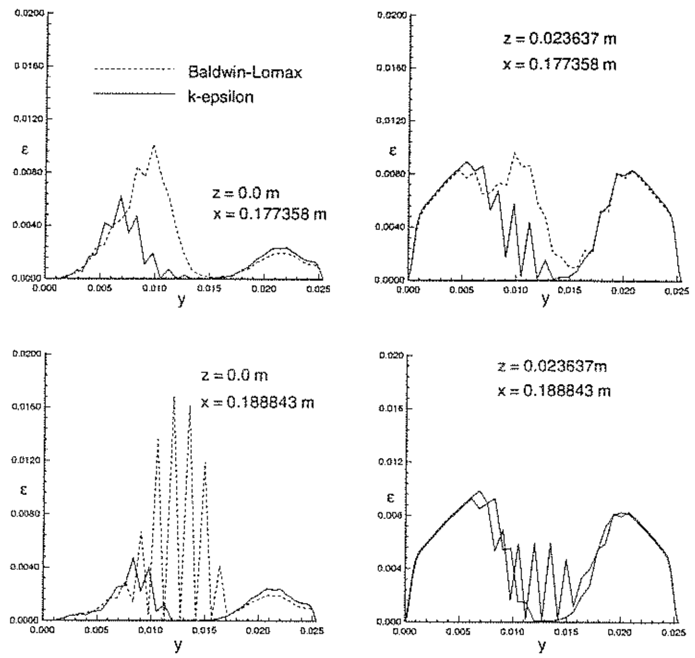

Fig. 3b Comparison of computed eddy viscosity distribution within the injection region

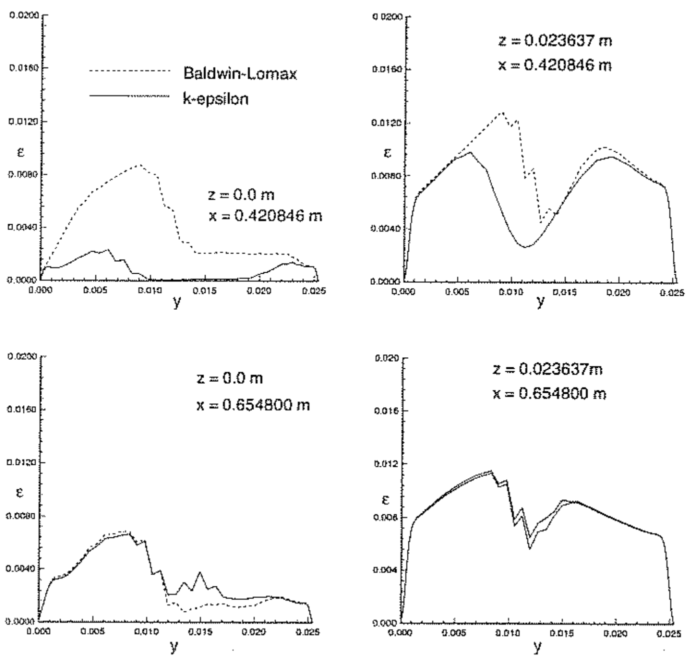

Fig. 3c Comparison of computed eddy viscosity distribution downstream of the injection region

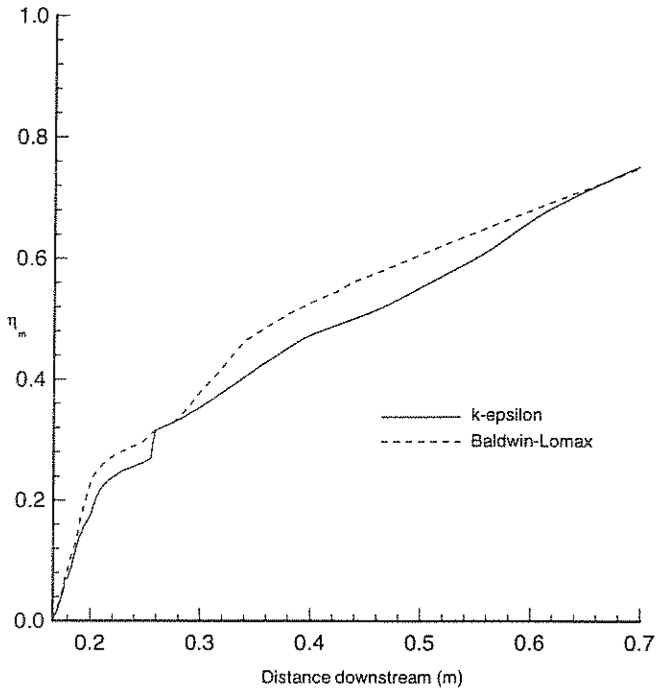

Fig. 4 A comparison of mixing efficiency

6. Conclusions

A computational comparison has been made with results obtained from tests with a generic combustor model at T5. Both the turbulence models predicted a wall pressure distribution that was in agreement with measured data. The Baldwin-Lomax model in general predicted larger values of eddy viscosity. The results from the eddy viscosity predictions were supported by the evaluation of the mixing efficiency. Similar to eddy viscosity predictions, the mixing efficiency predictions between the k-epsilon and Baldwin-Lomax turbulence models varied slightly within the injection region. Further downstream the Baldwin-Lomax turbulence model predicted higher values. However, both models predicted an identical value for mixing efficiency at the exit of the combustor. For the conditions considered in the present study, the use of the simpler albebraic Baldwin-Lomax model is considered sufficient.

7. References

Anderson, D.A., Tannehill, J.C. and Pletcher, R.H., *Computational fluid mechanics, and heat transfer*, Hemisphere Publishing Corporation, 1984.

Belanger, J., Studies of mixing and combustion in hypervelocity flows with hot injections, Ph.D. Thesis, California Institute of Technology, Pasadena, CA., April, 1993.

Chien, K., Predictions of channel and boundary-layer flows with a low-reynolds-number turbulence model, *AIAA Journal*, 20(1), 33-38, 1982.

Hornung, H. G., Performance data of the new free-piston shock tunnel at GALCIT", AIAA 92-3943, Nash ville, TN, July 6 - 8, 1992.

Jones, W. P. and Launder, B. E., The prediction of laminarization with a 2-equation model of turbulence, *International Journal of Heat and Mass Transfer*, 15(2),301-314, 1972.

Walters, R., McGrory, W., Slack, D., and Applebaum, M., *Gasp version 2.2 user's manual*, Aerosoft Inc., 1993.

Woods, D.M., Computation of mixing in hypersonic flows, M.S. Thesis, North Carolina A&T State University, Greensboro, 1997.