JOURNAL OF PROPULSION AND POWER
Vol. 13, No. 3, May–June 1997

Technical Notes

TECHNICAL NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Overexpanded Two-Dimensional–Convergent-Divergent Nozzle Flow Simulations, Assessment of Turbulence Models

A. Hamed* and C. Vogiatzis†
University of Cincinnati, Cincinnati, Ohio 45221

Introduction

THERE is renewed interest in two-dimensional–convergent-divergent (2D–CD) nozzles because of the advantages they offer over axisymmetric configurations for supersonic transport. These include higher performance, reduced afterbody drag, easier integration with airframes, and large mechanical area excursion capabilities. The performance of these nozzles can suffer at off-design conditions when the large nozzle area ratio reductions required during subsonic and transonic acceleration cannot be achieved under the mechanical and control system constraints. Because this can adversely affect the acceleration time to cruise, the fuel burnt, and the range, it is desirable to predict off-design performance with a high degree of accuracy.

The purpose of the present investigation is to assess the computational results obtained for overexpanded 2D–CD nozzles using five different turbulence models. Overexpanded nozzle flow predictions are challenging because of the large shock-induced separated flow regions in the divergent nozzle section. The 2D–CD nozzle tested under static conditions by Hunter[1] was selected for the numerical assessment because the large number of pressure taps gave a better definition of the shock location. The nozzle has a design pressure ratio of 8.8 for an exit Mach number of 2.1.

Approach

The implicit numerical solution of the compressible two-dimensional Navier–Stokes equations was obtained using the NPARC code.[2] The computations were performed using five different turbulence models, namely, Baldwin–Lomax and RNG algebraic models, Baldwin–Barth one-equation model, and the two-equation Chien k-ε and Wilcox k-ω turbulence models. A one-block 161×68 grid was selected after a grid refinement study. The Y^+ for the first grid point next to the wall was equal to 1.0 in the throat region, with at least 15 points inside the wall boundary layer. The computed Mach number contours at nozzle pressure ratio (NPR) of 2.41 are presented in Fig. 1 to illustrate the flow structure at overexpanded conditions.

Presented as Paper 95-2615 at the AIAA/ASME/SAE/ASEE 31st Joint Propulsion Conference, San Diego, CA, July 10–12, 1995; received Nov. 3, 1995; revision received Feb. 18, 1997; accepted for publication Feb. 24, 1997. Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Professor, Department of Aerospace Engineering. Fellow AIAA.
†Graduate Student, Department of Aerospace Engineering.

In all cases the numerical solution was advanced using local time stepping with a maximum attainable Courant–Friedrichs–Lewy number of 2.0. The internal thrust coefficient was determined from the integration of the axial momentum at the exit plane. A variation of less than 0.1% in thrust or mass flow over 1000 iterations was required to consider the solution converged. The convergence characteristics of the numerical solution obtained using the different turbulence models are shown in Fig. 2. One can see that the convergence characteristics at overexpanded conditions are strongly dependent on the turbulence model.

Results and Discussions

The computational results for the surface pressure distribution at NPR = 2.41 are presented in Fig. 3 together with Hunter's experimental data along the flap centerline.[1] The results exhibit a significant spread in the predicted shock location, depending on the turbulence model used. The algebraic and one-equation models predicted shock positions, respectively, upstream and downstream of the experimental location. Furthermore, their predicted postshock pressures were higher and contained overshoots not observed experimentally. The shock location was best predicted by the two-equation turbulence models; however, Wilcox's k-ω model was the closest to the

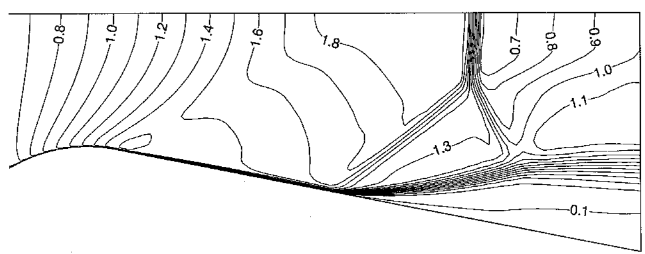

Fig. 1 Mach number contours, downstream of the throat, using the k-ω turbulence model (NPR = 2.41).

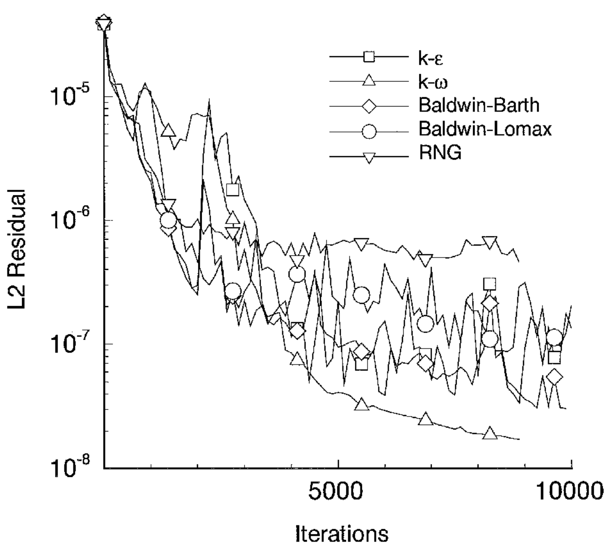

Fig. 2 Convergence history (NPR = 2.41).

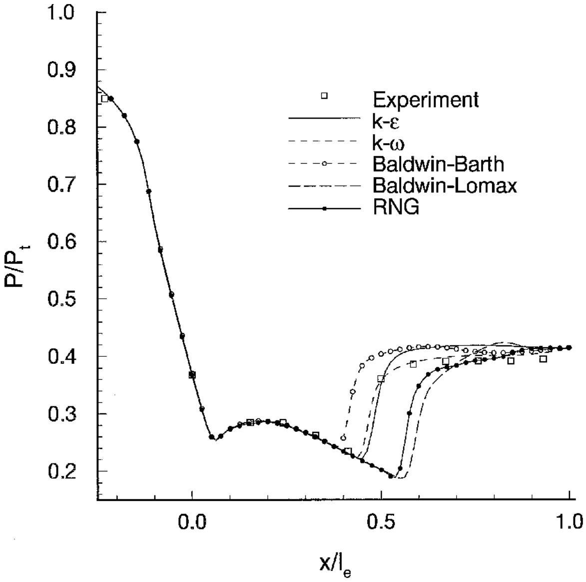

Fig. 3 Surface pressure distributions (NPR = 2.41).

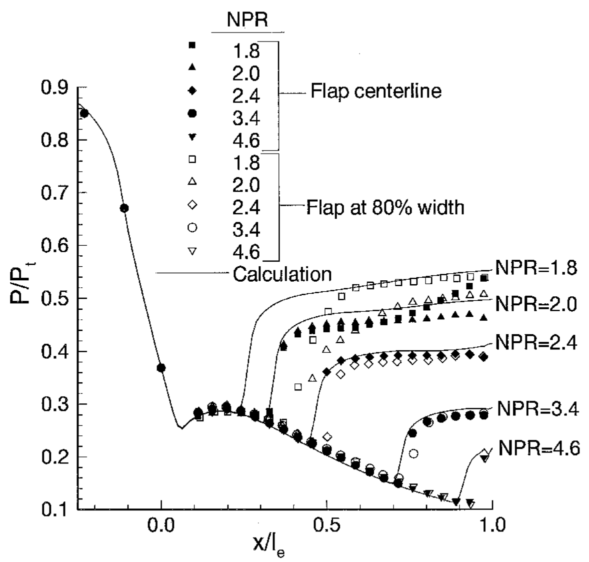

Fig. 4 Surface pressure distribution using Wilcox's k-ω turbulence model.

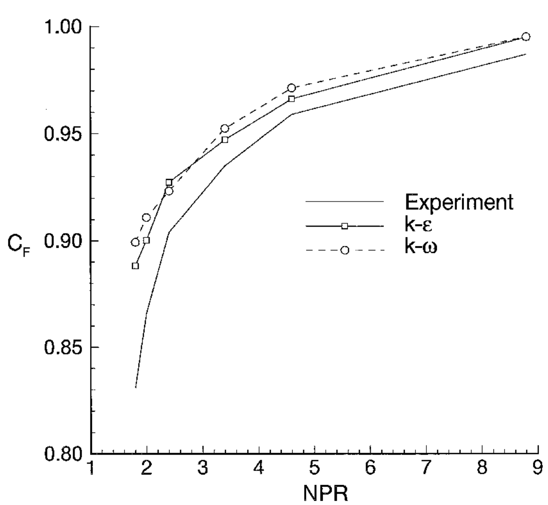

Fig. 5 Thrust coefficient predictions.

Table 1 Thrust coefficient at different pressure ratios

	NPR				
	2.0	2.4	3.4	4.6	8.8
Experiment	0.866	0.904	0.935	0.959	0.987
k-ε					
Thrust coefficient	0.900	0.927	0.947	0.966	0.995
% error in thrust	3.9%	2.5%	1.3%	0.7%	0.8%
k-ω					
Thrust coefficient	0.911	0.923	0.952	0.971	0.995
% error in thrust	5.2%	2.1%	1.8%	1.2%	0.8%

experimental results in predicting the pressure variation behind the shock. In general, the two-equation turbulence models gave the best overall agreement with the experimental results for the pressure distribution and thrust coefficient at overexpanded conditions. Figure 4 compares the computed pressure distribution over the flap using the *k*-ω model to the experimental results at the centerline and near the end wall at five overexpanded NPRs.

The computed thrust coefficients using the two-equation turbulence models are compared to the experimental results in Fig. 5 and Table 1. One can see that the thrust coefficient was predicted within 1.0 and 2% of the experimental values for NPRs above 50 and 30% design, respectively. At lower nozzle pressure ratios, two-dimensional flow predictions are inadequate because of the strong three-dimensional flow effects behind the shock, which are observed in the experimental results.

Acknowledgment

This work was sponsored by Ohio Aerospace Institute Subgrant 120520. The computational work was performed on the Cray Y-MP8 of the Ohio Supercomputer Center.

References

[1]Hunter, C. A., "An Experimental Analysis of Passive Shock-Boundary Layer Interaction Control for Improving the Off-Design Performance of Jet Exhaust Nozzles," M.S. Thesis, George Washington Univ., Washington, DC, Sept. 1993.

[2]Cooper, G. K., and Sirbaugh, J. R., "PARC Code: Theory and Usage," Arnold Engineering Development Center, TR-89-15, Arnold AFB, TN, Dec. 1989.

Lobed Mixers Using Simultaneous Laser-Induced Fluorescence and Mie Scattering

David M. Milam,* Mo Samimy,† and Steven Martens‡

Ohio State University, Columbus, Ohio 43210

Introduction

EFFICIENT mixing between two streams of fluids is important in many applications. The existence of large-scale

Received April 11, 1996; revision received Dec. 6, 1996; accepted for publication Jan. 2, 1997.

Copyright © 1997 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

*Graduate Student, Department of Mechanical Engineering; currently with Caterpillar Technology Center, Mossville, IL 61552.

†Professor, Department of Mechanical Engineering. Associate Fellow AIAA.

‡Postdoctoral Fellow, Department of Mechanical Engineering; currently with General Electric Aircraft Engines, Cincinnati, OH 45215. Member AIAA.