https://ntrs.nasa.gov/search.jsp?R=19950002779 2020-06-17T23:45:54+00:00Z

N95- 70899

VALIDATION OF TWO-EQUATION TURBULENCE MODELS FOR PROPULSION FLOWFIELDS

Manish Deshpande[1], S. Venkateswaran[2], Charles L. Merkle[3]
Propulsion Engineering Research Center
Department of Mechanical Engineering
The Pennsylvania State University

Summary

The objective of the study is to assess the capability of two-equation turbulence models for simulating propulsion-related flowfields. The standard $k-\epsilon$ model with Chien's low Reynolds number formulation for near-wall effects is used as the baseline turbulence model. Several experimental test cases, representative of rocket combustor internal flowfields, are used to catalog the performance of the baseline model. Specific flowfields considered here include recirculating flow behind a backstep [1,2], mixing between coaxial jets [3] and planar shear layers [4,5]. Since turbulence solutions are notoriously dependent on grid and numerical methodology, the effects of grid refinement and artificial dissipation on numerical accuracy are studied. In the latter instance, computational results obtained with several central-differenced and upwind-based formulations are compared. Based on these results, improved turbulence models such as enhanced $k-\epsilon$ models as well as other two-equation formulations (e.g., $k-\omega$) are being studied. In addition, validation of swirling and reacting flowfields are also currently underway.

Technical Discussion

The k and ϵ transport equations are solved coupled to the the preconditioned Navier-Stokes equations. Preconditioning ensures that the system remains well-conditioned at all flow Mach numbers and Reynolds numbers, thereby providing uniform convergence under a wide range of conditions (Ref. [6,7] for details). Furthermore, convergence difficulties associated with strongly stretched grids, which are characteristic of turbulent flows, are mitigated by using the ADI algorithm with proper local time-stepping and preconditioning [6,7]. Several spatial discretization schemes such as second-order central-differencing and first- and third-order upwind differencing are used. When central-differencing is used, the presence of odd-even splitting and/or oscillatory solution behavior in the vicinity of steep gradients, sometimes necessitate the judicious addition of second-order dissipation through the use of switches or flux-limiters [8]. Oscillatory solutions, in particular, are frequently observed in turbulence computations and usually seem to be related to large-scale unsteady flow processes such as

[1]Graduate Research Assistant
[2]Research Associate
[3]Professor

vortex-shedding. Upwind schemes which possess inherent dissipative properties usually suppress such unsteady behavior and may therefore be regarded as more robust schemes.

Results and Discussion

The first test case discussed is the Driver-Seegmiller backward-facing step flow [1] shown in Fig. 1. The velocity contours (Fig. 2) indicate a recirculation length of $x/H = 5.7$, which is in agreement with other published computational results using the $k - \epsilon$ model (expt. value is 6.3). The results in Fig. 2 were obtained with third-order upwind-biased discretization of the convective terms. Interestingly, central differencing does not yield a converged solution with the velocity contours suggesting that the solution is unsteady. Addition of second-order dissipation [7] renders the solution steady. This unsteadiness was however not encountered with any of the upwind schemes studied. Figure 3 shows the velocity and μ_t profiles plotted against experimental data at several axial stations for the different discretization schemes. The central-differenced and third-order upwind schemes agree fairly well with each other, while the first-order scheme shows significant discrepancies in μ_t and under-predicts the recirculation length. Additionally, the results are well-predicted when third-order is used for the flow equations and first-order upwind is used for the turbulence equations. This result suggests that using first-order accuracy in the turbulence equations does not undermine the overall accuracy of the calculation.

A grid refinement study was performed for the backstep flow of Kim, Kline and Johnston [2]. Grid sizes of 91^2, 181^2 and 361^2 were used. The coarsest grid had 9 points within $y^+ = 10$. Fig. 4 compares the velocity and k profiles for the three levels of refinement for the third-order upwind scheme. As the figure shows, the coarsest grid differs significantly from the finer grids, especially in the region just downstream of the reattachment point. The two finer grids agree quite well with each other, implying grid-independence. Second-order central differencing, once again, indicated an unsteady solution even for the coarsest grid case unless additional dissipation was included.

The next case considered is the experiment of Johnson and Bennett [3], involving two co-flowing jets in a confined sudden expansion. Figure 5 shows the converged velocity contours for this case while Fig. 6 shows comparisons of velocity profiles obtained from central-differenced and upwind calculations plotted against experimental data. Overall agreement is quite good except at the centerline in the near-injector region.

The final test case presented here is the turbulent planar shear layer experiments of Chang *et al* [5]. In Fig. 7, comparisons of the velocity and k profiles are presented for the non-reacting case using third-order upwind for the convective terms. The computations were performed using inviscid wall boundary conditions at the upper and lower edges of the domain. The computations agree fairly well with the experiments in the upstream stations. However, in the downstream stations, there is growing discrepancy in the predicted k profiles. A similar discrepancy was observed in the calculations reported in Ref. 5 and may be related to the choice of wall boundary condition.

The above results demonstrate that accurate turbulent flow solutions may be obtained with the proper choice of spatial discretization schemes. Several of the flowfields were observed to have convergence difficulties related to large-scale unsteadiness in the flowfield. Upwind-based schemes appear to be more robust in this regard, probably because of the dissipation that is inherent in these schemes. The results presented here will provide a baseline reference point for the selection of improved turbulence models for solving combustor-related flowfields. Additional calculations are underway for the computation of swirling and reacting flowfields.

Acknowledgements

This work was supported by Pratt & Whitney, West Palm Beach, Florida. The authors also wish to acknowledge Kevin Van Dyke and Andy Brankovic of Pratt and Whitney for their help in setting up some of the cases.

References

1. Driver, D. and Seegmiller, H., AIAA Journal, Vol. 23, No. 2, pp. 163-171.
2. Kim, J., Kline, S.J. and Johnston, J.P., J. Fluids Eng., Vol. 102, pp. 302-308, 1980.
3. Johnson, B. and Bennett, J., Proc. ASME Fluids Eng. Conf., Boulder, CO.
4. Chang, C.T., *et al*, AIAA 93-2381.
5. Lai, H.T., and Raju, M.M., AIAA 93-1773, 29th Joint Propulsion Conf., Monterey, 1993.
6. Buelow, P., Venkateswaran, S. and Merkle, C., AIAA 93-3367, Orlando, FL.
7. Venkateswaran, S., Proc. 14th Intl. Conf. on Num. Meth. in Fluid Dyn., Bangalore, India, 1994.
8. Jameson, A., AIAA 93-3359, Orlando, FL.

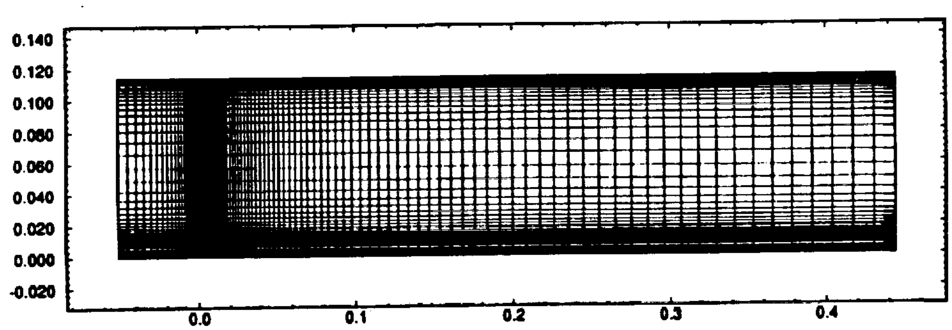

Figure 1: Computational Grid for Driver-Seegmiller Backstep Flow

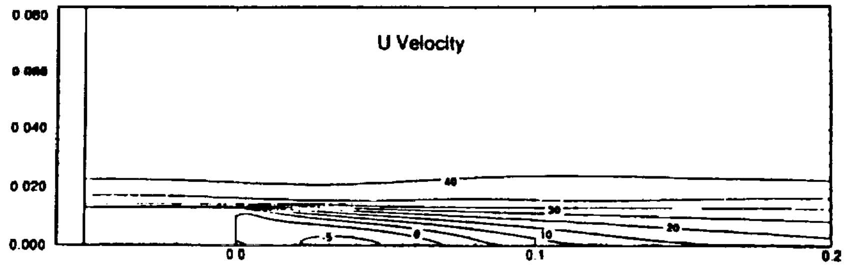

Figure 2: Velocity Contours for Driver-Seegmiller Backstep Flow

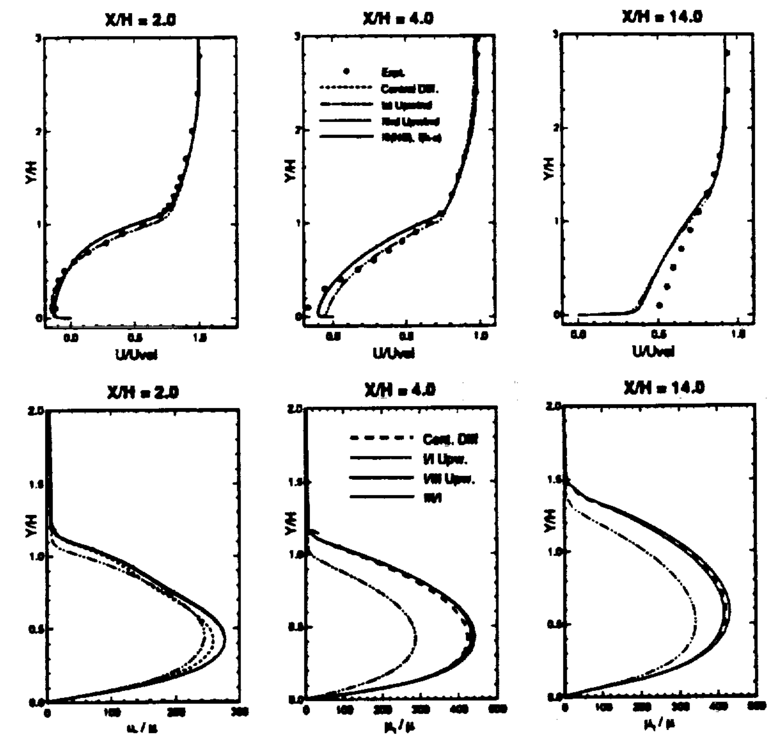

Figure 3: Velocity and μ_t profiles - Comparison between Schemes

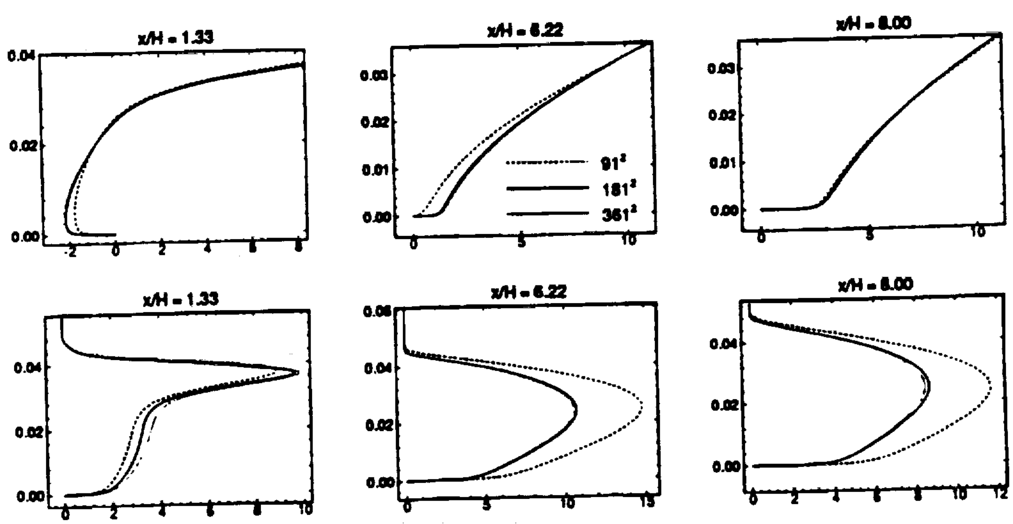

Figure 4: Velocity and k profiles for Kim *et al* backstep - Grid Refinement Study

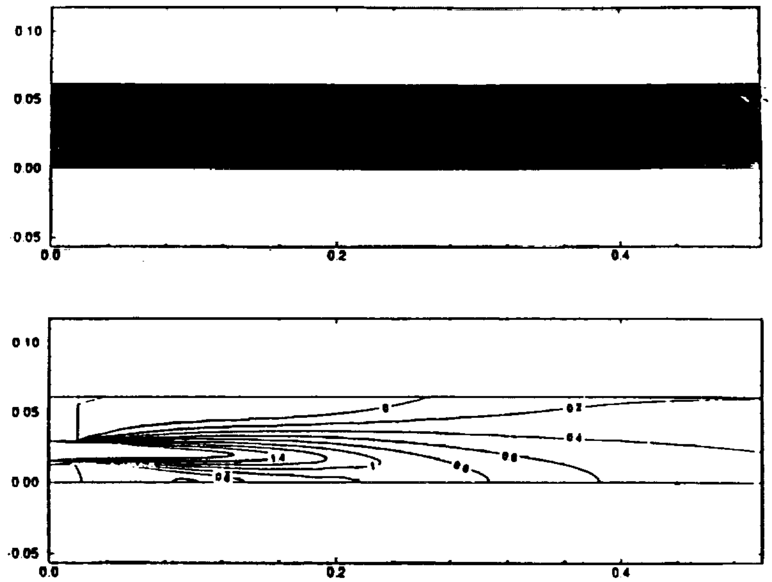

Figure 5: Computational Grid and Velocity Contours for co-axial jets.

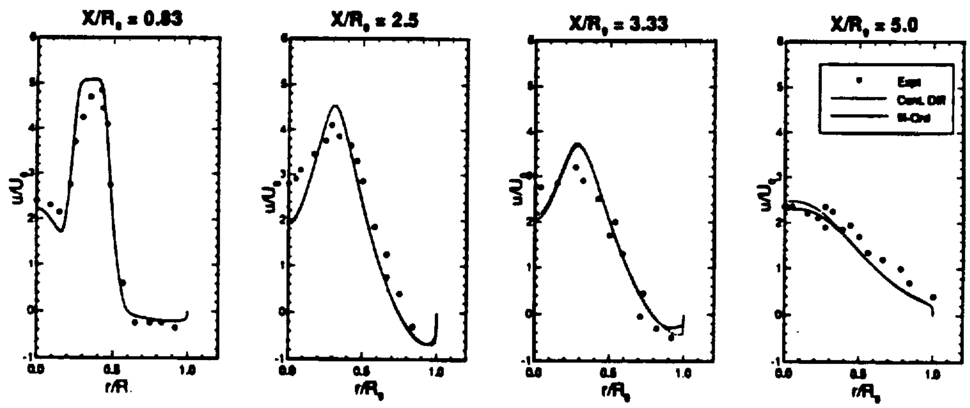

Figure 6: Velocity profiles for co-axial jets.

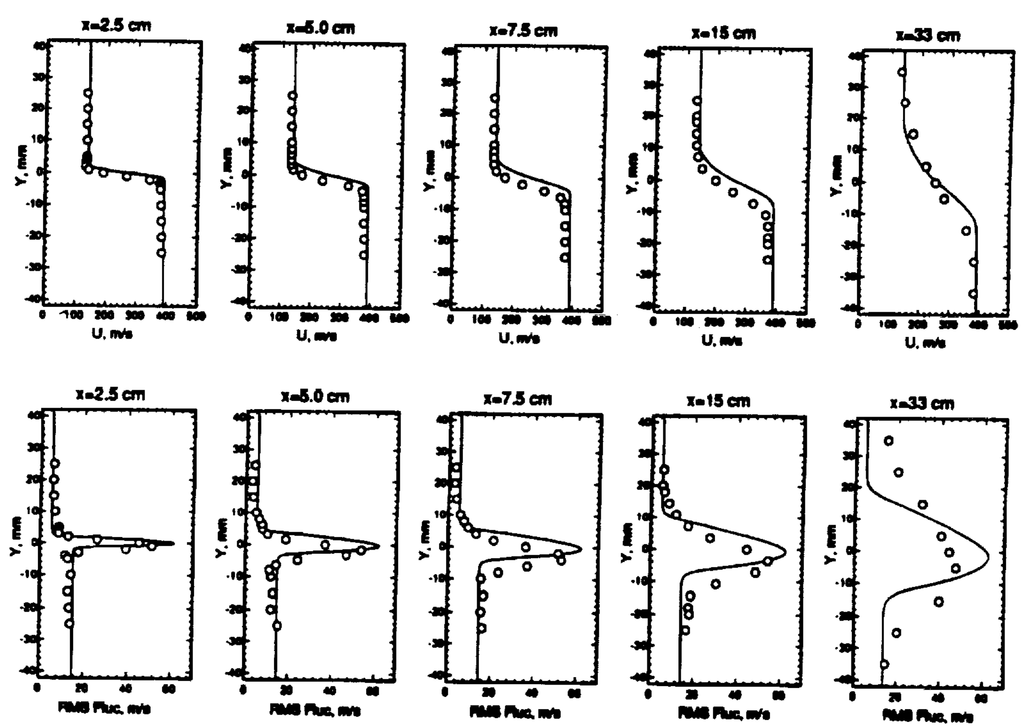

Figure 7: Velocity and k profiles for turbulent planar shear layer