Turbulence Modeling in a Hypersonic Inlet

W. F. Ng,* K. Ajmani,† and A. C. Taylor III‡
Virginia Polytechnic Institute and State University, Blacksburg, Virginia

A study is conducted to analyze the performance of different turbulence models when applied to flow through a Mach 7.4 hypersonic inlet. The analysis, which is two-dimensional, is done by comparing computational results from a Parabolized Navier-Stokes code and a full Navier-Stokes code with experimantal data. The McDonald-Camarate (MC) and Baldwin-Lomax (BL) models were the two zero-equation models used in the study. The Turbulent Kinetic Energy (TKE) model was chosen as a representative higher-order model. The MC model, when run with user-specified transition of the boundary layer, provides a solution that compares excellently with the data. The BL model predicts separation of flow in the inlet, which contradicts experimental findings. The TKE model does not perform any better than the MC and BL models, despite the fact that it is a higher-order turbulence model.

Nomenclature

f_f = Bushnell-Beckwith correction factor, Eq. (2)
H = velocity profile shape factor
P = static pressure
P_p = pitot pressure
P_{ref} = freestream static pressure
$P_{s\text{ref}}$ = freestream stagnation pressure
R = inlet cowl height
X = axial length along the inlet
Y = normal distance from centerbody
δ = boundary-layer thickness
θ = boundary-layer momentum thickness
μ_t = turbulent viscosity
μ_∞ = freestream viscosity

I. Introduction

THE recent upsurge in interest in hypersonics technology has led to much time and effort being dedicated to the development of accurate and efficient procedures to analyze and predict flows in a hypersonic inlet. By the application of advanced Computational Fluid Dynamics (CFD) techniques, very powerful and efficient computational codes have been developed to solve the governing equations for high-speed compressible viscous flows. A recent review paper summarized the current status of CFD techniques for scramjet engine analysis.[1]

However, the ability of any CFD technique to have a positive and realistic impact on aerospace vehicle development and design depends, among other factors, on the type of turbulence modeling being implemented in the computer code. A review of some recent turbulence modeling efforts for computational aerodynamics can be found in Ref. 2.

The principal aim of this paper is to show that proper turbulence modeling is very important. The paper examines the application of different eddy viscosity models on the calculation for a two-dimensional hypersonic inlet flowfield. These models are known to eliminate important information about the dynamic character of turbulence in the flow. Furthermore, most of these models have never been applied to a hypersonic inlet flowfield. Thus, it is important to examine and compare the performance of different models.

The issue of transition from laminar to turbulent flow cannot be separated from any study that attempts to deal with turbulence. Therefore, this paper also examines the effect of transition on the calculated flowfield. The switch from laminar to turbulent boundary-layer calculations is either turned on automatically (for models that are capable of doing so) or specified by the user at the locations as determined from experiments.

The paper begins with a brief description of the experimental results being used as the benchmark for determining the most successful turbulence model. This is followed by a brief outline of the turbulence models used in this study. The description of computations for the Parabolized Navier-Stokes (PNS) code and full Navier-Stokes (NS) code is also presented. Results of the study with the hypersonic inlet are then discussed. The relative merits of the models, with and without transition, are also presented.

II. Experimental Background and Support

The experimental investigation was conducted to determine the internal flow characteristics in model passages representative of hypersonic inlets. The geometry of the inlet is shown in Fig. 1. The freestream Mach number at the inlet entrance is 7.4. The model has an internal compression ratio of 8.0 and thus is also referred to as the P-8 inlet.

As shown in Fig. 1, the cowl shock wave interacts with the centerbody boundary layer, and a complex reflected wave system emerges downstream of the location of this interaction. The reflected system interacts with the cowl boundary layer and again enters the inviscid flowfield near the throat station. The most relevant (to this study) characteristic observed about the reflected shock wave system is that *no evidence of boundary-layer separation* was found anywhere in the inlet, including in the regions of high adverse pressure

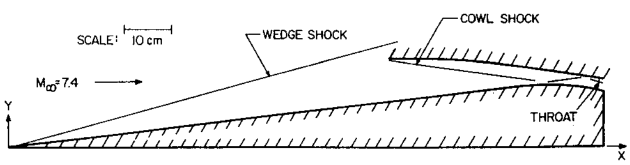

Fig. 1 Hypersonic inlet geometry

Received March 22, 1988; presented as Paper 88-2957 at the AIAA/ASME/SAE/ASEE 24th Joint Propulsion Conference, Boston, MA, July 11-13, 1988; revision received Oct. 28, 1988.

*Associate Professor, Department of Mechanical Engineering. Member AIAA.

†Graduate Research Assistant, Department of Mechanical Engineering.

‡Graduate Research Assistant, Department of Mechanical Engineering. Student Member AIAA.

gradients. The boundary-layer transition point on the centerbody was found experimentally to be at approximately 25% of the distance between the wedge leading edge and the throat station. The cowl boundary-layer transition is located approximately halfway between the cowl leading edge and the throat station. Further details of the experimental program are found in Ref. 3.

III. Turbulence Models

The eddy-viscosity turbulence models used for the present computations include the algebraic models of McDonald-Camarata (with the Bushnell-Beckwith correction) and Baldwin-Lomax. In addition, the one-equation Turbulent Kinetic Energy (TKE) model was also examined. These models have been in wide use for transonic and supersonic flows and have shown some success. Each model is also examined with reference to the way it handles the phenomenon of transition for laminar to turbulent flow.

A. McDonald-Camarata Model

The model uses an extended mixing-length hypothesis to calculate the turbulent shear-stress distribution in the boundary layer. Details of the model can be found in Ref. 4.

The original McDonald-Camarata turbulence model was developed for incompressible boundary layers only. In the present study, the original McDonald-Camarata model had been modified to account for high-speed effects in compressible turbulent boundary layers.[5] (This modification is referred to as the Bushnell-Beckwith high-speed correction.) The compressibility effects are approximated by assuming that the mixing length in the midregion of the boundary layer is a function of a velocity profile shape factor[6] H, given by

$$H = \delta / \theta \tag{1}$$

where δ and θ are the incompressible displacement and momentum thicknesses, respectively.

A correction factor f_f is introduced for the mixing length, and its magnitude is a function of the shape factor of the velocity profile at each computational point in the boundary layer. Expression f_f is given by

$$f_f = (-0.0305\,H + 0.1125)/0.09 \tag{2}$$

The range of the correction factor is 0.44–0.84. It was found that the correction factor significantly improves the computational results. This is to be expected, since as stated earlier, the original McDonald-Camarata model was developed for incompressible boundary layers only.

The McDonald-Camarata model is not capable of prediction transition. In order to simulate transition while using this model, initial computations must be done with fully laminar flow up to a particular axial location in the inlet. This location is selected from the range of transition suggested by the experimental measurements. The calculations are then restarted from this user-specified location while using options in the code that pertain to turbulent boundary-layer calculations. Hence, transition must be user-specified while using this turbulence model. This procedure of simulating transition will be referred to as manual or hand transition for the remaining portion of this paper.

B. Baldwin-Lomax Model

This model was chosen for the study because it is one of the most widely used turbulence models. However, as will be shown later, the model predicted *separation of flow* in the P-8 inlet, whereas it was observed that no separation was detected in experiments with this inlet.

The model is a zero-equation, eddy-viscosity, two-layer model in which the vorticity generated near solid surfaces is used to determine length scales in the turbulent boundary layer. Details of the model can be found in Ref. 7.

The model uses an arbitrary constant value of the turbulent viscosity ($\mu_t = 14\mu_\infty$) to determine the point of transition for the boundary layer. The maximum computed value of turbulent viscosity in a profile is compared with this constant, and if the latter is greater, the turbulent viscosity is set to zero everywhere in that profile (i.e., assume laminar flow). If the former is greater, computed values of the turbulent viscosity are used to determine the effective viscosity.

The Baldwin-Lomax (BL) model is therefore capable of simulating transition by itself, and the user need not specify the point of transition. However, it is found in this study that the BL model predicts transition much too early in the P-8 hypersonic inlet, and this failure is attributed to the empirical constant for transition, which was determined for external transonic flows.

C. Turbulent Kinetic Energy (TKE) Model

The TKE model is a one-equation eddy-viscosity model in which closure of the equations of flow is achieved by using information obtained from the mean turbulent field. A complete description of the Turbulent Kinetic Energy model is available in Ref. 8.

In all computations for this study, the McDonald-Camarata model is used for modeling the length scales for the TKE model. In addition, the boundary-layer thickness is prescribed and wall-shear values are used at all points in a profile. Solid wall boundary conditions are imposed by setting Turbulent Kinetic Energy equal to zero at the wall. For nonwall boundaries, the gradient of Turbulent Kinetic Energy normal to the boundary is set to zero. An initial value of the freestream TKE has to be specified by the user in order to start the boundary-layer calculations. As no experimental data were available on this issue, various guesses for the initial value of TKE were tried to get the closest possible match of the flowfield solution to the experimental pitot-pressure profiles.

The TKE model is capable of switching from laminar to turbulent boundary-layer calculations automatically. Transition is turned on by the code when the level of freestream disturbances exceeds the level of disturbance experienced in laminar flow. Hence, both fully turbulent flows and flows with transition may be executed using the TKE model.

IV. Description of Computations

A. Parabolixzed Navier-Stokes (PNS) Code

The computer code chosen for this study is the NASA-Lewis-Inlet code (PEPSIS). The code was originally developed by Buggeln et al.[9] and is well suited to viscous analysis of high-speed inlets. The solution methodology is documented in Ref. 9.

The computational grid used in the study had 100 Y points in the direction normal to the wall(s). Points were packed more densely near the walls to resolve the gradients in the boundary layers. The solution was marched downstream from the leading edge of the centerbody to the throat station in approximately 650 X steps. To ensure that the solutions are not highly grid-dependent, one set of computations was also done with 200 Y points. Except for more severe postshock oscillations in the 100-Y-point solution, the solutions from the 100- and 200-Y-point runs were essentially the same.

All computations were performed on the IBM 3090 vector processing facility. Approximate CPU time for a complete run was 150 s.

B. Full Navier-Stokes (NS) Code

A two-dimensional Navier-Stokes analysis was applied to the P-8 inlet using a code called NASCRIN. Details and documentation about this code are found in Ref. 10. In this study, *only fully turbulent flow* (without transition) was computed throughout the length of the entire inlet, and turbulence was modeled on both inlet walls (cowl and center body) using

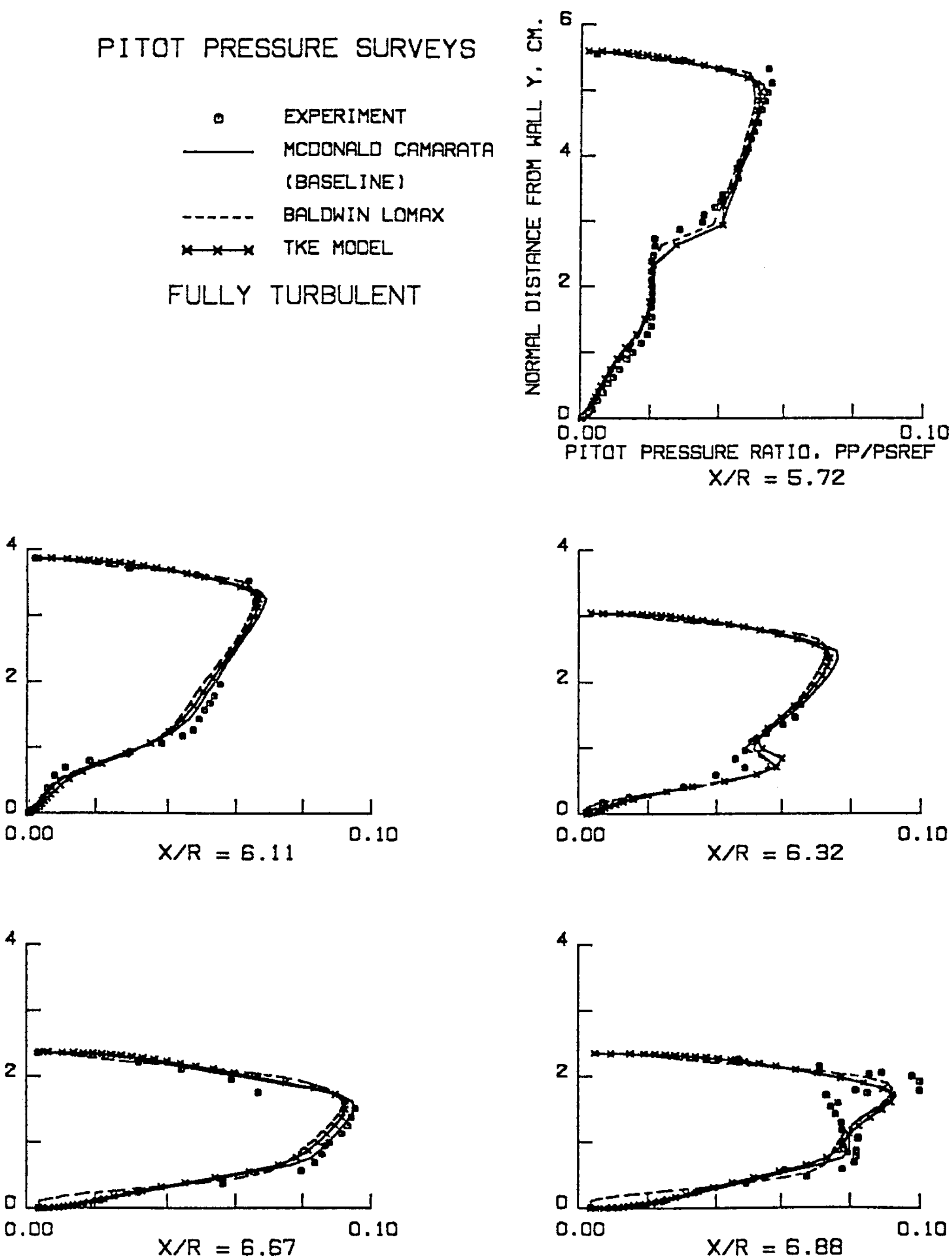

Fig. 2 Pitot-pressure profiles for fully turbulent run using McDonald-Camarata (baseline), Baldwin-Lomax, and TKE models.

the Baldwin-Lomax eddy-viscosity turbulence model. The cost of running the full NS solution prohibits us from studying different turbulence models. Results are obtained and compared with experimental data and also compared with the computed results from the PNS code.

The mesh upon which the computations were done was 101 points in the Y direction and a total of 178 points in the X direction. The calculations were done on an IBM 3090 vector processing computer. The code requires a total of 6.5 h CPU time to achieve global convergence. This represents a two-order-of-magnitude increase compared to the PNS code.

It should be pointed out that all calculations (PNS and NS) were performed with a ratio of specific heats (gamma) of 1.38 instead of 1.40. In Ref. 11 it was shown that the inviscid portion of the pitot pressure profiles in the P-8 inlet were very sensitive to the choice of gamma. However, the pitot pressure profiles in the boundary layer are relatively independent of the choice of gamma. Hence, the conclusions in the paper are not affected by the choice of gamma.

V. Results and Discussion

A. Parabolized Navier-Stokes (PNS) Calculations

The computed results may be broadly divided into two categories. Results for both these categories are presented in the form of comparisons of computational results from each test case with a "baseline" and experimentally measured pitot pressures. The pitot pressure was chosen because it provides detailed flow features inside the inlet. As will be shown later, comparisons based on surface-pressure measurements alone can result in misleading conclusions.

The first category is comparison of results from the different turbulence models while performing calculations for fully turbulent flow throughout the inlet. The second category is comparison of results from the different models while trying to simulate transition from laminar to turbulent flow. The baseline for both categories was chosen as the solution provided by fully turbulent calculations using the McDonald-Camarata model.

1. Fully Turbulent Runs

The baseline used for all comparisons is shown in Fig. 2. It is observed that at $X/R = 5.72$, the solution from the code misses the boundary-layer profile on the ramp. The location of the cowl shock wave is also missed. The solution also misses the boundary-layer profile on the cowl, particularly between $Y = 5.0$ and 5.5 cm. Although not visible in the figure (because of overlapping with the experimental data), post-shock oscillations are present in the solution. At $X/R = 6.32$, the solution again misses the location and the strength of the reflected shock wave from the centerbody. At $X/R = 6.88$, the solution compares rather poorly with experimental data.

The two comparisons using the baseline are the following:

1) Baldwin-Lomax Model (fully turbulent) vs baseline (Fig. 2): This model compares even more poorly with the experimental data than the baseline. It misses the boundary-layer profiles on the centerbody and also the shock locations for the cowl shock wave and the reflected shock from the centerbody. In addition, one can observe a sharp gradient in the pitot-pressure profile on the lower surface at $X/R = 6.32$, 6.67, and 6.88. An examination of the velocity distribution indicated the existence of a separation bubble. This was found to contradict directly the experimental findings, which stated that no regions of flow separation exist in the P-8 inlet. The extent of the separation in the direction normal to the wall is small, and hence the PNS code was able to march through the entire inlet.

2) TKE Model (fully turbulent) vs baseline (Fig. 2): This model, despite being a higher-order model than the baseline, fails to improve on the solution provided by the baseline (Fig. 2). The TKE model solution is almost the same as that of the baseline. However, there is no indication of flow separation, as was seen in the Baldwin-Lomax model.

In summary, it can be concluded that for fully turbulent flow conditions, calculations using the McDonald-Camarata model give the best solution among the three models.

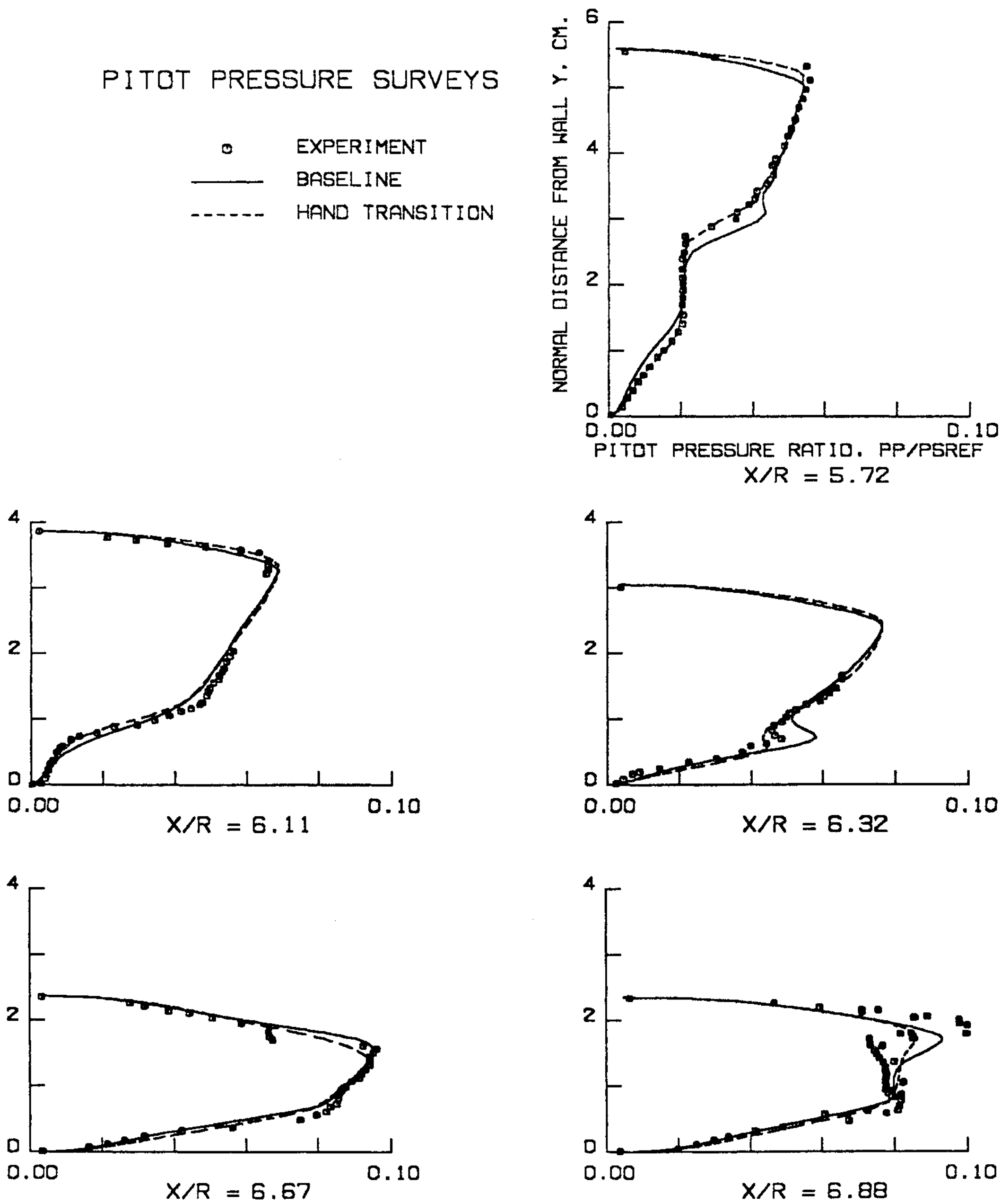

Fig. 3 Pitot-pressure profiles for transitional run using McDonald-Camarata model.

2. Transition Runs

In order to improve the solution from the fully turbulent runs, calculations for flows with transition from laminar to turbulent flow were performed for the McDonald-Camarata model. The results are presented in Fig. 3. The improvement in results is very encouraging. Almost perfect agreement with experimental data is achieved when the McDonald-Camarata model is used with user-specified or hand transition. The boundary-layer profiles on the centerbody and cowl surfaces are matched very well. The solution also picks up the locations of the cowl shock wave ($X/R = 5.72$) and the reflected shock wave from the centerbody ($X/R = 6.32$) quite accurately.

It may be observed that the solution becomes worse as we move down into the inlet, especially at $X/R = 6.88$, the throat station. This fact was examined in detail, but no reasonable explanation could be found. The poor performance may be due to the kind of turbulence modeling employed or something totally different altogether. One may argue that the discrepancies seen deep in the inlet are due to the suppression by the PNS code of influence from downstream stations on upstream stations in the flowfield. However, results from a fully NS simulation (presented later) did not perform any better. The possibility of three-dimensional effects causing these large variations was eliminated by results from a parallel study using three-dimensional simulation.[12]

When run with the transition option, the Baldwin-Lomax model does not perform any better than its corresponding fully turbulent run. In fact, the solutions for the fully turbulent run and with the transition option are almost identical to each other. This is because the model predicts transition very early, and thus, for the major portion of the flowfield, the flow is turbulent. The solution with transition also predicts separation, as is the case for the fully turbulent run.

The TKE model was also run with the transition option in an attempt to improve its solution. However, the results were again disappointing. No real improvement in the match with the experimental boundary-layer profiles or the shock-wave locations was observed. It thus may be safely said that using a higher-order turbulence model does not necessarily imply a better solution for this problem. Again, no hint of flow separation when using the TKE model is found anywhere in the flowfield.

A comparison using the surface static pressure on the centerbody and cowl surfaces was also made in the study. The results are shown in Fig. 4. The surface-pressure distribution from a fully turbulent run of the McDonald-Camarata model was compared with the hand transition run of the same model.

The comparison on the centerbody does not show any major differences in the two runs, with the hand transition run doing slightly better. However, the cowl surface comparison suggests that the fully turbulent run represents the flow physics better than the hand-transition run. Thus, if we were to rely only on comparisons based on surface pressure, we would conclude that fully turbulent calculations are better than calculations that account for transition in the flow. Comparisons of pitot pressure disprove this point.

B. Full Navier-Stokes (NS) Calculations

Results from the application of the NASCRIN code to the P-8 inlet were reasonably good. As mentioned previously, the only case run with the NS code is the Baldwin-Lomax model with fully turbulent flow. Figure 5 is a comparison of the full global results of the application of NASCRIN with computations using the PNS code, plotted with experimentally measured values of pitot pressure. For purposes of comparison between different codes, the PNS solution presented in Fig. 5 also uses the Baldwin-Lomax model with fully turbulent flow. Favorable comparison with experimental data is especially noted at the most upstream survey stations. At $X/R = 5.72$, postshock oscillations due to insufficient grid points are visible in the NS solution. Comparison of the NS computations with the experimental data becomes less favorable with increasing distances downstream. In addition, at $X/R = 6.11$ and 6.32, the PNS and NS solutions show different pitot-pressure profiles in the freestream. The reason for this is not well understood yet.

One key result to be mentioned from the present calculations was the prediction by the codes (both NS and PNS) of a flow separation bubble on the centerbody surface in the immediate region of the cowl-shock centerbody-reflection boundary-layer interaction ($X/R > 6.32$). Although the pitot pressure profiles in Fig. 5 for the NS solution do not exhibit the sharp gradient on the lower surface as in the PNS solution, an examination of the velocity distribution confirmed regions of reverse flow for the NS solution.

The erroneous prediction of flow separation by the code is attributed to failure of the Baldwin-Lomax turbulence model. It is felt that this model, which was originally developed to predict transonic external flows, can seriously fail in the regions of shock-boundary-layer interactions in high-speed internal flows. In applying the PNS code to the inlet, when using the Baldwin-Lomax model, the axial location of the onset of separation was predicted to be about the same place as predicted by the Navier-Stokes code. This also justifies the use of a PNS code for the present investigation even though there is a small separation bubble in the flowfield.

There is a reason to doubt whether the conclusions in the paper can be generalized to other hypersonic inlets, since only one test case was presented. It is hoped that further numerical investigation will be performed in the future, when test data for additional hypersonic inlet geometries become available.

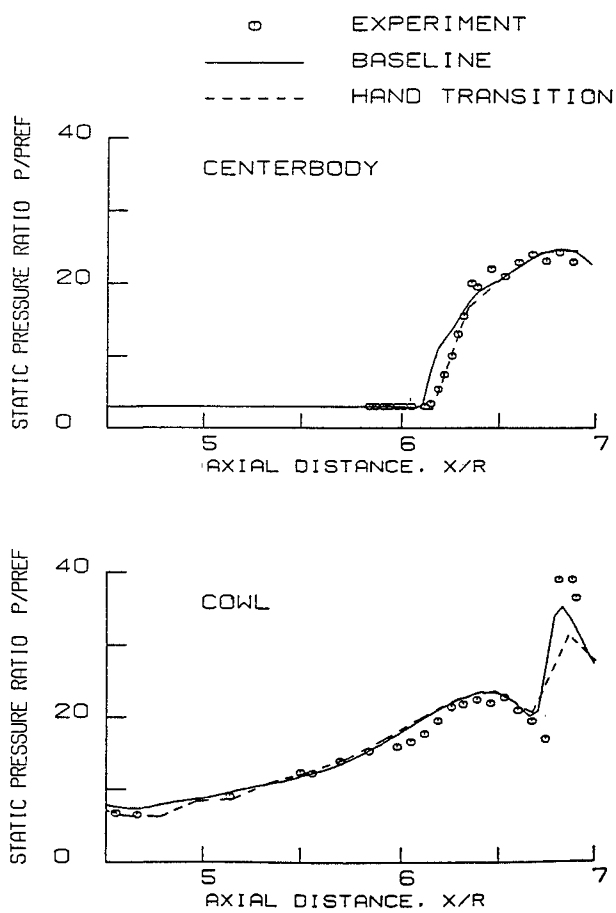

Fig. 4 Surface-pressure comparisons for fully turbulent and transitional runs using McDonald-Camarata model.

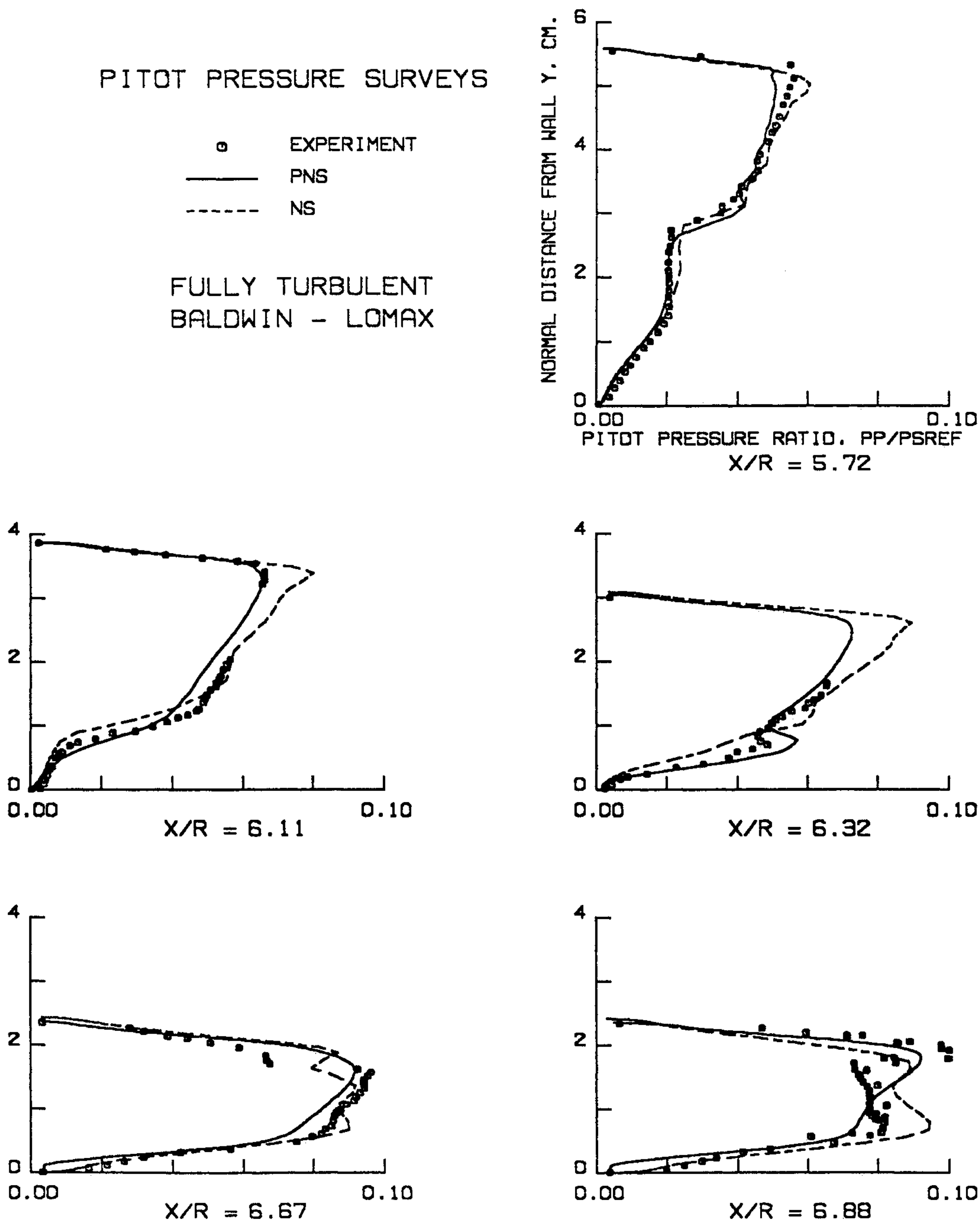

Fig. 5 Pitot-pressure profiles for PNS and NS solutions.

VI. Conclusions

A study was conducted to evaluate the performance of various turbulence models in a Mach 7.4 (P-8) hypersonic inlet. A PNS code and NS code were used to compare computational results with available experimental data. The McDonald-Camarata and Baldwin-Lomax models were the two zero-equation models used in the study. The TKE model was chosen as a representative higher-order model. The models were evaluated on the basis of comparisons of pitot-pressure profiles at various axial locations in the inlet. Comparisons for fully turbulent runs for the three models were followed up by an attempt to improve the solutions by introducing transition on the centerbody and cowl surfaces of the inlet. The conclusions based on the various findings of the study are summarized in the following paragraphs.

1) Calculations for *fully* turbulent flow conditions when using the McDonald-Camarata model show that this model, when compared with experimental data, gives the best solution among the three models.

2) In using the McDonald-Camarata model, the overall flow solution on the centerbody and cowl surfaces is improved when transition is forced in the code in the regions suggested by the experimental data.

3) The attempts of using the Baldwin-Lomax and TKE models to predict transition automatically from laminar to turbulent flow failed. Both models are found to predict transition very early on in the development of the boundary layer, and hence their fully turbulent and transition runs do not show any meaningful differences.

4) The Baldwin-Lomax model predicts separation of flow for both the NS and PNS codes where none is known to exist in the experiments. This suggests that the model is inadequate to deal with boundary layers developing in internal flows under strong adverse pressure gradients, particularly in the

regions of shock-boundary-layer interactions such as those found in hypersonic inlets.

5) Surface pressure comparisons show that when analyzing the performance of different turbulence models, reliance on a comparison of surface pressures only may lead to misleading results and conclusions. Hence, it is recommended that in such analyses, a much finer comparison, for example, of pitot pressures, must be conducted to determine the most effective model for the particular application.

Acknowledgments

The authors would like to express their sincere appreciation to Dr. Lou A. Povinelli for his support of this research. Funding for the research came from the NASA Lewis Research Center Office of Chief Scientist under Dr. Marvin E. Goldstein (Grant NAG 3-676). Technical monitor for the grant is Mr. Jerry R. Wood.

References

[1]White, M. E., Drummond, J. P., and Kumar, A., "Evolution and Application of CFD Techniques for Scramjet Engine Analysis," *AIAA Journal of Propulsion and Power*, Vol. 3, Sept.–Oct. 1987, pp. 423–439.

[2]Marvin, J. G., "Turbulence Modeling for Computational Aerodynamics," *AIAA Journal*, Vol. 21, July 1983, pp. 941–955.

[3]Seebaugh, W. R., "Hypersonic Flows in Large-Scale Inlet Models," *AIAA Journal of Aircraft*, Vol. 10, Jan. 1973, pp. 38–44.

[4]McDonald, H. and Camarata, F. J., "An Extended Mixing Length Approach for Computing the Turbulent Boundary Layer Development," *Proceedings, Stanford Conference on Computation of Turbulent Boundary Layers*, Vol. I, Stanford University, Stanford, CA, 1969, pp. 83–98.

[5]Benson, T. J., personal communication, Dec. 1985.

[6]Bushnell, D. M. and Beckwith, I. E., "Calculation of Nonequilibrium Hypersonic Turbulent Boundary Layers and Comparisons with Experimental Data," *AIAA Journal*, Vol. 8, Aug. 1970, pp. 1462–1469.

[7]Baldwin, B. and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[8]Schetz, J. A., *Foundations of Boundary Layer Theory for Momentum, Heat and Mass Transfer*, Prentice-Hall, Inc., Englewood Cliffs, NJ, 1984, pp. 204–212.

[9]Buggeln, R. C., Kim, Y. N., and McDonald, H., "Computation of Multi-Dimensional Viscous Supersonic Flow," NASA CR-4021, Contract NAS3-22027, Oct. 1986.

[10]Kumar, A., "User's Guide for NASCRIN—A Vectorized Code for Calculating Two-Dimensional Supersonic Internal Flow Fields," NASA Langley Research Center, Hampton, VA, NASA TM 85708, 1984.

[11]Ng, W. F., Benson, T. J., and Kunik, W. G., "Real Gas Effects on the Numerical Simulation of Hypersonic Inlet," *AIAA Journal of Propulsion and Power*, Vol. 2, July-Aug. 1986. pp. 381-382.

[12]Kunik, W. G., Benson, T. J., Ng, W. F., and Taylor, A. C., III, "Two- and Three-Dimensional Viscous Computations of a Hypersonic Inlet Flow," AIAA-87-0283, Jan. 1987.