NASA/TM—2002-211727

An Evaluation of Parameters Influencing Jet Mixing Using the WIND Navier-Stokes Code

Mary Ann Dembowski and Nicholas J. Georgiadis
Glenn Research Center, Cleveland, Ohio

August 2002

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at ***http://www.sti.nasa.gov***

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at 301–621–0134

- Telephone the NASA Access Help Desk at 301–621–0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

NASA/TM—2002-211727

An Evaluation of Parameters Influencing Jet Mixing Using the WIND Navier-Stokes Code

Mary Ann Dembowski and Nicholas J. Georgiadis
Glenn Research Center, Cleveland, Ohio

National Aeronautics and
Space Administration

Glenn Research Center

August 2002

The Aerospace Propulsion and Power Program at
NASA Glenn Research Center sponsored this work.

Available from

NASA Center for Aerospace Information
7121 Standard Drive
Hanover, MD 21076

National Technical Information Service
5285 Port Royal Road
Springfield, VA 22100

Available electronically at http://gltrs.grc.nasa.gov

An Evaluation of Parameters Influencing Jet Mixing Using the WIND Navier-Stokes Code

Mary Ann Dembowski and Nicholas J. Georgiadis
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Abstract

The WIND code, a Reynolds-averaged Navier-Stokes solver used for a variety of aerospace flow simulations, was investigated for a Mach 2 nozzle at a series of nozzle stagnation temperatures. Comparisons of WIND calculations are made to experimental measurements of axial velocity, Mach number, and stagnation temperature along the jet centerline. The primary objective was to investigate the capabilities of the two-equation turbulence models available in WIND—version 4.0, for the analysis of heated supersonic nozzle flows. The models examined were the Menter Shear Stress Transport (SST) model and the Chien k-ε model, with and without the compressibility correction due to Sarkar. It was observed that all of the turbulence models investigated produced solutions that did not agree well with the experimental measurements. The effects of freestream Mach number and turbulent Prandtl number specifications were also investigated.

Introduction

Computational fluid dynamics (CFD) codes have become an attractive option for the analysis of aerospace systems due to advances in flow solver and computer hardware technologies. For the investigation of one particular class of flows, that of aircraft engine exhaust nozzles, Reynolds-averaged Navier-Stokes (RANS) solvers have been used extensively. In the RANS approach, all effects of turbulence are replaced with a turbulence model. However, with realistic nozzle flows typically dominated by turbulent mixing, the accuracy of a flow simulation is usually determined by the capabilities of the turbulence model employed. Previous studies, such as those of Barber et al.[1] and Georgiadis et al.[2] have indicated that turbulence models available for exhaust nozzle flow simulations have difficulty in accurately predicting the effects of compressibility, high temperatures, and three dimensional flow features.

Due to these deficiencies in RANS methods, interest has arisen in more sophisticated techniques such as Large Eddy Simulation (LES). However, the routine use of LES methods to calculate nozzle and jet flows to a high degree of accuracy is not yet possible. This is due to the requirement of prohibitively large computational grids and very long computer run times using LES for even simple configuration nozzles at low Reynolds numbers.[3,4] As a result, RANS methods will still be the practical technique for calculating nozzle and jet flows in the foreseeable future, and it is necessary to identify the capabilities of currently available RANS methods.

In the current study, an assessment of the WIND code-version 4.0, a production-use RANS solver,[5] is made for a supersonic round nozzle that was tested for a series of nozzle stagnation temperatures. The primary objective of this study is to investigate the turbulence models most frequently used in WIND for nozzle and jet flow analyses. In addition, the effects of other computational settings are also investigated including variation of the turbulent Prandtl number, the Mach number set for the ambient air surrounding the jet, and incorporation of a correction for compressibility.

Experimental Nozzle Configuration

The present study uses the benchmark experiments performed at the NASA Langley Research Center (LaRC) Jet Noise Laboratory.[6] These experiments investigated an axisymmetric water-cooled Mach 2 nozzle with an exit diameter, D = 3.60 in. (91.44 mm). Five sets of data were collected from the cases, with the nozzle plenum stagnation temperature varied as indicated in Table 1. The nozzle operated at fully expanded conditions for this series of cases. The Reynolds number, based on D, varied from 1.3×10^6 to 8.3×10^6.

Table 1: Experimental nozzle flow conditions

Run no.	T_{tj}, °F	T_{tj}, °R
6	104	563.67
21	455	914.67
41	900	1359.67
51	1550	2009.67
61	2000	2459.67

Computational Model

The computational grid used for these nozzle calculations was a slightly modified version of that used for calculations discussed in reference 1. This axisymmetric grid, shown in figure 1, consisted of three zones as follows. The first zone represented the internal nozzle region, with 121 axial points and 81 vertical points. The grid was slightly clustered to the exit of the nozzle, where an interface to the downstream mixing zone was placed. The grid was clustered to the wall such that two points were placed within the laminar sublayer of the developing boundary layer along the nozzle wall. The second zone is positioned above the nozzle and represents the inflow of the ambient air. The original grid used in reference 1 had 9 axial points and 34 vertical points. Because initial calculations with the WIND code indicated stability problems in this region, this zone was modified by increasing the number of axial points from 9 to 41, and moving the physical position of the inflow slightly further upstream. This modification improved the stability characteristics of the calculations discussed later in this report. The third zone was the nozzle exhaust region downstream of zones one and two. This zone had 121 axial and 121 vertical points. The interface of zone three with zones one and two formed a contiguous, point to point match. In addition, 8 vertical points were placed at the location of the nozzle lip, which is represented by the vertical line between blocks one and two at the interface with block three. These additional points were carried through zone three to help resolve the flow development just downstream of the nozzle lip.

Grid sensitivity studies discussed in reference 1 indicated that the original grid was sufficient using five Navier-Stokes solvers utilized in that work. Since the WIND code used in the current study was derived from NASTD, one of the solvers in the former investigation, further grid sensitivity analyses were not conducted here.

Boundary condition types and zone interfacing were set for calculations with WIND using the GMAN pre-processor. An arbitrary inflow was specified for both the nozzle plenum (zone one) and the freestream inflow (zone two). The inner wall of the nozzle was set as a no-slip, adiabatic surface. The axis of symmetry is set as an inviscid wall, but was further specified as axisymmetric within the WIND run data file. The top and bottom faces of zones two and three were also set as inviscid walls. The right face of zone three was specified an outflow. In the WIND run data file, the actual flow conditions were specified corresponding to the boundary condition types set with GMAN, and are described next.

In the experiment, the surrounding air was at rest with the pressure and temperature corresponding to 14.7 psi and 530° R, respectively. The majority of cases were run assuming a small forward freestream velocity corresponding to Mach 0.01. To determine the effect of the ambient air settings, two other freestream conditions, corresponding to Mach 0.1 and Mach 0.2 were investigated for selected cases, as will be discussed in the results section. The nozzle plenum stagnation pressure was set to 115.02 psi for all cases, and the stagnation temperatures were set to the values indicated in table 1.

The turbulence models used within WIND version 4.0 for this study were the Menter two-equation model,[7,8] also known as the Shear-Stress Transport model (SST) and the Chien two-equation k-ε model.[9] The SST model applies the k-ω form of its equations in near wall-bounded regions and the k-ε form in free shear layer regions. In addition, the Chien model was used with and without the Sarkar compressibility correction.[10,11] The Sarkar compressibility correction effectively increases the rate of the turbulent kinetic energy dissipation and then lowers the resultant turbulent viscosity. Finally, the turbulent Prandtl number, which is frequently set to a default value of 0.9 in production flow solvers such as WIND, was varied from 0.5 to 0.9 while using the SST turbulence model.

Results

The first comparisons to be discussed investigated effects of the freestream Mach number. This value is ideally set to the conditions of the air surrounding the jet, but compressible flow solvers such as WIND have trouble converging the extreme differential between the slow freestream air and the supersonic jet. Beginning with the unheated nozzle case (104 °F) WIND calculations were obtained using the SST turbulence model for each of the three freestream Mach numbers, 0.01, 0.10, and 0.20. While Mach 0.01 models the ambient air stream most closely, figure 2 indicates that the jet spread rate is substantially overpredicted as was also reported in reference 1. As in the previous work, the overpredictions are due to deficiencies in the turbulence models for highly compressible flow. In figure 2 and all following figures, x/D = 0 represents the nozzle exit. As the freestream Mach number is increased to 0.1 and 0.2, the jet mixing rate is reduced. The conclusion that the Mach 0.2 case compares best with the experimental data should not be drawn, however, because the "closer" agreement is a result of turbulence model inadequacy being compensated for by the modified freestream conditions. Similar results were obtained for calculations obtained with the nozzle stagnation temperature set to 1550 °F, as shown in figure 3. The centerline stagnation temperature profiles do not show much sensitivity, which is to be expected since the freestream stagnation temperature does not vary much between Mach 0.01 and Mach 0.20 while holding the static temperature fixed.

The next set of results examine the effects of varying the turbulent Prandtl number. As mentioned previously, a value of 0.9 is the default value when using WIND and other similar production flow solvers. However, in free shear layers, a value for the turbulent Prandtl number closer to 0.5 has been suggested.[12] The 1550 °F nozzle stagnation temperature case was investigated using WIND again with the SST turbulence model and turbulent Prandtl number settings of 0.5, 0.7, and 0.9. Centerline profiles of axial velocity and Mach number shown in figures 4(a) and 4(b), respectively, indicate little effect of varying the turbulent Prandtl number. The centerline stagnation temperature, figure 4(c), does decay faster as the turbulent Prandtl number is reduced. These results cannot justify a particular setting for the turbulent Prandtl number but indicate that the turbulent thermal transport is significantly affected by the value used. Some efforts such as those of Kenzakowski et al.[13] have begun to investigate a variable turbulent Prandtl number model for more accurate treatment of complex exhaust nozzle flows with thermal transport as a dominant flow feature.

While the freestream Mach number and turbulent Prandtl number variations were investigated only for selected cases, the turbulence model investigations were conducted for all of the nozzle operating conditions listed in Table 1 using the SST and Chien k-ε turbulence models. The SST model is currently one of the most popular turbulence models used in WIND and similar production-use RANS solvers because it uses a k-ω formulation in wall-bounded regions and a k-ε formulation in free shear layers. The k-ε formulation used within SST differs slightly from the free shear layer form of standard k-ε models, with the most significant difference occurring in the ε equation diffusion coefficient, σ_ε. Models such as the Chien and Jones-Launder[14] formulations use σ_ε = 1.30, while the k-ε formulation used within SST uses σ_ε = 1.17. Reference 15 mentions that this is done in the SST approach to more accurately calculate the logarithmic portion of the boundary layer.

The Chien model as implemented in WIND provides the option to include the compressibility correction due to Sarkar.[10,11] As a result, three WIND solutions were obtained for each of the five operating conditions listed in Table 1, the first using the SST model, the second using the standard Chien model, and the third using the Chien model incorporating the Sarkar compressibility correction. All of these cases were run using a freestream Mach number of 0.01 and the default setting in WIND for the turbulent Prandtl number, 0.9.

Figures 5 through 9 compare centerline profiles of axial velocity, Mach number, and stagnation temperature for the three WIND solutions to the experimental data of Seiner. The stagnation temperature is not shown for the unheated case with nozzle plenum temperature set to 104 °F. For each operating point, the Chien k-ε solution obtained with no compressibility correction predicts the fastest mixing, and much faster than indicated by the experimental data. The SST solutions also consistently predict a faster mixing rate than the data. Compared with the baseline Chien model solutions, the SST results predict a longer potential core, but then a steeper slope in jet decay further downstream. These results are most likely due to the differences in σ_ε used by the Chien and SST k-ε formulations.

The Chien model results obtained with the Sarkar compressibility correction indicate the longest potential cores and slowest mixing of any of the solutions obtained for all of the temperatures investigated. In all cases, these solutions obtained with the compressibility correction overpredict the potential core length. Further downstream, the rate of decay is still slightly overpredicted for the unheated nozzle case, but this trend reverses for the higher temperatures.

Figure 10 provides the centerline velocity profile variations with temperature for the experimental data and computational approaches. While figures 5 to 9 indicated that none of the WIND solutions were able to match the experimental data at any particular operating point, figure 10 further indicates that not even the trends of jet decay variation with nozzle stagnation temperature that were observed in the experiments were able to be simulated adequately with the computational approaches employed here.

Conclusions

The results of this study indicate the current status of the WIND flow solver to simulate heated supersonic round nozzle exhaust flows. Two-equation turbulence models, such as the SST and the Chien k-ε formulations in WIND used in this study, are the most frequently used option within production flow RANS solvers for nozzle exhaust simulations. The results presented in this report indicate that models such as these do not predict supersonic nozzle flows accurately. These results are consistent with those in previous studies such as those of references 1 and 2. Simple corrections for compressibility, such as that due to Sarkar, reduce the overpredictions in jet mixing rate, but also result in jet potential core lengths that are longer than those observed in the experiments across the temperature range.

The results of the freestream Mach number dependence investigation indicate that for exhaust nozzle flows exiting into ambient air, it is necessary to set the freestream Mach number as low as possible. While doing so slows the convergence characteristics of RANS solvers such as WIND, failure to do this will likely result in artificially slower jet mixing rates. At Mach 0.2, the highest freestream Mach number investigated here, the jet mixing was the lowest of any investigated, and in general was closest to the experimental data. However, the conclusion that simulating jet flows with artificially elevated freestream Mach number should not be drawn from these results, because the "closer" agreement observed was the result of the compensation of the modified freestream conditions for the turbulence model inadequacy.

The turbulent Prandtl number investigation indicates that the setting of this parameter can significantly affect thermal transport in the jet plume. Because the turbulent Prandtl number is believed to vary significantly in high temperature jet flows, simple tuning of the turbulent Prandtl number for a particular case will not likely enable more accurate predictions. While some work has been initiated to formulate a variable turbulent Prandtl number model, it will likely be some time before a rigorous capability is demonstrated.

The limitations of the WIND code demonstrated here for supersonic nozzle flowfields, using standard two-equation turbulence models, are representative of nearly all currently available production-use RANS solvers. As a result, the calculations discussed in this report can be used to benchmark current capabilities against future developments. Efforts to improve the capability to predict nozzle flowfields by incorporating more sophisticated turbulence model approaches into WIND such as an explicit algebraic Reynolds stress model are underway. Improved thermal transport models, such as a variable turbulent Prandtl number formulation, should also be investigated.

References

(1) Barber, T.J., Chiappetta, L.M., DeBonis, J.R., Georgiadis, N.J., and Yoder, D.A., "Assessment of Parameters Influencing the Prediction of Shear Layer Mixing," *Journal of Propulsion and Power*, Vol. 15, No. 1, 1998, pp. 45–53.

(2) Georgiadis, N.J., Yoder, D.A., and DeBonis, J.R., "A Comparison of Three Navier-Stokes Solvers for Exhaust Nozzle Flowfields," AIAA Paper 99–0748, Jan. 1999.

(3) DeBonis, J.R. and Scott, J.N., "A Large-Eddy Simulation of a Turbulent Compressible Round Jet," AIAA Paper 2001–2254, May 2001.

(4) Georgiadis, N.J., Alexander, J.I.D., and Reshotko, E., "Development of a Hybrid RANS/LES Method for Compressible Mixing Layer Simulations," AIAA Paper 2001–0289, Jan. 2001.

(5) Nelson, C.C. and Power, G.D., "CHSSI Project CFD-7: The NPARC Alliance Flow Simulation System," AIAA Paper 2001–0594, Jan. 2001.

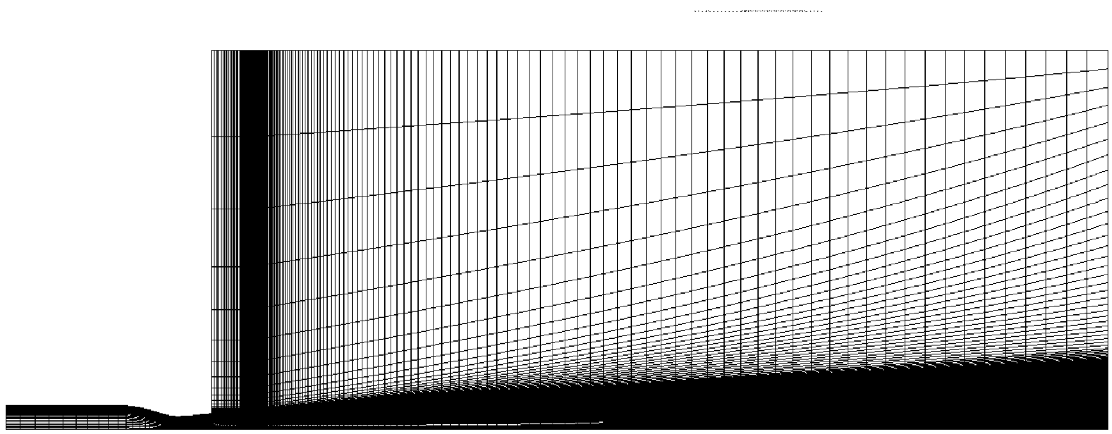
(6) Seiner, J.M., Ponton, M.K., Jansen, B. J., and Lagen, N. T., "The Effects of Temperature on Supersonic Jet Noise Emission," DGLR/AIAA 14th Aeroacoustics Conference, Aachen, Germany, AIAA Paper No. 92–02–046, May 1992.

(7) Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598–1605.

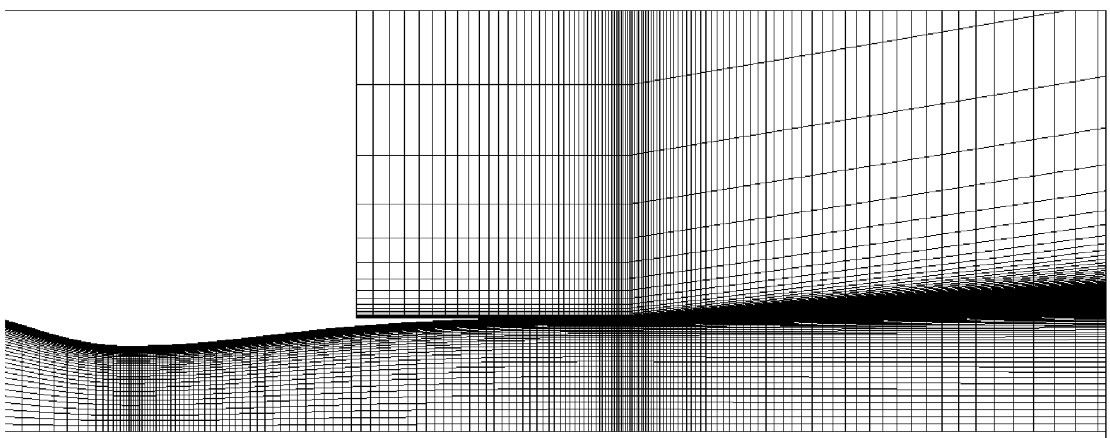
(8) Menter, F. R., "Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows," AIAA Paper 93–2906, July 1993.

(9) Chien, K.-Y., "Predictions of Channel and Boundary Layer Flows with a Low-Reynolds-Number Turbulence Model," *AIAA Journal*, Vol. 20, No. 1, 1982, pp. 33–38.

(10) Sarkar, S., Erlebacher, G., Hussaini, M.Y., and Kreiss, H.O., "The Analysis and Modeling of Dilatational Terms in Compressible Turbulence," *Journal of Fluid Mechanics*, Vol. 227, 1991, pp. 473–493.
(11) Sarkar, S., and Lakshmanan, B., "Application of a Reynolds Stress Turbulence Model to the Compressible Shear Layer," *AIAA Journal*, Vol. 29, No. 5, 1991, pp. 743–749.
(12) Wilcox, D.C., Turbulence Modeling for CFD, DCW Industries, Second Ed., 1998.
(13) Kenzakowski, D.C., Papp, J., and Dash, S.M., "Evaluation of Advanced Turbulence Models and Variable Prandtl/Schmidt Number Methods for Propulsive Flows," AIAA Paper 2000–0885, Jan. 2000.
(14) Jones, W.P., and Launder, B.E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," *International Journal of Heat and Mass Transfer*, Vol. 15, Feb. 1972, pp. 301–314.
(15) Menter, F.R. and Rumsey, C.L., "Assessment of Two-Equation Turbulence Models for Transonic Flows," AIAA Paper 94–2343, June 1994.

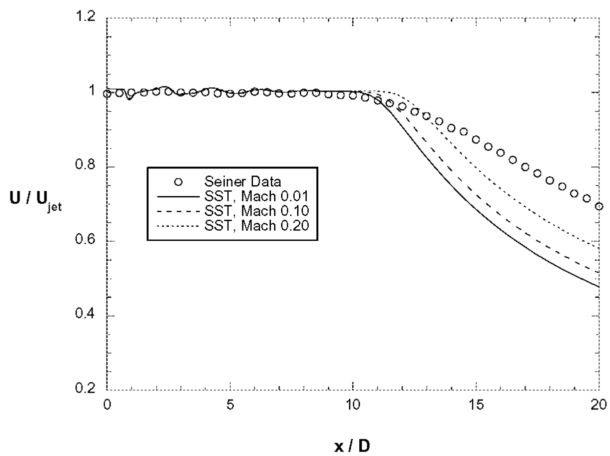


(a) entire grid

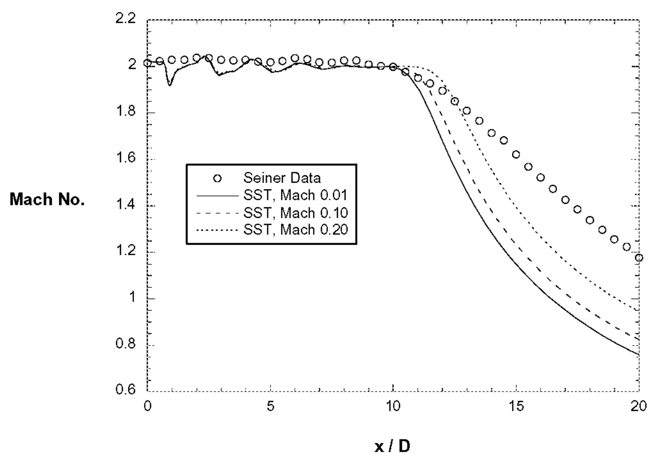


(b) detail near nozzle exit plane

Fig. 1. Computational grid

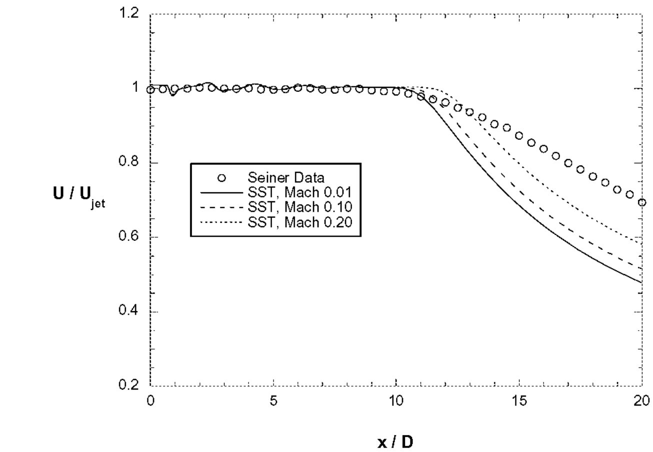


(a) centerline axial velocity

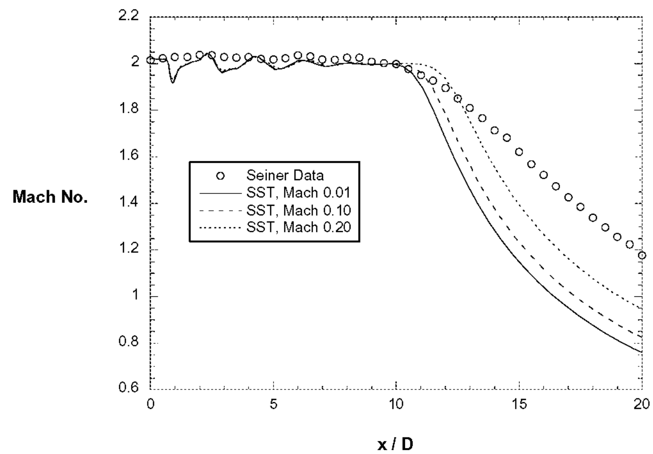


(b) centerline Mach number

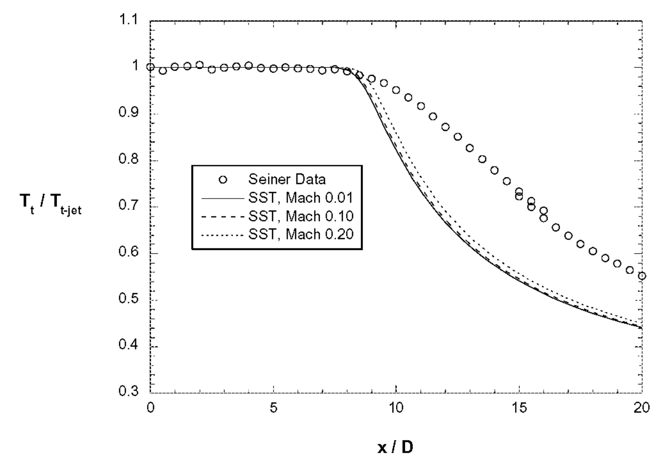
Fig. 2. Comparison of SST solutions at T_t = 104 F with varying freestream Mach number.



(a) centerline axial velocity

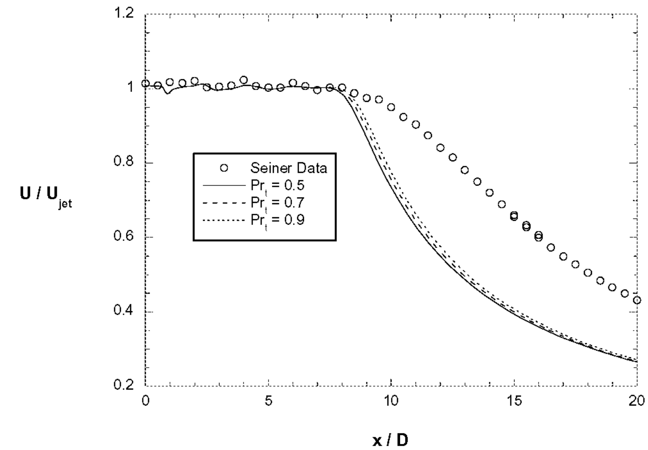


(b) centerline Mach number

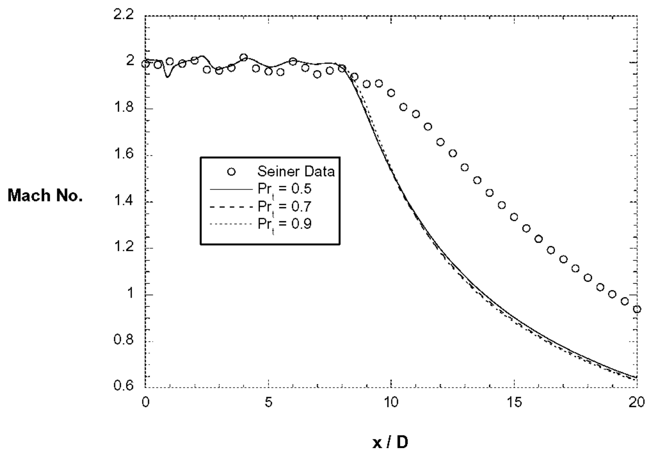


(c) centerline stagnation temperature

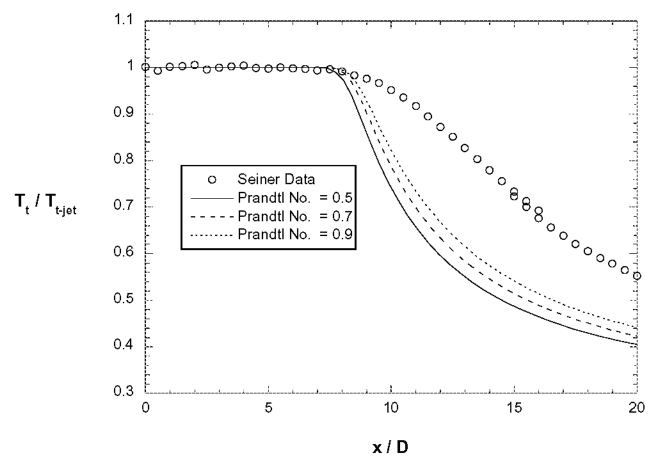
Fig. 3. Comparison of SST solutions at T_t = 1550 F with varying freestream Mach number.



(a) centerline axial velocity

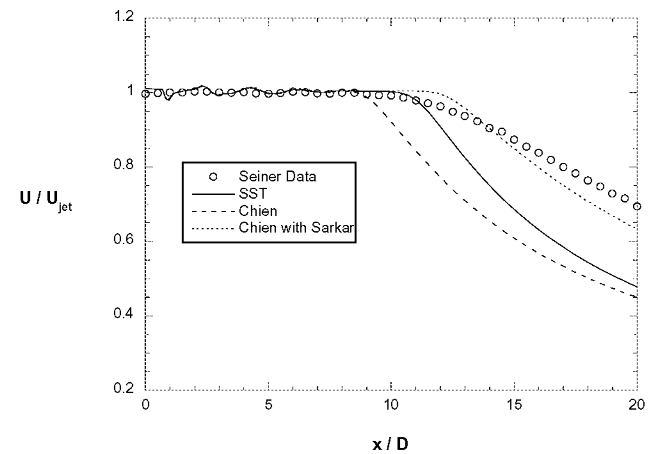


(b) centerline Mach number

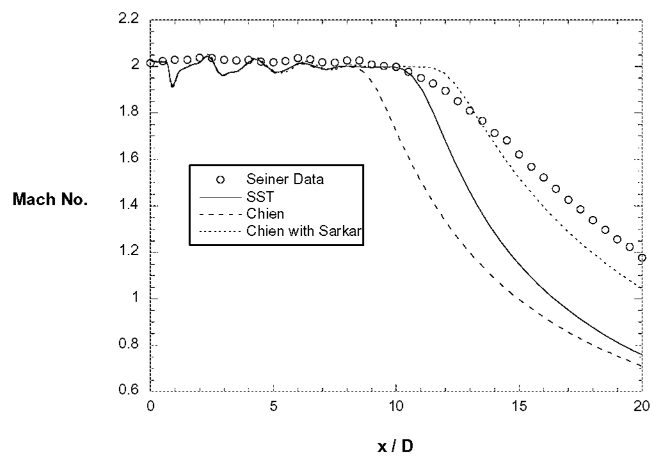


(c) centerline stagnation temperature

Fig. 4. Comparison of SST solutions at T_t = 1550 F with varying turbulent Prandtl number.

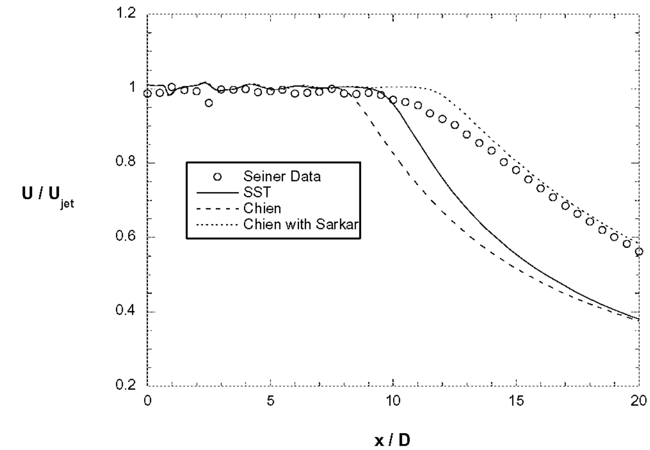


(a) centerline axial velocity

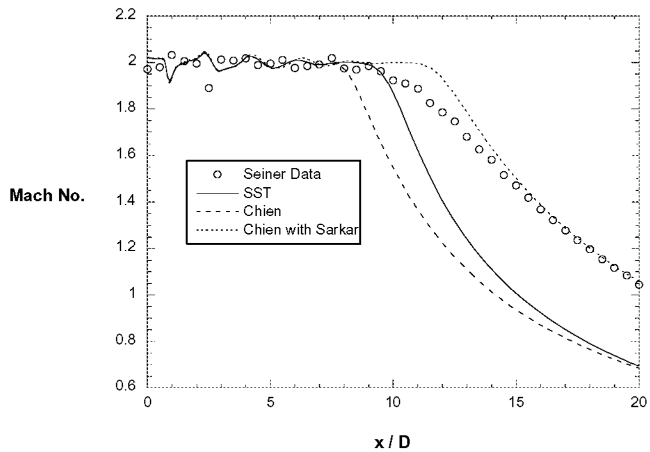


(b) centerline Mach number

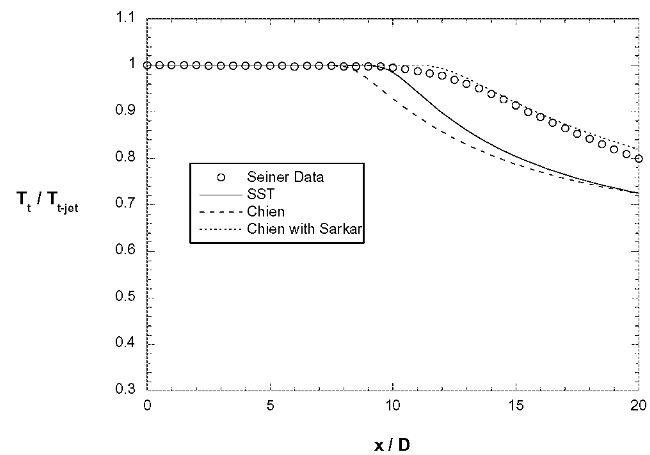
Fig. 5. Comparison of solutions at T_t = 104 F using different turbulence models.



(a) centerline axial velocity

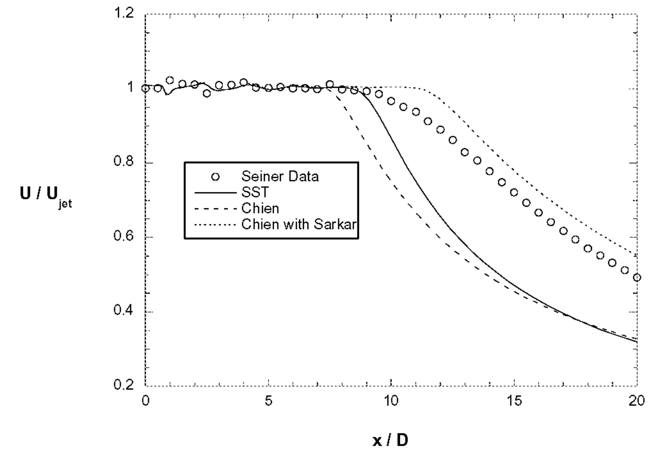


(b) centerline Mach number

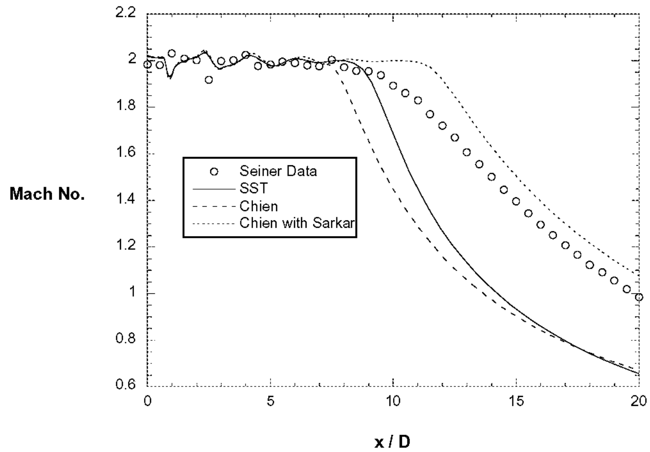


(c) centerline stagnation temperature

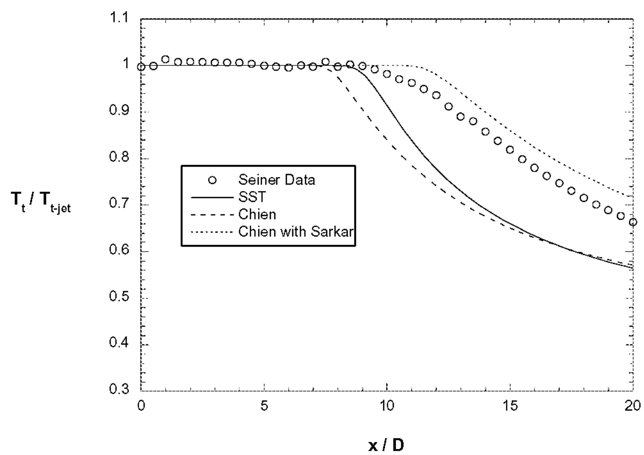
Fig. 6. Comparison of solutions at T_t = 455 F using different turbulence models.



(a) centerline axial velocity

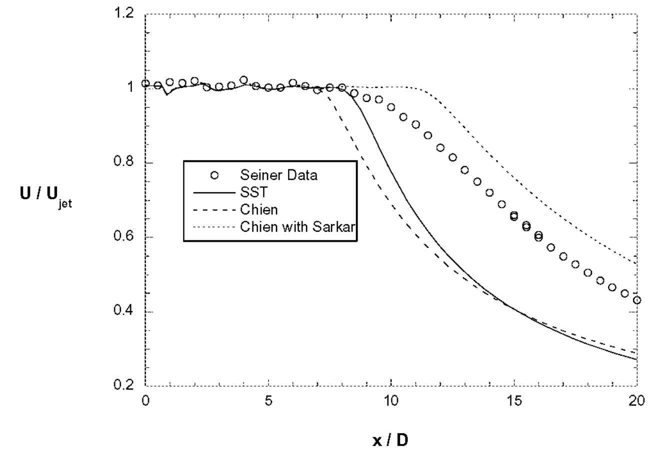


(b) centerline Mach number

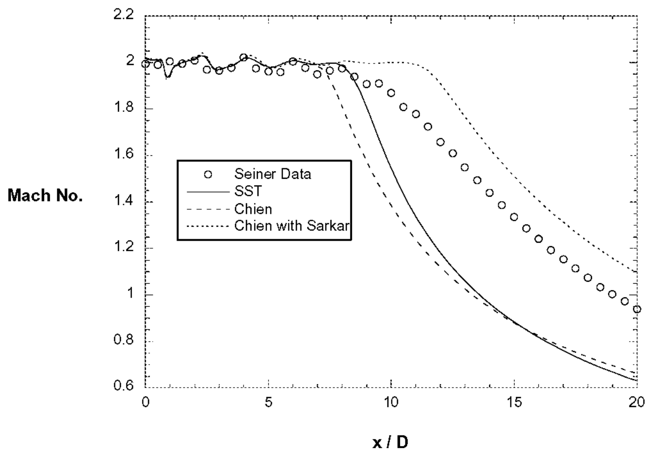


(c) centerline stagnation temperature

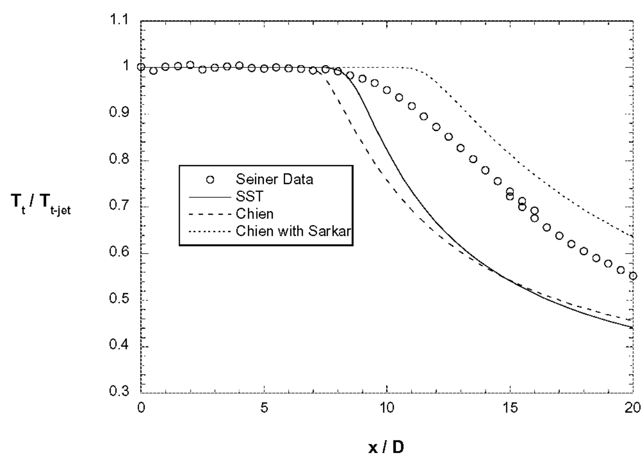
Fig. 7. Comparison of solutions at T_t = 900 F using different turbulence models.



(a) centerline axial velocity

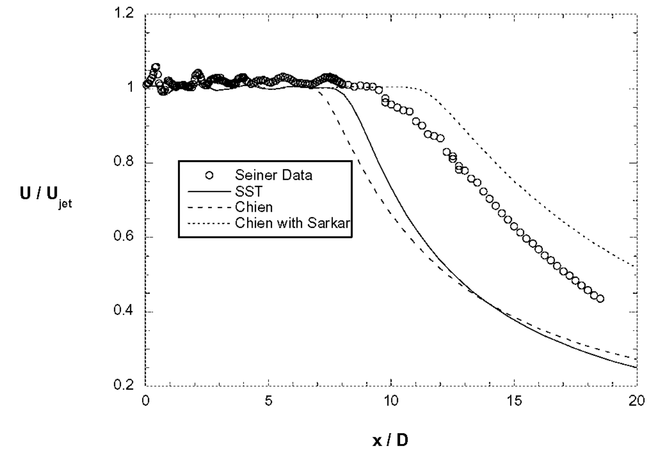


(b) centerline Mach number

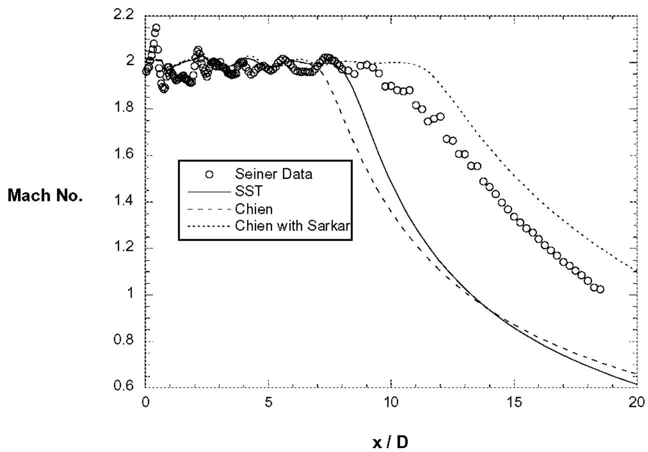


(c) centerline stagnation temperature

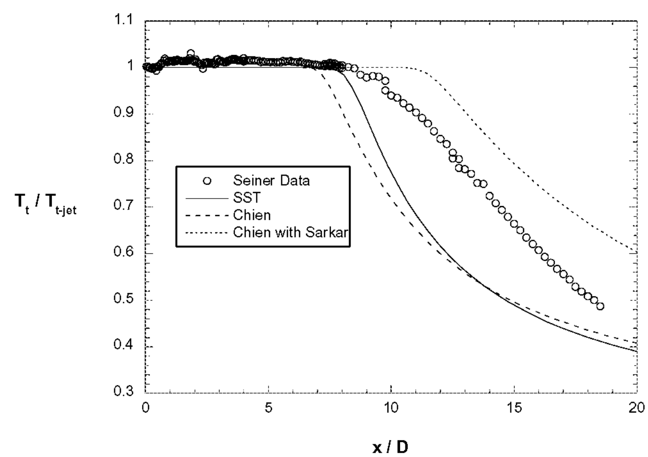
Fig. 8. Comparison of solutions at T_t = 1550 F using different turbulence models.



(a) centerline axial velocity

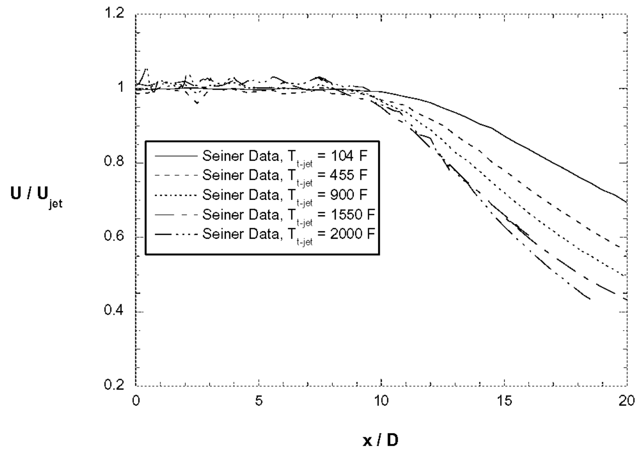


(b) centerline Mach number

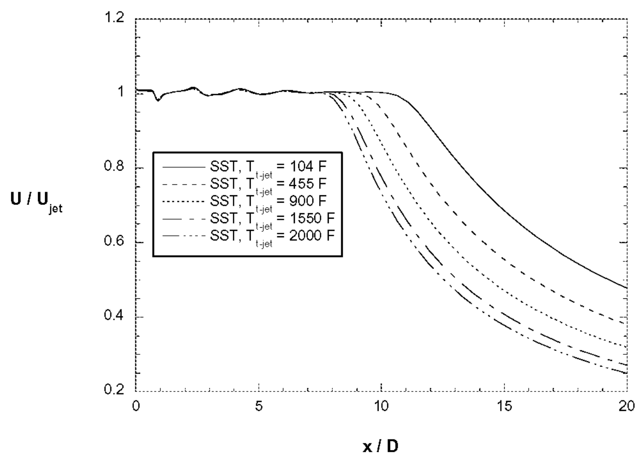


(c) centerline stagnation temperature

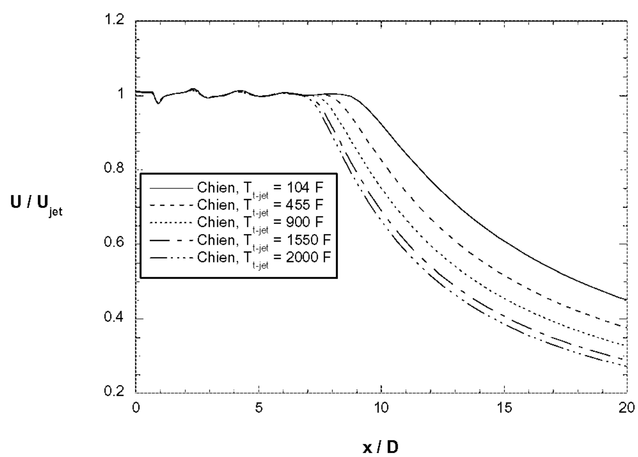
Fig. 9. Comparison of solutions at T_t = 2000 F using different turbulence models.



(a) Seiner data

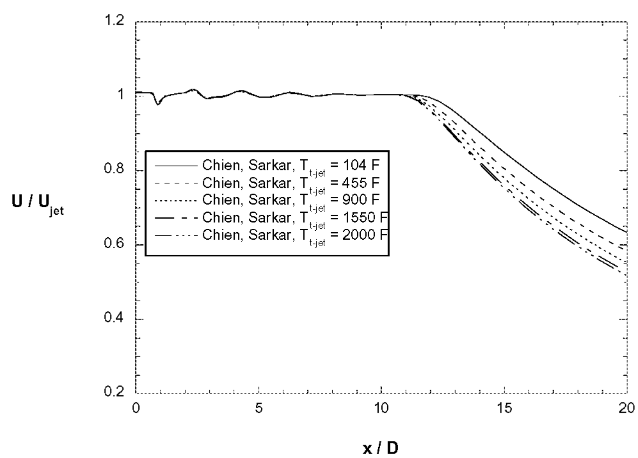


(b) SST calculations



(c) Chien calculations

Fig. 10. Centerline axial velocity with varying nozzle stagnation temperature.



(d) Chien (with Sarkar) calculations

Fig. 10. Centerline axial velocity with varying nozzle stagnation temperature (concluded).

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
August 2002

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
An Evaluation of Parameters Influencing Jet Mixing Using the WIND Navier-Stokes Code

5. FUNDING NUMBERS
WU–708–90–43–00

6. AUTHOR(S)
Mary Ann Dembowski and Nicholas J. Georgiadis

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E–13478

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM—2002-211727

11. SUPPLEMENTARY NOTES
Responsible person, Nicholas J. Georgiadis, organization code 5860, 216–433–3958.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Categories: 02 and 07 Distribution: Nonstandard
Available electronically at http://gltrs.grc.nasa.gov
This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***

The WIND code, a Reynolds-averaged Navier-Stokes solver used for a variety of aerospace flow simulations, was investigated for a Mach 2 nozzle at a series of nozzle stagnation temperatures. Comparisons of WIND calculations are made to experimental measurements of axial velocity, Mach number, and stagnation temperature along the jet centerline. The primary objective was to investigate the capabilities of the two-equation turbulence models available in WIND, version 4.0, for the analysis of heated supersonic nozzle flows. The models examined were the Menter Shear Stress Transport (SST) model and the Chien k-ε model, with and without the compressibility correction due to Sarkar. It was observed that all of the turbulence models investigated produced solutions that did not agree well with the experimental measurements. The effects of freestream Mach number and turbulent Prandtl number specifications were also investigated.

14. SUBJECT TERMS
Turbulent mixing; Supersonic jet; Reynolds-averaged Navier Stokes; Turbulence models

15. NUMBER OF PAGES
24

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	