21st Applied Aerodynamics Conference
23-26 June 2003, Orlando, Florida

AIAA 2003-4070

Temperature Corrected Turbulence Model for High Temperature Jet Flow

Khaled S. Abdol-Hamid*
S. Paul Pao†
Configuration Aerodynamics Branch, MS 499
NASA Langley Research Center
Steven J. Massey‡
Eagle Aeronautics, Inc.
Alaa Elmiligui‡
Analytical Services & Materials, Inc.

Abstract

It is well known that the two-equation turbulence models under predict mixing in the shear layer for high temperature jet flows. These turbulence models were developed and calibrated for room temperature, low Mach number, and plane mixing layer flows. In the present study, four existing modifications to the two-equation turbulence model are implemented in PAB3D and their accuracy is assessed for high temperature jet flows. In addition, a new temperature gradient correction to the eddy viscosity term is presented and validated. The new model was found to be in the best agreement with experimental data for subsonic, and supersonic jet flows at both low and high temperatures.

Introduction

Computational fluid dynamics (CFD) codes have become an attractive option for the analysis of aerospace systems due to advances in flow solvers and computer hardware technologies. With the re-emergence of jet noise as a significant contributor to total airplane noise, the accurate prediction of 3-D jet flows has again become an important research goal [1-3]. In recent years, sophisticated techniques such as Large Eddy Simulation (LES) have been used to simulate jet flow. 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 large computational grids and very long computer run time to analyze even a simple nozzle configuration at low Reynolds numbers [4-5].

Solving the Reynolds- averaged Navier-Stokes (RANS) equations is considered to be the practical approach for calculating steady state nozzle and jet flows, and it is necessary to enhance the capabilities of currently available RANS methods. Advanced turbulence models are needed to predict propulsion aerodynamic effects in transonic and supersonic free-stream conditions. Transport equations have been included in the standard, two-equation turbulence model of energy and dissipation rate (k-ε) [6]. The k-ε equations can be applied to the near-wall region as well as far away regions from wall boundaries. For flow regions far away from solid boundaries, the high Reynolds number form of the model can be used.

For transonic and supersonic flow propulsion applications, the local density variation in standard incompressible turbulence models does not adequately duplicate the experimentally observed reduction in growth rate of the mixing layer with increasing convective Mach number. However, substantial progress has been made in the development of appropriate compressibility corrections to the transport equation turbulence models, [7&8]. These corrections resulted from direct numerical simulation of homogeneous compressible turbulence. Notably, Sarkar et al [7] recognized the importance of including compressible dissipation in the two-equation turbulence model when computing high-speed flows. A simple correction was proposed for compressible dissipation that can be included easily in the existing two-equation turbulence models. The standard model is recovered when the model

---

* Aerospace Engineer, Associate Fellow AIAA
† Senior Research Scientist, Associate Fellow AIAA
‡ Senior Research Scientist, AIAA Member

Copyright © 2003 by the American Institute of Aeronautics and Astronautics, Inc. No copyright is asserted in the United States under Title 17, U.S. Code. The U.S. Government has a royalty-free license to exercise all rights under the copyright claimed herein for governmental purposes. All other rights are reserved by the copyright owner.

This material is declared a work of the U.S. Government and is not subject to copyright protection in the United States.

constants for these corrections are assumed to be zero.

For high-temperature jet flow, the standard turbulence models lack the ability to model the observed increase in growth rate of mixing layer [2-3]. Several researchers [9-14] modified one or more terms of the transport equations to compensate for the deficiency in predicting high temperature flows. All these modifications affect directly or indirectly the closure terms of the turbulent heat flux ($\rho\,\overline{u_i t}$) and stresses ($\rho\overline{u_i u_j}$). Theis and Tam [9] changed several coefficients in the turbulent transport equations. However, such extensive modifications of model coefficients completely changed the characteristics of the equations and it may cause deficiency in flow prediction accuracy for other flow problems. There were yet other attempts to affect the turbulence model through different modeling of turbulent heat flux term appearing in the average energy equation [12-14]. In general, these models are considered to be nonlinear sets of algebraic equations to better describe the heat flux term explicitly. These models had great success in simulating turbulent fully developed high temperature flows. In this paper, we will test the capabilities of such models in simulating high shear flows. This includes all the stages of jet flow development regions namely, core, mixing, and fully developed stages.

The objective of the present work is to present a simple modification for the turbulence model to simulate the physics of high temperature jet flows. Experimental studies by Seiner et. al. [2] and Thomas et. al. [3] showed that the high total temperature gradient led to faster mixing and spreading of jet flow. Based on these observations, a simple modification to the k-ε turbulence model is proposed. The modification is designed to simulate enhanced mixing only for high total temperature gradient jet flows. The modification will also take into account the commonly accepted compressibility effect corrections on mixing. New modifications are implemented in PAB3D [1] which is a three dimensional structured grid flow analysis computer code based on the RANS equations. The present study uses the benchmark experiments performed at the NASA Langley Research Center (LaRC) Jet Noise Laboratory [2] and several multistream subsonic jet flows [3] to validate and test present modifications.

Approach

The governing equations of the RANS formulation include the conservation equations for mass, momentum, energy, and the equation of state. In the present study, the perfect gas law is chosen to represent air properties. For turbulent flow, Reynolds stresses are modeled using the eddy viscosity concept. It was also observed that even using a more sophisticated algebraic stress model does not have much influence on mixing.

The mass, momentum, and energy conservation equations of the RANS equations can be written in a conservative form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial(\rho u_i u_j + p\delta_{ij})}{\partial x_j} = \frac{\partial(\tau_{ij} - \rho\overline{u_i u_j})}{\partial x_j}$$

$$\frac{\partial \rho e_0}{\partial t} + \frac{\partial(\rho e_0 u_i + p u_i)}{\partial x_i} = \frac{\partial(\tau_{ij} u_j - \rho\overline{u_i u_j} u_j)}{\partial x_i} - \frac{\partial(q_i - C_P \rho\,\overline{u_i t})}{\partial x_i} + \frac{\partial}{\partial x_i}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right]$$

To close the RANS equations, we will use the two-equation (k-ε) turbulence model [6] as follows:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k}{\partial x_i} = -\rho\overline{u_j u_i}\frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_i}\left[\mu_l + \frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right] - \rho\varepsilon$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \varepsilon}{\partial x_k} = -C_{\varepsilon 1}\rho\overline{u_j u_i}\frac{\partial u_i}{\partial x_j}\frac{\varepsilon}{k} + \frac{\partial}{\partial x_i}\left[\mu_l + \frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right] - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}$$

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon},$$

$$C_\mu = .09,\ C_{\varepsilon 1} = 1.44,\ and\ C_{\varepsilon 2} = 1.92$$

In this paper, we will examine five different modifications to the standard two equation turbulence model. The first four are directly related to the modeling of the turbulence heat flux terms ($\rho\,\overline{u_i t}$). The fifth modification is based on simple temperature gradient and

compressibility corrections to the eddy viscosity (μ_t). The modifications are:

1) The Simple Eddy Diffusivity (SED) is based on the Boussinesq viscosity model. This approach is used to model all the scalar diffusion terms appearing in the RANS and standard k-ε equations. For the heat flux term, the SED is written as follows:

$$\rho \overline{u_i t} = -\frac{\mu_\tau}{\sigma_\tau}\frac{\partial T}{\partial x_i} \qquad (1)$$

$\mu_t = \rho C_\mu k^2/\varepsilon$, $C_\mu = 0.09$, $\sigma_\tau = 0.9$

This is the simplest and most commonly used formulation in CFD codes to calculate the heat flux term.

2) So and Sommer [12] (hereinafter refered to as Sommer-So model) proposed the following algebraic heat-flux model including the effect of the mean-velocity gradient:

$$\rho \overline{u_i t} = -\rho k \tau_m C_{ji}\frac{\partial T}{\partial x_i} \qquad (2)$$

$$C_{ji} = -C_\lambda \delta_{ji} + \frac{1}{C_{1\theta}} \left\{(2C_\mu \tau_u + C_{2\lambda}\tau_m)S_{ji} + C_2 \tau_m \Omega_{ji}\right\}$$

$$C_2 = C_\lambda[1 - C_{2\theta}]$$

$$C_\lambda = 0.095, C_{1\theta} = 3.28,$$

$$C_{2\theta} = 0.4, \tau_u = \frac{k}{\varepsilon}, \tau_m = 0.83\tau_u$$

3) The Generalized Gradient Diffusion Hypothesis (GGDH) adds the stress effect into the heat flux term. A complete description of this model is discussed by Ronki and Gatski [13]

$$\rho \overline{u_i t} = -C_\tau \rho \overline{u_j u_i}\frac{k}{\varepsilon}\frac{\partial T}{\partial x_i} \qquad (3)$$

$$\rho \overline{u_j u_i} = 2\mu_\tau S_{ji} - \frac{2}{3}\delta_{ji}\rho k,$$

$C_\tau = 0.3$

4) This model adds more complexity to GGDH model by changing the constant C_τ to include the effect of mean-velocity gradient. Abe, et al.[14] describes the derivation of this model as follows:

$$\rho \overline{u_i t} = -C_{jl}\rho \overline{u_l u_i}\frac{k}{\varepsilon}\frac{\partial T}{\partial x_i} \qquad (4)$$

$$C_{jl} = C_\Delta\left\{-\delta_{jl} + \tau_m[C_{T32}S_{jl} + C_{T23}\Omega_{jl}]\right\}$$

$$C_\Delta \frac{C_{T1}\tau_m}{1 + 0.5\tau_m^2[C_{T23}^2\Omega^2 - C_{T32}^2 S^2]}$$

$C_{T1} = C_{T2} = 0.18$, $C_{T3} = 0.02$,

$C_{T23} = C_{T2} - C_{T3}$, $C_{T32} = C_{T2} + C_{T3}$

$$\tau_m = 0.75\frac{k}{\varepsilon}$$

$$S_{jl} = \frac{1}{2}\left[\frac{\partial u_j}{\partial x_l} + \frac{\partial u_l}{\partial x_j}\right] - \frac{1}{3}\delta_{jl}\frac{\partial u_j}{\partial x_l}$$

$$\Omega_{jl} = \frac{1}{2}\left[\frac{\partial u_j}{\partial x_l} - \frac{\partial u_l}{\partial x_j}\right]$$

5) Temperature Corrected Turbulence Model: A new approach developed by the authors of this paper to include the temperature effect in the turbulence model. We introduce in this study a temperature correction in the form of a variable C_μ which is a function of the total temperature gradient normalized by the local turbulence length scale:

$T_g = |\nabla(T_t)|\,(k^{3/2}/\varepsilon)/\,T_t$

To extend the model for high speed flow, we formulate the model to be also a function of turbulence Mach number:

$$M_\tau = \frac{\sqrt{2k}}{a}$$

where a is the local acoustic speed and k is the turbulence kinetic energy. The chosen generic form of the functional dependence of C_μ on T_g and M_τ is:

$$C_\mu = 0.09\left[1 + \frac{T_g^3}{0.041 + f(M_\tau)}\right] \qquad (5)$$

where,

$$f(M_\tau) = (M_\tau^2 - M_{\tau 0}^2)H(M_\tau - M_{\tau 0}),$$

H(x) is the Heaviside step function; and $f(M_\tau) = 0$ for no compressibility correction. We have selected $M_{\tau 0} = 0.1$ for the present model.

We have chosen to use the total temperature gradient for a number of empirical reasons:

■ The temperature correction is not coupled to Mach number effects because total temperature is not Mach number dependent. Hence, flow features that are not related to shear layer mixing, such as flow expansion, compression, and internal shocks, will not influence the turbulence model.

■ The total temperature gradient locates itself automatically in the shear layer where corrections are intended to occur.

■ The proposed turbulence model reverts back to the standard k-ε model in the boundary layer where the total temperature is constant. In general, the correction vanishes everywhere the total temperature gradient is absent in the flow.

■ This correction is well behaved over a very large temperature range in a shear flow. The smooth functional form of the correction allows for unrestricted application to shear flows with simple as well as complex configurations.

Test Cases and Comparisons

Two test cases were selected to evaluate the present modifications. The first test case is a supersonic axisymmetric jet operated at total design pressure condition. We selected this case to compare the present modification with the four other approaches in modeling the turbulence heat flux terms. This case covers a wide range of temperatures and the numerical solution will help in verifying the predicted effect of temperature on jet mixing. The second test case is a multi-stream subsonic jet configuration. This case provides a comparison in the subsonic flow and it also addresses the complexity of modeling multi-stream flows.

Supersonic Jet Flow

The present study uses the benchmark experiments performed by Seiner [2] at the NASA LaRC Jet Noise Laboratory. These experiments investigated an axisymmetric water-cooled Mach 2 nozzle with an exit diameter D = 3.60 in. (91.44mm). The simple geometry made it practical to run a larger number of numerical calculations and to more completely evaluate different turbulence models. Three sets of data were selected from the reported test results, with the nozzle plenum stagnation temperature ($T_{t,c}$) 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 x 10^6 to 8.3 x 10^6.

Table 1: Experimental nozzle flow conditions

Case	$T_{t,c}$ [°R]	$P_{t,c}$ [psi]	$T_{t,0}$ [°R]	$P_{t,0}$ [psi]	M_0
1	563	115	563	14.7	.02
2	1359	115	563	14.7	.02
3	2009	115	563	14.7	.02

The computational grid used for these nozzle calculations is generated by Dembowski and Georgiadis [15] as shown in figure 1. The computational mesh is an axisymmetric wedge shaped grid with 1 cell in the circumferential direction. The computational domain is divided into 3 blocks. Five PAB3D solutions were obtained for each of the three operating conditions as listed in Table 1, with the first using the SED, the second using Sommer-So [12], the third using the GGDH [3] model, the fourth using the Abe, et al. [14] model, and the fifth using the present temperature corrected model. All the flow simulation cases used the Sarkar [7] compressibility correction to compensate for the Mach number effect. All of these cases were run using a free stream Mach number of 0.01.

Figures 2-4 compare centerline profiles of stagnation pressure, Mach number, and stagnation temperature for the five PAB3D solutions and the experimental data of Seiner [2]. The stagnation temperature is not shown for the unheated case with nozzle plenum temperature set to 563°R. In figure 2, all the five modifications give good prediction of the stagnation pressure and Mach number as compared with experimental data for T_t=563°R. The difference in prediction between the cases is insignificant. This establishes that all the modifications reduced to the standard k-ε turbulence model. For flow conditions 2 and 3, the nozzle operated at stagnation temperatures of 1359°R and 2009°R, respectively. All the models that had changes to the heat flux term

significantly over predicted the stagnation pressure. The present temperature corrected model more accurately predicted the stagnation pressure data for both cases 2 and 3. In general, the present model gives the best comparison with experimental data for the stagnation pressure, the Mach number and the stagnation temperature distributions for all three cases. GGDH [13] also produces good predictions for the stagnation temperature profile shown in figures 3c and 4c but fails to accurately predict stagnation pressure and Mach number.

Multi-stream Subsonic Jet Flow

The second test configuration includes a separate fan and core nozzle flows at a bypass ratio of five with an external plug. One set of data was selected from the reported test results, with the flow condition as indicated in Table 2 for core, fan and free stream.

Table 2: Experimental Subsonic Condition

	T_t [°R]	P_t [psi]	M
Core	1498	21.72	------
Fan	647	24.36	------
Free Stream	530	14.7	0.28

This test configuration is part of a comprehensive investigation for jet exhaust noise due to the pylon-jet interaction similar to the configuration shown in figure 5. This nozzle was tested at NASA LaRC during the Advanced Subsonic Technology (AST) program in parallel with a similar test conducted at NASA Glenn (see figure 6). The computational domain for the solution extended from x/D_c = -6.3 to x/D_c = 31.6 in the axial direction and $6.3D_c$ in the radial direction, where D_c is the diameter of the baseline core nozzle, 12.80 cm. The origin, x/D_c = 0.0 was set at the exit of the fan nozzle so that the exit of the core nozzle is at about x/D_c = 0.5. The computational grid used in the present study is shown in Figure 7. The computational mesh is an axisymmetric wedge shaped grid with 2 cells in the circumferential direction, each cell being 3°. The computational domain is divided into 12 blocks where block one is the core nozzle, block two is the fan nozzle, and the remaining 10 blocks representing the free stream. The mesh has a total of 296928 cells. Grid points are clustered near the solid surfaces and around the shear layer. The value of y^+ for the first cell off the surface varied between 0.16 and 1.8

Computational solutions were obtained for standard k-ε model using SED, and the temperature corrected model. The simulated conditions were set to coordinate with the data presented in ref [3]. The computational results were conducted for a free stream Mach number of 0.28.

Comparison between computed stagnation temperature, using SED and present model, and experimental data is shown in figures 8 and 9. As shown in figure 9, the present model was able to more accurately predict the temperature flow field and match the experimental data while SED approach over predicted the temperature in the core region. A comparison between computed center line stagnation temperature and the experimental data is shown in figure 10. The SED approach was not able to match the experimental data accurately while the present model was within 1% of the experimental value for the first point and 4% for the second experimental point.

A grid sensitivity study was conducted to determine the effect of grid resolution on the accuracy of the present model solution. The flow field for the dual subsonic jet flow was computed for three grid levels; coarse (1/4 grid resolution), medium (1/2 grid resolution) and fine grid. The residual convergence history for all grid levels is shown in figure 11. The computing strategy was to first run the problem on a coarse grid and once convergence is achieved to interpolate the solution to the next grid level and run the solution on the finer grid level. The centerline stagnation temperature profiles for all grid levels are shown in figure 12. The temperature profiles essentially do not vary between grid levels and compare well with experimental. Further grid sensitivity analyses were not conducted since the solution was not sensitive to grid level. A comprehensive investigation of the dual subsonic jet flow is presented by the authors in reference [16 & 17]

Concluding Remarks

The motivation of this work is to provide more accurate aerodynamic and aero-acoustic predictions for 3D jet flows. In this paper, different modifications to the k-ε model for the simulation of high speed high temperature jet flows are presented and compared with experimental data. The present modification to the turbulent viscosity holds great promise in predicting high temperature jet flows while showing good predictions for the low temperature jet flows. The present model

accurately simulated different jet flow conditions which confirm the accuracy and robustness of the model.

Acknowledgment

This work is part of a larger ongoing study of complex nozzle configurations. A jet noise prediction method based on the computational information is also being pursued together with the computational results and the experiments. The third and fourth authors would like to acknowledge the support of NASA Langley Research Center for providing the funding needed to carry out this work

.

References

1- PAB3D Code Manual Originally developed by the Propulsion Aerodynamics Branch, now under cooperative program between the Configuration Aerodynamics Branch, NASA Langley Research Center and Analytical Services & Materials, Inc. Hampton, VA. See http://www.asm-usa.com/software/pab3d.html.

2- 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.

3- Thomas, R.H., Kinzie, K.W. and Pao, S. Paul, "Computational Analysis of a Pylon-Chevron Core Nozzle Interaction," AIAA Paper 2001-2185, May 2001.

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

5- 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.

6- Jones,W. P.; and Launder, B. E., "The Prediction of Laminarization With a Two-Equation Model of Turbulence," Int. J. Heat & Mass Transf., vol. 15, no. 2, Feb. 1972, pp. 301-314.

7- Sarkar, S.; Erlebacher, G.; Hussaini, M. Y.; and Kreiss, H. O., "The Analysis and Modeling of Dilatational Terms in Compressible Turbulence," J. Fluid Mech., vol. 227, June 1991, pp. 473-495.

8- Wilcox, David C., "Progress in Hypersonic Turbulence Modeling." AIAA 91-1785, June 1991. vol. 109, June 1987, pp. 156-160.

9- Theis, A. T., and Tam, C. K. W., "Computation of Turbulent Axisymmetric and Non-axisymmetric Jet Flows Using the K-ε Model," AIAA Journal, Vol. 34, No. 2, February 1996, pp. 309-316.

10- Tam, C. K.W., and Ganesan, A., "A Modified k-ε Model for Calculating the Mean Flow and Noise of Hot Jets," AIAA 2003-1064, January 2003.

11- Lebedev, A. B., Lyubimov, A. D., V.P.Maslov, V. P., Mineev, B. I., Secundov, A. N., and Birch, Stanley F., " The Prediction of Three-Dimensional Jet Flows for Noise Applications," AIAA-2002-2422, 2002.

12- So, R. M. C., and Sommer, T. P., "An Explicit Algebraic Heat-Flux Model for the Temperature Field," International Journal of Heat and Flow, 1995, Vol. 7, pp. 455-465.

13- Ronki, M., and Gatski, Thomas B., "Predicting Turbulent Convective Heat Transfer in Fully Developed Duct Flows," International Journal of Heat and Flow, Vol. 22, 2001, pp. 381-392.

14- Abe, K., Kondoh, T., and Nagano, Y., "A Two-equation Heat Transfer Model Reflecting Second-Moment Closures for Wall and Free Turbulent Flows," International Journal of Heat and Flow, Vol. 17, 1996, pp. 228-237.

15- Dembowski, M. A. and Georgiadis, N.J., "An Evaluation of Parameters Influencing Jet Mixing Using the WIND Navier-Stokes Code," NASA TM 2002-211727, 2002.

16- Massey, S.J., Thomas, R.H., Abdol-Hamid, K.S and Elmiligui, A., "Computational and Experimental Flowfield Analyses of Separate Flow Chevron Nozzles and Pylon Interaction," AIAA-2003-3212, 2003.

17- Hunter, C., Pao, S. and Thomas, R., "Development of a Jet Noise Prediction Method for Installed Jet Configurations," AIAA 2003-3169, 2003.

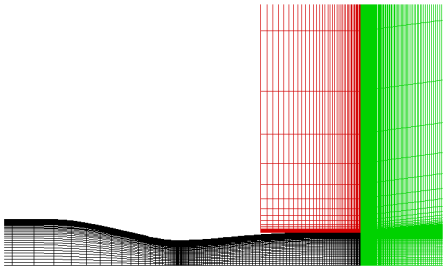
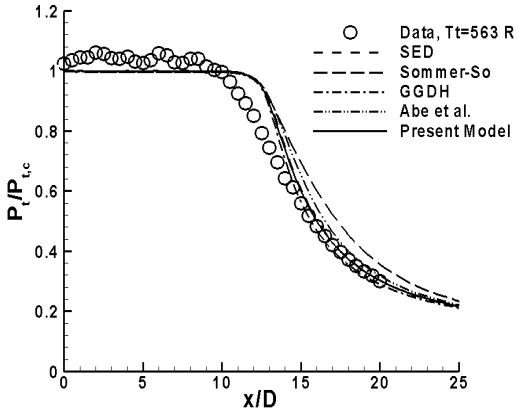
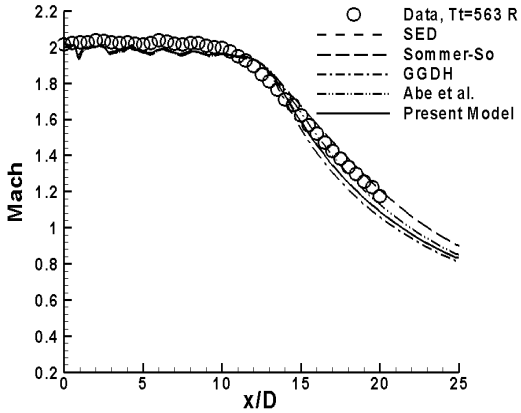


Figure 1: Computational grid for the Supersonic Jet Flow

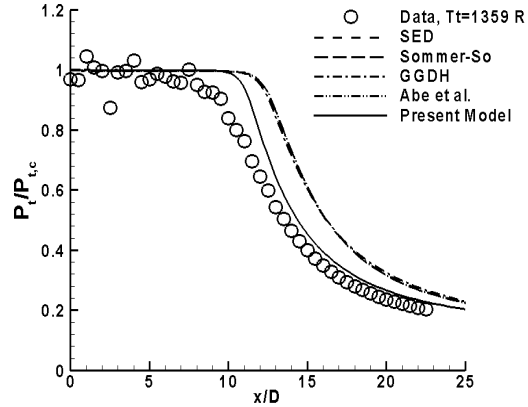


a) Centerline Stagnation Pressure

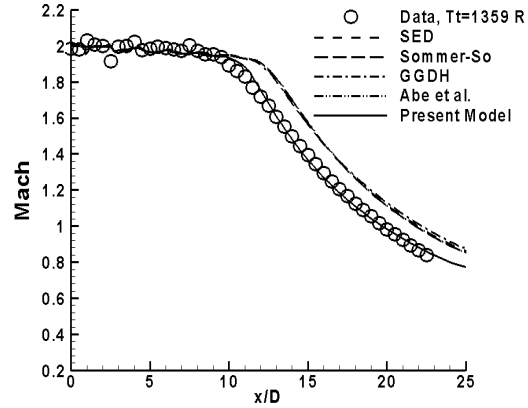


b) Centerline Mach Number

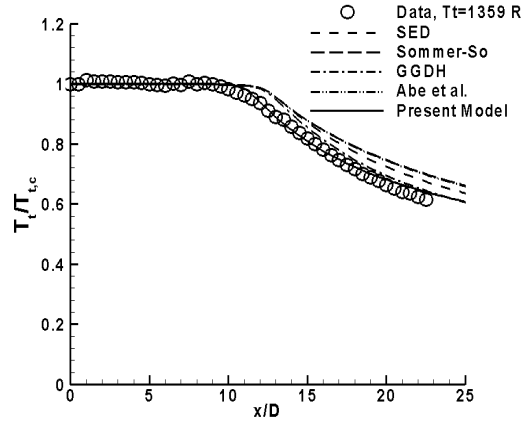
Figure 2: Comparison of solutions at T_t = 563.67 R using different turbulence models with Experimental Data [2]



a) Centerline Stagnation Pressure

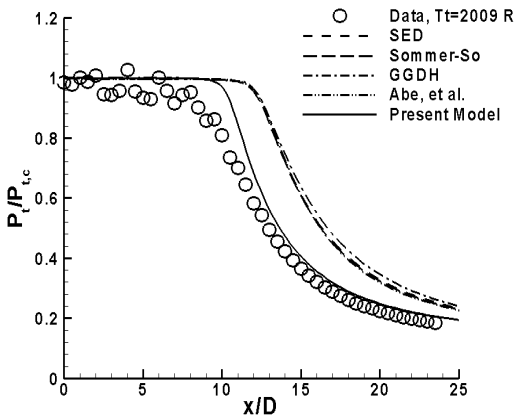


b) Centerline Mach Number

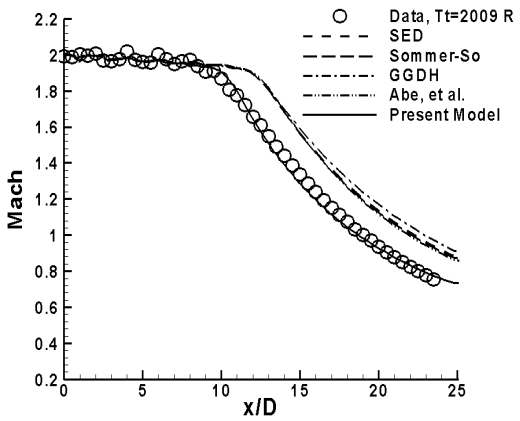


c) Centerline Stagnation Temperature

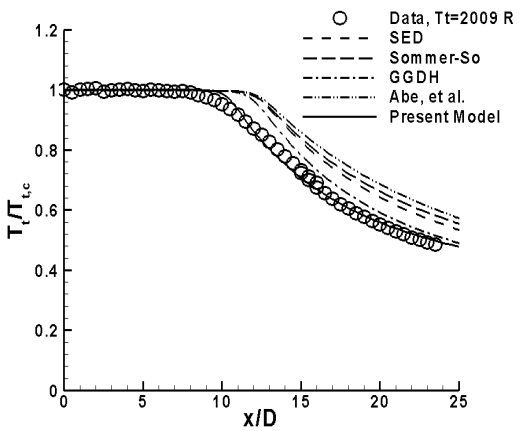
Figure 3: Comparison of solutions at T_t = 1359.67 R using different turbulence models with Experimental Data [2]



a) Centerline Stagnation Pressure



b) Centerline Mach Number



c) Centerline Stagnation Temperature

Figure 4: Comparison of solutions at T_t = 2009.67 R using different turbulence models with Experimental Data [2]

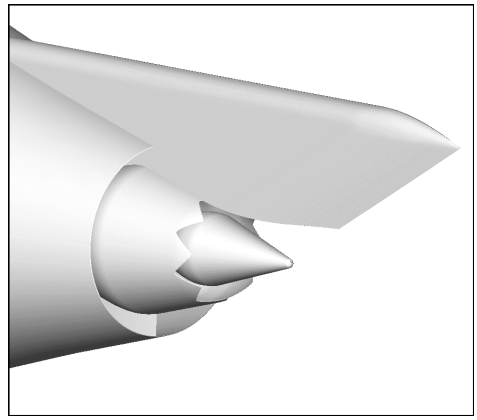


Figure 5: Configuration 4, 8-chevron core nozzle with round fan nozzle and pylon.

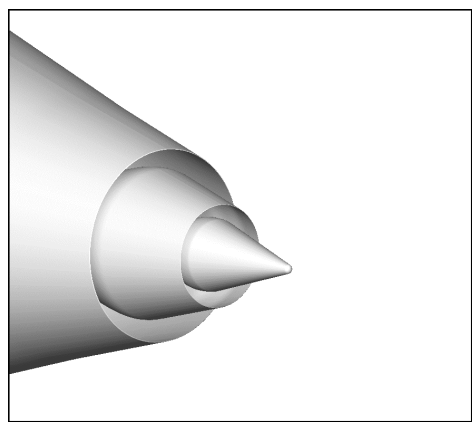


Figure 6: Configuration 1, baseline round core nozzle with round fan nozzle.

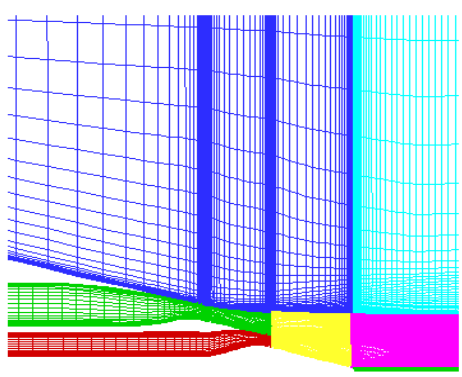
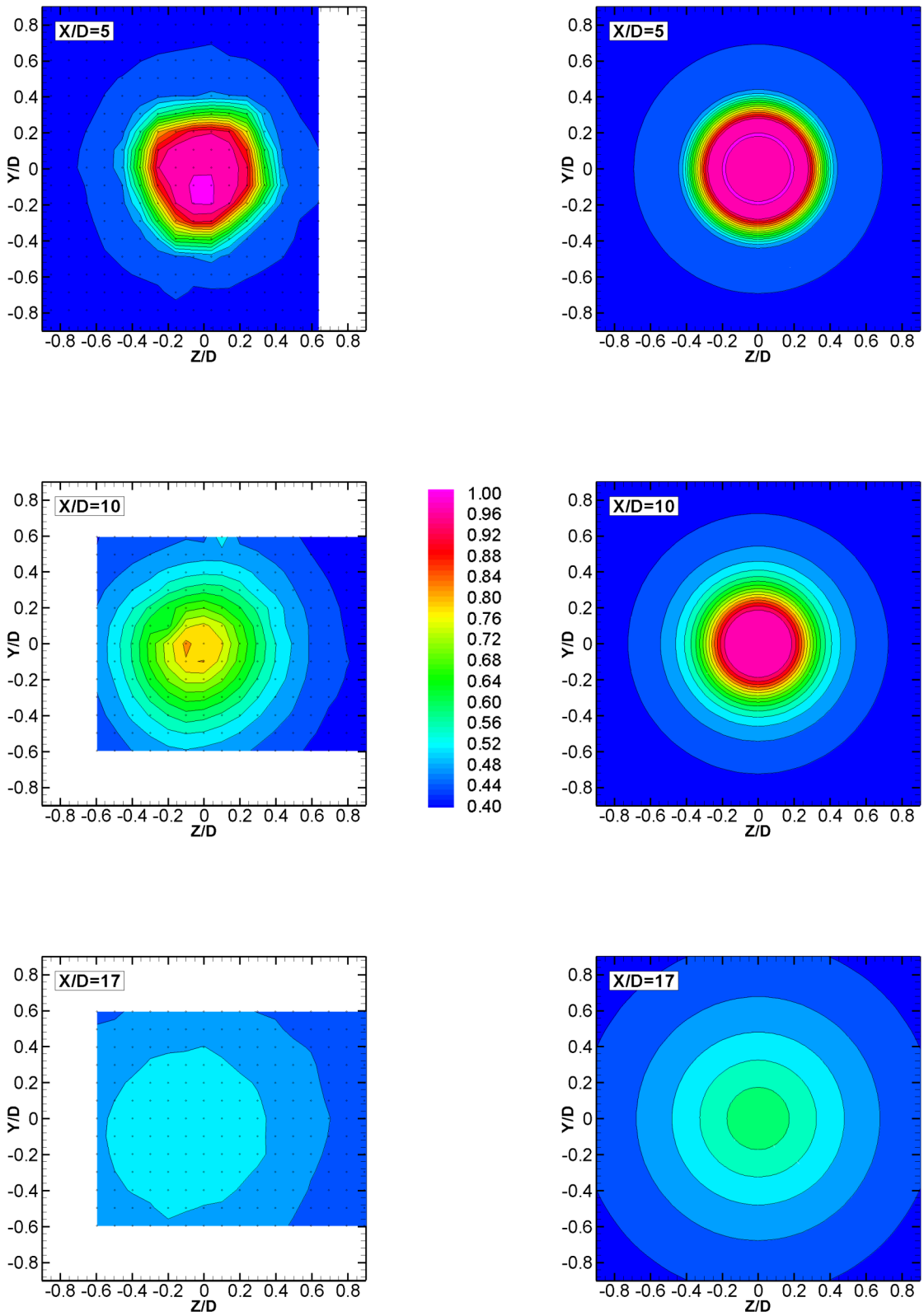
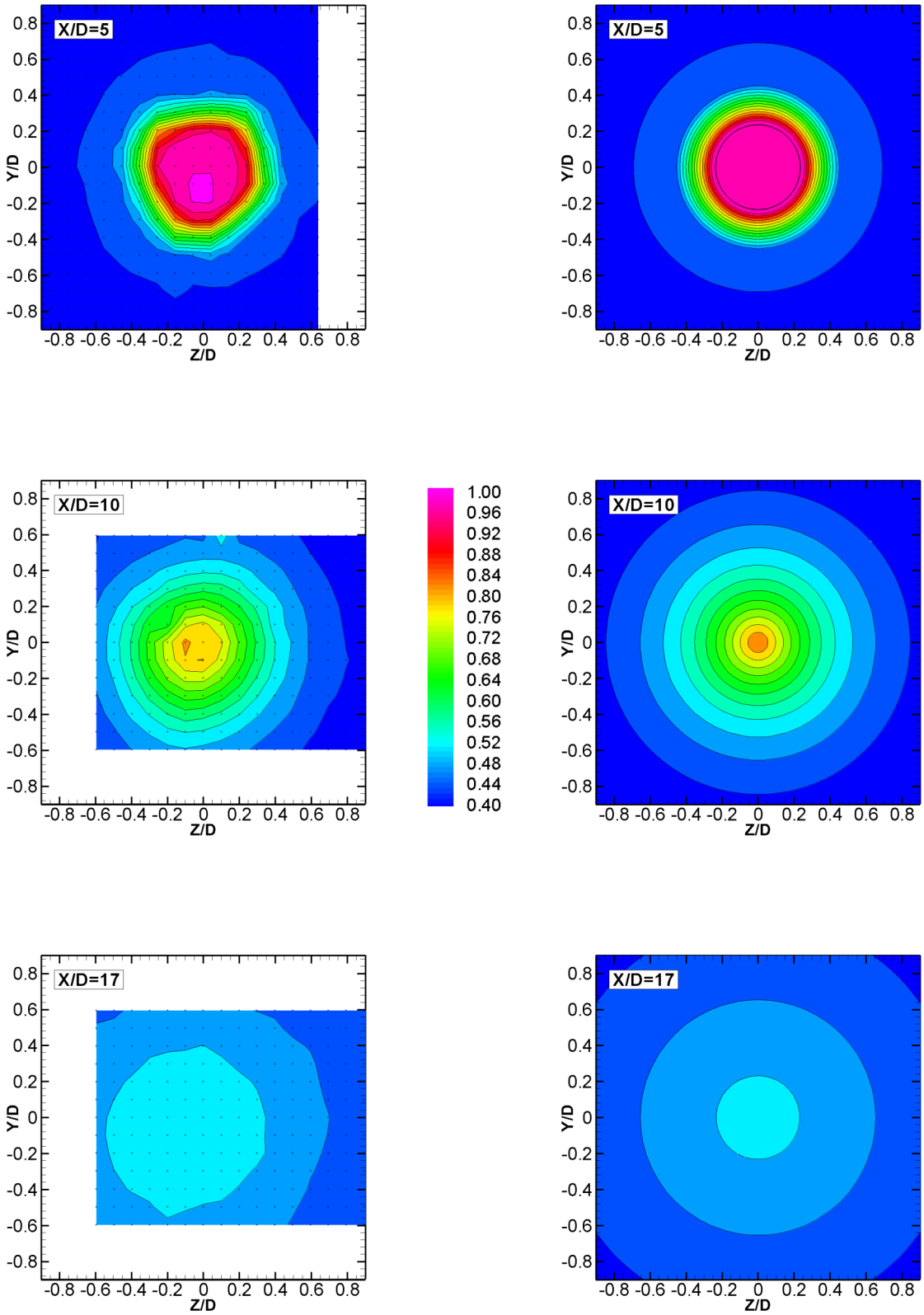


Figure 7: Computational Domain for the Muti-stream Jet Flow



a) Data [16] b) SED

Figure 8: Comparison of the Stagnation Temperature Prediction with Experimental Data [16]



a) Data [16] b) Present Model

Figure 8: Comparison of the Stagnation Temperature Prediction with Experimental Data [16]

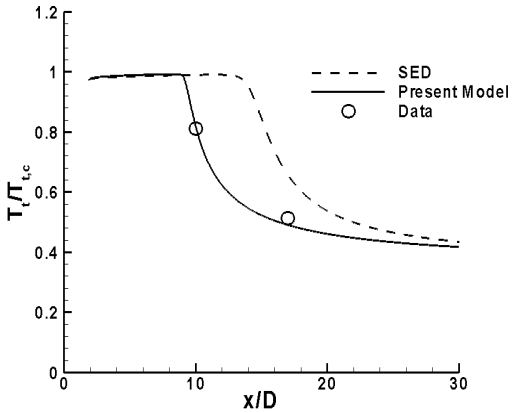


Figure 10: Comparison between Computed Center Line Stagnation Temperature and Experimental data [16].

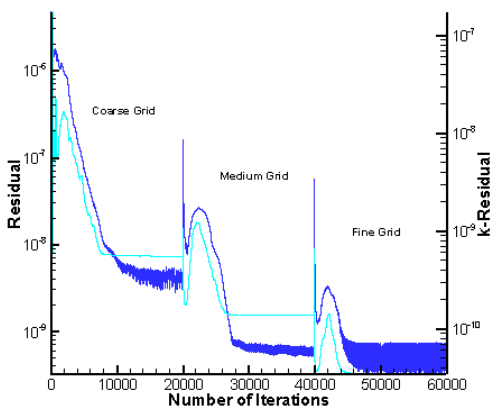


Figure 11: Convergence History for Different Grid Levels using Present Model

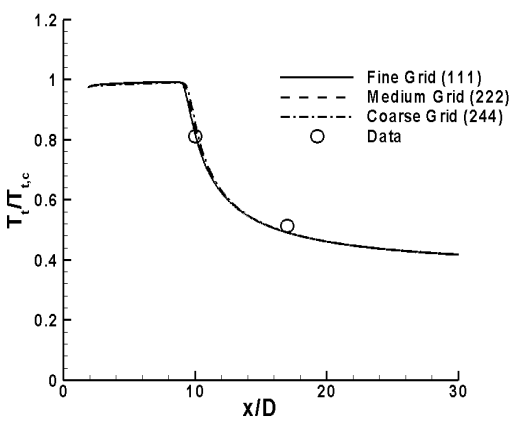


Figure 12: Comparison between Computed Results for Coarse, Medium and Fine Grid Level and Experiment Data [16].