Advanced Materials Research Vols. 984-985 (2014) pp 1204-1209 Submitted: 2014-04-23
© (2014) Trans Tech Publications, Switzerland Accepted: 2014-04-27
doi:10.4028/www.scientific.net/AMR.984-985.1204 Online: 2014-07-16

Comparative studies on various turbulent models with liquid rocket nozzle through computational tool

N.K.Mohamad Tanveer [1], C.Mohanraj[2*], K.Jegadeesan[1], S.Maruthupandiyan[1]

Student, PGP College of Engineering and Technology-Namakkal, India [1]

Assistant professor, PGP College of Engineering and Technology-Namakkal, India [1]

nktanveer@gmail.com[1,] mohanrajc@karunya.edu.in[2*],

Keywords: Supersonic flow, computational fluid dynamics, turbulence model

Abstract - Supersonic flows associated with missiles, aircraft, missile engine intake and rocket nozzles are often steady**.** In this present work, the computational analysis was conducted on C-D (convergent –divergent) nozzle for understanding the flow regime with various flow properties such as velocity and various turbulent models (spalert almaras, K-ε and K-ω). The Scale down model of C-D nozzle was chosen for this study and it was modelled computationally with Gambit software package. In this integrated component model, the inlet flow is assumed a two-dimensional, steady, compressible, turbulent and supersonic. The physics based mathematical model of the considered flow consists of conservation of mass, momentum and energy equations subject to appropriate boundary conditions as defined by the physical problem stated above. The system of the governing equations with turbulent effects is solved numerically using different turbulence models to demonstrate their numerical accuracy in predicting the characteristics of turbulent gas flow in such complex geometry. Fluent software package was used for solving gas flow equations with turbulence models. The Mach number was chosen for different cases of analyses were 1.2, 1.5 and 2. For each case, different turbulence were engaged and solved and all the results were compared finally.

Introduction

Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation. Its scope is not limited to fluid dynamics; CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can make use of various methods like the analytical method, experimental methods using prototypes. The analytical method is very complicated and difficult. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionised the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem is mathematically solved with computer assistance. Thus we would be able to predict the results of a problem before experimentation. Some of the earlier investigation related with our current work are: [1] P. Padmanathan et al, computational analysis was carried out for shock waves on C-D nozzle. It was proven computational that increase in the static pressure, density and the static temperature across the shock. Mach number decrease across the shock. [2] Ms.B.Krishna Prafulla et al, numerical studies were conducted for C-D nozzle. The results from computational analysis of flow properties had good agreement with experimental data from *Naval Science and Technological Laboratory, Visakhapatnam. [3]* Sibendu Soma et al, studies were carried out for Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions. The flame structure and stabilization are also noticeably influenced by orifice geometry. The flame lift-off lengths are the highest and lowest for the hydroground and conical nozzles, respectively the amount of soot produced is highest with a conical nozzle, while the amount of NOx produced is the highest with a hydroground nozzle.[4] Seyed Ehsan Rafiee et al, energy separation techniques were tested and achieved through using of C-D nozzle. The parameters are focused on the convergence ratio of nozzle, inlet pressure and number of nozzle intakes. The effect of the convergence ratio of nozzle is investigated in the range of 1–2.85.The experimental and simulated results were compared. [5] A. Balabel et al, turbulent models were assessed on 2 dimensional C-D rocket nozzles. From the results, the shear-stress transport (SST) k–ω model exhibits the best overall agreement with the experimental measurements.[6] K. Pougatch et al, compressible gas-liquid phase was modelled over C-D nozzle. It reveals that virtual mass force plays a major role in accelerating/decelerating flows with a relatively low interfacial drag. [7] Wen-Ya Li et al, optimization was done for design of convergent-barrel cold spray nozzle using numerical method. Particles can achieve a relatively low velocity but a high temperature under the same gas pressure using a CB nozzle compared to a convergent–divergent (CD) nozzle.

Mathematical modelling

The flow from a circular nozzle is governed by the steady state axisymmetric form of the fluid flow conservation equations. For variable density flows, the Favre averaged Navier-Stokes equations are more suitable and will be used in this work. No assumptions are made on the Navier-Stokes equation (parabolized) except for turbulence modelling. The total energy equation including viscous dissipation is also included and coupled to

Set with the perfect gas law. The thermodynamics and transport properties for air are held constant; theri influence was found to be not significant along the validation runs [Y. Bartosiewicz].

Continuity equation

$$\partial p/\partial t + \nabla.(\rho V)=0 \quad (1)$$

Momentum Equation

$$\partial (\rho u)/\partial t + \nabla.(\rho u V) = -\partial p/\partial x + \rho fx \quad (2)$$

$$\partial (\rho v)/\partial t + \nabla.(\rho v V) = -\partial p/\partial y + \rho f \quad (3)$$

$$\partial (\rho w)/\partial t + \nabla.(\rho w V) = -\partial p/\partial z + \rho fz \quad (4)$$

Energy Equation

$$\partial/\partial t\,[\rho(e + V2/2)] + \nabla.[\rho(e + V2/2)\, V] = \rho q - \partial/\partial x(up) - \partial/\partial y(vp) - \partial/\partial z(wp) + \rho f.V \quad (5)$$

Turbulence modelling

Spalart-Allmaras Model: Being a one equation model, the Spalart-Allmaras model solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has given good results for boundary layers subjected to adverse pressure gradients. The turbulent dynamic viscosity is computed from

$$\mu t = \rho v\, f v1 \quad (6)$$

K-$^{\varepsilon}$ Model

The standard K-ε model was considered for this simulation. It is simplest 2-equation model and suited for the well bounded cases. This turbulence model based on turbulent kinetic energy k and dissipation rate ε. The turbulent kinetic energy and dissipation rate has been obtained from following equations [8],

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k U) = div\left[\frac{\mu_t}{\sigma_k}\, grad\, k\right] + 2\mu_t E_{ij} - \rho\varepsilon \quad (7)$$

And

$$\frac{\partial(\rho\varepsilon)}{\partial t}+div(\rho\varepsilon U)=div\left[\frac{\mu_t}{\sigma_\varepsilon}grad\,\varepsilon\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}\,.\,E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (8)$$

Where the eddy viscosity $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$; C_μ is dimensionless constant
The values for the adjustable constants
$C_\mu = 0.09$; $\sigma_k = 1.00$; $\sigma_\varepsilon = 1.30$; $C_{1\varepsilon} = 1.44$; $C_{2\varepsilon} = 1.92$

Geometrical modelling

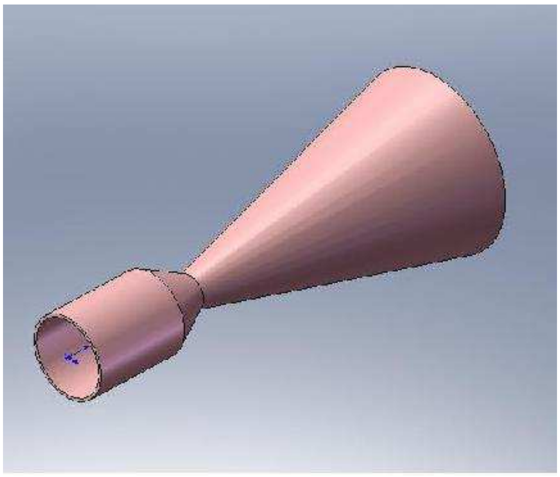

Fig 1, 3-Dimensional model of C-D Nozzle

The 3-d model was generated from solid works software package. The dimension of C-D nozzle was taken from Liquid propulsion system centre (division of ISRO Thiruvanadapuram). It is a scale down model and was used for testing with air- kerosene which has a maximum working pressure of 10 bar and therefore the secondary injection module was suitably altered in order to bypass the effects of flow separation. The secondary injection was initiated well before flow separation occurs.

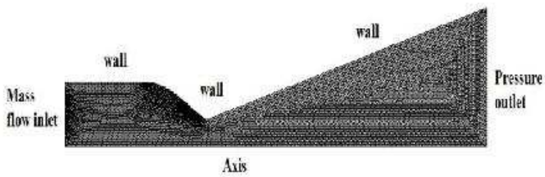

Fig 2, Computational Model of C-D nozzle

The 2- dimensional axisymmetric model was generated and meshed in Gambit pre-processor. Triangular elements and pave method of meshing was used for discretization. Totally 16616 elements and 8616 nodes were involved in discretization. Boundary types were mentioned in gambit pre-processor itself as shown in fig 3. For the purpose of decreasing solving time and cost, axi-symmetric model was given for this analysis.

A mathematical model comprises equations relating the dependent and the independent variables and the relevant parameters that describe some physical phenomenon. Typically, a mathematical model consists of differential equations that govern the behaviour of the physical system, and the associated boundary conditions. To start with fluent, it is necessary to know if the meshed geometry is correct, so is checked. To ensue with, we are to define the model, material, operating condition and boundary condition. Models are to be set in order to define if any energy equation is dealt with our study, if the flow is viscous.

Result and Discussions

This scaled down model of the Convergent and divergent nozzle is to be fabricated and will be tested with different operating parameters. In prior to that, computational analysis was done for different Mach number and various turbulence model. The mach number chosen for this analysis were 1.2, 1.5 and 2. Also turbulence model were one equation- spalart Allmaras, two equation model of K-epsilon and two equation model of K-omega. The flowing table-1 can describe the cases which were analysed.

TABLE -1
Various cases for analysis

Case No	Mach Number	Turbulence model
1	1.2	Spalart-Allmaras
2		K-Epsilon (K-ε)
3		K-Omega (K-ω)
4	1.5	Spalart-Allmaras
5		K-Epsilon (K-ε)
6		K-Omega (K-ω)
7	2	Spalart-Allmaras
8		K-Epsilon (K-ε)
9		K-Omega (K-ω)

The following contours are describing that, the turbulent kinetic energy distribution throughout nozzle at Mach number 2.

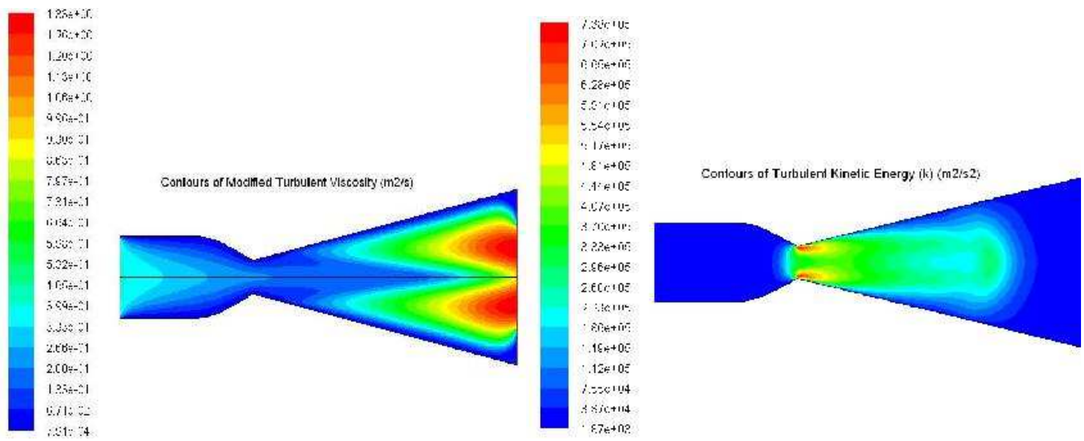

Fig 3, Turbulence kinetic energy contours for spalart-Allamaras model and K-Epsilon model (Mach no-2)

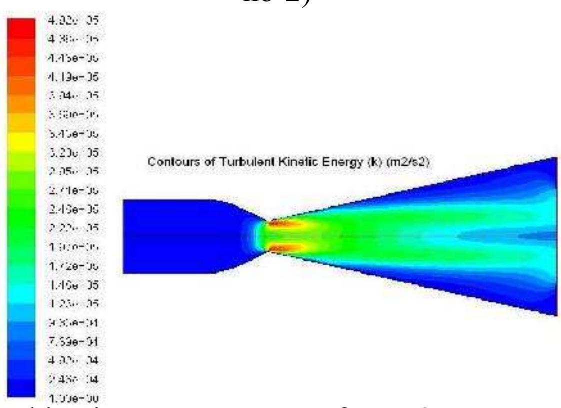

Fig 4, Turbulence kinetic energy contours for K-Omega model (Mach no-2)

From above figures, the maximum magnitude of kinetic energy was obtained from k-epsilon turbulence model than it goes with K-Omega and spalart allamaras. But Maximum distribution of turbulent kinetic energy was observed at throat section for both K-epsilon and K-omega. But it was observed at tail section of C-D nozzle. The frequency of energy vibration was higher with spalart allmaras than other two models.

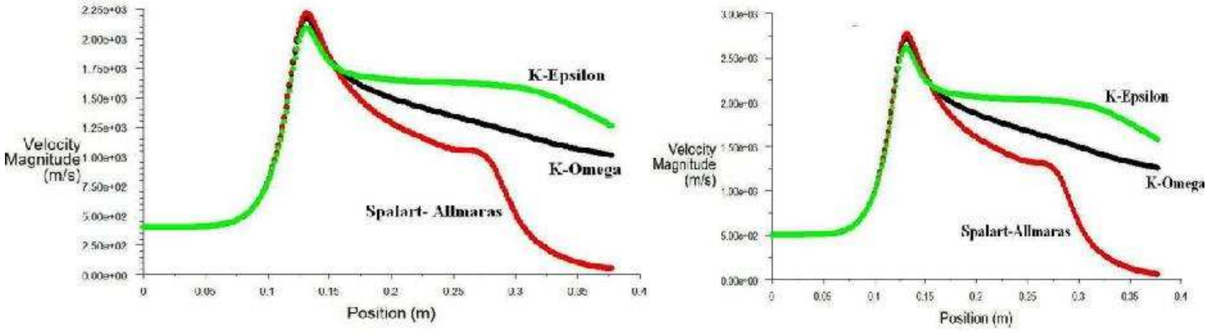

Fig 5, Velocity distribution plot at Mach no-1.2 and 1.5

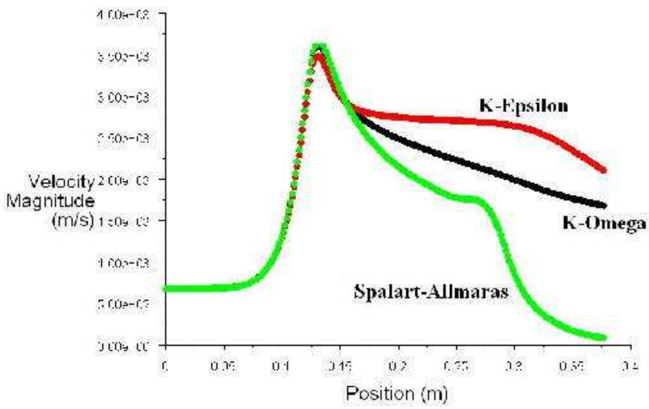

Fig 6, Velocity distribution plot at Mach no-2

Above all the plots are describing the velocity distribution of C-D nozzle for various Mach number and different turbulence model. All the distribution was quite similar to each other with different Mach number. The differences are very small. It reveals that, not much relation between Mach number and turbulence model. Variation was found between various turbulence models at same Mach number. K-epsilon and K-omega were had similar distribution except small deviation at tail of C-D nozzle. But Spalart allmaras model had immense digression form other two models. It might of large turbulence at tail section of C-D nozzle.

Conclusion

Near the wall, the Mach number is decreasing for all the nozzles. This is due to the viscosity and turbulence in the fluid. Flow separation and formation of a vortex after the shock can be attributed to adverse pressure gradient following shocks. The turbulence intensity was high at tail of C-D nozzle in sparart allmaras model. But other two models had more intensity at throat. Flow was separated at near to throat (ahead of throat) in K-epsilon and K-omega models. But the flow was attached to the more length of divergent wall in spalart allmaras model. The standard K- epsilon turbulence model provided the accurate results as compared to Spalart-Allmaras model and K-omega for same set of conditions and discretization schemes.

Reference

[1] P. Padmanathan, Dr. S. Vaidya Nathan, "Computational Analysis of Shockwave in Convergent Divergent Nozzle" International Journal of Engineering Research and Applications, Vol. 2, Issue 2,Mar-Apr 2012, pp.1597-1605

[2] Ms.B.Krishna Prafulla, Dr. V. Chitti Babu *2* and Sri P. Govinda Rao, "Cfd Analysis of Convergent- Divergent Supersonic Nozzle" International Journal of Computational Engineering Research, Vol, 03,Issue 5.

[3] Sibendu Soma, Anita I. Ramirez , Douglas E. Longman , Suresh K. Aggarwal, "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions" journal homepage: www.elsevier.com/locate/fuel

[4] Seyed Ehsan Rafiee, Masoud Rahimi, "Experimental study and three-dimensional (3D) computational fluid dynamics(CFD analysis) on the effect of the convergence ratio, pressure inlet and number of nozzle intake on vortex tube performance Validation and CFD optimization" Energy, Volume 63, 15 December2013, Pages195-204

[5] A. Balabel, A.M. Hegab, M. Nasr, Samy M. El-Behery, "Assessment of turbulence modeling for gas flow in two-dimensional convergent divergent rocket nozzle"Applied Mathematical Modelling, Volume35,Issue7, July2011, Pages3408-3422

[6] K. Pougatch, M. Salcudean, E. Chan, B. Knapper, "Modelling of compressible gas–liquid flow in a convergent divergent nozzle" Chemical Engineering Science, Volume 63, Issue 16, August 2008, Pages 4176-4188

[7] Y. Bartosiewicz, Zine Aidoun, "CFD-Experiments Integration in the Evaluation of Six Turbulence Models for Supersonic Ejectors Modeling" CETC-Varennes, Natural Ressources Canada, ybartosi@RNCan.gc.ca

[8]C.Mohanraj, Dr.P.K.Srividhya, "CFD Simulation of 20 kW Downdraft Gasifier"
International Journal of Current Engineering and Technology ISSN 2277 - 4106, Vol.3, No.1 (March 2013)

[9] K.M. Pandey, Virendra Kumar, Prateek Srivastava, "CFD Analysis of Twin Jet Supersonic Flow with Fluent Software" Current Trends in Technology and Sciences Volume 1, Issu 2 , (Sept.-2012)

[10] Biju Kuttan P, M Sajesh, "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes), Volume2, Issue 2,Pages196-207,2013

[11] Nadeem Akbar Najar, D Dandotiya, Farooq Ahmad Najar, "Comparative Analysis of K-ε and Spalart-Allmaras Turbulence Models for Compressible Flow through a Convergent-Divergent Nozzle" The International Journal Of Engineering And Science (IJES), Volume2, Issue8, Pages08-17, 2013

[12] Pardhasaradhi Natta, V.Ranjith Kumar, Dr.Y.V.Hanumantha Rao, "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd)" Engineering Research and Applications (IJERA) ISSN 2248-9622 ,Vol. 2, Issue 5, September- October 2012, pp.1226-1235

Modern Achievements and Developments in Manufacturing and Industry
10.4028/www.scientific.net/AMR.984-985

Comparative Studies on Various Turbulent Models with Liquid Rocket Nozzle through Computational Tool
10.4028/www.scientific.net/AMR.984-985.1204