Modelling 2-D Supersonic Jet from a Convergent-Divergent Nozzle using k-ε Realizable Turbulence Model

Bholu Kumar¹ and Shantanu Srivastava¹

¹Department of Mechanical Engineering, National Institute of Technology, Patna – 800006, INDIA
E-mail: bholumech@gmail.com

Abstract: The present study focuses on the numerical investigation of the Mach 1.86 supersonic jet from the 2-D convergent-divergent nozzle for NPR (nozzle pressure ratio) values of 5 to 8 with a step size of one, covering the overexpansion, correct expansion and the underexpansion conditions using Realizable k-ε turbulence model under the steady state condition. The computational model is constructed using the design modeler of ANSYS 16.0 workbench. The structured grids are generated into the ICEM module of the ANSYS workbench and the fluid flow analysis is done by FLUENT solver. The plot of non-dimensional total pressure along the jet centerline with respect to the non-dimensional downstream distance of the nozzle exit along the jet centerline identify the extent of jet mixing. The contours of Mach number with different NPRs visualize the effects of shock cells and shock strength prevail into the jet flow field due the favorable and adverse pressure gradients at the nozzle exit.

1. Introduction

The study of supersonic jet has gained importance in the recent past because of its applicability in the wide variety of engineering applications beginning from household appliances to high tech rockets. In terms of academic interest, studies on jets have provided insight into the understanding of the dynamics of free shear layers and vortical structures. The jet is a pressure driven free shear flow which shows the characteristics that its local width to the local axial distance bears a constant value, which is approximately 8 for jet Mach numbers less than 0.2. This ratio decreases with the increasing Mach numbers. The jet may also be defined as the fluid issuing from a nozzle into quiescent surrounding. If the jet is issuing into the ambient surround fluid of zero velocity, then it is called as submerged jet.

It has been established in the literature that for an early and rapid jet mixing there should be appropriate ratio of the large and small scale vortices present in the jet field. However, finding out this appropriate ratio is almost not possible, especially for a turbulent flow field like supersonic free jet. Hence, an indirect method which might be used for the quantification of the jet mixing is the rapid decay of the jet centerline velocity or the core length of the jet. This demonstrates that a jet with shorter core length will comparatively have better and enhanced mixing than an identical jet with a longer core length [3,4,5].
Due to the complexity of flow, jets are mostly studied experimentally. These complexities arise due to the entrainment of the jets, large perturbations present at the high Reynolds number and strong wave interactions in jets exiting at supersonic Mach numbers from typical convergent-divergent nozzles. However, the computational studies on the jet mixing characteristics of supersonic jets are investigated by some researchers. An account of computational investigations on the supersonic jet is the following.

Launder and Spalding [6] investigated the efficacy of k-\(\varepsilon\) turbulence model for the jet simulation from a convergent-divergent nozzle. It was found that, the two equation k-epsilon model well captured the turbulence in the free-shear flow without adjustments of constants and functions. However, the model can only accurately predict the features for low Reynolds number flows. Dash et. al. [7] upgraded the existing k-\(\varepsilon\) turbulence model by introducing the effect of vortex-stretching and compressible-dissipation extensions in order to put the results in agreement with the existing fluid dynamic data. Tam and Thies [8] investigated the k-\(\varepsilon\) turbulence model for supersonic jets. However, the investigation and results were limited to the simulation of velocity profiles and the Mach number profile, downstream of the nozzle exit. Evgenevna et. al. [9] evaluated the prediction capabilities of various two-parameter differential turbulence models. The work was confined to only correctly expanded supersonic jets and it was investigated that, k-\(\varepsilon\) realizable and transition SST turbulence models, showed the best results.

So far, it has been demonstrated that the k-\(\varepsilon\) realizable turbulence model and the transition SST turbulence model show promising results for the simulation of free jets from convergent-divergent nozzles. However, the computational cost and resources requirement involved for more sophisticated turbulence models were exorbitantly high. Using Area-Mach number relation [1], the two-dimensional convergent-divergent nozzle is designed to generate a shock-free Mach number of 1.86 at the nozzle exit with suitable nozzle pressure ratio (NPR) in the present investigation. The nozzle is designed to generate Mach 1.86 jet only due to the established experimental results of the Mach 1.86 jet, in order to validate it and then for further studies of controlled jet using computational analysis.

The present investigation aims at simulating the mixing extent of the Mach 1.86 jet from the convergent-divergent nozzle, at all the levels of expansion with Realizable k-\(\varepsilon\) turbulence model under steady state conditions, through a commercial software package without requiring High Performance Computation Facility. The study aspires to develop a computationally economic method for the simulation of simplified problem of high speed flows without the requirement of sophisticated computational facility. However, care is taken so as to make the accuracy of the computations within acceptable limits, close to the established experimental results. The comparison of the results is achieved with the experimental results of Shantanu and Rathakrishnan [10]. In the present work, a computational study is performed with the 2-D axisymmetric uncontrolled jets, (3-D being computationally costly). The NPR of the jet is varied from 5 to 8 with a step size of one. The mixing characteristics of overexpanded, correctly expanded and underexpanded uncontrolled supersonic jet is studied by plotting the non-dimensional total pressure with respect to the non-dimensional distance downstream of the nozzle exit along the jet centerline. In addition to this, Mach contours are also plotted at different NPRs for the qualitative aspects of the study and visualization of other jet features such as the barrel shock, the compression waves and the expansion fans.

2. Methodology

In the present study, an axisymmetric two dimensional computational model is constructed in ANSYS software of version 16.0. The domain extends to about 30D from the nozzle exit, along the jet centerline and 15D along the transverse direction to the jet axis, where D is the diameter of the nozzle exit plane geometry. The grid is generated using the ICEM module of the ANSYS workbench 16.0 and the flow of the jet, in two dimensions, is analyzed in FLUENT.
2.1. Numerical Domain

The sketch of computational domain for the axisymmetric model, is as shown in Figure 1. The convergent-divergent nozzle is constructed with convergent angle of $15^\circ$ and divergent angle of $7^\circ$. The diameter ($D$) of the nozzle exit is found to be 12.28 mm with reference to the nozzle throat diameter of 10 mm which is evaluated from the Area-Mach number relationship, for Mach 1.86 jet [1]. The computational domain extends to about 30D along the axis of jet and to about 15D in the perpendicular direction to the jet axis.

![Figure 1. Cartoon of the computational domain](image)

2.2. Boundary conditions

2.2.1. Nozzle Inlet (B1): The pressure inlet boundary condition is applied to the nozzle inlet boundary (B1) by varying the NPR from 5 to 8. A constant temperature of 300 K is prescribed.

2.2.2. Nozzle walls (B2 & B3): The nozzle walls B2 and B3 are specified by the wall boundary condition. The walls of nozzle are assumed to be adiabatic with no slip condition.

2.2.3. Pressure Far-field (B4, B5, and B6): The pressure far-field boundary condition is specified for the boundaries B4, B5, and B6, of the flow domain.

2.2.4. Pressure Outlet: The pressure outlet boundary condition is specified for the domain boundary B7.

2.2.5. Axis: In order to make computationally economical numerical investigation, the computational domain is made axisymmetric and thus the axis boundary condition is specified to the axis of the jet which is shown in Figure 1.

2.3. Mesh generation

The 2-D quadrilateral structured grids were generated, as shown in Figure 2, with grid sizes of 56016, 87237, 114274, 126036, 154680, 223581, and 349117. The variation of results for the above grid sizes has been done which is shown in the Figure 3. The Mach number of the jet issuing from the nozzle along the jet centerline is computed at the different spatial locations and the simulations were conducted at these different grid sizes to figure out the grid size which results in the solution becoming independent of it. The results are found to be independent of the grid size at all spatial locations along the jet centerline for the minimum grid size of 223581. Thus, the grid size of 223581 is adopted for the complete simulation of the mixing characteristics of the jet. The maximum skewness of 0.5 and a maximum aspect ratio of 2.38 are reported for the present grid size of 223581. The mesh is uniformly refined to capture the flow physics of the jet, when it is discharged from the nozzle. At the boundary of the domain (far-field), presence of the shear layer

![Figure 2. Computational Grid](image)
is almost absent and due to this the mesh is comparatively coarse at the boundaries. The presence of free shear layer in the jet vicinity due to the entrainment of the ambient fluid into the jet field, causing sharp changes into the fluids properties. Thus, fine refinement of the mesh is done at the axis of the jet and inside the nozzle only.

2.4. Numerical Procedure

The FLUENT solver which works on the Finite Volume method is used to solve the principal governing equations for mass, momentum and energy. These equations, in steady state conditions are solved with double precision accuracy, an approach to mitigate the effects of typical round-off errors. A second order upwind scheme is used for modelling the flow, the turbulent kinetic energy and other convective variables [11]. The principal governing equations for fluid flow are the following.

2.4.1. Continuity Equation (Conservation of Mass)

“The rate of change of mass of a fluid element is equal to the net rate of mass flow across its control volume”.

The two dimensional continuity equation for compressible flow of jet can be expressed as [12]

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0
\]  

(1)

Where, \( \vec{V} = u\hat{i} + v\hat{j} \); \( u \) and \( v \) are the components of velocity along the x and y direction respectively and \( \rho = \rho(x, y) \) is the density of air.

2.4.2. Momentum Equation

The momentum equation of the fluid element is governed from the Newton’s second law of the motion which states that “The sum of all forces acting on the fluid element is equal to the product of the mass of the fluid element and its acceleration”. In other words, the net forces acting on the fluid element is equal to the rate of change of momentum of the fluid element.

The equation of momentum in the direction of jet flow (x-direction) can be expressed as [12]

\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \vec{V}) = -\frac{\partial p}{\partial x} + \frac{\tau_{xx}}{\partial x} + \frac{\tau_{yy}}{\partial y} + \rho f_x
\]  

(2)
Similarly, the equation of momentum across the direction of jet flow (y-direction) can be expressed as
\[
\frac{\partial (\rho V)}{\partial t} + \nabla \cdot (\rho V V) = -\frac{\partial p}{\partial x} + \frac{\tau_{xy}}{\partial x} + \frac{\tau_{yy}}{\partial y} + \rho f_y
\]
(3)

Where, \( \tau_{xx} \) and \( \tau_{yy} \) denotes the shear stresses in the direction of X and Y respectively, \( \tau_{yx} \) and \( \tau_{yx} \) denotes the normal stresses in the direction of X and Y and acting normal to the direction of Y and X respectively. \( f_x \) and \( f_y \) are the body forces per unit mass along X and Y direction respectively.

2.4.3. Energy Equation

The energy equation is the statement of first law of thermodynamics which states that “The rate of change of energy inside a fluid element is equal to the net rate of heat transfer into the fluid element and the rate of work done on the fluid element”. The energy equation can be mathematically expressed as; [13]
\[
\frac{\partial}{\partial t} \left( \rho \left( e + \frac{V^2}{2} \right) \right) + \nabla \cdot \left( \rho V \left( e + \frac{V^2}{2} \right) \right) = -\rho \mathbf{V} \cdot \nabla p + \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k \frac{\partial T}{\partial y} \right) - \rho \mathbf{V} \cdot \left( \frac{\partial q}{\partial x} \right) + \frac{\partial \mathbf{q}}{\partial y} + \frac{\partial (\sigma_{xx})}{\partial x} \frac{\partial T}{\partial x} + \frac{\partial (\sigma_{yy})}{\partial y} \frac{\partial T}{\partial y} + \rho \mathbf{f} \cdot \mathbf{V}
\]
(4)

Where, \( q \), \( e \), \( k \), and \( p \) are the source term per unit mass, the internal energy of the fluid element, thermal conductivity of the material and the pressure respectively.

2.4.4. Turbulence Model

Realizable k-\( \varepsilon \) turbulence model is applied in the present investigation to capture the effects of turbulence, so as to quantify the mixing characteristics of the jet. The transport equations for \( k \) (turbulent kinetic energy) and \( \varepsilon \) (turbulent dissipation) in the Realizable \( k-\varepsilon \) turbulent model are [13]:
\[
\frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_j} \left( \rho k u_j \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \rho P_k + P_b - \rho \epsilon - \rho Y_M + S_k
\]
(5)
\[
\frac{\partial}{\partial t} \left( \rho \varepsilon \right) + \frac{\partial}{\partial x_j} \left( \rho \varepsilon u_j \right) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S_{\varepsilon} - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + \frac{\varepsilon}{k} + C_{\varepsilon} \frac{\varepsilon}{k} P_{\varepsilon} + S_{\varepsilon}
\]
(6)

Where, \( \mu \) and \( \mu_t \) denotes the coefficient of dynamic and eddy viscosity respectively. \( \sigma_k \) and \( \sigma_\varepsilon \) represents the turbulent Prandtl numbers for \( k \) and \( \varepsilon \) respectively. \( P_k \) and \( P_b \) are the generation of turbulent kinetic energy due to the mean velocity gradients and the generation of turbulent kinetic energy due to buoyancy respectively. \( Y_M \) represents the contribution of fluctuating dilatation in compressible turbulence to overall dissipation rate. \( S_k \) and \( S_{\varepsilon} \) are the user defined source terms.

Now, the model constant \( C_1 \) can be evaluated as;
\[
C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \eta = \frac{k}{\varepsilon}, \quad S = \sqrt{2S_{ij}S_{ij}}
\]

Where, \( S \) and \( S_0 \) denotes the strain tensor and mean strain rate respectively.
2.4.1.1. Modelling Turbulent Viscosity

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}
\]  

(7)

Where,

\[
C_\mu = \frac{1}{A_0 + A_s} \frac{KU}{\varepsilon}, \quad U^* = \sqrt{S_{ij}S_{ij} + \Omega_{ij}^2}, \quad \Omega_{ij} = \Omega_{ij} - 2\varepsilon_{ijk} \omega_k, \quad \Omega_{ij} = \Omega_{ij} - \varepsilon_{ijk} \omega_k
\]

Where, \(\Omega_{ij}\) is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity \(\omega_k\). The model constants \(A_0\) and \(A_s\) are given by:

\[
A_0 = 4.04, \quad A_s = \sqrt{6\cos \varphi}, \quad \varphi = \frac{1}{3} \cos^{-1} \left( \sqrt{6W} \right), \quad W = \frac{S_{ij}S_{ji}S_{ij}}{S^3}, \quad \hat{S} = \sqrt{S_{ij}S_{ij}}, \quad S = \frac{1}{2} \left( \frac{\partial u}{\partial x} + \frac{\partial \tilde{u}}{\partial x} \right)
\]

Model constants, \(C_\mu = 1.44, C_z = 1.9, \sigma_a = 1.0, \sigma_r = 1.2\)

2.5. Numerical Simulation

The numerical simulations were carried out in the FLUENT solver of ANSYS software of version 16.0. The density-based steady solver is adopted for the numerical simulations, as the jet in the present investigation is compressible and supersonic with Mach number 1.86. The implicit scheme and the convergence criteria of \(1\times10^{-5}\) are chosen for the convergence of the solutions. The computational economy is achieved by exploiting the symmetry of the nozzle, using axisymmetric formulation for the entire computational domain, as shown in Figure 1.

2.6. Computational Validation

In the present work, CFD validation is done with the experimental results of Shantanu and Rathakrishnan [10] at NPR 5 by plotting the centerline pressure decay (CPD) along the axis of the jet as shown in the Figure 5. The figure shows a quantitative comparison between the experimental and simulation results plotted as CPD at NPR 5. It was found that almost same number of shock cells are found in both the investigations viz., experimental and numerical. It is seen that there is a slight phase difference between the
CPD plots obtained experimentally and numerically. This could be attributed to the inability of the pitot probe to take measurements precisely from the origin point, due to physical interference of the probe with the physical model. In addition to this, the physical interference of the pitot probe leads to the generation of additional shocks in the flow field during experimentation. Due to the formation of shocks at the nozzle exit, the computational results show momentarily deceleration up to a distance of about 0.5D from the nozzle exit, which is otherwise not observed during experimentation. Thus, it is observed that, there would always be slight difference between the computational and experimental results due to the ability of computers to create an ideal experimental set up, which is not possible with physical experimentation. The flow accelerates thereafter due to the cumulative effect of the expansion fans and the relaxation offered by the sudden large space of free environment.

3. Results and Discussion

Figure 6. CPD of the overexpanded jet at NPR5
Figure 7. CPD of the overexpanded jet at NPR6
Figure 8. CPD of the underexpanded jet at NPR7
Figure 9. CPD of the underexpanded jet at NPR8

The core length of the jet is defined as the axial distance downstream of the nozzle exit along the jet centerline up to which the velocity of the jet remains supersonic, this is a direct indication of the extent of jet mixing. A shorter core length shows rapid mixing of the jet as compared to a similar jet with longer core. The characteristics decay zone begins after the jet core. A sharp characteristic decay indicates rapid mixing and the ability of the jet to attain self-similar profile rapidly. The CPD is a plot of non-dimensional total pressure variation, with respect to the non-dimensional distance along the jet centerline. The total pressure is made non-dimensional by dividing it with the upstream stagnation pressure. Whereas, the distance along centerline of the jet is made non-dimensional by dividing it with the equivalent nozzle exit diameter (D).
Figure 6 shows the CPD plot of the Mach 1.86 jet at NPR 5. It was found that the jet becomes overexpanded with an overexpansion level of about 21% at NPR 5. At this value of NPR, the flow of jet is dominated by the adverse pressure gradient at the nozzle exit. The flow along the downstream of the nozzle exit is influenced by the seven prominent shock cells.

This is also evident from the Mach contour plots of the jet, as shown in Figure 12. The jet core length extends to about 11D. The characteristic decay zone, extends from about 11D to 20D, along the jet centerline. It is nice to see that the characteristic decay of the jet exhibits sharp nature. The jet attains self-similar profile beyond X/D=20, which is evident from the flat nature of the CPD variation beyond X/D=20.

Figure 7 shows the CPD plot of the Mach 1.86 jet at NPR 6. It was found that the jet becomes near the correct expansion with a slight overexpansion level of about 5%. The flow of jet at the nozzle exit is dominated by the adverse pressure gradient, however, the level is drastically reduced compared to that at NPR 5. The jet flow along the downstream of the nozzle exit is influenced by the eight prominent shock cells. This is also evident from the Mach contour plots of the jet, as shown in Figure 12. The jet core length extends to about 12D. The characteristic decay zone, extends from about 12D to 20D, along the jet centerline. Thus, it might be inferred from the CPD plot that, with slight lowering in the level of overexpansion or the adverse pressure gradient at the nozzle exit, the jet shows faster mixing compared to that at lower NPR 5. The jet attains self-similar profile beyond X/D=20, which is evident from the flat nature of the CPD variation beyond X/D=20.

At the NPR 7 the jet becomes underexpanded with an underexpansion level of 11.5% as shown in Figure 8. The CPD of jet at NPR 8 becomes steeper than CPD of jet at NPR 7 which is clearly seen from the Figure 10. The seven prominent shock cells are also present at NPR 8 at NPR 7 as compared to NPR 5 but the shocks at NPR 7 is of stronger in strength which leads to faster mixing as compared to overexpanded jets. The jet attains self-similar profile beyond X/D=20, which is evident from the flat nature of the CPD variation beyond X/D=20.

At the NPR 8 the jet becomes highly underexpanded with an underexpansion level of 29% which is shown in the Figure 9. The CPD of jet at NPR 8 becomes steeper than CPD of jet at NPR 7 which is clearly seen from the Figure 10. The seven prominent shock cells are also present at NPR 8 as compared to NPR 7 but the first three shock cells at NPR 8 is of stronger strength which leads to steeper curve of CPD and thus
enhanced the jet mixing. The jet core length extends to about 14D. The characteristic decay zone, extends from about 14D to 20D, along the jet centerline. Due to the favorable pressure gradient at the nozzle exit Mach disk nucleates, which is shown in the Mach contour in Figure 13. Thus, it might be inferred from the CPD plot that, with rise in level of underexpansion the jet experiences stronger shocks and tends to have faster mixing as compared to overexpanded jets at low NPR.

The comparative plots of CPD and Mach number at different NPRs (5 to 8) are shown in Figures 10 and Figure 11. Thus the level of expansion plays a vital role in the mixing characteristics of the jet which is clearly evident from the plots of CPD at the nozzle exit. It is seen that the mixing extent improves with the increase in the NPR. However, the present investigation is focused on the mixing characteristics of the jet at all the levels of expansions. It is found that the extent of jet mixing improves with the increase in the favorable pressure gradient at the nozzle exit. The Mach contour plots of the jet at NPRs 5 to 8 are shown in Figures 12 and 13. It is seen that the number of shock cells increase with the increase in the NPR. Also, it is evident from the Mach contour plots that, the degree of jet mixing increases with the increase in the NPR or the favorable pressure gradient at the nozzle exit. The gradient of the characteristic decay becomes steeper with the increase in NPR, as shown in the plot for CPD (Figure 11) and is highest for the NPR 8. This clearly demonstrates that, with the increasing NPR, the rate of jet mixing enhances considerably. In another words it can be inferred that, with the increasing NPR, the jet velocity increases downstream of the nozzle exit. This leads to the enhanced rate with which the vortices generated at the jet boundary reach to the jet centerline causing rapid decay of the jet.

4. Conclusions

The present work focuses in simulating the mixing characteristics of a Mach 1.86 jet at the all levels of expansions (overexpansion, correct expansion, and under expansion) for different NPRs in the range of 5 to 8, with the step size of 1. The results are presented in the form of CPD plots along the jet centerline. The CPD is the plot of non-dimensional total pressure along the jet centerline, with respect to the non-dimensional distance along the jet centerline. It is found that,

a) The grid becomes independent above the grid size of 223581 and thus, this grid size is taken for the present study so as to attain computational economy.
b) The number of shock cells increase with the increasing NPR and consequently with the reducing level of adverse pressure gradient at the nozzle exit.

c) The core length of the jet increases with the increasing NPR which is due to the declining level of adverse pressure gradient at the nozzle exit.

d) The characteristic decay zone, which begins immediately after the potential core region and extends to the point along the jet centerline from which the jet attains self-similar profile, shortens and becomes steeper with the increase in the NPR.

e) The gradient of the characteristic decay zone increases with the increasing NPR which might be attributed to the enhanced rate with which the viscous activity reaches the jet centerline, consequently affecting the jet centerline velocity.

f) The jet attains self-similar profile beyond $X/D=20$, which clearly demonstrates that the viscous activity which originated at the jet boundary reaches the jet centerline.

References

[1] Rathakrishan E., Applied Gas Dynamics (Wiley India Pvt. Ltd.), 2010.
[2] Hussain A.K.M.F, Coherent Structures and Turbulence, Journal of Fluid Mechanics, Volume 173, pp 303-356, 1986.
[3] Shirie J. W. and Seubold J. G., Length of the Supersonic core in High-Speed Jets, AIAA Journal, Volume 5, Issue 11, pp 2062-2064, 1967.
[4] Ho Chih-Ming and Gutmark E., Vortex induction and mass entrainment in a small-aspect-ratio elliptic jet. Journal of Fluid Mechanics, Volume 179, pp 383-405, 1987.
[5] Gutmark E. and Wygnanski I., The planar turbulent jet, Journal of Fluid Mechanics, Volume 73, Issue 3, pp 465-495, 1976.
[6] Launder B.E. and Spalding D.B., The Numerical Computation of Turbulent Flows, Computer Methods in Applied Mechanics and Engineering, Volume 3, pp 269-289, 1974.
[7] Dash S.M, Kenzakowski D.C, Seiner J.M and Bhat T.R.S, Recent Advances in Jet Flow Field Simulation: Part I – Steady Flow. 15th AIAA Aeroacoustics Conference, (1993, October 25-27), Long Beach, CA.
[8] Thies Andrew T. and Tam Christopher K.W., Computation of Turbulent Axisymmetric and Nonaxisymmetric Jet Flows Using the k-ε Model. AIAA Journal, Volume 34, Issue 2, pp 309-316, 1996.
[9] Evgenevna Ilina E., Evgenevna Ilina T. and Viktorovich Butal P., Analysis of the Application of Turbulence Models in the Calculation of Supersonic Gas Jet, American Journal of Applied Science, Volume 11, Issue 11, pp 1914-1920, 2014.
[10] Srivastava Shantanu and Rathakrishnan E., Performance of Corrugated Limiting Tab in the Presence of Sharp Corners, International Review of Aerospace Engineering, Volume 7, Number 1, pp 1-7, 2014.
[11] Amina Meslem, Florin Bode, Cristina Croitoru, and Ilinca Nastase., Comparison of turbulence models in simulating jet flow from a cross-shaped orifice, European Journal of Mechanics B/Fluids, Volume 44, pp 100-120, 2014.
[12] Anderson J., Degrez G., Degroote J., Dick E., Grundmann R and Virendeels J., Computational Fluid Dynamics: An Introduction. Springer Verlag and Academic Press, 3rd edition, (2009), Berlin, Heidelberg.
[13] Wilcox, David C. Turbulence Modeling for CFD. DCW Industries, Inc., (1994), California.