International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 12 | Dec -2016 www.irjet.net p-ISSN: 2395-0072

Prediction of Turbulence Behavior over a 2D Double Wedge comparing K-Epsilon & K-Omega Model at Mach-4.2

Vamsi Krishna Chowduru[1], Yashas C R[2], Indraniil Roy[3]

[1]Assistant Professor, Department of Aerospace Engineering, IIAEM-Jain University, Bengaluru,India
[2] Visiting Professor, Department of Aerospace Engineering, IIAEM-Jain University, Bengaluru,India
[3] Visiting Professor, Department of Aerospace Engineering, IIAEM-Jain University, Bengaluru,India

---***---

Abstract - *With a Steep Angle at the leading edge of Wedge a mechanical device which coverts velocities to higher shock waves and huge turbulence. Our current research work focuses on behavior of Shock waves expanding at the Wedge entry and exit. Predicting the flow behavior at various angles the Wedge behaves with huge change in lift and drag divergence at leading and trailing edge of the Profile. Estimating the Wedge transforming the velocity from its steep slope to the width. With specific design requirements for a gauge pressure of 200KPa and 4.2 Mach the behavior of Flow at various flow angle can be analyzed. For the estimation of K-Epsilon & K Omega over the wedge at Steep Entry Angle can affect the flow velocity consequently at intervals. Computational Fluid Dynamics tools over a solid wedge is being used to predict the behavior of the flow characteristics.*

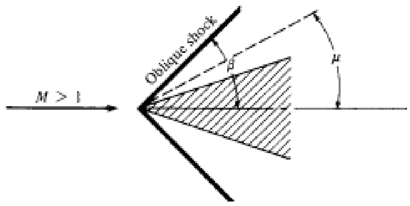
Key Words: Steep angle, Wedge, Shock waves, K-epsilon, K-Omega, Flow Behaviour.

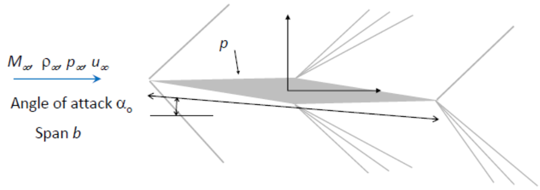
1.INTRODUCTION

A Supersonic wedge is designed for carrying out Flow behavior Turbulence models. Our paper currently focuses on the Behavior of Flow over the Wedge with Steep edge angle for analyzing the past leading edge for Turbulent Kinetic Energy and Eddy Dissipation. Assuming the design input of Shock waves, deflection angle and angle β of the shock with respect to the inflowing gas. Depending on angle of the wedge the reaction of flow creates continuous dispersion of Flow across the surface of the wedge. A Solid wedge when is inclined at various angles the puzzling behavior of Flow will not remain streamline. With corresponding surface interaction of Oblique shock wave the straight shock emanates the flow subsequently onto the wedge width. A Wedge can experience high end impacts on large width area based on the span and Angle of attach. Each turn the supersonic flow makes can be computed separately from every other angle, no wave reflections back onto surface. Considering the Mach angle with respect to the geometry with an expression as

$$\boxed{\mu = \sin^{-1}\frac{1}{M}}$$

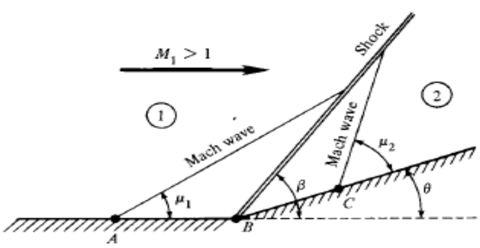
The Propagation of Weak Disturbances and the coalescence into a Mach wave can be shown using predicted Flow Behavior. For a Mach value greater than 1 the transformation of Flow and Shock angles is determined as shown below





1.1 Flow Behavior

From the edge of the Double Wedge surface the Leading edge experiences the immediate effect of velocity by which distribution of Flow is initiated. A Wedge can also experience Tangential velocity components based on the Mach number ahead in generating a shockwave. The Flow streamlines behind the shock are straight and parallel to the wedge surface. The Pressure of the surface of the wedge is constant and equal. Straight oblique shocks to the tip of the sharp edge in supersonic flow behavior in diverse relieving effect of surface. If the shock occurs from the same family of the flow the behavior seems to be intersecting and simply diverging.



$$\tan\theta = 2\cot\beta\left[\frac{M_1^2\sin^2\beta - 1}{M_1^2(\gamma+\cos 2\beta)+2}\right]$$

2. Developing a Design Model

2.1 Design Constraints

Considering various Aerodynamic coefficients the Design is developed to predict the Turbulence Behavior in Smaller angles of velocity reacting on the wedge surface. The Coefficient of pressure, Lift and drag are studied to estimate the right behavior of flow parameters. Understanding K-Epsilon and K Omega models for supersonic flows and the governing equations have been numerically solved using the CFD solver FLUENT. The K-Epsilon model has become one of the most widely used turbulence models as it provides robustness, economy and reasonable accuracy for a wide range of turbulent flows. Advancements have been made to the standard model which improves the performance and two variants in Fluent.

$$C_p \equiv \frac{p-p_\infty}{\frac{1}{2}\rho_\infty U_\infty^2}$$

$$C_L \equiv \frac{L/bc}{\frac{1}{2}\rho_\infty U_\infty^2}$$

$$C_D \equiv \frac{D/bc}{\frac{1}{2}\rho_\infty U_\infty^2}$$

The two transport equations are solved independently for the turbulent velocity and length scales. The significant comparison between the three models are; the turbulent Prandtl Numbers governing the turbulent diffusion of k and ε. Are the consecutive methods of improvement terms in the equation for ε .The method of calculating turbulent viscosity, the turbulent kinetic energy k, and its rate of dissipation ε, for this model are obtained by the following equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_K$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

where G_k represents the generation of turbulent kinetic energy the arises due to mean velocity gradients, Gb is the generation of turbulent kinetic energy that arises due to buoyancy, and Y_m represents the fluctuating dilution in compressible turbulence that contributes to the overall dissipation rate. S_e and S_k are turbulent Prandtl numbers for the turbulent kinetic energy and its dissipation rate. The turbulent (or eddy) viscosity at each point is related to the local values of turbulent kinetic energy and its dissipation rate by;

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

Where Cμ is constant and defined above.

The second turbulence model to be investigated is the K-Omega turbulence model. The K-Omega model has two variations that will both be shown below; the standard K-Omega model, and the shear stress transport (SST) model. The standard model is an empirical based model with transport equations for turbulent kinetic energy (k) and its specific dissipation rate (ω). These models have been modified during an attempt to improve its accuracy and as a result the transport equations used in FLUENT.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-Y_k+S_K$$

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}(\rho\omega u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\omega}\right)\frac{\partial\omega}{\partial x_j}\right]+G_\omega-Y_\omega+S_\omega$$

Where G_k represents the generation of turbulent kinetic energy that arises due to mean velocity gradients and G_ω is generation of ω which are defined in the exact manor as the K-Epsilon model. Y_k and Y_ω represents the dissipation of k (Kinetic Energy) and ω (Angular velocity) due to turbulence. The turbulent viscosity is defined using a damping coefficient (α*);

$$\mu_t = \alpha^*\frac{\rho k}{\omega}$$

This k-omega model was first developed in 1998 by Wilcox that has shown comparable results for far wakes, mixing layers and various jet types which make it applicable to wall bounded flows and free shear flows. This model has also incorporated modifications for compressibility, shear flow spreading, and low-Re number effects which are applicable in the present study.

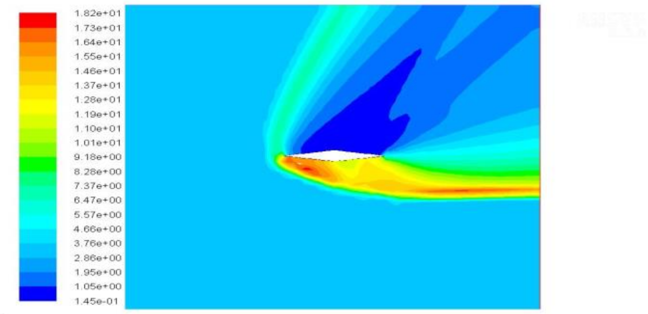
The detached Eddy simulation (DES) model is a cross between the Large Eddy Simulation (LES) and Reynolds Averaged Navier Stokes (RANS) coupling modeling approach. This model is generally used when it is advantageous to combine the RANS modeling approach with the LES approach when the use of the LES is determined to be too computationally expensive. In this case, with a large mesh size, the DES approach is significantly less expensive than an LES approach, but more expensive than RANS.

Unlike the Reynolds-averaged Navier Stokes (RANS) equation method, LES uses filtered Navier Stokes equations. Filtering the equations is a procedure by which the eddies with a scale smaller than the filter width (grid spacing) are filtered out leaving equations that model the dynamics of large scale eddies only. The LES model directly resolves large eddies in a time dependent simulation using the filtered equations essentially modeling less turbulence (and calculating more) thereby decreasing the inaccuracies created by modeling turbulence in small scales. In general these large eddies are similar in size to the characteristic length of the mean flow. Smaller Eddies that are usually responsible for dissipation of turbulent kinetic energy are “filtered” depending on the mesh size in FLUENT and modeled.

3. DESIGN DEVELOPMENT & ANALYSIS PREDICTIONS:

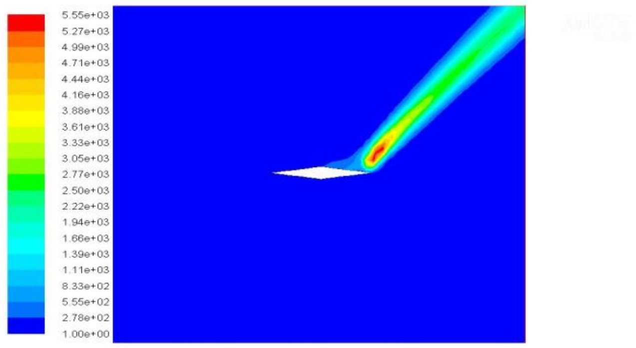
Computational fluid dynamics (CFD) uses numerical methods to solve and analyze problems that involve fluid flows. The governing equations discussed above have been solved with respect to the specified boundary conditions using the finite volume method as implemented in the commercial CFD code FLUENT. The Solution method is carried out from the Design in Workbench and Analyzed.

DENSITY CONTOUR:



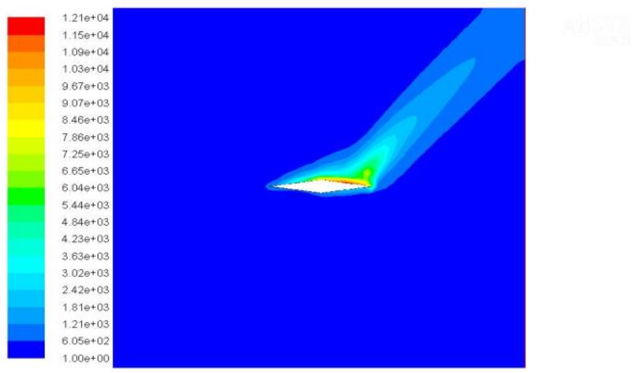
Contours of Density All (kg/m3) Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, ske)

TURBULENT KE(k):



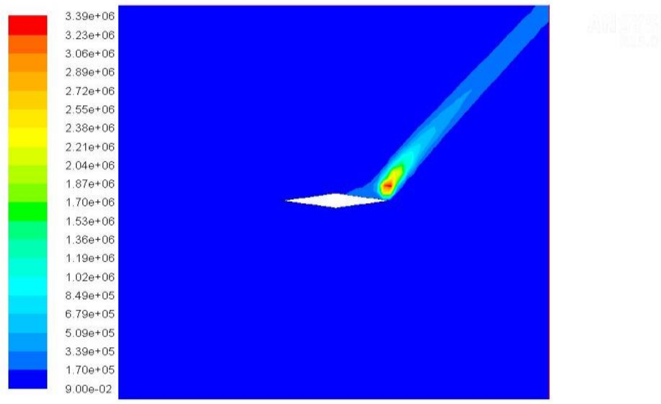
Contours of Turbulent Kinetic Energy (k) (m2/s2) Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, skw)

DISSIPATON RATE (ω):

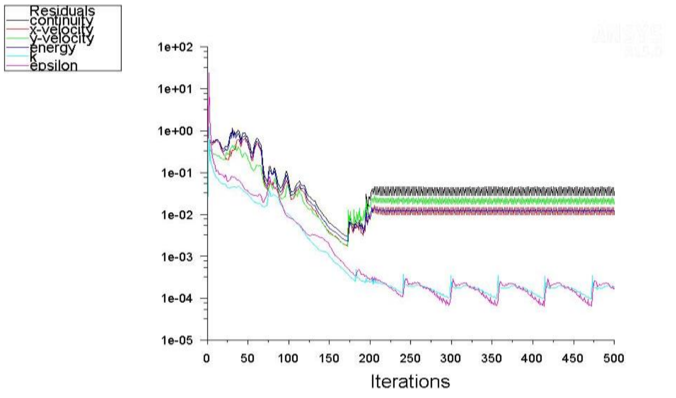


Contours of Specific Dissipation Rate (Omega) (1/s) Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, skw)

TURBULENT DISSIPATION RATE (ε):

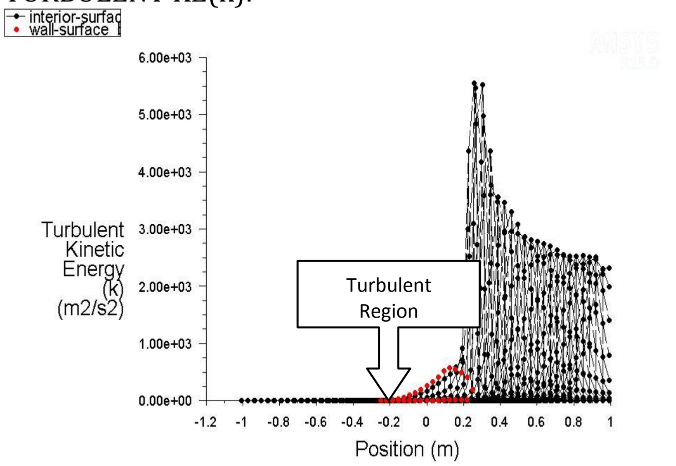


Contours of Turbulent Dissipation Rate (Epsilon) (m2/s3) Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, skw)



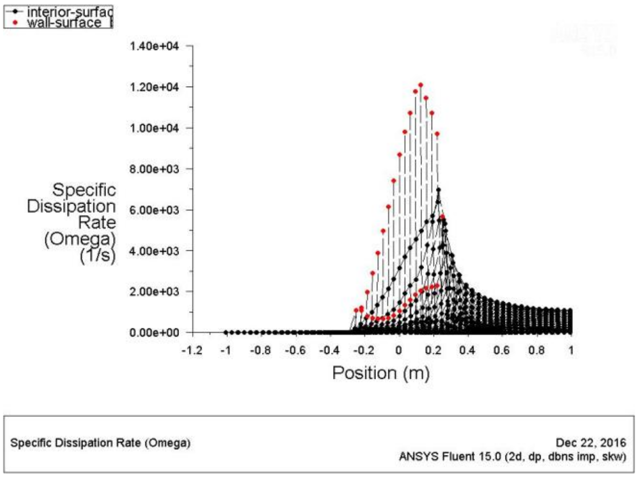
Scaled Residuals Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, ske)

TURBULENT KE(k):

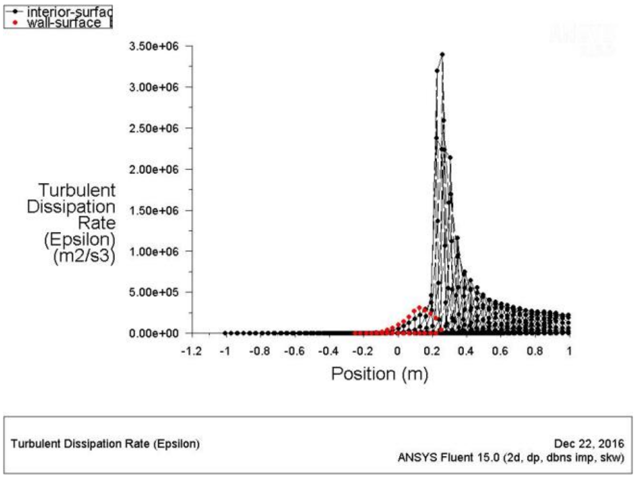


Turbulent Kinetic Energy (k) Dec 22, 2016
ANSYS Fluent 15.0 (2d, dp, dbns imp, skw)

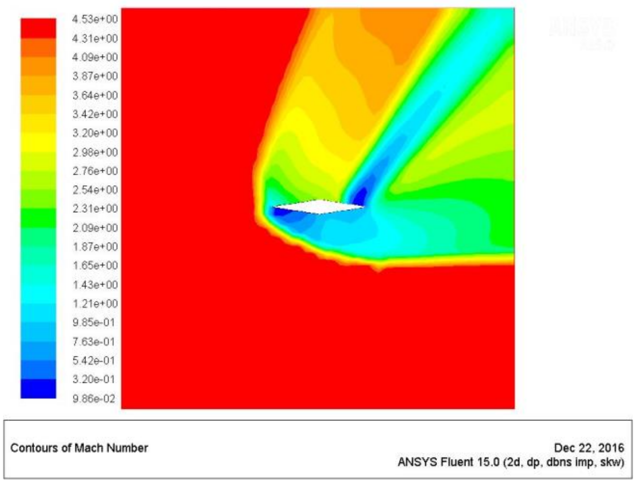
DISSIPATION REGION:

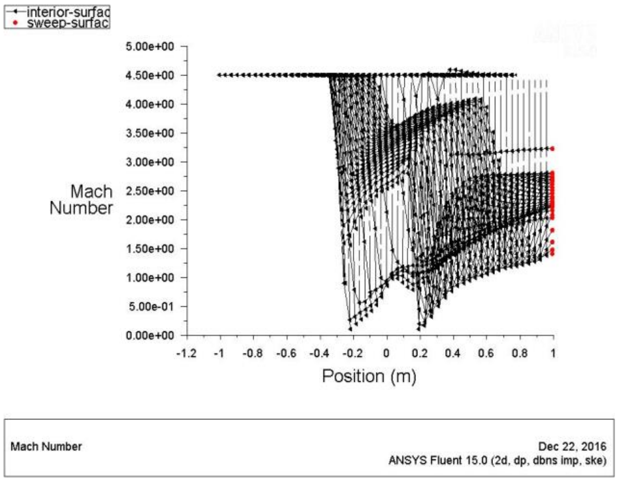


TURBULENT DISSIPATION REGION:



MACH BEHAVIOUR:





4. CONCLUSIONS

The Numerical implication for K Epsilon and K Omega have similar effects distribution of Turbulence under -0.2 to 0.2 position which implies that the effective turbulence behavior of smaller angle and effect on the width of the Wedge is accommodate optimum level of effects over the wedge. The maximum dissipation rate is observed over the upper surface of the wedge creates implicit effects on Solid Structure. Future study & experiments will be carried out for Steep angle for Turbulence in Supersonic conditions in various angle of attack of Single Wedge to Double Wedge for wider leading edge.

REFERENCES

[1] Bertin, J. J. & Hinkle, J. C. 1975 Experimental investigation of supersonic flow past double wedge configurations AIAA J. **13**, 897{901.

[2] Candler, G. V., Wright, M. J. & McDonald, J. D. 1994 Data-parallel lower-upper relaxation method for reacting flows. AIAA J. 32, **12**, 2380{2386.

[3] Edney, B. 1968 Anomalous heat transfer and pressure distributions on blunt bodies at hypersonic speeds in the presence of an impinging shock. Rep. 115. Flygtekniska Forsoksanstalten (The Aeronautical Research Institute of Sweden), Stockholm.

[4] Lamont, P. J. & Hunt, B. L. 1980 The impingement of under expanded, axisymmetric jets on perpendicular and inclined plates J Fluid Mech **100**, 471{511.

[5] Yakhot, V., S.A. Orszag, S. Thangam, T.B. Gatski and C.G. Speziale, 1992. Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A, 4(7): 1510-1520

[6] Bulat, P.V., O.N. Zasuhin and V.N. Uskov, 2012. On classification of flow regimes in a channel with sudden expansion. Thermo physics and Aeromechanics, 19(2): 223-246.

[7] Turbulent Flows – Fundamentals, Experiments and Modeling, G.Biswas and V.Eswaran, Narosa Publishing House, 2002.

[8] Turbulence Modeling for CFD – Wilcox, D.C, 1993

[9] Fluid Mechanics - An Introduction to the Theory of Fluid Flows, Franz Durst, Springer, 2008.

[10] S.C. Metcalf and C. J. Berry Boundary-layer interaction effects in low-density supersonic flow. Proc. 5th Int. Symp. on Rarefied Gas Dynamics. Vol. II (Ed: C. Brundin), pp 1087-1106. 1966.

BIOGRAPHIES:

Vamsi Krishna Chowduru is Assistant Professor from Department of Aerospace Engineering IIAEM-JU Has published papers on wide areas of Propulsion, Aerodynamics, CFD, and Waste Engineering.

Yashas CR has completed M.Tech in Aerospace Propulsion from IIAEM in 2016 has presented conference paper in Propulsion Research at IIT Bombay. Currently visiting Professor at IIAEM Jain University

Indraniil Roy completed his M.Tech in Aerodynamics and is currently a Visiting Professor in Dept of Aerospace Engineering IIAEM Jain University