iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 Issue: XI Month of publication: November 2017

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X1, November 2017- Available at www.ijraset.com

Numerical Investigations on Flow Characteristics over Backward Facing Sharp Edge Step through Hybrid RANS-LES

Dr. Nirmal Kumar Kund[1]
[1]*Associate Professor, Department of Production Engineering, Veer Surendra Sai University of Technology, Burla 768018, India*

***Abstract**: The present research involves the development of a 2D numerical model for examining the fluid flow behaviours (pertaining to density, vorticity and Mach number distributions) over a backward facing sharp edge step deploying the hybrid RANS-LES/Spalart-Allmaras turbulent model which also includes a viscosity-like variable. The model encompasses key issues like production, diffusion and destruction terms. The simulations are done with inflow free stream Mach number of 2.5 corresponding to free stream pressure and velocity of 15350 N/m^2 and 651.9 m/s^2, respectively. The simulation predictions of density, vorticity and Mach number are observed to be fairly consistent and also along the expected lines all over the whole flow regime. It is observed that both vorticity intensity and Mach number are quite high at the expansion fan near the lip of the separation as well as near the reattachment shock, whereas, the fluid density is quite low at the same. Furthermore, it is also noticed that the recirculation vicinity which is also otherwise called as dead air region has experienced the least fluid density, vorticity along with the Mach number owing to non-viscous rotation. In addition, both density field and Mach number benefit us in understanding whether the fluid flow is compressible or incompressible nature. Besides, the sudden change i.e. expansion/contraction within the flow field causes the vorticity generation resulting in uneven flow behaviours.*
***Keywords**: Density, Vorticity, Mach number, Backward Facing, Sharp Edge Step, Hybrid RANS-LES*

I. INTRODUCTION

Flow over backward-facing step is one of the imperative contexts and has increased unambiguous focus in view of not just easiness but for huge high-tech uses. In industrial aerodynamics, it is also used to investigate many complex structures, together with separation and reattachment. In the arena of research of high Mach number flow, the backward facing step is always taken as a complex configuration for ignition in a scramjet, where the recirculation vicinity has a major role in stabilizing the firing of the engine. Steps on the surfaces of hypersonic or supersonic aircrafts make the flow region more challenging and hence significant researches are really essential for refining the dynamic design of aircrafts.

II. LITERATURE REVIEW

Smith [1] executed experimental studies on the flow field and heat transfer downstream of a rearward facing step in supersonic flow. Launder and Sharma [2] applied the energy dissipation model of turbulence to investigate the flow field around a spinning disc. Armaly et al. [3] conducted both experimental and theoretical studies on backward facing step flow. Spalart and Allmaras [4] presented a one-equation turbulence model for evaluating aerodynamic flows. Anderson and Wendt [5] testified illustrious and complete descriptions of computational fluid dynamics. Neumann and Wengle [6] used both DNS and LES for investigating passively controlled turbulent flow of backward-facing step. Hamed et al. [7] done the numerical simulations of fluidic control for transonic cavity flows. Chen et al. [8] experimentally investigated on fine structures of supersonic laminar and turbulent flow over a backward-facing step through the Nano-based Planar Laser Scattering (NPLS). Liu et al. [9] numerically studied on the influences of inflow Mach number and step height on supersonic flows over a backward-facing step. Terekhov et al. [10] accomplished the experimental investigations on the separated flow structure behind a backward-facing step over and above the passive disturbance.
From the reported studies, to the best of author' information, it is observed that there is not a single extensive numerical study on flow over a backward facing sharp edge step by means of hybrid RANS-LES technique. With this standpoint, the current investigation establishes the numerical studies on flow characteristics over a backward facing sharp edge step by means of hybrid RANS-LES method. Furthermore, the numerical model also includes key features namely production, diffusion and destruction terms apart from the usual issues pertaining to the present physical problem. Additionally, the specified model also includes both compressibility and eddy viscous effects. The model is thoroughly demonstrated for the careful numerical studies on fluid flow

behaviours relating to flow over a backward facing sharp edge step by using the inflow free stream velocity and the corresponding Mach number as the important model parameters. Eventually, a 2D numerical model is developed to investigate the flow characteristics, pertaining to density, vorticity and Mach number distributions, over a backward facing sharp edge step using the hybrid RANS-LES/Spalart-Allmaras turbulent model which also includes a viscosity-like variable. Finally, the simulation predictions of density, vorticity and Mach number are found to be quite consistent and also along the expected lines throughout the entire flow regime.

III.DESCRIPTION OF PHYSICAL PROBLEM

Backward facing sharp edge step with extensive applications in industrial aerodynamics is studied in the current research. The geometric configuration together with initial and boundary conditions are referred from the experimental research report of Smith [1].

A. Geometric model

Figure 1 embodies the setup configuration for testing the backward facing sharp edge step flow over sharp edge geometry separating at a step height $H = 0.01125$ m, upstream distance from inlet to step $L_u = 0.1016$ m and downstream distance from sharp edge step to outlet $L_d = 0.2032$ m. The distance from downstream to upper boundary layer $Z = 0.15875$ m, spanwise distance $L = 0.3048$ m and width $B = 0.025908$ m. The separation and reattachment points are denoted by S and R respectively and are likely to be obtained after carrying out numerical simulation.

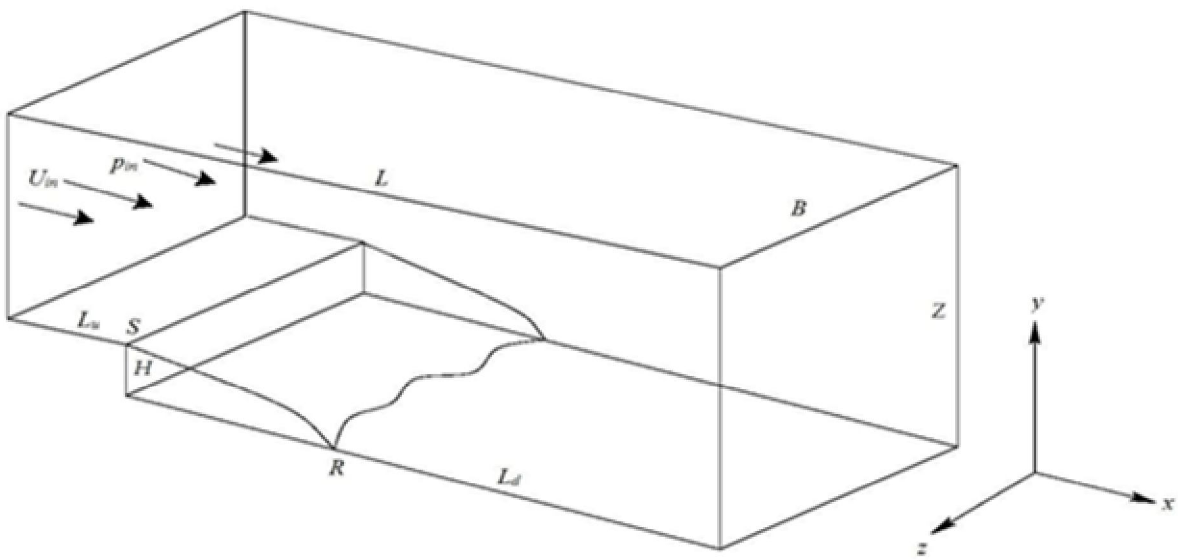


Figure1. Flow specification of backward facing sharp edge step

B. Initial and boundary conditions

The inflow free stream velocity $U_{in} = 651.9$ m/s, for which the identified static free stream pressure $p_{in} = 15350$ N/m^2 corresponds to the Mach number Ma = 2.5. At the left side ahead of the step, the initial temperature is maintained at 169.2 K. The initial conditions which are set on the upstream are really useful during the simulation along the spanwise direction, for obtaining the flow features beyond the step.

For the turbulence, the Spalart-Allmaras one-equation hybrid RANS-LES (otherwise termed as Detached Eddy Simulation, DES) model is considered.

The boundary conditions for the geometry shown in figure 2 are as mentioned underneath:

1) Pressure $p = 15.35$ kPa, everywhere else for pressure using hybrid RANS-LES model.
2) Temperature $T_{in} = 169.2$ K, everywhere else for temperature using hybrid RANS-LES model.
3) Velocity $U_{in} = 651.9$ m/s at the inlet, no-slip wall at the lower boundary, slip wall at the upper boundary and zero velocity gradient at the outlet are set for the present model.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X1, November 2017- Available at www.ijraset.com

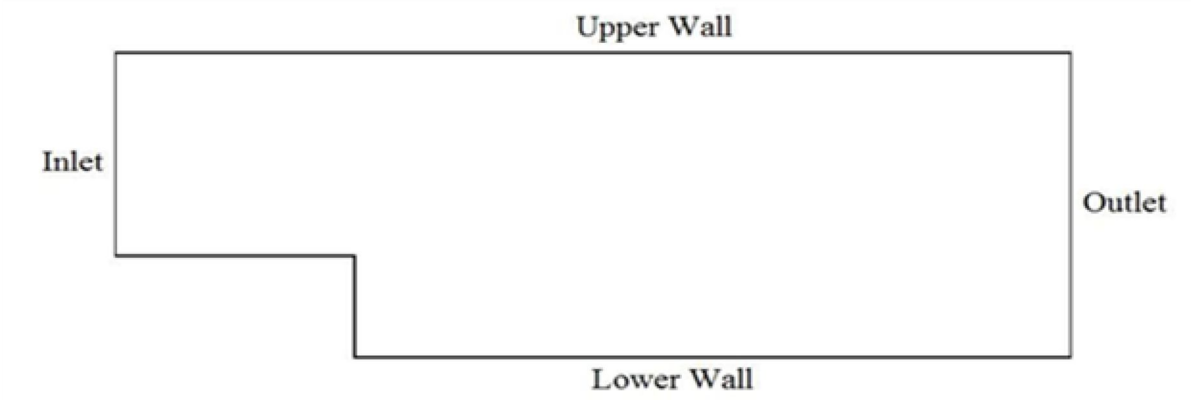


Figure2. Backward facing sharp edge step boundary representation

IV. MATHEMATICAL FORMULATION

A. Generalized governing transport equations

Very generalized governing transport equations of mass, momentum and energy stated in the conservative form of Navier-Stokes equation for compressible flow in association with the influences of turbulence are as follows.

Continuity: $$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_j)}{\partial x_j} = 0 \qquad (1)$$

Momentum: $$\frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}(2\mu S_{ij} + \tau_{tij}) \qquad (2)$$

Energy: $$\frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_j}(\bar{u}_j(\rho E + p)) = \frac{\partial}{\partial x_j}\left((k + k_t)\frac{\partial \bar{T}}{\partial x_j} + (2\mu S_{ij} + \tau_{tij})\bar{u}_i\right) + S_h \qquad (3)$$

Where, $$\left.\begin{aligned} u_i &= \bar{u}_i + u_i' \\ p &= \bar{p} + p' \\ T &= \bar{T} + T' \end{aligned}\right\} \qquad (4)$$

Total energy, $$E = e + k = h - \frac{p}{\rho} + \frac{v^2}{2} \qquad (5)$$

The Reynolds stress term is modeled in terms of the eddy viscosity and is represented by:

$$\tau_{tij} = 2\mu_t(S_{ij} - S_{nn}\delta_{ij}/3) - 2\rho k \delta_{ij}/3 \qquad (6)$$

The eddy viscosity is defined as a function of the turbulent kinetic energy k, and the turbulent dissipation rate ε, and is represented by:

$$\mu_t = c_\mu f_\mu \rho k^2/\varepsilon \qquad (7)$$

Besides, all the model terms/symbols/coefficients/functions have their usual meanings and values.

B. Hybrid RANS-LES Turbulence Modelling

The Hybrid RANS-LES/Spalart–Allmaras turbulence model also otherwise called as Detached Eddy Simulation (DES) model is a one-equation model for the eddy viscosity. The differential equation is derived from empiricism and arguments of dimensional

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X1, November 2017- Available at www.ijraset.com

analysis, Galilean invariance and selected dependence on the molecular viscosity. Grid resolution is not necessarily finer for this model, but, one can essentially capture the flow field with the related algebraic models.

The transport equation for the working variable (otherwise known as Spalart–Allmaras variable) i.e. viscosity-like variable ($\tilde{\nu}$) is represented by:

$$\frac{\partial(\rho\tilde{\nu})}{\partial t}+\tilde{u}_j\frac{\partial(\rho\tilde{\nu})}{\partial x_j}=c_{b1}\tilde{S}\rho\tilde{\nu}+\frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}(\mu+\rho\tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}+c_{b2}\frac{\partial\tilde{\nu}}{\partial x_j}\frac{\partial(\rho\tilde{\nu})}{\partial x_j}\right]-\rho c_{w1}f_w\left(\frac{\tilde{\nu}}{d}\right)^2 \tag{8}$$

The eddy viscosity is represented by: $\mu_t = \rho\tilde{\nu}f_{v1} = \rho\nu_t$ (9)

In addition, all the model terms/symbols/coefficients/functions have their usual meanings and values.

V. NUMERICAL PROCEDURES

A. Numerical scheme and solution algorithm

The said governing transport equations are transformed into generalized form represented by:

$$\frac{\partial}{\partial t}(\rho\varphi)+\nabla.(\rho\mathbf{u}\varphi)=\nabla.(\Gamma\nabla u)+S \tag{10}$$

The transformed governing transport equations are discretised by using a pressure based coupled framework involving finite volume method (FVM) with the SIMPLER algorithm, where ϕ denotes any conserved variable and S is a source term. The developed pressure based, fully coupled solver is utilized to predict flow characteristics of the associated flow variables relating to supersonic turbulent flow over a backward facing sharp edge step.

B. Choice of grid size, time step and convergence criteria

Figure 3 demonstrates that the grid of the computational domain is taken to be non-uniform and also grid is refined near the vicinity where the high gradient is expected. . In the current work, the simulation of the turbulence model with different wall distance from grid is performed on the computational domain. A comprehensive grid-independence test is carried out to develop an appropriate spatial discretization, and the levels of iteration convergence criteria to be used. As a consequence of this test, we have used 210 × 160 non-uniform grids for the final simulation. Corresponding time step taken in the simulation is 0.000001 seconds. Though, it is checked with smaller grids of 240 × 180 in numbers, it is found that a finer grid system does not change the results significantly.

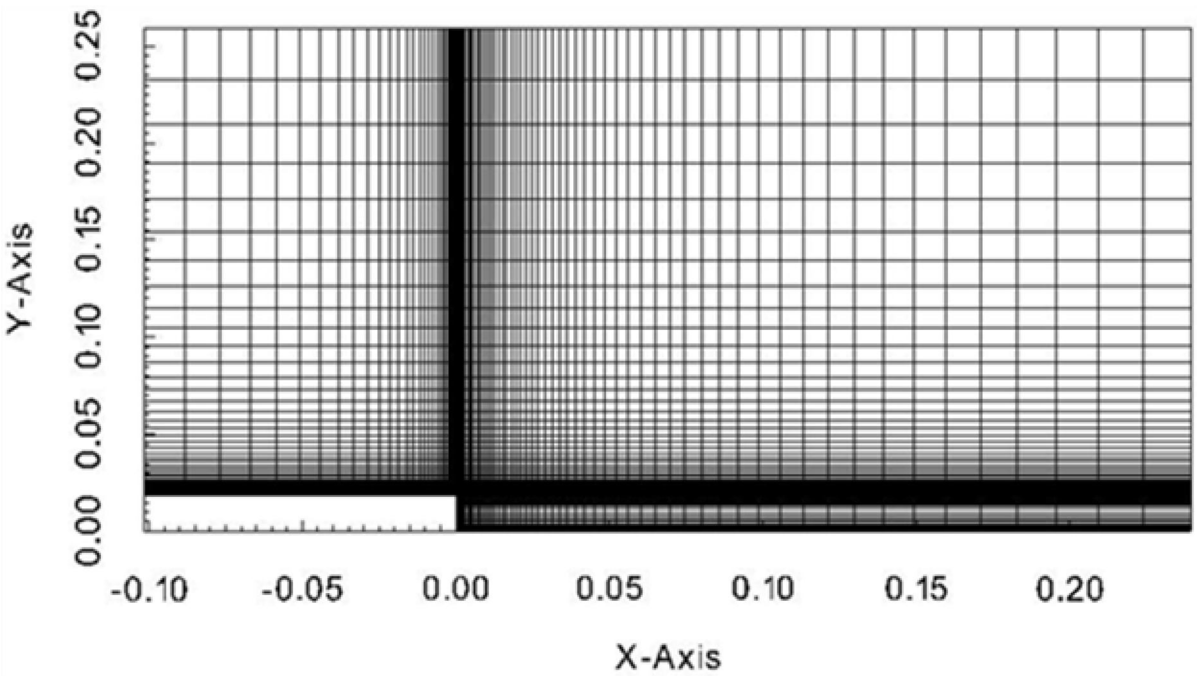


Figure 3. Mesh for backward facing sharp edge step

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X1, November 2017- Available at www.ijraset.com

Convergence in inner iterations occurs only when the condition $\left|\frac{\varphi-\varphi_{old}}{\varphi_{max}}\right| \leq 10^{-4}$ is held good concurrently for all variables, where φ symbolizes the field variable at a grid point at the present iteration level, φ_{old} denotes the corresponding value at the previous iteration level, and φ_{max} is the maximum value of the variable at the present iteration level in the whole domain.

VI. RESULTS AND DISCUSSION

With the previously designated model conditions, the numerical simulations are done for examining the fluid flow behaviours of the related flow variables on the subject of supersonic turbulent flow over a backward facing sharp edge step.

A. Density distributions

It is quite obvious that the high Mach inflow is the cause of more density gradient within the flow field. Figure 4 demonstrates the coloured density contour together with the corresponding vertical scale bar, showing the decrease in density within the vicinity of the expansion fan region, while, the reattachment shock region has gotten more density gradient. Furthermore, the recirculation vicinity which is also otherwise called as dead air region has felt the least density owing to non-viscous rotation. In addition, the supersonic turbulent flow over the backward facing sharp edge step has also gained the significant density fluctuations between the expansion fan and the reattachment shock wave regions. Moreover, it is rather apparent that because of the shock generation, density recovery behind the sharp edge step is also not flawless enough for smooth and sound fluid flow. Additionally, the physics behind the density gradient due to two parallel shocks can simply be understood from the coloured density field together with the corresponding vertical scale bar, as illustrated in figure 5. If the flow field experiences density variation more than five present then the flow is considered to be compressible flow which may be observed from the density field. In other words, the density field helps us in understanding whether the flow is compressible or incompressible nature.

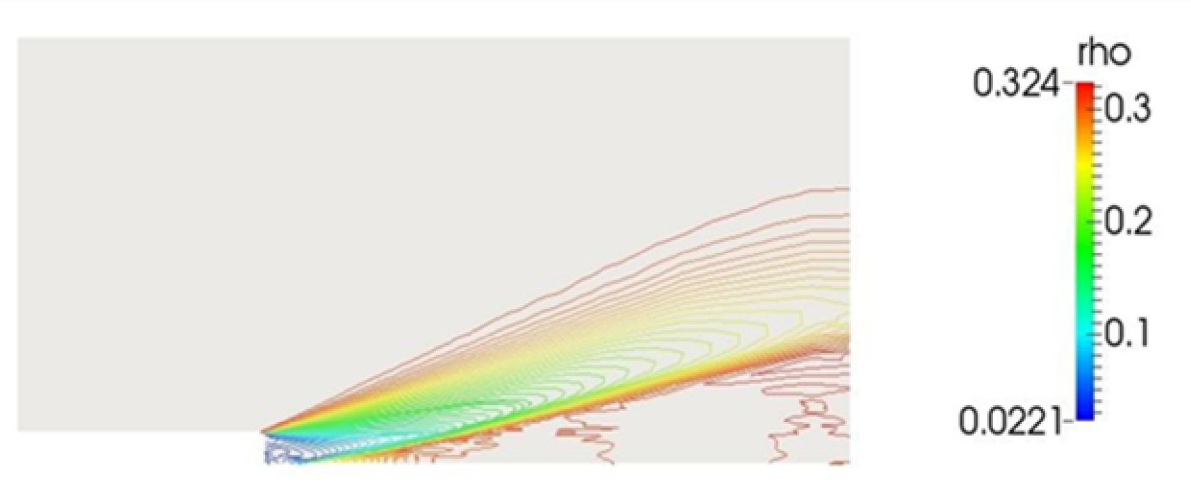


Figure 4. Density contour

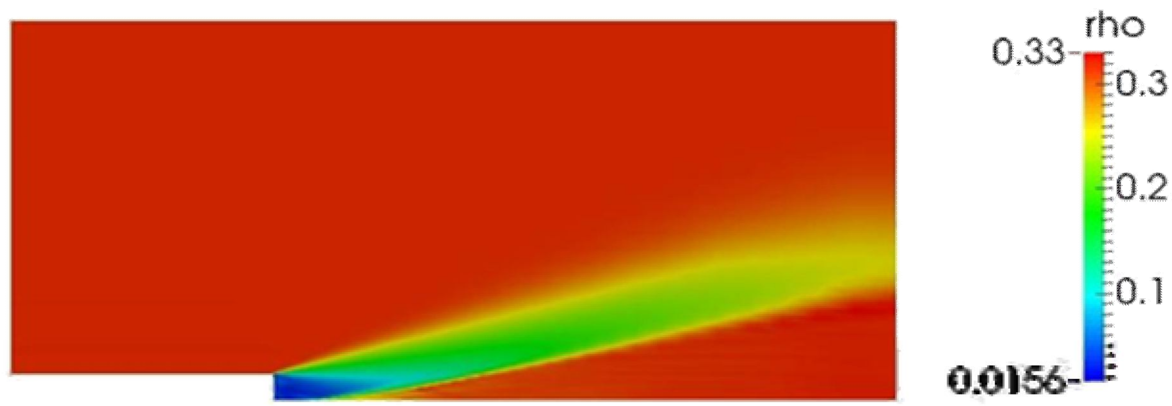


Figure 5. Density field

B. Vorticity distributions

Figure 6 demonstrates the coloured vorticity distribution together with the corresponding vertical scale bar, within the flow field. It may be observed that the vorticity intensity is quite high at the expansion fan near the lip of the separation as well as near the reattachment shock. Owing to viscous layer separation at the separation edge, the generation of lip shock has taken place, in addition, the interaction of shock and expansion fan has led to the generation of vorticity. The vorticity generation is due to sudden change (i.e. expansion or contraction) within the flow field. Furthermore, the vorticity generation becomes predominant because of the turbulent boundary layer separation.

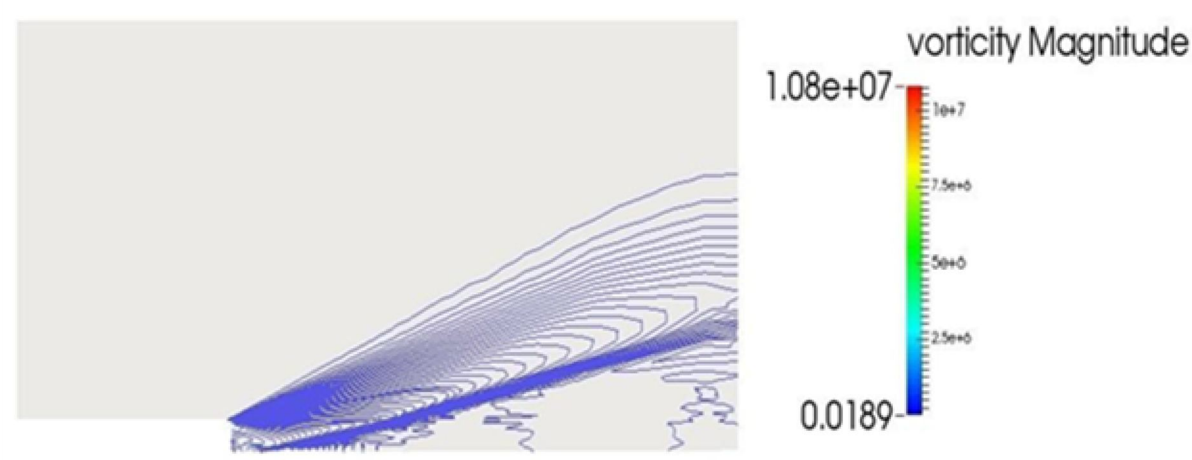


Figure 6. Vorticity distributions

C. Mach number distributions

Figure 7 shows the coloured Mach number contour together with the corresponding vertical scale bar, demonstrating the increase in Mach number within the vicinity of the expansion fan region, further, the reattachment shock region has also felt more Mach number gradient. Besides, there is a sudden increase in Mach number which is observed just ahead of the separation edge. Furthermore, the recirculation vicinity which is also otherwise termed as dead air region has attained the least Mach number (which is almost near to zero) owing to non-viscous rotation. Additionally, the supersonic turbulent flow over the backward facing sharp edge step has also gained the significant Mach number fluctuations between the expansion fan and the reattachment shock wave regions. Moreover, it is rather apparent that because of the shock generation, Mach number recovery occurs within the redeveloped boundary layer near the bottom wall just ahead of the reattachment point. However, after the reattachment shock the Mach flow continues in its usual direction. In addition, the physics behind the Mach number gradient due to two parallel shocks can simply be understood from the coloured Mach number field together with the corresponding vertical scale bar, as illustrated in figure 8. If the Mach number is more than 0.3 then the fluid region is considered to be compressible which may be observed from the Mach number field. In other words, the Mach number field helps us in understanding whether the fluid region is compressible or incompressible nature. However, no such changes or extra special effect is observed near the upper region of flow field.

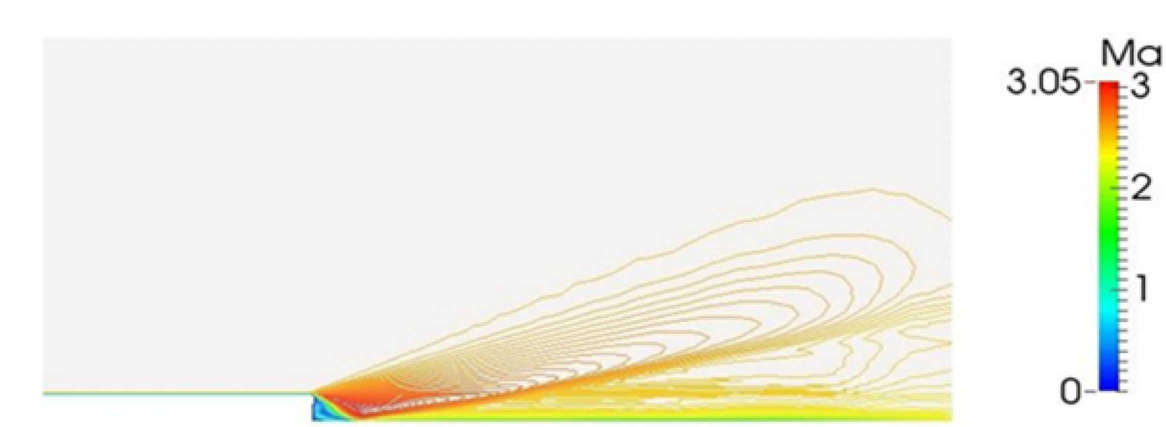


Figure 7. Mach number contour

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue X1, November 2017- Available at www.ijraset.com

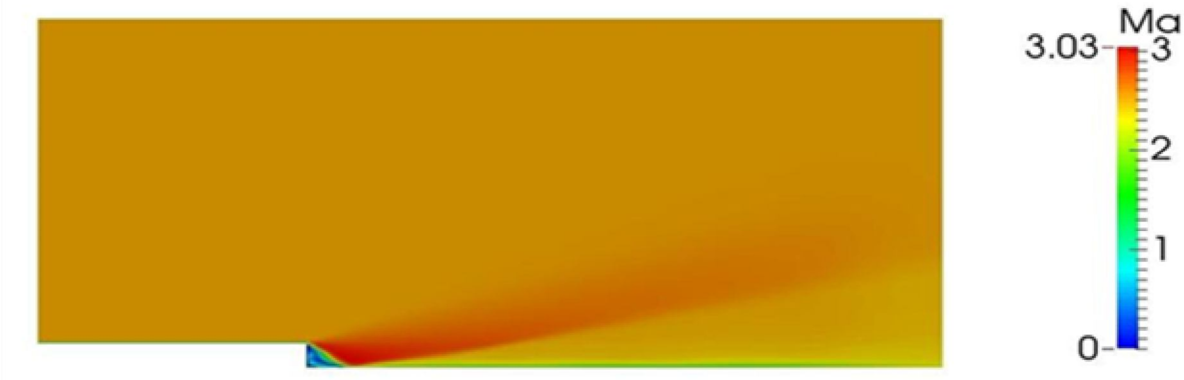


Figure 8. Mach number field

VII. CONCLUSIONS

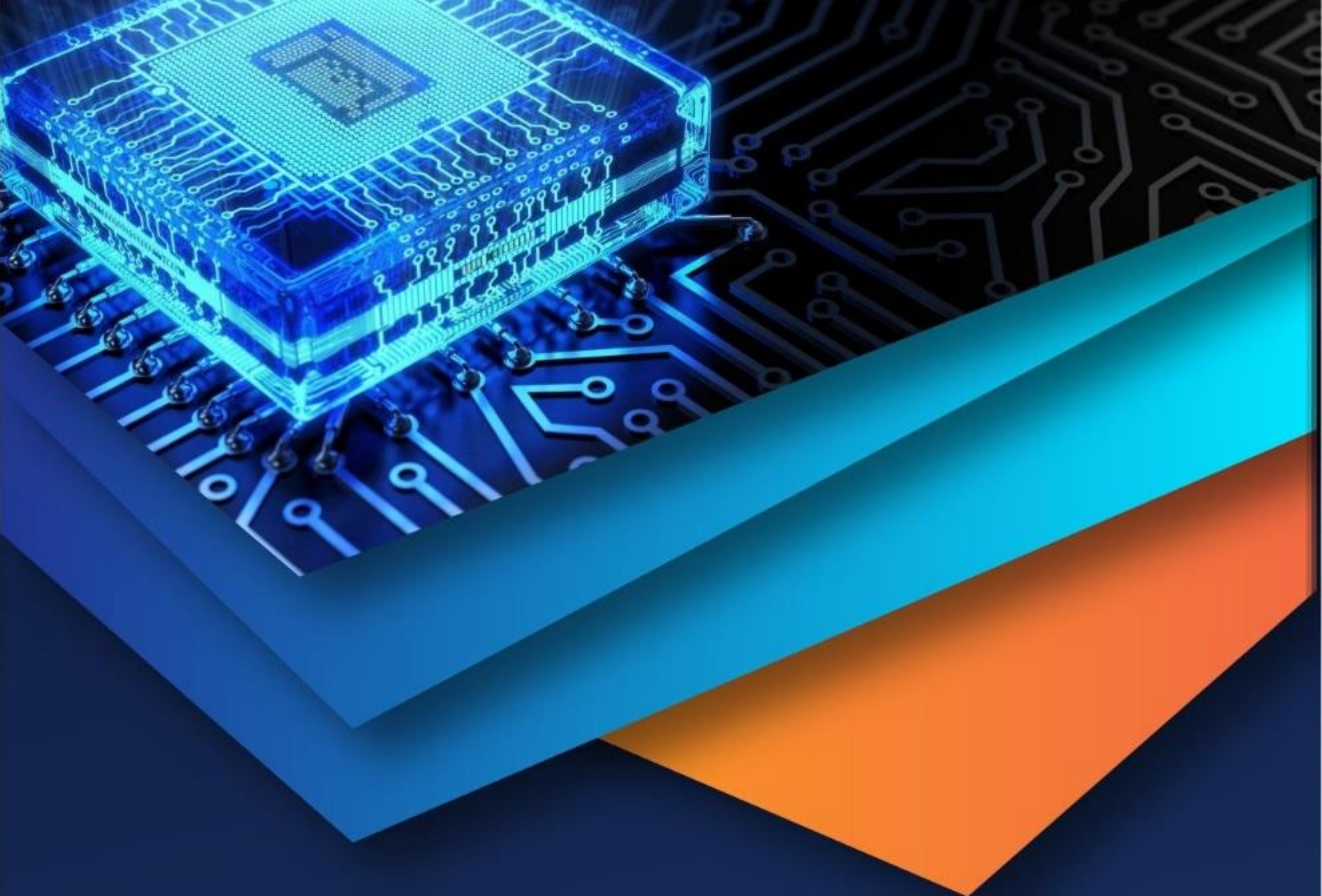
A two dimensional numerical model is developed to examine the fluid flow behaviours, relating to density, vorticity and Mach number distributions, over a backward facing sharp edge step with the hybrid RANS-LES/Spalart-Allmaras turbulent model which comprises a viscosity-like variable ($\tilde{\nu}$). The model also incorporates fundamental terms namely production, diffusion and destruction in to account. The numerical simulations are conducted with inflow free stream Mach number of 2.5 corresponding to free stream pressure and velocity of 15350 N/m^2 and 651.9 m/s^2, respectively. The simulation results of density, vorticity and Mach number are witnessed to be quite consistent and also along the lines of expectations within the entire flow regime. It is noticed that both vorticity intensity and Mach number are rather high at the expansion fan near the lip of the separation and near the reattachment shock, while, the fluid density is rather low at the same. In addition, it is also observed that the recirculation vicinity which is also otherwise termed as dead air region has felt the lowest fluid density, vorticity together with the Mach number because of non-viscous rotation. Furthermore, both density field as well as Mach number helps us in realizing whether the fluid flow is compressible or incompressible nature. Also, the sudden change (i.e. expansion or contraction) within the flow field leads to the vorticity generation causing the uneven flow behaviours. Additionally, the numerical modelling for fluid flow over a backward facing round step is in progress and is intended for the future to minimize recirculation region which can cause smaller shear layer formation and the shorter reattachment length, for the same model situations. Certainly, the current simulation predictions (of flow over the backward facing sharp edge step) will be really very much helpful to know the further flow features.

VIII. ACKNOWLEDGMENT

The author would like to thank the editor and the reviewers for their noble thoughts, valuable time together with contributions for giving insightful reviews to the research article.

REFERENCES

[1] Smith, Howard E. “The flow field and heat transfer downstream of a rearward facing step in supersonic flow.” No. ARL-67-0056. Aerospace Research Labs, Wright Patterson AFB, Ohio, (1967).
[2] Launder, B. E., and B. I. Sharma. "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc." Letters in heat and mass transfer Vol. 1, Issue 2 (1974): 131-137.
[3] Armaly B. F., Durst F., Pereira J. C. F., and Schoenung B., “Experimental and theoretical investigation of backward facing step flow,” Journal of Fluid Mechanics, Vol. 127, pp. 473–496, (1983).
[4] Spalart, Phillipe R., and Steven R. Allmaras. "A one-equation turbulence model for aerodynamic flows." (1992).
[5] Anderson, John David, and J. F. Wendt. Computational fluid dynamics. Vol. 206. New York: McGraw-Hill, (1995).
[6] Neumann, Jens, and Hans Wengle. "DNS and LES of passively controlled turbulent backward-facing step flow." Flow, turbulence and Combustion 71.1-4 (2003): 297-310.
[7] Hamed, A., K. Das, and D. Basu. "Numerical simulations of fluidic control for transonic cavity flows." AIAA Paper 429, (2004).
[8] Chen, Zhi, et al. "An experimental study on fine structures of supersonic laminar/turbulent flow over a backward-facing step based on NPLS." Chinese Science Bulletin, Vol. 57, Issue 6 (2012): 584-590.
[9] Liu, Haixu, et al. "Effects of Inflow Mach Number and Step Height on Supersonic Flows over a Backward-Facing Step." Advances in Mechanical Engineering (2013).
[10] V. I. Terekhov, Ya. I. Smul’skii, and K. A. Sharov, “Experimental study of the separated flow structure behind a backward-facing step and a passive disturbance,” Journal of Applied Mechanics and Technical Physics, Volume 57, Issue 1, (2016) pp 180–187.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY