Numerical Studies on Supersonic Turbulent Flow over a Backward Facing Sharp Edge Step Using Hybrid RANS-LES

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

Abstract – In the present study, a two dimensional numerical model is developed to investigate supersonic turbulent fluid flow over a backward facing sharp edge step by using both RANS (relating to the standard k-ε model) and hybrid RANS-LES (pertaining to the Spalart-Allmaras model involving viscosity-like variable) models. The model also involves additional important factors namely production, diffusion and destruction terms besides the very common aspects related to the present research problem. The numerical simulations are performed using the stated turbulence models with the inflow free stream Mach number of 2.5 along with the free stream pressure and velocity of 15350 N/m^2 and 651.9 m/s^2, respectively. The simulation predictions are compared with the corresponding experimental results available in the literature. Even though the simulation predictions from both RANS and hybrid RANS-LES models yield satisfactory results, however, the hybrid RANS-LES model gives fairly better and consistent results over the RANS model (which provides very good results within the vicinity of the wall region only) throughout the entire flow region and hence, only the hybrid RANS-LES is considered for further investigations. It is also noticed that the sudden viscous layer separation is the prime cause for the generation of reattachment shock. In addition, the uneven pressure recovery is owing to the sudden expansion flow over the sharp edge step. Further, the sudden expansion flow increases the intensity of the shock which leads to uneven flow behaviours. Definitely, the present test case results are very highly beneficial to understand the flow characteristics of supersonic turbulent fluid flow over the backward facing sharp edge step flows.

Index Terms – Supersonic, Turbulent Flow, Backward Facing, Sharp Edge Step, RANS, Hybrid RANS-LES

1. INTRODUCTION

The flow over backward-facing step is one of the important frameworks and has gained specific attention on account of not just simplicity but for voluminous technological applications. In applied aerodynamics, it is also used to study many complicated structures, including separation and reattachment. In the field of research of high Mach number flow, the backward facing step is always considered as a complex configuration for ignition in a scramjet, where the recirculation vicinity has a significant role in stabilizing the firing of the engine. Steps on the surfaces of hypersonic or supersonic aircrafts make the flow regime more complex and hence momentous investigations are very much required for improving the lively design of aircrafts.

2. LITERATURE REVIEW

Smith [1] carried out experimental investigations on the flow field and heat transfer downstream of a rearward facing step in supersonic flow. Launder and Sharma [2] used the energy dissipation model of turbulence to analyse 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] introduced a one-equation turbulence model for assessing aerodynamic flows. Anderson and Wendt [5] reported illustrious and comprehensive descriptions of computational fluid dynamics. Neumann and Wengle [6] used both DNS and LES for examining passively controlled turbulent flow of backward-facing step. Hamed et al. [7] performed the numerical simulations of fluidic control for transonic cavity flows. Chen et al. [8] studied experimentally on fine structures of supersonic laminar as well as turbulent flow over a backward-facing step by using Nano-based Planar Laser Scattering (NPLS). Liu et al. [9] investigated numerically on the influences of inflow Mach number and step height on supersonic flows over a backward-facing step. Terekhov et al. [10] performed the *e*xperimental studies on the separated flow structure behind a backward-facing step in addition to the passive disturbance.

From the stated investigations, to the best of author' knowledge, it is found that there is not a single comprehensive numerical study on flow over a backward facing sharp edge step using hybrid RANS-LES method. With this perspective, the present research demonstrates the numerical studies on flow behaviours over a backward facing sharp edge step using hybrid RANS-LES technique. In addition, the numerical model also involves additional important features namely production, diffusion and destruction terms besides the common issues relating to the present physical problem. Furthermore, the stated model also includes both compressibility and eddy viscous effects. The model is very well demonstrated for the meticulous numerical studies on fluid flow characteristics pertaining to flow over a backward facing sharp edge step by introducing the inflow free stream velocity along with the corresponding Mach number as the key model parameters. Ultimately, the present case of backward facing sharp edge step for both RANS (associated with the standard k-ε model) and hybrid RANS-LES (concerning Spalart-Allmaras model involving viscosity-like variable) predictions of fully supersonic turbulent flow are compared with experimental data of Smith [1]. Eventually, the model predictions with regard to the specified key model parameters are also along the expected lines and are in very good agreement with the corresponding experimental results.

3. DESCRIPTION OF PHYSICAL PROBLEM

Backward facing sharp edge step having wide range of applications in applied aerodynamics is investigated in the present research. The geometric configuration along with initial and boundary conditions are referred from the experimental research report of Smith [1].

3.1. Geometric Model

Figure 1 represents 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 represented by S and R respectively and are expected to be observed after performing numerical simulation.

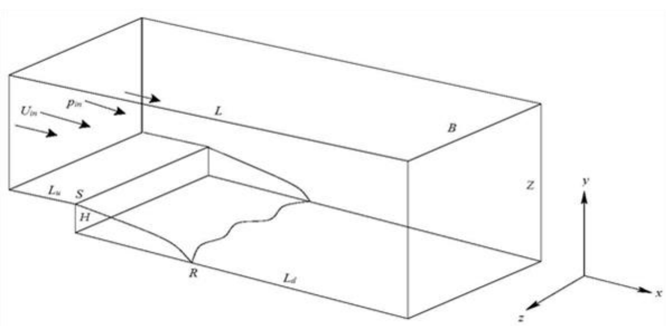

Fig 1. Flow specification of backward facing sharp edge step

3.2. Initial and Boundary Conditions

The inflow free stream velocity U_{in} = 651.9 m/s, for which the known 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 very much useful throughout the simulation along the spanwise direction, for getting the flow characteristics beyond the step.

For the turbulence, both RANS standard k-ε two-equation model and Spalart-Allmaras one-equation hybrid RANS-LES (otherwise termed as Detached Eddy Simulation, DES) model are taken into account.

The boundary conditions for the geometry represented by figure 2 are as follows:

- Pressure p = 15.35 kPa, everywhere else for pressure in case of both RANS and hybrid RANS-LES models.
- Temperature T_{in} = 169.2 K, everywhere else for temperature for both RANS and hybrid RANS-LES models.
- 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 both the models.

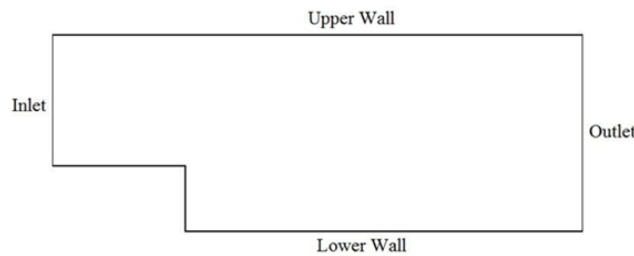

Fig 2. Backward facing sharp edge step boundary representation

4. MATHEMATICAL FORMULATION

The most generalized governing transport equations of mass, momentum and energy expressed in the conservative form of Navier-Stokes equation for compressible flow accompanying the influences of turbulence are as mentioned underneath.

Continuity:

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

Momentum:

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

Energy:

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

Where,

$$\left.\begin{aligned} u_i &= \bar{u}_i + u_i' \\ p &= \bar{p} + p' \\ T &= \bar{T} + T' \end{aligned}\right\} \tag{4}$$

Total energy,

$$E = e + k = h - \frac{p}{\rho} + \frac{v^2}{2} \tag{5}$$

The Reynolds stress term is modeled in terms of the eddy viscosity and is expressed as:

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

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

$$\mu_t = c_\mu f_\mu \rho k^2/\varepsilon \tag{7}$$

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

4.1. RANS Turbulence Modelling

The standard k-ε model is the most widely known and used extensively for two-equation eddy viscosity model. Transport equations are interpreted by two scalar properties of turbulence i.e., the transport equation k-equation is a model for the turbulent kinetic energy and the ε-equation is a model for the dissipation rate of turbulent kinetic energy.

The turbulent transport equations for the k-ε model are defined as follows:

Turbulent kinetic energy:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho\bar{u}_j k-\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) = \tau_{tij}S_{ij}-\rho\varepsilon+\phi_k \tag{8}$$

Energy dissipation:

$$\frac{\partial(\rho \varepsilon)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho\bar{u}_j \varepsilon-\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right) = c_{\varepsilon 1}\frac{\varepsilon}{k}\tau_{tij}S_{ij}-c_{\varepsilon 2}f_2\rho\frac{\varepsilon^2}{k}+\phi_\varepsilon \tag{9}$$

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

4.2. Hybrid RANS-LES Turbulence Modelling

The Spalart–Allmaras turbulence model is a one-equation model for the eddy viscosity. The use of this model is otherwise known as Hybrid RANS-LES modelling or Detached Eddy Simulation (DES) modelling. The differential equation is derived by using empiricism and arguments of dimensional analysis, Galilean invariance and selected dependence on the molecular viscosity. Grid resolution does not need to be finer for this model, however, one can essentially apprehend the velocity field gradient with the associated algebraic models.

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

$$\frac{\partial(\rho\tilde{\nu})}{\partial t}+\tilde{u}_j\frac{\partial(\rho\tilde{\nu})}{\partial x_j} = c_{b1}\tilde{S}\rho\tilde{\nu}+\frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}(\mu+\rho\tilde{\nu})\frac{\partial\tilde{\nu}}{\partial x_j}+c_{b2}\frac{\partial\tilde{\nu}}{\partial x_j}\frac{\partial(\rho\tilde{\nu})}{\partial x_j}\right]-\rho c_{w1}f_w\left(\frac{\tilde{\nu}}{d}\right)^2 \tag{10}$$

The eddy viscosity can be expressed as follows:

$$\mu_t = \rho\tilde{\nu}f_{v1} = \rho\nu_t \tag{11}$$

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

5. NUMERICAL PROCEDURES

5.1. Numerical scheme and solution algorithm

The aforesaid governing transport equations are converted into much generalized form as follows.

$$\frac{\partial}{\partial t}(\rho\varphi)+\nabla.(\rho\mathbf{u}\varphi)=\nabla.(\Gamma\nabla u)+S \tag{12}$$

The converted governing transport equations are discretized by expending a pressure based coupled framework relating to finite volume method (FVM) using the SIMPLER algorithm, where ϕ represents any conserved variable and S is a source term. The established pressure based, fully coupled solver is used to predict flow behaviours of the related flow variables in connection with supersonic turbulent flow over a backward facing sharp edge step.

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

Figure 3 shows that the grid of the computational domain is considered to be non-uniform and also grid is refined near the vicinity where the high gradient is expected. . In the present work, the simulation of both the turbulence models with different wall distance from grid is carried out on the computational domain. A comprehensive grid-independence test is performed to establish a suitable spatial discretization, and the levels of iteration convergence criteria to be used. As an outcome 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 observed that a finer grid system does not alter the results significantly.

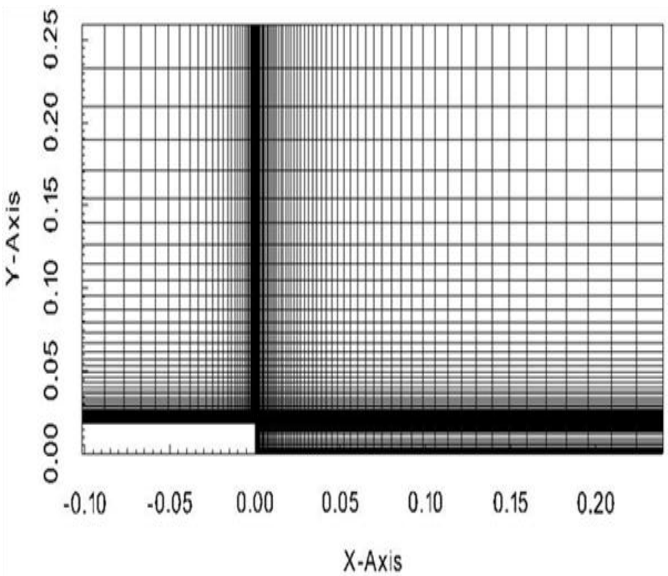

Fig 3. Mesh for backward facing sharp edge step

The convergence in inner iterations is declared only when the condition $\left|\frac{\varphi-\varphi_{old}}{\varphi_{max}}\right| \leq 10^{-4}$ is satisfied simultaneously for all variables, where φ stands for the field variable at a grid point at the current iteration level, φ_{old} represents the corresponding value at the previous iteration level, and φ_{max} is the maximum value of the variable at the current iteration level in the entire domain.

6. RESULTS AND DISCUSSIONS

With the already described model conditions, the numerical simulations are performed for investigating the fluid flow behaviours of the associated flow variables pertaining to supersonic turbulent flow over a backward facing sharp step.

6.1. Comparison of numerical predictions with experimental results

The numerical simulation predictions are conscientiously compared with the available experimental data of Smith [1] reported in the literature, in order to ascertain the simulation accuracy beforehand. The turbulence models taken into account (for the present case of fully supersonic turbulent fluid flow over the backward facing sharp edge step) for comparisons of the numerically predicted results (with the corresponding experimental data) are both RANS (relating to the standard k-ε model) and the hybrid RANS-LES (associated with the Spalart-Allmaras model involving the viscosity-like variable ($\tilde{\nu}$)). The model (between RANS and RANS-LES) rendering the relatively better simulation accuracy will be considered for further investigations.

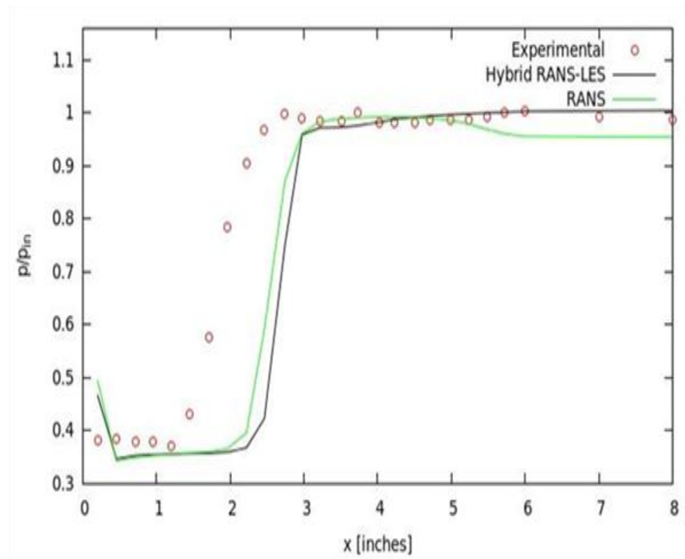

Fig 4. Pressure recovery comparison with experimental data

The comparisons of the numerical simulation research works for pressure recovery with the experimental data available in the literature reveals that the accuracy of the RANS model is limited to near wall region, whereas, the hybrid RANS-LES maintain the consistency in accuracy beyond the wall vicinity as demonstrated in figure 4. Therefore, the hybrid RANS-LES model gives rise to relatively superior and precise predictions than RANS model. Hence, only the hybrid RANS-LES is taken into considerations for further studies.

6.2. Pressure distributions

Figure 5 illustrates the coloured pressure contour along with the vertical scale bar, representing the decrease in pressure near the vicinity of the expansion fan region, whereas, the reattachment shock region has experienced more pressure gradient. In addition, the recirculation vicinity which is also known as dead air region has experienced the least pressure because of non-viscous rotation. Furthermore, the supersonic turbulent flow over the backward facing sharp edge step has also experienced the noticeable pressure fluctuations between the expansion fan and the reattachment shock wave regions. Additionally, it is quite evident that owing to the shock generation pressure recovery behind the sharp edge step is also not immaculate enough for smooth and flawless fluid flow. In addition, the physics behind the pressure gradient caused by two parallel shocks may easily be understood from the coloured pressure field together with the vertical scale bar, as demonstrated in figure 6.

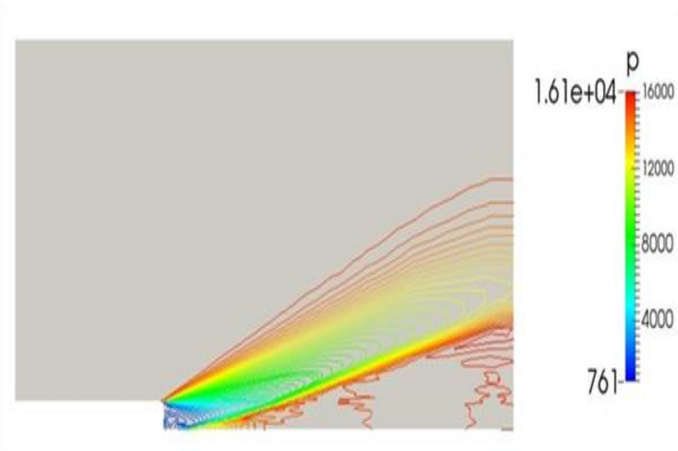

Fig 5. Pressure contour

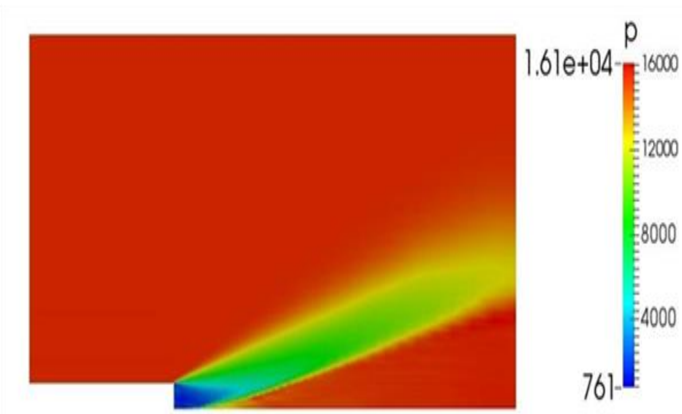

Fig 6. Pressure field

6.3. Velocity distributions

Figure 7 depicts the velocity profile along the spanwise direction relating to the supersonic turbulent fluid flow over the backward facing sharp edge step. It is quite apparent that the velocity accelerates due to sudden expansion over the sharp edge step leading to viscous layer separation and formation of shear layer which approaches to the bottom wall resulting in reattachment shock and redevelopment of boundary layer. In addition, the flow around the shear layer approaches to the bottom wall which will follow along the initial direction. However, a part of the flow reverses to the dead air region which causes it as a recirculation region. Furthermore, this circulation causes the increasing in the shear layer length which leads to flow field losses. In order to obtain the quite smooth and flawless flow the recirculation vicinity has to be minimized or eliminated. In addition, the coloured velocity vector along with the horizontal scale bar, within the fluid flow region is very well demonstrated in figure 8, which benefits for very clear understanding of the reattachment point flow physics near the bottom wall.

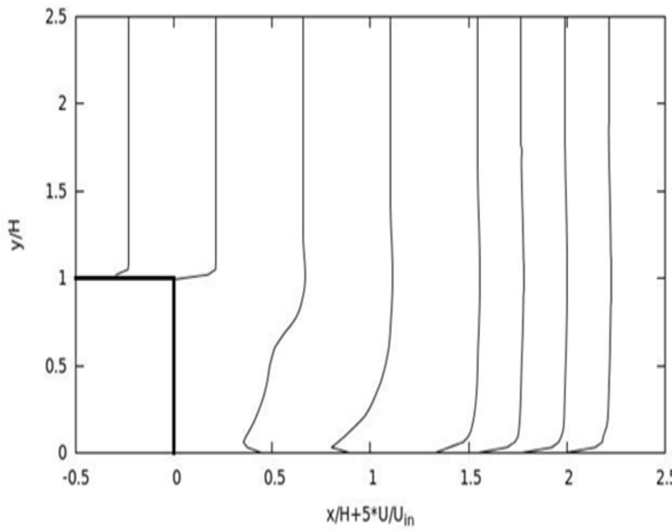

Fig 7. Velocity profile along spanwise direction

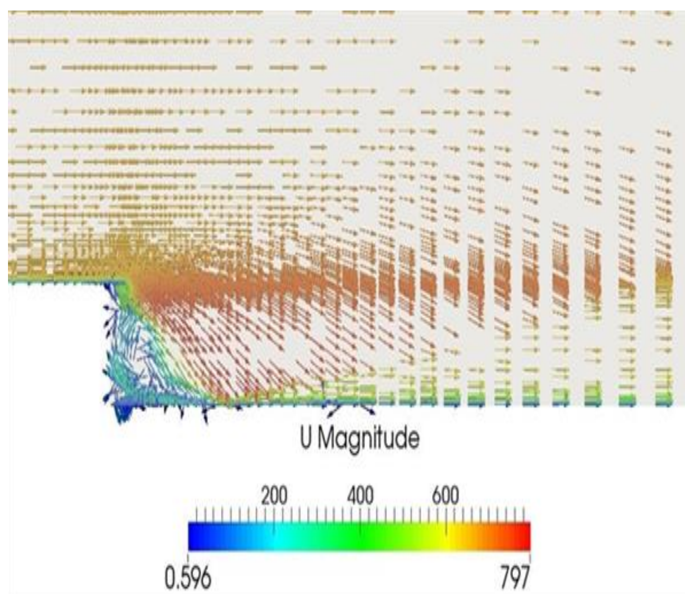

Fig 8. Velocity vector within fluid flow domain

7. CONCLUSIONS

A two dimensional numerical model is established to study the fully supersonic turbulent fluid flow characteristics over a backward facing sharp edge step. The model also involves additional key factors like production, diffusion and destruction terms besides the very usual aspects pertaining to the present physical research problem. The model is very well demonstrated for the painstaking numerical investigations on fluid flow behaviours with both the turbulent compressible fluid flow models namely RANS (associated with the standard k-ε model) and hybrid RANS-LES (relating to Spalart-Allmaras model involving viscosity-like variable ($\tilde{v}$)) by introducing the inflow free stream velocity along with the corresponding Mach number as the key model parameters. The simulation predictions from both the stated turbulent models are compared with experimental data available in literature. Eventually, the model predictions with regard to the specified key model parameters are also along the expected lines and are in very good agreement with the corresponding experimental results. However, the hybrid RANS-LES model is found to give reasonably better accuracy than the RANS model for the same computational cost and hence, only the former/first one is considered for all other studies. It is observed that the sudden viscous layer separation is the main reason for the generation of reattachment shock. Furthermore, the uneven pressure recovery is caused by sudden expansion flow over the sharp edge step. Besides, the sudden expansion flow increases the intensity of the shock which results in uneven flow characteristics. The development of recirculation region leads to shear layer formation and the reattachment length is also found in the sharp edge step. In addition, the numerical modelling for supersonic turbulent fluid flow over a backward facing round step is underway and is planned for the future to reduce recirculation region which can result in smaller shear layer formation and the shorter reattachment length, for the very similar test case conditions.

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.