Numerical simulation of supersonic separating-reattaching flow through RANS

Rahul Kumar Soni1, Nitish Arya1 and Ashoke De2
1Graduate Student, Department of Aerospace Engineering, IIT Kanpur, Kanpur, 208016, India
2Associate Professor, Department of Aerospace Engineering, IIT Kanpur, Kanpur, 208016, India

Email: ashoke@iitk.ac.in

Abstract. The present work reports turbulent supersonic flow subjected to separation-reattachment computed through Reynolds Averaged Navier-Stokes (RANS) based models. The modified density based solver, rhoCentralFoam in OpenFOAM framework, is employed for the flow simulation over a backward facing step and a rectangular cavity. The predictions by the different RANS models show good agreement with the empirical and numerical observations. Qualitatively, mean flow features such as boundary layer separation, re-circulation, and expansion fan and re-attachment shock are predicted with acceptable accuracy; while the quantitative assessment reveals the limitation of different models.

1. Introduction

Separating-reattaching flows have been a subject of various studies in the past due to the complicated flow features as well as due to the wide engineering applications. For example, backward facing steps and rectangular cavities have been used for various applications in the aerospace industry. Backward facing step being a classical problem has been widely studied for its interesting flow physics by various researchers [1-8]. In a compressible flow regime, it includes many features such as the expansion corner, flow separation recirculation zone etc. Hama (1968) in their investigation found that the flow separation occurs slightly below the corner forming a lip shock. The free stream above the corner gives rise to an expansion fan resulting in sharp pressure drop behind the step. The separated boundary layer develops into a free shear layer and on reaching the wall it is compressed and the low velocity part gives rise to a recirculation zone. The free stream layer which was earlier expanded by the corner is turned back to a direction more or less parallel to the original flow through a reattachment shock wave.

Like the backward facing step, cavities have also been known to increase the residence time of the flow inside the combustor. As put forward by various studies [9-15] the striking feature of the flow over a rectangular cavity is the shear layer, which amplifies the flow disturbance and subsequent scattering of these disturbances into acoustic waves at the downstream corner. The acoustic waves generated propagate upstream and create and excite further disturbances in the shear layer generating self-sustained oscillations at various resonant frequencies. The cavity flow may be broadly divided into four types open, closed transitional open and transitional closed as suggested by [12]. The present study only focuses on open cavity (L/D < 13).

A handful of available literature deals with the study regarding the supersonic application of RANS based turbulence models which offer lower computational cost and very much essential for the
problems of practical interests and industrial design. However, there is still a need for a systematic investigation of these flows. Therefore, the aim of the present work is to characterize the supersonic flow over the geometries subjected to separation and reattachment and simultaneously assess various RANS based turbulence models. Flow past two geometries is computed namely: backward facing step and rectangular cavity (L/D = 3). The details of these cases can be found in [5] & [11]. Both the cases, as reported widely in the literatures, pose very complicated flow physics involving boundary-layer separation, shock-boundary layer interaction and flow re-attachment. Overall, these cases appear to be ideal candidates for the assessment of the RANS based models in supersonic flow.

2. Numerical Details

2.1 Governing Equations

The Favre-averaged governing equations of fluid motion are discretized and solved using the Finite-Volume method. The equations are expressed as a set of partial differential equations (PDEs) which are derived by the application of the laws of conservation to fluid motion:

**Continuity:**

\[
\frac{\partial \rho}{\partial t} + \frac{\partial \bar{\rho}u_i}{\partial x_i} = 0
\]  

**Momentum:**

\[
\frac{\partial}{\partial t} (\bar{p} \bar{u}_i) + \frac{\partial}{\partial x_j} [\bar{p} \bar{u}_i \bar{u}_j + \bar{p} \delta_{ij} + \bar{\rho} \bar{u}_i u_{j''} - \tau_{ij}] = 0
\]  

**Energy:**

\[
\frac{\partial}{\partial t} (\bar{\rho} \bar{E}) + \frac{\partial}{\partial x_j} [\bar{p} \bar{u}_j \bar{E} + \bar{\rho} u_j \bar{p} + \bar{\rho} u_{i'} u_{j''} + \bar{q}_j - \bar{u}_i \bar{r}_{ij}] = 0
\]

Where (-) and (-) in above equation refer to time and Favre averaged quantity, \(\bar{r}_{ij} = 2\mu S_{ij} - \frac{2}{3} S_{kk} \delta_{ij}\) is considered positive in compression and strain rate tensor is, \(S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)\). Furthermore, total energy is defined as \(\bar{E} = \bar{e} + \frac{\bar{u}_k \bar{u}_k}{2} + k\); where \(\bar{e}\) and \(k\) being internal and turbulent kinetic energy respectively. The term \(\bar{\rho} u_{i'} u_{j''}\) in momentum equation leads to closure problem which upon utilizing the Boussinesq’s hypothesis is treated analogous to viscous stress and defined as \(-\bar{\rho} u_{i'} u_{j''} = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \rho k \delta_{ij}\) where \(\mu_t\) is evaluated by complementing the above set of equation with the transport equation of turbulence quantities.

For all the cases, a no-slip boundary condition is enforced at the top and bottom wall along with condition that the normal pressure gradient vanishes at wall. At the outlet, flow variables are extrapolated and non-reflecting boundary condition (NRBC) is imposed to allow outgoing waves to exit flow domain without reflection.

2.2 Numerical Scheme

Numerical simulation of high speed flow involves various challenges due to discontinuities, such as shock waves. Robust solver is required to address these issues in efficient manner. Some of the novel methods are Riemann solvers, Jacobian evaluation and characteristics decomposition. Although these methods are known to offer non-oscillatory and accurate solution but are difficult to implement especially on polyhedral mesh. The density based solver in OpenFOAM framework is based on finite volume discretization utilizing semi-discrete, non-staggered central schemes for collocated variables on polyhedral mesh. The transport equations are solved using operator-splitting approach where initially, convection of conserved variables are solved through explicit predictor equation and then diffusion of primitive variables are solved using implicit corrector equation. This approach offers the quick solution of convection terms through diagonal solver, while diffusion terms are solved implicitly enabling numerical stability. The solver utilizes central schemes proposed by Kurganov & Tadmor [16] which is an alternative approach to Riemann solver offering accurate non-oscillatory solution. In present
simulation, second order backward Euler scheme is utilized for the time integration whereas viscid and inviscid fluxes are discretised using central difference and TVD scheme. The parallel processing in OpenFOAM is achieved through the message passing interface (MPI) technique.

3. Results & Discussion

3.2 Backward Facing Step

The numerical results computed through various RANS models are compared against the experimental results. The streamwise velocity, static temperature and pressure profiles at x/h=3 are compared with experimental data, and reported in Fig. 1. Detailed result at other locations (x/h = 1.75 & 6.66) including grid independence can be found in the work of Soni et al. (2015). It is observed that the numerically predicted results match well with the experimental data for almost all the turbulence models, except in the region of y/h < 1.

It is also noticed that the results obtained by k-\(\varepsilon\) and SA are slightly different than those predicted by SST k-\(\varepsilon\) and RNG k-\(\varepsilon\) in the near wall region. The prediction of SST k-\(\varepsilon\) and RNG k-\(\varepsilon\) are in better agreement with experimental data. The boundary layer resulting from the formation of reattachment shock could be one reason for the difference observed in the pressure profile especially in case of k-\(\varepsilon\), which is known to perform poorly in adverse pressure gradient affected regions.

Streamwise velocity along the wall is presented in Fig. 1 to demonstrate the extent of reattachment length. The prediction of k-\(\varepsilon\) is lesser as compared to other models which are almost comparable. Since the experimental observation does not offer insight into the reattachment length, comparison with the numerical data is not possible. It is worth noticing that RNG and SST prediction follow each other closely. The minimum velocity prediction by S-A is lowest however other models predict close to each other.

3.3 Rectangular Open Cavity

To perform the grid independence study, two sets of grids are generated namely Grid1 and Grid2 with 75500 & 118750 cells, respectively. The normalized pressure distribution along cavity wall is presented in Fig. 2 for both grids. It can be witnessed that the solution for grid 2 appears to follow the experimental observation closely hence all the data reported in the present study corresponds to Grid2.

The velocity contours along with the streamlines for different turbulence models are presented in the Fig. 3. The recirculation region predicted by the k-\(\varepsilon\) and the RNG k-\(\varepsilon\) models does not follow the experimental observation accurately only SST k-\(\varepsilon\) model follows the observation of [11].

The Pressure distribution along the cavity wall predicted by the different turbulence models along with the experimental and numerical findings of Gruber et al. [2001] are presented in Fig. 4. The Pressure remains fairly constant along the front wall and decreases along the cavity floor. The compression of the shear layer at the rear end of the wall results in higher pressure at the rear wall of the cavity. The k-\(\varepsilon\) and RNG k-\(\varepsilon\) models show almost the same pressure distribution which is in accordance with the numerical and experimental results.

It is interesting to note that though the SST k-\(\varepsilon\) model predicts the qualitative data nicely, the pressure distribution near the wall shows small deviations from the empirical and numerical results. The deviations observed near the wall region for the SST k-\(\varepsilon\) model may be attributed to the poor performance of the wall model which could result from the shifting of the position of the first grid point from the logarithmic region of the boundary layer.
Figure 1: Comparison of numerical results at $x/h = 3$ and normalized velocity along wall

Figure 2: Normalized pressure distribution along cavity wall to demonstrate grid independence
Systematic assessment of RANS based models is carried out for turbulent supersonic flow over backward facing step and over a rectangular open cavity. For the backward facing step, the numerical and experimental results are in good agreement, except for pressure profile in the region $y/h < 1$, due to poor performance of most of the RANS based turbulence models in the separation region.

For the rectangular cavity, k-epsilon and RNG k-epsilon models under-predict the recirculation region near the front wall. Though SST k-$\omega$ model captures essential flow features associated with the cavity, there are discrepancies in the near wall predictions. Overall, SST k-$\omega$ appears to perform better in predicting the bulk flow features in the supersonic flow regime considered herein.

4. Conclusion

Figure 3: Streamwise Velocity Contours for different turbulence models

Figure 4: Normalized Pressure Distribution along the cavity wall
References
[1] Chapman D 1951 An analysis of the base pressure at supersonic velocities and comparison with experiment NACA TR 1051.
[2] Hama F R 1968 Experimental studies on the lip shock AIAA Paper 6 212-219.
[3] Donaldson I S 1967 On the separation of a supersonic flow at a sharp corner AIAA Paper 5 1086-1088.
[4] Correa S, & Warren R 1989 Supersonic sudden-expansion flow with fluid injection: An experimental and computational study AIAA Paper 89-389.
[5] McDaniel J C, Fletcher D G, & Hartfield R J Jr 1992 Staged transverse injection into mach 2 flow behind a rearward-facing-step: a 3D compressible flow test case for hypersonic combustor CFD validation AIAA Paper 1992-0827.
[6] Soni R K, Arya N, and De A 2015 Assessment of RANS based models in a supersonic flowSonip AIP Conference Proceedings 1648 030039.
[7] Arya N, Soni R K, and De A, 2015 Identification of coherent structures in a supersonic flow past backward facing step AIP Conference Proceedings 1648 030037.
[8] Arya N, Soni R K, and De A, 2015 Investigation of flow characteristics in supersonic cavity using LES American Journal of Fluid Dynamics 5 (3A) 24-32.
[9] Arunajatesan S, and Sinha N 2003 Hybrid RANS-LES modeling for cavity aeroacoustics Predictions International Journal of Computational Aeroacoustics 2 65–93.
[10] Gharib M, and Roshko A 1987 The effect of flow oscillations on cavity drag J. Fluid Mech. 177 501-530.
[11] Gruber M R, Baurle R A, Mathur T, and Hsu K Y 2001 Fundamental studies of cavity-based flameholder concepts for supersonic combustors J. Propul. Power 17 146-153.
[12] Plentovich E B, Stallings R L, Tracy M B 1993 Experimental Cavity Pressure Measurements at subsonic and transonic speeds NASA Technical Paper 3358.
[13] Rizzetta D P, Visbal M R 2003 Large-eddy simulation of supersonic cavity flow-fields including flow control AIAA Journal 41 1452–1462.
[14] Stallings R, Wilcox F 1987 Experimental cavity pressure distributions at supersonic speeds Technical Report NASA–TP–2683 NASA
[15] Zhang X, Edward J A 1988 Computational analysis of unsteady supersonic cavity flows driven by thick shear layers Aeronautical Journal 365-374.
[16] Kurganov A, Noelle S, and Petrova G 2000 Semidiscrete central-upwind schemes for hyperbolic conservation laws and hamilton – jacobi equations J. Comput. Phys 160 720–742.