Aerospace Science and Technology 58 (2016) 1–17

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Numerical study of flow physics in supersonic base-flow with mass bleed

Pratik Das, Ashoke De *

Department of Aerospace Engineering, Indian Institute of Technology, Kanpur, 208016, India

ARTICLE INFO

Article history:
Received 18 November 2015
Received in revised form 27 July 2016
Accepted 30 July 2016
Available online 4 August 2016

Keywords:
LES
Supersonic base bleed
POD

ABSTRACT

Large Eddy Simulation (LES) with dynamic sub-grid scale eddy viscosity model has been applied to numerically investigate the evolution of complicated flow structures in supersonic base flow with mass bleed. Mean flow properties obtained from numerical simulations, such as axial velocity, pressure on the base surface, have been compared with the experimental measurements to show that LES is able to predict the mean flow properties with acceptable accuracy. The data obtained from LES has been further analyzed to understand the evolution of coherent structures in the flow field. Periodical shedding of vortical structures from the outer shear layer has been observed and it has also been found that this vortex shedding is associated with the flapping of the outer shear layer. The frequency of flapping of the outer shear layer has been found out and the phase-averaged streamlines have been analyzed to further study the evolution of vortical structures associated with this flapping. The phase-averaged streamline plots clearly elucidate the evolution of vortical structures along the outer shear layer. Further, the study of these structures is investigated by performing Proper Orthogonal Decomposition (POD) analysis of the data, obtained along the central plane in the wake region. The POD results also seem to agree well with the observations made in the phase averaged streamline plots, as the concentrated energy and enstropy are observed in the outer shear layer with fewer POD modes.

1. Introduction

The aerodynamic performance of bullets, projectiles, and ballistic missiles are often compromised as they suffer massive pressure drag while traveling at supersonic speed. Due to design constraints, the design of this kind of object often features a blunt base with sharp corner similar to a cylindrical afterbody. As the turbulent boundary layer separates at the sharp corner, a low velocity, and a low-pressure recirculation region is formed which is separated from the supersonic flow outside by the compressible shear layer. This situation leads to the formation of partial vacuum just behind the base surface, thereby leading to massive pressure drag, otherwise known as base drag. Over the years, different passive techniques like boat-tailing, base cavity as well as active techniques like base burning, base bleeding etc. have been developed to reduce the base drag; yet the complicated fluid dynamics of the problem has always eluded practicing engineers and scientists from devising optimal parameters for application of these techniques. To obtain the optimized operating parameters for these techniques, it is crucial that the pressure on the base surface is predicted accurately as the key parameters are changed.

In this study, the complicated fluid dynamics associated with the base bleed technique has been numerically investigated. In this technique, a subsonic jet is injected from the base into the wake region of the cylindrical afterbody. As the low momentum fluid in the recirculation region gains momentum due to the mixing of higher momentum fluid from the jet, the recirculation region is pushed further downstream, away from the base surface and sudden rise in pressure on the base surface is also observed. How far the recirculation region is being pushed would depend on the mass flow rate of the bleeder jet, which also affects the average pressure on the base surface. The mass flow rate of the bleeder jet is quantified using a non-dimensional injection parameter (I), defined as the ratio of mass flow of the subsonic jet and the product of base area and mass flux of the free stream flow outside the recirculation region (Eq. (1)). Though the injection parameter does not take the momentum of the jet or the incoming boundary layer thickness on the after-body into account, in several previous experimental studies [1–3] on base bleed, it has been observed that the base pressure ratio has a strong dependence on the injection parameter. In mid-nineties, Herrin and Dutton [1] performed a detailed experimental study of supersonic base flow, followed by the

* Corresponding author. Fax: +91 5122597561.
E-mail address: ashoke@iitk.ac.in (A. De).

http://dx.doi.org/10.1016/j.ast.2016.07.016
1270-9638/

experiments studies performed by Mathur and Dutton [2,3]. They studied the effects of subsonic mass injection in the wake from the base on the base pressure. They have performed an extensive study of flow properties in the near wake region behind a cylindrical afterbody of 63.5 mm diameter with a bleed orifice of diameter 25.4 mm, in perfect axial alignment with a Mach 2.46 flow (approach Mach number). Their experimental facility was specifically designed to maintain the axial alignment while reducing the effects of support strings on the flow field. The experiments performed by Mathur and Dutton [2,3] provided an excellent database for validating the results obtained from CFD solvers and assessing the performance of different numerical techniques and mathematical models applied to solve this complicated problem. In their experiments, Mathur and Dutton [2,3] observed that the maximum base pressure ratio can be achieved at the injection parameter value of $I = 0.0148$.

In the past couple of decades several numerical studies of supersonic base flow have been performed [4–17], while only a few numerical studies of base bleed [18–22] were performed mostly focused on validating the experimental results obtained by Mathur and Dutton [2,3]. Sahu and Heavy [23] performed RANS (Reynolds Averaged Navier Stokes) simulation with limited success in predicting the distribution of axial velocity component along the centerline. They have also shown that the two equation k–ε turbulence model performed better than the Baldwin–Lomax algebraic model. Lee et al. [24] performed a similar RANS study while using the standard k–ω turbulence model in the Fluent solver. Bournot et al. [25] performed a numerical study of the base bleed and showed that the introduction of reactive particles in the subsonic bleeder jet may improve effectiveness by leading to higher pressure on the base surface. Shin and Choi [26] obtained good agreement with the experimental results when they performed DDES (Delayed Detached Eddy Simulation) study of the supersonic base flow and effect of the base bleed technique.

It is quite evident from the available literature that most of the previous numerical works were primarily focused on the predictions of mean flow field successfully, whereas little effort has been made to identify the evolution of flow structures in the near wake region of the base. Study of coherent flow structures in the supersonic regime is still a sparsely explored area of research. Current research is aimed towards a comprehensive study of flow structures that appear when a subsonic jet is injected from the base into the wake region of a cylindrical afterbody placed in supersonic flow. In the present study, two injection parameter values, i.e. $I = 0.0148$ and $I = 0.0226$ from the base bleed experiments of Mathur and Dutton [2,3], have been chosen as the test cases. LES of both the cases has been performed to resolve the evolution of large-scale structures in the wake region. POD in the wake region has also been carried out to successfully predict the most energetic flow structures present in the flow field. We have also compared our LES results with the experimental data obtained by Herrin & Dutton [1] for supersonic base flow without any flow control. Detailed analysis of our numerical study of base flow without flow control has already been reported and can be found in the published literature [27].

$$Injection\ parameter(I) = \frac{\dot{m}_{bleed}}{\pi R_{base}^2 \times \rho_{inf} \times U_{inf}} \tag{1}$$

$$T_{bleed} = T_{0,bleed} - \frac{\gamma - 1}{R\gamma} \times \frac{U_{bleed}^2}{2} \tag{2}$$

$$P_{bleed} = IRT_{bleed} \frac{A_{base} \times \rho_{inf} \times U_{inf}}{A_{bleed} \times U_{bleed}} \tag{3}$$

2. Numerical details

The Favre-filtered conservation equations for mass, momentum, energy and species transport are solved in the present work as used by the authors in the previous work [27–32]. To model the turbulent eddy viscosity (ν_t), LES is used so that the energetic larger-scale motions are resolved, and only the small-scale fluctuations are modeled. The sub-grid stress modeling is done using a dynamic Smagorinsky model [33,34]. For POD analysis, we have used the method of snapshots as detailed out in the literature [27,35–40] while considering the effect of compressibility through temperature correction. Usually, the aim of POD is to find out the set of orthonormal basis vectors of an ensemble of data in a lower dimension space to identify the most predominant structures in the data which are often hidden in the ensemble of the data. More details regarding the flow solver, numerical schemes, POD can be found in the literature [27–40].

2.1. Flow modeling using LES

To model the turbulence, LES is used where the large scale structures are resolved and the small scale structures are modeled. Hence, the Favre-filtered governing equations for the conservation of mass, momentum, energy and species transport are solved in the present work [27–32]. Dynamic Smagorinsky model is used for sub-grid stress modeling [33,34], where the gradient approximation is invoked to relate the unresolved stresses to resolved velocity field and given as:

$$\widetilde{u_i u_j} - \tilde{u}_i \tilde{u}_j = -2\nu_t \bar{S}_{ij} \tag{4}$$

Where

$$\nu_t = C_s^2 (\Delta)^2 |\bar{S}| \tag{5}$$

$$\bar{S}_{ij} = \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) \tag{6}$$

$$|\bar{S}| = \sqrt{2\bar{S}_{ik}\bar{S}_{ik}} \tag{7}$$

and S is the mean rate of strain. The coefficient C_s is evaluated dynamically [33,34]. More details regarding the governing equations can be found in the literature [27–32].

2.2. Numerical scheme

A density based, fully coupled FVM based solver has been used to solve the governing equations. A second order Low Diffusion Flux Splitting Scheme has been used to discretize the convective terms (Edwards [41]). All other spacial terms (i.e. diffusion terms) in the governing equations are discretized using second order central difference scheme, while the second order implicit temporal discretization is used. Moreover, the Low Mach number preconditioning (Weiss and Smith [42]) is used to effectively capture the different flow regime in the domain. The parallel processing is done using Message Passing Interface (MPI) technique. More details can be found in the literature [27–32].

2.3. Mathematical formulation used for POD analysis

The objective of POD technique is to find a set of orthonormal basis vectors of an ensemble of data in a lower dimension space such that every member of the ensemble can be decomposed relative to the orthonormal basis while minimizing the error between the ensemble and its projection on the new lower dimension space. POD technique serves two purposes: firstly, it performs reduction by projecting the higher dimensional data into a lower