Aerospace Science and Technology ••• (••••) •••–•••

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Numerical investigation of flow structures around a cylindrical afterbody under supersonic condition

Pratik Das, Ashoke De *

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

ARTICLE INFO

Article history:
Received 26 June 2015
Received in revised form 22 September 2015
Accepted 28 September 2015
Available online xxxx

Keywords:
LES
Supersonic base flow
POD

ABSTRACT

Large Eddy Simulation (LES) with dynamic Smagorinsky model has been applied to numerically investigate the complicated flow structures that evolve in the near wake of a cylindrical after body aligned with a uniform Mach 2.46 flow. Mean flow field properties obtained from numerical simulations, such as axial velocity, pressure on base surface, have been compared with the experimental measurements as well as with other published results. It has been found that standard *k*–epsilon model fails to predict the flow properties in the recirculation region where better agreement has been observed between the data obtained from LES and experimental measurements. Flow Statistics like turbulent kinetic energy and primary Reynolds' stress have also been calculated and compared with the results obtained from experiments in order to quantitatively assess the ability of LES technique to predict the turbulence properties of flow field in the highly compressible shear layer region. The data obtained from LES has been further analyzed to understand the evolution of coherent structures in the flow field. Proper Orthogonal Decomposition (POD) of the data obtained from central plane in the wake region has been performed in order to reveal the most energetic structures present in the flow field.

1. Introduction

Bullets, projectiles, missiles, launch vehicles and rockets traveling at supersonic velocity experience massive pressure drag, or otherwise known as base drag, due to the low pressure region created after flow separation behind the base of these objects. Due to design constraints often these objects feature a blunt base with sharp corner at the rear end and the geometrical features of the rear end of these objects often closely resemble with the geometrical features of a cylindrical after-body axially aligned with the flow direction. As the flow detaches at the sharp base corner, it forms a low velocity, low pressure recirculation region in the near wake of the base and the pressure on the surface of the blunt base is reduced. Over the past years several active and passive techniques, such as boat-tailing, base cavity, base bleed and base burning have been developed to increase the base pressure and reduce overall drag on the bodies traveling at supersonic speed. Yet, in order to design optimal techniques for base pressure recovery it is imperative to achieve a thorough understating of the complex fluid dynamic processes that occur in the wake region. Thus the base flow has been studied both numerically and experimentally, for a long time because of the determinant role of the design of the rear end of these objects traveling at supersonic speed in their flight capabilities. Despite having the apparently simple geometry, accurate prediction of the flow field in the near wake region of a cylindrical after body has long eluded engineers and scientists; as the several key features in the flow field such as unsteadiness in the flow field, rapid expansion near the base corner, presence of a strong compressible shear layer, presence of the recompression shock system and interaction of the turbulent flow field with the recompression shock system, leads to an extremely complicated flow physics.

In an attempt to understand the intricate physics of supersonic base flows, many experimental and numerical studies have been performed over the past few decades. Among these, most significant experimental studies have been performed by Herrin and Dutton [1]. They have performed an extensive study of mean and turbulent flow properties of the near wake region behind a cylindrical after-body of 63.5 mm diameter, in perfect axial alignment with a Mach 2.46 flow. 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 Herrin and Dutton [1] provide an excellent database for validating the results obtained from CFD (Computational Fluid Dynamics) solvers and assessing the performance of different numerical techniques and mathematical models applied to solve this complicated problem.

* Corresponding author. Tel.: +91 5122597863; fax: +91 5122597561.
E-mail address: ashoke@iitk.ac.in (A. De).

http://dx.doi.org/10.1016/j.ast.2015.09.032
1270-9638/© 2015 Published by Elsevier Masson SAS.

Nomenclature

C_p	Pressure coefficient
C_s	Smagorinsky constant
R_0	Radius of the cylindrical after-body
S_{ij}	Mean flow strain rate tensor
T	Temperature
U	Mean velocity
U_a	Axial component of mean velocity
k	Turbulent kinetic energy
p	Pressure
t	Physical time
u_i	Velocity vector
u_i'	Fluctuating component of velocity vector
Ω_{ij}	Rotation tensor
Δ	Filter size
ν	Kinematic viscosity

Super-scripts

$-$	Averaged variable

Sub-scripts

1	Free-stream condition
axial	Axial component of a vector
average	Time averaged variable
rms	Root mean square

In the past two decades several numerical studies have also been performed mostly focused on replicating the experimental data obtained by Herrin and Dutton [1]. Earlier attempts to numerically investigate the supersonic base flow were based on RANS (Reynolds Averaged Navier Stokes) based simulations with different turbulence models. Sahu [2] performed RANS study of base flow on 2D axisymmetric geometry and found that the two equation k–epsilon turbulence model was able to perform better than algebraic models. Chuang and Chieng [3] demonstrated that among three higher-order turbulence models, i.e. the low Reynolds-number form of a standard two-equation model, the two-layer algebraic stress model, and the Reynolds-stress model; the Reynolds Stress Model performed better than the other two. Benay and Servel [4] assessed the performance of two equation k–ω turbulence model in the context of supersonic base flow. Papp and Ghia [5] applied RNG turbulence model to simulate the axisymmetric base flow. Feo and Shaw [6] used commercial solver (FLUENT) to compare performance of Spalart–Allmaras, k–ω, and RSM (Reynolds Stress Models). From their study they found that RSM approach can render satisfactory results. Dharavath et al. [7] performed RANS based numerical study of massively separated flow and they observed that the renormalized group k–ε turbulence model performs better compared to the k–ω turbulence model. From all these previous studies it has been found that, though some of the RANS based turbulence models were able to predict the mean flow properties with moderate success, standard turbulence models failed to predict the flat pressure profile on the base surface and often overestimation of velocity has been observed in the near wake region. Standard turbulence models fail to predict the expansion of compressible shear layer formed after detachment of flow at the base corner. Under prediction of turbulent production term in the shear layer leads to prediction of a shorter recirculation region and eventually a much lower pressure level at the base surface. Turbulence models with compressibility correction were successful in predicting the mean flow properties but the pressure predicted on the base surface had radial variations, due to increased centerline velocity, on contrary to the flat base pressure profile observed by Herrin and Dutton [1]. Thus, the results obtained from RANS simulations have great model dependency.

With the advent of modern high performance computers in the dawn of twenty-first century, many researchers [8–14] have performed unsteady numerical simulations with advanced numerical techniques such as LES, DES (Detached Eddy Simulation) and many other hybrid techniques which require intensive computing power. Forsythe et al. [8] performed DES simulation with compressibility corrections of the base flow with similar geometry and boundary conditions. Kawai and Fujii [9,10] did a comparative study on performance of LES, MILES, RANS/LES hybrid methods. They considered two different values for Smagorinsky constant (C_s), 0.12 and 0.24 and had presented the conclusion that the higher value of Smagorinsky constant is optimal for compressible flow than incompressible flows. Simon et al. [11] compared the performance of LES, DES, and hybrid RANS/LES for the supersonic base flow case and discussed the effect of different numerical parameters relevant to hybrid methods on the results. Rodebaugh et al. [12] performed DDES calculations based on an extended k–ε RANS model to simulate various aero-propulsive flows and obtained acceptable match with the experimental data. From these afore-mentioned studies it is evident that advanced unsteady numerical simulations like LES, DES and other hybrid methods are able to predict the mean flow properties with reasonable accuracy while successfully predicting the shear layer thickness at different locations of the near wake region and the flat pressure profile on the base surface. More recently, Luo et al. [14] deployed relatively less computationally expensive 3-D PANS (partially averaged Navier–Stokes) models based on the Menter-SST turbulence model and the Wilcox k–ω model to simulate supersonic base flow and were able to obtain satisfactory results, however their study showed that the results were dependent on the resolution control parameter and increase in resolution or decrease in the value of the resolution control parameter did not always ensure improvement in the match between the computed results and the experimental data.

All of these previous studies have been primarily focused on how accurately these numerical methods are able to predict the mean flow-field in the near wake region of the cylindrical after body whereas a little effort has been made to identify the flow structures present in the turbulent wake region. Sandberg and Fasel [15] performed a DNS study of transitional supersonic base flow at a Mach no. of 2.46 where they simulated only half of the cylinder and kept the Reynolds number around 10^5 due to limitation in computational resources. In their study, they have investigated the effects of coherent structures associated with different azimuthal modes on the mean flow, in particular on the base pressure. It is quite evident from these available literatures that the most of these previous numerical works were mainly directed towards the investigation of mean flow field successfully, while a little attention was given to the evolution of flow structures in the wake region of the cylindrical after body placed in supersonic flow. Study of the flow structures in supersonic regime still remains a sparsely explored area of research. Hence, in the current work, the evolution of vortical structures in the wake of cylindrical after-body placed in a supersonic flow has been investigated; while LES is carried out in order to numerically resolve the large scale flow structures present in the wake region. We have chosen the experiment performed by Herrin and Dutton [1] as our test case for validation. Performance of RANS (k–ε two equations model) and LES have been assessed by comparing the results ob-

Please cite this article in press as: P. Das, A. De, Numerical investigation of flow structures around a cylindrical afterbody under supersonic condition, Aerosp. Sci. Technol. (2015), http://dx.doi.org/10.1016/j.ast.2015.09.032