Direct numerical simulation of a turbulent flow over an axisymmetric hill

Jony Castagnaa,

Yufeng Yaob, Corresponding author contact information, E-mail the corresponding author,

Jun Yaoc

http://dx.doi.org/10.1016/j.compfluid.2014.02.014

Highlights

We have further developed a fully 3-D high-order DNS code for a low-Re turbulent flow simulation.

We have advanced a previous LES study of same flow problem, but applying DNS approach.

We have carried out detailed study of 3d separated turbulent flow over an axisymmetric hill.

Obtained DNS (statistics) database would be useful for turbulence model development and future low-Re experiment.

Abstract

Direct numerical simulation (DNS) of a turbulent flow over an axisymmetric hill has been carried out to study the three-dimensional flow separation and reattachment that occur on the lee-side of the geometry. The flow Reynolds number is ReHReH = 6500, based on free-stream quantities and hill height (H). A synthetic inflow boundary condition, combined with a data feed-in method, has been used to generate the turbulent boundary layer approaching to the hill. The simulation has been run using a typical DNS resolution of View the MathML sourceΔx+=12.5,Δz+=6.5, and View the MathML sourceΔy1+=1.0 and about 10 points in the viscous sublayer. It was found that a separation bubble exists at the foot of the wind-side of the hill and the incoming turbulent boundary layer flow undergoes re-laminarization process around the crest of the hill. These lead to a significant flow separation at the lee-side of the hill, where a very large primary separation bubble embedded with a smaller secondary separations have been captured. The present low-Re simulation reveals some flow features that are not observed by high-Re experiments, thus is useful for future experimental studies.

Keywords

Turbulence simulation;

3D flow separation

Nomenclature

a constant

CpCp pressure coefficient

c∞c∞ speed of sound in freestream

didi distance from the inlet plane to the center of the hill axis

EtotEtot non-dimensional total energy

E edge of boundary layer

H hill height

I0I0 modified Bessel functions

J0J0 Bessel functions

K acceleration parameter

LrefLref reference length

LxLx computational domain lengths in x-direction

LyLy computational domain lengths in y-direction

LzLz computational domain lengths in z-direction

M Mach number

N nodal point

NxNx number of grid points in x-direction

NyNy number of grid points in y-direction

NzNz number of grid points in z-direction

Pr Prandtl number

P non-dimensional pressure

Re∞Re Reynolds number based on freestream quantities

ReHReH Reynolds number based on the hill height

Reδ∗Reδ∗ Reynolds number based on boundary layer displacement thickness

R radius

S saddle point

SijSij symmetric strain tensor

T non-dimensional temperature, non-dimensional time unit

T time

U+U+ streamwise mean velocity in wall unit

urmsurms turbulence intensities in x-direction

vrmsvrms turbulence intensities in y-direction

wrmswrms turbulence intensities in z-direction

uv~ Reynolds stress

uiui velocity components in x-direction

ujuj velocity components in y-direction

ukuk velocity components in z-direction

u∞u∞ freestream velocity

uτuτ friction velocity

x, y, z Cartesian coordinates

xixi Cartesian coordinates

xjxj Cartesian coordinates

xkxk Cartesian coordinates

y+y+ y coordinate in wall unit

Δx+Δx+ grid size in wall unit in x-direction

Δy+Δy+ grid size in wall unit in y-direction

Δz+Δz+ grid size in wall unit in z-direction

Δδ boundary layer thickness

Δijδij Kronecker delta

Ρρ non-dimensional density

Ττ shear stress tensor

Γγ ratio of specific heat capacity

Mμ dynamic viscosity

Nν kinetic viscosity

Ωω constant

ξ,η,ζ curvilinear coordinates

ββ constant

ΩijΩij asymmetric strain tensor

λ2λ2 second eigenvalue

1. Introduction

Three-dimensional flow separations can be found in many engineering applications, ranging from low-speed liquid flows to high-speed aerodynamic flows. In order to understand the underlying separation mechanism, experimental and numerical studies have been carried out for a turbulent flow over an obstacle, particularly a curved geometry such as a hill shape. Despite of these efforts, our current understanding is still quite limited due to the complexity of this kind of flow separation; i.e. the separation point is not fixed because of the smooth geometry surfaces. Furthermore, the separated flow contains some complicated three-dimensional flow topologies that are associated with a high-level of flow unsteadiness. These will cause further difficulties for accurately measuring the near-wall flow properties in the experiment. Recent advancement in computer power and numerical algorithm has made it possible to study this complex flow by the model-free direct numerical simulation (DNS), in which both the spatial and the temporal developments of small-scale turbulent eddies can be simulated accurately without the use of ad hoc turbulence models. This study will focus on numerical simulation of a low Reynolds number separated flow over an axisymmetric hill geometry.

A laminar boundary layer flow over an axisymmetric hill has been experimentally investigated by Ishihara et al. [1]. The obstacle was a cosine-squared cross-section and the ratio between the approaching boundary layer thickness δδ and the hill height H   was 9 and the Reynolds number 1.1×1041.1×104, based on the freestream quantities and the hill height. Despite little evidence was presented on the near-wall turbulence structures, the experiment has clearly shown that the flow separation and reattachment occurred on the lee-side of the hill. Further investigation of a turbulent flow over an axisymmetric hill was proposed and experimentally studied by Simpson et al. [2] at a high Reynolds number of 1.3×1051.3×105, based on the freestream quantities and the hill height. The ratio between the incoming turbulent boundary layer thickness and the hill height (δ/H=1/2δ/H=1/2) was much smaller than that of Ishihara et al. [1]. In Simpson's experiment, complicated vortical flow separations occur on the leeward-side of the hill and later they merge into two larger streamwise vortices downstream. By using advanced three-velocity-component laser Doppler velocimetry (LDV), in combination with other standard methods, accurate measurements of the mean velocities, the turbulence Reynolds stresses, the triple products and the skin friction in the near-wall region were obtained. The surface flow topology by the oil-flow visualization suggested the presence of multiple flow separation and re-attachment points occurring over a large area on the lee-side of the hill.

Following these earlier experiments, numerous numerical investigations have been carried out by various researchers using Reynolds-averaged Navier–Stokes (RANS) or large-eddy simulation (LES) approaches at either full or reduced experiment Reynolds number. The first numerical study of the Simpson hill problem was performed by Patel et al. [3] using LES technique. The simulated mean surface pressure, flow visualization and mean velocity profiles were presented and compared with the experiments of Simpson et al. [2]. Despite general good agreements achieved, some differences were observed, in particular the flow topology detected on the lee-side of the hill. Later, Wang et al. [4] performed a RANS study using five representative turbulence models including both the non-linear eddy-viscosity model and Reynolds-stress-transport model. Both a periodic 2D hill and a single 3D hill have been investigated in order to examine whether the predicted performance in three-dimensional conditions related to that in two-dimensional case. It was found that in the 2D hill geometry different separation behavior was predicted by different turbulent models, while for the 3D hill none of these models was able to capture the flow topology observed by Simpson et al. [2]. In fact, only a single vortex pair associated with a single separation line on the leeward side of the hill were detected. Similar conclusions were made in the work of Temmerman et al. [5], in which a second-moment-closure RANS modeling was presented. In an experiment of Byun et al. [6], measurement results from two different heights of H = δδ (small hill) and H = 2δ2δ (large hill) were presented. While the flow topology of the large hill was still quite similar to that observed by Simpson et al. [2], a single vortex pair was found for the small hill configuration, in consistency with the numerical prediction of Temmerman et al. [5]. Further experimental studies of Byun and Simpson [7] using a three-dimensional fiber-optic LDV have shown a similar flow topology to the numerical simulation [5], even for the large hill case of H = 2δ2δ, indicating that the difference with previous experimental work might be attributed to the gravity effect on the oil-flow mixture. Numerical investigations using RANS, LES and detached eddy simulation (DES) were carried out by Persson et al. [8]. Their results have shown that while the RANS fails to predict important flow features, both LES and DES are capable of reproducing the correct flow separation pattern. However, the near-wall grid resolution must be increased substantially, particularly in the spanwise direction, in order to produce correct predictions. By combining with a zonal near-wall approximation, LES study has been presented by Tessicini et al. [9]. While satisfactory results in terms of flow topology and mean quantities were achieved on a very fine grid (about 9.6 million points), simulation completely failed to capture flow separation on a coarse grid (about 1.5 million points). Patel and Menon [10] re-visited the problem by using LES approach and concluded that the separation and re-attachment processes are strongly controlled by the three-dimensional pressure gradient. A flow topology slightly different from those proposed by previous researchers was revealed, and the observed difference may be due to a low accuracy of the experimental devices employed. A recent paper by Garcia-Villalba et al. [11] examined turbulent flow separating from an axisymmetric hill-shaped obstacle by means of highly resolved large-eddy simulations with particular emphasis being placed on the separation region that is formed on the leeward side of the hill and in the near wake region. The connection between the wall topology of the mean flow and the secondary motions in the wake has been discussed and some aspects of flow unsteadiness were also analyzed, including the separation process, flow structures and coherent motions in the wake. To authors' knowledge, there is no published work on direct numerical simulation (DNS) of the hill flow, primarily due to the fact that a high Reynolds number DNS requires huge amount of computer resources that is difficult to meet even with the current HPC resource. However, simulation at a reduced Reynolds number as suggested by Patel and Menon [10] would be feasible and the results are also valuable to enhance our current understanding on this complex flow configuration. Hence, this study considers a reduced low-Re simulation and results will be compared qualitatively with those from the high-Re experiments. As a low Reynolds number flow will lead to a thicker boundary layer, this will increase the possibility of flow re-laminarization, particularly in the favorite pressure gradient region of the wind-side of the hill. while this happens, it will influence the downstream flow separations, which are the most important feature in this kind of hill flow problem.

In this study, a direct numerical simulation over an axisymmetric hill has been performed. A synthetic inflow boundary condition, combined with a data feed-in method, has been used to generate the turbulent boundary layer approaching to the hill. Simulation considers a flow Mach number of M = 0.6 and a low Reynolds number of ReHReH = 6500; i.e. 5% of the experimental conditions (see [2]); the former can permit larger time step for simulation using a compressible flow solver and the latter will significantly reduce the required computer resources. Simulation results at the low-Re condition will be compared with the high-Re experiment of Simpson et al. [2] and Byun and Simpson [7] qualitatively such as the flow topologies. Some quantitative turbulence properties will be presented as well.

2. Governing equations and mathematical formulation

2.1. Governing equations

The non-dimensional form of three-dimensional (3D) compressible Navier–Stokes equations in the Cartesian coordinate system can be written as:

equation(1)

View the MathML source$\frac{\partial \rho}{\partial t}+\frac{(\partial \rho u_i)}{\partial x_i}=0$,

Turn MathJax on

equation(2)

View the MathML source$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{1}{Re_\infty}\frac{\partial \tau_{ij}}{\partial x_j}$,

Turn MathJax on

equation(3)

View the MathML source∂Etot∂t+∂[(Etot+p)uj]∂xj=-1(γ-1)Re∞PrM2∂∂xjμ∂T∂xj+1Re∞∂uiτij∂xj,

Turn MathJax on

where View the MathML sourcexi,xj are the Cartesian coordinates and t is the time variable. The freestream quantities are used for dimensionalization and the non-dimensional physical variables are ρρ the density, View the MathML sourceui,uj the velocity components, p the static pressure, T the static temperature, and EtotEtot the total energy. μμ is the molecular viscosity of a fluid (calculated by using the power law μ=Tωμ=Tω with ω=0.76ω=0.76, same as previous studies using the same code [12]). The View the MathML sourceRe∞,Pr, and M are the Reynolds number, the Prandtl number and the Mach number, respectively with Re∞=ρ∞u∞Lref/μ∞Re∞=ρ∞u∞Lref/μ∞, where LrefLref is a reference length and M=u∞/c∞M=u∞/c∞, where c∞c∞ is the freestream sound speed). γγ is the ratio of specific heat capacity. An ideal gas law is used and for the air, γ=1.4γ=1.4 and Pr=0.72Pr=0.72.

The Newtonian fluid assumption is taken and this determines the correlation between the shear stress tensor τijτij and the mean velocity field, i.e.

equation(4)

View the MathML sourceτij=μ∂ui∂xj+∂uj∂xi-23δij∂uk∂xk.

Turn MathJax on

While a body-fitted coordinate system (ξ,η,ζξ,η,ζ) is applied, the above governing equations can be re-written in a general compact vector form and during a 3D curvilinear Jacobian transformation process, some unexpected numerical errors could be introduced when a central finite-difference scheme is applied [13]. The source of these numerical errors is mainly due to the non-cancelation of some additional terms introduced from the grid transformation. A few methods have been proposed in order to make sure the conservation of equations [13] and [14]. Here, an alternative treatment was proposed by present authors to implicitly ensure this condition. Validation of this approach can be found in a reference paper [15].

2.2. Simulation code and its numerical features

A direct numerical simulation (DNS) code was developed for analyzing the shock-wave and boundary layer interactions problem (named SBLI thereafter) [12]. The code uses a 4th-order central difference scheme for evaluating the spatial derivatives at interior points and a sTable 3rd-order scheme, based on the summation by parts (SBP) approach [16], for derivative calculations at boundary points. For the time integration, a 3rd-order explicit multi-stage Runge–Kutta algorithm is applied. In addition, an entropy splitting concept is used to enhance the numerical stability [17] and [18], and it is important for a DNS code that normally requires an extremely longer run time to reach statistically converged results. The code parallelization uses the MPI library and the scalability and portability have been tested on various high-performance computing platforms with very good linear performance. Although the SBLI code was initially developed for numerical simulation of transonic flow over a bump geometry [12] and later used to simulate a similar problem of an oblique impinging shock interacting with a spatially-developing boundary-layer flow [19], the code has proved to be remarkably adaptable and its variants (including a quasi-three-dimensional curvilinear multi-block version) have been successfully used for simulations of a broad range of transitional and turbulent flows, including supersonic turbulent channel flow, plane jet aero-acoustics, shock-wave and turbulent spot interactions, transitional heat transfer, transonic cavity flow, trailing-edge noise calculations, and most recently separation bubbles on an airfoil at incidence.

A full-three-dimensional curvilinear multi-block version of the code has been developed by using a global mapping technique for meshing in each individual domains and pre-processing for defining the interfaces between adjacent blocks [15] and [20]. The code validations are presented with some selected test cases including a pulse single traveling through an interface of different grid density, a freestream preservation on three-dimensional wavy grids, a laminar channel flow on a skewed grids, and stationary vortex development on a wavy grid. The code parallel performance is also shown on HPC platform using 1000 + processors. More details can be found in Yao et al. [21].

3. Turbulent flow over an axisymmetric hill

3.1. Problem definition, computational domain and boundary conditions

The problem considers a fully developed turbulent boundary layer flow over an axisymmetric hill geometry placed on a flat plate at a location $8.4H$ from the inlet (with H the hill height), where the boundary layer thickness at the hill wind-side foot is about half the hill height, in absence of the hill geometry. The flow Reynolds number is 6500 based on the hill height, this equivalent to about 500 based on the boundary layer displacement thickness at the hill foot, which is about a factor of 6.5 to the boundary layer thickness. Comparing to experiments of Simpson et al. [2] and Byun and Simpson [7], present simulation considers a reduced Reynolds number about 5% of the test value. The purpose is to carry out a proper resolved DNS study and at meantime to reveal consequence flow structure changes at this low Reynolds number. Fig. 1 gives a sketch of the hill geometry, where H is the hill height. The hill shape is defined by Simpson et al. [2] as:

equation(5)

View the MathML source$y(r)H=-\beta[J0(\Lambda)I0(r\Lambda/a)-I0(\Lambda)J0(r\Lambda/a)]$,

Turn MathJax on

where $y(r)y(r)$ is the shape function of the radius r . The coefficients are $\beta\beta$ = 1/6.048, $\Lambda\Lambda$ = 3.1926, a = $2H$. The $J0J0$ and $I0I0$ are the Bessel and modified Bessel functions, respectively. This formula results in a hill geometry of H in height at the center and $2H2H$ in radius at the ground surface.

Full-size image (12 K)

Fig. 1.

A sketch of turbulent flow over an axisymmetric hill geometry.

The computational domain has a dimension of View the MathML sourceLx,Ly,Lz in the streamwise, the wall normal and the spanwise directions, respectively. The distance between the center of the hill and the domain outlet is 11.6H11.6H, resulting in the streamwise domain length of Lx=20HLx=20H. The domain height is Ly=2.305HLy=2.305H and the domain width is Lz=10HLz=10H, which is sufficient to contain a wide range of turbulent fluctuation frequencies.

The dimensions of physical flow domain used in present and previous studies are summarized in Table 1. It can be seen that comparing to that of Wang et al. [4] and Tessicini et al. [9], present simulation considers a larger computational domain. Furthermore, the hill height H is twice the turbulent boundary layer thickness at the hill foot, in absent of the hill geometry; i.e. H=2δH=2δ. Based on a previous DNS study [22], a fully developed turbulent boundary layer has thickness about 6.5δ∗6.5δ∗. Thus the streamwise, wall normal and spanwise lengths are about 260δ∗δ∗, 42δ∗δ∗ and 130δ∗δ∗, respectively, which are sufficiently wider to contain all scales of turbulent eddy motions at this low Reynolds number [23]. Note that Patel et al. [3] applied a shorter spanwise length of 6H6H in their simulation and Persson et al. [8] used a moderate length of 10H10H, same as that used in the present simulation.

Table 1.

Computational domain size in comparison with those used by previous researchers.

Test case	LxLx	LyLy	LzLz	didia
Present DNS	20H	3.205H	10H	8.4H
Patel et al. [3]	9.5H	3.205H	6H	3.4H
Wang et al. [4]	16H	3.205H	11.67H	4.4H
Persson et al. [8]	12H	3.205H	10H	3.4H
Tessicini et al. [9]	16H	3.205H	11.67H	4.4H

a

didi is distance from the inlet plane to the center of the hill axis.

Uniform grids are used in both the streamwise and the spanwise directions, while a hyperbolic sinusoidal stretching function is applied in the wall-normal direction, similar to that used in a boundary layer flow over a channel bump [12]. Table 2 gives the number of grid points and the grid resolution in wall unit. Comparing to previous study using the same computer code [18], it is clear that present DNS have suitable grid resolutions; e.g. about 10 points in the sublayer (y+=10y+=10) and grid resolution in the streamwise and the spanwise directions are similar to previous DNS studies [12]. These estimations are based on the boundary layer properties of a prescribed inflow profile.

Table 2.

Computational grid and grid resolution of test case presented.

Test case	(Nx,Ny,NzNx,Ny,Nz)	Δx+Δx+	View the MathML sourceΔymin+	View the MathML sourceΔymax+	Δz+Δz+
Present DNS	(601, 161, 561)	12.17	1.06	25.37	6.52

The boundary conditions are defined as follows: a no-slip condition for the wall and the hill surfaces; a fully-developed turbulent boundary layer at the inlet plane, generated by an inflow data feed-in from a precursor simulation; a non-reflecting characteristic condition [24] at the outlet plane and upper surface; and a periodic condition for two side walls in the spanwise direction.

The inflow boundary layer has been generated using a precursor turbulent boundary layer simulation, obtained on a minimal computational domain in order to reduce the computational cost. After statistically convergence being achieved, the instantaneous flow field obtained near the exit plane is saved in a time sequence of data slice and later fed into the hill domain through an interpolation in time and space (see Fig. 2). The inlet condition is generated using the digital filter technique. This is based on the digital filter concept, initially proposed by Klein et al. [26], later further improved by Xie and Castro [27] and recently adapted to compressible flow by Touber and Sandham [28]. Details on the precursor simulation are given in the next section.

Full-size image (15 K)

Fig. 2.

Sketch of the interface between a precursor boundary layer simulation and hill flow simulation.

3.2. Precursor turbulent boundary layer simulation

A precursor turbulent boundary layer flow simulation at $Re\delta*=500Re\delta*=500$, same flow conditions as those for the hill flow simulation is carried out to generate the inlet conditions. The precursor simulation domain is $50\delta*\delta*$, $10\delta*\delta*$ and $8\delta*\delta*$ in the streamwise (x), the wall normal (y) and the spanwise (z) directions, where the $\delta*\delta*$ is the displacement thickness of the boundary layer at the inlet. A grid of $119\times61\times71119\times61\times71$ points is used, uniformly distributed in the View the MathML sourcex,z directions and stretched in the y direction. Based on the inflow conditions, the estimated grid resolutions are View the MathML source$\Delta x+=11.09,\Delta z+=6.24$ and the first point in the wall normal direction is about View the MathML source$\Delta y1+=0.99$ with a total of 10 points in the viscous sublayer region. A minimal domain in the spanwise direction with $Lz=8\delta*Lz=8\delta*$, large enough for the development of a turbulent boundary layer [25], has been used in order to reduce the computational cost. A two-point correlation is often used to find out the minimum width required for the computation and this has been confirmed in present simulations [15].

A synthetic approach of the digital filter technique [26] has been used. In this method, turbulent boundary layer properties (both the mean velocity and the Reynolds stress profiles) are prescribed, along with artificially generated random numbers as white noise. A filter procedure is then used to retain the appropriate values to reproduce the desirable turbulent correlations for given conditions.

The precursor simulation starts with a uniform flow field and after initial transient stage of about 100 time units (i.e. 2 through-flows), statistic samples are collected for every 100 time units until the ‘statistical’ convergence status achieved. This normally takes about 400 time units; i.e. about 8 through-flows.

The mean velocity in wall units, $U+U+$, the RMS of the turbulence intensities ($urms,vrms,wrmsurms,vrms,wrms$) and the Reynolds stress ($\langle uv\rangle \langle uv\rangle$) variations are shown in Fig. 3 and Fig. 4, where $y+y+$ is the y -coordinate in wall unit and $\delta\delta$ is the boundary layer thickness and the variables are normalized by the friction velocity $u\tau u\tau$. The results are compared with the benchmark DNS data of Spalart [23] at the same Reynolds number. It can be seen that the overall agreements are quite good, with slightly over-prediction of the log-layer slope and the streamwise turbulence intensity $urmsurms$.

Full-size image (10 K)

Fig. 3.

Mean streamwise velocity U+U+ at x/δ∗x/δ∗ = 45 and z/Hz/H = 0.

Full-size image (16 K)

Fig. 4.

Turbulence intensities at x/δ∗x/δ∗ = 45 and z/Hz/H = 0.

The simulation results at a position of x/δ∗≃45x/δ∗≃45 are saved at every 5-iteration intervals. This will result in a time sequence of two-dimensional instantaneous flow at a cross-section. In general, the precursor simulation has smaller cross-section, and the data needs to be manipulated by copying onto the inlet plane of the main simulation domain. The uniform freestream conditions are applied for the region above the upper surface of the precursor simulation (i.e. y>10δ∗y>10δ∗ in present study). Validation of this approach have been given in a previous study [15], in which a comparison between results obtained using the digital filter along the full inlet plane and those obtained using the above method are presented.

3.3. Results and discussion

The precursor simulation uses flow Reynolds number of Reδ∗=500Reδ∗=500, based on a unity boundary layer displacement thickness at the inlet. It was found that at the data export plane, the boundary layer thickness is about δ=6.5δ∗δ=6.5δ∗, indicating that for the hill simulation, the flow Reynolds number is about ReH≃6500ReH≃6500 based on the hill height H that is twice the boundary layer thickness δδ. This means that the present simulation Reynolds number is about 1/20 of the experiment Reynolds number; i.e. ReH=1.3×105ReH=1.3×105. After an initial transient period of about 300 time units (about 1.2 through-flows), statistical data are collected for further 900 time units for data analysis.

3.3.1. Flow structure and topology

Fig. 5 shows the flow separation boundaries superimposed with velocity vector plots. The simulation predict flow separations at the windward-side hill foot and around the crest of the hill, including an embedded secondary separation region. Note that dash line is from present DNS and solid line is from another DNS on a slightly smaller domain [15]. Fig. 6a gives a close-up view of the upstream windward-side hill foot, where a small recirculation region, is clearly visible. This flow recirculation will cause further thickening of the boundary layer while approaching the hill. Another close-up view of the leeward-side of the hill near the crest (see Fig. 6b) also reveals a small secondary flow recirculation bubble embedded inside a large primary flow recirculation bubble. Despite very small in size, this bubble seems to have some considerable influence on the flow separation behavior. The small pressure increment observed near the crest of the hill may also be related to the presence of this small bubble.

Full-size image (17 K)

Fig. 5.

Predicted separation bubble size and shape. Dashed line: present DNS, solid line: reference data [15].

Full-size image (41 K)

Fig. 6.

The secondary flow recirculation regions detected at the windward side of the hill foot and the leeward side of the hill near the crest (z/Hz/H = 0).

Fig. 7 gives the streamlines from the simulation. The flow recirculation increases in size and also separates earlier after the crest of the hill and reattaches further downstream at about x/Hx/H = 4.14. The center of the primary recirculation is at View the MathML sourcex/H=2.35,y/H=1.06, while the center of the small recirculation bubble which appears on the leeward-side of the hill is at x/Hx/H = 0.63 and y/Hy/H = 0.81. The flow topology analysis has identified four saddle points (see Fig. 8); i.e. View the MathML sourceS1,S2,S3 and S4S4 along the mid-plane (z = 0) at x/Hx/H = −2.4, x/Hx/H = −0.072, x/Hx/H = 0.93 and x/Hx/H = 4.25, respectively, and four nodal points (two for attachment and two for separation) as View the MathML sourceNa1,Na2 at the mid-plane and x/Hx/H = −1.6, x/Hx/H = 0.17, and Ns1Ns1 and Ns2Ns2 at x/Hx/H = 1.53 and z/Hz/H = ±±0.7, respectively. In the high-Re experiment of Simpson et al. [2], only one pair of nodal and saddle separation points were observed. However, in the present low-Re simulation, two pairs of nodal and saddle separation points are predicted. Overall, the flow topology satisfies the well-known rule of thumb (see Lighthill [29]) as:

equation(6)

$\sum N-\sum S=0,\sum N-\sum S=0,$

Turn MathJax on

where N and S are the number of nodal and saddle points, respectively.

Full-size image (21 K)

Fig. 7.

Streamline for the hill case of normal and vertical velocity at the middle plane (z=0z=0).

Full-size image (32 K)

Fig. 8.

Flow topologies in the near wall at y+y+ = 1.06.

The wall surface flow pattern in the lateral direction is visualized by skin friction lines as seen in Fig. 9. It shows that the flow circulates around the hill, and converges into a counter-rotating vortex pair (CRVP) on the leeward-side of the hill. Comparing the flow topology in Fig. 9 with the results obtained at high Reynolds number by Byun and Simpson [7] (see Fig. 10), the following differences have been observed: (a) for low Reynolds number simulation, the location of the foci separation points is much lower (close to the hill surface). Moreover, the intensity of the recirculation is very weaker, with the streamlines rotating only in the proximity of the separation point; (b) the saddle points are located further downstream, indicating a much larger recirculation zone than that observed by Byun and Simpson; (c) for high Reynolds number experiment, no separation or attachment points are observed at the crest the hill. An arbitrary streamline trace is presented in Fig. 11 and colored with the spanwise velocity magnitude. It illustrates a 3D flow recirculation path; i.e. starting from point 1 upstream of the hill, the flow circulates around the hill until point 2 and moves back towards the top of the hill (point 3); after that location, it convects downstream on the leeward-side of the hill, having gone through the CRVP (thus undergoes a 360° rotation) towards the bottom of the hill (point 4) and finally leaves the recirculation region and merges with the streamwise mean flow at point 5 downstream.

Full-size image (24 K)

Fig. 9.

Skin friction lines on the hill surface.

Full-size image (40 K)

Fig. 10.

Mean streamlines in the experiments of Byun and Simpson [7].

Full-size image (19 K)

Fig. 11.

An arbitrary mean velocity streamline colored by the spanwise velocity. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

3.3.2. Statistics of mean flow

Fig. 12 shows the computed mean surface pressure coefficient CpCp distributions. Downstream in the wake, the pressure recover to a higher value than inlet, due to the large recirculation which slow down the streamwise velocity, and also there are two low pressure regions on the leeward side of the hill at about View the MathML source$x/H=1.56,z/H=\pm0.74$. Moreover, there is a slightly higher pressure area just after the crest of the hill which, as explained later, will play the role of ‘pumping’ more fluids toward the hill center. These flow features are in qualitatively good agreement with those observed by Byun and Simpson [7] in the high-Re experiments.

Full-size image (29 K)

Fig. 12.

Predicted surface mean pressure coefficient CpCp distributions in the x-zx-z plane. Note that four circles at a diameter of 0.5, 1, 1.5 and 2 indicate different levels of the hill height.

Fig. 13 gives the streamwise CpCp distributions along the centerline of the hill z/H=0z/H=0. It can be seen that the pressure coefficient has a maximum value around 0.50.5 at the foot the hill (x/Hx/H = −2); i.e. the stagnation point. It decreases to a minimum value around -0.3-0.3 around the crest of the hill and recovered to about x/D=4.14x/D=4.14 in far-wake region. The small pressure increment just after the crest is also clearly shown. Comparing to the experiment of Byun and Simpson [7], good agreements are visible in upstream region up to x/H=-1x/H=-1, and then simulation result departs from the test data. The experiment reveals stronger flow acceleration and deceleration around the crest of the hill, while the far-field pressure decreases.

Full-size image (13 K)

Fig. 13.

Streamwise CpCp distributions along the centerline (z=0z=0) in comparison with the experimental data of Simpson et al. [2].

The streamwise mean velocity profile U+U+ along the wall normal direction y+y+ is shown in Fig. 14. The predicted boundary layer thickness is about δ=6.34δ∗δ=6.34δ∗ at the inlet plane. As discussed above, the presence of a small separation bubble at the windward side of the hill foot about x/H=-2x/H=-2 will cause a rapid reduction of the friction velocity at x/H=-3x/H=-3 by almost a half in comparison to the value at the inlet plane. Hence, at the edge of the boundary layer, the magnitude of the mean velocity in wall unit (normally evaluated by local boundary layer properties) is almost twice of the value at the inlet plane. At x/H=-1x/H=-1, the friction velocity recovers and further increases, leading to a lower u+u+ value. The appearance of a hill shape will also cause adverse effect on the boundary layer development.

Full-size image (14 K)

Fig. 14.

Mean streamwise velocity U+U+ along the vertical direction at different streamwise locations. The value refer to the middle plane (z=0z=0) and are compared with the law of the wall.

Fig. 15, Fig. 16 and Fig. 17 are the mean velocity profiles along the wall-normal direction. The velocities are normalized by the reference velocity u∞u∞ and presented at four streamwise locations of x/Hx/H = −2 (at wind-side of the hill foot); x/Hx/H = 0 (on the crest of the hill); x/Hx/H = 2 (at lee-side of the hill foot) and x/Hx/H = 4.14 (at the ”mean” reattachment location), respectively. A total of six spanwise locations of View the MathML sourcez/H=0,0.5,1,1.5,2,2.5 are given to illustrate the profile changes along the lateral direction.

Full-size image (51 K)

Fig. 15.

Streamwise mean velocity along the vertical direction at several spanwise locations of (a) x/Hx/H = −2, (b) x/Hx/H = 0, (c) x/Hx/H = 2 and (d) x/Hx/H = 4.14, respectively.

Full-size image (51 K)

Fig. 16.

Wall-normal mean velocity along the vertical direction at several spanwise locations of (a) x/Hx/H = −2, (b) x/Hx/H = 0, (c) x/Hx/H = 2 and (d) x/Hx/H = 4.14, respectively.

Full-size image (49 K)

Fig. 17.

Spanwise mean velocity along the vertical direction at several spanwise locations of (a) x/Hx/H = −2, (b) x/Hx/H = 0, (c) x/Hx/H = 2 and (d) x/Hx/H = 4.14, respectively.

3.3.3. Turbulence intensities

Fig. 18, Fig. 19 and Fig. 20 give the root-mean-square (RMS) turbulence intensities and the Reynolds stress (normalized by the friction velocity uτuτ and averaged in the spanwise direction) at three successively streamwise locations; i.e. the inlet plane (x/Hx/H = −8.4); flow reattachment location (x/Hx/H = 4.14) and the outlet plane x/Hx/H = 11.6, respectively, all along the centerline z/Hz/H = 0. At the inlet the predictions are in good agreement with the DNS of Spalart [23], but downstream there are some slight differences, in consistent with that seen in Fig. 14. The reason for this is probably due to the rapid decrease of the friction velocity, as a result of the presence of a small separation bubble at x/H=-2x/H=-2. At the reattachment point, the values are much larger than that of the inflow. The peak value for the urmsurms quantities is not close to wall surface, probably due to the strong turbulence activity behind the hill and the counter rotating vortex pair merging with the freestream. Finally, the RMS values at the exit of the domain are similar in magnitude to the incoming boundary layer, but the thickness of the boundary layer is much larger (nearly twice, i.e. equal to the hill height).

Full-size image (16 K)

Fig. 18.

Turbulence intensity and Reynolds shear stress distributions at x/Hx/H = −8.4 and z/Hz/H = 0 in comparison with Spalart DNS data.

Full-size image (16 K)

Fig. 19.

Turbulence intensity and Reynolds shear stress distributions at x/Hx/H = 4.14 and z/Hz/H = 0 in comparison with Spalart DNS data.

Full-size image (16 K)

Fig. 20.

Turbulence intensity and Reynolds shear stress distributions at x/Hx/H = 11.6 and z/Hz/H = 0 in comparison with Spalart DNS data.

3.3.4. Coherent flow structures

Fig. 21 gives the structures of instantaneous flow field illustrated by the iso-surfaces of the second invariants λ2=0.035λ2=0.035 of the symmetric tensor of instantaneous velocity flow field (see Jeong and Hussain [30]); i.e. λ2=S2+Ω2λ2=S2+Ω2 where View the MathML sourceSij¯=(1/2)(∂ũi/∂xj+∂ũj/∂xi) is a symmetric tensor and View the MathML sourceΩij¯=(1/2)(∂ũi/∂xj-∂ũj/∂xi) is anti-symmetric tensor. It can be seen that the flow structures occur in a larger area in the lee-side of the hill (consistent with larger separation bubble), but over the top of the hill they seem not presented. This indicates the flow may be partly re-laminarized in that local region. Further discussions can be seen below.

Full-size image (26 K)

Fig. 21.

Visualization of 3D flow structures using the criteria of second eigenvalue at a value of λ2λ2 = 0.035.

3.3.5. Flow re-laminarization around the hill crest

Due to a low Reynolds number used in the simulation and the strong favorable pressure gradient at the windward side of the hill, re-laminarisation may occur. In a previous study by Sandham et al. [12] for a turbulent flow over a bump geometry, the flow re-laminarisation issue has been studied by using a criteria related to a general acceleration parameter [31] as

equation(7)

View the MathML sourceK=veUe2∂Ue∂x,

Turn MathJax on

where the subscript ‘e ’ represents the boundary layer edge and the criterion for flow laminarisation occurs roughly for a value of K>3×10-6K>3×10-6. For the present hill simulation, the K value and its variation can be evaluated at the upper boundary of the domain (i.e. y=3.205Hy=3.205H). The results shown in Fig. 22 confirm that the K value has indeed exceeded the given laminarisation criterion. It is assumed that the flow laminarisation contributes to the larger separation bubble seen in Fig. 5. Fig. 23, Fig. 24 and Fig. 25 are shown the turbulent intensities on the windward-side of the hill obtained without averaging the results in the spanwise direction. The values are reduced while the prediction location moves towards the hill, confirming the laminarisation process.

Full-size image (7 K)

Fig. 22.

Laminarisation parameter K variations along the streamwise location and middle plane (z=0z=0) at y=3.205Hy=3.205H.

Full-size image (13 K)

Fig. 23.

Turbulent intensities at x/H=-1.5x/H=-1.5 and z/Hz/H = 0. The values are not normalized.

Full-size image (11 K)

Fig. 24.

Turbulent intensities at x/H=-1x/H=-1 and z/Hz/H = 0. The values are not normalized.

Full-size image (11 K)

Fig. 25.

Turbulent intensities at x/H=-0.5x/H=-0.5 and z/Hz/H = 0. The values are not normalized.

4. Conclusions

Direct numerical simulation of a turbulent flow over an axisymmetric hill shape is performed. Simulation considers a low Reynolds number 6500, which is about 1/20 of the experiment condition. Despite this difference, simulations are still able to capture some dynamic flow behaviors and the key flow topologies in the separation region, in good qualitative agreement with the high-Re test results. However, the predicted flow separation bubble at present low-Re seems much larger in size compared to that observed in the high-Re experiments by Simpson et al. [2] and Byun and Simpson [7]. The simulation also reveals a small separation at the windward-side of the hill near the hill foot and a secondary separation bubble embedded inside a large primary bubble at the leeward-side of the hill. It is found that the flow re-laminarization occurs in the vicinity of the crest of the hill. Consequently, this will alter the flow development and contribute to the earlier flow separation and a larger bubble formed immediately after the hill crest. These findings are useful for future experimental validation study at low Reynolds number.

Acknowledgement

The first author would like to acknowledge the finance support from the Kingston University London, while studying as a graduate student there. The computational time resource is supported by the UK Turbulence Consortia under the EPSRC grant (EP/D044073/1).

References

[1]

T. Ishihara, K. Hibi, S. Oikawa

A wind tunnel study of turbulent flow over a three-dimensional steep hill

J Wind Eng Ind Aerodynam, 83 (1999), pp. 95–107

[2]

R.L. Simpson, C.H. Long, G. Byun

Study of vortical separation from an axisymmetric hill

Int J Heat Fluid Flow, 23 (5) (2002), pp. , 582–591

[3]

Patel N, Stone C, Menon C. Large-eddy simulation of turbulent flow over an axisymmetric hill. In: The 41st AIAA aerospace sciences meeting & exhibit, Reno, NV. AIAA 2003-0967.

[4]

C. Wang, Y.J. Jang, M.A. Leschziner

Modelling two- and three-dimensional separation from curved surfaces with anisotropy-resolving turbulence closures

Int J Heat Fluid Flow, 25 (3) (2004), pp. 499–512

[5]

Temmerman L, Chen W, Leschziner MA. A comparative study of separation from a three-dimensional hill using LES and second-moment-closure RANS modeling. In: Fourth European congress on computational methods in applied sciences and engineering, ECCOMAS 2004, Finland; 2004.

[6]

G. Byun, R.L. Simpson, C.H. Long

Study of vortical separation from three-dimensional symmetric bumps

AIAA J, 42 (4) (2004), pp. 754–765

[7]

G. Byun, R.L. Simpson

Structure of three-dimensional separated flow on an axisymmetric bump

AIAA J, 44 (5) (2006), pp. , 999–1008

[8]

T. Persson, M. Liefvendahl, R.E. Bensow, C. Fureby

Numerical investigation of the flow over an axisymmetric hill using LES, DES, and RANS

J Turbul, 7 (1) (2006), pp. , 1–17

[9]

F. Tessicini, N. Li, M.A. Leschziner

Large-eddy simulation of three-dimensional flow around a hill-shaped obstruction with a zonal near-wall approximation

Int J Heat Fluid Flow, 28 (5) (2007), pp. , 894–908

[10]

N. Patel, S. Menon

Structure of flow separation and reattachment behind an axisymmetric hill

Int J Heat Fluid Flow, 24 (3) (2007), pp. 372–388

[11]

M. Garcia-Villalba, N. Li, W. Rodi, M.A. Leschziner

Large-eddy simulation of separated flow over a three-dimensional axisymmetric hill

J Fluid Mech, 627 (2009), pp. 55–96

[12]

N.D. Sandham, Y.F. Yao, A.A. Lawal

Large-eddy simulation of transonic turbulent flow over a bump

Int J Heat Fluid Flow, 24 (4) (2003), pp. 584–595

[13]

T.H. Pulliam, J.L. Steger

Implicit finite-difference simulations of three-dimensional compressible flow

AIAA J, 18 (2) (1980), pp. 159–167

[14]

P.D. Thomas, C.K. Lombard

Geometric conservation law and its application to flow computations on moving grids

AIAA J, 17 (10) (1979), pp. 1030–1037

[15]

Castagna J. Direct numerical simulation of turbulent flows over complex geometries. PhD thesis. Faculty of Engineering, Kingston University London; 2010.

[16]

M.H. Carpenter, J. Nordstrom, D. Gottlieb

A stable and conservative interface treatment of arbitrary spatial accuracy

J Comput Phys, 148 (2) (1999), pp. 341–365

[17]

H.C. Yee, N.D. Sandham, M.J. Djomehri

Low-dissipative high-order shock-capturing methods using characteristic-based filters

J Comput Phys, 150 (1) (1999), pp. 199–238

[18]

N.D. Sandham, Q. Li, H.C. Yee

Entropy splitting for high-order numerical simulation of compressible turbulence

J Comput Phys, 178 (2) (2002), pp. 307–322

[19]

Y.F. Yao, K.L. Narasimhan, N.D. Sandham, G.T. Roberts

The effect of Mach number on unstable disturbance in shock/boundary layer interactions

Phys Fluids, 19 (5) (2007), p. 054104

[20]

J. Castagna, Y.F. Yao

Multi-block high-order DNS code development for turbulent jet in cross-flow simulation

J Algorithm Comput Technol, 6 (4) (2012), pp. 593–622

Full Text via CrossRef

[21]

Yao YF, Shang Z, Castagna J, Sandham ND, Johnstone R, Sandberg RD, et al. Re-engineering a DNS code for high-performance computation of turbulence flows. AIAA 2009-0566.

[22]

Y.F. Yao, T.G. Thomas, N.D. Sandham, J.J.R. Williams

Direct numerical simulation of turbulent flow over a rectangular trailing edge

Theor Comput Fluid Dyn, 14 (5) (2001), pp. 337–358

[23]

P.R. Spalart

Direct simulation of a turbulent boundary layer up to Reθ=1410Reθ=1410

J Fluid Mech, 187 (1988), pp. 61–98

[24]

T.J. Poinsot, S.K. Lele

Boundary conditions for direct simulations of compressible viscous flows

J Comput Phys, 101 (1992), pp. 104–129

[25]

J. Jimenez, P. Moin

The minimal flow unit in near-wall turbulence

J Fluid Mech (1991), p. 225

[26]

M. Klein, A. Sadiki, A. Janicka

A digital filter based generation of inflow data for spatially developing direct numerical or large eddy simulations

J Comput Phys, 186 (2003), pp. 652–665

[27]

Z.T. Xie, I.P. Castro

Efficient generation of inflow conditions for large-eddy simulation of street-scale flows

Flow Turbul Combust, 81 (3) (2008), pp. 449–470

[28]

E. Touber, N.D. Sandham

Large-eddy simulation of low-frequency unsteadiness in a turbulent shock-induced separation bubble

Theor Comput Fluid Dyn, 23 (2009), pp. 79–107

[29]

M.J. Lighthill

Attachment and separation in three-dimensional flows

L. Rosenhead (Ed.), Laminar boundary layer theory, Sect. II 2.6., Oxford University Press, Oxford (1963), pp. 72–82

[30]

J. Jeong, F. Hussain

On the identification of a vortex

J Fluid Mech, 285 (2006), pp. 69–94

[31]

W.P. Jones, B.E. Launder

The prediction of laminarization with a two-equation model of turbulence

Int J Heat Mass Transf, 15 (1972), pp. 301–314