Hybrid Turbulence Model Simulations of Hemisphere-Cylinder Geometry

A. Gross^a, H.F. Fasel^b

^a*Department of Mechanical and Aerospace Engineering, New Mexico State University*

^b*Department of Aerospace and Mechanical Engineering, The University of Arizona*

Abstract

When low aspect ratio geometries such as submarines, torpedoes, or missiles are operated at large angles of attack three-dimensional separation will occur on the leeward side. Separation incurs losses and can result in undesirable unsteady forces. An improved understanding of three-dimensional separation is desirable as it may open the door to new methods for the control or prevention of separation. Numerical simulations of three-dimensional separation can provide detailed insight into instability mechanisms and the resultant flow structures. For most technical applications the Reynolds numbers are too high for direct numerical simulations and lower-fidelity approaches such as hybrid turbulence models become attractive. In this paper a hybrid turbulence model blending strategy is employed that adjusts the model contribution according to the local grid resolution. The strategy is validated for two-dimensional plane channel flow at $Re_\tau = 395$ and 2,000. The model is then employed for simulations of a hemisphere-cylinder geometry at 10 and 30 degrees angle of attack. Hybrid turbulence model simulations were carried out for a hemisphere-cylinder geometry at 10 and 30 degrees angle of attack. The simulations demonstrate satisfactory model performance over a wide range of Reynolds numbers ($5 \times 10^4 < Re_D < 5 \times 10^5$). A nose separation bubble is captured for the lower Reynolds numbers and leeward vortices are observed for 30° angle of attack regardless of Reynolds number.

Keywords: Three-Dimensional Separation, Computational Fluid Dynamics, Leeward Vortices, Hybrid Turbulence Model, Reynolds Number Scaling

Email address: `agross@nmsu.edu` (A. Gross)

Preprint submitted to Elsevier

April 2, 2015

1. Introduction

Separation for low aspect ratio geometries such as submarines, torpedoes, or missiles is always three-dimensional (3-D). Coherent structures can arise as a consequence of hydrodynamic instabilities. The instability mechanisms and the resultant structures are 3-D and their interaction with each other is highly complex. The accurate prediction of the nonlinear evolution and dynamical interactions of these structures is of crucial importance as they can strongly influence the global mean flow behavior (which in turn affects performance) and exert unsteady aerodynamic forces. An improved understanding of the flow physics governing 3-D separation in general, and the dynamics of the flow structures in particular, is desirable as this may lead to successful separation control strategies.

Figure 1: Hemisphere-cylinder at $Re = 5 \times 10^3$ and $\alpha = 30^\circ$. Skin-friction coefficient, $0 < c_f < 0.013$, and skin-friction lines. F: Focus, N: node, and S: saddle point.

Already the mean flow topology of 3-D separated flow regions is very complicated. The skin-friction line pattern, which is characterized by singular points and lines of separation (and attachment) which connect these singular points, provides a description of the flow topology [1, 2] (Fig. 1). “Nodes” are points where skin-friction lines converge (separation) or from which they diverge (attachment). Nodes about which skin-friction lines spiral are known as “focal points”. They form the roots of vortices that are often referred to as “horn vortices”. Points where “opposing” skin-friction lines meet and then spread outward sideways are called “saddle points”. Three-dimensional separation is indicated by the convergence of skin-friction lines onto a particular

limiting skin-friction line, the line of separation. At this line the streamlines are forced away from the surface.

Numerical simulations of 3-D separation can provide detailed insight into instability mechanisms and the resultant flow structures. However for flows of practical relevance the Reynolds numbers are often too high to be easily accessible by Direct Numerical Simulations (DNS). For simulations to become feasible the small scales have to be modeled. The lowest grid resolution requirement and lowest computational cost are incurred by Reynolds-Averaged Navier-Stokes (RANS) calculations. However, for bluff body flows as considered here, that exhibit considerable separation, unsteadiness, and energetic large-scale structures, steady RANS calculations are often not a good choice. Employing Large Eddy Simulations (LES), where only the dissipative range is modeled, is also often not practical, because of the very high resolution requirements (albeit not as stringent as for DNS), especially for the flow regions near walls. An idea pursued by many researchers is to switch or gradually blend to RANS near the wall. An alternative to RANS is Large-Eddy Simulation (LES), which, for massively separated flows, is generally more accurate, but also computationally more expensive. The LES grid needs to be sufficiently fine to resolve a significant part of the turbulent wavenumber spectrum. Although the resolution requirement is not as stringent as for DNS, the required near-wall grid resolution is considerably higher than for RANS and increases with Reynolds number. In this context, hybrid RANS/LES strategies which combine the advantages of RANS and LES are being developed. Speziale [3, 4] was among the first to propose a hybrid turbulence modeling approach that combines the advantages of RANS, LES, and DNS.

Hybrid RANS/LES simulations of configurations involving flow separation from smooth surfaces (such as submarine hulls) are challenging because the separation location is not fixed by a geometric discontinuity (such as a step). Several researchers have carried out hybrid simulations for bluff geometries and in the following a short overview of simulations for circular cylinders and a sphere are discussed. The circular cylinder is an attractive candidate for such investigations because of the immense available reference data in the literature (e.g., Williamson [5]). As the Reynolds number for the circular cylinder is increased beyond $Re \approx 300,000$ the boundary layer transitions to turbulence before separation. As a result separation is delayed, the wake becomes narrower, and the drag coefficient drops rapidly from about 0.5 to 0.2 (“drag crisis”).

Travin et al. [6] carried out Detached Eddy Simulations (DES) of a cylinder with laminar ($Re=50,000, 140,000$) and turbulent separation ($Re=140,000, 3 \times 10^6$). For the cases with turbulent separation the boundary layer was tripped. Good agreement was observed for the shedding frequency and the mean drag as well as the pressure, and skin friction distributions. The downstream extent of the recirculation region was about twice as long as in the experiment, the Reynolds stresses were about 30% off compared to the experiment, and the results were found to be grid dependent. Nevertheless, the DES meanflow results were in much better agreement with the experiment than reference results obtained from unsteady RANS (URANS) calculations.

Elmiligui et al. [7] employed a two-equation $k-\varepsilon$ model with RANS/LES transition function (dependent on grid spacing and turbulence length scale) and a modified Partially Averaged Navier-Stokes (PANS) model [8] for simulating the flow around a steady and rotating circular cylinder. For a steady cylinder case with turbulent separation ($Re=140,000$, high free-stream turbulence) accurate predictions of the drag coefficient and the Strouhal number were obtained with URANS calculations (based on the $k-\varepsilon$ model). For a case with laminar separation ($Re=50,000$) the hybrid turbulence models outperformed the URANS models.

Belme et al. [9] employed Variational Multiscale LES (VMS-LES) and hybrid RANS/LES (based on VMS-LES) for simulating the flow past a circular cylinder at Reynolds numbers between 20,000 and 500,000. The wake and drag predictions were in good agreement with experimental data. Moussaed et al. [10] employed the same hybrid model for simulating the flow over a circular cylinder at Reynolds numbers between $Re=1.4 \times 10^5$ and 6.7×10^5 . The model was in RANS mode in the attached boundary layer and switched to LES mode in the separated boundary layer and wake. The mean pressure distribution over the cylinder was in reasonable agreement with reference data. The solution was found to be only weakly affected by the grid resolution.

You and Kwon [11] carried out URANS and hybrid RANS/LES simulations of a circular cylinder for $Re=3.6 \times 10^6$. For the latter, both the DES and the Scale Adapted Simulation [12] model were employed. In all instances the cylinder suction peak was slightly underpredicted compared to the experiment. The Reynolds normal stresses (and turbulent kinetic energy) obtained with the different turbulence models were noticeably different.

Constantinescu and Squires [13] employed LES and DES for investigating a sphere at $Re=10,000$. For this Reynolds number the flow separates laminar

and the wake is turbulent. The computed drag coefficient, Strouhal number, separation location, and skin friction distributions were in good agreement with experimental data. Compared to results for a second-order-accurate discretization, results for a fifth-order-accurate upwind discretization showed a more pronounced dependence on the turbulence model coefficients. This is in agreement with Margolin and Rider [14] who showed that the numerical diffusion of certain lower-order upwind schemes can mimick the subgrid stress.

For the results presented in this paper, a hybrid turbulence model was employed that adjusts the model contribution locally according to the ratio of turbulence length scale and grid resolution. The model was tested for two-dimensional plane channel flow at $Re_\tau = 395$ and 2,000. The plane turbulent channel flow is a common test case for hybrid LES/RANS models (e.g., Batten et al. [15], Travin et al. [16], Temmermann et al. [17]).

Following the model validation, simulations were carried out for a hemisphere-cylinder geometry at 10 and 30 degrees angle of attack. In this paper a hybrid turbulence model blending strategy is employed for simulations of a hemisphere-cylinder geometry at 10 and 30 degrees angle of attack. These simulations serve two purposes: They demonstrate the model performance over a wide range of Reynolds numbers ($5 \times 10^4 < Re_D < 5 \times 10^5$) and they reveal how the mean flow topology changes with Reynolds number. Earlier research indicates considerable 3-D flow separation for the hemisphere-cylinder at angle of attack [18, 19, 20, 21, 22, 23, 24]. At low-Reynolds number conditions and for large angles of attack, a separation bubble forms on the fore-body. In addition, the cross-flow causes the boundary layers to separate, roll up, and form a pair of streamwise vortices on the leeward side of the body (leeward vortices). Three types of separation lines are common for hemisphere-cylinders: The bow separation line and the primary and secondary separation lines. The latter two are associated with the leeward vortices. At small angles of attack, foci terminate the bow separation line and horn vortices are present. At larger angles of attack, the leeward vortices appear to connect in the bow separation bubble and form a horseshoe vortex [20].

The paper is organized as follows: First the turbulence modeling is explained. Then details regarding the grid resolution and boundary conditions are discussed. The hemisphere-cylinder simulations constitute the main body of the paper and, **together with the turbulence model validation**, are presented in section 3. Finally a short summary and conclusions are provided.

2. Methodology

2.1. Navier-Stokes Code

A research computational fluid dynamics (CFD) code that was developed in our laboratory [25, 26] was employed for the present investigations. The compressible Navier-Stokes equations are solved in curvilinear coordinates. For robustness, especially on highly distorted grids, a finite volume formulation is employed. The convective terms of the Navier-Stokes equations were discretized with a fifth-order-accurate scheme. A fourth-order-accurate discretization was employed for the viscous terms. The turbulence model equations were discretized with second-order accuracy. An implicit second-order-accurate Adams-Moulton method was employed for time integration. To maintain numerical stability, for the $Re = 5 \times 10^6$ simulations the spatial accuracy of the discretization of the Navier-Stokes equations was reduced to second-order (using a symmetric total variation diminishing upwind discretization by Yee [27]) and a first-order-accurate time-integration was employed.

2.2. Turbulence Model

The hybrid turbulence model was based on the 1998 Wilcox k - ω model in the modified version by Rumsey and Gatski [28]. The transport equations for the Turbulence Kinetic Energy (TKE) and the specific dissipation rate, $\tilde{\omega}$, are

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i} \rho u_i k = \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_T) \frac{\partial k}{\partial x_i} \right] + P_k - \beta^* \rho k \tilde{\omega} \quad (1)$$

$$\frac{\partial \rho \tilde{\omega}}{\partial t} + \frac{\partial}{\partial x_i} \rho u_i \tilde{\omega} = \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_T) \frac{\partial \tilde{\omega}}{\partial x_i} \right] + \alpha \frac{\tilde{\omega}}{k} P_k - \beta \rho \tilde{\omega}^2, \quad (2)$$

with $\tilde{\omega} = 0.09\omega$ and production term,

$$P_k = \tau_{ij}^T u_{i,j}. \quad (3)$$

The eddy viscosity is obtained from

$$\mu_T = c_\mu \frac{\rho k}{\tilde{\omega}} \quad (4)$$

with $\varepsilon = \tilde{\omega} k$. Also, $\sigma_k = 2$,

$$\beta^* = \begin{cases} \frac{1+680\chi_k^2}{1+400\chi_k^2} & \chi_k > 0 \\ 1 & \text{else} \end{cases} \quad \chi_k = 0.09^2 \times \frac{1}{\tilde{\omega}^3} \frac{\partial k}{\partial x_j} \frac{\partial \tilde{\omega}}{\partial x_j}, \quad (5)$$

$\sigma_\omega = 2.204$, $\alpha = 0.575$, and $\beta = 0.83$. The Reynolds stresses were obtained from the explicit algebraic Reynolds-stress model (EASM) by Rumsey and Gatski [28],

$$\begin{aligned}\tau_{ij}^T &= 2\mu_T \left(S_{ij} - \frac{1}{3} u_{k,k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \\ &+ 2\mu_T \left[a_2 a_4 (S_{ik} W_{kj} + S_{jk} W_{ki}) - 2a_3 a_4 \left(S_{ik} S_{kj} - \frac{1}{3} S_{kl} S_{kl} \delta_{ij} \right) \right],\end{aligned}\quad (6)$$

with strain rate and vorticity tensors,

$$S_{ij} = \frac{1}{2} (u_{i,j} + u_{j,i}) \quad (7)$$

$$W_{ij} = \frac{1}{2} (u_{i,j} - u_{j,i}), \quad (8)$$

and $c_\mu = c_\mu(S_{ij}, W_{ij})$. c_μ was limited to be larger than 0.0005.

Similar to the k - ω -based DES model by Travin et al. [29], the RANS model was turned into a hybrid model by taking the model length scale,

$$l_T = \min(L_\Delta, L_T), \quad (9)$$

as the minimum of a cell length scale, $L_\Delta = \max(\Delta x, \Delta y, \Delta z)$, and the turbulence length scale,

$$L_T = \frac{k^{3/2}}{\varepsilon}. \quad (10)$$

For $L_\Delta < L_T$ this effectively increases the TKE destruction term,

$$D_k = -\beta^* \rho \frac{k^{3/2}}{l_T} = -\beta^* \rho \frac{k \tilde{\omega}}{f}, \quad (11)$$

and lowers the eddy viscosity

$$\nu_T = c_\mu \rho l_T k^{1/2} = c_\mu f \frac{\rho k}{\tilde{\omega}}. \quad (12)$$

The function

$$f = \min\left(1, \frac{L_\Delta}{L_T}\right) \quad (13)$$

allows for a convenient visualization of the model contribution. For $f = 1$ the model is in full RANS mode. For $f = 0$ the model contribution is zero and a DNS is recovered. For $0 < f < 1$,

$$D_k = -\beta^* \rho \frac{k^{\frac{3}{2}}}{L_\Delta} \quad (14)$$

$$\nu_T = c_\mu \rho L_\Delta k^{\frac{1}{2}}, \quad (15)$$

and a one-equation LES model in the form of the Yoshizawa model [30] is obtained.

The unresolved or modeled turbulent kinetic energy in hybrid simulations is affected by the local grid resolution as suggested by e.g. Johansen et al. [31] and De Langhe et al. [32]. For flow from a coarse grid into a fine grid region turbulent velocity fluctuations have to be added to maintain the correct turbulent kinetic energy balance. Such a procedure is suggested for example by Batten et al. [15]. For the present simulations a similar procedure was employed that is described in detail in Gross and Fasel [33].

2.3. Non-Dimensionalization

For the channel flow simulations the channel half height, $h/2$, was taken as reference length, L_{ref} , and the bulk velocity, v_b , was chosen as reference velocity, v_{ref} . The reference Mach number, M , was 0.3. For the hemisphere-cylinder simulations the body diameter, D , and the freestream velocity, v_∞ , were chosen as reference length and velocity, respectively, and the reference Mach number was 0.1. Length scales and velocities were made dimensionless with the body diameter, D , and the freestream velocity, v_∞ . Time was normalized by L_{ref}/v_∞ , the TKE was made dimensionless with v_∞^2 , and the specific dissipation rate, ω , was made dimensionless with $\rho_\infty v_\infty^2/\mu_\infty$. The eddy viscosity, μ_T , was made dimensionless with μ_∞ . The reference Mach number was $M=0.1$ and The Prandtl number was $Pr=0.72$.

2.4. Boundary Conditions

Walls were considered as adiabatic. The TKE was set to zero at the wall. The smooth wall boundary condition [34] was applied for obtaining, ω , at the wall,

$$\omega = \frac{1}{\text{Re}^2} \frac{N\mu}{\rho\Delta y^2}, \quad (16)$$

where Δy is the distance from the wall to the center of the first cell away from the wall. The parameter N was set to 1600. [34] This value guarantees that for near wall grid resolutions (in wall units) of $\Delta y^+ \leq 1$ the effective surface roughness is $k_s^+ \leq 5$, which corresponds to a hydraulically smooth surface. For the channel flow simulations flow periodicity was enforced in the streamwise and spanwise direction. For the hemisphere-cylinder simulations non-reflecting boundary conditions [35] were applied at the free-stream boundary and a symmetry boundary condition was employed at the symmetry plane ($z = 0$).

2.5. Hemisphere-Cylinder Simulations Cases

The hemisphere-cylinder geometry for the present simulations was derived by replacing the bow shape of a shortened DARPA Suboff model [36] with a hemisphere [24]. The length/diameter ratio of the hemisphere-cylinder geometry is 4.81 and the nose radius is $r = L/(2 \times 4.81) = 0.104L$. The $Re_D = 5,000$ results were obtained without turbulence model and have been published previously in Ref. [24]. They are here included for completeness. For the present simulations the Reynolds number was increased by one to three up to two orders of magnitude (Tab. 1). For comparison, the Reynolds numbers based on diameter for a Mk48 torpedo and a Los Angeles class submarine are roughly 10 million and 100 million. “Sustainer terms” as

Re_D	Re_L	α [°]
5×10^3 †	2.405×10^4	10, 30
5×10^4	2.405×10^5	10, 30
1.5×10^5	7.215×10^5	30
5×10^5	2.405×10^6	10, 30
5×10^6	2.405×10^7	10, 30

Table 1: Cases for hemisphere-cylinder simulations. †Results presented in Gross et al. [24].

suggested by Spalart and Rumsey [37] were added to the turbulence model equations to maintain the desired TKE and specific dissipation rate in the freestream. Also following Spalart and Rumsey, the freestream TKE and dissipation rate were set to $k_\infty = 10^{-6}v_\infty^2$ and $\omega_\infty = 5v_\infty/L_{ref}$. The corresponding freestream turbulence intensity is $Tu_\infty = \sqrt{2/3k} = 0.082\%$.

Figure 2: Hemisphere-cylinder. *a)* Surface and symmetry plane ($z = 0$) grid. *b)* Near-wall grid resolution in wall units, Δx^+ , Δy^+ , and Δz^+ , for $Re = 5 \times 10^4$ and $\alpha = 10^\circ$.

A Poisson grid generator [38] was employed to generate an O-grid in the $z = 0$ plane that was then “extruded” in the azimuthal direction. For the extrusion process, cells were clustered on the leeward side of the body where flow separation was expected (Fig. 2*a*). Assuming symmetry of the mean flow with respect to the $z = 0$ plane, only half of the flow was computed. This approach reduces the computational expense of the simulations but also enforces symmetry of the unsteady flow structures. In Gross et al. [24] results from a full-domain simulation (360° computational domain) and a half-domain simulation (180° computational domain) for the same hemisphere-cylinder geometry at $Re=5,000$ and $\alpha=10^\circ$ were compared. The unsteady flow structures and the mean flow field were found to be similar enough to justify half-domain simulations.

The outer (free-stream) boundary was located 50 body diameters from the center of the body. H-grids were inserted at the bow and stern to remove the grid singularities associated with the axisymmetric grids. Table 2 provides the block grid resolutions. For the $Re = 5 \times 10^3$ and $Re = 1.5 \times 10^5$ simulations, the block 1 grid had $438 \times 200 \times 64$ cells; the number of cells for blocks 2 & 3 was unchanged the same as for the other cases.

For the higher Reynolds number simulations Δy at the wall was reduced. The following estimates were made to determine by how much the near-wall grid resolution had to be increased: The flat plate turbulent skin-friction coefficient is $c_f \approx 0.0592 Re_x^{-1/5}$. With friction velocity, $v_\tau = \sqrt{1/2 v_\infty^2 c_f}$, the wall-normal coordinate in wall units becomes $y^+ = v_\tau y / \nu = Re \sqrt{c_f/2} y/D = Re^{9/10} \sqrt{0.0592/2} y/D$. Since approximately the same Δy^+ is desired at

Number of cells	
block 1	$354 \times 200 \times 64$
block 2	$16 \times 200 \times 32$
block 3	$16 \times 200 \times 32$
total	5,811,200

Table 2: Hemisphere-cylinder simulations. Number of cells per block.

Re	Δx^+	Δy^+	Δz^+
$\alpha = 10^\circ$			
5×10^3	0.12 ... 10	0.0030 ... 0.11	0.13 ... 89
5×10^4	1.0 ... 62	0.0056 ... 0.35	0.28 ... 400
5×10^5	6.5 ... 440	0.0047 ... 0.30	1.9 ... 2,800
5×10^6	74 ... 3,500	0.0054 ... 0.30	18 ... 22,000
$\alpha = 30^\circ$			
5×10^3	0.097 ... 18	0.0025 ... 0.21	0.16 ... 93
5×10^4	0.81 ... 96	0.0047 ... 0.54	0.37 ... 370
1.5×10^5	2.1 ... 250	0.02 ... 1.4	1.1 ... 870
5×10^5	5.3 ... 700	0.0062 ... 0.47	2.6 ... 2,400
5×10^6	110 ... 4,800	0.0081 ... 0.41	27 ... 19,000

Table 3: Hemisphere-cylinder simulations. Near-wall grid resolution.

the wall regardless of the Reynolds number it follows that $\Delta y / \Delta y_{50,000} = (50,000 / Re)^{9/10}$. Accordingly for $Re = 5 \times 10^5$ and 5×10^6 the near wall grid line spacing was reduced by factor 7.94 **times** and 63.3, respectively. The streamwise, wall-normal, and circumferential near-wall grid resolution in wall units, Δx^+ , Δy^+ , and Δz^+ , for the different cases are provided in Tab. 3. The numbers were obtained directly from the simulations (an example is shown in Fig. 2b).

2.6. Hemisphere-Cylinder Time Integration

The computational time steps and the time intervals used for computing the initial transients and for obtaining time-averaged data are listed in Tab. 4. With the chosen non-dimensionalization (body diameter and freestream velocity), a time interval of 4.81 corresponds to one “flow-over” time. For $Re = 5 \times 10^6$ the model contribution is large enough to damp out all unsteadiness and steady-state solutions were sought.

Re_D	α [°]	Δt	initial transient	time-averaging
5×10^3	10	0.001	18	12
	30	0.001	15.7	4
5×10^4	10	0.001	4	8
	30	0.001	15.7	12
1.5×10^5	30	0.001	6	9
5×10^5	10	0.0005	3	4
	30	0.0005	6.6	4
5×10^6	10	0.00025	-	-
	30	0.00025	-	-

Table 4: Hemisphere-cylinder. Computational time-step, Δt , and time intervals for initial transients and time-averaging.

3. Results

3.1. Validation of Turbulence Model for Channel Flow

Channel flow simulations were carried out for Reynolds numbers based on mean friction velocity, $\bar{v}_\tau$, and channel half-height, $h/2$, of $Re_\tau = \bar{v}_\tau h/2/\nu = 395$ and 2,000. The bulk Reynolds number, $Re_b = v_b D_h/\nu$, is based on the bulk velocity, $v_b = 1/A \int v dA$, and the hydraulic diameter, $D_h = 2h$. The friction factor, can be obtained from empirical relations, such as the one given by Petukhov, $f = 8(\bar{v}_\tau/v_b)^2 = (0.79 \ln Re_b - 1.64)^{-2}$. For the plane channel flow $f = 128(Re_\tau/Re_b)^2$ and bulk Reynolds numbers of $Re_b = 28,939$ ($Re_\tau = 395$) and 179,111 ($Re_\tau = 2,000$) are obtained. Because streamwise and spanwise flow periodicity was enforced, the turbulence was self-sustained. For the chosen non-dimensionalization the wall normal coordinate, streamwise velocity, and Reynolds shear stress in wall units are $y^+ = Re_\tau \times y$, $u^+ = Re_b/(4Re_\tau) \times u$, and $\overline{u'v'}^+ = (Re_b/(4Re_\tau))^2 \times \overline{u'v'}$. The laminar shear stress in wall units is $\tau_{xy}^+ = Re_b/(4Re_\tau^2) \times \partial u/\partial y$. In accordance with earlier simulations [15, 16, 17], the domain dimensions in the streamwise, wall-normal, and spanwise direction were chosen as $6.4 \times 2 \times 3.2$ and the number of cells in the respective directions were chosen as $64 \times 64 \times 32$. The near-wall grid line spacing in wall units, Δy^+ , was 0.8 for $Re_\tau=395$ and 1.0 for $Re_\tau=2,000$. Time-averages were computed over time-intervals of 5,000. Statistical quantities (w.r.t. to the time-averages which were averaged in the streamwise and spanwise direction) were computed over time-intervals of 5,000.

Figure 3: Channel flow: Iso-surfaces of $Q = 0.1$ shaded by streamwise velocity for $Re_\tau = 395$ (left) and 2,000 (right).

Instantaneous iso-surfaces of the vortex identification criterion [39] shaded

by the streamwise velocity are shown in Fig. 3. The vortex identification criterion,

$$Q = 0.5 (W_{ij}W_{ij} - S_{ij}S_{ij}) , \quad (17)$$

indicates regions where rotation dominates strain. Since for the hybrid approach the unresolved turbulence is modeled, the structures in Fig. 3 are the resolved flow structures. The computed bulk velocities are 1.071 ($Re_\tau=395$) and 1.087 ($Re_\tau=2,000$) and thus slightly larger than the reference bulk velocity, $u_b = 1$. This suggests that either the friction factor obtained from the Petukhov relationship is too high or that the resolved and unresolved Reynolds shear stresses are too low.

Figure 4: Channel flow: *a*) Velocity profiles in wall units and *b*) shear-stress profiles.

Velocity profiles in wall units and shear-stress profiles are provided in Fig. 4a. As a reference, the viscous sublayer relationship, $u^+ = y^+$, and the log-law, $u^+ = 5 + 1/0.41 \ln y^+$, were included in the velocity profiles. The velocity profiles in wall units are in reasonably good agreement with the reference data and show a slight shift at the y^+ -location where the resolved and unresolved Reynolds shear stresses balance each other (indicated by vertical lines). This location can be thought of as the location where the model bias changes from RANS to LES and vice versa. For the chosen grid resolution the RANS/LES “cross-over points” lie in the log-layer. The laminar shear stress, τ_{xy}^+ and the unresolved and resolved Reynolds shear stresses, $\overline{u'v'}_u^+$ and $\overline{u'v'}_r^+$, add up to $1 - y$ (reference line in Fig. 4b) which is the expected behavior.

3.2. Hemisphere-Cylinder Validation Case

For validation purposes, a hemisphere-cylinder simulation was carried out for the experimental conditions by Hoang et al. [22] ($Re = 1.5 \times 10^5$ and $\alpha = 30^\circ$). Velocity vectors for $Re = 1.5 \times 10^5$ and 5×10^5 at $s/r = 5.8$ are pro-

Figure 5: Hemisphere-cylinder at $\alpha = 30^\circ$. Velocity vectors at $s/r = 5.8$ for a) $Re = 1.5 \times 10^5$ and b) $Re = 5 \times 10^5$. c) Seven-hole probe measurements by Hoang et al. [22] for $Re = 1.5 \times 10^5$.

vided in Fig. 5. The corresponding x -location is $x = 0.5(1+s/r-\pi/2) = 2.61$. For the present geometry, at $x = 2.61$ the radius is less than 0.001% smaller than the nominal radius of 0.5 and the beginning taper already results in a slight flow acceleration. The numerical results reveal strong leeward vortices which are not apparent in the experimental data [22]. A possible explanation was provided by Hoang et al. [22] who noted that the seven-hole “probe generates a global interference to the flow. Large errors are registered in the circumferential components within the vortex. Differences as large as 46% are observed. Insertion of the probe at locations where the vortex is still small and weak may distort the flow and induce some sort of vortex break-down.” Velocity profiles for three different azimuth angles are provided in Fig. 6. According to Hoang et al. [22] their Laser Doppler Velocimetry (LDV) data are of higher quality than the seven-hole probe data (because of probe interference with the flow for the latter). The experimental LDV data are in good agreement with the simulation data for $Re = 5 \times 10^5$ than with

Figure 6: Hemisphere-cylinder at $\alpha = 30^\circ$. Radial profiles of azimuthal velocity at $s/r = 5.8$. Measurements by Hoang et al. [22] are for $Re = 1.5 \times 10^5$.

the simulation data and in adequate agreement with the present results for $Re = 1.5 \times 10^5$. A comparison of wall pressure coefficient distributions from the $Re = 5 \times 10^5$ simulation with the wind tunnel measurements by Hoang et al. [22] for $Re = 4.2 \times 10^5$ is provided in Fig. 7. The vertical line indicates the beginning of the tapering off of the geometry in the simulation. The quantitative agreement of the data is good. The characteristic pressure plateaus for $\vartheta \geq 120^\circ$ indicate a short nose separation bubble for both experiment and simulation. In the simulation the flow separates later. This can be explained by the turbulence model that was employed for the present simulations. The model cannot capture transition and in the simulation the boundary layer “transitions” to turbulence upstream of separation. Overall, the agreement of the present simulation data with the earlier measurements by Hoang et al. [22] was considered acceptable and the decision was made to investigate how the turbulence model behavior and the flow field were affected by the Reynolds number effects.

Figure 7: Wall pressure coefficient for $\alpha = 30^\circ$ obtained from simulation ($Re = 5 \times 10^5$, solid lines) and measurements by Hoang et al. [22] ($Re = 4.2 \times 10^5$, dashed lines).

3.3. Investigation of Reynolds Number Effects

Instantaneous iso-surfaces of the vortex identification criterion [39] for four **three** different Reynolds numbers and $\alpha = 10^\circ$ are provided in Fig. 8. The vortex identification criterion,

$$Q = \frac{1}{2} (W_{ij}W_{ij} - S_{ij}S_{ij}) , \quad (18)$$

indicates areas where rotation dominates strain. A nose separation bubble with resolved wake structures forms for $Re = 5 \times 10^3$ and 5×10^4 . For $Re = 5 \times 10^5$ unsteady flow structures are only observed in the body wake. For $Re = 5 \times 10^6$ all unsteadiness is modeled and the dominant resolved flow features are two short leeward vortices near the stern.

Figure 8: Hemisphere-cylinder at $\alpha = 10^\circ$. Iso-surfaces of $Q = 0.1$.

Iso-surfaces of the time-averaged u -velocity component help identify the shape of the nose separation bubble (Fig. 9). As the Reynolds number is increased the nose separation bubble is diminished in size. With increasing Reynolds number flow instabilities become stronger and the boundary layer is transitioning earlier (the turbulence model employed for the present simulations was not designed to capture transition). Both transition and the emergence of strong coherent flow structures with spanwise (i.e., circumferential) alignment resulting from instabilities of the turbulent mean flow accelerate reattachment.

Figure 9: Hemisphere-cylinder at $\alpha = 10^\circ$. Iso-surfaces of $u = 0$ (light grey) and $Q = 30$ (dark grey).

A “zoom-in” for $Re = 5 \times 10^3$ reveals that the jagged shape of the bubble which was also observed in time-averaged water tunnel Particle Image Velocimetry (PIV) data [40] can be (in the mean) associated with small streamwise vortices (Fig. 10).

Figure 10: Hemisphere-cylinder at $Re = 5 \times 10^3$ and $\alpha = 10^\circ$. Iso-surfaces of $u = 0$ (light grey) and $Q = 10$ (black).

Figure 11: Hemisphere-cylinder at $\alpha = 10^\circ$. Skin-friction coefficient, c_f , and skin-friction lines.

Alternatively, the skin-friction line pattern can be employed for analyzing the mean flow topology and identifying separated flow regions (Fig. 11). The arc-length, s , was measured from the point where the body center-line intersects with the bow, along lines of constant azimuth angle, ϑ . The azimuth angle, ϑ , was measured from the windward symmetry plane (the very top of the body is at $\vartheta = 180^\circ$ - also see Fig. 5). The geometry is tapered off for $x \geq 2.6$ which corresponds to $s/r = \pi/2 + 2(2.6 - 0.5) = 5.77$. The skin-friction line patterns illustrate a reduced downstream extent of the nose bubble with increasing Reynolds number for $Re = 5 \times 10^3$ & 5×10^4 and attached flow for $Re = 5 \times 10^5$ & 5×10^6 .

Instantaneous flow visualizations for $\alpha = 30^\circ$ are provided in Fig. 12. Pronounced leeward vortices are seen for all Reynolds numbers. As the Reynolds number is increased, less unsteady flow structures are resolved and the leeward vortices appear more coherent. The bow region is shedding for $Re \leq 5 \times 10^5$ hinting at a nose separation bubble.

Figure 12: Hemisphere-cylinder at $\alpha = 30^\circ$. Iso-surfaces of $Q = 0.1$.

An analysis of the mean flow does indeed reveal a nose separation bubble for $Re \leq 5 \times 10^5$ that is diminished in size with increasing Reynolds number, and pronounced leeward vortices (Fig. 13). Similar conclusions can be drawn based on the skin-friction line patterns (Fig. 14).

Figure 13: Hemisphere-cylinder at $\alpha = 30^\circ$. Iso-surfaces of $u = 0$ (grey) and $Q = 30$ (blue).

Figure 14: Hemisphere-cylinder at $\alpha = 30^\circ$. Skin-friction coefficient, c_f , and skin-friction lines.

Iso-contours of the unresolved eddy viscosity, $\mu_{T,u}$, and the model contribution, f , are provided in Fig. 15. No data are shown for $Re = 5 \times 10^3$ since these simulations were carried out without turbulence model. As the Reynolds number is increased by factor 10, the unresolved eddy viscosity increases by roughly one order of magnitude. The model contribution is one at the wall and in the freestream and overall also increases with Reynolds number.

Figure 15: Hemisphere-cylinder. Iso-surfaces of unresolved eddy viscosity, $\mu_{T,u}$, and model contribution, $0.5 < f < 1$, at $z = 0$.

An interesting observation was made when analyzing the time-dependent data for $Re = 5 \times 10^4$ and $\alpha = 30^\circ$. At irregular intervals large scale coherent structures originated from the nose bubble and traveled downstream (Fig. 16). The large structures weakened the leeward vortices and led to a modulation of the lift and drag coefficient (Fig. 17). Because the present time-series is relatively short the oscillation period cannot be estimated with certainty.

Finally, in Fig. 18 wall pressure distributions obtained from the simulations are compared. The $\vartheta = 180^\circ$ distributions for $\alpha = 10^\circ$ show a pressure

Figure 16: Hemisphere-cylinder at $Re = 5 \times 10^4$ and $\alpha = 30^\circ$. Iso-surface of $Q = 0$.

Figure 17: Hemisphere-cylinder at $Re = 5 \times 10^4$ and $\alpha = 30^\circ$. Lift and drag coefficient.

plateau which is indicative of separation for $Re = 5 \times 10^3$ and $Re = 5 \times 10^4$. For $\alpha = 30^\circ$ flow separation is observed for $Re \leq 5 \times 10^5$. The wall pressure distributions for $x = 2$ and $\alpha = 30^\circ$ show local minima near $155 < \vartheta < 165^\circ$.

These minima can be related to the leeward vortices.

Figure 18: Hemisphere-cylinder. Wall pressure coefficient for a) $\vartheta = 180^\circ$ and b) $x = 2$.

4. Conclusions

A hybrid turbulence model was first validated for a plane channel flow at $Re_\tau = 395$ and 2,000 and then employed for Hybrid turbulence model simulations of a hemisphere-cylinder geometry were carried out for at 10 and 30 degrees angle of attack. As the Reynolds number was increased from 5,000 to 500,000 5 million the model contribution increased and less and less of the unsteady flow structures were resolved. A 3-D laminar separation bubble

that was pronounced at the lower Reynolds numbers diminished in size with increasing Reynolds number and completely disappeared for $Re = 5 \times 10^6$. The bubble was captured in the simulations despite the fact that the underlying $k-\omega$ turbulence model was not designed to predict transition. More accurate results may be obtained when transition models (e.g., Likki et al. [41]) are employed. For $Re=50,000$ and $\alpha = 30^\circ$ an intermittent “bursting” of the nose bubble was observed which resulted in a fluctuation of the lift and drag coefficient of about 10%. This phenomenon may be relevant for practical applications and requires further investigation.

Acknowledgments

This work was funded by the Office of Naval Research (ONR) under grant number N00014-10-1-0404 with Dr. R. Joslin serving as program manager. Compute time was made available through a Department of Defense (DoD) High Performance Computing (HPC) Modernization Program challenge grant.

References

- [1] Lighthill, M.J., “Attachment and Separation in Three-Dimensional Flow,” in “Laminar boundary layers” Chapter II, ed. L. Rosenhead (Oxford: University Press), 1963, pp. 72-82.
- [2] Tobak, M., and Peake, D.J., “Topology of Three-Dimensional Separated Flows,” *Annual Review of Fluid Mechanics* **14**, 61 (1982).
- [3] Speziale, C.G., “A Combined Large-Eddy Simulation and Time-Dependent RANS Capability for High-Speed Compressible Flows,” Technical Report No. AM-97-022, Boston University, Sept. 1997.
- [4] Speziale, C.G., “Turbulence modeling for time-dependent RANS and VLES: a review,” *AIAA Journal* **36**, 173 (1998).
- [5] Williamson, C.H.K., “Vortex Dynamics in the Cylinder Wake,” *Annu. Rev. Fluid. Mech.* **28**, 477 (1996).
- [6] Travin, A., Shur, M., Strelets, M. and Spalart, P., “Detached-Eddy Simulations Past a Circular Cylinder,” *Flow, Turbulence and Combustion* **63**, 293 (1999).
- [7] Elmiligui, A., Abdol-Hamid, K.S., Massey, S.J., and Pao, S.P., “Numerical Study of Flow Past a Circular Cylinder Using Hybrid Turbulence Formulations,” *Journal of Aircraft* **47**, 434 (2010).
- [8] Girimaji, S.S., Sreenivasan, R., and Jeong E., “PANS Turbulence Model For Seamless Transition Between RANS, LES: Fixed-Point Analysis and Preliminary Results.” FEDSM2003-45336, Proceedings of ASME FEDSM’03 2003 4th ASME-JSME Joint Fluids Engineering Conferences, July 13-16, Honolulu, Hawaii, USA, 2003.
- [9] Belme, A., Dervieux, A., Koobus, B., Wornom, S., and Salvetti, M.-V., “Application of hybrid and VMS-LES turbulent models to aerodynamic simulations,” in Proceedings of ICAS Conference, Nice, 2010.
- [10] Moussaed, C., Salvetti, M. V., Wornom, S., Koobus, B., and Dervieux, A., “Simulation of the flow past a circular cylinder in the supercritical regime by blending RANS and variational-multiscale LES models,” *Journal of Fluids and Structures* **47**, 114 (2014).

- [11] You, J.Y., and Kwon, O.J., “Numerical Comparisons Between URANS and Hybrid RANS/LES at a High Reynolds Number Flow Using Unstructured Meshes,” *Int. J. Of Aeronautical & Space Sciences* **11**, 41 (2010).
- [12] Menter, F.R., and Egorov, Y., “A scale-adaptive simulation model using two-equation models,” *AIAA-Paper* 2005-1095 (2005).
- [13] Constantinescu, G.S., and Squires, K.D., “LES and DES Investigations of Turbulent Flow over a Sphere at $Re=10,000$,” *Flow, Turbulence and Combustion* **70**, 267 (2003).
- [14] Margolin, L.G., Rider, W.J., “A rationale for implicit turbulence modeling,” *Int. J. Numer. Meth. Fluids* **39**, 821 (2002).
- [15] Batten, P., Goldberg, U., and Chakravarthy, S., “Interfacing Statistical Turbulence Closures with Large-Eddy Simulation,” *AIAA Journal* **42**, 485 (2004).
- [16] Travin, A.K., Shur, M.L., Spalart, P.R., and Strelets, M.K., “Improvement of Delayed Detached-Eddy Simulation for LES with Wall Modelling,” *European Conference on Computational Fluid Dynamics*, Wesseling, P., Oñate, E., and Périaux, J. (Eds.), TU Delft, The Netherlands, 2006.
- [17] Temmermann, L., Hadžiabdić, M., Leschziner, M.A., and Hanjalić, K., “A hybrid two-layer URANS-LES approach for large eddy simulation at high Reynolds numbers,” *Int. J. of Heat and Fluid Flow* **26**, 173 (2005).
- [18] Bippes, H., “Experimental Investigation of Topological Structures in Three-dimensional Separated Flow,” *Boundary-Layer Separation IUTAM Symposium London*, 1986.
- [19] Wang, K.C., and Hsieh, T., “Separation Patterns and Flow Structures About a Hemisphere-Cylinder at High Incidences,” *AIAA paper* AIAA-92-2712, June 1992.
- [20] Hsieh, T. and Wang, K.C., “Three-dimensional separated flow structure over a cylinder with a hemispherical cap,” *Journal of Fluid Mechanics* **324**, 83 (1996).

- [21] Hoang, N.T., Rediniotis, O.K., and Telionis, D.P., “Symmetric and Asymmetric Separation Patterns over a Hemisphere Cylinder at Low Reynolds Numbers and High Incidences,” *Journal of Fluids and Structures* **11**, 793 (1997).
- [22] Hoang, N.T., Rediniotis, O.K., and Telionis, D. P., “Hemisphere Cylinder at Incidence at Intermediate to High Reynolds Numbers,” *AIAA Journal* **37**, 1240 (1999).
- [23] Ying, S.X., Schiff, L.B., and Steger, J.L., “A Numerical Study of Three-Dimensional Separated Flow past a Hemisphere Cylinder,” *AIAA paper AIAA-87-1207*, June 1987.
- [24] Gross, A., Jagadeesh, C., Fasel, H.F., “Numerical and Experimental Investigation of Unsteady Three-Dimensional Separation on Axisymmetric Bodies,” *International Journal of Heat and Fluid Flow* **44**, 53 (2013).
- [25] Gross, A., and Fasel, H., “High-Order Accurate Numerical Method for Complex Flows,” *AIAA Journal* **46**, 204 (2008).
- [26] Gross, A., and Fasel, H., “Modification of Ninth-Order Weighted Essentially Nonoscillatory Scheme for Mixed Subsonic/Supersonic Flow,” *AIAA Journal* **48**, 2698 (2010).
- [27] Yee, H.C., “Upwind and Symmetric Schok-Capturing Schemes,” *NASA TM 89464*, May 1987.
- [28] Rumsey, C.L., and Gatski, T.B. “Recent Turbulence Model Advances Applied to Multielement Airfoil Computations,” *Journal of Aircraft* **38**, 904 (2001).
- [29] Travin, A.K., Shur, M.L., Strelets, M.K., and Spalart, P.R., “Physical and Numerical Upgrades in the Detached-Eddy Simulation of Complex Turbulent Flows,” *Advances in LES of Complex Flows* **65**, 239 (2002).
- [30] Yoshizawa, A., “Statistical theory for compressible shear flows with the application of subgrid modelling,” *Physics of Fluids A* **29**, 2152 (1986).
- [31] Johansen, S.T., Wu, J., and Shyy, W., “Filter-based unsteady RANS computations,” *International Journal of Heat and Fluid Flow* **25**, 10 (2004).

- [32] De Langhe, C., Merci, B., and Dick, E., “Hybrid RANS/LES modelling with an approximate renormalization group. I: Model development,” *Journal of Turbulence* **6**, 2 (2005).
- [33] Gross, A., and Fasel, H.F., “Numerical Investigation of Separation for Airfoils,” AIAA Paper AIAA-2011-3438, June 2011.
- [34] Wilcox, D.C., ”Turbulence Modeling for CFD” Second Edition, DCW Industries, 1998.
- [35] Gross, A., and Fasel, H.F., “Characteristic Ghost-Cell Boundary Condition,” *AIAA Journal* **45**, 302 (2007).
- [36] Groves, N.C., Huang, T.T., and Chang, M.S., “Geometric Characteristics of DARPA Suboff Models,” David Taylor Research Center, Bethesda, MD, report DTRC/SHD-1298-01, March 1989.
- [37] Spalart, P.R., and Rumsey, C.L., “Effective Inflow Conditions for Turbulence Models in Aerodynamic Calculations,” *AIAA Journal* **45**, 2544 (2007).
- [38] Gross, A., and Fasel, H.F., “Multi-block Poisson grid generator for cascade simulations,” *Mathematics and Computers in Simulation* **79**, 416 (2008).
- [39] Hunt, J.C.R., Wray, A.A., and Moin, P., “Eddies, stream, and convergence zones in turbulent flows,” Report CTR-S88, Center For Turbulence Research, Stanford, CA, 1988.
- [40] Balthazar, M., Gross, A., and Fasel, H.F., “Experimental Investigation of Flows with Three-Dimensional Separation,” AIAA Paper AIAA-2014-1451, Jan. 2014.
- [41] Likki, S.R., Suzen, Y.B., Huang, P.G., völker, S., Menter, F.R., and Langtry, R.B., “A Correlation-Based Transition Model Using Local Variables - Part I: Model Formulation,” *Journal of Turbomachinery* **128**, 413 (2004).