UNSTEADY RANS AND DETACHED EDDY SIMULATIONS OF CAVITATING FLOW OVER A HYDROFOIL

Robert F. Kunz
Jules W. Lindau
Todd A. Kaday
Leonard J. Peltier

Applied Research Lab/The Pennsylvania State University

ABSTRACT
A three-dimensional, time-accurate, compressible, homogeneous multiphase Navier-Stokes method is applied to modeling developed cavitation flow over a hydrofoil at angle-of-attack. The configuration and cavitation numbers considered give rise to highly unsteady cavitation structures on the suction side of the foil, characterized by large recirculating (re-entrant) flow regions. Accordingly, a time-accurate approach is forthcoming. In the present work Unsteady Reynolds Averaged Navier-Stokes (URANS) and Detached Eddy Simulation (DES) modeling are pursued with the intent of elucidating the differences that these modeling approaches have in the prediction of important engineering quantities. Two-dimensional URANS simulation results are presented for cavitation numbers of 0.4 and 0.8. Though DES can only simulate the largest scales of turbulence when applied three-dimensionally, “2D DES” runs were performed for the same two cases in order to provide experience and guidance with the model. A fully three-dimensional DES simulation was performed at a cavitation number of 0.8. A very rich and interesting multiphase flow field is observed in this simulation. However, despite the large CPU requirements of this calculation, the results presented are not yet statistically stationary, making a complete quantitative statistical interpretation of the data set premature. Visualizations of 3D flow structures and short time-histories of flow variables are presented. The preliminary data suggest that the intensities of turbulence fluctuations supported by the 3D solution are less than those observed in the 2D cases, a physically plausible result. A possible mechanism for this behavior is discussed.

INTRODUCTION
The authors have been involved for several years in the development of a 3D, time accurate, homogeneous multiphase Reynolds-Averaged, Navier-Stokes computational tool, for application to a variety of developed and supercavitating configurations [1-5]. Though we limit our interest here to low speed sheet cavitation over a hydrofoil, this configuration is typical of a class of engineering flows with large scale cavitation, in that a number of physical complexities render computational simulation a challenge. In particular, these flows are inherently of high Reynolds number, are often highly unsteady, are characterized by high density ratios between constituents, contain discrete interfaces between constituents, and may encompass a wide range of Mach numbers simultaneously within the flow domain due to two-phase sound speed reduction effects.

These complexities have focused elements of our research (and that of other workers [6-17], in the area of large scale cavitation CFD, many of whom are represented in the present proceedings) on physical modeling (e.g., turbulence, mass transfer, two-phase constitutive relations), on numerical methods (e.g., all-Mach number multiphase preconditioning, high order discretization/flux limiters), and on validation of CFD methods against the limited available experimental data. The significant progress that has been made by the research community in these areas has led to the ability to compute a variety of engineering-relevant sheet- and super-cavitating flow fields with good accuracy [1-17].

Despite this progress, challenges remain in the CFD analysis of these flows. The focus here is on perhaps the most important modeling issue for low-speed sheet cavitation - turbulence. The well known limitations of modern transport RANS turbulence models to complex flows, becomes quite problematic for the highly unsteady, high density ratio, compressible multiphase flows characteristic of sheet cavitation. This has motivated a number of researchers to apply/test a number of different models and to adopt modifications to conventional turbulence models, yielding prediction improvements in several cases [11, 14, and several contributions in these proceedings, for example].

Detached Eddy Simulation (DES), and other hybrid RANS-LES approaches, has emerged recently as a potential compromise between RANS based turbulence models and Large Eddy Simulation. Specifically, for flows characterized by large scale separation and unsteadiness, DES has been shown by a number of workers to date to provide improved mean flow predictions [18-22].
DES is a generalization of RANS modeling. The underlying turbulence model for DES is a RANS subgrid model with the characteristic length scale reinterpreted in terms of the local grid scale. When the local mesh is fine relative to the turbulence mixing length, the DES model becomes an LES with a Smagorinsky-like subgrid closure [18]. In these resolved regions, modeling error is minimized. The solution remains RANS elsewhere and retains the solution efficiencies of RANS.

DES is formulated for and is particularly suited to regions of massive separation where RANS closures are challenged. With minimal increases in grid density, the governing equations are allowed to support the most energetic turbulence scales yielding improved solution fidelity. Near boundaries, where kinematic constraints limit the scale of turbulence eddies and where grid resolution demands for traditional LES are most demanding, DES reverts back to RANS. By design, RANS models can perform very well near solid surfaces.

The cavitating flow over a hydrofoil studied in this work is characterized by massive separation and is a reasonable candidate for DES. Our goal is to explore DES for these flows.

The paper is organized as follows: First, brief overviews of the differential model and numerical methods are provided. The hydrofoil configuration and the associated numerical modeling parameters are then described. 2D results are presented for cavitation numbers of 0.8 and 0.4. 3D DES results are then presented and discussed.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a_0,a_1,a_2,a_3,a_4$</td>
<td>Coefficients of airfoil section geometry specification polynomial</td>
</tr>
<tr>
<td>$c$</td>
<td>Chord length</td>
</tr>
<tr>
<td>$C_1, C_2, C_{DES}, C_a$</td>
<td>Turbulence model constants</td>
</tr>
<tr>
<td>$C_L, C_D$</td>
<td>Lift, drag coefficients: $C_L = \frac{L}{1/2 \rho U_c^2 c}$, $C_D = \frac{D}{1/2 \rho U_c^2 c}$</td>
</tr>
<tr>
<td>$C_p, \sigma$</td>
<td>Pressure coefficient and cavitation index: $C_p = \frac{p - p_{ref}}{1/2 \rho U_c^2}$, $\sigma = \frac{p_{ref} - p_v}{1/2 \rho U_c^2}$</td>
</tr>
<tr>
<td>$F_{DES}$</td>
<td>DES modeling parameter</td>
</tr>
<tr>
<td>$k, \varepsilon$</td>
<td>Turbulence kinetic energy and dissipation rate</td>
</tr>
<tr>
<td>$L_t$</td>
<td>Turbulent length scale</td>
</tr>
<tr>
<td>$P$</td>
<td>Turbulence model production term</td>
</tr>
<tr>
<td>$Pr_k, Pr_c$</td>
<td>Turbulant Prandtl Numbers for $k$ and $\varepsilon$</td>
</tr>
<tr>
<td>$p_v, p_{ref}$</td>
<td>Saturation vapor pressure, and reference pressure (Fig 1.)</td>
</tr>
<tr>
<td>$t, t^*$</td>
<td>Time, nondimensional time, $t^* = t/(c/U_c)$</td>
</tr>
<tr>
<td>$u_i, U_c$</td>
<td>Cartesian velocity, freestream velocity</td>
</tr>
<tr>
<td>$V_v, V_{cell}$</td>
<td>Total vapor volume, cell volume</td>
</tr>
</tbody>
</table>

**THEORETICAL FORMULATION**

**Governing Equations**

The governing differential model employed is the three-dimensional, homogeneous multiphase set of constituent continuity, and momentum equations, cast in conservation law form:

\[
\frac{\partial \rho_k}{\partial t} + \frac{\partial \rho_k u_i}{\partial x_i} = \sum_{k=1}^{n_{phase}} \Gamma_{kl} - \sum_{l=1}^{n_{phase}} \Gamma_{lk}, \quad k, l = 1, n_{phase}
\]

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \rho g_i + \frac{\partial \tau_{ij}}{\partial x_j}
\]  

In equation 1, the viscous stress tensor is given by:

\[
\tau_{ij} = \mu_{ij} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

with mixture density defined as:

\[
\rho = \sum_k \rho_k a_k
\]

The formulation is fully compressible, incorporating energy transport and generalized state relations for the liquid and vapor phases. As demonstrated in [2], incorporation of compressibility can have a significant impact on predicted cavity dynamics in re-entrant sheet cavity flows of interest here. In all of the computations presented in this paper, an isothermal state relation is employed, wherein sound speeds for each constituent are specified, and the energy equation is not solved. In [3] it is demonstrated that for the low speed flows considered here, an isothermal compressibility approximation is appropriate.

In this work 2 phases are considered, liquid and vapor. Mass transfer modeling is employed for the vaporization and condensation processes associated with natural cavitation; these models for $\Gamma_{\text{liquid-vapor}}$ and $\Gamma_{\text{vapor-liquid}}$ are detailed in [1, 4].

**Turbulence Modeling**

A high Reynolds number $k-\varepsilon$ turbulence model with wall functions and its Detached Eddy Simulation (DES) variant are employed. Transport equations for $k$ and $\varepsilon$ are written:
obtained by defining a modified turbulence energy destruction

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_{k \ell}} \frac{\partial k}{\partial x_j} \right) + \rho \varepsilon
\]

and the eddy viscosity is obtained from:

\[
\mu_t = \rho C_\mu k^2/\varepsilon
\]

In the context of the \(k-\varepsilon\) model, a DES variant is obtained by defining a modified turbulence energy destruction term as (see [20] for example):

\[
\rho \varepsilon \rightarrow \rho \varepsilon \cdot F_{\text{DES}}
\]

\[
F_{\text{DES}} = \text{MAX} \left[ \frac{L_1}{C_{\text{DES}} \Delta}, 1 \right], L_1 = \frac{k^{3/4}}{\varepsilon}, \Delta = \text{MAX}(\delta_x, \delta_y, \delta_z)
\]

**Numerical Method**

The numerical method is evolved from a single-phase, block-structured, pseudo-compressibility formulation. Roe-based flux difference splitting with 3rd order MUSCL interpolation is utilized for convection term discretization. An implicit procedure is adopted with inviscid and viscous flux Jacobians approximated numerically. A block-symmetric Gauss-Seidel iteration is used to solve the approximate Newton system at each pseudo-timestep.

The multiphase code retains these underlying numerics but has been evolved to incorporate full compressibility, species continuity transport, mass transfer, dual-timestepping, flux limiting and two-equation turbulence modeling. The code is structured multi-block and parallel. A good deal of effort has gone into development of the all-Mach number, multiphase preconditioning formulation used [2-3]. Further details on the numerics can be found in [1-5].

**RESULTS**

**Configuration and Numerical Modeling Details**

Figure 1 shows a schematic of the hydrofoil geometry as offered in [23]. The nominally two-dimensional configuration is mounted at a 7° angle-of-attack. The equation for the upper surface of the symmetric foil geometry is provided as:

\[
y/c = a_0 \sqrt{x/c} + a_1 x/c + a_2 (x/c)^2 + a_3 (x/c)^3 + a_4 (x/c)^4
\]

\[
\begin{align*}
a_0 &= 0.11858 \\
a_1 &= -0.02972 \\
a_2 &= 0.00593 \\
a_3 &= -0.07272 \\
a_4 &= -0.02207
\end{align*}
\]

The x-y plane multi-block structured mesh employed is illustrated in Figure 2. A total of 37,140 cells were employed in the 2D model. This choice of grid size is based on grid studies that have been performed and documented by the authors for similar developed cavitation analyses [1, for example]. The model was extended 10 chord lengths downstream from the nominal domain exit depicted in Figure 1, in order to facilitate the specification of a constant exit pressure boundary condition (The location of the cavitation reference pressure used in the simulations remains at point R in Figure 1). A near wall grid spacing of 0.0002m was specified in order to achieve \(y^+\) values \(\leq 30\) at the first cell center off of the hydrofoil and channel boundaries, consistent with the high Reynolds number form turbulence model employed. These computational domain was partitioned into 20 blocks and the 2-D simulations were run on 20 processors of local LINUX PC cluster systems. Figure 3 shows a close-up view of the airfoil region grid. Figure 4 shows a typical contour plot of predicted liquid volume fraction and velocity vectors, illustrating the complex unsteady nature of the predicted flow field and the sharpness that bubble interfaces are preserved.

For the 3-D simulations, the grid shown in Figure 2 was extruded in the z-direction by 1 chord. 100 equally spaced cells (\(\Delta z = .001\ m\)) were employed in this spanwise direction for a total of 3,714,000 cells. Symmetry boundary conditions were specified at \(z/c = 0.0, 1.0\). The 3-D computational domain was partitioned into 100 blocks and the simulations were run on 100 T3E processors available through the US High Performance Computing and Modernization Office.

All of the simulations (except single phase) were carried out in a time accurate fashion. Specifically, for all of the 2D simulations, a timestep of \(\Delta t^* = \Delta t/(c/U_\infty) = 0.0005\) was specified, based on a series of URANS studies performed on the \(\sigma=0.8\) case. A formal \(\Delta t\) study was not performed, since as seen below, even after 60,000 timesteps, the simulations are not fully stationary. Still, this value resolves each dominant re-entrant ejection event with on the order of 10,000 timesteps. For the 2D simulations, 10 pseudo-timesteps are run for each physical timestep, as justified in [4].

The inflow velocity was set to a constant value of 6m/s. An inlet turbulence intensity of 0.01 and turbulence length scale of .0005* the channel height were specified. The liquid and vapor densities were set to 1000 and 1 kg/m\(^3\) respectively. The constant liquid and vapor sound speeds were set to 1500 and 420 m/s respectively. The cavitation number was defined in terms of the average value of \(p_R\) (see Figure 1), by accumulating the average of this time-varying quantity as the simulations proceeded.

**2D Results**

Five sets of 2D simulations were performed. A single phase computation was carried out as well as two simulations each at cavitation numbers of 0.4 and 0.8. The single phase calculation converged to a steady state, but the multiphase simulations exhibited highly unsteady behavior. Unsteady RANS analysis was performed for each of \(\sigma=0.4, 0.8\). Also, a “2D DES” simulation was performed at each of these cavitation numbers. Though a 2D simulation cannot capture the energy cascade of turbulence, these “2D DES” runs were performed to
provide experience and guidance with the model, before applying it to a 3D simulation described below.

Figures 5 through 9, and Table 1, provide details of the 2D results that were obtained. The quantities to be computed by participants in this session are time averaged lift and drag coefficients, and various cavity shape parameters. Though lift and drag are unambiguously computed, determination of the average cavity shape can be somewhat discretionary. Specifically, in Figure 5, the time average cavity shape (defined as the time average contour of \( \alpha = 0.9 \)) is shown for each of the 2D cases. The cavitation detachment location near the leading edge is easily extracted from this plot. The maximum cavity length, though uniquely defined, is less informative than when simple cavity models are applied, due to the complex shape of the cavity near its trailing edge. Similar ambiguities apply in extracting the maximum cavity thickness. Nevertheless, rather than attempt to smooth or fair the predicted average cavity shape in order to obtain a comparison measure more representative of simpler cavity models, the authors have reported the absolute maxima shown in Figure 5. The average total vapor volume \( (V_v = \sum \alpha_v V_{cell}) \) is provided in Table 1, for the four 2D cases, as an additional indicator of bubble size. (Reported volumes assume a spanwise extent of 0.1m).

The 2D \( \sigma=0.8 \) results are reported in Figures 6 and 7. In Figure 6a, the 2D URANS lift and drag histories are seen to exhibit highly unsteady behavior. An average lift coefficient of \( \bar{C}_L = 0.492 \) and drag coefficient of \( \bar{C}_D = 0.080 \) are obtained, representing a 21% decrease and 230% increase respectively compared to the single phase values. A dominant Strouhal frequency of approximately 0.10->0.12 is observed. Accordingly, despite the 60,000 timesteps run for a non-dimensional record of \( t^* = 30 \), the simulations are not yet fully statistically stationary. It is observed that the lift and drag can become negative for short durations at this cavitation number. Figure 6b shows the minimum and maximum (over time) and average pressure coefficient distributions on the foil, along with the single phase results. Significant average unloading on the suction side is observed and is principally responsible for the loss of lift. Minimum \( C_P \) values of -0.8 occur along the entire suction surface consistent with the nearly complete enveloping of the suction surface by the cavity at certain times in the transient at the cavitation number run. Maximum pressures on the suction surface are seen to exceed that on the pressure surface on the aft half-chord of the blade, consistent with the instantaneous negative lift values observed in Figure 6a. Figure 6c shows a sequence of predicted instantaneous vapor volume fraction, \( \alpha_v = 0.1 \), for the last one-sixth of the transient, illustrating the highly unsteady nature of the cavity.

Figures 7a-c report the “2D DES” results for \( \sigma=0.8 \). As expected, a significantly richer unsteadiness is observed in the \( C_L \) and \( C_D \) histories. Average lift is almost unchanged, drag increases by 21% compared to URANS. There is less of a clear dominant frequency associated with the “2D DES” simulation. Predicted pressure distributions in Figure 7b show similar trends to URANS. Inspection of the time sequence in Figure 7c shows that the DES cavity is composed of more smaller vapor structures, and that the quasi-periodic “pinching off” of the vapor bubble by the re-entrant jet closer to the leading edge gives rise to much larger vapor structures being transported downstream. Inspection of Figure 5, indicates that the predicted average bubble size is somewhat increased for the “2D DES” computation, as quantified in Table 1 by a 59% increase in vapor volume.

The 2D \( \sigma=0.4 \) results are reported in Figures 8 and 9. Again, highly unsteady lift and drag histories are observed. Average \( C_L \) has now decreased to 0.198, a 68% reduction from single phase. \( C_D \) is reduced from the \( \sigma=0.8 \) value to .068, still a 183% increase over single phase. An apparently slightly lower dominant Strouhal frequency of \( 0.08->0.1 \) is observed, compared to the \( \sigma=0.8 \) URANS result. Lost lift is due to a virtually flat mean \( C_P \) distribution on the suction side due to the envelopment of the suction surface by the now much larger cavity, i.e., \( -C_P \approx 0.4 \). Figure 8c again shows a sequence of predicted instantaneous vapor volume fraction, \( \alpha_v = 0.1 \), for the last one-sixth of the transient. Like the \( \sigma=0.4 \) calculations, we observe a richer unsteadiness when turning on the “2D DES” but now there is observed a slight decrease in lift and virtually no change in drag compared to the URANS results. Inspection of Figure 5, again indicates that the predicted average bubble size is somewhat increased for the “2D DES” computation, as quantified in Table 1 by a 35% increase in vapor volume.

Table 1 provides a summary of the predicted lift and drag coefficients and various bubble size parameters for the 2D runs.

**TABLE 1. PREDICTED CAVITATION PARAMETERS**

<table>
<thead>
<tr>
<th>Time average quantity</th>
<th>Single Phase</th>
<th>( \sigma=0.8 )</th>
<th>( \sigma=0.4 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_L )</td>
<td>0.624</td>
<td>0.492</td>
<td>0.501</td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.024</td>
<td>0.080</td>
<td>0.097</td>
</tr>
<tr>
<td>( L_{g/c} )</td>
<td>NA</td>
<td>.0045</td>
<td>.0050</td>
</tr>
<tr>
<td>( L_{max/c} )</td>
<td>NA</td>
<td>1.03</td>
<td>1.09</td>
</tr>
<tr>
<td>( t_{max/c} )</td>
<td>NA</td>
<td>0.17</td>
<td>0.22</td>
</tr>
<tr>
<td>( V_v (10^{-4} m^2) )</td>
<td>NA</td>
<td>2.74</td>
<td>4.38</td>
</tr>
</tbody>
</table>

**3D Results**

For the 3D DES simulations, the timestep was doubled to \( \Delta t^* = 0.0010 \) and the number of inner iterates halved to five in order to return a factor of 4 benefit in the solution run times. Only the \( \sigma=0.8 \) case has been run out to date. A large timestep URANS run was first performed to initialize the simulation. This was followed by a 30,000 timestep DES run with one pseudo-timestep per physical timestep. The “final” DES run reported here was run for 30,000 timesteps using 5 pseudo-timesteps per physical timestep. This required approximately 20,000 processor hours on a Cray T3E, for a total wall clock turnaround time of about a week on 100 processors.

The resulting flow field prediction exhibits a highly three-dimensional unsteady character as indicated in Figure 10. There, a composite of \( \alpha_v = 0.9 \) isosurfaces are presented for 21
timesteps between $t^*=15$ and $25^*$. Resolved large scale features include spanwise, streamwise and horseshoe vortical structures, a spanwise oscillatory pinching off of the vapor bubble by a re-entrant jet in the leading edge region, merging and splitting of geometrically complex vapor structures, and spanwise wave structures propagating axially along the cavity interface near the leading edge.

Figure 11 shows an instantaneous view of the predicted $\alpha_L=0.9$ isosurface along with a number of particle pathlines colored by static pressure, indicating the complex vortical nature of the DES flow, including significant axial vorticity which vapor structures tend to track.

A visual comparison of the lift and drag coefficient time histories from the 2D (Fig. 7a) and 3D (Fig. 12) DES solutions shows that they approach a similar mean value, and exhibit a range of maxima to minima of approximately 1.2 to -0.2. One can see clearly in the 3D result a low frequency oscillation that suggests that we have achieved only a few large-eddy turnover times in the solution, too few to yield a reliable mean statistic.

The striking difference between the 2D and 3D runs is the decreased fine-scale content in the 3D data. A number of factors have been identified that may contribute to this behavior. We note that the time step size for the 3D case is a factor of 2 larger than for the 2D case leading to increased temporal filtering which would reduce the fine scale content. The observation that the 3D solution has not yet reached a statistically steady state also suggests that its finer scales have not fully energized.

Another potential cause for the decrease in fine scale content in the 3D simulation may be near-surface physics. The primary mechanism for producing fluctuations is gradient production in the longitudinal velocity variance budget. The cross-stream and wall normal fluctuations receive their energy from pressure redistribution. The production is similar in magnitude in the present 2D and 3D simulations, but because global production must balance global dissipation, the loss of the transverse direction in the 2D case means that this balance is achieved through increased destruction in the longitudinal and wall-normal directions. This increase can only be accomplished by amplified fine-scale gradients which come from increased fine-scale activity. This latter conjecture will be analyzed in detail once a sufficient sample from the 3D case is collected.
Figure 6a. Lift and drag coefficient history for 2D URANS simulation, $\sigma=0.8$.

Figure 6b. Predicted pressure coefficients for 2D URANS simulation, $\sigma=0.8$.

Figure 6c. Predicted instantaneous and average $\alpha_{\text{vapor}}=0.1$ contours over final $t^*=5$ for 2D URANS simulation, $\sigma=0.8$. 
Figure 7a. Lift and drag coefficient history for “2D DES” simulation, $\sigma=0.8$.

Figure 7b. Predicted pressure coefficients for “2D DES” simulation, $\sigma=0.8$.

Figure 7c. Predicted instantaneous and average $\alpha_{\text{vapor}}=0.1$ contours over final $t^* = 5$ for “2D DES” simulation, $\sigma=0.8$. 
Figure 8a. Lift and drag coefficient history for 2D RANS simulation, $\sigma=0.4$.

Figure 8b. Predicted pressure coefficients for 2D URANS simulation, $\sigma=0.4$.

Figure 8c. Predicted instantaneous and average $\alpha_{\text{vapor}}=0.1$ conotours over final $t^*=5$ for 2D URANS simulation, $\sigma=0.4$. 
Figure 9a. Lift and drag coefficient history for “2D DES” simulation, $\sigma=0.4$.

Figure 9b. Predicted pressure coefficients for “2D DES” simulation, $\sigma=0.4$.

Figure 9c. Predicted instantaneous and average $\alpha_{vapor}=0.1\con$otours over final $t^*=5$ for “2D DES” simulation, $\sigma=0.4$. 

9
Figure 10. Composite of $\alpha_1=0.9$ isosurfaces for 21 timesteps between $t^*=15$ and $25^*$ for DES computation of $\sigma=0.8$ cavitating hydrofoil case.
CONCLUSION

A cavitating hydrofoil case was analyzed using a homogeneous multiphase CFD method. The geometry and test conditions were defined according to the CAV2003 workshop: “Models and CFD Tools for Computation of Cavitating Flows;” [23]. Unsteady RANS using a conventional k-ε model and its Detached Eddy Simulation variant were performed. Time average lift, drag and cavitation bubble geometry parameters were reported. The transients were run for a non-dimensional time of $t/(c/U_\infty) = 30$, which though not fully statistically stationary, is enough to resolve several ($\approx 3$) global re-entrant jet turnover cycles.

The DES simulation exhibits a richly unsteady three-dimensional behavior as expected. However, despite the high CPU requirements of this simulation presented, the results are far from statistically stationary. The lift and drag histories exhibit less richness than the 2D simulations. Several possible physical and numerical explanations for this were postulated and as we evolve the DES simulation we hope to fully understand this behavior.

ACKNOWLEDGMENTS

This work was supported by the Office of Naval Research, grant #N00014-01-1-0325, with Dr. Kam Ng as contract monitor. The third author was supported under an Applied Research Laboratory Undergraduate Summer Internship in Fluid Mechanics Research. High performance computing resources were provided through a Challenge program award from the US Department of Defense High Performance Computing Modernization Program.

REFERENCES


