31st Symposium on Naval Hydrodynamics
Monterey, California, 11-16 September 2016

Towards Large Eddy Simulation of Hull-attached Propeller in Crashback

Praveen Kumar and Krishnan Mahesh
(Aerospace Engineering & Mechanics, University of Minnesota, USA)

ABSTRACT

Crashback is an extreme off-design operating condition in which the vessel moves in the forward direction while the propeller rotates in the reverse direction. Previously, large-eddy simulation (LES) has been shown to accurately simulate propeller crashback. Verma et al. (2012) studied the effect of hull in crashback by simulating flow over the stern portion of an axisymmetric hull with a propeller attached to it. They reproduced the experimentally observed effect of hull to drastically increase the side-force magnitude at higher negative advance ratio and explained the reason behind this behaviour. In this paper, we present our work towards simulating entire hull with a propeller in crashback. This problem has two major challenges: the boundary layer on the hull and massively separated unsteady flow in the vicinity of propeller. We perform wall-resolved LES of flow over axisymmetric hull alone at $Re = 1.1$ *million* based on hull length, which shows good agreement with experiment for pressure and skin-friction coefficient as well as drag. The hull's wake is sensitive to the axisymmetric turbulent boundary layer on the hull. In order to study the hull boundary layer, the recycle-rescale methodology originally proposed by Lund et al. (1998) for generating turbulent inflow for spatially growing flat plate turbulent boundary layer, is extended for generating inflow for thin axisymmetric turbulent boundary layers and implemented on unstructured grid. The simulations are validated with the data available in literature showing good agreement.

INTRODUCTION

A marine vessel is quickly decelerated by rotating the propeller in reverse direction, thus yielding negative thrust. This extreme off-design operating condition in which the vessel moves in the forward direction while the propeller rotates in the reverse direction is called crashback. The flow in crashback is highly unsteady and massively separated which makes it difficult to simulate using traditional methods. The large unsteady forces generated in crashback can potentially damage blade structure. The side-forces can affect maneuverability of the vessel. Thus, understanding crashback is critical to better design and operation of marine vessels.

The use of computational fluid dynamics (CFD) for the development of engineering devices is becoming more popular in recent years with the advent of powerful supercomputers. Direct numerical simulation (DNS) of flows at high Reynolds number is not computationally feasible (Moin and Mahesh, 1998) and RANS fails for crashback flows (Davoudzadeh et al., 1997; Chen and Stern, 1999). LES is an intermediate method between DNS and RANS in which the large scales of motion are resolved and the effect of small scales is modeled. Mahesh et al. (2015) showed the capability of LES to accurately simulate a variety of complex marine flows including crashback. Marine flows have very high Reynolds numbers and therefore a broad range of length and time scales. The properties of the numerical algorithm used to solve the LES equations are important to ensure that the numerical dissipation not damp the smallest resolved scales and overwhelm the interscale transfer represented by the the subgrid model. Mahesh et al. (2004) developed a numerical algorithm for LES of complex flows which emphasizes discrete conservation of the kinetic energy, thus ensuring robustness at high Reynolds number without any numerical dissipation. This algorithm will be used to perform the LES reported in this paper.

The simulation of hull-attached propeller in crashback has two major challenges: the thin boundary layer on the hull and the complex flow due to reverse-rotating propeller. The evolution of turbulent boundary layer on the hull needs to be captured accurately. This requires very fine grid resolution near wall to resolve the fine near-wall structures which determine drag. The state of boundary layer determine the location of flow separation, which in turns determine the thickness of wake of hull. The separated flow in the stern region of hull interacts with the flow field of reverse rotating propeller, which is typically mounted on the stern. This interaction between the incoming wake of hull and the fluid pushed upstream by the reverse rotating propeller forms ring vortex, a characteristic feature of crashback flows (Jang and Mahesh, 2013; Verma et al., 2012; Jang and Mahesh, 2012; Kumar and Mahesh, 2015). This unsteady separated flow is challenging because of the complex geometry of the propeller and the low frequencies due to flow separation. Thus, LES of hull-attached propeller in crashback is demanding in terms of grid generation for the complex propeller geometry and the need to run for long periods of time to capture important flow physics. The present paper is focussed on the evolution of boundary layer on the hull.

Turbulent boundary layers are one of the most studied canonical fluid problems but most of the studies are devoted to flat plate turbulent boundary layers (FPTBL). A variety of hydrodynamic engineering applications involve axisymmetric turbulent boundary layers (ATBL). The complexity in ATBL comes from the fact that there is an additional parameter for curvature effects. Based on two length parameters namely δ/a and a^+ which are ratio of boundary layer thickness to the radius of curvature and radius of curvature in wall units respectively, three flow regimes can be identified (Piquet and Patel, 1999): (i) both δ/a and a^+ are large, (ii) large δ/a and small a^+ and (iii) small δ/a and large a^+. Large effect of curvature is felt in the first flow regime. The second flow regime behaves like an axisymmetric wake. Most of the past experiments and computations are devoted to these two flow regimes. The flow regime of our interest i.e. flow over hull is the third one where δ/a is small (<1) and a^+ is large. Afzal and Narasimha (1976) modified the classical law of the wall to include the effect of curvature which was applicable to thin ATBL with the error being of order $1/a^+$. Due to lack of many DNS or experimental data for thin ATBL at high Reynolds number, we will use the modified law of the wall to validate our results.

Figure 1 is a picture taken from water tunnel showing a fully-appended hull with propeller attached to it. The generic submarine hull (DARPA suboff) geometry DTMB model 5470 (Groves et al., 1989) is shown in figure 2. It has a fairwater sail near the front portion and four appendages near the end. This hull has been used in numerous experiments (Cook, 1990; Atsavapranee et al., 2004) conducted by US Navy. Huang et al. (1992) con-

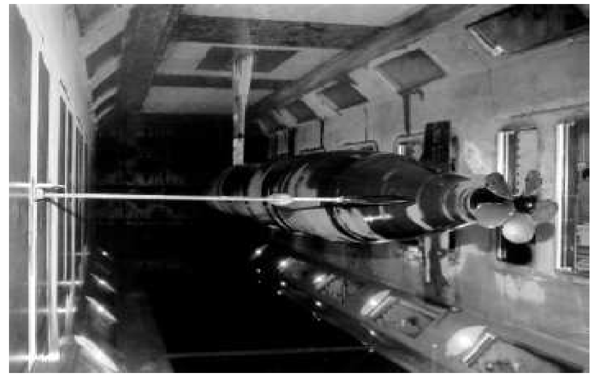


Figure 1: Fully-appended hull with propeller P4381 in water tunnel (Bridges, 2004).

Figure 2: Geometry of DARPA suboff DTMB model 5470 (Groves et al., 1989).

ducted experiments for flow over suboff without appendages. They measured pressure and skin-friction coefficients on the hull. Jimenez et al. (2010b) provides extensive experimental data for the hull without appendages at wide range of Reynolds number ranging from 1.1 *million* to 67 *million*. They measured wake profile at $3, 6, 9, 12$ and 15 diameters downstream from the stern in the midline plane. In the present paper, we use DARPA Suboff without appendages as the hull geometry because of availability of experimental data (Huang et al., 1992; Jimenez et al., 2010b).

The objectives of the present paper are to: (i) simulate wall-resolved attached flow over hull alone and (ii) extend and validate recycle-rescale method of turbulent inflow generation for thin ATBL. The use of recycle-rescale method for computational studies of flow over stern and the wake of hull is evaluated and discussed.

ZERO-PRESSURE-GRADIENT THIN AXISYMMETRIC TURBULENT BOUNDARY LAYER (ATBL)

Unlike channel or pipe flow, in spatially evolving boundary layers the boundary layer thickness and the wall shear stress are a function of streamwise distance which makes the flow inhomogenous in streamwise direction. The pioneering work for turbulent inflow generation for a spatially developing turbulent boundary layer was done by Spalart (1988). He used a set of coordinate transformations to minimize the streamwise inhomogeneity so that the periodic boundary conditions in that direction can still be used. However, this method introduces additional terms in the Navier–Stokes equations causing additional complications. Lund et al. (1998) modified the Spalart method to make it simpler and easy to implement. Instead of using growth terms, Lund et al. (1998) proposed the so-called recycle-rescale method where flowfield at a streamwise location downstream is rescaled as per known well-established theoretical boundary layer scaling laws and re-introduced at the inflow. This way, the simulation generates its own inflow. This algorithm has been shown to avoid long development region needed to generate developed boundary layer, which is otherwise needed for the boundary layer to develop starting from a parallel flow with random fluctuations. The details of the recycle-rescale method for inflow generation is briefly described as follows.

The velocities at the inflow plane, $x = x_{in}$ are written as:

$$\begin{aligned} u(y,z,t) &= \beta[\gamma\bar{U}(x_r, y_r^o, t) + (1-\gamma)U_\infty \\ &+ \gamma u'(x_r, y_r^o, z_r, t)] \\ &+ (1-\beta)[\gamma\bar{U}(x_r, y_r^i, t) \\ &+ \gamma u'(x_r, y_r^i, z_r, t)], \quad (1) \\ v(y,z,t) &= \beta[\bar{V}(x_r, y_r^o, t) + \gamma v'(x_r, y_r^o, z, t)] \\ &+ (1-\beta)[\bar{V}(x_r, y_r^i, t) \\ &+ \gamma v'(x_r, y_r^i, z, t)] \quad (2) \\ w(y,z,t) &= \beta\gamma w'(x_r, y_r^o, z, t) \\ &+ (1-\beta)\gamma w'(x_r, y_r^i, z, t) \quad (3) \end{aligned}$$

where the $\bar{(\cdot)}$ is the spanwise average through time, 'r' denotes the recycle plane, 'i' denotes the inner scale, 'o' denotes the outer scale. The inner scales are based on the $y^+ = yu_\tau/\nu$ scaling and the outer scales are based on $\eta = y/\delta_{99}$. β is the Lund's weighting function as given in eq. 4, which is used to blend the inner and outer scales. The values of constants in eq. 4 are $a = 4$ and $b = 0.2$.

$$\beta(\eta) = \frac{1}{2}\left\{\frac{1 + tanh\left(\frac{a(\eta-b)}{(1-2b)\eta+b}\right)}{tanh(a)}\right\} \quad (4)$$

The mean velocity profile is obtained by spanwise averaging at every timestep and then averaging over a sliding time window. The averaging time window is initially set to $T = A\delta_{99,i}|_0/U_\infty$ where $A = 10$ to discard the transients and then switched to 100 once the transients die out. The running average,

$$\mathcal{F}(t) = \left(1 - \frac{\Delta t}{T}\right)\mathcal{F}(t-\Delta t) + \frac{\Delta t}{T} f(t) \quad (5)$$

where f is the instantaneous spanwise average. Finally, the averaging is switched to a simple running average with $T = T_0 + t - t_0$ where, t is the time in the simulation, t_0 is the time at which the running averaging was initiated and T_0 is the value of the averaging interval used prior to t_0. This mean boundary layer velocity profile is used to evaluate the scaling parameters $(\theta, \delta_{99}, u_\tau)$ as a function of streamwise location.

In order to generate the desired turbulent inflow, the inflow parameters $u_{\tau,i}$, θ_i and $\delta_{99,i}$ are specified. The value of θ_i is kept fixed whereas $u_{\tau,i}$ is computed at every timestep using eq. 6, knowing the values of flow parameters at the rescale plane.

$$u_{\tau,i} = u_{\tau,r}\left(\frac{\theta_r}{\theta_i}\right)^{(1/8)}, \quad (6)$$

The rescale parameter γ is defined as:

$$\gamma = \frac{u_{\tau,i}}{u_{\tau,r}} = \left(\frac{\theta_r}{\theta_i}\right)^{(1/8)} \quad (7)$$

which is used to construct velocities at the inflow plane. The constructed inflow plane velocity profiles are adjusted through a Newton–Raphson scheme to obtain $\delta_{99,i}$. This way any general inflow-outflow code can be converted into a self-contained turbulent inflow generator.

In the present work, we extend the original recycle-rescale method to axisymmetric flow and implement it in unstructured framework for the generation of turbulent inflow. As mentioned in previous section, ATBL has an additional length parameter to take into account the curvature effects. However for the problem of our interest, the boundary layer is thin and at high Reynolds number i.e. small δ/a and large a^+. In this flow regime, the recycle-rescale method for flat plate turbulent boundary layers can be extended to ATBL by simply changing wall-normal (y) and spanwise (z) coordinates to radial (r) and azimuthal (θ) coordinates respectively.

SIMULATION DETAILS

Numerical method

In LES, large scales are directly accounted for by the spatially filtered Navier–Stokes equations, whereas the effect of small scales is modeled. The spatially filtered incompressible Navier–Stokes equations are formulated for the absolute velocity vector in the inertial frame as follows:

$$\begin{aligned} \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\bar{u}_i \bar{u}_j) &= -\frac{\partial \bar{p}}{\partial x_i} \\ &+ \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j}, \\ \frac{\partial \bar{u}_i}{\partial x_i} &= 0 \end{aligned} \tag{8}$$

where u_i is the velocity in the inertial frame, p is the pressure, ν is the kinematic viscosity, the overbar denotes the spatial filter and $\tau_{ij} = \overline{u_i u_j} - \bar{u}_i \bar{u}_j$ is the sub-grid stress. The dynamic Smagorinsky model proposed by Germano et al. (1991) and modified by Lilly (1992) is used to model the subgrid stress terms.The Langrangian time scale is dynamically computed based on surrogate-correlation of the Germano-identity error. This approach extended to unstructured grid has shown good performance for a variety of cases including flow past a marine propeller in crashback (Verma and Mahesh, 2012).

Eq. 8 is solved by a numerical method developed by Mahesh et al. (2004) for incompressible flows on unstructured grids. The algorithm is derived to be robust without any numerical dissipation. It is a finite volume method where the cartesian velocities and pressure are stored at the centroids of the cells and the face normal velocities are stored independently at the centroids of the faces. A predictor–corrector approach is used. The predicted velocities at the control volume centroids are first obtained and then interpolated to obtain the face normal velocities. The predicted face normal velocity is projected so that the continuity equation in eq. 8 is discretely satisfied. This yields a Poisson equation for pressure which is solved iteratively using a multigrid approach. The pressure field is used to update the cartesian control volume velocities using a least-square formulation. Time advancement is performed using an implicit Crank–Nicholson scheme. The algorithm has been validated for a variety of problems over a range of Reynolds numbers (see Mahesh et al., 2004). Chang et al. (2011) used this algorithm to peform wall-resolved LES of weakly separated flows on a range of Reynolds number.

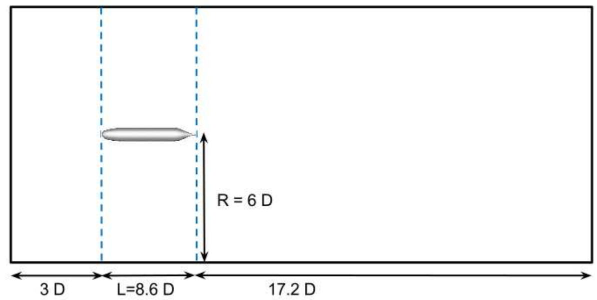


Figure 3: The computational domain used for simulations of flow over hull.

Problem Description

Flow over hull

Simulations are performed for flow over barehull at a Reynolds number of 1.1 *million* based on length of hull. The computational domain used for these simulations consists of a circular cylinder of diameter $12D$ where, D is the maximum diameter of the hull. The hull is situated on the axis of this cylindrical domain such that the nose of the hull is the origin of the reference coordinate system. The domain extends $3D$ upstream of the nose and $17.2D$ downstream of the stern of the hull as shown in figure 3. The preliminary simulations (not shown here) show that the radial and inflow confinement effects are negligible for this domain. Freestream boundary conditions are prescribed at the inlet and the lateral boundaries and convective boundary conditions are imposed at the exit. The computational grid contains 535 million hexahedral control volumes. The grid is clustered near the hull with a first spacing of $0.0003D$ and a growth ratio of 1.01 to capture the hull boundary layer.

ATBL using recycle-rescale method

The ATBL is simulated using the recycle-rescale method. The computational domain is a cylinder of length $10a$ and outer radius $2a$, where a is the radius of cylinder. The axial, radial and azimuthal directions contain 300, 100 and 200 control volumes respectively. The grid is uniform in streamwise (x) and azimuthal (θ) direction with a gentle growth in wall-normal (r) direction. Simulations are performed for

Table 1: Simulation parameters for ATBL using recycle-rescale methodology.

Case	$Re_{\theta,in}$	θ_{in}/a	Re_a
1	356	0.017	20800
2	1400	0.07	20000

spatially developing boundary layer with two different sets of parameters using recycle-rescale methodology. The Re_θ is the Reynolds number based on momentum thickness (θ) which is defined as:

$$Re_\theta = \frac{U\theta}{\nu} \quad (9)$$

where U is the free-stream velocity and ν is the kinematic viscocity of the fluid. The value of Re_θ at the inflow for the two cases are 356 and 1400 respectively. The parameters for the Case 1 are chosen to match with that of Woods (2006) who performed simulations of flow over circular cylinders at a variety of parameters. The parameters of Case 2 are chosen to be representative of flow conditions on hull at a Reynolds number of 1.1 *million*. The recycle plane is located at $x = 8.25a$ for both cases. The parametric details of the simulations run for validation is listed in table 1.

RESULTS

LES of hull

Simulations are performed for flow over hull at a Reynolds number of 1.1 *million* (based on hull length and the freestream velocity) using LES. The boundary layer is tripped at the same streamwise location as that of the experiments performed by Jimenez et al. (2010b) i.e at $x = 0.75D$. This is done by imposing 5 percent steady wall-normal velocity perturbation at the required streamwise location. This triggers the transition of the boundary layer to turbulence with negligible addition of mass in the domain. Figure 4(a) shows contours of instantaneous axial velocity in the midline (xy) plane. The flow accelerates in the bow region of the hull due to favorable pressure gradient. The boundary layer is tripped following which it becomes turbulent. The turbulent boundary layer becomes developed and evolves under zero pressure gradient on the mid portion of the hull. The hull boundary layer encounters adverse pressure gradient in the stern region where it separates to form wake. Overall, the flow is mostly attached. The axial evolution of hull boundary layer is visualized in the zoomed-in view on the hull in figure 4(b).

Figures 5(a) shows pressure coefficient (C_p) on the hull compared to experiments (Huang et al., 1992; Jimenez et al., 2010b) available in literature. Note that the experiments of Huang et al. (1992) were conducted at $Re = 12$ *million*. C_p is insensitive to Re for higher Re. The experiments of Jimenez et al. (2010b) suffer from confinement effects due to tunnel wall and measurement apparatus also (Smits, personal comm.). The wind tunnel used in their experiments has a diameter of $4.8D$ where D is the maximum diameter of hull. This is why our computed pressure coefficient show good agreement with the measurements of Huang et al. (1992), who have corrected their data for tunnel effects. Note that the spike seen in the plot of pressure coefficient (at $x = 0.75D$) is due to tripping. Figures 5(b) shows skin-friction coefficient (C_f) on the hull. For high Re attached flow, $C_f \sim Re^{-0.2}$. The C_f obtained from LES is compared to the scaled experimental data of Huang et al. (1992) (scaled to $Re = 1.1$ *million* based on Re scaling), showing good agreement. The large difference between the C_f from LES and the scaled value from Huang et al. (1992) near the end of stern is due to the fact that the scaling law is valid only for attached boundary layer but the flow separates in the stern region of the hull. Figure 5(c) shows the mean velocity profile at $x = 3.6D$ on the hull. The existence of the viscous sublayer and log layer (law of the wall) in the mean velocity profile indicates fully developed turbulent boundary layer and adequate near-wall resolution of the grid. The time-averaged axisymmetric wake of hull is further averaged in azimuthal direction to obtain mean wake profile, which is compared to measurements of Jimenez et al. (2010b) in figure 6. Jimenez et al. (2010b) computed drag coefficient from the measured wake profile ignoring the drag contribution of pressure and normal stress. So, the computed viscous drag is compared to the drag coefficient reported by Jimenez et al. (2010b). The viscous drag coefficent predicted by LES is 0.141 which shows good agreement with the experimental value (0.142) at $Re = 1.1$ *million*.

ATBL using recycle-rescale method

Figure 7 shows instantaneous axial velocity in xy plane for Case 1 as listed in table 1. The contours of instantaneous axial velocity and azimuthal vorticity in yz plane are shown at $x = 5a$ in figure 8. The mean axial velocity profile is compared to the DNS of Woods (2006), which is at same parameter set

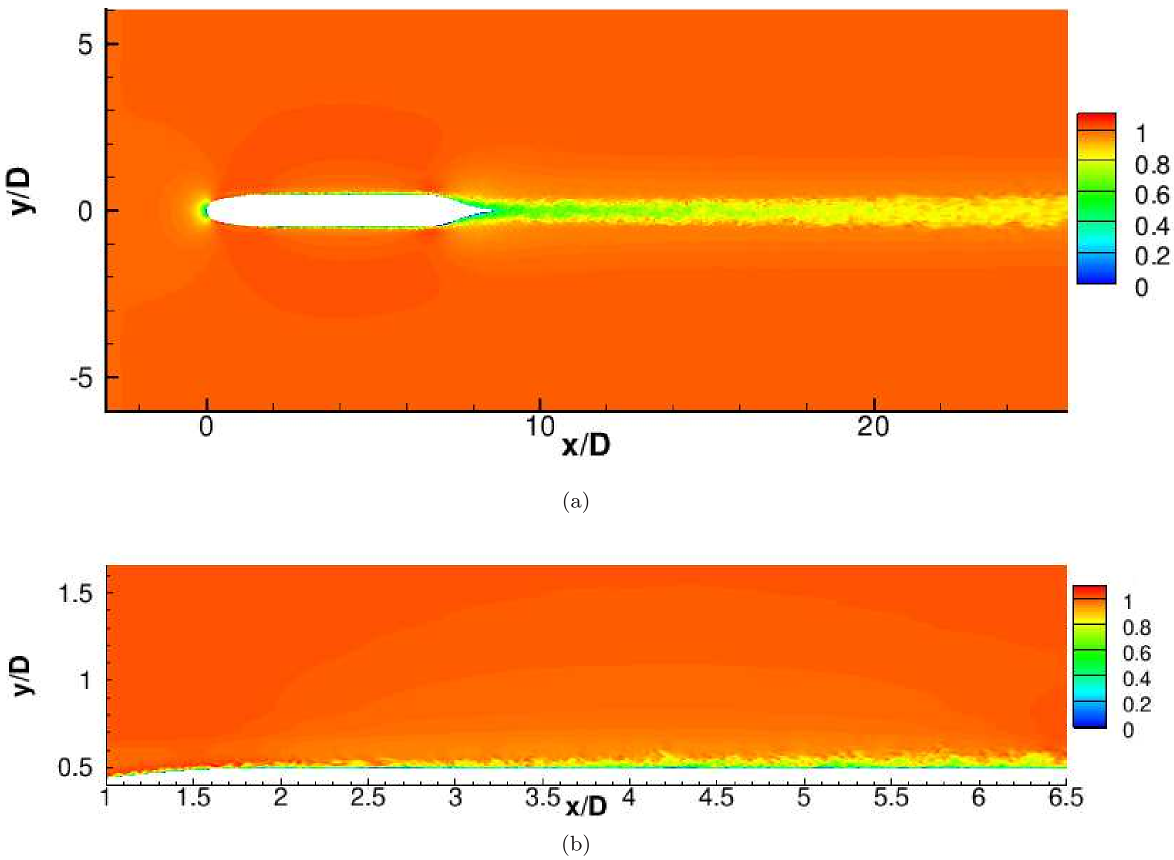
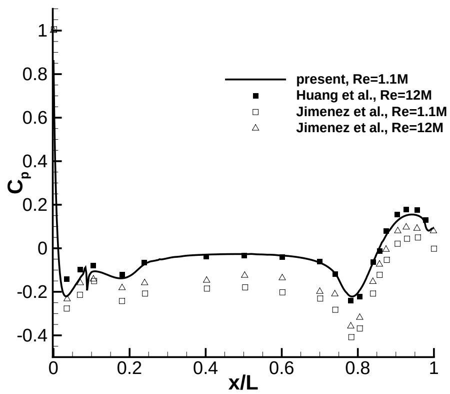


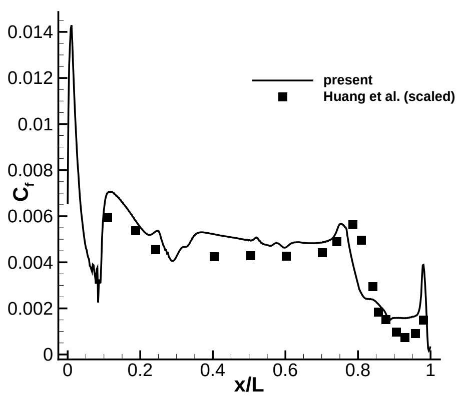
Figure 4: (a) Instantaneous axial velocity contours for flow over barehull at $Re = 1.1$ *million*. (b) Zoomed-in view showing axial evolution of the hull boundary layer.

in figure 9. The theoretical law of the wall (Afzal and Narasimha, 1976) is also shown for comparison. The computed mean velocity profile is slightly smaller than the DNS data in the buffer region of the boundary layer. This is because $Re_\theta = 356$ is at the inflow. A small region near the inflow is known to be affected by the memory of the synthetic profile created by recycle-rescale method, which persists for first few boundary layer thicknesses. This effect can also be seen in the plots of the Reynolds number (Re_θ) and shape factor $H = \delta^*/\theta$ as a function of axial location (figure 10). Simens et al. (2009) noted similar trends in their simulations of turbulent boundary layers using recycle-rescale method. Overall, the result shows good agreement with DNS and theoretical values.

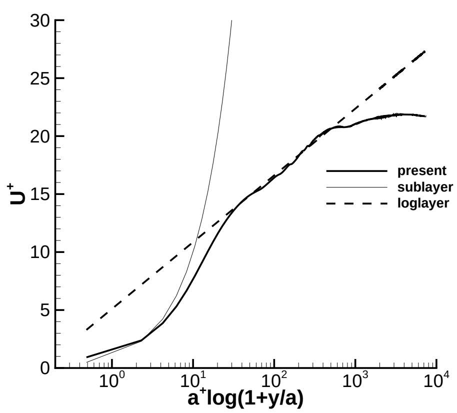
The computed shape factor (H) and boundary layer edge velocity in wall units ($U_0^+ = U_0/u_\tau$, where $U_0 = U(y = \delta_{99})$) are compared to the scaling laws and asymptotic relations available in literature for FPTBL (Monkewitz et al., 2008) and ATBL (Woods, 2006; Monte et al., 2011; Jordan, 2014). Note that the quantity U_0^+ is related to C_f as $U_0^+ = \sqrt{2/C_f}$. The axial evolution of H and U_0^+ show similar slope as compared to the correlations away from the inflow. The computed H is higher than the predicted value for ATBL (Jordan, 2014) but lower than that of FPTBL (Monkewitz et al., 2008). Note that the correlation given by Jordan (2014) is based on simulation database for thick ATBL ($\delta/a > 1$) at high Re. The computed value of U_0^+ is lower than the predicted values for FPTBL (Monkewitz et al., 2008), simlar to the trend shown by H. However, the computed values are higher than the predictions of Woods (2006) but lower than that of Monte et al. (2011). Monte et al. (2011) improved the U_0^+ correlation given by Woods (2006) with their simulation database, which showed better match with the experimental data for ATBL. Note that both Woods (2006) and Monte et al. (2011) used simulations for thick ATBL ($\delta/a > 1$) as well for curve-fitting to obtain these correlations. Another important thing to note is that the recycle-



(a)



(b)



(c)

Figure 5: Flow over hull: (a) C_p, (b) C_f and (c) mean velocity profile on the hull at $x = 3.6D$. Data from experiments (Huang et al., 1992; Jimenez et al., 2010b) are also shown for comparison.

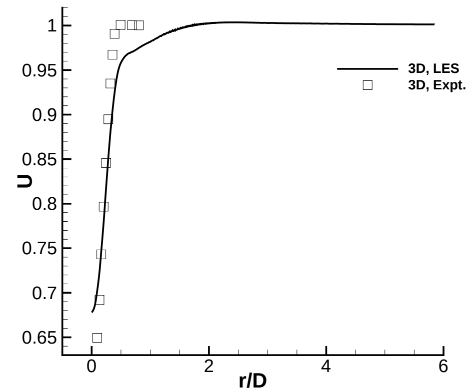


Figure 6: Circumferentially averaged mean axial velocity profile at $3D$ downstream of the hull. Symbols are experimental data of Jimenez et al. (2010b).

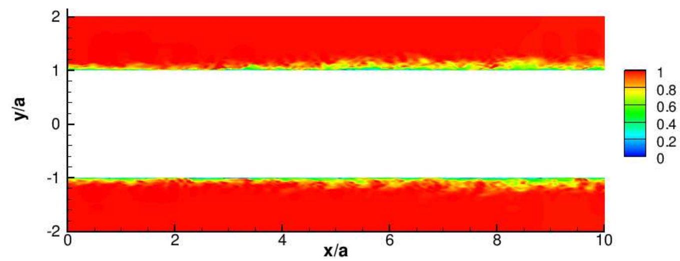
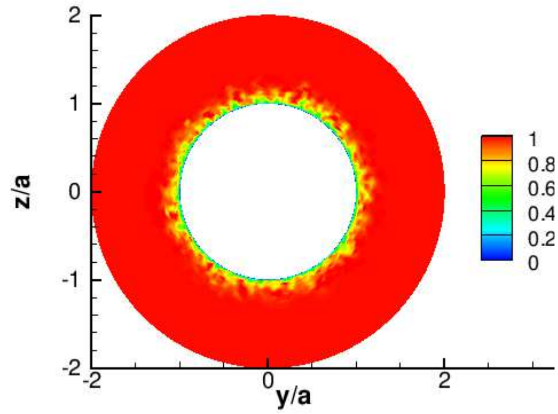


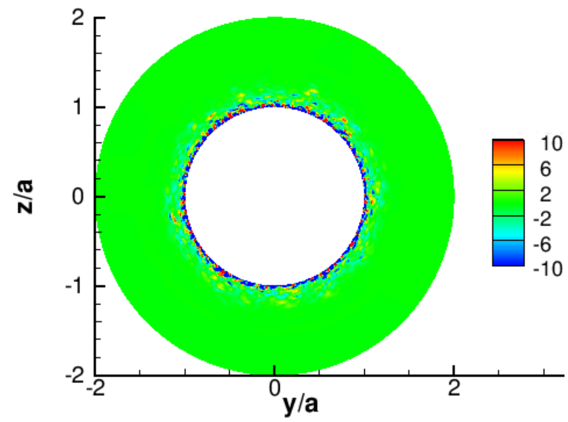
Figure 7: Case 1 ($Re_{\theta,in} = 356$): Instantaneous axial velocity in xy plane.

rescale method is valid at high Re_θ where the boundary layer is fully-turbulent. The Re_θ of Case 1 is quite small compared to Lund et al. (1998). Thus, the quantitative mismatch in the values of H and U_0^+ when compared to the correlation available in the literature can also be due to using recycle-rescale method at low Re. The C_f values for ATBL is known to be higher than that of FPTBL for comparable Re_θ (Lueptow, 1990). This explains the trend shown by U_0^+ to be lower than that of FPTBL.

Case 2 represents a parameter set which is comparable to hull boundary layer. Figure 11 shows instantaneous axial velocity in xy plane for this case. The contours of instantaneous axial velocity and azimuthal vorticity in yz plane are shown at $x = 8a$ in figure 12. The mean axial velocity profile is compared to that of flat plate DNS at the same Re_θ (Jimenez et al., 2010a) in figure 13. The results show that for same Re_θ, ATBL has higher skin-friction than its flat plate counterpart which is well-known



(a)



(b)

Figure 8: Case 1 ($Re_{\theta,in} = 356$): (a) Instantaneous axial velocity and (b) azimuthal vorticity in yz plane at $x = 5a$.

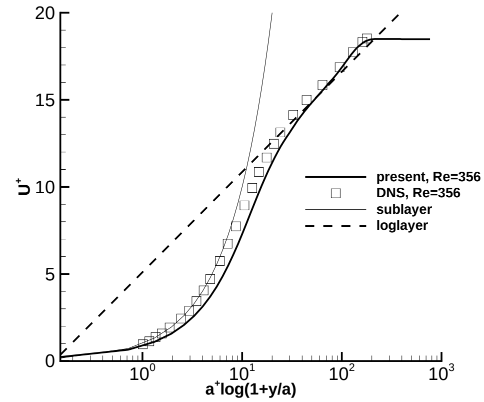


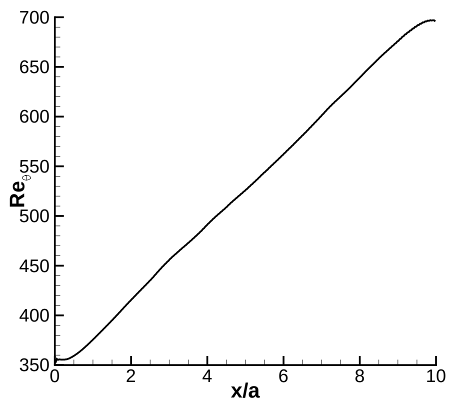
Figure 9: Case 1 ($Re_{\theta,in} = 356$): Mean axial velocity at $Re_\theta = 356$ from Case 1. Symbols are DNS data of Woods (2006).

(Lueptow, 1990). The evolution of boundary layer quantities (Re_θ, H and U_0^+) is shown in figure 14. These quantities show trends similar to Case 1 when compared with the correlations available in the literature. However, there is a smaller transient region close to the inflow clearly seen in the plot of H as compared to Case 1. This supports the argument that the larger transient region near inflow in Case 1 can be the result of using recycle-rescale method at low Re.

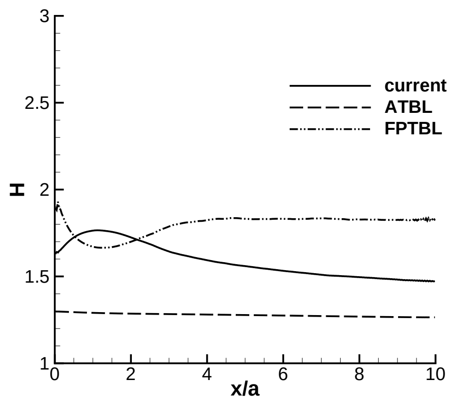
CONCLUSION AND FUTURE WORK

The problem of hull-attached propeller in crashback is split into two major challenges: ATBL on the hull and flow in the vicinity of propeller. This paper focusses on ATBL on the hull. Wall-resolved LES is shown to accurately predict pressure and skin-friction coeffcient as well as overall drag. The boundary layer on the hull is fully developed. The wake of the hull is sensitive to the boundary layer thickness on the hull. The recycle-rescale method is extended to ATBL and implemented on unstructured grid to simulate thin ATBL at high Reynolds number. The simulation results are compared to the data available in the literature as well as theory.

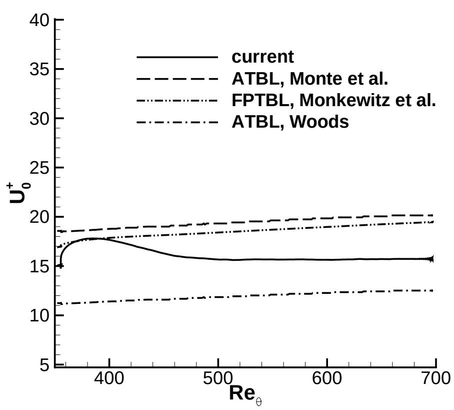
The recycle-rescale method can be a very useful tool to study wake of axisymmetric hull because of lesser computational cost as compared to the LES of entire hull. The flow field on the stern is crucial because propeller is mounted on the stern. This method can in principle be used to study flow field near stern and wake characteristics of the hull



(a)



(b)



(c)

Figure 10: Case 1 ($Re_{\theta,in} = 356$): Spatial evolution of ATBL: (a) Re_θ, (b) H and (c) U_0^+. Results are compared with correlations available in literature for FPTBL (Monkewitz et al., 2008) and ATBL (Woods, 2006; Monte et al., 2011)

.

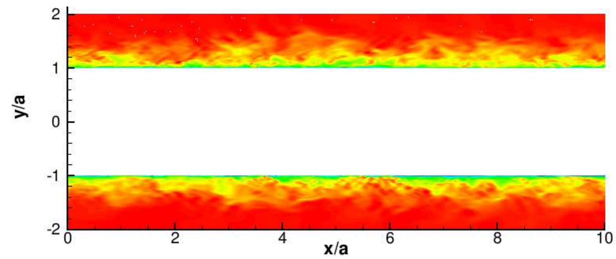
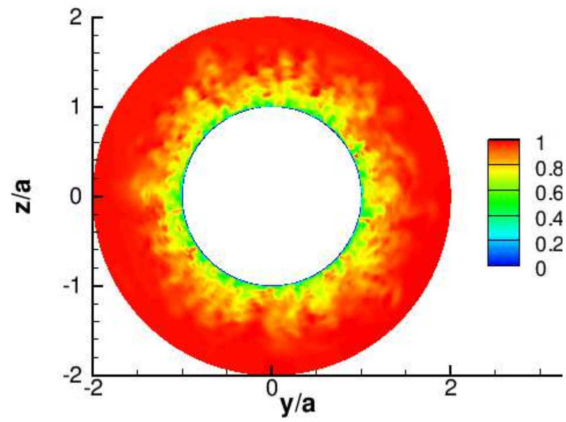
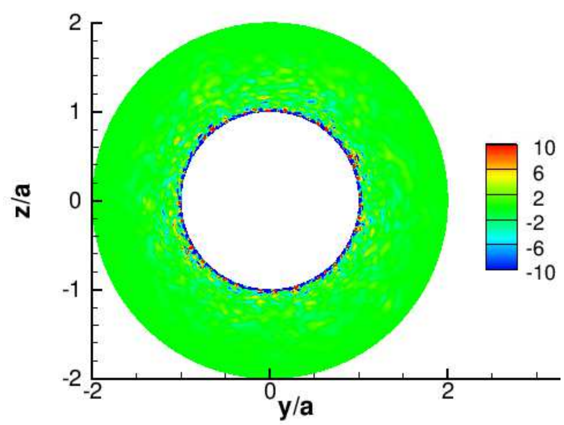


Figure 11: Case 2 ($Re_{\theta,in} = 1400$): Instantaneous axial velocity in xy plane.



(a)



(b)

Figure 12: Case 2 ($Re_{\theta,in} = 1400$): (a) Instantaneous axial velocity and (b) azimuthal vorticity in yz plane at $x = 8a$.

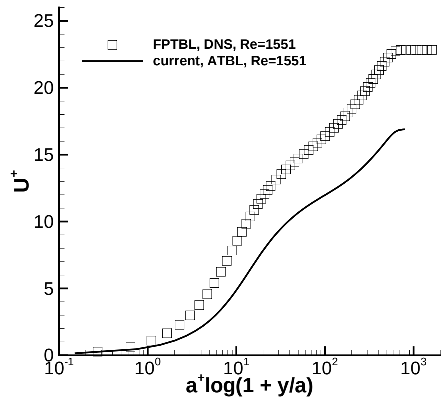


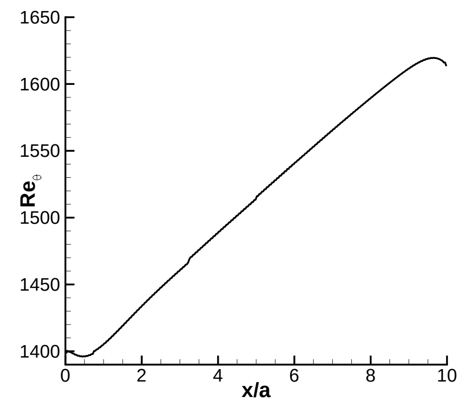
Figure 13: Case 2 ($Re_{\theta,in} = 1400$): Mean axial velocity at $Re_\theta = 1551$. Symbols are the DNS data of Jimenez et al. (2010a).

without having to simulate the entire upstream portion of the hull. However, it can not be used for flow over axisymmetric hulls at non-zero angle of attack or for hulls with sail near the bow region, due to loss of axisymmetry of the flow field. Note that further reduction in computational cost can be achieved by simulating only a sector of ATBL because the azimuthal correlations are known to be only a fraction of the entire circumference. However, we simulate the entire circumference to avoid the dependence of the computational domain size (sector angle) on the parameters of ATBL.

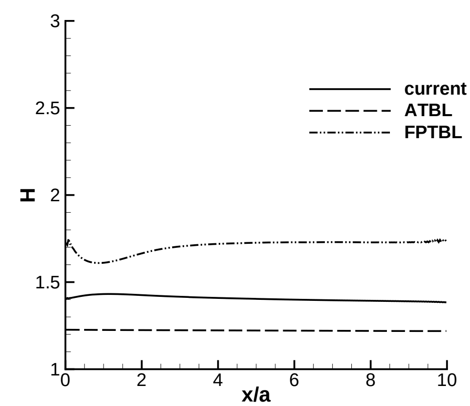
Simulations will be performed in future to generate inflow with the parameters matched exactly with the hull boundary layer using recycle-rescale method and this will be to simulate hull's wake for a range of Re in order to understand the effect of hull boundary layer on flow over stern and the wake characteristics.

ACKNOWLEDGEMENT

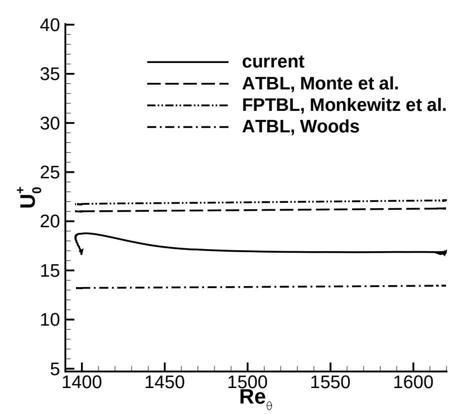
This work is supported by the United States Office of Naval Research (ONR) under ONR Grant N00014-14-1-0289 with Dr. Ki-Han Kim as technical monitor. The computations were made possible through the computing resources provided by the U.S. Army Engineer Research and Development Center (ERDC) in Vicksburg, Mississippi on the Cray XE6, Copper and Garnet of High Performance Computing Modernization program (HPCMP). This research partly used the computer time provided by the Innovative and Novel Computational Impact



(a)



(b)



(c)

Figure 14: Case 2 ($Re_{\theta,in} = 1400$): Spatial evolution of ATBL: (a) Re_θ, (b) H and (c) U_0^+. Results are compared with correlations available in literature for FPTBL (Monkewitz et al., 2008) and ATBL (Woods, 2006; Monte et al., 2011)
.

on Theory and Experiment (INCITE) program on the resources of the Argonne Leadership Computing Facility (ALCF), which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357. The authors also acknowledge the Minnesota Supercomputing Institute (MSI) at the University of Minnesota for providing resources that contributed to the research reported in this paper.

References

Afzal, N. and Narasimha, R. "Axisymmetric turbulent boundary layer along a circular cylinder at constant pressure". Journal of Fluid Mechanics, 74(01):113–128, 1976.

Atsavapranee, P., Forlini, T., Furey, D., Hamilton, J., Percival, S., and Sung, C.H. "Experimental measurements for CFD validation of the flow about a submarine model (ONR Body-1)". In Proceedings of the 25th Symposium on Naval Hydrodynamics, St. John's, Canada, pages 8–13, 2004.

Bridges, D.H. "A detailed study of the flowfield of a submarine propeller during a crashback maneuver". Technical Report MSSU-ASE-04-1, Department of Aerospace Engineering, Mississippi State University, 2004.

Chang, P.A., Vargas, A., Lummer, D., Jiang, M., and Mahesh, K. "Fully-resolved les of weakly separated flows". In 20th AIAA Computational Fluid Dynamics Conference. AIAA, 2011.

Chen, B. and Stern, F. "Computational fluid dynamics of four quadrant marine propeller flow". Journal of Ship Research, 43:4:218, 1999.

Cook, B.L. "Resistance for DARPA SUBOFF as represented by model 5470". David Taylor Research Center, Bethesda, Md, 1990.

Davoudzadeh, F., Taylor, L. K., Zierke, W. C., Dreyer, J. J., McDonald, H., and Whitfield, D. L. "Coupled navier–stokes and equations of motion simulation of submarine maneuvers, including crashback". In Proceedings of the 1997 ASME Fluids Engineering Division Summer Meeting, New York, 1997.

Germano, M., Piomelli, U., Moin, P., and Cabot, W. H. "A dynamic subgrid–scale eddy viscosity model". Physics of Fluids A, 3:7:1760, 1991.

Groves, N.C., Huang, T.T., and Chang, M.S. Geometric characteristics of DARPA Suboff Models:(DTRC Model Nos. 5470 and 5471). David Taylor Research Center, 1989.

Huang, T., Liu, H. L., Grooves, N., Forlini, T., Blanton, J., and Gowing, S. "Measurements of flows over an axisymmetric body with various appendages in a wind tunnel:the DARPA Suboff experimental program". In Proceedings of the 19th Symposium on Naval Hydrodynamics, 1992.

Jang, H. and Mahesh, K. "Large eddy simulation of crashback in ducted propulsors with stator blades". In Proceedings of the 29th Symposium on Naval Hydrodynamics, 2012.

Jang, H. and Mahesh, K. "Large eddy simulation of flow around a reverse rotating propeller". Journal of Fluid Mechanics, 729:151–179, 2013.

Jimenez, J., Hoyas, S., Simens, M.P., and Mizuno, Y. "Turbulent boundary layers and channels at moderate reynolds numbers". Journal of Fluid Mechanics, 657:335–360, 2010a.

Jimenez, J.M., Hultmark, M., and Smits, A.J. "The intermediate wake of a body of revolution at high Reynolds numbers". Journal of Fluid Mechanics, 659:516–539, 2010b.

Jordan, Stephen A. "On the axisymmetric turbulent boundary layer growth along long thin circular cylinders". Journal of Fluids Engineering, 136 (5):051202, 2014.

Kumar, P. and Mahesh, K. "Analysis of marine propulsor in crashback using large eddy simulation". In Fourth International Symposium on Marine Propulsors, Texas, USA, 2015.

Lilly, D. K. "A proposed modification of the Germano subgrid–scale closure model". Physics of Fluids A, 4:3:633, 1992.

Lueptow, R.M. "Turbulent boundary layer on a cylinder in axial flow". AIAA journal, 28(10): 1705–1706, 1990.

Lund, T.S., Wu, X., and Squires, K.D. "Generation of turbulent inflow data for spatially-developing boundary layer simulations". Journal of Computational Physics, 140(2):233–258, 1998.

Mahesh, K., Constantinescu, G., and Moin, P. "A numerical method for large–eddy simulation in complex geometries". Journal of Computational Physics, 197:1:215, 2004.

Mahesh, K., Kumar, P., Gnanaskandan, A., and Nitzkorski, Z. "LES applied to ship research". Journal of Ship Research, 59(4):238–245, 2015.

Moin, P. and Mahesh, K. "Direct numerical simulation: a tool in turbulence research". Annual Review of Fluid Mechanics, 30(1):539–578, 1998.

Monkewitz, P.A., Chauhan, K.A., and Nagib, H.M. "Comparison of mean flow similarity laws in zero pressure gradient turbulent boundary layers". Physics of Fluids (1994-present), 20(10):105102, 2008.

Monte, S., Sagaut, P., and Gomez, T. "Analysis of turbulent skin friction generated in flow along a cylinder". Physics of Fluids (1994-present), 23(6): 065106, 2011.

Piquet, J. and Patel, V.C. "Transverse curvature effects in turbulent boundary layer". Progress in aerospace sciences, 35(7):661–672, 1999.

Simens, M.P., Jiménez, J., Hoyas, S., and Mizuno, Y. "A high-resolution code for turbulent boundary layers". Journal of Computational Physics, 228 (11):4218–4231, 2009.

Spalart, P.R. "Direct simulation of a turbulent boundary layer up to $R_\theta = 1410$". Journal of Fluid Mechanics, 187:61–98, 1988.

Verma, A. and Mahesh, K. "A lagrangian subgrid-scale model with dynamic estimation of lagrangian time scale for large eddy simulation of complex flows". Physics of Fluids (1994-present), 24(8):085101, 2012.

Verma, A., Jang, H., and Mahesh, K. "The effect of an upstream hull on a propeller in reverse rotation". Journal of Fluid Mechanics, 704:61–88, 2012.

Woods, M. J. Computation of axial and near-axial flow over a long circular cylinder. University of Adelaide, Australia, 2006.

DISCUSSION

Peter Chang,
Naval Surface Warfare Center, Carderock.

1. Congratulations on a nice written paper which represents an advance in the simulation of submarine crashback.

2. You offer two methods for generating the flows in the stern: (1) directly using wall-resolved LES of the complete hull and (2) using recycle-rescale (RR). Please comment on the advantages of one over the other in terms of computational resources and turbulence resolution. Given that the spanwise correlations are only a fraction of the circumference would you consider performing RR on just a sector of the hull?

3. Where on the Suboff hull are the ATBL (recycle-rescale) results comparable? It looks like your Case 2 might be comparable? How do the results compare?

AUTHOR'S REPLY

Thank you for your questions which are addressed below.

1. Thank you for your valuable comments.

2. The RR method is computationally cheaper than wall-resolved LES of entire hull and it can be used as an alternative if only flow in the stern and wake region are of interest. In principle only a sector can be used as the spanwise correlations are only a fraction of the circumference, but we chose to have full circumference in order to avoid the dependencies of flow domain with the inflow parameters.

3. We chose the parameters of Case 2 to be representative of a hull boundary layer but the parameters are not matched with that of wall-resolved LES simulations. Hence, at this point we can not expect them to match. However, the simulation of flow over stern with inflow parameters matched to that of full hull simulations will be performed and compared as part of the future work.

DISCUSSION

Michael Mattson,
CSRA Inc.

Thank you for writing an excellent paper on such a challenging and important topic. I am pleased with the progress the authors have made and am eager to see continued advancement. Even for problems at model scale Reynolds numbers, wall-resolved Large Eddy Simulation (LES) of attached flow over a body-of-revolution hull is prohibitively expensive due to mesh resolution requirements.

1. For hulls at zero angle-of-attack, the recycle-rescale approach seems to be an appropriate simplification in order to reduce the size of the grid. Can the authors comment on the possibility and practicality of simulating flows using the recycle-rescale approach with a non-zero angle-of-attack?

2. The low-Reynolds number case for the recycle-rescale approach shows a large stream-wise transient in shape factor section near the inflow (figure 10b). Both cases show a short transient region for momentum thickness (figures 10a and 14a), where a constant slope is expected. Does the skin friction coefficient also show such features?

3. The results from the entire hull (figure 5b) show a "dip" in the skin-friction at a stream-wise distance of approximately 0.3 hull lengths, which is not expected based on the experimental results. Please comment.

AUTHOR'S REPLY

Thank you for your comments and questions which are addressed below.

1. This recycle-rescale method is applicable only to thin axisymmetric turbulent boundary layers in present form. The flow over hull with a non-zero angle-of-attack is not axisymmetric, hence this method can not be used.

2. There is indeed a small transient region near inflow for both cases with low-Reynolds number case (Case 1) showing a larger stream-wise transient in shape factor (figure 10b). The skin friction coefficient (C_f) is related to U_0^+ as $U_0^+ = \sqrt{2/C_f}$, which is shown in figures 10c

and 14c for the two cases respectively showing similar features.

3. The skin-friction results of Huang et al. (1992) is rescaled to $Re = 1.1$ *million*, which is shown along with our LES results (figure 5b). This scaling is based on the assumption that the boundary layer is attached and evolving under zero pressure gradient. This small "dip" is present near bow region ($x/L = 0.3$) which may have small favourable pressure gradient. Nevertheless, the skin-friction values are in reasonable agreement with the scaled experimental results, which is reflected in good agreement in overall drag as well.