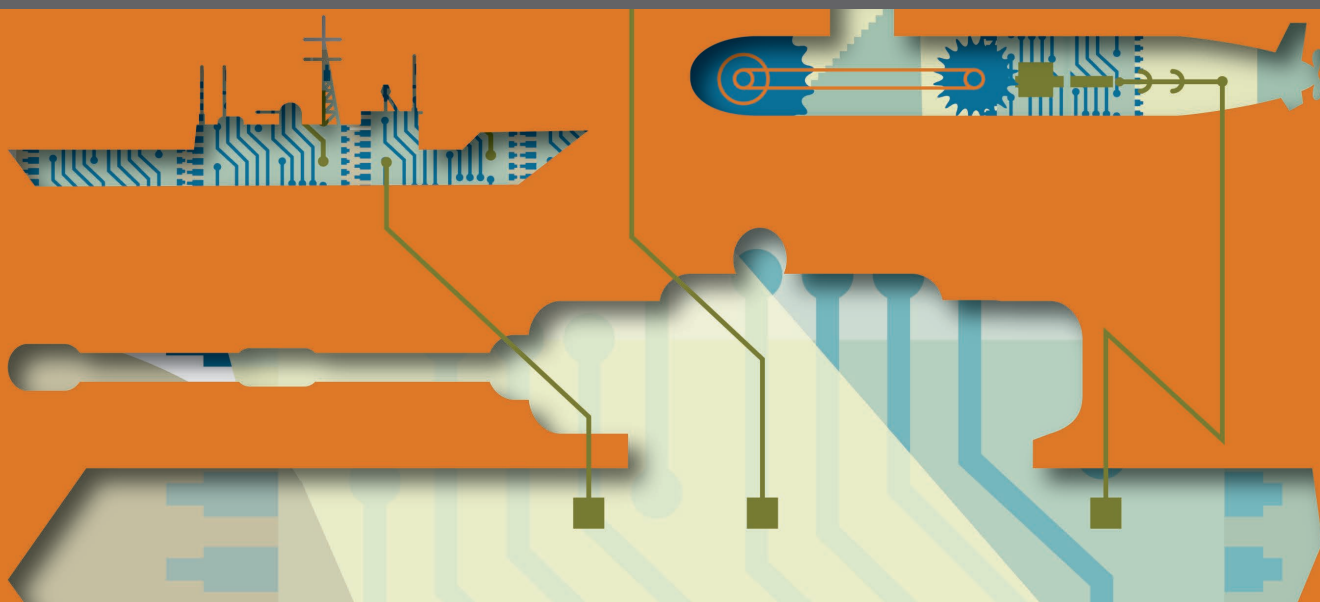




A Scalable and Extensible Computational Fluid Dynamics Software Framework for Ship Hydrodynamics Applications: NavyFOAM

Sung-Eun Kim, Hua Shan, Ronald Miller, Bong Rhee, Abel Vargas, Shawn Aram, and Joseph Gorski | Naval Surface Warfare Center Carderock Division

The main challenge facing simulation-based hydrodynamic design of naval ships comes from the complexity of the salient physics involved around ships, which is further compounded by the multidisciplinary nature of ship applications. Simulation of the flow physics using “first principles” is computationally very expensive and time-consuming. NavyFOAM offers a solution.

The ultimate interest of the US Navy in ship hydrodynamics lies in making the best use of the associated technology of today and the future to build and maintain the fleet. To that end, the hydrodynamic performance of the fleet, including both surface ships and undersea vehicles, should satisfy a wide range of strategic, economic, and tactical requirements. The CREATE-Ships Project sees computational fluid dynamics (CFD) as an enabler that will help realize the lofty goal of simulation-based design of ships. The role of NavyFOAM as a product in the CREATE-Ships Project is to provide ship design engineers with a high-fidelity computational capability that allows them to assess alternative hull form designs with the same level of confidence they would have utilizing model tests instead. NavyFOAM will eventually replace empirical methods that often have proven to be inadequate for novel designs that depart from the Navy’s historical experience. Instead, it will provide a high-fidelity, physics-based, computation-intensive hydrodynamics tool geared toward the next generation of massively parallel computers. NavyFOAM will be capable of modeling the detailed physics necessary for accurately predicting the hydrodynamics performance of naval vessels operating in extreme environmental

conditions, including resistance, propulsion, maneuvering and control, six degrees of freedom (6-DOF) of ship motions, fluid-structure interactions, and seaworthiness.

Challenges and Needs

CFD today is increasingly called upon to tackle ship applications involving complex physics and complex geometry. To effectively impact ship design, CFD must provide reliable predictions of the relevant engineering quantities of interest (QOIs) in the areas of resistance, powering, maneuvering, and seakeeping. The foremost challenge comes from the complexity of the salient physics involved. Turbulence and two-phase flows are among the two most prominent yet challenging features common to all ship hydrodynamics applications. In addition, CFD often has to be integrated with other domain codes to address multidisciplinary applications such as fluid-structure interaction and hydroacoustics. Simulation of these complex flow physics using “first principles” is computationally very expensive and prohibitively time-consuming. At the moment, “time to solution” is a major barrier that impedes wider adoption of physics-based simulations in ship design. Despite the formidable computing power and speed, there’s still a wide gap between the requirement and the reality in terms of solution turnaround time. Another difficulty impacting time to solution comes from the complex geometry and topology of ship hull and appendages, including bulbous bow, sonar dome, stern-end bulb, transom, shafts, struts, bilge keels, propeller, rudder assembly, sails, and stern fins. Complex geometry translates into difficulty making computational grids that properly resolve regions of interest such as the air-water interface, turbulent boundary layers on ship hull and appendages, and the stern and near-wake regions.

In developing CFD software, there are also challenges associated with software engineering. First, writing a multiphysics CFD software from the ground up is a fairly expensive proposition and can take multiple years of team effort executed by experts in subdisciplines of CFD and computer science. In addition, the multiphysics, multidisciplinary nature of ship hydrodynamics applications such as fluid-structure interaction and hydroacoustics poses additional challenges to software development and maintenance that arise from the need to glue together multiple codes and transfer data between them. The complexity of the

physics tends to be directly carried over to CFD software. Software complexity—aptly conveyed by the phrase “spaghetti codes”—is a serious issue that many legacy CFD codes face today, negatively impacting their overall efficacy in terms of software development, deployment, maintenance, and extensions. Thus, software “extensibility” is an important attribute that modern CFD software has to possess because ship hydrodynamics has to deal with diverse issues to meet both the current and future needs of the fleet. All of this makes software design and architecture critically important issues in developing modern CFD software. These issues and requirements were considered in determining the software framework and architecture design when the CREATE-Ships Project embarked on the development of NavyFOAM.

NavyFOAM Code Development

To address the issues and overcome the challenges alluded to earlier, the CREATE-Ships Project decided to build NavyFOAM using OpenFOAM.¹ OpenFOAM is a freely available open source CFD software toolkit written in C++ that draws heavily on object-oriented programming (OOP). OOP significantly expedites development and maintenance of complex computational mechanics software for multiphysics, multidisciplinary applications such as ship hydrodynamics. OpenFOAM also makes heavy use of programming techniques offered by the C++ language such as function and operator overloading, inheritance, generic programming using templates, and dynamic binding using virtual base classes, all of which allow designers to easily assemble top-level applications using lower-level objects. Thus, the suite of numerical methods libraries, physical model libraries, top-level applications solvers, and utility (pre- and postprocessing) programs are tightly integrated in the framework in a modular manner.

NavyFOAM Features and Capabilities

In NavyFOAM, a large number of new features and capabilities have been added on top of the native OpenFOAM, including a large number of new top-level solvers for single- and multiphase flows, libraries of new numerical schemes and algorithms, advanced physical models,^{2,3} and custom postprocessing utilities. We refer to this framework as NavyFOAM (HPCMP CREATE-SH) to distinguish it from the native OpenFOAM.

The Navier-Stokes solvers in OpenFOAM (and therefore NavyFOAM) employ a cell-centered

finite-volume method based on a multidimensional linear reconstruction scheme that permits use of arbitrary polyhedrals. The ability to use unstructured grids is critical to handling complex geometry. The spatial discretization schemes in NavyFOAM offer second-order accuracy for arbitrary polyhedral grids. Several new discretization schemes, including interpolations schemes, flux reconstruction schemes, and advection discretization schemes, have been implemented in NavyFOAM to enhance spatial accuracy and robustness for unstructured grids.

To advance solutions in time, OpenFOAM employs semi-implicit projection algorithms. NavyFOAM additionally offers variants of those algorithms aimed at faster turnarounds of transient solutions. The solvers are fully parallelized using domain decomposition and the message-passing interface (MPI), and are scalable on manycore modern computers such as Linux clusters.

NavyFOAM offers a suite of RANS turbulence models mainly comprising one- and two-equation eddy-viscosity models and algebraic and differential Reynolds stress models. In addition, NavyFOAM offers a number of eddy-resolving turbulence models for large eddy simulation (LES) and hybrid RANS/LES simulations. All the turbulence models in NavyFOAM are coded up using a virtual base class and actual implementations. Thanks to the virtual base class defining the interface, the top-level RANS solvers can be assembled independently without knowing the details of turbulence models.

V&V and Ship Applications

NavyFOAM has been unit-tested for all the newly added codes (classes). Specifically, the new features have been tested using an integration-test suite along with a regression test suite. NavyFOAM has also been validated for a number of Navy surface ship and underwater vehicle models. Sensitivity of numerical solutions to grids, discretization schemes, time-step sizes, turbulence models, and iterative algorithms have been studied for all validation cases, allowing us to quantify the level of uncertainty for each test case and to develop best practices for the corresponding applications. The best practices established in this manner have been captured and documented in tutorials available to users. In the past few years, the NavyFOAM development team has participated in international workshops on CFD in ship hydrodynamics.⁴

Navy ships such as frigates, destroyers, the Littoral Combat Ship (LCS), and amphibious combat

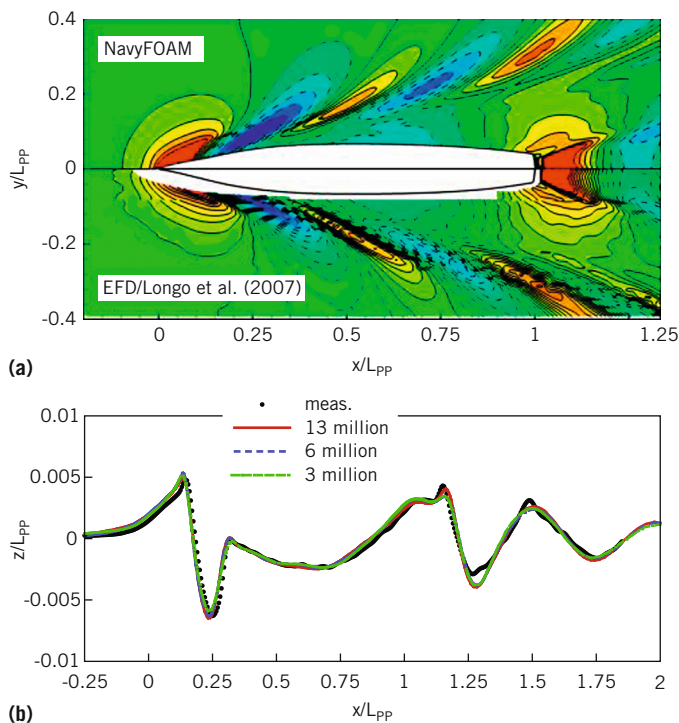


Figure 1. Validation for NavyFOAM's capability to predict resistance and wave pattern of a surface ship model in calm water using RANS computations of waves for the DTMB-5415 model: (a) contour of wave elevation, and (b) wave elevation along a longitudinal cut at 0.082 ship length from the ship center plane.

vehicles (ACV) all operate at the interface of water and air in the presence of ambient sea waves and waves generated by the ships themselves. Calm water resistance is the most basic quantity of interest that helps ship designers determine the powering requirements of surface ships, constituting an important "use case" for NavyFOAM, which has been validated with several calm-water resistance datasets obtained from model tests in towing tanks. One dataset was from the DTMB-5415 model, a destroyer hull form that has been widely studied both experimentally and numerically. It was selected as a test case at the 2010 Gothenburg CFD Workshop for Ship Hydrodynamics.⁴

NavyFOAM computations were made for a 3.048 m-long bare hull model with no appendages (bilge keels), which was tested at the 10-foot wide University of Iowa towing tank. The Reynolds and the Froude numbers are $Re_L = 5 \times 10^6$ and $Fr = 0.28$, respectively. The RANS computations were made using three systematically refined meshes: 3 million, 6 million, and 13 million elements, all of which resolve the near-wall region down to the hull surface. Figure 1a depicts the predicted wave

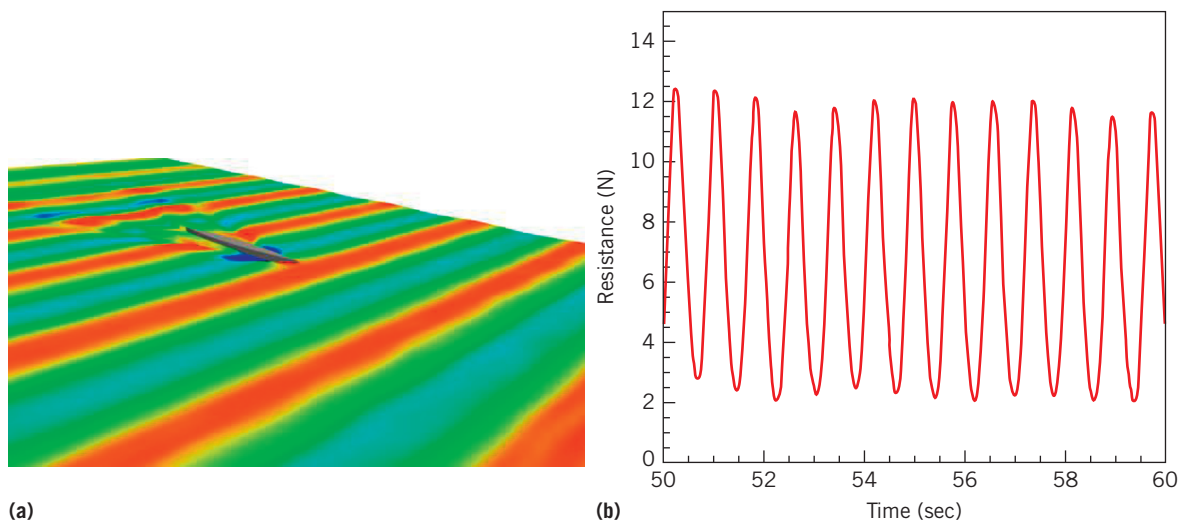


Figure 2. Wigley III ship model cruising in head sea simulated using RANS computation including the effects of ship motion in response to waves: (a) wave pattern illustrated with contour of wave elevation, and (b) time history of ship resistance (drag) oscillating due to the waves and ship motion.

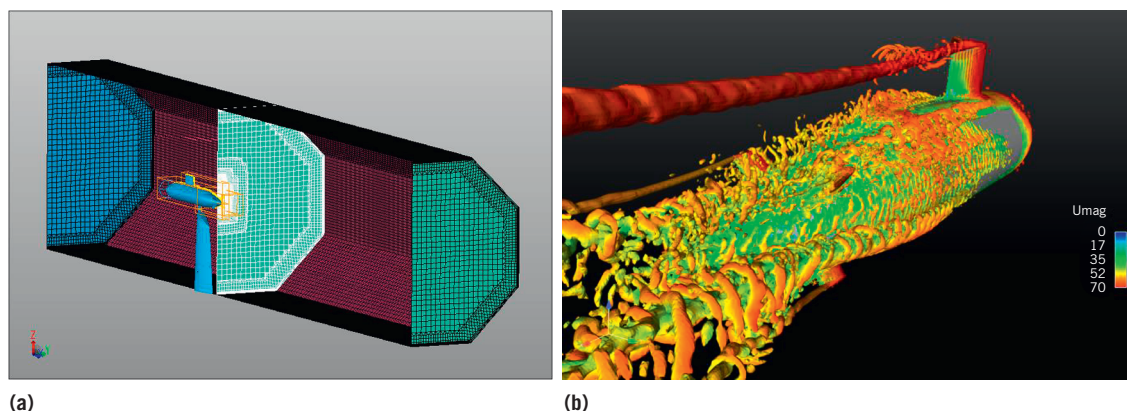


Figure 3. Joubert submarine model: (a) computational grid and (b) hybrid RANS/LES simulation of the flow around the model.

pattern along with the measured pattern. Figure 1b shows the wave elevations along a longitudinal cut at 0.082 ship-length predicted with the three different grids. The results clearly show grid-convergence of the predictions, all of which are in good agreement with the data. The percentage errors relative to the measured resistance are 5.4 percent, 4.6 percent, and 4.1 percent for the coarse, medium, and fine grid, demonstrating a monotonic convergence and reasonable accuracy of the predictions.

A ship's resistance and engine power to sustain a ship's speed in waves is augmented mainly due to complex nonlinear interaction between the ship hull and waves. Propulsive efficiency of a ship in waves is also affected by the time-varying relative motion between its propulsor and the

ambient flow. Numerical methods based on linear potential theory have been widely used to calculate added resistance. A more physics-based avenue to predicting added resistance is now offered by coupled solutions of two-phase unsteady Reynolds-averaged Navier-Stokes (URANS) equations and the 6-DOF rigid-body motion (RBM) equation.⁵ Ship speed loss for added power due to waves can be predicted by including propulsors in the computations. With enough details of the flow-fields and ship motion available, coupled URANS-RBM computations can not only predict added resistance and speed loss but can also provide valuable insights into the underlying physical mechanisms. Figure 2 illustrates a surface ship model called "Wigley Hull III" traveling straight at a constant

speed of 1.627 m/s ($Fr = 0.3$) in incoming head waves of a 3-m wave length (equal to the ship length, $\lambda/L = 1.0$) and a wave height of 0.04 m. Figure 2 shows the wave pattern around the ship and the time history of the ship's resistance.

The flows around underwater vehicles are featured by 3D turbulent shear flows such as the hull boundary layer with crossflow, open separations, and ensuing stream-wise vortices. NavyFOAM has been demonstrated to be capable of predicting the complex flows around underwater vehicles and the resulting forces and moments accurately and efficiently.^{3,5} Figure 3 depicts a research submarine model (Joubert) and the computational grid set up for NavyFOAM computations. As shown, the tunnel wall and the mounting support are included in the computations, which allows experimentalists to assess the effects of the tunnel blockage and mounting supports. Turning of a body further compounds the flow. Turning rate and drift angle generate prominent crossflow and stream-wise vortices. These vortices have an important effect on the forces and moments on the body. Figure 3b illustrates the vortical flow around the Joubert model at the drift angle of 10° . The simulation was made using a hybrid RANS/LES (HRLES) turbulence model designed to resolve large-scale coherent turbulent eddies.⁶ The figure portrays the formation and evolution of multiple vortices and large-scale turbulent structures originating from different parts of the body such as the main hull, the sail, and the four stern appendages.

The safe operational limits of naval vessels are much broader than the normal operational design conditions. Consequently, ship designers need a computational capability that can accurately predict the performance of propulsors not only near normal design conditions but also at off-design conditions. One such situation happens when an underwater vehicle executes an emergency stopping maneuver by reversing the direction of the propeller rotation. This maneuver is referred to as crashback. During a crashback maneuver, propeller blades are subjected to severe hydrodynamic loading, which could lead to structural failure of the blades. Figure 4 illustrates the flow around a 4.36 m-long submarine model that undergoes a crashback maneuver that was initiated from a straight-ahead, self-propulsion condition ($U_s = 2.5$ m/s) at a given propeller rpm ($n = 560$ rpm) that was obtained from a 1-DOF precursor simulation of the straight-ahead cruise. The direction (sense) of the propeller rotation is reversed from

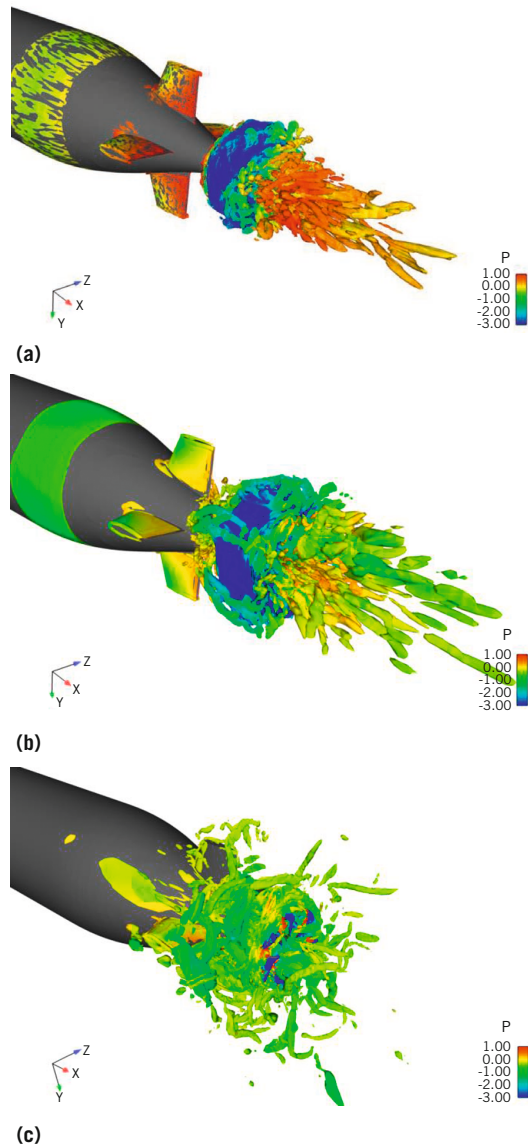


Figure 4. Turbulent vortices around the propeller on the fully appended SUBOFF body in a crashback maneuver visualized using isosurface of vortex strength colored by pressure: (a) $t = 1.0$ sec ($U = 2.2$ m/s), (b) $t = 2.5$ sec ($U = 2.0$ m/s), and (c) $t = 13.0$ sec ($U = 1.6$ m/s).

(+) 560 rpm to (−) 560 rpm. Figure 4 illustrates the flow structures around the propeller at three time instants ($t = 1.0, 2.5$, and 13.0 seconds) after the crashback is initiated. One feature that's clearly noticeable is a ring-shaped, vortical structure (low-pressure region shown in blue in the figures) around the propeller blades.

After reaching the maximum strength in the early phase of the crashback, the vortex becomes gradually weaker as the vehicle slows down. The

coupled HRLES and 3-DOF RBM solution captures the time evolutions of the forces and moments on the hull and the propeller, and vehicle velocities (surge, sway, and yaw), positions, and heading angles throughout the crashback maneuver. Although not shown here, the simulation revealed that the extrema of the axial and transverse force components on the propeller blades mostly occur in the initial stage of the crashback maneuver, and that the vehicle ultimately veers off from a straight course due to the side force and yaw moment from the propeller's blades.

Developed using modern software development processes and OOP practices, NavyFOAM today offers a mature CFD capability tailored to naval hydrodynamics applications. The main strength of NavyFOAM is modularity and flexibility that facilitates collaboration among developers and customization and extension of the software. NavyFOAM has been extensively validated for a wide range of ship hydrodynamic applications. Over the past few years, NavyFOAM has been used for acquisition and design programs including the DDG-1000 program and the Ohio Replacement program for the US Navy, and the Amphibious Combat Vehicle program for the US Marine Corp. NavyFOAM has also been adopted for a number of R&D programs with demonstrated robustness and accuracy. The ingredients of NavyFOAM such as numerical methods and physical models will continuously be upgraded to keep up with the state of the art in the respective areas. Future efforts will focus on the preparation for the next generation of high-performance computers. ■

Acknowledgments

The development of NavyFOAM has been mainly funded by the Department of Defense High Performance Computing Modernization Program (HPCMP) under the CREATE Ship's Hydrodynamics Program. We also acknowledge the financial supports from the Naval Surface Warfare Center Carderock Division (NSWCCD) and the Office of Naval Research.

References

1. H.G. Weller et al., "A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques," *Computers in Physics*, vol. 12, no. 6, 1998, pp. 620–631.
2. S.-E. Kim, S. Schroeder, and H. Jasak, "A Multi-phase CFD Framework for Predicting Performance of Marine Propulsors," *Proc. 13th Int'l Symp. Transport Phenomena and Dynamics of Rotating Machinery*, 2010.
3. S.E. Kim, B.J. Rhee, and R.W. Miller, "Anatomy of Turbulent Flow around DARPA SUBOFF Body in a Turning Maneuver Using High-Fidelity RANS Computations," *Int'l Shipbuilding Progress*, vol. 60, no. 1, 2014, pp. 207–231.
4. L. Larsson, F. Stern, and M. Visonneau, *Numerical Ship Hydrodynamics: An Assessment of the Gothenburg 2010 Workshop*, Springer, 2014.
5. S.E. Kim, B.J. Rhee, and H. Shan, "URANS Simulation of Propeller-Hull Interactions on an Underwater Body in Maneuver," *Proc. 30th Symp. Naval Hydrodynamics*, 2014.
6. S.-E. Kim and B. Rhee, "Eddy-Resolving Simulation of Turbulent Flows around Undersea Vehicles—A Quest for a Practical Approach," presented at the 31st Symp. Naval Hydrodynamics, 2016.

Sung-Eun Kim, the corresponding author, is a technical lead of the CREATE SHIPS program and head of the Computational Fluid Dynamics R&D Branch at the Naval Surface Warfare Center Carderock Division. His research interests include ship hydrodynamics, turbulence modeling and simulation, and multidisciplinary computational mechanics. Kim received a PhD in mechanical engineering from the University of Iowa. Contact him at sungeun.kim@navy.mil.

Hua Shan is an engineer in the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include turbulence modeling and simulation, computational hydroacoustics, and software development. Shan received a PhD in fluid mechanics from Tsinghua University in China. Contact him at hua.shan@navy.mil.

Ronald Miller is an engineer in the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include ship hydrodynamics, software development, and fluid-structure interaction. Miller received an MS in ocean and marine engineering from the George Washington University. Contact him at ronald.w.miller@navy.mil.

Bong Rhee is an engineer in the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include submarine hydrodynamics, turbulence modeling, and simulation. Rhee received a PhD in mechanical engineering from the George Washington University. Contact him at bong.rhee@navy.mil.

Abel Vargas is an engineer in the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include ship hydrodynamics, application of CFD to surface roughness modeling, and ship design. Vargas received a PhD in mechanical and aerospace engineering from the George Washington University. Contact him at abel.vargas@navy.mil.

Shawn Aram is an engineer in the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include flow control, fluid-structure

interaction, and ship hydrodynamics. Aram received a PhD in mechanical engineering from the Johns Hopkins University. Contact him at shawn.aram@navy.mil.

Joseph Gorski is a director of science and technology for the Naval Architecture and Engineering Department at the Naval Surface Warfare Center Carderock Division. His research interests include computational fluid dynamics for ships and submarines. Gorski received a PhD in mechanical engineering from the University of Maryland. Contact him at joseph.j.gorski@navy.mil.

Take the CS Library wherever you go!



IEEE Computer Society magazines and Transactions are now available to subscribers in the portable ePub format.

Just download the articles from the IEEE Computer Society Digital Library, and you can read them on any device that supports ePub. For more information, including a list of compatible devices, visit

www.computer.org/epub



IEEE

IEEE  computer society