High-Fidelity Simulations of Bubble, Droplet, and Spray Formation in Breaking Waves

By Zhaoyuan Wang, Jianming Yang, Frederick Stern, IIHR-Hydroscience & Engineering, University of Iowa, Iowa City, Iowa

Background and Objective

Plunging wave breaking is of great importance to ship hydrodynamics, including strong turbulence with large amounts of air entrainment, bubbles, droplets, jets, and spray. Previous experimental fluid dynamics (EFD) and computational fluid dynamics (CFD) studies on plunging wave breaking are mainly focused on the global structures of the wave breaking, such as wave elevation, jet, air cavity, and wave breaking process. Moreover, due to the technical difficulties, the experimental measurements usually can only be done in the water region, and detailed description of the energetic wave breaking region is not available. With the development of the CFD technology, detailed studies of the two-phase region become possible.

The experimental study (Deane and Stokes, 2002) shows that two distinct flow conditions that drive bubble creation in breaking waves: jet plunging and collapse of the cavity. The bubble sizes are from 2.0 mm down to at least 0.1 mm for the jet plunging period, larger bubbles from 2.0 mm to ⩾ 10.0 mm are created due to the collapse of the air cavity. In the recent study by Koo et al. (2012), plunging wave breaking over a submerged bump was simulated using the exact experimental flow conditions provided in Kang et al. (2012). The simulations were carried out on a 2D domain and focused on the overall plunging wave breaking process. Flows around a wedge-shaped bow were simulated by Wang et al. (2010) to study the wave breaking mechanism and main structures of ship bow waves.

The objective of the present study is to investigate the small scale features of breaking waves such as the spray formation, air bubble and water droplet size and distribution via numerical simulations. In order to resolve the small structures of the wave breaking at the scale of several micrometers, large-scale parallel computing (several billion grid points) is needed. In the present study, wave breakings around a wedge-shaped bow and over a submerged bump are simulated. The grid sizes used in these two simulations are 1.0 billion (1536×768×848) and 2.2 billion (1920×896×1280), respectively.

Approach

The challenges and difficulties for simulations using huge grid sizes (billions of grid points) lie in the data pre/ post-processing, data storage and transfer and computational speed, etc. Grid generation and grid file reading is easy for Cartesian grid since only one-dimensional arrays are required for grid coordinates. It is very difficult to generate billion-point grids for complex geometries using curvilinear or unstructured grids with current mainstream grid generation software. It is too time-consuming to transfer even a single data file (size can be tens of GB for billion-point grids) between a local PC and HPC and it is impossible to store all the data files on a local PC, which is difficult and inconvenient for data visualization, analysis and animation. Moreover, large grid size requires more memory and processors and smaller grid spacing needs a smaller time step, which leads to even longer or unacceptable total computational time.

An orthogonal curvilinear grid solver, CFDShip-Iowa Version 6.2 (Wang et al., 2012a), is used for the computations. This solver was extended from the Cartesian grid solver (Version 6.1) (Yang and Stern, 2009) for two-phase incompressible viscous flows recently developed at the University of Iowa. In this solver, both gas and liquid phases are considered for the strong interactions between two phases, such as spray dispersion and bubble entrainment. The level-set based ghost fluid method is adopted for sharp interface treatment and the volume-of-fluid (VOF) method is used for the interface tracking

In order to speed up the computations and improve the accuracy and efficiency, some enhanced technologies have been implemented such as a new VOF method (Wang et al., 2012b), semi-Lagrangian advection schemes for both the Navier-Stokes and VOF equations (Wang et al., 2012c), parallel grid/solution files reading/writing using MPI2-I/O (Yang et al., 2008), optimized memory usage. Due to relatively simple geometries of the bump and wedge, 2D grids were generated first using Gridgen on a local desktop and translationally extruded to 3D grids on HPC. The water/air interface is extracted as PLY polygon file format for post-processing.

Simulation Conditions and HPC Setup

The wedge geometry is similar to the large wedge model used by Waniewski et al. (2002). The side length of the wedge is L = 0.75 m, and the height of the wedge is H = 1 m. The half wedge angle is θ = 26º and the flare angle φ = 0º. For the case considered here, the water depth is d = 0.0745 m and the upstream velocity is U = 2.5 m/s, the corresponding Reynolds number, $\mathrm{Re} = \rho U d / \mu = 1.64\times10^5$, and the Froude number, $Fr = U / \sqrt{gd} = 2.93$. The geometry of the bump is the same as that in the EFD and CFD studies (Kang et al., 2012; Koo et al., 2012). The constant velocity imposed at the inlet boundary is u = 0.87 m/s for water and zero for air. The initial interface elevation is 0.2286 m and a uniform velocity field is prescribed in the water domain with the air phase at rest.

The wedge flow case with 1.0 billion grid points was performed first on Navy DSRC Einstein (Cray XT5) from 18 September 2010 through 19 October 2010, and then moved to ERDC Diamond (SGI Altix Ice). The simulation ran smoothly on both platforms using 1024 cores, and was also able to run on 512 cores (64 nodes) on Einstein. This was the first largest case that had ever been conducted in the Ship Hydrodynamics group at the University of Iowa. In March 2012, the bump flow case with 2.2 billion grid points was performed on Einstein first using 1024 cores (128 nodes) but it ended up with “out of memory” (OOM) error messages. So the number of the cores was increased to 2048 (256 nodes), which was the maximum number of cores that could be used for the standard queue on Einstein. The code was terminated again with the same OOM error messages reported. This is because the amount of memory required by the multigrid Poisson solver has some dependence on the global grid size.

The total processes were then reduced to1024 on 256 nodes (2048 cores), and the simulation was running without OOM errors any more. However, this would waste much CPU time since only half of the cores on each node were used and the CPU-hours would be charged for the total cores requested. The code might be able to run 4096 processes on 512 nodes (4096 cores) on Einstein, and then the CPU time would not be wasted. A request was sent to the Consolidated Customer Assistance Center (CCAC) for 4096 cores on Einstein on 12 April 2012. The request was approved with a “special queue” assigned. The code ran smoothly without any errors with 4096 processes on 512 nodes (4096 cores). The simulation was started on 18 April 2012 and terminated on 27 April 2012. The total size of all the data files generated in this simulation was about 51 TB, which was approximately 10% of the entire scratch file system disk space on Einstein.

Figure 1. Wave breaking around a wedge-shaped bow.

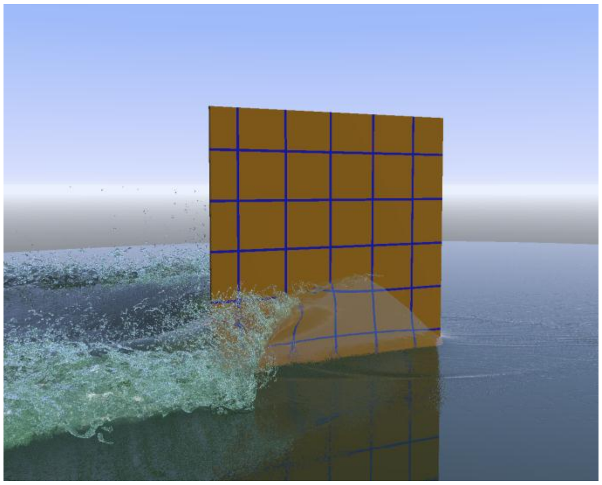

Figure 2. Wave profile of the wedge flow.

Simulation Results

The wedge geometry and the computed wave profile are shown in Fig. 1. Figure 2 shows the close-up view of the computed wave profile, which is very similar to the experimental observation (Waniewski et al., 2002), such as the thin liquid sheet at the leading edge of the bow, overturning jet, jet plunging onto the free surface, and splashes at the wake. As the liquid sheet overturns, the sheet is stretched and fingered up, and some ‘‘cylindrical drops’’ then pinch off from the liquid sheet, when the detached drops impact onto the water surface, a spray region is created. In the experimental study conducted by Deane and Stokes (2002), the diameters of most observed bubbles due to the fragmentation process are greater than 2.0 mm. Mean drop size observed in experiments by Karion et al. (2004) is 2.3 mm. For small size bubbles/droplets, surface tension force is dominant and further fragmentation is difficult. In the present study, the grid spacing is 0.125 mm near the wedge and 1.0 mm in the plunging region. With the current grid, the minimum drop size is 0.8 mm near the wedge. The droplets and bubbles near the wedge can be effectively captured. Further grid refinement (3 to 4 billion grid points) is needed to increase the resolution in the wake region.

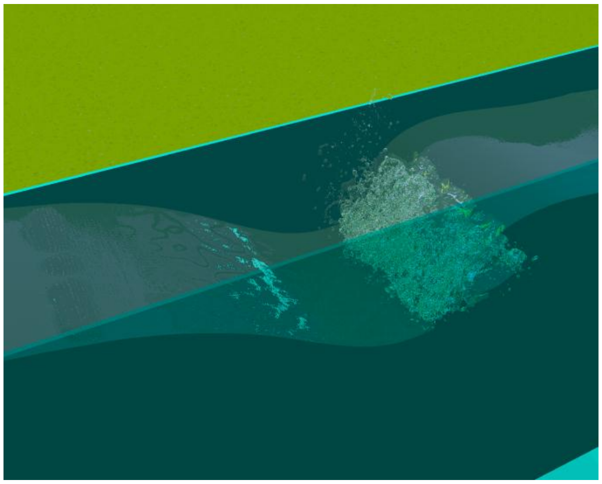

Figure 3. Overview of the plunging wave breaking over a submerged bump.

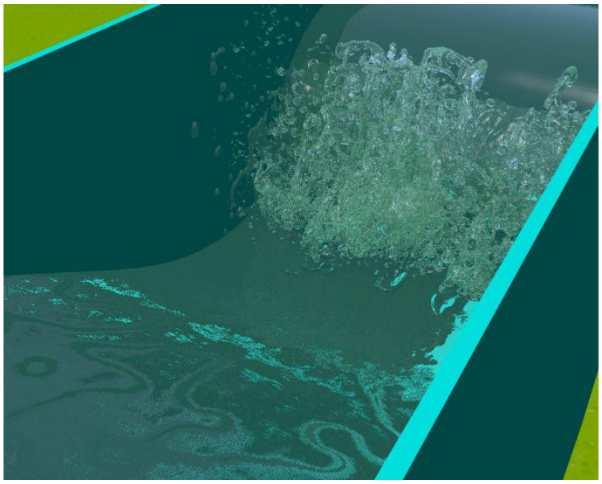

Figure 4. Close-up view of the wave breaking over a submerge bump.

Figure 3 shows the overall plunging wave breaking downstream of the bump. The major events of the plunge wave breaking are demonstrated, *i.e.*, maximum height, first plunge, oblique splash, and vertical jet. The computational results match the experiments very well. Figure 4 shows the close-up of the bubbles and droplets in the breaking waves downstream of the bump, where detailed small interface structures are well demonstrated. With current grid resolution, the experimental observed mean droplet/bubble size can be effectively captured. Further grid refinement (up to 16 billion points) is needed in order to capture the minimum drop/bubble size observed in the experiments.

Conclusions and Future Work

Breaking waves around a wedge-shaped bow and over a submerged bump were numerically simulated using billions of grid points. An orthogonal curvilinear grid solver, CFDShip-Iowa Version 6.2, is used for the computations with enhanced technologies such as a new VOF method, semi-Lagrangian advection schemes for both the Navier-Stokes and VOF equations to improve the accuracy and efficiency. The computational results match the experimental observations very well.

In the future work, the grid resolution for the wedge flow will be increased (3 to 4 billion grid points) in order to effectively capture more droplets and bubbles away from the wedge. Further grid refinement (up to 16 billion points) is also needed for the bump flow case in order to capture the minimum drop/bubble size observed in the experiments. To accomplish these goals, much more HPC resources are needed, e.g, if 3 billion grid points are used for the wedge flow case more than 2.8 million CPU-hours are required, which accounts for about 20% of our current total annual project allocation. Therefore, procurement of additional CPU-hours is required to carry out these large grid simulations.

Acknowledgments

This research was sponsored by the Office of Naval Research under Grant N000141-01-00-1-7, under the administration of Dr. Patrick Purtell. The simulations were performed at the Department of Defense (DoD) Supercomputing Resource Centers (DSRCs) through the High Performance Computing Modernization Program (HPCMP). The authors would like to give special thanks to Navy DSRC for the help on the “special queue”, without which the simulations would not have been finished.

References

G.B. Deane, M.D. Stokes, Scale dependency of bubble creation mechanisms in breaking waves, Nature, Vol. 418, 2002, pp. 839–844.

D. Kang, S. Ghosh, G. Reins, B. Koo, Z. Wang, F. Stern, Impulsive plunging wave breaking downstream of a bump in a shallow water flume—Part I: Experimental observations, J. Fluids Struct., Vol. 32, 2012, pp. 104-120.

B. Koo, Z. Wang, J. Yang, F. Stern, Impulsive plunging wave breaking downstream of a bump in a shallow water flume—Part II: Numerical simulations, J. Fluids Struct., Vol. 32, 2012, pp. 121-134.

Z. Wang, J. Yang, F. Stern, Numerical simulations of wave breakings around a wedge-shaped bow, 28th Symposium on Naval Hydrodynamics, September 12 ~17, 2010, Pasadena, California.

Z. Wang, J. Suh, J. Yang, F. Stern, Sharp Interface LES of Breaking Waves by an Interface Piercing Body in Orthogonal Curvilinear Coordinates, AIAA-2012-1111, 50th AIAA Aerospace Sciences Meeting and Exhibit, January 2012a, Nashville, Tennessee.

J. Yang, F. Stern, Sharp interface immersed-boundary/level-set method for wave-body interactions, J. Comp. Phys., Vol. 228, 2009, pp. 6590-6616.

Z. Wang, J. Yang, F. Stern, A new volume-of-fluid method with a constructed distance function on general structured grids, J. Comp. Phys. Vol. 231, 2012b, pp.3703-3722.

Z. Wang, J. Yang, F. Stern, A simple and conservative operator-splitting semi-Lagrangian volume-of-fluid advection scheme, J. Comp. Phys., Vol. 231, 2012c, p. 4981-4992.

J. Yang, S. Bhushan, J. Suh, Z. Wang, B. Koo, N. Sakamoto, T. Xing, F. Stern, Large-Eddy Simulation of Ship Flows with Wall-Layer Models on Cartesian Grids, 27th Symposium on Naval Hydrodynamics, October 2008, Seoul, Korea

T.A. Waniewski, C.E. Brennen, F. Raichlen, Bow wave dynamics, J. Ship Res., Vol. 46, 2002, pp. 1–15.

A. Karion, T.C. Fu, T.W. Sur, J.R. Rice, D.C. Walker, D.A. Furey, Experiment to examine the effect of scale on a breaking bow wave, Carderock Division, Naval Surface Warfare Center, Hydromechanics Research and Development report, 2004, NSWCCD-50-TR-2004/060.