Dr. Hamn-Ching Chen

Dr. Hamn-Ching Chen, A.P. & Florence Wiley Professor of Civil Engineering and Professor of Ocean Engineering at Texas A&M University, received his B.S. and M.S. degrees in Power Mechanical Engineering from National Tsing Hua University, Taiwan, in 1976 and 1978, respectively; and his Ph.D. in Mechanical Engineering from the University of Iowa in 1982. After working as a research scientist at the Iowa Institute of Hydraulic Research (1982-88) and a senior research scientist at Science Applications International Corporation (1988-90), he joined the faculty of Texas A&M University in 1991. He is the originator and primary developer of the Finite-Analytic Navier-Stokes (FANS) and CHimera finite Analytic Method Potential-flow Solver (CHAMPS) codes which have been widely used for applications in ocean, civil, mechanical, and hydraulic engineering problems. His research interests include computational fluid dynamics, turbulence modeling, submarine hydrodynamics, ship berthing operations, passing ship effects, vortex-induced vibrations of deepwater risers, free-span offshore pipelines, nonlinear pipe-soil interactions, vortex-induced motions of deep-draft semisubmersibles, hurricane wave impact on offshore structures, floating offshore wind turbines, bridge scour, channel migration, and internal cooling and film cooling of turbine blades. Dr. Chen is a Fellow of ASME and an Associate Fellow of AIAA. He was the Chair of ASCE Turbulence Committee (2000-2002), and an associate editor for the ASCE Journal of Engineering Mechanics (2000-2002) and ASCE Journal of Waterway, Port, Coastal and Ocean Engineering (2002-2011). Currently, he is an Editor for the International Journal of Offshore and Polar Engineering (2011-present), and a member of Editorial Board for Ocean Systems Engineering Journal.

Keynote Presentation 2

CFD Simulation of Extreme Wave Impact on Ships and Offshore Structures using Coupled Level-Set and Volume-of-Fluid Method

A local-analytic-based Finite-Analytic Navier-Stokes (FANS) numerical method has been employed for computational fluid dynamics (CFD) simulation of fluid-structure interaction problems including violent free surface flows and extreme wave impacts. The method solves unsteady Navier-Stokes equations in moving curvilinear coordinate system using a chimera domain decomposition approach with moving overset grids. Simulation of moving bodies and fluid-structure interaction problems can be readily accomplished by allowing for arbitrary motions among various computational grid blocks without tedious grid-regeneration or mesh deformations. For problems involving violent free surface flows, a coupled level-set and volume-of-fluid (CLSVOF) method has been incorporated to provide detailed resolution of the air-water interface including breaking waves with small droplets and air bubbles. The coupled level-set and volume-of-fluid (CLSVOF) method is widely accepted as one of the most advanced interface-capturing methods. In this implementation, the level-set function is solved using the 5th-order WENO (weighted essentially non-oscillatory) scheme, and the VOF function is calculated using piecewise linear interface construction (PLIC) with mixed Euler-implicit Lagrangian-explicit (EI-LE) advection scheme. For overset grids involving embedding or overlapping grids, the VOF function depends on the cell size and cannot be interpolated between two grid blocks. A new inter-grid VOF interpolation technique has been developed to enforce mass conservation and maintain smooth interface across non-matching block boundaries. The new CLSVOF method has been employed for time-domain simulation of extreme wave impact and greenwater on two offshore platforms under Hurricane Katrina with highly localized impact force produced by random 3D short-crested waves. The CLSVOF simulation results successfully captured the breakup of overtopping wave into numerous small droplets after the massive wavefront slams into the platform deck and top-side equipment. More recently, the coupled FANS-CLSVOF method was further generalized for wave-current-structure interaction problems involving moving overset grids. CFD simulation of heave and pitch motions of a containership in random waves has been performed under both the head sea and following sea conditions. Both bow slamming at a speed of 6 knots, and stern slamming at 0 and 5 knots were investigated. These simulation results clearly demonstrated that the CLSVOF method can be effectively used for time-domain simulation of extreme wave impacts on floating offshore structures and moving vessels.