Refining Estimation of Design Loads for Wave Slam Events Experienced by Offshore Wind Turbines

Andrei Horvat, Ian Prowell, Dan Dolan, Ralph L. Nichols, P.T Gayes, Len Pietrafesa, Tingzhuang Yan, Frederick Driscoll, and Walt Musial

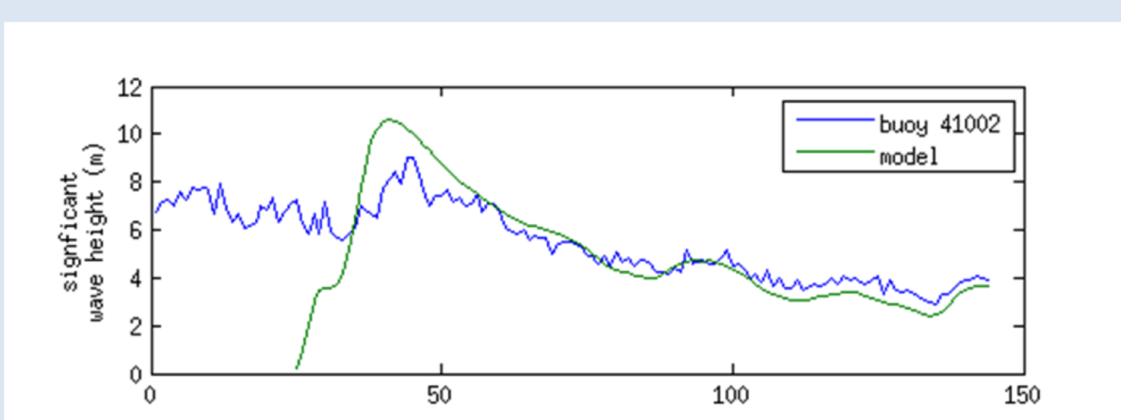
PROJECT SUMMARY

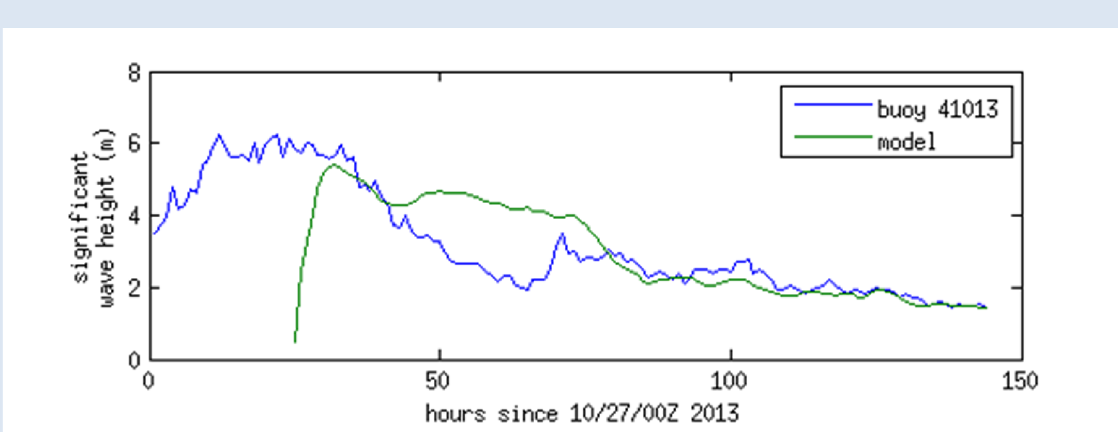
Offshore wind farms located in a shallow water environment and in areas prone to hurricanes may experience extreme breaking waves. Although little is known about their characteristics, breaking waves may represent a dominant load that a wind turbine structure is subjected to over its design lifetime. Existing analysis approaches may have limited applicability, especially for large diameter monopiles. The main goal of the research program is to improve methodology for determining the wave loading. Reducing the uncertainty associated with the load spectrum a structure can be subjected to will enable a lower design safety factors, and therefore reduce the cost of offshore wind power generation systems.

This poster focuses on a CFD study to determine the characteristics of slam loads based on anticipated wave behavior from modeling and physical observation. The model will use a numerical representation of a wave tank with a flap-type wave generator to create virtual breaking waves to analyze slam forces on a monopile foundation. Buoy data from of the North Carolina coast will be used to validate the developed CFD model. The results will be used to assess the applicability of and suggest revisions to the wave load equations for large diameter monopiles.

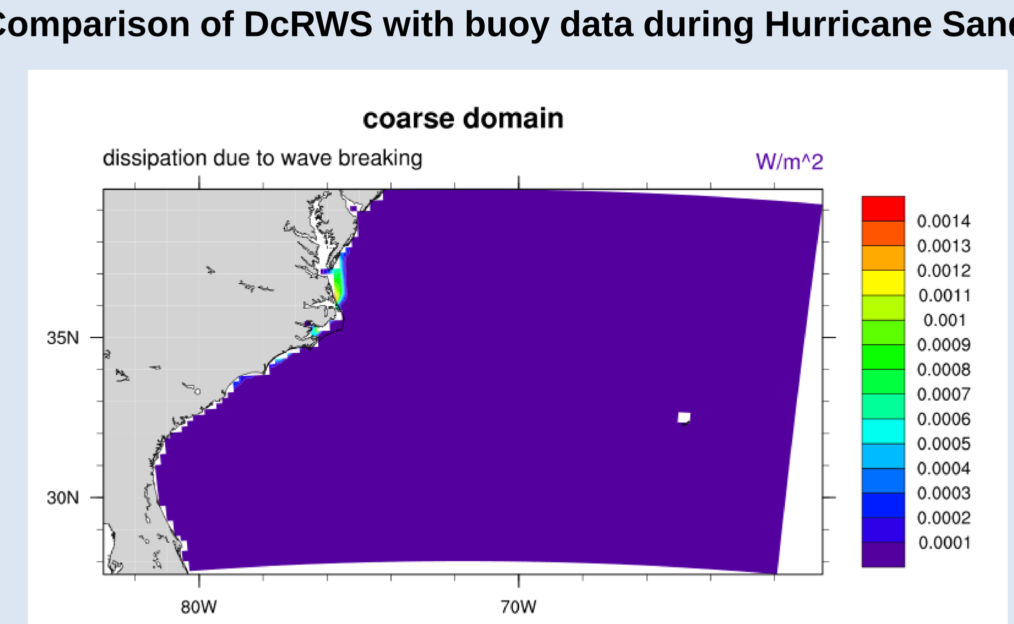
VERIFICATION

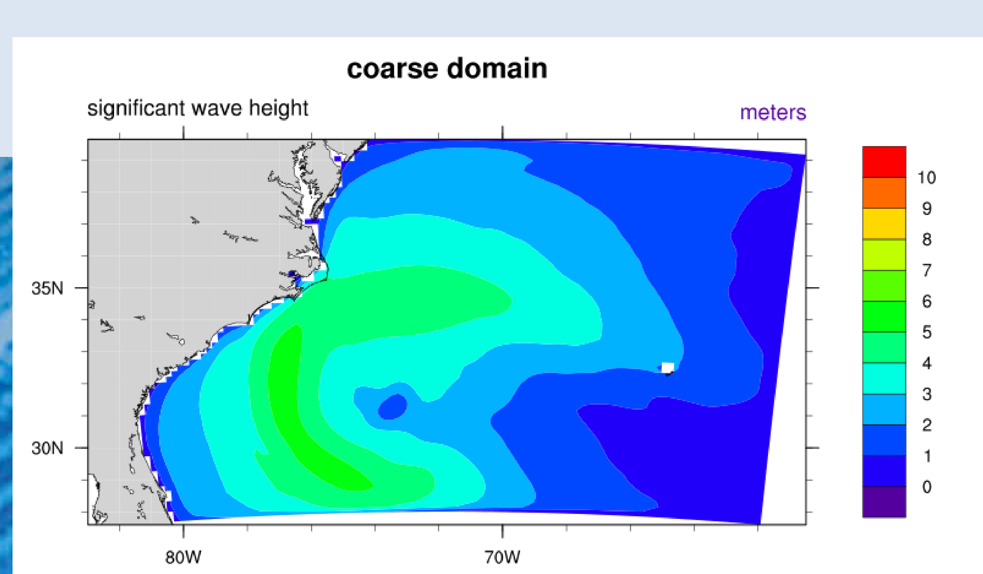
A nested dynamically coupled met-ocean model, DcRWS, was used to assess the variability of significant wave height at selected locations during Hurricane Sandy, Figure 3. Significant wave height predictions compared well with the observations from buoys, Figure 4. Time series for significant wave height and wave breaking dissipation energy was output for 3 locations of interest within the high resolution (0.72 km) model, Figure 5. Wilmington West showed wave breaking dissipation energy while New East Bank and Wilmington East did not. Wilmington West has the shallowest water at 3.48m and New East Bank is the deepest at 10.23m.





Comparison of DcRWS with buoy data during Hurricane Sandy





CFD MODELING OF BREAKING WAVE BEHAVIOR

Computational Fluid Dynamics (CFD) solves a transient form of Navier-Stokes equations. Although the process is computationally demanding, it allows recreation of all aspects of wave behavior in a simulation environment. Computational results of CFD models allow calculation of forces and moments that can be then used directly in the structural design analysis, or to obtain force coefficients for lower order, dynamic models. Lately, CFD models are also directly coupled to FEA structural analysis to provide a coupled fluid-structure interaction (FSI) simulation environment. The objective of the on-going project is to recreate breaking wave conditions on a monopile in order to analyse time variation of horizontal and vertical forces, and associated moments in the intermediate and shallow depth water environment.

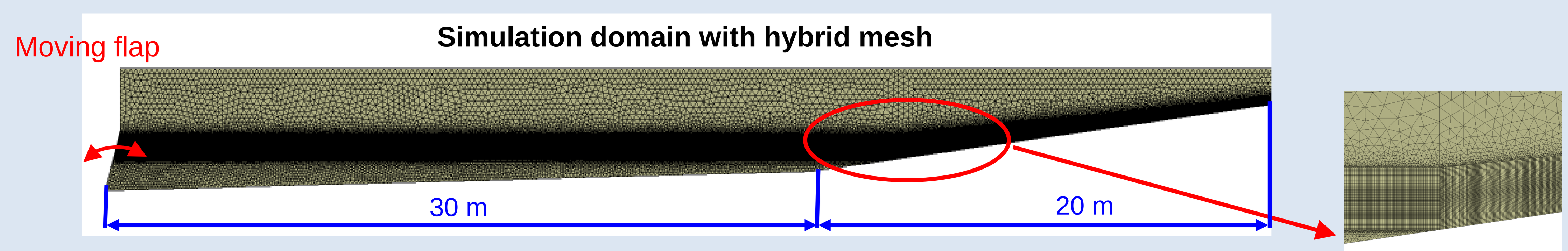
A computational (CFD) environment of a wave tank was used to assess characteristics of different wave generators (flap and piston types), wave propagation with minimum dissipation, wave breaking initiation and wave termination without reflection. Two-dimensional CFD simulations of propagating periodic waves were conducted for the initial (non-dimensional) wave height (H/d) of 0.33. Due to the positive slope, the wave height was further magnified to reach the wave breaking point. The computational results correspond well to the theoretical predictions (Battjes, 1974).

Currently, the three-dimensional CFD simulations are being performed for the water depth of 18 m. The results of these simulations will confirm the wave breaking point location and provide the initial conditions for the CFD model with a monopile.

MODELING DOMAIN

A periodic moving surface was positioned at one end of the simulation domain to model a wave generator. The propagating waves attenuate as they propagate away from the wave generator, which reduces their amplitude. To compensate for the amplitude reduction, an initial slope of 1.5º was introduced.

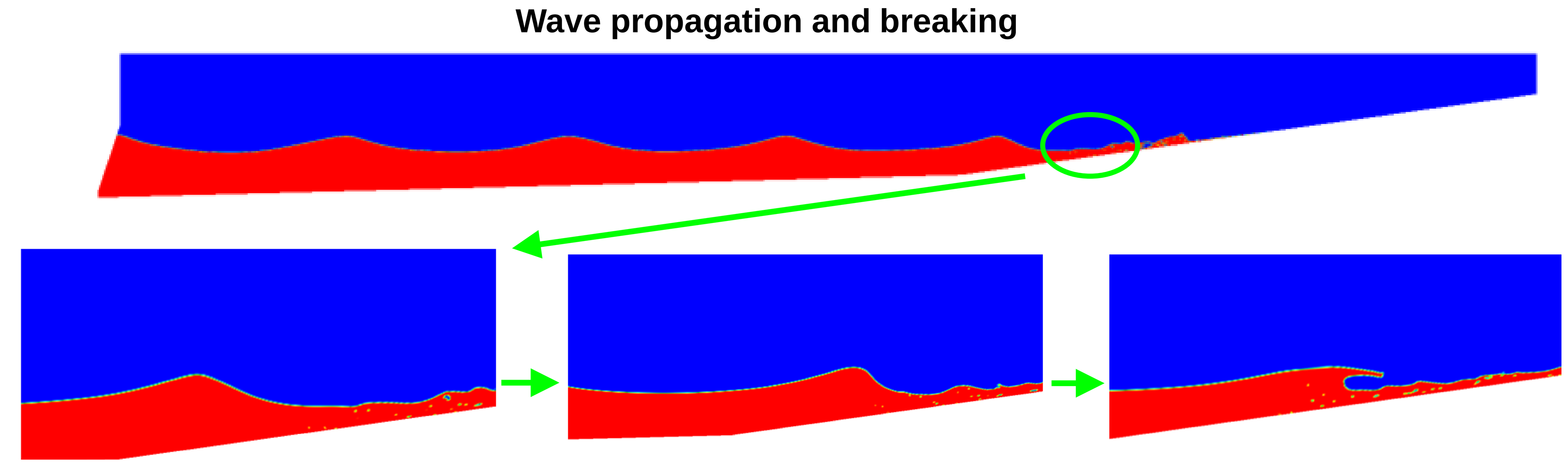
The wave amplitude will amplify only if the introduced slope is steep enough. This effect has both physical (i.e. viscosity, wall friction, interfacial drag) and numerical causes (i.e. introduced source terms, grid resolution, numerical dissipation and dispersion, under-relaxation). It was established that for the implemented CFD model, the slope angle needs to steeper than 7-8º to force wave breaking.



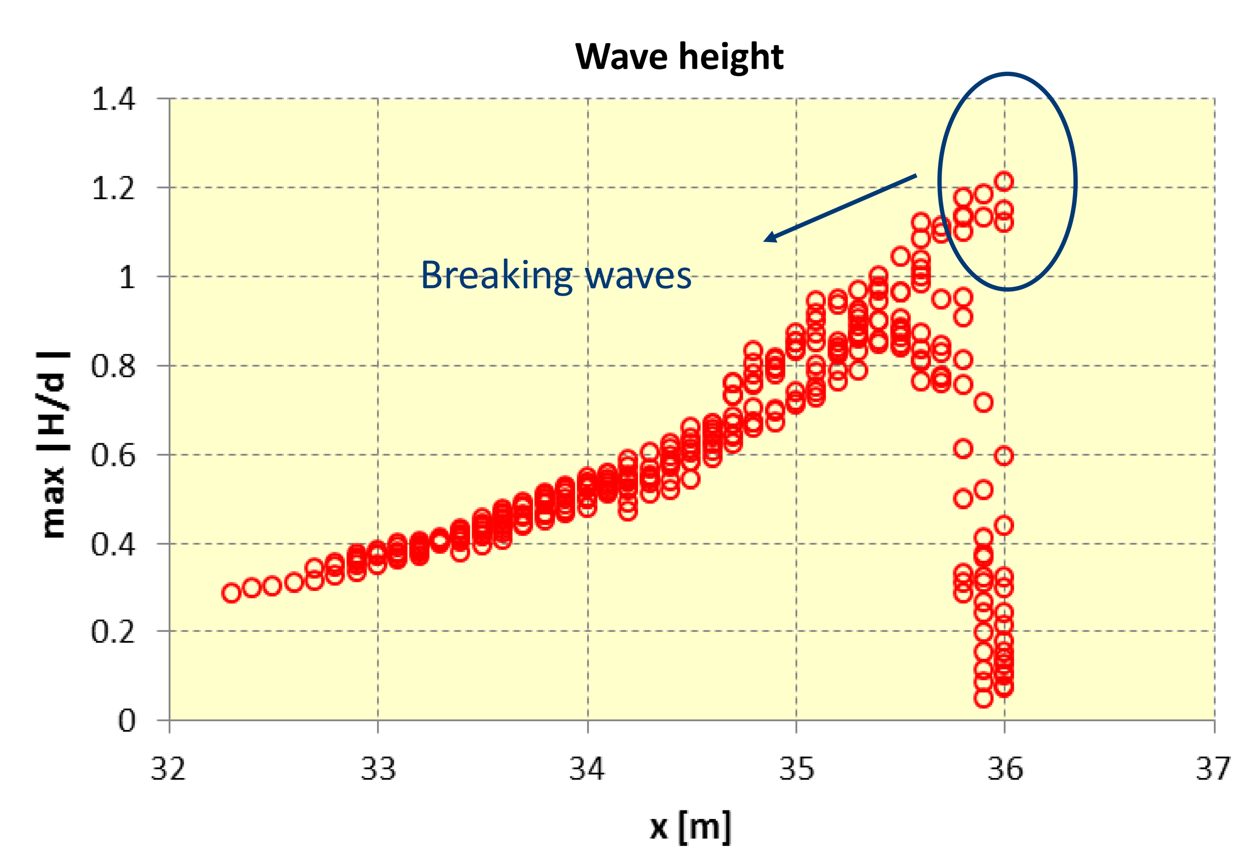
SIMULATION RESULTS

Transient CFD simulations were performed for the water depth of 1.8 m. Commercial CFD software packages ANSYS Fluent and CFX were used. Significant effort was invested to find a suitable multiphase formulation (Eulerian approach, Volume of Fluid – VOF, Level Set). The k-ω turbulence model with strong interface turbulence damping was used for the simulations. The mesh movement associated with the wave generator was introduced through a User-Defined Function (UDF) in the Fluent model or CFX Command Language (CCL) in the CFX model.

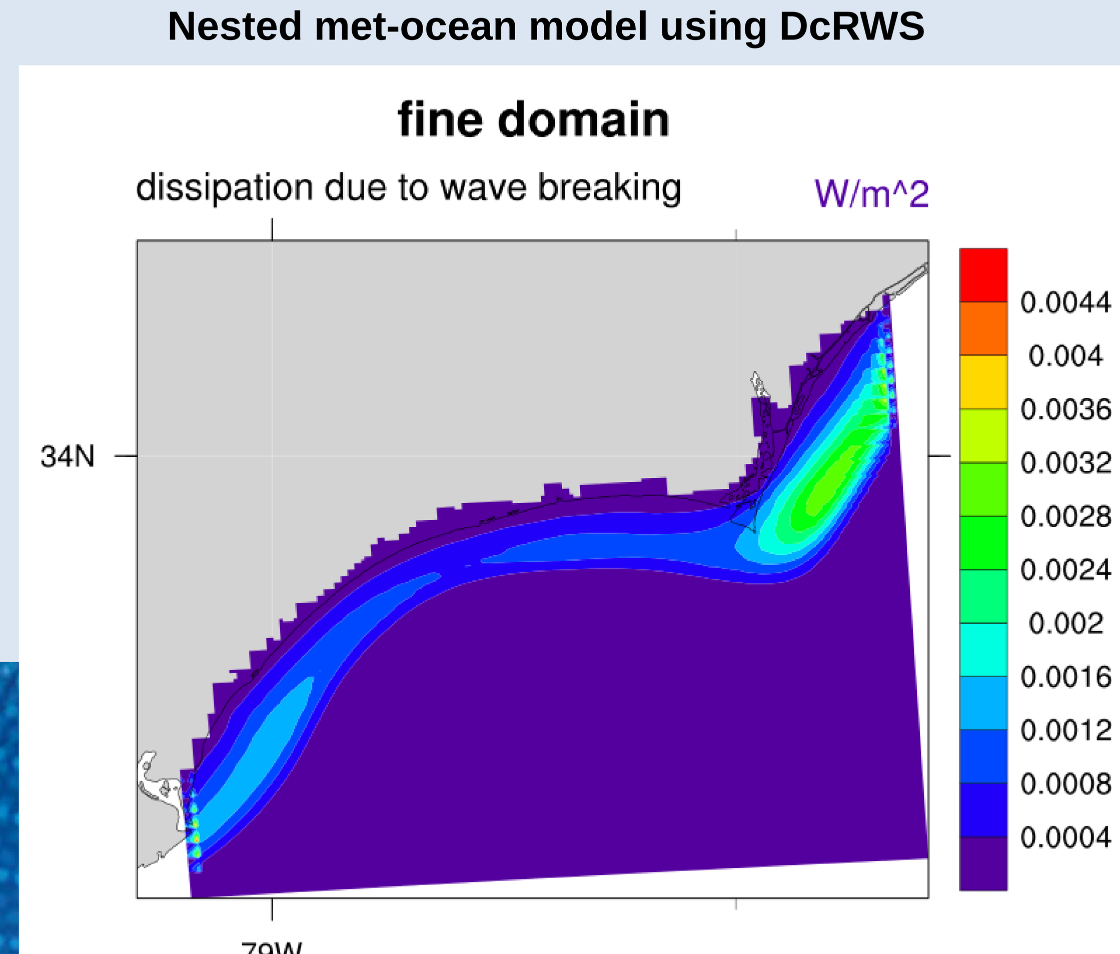
Wave propagation and breaking



Location (x) and wave height (H) were calculated for the propagating wave. Maximum relative values (H/d) were recorded to detect breaking waves. The waves break near x = 36 m, when their relative height (H/d) reaches 1.0 -1.2. These values were also theoretically and experimentally confirmed by Battjes (1974).



Nested met-ocean model using DcRWS



ACKNOWLEDGEMENTS

Funding for this work was provided by the U.S. Department of Energy Wind and Water Power Technology Office under contract DE-AC09-96SR18500.

CONTACT INFORMATION
1111 Broadway 6th Floor Oakland, CA, USA +1- 510-836-3002

Andrei Horvat AHorvat@MMIEngineering.com
Ian Prowell IProwell@MMIEngineering.com
Dan Dolan DDolan@MMIEngineering.com