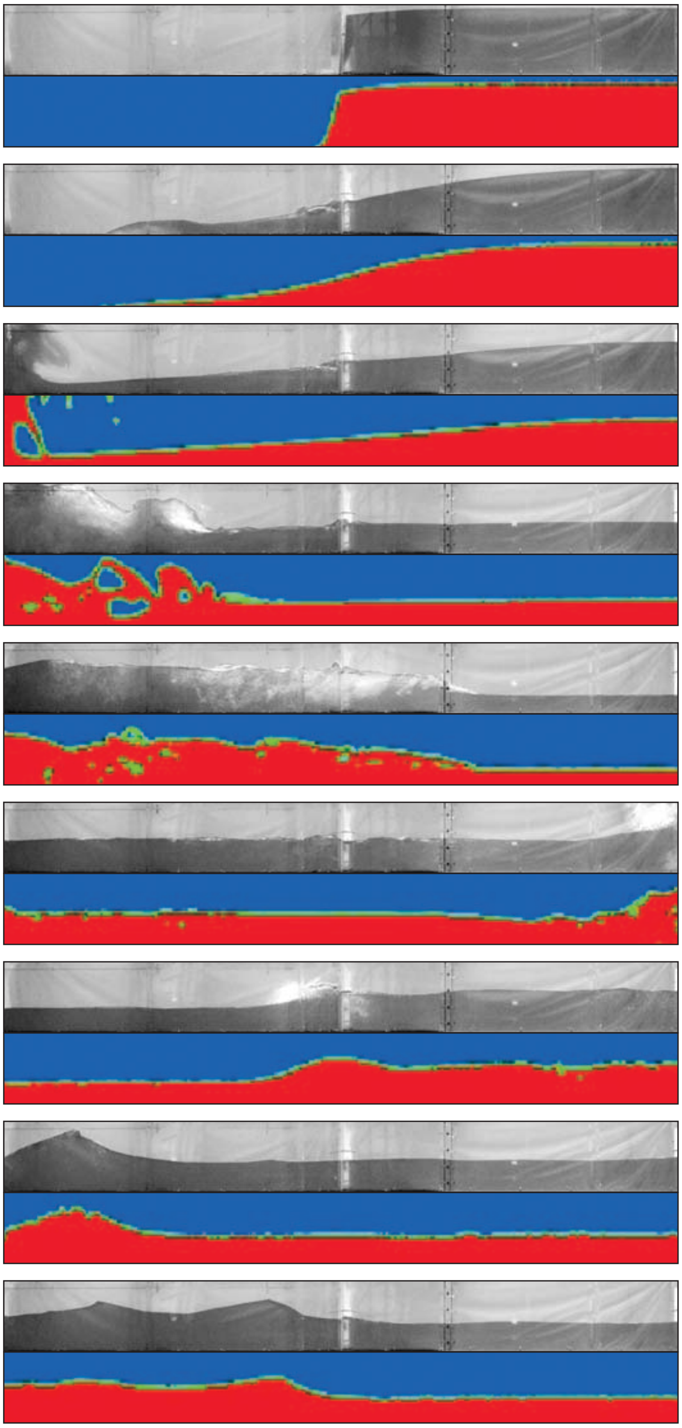
First Wave of Simulation

By Christian Schmittner, Technical University of Berlin, Berlin, Germany

FOR THE INVESTIGATION OF LOADS on offshore structures like oil rigs, it is essential to model the propagation of steep breaking water waves over several wave lengths. In order to make numerical simulations comparable to model tests from wave basins, the waves simulated numerically have to be identical to the waves measured in the experiments. Since the propagation characteristics of steep, breaking water waves are highly non-linear, a precise numerical model is required.

To check the ability of FLUENT to model extreme wave conditions, a breaking dam problem was simulated and compared to measurements from a wave tank. The test section of the tank used is 4.6m long, 0.5m deep, and 0.3m wide. A removable wall is installed across the middle of the tank. Before the start of the experiment, the left-hand section is left empty and the right-hand section is filled with water up to a level of 0.46m. The wall is then abruptly removed, causing the water block to disintegrate and a fast running wave to be generated. The wave hits the left wall of the tank, and the water shoots up, splashing against the top of the tank. The initial wave breaks, but a reflected wave begins to propagate in the opposite direction. This wave hits the right wall, where it is once again reflected. In the experiment, the waves were followed for more than 10 seconds, or about 3 full reflections. Two synchronized video cameras were used to film the experiment, one for the left side and one for the right side of the tank.

For the numerical simulation, the VOF model in FLUENT was used. A 2D mesh of approximately 20,000 quadrilateral cells was created in GAMBIT. The initial condition was a resting zone of water in the right half of the tank, corresponding to the half-filled basin in the experiment. The remainder of the fluid domain was filled with air. A time step of 5ms was used for most of the transient calculation. The results were exported as jpeg graphics and converted into a video. To compare the experiment against the numerical simulation, transparent video files were created so that the CFD predictions and experimental recordings could be overlaid. This allowed for a comparison to be made during each frame of the composite animation. The results were encouraging. FLUENT was found to predict the water run-up at the side walls of the tank quite well, both in time and in scale. The wave breaking phenomena, splashes, and drops were also captured surprisingly well. As a result of this success, FLUENT is now being used to calculate the propagation of water waves in a larger wave tank, 80m long and 1.5m deep. The dynamic mesh model is being used to simulate a moving wave board in the CFD model, and the results to date are in very good agreement with measurements. ■



Images show the comparison of experiment (grey) and simulation (color) at different times following the dam break; white regions correspond to breaking waves in the experiment