A THREE-DIMENSIONAL NUMERICAL MODEL FOR WAVE-STRUCTURE INTERACTIONS

Philip L.-F. Liu, Tso-Ren Wu and Weihua Mo
School of Civil and Environmental Engineering, Cornell University
Ithaca, NY 14853, USA

607-255-9004 (FAX), pll3@cornell.edu; tw36@cornell.edu; wm54@cornell.edu

Introduction

In designing a coastal structure in the nearshore region, such as the piles for a pier, a water intake structure or a submerged discharge pipeline, wave forces must be considered. In many situations, the waves may be breaking and the interactions between the waves and structure may also generate very complex three-dimensional flow fields. In the past, our knowledge of wave-structure interaction has relied heavily on laboratory experiments and empirical or semi-empirical correlations. However, it is well known that laboratory experiments suffer from constraints on the range of physical parameters may take on and scaling effects not to mention the cost of doing careful experiments. In recent years, because of the rapid advancement of computational power, several three-dimensional Navier-Stokes equation solvers have been developed and modified so that they can be used for wave hydrodynamics studies. With proper treatment of the breaking wave induced turbulence and boundary conditions, these models have the potential to become useful research tools for understanding the physical processes and engineering tools for the design of coastal structures.

Numerical model

In this paper, a Navier-Stokes equation solver for multiple fluids, known as Truchas (developed by Los Alamos National Laboratory), is modified to simulate 3D free surface flows. Two approaches have been adopted in modeling turbulence generated by flow separation and wave breaking. The first one employs the $k-\varepsilon$ turbulence model, using the nonlinear Reynolds stress and strain relationship, which can be found in Lin and Liu (1998 a, b). The other one adopts the Large Eddy Scale (LES) approach with the simple Smogarinsky subgrid model (see Liu et al. 2005). The modified Truchas code has the capability of solving 3D multi-phase high-density ratio fluid problems (Bussmann *et al.,* 2002) with unstructured grids. In the numerical model, the RANS or the filtered Navier Stokes equations are solved by a finite volume two-step projection method. The piece-wise linear Volume of Fluid method (Rider and Kothe, 1998) is used to track the free-surface. To study wave-structure interactions, appropriate boundary conditions are developed. Various wave generation mechanisms have been developed and implemented in the model and open boundary conditions, including the sponge layer method, are also employed. The details of the model will be discussed in the presentation.

Preliminary Results

We present here preliminary results for two wave-structure interaction studies. We note that while these two cases are limited to stationary structures, the model has the capability of dealing with a moving solid body (Liu et al. 2005).

1. Numerical simulation of solitary wave passing a circular cylinder

The first case concerns a solitary wave interacting with a circular cylinder. A series of experiments examining a solitary wave passing a slender circular cylinder was performed in the large tsunami wave basin at the Oregon State University (OSU). The plane view of the flume is shown in Figure 1.

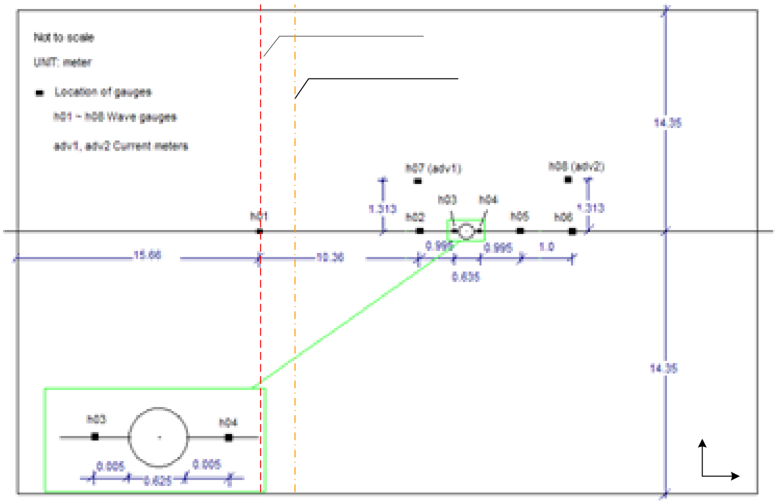

Figure 1: Plane view of the wave basin

The solitary wave travels a long distance before it hit the cylinder. The instrument positions are listed in Table 1. There are 8 wave gauges, 2 current ADVs, and 5 pressure ports right on the front face of the cylinder.

ADVs				Pressure Ports (x = 27.02m, y = 14.35m)					
	x	y	z		p01	p02	p03	p04	p05
v01	26.02	15.663	0.205	z	0.2	0.4	0.6	0.8	1.0
v02	28.645	15.663	0.205						

Wave Gauges

	h01	h02	h03	h04	h05	h06	H07	h08
x	15.66	26.02	27.015	27.65	28.645	29.645	26.02	28.645
y	14.35	14.35	14.35	14.35	14.35	14.35	15.663	15.663

Table.1 Instrument Positions (Unit: meters)

In the numerical simulations, we divided the basin into two subsections: the first subsection comprises most of the flume before the cylinder, including the wave gauge #1, and the second subsection contains the region surrounding the cylinder, starting from the position of wave gauge #1 (See Fig 1). And the computation is divided into two steps. The first step calculation is carried out in the first subsection. Since the incoming wave is uniform in spanwise direction, we can use

2-D numerical simulation instead of 3-D calculation. The purpose of this calculation is to collect the relevant data (velocity, surface elevation) at the position of wave gauge #1, which will then be used as inflow boundary conditions in the 2nd step calculation. Currently, we simulate the wave paddle movement according to Goring's solitary wave generation theory, which is also used in the experiments, to numerically generate the solitary wave. The mesh is rectangular orthogonal non-uniform mesh. The 2nd step calculation is a 3-D calculation, and it uses linear interpolation of the data obtained at 1st step to send in the solitary wave. The mesh is unstructured mesh generated by Cubit software.

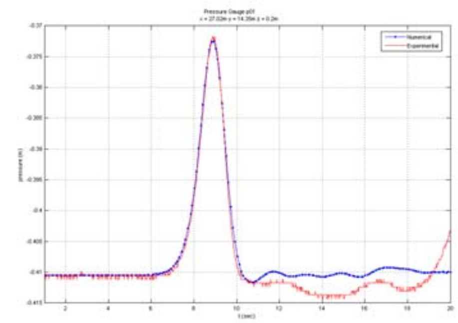

Fig.2 Comparisons for pressure data and numerical results.

Because of the page limit, we will only show one comparison here. In Figure 2 displays the data and numerical results for pressure gage p01on the cylinder for the case *H/h* = 0.1. The agreement is very good for the nonbreaking wave case. In the conference, we will present the comparisons for velocity components and free surface elevations for both non-breaking and breaking solitary waves.

2. Numerical simulation of a broken bore passing a square cylinder

The second example concerns a 3D simulation of a wall of water (bore) impinging on a vertical square column. Initially, water of 30-cm deep was impounded behind a gate and the other side was kept "almost" empty (less than 0.5 cm deep). A 12-cm by 12-cm square column was placed upright at a distance of 50 cm away from the gate; there was a wall at 58-cm downstream from the column. The gate was lifted almost instantaneously with the aid of a pneumatic cylinder with a solenoid valve - the operating air pressure is 100 psi. Numerical results have been obtained and partial comparisons with laboratory experimental data are shown here. In Figure 3 the longitudinal forces are compared. The over all agreement is very good. We note that both the $k-\varepsilon$ turbulence model and LES (Large Eddy Simulation) model have been implemented successfully. Finally, several snap-shots of the free surface profiles obtained from the numerical simulation are shown in Figure 4. More detailed comparisons and discussion will be presented in the meeting.

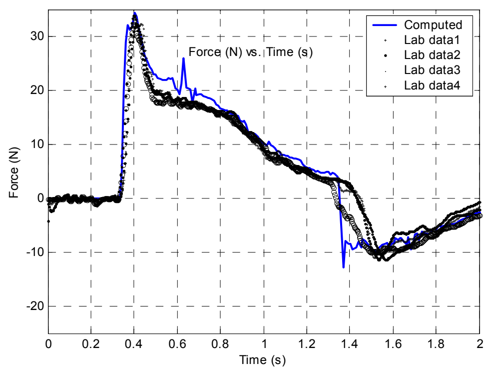

Figure 3. Comparisons for wave forces acting on the cylinder.

Acknowledgement

The authors gratefully acknowledge the financial support of the Office of Naval Research and the National Science Foundation for their support of this research.

References

Bussmann, M., Kothe D. B., and Sicilian, J. M. Modeling high density ratio incompressible interfacial flows, Proceedings of FEDSM'02, 2002 ASME Fluids Engineering Division Summer Meeting. FEDSM2002-31125, 1-7.

Lin, P. and, P. L.-F. 1998a A numerical study of breaking waves in the surf zone. *J. Fluid Mech.*, 359, 239-264.

Lin, P. and P. L.-F. 1998b Turbulence transport, vortices dynamics, and solute mixing under plunging breaking waves in surf zone. *J. Geophys. Res.,* 103(C8), 15677-15694.

Liu, P. L.-F., Wu, T.-R., Raichlen, F., Synolakis, C., and Borrero, J. 2005 "Runup and rundown generated by three-dimensional sliding masses", *J. Fluid Mech.,* (in press)

Rider, W. J. and Kothe, D. B. 1998 Reconstructing volume tracking, *Journal of Computational Physics* 141(1), 112-152.

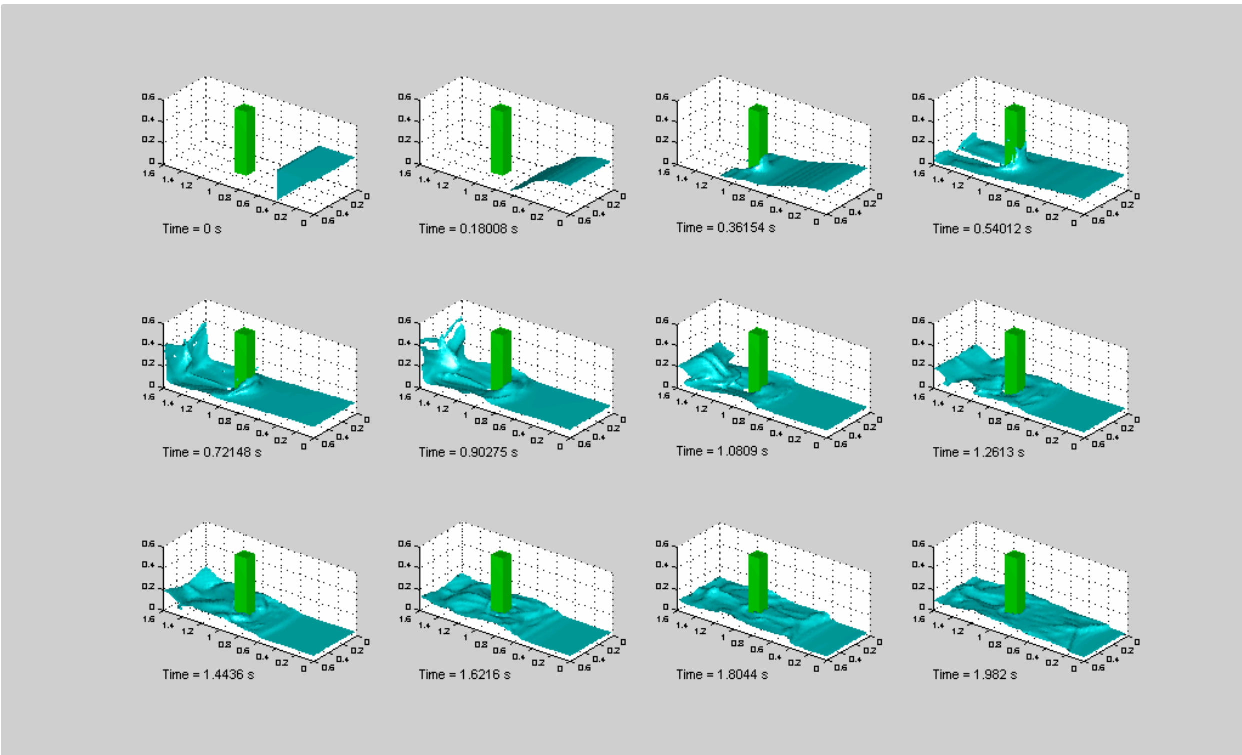

Fig.4 Snap shots of free surface profiles from numerical simulation