

Multigrid Technique for Numerical Simulations of Water Impact Phenomena

Yoshiki Nishi, Changhong Hu and Masashi Kashiwagi*
Research Institute for Applied Mechanics, Kyushu University, Kasuga, Fukuoka, Japan

A method of computational fluid dynamics is investigated for numerical simulations of strongly nonlinear phenomena with the large deformation of the free surface. The numerical solution method of this study is based on the CIP (Constrained Interpolation Profile) scheme, and the multigrid technique for solving a pressure Poisson's equation with high efficiency and robustness. The method developed was applied to dam breaking and sloshing in a tank. As a result, the numerical computations confirmed that the method developed provides fairly efficient and robust computations in 3-dimensional simulations of violent free-surface flows involving strong water impacts.

INTRODUCTION

From the perspective of seakeeping performance in rough seas, both global motions of a ship and local wave loads on structural materials should be concurrently clarified. Under the condition of high wave heights, several phenomena such as slamming, water on deck, water impact by green water, and violent sloshing in tanks continuously occur. Thus, theories or numerical methods of seakeeping problems should be able to predict the 2 points noted above with high efficiency and high robustness in order to use results obtained from numerical computations in practical ship design.

The physical phenomena noted above involve large deformations of the free surface such as wave breaking and water spray. This means that these have strong nonlinearity. Thus, Computational Fluid Dynamics (CFD), which directly solves primitive forms of hydrodynamic equations, is useful for responding to the needs listed above.

Recently, several newly developed schemes have been proposed that are capable of simulating fine structures of the free surface in a multiphase flow field (liquid, gas and solid). For example, Kleefsman et al. (2004, 2005) simulated wave impact problems using the Comflo that is based on the finite difference method and VOF for capturing free surfaces. In addition, particle methods have been actively developed for violent free-surface simulations: SPH (Smoothed Particle Hydrodynamics, Iglesias et al., 2004, and Oger et al., 2006), and MPS (Moving Particle Semi-Implicit, Koshizuka et al., 1998, and Sueyoshi, 2005).

This study concerns a numerical method based on the CCUP (CIP-Combined and Unified Procedure) method (Yabe et al., 2001), which has the following features:

- compact upwind scheme with a resolution inside a cell, which can accurately reproduce the advection of matter; and,
- pressure-based algorithm that can treat the multiphase flow field by one set of governing equations.

The first feature corresponds to the CIP (Constrained Interpolation Profile) scheme proposed by Takewaki et al. (1985).

Several numerical models based on this method have been developed and applied to the highly nonlinear free-surface problems related to ship hydrodynamics (e.g. Hu and Kashiwagi, 2004; Hong et al., 2005). These previous studies applied this scheme to 2-dimensional dam breaking, oscillatory motions of a floating body in waves, and a ship's wave in a waterway; they reported that numerical results agree well with experiments with respect to the shapes of the free surfaces and time variations of pressure on a wall.

On the other hand, it is also a fact that many CFD methods have some problems pertaining to computational efficiency and robustness, which are required in practical engineering. The main purpose of this study is to present a possible solution by focusing on an iterative solution method to solve a pressure Poisson's equation.

A finite differentiation of this equation leads to a linear simultaneous equation with huge numbers of known coefficient and unknown variables. The iterative operation consumes most of the computational time. It follows that it often becomes a bottleneck in the development of CFD methods. Hence, the advancement of the iterative method can contribute to the improvement of developing CFD-based ship hydrodynamics.

The multigrid method (MG) has a characteristic that contributes to the overcoming of difficulties involved in conventional iterative methods noted above. We developed a numerical calculation method based on the CCUP combined with the multigrid technique, and applied it to a 2-D dam-breaking problem (Nishi et al., 2005). The result obtained from this previous work confirmed that the MG is a quick solver, and that as a preconditioner in GMRES (Generalized Minimal RESidual) algorithm (Saad and Schultz, 1986) it performs fairly well.

Three-dimensional effects must be taken into account especially when we pay attention to a local phenomenon involving impact loads. The difficulty of iterative solution methods increasingly stands out in 3-D problems, compared with 2-D problems. The objective of this study, then, is to present a useful strategy for the implementation of a 3-D computation using the MG technique, and to apply the newly developed code to 3-D computations of free-surface problems: dam breaking and sloshing.

NUMERICAL SOLUTION METHOD

The CCUP Method

The CCUP, proposed by Yabe et al. (2001) for hydrodynamic simulations for both compressible and incompressible fluid, is the

*ISOPE Member.

Received February 22, 2006; revised manuscript received by the editors June 20, 2006. The original version (prior to the final revised manuscript) was presented at the 16th International Offshore and Polar Engineering Conference (ISOPE-2006), San Francisco, May 28–June 2, 2006.

KEY WORDS: Multigrid, free-surface flows, computational efficiency, computational robustness, water impact.