CFD PARAMETRIC STUDY FOR 2D WATER ENTRY

H.H. Lee*1 and S.H. Rhee2

A parametric study for for the water entry of a two dimensional symmetric wedge with deadrise angle of 10 degrees was carried out to find out the most dominant parameter. Water entry problem with constant velocity is simplified as the stationary wedge in the way of the upcoming water surface. The calculated impact loads showed that the effect of the viscosity was not so important in this problem. For a given grid system a suitable time step size can be found. The most sensitive parameter was found to be the grid size.

Key Words : Water Entry, Slamming, Impact, Viscosity, Grid Size, Time Step Size

1. INTRODUCTION

Commercial vessels are getting large, most evident in the containerships. Modern containerships have a large flare in the stem and a wide flat stern shape in order to increase the deck area and to improve the propulsive efficiency. This kind of hull form can be more susceptible to the slamming loads so the structural design considering the slamming impact loads is more necessary.

Wagner[1] derived a well-known theoretical solution for the impact pressure distribution acting on the wedge entering water with constant velocity by using a momentum theory. Chuang[2] performed many experiments and suggested an empirical formula. Zhao et al.[3] performed experiments and developed theoretical method to estimate the impact loads for arbitrary shaped bodies. Muzaferija et al.[4] used a computational fluid dynamics (CFD) tool to solve the slamming problem. Yang et al. [5] showed a possibility of using the commercial CFD tool with some improvement. But it was observed that the CFD results are dependent on the selection of modeling parameters.

As computing power increases rapidly and more advanced numerical schemes are developed, it becomes possible to use CFD tools for this kind of impact problems. However, it is well known that CFD results are sensitive to the numerical parameters used in the computations.

1 비회원, 현대중공업(주) 선박성능연구실
2 정회원, 서울대학교 조선해양공학과
* Corresponding author, E-mail: hhlee@hhi.co.kr

In the present study, parameters that may influence the impact load computations such as viscosity, grid size and time step size are investigated systematically by using a CFD code solving the Reynolds-averaged Navier-Stokes (RANS) equations.

2. PROBLEM DESCRIPTION

A simple two dimensional (2D) rigid wedge model is chosen as a test body in order to check the effect of the numerical parameters more clearly. Also it helps the CFD parametric study results not to be tainted by the geometric complexity. Specifically, a 2D symmetric wedge, which is 0.6 m wide, 0.4 m deep and deadrise angle of 10 degrees, was selected as a test geometry. For this model physical experiments have been done previously[5].

Slamming takes place when there are relative motions between a solid body and a fluid, e.g., a spacecraft lands on calm water, waves hit a fixed offshore platform and a ship advances in waves. In this study, the simple wedge is fixed in space and the calm water surface is set to move upward with constant speed from 0.1 m below the tip of the wedge apex.

3. COMPUTATIONAL METHOD

Fluid flow is assumed to be incompressible and the behavior of the fluid is modeled by the unsteady RANS equations. The associated continuity and momentum equations were solved by using a commercial CFD program, FLUENT.

The size of the entire calculation domain and the

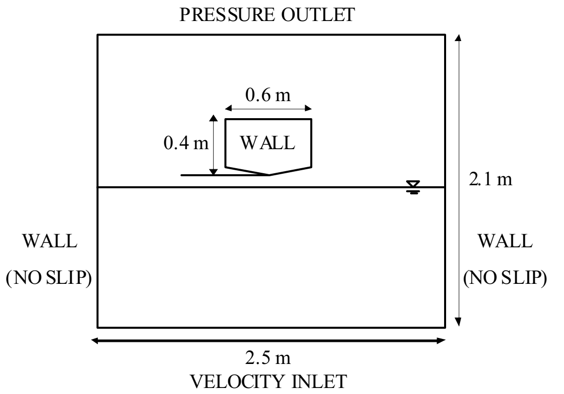

Fig. 1 Schematic view of the wedge model, the calculation domain and the boundary conditions

boundary conditions used in the problem are shown in Fig. 1. The size of the domain is the same as the size of the physical water basin used in the experiments by Yang et al.[5]. Grid generation is done by using GAMBIT.

The movement of the fresh water was controlled by giving a constant velocity of 1.83 m/s at the bottom boundary as a velocity inlet. The left wall, right wall and the wedge boundaries are treated as a stationary wall with no slip condition. The pressure outlet condition was imposed on the top boundary.

In order to capture the violent free surface movement due to the impact, the piecewise linear volume-of-fluid (VOF) scheme was used with two fluids (air and fresh water). The SIMPLE algorithm was used for the velocity-pressure coupling. The pressure and the momentum equations are discretized with pressure staggered option (PRESTO) and quadratic upwind interpolation for convective kinematics (QUICK) scheme respectively. First order implicit scheme is used for the time integration. Under-relaxation factors are set as 0.3 and 0.7 for pressure and momentum equations respectively.

Three different structured grids (coarse, medium and fine grids) with similar pattern were prepared as shown in Fig. 2. Grid refinement was done near the wedge and around the free surface where the fluid motion is supposed to be violent. The details on these grids are summarized in Table 1.

Six pressure sampling points were defined with even spacing as shown in Fig. 2, where the locations from P1 to P5 were the same as those of the pressure transducers in the physical experiments(Table 2).

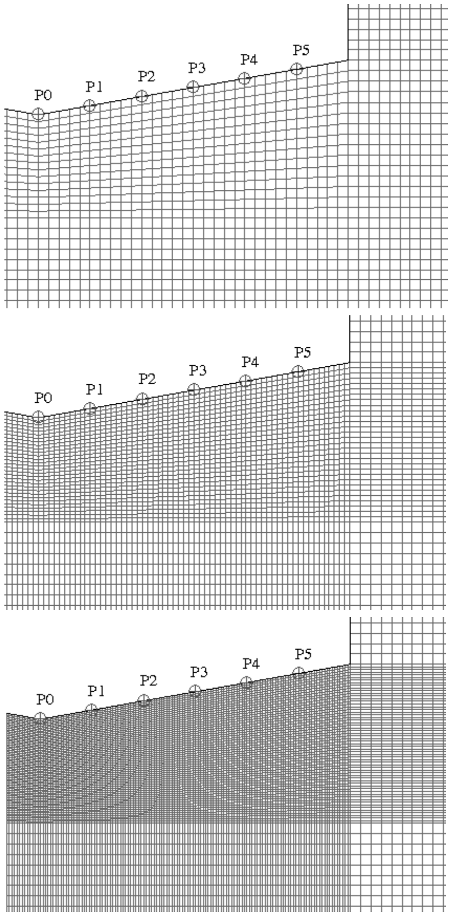

Fig. 2 Three different grid system: coarse (top), medium (middle) and fine (bottom) grid

4. Effect of Numerical Parameters

4.1 Effect of Viscosity

The viscosity is selected as the first parameter to be varied in the present parametric study. Nearly every theoretical work and the boundary element methods are based on the ideal fluid which has no viscosity, and these methods produce quite good results. One of the main

Table 1 Three different grids

Grid Type	Smallest grid size (mm)	# of cells
Coarse	10.0	45400
Medium	5.0	56950
Fine	2.5	85450

reasons of this is that the very short time duration of the impact makes it difficult for the viscosity effect to spread spatially.

Three different fluid flows such as inviscid, laminar and turbulent (realizable k-ε model with standard wall function) flows are modeled using the medium level grids with time step size of 1 millisecond (ms). The pressure time histories for both inviscid and laminar flows are almost identical but the results from turbulent flows are somewhat different(Fig. 3). At the initial stage of the impact the pressure in turbulent flows is larger than that of inviscid or laminar flows but this trend reverses as the jet flow develops. As can be inferred from the pressure results, the vertical force, which is the integration of the pressures on the body surface, in turbulent flows is smaller than the other cases as shown in Fig. 4.

4.2 Effect of Grid Size

It is well known that CFD results are sensitive to grids. The grids should be nicely discretized such as to best follow the fluid flows. In this study, after setting the basic structure of the grid system fixed only the number of grids is changed in three levels (coarse, medium and fine). Calculations are done in laminar flows with time step of 0.1 ms.

The pressure time histories from three grids are quite different as can be seen in Fig. 5. As the size of the grid gets smaller the calculated pressures get larger and the jet travels on the wedge surface faster. The results from "fine" grid show very fluctuating pressures and this kind of behavior is clear as time goes on. This may be explained by using the CFL (Courant-Friedrichs-Lewy) condition. Since the time step size is fixed and the flow velocity is also fixed in reality, a finer grid will increase the Courant number and this will worsen the CFL condition. The vertical forces also increases as the size of the grid gets smaller but the fluctuating behavior is more clear in the coarse grid(Fig. 6). Further grid refinement has not been done because the stability problem is foreseen clearly.

Table 2 Pressure sampling locations

Name	Horizontal distance from the apex (mm)	Vertical distance from the apex (mm)
P0	0.0	0.0
P1	50.0	8.8
P2	100.0	17.6
P3	150.0	26.4
P4	200.0	35.3
P5	250.0	44.1

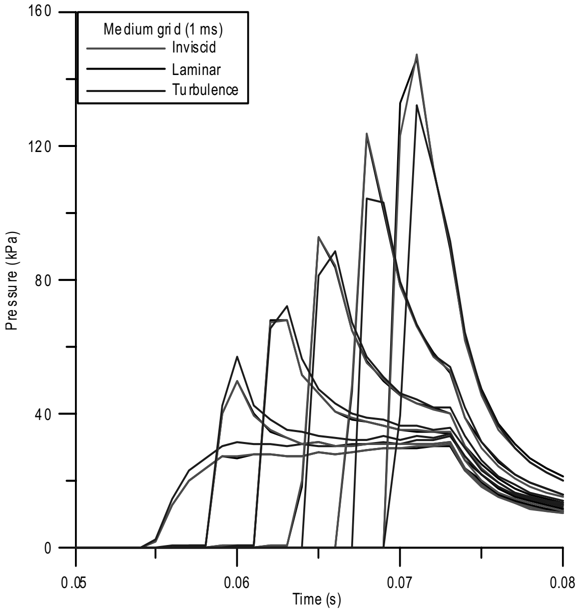

Fig. 3 Comparison of pressures (viscosity effect)

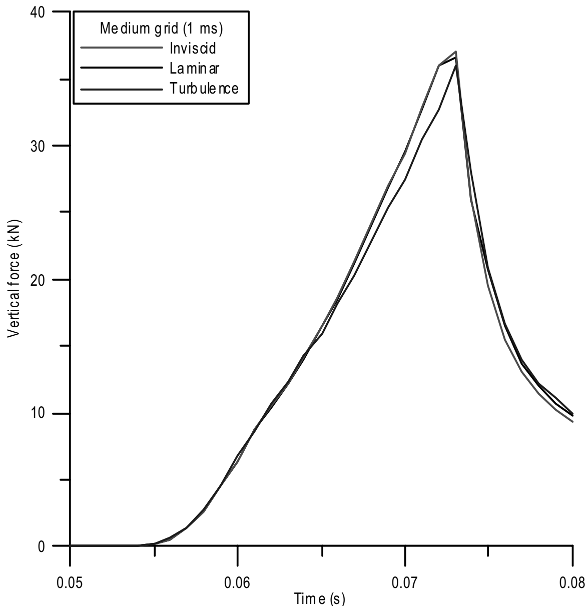

Fig. 4 Comparison of vertical forces (viscosity effect)

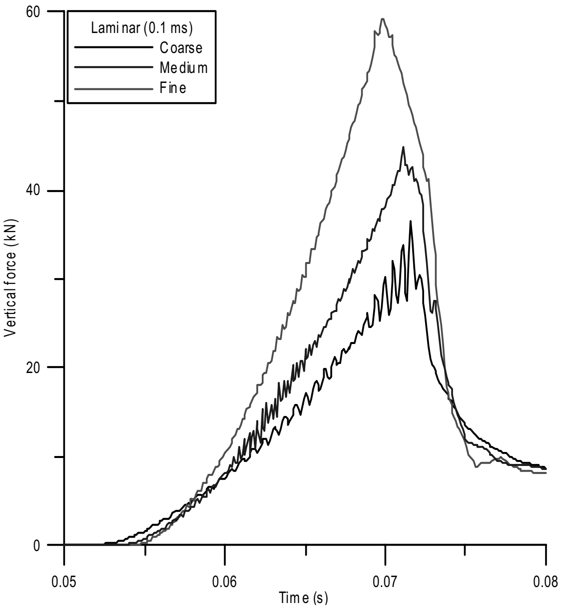

Fig. 5 Comparison of vertical forces (grid size effect)

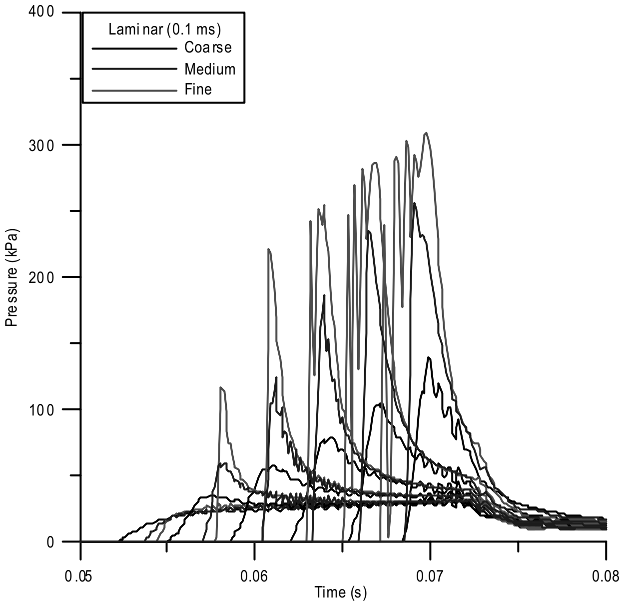

Fig. 6 Comparison of pressures (grid size effect)

4.3 Effect of Time Step

The size of the time step also plays an important role in numerical simulations. Generally speaking, it should be sufficiently small enough to capture the physical flow characteristics. Three different time step sizes such as 1.0, 0.2 and 0.1 ms have been adopted for medium grid simulations in laminar flows.

The calculated pressure time histories in Fig. 7 show that the results converges when the time step size is smaller than 0.2 ms. Larger time step size under-predicted the pressures. The same can be said for the calculated vertical forces in Fig. 8.

5. Comparison with the Other Results

Wagner[1] derived a theoretical equation for a wedge entering into water with constant speed which can be used for predicting the maximum pressure on a wedge surface as follows;

$$P_{max} = \frac{1}{2}\rho V^2\left(\frac{\pi}{2\tan\Theta}\right)^2 \qquad (1)$$

where P_{max} is the maximum impact pressure, ρ the density of water, V the initial vertical velocity and Θ the deadrise angle of the wedge. In our case, the maximum pressure is predicted about 133 kPa by using Wagner's method.

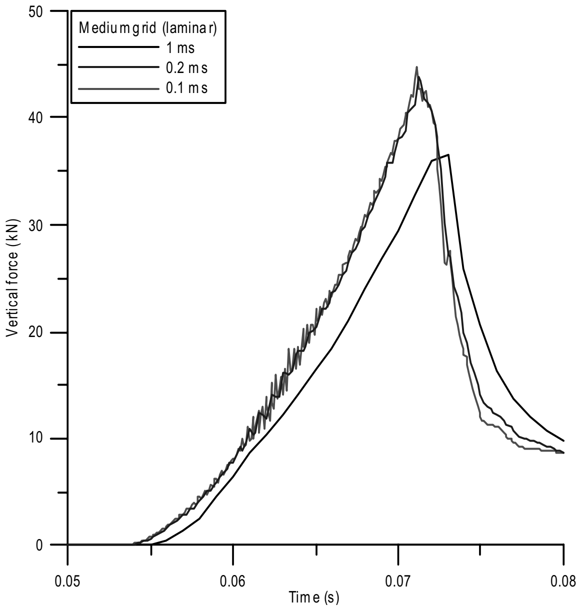

Fig. 7 Comparison of vertical forces (time step size effect)

Stavovy and Chuang[6] performed a series of drop tests

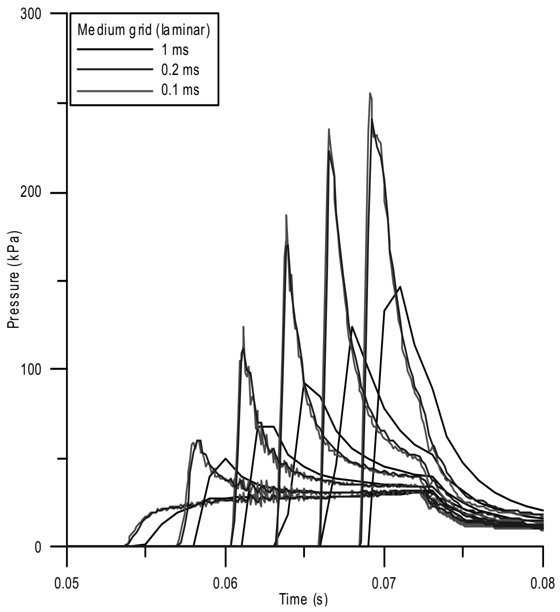

Fig. 8 Comparison of pressures (time step size effect)

and provided pressure coefficient (C_P). For a wedge of 10 degrees deadrise angle C_P is about 46.7 and this corresponds to the maximum pressure of about 78 kPa.

Free fall experiments have been performed in Hyundai Maritime Research Institute and the details can be found in Yang et al. [5]. The maximum pressure from these experiments is about 84 kPa at P2 location.

All the CFD results of this paper show that the impact pressure grows monotonically as the measuring points is located farther from the initial impact position. But the measured results from free fall experiments show rise and fall. This discrepancy is caused by the simplification of the problem to a constant inlet velocity.

6. CONCLUSIONS

Systematic numerical calculations have been performed for a two dimensional simple wedge model with deadrise angle of 10 degrees for a parametric study for the fluid viscosity, grid size and time step size which can influence the impact load calculations.

The pressures and vertical forces in inviscid and laminar model showed similar results and those in turbulent model showed slightly different results. This may be due to the very short time duration and show that the effect of viscosity does not spread much.

The size of the grid was found to be the most sensitive parameter which influences the impact load calculations. Finer grid resulted in larger pressures and vertical forces. For a given grid size, an appropriate time step size could be found relatively easily.

REFERENCES

[1] 1932, Wagner, V.H., "Über Stoß- und Gleitvorgänge an der Oberfläche von Flüssigkeiten," *Zeitschrift für angewandte Mathematik und Mechanik*, Vol.12, No.4, pp.193-215.

[2] 1967, Chuang, S.L., "Experiments on Slamming of Wedge-Shaped Bodies," *Journal of Ship Research*, Vol.12, No.3, pp.190-198.

[3] 1996, Zhao, R., Faltinsen, O. and Aarsnes, J., "Water entry of arbitrary two-dimensional sections with and without flow separation," *Proc. 21st Symposium on Naval Hydrodynamics*, Trondheim, pp.408-423.

[4] 1998, Muzaferija, S., Peric, M., Sames, P.C. and Schellin, T.E., "A Two-Fluid Navier-Stokes Solver to Simulate Water Entry," *Proc. 22nd Symposium on Naval Hydrodynamics*, Washington D.C., pp.638-349.

[5] 2007, Yang, S.H., Lee, H.H., Park, T.H., Lee, I.H. and Lee, Y.W., "Experimental and Numerical Study on the Water Entry of Symmetric Wedges and a Stern Section of Modern Containership," *10th Symposium on Practical Design of Ships and Other Floating Structures*, Houston, pp.518-526.

[6] 1976, Stavovy, A.B. and Chuang, S.L., "Analytical determination of slamming pressures for high-speed vehicles in waves," *Journal of Ship Research*, Vol.20, No.4, pp.190-198.