한국해양공학회지 제19권 제4호, pp 15~20, 2005 (ISSN 1225-0767)

Slamming Load Analysis

HEE-SUNG LEE*, SUN-HONG KWON**, JONG-HYUN LEE**, BYUNG-HOON JUNG***, AND KI-JONG SONG***
*Sam Sung Heavy Industries Co. Ltd., Geoje, Korea
**Department of Naval Architecture and Ocean Engineering, Pusan National University, Busan, Korea
***Hyundai Heavy Industries Co. Ltd., Ulsan, Korea

KEYWORDS: : Slamming, Pneumatic Cylinder, LM-guide, In-hose Code, FLUENT

ABSTRACT: *This paper presented the development of a new experimental and numerical technique for the investigation of slamming impact. The in-house code was developed, based on the boundary element method. The application of commercial code was attempted using FLUENT. Pneumatic cylinder and LM-guide were introduced to closely simulate the free fall phenomenon. It was demonstrated that the proposed experimental and numerical studies can be useful tools in investigating slamming phenomenon.*

1. INTRODUCTION

When a ship travels in a rough sea, it frequently experiences various types of impacts from waves. The impact of a ship striking a water surface often causes extremely large load with high pressure, which shows transient behavior. It can result in substantial damage to the ship structures. Thus, the water entry problem has attracted much research for its practical importance in the field of naval architecture and ocean engineering (Greenhow ,1987; Arai et al.,1994).

This paper presents a comparison study of experimental data and numerical results. The potential code was developed by the present authors. This potential code was based on the boundary element method (BEM). The commercial code FLUENT was used to solve the water entry problem in termsof viscosity. The present study was carried out under a two-dimensional analysis. Most theoretical studies pertaining to the water entry impact load have been done on two-dimensional analysis for complicated body geometry (Korobkin,1996; Dobrovol'skaya,1969; Armand and Cointe,1986;Watanabe,1986; Zhao et al.,1996). The interaction between the body and fluid was simulated with a user-defined function when the FLUENT was utilized.

Experimental tests on free fall have been conducted to investigate the slamming phenomenon, as that method is conducive to inducing the velocity that we desire (Garabedian,1953). However, free fall has several defects. First of all, a guide-line with the appropriate degree is necessary, and it is difficult to make this guide-line. The friction is substantial in free fall test using the equipment. Second, the movement distance of the drop test model is longer to increase the velocity, so the line between sensor and amplitude is longer, creating a serious noise that makes it difficult to determine the peak values of impact forces. Third, in the case of the free fall test, it is difficult to replicate the experiments. To overcome these defects, pneumatic cylinder was introduced (Howison et al.,1991; Zhao and Faltinsen,1993; Mei et al.,1999). In other words, we tried to induce the same velocity with free fall, using a pneumatic cylinder. The velocity was measured by a high-speed camera that takes pictures at a rate of 500 frames per second. The models are wedge shaped and were made with three holes on the bottom. The pressures of the model were measured, separately, according to the locations of holes. The pressure time history of pneumatic cylinder was quite different, compared to that of the free fall test, and there was a difference in peak pressure values. For example, the pressures of the 20-degree wedge model were significantly less than those of the 10-degree wedge model. All those computational and numerical results are compared.

2. IN-HOUSE CODE BASED ON BOUNDARY ELEMENT METHOD

Let's describe the boundary value problem to be solved, before we mention the developed potential code. The present paper deals with two dimensional analysis. The z axis takes the positive direction upward. z=0 location represents undisturbed free surface, which corresponds to t=0. The fluid

---

제 1제자 권순홍 연락처: 부산광역시 장전동 산60번지
051-510-2347 shkwon@pusan.ac.kr

is assumed to be inviscid and incompressible. The additional assumption of irrotationality yields the Laplace equation as the governing equation. The mathematical expression of the governing equation is

$$\nabla^2\phi=0 \qquad \text{in the fluid region} \qquad (1)$$

The boundary condition on the wet body surface SB is

$$\frac{\partial\phi}{\partial n}=Vgn \qquad \text{on SB} \qquad (2)$$

where V represents the body velocity, and the outward normal into the fluid domain is denoted as n. The two boundary conditions for the free surface SF are kinematic and dynamic boundary conditions, which can be written as follows:

$$\frac{Dx_F}{Dt}=\nabla\phi \qquad \text{on SF} \qquad (3)$$

$$\phi=0 \qquad \text{on SF} \qquad (4)$$

The new position of the free surface can be obtained by integrating the kinematic boundary condition (3). The dynamic boundary condition.

On the side wall boundary SW and tank bottom boundary SD, the following boundary conditions are imposed

$$\frac{\partial\phi}{\partial n}=0 \qquad \text{on SW and SD} \qquad (5)$$

To solve the given boundary problem using BEM, the following equation is used, which can be derived from Green's second identity.

$$\phi=\int_s\frac{\partial\phi}{\partial n}G(x,y,\xi,\eta)ds+\int_s\phi\frac{\partial G(x,y,\xi,\eta)}{\partial n}ds \qquad (6)$$

where (ξ,η) denotes the position of source point, G is the Green function.

The fundamental solution adopted here is the well known logarithmic function. Its mathematical expression is as follows:

$$G=\frac{1}{2\pi}\ln\sqrt{(x-\xi)^2+(y-\eta)^2} \qquad (7)$$

The above integral equation can be solved by dividing the boundary into straight line segments, over which the values of the potential and its derivatives are assumed to be constant. The free surface boundary near the body was densely segmented, so that the complex displacement of the boundary can be described.

Only half of the computational domain was considered, due to the symmetry of the present problem. The initial free surface is assumed to be calm. The initial potential values of the free surface were set to be zero. To initiate the BEM computation, a small portion of the body must be submerged.

The constant body velocity was set to be 1 for all the computations. Every scale used in this computation has been normalized in this study. The number of meshes on the body, free surface, and wall were 96, 304, and 20, respectively. The time increment on all computations was 0.0005 seconds.

Meshes are composed of different sizes, according to the location of boundary. The size of meshes located at the intersection part of body and free surface is smaller than that of other parts.

Fig.1, Fig.2 and Fig.3 show the results, compared with Zhao & Faltinsen's numerical results (Faltinsen,1990). The BEM code developed in this study shows quite good agreements with Zhao & Faltinsen's calculation results.

3. COMMERCIAL CODE APPLICATION

The viscous computation for the water entry problem was carried out using a commercial code FLUENT. The computational domain in FLUENT is discretized into a finite number of tetrahedral cells. The movement of the air liquid interface is traced by the distribution of the volume fraction of water in a computational cell. The slamming phenomenon was described by forcing the uniform flow approach to the body. The flow was assumed to be laminar. Any interested readers in FLUENT may visit the web site of the FLUENT. Dense meshes are distributed near the body and the free surface around the body surface.

To take into account the interaction between the body and fluid, the user defined function has been adopted. A concise introduction to the user defined function is given below. A user defined function, or UDF, is a function that is programmed to be dynamically loaded with the FLUENT solver to enhance the standard features of the code [FLUENT manual]. UDFs are written in the C programming language. They are defined using DEFINE macros, which are supplied by Fluent Inc. They access data from the FLUENT solver using predefined macros and functions, also supplied by Fluent Inc. Every UDF contains the udf.h file inclusion directive (#include "udf.h") at the beginning of the source code file, which allows definitions for DEFINE macros and other Fluent provided macros and functions to be included during the compilation process. UDFs are executed as either interpreted or compiled functions in FLUENT. Values that are passed to the solver by a UDF, or returned by the solver to a UDF, must be specified in SI units.

The unstructured grid is shown in Fig.4. The program, which adopts a user defined function, yields a varying body velocity when it interacts with fluid. The traditional method assumed constant body velocity. The results shown in Fig.5 and Fig.6 show the differing results between the two approaches.

4. EXPERIMENTS

- **Model**

The specimen is a wedge shaped model, measuring 29cm(L)×29cm(B)×0.25cm(T). The material of the specimen is stainless. The deadrise angles of the models are 10 degrees and 20 degrees, respectively. There are 3 small holes to fix the pressure gauges. Fig. 7 shows the entire shape and 3 holes holding the sensors. The sensors are located at 3 points, A, B and C.

- **Pneumatic Cylinder Test**

Pneumatic cylinder can be discharged to 3m/s. This cylinder was produced by SMC in Japan. The inner diameter is 40mm, and the shooting distance is 100mm. Fig.8 shows the shape of the cylinder.

- **LM GUIDE Test**

The LM GUIDE was introduced to carry out the free fall test in this experiment. The LM GUIDE consisted of SBR and SBS. There are ball bearings on the inner side of SBR, in order to reduce the frictional resistance between SBR and SBS. Fig.9 shows the shape of the LM GUIDE.

The typical time histories of the above two experiments were shown in Fig.10. The measured pressures were compared with those of Chuang, as shown in Fig.11. The comparison indicated that the experimental results obtained from the proposed technique showed less deviation than those of Chuang.

5. Comparison between Numerical and Experimental Results

Fig. 12 shows the comparison between experimental results and numerical results. The results are compared at 10 and 20 degree deadrise angles. All pointed values represent the mean value of non-dimensionalized values.

In the case of the 10 degree deadrise angle, the calculation result of FLUENT with constant velocity is the highest value, while the result of the free fall test is the lowest value. In the case of the 20 degree deadrise angle, the calculation result of FLUENT with free fall is the highest value, while the result of the free fall test is the lowest value. The calculation results of BEM are always located at the middle of all values, and the results of the pneumatic cylinder test are higher than the results of the free fall test. Thus, the numerical results are generally higher than the experimental results.

The experimental and numerical results of the 10 degree deadrise angle are compared with the results of Chuang's drop test. The two points represent the maximum and minimum values of Chuang's results. Most of our results are located in the range of Chuang's test results.

6. Concluding Remarks

This study presents the results of two kinds of numerical computations and two kinds of experiments. One represents the potential code, and the other is represented by the viscous one in the numerical calculations. The potential code was written by BEM, which is the typical computational tool for the potential problem. The commercial code, FLUENT, was utilized to simulate the viscosity and free fall motion. The free fall test is the traditional method to determine the slamming impact, but we introduced the LM Guide to make the test easy and exact. The pneumatic cylinder test is a new and simple method proposed in our research.

The developed BEM code has shown to be in good agreement with Faltinsen's results. It seems that the developed BEM code complies with the slamming phenomenon. The required mesh generation and imposition of the boundary condition seem suitable to describe the slamming phenomenon. When it comes to the FLUENT code, the numerical values are higher than BEM and experimental results. One promising feature, derived from the FLUENT calculation, is that we can see the generation of jet flow and its corresponding free surface evolution. This can not be achieved with BEM, since it can not simulate the separated flow. FLUENT can simulate free fall motion by control of the velocity using UDF.

All experimental results are lower than numerical results; however, we have to consider the error of data acquisition. Nevertheless, these results are in good agreement with each other.

All experimental and numerical results show that the deadrise angle is the most important factor of slamming impact.

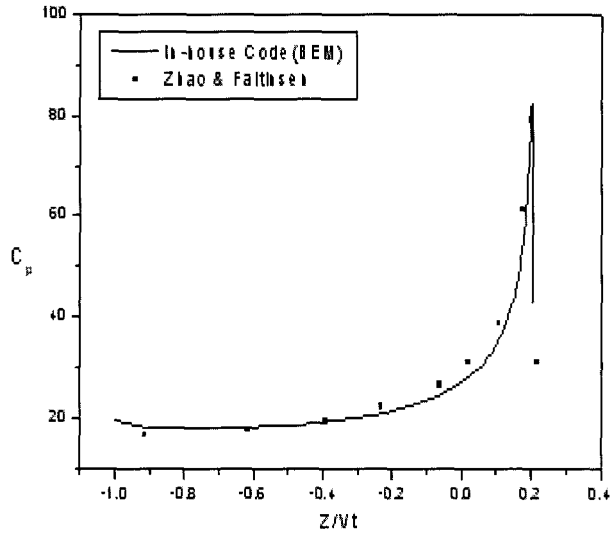


Fig. 1 Impact pressure distribution (deadrise Angle 10 °)

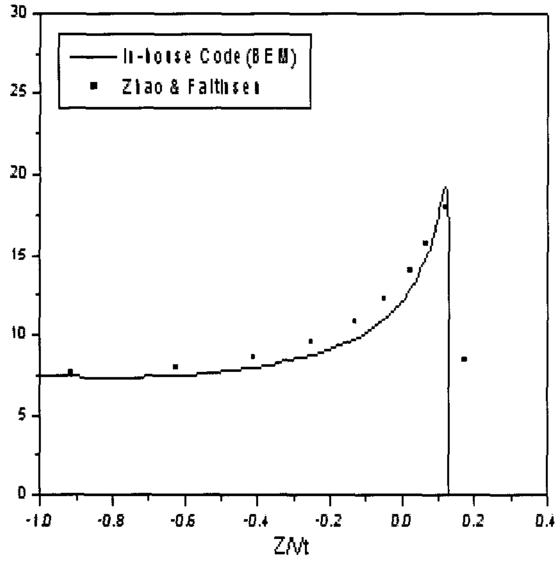


Fig. 2 Impact pressure distribution (deadrise angle 20 °)

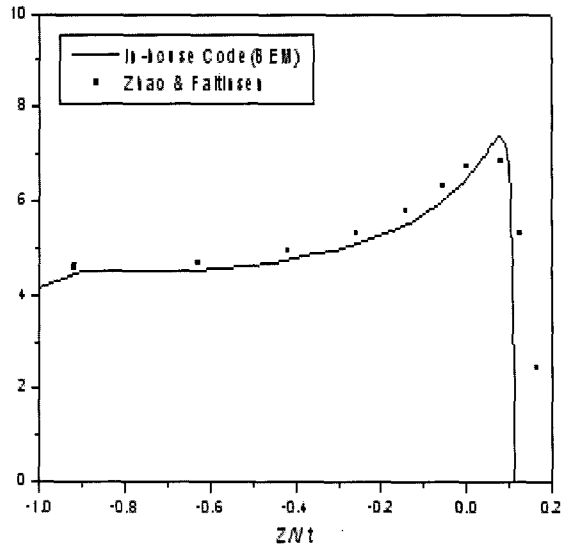


Fig. 3 Impact pressure distribution (deadrise angle 30 °)

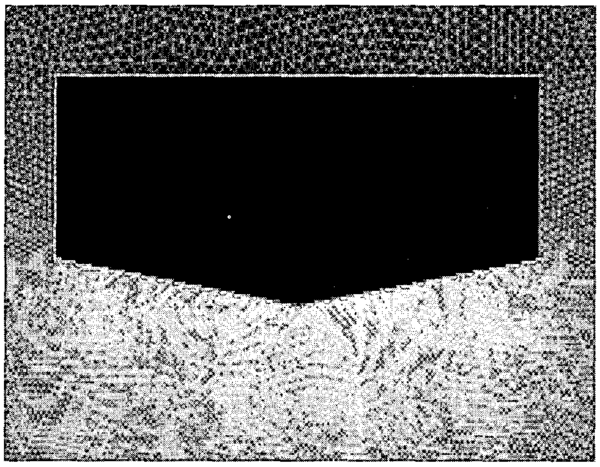
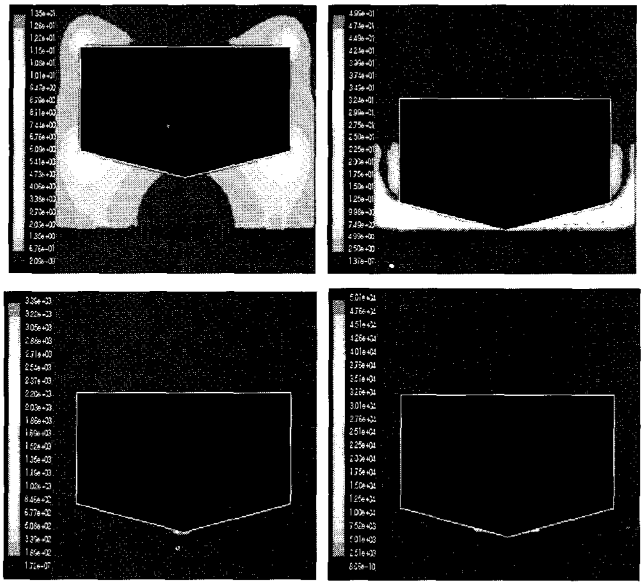


Fig. 4 Unstructured grid for wedge (deadrise angle 10 °)



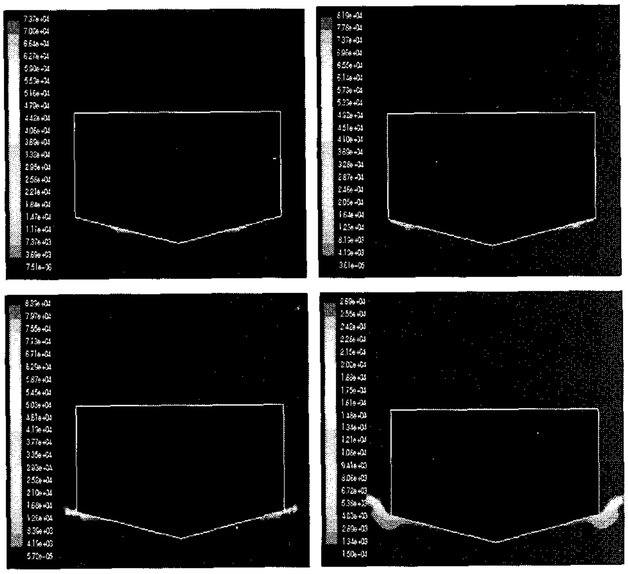


Fig.5. Pressure contour for wedge
(deadrise angle 10 °, V=1.0m/sec)

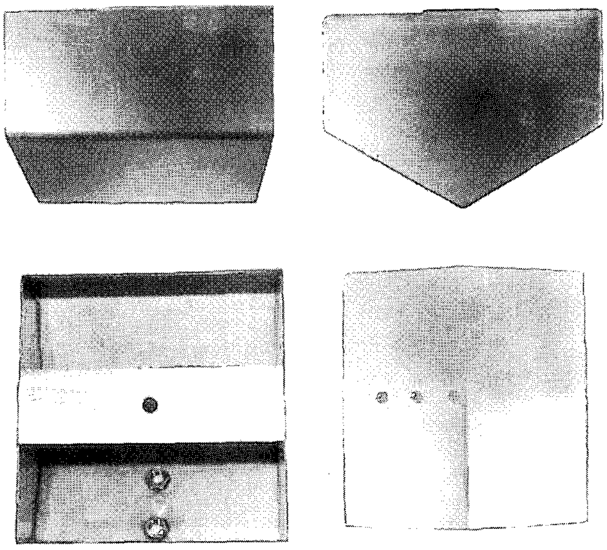


Fig. 7 Photos of specimen

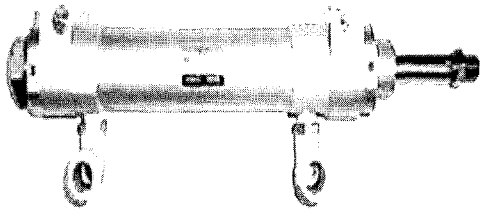


Fig. 8 Photo of pneumatic cylinder

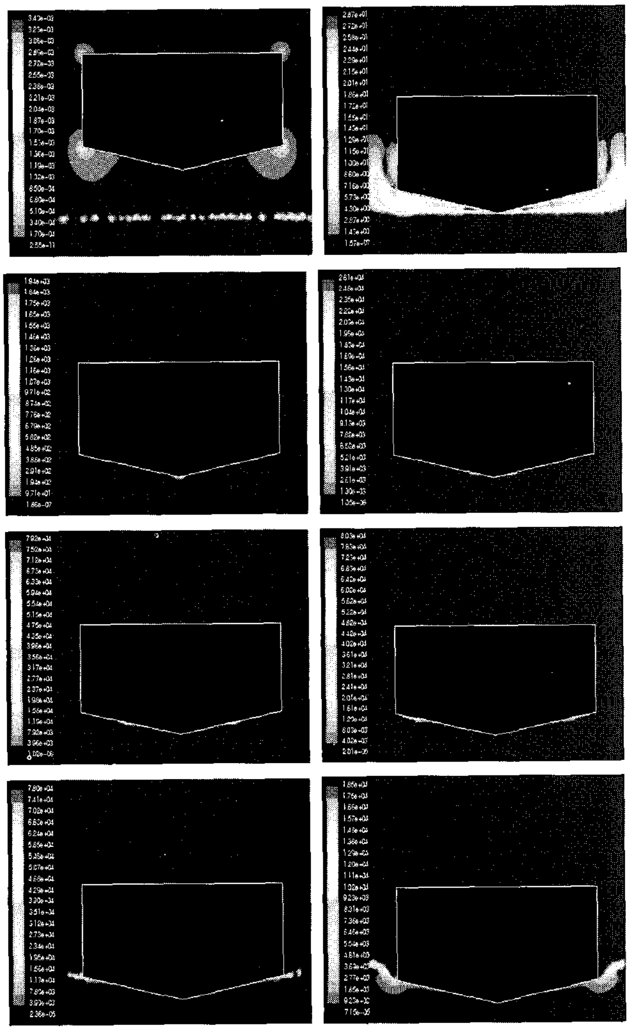


Fig. 6 Pressure contour for wedge
(deadrise angle 10 °, free fall)

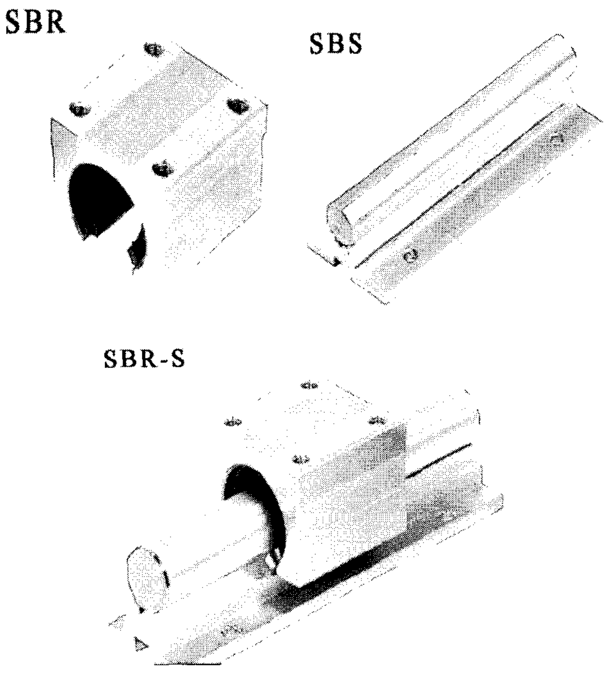


Fig. 9 Photos of LM-GUIDE

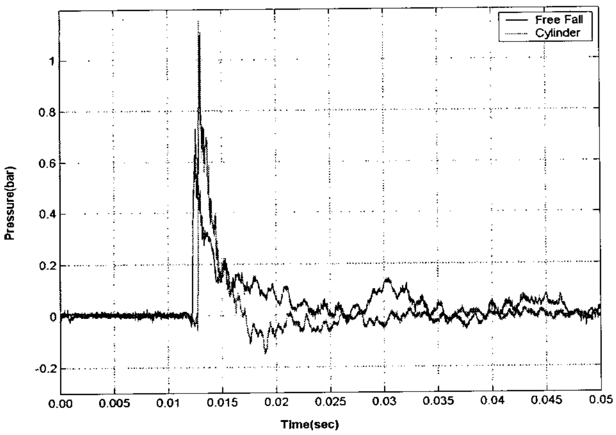


Fig.10. Comparison of pressures time history (point A, V=2.5m/sec, deadrise angle of 10 degree)

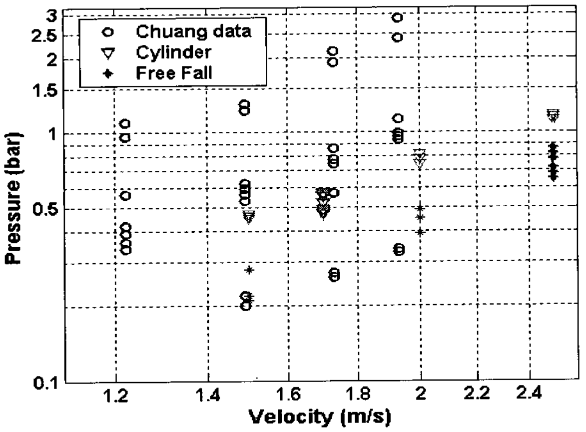


Fig. 11 Comparison with Chuang

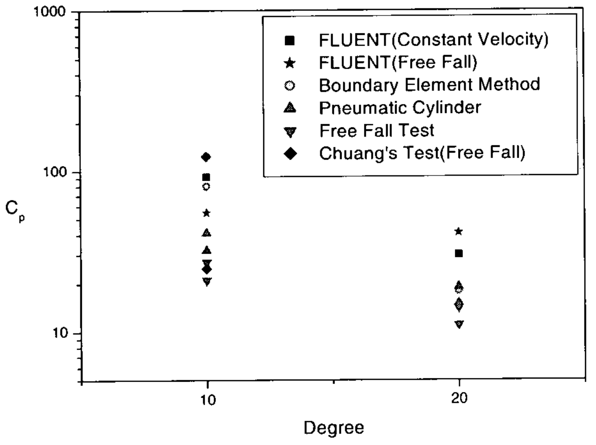


Fig. 12 Comparison with the results of wedge

ACKNOWLEDGMENT

The present work has been supported by the Hyundai Heavy Industries Co. Ltd. and the Advanced Ship Engineering Research Center of Pusan National University in Korea.

REFERENCES

Arai, M., Cheng, L.Y., and Inoe, Y. (1994). "A Computing Method for the Analysis of Water Impact of Arbitrary Shaped Bodies," J. Soc. Naval Arch. Japan, pp 176.

Armand, J.L. and Cointe, R. (1986). "Hydrodynamic Impact Analysis of a Cylinder," Proc. Fifth Int. Offshore Mech. And Arctic Engng. (OMAE). Tokyo, Japan, Vol 1, pp. 609-634.

Dobrovol'skaya, Z.N. (1969). "On Some Problems of Similarity Flow of Fluid with a Free Surface," J. Fluid Mech., Vol 36, No 4, pp 805-829.

Faltinsen, O.M. (1990). "Sea Loads on Ships and Offshore Structures," Cambridge University Press.

Garabedian, P.R. (1953). "Oblique Water Entry of a Wedge," Commun. Pure Appl. Maths 6, pp 157-165.

Greenhow, M. (1987). "Wedge Entry into Initially Calm Water," Appl. Ocean Res., Vol 9, pp 214-223.

Howison, S.D., Ockendon, J.R., and Wilson, S.K. (1991). "Incompressible Water-entry Problem at Small Deadrise," J. Fluid Mech., Vol 222, pp 215-230.

Korobkin, A. (1996). Water Impact Problems in Ship Hydrodynamics, Advances in Marine Hydrodynamics (Editor:M.Okushu), pp 323-371.

Mei, X., Liu, Y., andYue, D.K.P. (1999). "On the Water Impact of General Two-dimensional Sections," Applied Ocean Research, Vol 21, pp 1-15.

Watanabe, T. (1986). "Analytical Expression of Hydrodynamic Impact Pressure by Matched Asymtotic Expansion Technique," Trans. West-Japan Soc. Naval Arch., Vol 71, pp 77-85.

Zhao, R., and Faltinsen, O. (1993). "Water Entry of Two-dimensional Bodies," J. Fluid Mech. Vol 246, pp 593-612.

2005년 6월 25일 원고 접수
2005년 8월 16일 최종 수정본 채택