Water Exit of a Wedge-Shaped Body

Dominic J. Piro† and Kevin J. Maki

Department of Naval Architecture and Marine Engineering
University of Michigan, Ann Arbor, MI, 48109 USA
[djpiro, kjmaki]@umich.edu

† Presenting Author

Introduction

The problem of free-surface impact has been studied intensely for many decades for the purpose of understanding the performance of vehicles such as seaplanes, planing craft, space craft that land in the ocean, and ships in general. While impact is a very important problem, the subsequent exit of the body is also relevant for the performance of many of these vehicles. Indeed, the exit of a body from the free surface may impose greater loads on the structure than during the impact phase, and thus requires consideration [1, 2]. Furthermore, the extremely useful class of hydrodynamic methods based on the $2D$+t and strip theories are often applied to bodies in which the sections near the stern of the vessel appear to be exiting the water. Often it is assumed that the force generated during exit is negligible, and though under many circumstances this may be true, it is felt that a detailed study of the exit process will ultimately be useful in confirming this assumption.

In this work the constant velocity exit of a wedge-shaped body is studied using a fully nonlinear numerical method. The geometry is shown in Figure 1 and can be described by the deadrise angle β and beam B. Also, the initial distance from the keel to the free-surface is denoted d. The body is assumed to have infinite span. A single deadrise angle of 10 degrees is chosen because this value represents many marine craft. The chine of the section is far enough above the initially calm-free-surface plane such that it does not influence the solution. The numerical method requires discretization of the fluid domain, which is obviously finite, although it is large enough such that the solution is effectively that of an infinite-fluid domain. Particular attention is paid to the finite time over which the body accelerates to a content value of velocity, and to the influence of gravity. Results of the force on the body, and pressure distribution on the body are included in the abstract. Additional details such as the flow field and free-surface evolution will be shown in the presentation.

Methodology

The fluid flow is described by solutions to the Navier-Stokes equations. The numerical method finds approximate solutions to the Navier Stokes equations using a solver from the OpenFOAM CFD library. The method is based on the finite-volume discretization on arbitrary polyhedral discretization, although in this work all finite volumes are hexahedral (or quadrilateral areas in two dimensions). The Volume of Fluid (VOF) scheme is used to capture the position of the interface between the air and water phases. This method allows for very complex topology of the free surface such as that associated with wave breaking and air entrainment.

The Navier-Stokes equations for an incompressible fluid are solved in a non-interial body-fixed coordinate system (Equation 1). This procedure introduces a source-term to the governing equations for momentum to account for the acceleration of the coordinate system, and eliminates the need to move the fluid grid. In Equation 1, $\vec{u}$ is the velocity of the fluid relative to the body, and $\vec{u}_b$ is the velocity of the body relative to an inertial coordinate frame. The fluid pressure is denoted p, the gravity vector $\vec{g}$, and fluid density and molecular viscosity are ρ and μ.

$$\frac{\partial \rho \vec{u}}{\partial t} + \nabla \cdot \rho \vec{u}\vec{u} = -\nabla p + \rho \vec{g} + \mu \nabla^2 \vec{u} - \rho \frac{d\vec{u}_b}{dt} \tag{1}$$

The discretization of the governing equations is second-order in space and time.

The body boundary condition is no-slip, or $\vec{u} = 0$. The velocity is specified on the bottom boundary to be opposite the body velocity u_b, and the top boundary a zero-gradient condition is applied. The centerline is a symmetry boundary in all variables. The pressure is zero gradient everywhere except the top, where it is a fixed value of zero. The VOF variable which can be interpreted to represent the volume fraction of water in a cell is set to one at the bottom (signifying all water) and zero at the top, and zero-normal gradient on the body.

The governing equations can be parameterized using the Froude and Reynolds numbers defined with the initial-penetration depth d as the length scale, the constant-exit velocity U. A small amount of viscosity that would be similar to water at a model scale is used. The value of gravity is systematically varied from zero to a very large value to investigate the dependence on the Froude number.

Results

Validation with Experiments Results are first presented for the purpose of validation of the present numerical method by simulating the constant velocity exit of a wedge that was experimentally studied by Tveitnes et al. [3]. In their experimental set-up, the wedge is initially submerged with the chines wetted, which is different than the case that is of primary interest in which the chines are always dry.

The experiments presented by Tveitnes et al. [3] include a case of a 10 degree deadrise wedge where the Froude number is $\mathcal{F} = U/\sqrt{dg} = 0.12$, and the Reynolds number is $\mathcal{R} = Ud\rho/\mu = 730,000$ (by assuming common values for the material properties of water). As is shown in the later portion of this section, this case is dominated by hydrostatics. Experimental data are not available for a larger Froude number.

The present numerical method is used on three different numerical discretizations so that the discretization error, which can be large for this style of numerical method, may be studied. The three discretizations use 50, 100, and 200 elements along the body (denoted coarse, medium, and fine).

The vertical force time series for the experiment and the simulation on each grid is shown in Figure 2. Here, the force is made non-dimensional using the a nominal value of the hydrostatic force and the conventional hydrodynamic factor:

$$F' = \frac{F_y}{1/2\rho U^2 B + \rho g A_0} \tag{2}$$

where A_0 is the initial displaced area. In this figure time is made dimensionless using the time scale d/U. The force time series from the simulations compare well with the experimental force. Also, the medium and fine grids give similar time series which shows that either discretization is suitable for this problem.

Effect of Initial Acceleration of the Body The physical problem of a body that is exiting the water with constant velocity requires some specification on how it arrived to the final velocity. Since we wish to study the problem under the most simple terms using initially quiescent conditions, we must describe the acceleration of the body. Also, it is desired that this acceleration does not influence the solution during most of the time range of interest.

The body velocity is prescribed using the following expression:

$$\vec{u}_b(t) = \begin{cases} \frac{1}{2}U\left(1 - \cos\left(\frac{\pi}{t_r}t\right)\right)\hat{\jmath} & \text{for} \quad 0 < t < t_r \\ U\hat{\jmath} & \text{for} \quad t \geq t_r \end{cases} \tag{3}$$

where t_r is the relaxation time, or the time to reach constant velocity.

Figure 3 shows the force time series for five values of the relaxation time. The horizontal axis is time made dimensionless with t_r, and the force is the dimensionless force defined in Equation 2 multiplied by the factor $t_r U/d$. This additional factor scales the force by a measure of the acceleration. Five values of $t_r U/d$ from 0.0001 to 1.0 are shown. It can be seen that for values of $t_r U/d < 0.01$, that the results are no longer dependent on the relaxation time. For all subsequent results, the relaxation time parameter is chosen to be $t_r U/d = 0.01$.

Effect of Froude Number The effect of gravity on water exit of a wedge is studied by varying the value of the Froude number. The values of $\mathcal{F}$ used in this study are 0.01, 0.05, 0.1, 0.5, 1.0, 2.0, 5.0, and ∞.

Figure 4 shows non-dimensional vertical force for each Froude number. This non-dimensionalization allows for the force time-series from all cases to be plotted together. The three lowest Froude number cases match well at the bottom lines of the plot, where gravitational force dominates. The non-dimensional force increases with Froude number and appears to converge towards the infinite Froude number case, where fluid inertia dominates. The range of Froude numbers between 0.5 and 2 exhibit both behaviors.

The distribution of pressure coefficient $C_p = p/0.5\rho U^2$ is shown for five time instances (0.25, 0.50, 0.75, 1.00, and 1.25)$\times d/U$ at three Froude Numbers in Figure 5. The variable ξ is used to denote distance along the wedge from the keel such that $\xi \sin(\beta)/d = 1$ is the initial position of the free surface. Figure 5(a) shows that for $\mathcal{F} = 0.01$, the pressure is mostly hydrostatic. At the other extreme, Figure 5(c) shows the purely hydrodynamic pressure distribution for infinite $\mathcal{F}$. The pressure is nearly constant near the keel and curves down towards the intersection of the free surface and the wedge. Also evident is that the wedge is still wetted after the time when the calm free surface clears the keel. Figure 5(b) shows both hydrostatic and hydrodynamic effects for the intermediate value of $\mathcal{F} = 1.0$.

Conclusions

By comparing to experimental data, the simulation strategy is shown to work well in predicting the vertical force on a wedge exiting the water at constant velocity. The effects of the start-up of the simulations are minimized by testing several different values of acceleration relaxation time, and choosing the a small value that shows behavior is effectively instantaneous.

The effect of gravity is studied by simulating a wide range of Froude number. The pressure and force is dominantly hydrostatic for $\mathcal{F} < 0.1$. For $\mathcal{F} > 5$ gravity is no longer important and the force is purely hydrodynamic.

Acknowledgements

The authors would like to gratefully acknowledge the support of a grant from the US Office of Naval Research, Award # N00014-10-1-0301 and # N00014-11-1-0846, under the technical direction of Ms. Kelly Cooper.

Bibliography

[1] D. Dessi and R. Mariani. Slamming load analysis of a fast vessel in regular waves: a combined experimental/numerical approach. In *Twenty-Sixth Symposium on Naval Hydrodynamics*. Rome, Italy, 2006.

[2] D. J. Piro and K. J. Maki. Hydroelastic wedge entry and exit. In *Eleventh International Conference on Fast Sea Transport*. Honolulu, Hawaii, USA, 2011.

[3] T. Tveitnes, A. C. Fairlie-Clark, and K. Varyani. An experimental investigation into the constant velocity water entry of wedge-shaped sections. *Ocean Engineering*, 35(14-15):1463–1478, 2008.

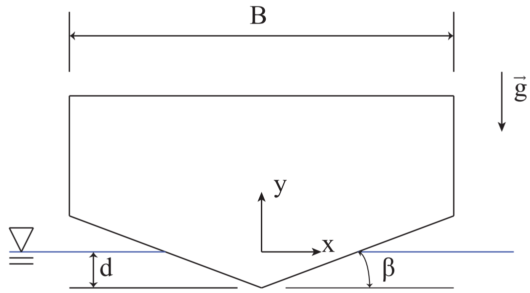

Figure 1: Geometry of wedge used in exit problem.

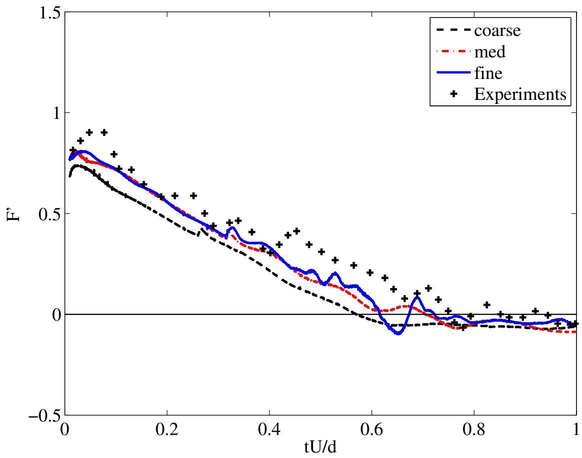

Figure 2: Comparison of Vertical Force Between Simulations and Experiments [3].

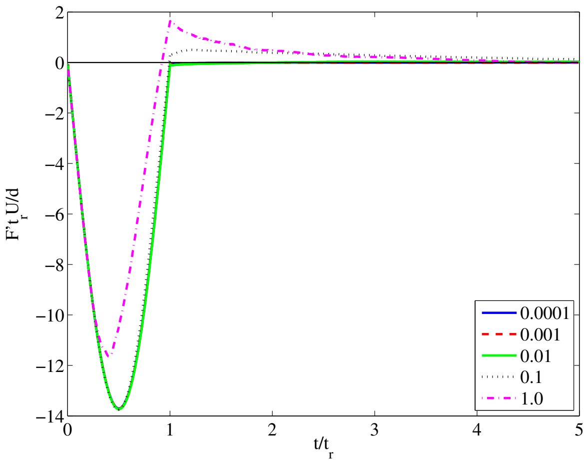

Figure 3: Non-dimensional force versus non-dimensional time for different values of t_r'.

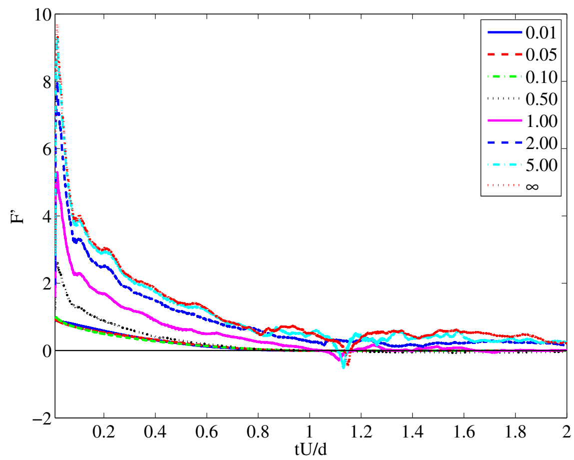

Figure 4: Non-dimensional force versus non-dimensional time for different Froude Numbers.

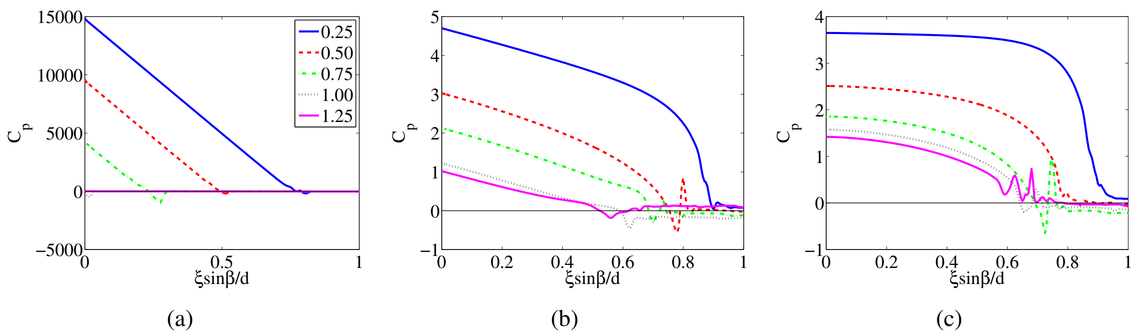

Figure 5: Pressure coefficient distribution for Fr = 0.01 (a), Fr = 1.00 (b), and Fr = ∞ (c)