ACCURATE DESCRIPTION OF THE JET FLOW DEVELOPING DURING WATER IMPACT

A. Iafrati and **D. Battistin**, INSEAN (Italian Ship Model Basin),
E-mail: a.iafrati@insean.it, d.battistin@insean.it

SUMMARY
The jet flow originated during the water impact of rigid bodies is investigated. In the framework of a potential flow assumption, a numerical model is developed which allows an efficient and accurate description of the unsteady flow taking place within the thin jet layer. To this aim, the modeled part of the jet is discretized with small control volumes and, inside each one, the velocity potential is approximated by local harmonic expansions, coefficients of which are recovered by enforcing boundary conditions on the body and on the free surface. An additional matching condition between contiguous volumes is also applied to achieve regularity. The model is incorporated within a boundary element solver, which is used to determine the solution in the remaining part of the fluid domain. For validation, the model is used to simulate the flow generated by the wedge impact with constant entry velocity. A rather good agreement with the similarity solution is achieved both in terms of free surface shape and pressure distribution.

1. INTRODUCTION

Water impact is rather challenging due to the peculiar features of the resulting flow, such as the development of thin jet layers, the highly localized pressure peaks, the possible occurrence of flow detachment from the body contour. In this framework, numerical models based on potential flow assumptions have been found efficient and accurate but the description of the flow close to the thin jet, developing about the intersections between the body contour and the free surface, is not straightforward. First attempts tried to follow all the details of the free surface motion in the thin jet region but results have been found dependent on the assumptions made at the jet tip [4, 3]. However, when attention is mainly related to the hydrodynamic loads originated during water entry, owing to the low pressure taking place in the jet region, a very refined description of the flow is not needed. On the basis of this consideration, recently, numerical models that cut off the jet from the computational domain have been introduced [5, 2, 1]. These models have been found rather stable and reliable for a wide range of deadrise angles and have been also applied with success to the impact of arbitrary shaped bodies. On the other hand, a major drawback of these models is the complete lack of information about the jet, which could separate or not from the body contour, therefore producing very different distributions of pressure. This is crucial when the position of the separation point is not known a priori, as it is for smooth convex contours like a circular cylinder, for instance.

In the present work the fully nonlinear boundary element method, previously developed by the authors [1], is extended to provide an improved description of the flow in the jet region. To this aim, the jet region is divided into small control volumes and, within each one, local Taylor expansions of the velocity potential are used, coefficient of which are recovered by enforcing the boundary conditions and suitable matching relations among control volumes. In this way the use of a boundary integral representation in a very thin region is avoided, thus allowing a stable and accurate description of the flow. It is expected that, once the flow inside the thin jet layer is correctly described, the prediction of separation phenomena becomes feasible.

In the following, the numerical model is described and some results obtained for the impact of wedge shaped bodies are shown as a purpose of validation.

2. NUMERICAL MODEL

The numerical simulation of the water impact problem is faced in the framework of inviscid and incompressible fluid in irrotational flow. Moreover, surface tension and gravity effects are also neglected. Let y the vertical axis oriented upward, and x the horizontal axis lying on the undisturbed water plane, the vertical impact of a symmetric two-dimensional body with an entry velocity $\boldsymbol{w} = (0, -V)$ is governed by the following equations:

$$
\begin{array}{rcll}
\Delta\varphi & = & 0 & \Omega \\
\varphi_n & = & -V n_y & S_B \\
\dfrac{D\varphi}{Dt} & = & \dfrac{|\nabla\varphi|^2}{2} & S_S \\
\dfrac{D\boldsymbol{x}}{Dt} & = & \boldsymbol{u} & S_S
\end{array}
\qquad (1)
$$

where the symbols are defined as in Fig.1. The pressure is related to the velocity potential through the unsteady Bernoulli equation:

$$p = -\rho\left(\varphi_t + \frac{|\nabla\varphi|^2}{2}\right) \qquad (2)$$

This initial-value problem is solved through a mixed Eulerian-Lagrangian approach which makes use of a boundary integral formulation to solve the boundary value problem for the velocity potential at each time step. To this aim, an integral equation is solved, providing the unknown value of the potential on the body surface and that of its normal derivative on the free surface:

$$\frac{\varphi}{2} = -\int_{\partial\Omega} (\varphi G_n - \varphi_n G)\, dS \qquad (3)$$

$G = \frac{1}{2\pi} \ln |P - Q|$ being the free space Green function for the Laplace operator.

In discrete form, the boundary of the fluid domain is represented by straight line panels and a piecewise distribution of the velocity potential and of its normal derivative is used along them. By integrating in time the last two equations in (1) for the panel centroids, the free surface position and the distribution of the velocity potential along it are updated. Once the position of the free surface centroids is updated, a cubic spline is passed through them to reconstruct the distribution of panel vertices. Distribution of panels is reconstructed at each step in order to guarantee a good accuracy in thin and highly curved region.

In order to determine the pressure distribution on the body, the time derivative of the potential φ_t is calculated at each time step, after the velocity potential and its normal derivative are updated. The approach is exactly the same as for the potential itself, but for the boundary conditions which are

$$\varphi_t = -|\nabla\varphi|^2/2$$

on the free surface and

$$\varphi_{tn} = \boldsymbol{n} \cdot \boldsymbol{a} - w_\tau (u_n)_\tau + w_n (u_\tau)_\tau - k_\tau \boldsymbol{w} \cdot \boldsymbol{u} \qquad (4)$$

on the body contour [1]. In equation (4) $\boldsymbol{\tau}$ is the tangent along the body contour, k_τ is the local curvature, $\boldsymbol{w}$ and $\boldsymbol{u}$ denote the local velocity of the body and of the fluid particles at the same position and, finally, $\boldsymbol{a}$ is the local acceleration of the body.

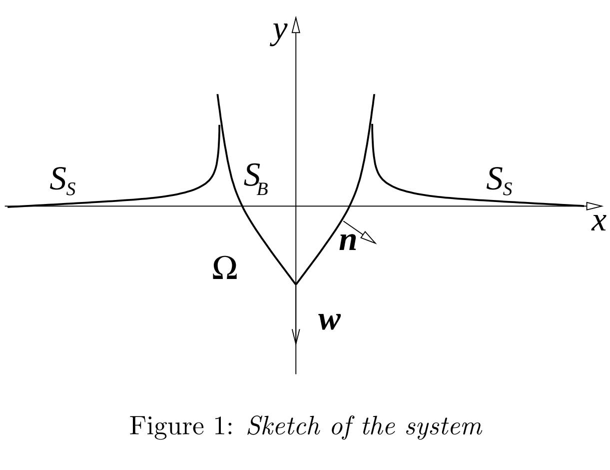

Figure 1: *Sketch of the system*

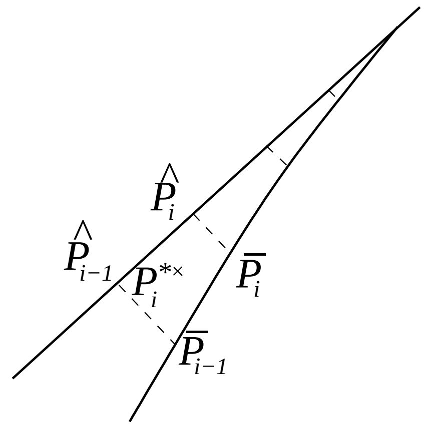

Figure 2: *Sketch of the jet region*

The use of boundary integral approaches to describe the flow inside thin jet layers is not convenient. Actually, to get a good accuracy, the panel size must be smaller than the local thickness of the jet and, moreover, since a small angle usually occurs at the intersection, a region always remains near the tip which is not properly discretized. To overcome this difficulty a jet model is developed and coupled with the boundary integral approach, which continues to be used in the bulk of the fluid domain. The idea is to decompose the jet region into small control volumes and to use local Taylor expansion to approximate the velocity potential. The coefficient of the expansions are assigned by enforcing the body and free surface boundary conditions along with some matching constraint introduced to link the expansions among contiguous elements.

To explain how the model works, let consider a control volume V_i in the jet region, bounded by four vertices (which coincide with panel centroids) $\hat{P}_i$, $\hat{P}_{i-1}$, $\bar{P}_i$ and $\bar{P}_{i-1}$ (see Fig.2). The velocity potential inside V_i is approximated as an harmonic expansion

$$\varphi_i^J(x, y) \simeq A_i + B_i(x - x_i^*) + C_i(y - y_i^*) + \qquad (5)$$
$$\frac{D_i}{2}\left[(x - x_i^*)^2 - (y - y_i^*)^2\right] + E_i(x - x_i^*)(y - y_i^*)$$

where (x_i^*, y_i^*) are the coordinates of the volume centroid P_i^*. The coefficients are recovered enforcing the boundary conditions:

$$\begin{aligned} \varphi_{in}^J(P) &= w_n(P), \qquad & P = \hat{P}_i, \hat{P}_{i-1} \\ \varphi_i^J(P) &= \varphi(P), \qquad & P = \bar{P}_i, \bar{P}_{i-1} \end{aligned} \qquad (6)$$

plus a fifth condition that enforces continuity with the control volume $V_{(i-1)}$ either in terms of the potential along the body contour or in terms of its normal derivative along the free surface:

$$\varphi_{in}^J(\bar{P}_{(i-1)}) = \varphi_{(i-1)n}^J(\bar{P}_{(i-1)}) \ , \qquad (7)$$

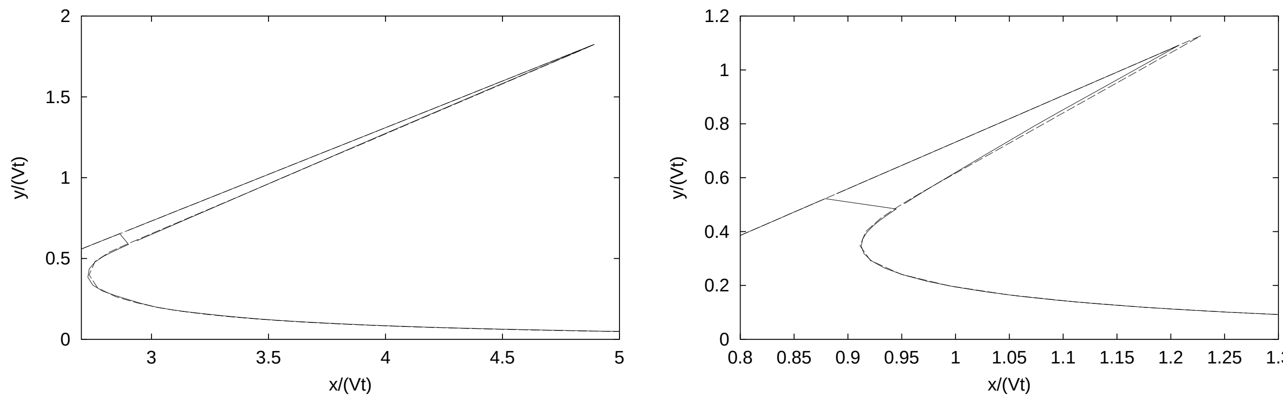

Figure 3: *Comparisons between the free surface configuration obtained by the proposed model* (solid line) *and by the similarity solution* (dash line). *The deadrise angle of the impacting wedge is* $30°$ *(left) and* $60°$ *(right). The transverse segment denotes the boundary between the bulk domain and the modeled part of the jet (intermediate region included).*

$$\varphi_i^J(\hat{P}_{(i-1)}) = \varphi_{(i-1)}^J(\hat{P}_{(i-1)}) \ . \tag{8}$$

Tests done by using the two different matching conditions have shown that results are essentially independent of the choice.

Such model for the jet region is coupled with the classical boundary integral representation on the rest of the fluid domain. The discretized version of equation (3) is collocated on the panel centroids belonging to the boundary of the bulk region; when integrating along panels lying in the jet region, that is along the faces of control volume which lie on the free surface and on the body contour, the definition (5) is used. Together with these equations, those coming from the boundary conditions (6) and from the matching condition (7) are solved. The unknowns of the resulting linear system are: the value of the potential along the body contour in the bulk region, the value of its normal derivative along the free surface in the bulk region and the value of the coefficients $(A_i, \ldots, E_i)$ for each of the control volumes in the jet region. To make smoother the transition between the bulk of the domain and the jet region, an intermediate region is interposed, where a weighted average of the potential is used:

$$\varphi(P) = (1 - l_i)\varphi^B(P) + l_i \varphi_i^J(P) \ \ , P \in V_i \tag{9}$$

φ^B denoting the velocity potential given by the boundary integral representation. This intermediate region cover a limited number of control volumes, with the weight function l_i linearly varying from zero, at the matching with the bulk domain, to one, at the matching with the fully modeled zone. A similar average is applied to the matching conditions. Extending the definition of the weight function l, by putting $l = 0$ in the bulk region and $l = 1$ in the jet region, the integral equation, after substitution of (5) and (9), reads

$$\frac{1}{2}\left[(1-l)\varphi^B + l\varphi^J\right] = -\int_{\partial\Omega}\left\{\left[(1-l)\varphi^B + l\varphi^J\right] G_n - \left[(1-l)\varphi_n^B + l\varphi_n^J\right] G\right\} dS \tag{10}$$

Concerning the calculation of the pressure on the body, identical expansions are used for the time derivative of the potential φ_t, and a similar linear system is assembled and solved. The solution provides the unknown field φ_t along the body, which allows to evaluate the pressure by means of the unsteady Bernoulli equation (2).

The separation of the fluid domain into the three regions, bulk, intermediate and jet is made through the following steps: first, starting from the tip, the angle between the free surface and the body is monitored, and the region where this angle is smaller than a (large) fixed value, say $30°$, is selected. The extension of this region f_J, defined as the curvilinear abscissa along the body from the tip, is calculated. Then, the modeled part of the jet region is assumed to be a fraction of this zone, usually $f_M = 0.5 f_J \div 0.8 f_J$. Correspondingly, the extension of the intermediate region is assumed to be $f_I = 0.2 f_J \div 0.5 f_J$.

3. VALIDATION OF THE PROPOSED MODEL

In order to validate the proposed model, the flow generated by the wedge impact with constant entry velocity is simulated and comparisons are established with the self-similar solution obtained by solving the boundary value problem written in a suitable set of non-dimensional variables. Comparisons are established in terms of the free surface shape and pressure distribu-

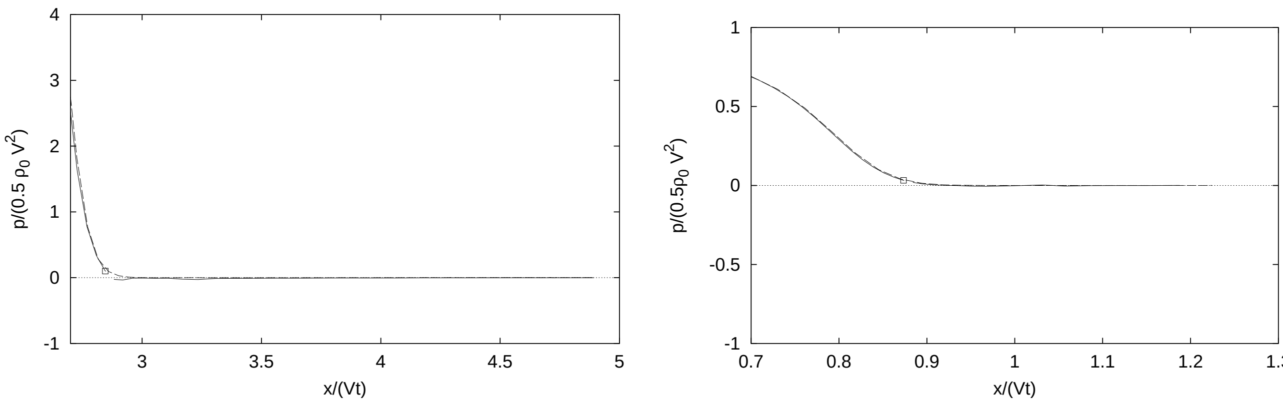

Figure 4: *Close up view of the pressure distributions in the jet region obtained by the proposed model* (solid line) *and by the similarity solution* (dash line)*. The deadrise angle of the impacting wedge is* 30° *(left) and* 60° *(right). The square symbol along the curves represents the boundary between the bulk domain and the modeled part of the jet (intermediate region included).*

tion along the body surface for two different values of the deadrise angle of the wedge, 30° and 60°.

At the beginning of the simulation the wedge is slightly immersed in an elsewhere undisturbed free surface, and the jet model is still not activated. The free surface, during the first time steps, raises on the body and develops the jet layer along it; at that moment the jet model is activated, and the simulation continues until the self similar solution is reached.

In the numerical simulation typical values used for f_M and f_I are 0.8 and 0.35, respectively. The panel distribution is built by choosing the smallest panel amplitude at the matching point between the modeled part and the bulk of the fluid. The smallest panel size is usually assumed to be one third of the local jet thickness. Hence, the panel size is progressively increased moving along the free surface and the body. A similar procedure is followed to choose the width of the control volumes in the modeled part of the jet.

In Fig.3 the non-dimensional free surface shape is shown. The agreement with the similarity solution is rather satisfactory, although the proposed model predicts a smaller wetted area for the 60° case. A very good agreement is found at the jet root where, also, a smooth transition from the modeled part of the jet to the bulk of the fluid is obtained.

In Fig.4 the comparison is presented in terms of non-dimensional pressure distribution. Attention being focused at the details of the solution in the jet region, only a close up view about the modeled part is shown, where pressure is very low. In the 60° case the agreement is very good, while for the 30° case a small disagreement occurs just at the first few elements of the modeled part of the jet.

References

[1] D. Battistin & A. Iafrati 'Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies.', *accepted for publication on the Journal of Fluid and Structures*, 2002.

[2] E. Fontaine & R. Cointe 'Asymptotic theories of incompressible water entry.' *Proceedings of AGARD FDP Workshop on High Speed Body Motion in Water*, Kiev, Ukraine, 1997.

[3] M. Greenhow 'Wedge entry into initially calm water.' *Applied Ocean Research*, **9**, 214-223, 1987.

[4] B. Yim 'Numerical solution of two-dimensional wedge slamming with a nonlinear free-surface condition.' *Proceedings of the* 4^{th} *International Conference on Numerical Ship Hydrodynamics*, Washington (DC), USA, 1985.

[5] R. Zhao & O.M. Faltinsen 'Water entry of two-dimensional bodies.' *Journal of Fluid Mechanics*, **246**, 593-612, 1993.