Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science

Simulation of free-surface flow in a tank using the Navier-Stokes model and unstructured finite volume method

X Zhang, N M Sudharsan, R Ajaykumar and K Kumar Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science 2005 219: 251 DOI: 10.1243/095440605X8496

> The online version of this article can be found at: http://pic.sagepub.com/content/219/3/251

> > Published by: **SAGE** http://www.sagepublications.com

> > > On behalf of:



Institution of Mechanical Engineers

Additional services and information for Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science can be found at:

Email Alerts: http://pic.sagepub.com/cgi/alerts

Subscriptions: http://pic.sagepub.com/subscriptions

Reprints: http://www.sagepub.com/journalsReprints.nav

Permissions: http://www.sagepub.com/journalsPermissions.nav

Citations: http://pic.sagepub.com/content/219/3/251.refs.html

>> Version of Record - Mar 1, 2005

What is This?

Simulation of free-surface flow in a tank using the Navier–Stokes model and unstructured finite volume method

X Zhang^{1,2*}, N M Sudharsan^{1,3}, R Ajaykumar¹, and K Kumar¹

¹Computational Fluid Dynamics Division, Institute of High Performance Computing, Singapore Science Park II, Singapore

²LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing, People's Republic of China

³Department of Mechanical Engineering, College of Engineering, Anna University, Guindy, Madras, India

The manuscript was received on 2 September 2003 and was accepted after revision for publication on 29 October 2004.

DOI: 10.1243/095440605X8496

Abstract: Modelling free-surface flow has very important applications in many engineering areas such as oil transportation and offshore structures. Current research focuses on the modelling of free surface flow in a tank by solving the Navier–Stokes equation. An unstructured finite volume method is used to discretize the governing equations. The free surface is tracked by dynamically adapting the mesh and making it always surface conforming. A mesh-smoothing scheme based on the spring analogy is also implemented to ensure mesh quality throughout the computation. Studies are performed on the sloshing response of a liquid in an elastic container subjected to various excitation frequencies. Further investigations are also carried out on the critical frequency that leads to large deformation of the tank walls. Another numerical simulation involves the free-surface flow past as submerged obstacle placed in the tank to show the flow separation and vortices. All these cases demonstrate the capability of this numerical method in modelling complicated practical problems.

Keywords: finite volume, finite element, free surface, fluid-structure interaction

1 INTRODUCTION

The modelling of incompressible free-surface flow has large-scale commercial applications such as the design of tanks for oil transportation and offshore structures. The difficulty of this type of problems lies in the boundary condition. Not only does the free surface form a part of the boundary for the computation domain but also its shape and position are coupled with the solution to the fluid system. Further complexity may also be added if structural interaction is also considered. The structural response alters the fluid boundary, which in turn causes a change in the flow field and force exerted on the structure. This strongly coupled nature makes the problem very challenging. Solution to such a problem with the ideal fluid assumption (potential

*Corresponding author: LNM, Institute of Mechanics, Chinese Academy of Sciences, Beijing 100080, People's Republic of China. flow) can be achieved with less computational effort. However in some applications, viscous effect needs to be accounted for, e.g. the drag force exerted on a ship, boundary layer separation, and recirculating flows. Thus the solution of a full Navier-Stokes equation with a free surface is sometimes necessary although it is computationally very expensive. In this paper, the Navier-Stokes equation is solved using the unstructured finite volume method (FVM) with an arbitrary Lagrangian-Eulerian formulation. The mesh adapts dynamically to the moving boundary and forces the cell faces always to coincide with the free surface. The elastic walls of the tank are simplified as one-dimensional beam elements and their responses are computed using a finite element (FE) formulation.

In the design of an engineering system, it is necessary to model the system as close to reality as possible, enforcing the correct input parameters and boundary conditions. This would ensure a safe and economical design. For example, in the design of a tank carrying liquid, it is necessary to compute the dynamic forces exerted by the liquid on the tank. The non-linear free surface modelling has to be accurate as this alters the pressure head, which in turn affects the structural response. The solution becomes more involved in a viscous fluid when the assumption of an ideal fluid with irrotational flow is no longer valid. This requires the solution of a full Navier-Stokes equation. As a first step, the full Navier-stokes equation is solved with 'zero viscosity' to represent an ideal fluid and results are compared with potential flow solutions. Then viscosity is brought in to simulate a 'real' fluid. In this work, three numerical studies involving free surface are performed: (a) the dynamic response of a liquid in a container subjected to agitation; (b) the behaviour of the container side walls, modelled as elastic beams; (c) the simulation of free-surface flow past a rectangular obstacle in a tank.

In all the aforementioned simulations, the velocity and pressure fields are obtained by solving the Navier-Stokes equation. For the fluid-structure interaction (FSI) problem, this pressure is integrated over the structural area and the resulting force is applied to calculate the structural response. The Navier-Stokes equation is solved using a discretization based on finite volume (FV) linear elastic behaviour. In the case of a tank the structure is modelled as a beam element assuming it to be a plate extending in the z-direction. For the wavestructure interaction the structure is modelled as a plane strain element. The structural response is obtained by discretizing the structural domain into PEs and solving using the standard Galerkin FE formulation. As the response is dynamic in nature, the Newmark method is used for time integration. The time steps for the fluid and structural solvers are different, but the transfer of boundary conditions occurs at exactly the same time instant to ensure that the solutions progress by the same time interval.

A brief review of the literature in this research area will be given in section 2. Section 3 focuses on the numerical model and solutions including the Navier–Stokes equation, the static equilibrium equation of an elastic beam, and the spring analogy smoothing algorithm. Numerical results are presented in section 4. Finally, conclusions appear in section 5.

2 LITERATURE REVIEW

The literature regarding the representation of the free-surface profile is summarized first. There are two main methods of capturing the free-surface profile, namely interface capturing [1, 2] and interface

tracking [3, 4]. In the interface capturing method, the solution domain includes the region occupied by both fluids (in this case, air and liquid). The free-surface position at various time intervals is obtained by solving the volume fraction of one fluid. Some other passive scalars with similar physical meaning are also used in some approaches, e.g. the level set function in the level set method by Osher and Sethian [5]. In the interface tracking method, only one fluid is solved and the mesh moves and adapts itself to the free-surface profile. However, this method is not effective when the topology change is abrupt with even breaking waves or overturning waves. In this work the interface tracking methodology is followed. The method explained by Muzaferija and Peric [6] is adopted with a slight variation.

FSI problems are, in general, solved using two types of formulation. They are the monolithic formulation and the sequential formulation. In a monolithic formulation, the fluid and the structural equations are combined and the resulting equation is solved by an iterative method. For complex problems, the monolithic coupled approach is computationally involved and expensive both mathematically and economically. Hence, an alternative approach called the sequential coupled field formulation is used. In this method, the interaction between the fluid and structural codes is limited to the exchange of surface loads and surface deformation information. The relevant boundary conditions are updated for subsequent computation. A review of the various literature on FSI has been presented by Sudharsan *et al.* [7].

In the present work the analysis of FSI in a rectangular tank and the structural response due wave-structure interaction are considered. Although there is abundant literature involving cylindrical tanks, there is limited literature involving rectangular tanks. Housner [8, 9] and Haroun [10] studied the dynamic response of rectangular fluid container, which, however, did not fully take into account the flexibility of the container. Kim et al. [11] performed similar analysis by incorporating the modal response of the structure and found that the pressure distribution is amplified due to the elastic response of the structure. However, the effect of sloshing motion was not completely considered. Koh et al. [12] performed an FSI analysis, using the boundary element method (BEM)-finite element method (FEM), on a rectangular tank subjected to a seismic response. However, the effect of free-surface sloshing was taken into account using the linear free-surface boundary condition only. Ortiz and Barhorst [13] modelled the FSI of a rigid tank and a rigid tank coupled to either a flexible or a rigid body. They considered the fluid to be a viscous incompressible fluid

and as a potential flow with modified Rayleigh damping. The flexibility of the container was not considered. Pal et al. [14] performed an FSI analysis using a sequential coupled approach, assuming the structure to be a composite and the sloshing response to be linear. Very recently, Pal et al. [15] incorporated non-linear slosh dynamics, but the interaction results were provided for a cylindrical container using a two-dimensional FE approach. Bermudez et al. [16] performed FE computations of sloshing modes in a container with an elastic baffle using linear velocity potential formulation in the frequency domain. Thus, based on the above review, to the present authors' best knowledge, it was observed that the transient dynamic response of a rectangular elastic tank incorporating fully non-linear free surface conditions with a viscous fluid using the full Navier-Stokes equation has not yet been attempted. Thus the dynamic response of a rectangular elastic container with a viscous fluid subjected to external forces in analysed. The free surface flow of a viscous fluid past a two-dimensional bluff body is also analysed and presented.

3 MATHEMATICAL MODEL AND NUMERICAL SOLUTION

3.1 Navier-Stokes equations

The motion of an incompressible fluid is governed by the following conservation laws written in an integral form. The continuity equation is

$$\int_{\partial S} (\boldsymbol{v} - \boldsymbol{v}_{s}) \cdot \boldsymbol{n} \, \mathrm{d}S = 0 \tag{1}$$

The momentum equation is

$$\frac{\partial}{\partial t} \int_{V} \rho \mathbf{v} \, \mathrm{d}V + \int_{\partial S} \rho \mathbf{v} (\mathbf{v} - \mathbf{v}_{s}) \cdot \mathbf{n} \, \mathrm{d}S$$
$$= \int_{\partial S} \mathbf{T} \cdot \mathbf{n} \, \mathrm{d}S + \int_{V} f_{\mathrm{b}} \, \mathrm{d}V$$
(2)

The space conservation law is

$$\frac{\partial}{\partial t} \int_{V} \mathrm{d}V - \int_{\partial S} \mathbf{v}_{\mathrm{s}} \cdot n \, \mathrm{d}S = 0 \tag{3}$$

where *V* is an arbitrary volume occupied by the fluid, ∂S is the boundary which surrounds the fluid volume, ρ is the density (which is a constant for an incompressible fluid); $\boldsymbol{\nu}$ and $\boldsymbol{\nu}_{s}$ are the velocities of the fluid and control surface respectively, *T* is the stress tensor $f_{\rm b}$ is the body force, and *n* is the unit outward-normal vector on the integral surface d*S*.

The stress in the fluid is related to the rate of deformation by the Stokes law

$$T = pI + \mu [\nabla v + (\nabla v)^{\mathrm{T}}]$$
(4)

where *p* is the static pressure and μ is the dynamic viscosity. Substituting equation (4) in equation (2), the momentum equation can be written as

$$\frac{\partial}{\partial t} \int_{V} \rho \mathbf{v} \, \mathrm{d}V + \int_{\partial S} \rho \mathbf{v} (\mathbf{v} - \mathbf{v}_{\mathrm{s}}) \cdot \mathbf{n} \, \mathrm{d}S$$
$$= -\int_{V} \nabla p \, \mathrm{d}V + \int_{\partial S} \mu (\nabla \mathbf{v}) \cdot \mathbf{n} \, \mathrm{d}S + \int_{V} \mathbf{f}_{\mathrm{b}} \, \mathrm{d}V \qquad (5)$$

Note that Gaussian theorem is used to convert the surface integral into a volume integral.

If gravity is the only body force, this term could be incorporated into the pressure gradient term by introducing a potential function *H* according to

$$f_{\rm b} = -\nabla H$$

$$H = \rho g y \tag{6}$$

Here, y is the vertical coordinate and g is the gravitational acceleration. Thus equation (5) can be rearranged as

$$\frac{\partial}{\partial t} \int_{V} \mathbf{v} \, \mathrm{d}V + \int_{\partial S} \mathbf{v} (\mathbf{v} - \mathbf{v}_{\mathrm{s}}) \cdot \mathbf{n} \, \mathrm{d}S$$
$$= -\int_{V} \nabla \left(\frac{p^{*}}{\rho}\right) \mathrm{d}V + \int_{\partial S} \mathbf{v} (\nabla \mathbf{v}) \cdot \mathbf{n} \, \mathrm{d}S \tag{7}$$

where $p^* = p + H$ which is called the *piezometric* pressure in some references.

In free-surface flows, usually the dominating terms are the pressure gradient term and the gravity term. It is their difference that drives the flow. This could lead to large round-off errors if the original formulation of equation (5) is used in the discretization. This problem could be solved by using the reformulated form [i.e., equation (7)], as proved to be true in the present numerical experiments. A similar issue has also been addressed in references [**3**] and [**17**].

3.2 Numerical schemes for the fluid

The FVM is used to discretize the governing equations. First, the solution domain is divided into small control volumes (CVs) or cells. All unknowns are stored at the centre of these cells. In this work, only triangular cells are used in the discretization. Linear distribution of dependent variables and midpoint rules are used to calculate the surface and volume integrals. An algebraic linear system is obtained as a result of such discretization on all the CVs. The pressure is calculated using the SIMPLE algorithm. Since the collocated storage is adopted for all dependent variables, a Rhie–Chow interpolation is used to avoid the possible wiggles in the pressure field. All the resulting linear systems are solved using Krylov-type iterative solvers. The numerical procedure is only summarized briefly here; for full details of this numerical scheme, readers should refer to the Ferziger and by Peric [18] or the paper by Demirdzi and Muzaferija [19]. The discretization of the momentum equation (7) is explained in the form of a generic transport equation of a variable ϕ according to

$$\frac{\partial}{\partial t} \int_{V} \phi \, \mathrm{d}V + \int_{S} [\phi(\mathbf{v} - \mathbf{v}_{\mathrm{s}}) - \mathbf{\Gamma}_{\phi} \nabla \phi] \cdot \mathbf{n} \, \mathrm{d}S$$
rate of change
$$= \int_{S} Q_{\phi} \, \mathrm{d}S \tag{8}$$

where ϕ represents the velocity components v_i (i = 1, 2), Γ_{ϕ} is the diffusive coefficient (1/*Re*) and the source term is $Q_{\phi} = -(p^*/\rho)n_i$.

The rate of change is discretized by a backward Euler (fully implicit) scheme

$$\frac{\partial}{\partial t} \int_{V} \phi \, \mathrm{d}V \approx \frac{1}{\Delta t} [(\phi V)^{n} - (\phi V)^{n-1}] \tag{9}$$

where *n* and n-1 denote the time step counter.

The convective term is discretized as

$$\int_{S} \phi(\mathbf{v} - \mathbf{v}_{\rm s}) \cdot \mathbf{n} \, \mathrm{d}S \approx \dot{n}_{j} \phi_{j}^{\prime} \tag{10}$$

where \dot{m}_j is the volume flux across face *j*. It is defined as

$$\dot{m}_j = \int_{S_j} (\boldsymbol{v} - \boldsymbol{v}_s) \cdot \boldsymbol{n} \, \mathrm{d}S \approx A_j (\boldsymbol{v}_j' - \boldsymbol{v}_s) \cdot \boldsymbol{n}_j \tag{11}$$

where A_j is the area of face j, v'_j is the face velocity. The mesh velocities v_s are chosen in such a way that they satisfy the space conservation law (SCL); otherwise a non-physical mass source term will be introduced into the discretized form of equation (8). This issue has been discussed in detail by Demiridzic and Peric [**20**]. The SCL can be expressed in the form

$$\frac{V_{P_0}^n - V_{P_0}^{n-1}}{\Delta t} = \sum_{j=1}^3 \int_{S_j} \boldsymbol{v}_s \cdot \boldsymbol{n} \, \mathrm{d}S = \sum_j^3 \left(\frac{\Delta V_j}{\Delta t}\right) \tag{12}$$

where ΔV_j is the volume swept by face *j* in the time interval Δt . In the code, the mesh velocities are not computed directly from equation (12); instead they are determined by the current mesh position and the mesh position after the time interval Δt (Fig. 1). Thus the flux correction due to the mesh movement can be expressed as

$$A_j \mathbf{v}_{\mathrm{s}} \cdot \mathbf{n} \approx \frac{1}{2} \left[(A_j \mathbf{n})^n + (A_j \mathbf{n})^{n-1} \right] \cdot \frac{(X_j^n - X_j^{n-1})}{\Delta t}$$
(13)

where X_j^{n-1} and X_j^n are the face-centre position vectors at the time steps n-1 and n respectively (see Fig. 1). A similar formulation has been used by Perot and Nallapati [**21**] in simulating free-surface flow.

 ϕ_j' in equation (10) is the variable value interpolated to face *j* using a blending scheme

$$\phi'_j = \phi_j^{(1)} + \gamma_\phi(\phi_j^{(2)} - \phi_j^{(1)}) \tag{14}$$

where the superscripts (1) and (2) denote firstorder interpolation and second-order interpolation respectively. The first-order interpolation is just a simple 'upwind' scheme. The second-order scheme uses the gradient of the variable ϕ and a Taylor expansion to evaluate the value of ϕ on the face centres. In this paper, the gradient of ϕ is constructed using a least-squares approach. γ_{ϕ} is a blending factor which is set to 0.95 in this paper.

The diffusion term is discretized as

$$\int_{S_{j}} -\Gamma_{\phi} \nabla \mathbf{\phi} \cdot \mathbf{n} \, \mathrm{d}S \approx -\Gamma_{\phi_{j}} \frac{A_{j}}{L_{j}} \Big((\phi_{\mathrm{P}_{j}} - \phi_{\mathrm{P}_{0}}) + [(\nabla \phi)_{\mathrm{P}_{j}} \cdot \mathbf{\tau}_{1} - (\nabla \phi)_{\mathrm{P}_{0}} \cdot \mathbf{\tau}_{2}] \Big)$$
(15)

where L_j is the distance from the centre of cell P₀ to that of cell P_j projected on to the normal direction on face *j*. τ_1 and τ_2 are two vectors in the tangential direction of face *j* (Fig. 2). The first term on the right-hand side of equation (15) is the 'normal



Fig. 1 SCL and the calculation of mesh velocity



Fig. 2 Cells used in the discretization. The dashed line is perpendicular to the cell face. It makes an angle (not necessarily 90°) with the line connecting cells P_0 and P_1

diffusion' and the second term is the 'cross diffusion' which is a correction on non-orthogonal meshes.

Linear systems are obtained as a result of the discretization of the two velocity components, e.g.

$$a_{\mathbf{P}_{0}}^{C}\phi_{\mathbf{P}_{0}} = \sum_{j=1}^{3} a_{\mathbf{P}_{0}}^{j}\phi_{\mathbf{P}_{0}}^{j} + b_{\mathbf{P}_{0}}$$
(16)

Here the superscript *C* denotes the diagonal element of the coefficient matrix and *j* denotes the neighbouring cells which share a common face with cell P_0 . The contributions to the coefficient matrix are the mass matrix, the upwind difference of the convective term, and the 'normal diffusion'. The source term b_{P_o} in equation (16) has three contributions: the pressure gradient, the 'cross diffusion', and the departures of the convective flux from the upwind differencing (deferred correction).

The SIMPLE algorithm is used to calculate the pressure. A pressure-correction equation is derived from the continuity equation (1) and is given by

$$\sum_{j=1}^{3} \overline{\left(\frac{1}{a_{\mathbf{P}_{0}}^{C}}\right)}_{j} (\nabla p')_{j} A_{j} \mathbf{n}_{j} = \sum_{j=1}^{3} \dot{m}_{j}$$

$$\tag{17}$$

where p' is the pressure correction and the over bar indicates arithmetic averaging from cell to face.

The left-hand side of equation (17) is a Laplacian operator and is treated similarly to the diffusive term in the momentum equation. Some corrections for the mesh non-orthogonality are also considered. After obtaining the pressure correction p', the pressure and velocity are corrected by

$$p^{m} = p^{m-1} + \beta_{p} P^{m}$$
$$v^{m} = v^{m-1} \frac{1}{a_{P_{0}}^{C}} \sum_{j=1}^{3} p_{j}^{m} A_{j} n_{j}$$
(18)

where β_p is a relaxation factor for pressure (Ferziger and Peric [18] suggested that $\beta_p = 0.2$ for fluid), *m* is the pressure-correction loop counter at time step *n*. After these corrections, the coefficient matrix and source term in equation (16) are computed using the newly updated *p* and *v*. A new velocity is obtained by solving equation (16) again. This velocity is then substituted into (17) to compute a new pressure correction. This pressure-correction procedure is repeated until the convergence criterion is met. The convergence criterion used in this paper is that solution to the subsequent time steps are obtained following the same procedure.

Attention is to be paid to the face velocity v'_i which is

$$\max_{k=1}^{\text{tellnumber}} \left(\left| \sum_{j=1}^{3} \dot{m}_{j} \right| \right) \leq 10^{-5}$$

used to calculate the volume flux. This velocity cannot be obtained by a simple average of the values in the neighbouring cells. Instead, a Rhie–Chow interpolation that introduces some dependence on the pressure is used.

3.3 Boundary conditions for the fluid

In order to solve the momentum and continuity equations, boundary conditions must be provided. For the partial differential equations that govern the fluid motion, usually two types of boundary condition are specified at the boundaries. The first type is a *Dirichlet* boundary condition, where the value of the variable is prescribed. The second type is a *Neumann* boundary condition where the gradient of the variable is prescribed.

The boundary conditions of the problems studied in this work are as follows.

- 1. On solid walls, the normal velocity component is always zero (Dirichlet) and the tangential components are zero gradient (Neumann) for inviscid fluid and zero (Dirichlet) for viscous fluid. Pressure is always zero gradient for both inviscid and viscous fluids. If the wall is considered to be an elastic wall, then the fluid velocity must be the same as the velocity of the structure (or mesh moving velocity).
- 2. On the inlet (piston boundary), the velocity is prescribed as a function of time (Dirichlet condition

for all components). The pressure is also zero gradient.

Some attention should be paid to the free-surface boundary condition. On the free surface, two conditions should hold, namely the kinematic condition and the dynamic condition. The kinematic condition states that no fluid passes through the free surface according to

$$\dot{m}_{\rm fs} = \rho [(\boldsymbol{v} - \boldsymbol{v}_{\rm s}) \cdot \boldsymbol{n}]_{\rm fs} = 0 \tag{19}$$

where the subscripts fs stands for the free surface.

The dynamic condition states that the fluids in contact with the free surface are in the state of dynamic equilibrium. If both the viscous effect and the surface tension effect are neglected, this condition is reduced to the statement that the pressures on both sides of the interface are equal. Since a constant pressure (zero in this work) is always assumed for the air above the free surface, this condition is written as

$$p_{\rm fs} = 0 \quad \text{or} \quad p_{\rm fs}^* = H_{\rm fs}$$
 (20)

In flows with the presence of a free surface, usually the position of the free surface is part of the solution and cannot be pre-determined. In this work, a moving-grid technique is used to track the free surface. An iterative correction procedure has to be implemented to make sure that equations (19) and (20) are satisfied simultaneously at the free surface. Since the pressure is prescribed (zero) on the free surface, the velocity on it will be corrected in the SIMPLE algorithm in order to satisfy the continuity equation. Thus the position of the free surface must also be corrected to compensate for the non-zero flux across the free surface resulting from this velocity correction. The movement of the free surface is achieved by moving the 'control points' instead of the mesh nodes (Fig. 3). These 'control points' are the mesh face centres on the free surface. The mesh node position is then determined by the linear interpolations of these control points. The displacement of the control points along the vertical direction is

$$\Delta y = \gamma \frac{\dot{m}_{\rm fs} \Delta t}{\rho A_{\rm f}(\boldsymbol{n} \cdot \boldsymbol{e}_2)} \tag{21}$$



Fig. 3 Corrections to the free-surface position

Where γ is a relaxation factor, e_2 is the unit vector in the vertical (*y*) direction, A_f is the face area, **n** is the face unit normal vector pointed outwards, \dot{m}_{fs} is the non-zero mass flux across the free surface resulting from the velocity corrections, and Δt is the time step.

To summarize, there are three loops in the computer code: the first is the time-advancing loop; the second is the pressure-correction loop in the SIMPLE algorithm; the third is the loop for tracking the free surface. The iterative procedure will continue until all the corrections become negligibly small. The flow chart of this algorithm is presented in Fig. 4. The details of this free-surface tracking method can be found in the paper by Muzaferija and Peric [**6**].

3.4 Mesh smoothing scheme

Another issue that needs to be addressed is the mesh movement. The movement of the free surface or the structure will lead to a change in position of the boundary nodes in the fluid domain. If only these nodes are moved while the interior nodes are intact, poor quality or even overlapping cells may appear. This could affect the accuracy of numerical algorithms or even prevent the computation from proceeding. A spring-analogue smoothing technique is implemented in the code to control the mesh quality throughout the computing procedure. In this algorithm, each face is replaced by a spring under tension. The mesh smoothing procedure is completed when all the nodes of the mesh reach an equilibrium state. This method avoids topology change and thus is much easier to implement than the point insertion and elimination techniques. The movement of the mesh point is determined by

$$\frac{\mathrm{d}\boldsymbol{r}^{(i)}}{\mathrm{d}t} = \sum_{j=1}^{\mathrm{nb}} \boldsymbol{F}^{(ij)} = \sum_{j=1}^{\mathrm{nb}} k^{(ij)} (\boldsymbol{r}^{(j)} - \boldsymbol{r}^{(i)})$$
(22)

where $\mathbf{r}^{(i)}$ and $\mathbf{r}^{(j)}$ are the position vectors of nodes *i* and *j* respectively, $\mathbf{F}^{(ij)}$ is the pulling force exerted on node *i* by spring *ij*, k^{ij} is the stiffness coefficient of spring *ij*, and nb stands for all the neighbouring nodes that are connected to node *i* (Fig. 5). Since the unknown \mathbf{r} appears on both sides of equation (22), the solution has to be obtained using an iterative method.

3.5 Equation for the elastic beams

The transient response of the elastic tank walls to a sloshing liquid is studied. The sidewalls are assumed to be plates extending to the third dimension and are modelled as elastic beams. The governing equation



Fig. 4 Flow chart for the algorithm in the code

for the beam is

$$m\frac{\mathrm{d}^2 u}{\mathrm{d}t^2} + \frac{\mathrm{d}^2}{\mathrm{d}x^2} \left(EI\frac{\mathrm{d}^2 u}{\mathrm{d}x^2} \right) = f \tag{23}$$

where *m* is the mass per unit length, *u* is the displacement which has two components, *f* is the force per unit length acting on the beam, *E* is the modulus of rigidity, and *I* is the moment of inertia. The following values are considered in this simulation: density of steel, 7800 kg/m³; modulus of rigidity, 200 GPa;



Fig. 5 Spring analogy model for smoothing out the mesh movement

moments of inertia, 2×10^{-3} and 2×10^{-4} m⁴. Comparison is also made for a material whose properties are those of high-density polyethylene (HDPE), i.e. a density of 944 kg/m³, a rigidity modulus of 0.8 GPa and moments of inertia of 2×10^{-1} and 2×10^{-2} m⁴. The standard Galerkin method is used to obtain the element stiffness and mass matrix. The matrix form of the equation is written as

$$\mathbf{M}\ddot{\boldsymbol{u}} + \mathbf{K}\boldsymbol{U} = \boldsymbol{F} \tag{24}$$

The structure has to be integrated in time simultaneously with the fluid. Here, equation (24) is solved using the Newmark method. The two variables α and β in the Newmark method are chosen as $\alpha = \frac{1}{2}$ and $\beta = \frac{1}{4}$. These values lie within the stability limits given by $\beta \ge \frac{1}{4}\alpha + \frac{1}{2}^2$. Although the time step chosen for the structural solver is different from the fluid solver to ensure numerical stability, the transfer of unknowns and updating of boundary conditions occurs exactly at the same time instant. In the present simulation the fluid solutions are stepped every $2\pi/240$ seconds and the structure is cycled four times within this cycle (i.e. $2\pi/960$ s).

Data transfer between the FV code and FE code is illustrated in Fig. 6. It is seen that the beam element



Fig. 7 The computational mesh for flow over a cylinder

nodes are also the vertices of the triangular FV cells. The ghost cell is located at the centre of boundary faces. In the FE code, variables (displacement, velocity, etc) are defined at element nodes; they can easily be interpolated to the ghost cell position by arithmetically averaging. These averaged quantities then act as boundary condition to the FV code. Conversely, variables from the FV code (e.g. pressure) are interpolated to the element nodes and act as the load in the FE code.

4 NUMERICAL RESULTS

4.1 Code validation

The fluid equations are solved using an in house computational fluid dynamics code UNCFV3D. Before going further in the free surface simulation, some validation cases are tested. One of these cases will be presented here. This case involves laminar flow past a circular cylinder. The Reynolds number is 100 based on uniform inlet velocity and diameter of the cylinder. This problem is inherently unsteady because of the existence of shedding vortices in the wake region. The computational mesh is presented in Fig. 7. Instantaneous velocity vector plots and streamlines are presented in Fig. 8. The periodic behaviour of the shedding vortices is examined at one sampling point in the flow field. This sampling point is located on the centre-line and about one



Fig. 6 Data transfer between the FV code and the FE code



Fig. 8 Instantaneous streamlines and velocity vector plots



Fig. 9 The vortex shedding frequency: (a) vertical velocity component versus time; (b) Fast Favier transform of (a)

diameter away from the lee side of the cylinder. In Fig. 9(a), the vertical velocity component at this point is plotted as a function of time. This picture demonstrated a perfectly periodic behaviour. By taking the fast fourier transform Fig. 9(a), the frequency is shown to be 0.16 Hz in Fig. 9(b). This value matches the experimental data [22] very well. Next the free-surface validation is attempted.

4.2 Free-surface validation

The free-surface profile obtained in a numerical wave tank is simulated using the FV program developed by the present authors. As the code has been validated for viscous flows, the code's performance under the 'zero-viscosity' condition is tested. Although the effects of viscosity on the free surface can be compared with those published by Wu et al. [23], the same simulation has not been attempted in the present work. Only the results from potential flow solutions are compared with the present code. A numerical tank with a length-to-depth ratio of 40, as simulated by Wu and Taylor [24], is adopted here for comparison. The tank has a wave-maker piston on the left-hand side and a free surface on the top. The wave-maker piston undergoes a sinusoidal motion with an amplitude A = 10 per cent of water depth ('zero-viscosity' computation) and $\omega^* = 1$. For this case the wavelength obtained $\delta_t = A \sin (\omega t)$ using linearized theory is 5.2 times the depth d. Figure 10 presents the free-surface profile after 3.8 cycles. As the flume length is 40, it will accommodate 7.6 wavelengths or, in other words, it would take 7.6 cycles for the particle in contact with the right-hand wall to be disturbed. Thus 3.8 cycles refers to half this value. From the figure it can be seen that the normalized $\omega^* = \omega \sqrt{d/g}$ freesurface profile is within the range. The wavelength of 5.2d (approximately) is also obtained. From the figure it is also seen that the free surface in the first half of the flume is disturbed corresponding to 3.8 cycles. Figure 11 presents the particle response 6d away from the wave-maker piston. It can be observed that the particle responds after approximately 1.15 wave ± 1 cycles corresponding to its position (6*d*-5.2d), with subsequent peaks occurring every cycle. Thus the results obtained for the numerical wave



Fig. 10 Free-surface profile in a numerical wave tank after 3.8 cycles



Fig. 11 Response of a particle 6*d* from the wave-maker piston

tank is validated. Next, the free-surface response of a liquid in a tank subjected to a periodic oscillation is computed. The results are compared with those published in the literature. A tank is assumed to undergo a sinusoidal transverse displacement of the form Thus the velocity V will be $V = A\omega \cos(\omega t)$. The parameters chosen for this simulation are the same as used by Wu et al. [25] in their simulation, namely a tank with a ratio of depth d to breath 1 of 1 to 2, an amplitude A = 0.0186 and a non-dimensional frequency of 1.2. The analytical solution for this case is obtained from the derivation provided by Faltinsen [26]. Figures 12 and 13 present the free-surface profiles after 2.5 cycles and 3 cycles respectively. The results published by Wu et al. [25] are digitized and superimposed in the present figures. It is seen from the Figures that the results obtained in the present method are comparable with the published results.



Fig. 12 Free-surface profile comparison after 2.5 cycles



Fig. 13 Free-surface profile comparison after 3.0 cycles

4.3 Structure response of the tank sidewalls

The structural response of an elastic tank containing a viscous fluid undergoes a transverse excitation, whose velocity *V* is of the form $V = A\omega \cos(\omega t)$. The amplitude A is varied from 1 to 4 per cent of water depth d and $\omega^* = 0.5, 0.75, 1.0, \text{ and } 1.25$. The kinematic viscosity of the arbitrary fluid is fixed at 0.01. The corresponding Reynolds number $Re = A\omega d/\nu$ varies from 1.57 to 15.66. Figures 14, 15, and 16 present the deformations of the left sidewall for excitation amplitudes, A of 1 per cent, 2 per cent, and 4 per cent respectively of water depth d. It is to be noted that the deformations have been magnified and are not to scale; this is only to show the difference in deformation patterns. The simulations are performed for the structural properties of steel and for a moment of inertia of $2.0 \times 10^{-3} \text{ m}^4$ for various non-dimensional circular frequencies ω^* . From the figures the following



Fig. 14 Left sidewall deformations for various frequencies and an amplitude *A* of 1 per cent of *d*

observations can be made. There is a general trend in the magnitude of deformation and the excitation amplitude. From Fig. 14 it can be seen that the deformation is large for $\omega^* = 1$ compared with other frequencies. The same observation is valid for the higher amplitude of 2 per cent (Fig. 15) with a larger deformation. It is interesting to compare Fig. 14 and 15. Although a larger deformation is seen for the higher excitation frequency $\omega^* = 1.25$ for the 4 per cent amplitude case, the deformation



Fig. 15 Left sidewall deformations for various frequencies and an amplitude *A* of 2 per cent of *d*



Fig. 16 Left sidewall deformations for various frequencies and an amplitude *A* of 4 per cent of *d*

for $\omega^* = 1$ (Fig. 16) is less than that obtained for the 2 per cent amplitude (Fig. 15). It appears that the excitation amplitude and frequency of 2 per cent of water depth and $\omega^* = 1$ give the maximum deformation for this particular cross-section. The thickness of the structure is reduced in such a way that the moment of inertia is reduced by one order of magnitude to 2.0×10^{-4} m⁴. Figures 17, 18, and 19, present the deformations of the left sidewall for excitation amplitudes *A* of 1 per cent, 2 per cent, and 4



Fig. 17 Left sidewall deformations for various frequencies and an amplitude *A* of 1 per cent of *d* (thin structure)



Fig. 18 Left sidewall deformations for various frequencies and an amplitude *A* of 2 per cent of *d* (thin structure)

per cent respectively of water depth *d*. From Fig. 17 to 19, it can be observed that the deformations are larger than those obtained for the previous cases (Figs 14 to 16). However, the critical frequency seems to have shifted to $\omega^* = 0.75$. In this case too the excitation amplitude of 2 per cent of water depth seems to have the largest effect on the structure. Figure 20 presents the deformations on the right sidewall for excitation amplitudes *A* of 1 per cent and 2 per cent of the water depth for a moment of investor of 2.0×10^{-4} m⁴ The critical



Fig. 19 Left sidewall deformations for various frequencies and an amplitude *A* of 4 per cent of *d* (thin structure)



Fig. 20 Right sidewall deformations for various frequencies and amplitude *A* of 1 per cent and 2 per cent of *d* (thin structure)

frequency $\omega^* = 0.75$ is realized here too. However, it was also noted that for a moment of inertia of 2.0×10^{-3} m⁴ the deformation occurred at $\omega^* = 0.75$, instead of $\omega^* = 1$ as observed for the left sidewall. The critical frequency where maximum deformation occurred using a potential flow code for the same set of parameters was found to be $\omega^* = 1.2$ [27]. This shift could be attributed to the damping effects of viscosity. As it was seen that the excitation amplitude of 2 per cent caused the largest deformation, this case is further tested by changing the material property to that of HDPE. Figure 21 presents the deformations of the left and right sidewalls for HDPE whose moment of inertia is 1.0×10^{-1} m⁴, for various frequencies and A = 2 per cent of d.



Fig. 21 Left and right sidewall deformations of HDPE for various frequencies and an amplitude *A* of 2 per cent of *d*

From this figure too, the frequency $\omega^* = 1$ causes the maximum deformation. The moment of inertia is then doubled and simulated for $\omega^* = 1$. Figure 22 presents the deformations of the left and right wall for the various cases studied. It is seen from this figure that the deformation for a moment of inertia of 2.0×10^{-1} m⁴ is less. Thus there are a critical frequency and critical amplitude that cause the maximum deformation. It is also seen that the critical values are dependent on the structural properties. The structural deformation thus alters the flow pattern and, because of that effect, the pressure. Thus the system is interdependent and hence the importance of analysing the FSI as one system can be seen here.



Fig. 22 Comparison of wall deformations for various material properties (M.I., moment of inertia)

4.4 Free-surface flow over a submerged rectangular obstacle

This study concerns the free-surface flow over a submerged obstacle. In this problem, a rigid rectangular obstacle is placed in a tank. The dimensions of the tank and the obstacle are 40×1 and 0.125×0.75 respectively. The distance from the left end of the tank of the left end of the rectangular obstacle is 6.0. The computational domain is meshed with 5788 triangular cells and the mesh is shown in Fig. 23. A piston-type boundary condition (same as in the previous section) is applied on the left-hand side. A slip wall boundary condition is applied for the right-hand side. The bottom wall of the tank and the surface of the obstacle are treated as non-slip



Fig. 23 Mesh for simulating free-surface flow over a submerged obstacle (the X-to-Y ratio is 0.3)

walls. The Reynolds number $Re = A\omega d/\nu$ for this simulation is 31.3. A series of instantaneous velocity plots every quarter-cycle for one full cycle is given in Fig. 24. The wave dispersion and the reversing velocity vector can be observed as time progresses. The instantaneous streamlines after an elapsed time of 5.5 cycles are plotted in Fig. 25. The flow separation and formation of vortices immediately after

the obstacle can be clearly seen. The flow separation and vortices will alter the pressure field and thus a simple potential flow solution may not be valid. Thus for this class of problems it is necessary to implement a full Navier–Stokes equation. The limitation of the present code is that it cannot handle a turbulence model; hence, a simulation is currently not possible for high-Reynolds-number flows.



Fig. 24 Wave dispersion and velocity vectors for a flow over an obstacle



Fig. 25 Contours of streamlines after 5.5 cycles

However, as a preliminary investigation, it is observed that some major invscid effect (wave dispersion) and viscous effects (large vortices formed by the roll-up of separated shear layer on the lee side of the obstacle) have been successfully captured in the simulation. To summarize, this simulation shows the capability of this numerical method in applications in marine structural design.

5 CONCLUSIONS

In this paper, a numerical simulation of free-surface flow in a container is presented by using a self-developed moving-mesh FV code. In the first sloshing case, the container walls are considered as rigid, and the free-surface profile of an inviscid fluid has been captured. The computed amplitude matches the simulation in the literature. In the second sloshing case, the sidewalls are treated as elastic beams and the fluid considered as viscous. An FSI problem is solved by coupling with an FE code. The structure response to the fluid motion is studied and the critical frequency of agitation related to a large deformation of the sidewall is identified. This critical frequency is found to vary with the structural rigidity and is different from that obtained by a potential flow treatment. The third simulation is a viscous free-surface flow past a submerged obstacle in a tank. The flow field shows both an inviscid effect such as wave dispersion, and a viscous effect such as separation bubbles. All these simulations demonstrate the capability and efficiency of this numerical method in modeiling both free-surface and FSI problems.

REFERENCES

- **1 Hirt C. W.** and **Nichols, B. D.** Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Physics*, 1981, **39**, 201–225.
- 2 Lafuarie, B., Nardone, C., Scardovelli, R., Zaleski, S., and Zanetti, G. Modelling merging and fragmentation in multiphase flows with SURFER. *J. Comput. Physics*, 1994, **113**, 134–147.
- **3** The, J. L., Raithby, G. D., and Stubley, G. D. Surfaceadaptive finite volume method for solving free surface flows. *Numer. Heat Transfer B*, 1994, **26**, 367–380.
- **4 Farmer, J., Martinelli, L.,** and **Jameson, A.** Fast multigrid method for solving incompressible hydrodynamics problems with free surfaces. *Am. Inst. Aeronaut. Astron. J.*, 1994, **32**, 1175–1182.
- **5** Osher, S. and Sethian, J. A. Fronts propagating with curvature-dependent speed: algorithms based on Hamilton–Jacobi formulations. *J. Comput. Physics*, 1988, **79**, 12–49.
- 6 Muzaferija, S. and Peric, M. Computation of free surface flow using the finite volume method and moving grids. *Numer. Heat. Transfer B*, 1997, 32, 369–384.
- 7 Sudharsan, N. M., Kantharaj, M., and Kumar, K. Preliminary investigation on non-linear FSI of an offshore structure. *Proc. Instn Mech. Engrs, Part C: J. Mechanical Engineering Science*, 2003, 217, 759–765.
- 8 Housner, G. W. Dynamic pressures on accelerated fluid containers. *Bull. Seismol. Soc. Am.*, 1957, 47, 15–35.
- 9 Housner, G. W. The dynamic behaviour of water tanks. Bull. Seismol. Soc. Am., 1963, 53(2), 381–389.
- **10 Haroun, M. A.** Stress analysis of rectangular walls under seismologically induced hydrodynamic loads. *Bull. Seismol. Soc. Am.* 1984, **74**(3), 1031–1041.
- 11 Kim, J. K., Koh, H. M., and Kwahk, I. J. Dynamic response of rectangular flexible fluid containers. *J. Eng Mechanics*, 1996, 122(9), 807–817.
- **12 Koh, H. M., Kim, J. K.,** and **Park, J.-H.** Fluid–structure interaction analysis of 3-D rectangular tanks by a variationally coupled BEM–FEM and comparison with test results. *Earthquake Engng Struct Dynamics*, 1998, **27**, 109–124.
- 13 Ortiz, J. L. and Barhost, A. A. Modelling fluidstructure interaction. *J. Guidance, Control Dynamics,* 1997, **20**(6), 1221–1228.
- 14 Pal, N. C., Sinha, P. K., and Bhattacharyya, S. K. Finite element coupled slosh analysis of rectangular liquid filled laminated composite tanks. *J. Reinf. Plast. Composites*, 1999, 18, 1375–1407.
- **15 Pal, N. C., Bhattacharyya, S. K.,** and **Sinha, P. K.** Non-linear slosh dynamics of liquid-filled laminated composite container: a two-dimensional finite element approach. *J. Sound Vibr.*, 2003, **261**, 729–749.
- **16 Bermudez, A., Rodriguez, R.,** and **Santamarina, D.** Finite element computation of sloshing modes in containers with elastic baffle plates. *Int. J. Numer. Meth. Engng*, 2003, **56**, 447–467.
- 17 Apsley, D. and Hu, W. CFD simulations of two-and three-dimensional free surface flow. *Int. J. Numer. Meth. Fluids*, 2003, **42**, 465–491.

- 18 Ferziger, J. H. and Peric, M. Computational Methods for Fluid Dynamics, 1996 (Springer-Verlag, Berlan).
- 19 Demirdzic, I. and Muzaferija, S. Numerical method for coupled fluid flow, heat transfer and stress analysis using unstructured moving meshes with cells of arbitrary topology. *Computer Meth. App. Mechanics Engng*, 1995, 125, 235–255.
- **20 Demirdzic, I.** and **Peric, M.** Finite volume method for prediction of fluid flow in arbitrary shaped domains with moving boundaries. *Int. J. Numer. Met. Fluids*, 1990, **10**, 771–790.
- **21 Perot, B.** and **Nallapti, R.** A moving unstructured staggered mesh method for the simulation of incompressible free-surface flows. *J. Comput. Physics*, 2003, **184**(1), 192–214.
- 22 Williamson, C. H. K. Defining a universal and continuous Strouhal–Reynolds number relationship for the laminar vortex shedding of a circular cylinder. *Physics Fluids*, 1988, **31**, 2742–2744.
- 23 Wu, G. X., Entock, T. R., and Greaves, D. M. Viscous effect on the transient free surface flow in a two dimensional tank. *J. Engng Math.*, 2001, **40**, 77–90.
- 24 Wu, G. X. and Taylor, R. E. Finite element analysis of two-dimensional non-linear transient water waves. *Appl. Ocean Res.* 1994, 16, 363–372.
- 25 Wu, G. X., Ma, Q. W., and Taylor, E. Numerical simulation of sloshing waves in a 3-D tank based on finite element method. *Appl. Ocean Res.*, 1998, 20, 337–355.
- **26 Faltinsen, O. M.** A numerical nonlinear method of sloshing in tanks with two-dimensional flow. *J. Ship Res.*, 1978, **22**(3), 193–202.
- 27 Sudharsan, N. M., Ajaykumar, R., Kantharaj, M., and Kumar, K. Transient sloshing response of a liquid in an elastic container. *J. Comput. Meth. Sci. Engng*, 2004 (submitted).

APPENDIX

Notation

$a_{\rm PO}^{j}$	coefficient of the matrix in the linear
	system resulting from the
	discretization
$A_{\rm f}$	face area on the free surface
A _i	face area at the face <i>j</i>
$\dot{b_{\rm P0}}$	source term in the linear system
	resulting from the discretization
e ₂	unit vector in the vertical direction
Ε	Young's modulus
f b	body force on the fluid
F	force on the beam element

$F^{(ij)}$	force exerted on the pseudo-spring
	connecting nodes <i>i</i> and <i>j</i>
g	gravitational acceleration
H	potential function
Ι	moment of inertia
$k^{(ij)}$	stiffness coefficient of the pseudo-
	spring connecting node <i>i</i> and <i>i</i>
Li	distance between two neighbouring
<i></i>	cells
m	mass per unit length of the beam
	element
me.	mass flux across the free surface
m:	mass flux across the face i
n	cell face normal vector
n	static pressure of the fluid
p'	pressure correction
$p \\ n^*$	piezometric pressure of the fluid
ρ Ο	source term in the generic transport
$Q\phi$	aduation
$\mathbf{r}^{(i)}$ $\mathbf{r}^{(j)}$	position vectors of much nodes <i>i</i> and <i>i</i>
1,1	rospoctively
Т	stross topsor of the fluid
1 11	displacement in the beam element
0	valocity of the fluid
v	velocity of the mosh
$v_{\rm s}$	fluid velocity at face i
v_j v_n v_{n-1}	volume of coll P at time stops $n-1$
v p0, v p0	and <i>n</i> respectively
\mathbf{X}^{n-1} \mathbf{X}^n	position vectors of face <i>i</i> at time steps
m j , m j	n-1 and <i>n</i> respectively
ß	relayation factor for pressure
ρ_p	relaxation factor in the free-surface
Ŷ	correction
Q	blending factor
γφ Γ.	diffusive coefficient in the generic
ιφ	transport equation
Λt	time sten
ΔV	area swept by face <i>i</i>
ΔV_J	change in v position in the correction
Δ_y	procedure for tackling the free surface
	dynamic viscosity of the fluid
μ V	kinematic viscosity of the fluid
0	density of the fluid
P T_1 T_2	two vectors in the tangential direction
•1, •2	at the faces
ф	control variable in the generic
+	transport equation
ф':	control variable at face i
Ψ 1	