Computers and Structures 158 (2015) 346–354

Contents lists available at ScienceDirect

Computers and Structures

journal homepage: www.elsevier.com/locate/compstruc

Transient solution of 3D free surface flows using large time steps

Soyoung You, Klaus-Jürgen Bathe *

Department of Mechanical Engineering, Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, MA 02139-4307, USA

ARTICLE INFO

Article history:
Received 20 March 2015
Accepted 9 June 2015

Keywords:
Finite elements
ALE
FCBI scheme
Sloshing
Incompressible flows
Mass conservation

ABSTRACT

This paper presents an Arbitrary Lagrangian Eulerian (ALE) formulation derived from the Reynolds transport theorem to accurately solve certain problems of three-dimensional unsteady Newtonian flows with free surfaces. The analysis problems addressed are those without breaking waves or waves spilling over obstructions. The proposed method conserves mass very accurately and obtains stable and accurate results with large time steps, and even when using rather coarse meshes. The continuum mechanics equations are formulated and the three-dimensional Navier–Stokes equations are solved using a 'flow condition based interpolation' (FCBI) scheme for a tetrahedral finite element using finite volume concepts. Various example solutions are given to indicate the effectiveness of the solution schemes.

1. Introduction

Free surface fluid flow analyses solve problems with continuously moving fluid domains. Many industries require free surface solutions, such as vehicle dynamics and earthquake engineering. If free surfaces are not correctly calculated and designed for, the dynamic system may be affected with possible dangerous consequences, for example, an instability may arise due to the fluid motion in large fuel tanks.

Because of the importance of correct free surface solutions, many researchers have attempted to develop methods to calculate incompressible free surface flows in various fields, see for example Refs. [1–10] and the references therein. The Volume of Fluid (VOF) method is a well-known scheme for an Eulerian approach and uses density functions. This approach can ensure mass conservation but a serious disadvantage of the method is that it does not accurately capture free surfaces and interfaces especially in the calculation of three-dimensional flow problems. Another widely used free surface flow calculation method is the level set scheme. This method makes it relatively easy to capture a free surface accurately, using a function which has zero value contour on the free surface as an identifier. However, while the method has desirable capabilities to establish free surfaces and interfaces, difficulties arise in conserving the total mass of the fluid. Of course, these two approaches can also be combined to reach a more effective scheme. An important Lagrangian approach for free surface analyses is the Smoothed Particle Hydrodynamics (SPH) method. The SPH scheme is attractive because a simulation does not require a mesh. However, the disadvantages are that artificial constants such as smoothing factors are used and the method may induce spurious oscillations; thus, it can be difficult to find an accurate solution.

In this paper, we develop an improved numerical method that accurately establishes the free surfaces and robustly achieves mass conservation without requiring any *a posteriori* mass conservation treatment. The formulation uses an arbitrary Lagrangian–Eulerian (ALE) method with a special focus on the condition of accurate mass conservation during long-time response.

The finite element method is employed because of its strong mathematical foundation and the possibility to directly evaluate the Jacobians used for the Newton–Raphson iterations [11]. For the effective solution of the three-dimensional fluid flows governed by the Navier–Stokes equations, we develop a weak formulation of the tetrahedral MINI element (slightly modified) with step weighting functions and flow-condition-based interpolations (FCBI) for the trial functions in the convective terms [12–14]. This approach ensures that the inf–sup condition for modeling incompressible response is passed and stability is maintained regarding the convective terms for high Reynolds number flows. The contribution in the paper is the formulation and the specific 3D element given for the free surface flow conditions considered herein.

In the next sections we first present the finite element formulation and then we give illustrative example solutions.

2. Finite element formulation

In this section, we present a finite element ALE formulation for the transient solution of incompressible fluid flows with free

* Corresponding author.
E-mail address: kjb@mit.edu (K.J. Bathe).

http://dx.doi.org/10.1016/j.compstruc.2015.06.011
0045-7949/© 2015 Elsevier Ltd. All rights reserved.

surfaces or interfaces. ALE approaches have of course been amply pursued, see e.g. [15–18], but we focus here on using the Reynolds transport theorem in implicit time integration to achieve a formulation that is effective when using large time steps and coarse finite element meshes.

2.1. Governing equations

Considering the complete fluid flow domain, we have the kinematic relation for the free surface

$$(\underline{u} - \underline{u}_m) \cdot \underline{n} = \quad \text{on} \quad S_f \times [0, T] \tag{1}$$

and the mass conservation equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \underline{u} = 0 \quad \text{in} \quad V_f \times [0, T] \tag{2}$$

where $\underline{n}$ is the unit normal vector on the free surface S_f (and also used on any fluid domain), V_f denotes the complete fluid domain (see Fig. 1), ρ is the mass density, $\underline{u}$ is the fluid velocity, $\underline{u}_m$ is the velocity of an underlying medium of observation, which in the ALE formulation is the mesh velocity, and T denotes the time span considered. We assume zero surface tension, and while not explicitly noted variables are of course a function of the spatial coordinates $\underline{x}$ and time t.

Eq. (1) is the kinematic relation on the free surface that is included to satisfy the condition of mass conservation and correctly identify the moving free surface. The mass conservation condition for the interior, Eq. (2) looks as used in compressible flows but we use it here for incompressible flows because we know that the density at a fixed point may change in time due to the motion of the free surface through the fixed point.

In the fluid domain, we name Ω the moving control volume and Γ the surface that encloses the control volume. Using the Reynolds transport theorem, the mass conservation equation in a moving control volume is

$$\frac{d}{dt} \int_\Omega \rho \, d\Omega + \int_\Gamma \rho (\underline{u}_c \cdot \underline{n}) d\Gamma = 0 \tag{3}$$

where $\underline{n}$ is here the outward pointing unit normal to Γ, and $\underline{u}_c$ denotes the convective velocity given by

$$\underline{u}_c = \underline{u} - \underline{u}_m \tag{4}$$

Hence we have

$$\int_\Gamma \rho (\underline{u} \cdot \underline{n}) d\Gamma = -\frac{d}{dt} \int_\Omega \rho \, d\Omega + \int_\Gamma \rho (\underline{u}_m \cdot \underline{n}) d\Gamma \tag{5}$$

The momentum equation using the Reynolds transport theorem is

$$\frac{d}{dt} \int_\Omega \rho \underline{u} d\Omega + \int_\Gamma \rho \left(\underline{u}\,\underline{u}_c + p\underline{I} - \mu (\nabla \underline{u} + \nabla \underline{u}^T) \right) \cdot \underline{n} d\Gamma = \int_\Omega \underline{g} \rho d\Omega \tag{6}$$

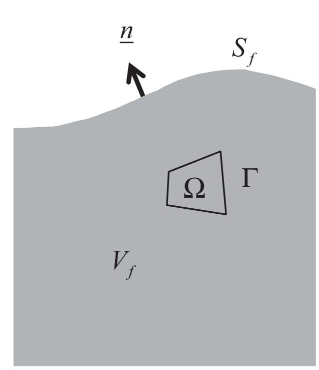

Fig. 1. Complete fluid domain with a free surface.

where $\underline{g}$ is the gravitational acceleration vector. Note that the inertial term accounts for the time rate of change of control volume size. In Eq. (6), we used the stress $\underline{\tau}$ given as

$$\underline{\tau} = \underline{\tau}(\underline{u}, p) = -p\underline{I} + \mu \left\{ \nabla \underline{u} + (\nabla \underline{u})^T \right\} \tag{7}$$

with $\underline{I}$ the identity tensor, p the pressure and μ the viscosity.

The essential boundary conditions are

$$\underline{u} = \underline{u}^S, \quad \underline{x} \in S_v \tag{8}$$

and the natural boundary conditions are

$$\underline{\tau} \cdot \underline{n} = \underline{f}^S, \quad \underline{x} \in S_f \tag{9}$$

where $\underline{u}^S$ is the prescribed velocity on the boundary S_v, $\underline{f}^S$ is the prescribed traction on the boundary S_f, with $S = S_v \cup S_f$ and $S_v \cap S_f = \emptyset$, for the fluid domain, where S denotes the complete boundary.

To solve the momentum and mass conservation equations, Eqs. (5) and (6), we use a Petrov–Galerkin variational formulation in the subspaces U_h, V_h and W_h for the velocities and subspaces P_h and Q_h for the pressure p. The finite element formulation is:

Find $\underline{u} \in U_h$, $\underline{v} \in V_h$ and $p \in P_h$ such that for all $w \in W_h$ and $q \in Q_h$

$$\frac{d}{dt} \int_\Omega q \rho \, d\Omega + \int_\Gamma q \rho [(\underline{u} - \underline{u}_m) \cdot \underline{n}] d\Gamma = 0 \tag{10}$$

$$\frac{d}{dt} \int_\Omega w[\rho \underline{u}] d\Omega + \int_\Gamma w[\rho \underline{v}\{(\underline{u} - \underline{u}_m) \cdot \underline{n}\}] d\Gamma - \int_\Gamma w[\underline{\tau}(\underline{u}, p) \cdot \underline{n}] d\Gamma = \int_\Omega w[\rho \underline{g}] d\Omega \tag{11}$$

In Eqs. (10) and (11), the trial functions in U_h and in P_h are the conventional finite element interpolations for velocity and pressure, respectively. We select these to satisfy the inf–sup condition of the analysis of incompressible media [19]. The advection term, which is not considered in the Stokes flow assumptions, requires different trial functions in V_h from the functions in U_h. The trial functions in V_h should lead to stability of the method when higher Reynolds number flows are considered and we use the flow-condition-based interpolation approach [12]. Step weight functions are chosen in the spaces W_h and Q_h, to achieve local conservation of momentum and mass, respectively. Hence the formulation is in fact a hybrid between the traditional finite element and finite volume formulations.

2.2. 3D tetrahedral MINI element

The motivation for the development of the tetrahedral element is to be able to generate meshes for complicated 3-D geometries. However, for simple geometries we can use meshes based on hexahedra that are subdivided into tetrahedra. One hexahedron is divided into 6 tetrahedral elements, see Fig. 2.

To establish an FCBI scheme for tetrahedral grids that can be used to solve problems with complex geometries, we develop the MINI tetrahedral element using interpolations to satisfy the inf–sup condition, to give stability in the convective terms, and to satisfy mass and momentum conservation locally [11,12,19].

In a slight modification, instead of using the usual cubic bubble for the MINI element, we use a linear hat function [11].

Fig. 3 shows a MINI element in which the velocity is defined at five nodes, at the local node numbers 1–5, while the pressure is defined at four nodes, at the local node numbers 1–4, in order to satisfy the inf–sup condition [11]. The flux is calculated with interpolated values at the centers of the surfaces of the control volumes for the nodes.