Computers and Fluids 136 (2016) 11–34

Contents lists available at ScienceDirect

Computers and Fluids

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

Smoothed particle hydrodynamics method for fluid flows, towards industrial applications: Motivations, current state, and challenges

M.S. Shadloo[a,b,*], G. Oger[b], D. Le Touzé[b]

[a] CORIA-UMR 6614, Normandie University, CNRS-University & INSA of Rouen, 76000 Rouen, France
[b] Ecole Centrale de Nantes, LHEEA Lab. (ECN/CNRS), Nantes, France

ARTICLE INFO

Article history:
Received 6 July 2015
Revised 11 April 2016
Accepted 28 May 2016
Available online 30 May 2016

Keywords:
Computational fluid dynamics (CFD)
Smoothed particle hydrodynamics (SPH)
Meshless methods
Interfacial flows
Industrial applications

ABSTRACT

Smoothed Particle Hydrodynamics (SPH) is a relatively new meshless numerical approach which has attracted significant attention in the last two decades. Compared with the conventional mesh-based computational fluid dynamics (CFD) methods, the SPH approach exhibits some unique advantages in modeling multiphysic flows and associated transport phenomena due to its capabilities of handling complex boundary evolution as well as modeling complicated physics in a relatively simple manner. On the other hand, as SPH is still a developing CFD method, it is crucial to identify its advantages and limitations in modeling realistic multiphysic flow problems of real life and of industrial interest. Toward this end, this work aims at summarizing the motivations behind utilizing the SPH method in an industrial context, making the state-of-the-art of the present application of this method to industrial problems, as well as deriving general conclusions regarding its assets and limitations and stressing the remaining challenges in order to make it an hand-on computational tool.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

1.1. Motivation

Predicting the dynamics of fluids is achieved by three general means: theoretical, experimental, and numerical. Experimental approaches have the advantage of representing the reality and are always used as ultimate reference. On the other hand, they are faced to some limitations, namely scaling difficulties when performed at model scale, measurement difficulties which can induce errors in the representation of the actual flow, and costs which limit the extent of configurations which can be studied. When it comes to complex multiphysic situations governed by multiple characteristic dimensionless numbers, setup, scaling and measurement rise more and more difficulties. As for theoretical approaches, they are limited to simplified physics/configurations and provide useful ideal exact solutions to serve as reference.

With the advent of computational hardware, numerical modeling of these complex multiphysic flows is rapidly growing and has now become common practice on some industrial problems where the simulation tools have been validated enough to be seen as reliable. However, the adequacy, effectiveness, efficiency and accuracy of a given numerical tool directly depends on its underlying numerical methods, and no universal tool exists. Rather, the variety of the methods increases throughout time and each method finds applications for which it is best suited. This draws application domains of which boundaries are constantly moving, with an overall fast growing use on industrial problems. More and more complex problems can now be solved numerically in a reliable manner.

The accurate treatment of difficulties inherent to numerical modeling of fluid flow systems is essential for determining the success of the entire method for real life and industrial applications. The requisites are that: (i) the method should be able to represent different material species, (ii) it should correctly and effectively couple these different species, (iii) it should model precisely the physical continuity/discontinuity conditions at the interfaces between these species, (iv) it should accurately conserve the conservative variables of the problem, especially in cases where the interfaces between species have complex deformations involving fragmentations/reconnections, and finally (v) it should be easily extendable to deal with more complicated multiphysic phenomena. Furthermore, a good methodology should lend itself to three-dimensional modeling and efficient massively parallel computing in order to handle realistic scale complex industrial problems in a reasonable time.

In this context, the Smoothed Particle Hydrodynamics (SPH) is a relatively new method, at least in its application to industrial problems [1]. This is due to its relatively large computational cost, to an

---

[*] Corresponding author.
E-mail address: msshadloo@coria.fr (M.S. Shadloo).

http://dx.doi.org/10.1016/j.compfluid.2016.05.029
0045-7930/© 2016 Elsevier Ltd. All rights reserved.

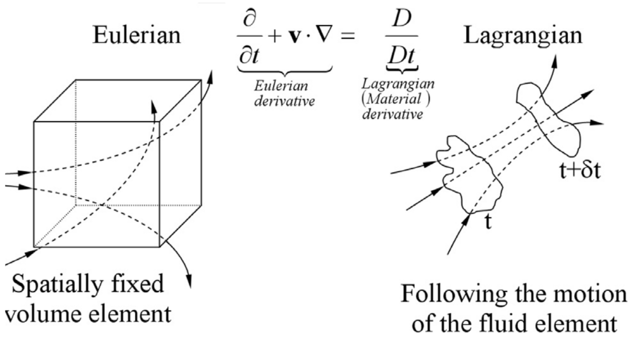

Fig. 1. Eulerian and Lagrangian representation of fluid flow equations.

accuracy which has only recently been risen to a satisfactory level to open the door to real-life application, and to the fact that the applications addressed by the method are physically and/or geometrically complex, thus unreachable with limited hardware.

In the following, we concisely present the frequently used numerical strategies in Computational Fluid Dynamics (CFD) literature for the numerical simulation of fluid flows and elaborate on their differences, similarities, advantages and drawbacks, without going to their detailed implementations. Then, a review of the literature permits to determine whether the SPH method could be a successful alternative approach for leading edge of numerical modeling for industrial applications. A state-of-the-art of its application to industrial problems is made and general conclusions are derived regarding its current strengths and limitations, and its future challenges to become a truly established CFD method.

1.2. Numerical methods for interfacial flows

Interfacial flows are of the most challenging and difficult to represent by CFD, whereas they are present in many industrial problems such as, e.g., cavitation on blades, boiling heat transfer in industrial processes, oil-water separation, wave energy converters, air entrainment at ocean surfaces, bubble reactors and tire grooves among others. Nevertheless, because of the complexity of these problems mainly associated with the necessity of accurately describing complex interface evolutions, most of the literature discusses only of simple problems. As can be inferred, the interface evolution is crucial to the modeling of such flows and thus, needs to be modeled correctly and studiously in order to obtain reliable simulation results. Note that, the term “interface” and/or “interfacial flow” is used in general sense to specify all kind of interfacial phenomena such as free surface flows, multi-phase flows, fluid-structure interaction, *etc.*; however in the following the discussion will be more focused on the fluid parts and interfacial phenomena treatment.

The ongoing attempts of modeling interfacial fluid flows resulted in the availability of a numerous amount of papers with different numerical approaches. This is the topic of different review papers, e.g. by, Anderson et al. [2], Cuvelier and Schulkes [3], Floryan and Rasmussen [4], Hou [5], Scardovelli and Zaleski [6], Tsai and Yue [7] and Shyy et al. [8].

Numerical methods for fluid flow can be categorized into three distinct classes based coordinate system utilized, namely, Eulerian, Lagrangian and mixed Eulerian-Lagrangian. Eulerian methods generally employ a reference coordinate system wherein phase properties are transmitted from one cell into another. In Lagrangian methods unlike Eulerian methods, moving coordinate system is utilized whereby the phase elements (can be represented by numerical cells or particle) move along its motion while containing identical phase species (see Fig. 1). In between, mixed Eulerian-Lagrangian are the numerical schemes that employ both Eulerian and Lagrangian concepts.

Since the Eulerian treatment of the interfacial phenomena is not of interest of the current review paper, we will not detail these methods as the rest of this work is dedicated to Lagrangian techniques, more specifically SPH method. Interested readers are referred to [9,10].

1.2.1. Meshless particle methods

Using meshless particle methods is the second popular approach circumventing the mesh tangling problem. In these methods, grids are completely abandoned. In this group, the discrete flow is represented by replacing the conventional mesh with a finite number of particles which can carry the fluid characteristic properties such as position, mass, velocity, and other hydrodynamics properties. Hence, the fluid system evolution is governed by interactions between these particles. The particles are explicitly associated with different materials, and thus the interface between species can be easily tracked. The Boltzmann lattice-gas algorithms can be classified in the category of particle methods [11–13]. Although they have a natural ability to treat the interfacial flows, these methods suffer from some uncertainties which are: (i) the concerns towards the reliability of physical models for flow viscosity and also inter-particle force representation and (ii) how to properly model the interfacial jump conditions with high density and viscosity ratio and in the presence of surface tension.

In the scope of meshless particle methods, SPH (or alternatively moving particle semi-implicit (MPS) method [14]) is a solution towards achieving a realistic physical model for interfacial flows. Benefiting from a smoothing kernel function, physical quantities are interpolated in a discrete form [15,16]. Nevertheless, common to every numerical method, the SPH method in its standard representation has also some shortcomings around: (i) accuracy of flow variable approximation as an optimized point between the interpolation accuracy and numerical diffusion, and (ii) modeling of large ratios of density/viscosity discontinuity at the interface. Additionally, particle clustering in some region may cause insufficient particle resolution in some other region and hence, comparing to grid-based Lagrangian methods, particle methods may suffer from the accurate representation of the interface.

It is noted that there are many modified versions of standard SPH in which these shortcomings tried to be resolved [17–23]. For instance the density ratios up to thousand and the viscosity ratios of the order of hundred are reached [24–26]; or the particle clustering problem is hindered by using artificial particle displacement technique [27–30], background pressure field [31,32], or some other methods [33,34]. However, information regarding the detail of these methods is not in the scope of the current work and interested readers are referred to Randles and Libersky [35], Monaghan [36], Cleary et al. [37], Liu and Liu [38] and references therein.

1.3. Computational strategy

Here, we choose purely Lagrangian meshless particle approach since it enables us to use movable particles on any arbitrarily computational domain. It is designed to solve situations where meshes have difficulties. As examples for very large deformations of the fluid domain, large motions of multi-bodies (especially contact between 2 bodies in the flow), and to avoid costly mesh generating/handling (human cost especially). Second, we rely on this fact that the interfaces between different materials can be easily followed in order to be able to deal with complex changes in interface topology including interface break-up, and merger phenomena. The latter is due to the Lagrangian nature of the method for fluid representation. This enables us to naturally and accurately

Download English Version:

https://daneshyari.com/en/article/7156627

Download Persian Version:

https://daneshyari.com/article/7156627

Daneshyari.com