Computers & Fluids 88 (2013) 418–429

Contents lists available at ScienceDirect

Computers & Fluids

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

A Smoothed Particle Hydrodynamics method with approximate Riemann solvers for simulation of strong explosions

Fedir V. Sirotkin, Jack J. Yoh *

School of Mechanical and Aerospace Engineering, Seoul National University, Seoul 151-742, Republic of Korea

ARTICLE INFO

Article history:
Received 4 April 2013
Received in revised form 12 September 2013
Accepted 27 September 2013
Available online 11 October 2013

Keywords:
Smoothed Particle Hydrodynamics
Riemann solver
Shock physics
Interface tracking

ABSTRACT

A new modification of the Smoothed Particle Hydrodynamics (SPH) method is presented. The performance of the proposed method is checked when used together with Local Lax-Friedrichs (LLF), Harten, Lax, van Leer (HLL) and exact Riemann solvers, and we derive the SPH equations for each applied solver. The validation problems include Sod's problem, Sjögreen test, blast wave tests, and collision of strong shocks problem. On the basis of our results, we conclude that the application of HLL and LLF approximate Riemann solvers is preferable, with LLF solver having a smallest compuational load. The conservative properties of the method are also strong. The linear momentum, the total mass, and the total energy are conserved within machine accuracy for all applied Riemann solvers.

1. Introduction

Smoothed Particle Hydrodynamics (SPH) is a gridless Lagrangian particle method, and because of its simplicity and robustness, SPH is a useful tool for applications in numerous areas, including fluid dynamics, magnetohydrodynamics, free surface and interfacial flows, multi-phase flows, high-velocity impacts, penetration, shock damage in solids, and explosion phenomena [1]. Regarding simulations of explosions phenomena, SPH was successfully applied for underwater explosion [2,3], detonation of TNT charge of various geometry [3], and detonation of heterogeneous explosives [4].

Since it was introduced, the SPH has been the subject of extensive research to address major technical difficulties when applied for simulations of strong explosions. In particular, when SPH with the conventional artificial viscosity concept is applied to model flows which evolve strong shocks, the resulting shock fronts are smeared while near contact discontinuities strong glitches are present. Shock profiles appear to be more blurred and broad than when grid based methods are applied. The low accuracy at boundaries and regions with high gradients is related to the applied shock capturing scheme and consistency of the SPH approximation. SPH methods which enforce consistency of the particle approximation show a much better accuracy at boundaries, but fail near contact discontinuities [5].

In the more involved reformulations of SPH, which can handle strong shock phenomena, the pairwise particle interaction is determined by solving the Riemann problem between each pair of interacting particles [6–8]. This technique is analogous to that used in grid based Godunov type methods where the Riemann problem is solved at each cell interface to calculate the inter-cell flux.

In the SPH version of [8], the particle consistency is restored and Godunov-type scheme is applied. In this method the evolution equations are obtained by direct convolution of the exact equations with the kernel function, while evaluation of the spatial integrals is performed by interpolating the specific volume around each pair of particles. Unfortunately, when applied to blast wave and shock wave problems, the resulting pressure and energy profiles show strong spikes near contact discontinuities [7]. The amplitude of these spikes depends weakly on the accuracy of the interpolation procedure which is necessary for evaluating the spatial integrals.

Parshikov and Medin [6] applied Godunov-type SPH in conjunction with the approximate state Riemann solver. In contrast to [8] where the density summation approach is applied, they evaluate the density flux from the continuity equation. For mild problems, such as Sod's shock tube test, their method does not produce spikes near contact discontinuities. For the blast wave problem their method requires a fine tuning of ad hoc coefficients in the continuity equation which cannot be justified.

Cha and Whitworth [7] derived four different versions of Godunov-type SPH. They observed observed that particles which represent colliding flows do not penetrate across contact discontinuity even in the case of supersonic flows. For the Godunov-type SPH this effect is brighter. However, the runs of the strong blast wave test exhibit strong spikes near the contact discontinuity for all kind of methods they applied.

* Corresponding author. Tel.: +82 2 880 9334; fax: +82 2 887 2662.
E-mail addresses: fyodor@snu.ac.kr (F.V. Sirotkin), jjyoh@snu.ac.kr (J.J. Yoh).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.09.029

Murante et al. [9] improved significantly the accuracy of the Godunov SPH method by applying MUSCL [10] approach when the second order accuracy is achieved with the use of piecewise linear reconstruction. Despite improvements they made, spurious oscillations near contact discontinuities are still present.

The reliability and convergence of any kind of numerical method are both important. However, it is not always possible to achieve both especially in the wide range of solutions. If numerical artifacts exist when first order method is applied for simple one-dimensional test problem such as Sod shock tube test, it is hard to believe that the second order extension of the method is able to describe complex physical phenomena in one or two-dimensional cases. The main intention of this work is select numerical scheme which produce reliable results with acceptable convergence within framework of conventional SPH.

In this article we derive the SPH equations using finite volume approach which is frequently applied for Lagrangian methods on unstructured meshes. The performance of the code is checked with various exact and approximate Riemann solvers. We apply Local Lax-Friedrichs (LLF), Harten, Lax, van Leer (HLL) and exact Riemann solvers. Our intention is to select approximate solvers which do not produce spurious oscillations near contact discontinuities on the one hand. On the other hand, we seek to find solvers which do not require significant modifications of existing SPH codes. We check the performance of selected Riemann solvers with the one-dimensional tests and compare obtained results against exact solutions. We found that, when exact Riemann solver is applied, the resulting pressure and energy profiles show strong spikes which do not dissapear when the temporal and spatial resolution increases, while results with LLF and HLL solvers do not show such artifacts. The latter outperformed the exact Riemann solver in terms of computational load. We conclude therefore, that LFF and HLL solvers are more suitable for the subsequent second order extension of the method.

The paper is organized as follows. In Section 2 we write down the general form SPH hydrodynamic equations. The SPH equations for exact, Local Lax-Friedrichs (LLF), Harten, Lax, van Leer (HLL) Riemann solvers are derived in Sections 2.1, 2.2 and 2.3, respectively. The Section 2.4 describes proposed time-step criteria. The applied interface tracking technique is explained in the Section 2.5. Then in Section 3 we show the results of a few model calculations when derived equations applied for the conventional one-dimensional tests. Section 3.4 provides the data which gives the information about the computational load when different Riemann solvers are applied. In Section 3.5 the actual convergence rate of derived equations is studied. The results are discussed in Section 4, and in Section 5 we give conclusions.

2. The SPH equations

In the SPH the interpolation of a quantity A, which is a function of the spatial coordinates, is based on the integral interpolant

$$A(\mathbf{r}) = \int A(\mathbf{r}')W(\mathbf{r}-\mathbf{r}', h)\mathrm{d}\mathbf{r}' \tag{1}$$

where the function W is the kernel. The interpolant reproduces A exactly if the kernel satisfies the following conditions:

$$\int W(\mathbf{r}-\mathbf{r}', h)\mathrm{d}\mathbf{r}' = 1, \tag{2}$$

and

$$\lim_{h\to 0} W(\mathbf{r}-\mathbf{r}', h) = \delta(\mathbf{r}-\mathbf{r}'), \tag{3}$$

where $\delta(\mathbf{r}-\mathbf{r}')$ is the Delta function. Since kernel is normalized to 1, the constants are interpolated exactly [1].

To apply this interpolation to a fluid, we divide it into a set of small mass elements. The element i has a mass m_i, density ρ_i, position $\mathbf{r}_i$ and volume υ_i=m_i/ρ_i. The value of A at particle i is denoted by A_i. The integral (1) can then be approximated by a summation over the mass elements. This gives the summation interpolant

$$A_i = \sum_{j=1}^{N} \upsilon_j A_j W(\mathbf{r}-\mathbf{r}_j, h), \tag{4}$$

where the summation is over all the particles but, in practice, it is only over near neighbors if W falls off rapidly with distance. Typically, h is close to the particle spacing and the kernel W is effectively zero beyond a some distance. For subsequent discussion it is important to note that A_i is the weighted *average*.

The Euler equations for unsteady compressible flow in the Lagrangian reference frame may be written in integral form as

$$\frac{\partial}{\partial t}\int_{V(t)} \mathbf{U}\mathrm{d}V + \oint_{S(t)} (\mathbf{n}\cdot\mathbf{F})\mathrm{d}\mathbf{S} = 0 \tag{5}$$

where $V(t)$ is a time-dependent control volume enclosed by the boundary $S(t)$, $\mathbf{U}$ is the vector of dependent variables, $\mathbf{n}$ is the outward unit vector normal to the boundary, and $\mathbf{F}$ is the flux vector. The forms used for $\mathbf{U}$ and $\mathbf{F}$ are defined as:

$$\mathbf{U} = \begin{bmatrix} 1 \\ \mathbf{u} \\ E \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} -\mathbf{u} \\ \mathbf{I}p \\ p\mathbf{u} \end{bmatrix}, \tag{6}$$

where ρ is the density, $\mathbf{u}$ is the velocity vector, p is the pressure, $\mathbf{I}$ is the unit tensor, $E = \varepsilon + \frac{1}{2}u^2$ is the specific energy, and ε is the specific internal energy. The set of Eq. (5) is closed with the equation of state:

$$p = (\gamma - 1)\rho\epsilon, \tag{7}$$

where γ denotes the ratio of specific heats.

Enforcing the integral conservation law for a particular particle, i, Eq. (5) can be written as

$$\frac{\partial}{\partial t}\int_{\upsilon_i} \mathbf{U}\mathrm{d}V + \int_{\upsilon_i} (\nabla\cdot\mathbf{F})\mathrm{d}V = 0 \tag{8}$$

where we used the divergence theorem. In what follows we assume that any vector or scalar variable inside of volume υ_i is equal to the volume average of this variable. Defining volume average for conserved variables and the divergence of the flux vector as

$$\mathbf{U}_i = \frac{1}{\upsilon_i}\int_{\upsilon_i} \mathbf{U}\mathrm{d}V, \tag{9}$$

$$(\nabla\cdot\mathbf{F})_i = \frac{1}{\upsilon_i}\int_{\upsilon_i} (\nabla\cdot\mathbf{F})\mathrm{d}V. \tag{10}$$

the dependent variables at time levels n and $n+1$ can be related by

$$\mathbf{U}_i^{n+1} = \frac{\rho_i^{n+1}}{\rho_i^n}[\mathbf{U}_i^n - \Delta t(\nabla\cdot\mathbf{F})_i^n], \tag{11}$$

where Δt is the time step. Eq. (11) can be used to update $\mathbf{U}$ explicitly. The resulting algorithm is first order accurate in time. Equations similar to (11) are frequently applied in the case of unstructured meshes [11,12].

Directly applying SPH particle approximation to the divergence of the flux vector yields the following equation

$$(\nabla\cdot\mathbf{F})_i = \sum_{j=1}^{N} \upsilon_j(\mathbf{AF}_j^{1D})\cdot\nabla_i W_{ij}, \tag{12}$$