Computers and Fluids 128 (2016) 65–76

Contents lists available at ScienceDirect

Computers and Fluids

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

A cell-centered Lagrangian method based on local evolution Galerkin scheme for two-dimensional compressible flows☆

Yutao Sun[a], Ming Yu[a,*], Zupeng Jia[a], Yu-Xin Ren[b]

[a] Key Laboratory of Computational Physics, Institute of Applied Physics and Computational Mathematics, Beijing 100094, China
[b] Department of Engineering Mechanics, Tsinghua University, Beijing 100084, China

ARTICLE INFO

Article history:
Received 1 May 2015
Revised 19 January 2016
Accepted 22 January 2016
Available online 29 January 2016

Keywords:
Lagrangian method
Cell-centered scheme
Local evolution Galerkin scheme
Finite volume scheme

ABSTRACT

The paper presents a new cell-centered Lagrangian method for two-dimensional compressible flows. The main feature of the method is that the velocity and pressure at the cell vertex are computed using the local Galerkin evolution scheme for solving the linearized flow equations in terms of the bicharacteristic theory, and then the velocity and pressure are used to update the grid coordinates and evaluate the numerical flux across the cell interface. The local Galerkin evolution operator in terms of the Lagrangian description is developed, which gives the solutions evolving for an infinite small time interval from the initial conditions and still maintaining the genuine multidimensional nature of hyperbolic system. Meanwhile, the present method can preserve geometry compatibility. Several numerical results demonstrate that the method possesses of good property of convergence, symmetry and robustness, and has the capability to handle the multimaterial flows.

1. Introduction

In multimaterial flow simulation, a Lagrangian method embeds a computational mesh in material and solves for the positions of grid points at discrete time intervals. Since the mesh is embedded in material, the motion of the material as well as the multimaterial interfaces is inferred from the motion of the mesh. Therefore, the Lagrangian method can accurately compute the motion of the material interfaces. Its main difficulty lies in the determination of the velocity at the cell vertex, especially in multidimensional cases. The traditional way to overcome this difficulty is to use a staggered-grid discretization, in which the velocity is defined at vertex of cell and the other variables, such as density, pressure and specific internal energy, are defined at the center of cell [1]. Besides inconsistency of locations in defining the physical variables, the staggered-grid scheme has asynchronous time advancement between the momentum equation and the mass and internal energy equations. In its initial version, this scheme cannot preserve the conservation of the total energy, and maybe produce non-physical oscillations induced by the artificial viscosity in the vicinity of shock waves. In the past decade, many studies have been devoted in solving these problems and in improving the conservation, accuracy, monotonicity and robustness of the staggered-grid Lagrangian methods [2-5]. With these improvements, the staggered-grid Lagrangian scheme gradually becomes an accurate and robust method.

A highly promising alternative to the staggered-grid scheme is to use a conservative cell-centered discretization, in which the primary variables including the density, momentum (velocity) and total energy are defined at the center of a cell. The cell-centered scheme is constructed by integrating directly the system of conservation laws on each moving cell with finite volume discretization. Therefore, it preserves the conservation of the momentum and total energy. Besides the consistency of locations in defining physical variables, the cell-centered scheme has synchronous time advancement among the flow governing equations. Moreover, under the framework of finite volume discretization, the artificial viscosity and hourglass viscosity are not needed when the Riemann solver is used. The idea of cell-centered scheme was firstly introduced by Godunov [6] in one-dimensional gas dynamics and then extended to multidimensional flows. In multidimensional cases, there are three typical approaches to determine the vertex velocity of a cell. The first approach is a pure mathematical arithmetic, such as the one proposed by Dukowicz [7], in which an exact or approximate one-dimensional Riemann problem in the direction normal to the cell interface is solved, and then the vertex velocity is computed by the least squares procedure that minimizes the difference between the normal velocity predicted by the Riemann solver and the normal projection of the vertex velocity. The main drawbacks of this

☆ Research supported by Natural Science Foundation of China Grant 11272064.
* Corresponding author. Tel.: +861059872198.
E-mail address: yu_ming@iapcm.ac.cn (M. Yu).

http://dx.doi.org/10.1016/j.compfluid.2016.01.013
0045-7930/

approach are the usual production of an artificial mesh motion and an inconsistent numerical flux across the cell interface with the mesh motion. The second approach is a dimensional-splitting or "grid-aligned" solver, such as the one developed by Cheng and Shu [8], in which the left and right states of velocity across the cell interface are split into normal and tangential component along the cell interface, and then the tangential velocity of the vertex along the cell interface is computed by the arithmetic average, and the normal velocity is computed by a one-dimensional Riemann solver. The main drawbacks of this approach are that the multidimensional effect of flow is not taken into account and the numerical flux still is not consistent with mesh motion. The third approach is a nodal acoustic Riemann solver, such as the ones suggested by Després and Mazeran [9] and Maire [10–16], in which the vertex velocity is computed in a coherent manner in terms of the numerical fluxes across the cell interface. On basis of theoretically analysis of the Lagrangian gas dynamics equations written in the fully Lagrangian form, Després and Mazeran derived a conservative and entropy consistent two-dimensional Lagrangian scheme of the finite volume type. However, even in the case of one-dimensional flows solved by two-dimensional meshes, this scheme computed a vertex velocity depending on the cell aspect ratio. Maire studied this issue and developed an improved scheme that retains the good features of Després and Mazeran scheme but resolves the aspect ratio problem. In the scheme of Maire, the vertex velocity and numerical flux across the cell interface are not computed independently as usual but in a consistent manner with an original solver located at the node. The main feature of this scheme is the introduction of four pressures on every interface, two for every vertex on every side of the interface, and the four pressures are utilized to compute the velocity and pressure at the vertex by means of a linear system derived from the local momentum and total energy conservation and the local entropy inequality. The numerical results show that, in the case of a one-dimensional flow solved on two-dimensional meshes, or for flows in a cylindrical geometry, this scheme recovers the classical Godunov approximate Riemann solver. Subsequently, Maire extends his scheme to higher order accuracy and unstructured grids cases. Maire's scheme has an impressive simulating capability, but still has some very prominent drawbacks. The introduction of four pressures on every interface leads to a nonequilibrium of numerical fluxes on two sides of a cell interface, and the sufficient conditions satisfied by each vertex for the local momentum and total energy conservation and the local entropy inequality are excessively strict, moreover, the approximate acoustic solver and a corresponding weighted least squares procedure do not accommodate extensive flow physics.

Apparently, it is highly preferable to construct the Riemann solver at the cell vertex directly from the characteristic property of multidimensional compressible flow equations. To design this "genuinely multidimensional" upwind technique, the evolution Galerkin scheme [17] may be adopted, in which the exact integral equations for the linear or linearized hyperbolic system were derived from the general theory of bicharacteristics in terms of the primitive physical variables. These integral equations can be solved approximately to obtain the solution at the cell vertex and to evaluate the numerical fluxes across the cell interface. This vertex solver from the bicharacteristic theory essentially is a multidimensional Riemann solver or a generalization of the original idea of Godunov to multidimensional hyperbolic conservation laws. The idea has been studied extensively from theoretical as well as numerical point of view and applied to various scientific and engineering problems involving the compressible flow equations in the Eulerian formulation [18–24]. Traditionally, the evolution Galerkin operator gives the evolution of the approximate solution within a certain time interval. In order to simplify the solution procedure and apply it in the efficient semi-discrete finite volume scheme, the local evolution Galerkin operator has been proposed by Sun and Ren [23], in which the solutions that are evolved for an infinitely small time interval are derived and are used as an multidimensional Riemann solver of the semi-discrete finite volume scheme. The semi-discrete finite volume scheme decouples the temporal discretization and the spatial discretization while maintaining the genuine multidimensional nature of the original evolution Galerkin scheme.

In the present paper, the local evolution Galerkin scheme is extended to the two-dimensional compressible flows in the Lagrangian formulation. This new cell-centered Lagrangian method is proposed under the finite volume framework, in which the velocity and pressure on the vertex of a cell are computed on the basis of the local Galerkin evolution operator for solving the linearized compressible flow equations in terms of the primitive variables, and then the velocity and pressure are used to update the grid coordinates and evaluate the numerical flux across the cell interface.

The main difference between the new method in this paper and the method in [10] is on the construction of the nodal solver. In [10], the nodal solver is derived from the global conservation of momentum and a local entropy inequality, which is equivalent to a weighted least squares procedure. In this paper, the nodal solver is constructed using the local Galerkin evolution scheme for solving the linearized flow equations in terms of the bicharacteristic theory, which is essentially a multidimensional Riemann solver taking "multidimension effect" into account in a nature way. Furthermore, in the method of [10], four pressures are defined on each edge, two for each node on each side of the edge, where a "grid-aligned" Riemann solver (acoustic approximate solver) is adopted and the numerical flux across interface is nonequilibrium. While in the nodal solver of this paper, the pressure is defined on each node, so a really nodal solver is applied and the numerical flux across interface can be equilibrium. Compared to the method in [10], the new method in this paper has smaller absolute error and numerical dissipation.

The paper is organized as follows. In Section 2, we give the cell-centered finite volume method for compressible flows equations in the Lagrangian formulation. In Section 3, the vertex solver to compute velocity and pressure at vertex of cell by local evolution Galerkin operator is derived. In Section 4 the global description of the present algorithm is shown. In Section 5 several numerical tests are shown to demonstrate the convergence, symmetry, accuracy and robustness of this new scheme, and to validate its capability to unstructured grids and multi material flows. Some main conclusions are presented in Section 6.

2. Numerical methods

2.1. Governing equations of compressible flow

The governing equations of compressible flow in Lagrangian formulation are follows:

$$\frac{d}{dt}\int_{\Omega(t)} d\Omega = \int_{\partial\,\Omega(t)} \boldsymbol{u}\cdot d\boldsymbol{l} \tag{1.1}$$

$$\frac{d}{dt}\int_{\Omega(t)} \rho\, d\Omega = 0 \tag{1.2}$$

$$\frac{d}{dt}\int_{\Omega(t)} \rho\,\boldsymbol{u}\, d\Omega = -\int_{\partial\,\Omega(t)} p\, d\boldsymbol{l} \tag{1.3}$$

$$\frac{d}{dt}\int_{\Omega(t)} \rho\, E\, d\Omega = -\int_{\partial\,\Omega(t)} p\,\boldsymbol{u}\cdot d\boldsymbol{l} \tag{1.4}$$

where ρ is density, u and v are component velocity, p is pressure, E is specific total energy, $E = e + (u^2+v^2)/2$, e is specific internal