Engineering Analysis with Boundary Elements 56 (2015) 106–118

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

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

# A novel meshless method for incompressible flow calculations

Kamiar Zamzamian [a,*], M.Y. Hashemi [b]

[a] Department of Mechanical Engineering, Tabriz Branch, Islamic Azad University, Tabriz, Iran
[b] Department of Mechanical Engineering, Azarbaijan Shahid Madani University, Tabriz, Iran

ARTICLE INFO

*Article history:*
Received 10 March 2014
Received in revised form
28 January 2015
Accepted 18 February 2015
Available online 12 March 2015

*Keywords:*
Incompressible flow
Artificial compressibility
Least-squares meshless method
Multi-dimensional characteristic based scheme

ABSTRACT

This paper presents a novel explicit meshless solver for incompressible flows using the multidimensional characteristic relations of artificial compressibility equations. The main objective of this research is using the recently introduced multi-dimensional characteristic based (MCB) scheme in order to prevent the instabilities and comparing it with the results obtained by the one dimensional characteristic based (CB) method. An explicit four-stage Runge–Kutta scheme was used for meshless calculations and local time stepping and residual smoothing are used to accelerate convergence. The MCB and CB meshless methods are used for solving two incompressible flows including flow in a lid-driven cavity and the steady and unsteady flows past a circular cylinder. The results obtained using new proposed MCB meshless method show high accuracy and better convergence rate respect to CB scheme and the results are in good agreement with standard benchmark solutions in the literature.



## 1. Introduction

The various finite difference methods (FDM), finite volume methods (FVM) and finite elements methods (FEM) have been developed for incompressible fluid flow simulation in computational fluid dynamics (CFD). The main problem of CFD for incompressible flow calculation is the generation of quality mesh around complex geometries because low speed of gases and fluid flows are conflict with complex geometries such as air-condition systems [1,2], heat-exchanger [3,4], electronic equipment cooling [5], ocean freight [6] and etc.

In general the numerical mesh generation methods are classified as structured and unstructured methods that is each of these methods has its own advantages and disadvantages [7,8]. Difficulties in generating quality meshes, particularly in complex geometry cases have recently attracted meshless methods. Lohner has shown that generation of points in on domain by the advancing front technique is an order of magnitude faster as compared to an unstructured mesh for a 3D configuration [9,10]. Meshless methods have advantages regarding the moving boundary and large deformations compared with mesh-based algorithms. In which the spatial domain is discretized using a set of points, as opposed to the cells of a finite volume grid. Clouds are then used to solve the governing equations.

The flow derivatives are calculated using different approximation methods like smooth particle hydrodynamics (SPH) [11], generalized finite difference method (GFDM) [12,13], element-free Galerkin method (EFGM) [14–16], radial basis function method (RBFM) [17,18], reproducing kernel particle method (RKPM) [19], meshless local Petrov–Galerkin approach (MLPG) [20,21], etc. Meshless method does not involve remeshing process and easy to realize adaptivity strategy. Lohner et al. used the finite point method (FPM) for compressible flow solution [22]. Recently Ortega et al. developed the finite point method for solving compressible flow problems involving moving boundaries and adaptivity [23–25]. The least-squares meshfree method (LSMFM) was used by Hashemi and Jahangirian for compressible viscous and inviscid flow calculations [26,27] and the convergence behavior and approximation accuracy on Stokes problem by LSMFM have been presented [28]. An upwind least-squares based meshless method was analyzed and used by Su et al. for high Reynolds number flow calculations [29].

Different schemes were proposed for the solution of compressible and incompressible flows via finite difference and finite volume methods. The method of solving low-speed or incompressible flows by the artificial compressibility (AC) correction was first introduced by Chorin [30] in obtaining steady state solutions. In this method, a time derivative of the pressure is added to the continuity equation and a coupling system of equations for pressure and velocity is obtained. By reviewing the literatures, it is found that different schemes for discretization of AC equations have been used in FDM and FVM such as central schemes[31,32], Godunov-type schemes[33,34] characteristic based schemes (CB) [35–39] or multi-dimensional characteristic based schemes (MCB) [40,41].

* Corresponding author.
*E-mail addresses:* zamzamian@iaut.ac.ir (K. Zamzamian), m.y.hashemi@azaruniv.ac.ir (M.Y. Hashemi).

http://dx.doi.org/10.1016/j.enganabound.2015.02.009
0955-7997/

This work presents an explicit meshless solver for incompressible fluid flow. To achieve the discretized form of equation, the Taylor series least-squares method is used for approximation of derivatives at each node which leads to a central difference spatial discretization. For stable computation of hyperbolic problems, some kind of numerical dissipation is needed to guarantee the stability such as upwind schemes. The implementation of upwind schemes with high order of accuracy is easy in unstructured grids and meshless methods. Therefore in this paper, the upwind least-squares meshless method is used.

The main objective of this research is using the MCB scheme in order to prevent the instabilities in meshless method. The explicit four-stage Runge–Kutta scheme was used for meshless calculations and local time stepping and residual smoothing are used to accelerate convergence. The results presented in this paper are for the solution of two benchmark problems including a lid-driven cavity flow and two-dimensional steady and unsteady flows past a circular cylinder.

After introducing the governing equations in Section 2, discretized form of equations in the meshless manner is presented in Section 3. the using of 1D characteristic in least-squares meshless method is explained in Section 4. The basis of new idea including characteristic paths and compatibility relations for artificial compressibility equations and relations in meshless algorithm is presented in Section 5. The boundary treatment and time discretization methods are given in Sections 6 and 7 and finally the obtained results using new scheme and discussions about them are presented in Section 8.

## 2. Governing equations

The non-dimensional conservation form of the Navier–Stokes equations for two-dimensional incompressible flows modified by the AC correction can be expressed as [38]:

$$\frac{\partial \mathbf{W}}{\partial t}+\frac{\partial \mathbf{F}}{\partial x}+\frac{\partial \mathbf{G}}{\partial y}=\frac{1}{Re}\left(\frac{\partial \mathbf{R}}{\partial x}+\frac{\partial \mathbf{S}}{\partial y}\right) \tag{1}$$

where

$$\mathbf{W}=\begin{bmatrix} p \\ u \\ v \end{bmatrix},\quad \mathbf{F}=\begin{bmatrix} \beta u \\ u^2+p \\ uv \end{bmatrix},\quad \mathbf{G}=\begin{bmatrix} \beta v \\ vu \\ v^2+p \end{bmatrix},$$
$$\mathbf{R}=\begin{bmatrix} 0 \\ \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \end{bmatrix},\quad \mathbf{S}=\begin{bmatrix} 0 \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial y} \end{bmatrix},\quad Re=\frac{\rho VL}{\mu} \tag{2}$$

Here $\mathbf{W}$ is the vector of primitive variables, $\mathbf{F}$, $\mathbf{G}$ and $\mathbf{R}$, $\mathbf{S}$ are convective and viscous flux vectors, respectively. The artificial compressibility parameter and Reynolds number are shown by $\beta$ and $Re$, respectively. In Reynolds definition, $L$ is reference length, $V$ is reference velocity and $\mu$ is viscosity of fluid that is constant in flow field.

## 3. Discretization of equations

The least-squares meshless method is used to discretization of the flow equations in the conservation form. The spatial derivatives of the function by using the least-squares method are as following [42]:

$$\left(\frac{\partial \phi}{\partial x}\right)_i=2\sum_{j=1}^{m} a_{ij}(\phi_{j+(1/2)}-\phi_i),\quad \left(\frac{\partial \phi}{\partial y}\right)_i=2\sum_{j=1}^{m} b_{ij}(\phi_{j+(1/2)}-\phi_i) \tag{3}$$

where $\phi$ is general symbol for flow variables, $j+\frac{1}{2}$ is the mid-point of the edge $ij$ and $j$ is in a cloud of point i (Fig. 1). The coefficients in Eq. (3) can be calculated as

$$a_{ij}=\frac{\omega_{ij}\Delta x_{ij}\left(\sum_{k=1}^{m}\omega_{ik}\Delta y_{ik}^2\right)-\omega_{ij}\Delta y_{ij}\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}\Delta y_{ik}\right)}{\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}^2\right)\left(\sum_{k=1}^{m}\omega_{ik}\Delta y_{ik}^2\right)-\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}\Delta y_{ik}\right)^2},$$
$$b_{ij}=\frac{\omega_{ij}\Delta y_{ij}\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}^2\right)-\omega_{ij}\Delta x_{ij}\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}\Delta y_{ik}\right)}{\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}^2\right)\left(\sum_{k=1}^{m}\omega_{ik}\Delta y_{ik}^2\right)-\left(\sum_{k=1}^{m}\omega_{ik}\Delta x_{ik}\Delta y_{ik}\right)^2} \tag{4}$$

where $\omega$ is an arbitrary weighting function such as normalized Gaussian [43,44] (Fig. 2),

$$\omega_{ij}=\frac{e^{-(r_{ij}/c)^2}-e^{-(r_d/c)^2}}{1-e^{-(r_d/c)^2}},\quad r_{ij}=\sqrt{\Delta x_{ij}^2+\Delta y_{ij}^2},\quad r_d=(1+\epsilon_g)r_{max},$$
$$c=\eta r_d,\quad r_{max}=\mathrm{Max}\{r_{i1},r_{i2},\ldots,r_{im}\},\quad \Delta x_{ij}=x_j-x_i,\Delta y_{ij}=y_j-y_i \tag{5}$$

where $r_{max}$ is the maximum value of $r_{ij}$ for point $i$. In practice, $\epsilon_g=1$ and $\eta=0.5$ have given the most accurate results [43].

Applying the least-squares approximations given by Eq. (3) to each component of flux functions in Eq. (1), a semi-discrete form of the Navier–Stokes equations at point $i$ is obtained:

$$\frac{\partial \mathbf{W}_i}{\partial t}+2\left[\sum_{j=1}^{m} a_{ij}(\mathbf{F}_{j+(1/2)}-\mathbf{F}_i)+\sum_{j=1}^{m} b_{ij}(\mathbf{G}_{j+(1/2)}-\mathbf{G}_i)\right]=\mathrm{RHS}$$
$$\mathrm{RHS}=\frac{2}{Re}\left[\sum_{j=1}^{m} a_{ij}(\mathbf{R}_{j+(1/2)}-\mathbf{R}_i)+\sum_{j=1}^{m} b_{ij}(\mathbf{S}_{j+(1/2)}-\mathbf{S}_i)\right] \tag{6}$$

If arithmetic averaging of primitive variables and their derivations are used at mid-point to calculating the convective and viscous fluxes, the flow equations discretization lead to checkerboard pattern and the above equation represents an unstable discretization. Therefore, it is necessary to modify the variables and their gradients at mid-points to remove solution instability. For the carrying out the checkerboard pattern in viscous flux calculating the derivation of any variable ($\phi$) as follows [45]:

$$\nabla \Phi_{j+(1/2)}=\overline{\nabla \phi_{j+(1/2)}}-\left(\overline{\nabla \phi_{j+(1/2)}}\bullet \vec{s}_{ij}-\frac{\phi_j-\phi_i}{|r_{ij}|}\right)\vec{s}_{ij},$$
$$\vec{r}_{ij}=\Delta x_{ij}\vec{i}+\Delta y_{ij}\vec{j} \tag{7}$$

where $\vec{s}_{ij}$ is the unit vector between i and j or $(\vec{r}_{ij}/|\vec{r_{ij}}|)$ and $\overline{\nabla \phi_{j+(1/2)}}$ is the average of the gradient at mid-point $((\nabla\phi_i+\nabla\phi_j)/2)$. The $\nabla\phi_i$ is evaluated as

$$\nabla\phi_i=\left[\sum_{j=1}^{m} a_{ij}(\phi_j-\phi_i)\right]\vec{i}+\left[\sum_{j=1}^{m} b_{ij}(\phi_j-\phi_i)\right]\vec{j}. \tag{8}$$

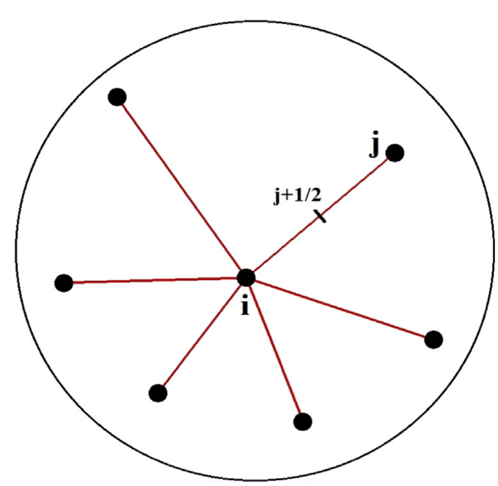


**Fig. 1.** Schematic of point and its neighbors.