Application of Local Radial Basis Function-based Differential Quadrature Method in Micro Flows

D. Xu*, H. Ding** and C. Shu***

*Institute of High Performance Computing, Singapore, xud@ihpc.a-star.edu.sg
**National University of Singapore, Singapore, mpedingh@nus.edu.sg
***National University of Singapore, Singapore, mpeshuc@nus.edu.sg

ABSTRACT

Numerical simulation of flows in micro geometry is one of the most appealing fields in computational fluid dynamics due to the advent of MEMS. In the present study, a local Radial Basis Function-based Differential Quadrature (RBF-DQ) method is applied to study the flow in micro cavity in the slip flow regime. RBF-DQ method is a natural mesh-free approach. The weighting coefficients of RBF-DQ method are determined by taking Radial Basis Functions (RBF) as test functions instead of using high order polynomials. RBF-DQ method has a unique feature of discretizing derivatives at a knot by a weighted linear sum of functional values at its neighboring knots, which may be distributed randomly. In the paper, two-dimensional incompressible flows in a micro lid-driven cavity are simulated with Knudsen number $Kn < 0.1$. Maxwell's first-order formula is adopted to treat slip boundary condition on solid walls.

Keywords: Micro flow, RBF-DQ method, meshless method, CFD

1. INTRODUCTION

In recent years, the theory of radial basis function has obtained intensive attention and enjoyed considerable success as a technique for interpolating multivariable data and functions. Its "true" mesh-free nature motivates researchers to use it to deal with partial differential equations. The first trial of exploration was made by Kansa [1]. Subsequently, Fornberg [2], Hon and Wu [3], Chen et al [4], Fasshauer [5], Chen and Tanaka [6] also made a great contribution in its development. It should be noted that most of above works related to the application of RBF for the numerical solution of PDE are actually based on the function approximation instead of derivative approximation. In other words, these works directly substitute the expression of function approximation by RBF into a PDE, and then change the dependent variables into the coefficients of function approximation. The process is very complicated, especially for non-linear problems. To overcome this difficulty, Wu and Shu [7], Shu et al. [8] recently developed a RBF Differential Quadrature (RBF-DQ) method. RBF-DQ method directly approximates derivatives of a PDE, which combines the mesh-free nature of RBF with the derivative approximation of Differential Quadrature (DQ) method. In this paper, we concentrate on the Multi Quadric (MQ) RBF to exploit its excellent performance in the function approximation. In a local MQ-DQ approach, a spatial derivative at a knot is approximated by a linear weighted sum of the functional values in the supporting region around the knot. The method shows its consistency to both linear and nonlinear problems. The weighting coefficients in derivative approximation are determined by MQ approximation of the function and linear vector space analysis. Some fundamental issues of this method are shown in [8]. This paper further explores the applicability of local MQ-DQ method for simulation of micro incompressible flows.

2. NUMERICAL METHODOLOGY

Following the work of Shu et al. [8], the *n*th order derivative of a smooth function $f(x,y)$ with respect to *x*, $f_x^{(n)}$, and its *m*th order derivative with respect to *y*, $f_y^{(m)}$, at (x_i, y_i) can be approximated by local MQ-DQ method as

$$f_x^{(n)}(x_i, y_i) = \sum_{k=1}^{N} w_{i,k}^{(n)} f(x_k, y_k) \qquad (1)$$

$$f_y^{(m)}(x_i, y_i) = \sum_{k=1}^{N} \overline{w}_{i,k}^{(m)} f(x_k, y_k) \qquad (2)$$

where *N* is the number of knots used in the supporting region, $w_{i,k}^{(n)}$, $\overline{w}_{i,k}^{(m)}$ are the DQ weighting coefficients in the *x* and *y* directions. The determination of weighting coefficients is based on the analysis of function approximation and the analysis of linear vector space.

In the local MQ-DQ method, MQ approximation is applied locally. At any knot, there is a circular supporting region, in which there are *N* knots randomly distributed. The function in this region can be locally approximated by MQ RBF as

$$f(x,y) = \sum_{j=1, j\neq i}^{N} \lambda_j g_j(x,y) + \lambda_i \qquad (3)$$

$$g_j(x,y) = \sqrt{(x-x_j)^2 + (y-y_j)^2 + c_j^2} - \sqrt{(x-x_i)^2 + (y-y_i)^2 + c_i^2} \qquad (4)$$

where λ_j is a constant, c_j is the shape parameter. The weighting coefficient matrix of the *x*-derivative can be determined by

$$[G][W^n]^T = \{G_x\} \qquad (5)$$

where $[W^n]^T$ is the transpose of the weighting coefficient matrix $[W^n]$, and

$$[W^n] = \begin{bmatrix} w_{1,1}^{(n)} & w_{1,2}^{(n)} & \cdots & w_{1,N}^{(n)} \\ w_{2,1}^{(n)} & w_{2,2}^{(n)} & \cdots & w_{2,N}^{(n)} \\ \vdots & \vdots & \ddots & \vdots \\ w_{N,1}^{(n)} & w_{N,2}^{(n)} & \cdots & w_{N,N}^{(n)} \end{bmatrix},$$

$$[G] = \begin{bmatrix} 1 & 1 & \cdots & 1 \\ g_1(x_1,y_1) & g_1(x_2,y_2) & \cdots & g_1(x_N,y_N) \\ \vdots & \vdots & \ddots & \vdots \\ g_N(x_1,y_1) & g_N(x_2,y_2) & \cdots & g_N(x_N,y_N) \end{bmatrix}$$

$$[G_x] = \begin{bmatrix} 0 & 0 & \cdots & 0 \\ g_x^n(1,1) & g_x^n(1,2) & \cdots & g_x^n(1,N) \\ \vdots & \vdots & \ddots & \vdots \\ g_x^n(N,1) & g_x^n(N,2) & \cdots & g_x^n(N,N) \end{bmatrix}$$

The elements of matrix [G] are given by equation (4). For matrix [G_x], we can successively differentiate equation (4) to get its elements. For example, the first order derivative of $g_j(x,y)$ with respect to x can be written as

$$\frac{\partial g_j(x,y)}{\partial x} = \frac{x-x_j}{\sqrt{(x-x_j)^2+(y-y_j)^2+c_j^2}} - \frac{x-x_i}{\sqrt{(x-x_i)^2+(y-y_i)^2+c_i^2}} \tag{6}$$

With the known matrices [G] and [G_x], the weighting coefficient matrix $[W^n]$ can be obtained by using a direct method of LU decomposition. The weighting coefficient matrix of the y-derivative can be obtained in a similar manner. Using these weighting coefficients, we can discretize the spatial derivatives, and transform the governing equations into a system of algebraic equations, which can be solved by iterative or direct method.

From the procedure of DQ approximation for derivatives, it can be observed that the weighting coefficients are only dependent on the selected RBF and the distribution of the supporting points in the support domain, Ω_{x_i}. In the process of numerical simulation, they are only computed once, and stored for future use. It should be noted that the computed coefficients can be applied to both linear and nonlinear problems.

The local MQ-DQ method provides an interesting and effective way to discretize the differential operators in the partial differential equations. However, to solve the Navier-Stokes equations in primitive-variable form, special splitting technique is required to deal with the difficulty arising from lack of an independent equation for the pressure, whose gradient contributed to the momentum equations. In this study, fractional step method is adopted to solve this difficulty.

3. FRACTIONAL STEP METHOD

Considering an unsteady, viscous, incompressible flow, the non-dimensional governing equations in terms of the primitive variables can be written as

Non-dimensional momentum equation:

High-speed flows:

$$\frac{\partial \mathbf{u}}{\partial t_h^*} + \mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}}\Delta\mathbf{u} \tag{7}$$

where the convective time scale $t_h = L/U$ is used.

Low-speed flows:

$$\frac{\partial \mathbf{u}}{\partial t_l^*} + \mathrm{Re}\,\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p + \Delta\mathbf{u} \tag{8}$$

where the convective time scale $t_l = L^2/\nu$ is used.

Continuity equation:

$$\nabla\cdot\mathbf{u} = 0 \tag{9}$$

where Re denotes the Reynolds number, defined as

$$\mathbf{Re} = \frac{UL}{\nu}$$

where L is the reference length, U the reference velocity and ν the kinematic viscosity. Solution of above Navier-Stokes equations confronts some difficulties like the lack of an independent equation for the pressure and nonexistence of a dominant variable in the continuity equation. One way to circumvent these difficulties is to uncouple the pressure computation from the momentum equations and then construct a pressure field so as to enforce the satisfaction of the continuity equation, which is named fractional step method or projection method.

In this paper, a two-step fractional formulation is applied with a collocated/non-staggered arrangement. For a multi-time-stepping scheme, the solution is advanced from time level "n" to "$n+1$" through a predicting advection-diffusion step where pressure term is dropped from the momentum equations. In the advection-diffusion equations, convective and diffusive terms are discretized by using Adam-Bashforth scheme and Crank-Nicolson scheme, respectively. It is known that the implicit treatment of diffusion term eliminates the viscous stability constraint which can be quite severe in numerical computations of viscous flow.

As an example, the following shows how the fractional step procedure is performed for high speed flows. For a time increment, $\Delta t = t^{n+1} - t^n$, the algorithm of fractional step method consists of two steps: Firstly, an intermediate velocity $\mathbf{u}^*$ is predicted by the advection-diffusion equation, which drops the pressure term. That is, for each interior node in the domain, the intermediate velocity $\mathbf{u}^*$ can be calculated by

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -\left[\frac{3}{2}H(\mathbf{u}^n) - \frac{1}{2}H(\mathbf{u}^{n-1})\right] + \frac{1}{2\,\mathrm{Re}}L\left(\mathbf{u}^* + \mathbf{u}^n\right) \tag{10}$$

where H denotes the discrete advection operator, L the discrete Laplace operator. Superscript $n\text{-}1$, n and $n+1$ denote the time levels. Secondly, the complete velocity $\mathbf{u}$ at t^{n+1} is corrected by including the pressure field,

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^*}{\Delta t} = -Gp^{n+1} \tag{11}$$

where G is the discrete gradient operator. The final velocity field should satisfy the continuity equation,

$$D\,\mathbf{u}^{n+1} = 0 \tag{12}$$

where D is the discrete divergence operator. Substituting Eq. (12) into Eq. (11) leads to the following Poisson equation for pressure

$$Lp^{n+1} = \frac{1}{\Delta t}(D\,\mathbf{u}^*) \tag{13}$$

Finally, the velocity $\mathbf{u}^{n+1}$ is updated by the solution of pressure equation (13). It should be emphasized that all the discrete operators mentioned above are discretized by the LMQDQ method.

Numerical experiments for micro flow in micro geometry

Flows in micro cavity in the slip regime are simulated in the study. The flows are considered to be incompressible, viscous, and driven by the moving upper wall. The flows therefore are described by those equations (7) - (9).

For the slip models on the solid walls, Maxwell's first-order formula is adopted, which is applicable for Knudsen number $Kn < 0.1$. Knudsen number is defined:

$$Kn = \frac{\lambda}{L} \tag{14}$$

where λ is mean free path of molecule, and L is characteristic geometry length. The physical boundary conditions of the problem can be specified as

$$u = 0,\ v = k_n\frac{\partial v}{\partial x}, \text{ on left boundary } x = 0 \tag{15a}$$

$$u = 0,\ v = -k_n\frac{\partial v}{\partial x}, \text{ on right boundary } x = 1 \tag{15b}$$

$$u = 1 - k_n\frac{\partial u}{\partial y},\ v = 0, \text{ on upper boundary } y = 1 \tag{15c}$$

$$u = k_n\frac{\partial u}{\partial y},\ v = 0, \text{ on lower boundary } y = 0 \tag{15d}$$

In the paper, numerical experiments are carried out at various Knudsen numbers. For convenience, uniform grid generation technique is adopted to generate the nodes in the domain. Based on time-dependent algorithm discussed in the previous section, numerical simulations for the micro lid-driven flows in a micro square cavity with Knudsen numbers of 0.005, 0.03 and 0.08 are carried out. The flow regime is visualized in the terms of pressure contours and streamlines which are shown in Figures 1 to 3. It can be observed that with the increasing of *Kn* number, the center of primary vortex still locates at the same position. The pressure contours are all more or less similar regardless of the changes of Knudsen numbers. However, due to lack of data from other researchers, the numerical results are not validated.

4. CONCLUTIONS

Flows in a micro square cavity with a moving lid in slip flow regime are numerically simulated using developed Local Radial Basis Function-based Differential Quadrature method with meshless approach. It demonstrates the broad applications of the method. Numerical results show that for slip flow in micro cavity, the Knudsen number does not have great effects to velocity and pressure contours.

REFERENCES

[1] E. J. Kansa. "Multiquadrics – a scattered data approximation scheme with applications to computational fluid-dynamics –I. Surface approximations and partial derivative estimates", Computers Mathematics application, 19, No (6-8) pp127-145 (1990).

[2] B. Forberg and T. A. Driscoll, "Interpolation in the limit of increasingly flat radial basis functions", Comput. Math. Appl., 43, 413-422 (2002).

[3] Y. C. Hon and Z. M. Wu, "A quasi-interpolation method for solving stiff ordinary differential equations", International Journal for Numerical Methods in Engineering, 48, 1187-1197 (2000).

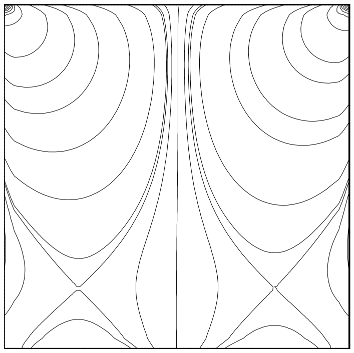
[4] C.S. Chen, C. A. Brebbia and H. Power, "Dual reciprocity method using for Helmholtz-type operators", Boundary Elements, Vol. 20, pp. 495-504 (1998).

[5] G.E. Fasshauer, "Solving partial differential equations by collocation with radial basis functions", Surface fitting and multiresolution methods, eds. A. L. Mehaute, C. Rabut and L. L. Schumaker, pp.131-138 (1997).

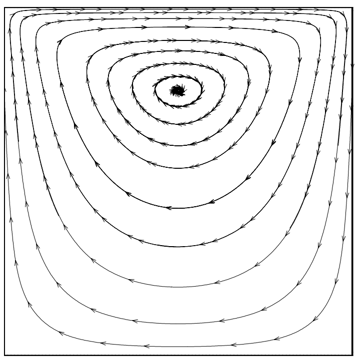
[6] W. Chen and M. Tanaka, "A meshless, integration-free, and boundary-only RBF technique", Comput. Math. Appl., 43, 379-391 (2002).

[7] Y. L. Wu and C. Shu, "Development of RBF-DQ Method for Derivative Approximation and Its Application to Simulate Natural Convection in Concentric Annuli", Computational Mechanics, 29, 477-485 (2002).

[8] C. Shu, H. Ding and K.S. Yeo, "Local radial basis function-based differential quadrature method and its application to solve two-dimensional incompressible Navier–Stokes equations", Comput. Methods Appl. Mech. Engrg. 192, 941-954 (2003).

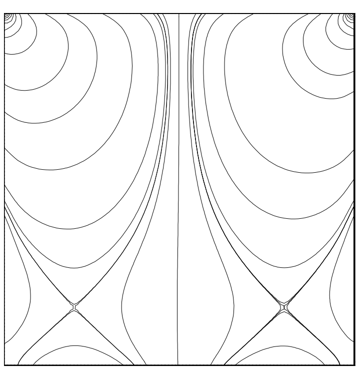


(a)

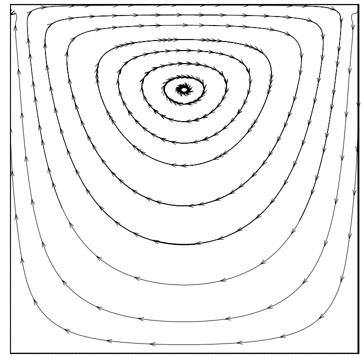


(b)

Fig.1 (a) Pressure contours; (b). Streamlines
Kn=0.005, Re=0.59, Vortex center at x=0.5, y=0.76

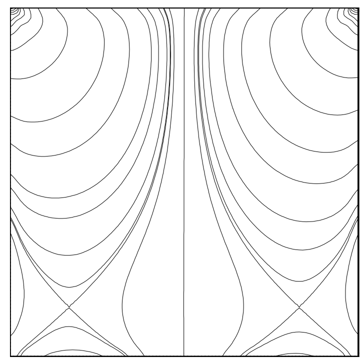


(a)

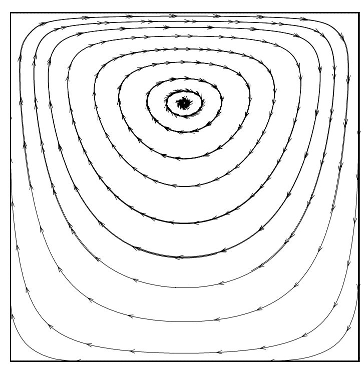


(b)

Fig.2 (a). Pressure contours; (b). Streamlines
Kn=0.03, Re=0.1, Vortex center at x=0.5, y=0.76



(a)



(b)

Fig.3 (a). Pressure contours; (b). Streamlines
Kn=0.08, Re=0.037, Vortex center at x=0.5, y=0.76