 ICCES, vol.6, no.1, pp.13-18

A Meshless Radial Basis Function Method for Fluid Flow with Heat Transfer

K Nagamani Devi[1], D.W. Pepper[2]

Summary

Over the past few years, efforts have been made to solve fluid flow and heat transfer problems using radial basis functions. This approach is meshless, easy to understand, and simple to implement. Preliminary results indicate accuracies on the order of finely meshed conventional techniques, but with considerably less computational effort. In this study, a projection-based technique is used to solve the primitive equations of motion and energy using radial basis functions. Three benchmark test cases are examined: (1) lid-driven cavity flow, (2) natural convection in a square enclosure, and (3) flow with forced convection over backward facing step. Results are compared with COMSOL and FLUENT – two popular commercial CFD packages.

Introduction

In the rapidly developing branch of meshfree numerical methods, there is no need to create a mesh, neither in the domain nor on its boundary. A number of mesh reduction techniques such as the dual reciprocity boundary element method [1], meshfree techniques such as the method of fundamental solutions [2], and meshfree local Petrov Galerkin methods (MLPG) [3] have been developed for transport phenomena and solution of the Navier-Stokes equations. This paper focuses on the simplest class of mesh-free methods being employed today known as radial basis function methods [4].

A common feature of meshless methods is that neither domain nor surface polygonization is required during the solution process. Recently, advances in the development and application of meshless techniques show they can be strong competitors to the more classical finite difference, finite volume, and finite element approaches [5].

Governing Equations

Assuming incompressible laminar flow with convective heat transfer effects and employing scaling relations to non-dimensionalize the equations of momentum and energy, the non-dimensional form of the governing equations can be written as

$$\nabla \cdot V = 0 \tag{1}$$

$$\frac{\partial V}{\partial t} + V \cdot \nabla V = -\nabla p + C_{visc} \nabla^2 V + B \tag{2}$$

[1]Research Assistant, NV Ctr. for Adv. Comput. Meth., UNLV, Las Vegas, NV 89154
[2]Professor and Director, NCACM, UNLV, Las Vegas, NV 89154

$$\frac{\partial T}{\partial t} + V \cdot \nabla T = C_T \nabla^2 T \tag{3}$$

where the Reynolds number, Rayleigh number, Prandtl number and Peclet number are defined as

$$Re = \frac{\rho V L}{\mu},\ Ra = \frac{g\beta (T_h - T_c) L^3}{\alpha \upsilon},\ \Pr = \frac{\upsilon}{\alpha},\ Pe = \frac{VL}{\alpha} \tag{4}$$

with the body force defined as B = PrRaT in the y-direction for natural convection problems. For all the other cases, B = 0. The coefficients in the governing equations are defined as: for 2-D lid driven cavity, C_{visc} = 1/Re; for natural convection in a differentially heated enclosure C_{visc} = Pr, C_T = 1; flow with forced convection over backward facing step, C_{visc} = 1/Re; C_T = 1/Pe.

The Projection Method

Instead of solving a pressure Poisson equation as typically done in many numerical CFD approaches, a simplified local pressure-velocity coupling (LPVC) algorithm is employed. The method represents a local variant of an already developed global solution for coupled heat transfer and fluid flow problems. This local variant was already developed for diffusion problems, convection-diffusion solid-liquid phase change problems and subsequently successfully applied in the industrial process of direct chill casting [6]. In order to solve such problems, the time dependent equations are employed. An implicit time scheme using a simple finite difference approximation is adopted to calculate the time derivative and the Navier-Stokes equations solved iteratively. The LPVC algorithm, where pressure correction is estimated from local mass continuity violation, is used to drive the intermediate velocity towards a divergence-free velocity.

The RBF method

Radial basis functions (RBFs) are increasingly being used as an alternative to traditional discretization schemes employed in FDM/FVM/FEM. A major advantage with using RBFs is that the points on the grid do not need to be uniform in anyway. A random scattering of data points can be used just as easily as a uniform grid. One of the most popular choices for RBFs is the multiquadrics (MQ) approximation first introduced by Hardy [7]. In Kansa's method [8], a function is first approximated by an RBF, and its derivatives are then obtained by differentiating the RBF.

Since multiquadrics (MQ) are infinitely smooth functions, they are often chosen as the trial function for ϕ (some form of RBF), i.e.,

$$\phi(r_j) = \sqrt{r_j^2 + c^2} = \sqrt{(x - x_j)^2 + (y - y_j)^2 + c^2} \tag{5}$$

where c is a shape parameter provided by the user.

The momentum and energy equations are discretized using a linear combination of RBFs and can be expressed in the forms

$$\sum_{j=1}^{N} \hat{V}_j^{n+1}\phi_j(x_i,y_i) = \sum_{j=1}^{N} V_j^n\phi_j(x_i,y_i) + \Delta t\left[C_{visc}\sum_{j=1}^{N} V_j^n\nabla^2\phi_j(x_i,y_i)\right.$$
$$\left. - \sum_{j=1}^{N} P_j^n\nabla\phi_j(x_i,y_i) - \sum_{j=1}^{N} V_j^n\phi_j(x_i,y_i)\sum_{j=1}^{N} V_j^n\nabla\phi_j(x_i,y_i) + \sum_{j=1}^{N} B_j^n\phi_j(x_i,y_i)\right] \quad (6)$$

$$\sum_{j=1}^{N} T_j^{n+1}\phi_j(x_i,y_i) = \sum_{j=1}^{N} T_j^n\phi_j(x_i,y_i) +$$
$$\Delta t\left[C_T\sum_{j=1}^{N} T_j^n\nabla^2\phi_j(x_i,y_i) - \sum_{j=1}^{N} V_j^n\phi_j(x_i,y_i)\sum_{j=1}^{N} T_j^n\nabla\phi_j(x_i,y_i)\right], \quad i = 1,2,\ldots,N_I \quad (7)$$

where N_I denotes the total number of interior points and N denotes total number of points, Δt denotes the time step, superscript $n+1$ is the unknown value to be solved, and superscript n is the current known value. Discretized forms for the pressure correction and velocity correction equations, and the intermediate pressure and velocity relations, can be written in similar fashion.

Test Case Results

Lid-Driven Cavity Flow

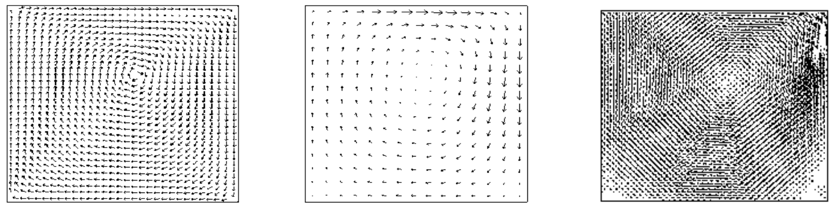
The lid-driven cavity is one of the most frequently employed benchmark cases to evaluate accuracy and feasibility of numerical algorithms and commercial CFD software. Many papers are available in the literature [9].

The boundary conditions for flow in a lid-driven cavity ($0 \leq x \leq 1$, $0 \leq y \leq 1$) include a top lid that is moving at a unit horizontal velocity with no-slip conditions on all other walls.

The computational results for various Reynolds numbers for the lid-driven flow in a square cavity were compared with those obtained by COMSOL and FLUENT. Both uniform and random point distributions of 31 X 31 were used for the RBF approximations. Figure 1 shows the comparison of velocity vectors in the square cavity for Re = 100 using the meshless method with velocity vectors using COMSOL and FLUENT. The meshless results are in excellent agreement with the two commercial packages.

Natural Convection in a Square Enclosure

Natural convection in a square enclosure is another very popular benchmark problem which has been studied extensively over the past 30 years [10].

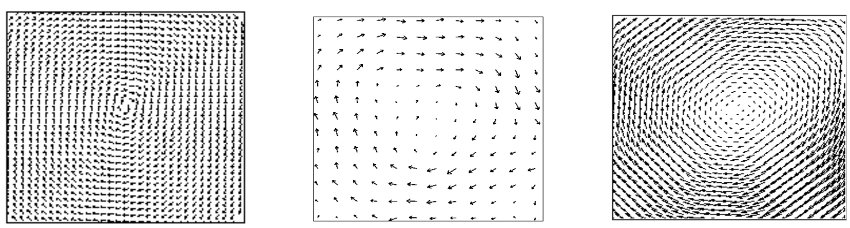


(a) (b) (c)

Figure 1: Velocity vectors in the square cavity using (a) Meshless (b) COMSOL (c) FLUENT

The boundary conditions for a differentially heated square enclosure ($0 \le x \le 1$, $0 \le y \le 1$) are prescribed as a hot left vertical wall with a cold right vertical wall and no-slip velocities everywhere; the top and bottom walls are insulated. The domain of the problem is filled with air (Prandtl number = 0.71).

Assuming constant initial temperature with pressure and velocity set to zero, steady-state is achieved through time transient. Results for various Rayleigh numbers for the natural convection in a square cavity were compared with those of COMSOL and FLUENT. A uniform point distribution of 31 x 31 was used for the RBF approximations. Figure 2 shows the comparison of velocity vectors in the square cavity for Ra = 1000 using the meshless method with velocity vectors using COMSOL and FLUENT. Meshless results are in excellent agreement.



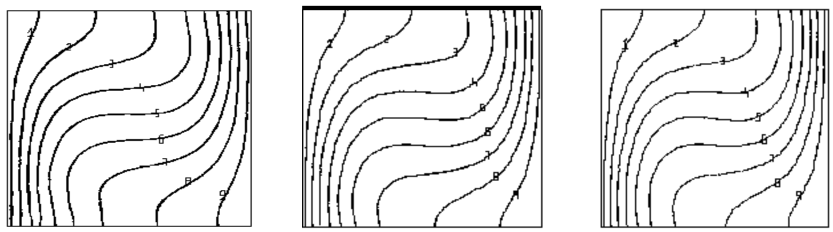
(a) (b) (c)

Figure 2: Velocity vectors for natural convection for Ra = 10^3 in a square cavity using (a) Meshless (b) COMSOL (c) FLUENT

In Figure 3, simulation results of the temperature contours ranging from 0 to 1 with 0.1 as the interval for Ra = 10^4 are compared with results from COMSOL and FLUENT. Meshless results are again in excellent agreement.

Flow with forced convection over backward facing step

Two-dimensional flow over a backward facing step is also a well known benchmark case that has been studied extensively over many years – the problem is easy to set up with known (expected) results at various Reynolds numbers. Early research work for this problem focused on the fluid pattern. The boundary conditions

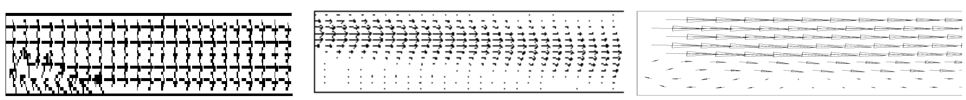


(a) (b) (c)

Figure 3: Isotherms for Natural convection in a square cavity for Ra = 10^4 using (a) Meshless (b) COMSOL (c) FLUENT

for this problem are described in Blackwell and Pepper [11].

A constant heat flux is introduced into the upper and lower channel walls immediately downstream of the step. Flow over the two-dimensional backward facing step is simulated for Re = 800 and Pr=0.71. A uniform distribution of 284 nodes was used for the RBF approximations. Figure 4 shows the comparison of velocity vectors for Re = 800. The meshless results are in excellent agreement.



(a) (b) (c)

Figure 4: Velocity vectors for backward facing step using (a) Meshless (b) COMSOL (c) FLUENT

Temperature contours for Re = 800 are shown in Figure 5. The isotherms are nearly identical for all three models.

(a) (b) (c)

Figure 5: Isotherms for backward step flow using (a) Meshless (b) COMSOL (c) FLUENT

Conclusion

A simplified RBF approach has been developed for the calculation of coupled heat transfer with fluid flow using a local pressure correction scheme. The algorithm is simple to implement, fast and robust. Using only a one step pressure correction, the algorithm needs only a small number of calculations per iteration cycle. The number of points required to obtain comparable accuracy is much less than mesh-based methods, Excellent agreement was achieved using model results obtained by COMSOL and FLUENT.

References

1. Sarler, B.; Kuhn, G. (1999): "Primitive variables dual reciprocity boundary element method solution of incompressible Navier-Stokes equations", *Engineering Analysis with Boundary Elements*, Vol. 23, pp. 443-455.
2. Sarler, B. (2002): "Towards a mesh-free computation of transport phenomena", *Engineering Analysis with Boundary Elements*, Vol. 26, pp. 731-738.
3. Lin, H.; Atluri, S.N. (2001): "Meshless local Petrov Galerkin Method (MLPG) for convection-diffusion problems", *CMES: Computer Modelling in Engineering and Sciences*, Vol. 1, pp. 45-60.
4. Buhman, M.D. (2000): *Radial Basis Functions*, Cambridge University Press, Cambridge.
5. Huang, H.C.; Usmani, A.S. (1994): "*Finite Element Analysis for Heat Transfer*", Springer-Verlag: London, UK.
6. Vertnik, R.; Sarler, B. (2006): "Meshfree Explicit Local Radial Basis Function Collocation Method for Diffusion Problems", *Computer and Mathematics with Applications*, vol. 51, pp. 1269-1282.
7. Hardy, R.L. (1971): "Multiquadric equations of topography and other irregular surfaces", *J. Geophy. Res.*, vol. 76, pp. 1905-1915.
8. Kansa, E.J. (1990): "Multiquadrics – a scattered data approximation scheme with application to computational fluid dynamics, part I", *Computers and Mathematics with Applications*, Vol. 19, pp. 127-145.
9. Koseff, J.R.; Street, R.L. (1984): "Visualization studies of a Shear Driven Three Dimensional Recirculation Flow", *J. Fluids Eng.*, vol. 106, no.1, pp. 21-29.
10. Shu, C.; Wee, K.H.A. (2002): "Numerical simulation of natural convection in a square cavity by simple-generated differential quadrature method", *Computers & Fluids*, vol. 31, pp. 209-226.
11. Blackwell, B.F.; Pepper, D.W. (1992): "Benchmark Problems for Heat Transfer Codes, HTD", vol. 21, pp. 190-198.