 ICCES, vol.3, no.4, pp.251-256

Further Investigation of An Adaptive Three-dimensional Mesh Refinement Method with a Central Vortex Velocity Field

Zhenquan Li[1]

Summary

An adaptive three-dimensional mesh refinement method based on the law of mass conservation has been introduced and tested using some analytical velocity fields as accurate in identifying singular point, asymptotic plane and drawing closed streamlines. This paper further investigates the adaptive mesh refinement method using a velocity field that is due to a uniform strain and a point vortex. Similar results have been obtained.

Introduction

Adaptive mesh refinement is one of the key issues in Computational Fluid Dynamics. There are a large number of publications on mesh adaptive refinements and their applications. The Berger-Oliger method is one of the well-known adaptive mesh refinements [6]. The refinement criterion for this method is local truncation errors. As the solution progresses mesh points with high local truncation errors are flagged. Fine meshes are created such that all the flagged points are interior to some fine mesh. The method suits for solving hyperbolic partial differential equations on structured computational domains and the refinement factor is the same in both space and time. The method has been extended to other applications [e.g. 5, 4, 1]. The other common methods include h-refinement (e.g. [13]), p-refinement (e.g. [2]) or r-refinement (e.g. [14]), with various combinations of these also possible (e.g. [7]). The overall aim of any adaptive algorithm is to allow a balance to be obtained between accuracy and computational efficiency.

From a different point of view, an adaptive mesh refinement method based on the law of mass conservation for three-dimensional incompressible or steady flows has been created [10, 11]. The corresponding two-dimensional adaptive mesh refinement method was proposed in [8, 9]. We assume that f is a scalar function depending only on spatial variables such that its product with the linear interpolation of velocity fields at the nodes of a tetrahedron satisfies the law of mass conservation on the tetrahedron. The criteria for mesh refinement are the conditions for scalar functions f. The adaptive mesh refinement method for three-dimensional incompressible or steady flows in [10] has been investigated through some analytical velocity fields and the corresponding CFD velocity fields that take the values of the analytical velocity fields at the nodes of meshes. The advantages of the adaptive

[1]School of Computing, Information and Mathematical Sciences, The University of the South Pacific, Suva, FIJI

mesh refinement method include: (1) identify singular point, asymptotic plane; (2) draw accurate closed streamlines; (3) control the accuracy by a threshold number T.

We will further investigate the adaptive mesh refinement method in [10, 11] with a velocity field that due to a uniform strain and a point vortex [3].

The Velocity Field

Let a non-axisymmetric strain be

$$U^* = (\alpha x^*, \beta y^*, \gamma z^*),\ \alpha+\beta+\gamma=0,\ \alpha \le \beta \le \gamma \tag{1}$$

where an asterisk denotes dimensional variables [3]. We suppose that the magnetic Prandtl number is small:

$$P_m = \nu/\eta \ll 1$$

where ν is the kinematic viscosity and η is the magnetic diffusivity. Hence, we have the point vortex in polar coordinates

$$\psi(r,\theta) \sim -\frac{1}{2\pi}\ln r. \tag{2}$$

The components of the dimensionless velocity fields that due to the uniform strain (1) and the point vortex (2) in cylindrical coordinate system (r, θ, z) are

$$u_r = -\frac{1}{2}\varepsilon_m r(1+\lambda\cos 2\theta),\ u_\theta = \frac{1}{2\pi r} + \frac{1}{2}\varepsilon_m \lambda r \sin 2\theta,\ u_z = \varepsilon_m z$$

where the parameter ε_m is the reciprocal of a magnetic Reynolds number and λ is a non-negative parameter measuring the non-axisymmetry of the strain field (1). The details of the velocity field can be found in [3].

The Adaptive Mesh Refinement Method

This section reviews the adaptive mesh refinement method proposed in [10]. The adaptive refinement method is for each element in a mesh. Fig.1 is a hexahedral element of a mesh. The conditions for mesh refinement (MC) are for tetrahedra only. The following process describes how to use the conditions (MC) to refine a hexahedral element in a given initial mesh.

Let V_l be the linear interpolation of a CFD velocity field at a tetrahedron. It is unique if the volume of the tetrahedron is non-zero [10, 12].

The refinement process is as follows.

1. Subdivide a hexahedron into five tetrahedra as shown in Fig. 2 and check if V_l satisfies the law of mass conservation on all these tetrahedra. If yes, no refinement for the hexahedron is required. If no, go to Step 2.

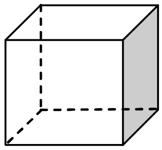

Figure 1: A hexahedral element.

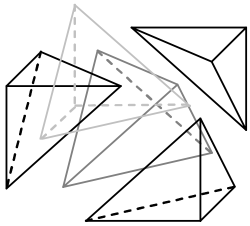

Figure 2: Tetrahedral subdivision of a hexahedron.

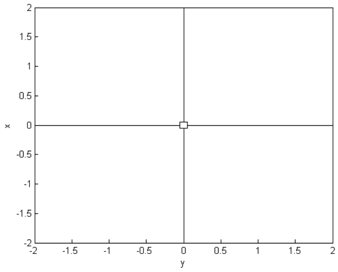

Figure 3: Initial mesh

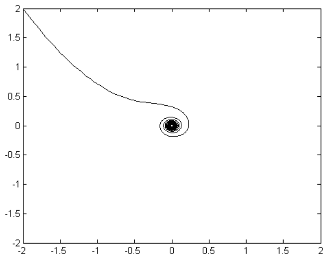

Figure 4: A streamline for λ=0.3 and $\varepsilon_m = 1$.

2. Apply the conditions (MC) to all of the tetrahedra in Fig. 2. If the conditions (MC) are satisfied on these tetrahedra, there is no need to subdivide the hexahedron. Otherwise, we subdivide the hexahedron into a number of small hexahedral elements such that the lengths of all sides of the small hexahedral elements are truly reduced (e.g. half).
3. Take the smaller hexahedra in the subdivided hexahedron as new elements of the mesh by replacing the initial element in Fig. 1 and repeat these three steps until a pre-specified threshold number T is reached.

In this paper, we subdivide a hexahedron into eight equal hexahedra as the same as in [10].

Refined Meshes

This section presents the refined meshes for the velocity field described in Section 2. Only the projections of the refined meshes on the (x, y) plane are shown as in [3] because we are interested only in the structure of the velocity field on the plane. These refined meshes identify the complicated regions of the velocity field by comparing the streamlines and the refined meshes. Because the velocity field has no definition at the origin, a square with side 0.1 unit and center at origin is cut off from the square [-2 2]$\times$[-2 2]. The initial mesh for all refinements is shown in

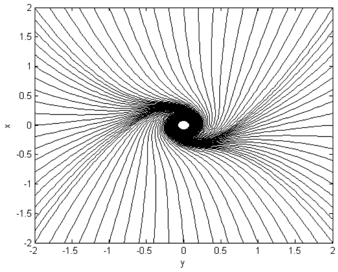

Figure 5: Streamlines for λ=0.3.

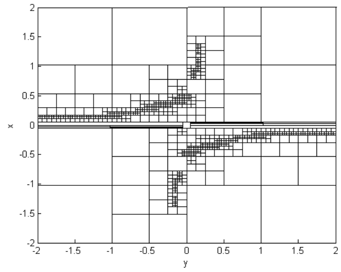

Figure 6: Refined mesh for λ=0.3.

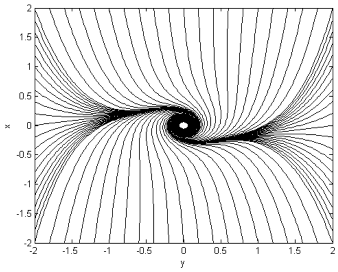

Figure 7: Streamlines for λ=0.6.

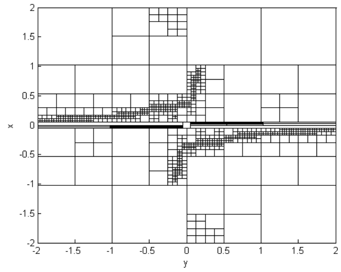

Figure 8: Refined mesh for λ=0.6.

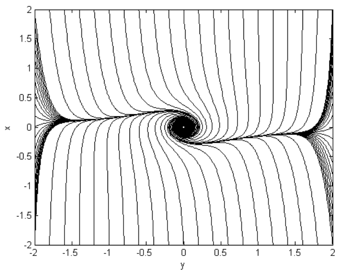

Figure 9: Streamlines for λ=0.9.

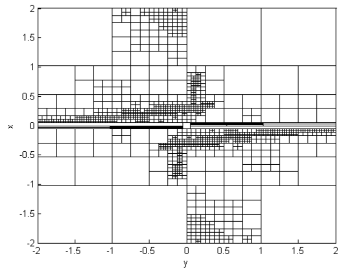

Figure 10: Refined mesh for λ=0.9.

Fig. 3. Fig. 4 shows the streamline of the velocity field with seed point (2, -2). Fig. 5, Fig. 7, and Fig. 9 show the streamlines of the velocity fields drawn by Matlab function ODE45 with the seed points on the boundaries of the square [-2 2]×[-2 2]. Fig. 6, Fig. 8, and Fig. 10 show the refined meshes for T=6, $\varepsilon_m = 1$ and $\lambda = 0.3$, 0.6 and 0.9, respectively.

From Fig. 5, 7, and 9, the projections of the streamlines increasingly strongly converge toward the origin, i.e., the z axis in three-dimensions, as λ increases. From Fig. 6, 8, and 10, we may understand that the refined meshes in the middle regions shrink toward the origin, i.e., the z axis, when λ increases; also the two high density regions of nodes above and below x-axis move toward x-axis on both sides when λ increases. The latter characteristic of the refined meshes is similar to that of the streamlines shown in Fig. 5, 7, and 9. We may conclude from the examples in [10] that we could draw accurate streamlines if the refined meshes are used.

Acknowledgement

The author is grateful to Prof. Konrad Bajor, University of Warsaw, Poland for sharing his knowledge about the velocity field discussed in this paper.

References

1. Almgren, A., Bell, J., Colella, P., Howell, L., Welcome, M. (1998): "A conservative adaptive projection method for the variable density incompressible Navier-Stokes equation", *J. Comp. Phys.*, Vol. 142, pp.1–46.
2. Babuska, I., Szabo, B. A., Katz, I. N. (1981): "The p-Version of the Finite Element Method", *SIAM. J. on Numer. Anal.*, Vol.18, pp. 515–545.
3. Bajer, K., Moffactt, H.K. (1997): "On the Effect of a Central Vortex on a Stretched Magnetic Flux Tube", *J. Fluid Mech.*, Vol. 339, pp. 121-142.
4. Bell, J., Berger, M., Saltzman, J., Welcome, M. (1994): "Three dimensional adaptive mesh refinement for hyperbolic conservation laws", *J. Sci. Comp.*, Vol. 15, pp.127–138.
5. Berger, M.J., Colella, P. (1989): "Local adaptive mesh refinement for shock hydrodyamics", *J. Comp. Phys.*, Vol. 82, pp. 64–84.
6. Berger, M. J. and Oliger, J. (1984): "Adaptive Mesh Refinement for Hyperbolic Partial Differential Equations", *J. Comp. Phys.*, Vol. 53, pp. 484–512.
7. Capon, P.J., Jimack P.K. (1995): "An Adaptive Finite Element Method for the Compressible Navier-Stokes Equations", In: Baines, M.J., Morton, K.W. (eds.): *Numerical Methods for Fluid Dynamics 5*, OUP, pp. 327–334.
8. Li, Z. (2004): "An Adaptive Two-Dimensional Mesh Refinement Method for the Problems in Fluid Engineering", *Proc. of CIS'04. Lecture notes in Computer Science 3314*, pp. 118-123.
9. Li, Z. (2006): "An Adaptive Two-dimensional Mesh Refinement Method Based on the Law of Mass Conservation", submitted to *Computer Modeling in Engineering & Sciences*.
10. Li, Z. (2006): "An Adaptive Three-dimensional Mesh Refinement Method Based on the Law of Mass Conservation", submitted to *Computer Modeling in Engineering & Sciences*.

11. Li, Z. (2005): "An Adaptive Three-dimensional Mesh Refinement Method for the Problems in Fluid Engineering", *Proceedings of ICCES05*-INDIA, pp. 381-386.
12. Li, Z. (2006): "An Adaptive Streamline Tracking Method for Three-Dimensional CFD Velocity Fields Based on the Law of Mass Conservation", to appear in *J. of Flow Visualization and Image Processing*.
13. Lohner, R. (1987): "An Adaptive Finite Element Scheme for Transient Problems in CFD", *Comp. Meth. in Appl. Mech. and Eng.*, Vol. 61, pp. 323-338.
14. Miller, K. and Miller, R. (1981): "Moving Finite Elements. Part I", *SIAM J. on Numer. Anal.*, Vol. 18, pp. 1019-1032.