

Accuracy analysis of a mesh refinement method using benchmarks of 2-D lid-driven cavity flows and finer meshes

Zhenquan Li

Received: 1 August 2013 / Accepted: 31 January 2014 / Published online: 13 February 2014
© Springer International Publishing Switzerland 2014

Abstract Lid-driven cavity flows have been widely investigated and accurate results have been achieved as benchmarks for testing the accuracy of computational methods. This paper verifies the accuracy of a mesh refinement method numerically using two-dimensional steady incompressible lid-driven flows and finer meshes. The accuracy is shown by comparing the coordinates of centres of vortices located by the mesh refinement method with the corresponding benchmark results. The accuracy verification shows that the mesh refinement method provides refined meshes that all centres of vortices are contained in refined grids based on the numerical solutions of Navier-Stokes equations solved by finite volume method except for one case. The well known SIMPLE algorithm is employed for pressure–velocity coupling. The accuracy of the numerical solutions is shown by comparing the profiles of horizontal and vertical components of velocity fields with the corresponding components of the benchmarks and also streamlines. The mesh refinement method verified in this paper can be applied to find the accurate numerical solutions of any mathematical models containing continuity equations for incompressible fluid or steady state fluid flows or heat transfer.

Keywords Mesh refinement · Lid-driven cavity flow · Finite volume method

1 Introduction

Meshing is the process of breaking up a physical domain into smaller sub-domains (called elements or cells or grids) in order to evaluate the numerical solutions of differential equations. Adaptive mesh refinement is a computational technique to improve

Z. Li (✉)
School of Computing and Mathematics, Charles Sturt University,
Thurgoon, NSW 2640, Australia
e-mail: jali@csu.edu.au

the accuracy of numerical solutions of differential equations by starting the calculations on a coarse basic mesh (initial mesh) and then refining this mesh where less accuracy may occur locally.

There are a large number of publications on adaptive mesh refinements and their applications. Some of adaptive refinement methods use a refinement criterion which is based on local truncation errors (e.g. Almgren et al. [1]; Bell et al. [2]; Berger and Olinger [3]; Berger and Colella [4]). Other common adaptive mesh refinement methods include h -refinement (e.g. Lohner [18]; Speares and Berzins [21]), p -refinement (e.g. Bell et al. [2]; Zienkiewicz et al. [23]) or r -refinement (e.g. Miller and Miller [19]; Mosher [20]), with different combinations of these also possible (e.g. Capon and Jimack [5]; Demkowicz et al. [6]). The overall aim of these adaptive algorithms is to allow a balance to be obtained between accuracy and computational efficiency. The h -refinement is a method where meshes are refined and/or coarsened to achieve a prescribed accuracy and efficiency. The p -refinement is a method where the accuracy orders are assigned to grids to achieve exponential convergence rates and r -refinement is a method where grids are moved and redistributed to track evolving non-uniformities. In summary, all these mesh refinement methods are proposed based on the quantitative considerations of numerical solutions of differential equations.

We proposed adaptive mesh refinement methods from a different point of view for 2-D velocity fields (Li [14]) and for 3-D velocity fields (Li [13]) based on a theorem in qualitative theory of differential equations (Theorem 1.14, page 18, Ye et al. [22]). The theorem indicates that a divergence free vector field has no limit cycles or one sided limit cycles, that is, the trajectories (or streamlines) of divergence free vector fields are closed curves in bounded domains (singular points are streamlines). The adaptive mesh refinement methods adaptively refine the mesh based on the information of numerical velocity fields evaluated on it. Using numerical velocity fields obtained by taking the vectors of the analytical velocity fields at nodes of meshes, examples showing the accuracy of the methods include: locating the singular points and asymptotic lines for two-dimensions [14]; the singular points and asymptotic plane for three-dimensions [13]; and drawing closed streamlines (Li [11, 12]) using the refined meshes with a pre-specified number of refinements of the initial meshes. We have verified that the accuracies of the proposed adaptive mesh refinement method in identifying asymptotic line for T from 1 to 5, and identifying singular points and drawing closed streamlines for T from 1 to 4 by two analytical velocity field examples [14]. We have also verified that the larger the value of T , the more accurate are the location of the asymptotic line, the coordinates of the singular points and the closed streamline [14]. Identification of accurate locations of singular points and asymptotic lines (planes), and drawing closed streamlines are some of the accuracy measures for computational methods.

We showed that the adaptive mesh refinement method for 2-D velocity fields provides accurate estimates for the singular points of 2-D steady incompressible lid-driven cavity flows using the numerical velocity fields (Li and Lal [16]). The numerical velocity fields are obtained by solving the Navier–Stokes equations with the boundary conditions numerically using a second order collocated finite volume method (GSFV) with a splitting method for time discretization (Faure et al. [8]). We applied the adaptive mesh refinement method to the initial meshes and the numerical velocity fields, and

take the centres of refined grids in the vortex regions as the estimates of the singular points. The comparison of the estimates with the benchmarks shows that the estimates for the singular points are accurate.

The accuracy of the mesh refinement method using two-dimensional lid-driven cavity flow and a different finite volume method has been established using coarser meshes (65×65 for $Re = 1,000$ and 85×85 for $Re = 2,500$) [17]. It shows that mesh refinement is necessary if we want the relative error of the centre coordinates of tertiary vortices less than 40%. This paper investigates whether mesh refinement is needed for finer initial meshes.

2 Algorithm of mesh refinement and finite volume method

In this section, we summarize the adaptive mesh refinement method based on the continuity equation (Li [14]) and the finite volume method used (Ferziger and Peric [9]).

Assume that $\mathbf{V}_l = \mathbf{A}\mathbf{X} + \mathbf{B}$ is a vector field obtained by linearly interpolating the vectors at the three vertexes of a triangle, where

$$\mathbf{A} = \begin{pmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{pmatrix}, \quad \mathbf{B} = \begin{pmatrix} b'_1 \\ b'_2 \end{pmatrix}$$

are constant matrix and vertical vector respectively, and $\mathbf{X} = (x_1, x_2)^T$. \mathbf{V}_l is unique if the area of the triangle is not zero [15]. Mass conservation for an incompressible fluid means that

$$\nabla \cdot \mathbf{V}_l = \text{trace}(\mathbf{A}) = 0. \quad (1)$$

Let f be a scalar function depending only on spatial variables. We assume that $f\mathbf{V}_l$ satisfies equation (1) and then calculate the expressions of f . Li [14] gives the expressions of f for the four different Jacobian forms of coefficient matrix \mathbf{A} in Table 1. The conditions (MC) (MC is the abbreviation of mass conservation) are the functions f in Table 1 not equaling zero or infinity at any point on the triangular domains when $f\mathbf{V}_l$ satisfies Eq. (1) on these triangular domains.

We review the algorithm of adaptive mesh refinement for quadrilateral mesh used in this paper. The algorithm is also applicable to triangular meshes. The following grid refinement algorithm describes how to use the conditions (MC) to refine a quadrilateral grid in a given mesh. To avoid an infinite refinement of the mesh, we choose a pre-specified threshold number of refinements T based on the accuracy requirements. The algorithm of grid refinement is:

- Step 1 Subdivide a quadrilateral grid into two triangles, obtain \mathbf{V}_l by linearly interpolating the given numerical velocity field and check if \mathbf{V}_l satisfies Eq. (1) on both triangles. If yes, there is no refinement for the quadrilateral grid. If no, go to Step 2.
- Step 2 Apply the conditions (MC) to both of the triangles. If the conditions (MC) are satisfied on both triangles, there is no refinement for the quadrilateral grid.

Table 1 Locations of the centre of vortices

Vortex type	Reynolds numbers		
	Re = 1,000 (99 × 99)	Re = 2,500 (121 × 121)	Re = 5,000 (139 × 139)
Primary vortex	(0.5316, 0.5659)	(0.5202, 0.5446)	(0.5155, 0.5357)
	(0.5300, 0.5650)	(0.5200, 0.5433)	(0.5150, 0.5350)
BR1	(0.8634, 0.1128)	(0.8318, 0.0910)	(0.8037, 0.0729)
	(0.8633, 0.1117)	(0.8350, 0.0917)	(0.8050, 0.0733)
BL1	(0.0839, 0.0779)	(0.0845, 0.1108)	(0.0731, 0.1370)
	(0.0833, 0.0783)	(0.0850, 0.1100)	(0.0733, 0.1367)
BR2	–	(0.9851, 0.0056)	(0.9743, 0.0201)
	(0.9917, 0.0067)	(0.9900, 0.0100)	(0.9783, 0.0183)
BL2	(0.0075, 0.0075)	(0.0090, 0.0083)	(0.0079, 0.0099)
	(0.0050, 0.0050)	(0.0067, 0.0067)	(0.0083, 0.0083)
TL1	–	(0.0441, 0.8904)	(0.0641, 0.9105)
	–	(0.0433, 0.8900)	(0.0633, 0.9100)
BR3	–	–	(0.9397, 0.0043)
	–	–	(0.9983, 0.0017)

Otherwise, we subdivide the grid into a number of small grids such that the lengths of all sides of the small grids are truly reduced (e.g. connecting the mid-points of opposite sides of a quadrilateral by line segments produces four small quadrilaterals and the lengths of the sides of the four small quadrilaterals are truly reduced).

The algorithm of adaptive mesh refinement is:

- Step 1 Evaluate the numerical velocity field for a given initial mesh;
- Step 2 Refine the grids of the mesh one by one using the above grid refinement algorithm;
- Step 3 Take the refined mesh as initial mesh and go to Step 1 until a satisfactory numerical velocity field is obtained or the threshold number T is reached.

In Sect. 3, we use the finite volume method with SIMPLE algorithm for pressure–velocity coupling to evaluate numerical velocity fields [9]. This finite volume method has different arrangement for pressure–velocity from the finite volume method we used before [8].

In this paper, we subdivide a quadrilateral by connecting the mid-points of the two opposite sides of a quadrilateral and the threshold number $T = 1$, i.e., we subdivide a cell once only for testing the accuracy of the refinement method.

3 Accuracy analysis by comparisons with benchmarks

We take the results obtained by using a mesh with 601×601 uniform grids, stream function and vorticity as the benchmarks (Erturk et al. [7]). We consider the accuracy of the mesh refinement method in the following two aspects:

- the variations of the refined meshes according to the comparison of horizontal and vertical profiles of the numerical velocity fields with the corresponding benchmarks.
- inclusion of the centres of vortices located in the benchmarks in the refined mesh.

3.1 Variation of refined meshes

We consider the refined meshes for two-dimensional lid-driven cavity flows for different mesh sizes and Reynolds number $Re = 1,000, 2,500, \text{ and } 5,000$, respectively. We show horizontal profiles at $x = 0.5$ and vertical velocity profiles at $y = 0.5$ as the corresponding benchmarks are known, streamlines of \mathbf{V}_l , and refined meshes. The streamlines are generated by *Matlab* function `streamline`. A grid is said to be a refined grid if a cross is drawn inside.

One of the possible comparisons is the adaptive mesh refinement which refines everywhere that solution gradients are large (Henderson [10], 293–299). The refinement criteria enforce

$$\| \nabla u^{(k)} \| \leq \epsilon \| u^h \|_1$$

everywhere in the mesh, where $\| \cdot \|$ is the L_2 norm, $\| \cdot \|_1$ is the H^1 norm, ϵ is the discretization tolerance, u^h is finite-dimensional approximation for u , and k in $\| \nabla u^{(k)} \|$ is the number of subdomains. Figure 5.7 of [10] shows the refined meshes for $\epsilon = 10^{-3}, 10^{-4}, 10^{-5}, \text{ and } 10^{-6}$ for lid-driven cavity flow at $Re = 1,000$. Even though there might be some relations between the refined meshes and the vorticity field as ϵ decreases, no one provides any information on the pattern of the flow field such as locations of the centres of vortices and separation curves of the regions (e.g., primary and secondary vortex regions).

3.1.1 $Re = 1,000$

We show the figures for $Re = 1,000$ generated from a mesh with 99×99 uniform grids. From Fig. 1, the horizontal profile u of the numerical velocity field at $x = 0.5$ shows a slight difference with the corresponding benchmark. From Fig. 2, the vertical profile v of the numerical velocity field at $y = 0.5$ fits the benchmark well. However, the horizontal and vertical profiles reflect the local accuracy of the numerical velocity field. The streamlines in Fig. 3 provide the global accuracy of the numerical velocity field depending on the accuracy of streamline generation method. The streamlines in Fig. 3 are not closed (spiral lines) so we conclude that the velocity field \mathbf{V}_l does not satisfy Eq. (1) [7,22] or f does not satisfy the condition (MC) on some grids in the regions. Figure 4 shows the refined mesh. There are three isolated grids in the refined mesh: one in the primary region and two in the secondary regions. The refined grid in the primary regions contains the centre of primary vortex, and the isolated refined grid on the bottom left side contains the centre of the bottom left secondary vortex, and the isolated refined grid on the bottom right side contains the centre of the bottom right secondary vortex (refer to Table 1). Even though the centres of tertiary vortices

Fig. 1 Horizontal profile of velocity field at $x = 0.5$ for mesh size 99×99

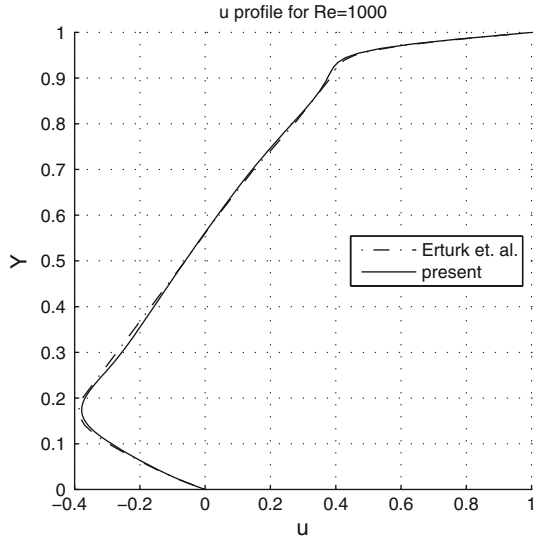
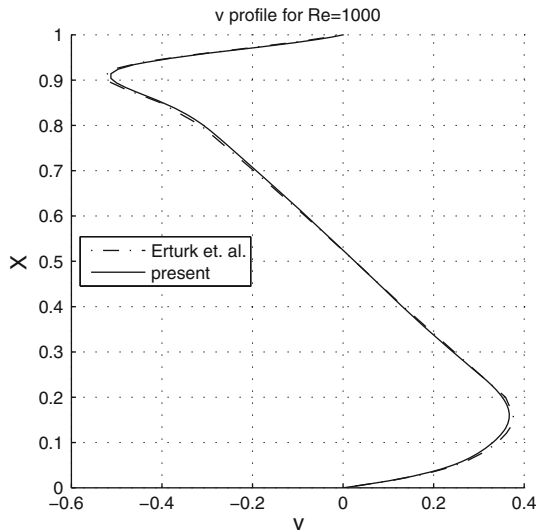


Fig. 2 Vertical profile of velocity field at $y = 0.5$ for mesh size 99×99



are included in the refined grids, we can not identify the grids in the refined mesh. Further mesh refinement is needed for more information on this matter.

3.1.2 $Re = 2,500$

We show the figures for $Re = 2,500$ generated from a mesh with 121×121 uniform grids. From Fig. 5, the difference between the horizontal profile u of the numerical velocity field at $x = 0.5$ and the corresponding benchmark is small. From Fig. 6,

Fig. 3 Streamlines for mesh size 99×99

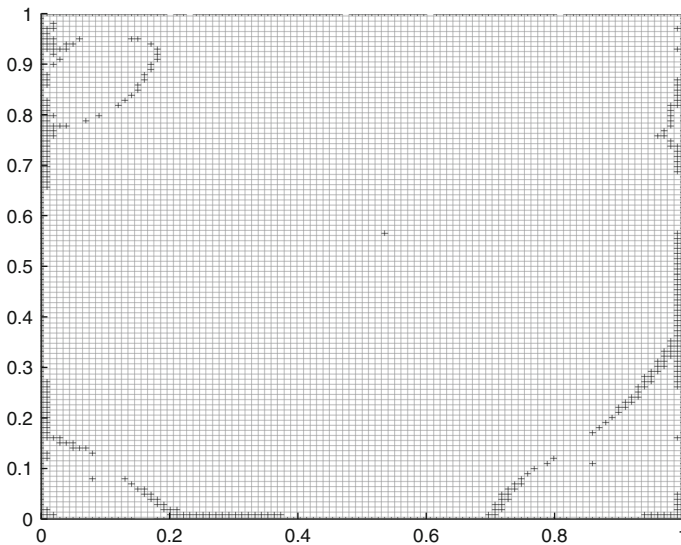
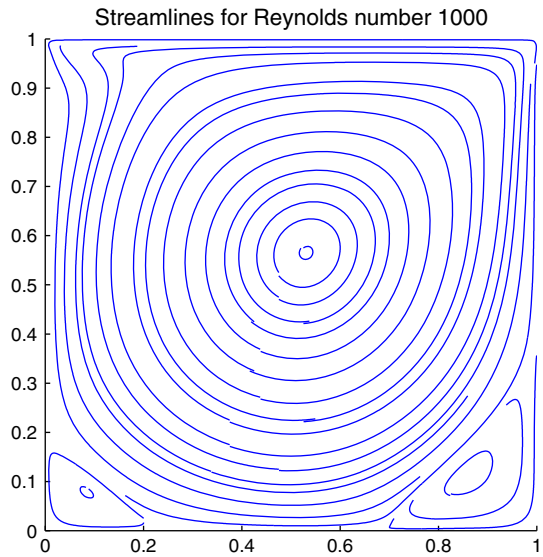


Fig. 4 Refined mesh for $Re = 1,000$ with mesh size 99×99

the vertical profile v of the numerical velocity field at $y = 0.5$ fits the corresponding benchmark well. The streamlines in the primary vortex region in Fig. 7 are almost closed with very small errors. If the errors come from the process of generating of streamline, we conclude that the velocity field \mathbf{V}_I satisfies Eq. (1) or f satisfies the condition (MC) and there is no refinement in the region. If the errors come from the numerical velocity field, there are refinements in the region. There is no refinement

Fig. 5 Horizontal profile of velocity field at $x = 0.5$ for mesh size 121×121

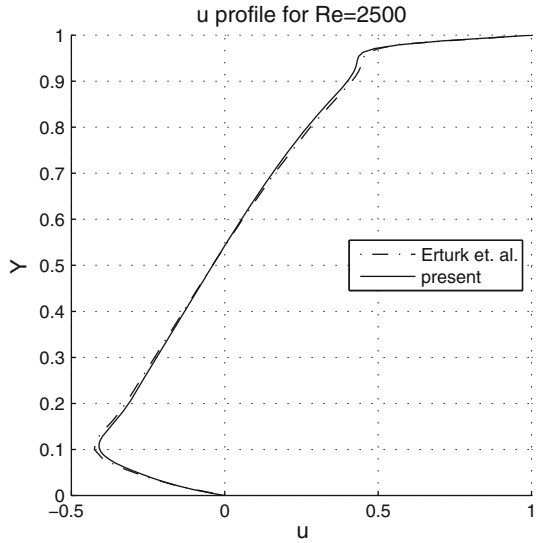
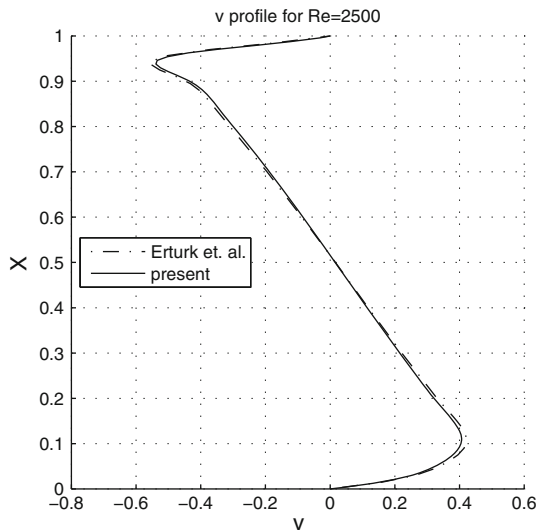


Fig. 6 Vertical profile of velocity field at $y = 0.5$ for mesh size 121×121



in the primary vortex region in Fig. 8 so the errors shown in Fig. 7 come from the generation of the streamlines. The difference between the coordinates of the centre of primary vortex from the benchmark and linearly interpolated velocity field \mathbf{V}_l is shown in Table 1. Even though the centre of primary vortex is not shown in Fig. 8 due to the accuracy of the linearly interpolated velocity field in the primary region, the centre of an extra tertiary vortex is shown in the bottom right corner. The three isolated refined grids in the two bottom corners include the centres of two secondary and one tertiary vortices (refer to Table 1).

Fig. 7 Streamlines for mesh size 121×121

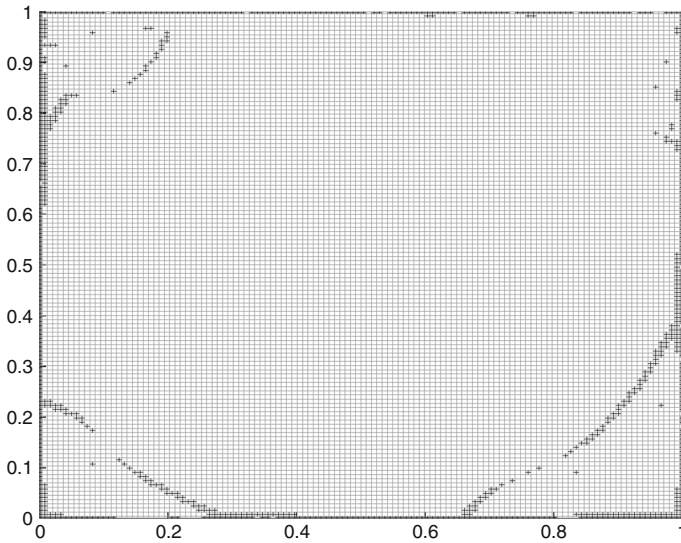
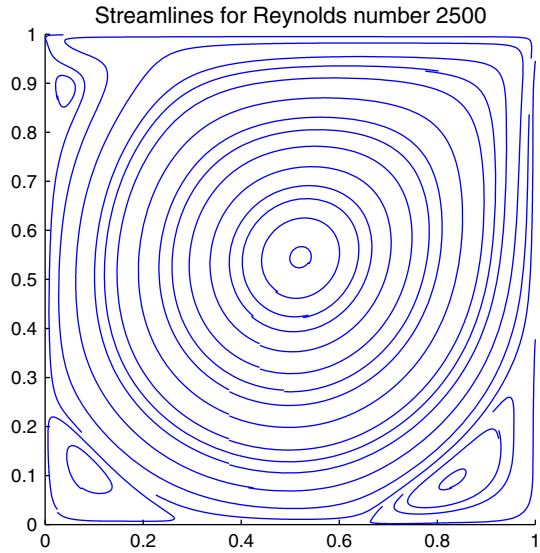


Fig. 8 Refined mesh for $Re = 2,500$ with mesh size 121×121

3.1.3 $Re = 5,000$

This section shows the figures for $Re = 5,000$ generated from a mesh with 139×139 uniform grids and the analysis. Figure 9 shows the difference between the horizontal profile u of the numerical velocity field at $x = 0.5$ and the corresponding benchmark. The difference is not small. From Fig. 10, the difference between the vertical profile v of the numerical velocity field at $y = 0.5$ and the corresponding benchmark is similar

Fig. 9 Horizontal profile of velocity field at $x = 0.5$ for mesh size 139×139

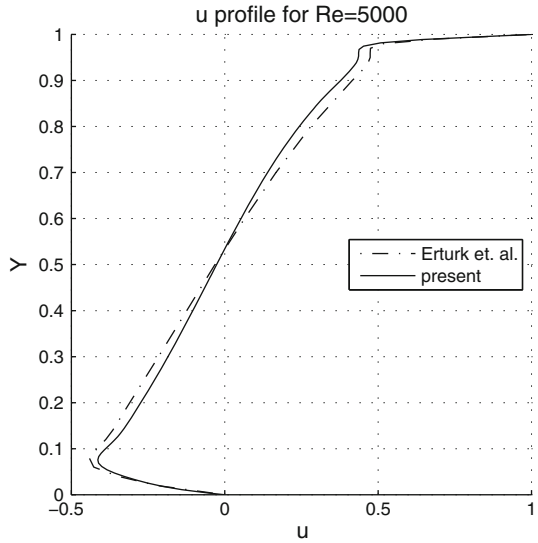
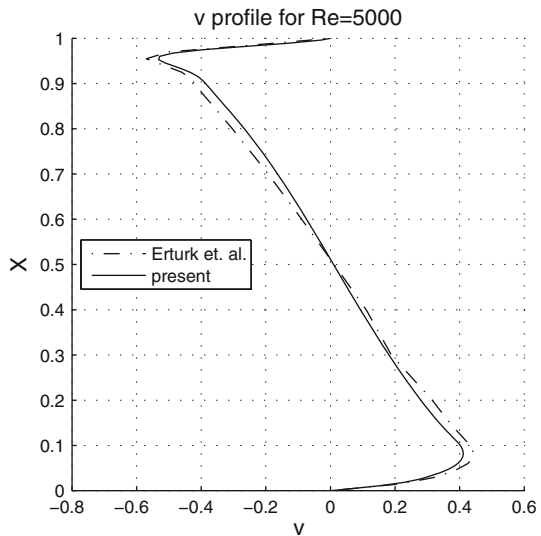


Fig. 10 Vertical profile of velocity field at $y = 0.5$ for mesh size 139×139



to that of the profile of u . The streamlines in the primary vortex region in Fig. 11 are also almost closed with very small errors. Combining the horizontal and vertical profiles of the velocity field with the streamlines in the primary vortex region, we conclude that the numerical velocity field is not accurate enough for a reasonable accuracy requirement. The accuracy can be improved by increasing the mesh size. Figure 12 confirms the conclusion. As the same case as $Re = 1,000$, there is a refined cell in the primary vortex region in Fig. 12. However, the refined mesh shows more complicated structure in the corners. The isolated refined grid in the primary region contains the

Fig. 11 Streamlines for mesh size 139×139

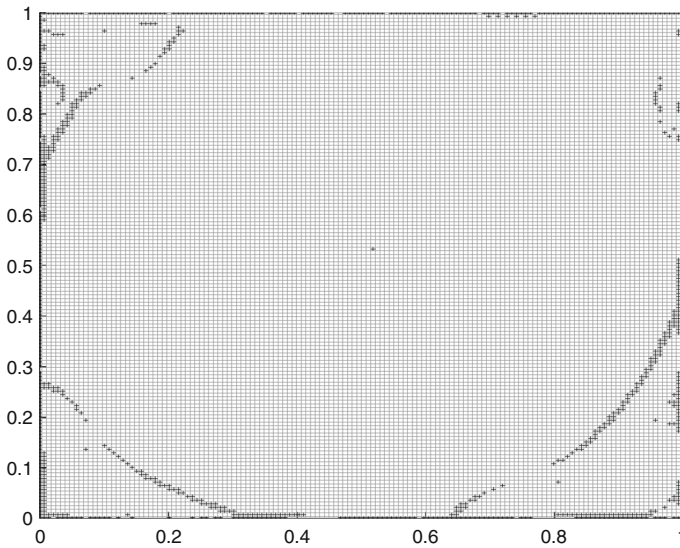
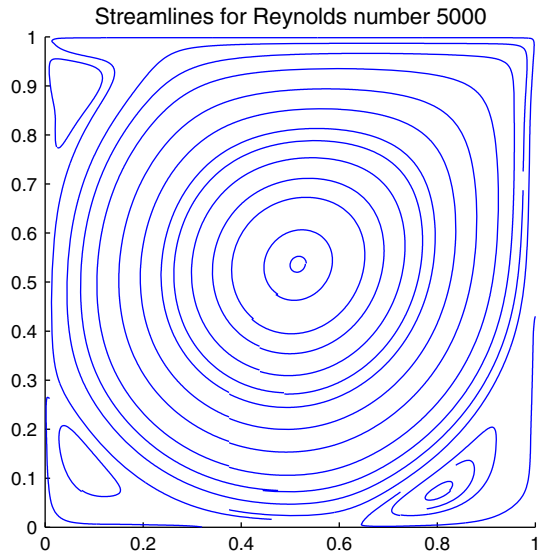


Fig. 12 Refined mesh for $Re = 5,000$ with mesh size 139×139

centre of primary vortex. The three isolated refined grids in the two bottom corners include the centres of two secondary and one tertiary vortices (refer to Table 1).

3.1.4 Vortex centre locations

This subsection shows the comparison of the centres of vortices between the benchmarks and the corresponding estimates obtained in this paper.

Table 2 Relative errors of the estimates of vortex centres

Vortex type	Reynolds numbers		
	Re = 1,000 (99 × 99)	Re = 2,500 (121 × 121)	Re = 5,000 (139 × 139)
Primary vortex	0.0024	0.0017	0.0012
BR1	0.0013	0.0039	0.0017
BL1	0.0063	0.0068	0.0023
BR2	–	0.0067	0.0045
BL2	0.5000	0.2957	0.1405
TL1	–	0.0010	0.0010
BR3	–	–	0.0588

Table 1 presents that coordinates of centres of vortices in the benchmark (the second line for each type of vortex) and the corresponding coordinates for $Re = 1,000$, $2,500$, and $5,000$ from the linearly interpolated velocity fields \mathbf{V}_l . In this table, the abbreviations BR, BL and TL refer to bottom right, bottom left and top left corners of the cavity, respectively. The numbers following these abbreviations refer to the vortices that appear in the flow, which are numbered according to size (e.g. BR1 refers to bottom right secondary vortex, and BR2 refers to bottom right tertiary vortex, etc.).

Even though the estimate for coordinates of the centre of primary vortex for $Re = 5,000$ is more accurate than that for $Re = 2,500$ because of the finer mesh, the streamlines in the primary vortex region for $Re = 5,000$ in Fig. 11 have larger error than those for $Re = 2,500$ in Fig. 7. The reason is that the coordinates of centres of the velocity fields are estimated by the coordinates of the centres of the linearly interpolated velocity fields \mathbf{V}_l . The coordinates of the centre of tertiary vortex in the bottom right corner for $Re = 1,000$ is not listed in Table 1 and not shown in refined grids in Fig. 4. Further refinement may find the centre as indicated in the examples of the analytical velocity fields [14]. Table 2 shows the relative errors of the estimated centre locations of vortices. Except for bottom left secondary vortices (BL2), all estimates are acceptable for relative error less than 1%. Further refinement is needed to find more accurate estimate locations of bottom left secondary vortices.

3.2 Refined grids containing centres of vortices

We take $Re = 2,500$ as an example to verify if the centres of vortices are contained in refined grids of refined mesh except the centre of the primary vortex. If the centres of vortices are included in refined grids, further refinements of the mesh will provide more accurate estimate locations of the centres. Figures 13, 14 and 15 show the sub plots of bottom left, bottom right and top left corners of refined mesh for $Re = 2,500$. The red dots are the centres given in the benchmark [7]. We conclude that the centres are contained in refined grids in these enlarged sub plots clearly. Figure 13 shows the refined mesh and the centres of vortices in region $[0, 0.3] \times [0, 0.3]$. The centre

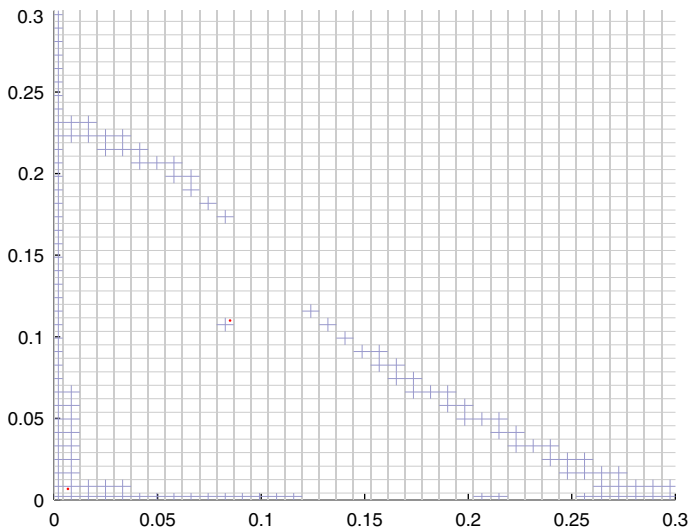


Fig. 13 Sub plot of *bottom left* corn of refined mesh for $Re = 2,500$

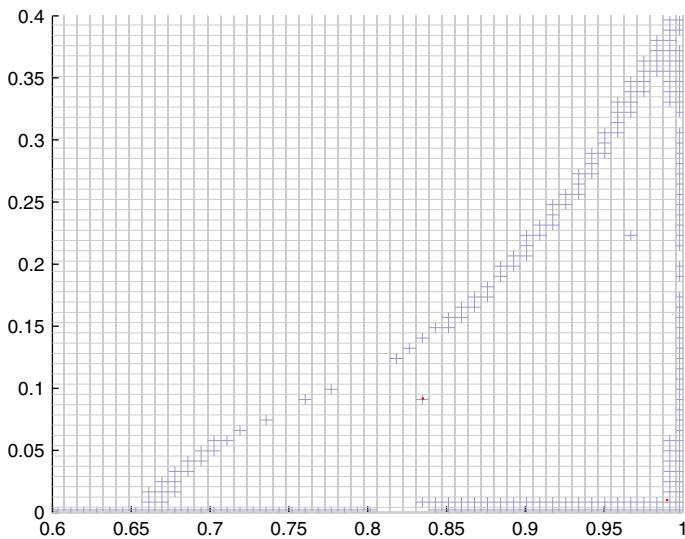


Fig. 14 Sub plot of *bottom right* corn of refined mesh for $Re = 2,500$

of the secondary vortex is not located at the centre of the refined grid. Figure 14 shows the refined mesh and the centres of vortices in region $[0.6 \ 1] \times [0 \ 0.4]$. The centre of the secondary vortex is almost located at the centre of the refined grid. Figure 15 shows the refined mesh and the centre of the secondary vortex in region $[0 \ 0.3] \times [0.7 \ 1]$. The centre of the secondary vortex is not located at the centre of refined grid.

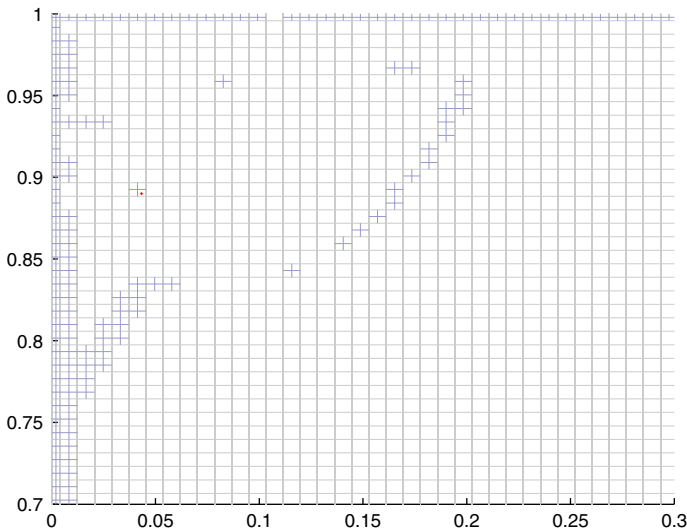


Fig. 15 Sub plot of *top left* corn of refined mesh for $Re = 2,500$

4 Discussion

We considered the adaptive mesh refinement method using three cases for two dimensional lid-driven cavity flows. We use horizontal and vertical profiles of velocity fields at $x = 0.5$ and $y = 0.5$ respectively and the streamlines generated by *Matlab* for determining the accuracy of the numerical velocity fields. We then consider whether the refined meshes can locate the centres of vortices. Besides the centre of primary vortex for $Re = 2,500$ which has been estimated accurately, the other centres of vortices locate in the refined grids in the refined meshes. Further refinement is necessary if we need more accurate coordinates of centres of BL2 with smaller relative errors for these three cases.

References

1. A.S. Almgren, J.B. Bell, P. Colella, L.H. Howell, M.L. Welcome, A conservative adaptive projection method for the variable density incompressible Navier–Stokes equations. *J. Comput. Phys.* **142**, 1–46 (1998)
2. J. Bell, M. Berger, J. Saltzman, M. Welcome, Three-dimensional adaptive mesh refinement for hyperbolic conservation laws. *SIAM J. Sci. Comput.* **15**, 127–138 (1994)
3. M.J. Berger, J. Olinger, Adaptive mesh refinement for hyperbolic partial differential equations. *J. Comput. Phys.* **53**, 484–512 (1984)
4. M.J. Berger, P. Colella, Local adaptive mesh refinement for shock hydrodynamics. *J. Comput. Phys.* **82**, 64–84 (1989)
5. P.J. Capon, P.K. Jimack, An adaptive finite element method for the compressible Navier-Stokes equations, in *Numerical Methods for Fluid Dynamics*, vol. 5, ed. by M.J. Baines, K.W. Morton (OUP, Oxford, 1995)
6. L. Demkowicz, J.T. Oden, W. Rachwicz, O. Hardy, An h–p Taylor–Galerkin finite element method for the compressible Euler equations. *Comput. Methods Appl. Mech. Eng.* **88**, 363–396 (1991)

7. E. Erturk, T.C. Corke, C. Gökcöl, Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers. *Int. J. Numer. Methods Fluids* **48**, 747–774 (2005)
8. S. Faure, J. Laminie, R. Temam, Colocated finite volume schemes for fluid flows. *Commun. Comput. Phys.* **4**, 1–25 (2008)
9. J.H. Ferziger, M. Peric, *Computational Methods for Fluid Dynamics*, 3rd edn. (Springer, Berlin, 2002)
10. R.D. Henderson, Adaptive spectral element methods for turbulence and transition, in *High-Order Methods for Computational Physics*, ed. by T.J. Barth, H. Deconinck (Springer, Berlin, 1999)
11. Z. Li, An adaptive streamline tracking method for two-dimensional CFD velocity fields based on the law of mass conservation. *J. Flow Vis. Image Process.* **13**, 1–14 (2006)
12. Z. Li, An adaptive streamline tracking method for three-dimensional CFD velocity fields based on the law of mass conservation. *J. Flow Vis. Image Process.* **13**, 359–376 (2006)
13. Z. Li, An adaptive three-dimensional mesh refinement method based on the law of mass conservation. *J. Flow Vis. Image Process.* **14**, 375–395 (2007)
14. Z. Li, An adaptive two-dimensional mesh refinement method based on the law of mass conservation. *J. Flow Vis. Image Process.* **15**, 17–33 (2008)
15. Z. Li, G. Mallinson, Mass conservative fluid flow visualisation for CFD velocity fields. *KSME Int. J.* **15**, 1794–1800 (2001)
16. Z. Li, R. Lal, Sensitivity analysis of a mesh refinement method using the numerical solutions of 2-D steady incompressible driven cavity flow. Submitted for publication
17. Z. Li, Accuracy analysis of an adaptive mesh refinement method using benchmarks of 2-D steady incompressible lid-driven cavity flows. Submitted for publication
18. R. Lohner, An adaptive finite element scheme for transient problems in CFD. *Comput. Methods Appl. Mech. Eng.* **61**, 323–338 (1987)
19. K. Miller, R. Miller, Moving finite elements. Part I. *SIAM J. Numer. Anal.* **18**, 1019–1032 (1981)
20. M.C. Moshier, A variable node finite element method. *J. Comput. Phys.* **57**, 157–187 (1985)
21. W. Speares, M. Berzins, A 3-D unstructured mesh adaptation algorithm for time-dependent shock dominated problems. *Int. J. Numer. Method Fluids* **25**, 81–104 (1997)
22. Y. Ye et al., *Theory of Limit Cycles* (American Mathematical Society Press, Providence, RI, 1986)
23. O.C. Zienkiewicz, D.W. Kelly, J.P. Gago, The hierarchical concept in finite element analysis. *Comput. Struct.* **16**, 53–65 (1983)