Development of Computer Program for Two-Dimensional Complex Geometry Flows

Natta Chunso[*1], Thara_Angskun[2], and Keerati Suluksna[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Suranaree University of Technology,
Nakhon Ratchasima, Thailand 30000
[2] Department of Information Technology, Faculty of Social Technology, Suranaree University of Technology,
Nakhon Ratchasima, Thailand 30000
* Corresponding Author: Tel. 044-224410, Fax: 044-224613,
E-mail: ahaaa_natta@hotmail.com

Abstract

This research proposes a CFD computer program for simulating the two-dimensional steady laminar flow based on finite volume method and triangular unstructured mesh. The numerical algorithm is developed based on SIMPLE algorithm for solving the Navier-Stokes equations, which are consisted of momentum equations and continuity equation in form of the pressure correction equation. The strategies of CDS and UDS schemes are respectively used to discretise the convection and diffusion terms of those governing equations. A lid driven cavity flow with Reynolds number of 100, 400, 1000, 3200 and 5000 is investigated to validate the developed computer program. The predicted results show that the in-house computer program gives good results compared with the reference data.

Keywords: Computational Fluid Dynamics, Finite Volume Method, Triangular Unstructured Mesh, Cell-Centered Grid

1. Introduction

Flow phenomena play an important role in many engineering applications such as flow around the vehicles, the ventilation of air inside the buildings. An understanding in their behaviors makes the engineers to optimize in designing the systems. Unfortunately, the governing equation of flow is controlled by a very complex form of the Navier-Stokes equations that are impossible to completely solve by using any analytical methods to obtain an exact solution. As a result, the numerical method called CFD (Computational Fluid

Dynamics) is taken into account for solving the flow problems by providing an approximation solution rather than those of using analytical method for the exact solution. Based on CFD concept, the flow domain will be divided into a finite number of the control volumes (CVs). The flow equations are then applied to all those CVs with the specified numerical schemes and hence a system of the algebraic equations is obtained. This can be solved numerically by means of computer programming. Generally, the technique of discretisation which is typically used in CFD process is the finite volume method (FVM). This

method is usually found as a strategy-based in developing of various popular CFD commercial software such as FLUENT, CFX, STAR CMM+, etc. Another shortcoming in solving the flow is the complexity of the domain shape which is usually found in many flow applications. With the complex geometry, it is very difficult or impossible to discretise such domain with using of regular mesh. As a result, the use of unstructured grid technique seems to be a suitable one of the choice here to overcome the problem. Due to the irregular shape of the unstructured mesh, however, the special treatments will be included in the solving process.

In the presented work, the numerical strategies and methodologies which are used to develop the computer program are described. The two-dimensional steady and incompressible flow is considered. The Navier-Strokes equations are discretised based on FVM and unstructured mesh. A technique of auxiliary node is adopted here to provide the orthogonal between the node-node connecting lines and the cell faces at the mid-face locations. The advantages of this technique are omitting of a cross diffusion term in the discretised equations resulting from the non-orthogonally effect and also the second order accuracy of the cell faces gradient calculation is preserved. The developed computer program has been validated with the driven cavity flow with Reynolds Number of 100, 400, 1000, 3200, and 5000, respectively. The predicted results have been taken to compare with the numerical results predicted by FLUENT software. It is found that the presented results give good agreement with those reference data.

2. The Numerical Strategies

For the two-dimensional flow under the conditions of steady state and incompressible, the flow equations are governed by the momentum equations and the continuity equation. Those can be written in general form of the transport equation as follow:

$$div\left(\rho \vec{V} \phi \right) = div\left(\Gamma . grad\left(\phi \right) \right) + S_\phi \quad (1)$$

Where ρ denote the fluid density, $\vec{V}$ the velocity vector, ϕ the flow variable, Γ the diffusion coefficient, and S_ϕ the source/sink term.

2.1 Finite Volume Method (FVM)

FVM is the discretisation technique for transforming the governing equation in PDE form to be the equation in algebraic form. This method takes integration of the governing equation over each control volumes and then considers the balancing of the fluxes through the surfaces of the control volumes as followed:

$$\int_{CV} div\left(\rho \vec{V} \phi \right) d\Omega$$
$$= \int_{CV} div\left(\Gamma grad \phi \right) d\Omega + \int_{CV} S_u \, d\Omega \quad (2)$$

Where $d\Omega$ is the control volume. The volume integral (Eq. (2)) will be transformed to be surface integral by using Gauss's theorem.

$$\int_S \hat{n} \cdot (\rho \vec{V} \phi) dS$$
$$= \int_S \hat{n} \cdot (\Gamma . grad \phi) dS + \int_{CV} S_\phi \, d\Omega \quad (3)$$

For the control volume with n faces, the Eq. (3) can be written as the summation of the properties for all those surfaces as follow:

$$\sum_{all \, surface} \hat{n}_i \cdot (\rho \vec{V} \phi_i) A_i$$
$$= \sum_{all \, surface} \hat{n}_i \cdot (\Gamma grad \phi) A_i + S_\phi \Omega \quad (4)$$

All terms in Eq. (4) will be implemented with the specified numerical schemes in order to discretise those terms to be the algebraic equation.

2.2 Discretisation of the Diffusion Term

From Eq. (4), the diffusion term has been approximated by using the central differencing scheme (CDS). However, the second order accuracy of this scheme will be provided only if it is treated in the direction of normal-to-surface. As a result, the normal gradient calculation is taken in to account for approximating as follow:

$$
\sum_{all\ surface} \Gamma(\hat{n}_i \cdot grad\phi)A_i
$$
$$
= \sum_{all\ surface} \Gamma A_i \left(\frac{\partial \phi}{\partial n}\right)_i \tag{5}
$$

From Fig. 1, it can be seen that the connected line between the centered nodes P and A is crossed the cell face line at point i. It is not lay through the mid-face location, point i′. This condition leading to a degrading of the accuracy in computation the gradient at the CVs surface and also produces instability in the calculation as well.
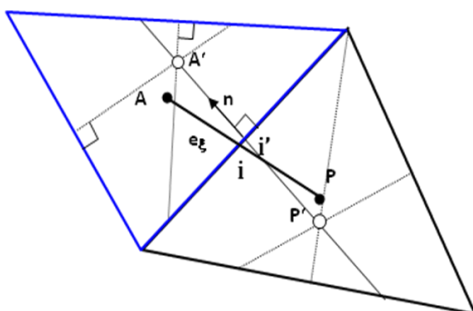

Fig. 1 Construction of the auxiliary nodes

To overcome this shortcoming, the technique of auxiliary node is adopted here. The auxiliary nodes P′ and A′ can be easily constructed by drawing the three straight lines

normally form the mid-faces inward the cell volume. The intersection between those lines is the location where the auxiliary node is placed. However, the technique of auxiliary node trend to fail if the twist angle of the connected line P-A with respect to the cell face is larger. This angle depends on the shape of the triangular cell. As a result, it is necessary to control the angles inside the triangular cell not exceed the range of 54-72 degree.

2.3 Discretisation of the Convection Term

Physically, the convection term is directly related to the mass flux through the cell face. To obtain the mass fluxes at each cell faces, the normal velocity at the cell face must be found. In the presented work, the technique of the so-called Rhie-Chow interpolation is adopted to apply for those normal velocities at cell faces:

$$
v_f = \overline{v_f} - \overline{\left(\frac{\Delta\Omega_f}{a_p}\right)_i}\left(\left(\frac{\partial p}{\partial n}\right)_{i'} - \overline{\left(\frac{\partial p}{\partial n}\right)_{i'}}^{old}\right) \tag{7}
$$

Obviously, it can be seen that the Rhie-Chow interpolation is formulated in terms of the pressure gradient in normal direction ($\partial p/\partial n$), the average value of the velocity at cell face, the average value of the cell volume and the central coefficient which is obtained from the discretised momentum equations.

2.4 Computation Procedures

The SIMPLE algorithm of Patankar and Spalding (1972) is employed to solve these flow equations here. The calculation procedure is started with initializing the field variables with a small value except for the pressure that is initiated with zero. To initiate the SIMPLE calculation process a pressure field is guessed

and employed to sequentially solve the momentum equation to yield the velocity field. The velocity field is used to determine the mass fluxes through each cell face and subjected to the constraint that it must satisfy the continuity equation. In this step, the Rhie-Chow interpolation is used to determine the mass fluxes. The pressure-correction equation is performed by using the mass imbalance arising from the incorrect velocity field as the source term so that the pressure-correction field can be obtained at all nodes. Once the pressure-correction field is know, the correct pressure and velocity fields can be obtained by updating them with the pressure correction.

During the SIMPLE iteration, the discretised equation is numerically solved by using the point relaxation of Gauss-Siedel method. The number of sweep for each equation solved should not be the same. Only one sweep is sufficient for momentum but for pressure the 5 sweeps are required. All variables will be weighted with the appropriate values of under-relaxation factor to avoid the solution wiggle, in this work the value of 0.2 is used to stabilize all variables except for pressure the value of 0.01 is used. The algorithm for the simulation can be summarized as follows:

(1) Initialize the velocity and pressure fields.

(2) Solve the momentum equations to yield velocity fields.

(3) Solve the pressure correction equation to yield pressure correction field.

(4) Correct the pressure and velocity by the pressure correction.

(5) Repeat steps (2)-(4) until the solution converges.

3. Computational Strategies

In this work, the developed computer program has been validated with the driven cavity flow. This flow is usually found as the simple problem for validating the computer code. The flow domain is constructed by a 1x1 m^2 square geometry as shown in Fig. 2. All boundaries are walls. The top wall is moved from left to right with the speed of 1 m/s and the rest walls are fixed. The cavity is full filled with the specified fluid. The fluid property is specified corresponding to the Reynolds number requirement.
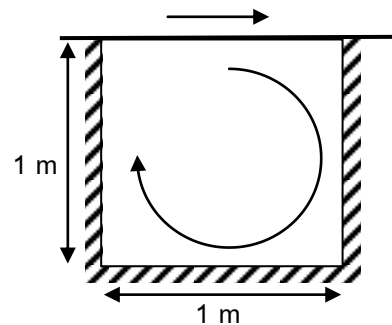

Fig. 2 Domain of a square cavity flow

The considered domain described above has been generated by using Gambit software. The domain has been discretised to the finite number of control volumes based on triangular unstructured mesh. In the process, each edges of the domain are divided with the number of 20 and the 902 CVs are obtained as shown in Fig. 3.
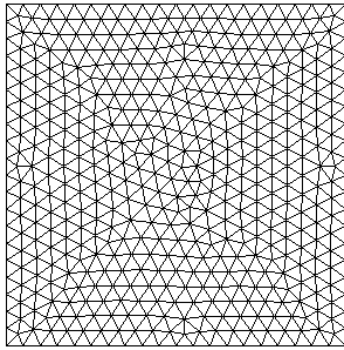
Fig. 3 Mesh configuration

4. Results and Discussion

Determine the fixed-length velocity vectors of the flow with Re=1000 as shown in Fig. 4. It can be seen that the fluid in the upper zone of the cavity is forced to drive from left to right due to the shear driven effect of the lid. The flow comes down in the right zone and then goes up in the left zone like the circulation. In the figure, there are two patterns of the circulations in the flow field, the primary and secondary circulations. The primary circulation moves around the cavity core with the vertex center of (0.62, 0.65). The secondary circulations with a smaller size are appears at the two bottom corners of the cavity.
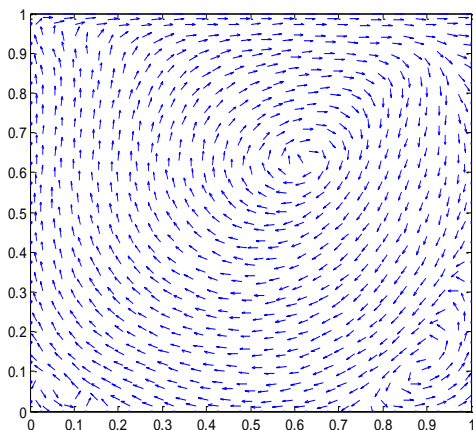
Fig. 4 Velocity vector of Re=1000

Figs. 5-9 display comparison of velocity on vertices from the presented work and Fluent software that the upper is a comparison of x-velocity and the lower is a comparison of y-velocity. The dashed-lines represent the presented results, and the solid lines come from Fluent. The presented results are satisfactory with Fluent software.
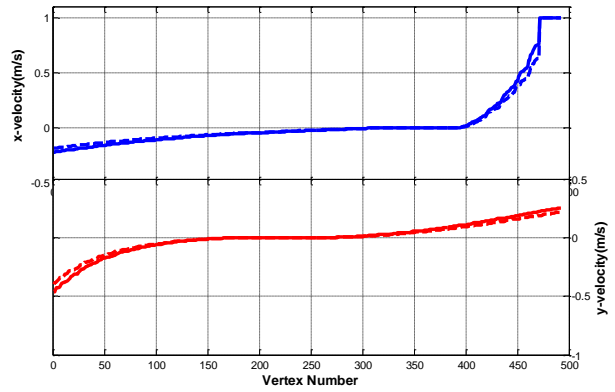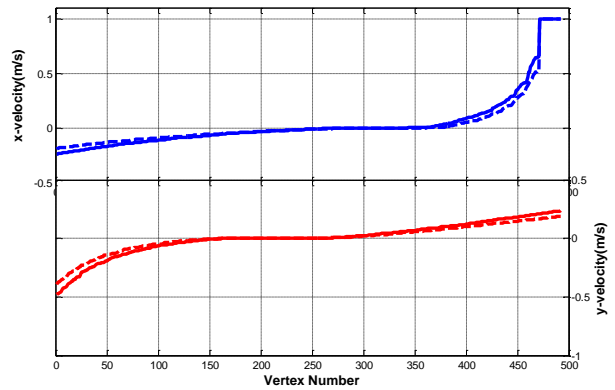

Fig. 5 Comparison of velocity at Re100
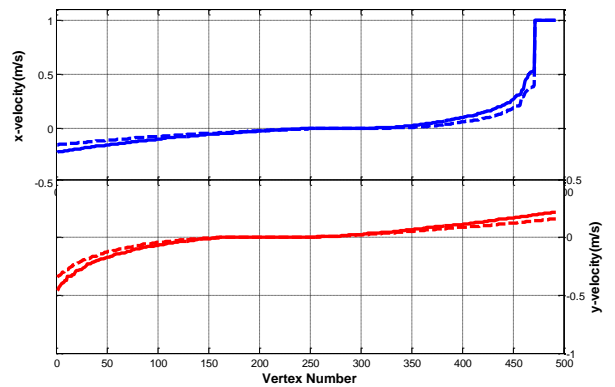

Fig. 6 Comparison of velocity at Re 400

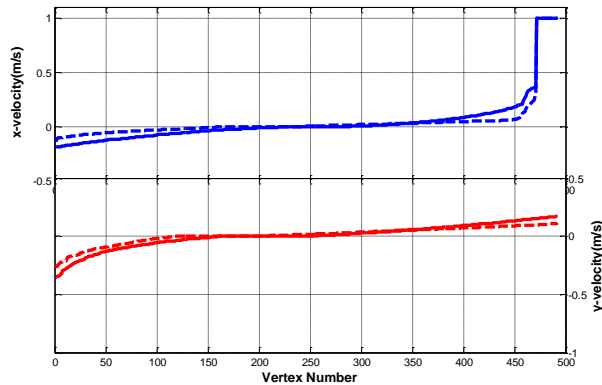Fig. 7 Comparison of velocity at Re 1000
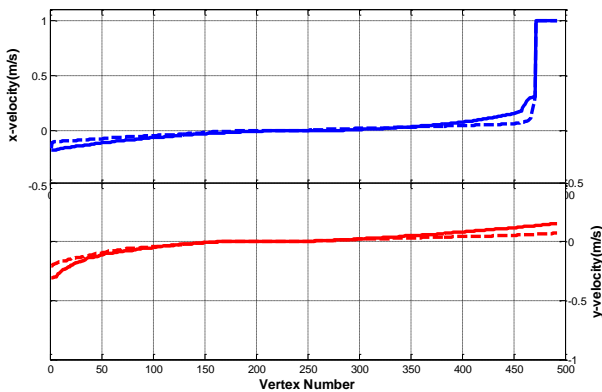
Fig. 8 Comparison of velocity at Re 3200

Fig. 9 Comparison of velocity at Re 5000

5. Conclusion

The research work presents the CFD in-housed software. The software is developed for the two-dimensional steady state and incompressible flow. The numerical technique of FVM and triangular unstructured mesh are implemented. The upwind differencing scheme is employed to approximate the convection and diffusion terms. The Rhie-Chow interpolation is used to calculate the mass fluxes throughout the cell faces. The shear driven cavity flow is used as valuable tool for validating the developed software. The results shown that the developed software gives the satisfactory results compared with Fluent.

6. Acknowledgement

7. Annotation

7.1 Symbols

A_i area of face

a_p cofactor of node P

$\hat{e}_\xi$ unit vector of node P to A

$\hat{n}_i$ outward unit normal vector of face

p pressure

u x-velocity

v y-velocity

$\vec{V}$ velocity vector

v_f normal face velocity

μ viscosity

Ω volume

ρ density

7.2 Subscript

f   considering on face

8. References

[1] Ghia, U., Ghia, K.N., Shin, and C.T.(1982). High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method, *Journal of Computational Physics*, vol. 48, 1982, pp. 387-411.

[2] J. H. Ferziger, M. Peric (2002), Computational Methods for Fluid Dynamics, 3rd edition, Springer.

[3] Suhas V. Patankar (1980), Numerircal Heat Transfer and Fluid Flow, Taylor&Francis Publishers.

[4] J. Blazek (2005), Computational Fluid Dynamics: Principles and Applications, 2nd edition, Elsevier Publisher.

[5] H. K. Versteeg, W. Malalasekera (2007), An Introduction Fluid Dynamics The Finite Volume Method, 2nd ed., Pearson Prentice Hall.

[6] Damrongsak Yambangwai, Finite Volume

Method For The Prediction Of Incompressible

Flow Through A Bounded Domain With

Specified Pressure As Boundary Condition,

Suranaree University of Technology, Thesis

Report (2008).