J. Basic. Appl. Sci. Res., 3(8)789-796, 2013
© 2013, TextRoad Publication

ISSN 2090-4304
Journal of Basic and Applied Scientific Research
www.textroad.com

Effect of Dependent Variable Selection on the Solution of Fluid Flow Problems in Complex Geometries

Abdollah Hassani

Islamic Azad University, South Tehran Branch, No. 209, Iranshahr St., Tehran, Iran

ABSTRACT

It is the aim of this paper to compare the effect of choosing different dependent variables in the solution of steady incompressible Navier-Stokes equations by, finite-volume method in general non-orthogonal curvilinear coordinates. A staggered grid system is used and comparison is made between cartesian cell face velocity components and covariant cell face velocity projections. The skewed lid driven cavity is used as the benchmark test case to analyze the effect of different schemes, considered. According to the results, Cartesian cell face velocity based method is a better choice when the grids are nearly orthogonal while the covariant cell face velocity projection based one is a better choice for highly non- orthogonal grids.
KEYWORDS: Body Fitted Coordinates, Staggered Grid, Cartesian, Covariant, Navier-Stockes.

INTRODUCTION

A very accurate method to capture the boundaries of complex geometries when dealing with fluid flow problems is body fitted non-orthogonal grids. With body fitted curvilinear coordinates, there are different choices available for pressure velocity coupling method (SIMPLE, SIMPLER, SIMPLEC, PISO,...), grid arrangement (staggered, non-staggered) and dependent variables in the momentum equations (cartesian, covariant, contravariant).

Demirdzic et al.[1]used finite volume method based on staggered grid in terms of contravariant velocity components. Rhie and Chow[2]applied the strong conservation form of the equations on a non-staggered grid system. Shyy et al.[3]used staggered grid system and one cartesian velocity at each cell face as the dependent variable. The use of non-staggered grid with cartesian velocity components as the dependent variables made the basis of FLOW- 3D code in 1987[4]. Demirdizic et al.[5] used the contravariant velocity with a staggered grid system. Karki and Patankar[6]and Davidson et al.[7]used the covariant components of velocity in a staggered grid system. Demirdizic et al.[8]used the cartesian velocity components together with a collocated grid arrangement. Shyy and Vu[9]concluded that using curvilinear velocity vectors introduce additional source terms in the governing equations while cartesian velocity vectors allows these equations to retain the full conservation form. Davidson and Farhanie[10]developed a code entitled CALC-BFC based on cartesian velocity components and non-staggered grid arrangement. Melaaen[11]made a comparison between two finite volume methods, one of them based on a non-staggered grid with cartesian velocity components and the other a staggered grid with covariant velocity components. Lee et al.[12]employed contravariant velocity based method in a non-staggered grid system to compute pulsatile flows. Choi et al.[13]compared the results of using covariant and contravariant velocity components in a non-staggered grid system. Tony et al.[14]used the contravariant velocity components in a staggered grid system. Xu and Zhang[15]investigated the effect of grid skewness on the under relaxation factor in a non-staggered grid system and compared the results in four different skew angles of 30, 45, 60 and 90^o. Wang and Komori[16]made a comparison between choosing cartesian and covariant velocity components on non-orthogonal collocated grids. Xu and Zhang[17]solved the steady laminar incompressible flow in a non-orthogonal non-staggered grid with contravariant velocity. In a similar work to what has done by Wang and Komori, Lai and Yan[18], concluded that there is no obvious difference on the convergence rate and accuracy by different cell face velocity selection. Shklyar and Arbel[19]extended the SIMPLE algorithm in general curvilinear coordinates in a way that two staggered grids were used to discretize the physical domain. Erturk and Dursun[20]solved the two dimensional steady incompressible Navier-Stokes equations in skewed cavity flow by non-orthogonal grid using a stream function-vorticity method. Cheng et al.[21]developed an improved SIMPER algorithm in non-staggered, non-orthogonal grid system named CLEARER. Traore et al.[22] developed a finite volume scheme on non-orthogonal mesh based on an

*Corresponding Author: Abdollah Hassani, Islamic Azad University, South Tehran Branch, No. 209, Iranshahr St., Tehran, Iran. E-mail: pr.hassani@gmail.com

iterative technique known as deferred correction. Sun et al.[23] implemented the IDEAL algorithm on a three dimensional collocated grid system. This algorithm uses a doubly-iterative process for pressure correction. Hwang[24] proposed a solution procedure for calculating incompressible Navier-Stokes equations on staggered polygonal grids. Pascau [25] extended the pressure-weighted interpolation method to unsteady flows. Singh and You[26] developed a second-order accurate finite volume method for a solution of the incompressible Navier-Stokes equations on complex multi-bock structured curvilinear grids. Cheng et al.[27] proposed an algorithm named Coupled and Linked Equation Algorithm Revised-ER(CLEARER) where, the second relaxation factor is introduced in constructing the contravariant interface velocities. It is claimed that this algorithm can deal with the grid non-orthogonality especially at high Reynolds numbers. Li et al[28] compared the performance of both momentum interpolation and linear interpolation in discretization of momentum and other scalar equations on a non-staggered grid system. Yu et al.[29] solved the Navier-Stokes equation by a finite-volume method based on non-staggered unstructured triangular grids. They showed that the convergence rate of the cell-vertex scheme is faster than that of the cell-centered scheme. Bu et al[30] compared two momentum interpolation methods in curvilinear non-staggered grids in single and two phase flows. Sun et al.[31] applied the previously described IDEAL algorithm to unsteady two phase flows. In a recent work, Shih et al.[32] conducted a comprehensive survey of the literature for numerical solution of heat transfer and fluid flow problems from 2010 to 2011.

In the present study, effects of cell face velocity selection in the two dimensional incompressible flow on non-orthogonal staggered grids is investigated. A comparison is made between cartesian cell face velocity components and covariant cell face velocity projections by applying the methods to a well-known test case, namely skewed lid driven cavity.

MATHEMATICAL DESCRIPTION

The steady state form of the 2-D governing equations in general curvilinear coordinates for the dependent variable ϕ can be written as [3]:

$$\frac{1}{J}\frac{\partial}{\partial\xi}(\rho U\phi)+\frac{1}{J}\frac{\partial}{\partial\eta}(\rho V\phi)=\frac{1}{J}\frac{\partial}{\partial\xi}\left[\frac{\Gamma}{J}\left(q_1\phi_\xi-q_2\phi_\eta\right)\right]+\frac{1}{J}\frac{\partial}{\partial\eta}\left\{\frac{\Gamma}{J}\left(-q_2\phi_\xi+q_3\phi_\eta\right)\right\} + S(\xi,\eta) \quad (1)$$

where,

$$U=uy_\eta-vx_\eta \qquad ,V=vx_\xi-uy_\eta$$
$$q_1=x_\eta^2+y_\eta^2 \quad ,q_2=x_\xi x_\eta+y_\xi y_\eta, \quad q_3=x_\xi^2+y_\xi^2 \quad ,J=x_\xi y_\eta-x_\eta y_\xi \quad (2)$$

U,V are the contravariant velocity components or velocity components along the (ξ,η) lines respectively, Γ is the diffusion coefficient, $S(\xi,\eta)$ is the source term of ϕ and q_1 to q_3 are metrics of transformation and *J* is the jacobian.

The continuity equation in generalized coordinates (ξ,η) can be written as:

$$\frac{\partial}{\partial\xi}(\rho U)+\frac{\partial}{\partial\eta}(\rho V)=0 \quad (3)$$

Discretization of governing equations

After dividing the physical plane into quadrilateral control volumes, the discretization equations are obtained by integrating the governing equations over the control volumes.

The resulting algebraic equations for the variable ϕ can be written in the following general form:

$$A_P\phi_P=A_E\phi_E+A_w\phi_w+A_N\phi_N+A_s\phi_S+b_\varnothing \quad (4)$$

where, E, W, N, and S refer to east, west, north and south neighbors of the central point P, respectively as shown in Fig. 1. $b_ø$ includes all the terms calculated explicitly including the source terms and the terms containing cross derivatives.

Integrating the continuity equation on each control volume results in:

$$(\rho U)_e-(\rho U)_w+(\rho V)_n-(\rho V)_S=0 \quad (5)$$

In curvilinear coordinates it is possible to use the cartesian, contravariant or covariant velocity components as the dependent variable *ø*. Although mathematically these options would yield the same result, in practical calculation with finite degree of numerical accuracy, these choices may not yield the same or even comparable results.

Momentum equations with cartesian velocity components

Equation 4 for the momentum equations with cartesian velocity components at the cell faces which, are shown in Fig. 2 takes the form:

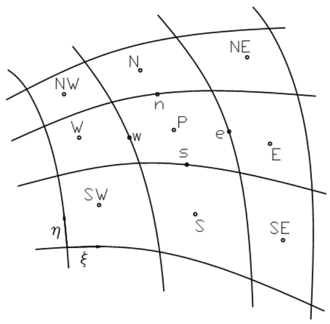


Fig.1 A control volume in curvilinear coordinates

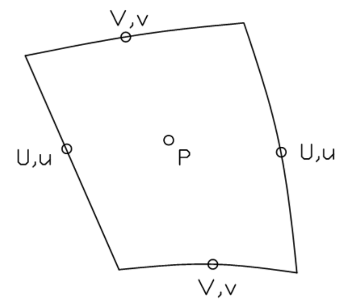


Fig. 2 Cell face velocity components

$$A_P^u u_P = \sum A_{nb}^u u_{nb} + S^u + \left(- y_\eta P_\xi + y_\xi P_\eta\right) \quad (6)$$

$$A_P^u v_P = \sum A_{nb} v_{nb} + S^v + \left(- x_\eta P_\xi + x_\xi P_\eta\right) \quad (7)$$

The main difference between the present formulation and that SIMPLE method described by Patankar[33]is the contribution of additional neighboring points (NE, NW, SE, and SW) due to the curvilinear coordinates selected and the need for correcting contravariant velocities in addition to cartesian velocities. By following the spirit of the SIMPLE algorithm as it is applied in cartesian coordinates, the velocity corrections at the neighboring points are neglected. Shyy et al.[3]found that dropping these terms led to a more efficient and stable algorithm because the resulting pressure correction equation (which becomes 5-Point instead of 9-Point) was easier to solve. The corrected contravariant velocity components, which are needed in continuity equation, are derived from corrected cartesian velocity components by equation(2). The final form of the pressure correction equation becomes:

$$A_P^P P_p' = \sum A_{nb}^p P_{nb}' + R \quad (8)$$

where, $R = U_e^* - U_w^* + V_n^* - V_s^*$ is the mass residual.

The iteration process for assessing the final converged solution follows the same procedure described by patankar[33].

Momentum Equations with Covariant Velocity Projection

The momentum equations for the covariant cell face velocity projections are obtained through algebraic manipulation of the momentum equations for the cell face cartesian velocity components. It must be noted that covariant velocities used in the equations are in fact projections, not vectors, while contravariant velocities are vector components.

The relationship between covariant velocity projections and cartesian velocity components is defined as follows:

$$u_\xi = \left(ux_\xi + vy_\xi\right)/\left(q_3\right)^{1/2} \quad , \quad v_\eta = (ux_\eta + vy_\eta)/(q_1)^{1/2} \quad (9)$$

Writing the momentum equations for each of the cartesian cell face velocities and substituting in the above equations, one can obtain the momentum equations for the cell face covariant velocities. For example, considering the east face of the control volume, by use of equations(6) and (7) we have:

$$A_e^u u_e = \sum A_{nb}^u u_{nb} + S^u + \left(- y_\eta P_\xi + y_\xi P_\eta\right) \quad (10)$$

$$A_e^v v_e = \sum A_{nb}^u v_{nb} + S^v + \left(x_\eta P_\xi - x_\xi P_\eta\right) \quad (11)$$

Substituting the above equations into equation4, one can obtain:

$$A_e u_{\xi,e} = H'_e + \left(BP_\xi\right)_e + \left(\frac{1}{\sqrt{q_3}} x_\xi S^u\right)_e + \left(\frac{1}{\sqrt{q_3}} y_\xi S^v\right)_e \tag{12}$$

where,

$$H'_e = \frac{x_\xi\left(\sum A^u_{nb} u_{nb}\right) + y_\xi\left(\sum A^v_{nb} v_{nb}\right)}{\sqrt{q_3}} \quad , \quad B = \frac{-J}{\sqrt{q_3}}$$

Velocity components retained in H'_e are a combination of cartesian velocities while the left hand side is in terms of covariant velocities. By use of a locally fixed coordinate system, adding and subtracting a locally constant term H_e to the right-hand side of Eq. (12), the result is[10]:

$$A_e u_{\xi,e} = H_e + \left(BP_\xi\right) + \left(\frac{1}{\sqrt{q_3}} x_\xi S^u\right)_e + \left(\frac{1}{\sqrt{q_3}} y_\xi S^v\right)_e + \left(H'_e - H_e\right) \tag{13}$$

where,

$$H_e = \sum A^u_{nb} u_{\xi,nb}$$

The last term of the right-hand side of the above equation is treated explicitly. A similar equation can be obtained for v_η.

Similar to the previous case, SIMPLE algorithm is employed to solve the above equations. The continuity equation must be written in terms of covariant velocities. The relation among covariant velocity projections and contravariant velocity components is defined as follows:

$$U = \alpha_\xi u_\xi - \beta_\xi u_\eta \quad , \quad V = \alpha_\eta u_\eta - \beta_\eta u_\xi \tag{14}$$

where,

$$\alpha_\xi = \frac{1}{J}(q_1)(q_3)^{\frac{1}{2}} \ , \ \beta_\xi = \frac{1}{J}(q_2)(q_1)^{\frac{1}{2}} \ , \ \alpha_\eta = \frac{1}{J}(q_3)(q_1)^{\frac{1}{2}} \ , \ \beta_\eta = \frac{1}{J}(q_2)(q_3)^{\frac{1}{2}}$$

Substituting the above equations into the continuity equation, one can obtain:

$$\left(\alpha_\xi u_\xi\right)_e - \left(\alpha_\xi u_\xi\right)_w + \left(\alpha_\eta v_\eta\right)_n - \left(\alpha_\eta v_\eta\right)_s = R1 \tag{15}$$

where, R1 is an additional term due to selecting covariant velocities as the dependent variable and is defined as:

$$R1 = \left(\beta_\xi v_\eta\right)_e - \left(\beta_\xi v_\eta\right)_w + \left(\beta_\eta u_\xi\right)_n - \left(\beta_\eta u_\xi\right)_s$$

Following the guidelines of the previous section, the pressure correction equation can be obtained as follows:

$$A_P P'_p = \sum A^p_{nb} P'_{nb} + R1 + R2 \tag{16}$$

where,

$$R_2 = \left(\alpha_\xi u_\xi\right)_w - \left(\alpha_\xi u_\xi\right)_e + \left(\alpha_\eta v_\eta\right)_s - \left(\alpha_\eta v_\eta\right)_n$$

The solution procedure is the same as the standard SIMPLE algorithm of Patankar[23].A FORTRAN code was developed to provide numerical solutions to this problem.

RESULTS AND DISCUSSIONS

The test case chosen for comparison is the steady two–dimensional flow inside a skewed (tilted) lid driven cavity. The problem is used to investigate the effects of non-orthogonality of the numerical grid on the convergence and stability of the solutions. The predicted results of the present methods are compared with the benchmark solution presented by Demirdzic et al.[34]and also a more recent research by Erturk et al.[20]. Demirdzic benchmark solution which is obtained for a highly dense 321x321 grid is accepted as a tool to check the accuracy and robustness of new solution methods which handle complex geometries. Erturk results which are obtained on a more refined 513x513 mesh agree well with the previous one. Their solution is for two dimensional incompressible, steady, laminar conditions. The domain of the problem is shown in Fig. 3.

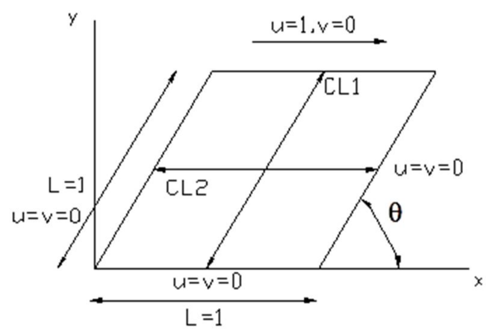


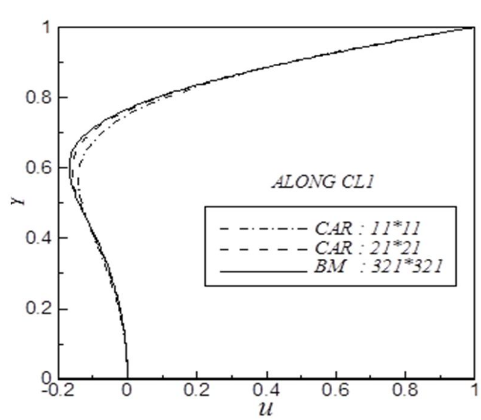
Fig. 3 Geometry for the test problem

In the following discussion and figures, the symbols "CAR" and "COV" and "BM" are used to denote the methods based on cartestian velocities, covariant velocities and benchmark solutions[34], respectively.

A comparison is made in Fig.4 and Fig. 5 for the velocity profiles along the horizontal and vertical centerlines of the cavity at Re=100 and θ=45˚. It can be seen that even by a 21*21 grid the difference between the results of both methods and the benchmark solution is very small. A similar comparison at θ=45˚ and at Re=1000 shows that the number of grid points required for assessing the converged solution in this case is increased, and even with a 81*81 grid, some differences between the results are evident. It is found that by increasing the number of grid points to 161*161, the results of both present methods converge to the benchmark solution. It must be mentioned here that the under relaxation factor[23]g the Reynolds number there is a need to more under-relax the momentum equation, especially in COV method, otherwise the method diverges. The relaxation factor for pressure correction equation[33]of the momentum equation used in the above two test cases are 0.7 and 0.4, respectively. In fact by increasing for both methods was constant at 0.3.

A comparison is also made for the effect of grid density at a fixed Reynolds number for the two methods at Re=1000 and θ=45˚. Almost in all cases the number of iterations for the CAR method is more than the COV method, but as before the time of computation is more for the latter. The convergence criterion is taken as 1E -7for the maximum residual of the continuity equation. It is found that grid density has a pronounced effect on the number of iterations, especially in the CAR method.

In order to investigate the effect of grid non-orthogonality on the convergence histories of the two methods, in Fig.6 the maximum residual of the continuity equation is shown as a function of iteration number. Two angles of 30 and 60 degrees are considered. The normalized CPU time required in each case is also shown in the figures.



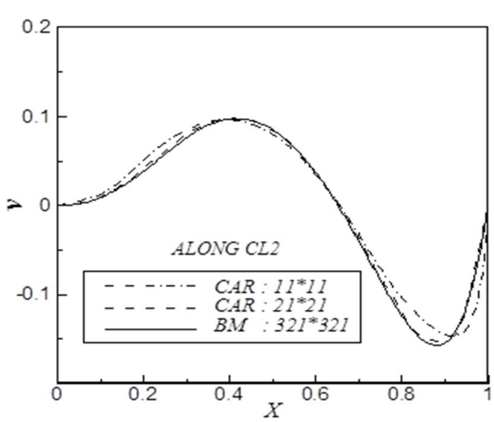


Fig. 4 Velocity profiles along *CL1* at Re=100, Θ=45°Fig. 5 Velocity profiles along *CL2* at Re=100, Θ=45°

According to the results, the number of iteration is always less in the case of covariant velocity method. But it is seen that the CPU time required for this method is generally more than the cartesian method. This is due to the more complex computations in the former method, which makes the time per iteration longer in comparison with the later.

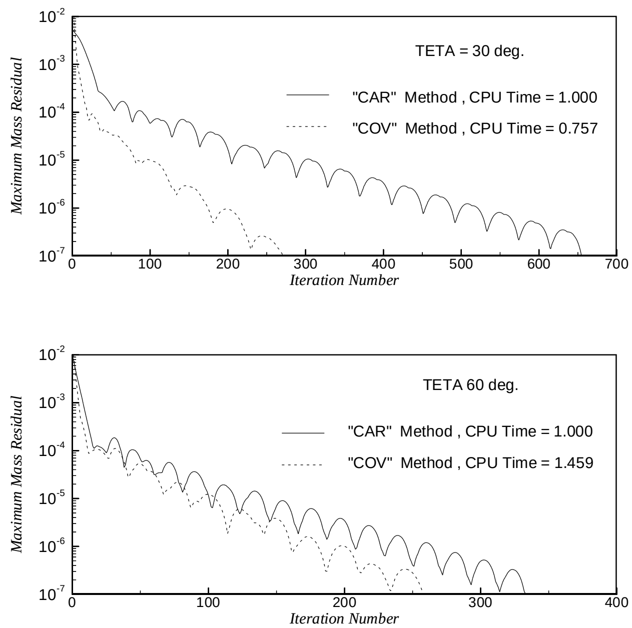


Fig. 6 Comparison of convergence rate at two different skewnessangle between cartesian and covariant methods

CONCLUSIONS

The relative performance of the solution scheme based on the covariant cell face velocities and the scheme based on the cartesian cell face velocities is compared through application to a test problem. The following conclusions are obtained from numerical calculations:

- The accuracy of the converged solution does not show any obvious difference between the various choices of cell face velocities. This is similar to the results shown by Wang and Komori[16]and Lai and Yan[18].

- In highly non–orthogonal grid systems, the number of iterations to attain a definite convergence criterionis always smaller for the covariant-based method. This also agrees with the results shown by Wang and Komori[16]. Lai and Yan[18],found that there was no obvious difference in convergence rate between the two methods but, it seems to be due to unavailability of their data for the 30 degree inclination angle where the difference arises.

- The time of one iteration for the covariant-based method is always more than the cartesian based method.

- The combinedeffect of the two previous results makes the total CPU time required for the COV method to be more than the one required for CAR method, when the grids are highly non-orthogonal. The reverse is true, when the grids are not highly non-orthogonal.

- When the Reynolds number is increased, the number of grids needed to attain the converged solution is increased quickly.

- Byincreasing grid number and skewness of the grids, the rate of convergence tends to decrease, sharply.

- In general, it seems that the CAR method is a better choice when, the grids are nearly orthogonal while, the COV method is a better choice for highly non- orthogonal grids.

ACKNOWLEDGMENT

The author would like to acknowledge the financial support of Islamic Azad University, South Tehran Branch (Under contract No. B/16/598) for this research.

REFERENCES

[1] Demirdzic, I. and Gosman, A. D. and Issa, R. I.1980. A Finite-Volume Method for the Predication of Turbulent Flow in Arbitrary Geometries. Lecture Notes in physics, 141: 141-150.

[2] Rhie, C. M. and Chow, W. L.1983. A Numerical Study of the Turbulentflow Past an Isolated Airfoil with Trailing Edge Seperation. AIAA J.,21(11): 1525-1532.

[3] Shyy, W. and Tong, S. S. and Correa, S. M. 1985. Numerical Recirculation Flow Calculation Using A Body-Fitted Coordinate System. Numerical Heat Transfer. 8: 99-113.

[4] Burns, A. D. and Wilkes, N. S.1987. A Finite Difference Method for the Computation of Fluid Flows in Complex Three Dimensional Geometries.ERE-R-12342.

[5] Demirdzic, I. and Gossman, A. D.and Issa, R. I. and Peric M.1987. A Calculation Procedure for Turbulent Flow in Complex Geometries.Comput. Fluids,15: 251-273.

[6] Karki, K. C. and Patankar, S. V.1988. Calculation Procedure for Viscous Incompressible Flows in Complex Geometry. Numerical Heat Transfer, 14: 295-307.

[7] Davidson, L. and Hedberg, P. L.1989. Mathematical Derivation of a Finite Volume Formulation for Laminar Flow in Complex Goemetries.Int. J Numerical Methods in Fluids, 9: 513-540.

[8] Demirdizic, I. and Peric, M.1990. Finite Volume Method for Prediction of Fluid Flow in Arbitrary Shaped Domains with Moving Boundaries. Int. J. Numerical Methods in Fluids, 10: 771-790.

[9] Shyy, W. and Vu, T. C., 1991. On the adoption of velocity variable and grid system for fluid flow computation in curvilinear coordinates. Journal of Computational Physics, 92(1): 82-105.

[10] Davidson, L. and Farhanieh, B.1992. CALC-BFC, A Finite-Volume Code Employing Collocated Variable Arrangement and Cartesian Velocity Components for Computation of Fluid Flow and Heat Transfer in Complex Three-Dimensional Geometries, Rept. 92/4 Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenberg.

[11] Melaaen, M. C.1992. Calculation of Fluid flows with Staggered and Non-Staggered Curvilinear Non-Orthogonal Grids, The Theory. Numerical Heat Transfer, 21(B): 1-19.

[12] Lee, D. and Chiu, J. J.1992. Covariant Velocity-Based Calculation Procedure with Non-Staggered Grids for Computation of Pulsatile Flows. Numerical Heat Transfer, 21(B): 269-286.

[13] Choi S. K. and Nam H. Y. and Cho M.1993. Use of the Momentum Interpolation Methods for Numerical Solution of Incompressible Flows in Complex Geometries. Numerical Heat Transfer, 23(B): 21-41.

[14] Tony W. H. and SheuLess S. M.1996. A Segregated Solution Algorithm For Incompressible Flows In General Coordinates. Int. J. Numerical Methods in Fluids, 22: 515-548.

[15] Xu, H. and Zhang, C. 1999. Study of the effect of the non-orthogonality for non-staggered grids-the results. Int. J. Numerical Methods in Fluids, 29:625-644.

[16] Wang Y. and Komori S.1999. Comparison of Using Cartesian and Covariant Velocity Components on Non-Orthogonal Collocated Grids. Int. J. Numerical Methods in Fluids, 31: 1265–1280.

[17] Xu, H. and Zhang, C. 2000. Numerical calculations of laminar flows using contravariant velocity fluxes. Computers and Fluids, 29: 149-177.

[18] Lai H. and Yan Y. Y.2001. The Effect of Choosing Dependent Variables and Cell Face Velocities on Convergence of the SIMPLE Algorithm Using Non-Orthogonal Grids. Int. J. Numerical Methods for Heat & Fluid Flow, 11(6): 524-546.

[19] Shklyar, A. and Arbel, A. 2003. Numerical method for calculation of the incompressible flow in general curvilinear co-ordiantes with double staggered grids. Int. J. Numerical Methods in Fluids, 41: 1273-1294.

[20] Erturk, E. and Dursum, B. 2005. Numerical solutions of 2-D Steady Incompressible flow in a driven skewed cavity. ZAMM Z. Angew. Math. Mech., 87(5): 377-392.

[21] Cheng, Y. P., Lee, T. S., Low, H. T. and Tao, W. Q. 2007. An efficient and robust numerical scheme for the SIMPLER algorithm on non-orthogonal curvilinear coordinates: CLEARER. Numerical Heat Transfer, Part B., 51: 433-461.

[22] Traore, P., Ahipo, Y. M., Louste, C. 2009. A robust and efficient finite volume scheme for the discretization of diffusive flux on extremely skewed meshes in complex geometries. Journal of Computational Physics, 228: 5148-5159.

[23] Sun, D. L., Qu, Z. G., He, Y. L., Tao, W. Q. 2009. Implementation of an efficient segregated algorithm-IDEAL on 3D collocated grid system. Chinese Sceince Bulletin, 54(6): 929-942.

[24] Hwang, Y. H. 2010. Development of a Compact and Accurate Discretization for Incompressible Navier-Stokes Equations Based on an Equation-Solving Solution Gradient, Part III: Fluid Flow Simulations on Staggered Polygonal Grids. Numerical Heat Transfer, 58(B): 193-215.

[25] Pascau, A. 2011. Cell face velocity alternatives in a structured collocated grid for the unsteady Navier-Stokes equations. International Journal for Numerical Methods in Fluids, 65: 812-833.

[26] Singh, S., You, D. 2011. A multi-block ADI finite-volume method for incompressible Navier-Stokes equations in complex geometries. Journal of Computational Physics, 230: 7400-7417.

[27] Cheng, Y. P., Lee, T. S., Sui, Y., Wang, L. P. Numerical simulation of 2D lid-driven cavity flow with CLEARER algorithm on extremely highly skewed grids at high Reynolds numbers. International Journal for Numerical Methods in Fluids, 65: 1201-1216.

[28] Li, W., Yu, B., Wang, X. R. and Sun, S. Y. 2012. Calculation of cell face velocity of non-staggered grid system. Applied Mathematics and Mechanics, 33(8): 991-1000.

[29] Yu, G., Yu, B., Zhao, Y., Wei, J. 2012. Comparative studies on accuracy and convergence rate between the cell-centered scheme and cell-vertex scheme for triangular grids. International Journal of Heat and Mass Transfer, 55: 8051-8060.

[30] Bu, L., Zhao, J., Tseng, K.J. 2012. Study on Momentum Interpolation Methods with Curvilinear Collocated Grids in Single-Phase and Two-Phase Flows. Numerical Heat Transfer, 61(B): 298 – 310.

[31] Sun, D. L.,Xu, J. L., Ding, P., Tao, W. Q. 2013. Implementation of the Ideal Algorithm on Unsteady Two-Phase Flows and Application Examples.Numerical Heat Transfer, 63(B): 204-221.

[32] Shih, T. M., Zheng, Y., Arie, M., Zheng, J. C. 2013. Literature survey of numerical heat transfer(2010-2011). Numerical Heat Transfer, 64(A): 435-525.

[33] Patankar, S. V. 1980. Numerical Heat Transfer and Fluid Flow, Hemisphere, Washington, DC.

[34] Demirdzic I., Lilek Z. and Peric M.1992.Fluid Flow And Heat Transfer Test Problems For Non-Orthogonal Grids:Bench-Mark Solutions. Int. J. Numerical Methods in Fluids, 15: 329-354.