Application of a Coordinate Transformation and Discretization Method for Computational Fluid Dynamics

Douglas F. Hunsaker
Utah State University

An overview of the computational methods implemented in a two-dimensional laminar flow solver is presented. The methods discussed include coordinate transformations making the code capable of solving flow in non-rectilinear domains, a discretization method implemented in the computational domain, and a pressure-coupling method which is used to enforce the continuity equation. Results of the numerical solver for laminar flow are presented and discussed.

I. INTRODUCTION

The author's research is focused on evaluating the behavior of a two-equation turbulence model proposed by Dr. Warren Phillips at Utah State University. Last year at this same conference, an overview of that model was presented by the author under the paper title, "Evaluation of an Alternate Incompressible Energy-Enstrophy Turbulence Model" (Hunsaker 2008). The model focuses on implementing transport equations of two transport properties of turbulent flow, namely turbulent kinetic energy and enstrophy. Many of the current turbulence models are based on the transport of dissipation, which is not a property of transport. Therefore, it is expected that this alternate turbulence model will perform better than many of the two-equation turbulence models currently in use. The purpose of the author's dissertation research is to evaluate this turbulence model.

It is expected that the alternate turbulence model will be tested in a numerical code written by the author. This code is currently under development using the programming language C++ and is called an Incompressible Computational Enstrophy Structured Solver (ICESS). The code is capable of solving two-dimensional flow problems in curvilinear coordinates for laminar flow and will eventually incorporate turbulent flow. The advantage of writing the code from scratch is that every aspect of the code will be controllable by the author. This paper is a very concise overview of the methods the author has implemented in this computational fluid dynamics (CFD) code and includes a discussion of the laminar results of the code.

II. COMPUTATIONAL METHOD

A. Coordinate System

Complex geometries are often modeled using a body-fitted coordinate system known as curvilinear coordinates. In two dimensions, this coordinate system is created by assuming that a computational domain (ξ,η) can be defined by a transformation of the physical domain (x,y) where

$$\begin{aligned} \xi &= \xi(x,y) \\ \eta &= \eta(x,y) \end{aligned} \tag{1}$$

This type of transformation allows the governing equations to be discretized and solved in the computational domain. This method has been widely used for CFD. See for example the work of Rhie (1981) or Shyy, Udaykumar, Rao, and Smith (1996). Details on this coordinate system can be found in Appendix B of the author's dissertation (Hunsaker 2009). Partial derivatives of any continuously differentiable scalar, ϕ, in the physical domain can be written as partial derivatives in the computational domain according to

$$\frac{\partial \phi}{\partial x} = \phi_{,x} = \frac{y_{,\eta}\phi_{,\xi} - y_{,\xi}\phi_{,\eta}}{J}$$
$$\frac{\partial \phi}{\partial y} = \phi_{,y} = \frac{x_{,\xi}\phi_{,\eta} - x_{,\eta}\phi_{,\xi}}{J} \tag{2}$$

where J is the Jacobian scalar of the coordinate transformation and is defined as

$$J \equiv x_{,\xi} y_{,\eta} - x_{,\eta} y_{,\xi} \tag{3}$$

Any scalar equation can be transformed from the physical domain to the computational domain by multiplying the equation by the Jacobian of the transformation. In this way, the two-dimensional steady-state general scalar transport equation can be written in curvilinear coordinates as

$$\frac{\partial}{\partial \xi}(\rho \overline{V}_{\xi} \phi) + \frac{\partial}{\partial \eta}(\rho \overline{V}_{\eta} \phi) = \frac{\partial}{\partial \xi}\left[\frac{\Gamma}{J}(q_{11}\phi_{,\xi} + q_{12}\phi_{,\eta})\right] + \frac{\partial}{\partial \eta}\left[\frac{\Gamma}{J}(q_{12}\phi_{,\xi} + q_{22}\phi_{,\eta})\right] + S_{\phi} \tag{4}$$

where Γ is the diffusion coefficient, S_{ϕ} represents the sources in curvilinear coordinates and

$$\overline{V}_{\xi} = f_{11}\overline{V}_x + f_{12}\overline{V}_y$$
$$\overline{V}_{\eta} = f_{21}\overline{V}_x + f_{22}\overline{V}_y \tag{5}$$

are the contravariant velocity components in the curvilinear coordinate plane. The quantities f_{11}, f_{12}, f_{21}, f_{22}, q_{11}, q_{12}, and q_{22}, are functions of the coordinate transformations. The source terms are converted from the physical domain to the computational domain in the same manner. The details of these transformations are included in Appendix B of the author's dissertation (Hunsaker 2009).

B. Solution Methods

1. General Transport Equations

The transport equations for all scalar transport properties are solved using the finite-volume method. A collocated grid arrangement is used and all flow properties are defined at the cell centers. Therefore, for any given cell, P, the flow properties of neighboring cells are defined at the E, W, N, and S (east, west, north, and south respectively) cell centers. Because the grid spacing in the computational domain is arbitrary, a grid spacing of unity is employed here. All boundary conditions are defined at nodes coincident with the edges of the boundaries such that all boundary cells have an area of zero. Figures 1 and 2 show the relationship between the physical domain and computational domain.

The convection terms are discretized to allow deferred correction between first-order upwinding and second-order central-differencing. All other terms in the equations are discretized in the computational domain using the second-order central differencing method. Appendix C of the author's dissertation (Hunsaker 2009) contains details on the discretization scheme employed including specifics on how boundary cells are treated. The discretized equations are solved using an iterative process which relaxes the new solution based on the previous solution and an under-relaxation factor. For work by others using similar solution schemes, see Rhie (1981), Versteeg and Malalasekera (1995), Ferziger and Peric (1996).

2. Pressure

To complete the solution process, the continuity equation must be coupled with the momentum equations. This is done here by implementing a well-known algorithm known as the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm. Details on this algorithm can be found in Appendix D of the author's dissertation (Hunsaker 2009). The method was first suggested by Patankar and Spalding (1972), and can be found in many CFD books. See for example Versteeg and Malalasekera (1995a). The method solves for the pressure distribution using a combination of the continuity equation momentum equations in a guess-and-correct manner. The method was originally suggested for staggered grid arrangements and can induce extreme pressure oscillations in collocated grid arrangements. Rhie and Chow (1983) suggest a correction to the way velocities are interpolated to cell faces in order to smooth out pressure fluctuations. This correction is implemented in the present work and is detailed in Appendix D of the author's dissertation (Hunsaker 2009).

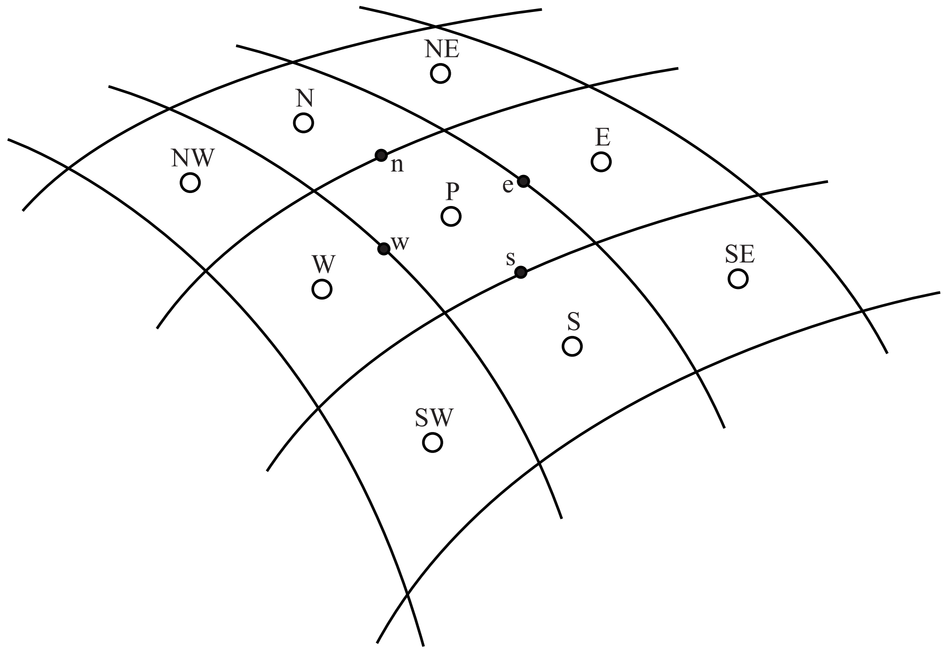

Figure 1. Grid discretization in physical domain.

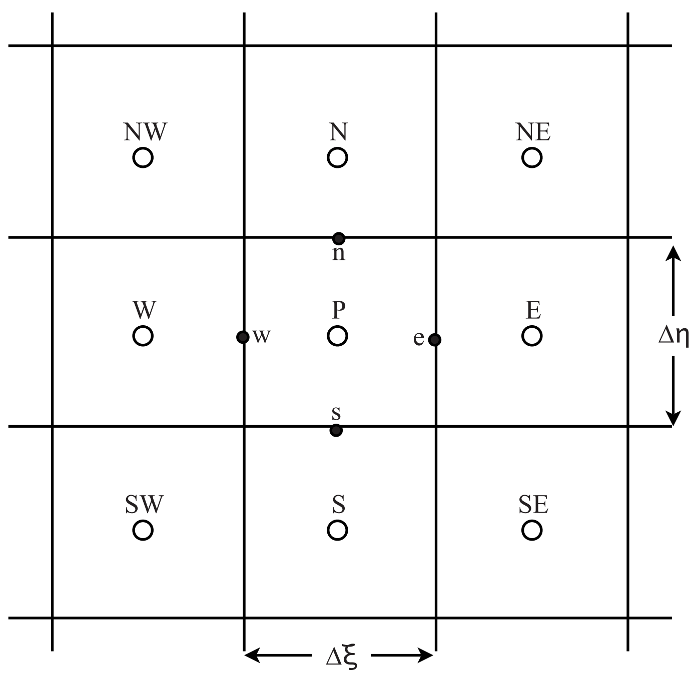

Figure 2. Grid discretization in computational domain.

3. Solution Procedure

The transport equations and the pressure corrections are solved in an iterative manner. Inner loop iterations on each of the transport equations are performed using relaxation factors. The pressure correction equation must be solved to a higher degree of accuracy than the transport equations, and thus 100 iterations are employed on the pressure equation while only 10 iterations are used on the transport equations. An outer loop sweeps through the inner-loop iterations in the following order: x-momentum, y-momentum, pressure (SIMPLE), turbulent kinetic energy, second differential equation (ε, ω, ζ, etc.). Before repeating the process, the turbulent viscosity, often referred to as the closing equation, is recalculated. Iterations of the outer loop are repeated until the residuals of each equation fall below specified criteria.

III. PRELIMINARY RESULTS

This section contains preliminary results of the code written by the author for this project. The code implements the solution methods presented above and has been tested for laminar flow implementing the curvilinear coordinate system. Richardson extrapolation (Richardson 1910, Richardson and Gaunt 1927) has been used to estimate the results for a fully grid-converged solution from the solutions of coarse, medium, and fine grids. The Richardson extrapolation method followed in this work is the method presented in the *Journal of Fluids Engineering* "Statement on the Control of Numerical Accuracy" (http://journaltool.asme.org/Content/JFENumAccuracy.pdf). The results of the code are compared with results from Fluent and OpenFOAM.

A case often used for testing the initial phases of a CFD code is the lid-driven cavity case. Figure 3 shows the results from ICESS for the x-velocity along the vertical centerline of the lid-driven cavity at a Reynolds number of 100. In this case, grid sizes of 2500, 10000, and 40000 cells were used for the coarse, medium, and fine grids respectively. All grids had uniform spacing.

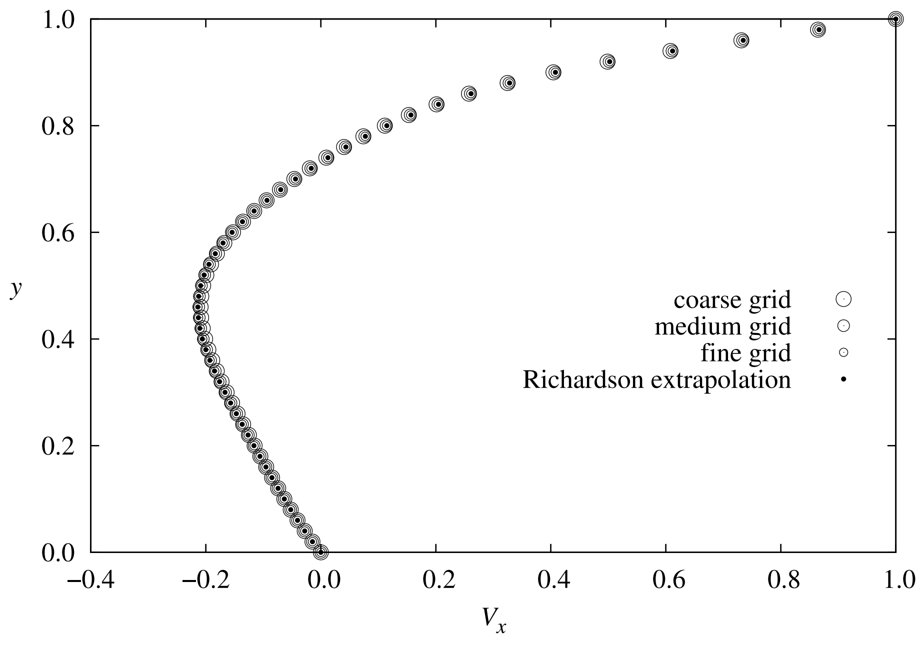

Figure 3. ICESS grid refinement results of the *x*-velocity profile along the vertical centerline of the lid-driven cavity at a Reynolds number of 100.

To test the implementation of the curvilinear coordinate transformation, the lid-driven cavity case was rotated 45 degrees and a grid-refinement study was again conducted. The case was run at the same Reynolds number using the same grid sizes. Figure 4 shows the Richardson extrapolation solutions for the x-velocity along the vertical centerline of the cases for both the unrotated and rotated cases. The results suggest that the curvilinear coordinate transformation implemented in the code is working correctly.

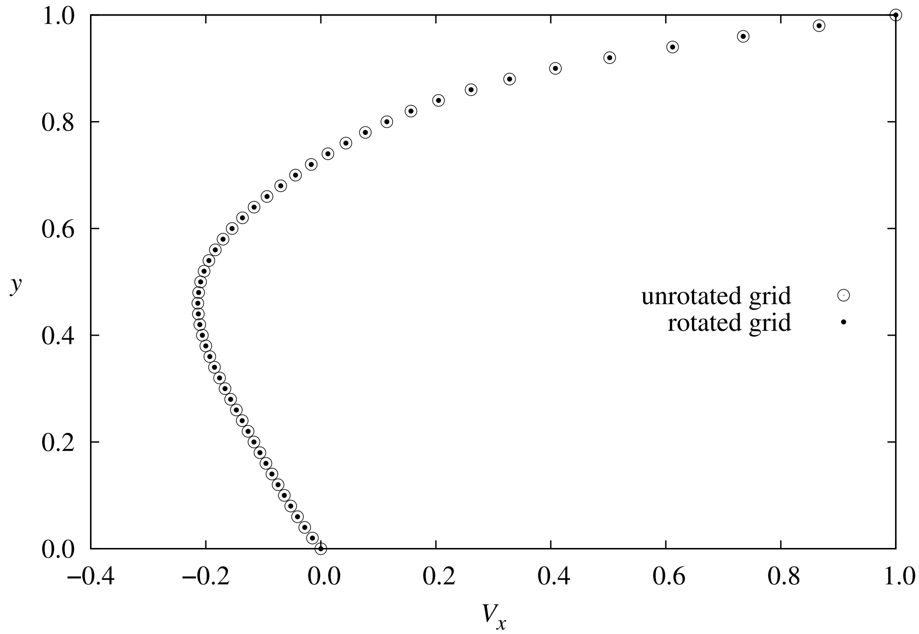

Figure 4. Richardson extrapolation results for the *x*-velocity profile along the vertical centerline of the lid-driven cavity for an unrotated case and a case rotated by 45 degrees.

Grid refinement studies for the lid-driven cavity case at a Reynolds number of 100 were also conducted in the Fluent and OpenFOAM software packages using the same grids as those used for the grid-refinement study in the author's code. Figure 5 shows the extrapolated results from Fluent, OpenFOAM, and ICESS. The results suggest that the laminar portion of the author's code is working correctly.

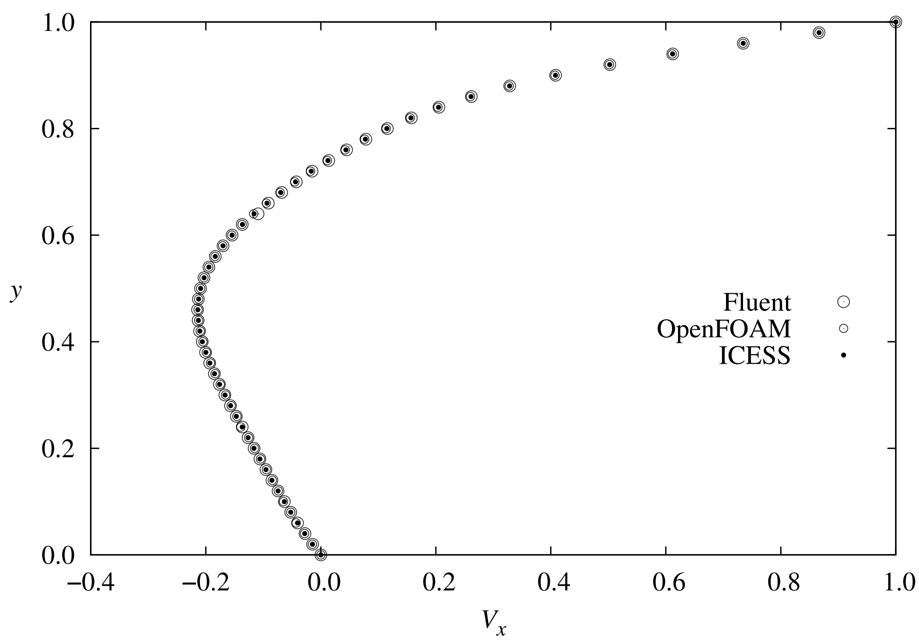

Figure 5. Richardson extrapolation results for the *x*-velocity profile along the vertical centerline of the lid-driven cavity. Comparison of results from Fluent, OpenFOAM, and ICESS.

Notice that at $y = 0.64$, the Richardson extrapolation from Fluent in Figure 5 seems inconsistent with the others. This is likely caused by the low precision to which Fluent reported the solutions for the grids.

Although Fluent calculated the solution using double precision numbers, the results were reported to less than single precision accuracy. The Richardson extrapolation is very sensitive to differences between solutions. When the differences between solutions are small and the solutions are reported only to low precision, the differences calculated in the Richardson algorithm are very similar between the coarse and medium solutions and the medium and fine solutions. This similarity in differences causes the Richardson extrapolation to estimate that the grid is still far from converged, and the final grid-resolved solution to lie significantly outside of the reported results.

IV. CONCLUSIONS

A computer code has been developed capable of solving two-dimensional laminar flowfields. The code has been validated against two commercially-available software packages and has shown to be accurate at least for the lid-driven cavity case. Further work will include expanding the code to make it capable of solving two-dimensional turbulent flow. The final work will evaluate the performance of an alternate turbulence model suggested by Dr. Warren Phillips.

REFERENCES

Ferziger, J. H., and Peric, M. (1996), "Finite Volume Methods," *Computational Methods for Fluid Dynamics*, Springer-Verlag Berlin Heidelberg, pp. 71–84.

Hunsaker, D. F. (2008), "Evaluation of an Alternate Incompressible Energy-Enstrophy Turbulence Model," Proceedings of the Rocky Mountain Space Grant Consortium Symposium, Salt Lake City, Utah, May 2008.

Hunsaker, D. F. (2009), "Evaluation of an Alternate Incompressible Energy-Enstrophy Turbulence Model," Ph.D. Dissertation DRAFT, Department of Mechanical and Aerospace Engineering, Utah State University.

Patankar, S. V. and Spalding, D. B. (1972), "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-dimensional Parabolic Flows," *Int. Journal of Heat Mass Transfer*, Vol. 15, p. 1787.

Rhie, C. M. (1981) "A Numerical Study of the Flow Past an Isolated Airfoil with Separation," Ph.D. Thesis, Dept. of Mechanical and Industrial Engineering, University of Illinois at Urbana-Champaign.

Rhie, C. M., and Chow, W. L. (1983), "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," *AIAA Journal*, Vol. 21, No. 11, Nov. 1983, pp. 1525–1532.

Richardson, L. F. (1910), "The Approximate Arithmetical Solution by Finite Differences of Physical Problems Involving Differential Equations, with an Application to the Stresses in a Masonry Dam," *Transactions of the Royal Society of London*, *A*, Vol. 210, Jan., pp. 307–357.

Richardson, L. F., and Gaunt, J. A. (1927), "The Deferred Approach to the Limit," *Transactions of the Royal Society if London*, *A*, Vol. 226, Jan., pp. 299–361.

Shyy, W., Udaykumar, H. S., Rao, M. M., and Smith, R. W. (1996), "Governing Equations and Solution Procedure," *Computational Fluid Dynamics with Moving Boundaries*, Taylor and Francis, Philadelphia, PA, pp. 21–23.

Versteeg, H. K., and Malalasekera, W. (1995), "The Finite Volume Method for Convection-Diffusion Problems," *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson Education Limited, Harlow, England, pp. 134–178.

Versteeg, H. K., and Malalasekera, W. (1995a), "The Finite Volume Method for Convection-Diffusion Problems," *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson Education Limited, Harlow, England, pp. 186–190.