

# A Finite Difference Calculation Procedure for the Navier-Stokes Equations on a Staggered Curvilinear Grid

R. M. Barron and B. Zogheib

**Abstract**—A new numerical method for solving the two-dimensional, steady, incompressible, viscous flow equations on a Curvilinear staggered grid is presented in this paper. The proposed methodology is finite difference based, but essentially takes advantage of the best features of two well-established numerical formulations, the finite difference and finite volume methods. Some weaknesses of the finite difference approach are removed by exploiting the strengths of the finite volume method. In particular, the issue of velocity-pressure coupling is dealt with in the proposed finite difference formulation by developing a pressure correction equation in a manner similar to the SIMPLE approach commonly used in finite volume formulations. However, since this is purely a finite difference formulation, numerical approximation of fluxes is not required. Results obtained from the present method are based on the first-order upwind scheme for the convective terms, but the methodology can easily be modified to accommodate higher order differencing schemes.

**Keywords**—Curvilinear, finite difference, finite volume, SIMPLE.

## I. INTRODUCTION

TWO-dimensional viscous incompressible flow equations are usually expressed in one of two different formulations, based on the dependent variables used. First is the primitive variable formulation, in which the equations are expressed in terms of the pressure and velocity. The second form of the equations is the vorticity-streamfunction equations which are derived from the Navier-Stokes equations by incorporating the definitions for the vorticity and streamfunction.

Over the last quarter century there has been much research devoted to the numerical solution of the incompressible Navier-Stokes (N-S) equations reported in the literature, see for example [1]-[7]. While there are finite difference, finite element and finite volume methodologies available, the majority of the fundamental research work in this field has been based on finite difference formulations, whereas the

finite volume method dominates commercial codes used in industry.

Primitive variables methodologies that are used to simulate the incompressible Navier-Stokes equations, in which mass conservation is used to derive an equation for the pressure field, invariably solve on a staggered grid arrangement. The scalar properties such as pressure are located at a different set of grid points than the velocity components. This staggered arrangement is a well known approach and has been used successfully with a variety of methodologies [8]-[10]. The main reason for choosing this arrangement is that it prevents odd-even coupling or what is known as checkerboarding between the pressure field and the velocity fields [9].

Finite difference primitive variable formulations have been used with success by different researchers, such as [10]-[12]. For the case of the incompressible Navier-Stokes equations written in primitive variables, the steady state solution is obtained either by taking an unsteady solution to the limit of large time, or by "directly" solving the steady equations. In most solution schemes for incompressible steady flows, the pressure field is obtained from a Poisson equation which is derived from the momentum equations and the continuity equation. However, this gives rise to issues concerning the proper boundary conditions for pressure. Unsteady methods include the work of [13], also known as the artificial compressibility method, the fractional step method [14], and the SOLA algorithm [15]. Chorin's method [13], which is the basis for many current finite difference formulations for incompressible flow (eg., [16]), avoids the problems associated with the Poisson pressure equation by introducing a time derivative of pressure into the conservation of mass equation.

In the finite volume world, the popular SIMPLE algorithm, see [17], uses a segregated solution technique in which the pressure field and velocity fields are solved for separately within an iteration cycle (i.e. a complete sweep of the flow field). The necessary pressure-velocity coupling for the satisfaction of mass conservation is attained through the solution of a "pressure correction" equation, derived from the continuity equation by means of certain simplifying assumptions.

In the present work, the new finite difference scheme developed in [18] to solve the Navier-Stokes equation on a

R. Barron is with the Department of Mathematics and Statistics, University of Windsor, Windsor, ON N9B3P4 Canada.

B. Zogheib is with the Division of Math, Science, and Technology, Nova Southeastern University, Fort Lauderdale, FL 33314USA (Phone: 954-262-7931; e-mail: zogheib@nova.edu)

Cartesian grid is extended to a curvilinear grid. The basic idea is that the method is implemented with a SIMPLE-type algorithm for the pressure field calculation similar to that used in finite volume methods, whereas the discretized equations are developed as a purely finite difference formulation. The convective terms in the momentum equations are approximated using first-order upwind finite difference formulae. There are several established higher-order schemes, such as QUICK [19], SMART [20], VONOS [21] and CUBISTA [22], which have been implemented in finite volume formulations to approximate cell face values needed to evaluate the integral fluxes. However, these schemes do not carry over to finite difference methods. Higher-order finite difference approximations for the first derivatives in the convective terms could be used, but the present results are accurate enough to justify the main conclusion of our work, ie., within the family of finite difference methods, this new approach is a viable alternative for handling the pressure compared to the artificial compressibility and Poisson equation methods.

## II. NUMERICAL PROCEDURE

### A. Governing Equations

The Navier-Stokes equations for the two-dimensional, steady, incompressible, viscous flow in terms of curvilinear coordinates  $(\xi, \eta)$ , in the non-conservative dimensional form, are

$$\xi_x u_\xi + \eta_x u_\eta + \xi_y v_\xi + \eta_y v_\eta = 0 \quad (1)$$

$$\begin{aligned} (u\xi_x + v\xi_y) u_\xi + (u\eta_x + v\eta_y) u_\eta = \\ -\frac{\xi_x}{\rho} p_\xi - \frac{\eta_x}{\rho} p_\eta + \nu \left\{ (\xi_x^2 + \xi_y^2) u_{\xi\xi} \right. \\ \left. + (\eta_x^2 + \eta_y^2) u_{\eta\eta} + 2(\xi_x \eta_x + \xi_y \eta_y) u_{\xi\eta} \right. \\ \left. + (\xi_{xx} + \xi_{yy}) u_\xi + (\eta_{xx} + \eta_{yy}) u_\eta \right\} \end{aligned} \quad (2)$$

$$\begin{aligned} (u\xi_x + v\xi_y) v_\xi + (u\eta_x + v\eta_y) v_\eta = \\ -\frac{\xi_y}{\rho} p_\xi - \frac{\eta_y}{\rho} p_\eta + \nu \left\{ (\xi_x^2 + \xi_y^2) v_{\xi\xi} \right. \\ \left. + (\eta_x^2 + \eta_y^2) v_{\eta\eta} + 2(\xi_x \eta_x + \xi_y \eta_y) v_{\xi\eta} \right. \\ \left. + (\xi_{xx} + \xi_{yy}) v_\xi + (\eta_{xx} + \eta_{yy}) v_\eta \right\} \end{aligned} \quad (3)$$

where  $u$  and  $v$  are the velocity components in the  $\xi$  and  $\eta$  direction respectively,  $p$  is the pressure,  $\rho$  is the constant density,  $\nu$  is the viscosity  $\xi_x$ ,  $\xi_y$ ,  $\eta_x$  and  $\eta_y$  are the metrics of transformation [23].

### B. Differencing Scheme

In the present work, the first order upwind differencing scheme is used to approximate the convective terms in the momentum equations, while second order central differencing is used for the diffusion terms. It is not argued here that first order upwinding is superior to other schemes. It has been used for its simplicity. Higher order schemes, which should give more accurate results, can be easily implemented into the procedure developed in this paper as shown in [18] on a Cartesian grid.

### C. Discretized Equations

A staggered grid is used to store the velocity components  $u$  and  $v$  and the pressure  $p$ . The variables  $u$  and  $v$  are stored at the  $i-1, j$  and  $i, j+1$  locations respectively and  $p$  is stored at  $i, j$ . Thus, the  $u$ -momentum equation (2) is discretized at  $i-1, j$ , the  $v$ -momentum equation (3) at  $i, j+1$ , and the continuity equation (1) at  $i, j$ . In general form, using finite volume notations, the finite difference equations can be expressed as

$$\begin{aligned} a_P^{\text{int}} u_{i-1,j} + a_N^{\text{int}} u_{i-1,j+2} + a_S^{\text{int}} u_{i-1,j-2} + a_W^{\text{int}} u_{i-3,j} \\ + a_E^{\text{int}} u_{i+1,j} = \xi_x \frac{\hat{p}_{i-2,j} - \hat{p}_{i,j}}{2\rho} + \hat{S}_u^{\text{int}} \end{aligned} \quad (4)$$

$$\begin{aligned} b_P^{\text{int}} v_{i,j+1} + b_N^{\text{int}} v_{i,j+3} + b_S^{\text{int}} v_{i,j-1} + b_W^{\text{int}} v_{i-2,j+1} \\ + b_E^{\text{int}} v_{i+2,j+1} = \eta_y \frac{\hat{p}_{i,j} - \hat{p}_{i,j+2}}{2\rho} + \hat{S}_v^{\text{int}} \end{aligned} \quad (5)$$

Where

$$\begin{aligned} \hat{S}_u^{\text{int}} &= \frac{\eta_x (\hat{p}_{i-2,j-2} + \hat{p}_{i,j-2} - \hat{p}_{i-2,j+2} - \hat{p}_{i,j+2})}{8\rho} \\ &+ \nu \left( \frac{\xi_x \eta_x + \xi_y \eta_y}{8} \right) (\hat{u}_{i+1,j+2} - \hat{u}_{i-3,j+2} - \hat{u}_{i+1,j-2} + \hat{u}_{i-3,j-2}) \\ \hat{S}_v^{\text{int}} &= -\frac{\xi_y (\hat{p}_{i+2,j+2} + \hat{p}_{i+2,j} - \hat{p}_{i-2,j} - \hat{p}_{i-2,j+2})}{8\rho} \\ &+ \nu \left( \frac{\xi_x \eta_x + \xi_y \eta_y}{8} \right) (\hat{v}_{i+2,j+3} - \hat{v}_{i-2,j+3} - \hat{v}_{i+2,j-1} + \hat{v}_{i-2,j-1}) \end{aligned}$$

The sub-index “E” in the coefficient  $a_E^{\text{int}}$  means that the coefficient is evaluated at the east neighbour of a  $u$ -node and “int” means at interior nodes, i.e., away from the boundaries. The same notation follows for the other coefficients. The caret above a variable indicates quantities calculated at the previous iteration.

Introducing corrections to the approximate values of  $u$ ,  $v$  and  $p$ , and following a procedure similar to SIMPLE, which is commonly used in finite volume formulations, the finite difference equation for the pressure correction  $p'$  is

$$\frac{\xi_x}{2} \left\{ u_{i+1,j}^* + \frac{\xi_x}{\rho} \frac{p_{i,j}' - p_{i+2,j}'}{2a_{i+1,j}} \right\} - \left\{ u_{i-1,j}^* + \frac{\xi_x}{\rho} \frac{p_{i-2,j}' - p_{i,j}'}{2a_{i-1,j}} \right\} + \frac{\eta_y}{2} \left\{ v_{i,j+1}^* + \frac{\eta_y}{\rho} \frac{p_{i,j}' - p_{i,j+2}'}{2b_{i,j+1}} \right\} - \left\{ v_{i,j-1}^* + \frac{\eta_y}{\rho} \frac{p_{i,j-2}' - p_{i,j}'}{2b_{i,j-1}} \right\} = -\eta_x \hat{u}_\eta \Big|_{i,j} - \xi_y \hat{v}_\xi \Big|_{i,j}$$

where  $u^*$  and  $v^*$  are the current iterates of the velocity. This equation can be written as

$$c_p^{\text{int}} p_{i,j}' + c_E^{\text{int}} p_{i+2,j}' + c_W^{\text{int}} p_{i-2,j}' + c_N^{\text{int}} p_{i,j+2}' + c_S^{\text{int}} p_{i,j-2}' = \xi_x \frac{u_{i-1,j}^* - u_{i+1,j}^*}{2} - \eta_y \frac{v_{i,j+1}^* - v_{i,j-1}^*}{2} + \hat{S}_p^{\text{int}} \quad (6)$$

Where

$$\hat{S}_p^{\text{int}} = -\eta_x \left( \frac{\hat{u}_{i+1,j+2} + \hat{u}_{i-1,j+2} - \hat{u}_{i+1,j-2} - \hat{u}_{i-1,j-2}}{8} \right) - \xi_y \left( \frac{\hat{v}_{i+2,j+1} + \hat{v}_{i+2,j-1} - \hat{v}_{i-2,j+1} - \hat{v}_{i-2,j-1}}{8} \right)$$

Because of the use of a staggered grid, the values of  $v$  in the  $\xi$  momentum equation and  $u$  in the  $\eta$  momentum equation, appearing as coefficients of the convective derivatives, are not available at the desired points. These velocities are computed using the four surrounding grid points at which they are stored.

#### D. Boundary Conditions

Boundary conditions are problem dependent, and are easily implemented in this formulation. For example, in the backward facing step problem, the following boundary conditions are applied; no slip boundary conditions are applied at the walls simply by setting velocity equals to zero. Inlet velocity values are prescribed and a parallel flow is applied at the outlet. The boundary conditions for  $p'$  are treated in a similar way as in a finite volume procedure.

#### E. Matrix Solver

The discretized equations (4)-(6) for  $u$ ,  $v$  and  $p$  are solved using successive line relaxation and the tri-diagonal matrix algorithm [9].

### III. RESULTS

Different Reynolds numbers in the case of backward facing step are compared against numerical and experimental results. The flow is simulated on a mesh where clustering is imposed close to the lower wall boundary. The results obtained are in

good agreement with those found in literature. To confirm that the results obtained are independent of grid resolution, successive mesh refinement is tested. For the flow over a backward facing step at  $Re = 200$ , the reattachment length and number of iterations obtained using  $201 \times 41$  and  $401 \times 81$  nodes are almost identical. Typical streamline patterns for these flows are shown in Fig. 1.

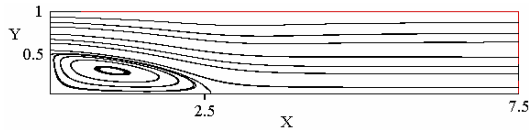


Fig. 1 Streamlines for flow over a backward facing step;  $Re = 200$

In the absence of an exact reference solution, the results obtained in [24] have been taken as the benchmark, as recommended in [25] for the flow in a complex channel. Cliffe, Jackson, and Greenfield [24] used a finite element method in primitive variables, a Newton-Raphson linearization scheme and the frontal solution method for the resulting linear system. The results are also compared to those obtained in [26] who used a velocity-vorticity approach in streamfunction coordinates to solve this problem. Fig. 2 shows the vorticity at the wall using the proposed method compared to those found in [24] and [26]. Fig. 3 shows pressure at the wall using the proposed method compared to those found in [24] and [27]. The circulation zone obtained by using FLUENT is shown in Fig. 4. The circulation zones obtained by FLUENT and the proposed method, Fig. 5, are almost identical.

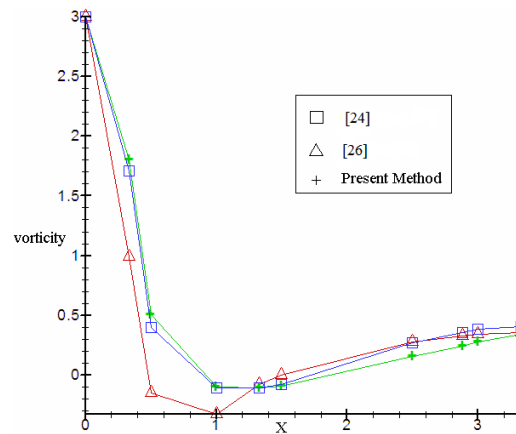
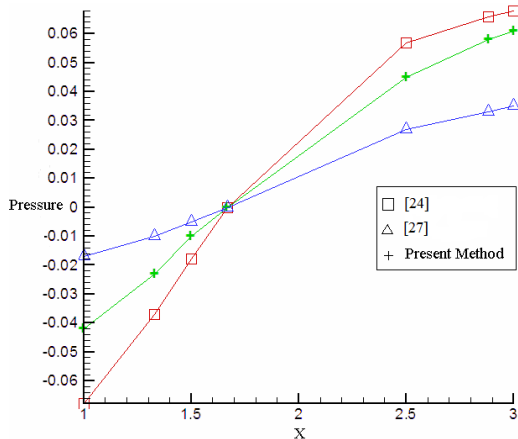
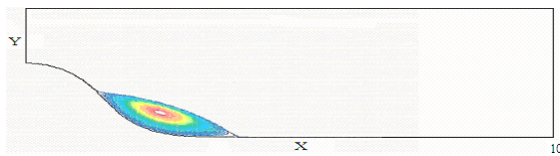
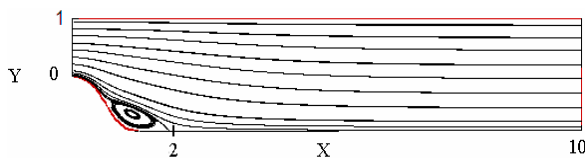


Fig. 2 Vorticity along the lower wall,  $Re = 10$

Fig. 3 Pressure along the lower wall,  $Re = 10$ Fig. 4 Streamlines for  $Re = 10$  using FLUENTFig. 5 Streamlines for  $Re = 10$  using present method

#### IV. CONCLUSIONS

This paper presents a new numerical algorithm for solving the two-dimensional, steady, incompressible, viscous flow equations on a staggered curvilinear grid by essentially taking advantage of the best features of the two well-established numerical formulations, the finite difference and finite volume methods. Under the finite difference umbrella, a velocity-pressure correction scheme has been developed, analogous to the SIMPLE method in finite volume formulations.

The numerical methodology developed in this paper can be easily extended to second orders and higher. It can be also extended to three-dimensional domains, and to the modeling of more complicated physical phenomena, such as unsteady turbulent flows.

#### REFERENCES

- [1] T. F. Miller and F. W. Schmidt, "Use of a pressure-weighted interpolation method for the solution of the incompressible Navier-Stokes equations on a non-staggered grid system," *Numerical Heat Transfer*, vol. 14, pp. 213-233, 1988.
- [2] G. D. Thiart, "Finite difference schemes for the numerical solution of fluid flow and heat transfer problems on nonstaggered grids," *Numerical Heat Transfer*, vol. 17, Part B, pp. 43-62, 1990.
- [3] I. E. Barton and R. Kirby, "Finite difference scheme for the solution of fluid flow problems on non-staggered grids," *International Journal for Numerical Methods in Fluids*, vol. 33, pp. 939-959, 2000.
- [4] Z. Tian and Y. Ge, "A fourth-order compact finite difference scheme for the steady stream function-vorticity formulation of the Navier-Stokes/Boussinesq equations," *International Journal for Numerical Methods in Fluids*, vol. 41, pp. 495-518, 2003.
- [5] M. Ben-Artzi, J. P. Croisille, and D. Fishelov, "Trachtenberg S. A pure-compact scheme for the streamfunction formulation of Navier-Stokes equations," *Journal of Computational Physics*, vol. 205, pp. 640-664, 2005.
- [6] A. K. De and K. Eswaran, "A high-order accurate method for two-dimensional incompressible viscous flows," *International Journal for Numerical Methods in Fluids*, vol. 53, pp. 1613-1628, 2007.
- [7] B. Zhu, "Finite volume solution of the Navier-Stokes equations in velocity-vorticity formulation," *International Journal for Numerical Methods in Fluids*, vol. 48, pp. 607-629, 2005.
- [8] F. H. Harlow and J.E. Walsh, "Numerical Calculations of Time-dependent Viscous Incompressible Flow of Fluid with Free Surface," *Physics of Fluids*, vol. 8, pp. 2182-2189, 1965.
- [9] S. Patankar, "Numerical Heat Transfer and Fluid Flow", Hemisphere, New York, 1980.
- [10] S. V. Patankar and D.B. Spalding, "A Calculation Procedure for Heat and Mass Transfer in Three Dimensional Parabolic Flows," *Int. J. Heat and Mass Transfer*, vol. 15, pp. 1787-1806, 1972.
- [11] W. R. Briley and H. McDonald, "Analysis and Computation of Viscous Subsonic Primary and Secondary Flows," *AIAA Paper* 79-1453, 1979.
- [12] K. N. Ghia and J.S. Sokhey, "Laminar Incompressible Viscous Flow in Curved Ducts of Rectangular Cross Sections," *J. of Fluids Engineering*, vol. 99, pp. 640-645, 1977.
- [13] A. J. Chorin, "A Numerical Method for Solving Incompressible Viscous Flow Problems", *J. of Computational Physics*, vol. 2, 12-26, 1967.
- [14] J. Kim and P. Moin, "Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations," *J. of Computational Physics*, vol. 59, pp. 308-323, 1985.
- [15] C. W. Hirt, B. D. Nichols and N.C. Romero, "SOLA - A Numerical Solution Algorithm for Transient Fluid Flow," *Los Alamos Scientific Laboratory Report* LA-5852, 1975.
- [16] C. H. Tia and Y. Zhao, "A finite volume unstructured multigrid method for efficient computation of unsteady incompressible viscous flows," *International Journal for Numerical Methods in Fluids*, vol.46, pp. 59-84, 2004.
- [17] S. Patankar, "A Calculation Procedure For Two-Dimensional Elliptic Situations," *Numerical Heat Transfer*, vol. 4, pp. 409-425, 1981.
- [18] B. Zogheib and R. Barron, "Velocity-Pressure Coupling in Finite Difference Formulations for the Navier-Stokes Equations," Accepted for publication, *International Journal for Numerical Methods in Fluids*, to be published.
- [19] B. P. Leonard, "A stable and accurate convective modelling procedure based on quadratic upstream interpolation," *Computational Methods in Applied Mechanical Engineering*, vol. 19, pp. 59-98, 1979.
- [20] P. H. Gaskell and A. K. C Lau, "Curvature-compensated convective transport: SMART, a new boundedness-preserving transport algorithm," *International Journal for Numerical Methods in Fluids*, vol. 8, pp. 617-641, 1988.
- [21] A. Varonos and G. Bergeles, "Development and assessment of a variable-order non-oscillatory scheme for convection term discretization," *International Journal for Numerical Methods in Fluids*, vol. 26, pp. 1-16, 1998.
- [22] Alves MA, Oliveira PJ, Pinho FT. A convergent and universally bounded interpolation scheme for the treatment of advection. *International Journal for Numerical Methods in Fluids*, vol. 21, pp. 47-75, 2003.
- [23] K. A. Hoffmann and S. T. Chiang, "Computational Fluid Dynamics," vol. 2, Wichita, KS, 1998.
- [24] K. A. Cliffe, C. P. Jackson and A. C. Greenfield, "Finite Element Solutions for Flow in a Symmetric Channel with a Smooth Expansion," *AERE-R*, 10608, UK, 1982.
- [25] M. Napolitano and P. Orlandi, "Laminar Flow in Complex Geometry: A Comparison," *International Journal for Numerical Methods in Fluids*, vol. 5, pp. 667-683, 1985.
- [26] P. S. Carson, "Computation of Laminar Viscous Flows Using Von Mises Coordinates," *Ph.D. Thesis*, University of Windsor, Canada, 1994.
- [27] A. K. Rostagi, "Hydrodynamics and Mass Transport in Pipelines Perturbed by Welded Joint," *DNV Technical Report* No. 82.0152, 1982.