11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
E. Oñate, J. Oliver and A. Huerta (Eds)

A RAPIDLY CONVERGENT ALGORITHM FOR THE SOLUTION OF NAVIER-STOKES EQUATIONS

SEVERINO KRIZMANIĆ, ZDRAVKO VIRAG AND MARIO ŠAVAR

Faculty of Mechanical Engineering And Naval Architecture (FMENA)
University of Zagreb
Ivana Lučića 5, 10000 Zagreb, Croatia
e-mail: severino.krizmanic@fsb.hr, http://www.fsb.unizg.hr/hydro

Key Words: *Computational Fluid Dynamics, Finite Volume Method, Unstructured Grid, Pressure-Velocity Coupling, Segregated Method, Method Of Loops*

Abstract. In this paper a novel pressure-velocity coupling algorithm for the solution of the Navier-Stokes equations by finite volume method on collocated grids is presented. In order to achieve pressure-velocity coupling, SIMPLE-like methods generally combine an approximate divergence of the pressure gradient field obtained from the momentum equations and the continuity equation. In formulations on collocated grids, the working variables are cell velocity components and cell pressure, while in order to suppress checker-board solutions, the cell face mass fluxes used in continuity equation are obtained by using special interpolations. The convergence rate of SIMPLE-like methods strongly rely on under-relaxation factor values, especially in segregated algorithm formulations. Compared to SIMPLE-like methods, novel pressure-velocity coupling method presented in this paper has several fundamental differences. Novel method uses cell face mass fluxes as working variables and irrotationality condition for the pressure gradient, while the cell velocity components are obtained by using the continuity equation. The resulting new pressure-velocity coupling algorithm approaches the final solution in an segregated iterative procedure by correcting cell face mass fluxes until the pressure gradient field becomes irrotational. The mass fluxes corrections applied in the new algorithm preserve the solution of the continuity equation, so that continuity equation has to be solved only once, at the beginning of the iteration procedure. Due the presented properties, the new algorithm "naturally" bypasses the checker-board problem in the collocated variable arrangement and doesn't require a stabilization procedure in the iterative solution process. The new pressure-velocity coupling algorithm was coded into own computer program and compared with the SIMPLE algorithm implemented within a commercial software package. The conducted tests covered 2D and 3D incompressible, inviscid and laminar flow problems having various boundary conditions and grid sizes and steady free convection problems solved using Boussinesq approximation. In conducted tests, the convergence rate of the new algorithm showed to be independent on grid size. In strict comparisons performed, both algorithms used same differencing scheme and identical grids. The new pressure-velocity algorithm shows a significantly higher convergence rate and CPU time efficiency.

1 INTRODUCTION

Nowadays Computational Fluid Dynamics (CFD) practice makes use of various methods for the solution of the flow-field equations, applying mostly SIMPLE–like [1],[2],[4],[5] methods for the solution of incompressible flows. Since it's first appearance ([1], Patankar&Spalding, 1972) these methods had been extended to application on collocated (unstructured) grids ([3], Rhie&Chow, 1983) so as to the solution of compressible flows. Fast development of computational technology has channeled the research efforts within the field of CFD towards extension of its applicability to physically and numerically ever more demanding problems, leaving the problem of pressure-velocity coupling somewhat neglected. On collocated grids all SIMPLE-like methods still require non-physical pressure boundary conditions and use of arbitrary interpolations [3] with a possible negative consequence to the solution accuracy, while the convergence rate of all methods of this family strongly relies on method's under-relaxation factors (i. e. their proper values). The theoretical research [7] here presented, has led to a new pressure-velocity coupling method that yields a novel algorithm for the solution of the flow-field equations. The new algorithm requires no under-relaxation factors for the solution of the equations of flow and has a significantly higher convergence rate, while keeping the segregated solving procedure.

2 THE EQUATIONS OF INCOMPRESSIBLE FLOW

The presented work focuses on a search for a more efficient method for the solution of the flow-field with the ability of resolving the equations of flow in general case. The key issue here is a successful resolution of the pressure-velocity coupling problem which arises when reaching the limit of incompressible flow. Therefore, all SIMPLE-like methods are originally formulated for the case of incompressible flow while the resolution of compressible flows presents an extension. The same case is with the present method, where the extension to compressible flows is found to be straightforward and will be a subject of future researches.

In incompressible case, the fluid flow is governed by the continuity equation:

$$\frac{\partial v_i}{\partial x_i}=0 \tag{1}$$

and momentum equations:

$$\frac{\partial(\rho v_i)}{\partial t}+\frac{\partial(\rho v_j v_i)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial(\rho v_i)}{\partial x_j}\right);\ i=1,2,3 \tag{2}$$

here, written with denotations: t – time, x_i – cartesian coordinate, ρ and μ – constant fluid density and dynamic viscosity, v_i – velocity field and p – pressure field.

3 NUMERICAL SOLUTION BY SIMPLE – LIKE METHODS

As a subset of primitive variable methods, the SIMPLE – like methods consider the system (1)-(2), with variables being the pressure field p and velocity components fields v_i and where each of the equations (2) is considered as being the equation for the respective component of the velocity field. By applying finite volume discretization method, the discretized equations in matrix presentation are:

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{bmatrix} v_i \\ p \end{bmatrix} = \begin{bmatrix} b_{i,\text{mom.}} \\ b_{\text{cont.}} \end{bmatrix} \tag{3}$$

Apart from the velocity field being non-linear in the convection term in equations (2), another adversity derives from the very nature of incompressible flow, where the equation for the pressure field is lacking. By reaching the limit of constant density, the continuity equation turns to a kinematic constraint for the velocity field (1), while the pressure field is present exclusively in momentum equations (2), in form of a gradient. The consequence is, a zero sub-matrix D in the system of discretized equations (3). Projection methods (of which SIMPLE-like methods are considered as being a subset) overcome this problem generally, by taking the divergence of the pressure gradient field and combining it with the continuity equation, which is equivalent to obtaining Schur's complement in the system of discretized equations (3). In order to keep numerical efficiency (inversion of sub-matrix A needed), SIMPLE-like methods use an approximate inverse, thus further reducing the implicitness of the iterative solution procedure. Here, an additional under-relaxation factor for the pressure is introduced to enhance the stability of the iterative procedure, which has a negative impact to its convergence rate. Another adversity arises in case of SIMPLE-like methods applied on collocated grids (the preferred - unstructured grids over staggered, structured 2D grids). The discretization of the pressure and velocity fields on same finite volumes produces algebraic equations that for the corresponding volume do not contain the values for the pressure - in the discretized pressure gradient term in equations (2) and for the velocity - in the discretized continuity equation (1). In order to avoid consequent checker-board solutions, a suitable interpolation procedure is applied [3], which is arbitrary (non-physical). This way, the continuity equation is satisfied by using velocity values that are interpolated to finite volume's faces, along with the difference between values of the pressure gradient obtained directly from the momentum equations and those obtained by differencing the discretized pressure field. Here, in order to obtain dimensional consistency, this difference is pre-multiplied with an appropriate dimensional factor, which is also obtained arbitrarily. In areas of strong pressure changes this procedure can cause wrong solution for the continuity equation in affected volumes. Often happens that, once the continuity equation is satisfied by using such interpolated velocity values, this error gets overlooked. Another adversity of SIMPLE – like methods applied on collocated grids is the discrepancy between the number of discretized pressure field values and the number of available equations. In order to close the momentum equations for boundary finite volumes, an arbitrary extrapolation of the pressure field towards the boundaries of the computational domain is applied, with a possible further impact to the solution accuracy.

4 NEW PRESSURE-VELOCITY COUPLING METHOD

With the SIMPLE algorithm on staggered grids, the continuity and momentum equations are discretized on theirs respective grids. Here, the consistency between number of variables and corresponding equations is preserved and no special arbitrary interpolations are needed. Following the idea of this original [1] formulation of the SIMPLE algorithm, with the new algorithm - a switch on choice of variables is performed, while keeping this consistency on collocated grids, also. Instead of velocity field valued at the finite volume's centers – the mass fluxes trough the finite volume's faces F are considered to be the variables. Here, for the discretized momentum equations - an interpolation is needed to obtain the discretized velocity field valued at the volume's centers:

$$v_i = \frac{1}{\rho \Delta V} \sum_{k=1}^{nb} x_i^k F^k \, , \tag{4}$$

where nb designates the number of volume's faces, while x_i^k designates the vectored distance from the volume's center to the respective face center. This interpolation is physical, since it is obtained directly from the continuity equation.

From the perspective of the SIMPLE-like methods, the velocity field obtained from the momentum equations (2) will satisfy the continuity equation (1) if solved with the correct pressure field. The perspective of the new method is that from all possible solutions for mass fluxes that satisfy continuity equation, solution of a given problem is the one that defines a unique pressure field. The pressure gradient field defines unique pressure field if it is irrotational. Therefore, oppositely to divergence of the pressure gradient, applied generally in all projection methods to achieve the pressure-velocity coupling, the new algorithm applies a irrotationality condition to the pressure gradient field thus eliminating the need for any (unfavourable) discretization of the pressure field on a collocated grid. Once the F and v_i are known, the pressure gradient values at volumes centers are calculated directly from the discretized momentum equations (2). By applying (4) to (2) for a finite volume with center A (see Fig. 1), this is formally written as:

$$\left.\frac{\partial p}{\partial x_i}\right|_{\mathrm{A}} = f_i\left(F^k\right) \; ; \; k = 1, 2, \ldots, nb_{\mathrm{A}} \tag{5}$$

The irrotationality condition is rewritten in form of zero circulation condition for (5) integrated along a sufficient number of closed loops. By this procedure additional quantity named "loop pressure drop" - Δp_L is formed and adhered to every loop thus created.

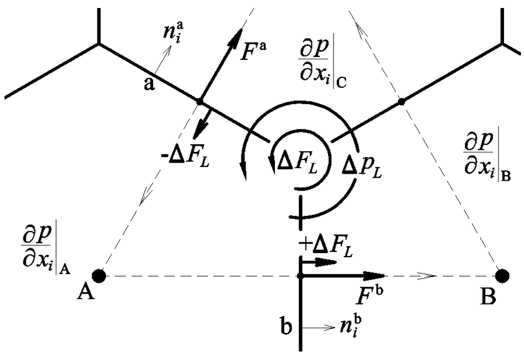

Figure 1: FLOP method variables assembly

Next, named loop pressure drops Δp_L-s, are rewritten in form of functions of ΔF_L – the unknown corrections of fluxes adhered to loops, required to achieve a zero value of loop pressure drop in all loops ($\Delta p_L = 0$). The resulting system for unknown corrections of fluxes ΔF_L is solved by a Newton's method. As depicted on Fig. 1. the corrections of fluxes ΔF_L are such assembled that continuity equation remains satisfied after each correction. This has a strong reflection to the solution algorithm's procedure, as shown on Fig. 2. For the sake of brevity, the new method will be denoted as FLOP (Flux LOoping for Pressure drop).

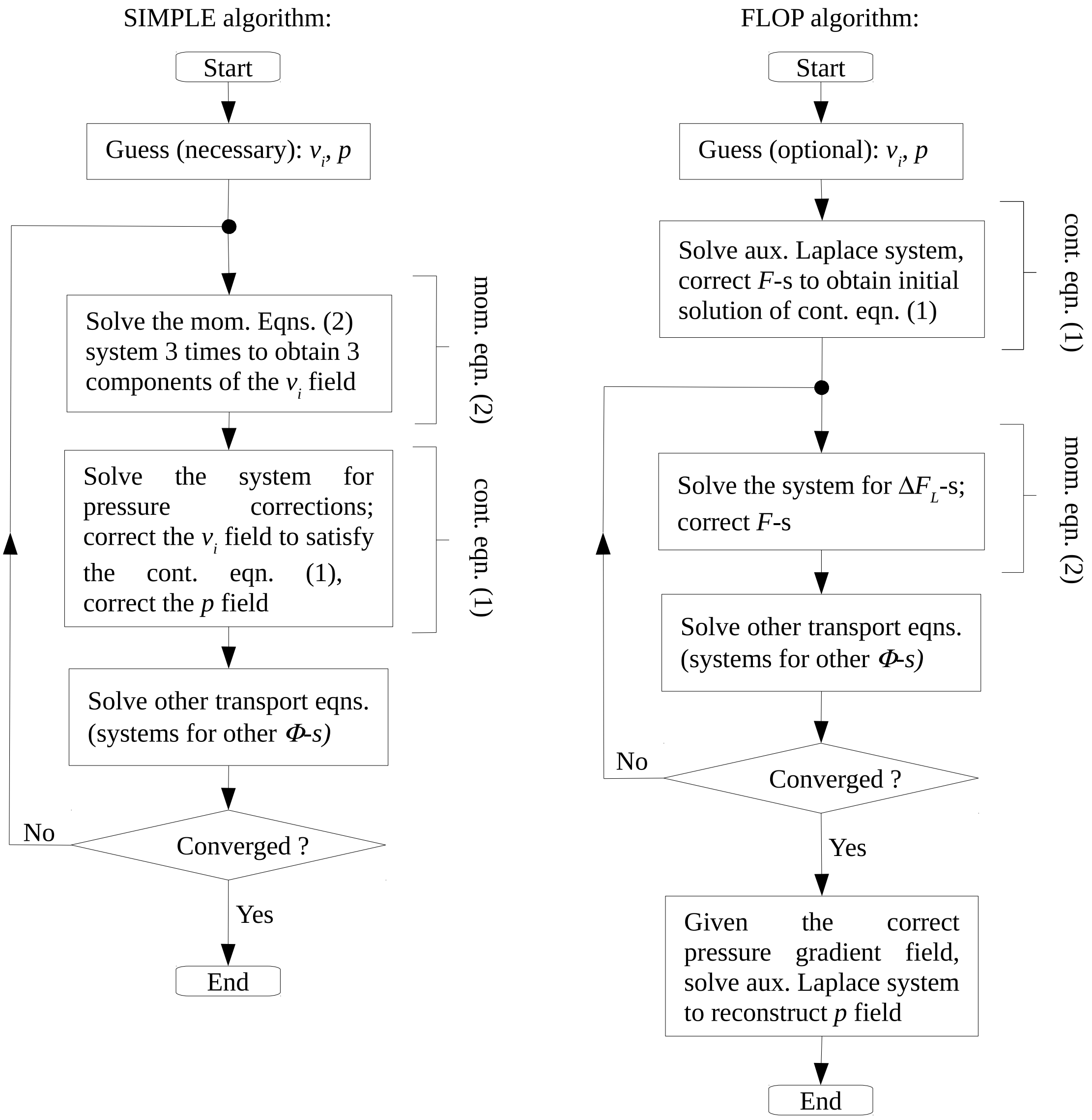

Figure 2: SIMPLE and FLOP algorithms procedures

By solving the Laplace's equation, the FLOP algorithm procedure starts from mass fluxes F which satisfy the continuity equation. Thereafter, the algorithm (due to linearization of the convection terms in (2)) iterates on the solution for corrections ΔF_L-s, to obtain a zero (sufficiently small) Δp_L in all loops, which then is - the solution of the problem. Finally, the actual pressure field is reconstructed from such obtained pressure gradient field. The comparison of SIMPLE and FLOP algorithms procedures is given on Fig. 2.

3 COMPARISONS OF FLOP AND SIMPLE ALGORITHMS

The new FLOP algorithm was coded into own computer program and compared with the SIMPLE algorithm, professionally (presumably - optimally) coded within a commercial software package.

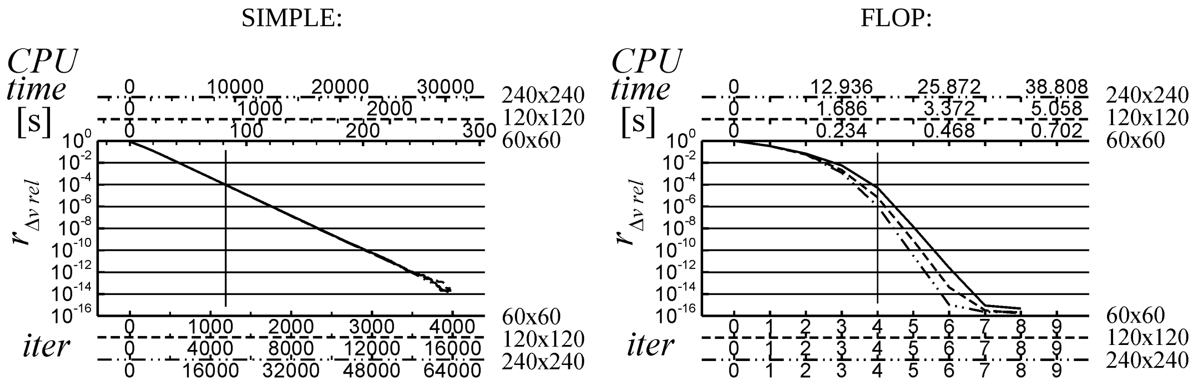

Figure 3: The convergence histories of the solution procedures for 2D lid-driven cavity laminar flow at Re = 100, solved for three different mesh sizes. The convergence histories are given in terms of the average relative difference in velocity field in respect to the own solution obtained at machine precision - $r_{\Delta v\ rel}$.

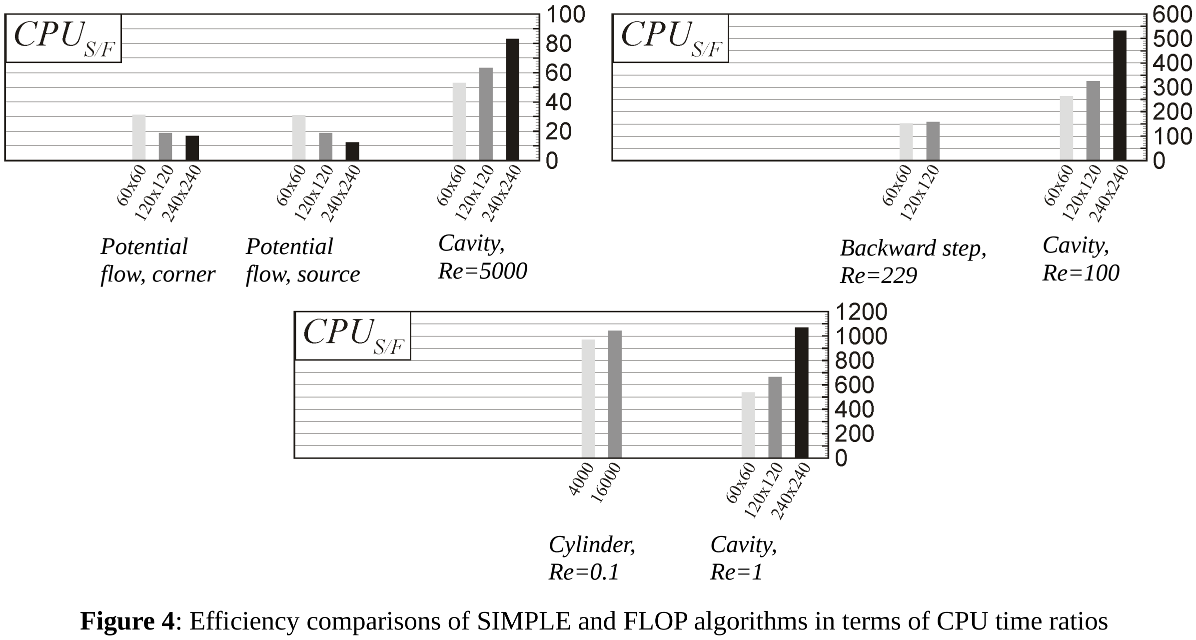

Figure 4: Efficiency comparisons of SIMPLE and FLOP algorithms in terms of CPU time ratios (SIMPLE/FLOP).

The conducted tests covered incompressible, inviscid and laminar flow problems ranging from low to high (at the verge of steadiness) Reynolds numbers for 2D and 3D problems, with various boundary conditions and grid sizes, so as steady free convection (Boussinesq approximation) problems solved over a wide range of Rayleigh numbers. In strict comparisons performed, both algorithms used the same EDS [6] differencing scheme and identical grids. The comparisons were made on a basis of solution for the velocity field at the same level of accuracy in respect to solution obtained at the limit of machine precision. A typical example is the one of a well known 2D lid-driven cavity laminar flow at Re = 100, for which comparison is depicted on Fig. 3, while the overall efficiency comparison in terms of CPU time ratios is presented on Fig 4.

5 CONCLUDING REMARKS

For all conducted comparisons, the algorithms behavior is similar to one depicted on Figs. 3 and 4: a significantly higher convergence rate and efficiency of the FLOP algorithm is noted, along with it's independency of number of iterations required for the achievement of the prescribed accuracy on mesh size. For the SIMPLE algorithm, this dependency is found to be linear for all viscous flows. Thus, along with a higher efficiency of the FLOP algorithm, the comparisons show it's further increase with increasing mesh size. Due to fulfillment of the continuity equation constraint of it's iterative process, the new FLOP algorithm iterates (Fig. 2) solely on momentum equations, while the continuity equation has to be solved only once. Having this property and due to only physical interpolations used, it does not require any stabilization procedure. This is opposite to all SIMPLE–like algorithms, where the procedure iterates (Fig. 2) between momentum and continuity equations, thus perturbing the solution of the preceding one and where a stabilization is needed trough under-relaxation factors for which optimal values are problem dependent. Being a novel method, current adversity of the FLOP algorithm is the matrix of the linear system for ΔF_L-s, for which solution a direct solver is used. That issue should be hopefully resolved within future researches

REFERENCES

[1] Patankar S.V., Spalding D.B., A Calculation Procedure for Heat, Mass and Momentum Transfer in Three Dimensional Parabolic Flows, *Int. J. Heat Mass Transfer*, Vol. 15, 1787-1806, 1972.

[2] Patankar S.V., A Calculation Procedure for Two-Dimensional Elliptic Situations, *Numerical Heat Transfer*, Vol. 4 (1981), 409-425.

[3] Rhie C.M., Chow W.L., A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. *AIAA J.*, Vol. 21 (1983), 1525-1532, 1983.

[4] Doormaal J.P., Raithby G.D., Enhancement of SIMPLE Method for Predicting Incompressible Fluid Flows, *Numerical Heat Transfer*, Vol. 7, 147-163, 1984.

[5] Issa R.I., Solution of Implicitly Discretized Fluid Flow Equation by Operator-Splitting, *J. Comput. Phys.*, Vol. 62 (1985), 40-65.

[6] Spalding D.B., A Novel Finite-Difference Formulation for Differential Expressions Involving Both First and Second Derivatives, *Int. J. Num. Methods Eng.*, Vol. 4 (1972), 551-559.

[7] Krizmanić, S., A New Pressure-Velocity Coupling Algorithm (in Croatian), Ph.D. thesis, *Faculty of Mechanical Engineering And Naval Architecture (FMENA),* (2011), University of Zagreb.