Comparison between the NWF and DC methods for implementing HR Schemes within a Fully Coupled Finite Volume Solver

F. Moukalled, A. Abdel Aziz, and M. Darwish

Citation: AIP Conference Proceedings **1168**, 581 (2009); doi: 10.1063/1.3241529
View online: http://dx.doi.org/10.1063/1.3241529
View Table of Contents: http://aip.scitation.org/toc/apc/1168/1
Published by the American Institute of Physics

Comparison between the NWF and DC methods for implementing HR Schemes within a Fully Coupled Finite Volume Solver

F. Moukalled, A. Abdel Aziz, and M. Darwish

Department of Mechanical Engineering
American University of Beirut
P.O.Box 11-0236
Riad El Solh, Beirut 1107 2020
Lebanon

Abstract. This paper reports on the performance of a high resolution implemented as part of an implicit fully coupled velocity-pressure algorithm for the solution of laminar incompressible flow problems. The numerical implementation of high resolution convective schemes follows two techniques; (i) the Deferred Correction (DC) approach, and (ii) the Normalized Weighting Factor (NWF) method. The superiority of the NWF method over the DC approach is demonstrated by solving the sudden expansion in a square cavity problem. Results indicate that the number of iterations needed by the NWF solver is grid independent. Moreover, recorded CPU time values reveal that the NWF method substantially reduces the computational cost.

Keywords: High resolution schemes, Finite Volume Method, Pressure-Based Method, and Coupled Solver.
PACS: 2008

INTRODUCTION

Recently, the authors of this article reported on a fully coupled algorithm for the solution of incompressible flows [1, 2] with acceleration rates at least an order of magnitude higher than the ones achieved with the more popular segregated approach. In their algorithm however, the convective flux in the momentum equations was discretized using the first order UPWIND scheme [3]. First-order schemes are numerically stable but highly diffusive. This numerical diffusion is desirable for numerical stability but often leads to highly inaccurate results and causes smearing of sharp gradients. To overcome this shortcoming and to increase the accuracy of the predicted results, researchers have developed a variety of higher-order schemes [4-6]. The difficulties associated with the development of reliable higher order schemes stem from the conflicting requirements of accuracy, stability, and boundedness. Solutions predicted with high order schemes are more accurate than the first-order upwind scheme and more stable than the second-order central difference scheme, but tend to provoke oscillations. To suppress oscillations, several techniques were advertised leading to new families of High Resolution (HR) schemes (i.e. high order bounded schemes), and the one adopted here is the composite flux limiter approach applied in the context of the Normalized Variable Formulation (NVF) [3]. The Deferred Correction (DC) procedure [7] remained the preferred technique for the numerical implementation of these schemes, because it allows the use of codes originally intended for low order schemes by the addition of a source term that accounts for the difference in interpolated values between the high resolution and low order scheme, at the price of a reduced convergence rate. The Normalized Weighting Factor (NWF) method [8], which is fully implicit, was developed to overcome this issue and accelerate convergence. While successful in solving for scalar transport equations, the NWF resulted in oscillations when dealing with flow problems.

As the aim of any coupled solver is to accelerate convergence, the effect of using high resolution schemes implemented via the DC method on the convergence rate is of primary importance. To this end, the objectives of this paper are twofold: (i) to study the effect of implementing HR schemes using the DC method on the convergence of the coupled solver, and (ii) to extend the applicability of the NWF to flow problems by implementing it within the coupled solver and to compare its performance with the DC method.

CP1168, Vol. 2, *Numerical Analysis and Applied Mathematics, International Conference 2009*
edited by T. E. Simos, G. Psihoyios, and Ch. Tsitouras

In what follows, the discretization procedure of the governing conservation equations, the coupled solvers, and the DC and NWF methods are briefly reviewed. Then the effect of using HR schemes applied using the DC approach and NWF method on the performance of the coupled algorithm is assessed in a test problem.

FINITE VOLUME FORMULATION

The conservation equations governing steady, laminar incompressible Newtonian fluid flow are given by

$$\nabla \cdot (\rho \mathbf{v}) = 0 \tag{1}$$

$$\nabla \cdot (\rho \mathbf{v}\mathbf{v}) = \nabla \cdot (\mu \nabla \mathbf{v}) - \nabla \cdot (p\mathbf{I}) \tag{2}$$

Integrating the transport equations over a control volume, transforming the volume integrals of the diffusion and convection terms into surface integrals using the divergence theorem, and evaluating these integrals by representing the variables at the control volume faces in terms of nodal values, the discretized forms of the momentum and continuity equations are respectively given by

$$\mathbf{v}_P + \sum_{F=NB(P)} \mathbf{A}_F^{\mathbf{v}} \mathbf{v}_F = \mathbf{B}_P^{\mathbf{v}} - \mathbf{D}_P \nabla p_P \quad \text{and} \quad \sum_{f=nb(P)} \dot{m}_f = 0 \tag{3}$$

where *NB(P)* refers to the neighbors of the *P* grid point and *nb(P)* refers to the faces of the *P* control volume.

High Resolution Schemes

When evaluating the convection flux using the upwind scheme, the value of the dependent variable at the control volume face is taken as the value at the main grid point on the upwind side. Mathematically, this is written as

$$(\dot{m}\phi)_f = \phi_P \left\| \dot{m}_f, 0 \right\| - \phi_F \left\| -\dot{m}_f, 0 \right\| \tag{4}$$

When using a HR scheme, the value of ϕ at a control volume face is written as a composite functional relationship of the values at several grid points upstream and downstream of the face. Without going into details and using normalized variables [3], the functional relationship for the SMART scheme [5] used in this work is given by

$$\phi_f = \begin{cases} 4\tilde{\phi}_C & 0 < \tilde{\phi}_C < 1/6 \\ \frac{3}{4}\tilde{\phi}_C + \frac{3}{8} & 1/6 < \tilde{\phi}_C < 5/6 \\ 1 & 5/6 < \tilde{\phi}_C < 1 \\ \tilde{\phi}_C & elsewhere \end{cases} \tag{5}$$

Deferred Correction (DC) Procedure

In the DC procedure the HR scheme is implemented by the splitting the convection flux into an implicit part, expressed through first order upwind differencing scheme (UDS) and an explicit part (a source term in the algebraic equation), which equals the difference between the UDS and HR approximations, i.e.:

$$\dot{m}_f \phi_f = \dot{m}_f \phi^U + \dot{m}_f \left(\phi^{HR} - \phi^U\right) \tag{6}$$

Normalized Weighting Factor (NWF) Method

From equation (5) it follows that the functional relationship of HR schemes can be written as

$$\phi_f = m\tilde{\phi}_C + k \tag{7}$$

where the expressions of m and k are obtained from the original equations. In the NWF method, equation (7) is used to express the ϕ value at a control volume face implicitly in terms of values at the neighboring grid points. The right hand side of Eq. (7) is substituted in the discretized leading to an implicit description of ϕ_f.

The Coupled Algorithm

The low convergence rate of the segregated method is a due to the explicit treatment of the pressure gradient in the momentum equation and the velocity field in the continuity equation. The coupled algorithm overcomes this deficiency by treating both terms implicitly. For that purpose the pressure gradient term in the momentum equations is integrated over the faces of the control volume and is evaluated implicitly. The pressure equation is derived from the continuity equation by expressing the velocity at the control volume face using the Rhie-Chow interpolation. The resulting system of momentum and continuity equations in 2-D is written as

$$\begin{aligned}
&a_P^{uu}u_P + a_P^{uv}v_P + a_P^{up}p_P + \sum_{F=NB(P)} a_F^{uu}u_F + \sum_{F=NB(P)} a_F^{uv}v_F + \sum_{F=NB(P)} a_F^{up}p_F = b_P^u \\
&a_P^{vv}v_P + a_P^{vu}u_P + a_P^{vp}p_P + \sum_{F=NB(P)} a_F^{vv}v_F + \sum_{F=NB(P)} a_F^{vu}u_F + \sum_{F=NB(P)} a_F^{vp}p_F = b_P^v \\
&a_P^{pp}p_P + a_P^{pu}u_P + a_P^{pv}v_P + \sum_{F=NB(P)} a_F^{pp}p_F + \sum_{F=NB(P)} a_F^{pu}u_F + \sum_{F=NB(P)} a_F^{pv}v_F = b_P^p
\end{aligned} \tag{8}$$

A system of equations involving velocity components and pressure is obtained for each control volume and when expressed over the entire computational domain yields a system of equations of the form

$$\mathbf{A}\Phi = \mathbf{B} \tag{9}$$

where all variables ($\mathbf{v}$, p) are now solved simultaneously.

The overall coupled algorithm can be summarized as follows:

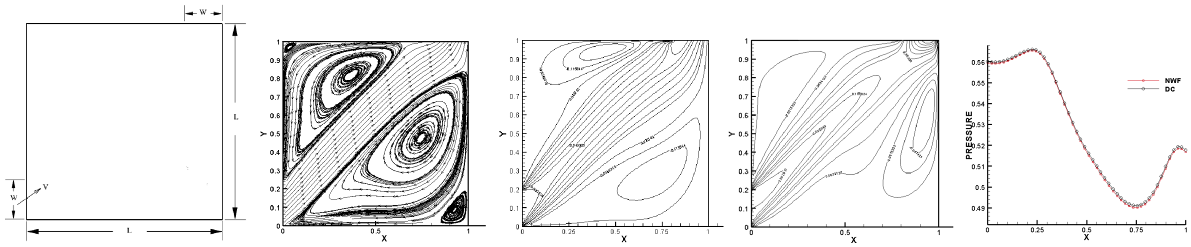
1. Start with the n^{th} iteration values
2. Assemble and solve the momentum and continuity equation for $\mathbf{v}^*$and p^*
3. Assemble $\dot{m}_f$ using the Rhie-Chow interpolation
4. Return to the first step and repeat until convergence

RESULTS AND DISCUSSION

The performance of the HR coupled algorithm is assessed in this section by presenting solutions to the sudden-expansion flow in a square cavity problem. The results are generated using quadrilateral control volumes on three grid sizes with cell values of 10^4, $5\text{x}10^4$, and $3\text{x}10^5$. The performances of the NWF and DC are assessed in terms of the number of iterations and time required to reach convergence. The same initial guess was used for all grid sizes. The physical situation, which represents a square cavity of side L (W=L/5), is depicted in Figure 1(a). The inlet velocity vector is $\mathbf{v}\left(\sqrt{2},\sqrt{2}\right)$ and the Reynolds number based on L is set at 750. In Figs. 1(b) through 1(e), the streamlines over the domain, contours of the u- and v-velocity components, and a comparison of the static pressure at y=0.5 computed using both the NWF and DC methods are presented.

Due to its full implicitness in discretizing the convection term, the NWF method saves a lot of computational time as compared to the DC method. This is clearly shown in Tables 1 and 2, which compare the performance of both methods. The NWF accelerates the solution of the problem by approximately 50%.

A summary of the number of iterations and CPU time needed by both the DC and NWF methods are presented for all grid sizes in Table 1. The number of iterations required by the NWF method is almost grid independent while the iterations in the DC approach varies with the grid size. In terms of computational times, good savings are achieved using the NWF approach with the amount increasing with increases to the grid size. The reduction factor (Table 2) reaches a value of 1.78 on the densest grid ($3x10^5$ cells). This represents a significant decrease in computational time.



(a) (b) (c) (d) (e)

FIGURE 1 (a) Physical domain, (b) streamlines, (c) u-velocity contours, (d) v-velocity contours, and (e) comparison of computed gauge pressure at y = 0.5 by both the NWF and DC methods.

Table 1. Convergence requirements by the DC and NWF methods

Grid Density	NWF		DC		DC / NWF
	Iterations	Time	Iterations	Time	Time Ratio
10000	26	57.1	32	81.25	1.42
50000*	24	288.5	25	368.2	1.3
300000	26	1975.4	37	3530.8	1.78

*: for the 50000 elements grid, a source under-relaxation of 0.85 had to be used to reach convergence.

CLOSING REMARKS

The performance of a high resolution scheme implemented within an implicit fully coupled velocity-pressure algorithm was assessed for the case when the high resolution convective scheme is implemented following either the DC approach or the NWF method. Results for the sudden expansion in a square cavity problem over a range of grids revealed that the NWF method is computationally more efficient than the DC approach substantially reducing the computational cost.

ACKNOWLEDGMENTS

This work was supported by the University Research Board of the American University of Beirut through Grant #DDF 113040-988323.

REFERENCES

1. M. Darwish, I. Sraj, and F. Moukalled, *Numerical Heat Transfer, Part B,* **52**, 353-371 (2007).
2. M. Darwish, I. Sraj, and F. Moukalled, *Journal of Computational Physics*, **28**,180-201, (2009).
3. M. Darwish and F. Moukalled, *Numerical Heat Transfer, Part B,* **26**: 79 – 96 (1994).
4. B. P. Leonard, *Computer Methods in Applied Mechanics and* Engineering, **19**, 59-98 (1979).
5. P. H. Gaskel and A.K.C. Lau, *International Journal for Numerical Methods in Fluids,* **8**, 617-641 (1988).
6. W. Shyy, *Journal of Computational Physics,* **57**, 415-438 (1985).
7. S.G. Rubin and P.K. Khosla, *Journal of Computational Physics,* **27**, 153-168 (1982).
8. M. Darwish and F. Moukalled, *Numerical Heat Transfer, Part B,* **30**: 217 – 237 (1996).