A Comparison of Characteristic-Based Split Algorithm and Multi-Grid method for Solution of Incompressible Navier-Stokes Equations.

T. M. Shah and N. Talat
Department of Mathematics, Air University, Islamabad, Pakistan.
dr.tasneem@mail.au.edu.pk, naziatalat82@yahoo.com

Abstract-- The split follows the process initially introduced by Chorin [1,2] for incompressible flow problems in the finite difference context. Similar extensions of the split to finite element formulation for different applications of incompressible flows have carried out by many authors [3]. The same split have also done by the author of this paper using multi-grid methods to incompressible flows [4]. The characteristics-Based Split (CBS) Algorithm has first introduced by Zienkiewicz and Codina [5] to solve the fluid dynamics equations of both compressible and incompressible forms. This new form is applicable to fully compressible flows in both explicit and semi-implicit forms. In the case of real compressibility such as in gas flows, the computational advantages of the explicit form compare well with other currently used schemes and the additional cost due to splitting the operator is insignificant. Generally for an ideal case, results are considerably improved throughout a large range of aerodynamic problems. However, a further advantage is that both subsonic and supersonic problems can be solved by the same code [5].

Similar, the case in Multi-Grid code developed in [4] can cope with large range of aerodynamic / hydrodynamic and subsonic / supersonic problems can be solved by the same code. This gives a true comparison between the two strategies.

In the current paper, a comparative analysis of Multi-Grid solution and solution obtained by CBS algorithm for the incompressible flow problems is done. Results are obtained and presented for discussions.

Index Term-- CBS algorithm, Mesh Generation, Multi-Grid method, Finite element method, Navier-Stokes equations, Benchmark Problems.

I. INTRODUCTION

The authors have solved problems using finite difference with multi-grid and finite element methods. This paper has a comparative analysis of solutions of both methods, solving incompressible Navier-Stokes equations by splitting mass and momentum equations. Since the problem of solid and fluid behavior are in many respect similar as stresses and material displaced occur in both the cases. The equations governing fluid flow and solid mechanics appear to be similar with the velocity vector replacing the displacement. However, there is one further difference i.e., even when the flow has a constant velocity (steady state) convective acceleration occurs. This convective acceleration provides terms, which make the fluid mechanics equations non self-adjoint. For self-adjoint forms, the approximating equations derived by the Galerkin process give the minimum error in the energy norm and thus are in a sense optimal. This is no longer true in general in fluid mechanics, though for slow flows, the situation is somewhat similar.

A finite element treatment has been introduced to the equations of motion for various problems of fluid mechanics [6]. Much of the activity in fluid mechanics has a finite difference formulation and more recently finite volume technique with multi-grid methods. Competition between the newcomer of finite elements and established techniques of finite difference have appeared on the surface and led to a much slower adoption of the finite element process in fluid mechanics than in structures.

A methodology appears to have intermediate position is that of finite volumes, which were initially derived as a subclass of finite difference methods, these are simply another kind of finite element form in which sub-domain collocation is used.

This paper is an effort to compare the solutions to benchmark problems of both the methods, finite volume with multi-grid and recently developed characteristics-based split with finite element. For the purpose of comparison, we consider incompressible Navier-Stokes equations in a lid driven cavity. Solutions obtained from CBS algorithm (steady state) are compared with the solutions obtained from multi-grid methods.

Governing Equations:

The governing equations in fully conservative standard form are as below:

\[
\frac{\partial p}{\partial t} = \frac{1}{c^2} \frac{\partial p}{\partial t} = - \frac{\partial U_i}{\partial x_i} \quad (1)
\]

where \(c\) is the speed of sound and depends on \(E, p\) and \(\rho\) assuming constant entropy.
\( \gamma \) is the ratio of specific heats equal \( \frac{c_p}{c_v} \). For a fluid with a small compressibility

\[
\gamma = \frac{K}{\rho}
\]

where \( K \) is the bulk modulus.

**Momentum conservation:**

\[
\frac{\partial U_i}{\partial t} = - \frac{\partial}{\partial x_j} \left( u_j U_i \right) + \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial p}{\partial x_i} + \rho g_i
\]

The mass flow fluxes is defined as

\[
U_i = \rho u_i
\]

**Energy conservation:**

\[
\frac{\partial (\rho E)}{\partial t} = - \frac{\partial}{\partial x_j} \left( u_j \rho E \right) + \frac{\partial}{\partial x_j} \left( k \frac{\partial T}{\partial x_j} \right) - \frac{\partial}{\partial x_i} \left( u_i p \right) + \frac{\partial}{\partial x_i} \left( \tau_{ij} u_j \right)
\]

**Equation of state:**

\[
p = \rho RT
\]

where following nomenclatures are used

- \( U_i \) Velocity components
- \( \rho \) Density
- \( E \) specific energy
- \( p \) pressure
- \( T \) absolute temperature
- \( \rho g_i \) Body forces and other source terms
- \( K \) thermal conductivity
- \( \tau_{ij} \) Deviotropic stress
- \( R \) universal gas constant

The above dimensional equations as well as non-dimensional forms are convenient to apply CBS-algorithm.

**CBS-Algorithm:**

The split follows the process initially introduced by Chorin [1] for incompressible flow problems in the finite difference context. A similar extension of the split to finite element formulation for different applications of incompressible flows has been carried out by other authors [3]. Here the split carried out to solve fluid dynamics equations of both compressible and incompressible forms using characteristic-Galerkin procedure [2]. The algorithm was first introduced in 1995 by Zienkiewicz-Codina [5].

The split provides a fully explicit algorithm even in incompressible case for steady state problems now using an ‘artificial’ compressibility which does not affect the steady state solution.

**The Split Discretization:**

Equation (2) can be discretized in time using the characteristic-Galerkin process. Rewriting Equation (2) in the form to which characteristic-Galerkin method can be applied

\[
\frac{\partial U_i}{\partial t} = - \frac{\partial}{\partial x_j} \left( u_j U_i \right) + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + Q_{i}^{n+\theta_t}
\]

with \( Q_{i}^{n+\theta_t} \) being treated as known quantity evaluated at \( t = t^n + \theta_t \Delta t \) in a time increment \( \Delta t \). In Equation (5)

\[
\frac{\partial p^{n+\theta_t}}{\partial x_i} = - \frac{\partial}{\partial x_i} \left( u_j \right) + \frac{\partial}{\partial x_j} \left( u_j p^{n+\theta_t} \right)
\]

(6)

with

\[
\frac{\partial p^{n+\theta_t}}{\partial x_i} = \frac{\partial p^n}{\partial x_i} + \left( 1 - \theta_t \right) \frac{\partial p^n}{\partial x_i}
\]

(7)

The final form becomes:

\[
U_i^{n+1} - U_i^n = \Delta t \left[ - \frac{\partial}{\partial x_j} \left( u_j U_i \right) + \frac{\partial \tau_{ij}^n}{\partial x_j} + Q_{i}^{n+\theta_t} - \left( \rho g_i \right)^n \right]
\]

(8)

Introduce the “Split” in which we introduce a suitable approximation for \( Q \) which allows the calculation to proceed before \( p^{n+1} \) is evaluated.

Introduce an auxiliary variable \( U_i^{*} \) such that

\[
\Delta U_i^{*} = U_i^{*} - U_i^n
\]

\[
\Delta U_i^{*} = \Delta t \left[ - \frac{\partial}{\partial x_j} \left( u_j U_i \right) + \frac{\partial \tau_{ij}}{\partial x_j} - \rho g_i + \Delta t \frac{\partial}{\partial x_j} \left( u_j U_i \right) + \rho g_i \right]
\]

(9)
\[ \Delta U_i = U_i^{n+1} - U_i^n = \Delta U_i^* - \Delta t \frac{\partial P^{n+\theta_i}}{\partial x_i} - \frac{\Delta t^2}{2} u_k \frac{\partial Q^n}{\partial x_k} \]  

(10)

\[ \Delta \rho = \left( \frac{1}{\epsilon^2} \right)^n \Delta \rho = -\Delta t \left[ \frac{\partial U_i^n}{\partial x_i} + \theta_i \frac{\partial \Delta U_i^*}{\partial x_i} - \Delta \theta \left( \frac{\partial^2 p^n}{\partial x_i \partial x_i} + \theta_2 \frac{\partial^2 \Delta \rho}{\partial x_i \partial x_i} \right) \right] \]  

(11)

The Galerkin form of equations:

The Standard Galerkin approximation yields in the following form

\[ \int \Omega N^k_u \Delta U_i^* d\Omega = -\Delta t \int \Omega \frac{\partial}{\partial x_j} \left( u_j U_i \right) d\Omega - \int \Omega \frac{\partial^2}{\partial x_i \partial x_j} \tau_{ij} d\Omega - \int \Omega N^k_r \left( \rho g_i \right) d\Omega \]

\[ + \frac{\Delta t^2}{2} \left[ -\int \Omega \frac{\partial}{\partial x_i} \left( u_j N^k_u \right) \left( -\frac{\partial}{\partial x_j} \left( u_j U_i \right) + \rho g_i \right) d\Omega \right] \]

\[ + \Delta t \left[ \int \Gamma N^k_r \tau_{ij} n_j d\Gamma \right] \]  

(12)

The weak form of the density-pressure equation becomes

\[ \int \Omega N^k_p \Delta \rho d\Omega = \int \Omega \frac{1}{\epsilon^2} \Delta \rho d\Omega \]

\[ - \Delta t \int \Omega \frac{\partial N^k_p}{\partial x_i} \left[ U_i^n + \theta_i \left( \Delta U_i^* - \theta_1 \frac{\partial P^{n+\theta_i}}{\partial x_i} \right) \right] d\Omega \]

\[ - \Delta t \theta \int \Gamma N^k_r \left( U_i^n + \Delta U_i^* - \Delta t \frac{\partial P^{n+\theta_i}}{\partial x_i} \right) n_i d\Gamma \]  

(13)

The weak form of the correction step is

\[ \int \Omega N^k_u \Delta U_i^{n+1} d\Omega = \int \Omega N^k_u \Delta U_i^* d\Omega - \Delta t \int \Omega \left( \frac{\partial p^n}{\partial x_i} + \theta_2 \frac{\partial \Delta \rho}{\partial x_i} \right) d\Omega \]

\[ - \frac{\Delta t^2}{2} \int \Omega \frac{\partial}{\partial x_i} \left( u_j N^k_u \right) \frac{\partial p^n}{\partial x_j} d\Omega \]  

(14)

The weak form of the energy equation is

\[ \int \Omega N^k_E \Delta \left( \rho E \right)^{n+1} d\Omega = \int \Omega \frac{\partial}{\partial x_i} \left( u_j \left( \rho E + p \right) \right) d\Omega - \int \Omega \frac{\partial^2}{\partial x_i \partial x_j} \left( \tau_{ij} u_j + k \frac{\partial T}{\partial x_i} \right) d\Omega \]

\[ + \frac{\Delta t^2}{2} \int \Omega \frac{\partial}{\partial x_i} \left( u_j N^k_E \right) \left[ \frac{\partial}{\partial x_j} \left( -u_i \left( \rho E + p \right) \right) \right] d\Omega \]

\[ + \Delta t \int \Gamma N^k_E \left( \tau_{ij} u_j + k \frac{\partial T}{\partial x_i} \right) n_i d\Gamma \]  

(15)

The lid driven cavity problem has been studied by many investigators Ghia [2], Zienkiewize and Codina [5] and results are produced in figures (1-7) which are self explanatory.
Fig. 1: Lid-driven cavity. Geometry, boundary conditions and mesh.

Fig. 3: Lid-driven cavity. Pressure distribution along horizontal mid-plane for different Reynolds numbers (semi-implicit form).

Fig. 4: Lid-driven cavity. Streamlines and pressure contours for different Reynolds numbers (semi-implicit form).

Fig. 2: (a) Stokes flow, viscosity 1. (b) Re = 400. (c) Re = 1000. (d) Re = 5000. (e) Re = 4000. (f) 4400 iterations. (g) Re = 1000. 6100 iterations. (h) Re = 5000. 48000 iterations.
The Multi-grid method:

To describe the process, let us consider the problem of

\[ L\phi = f \quad \text{in} \quad \Omega \]  

(16)

which we discretize incorporating the boundary conditions suitably. On a coarse mesh the discretization results in

\[ K^c\tilde{\phi} = f^c \]  

(17)

which can be solved directly or iteratively and generally will converge quite rapidly if \( \tilde{\phi}^c \) is not a big vector. The fine mesh discretization is written in the form

\[ K^f\tilde{\phi}^f = f^f \]  

(18)

and we shall start the iteration after the solution has been obtained on the coarse mesh. Here we generally use a prolongation operator, which is generally an interpolation from which the fine mesh values at all nodal points are described in terms of the coarse mesh values. Thus

\[ \phi_i^f = P \phi_i^{c,-1} + \Delta\phi_i^f \]  

(19)

where \( \Delta\phi_i^f \) is the increment obtained in direct iteration. If the meshes are nesting then of course the matter of obtaining \( P \) is fairly simple, but interpolating from a coarser to a finer mesh even if the points are not coincident can do this quietly generally. Obviously the values of the matrices \( P \) will be close to unity whenever the fine mesh points lie close to the coarse mesh ones. This leads to an almost hierarchical form. Once the prolongation to \( \phi^f \) has been established at a particular iteration \( I \) the fine mesh solutions can be attempted by solving

\[ K^f\Delta\phi^f = f^f - R^f \]  

(20)

where the residual \( R \) is easily evaluated from the actual equations. The solution need not to be complete and can well proceed for a limited number of cycles after which a return to the coarse mesh is again made to cancel out major low-frequency errors. At this stage it is necessary to introduce a matrix \( Q \), which transforms values from the fine mesh to the coarse mesh. We now write for instance
\[ \tilde{\phi}_i^c = Q\tilde{\phi}_i^f \]  
(21)

where \( Q \) is, of course, \( P^T \). In a similar way we can also write

\[ R_i' = QR_i^f \]  
(22)

where \( R_i \) are residuals. The above interpolation of residuals is by no means obvious but intuitively at least correct and the process is self-checking, as now we start a coarse mesh solution written as

\[ K(\tilde{\phi}_{i+1}^c - \tilde{\phi}_i^c) = R_i^c \]  
(23)

At this stage we solve for using \( \tilde{\phi}_{i+1}^c \) the values of previous iterations of \( \tilde{\phi}_i^c \) and putting the collected residuals on the right-hand side. This way of transferring residuals is by no means unique but has established itself well and the process is rapidly convergent. The problem lid driven cavity has been studied by Shah et. al. [4]. Results are produced in figures (9-12) are also self explanatory.

Comparative Analysis:

The two approaches CBS and multi-grid has been applied to incompressible Navier-Stokes problems in two-dimensional driven cavity. Solutions obtained by CBS though time dependent but we choose steady state solutions only for the comparison with multi-grid steady state solutions.

The multi-grid solutions are much faster in terms of convergence and computational complexity than CBS. Hence it is more efficient if the multi-grid methods are applied together with CBS algorithm. This was accelerated the convergence. However, multi-grid methods are designed meshes which are ‘nested’ i.e., in which coarser and finer mesh nodes coincide, this need not be the case generally, so care must be given for different meshes of varying density used at various stages. Moreover, physical problems are generally in three dimensions, so it would be more appropriate to use CBS-algorithm in developing three-dimensional codes for solving realistic problems. It is clear that the number of unknowns in three-dimensional problems will increase very rapidly and it is common when using linear order elements to encounter several million variables as unknown, and for this reason iterative processes are necessary.

Multi-grid method is one of the fastest iterative schemes applied to both two and three-dimensional problems. Effort must be done to use hybrid scheme of multi-grid and CBS-algorithm.

II. CONCLUSION

The incompressible Navier-Stokes equations have been solved for benchmark problems like driven cavity. The solutions were obtained by the two approaches, CBS and Multi-grid methods and compared. The comparison is only restricted to the rate of convergence not in terms of solutions because CBS solved time dependent Navier-Stokes equations whereas multi-grid solutions are only steady state. However, it was concluded that the multi-grid methods process is much faster than the CBS. Hence the two approaches if apply together would definitely accelerate the iterative convergence. It is more appropriate if the realistic problems be solved in three-dimension using CBS-algorithm with multi-grid method while doing this, attention must be given to the elements when using coarser and finer elements. Elements of the finer vertices must coincide with the elements of the coarser vertices. Such techniques are of considerable value and will explore in future work for fluid dynamics problems.

III. Scope of the present Research

The present comparative analysis was done to the rate of convergence of the two solvers not the accuracy of the solutions. Since both CBS and multi-grid are considered to be the fast solvers so both could easily be applied to solve industrial problems of thermo-fluids, chemical industries and most effective in aero-space industries. The author suggested that it would be more appropriate to use combined solver for aerodynamics problems for example blade to blade fluid flow analysis of a jet engine which have taken huge computational time [7].

REFERENCES


