Engineering Analysis with Boundary Elements 72 (2016) 1–10

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

journal homepage: www.elsevier.com/locate/enganabound

A direct velocity-pressure coupling Meshless algorithm for incompressible fluid flow simulations

Andrés Vidal [a,*], Alain J. Kassab [a], Eduardo A. Divo [b]

[a] Department of Mechanical and Aerospace Engineering, University of Central Florida, Orlando, FL 32816, USA
[b] Department of Mechanical Engineering, Embry-Riddle Aeronautical University, Daytona Beach, FL 32114, USA

ARTICLE INFO

Article history:
Received 3 March 2015
Received in revised form
17 June 2016
Accepted 22 July 2016

Keywords:
Computational Fluid Dynamics
Meshless
RBF
Direct velocity-pressure coupling

ABSTRACT

A localized radial-basis function (RBF) Meshless algorithm, with a direct velocity-pressure coupling scheme, is presented for fluid flow simulations. The proposed method is a combination of several efficient techniques found in different Computational Fluid Dynamic (CFD) procedures and has very low numerical diffusion. The fundamental idea of this method lays on several important inconsistencies found in three of the most popular techniques used in CFD, segregated procedures, streamline-vorticity formulation for 2D viscous flows, and the fractional-step method, very popular in Direct Numerical Simulation (DNS) and Large-Eddy Simulation (LES). The proposed scheme uses the classical segregated point distribution for all primitive variables, and performs all necessary interpolations with the accurate RBF technique. The viscous term is estimated using standard second order finite differences, while the convection term is discretized using the low-diffusion flux limiters. The velocity-pressure coupling is performed with the flow equations in their original form, and using a direct velocity-pressure coupling scheme. This way of solving the flow equations has no approximations in the boundary conditions. The method is validated with the 2D lid-driven cavity problem and very good agreement is found with classical data.

1. Introduction

Since the first successful velocity-pressure coupling procedure presented by Harlow and Welch [1], and the later development of the *Projection Method*, the science of Computational Fluid Dynamics (CFD) has become a fundamental tool for engineering calculations and design. In the early 1980's [2], the Finite-Volume Method (FVM) was introduced and became very popular since the formulation uses very simple mathematical tools, so the method became widely accessible. At the beginning, most calculations were performed in geometries where the flow was parabolic, and the results agreed very well with experimental data. When the same procedure was used to solve elliptic flow problems, some differences appeared with experimental data, but there were no major errors between simulations performed with different numerical methods. In the 1980's, the Finite Element Method (FEM), very popular in structural analysis, was extended to fluid flow computations, and the calculation in irregular geometries became possible with the use of a coordinate transformation of the geometry, keeping the velocity vector in the original Cartesian form. With this procedure, the flow equations were solved in their original form but the cost of the solution of the corresponding linear system was excessive. In spite of the use of a cheaper alternative called the *frontal method* [3], the FEM procedure was not very popular to solve fluid flow problems. In the 1990's, the FVM method was extended to general curvilinear coordinate systems, or for short, body-fitted. When the segregated velocity-pressure coupling approach was used in irregular geometries, keeping the velocity vector in the Cartesian form, there were many cases in which convergence was impossible. The only solution was to perform a full transformation of the coordinates and velocity vector. This methodology is the most frequent approach in the solution of today's engineering problems (i.e. [4]). Of course, some researchers developed hybrid procedures, as for example, the combination of FVM and FEM.

Unfortunately, the approach of transforming any fluid flow problem into a general curvilinear system (orthogonal or non-orthogonal) imposes some additional numerical diffusion since all metric coefficients are computed using the standard finite differencing scheme. These errors make the Direct Numerical Simulations (DNS) and Large-Eddy Simulations (LES) simulations impractical in complex geometries. Basically, almost all DNS and LES simulations are done in Cartesian coordinates, limiting the scope

* Corresponding author.
E-mail addresses: Andres.Vidal@knights.ucf.edu (A. Vidal), Alain.Kassab@ucf.edu (A.J. Kassab), Eduardo.Divo@erau.edu (E.A. Divo).

http://dx.doi.org/10.1016/j.enganabound.2016.07.014
0955-7997/

of problems to be solved. In order to avoid the transformation of the domain and/or velocity vector into a curvilinear system, a new technique called *Meshless* was developed by many authors starting in the early 1990's. The works of Batina [5], Kansa [6,7] and Belytschko [8,9] were fundamental in the development of the Meshless technique. Today, the main idea of any Meshless procedure is to solve the flow equations in Cartesian coordinates and employ high order interpolations for irregular geometries.

The common issue with all these popular techniques is that the velocity-pressure coupling is done following the same projection method. With some minor modifications, the original flow equations are transformed into a series of consecutive and explicit equations for velocity, pressure and mass correction, this last one needing to satisfy the mass balance. The transformation of the flow equations into an equivalent system of Poisson or Poisson-like equations is an incredibly simple solution for the problem of how to link pressure to velocity without an explicit equation for pressure. This idea became the core of the development of basically all commercial CFD software industry today.

However, the Boundary Element Method (BEM) together with the stream function-vorticity formulation is able to overcome the difficulties of the classical segregated procedure with primitive variables, obtaining good results [10–14].

The normal approach in CFD, for DNS and LES simulations, has been to extend the current CFD procedures and perform some minor changes intended to reduce the so-called *numerical diffusion error*. In almost all procedures that transform the original flow equations into a system of segregated equations, the simplification of the boundary conditions has been fundamental to solve any fluid flow problem. The simplification, or simply speaking, the set of boundary conditions of the transformed fluid flow equations, has always produced considerable controversy. For example, the Poisson equation for pressure with the boundary conditions obtained from the Navier-Stokes equations, make the convergence of any segregated procedure too slow for high demanding problems. In the particular case of the wall, one practical solution is to set the normal derivative of pressure to zero. For parabolic problems, the above simplification is correct, and the convergence is fast. In the case of general elliptic problems, even with a grid that is fine enough so that the first nodes are inside the boundary layer, it has not yet been proven that this consideration is absolutely correct.

In spite of the improvement in all CFD techniques, even today the solution of complex elliptic problems, such as the 2D/3D backward-facing step or lid-driven cavity, is still a mayor challenge. These two cases have produced by far the largest amount of differences in numerical results between procedures. Many authors have explained this effect as *bifurcation of the solution*.

The objective of this work is to present a hybrid approach, by combining the direct velocity-pressure formulation presented in [15] and a high order interpolation technique.

In more detail, the idea of this work is to present an alternative approach, the use of a localized RBF Meshless procedure to solve the flow equations in the original form, so that there is no simplification or approximation of any boundary condition, and solving the corresponding linear system using any standard matrix procedure. Additionally, the proposed numerical procedure tries to minimize the generation of additional numerical diffusion, a very popular approach to improve stability of the numerical scheme. The velocity-pressure coupling procedure is the same one developed in [15]. The staggered point distribution approach (or grid) is selected and the RBF scheme is chosen to perform any necessary interpolation. Finally, in order to keep the numerical diffusion at a very low level, the well known flux-limiting scheme will be used in the discretization of the convection terms.

It is convenient to underline that meshless means volume-less or element-less. Essentially, the equations are not integrated in any control-volume or element. In a meshless approach, the idea is to scatter points in space and then use a generalized form finite-differencing in Cartesian coordinates to compute the derivatives.

2. Velocity-pressure coupling

The main objective of any CFD procedure in the solution of incompressible flow problems is to compute velocity and pressure from momentum and continuity equations.

$$\nu\,\nabla^2\vec{V} - \left(\vec{V}\cdot\nabla\right)\vec{V} - \frac{1}{\rho}\nabla p = \vec{0}$$
$$\nabla\cdot\vec{V} = \vec{0} \tag{1}$$

This set of equations is quite simple, but unfortunately, there is no explicit equation for pressure and the implementation becomes difficult for non expert programmers. The development of a robust procedure able to compute velocity and pressure from those incompressible flow equations, is called *velocity-pressure coupling problem*.

The most popular approach, the projection method, also called the segregated approach, solves the flow equations by a series of Poisson-like equations. The SIMPLE method developed by Patankar [2] and its variants have been implemented in most commercial packages.

However, the mass correction equation, key step in the segregated coupling scheme, works correctly for parabolic flows, with important errors for elliptic flows if the grid is not fine enough. Additionally, the pressure equation may lead to incorrect values if the grid is too stretched in one particular direction. More details about these issues and the analysis on some particular methods can be found in [16].

2.1. The fully coupled procedure

The most straightforward way to solve the equations of motion is by solving the discrete linear system, momentum and continuity, with a sparse direct matrix solver. The fact that there is no pressure term in the continuity equation makes the condition number of that full system extremely large. Additionally, the amount of resources needed to solve a sparse linear system of several millions of equations is prohibitive in 3D simulations. One way to reduce the enormous computational resources needed for this problem is by dividing the entire region in small blocks or sub-domains.

In a given block, the linear system of equations to be solved can be written as, in 2D (where $\vec{p}$ is a vector containing the pressure in all grid points):

$$\mathbf{A}\vec{u} + \mathbf{B}\vec{p} = \mathbf{b}^u, \quad \mathbf{A}\vec{v} + \mathbf{C}\vec{p} = \mathbf{b}^v, \quad \mathbf{D}\vec{u} + \mathbf{E}\vec{v} = 0 \tag{2}$$

If boundary conditions are excluded a pressure equation can be obtained by simple matrix manipulation:

$$\left(\mathbf{DA^{-1}B} + \mathbf{EA^{-1}C}\right)\vec{p} = \mathbf{DA^{-1}b^u} + \mathbf{EA^{-1}b^v} \tag{3}$$

The resulting matrix is diagonally-dominant and has non-zero diagonal coefficients and consequently, a low condition number.

2.2. A local coupling procedure

A useful alternative that solves the problems associated with both segregated and direct full coupling procedures is presented in [15]. In general, this scheme uses the segregated grid arrangement in the same way as the finite volume method. The fundamental aspect of this coupling approach is that the velocity-pressure