Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

A THREE DIMENSIONAL SIMULATION OF HYDROCYCLONE BEHAVIOUR

T. DYAKOWSKI[1], A.F. NOWAKOWSKI[1], W. KRAIPECH[1] and R.A. WILLIAMS[2]

[1]Department of Chemical Engineering,
University of Manchester Institute of Science and Technology,
PO Box 88, Manchester M60 1QD, UK

[2]Centre for Particle and Colloid Engineering
School of Process Environmental and Material Engineering,
University of Leeds, Leeds LS2 9JT, UK

ABSTRACT

In this work the mixed finite element approximation to the incompressible Navier-Stokes equations is discussed. The approach is based on a segregated finite element scheme and unstructured grids. Stable and second order accurate tetrahedral elements were used for spatial discretization of computational domain. The algorithm is applied to study the flow in hydrocyclones in three dimensions.

INTRODUCTION

Over the past few years there has been significant advancement in the techniques to simulate incompressible flow. However, numerical simulation of the flow in hydrocyclones is still not used for designing purposes. One of the reasons behind this is a lack of capability of existing methods to accept complex geometries in an integrated manner. The philosophy underpinning this work was to avoid the introduction of any numerical technique, which would restrict geometric flexibility. A finite volume method is not well suited to handle unstructured meshes, which are needed for modelling inlet geometry of hydrocyclones. It should be emphasized that the majority of existing numerical algorithms is mainly restricted to an axi-symmetrical case and, for such a case a finite volume technique was applied, Dyakowski and Williams [1]. When the finite element method is used, the formulation of the equations imposes no restriction in the mesh topology. Flows in complex geometries can be simulated using an irregular mesh where required.

The paper describes the development of the segregated mixed finite element scheme. In segregated velocity-pressure formulations of the Navier-Stokes equations, velocities and corresponding pressure field are uncoupled and computed in an iterative sequence. In contrast, in couple formulations the governing variables are simultaneously treated. The advantages of the uncoupled/iterative methods are twofold: (i) reduced computational cost (memory and CPU) and (ii) a significantly larger radius of convergence, Gresho [2]. The disadvantage is a significant increase in the number of iterations to achieve convergence.

In this investigation, to ensure that the Brezzi-Babuska div-stability condition is satisfied, Taylor-Hood interpolation at element level is used, Zienkiewicz [3].

GOVERNING EQUATIONS

As a formulation for incompressible flow problems the Navier-Stokes equations are considered in three dimensions

$$\boldsymbol{u} \cdot \text{grad}\, \boldsymbol{u} + \text{grad}\ p - \nu \Delta \boldsymbol{u} = 0 \quad \text{in } \Omega \tag{1}$$

together with the incompressibility constraint

$$\text{div}\, \boldsymbol{u} = 0 \qquad \text{in } \Omega \tag{2}$$

where $\boldsymbol{u} = (u, v, w)$ denotes the velocity field, p the pressure and ν is the given constant kinematic viscosity. The constant density has been absorbed into the pressure. The inclusion of body force term $\boldsymbol{f}$ on the right hand side would not change anything in the subsequent analysis. The statement of the problem is made complete by the specification of the suitable boundary condition. It is enough, if the velocity vector is given on all boundaries, provided that the boundary condition satisfies the incompressibility constraint that the same amount of fluid enters and leaves the domain Ω.
That is

$$\boldsymbol{u} = \boldsymbol{g} \qquad \text{on } \partial\Omega \tag{3}$$

The boundary velocity $\boldsymbol{b}$ must satisfy the global condition (2)

$$\int_{\Omega} \text{div}\, \boldsymbol{u}\, d\Omega = \oint_{\partial\Omega} \boldsymbol{g} \cdot \boldsymbol{n}\, ds = 0 \tag{4}$$

where $\boldsymbol{n}$ is the outward unit normal vector to the boundary $\partial\Omega$.

The system of equation (1)-(2) with the boundary conditions (3) and (4) results in a solution determined up to an additive constant in pressure.

MIXED FINITE ELEMENT APPROXIMATION

Weak Form

The mixed finite element method has been applied to the governing equations. Galerkin type weak formulation of (1) and (2) has the form

$$\int_{\Omega} \varphi \cdot \mathbf{f}_1 d\Omega = 0 \qquad \text{in } \Omega \tag{5}$$

$$\int_{\Omega} \mu f_2 d\Omega = 0 \qquad \text{in } \Omega \tag{6}$$

where (φ, μ) are weight functions, which are equated to the interpolation functions used for $(p, \boldsymbol{u})$ For convenience the left sides of equation (1) and (2) are replaced by $\boldsymbol{f}_1$ and f_2 respectively.

Finite Element Model

In order to prevent an over-constrained system of discrete equations the interpolation used for pressure must be at least one order lower than that used for velocity field Sani [4,5]. The Taylor-Hood P_2P_1 type of tetrahedral elements are used in this approach to ensure second order accuracy and to satisfy the crucial div-stability condition, Gunzburger [6], Gresho [7]. The choice of these elements means that velocity (each component) is approximated by piecewise continuous quadrilateral polynomial and pressure by piecewise continuous linear polynomial. Both velocity and pressure are continuous across element boundaries and each tetrahedral element contains ten velocity nodes and four pressure nodes. Dependent variables, pressure and velocity, can be written by expansion

$$p^e = \sum_{j=1}^{4} p_j^e \mu_j^e = \boldsymbol{\mu}^T \mathbf{p} \tag{7a}$$

$$u_i^e = \sum_{j=1}^{10} u_j^e \varphi_j^e = \boldsymbol{\Phi}^T \mathbf{u}_i \tag{7b}$$

where $\boldsymbol{\mu}$ and $\boldsymbol{\Phi}$ are vectors of interpolation (shape) functions, and $\mathbf{u}_i$ and $\mathbf{p}$ are vectors of nodal values of velocity components and pressure, respectively. Substitution of equations (7a) and (7b) into momentum (1) and continuity (2) equations results in the finite element equations. The derived equations can be written symbolically in matrix form as.

$$\begin{bmatrix} \mathrm{K}_{uu} & \mathrm{K}_{uv} & \mathrm{K}_{uw} & -\mathrm{C}_x \\ \mathrm{K}_{vu} & \mathrm{K}_{vv} & \mathrm{K}_{vw} & -\mathrm{C}_y \\ \mathrm{K}_{wu} & \mathrm{K}_{wv} & \mathrm{K}_{ww} & -\mathrm{C}_z \\ \mathrm{C}_x^T & \mathrm{C}_y^T & \mathrm{C}_z^T & 0 \end{bmatrix} \begin{bmatrix} u \\ v \\ w \\ p \end{bmatrix} = \begin{bmatrix} \mathrm{F}_x \\ \mathrm{F}_y \\ \mathrm{F}_z \\ \mathrm{b}_c \end{bmatrix} \tag{8}$$

where **K** sub-matrices represent the combined effects of convection and diffusion; **C** matrices are the pressure gradient operators and their transposes; $\mathbf{C}^{\mathbf{T}}$ matrices appearing in the continuity equation are the velocity divergence operators. The vector $\mathbf{b_c}$ on the right hand side of the continuity equation represents the contribution to this equation from the non-zero Dirichlet velocity boundary conditions. The vector **F** contains surface-flux type contributions from the natural boundary conditions as well as a body force.

Equation Solver

As an equation solver the segregated type of iterative algorithm has been employed. The applied pressure projection algorithm was proposed by Horoutunian et al., [8]. It is a consistent finite element counterpart of the SIMPLER algorithm first introduced by Patankar [9]. The primary variables were de-coupled directly from momentum and continuity equations (10) at the discretized level. The algorithm comprises the following main steps:

(0) given an initial or guess solution field (u, v, w, p) for i = 0,1,2... until convergence the following steps should be taken

(1) solve for pressure p

$$\left[C_x^T\left(\widetilde{K}_u^{-1}\right)^* C_x + C_y^T\left(\widetilde{K}_v^{-1}\right)^* C_y + C_z^T\left(\widetilde{K}_w^{-1}\right)^* C_z\right] p^{i+1/2}$$
$$= -C_x^T\left(\widetilde{K}_u^{-1}\right)^*\left[f_u^* - K_u^* u^i\right] -$$
$$C_y^T\left(\widetilde{K}_v^{-1}\right)^*\left[f_v^* - K_v^* v^i\right] - C_z^T\left(\widetilde{K}_w^{-1}\right)^*\left[f_w^* - K_w^* w^i\right]$$

(2) relax pressure via

$$p^{i+1} = \alpha_p p^i + \left(1-\alpha_p\right) p^{i+1/2}$$

(3) solve x-momentum equation for u

$$\left[\left(\frac{\alpha_u}{1-\alpha_u}\right)\widetilde{K}_u + K_u\right]^* u^{i+1/2} = f_u^* + C_x p^{i+1} + \left(\frac{\alpha_u}{1-\alpha_u}\right)\widetilde{K}_u^* u^i$$

(4) solve y-momentum equation for v

$$\left[\left(\frac{\alpha_v}{1-\alpha_v}\right)\widetilde{K}_v + K_v\right]^* v^{i+1/2} = f_v^* + C_y p^{i+1} + \left(\frac{\alpha_v}{1-\alpha_v}\right)\widetilde{K}_v^* v^i$$

(5) solve z-momentum equation for w

$$\left[\left(\frac{\alpha_w}{1-\alpha_w}\right)\widetilde{K}_w + K_w\right]^* w^{i+1/2} = f_w^* + C_y p^{i+1} + \left(\frac{\alpha_w}{1-\alpha_w}\right)\widetilde{K}_w^* w^i$$

(6) solve SCPE for pseudo-pressure p^s

$$\left[C_x^T\left(\widetilde{K}_u^{-1}\right)^* C_x + C_y^T\left(\widetilde{K}_v^{-1}\right)^* C_y + C_z^T\left(\widetilde{K}_w^{-1}\right)^* C_z\right]p^{i+1/2} =$$
$$-C_x^T u^{i+1/2} - C_y^T v^{i+1/2} - C_z^T w^{i+1/2} + b_c$$

(7) mass adjust velocity field via

$$u^{i+1} = u^{i+1/2} + \left(\widetilde{K}_u^{-1}\right)^* C_x p^s$$
$$v^{i+1} = v^{i+1/2} + \left(\widetilde{K}_v^{-1}\right)^* C_y p^s$$
$$w^{i+1} = w^{i+1/2} + \left(\widetilde{K}_w^{-1}\right)^* C_z p^s$$

Where:
$$f_u = F_u - K_{uv}v - K_{uw}w$$
$$f_v = F_v - K_{vu}u - K_{vw}w$$
$$f_w = F_w - K_{wu}u - K_{wv}v$$

In the above equations the superscripts i, i+1/2 and i+1 denote previous, intermediate, and latest iterate levels, respectively, while the superscript * denotes an expression involving the latest available field variables. The $\widetilde{K}$ matrices are incomplete versions of the full K matrices. It is necessary to stress that equations have been derived at the discretized level from manipulations on the dicretized form of the momentum (5) and continuity equation (6). As a consequence the boundary conditions on pressure implicitly implied in equation (1) through the boundary conditions on velocity, are automatically and consistently imposed in pressure equation - step (1). The equations are characterised by generic system **Ax = b** and are solved sequentially and repeatedly during the course of iteration. At the beginning of a given iteration, an approximation to the pressure is obtained from the solution of a simplified pressure equation using the latest available field variables. The components of the momentum equations are then solved in sequential manner using the most recent field data. Finally, at the end of the whole sequence, the velocity field is corrected to satisfy the discretized continuity equation.

PRELIMINARY RESULTS

The driving force of the presented paper is the need for modelling the complex inlet geometry. Figure 1 shows the grid being developed for simulation of the head entry region with inclined walls. The grid consists of tetrahedral elements enabling the matching of the computational domain to the physical boundaries.

A comparison between the proposed unstructured grid against the multi-block curvilinear grid (He et al [10]) is presented in Figure 2. It should be emphasized, that Slack and Boysan [11] also applied the concept of the unstructured grid in their latest version of Fluent software.

A global refinement and adjustment of the proposed grid can be obtained as follows. Firstly, global grid parameters and the number of subdivisions can be changed (Figure 3). Secondly, each tetrahedron can be subdivided into twelve smaller tetrahedrons by giving the centroid of the large tetrahedron to the vertices and centroids of the faces. The drawback of the proposed method is that the matrices of the algebraic equations do not have a convenient structure.

CONCLUSIONS

A numerical algorithm to study incompressible fluid flow has been introduced. This algorithm will be implemented to study the flow in hydrocyclones. Three-dimensional unstructured grids based on tetrahedral elements have been developed. The accurate representation of a computational domain allows researching into how changes in the shape of hydrocyclone will influence its operating performance. The more advanced computations may exceed the current workstation capabilities. It has always been the problem of the FEM that larger computational times have been associated with it. This is especially the case in the area of incompressible and turbulent fluid flow. However, the ability of modern supercomputers allows the approximation of three dimensional flow in hydrocyclones to be attempted.

REFERENCES

1. DYAKOWSKI, T. and WILLIAMS, R.A., Modelling turbulent flow within a small-diameter hydrocyclone, *Chem. Eng. Science*, 48 (1993) 1143.
2. GRESHO, P.M., SANI, R.L., *Incompressible Flow and the Finite Element Method*, John Wiley and Sons, Chichester, 1998
3. ZIENKIEWICZ, O. and TAYLOR, R., *The Finite Element Method, Basic formulation and Linear Problems,* New York, 1989, pp 324-327
4. SANI, R.L., Gresho, P.M., LEE, R.L. and GRIFFITHS, D.F., The cause and cure (?) of the spurious pressures generated by certain FEM solutions of the incompressible Navier-Stokes equations: Part 1: *International Journal of Numerical Methods in Fluids*, 1, 17-43 (1981).
5. SANI, R.L., GRESHO, P.M., LEE, R.L. and GRIFFITHS, D.F., The cause and cure (?) of the spurious pressures generated by certain FEM solutions of the incompressible Navier-Stokes equations: Part 2: *International Journal of Numerical Methods in Fluids*, 1, 171-204 (1981).
6. GUNZBURGER, M.D., *Finite element Method for Viscous Incompressible Flow*, Academic Press, San Diego, 1989
7. P.M. Gresho, Incompressible fluid dynamics: Some fundamental formulation issues: *Annu. Rev. Fluid Mech*. 23, 413-453 (1991)
8. HAROUTUNIAN, V., ENGELMAN, M.S. and HAASBANI, I., Segregated Finite element algorithms for the numerical solution of large-scale incompressible flow problems: *International Journal of Numerical Methods in Fluids*, 17, 323-348 (1993).
9. PATANKAR, S.V., *Numerical Heat transfer and Fluid Flow*, Hemisphere, Washington, DC, 1980.
10. HE, P., SALCUDEAN, M. and GARTSHORE, I.S., A Numerical Simulation of Hydrocyclones, *Trans. IchemE*, **77**, Part A, 429-441, (1999).
11. SLACK, M.D. and BOYSAN, F., Advances in Cyclone Modelling using Unstructured Grids, *Scandinavian FLUENT UGM*, Gothenburg, 1-9, (1998).

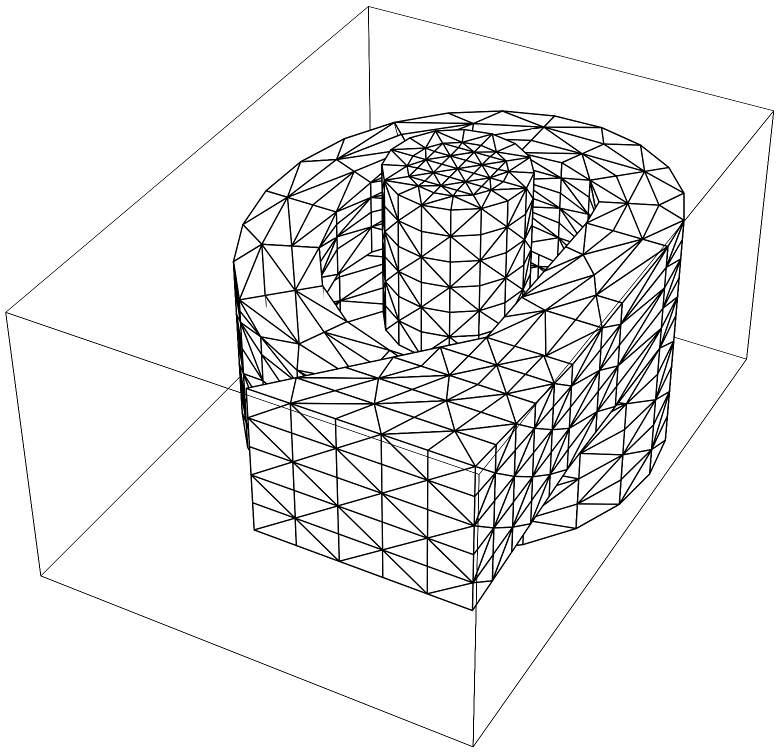

Figure 1. Mesh in the profiled head entry region of hydrocyclone

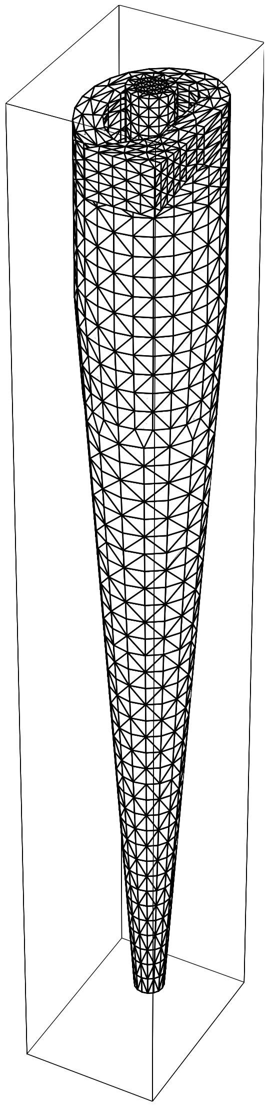

Figure 2: (a) The unstructured grid consisting of tetrahedral elements within hydrocyclone

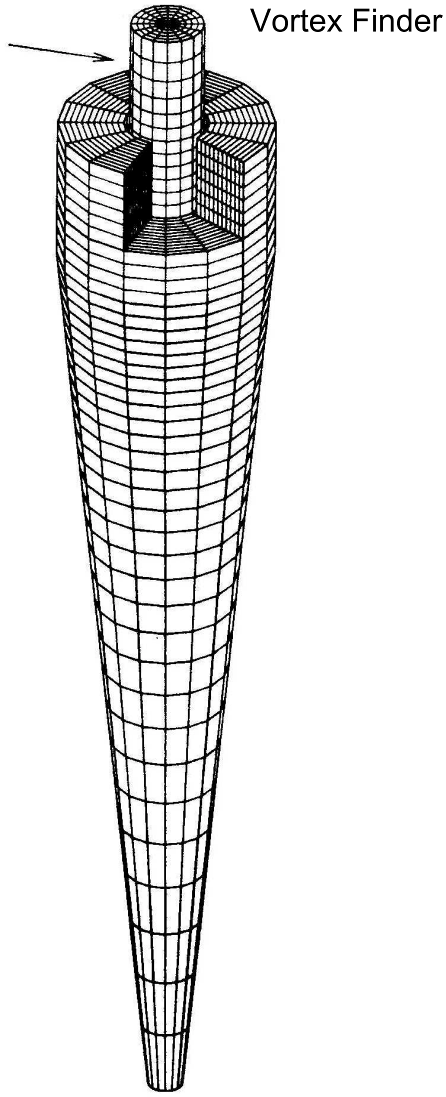

Figure 2: (b) Three segment mesh at the z – r plane (He et al., 1999)

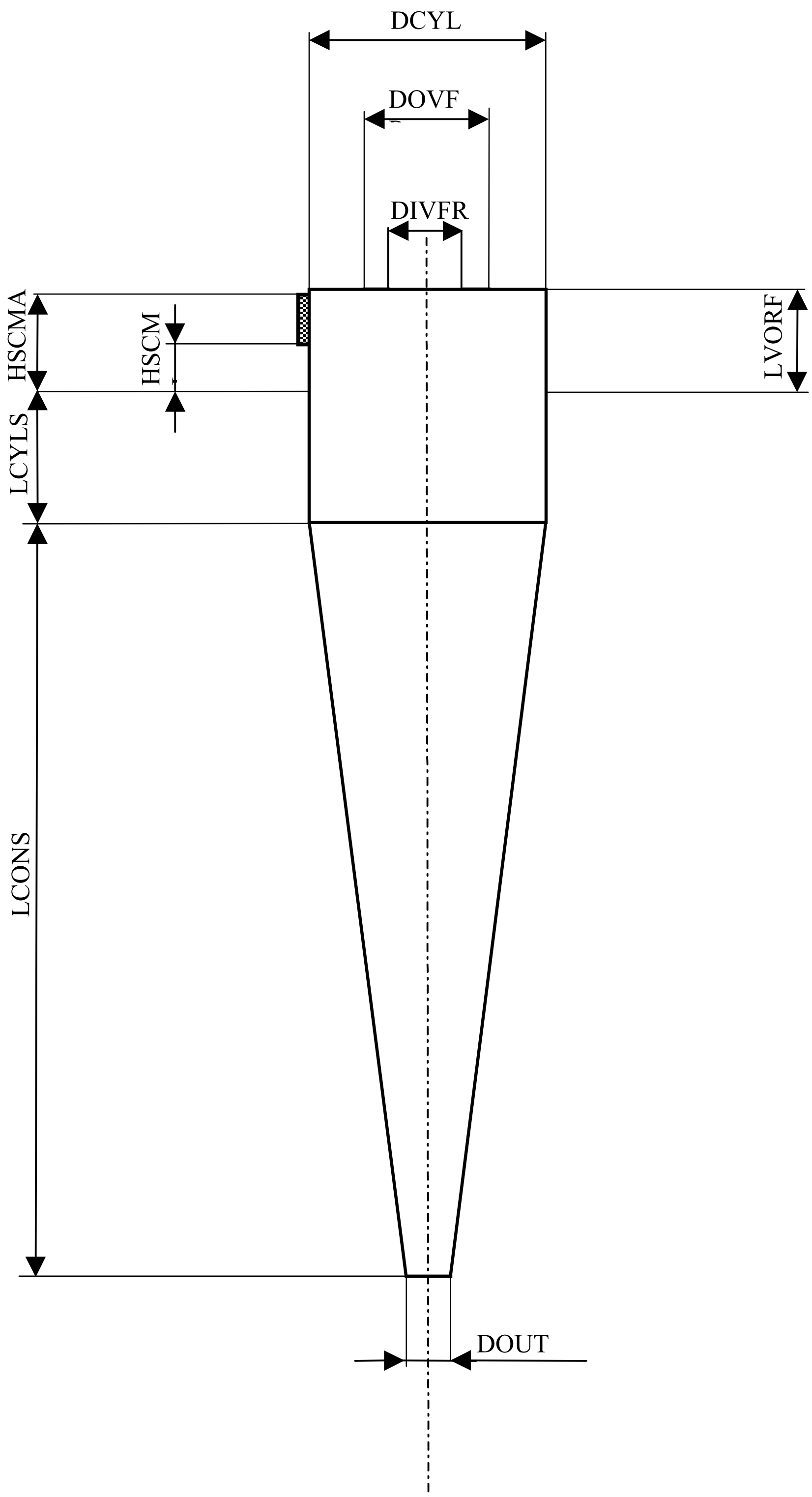

Figure 3: Hydrocyclone configuration – the shown parameters can be modified