Numerical Calculation and Visualization of Three-dimensional Flows of Engineering Interest

I. Cuesta, *J. Pallares*, *F.X. Grau*, and *Francesc Giralt*, SEECAT, Universitat Rovira i Virgili, Tarragona, Spain.

ABSTRACT: *The FORTRAN CFD code 3DINAMICS has been developed to simulate momentum, mass, and heat transfer in laminar and turbulent unsteady three-dimensional flows. The code has been successfully applied to the simulation of several engineering flows, particularly we present four different ones: lid-driven cavity, natural convection in a cubical cavity, hydrodynamics in shallow ponds and three-dimensional transient flow in a dump combustor. Once this results have been obtained they have been interpreted converting the numerical files into pictures. In addition to the classical flow visualizations the current work explores two alternative magnitudes to describe the structural characteristics of flows.*

Introduction

Visualization of complex flow fields has become indispensable for computational and experimental fluid flow researches, providing insights into data that would otherwise be impossible. Scientific visualization is therefore one of the fastest growing and most important areas of high performance computing.

The visualization of three dimensional dynamic fields is still a challenge in CFD, mainly because of their vectorial character. The objective of such visualization is to portray an image of the flow structure in order to allow the identification and analysis of complex flow patterns.

Several flow visualization techniques have been developed for analyzing flow fields. Such techniques may be classified into two groups according the nature of the variable of flow that is to be represent. The first group correspond to de vector variables (for instance velocity) and the second contains de representation of scalar variables (pressure, temperature, etc.).

The most classical way to show flow patterns has been the representation of the direction and magnitude of the velocity vectors and the lagrangian representation of the dynamic field by particle tracing: streamlines, pathlines, streaklines, etc.

The purpose of this study is to characterize numerically four different flows; the lid-driven cavity, the natural convection in a cubical cavity, the hydrodynamics in shallow ponds and the transient flow in a dump combustor. The topology of the different types of structures is examined incorporating the representation of the second invariant of the velocity gradient into the previously standard plotting techniques.

Governing equations

In all of the four cases the flow is considered incompressible and the variation of fluid properties with temperature has been neglected, with the only exception of the buoyancy term for natural convection flow, for which the Boussinesq approximation has been adopted. As a result, the Navier-Stokes and the energy transport equations are coupled only by the body force term, where linear dependence of density with temperature is assumed.

All flows evolve in time t and are described in terms of the velocity u_i at any position x_i, pressure P, and temperature T (for non-isothermal calculations). The set of elliptic partial differential equations governing a single-phase, constant property incompressible flow in cartesian coordinates may be written as:

Continuity equation

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

Momentum

$$\frac{\partial u_i}{\partial t} + \frac{\partial (u_j u_i)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \, \partial x_j} + g_i \tag{2}$$

Energy

$$\frac{\partial T}{\partial t} + \frac{\partial (u_j T)}{\partial x_j} = \alpha \frac{\partial^2 T}{\partial x_j \, \partial x_j} \tag{3}$$

where ν is the fuid kinematic viscosity, g_i the gravitational acceleration and α the thermal diffusivity.

Numerical method and computational procedure

Numerical solutions of the governing equations in terms of the primitive variables were obtained by using the CFD code 3DINAMICS (Cuesta, 1993; Cuesta et al., 1995). In this code, equations (1-3) are discretized following a control volume scheme (Patankar, 1980) with non-uniform three-dimensional staggered grids. The convective fluxes across the surfaces of the control volumes were discretized using a second order QUICK scheme (Leonard, 1979). 3DINAMICS incorporates the non-uniform grid version of QUICK proposed by Perng and Street (1989) to enhance convergence. Second order approximations were also implemented for the diffusive fluxes with a centred scheme. The SMAC algorithm Amsden and Harlow (1970) was applied to compute the velocity-pressure coupling.

A critical step in the pressure algorithm, that affects the global performance of the code, is the optimal solution of the Poisson equation. It is for this reason that a preconditioned version of the Conjugate Gradient Method (CGM) was adopted in the present study.

Time marching integration is carry out by means of two different, second order accurate, schemes namely the semimplicit ADI, and the fully explicit Adams-Bashforth.

Results and discussion

The most common approach to visualize three-dimensional structures in unsteady flows is to plot vector fields, streamlines, streaklines and particle lines. In this work the second invariant of the gradient of the velocity tensor (P) is also used to describe the structural characteristics of the flow (see Hunt et al., 1988). This magnitude is defined as

$$\Pi = \frac{\partial u_i}{\partial x_j}\frac{\partial u_j}{\partial x_i} = S_{ij}S_{ji} + \Omega_{ij}\Omega_{ji} \qquad (4)$$

where S_{ij} is the strain tensor and Ω_{ij} the rotation tensor

Lid-driven Cavity Flow

The classical 3D driven cavity flow is used to show the ability of some visualization techniques. For Reynolds numbers higher than 1000 the physics of the flow is complex and exhibits significant transverse motions, i.e. Taylor-Görtler-like vortices. Figure 1 shows for Re = 2000 the three-dimensional particle traces that show clearly the location and size of the main eddy.

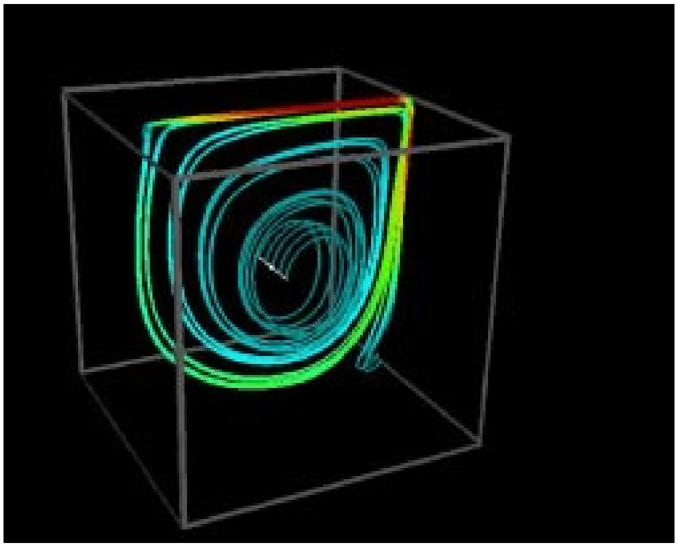

Figure 1. Particle traces of the lid-driven cavity flow at Re 2000.

Natural convection in a cubical cavity heated from below

This flow is the result of natural convection in a cubical cavity where buoyancy is induced by imposing a moderate temperature difference between the bottom (hot) and the top (cold) plates, while vertical walls are perfectly adiabatic. The topology of the types of stable convective structures are examined and characterized by using the alternative magnitude of the second invariant of the velocity gradient tensor, Π. Figure 2 shows the isosurfaces Π for one of the four structures obtained at Ra = 9000.

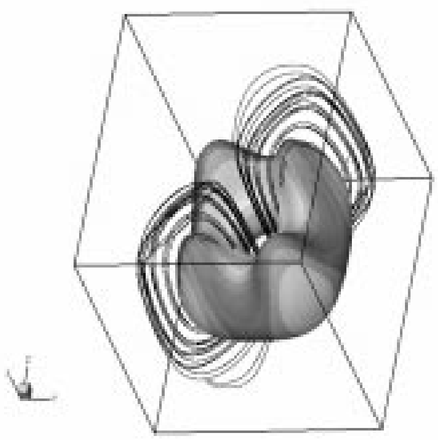

Figure 2. Isosurface Π for one of the structures at Ra = 9000.

Hydrodynamics in shallow ponds due to surface shear

In this case, numerical results correspond to the 2D laminar isothermal flow in a shallow rectangular enclosure of uniform depth filled with water. The flow circulation in the cavity is induced by a constant surface stress (wind) acting along the surface of air-water interface. Solutions have been obtained for several values of the Reynolds number (defined in terms of the horizontal length of the pond, the surface velocity, and the dynamic viscosity of water) ranging from 64 to 6480. As an example, Figure 3 shows the flow pattern for Re 6480.

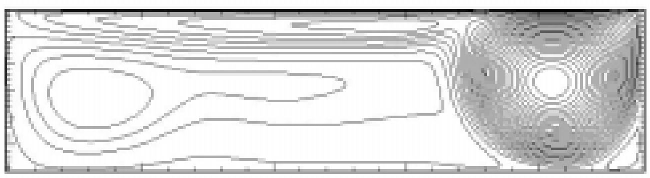

Figure 3. Flow pattern of the shallow pond for Re 6480

Transient recirculating flow in a dump combustor

This numerical study deals with the recirculating flow in a low speed dump combustor. The motivation of this study is to understand the nature of the flow instabilities that occur under certain operating conditions. Numerical simulations show that for Reynolds numbers below 3000 the flow is steady and 2D. The critical Reynolds number is 3000 and corresponds to an inlet velocity of 3 m/s. When this velocity is reached, the flow becomes unsteady and exhibits a periodic side-to-side oscillation. As shown in Figure 4, the computed stream-lines is a good technique to capture and visualize the previously described flow oscillations.

Concluding Remarks

This numerical study presents a detailed description of the topology of the different structures present in several engineering flows. It is interesting to note the ability of the second invariant of the velocity gradient tensor to portray the structural characteristics of the flow.

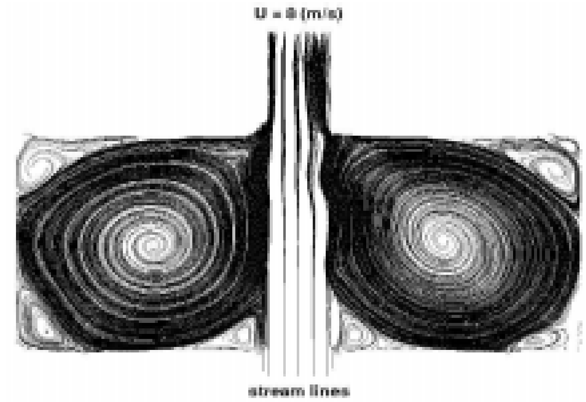

Figure 4. Stream lines for air at 8 m/s in the combustor

Acknowledgements

Most of the computations were performed on a Cray C90 16E at the Cray Research Inc. Supercomputer Center in Minnesota. The authors would like to acknowledge this company for providing the cpu time needed. The authors acknowledge the financial support received from DGICYT project no. PB93-0656-C02-01, as well.

References

[1] Cuesta, I. 1993. Estudi numèric de fluxos laminars i turbulents en una cavitat cúbica, Tesi Doctoral, Universitat Rovira i Virgili, Tarragona.

[2] Cuesta, I., Grau, F. X. & Giralt, F. 1994. Simulación numérica de flujos 3D no estacionarios mediante el código de segundo orden 3DINAMICS, Acepted to Anales de Química.

[3] Patankar, S.V. 1980. Numerical heat transfer and fluid flow. Series in Comp. Meths. in Mech. and Term. Sc., Mc Graw Hill.

[4] Leonard, B.P. 1979. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. *Comp. Meth. in App. Mech. and Eng*. **19**, 59-88.

[5] Perng, CH-Y. and Street, R.L. 1989. Three-Dimensional Unsteady Flow Simulations: Alternative Strategies for a Volume-Averaged Calculation, *Int. J. for Numerical Methods in Fluids*, **9**, 341.

[6] Amsden, A. A. and Harlow, F. H. 1970. The SMAC method: A numerical technique for calculating incompressible fluid flows, Los Alamos Scientific Laboratory of the University of California, LA-4370.

[7] Hunt, J.C.R., Wray, A.A. and Moin, P. 1988. Eddies, streams and convergence zones in turbulent flows, *Proc. 1988 Summer Program, NASA-Stanford Center for Turbulence Research*, 193-208.