UNIVERSIDADE FEDERAL DO PARANÁ
Setor de Tecnologia
Departamento de Engenharia Mecânica - DEMEC

TM-701 DINÂMICA DOS FLUIDOS COMPUTACIONAL I – 2010/2

10º TRABALHO COMPUTACIONAL – 18 Ago 10

Até 15 Set 10 = esclarecimento de dúvidas; **Até 17 Set 10 = entrega**

Implementar um programa computacional para resolver numericamente, através do método de volumes finitos, o modelo matemático constituído pelas equações de conservação da quantidade de movimento linear em x e y (**equações de Burgers 2D**), relativo ao escoamento bidimensional de fluido incompressível com propriedades constantes, definido por

QMLx:
$$\rho\frac{\partial(u^2)}{\partial x} + \rho\frac{\partial(uv)}{\partial y} = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) - \frac{\partial p}{\partial x}$$

QMLy:
$$\rho\frac{\partial(uv)}{\partial x} + \rho\frac{\partial(v^2)}{\partial y} = \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) - \frac{\partial p}{\partial y} - B(x,y,Re=1)$$

Modelo numérico:
- Empregar o modelo descrito no capítulo 9 das notas de aula, que envolve arranjo co-localizado de variáveis, malha uniforme em cada direção, condições de contorno com volumes fictícios e solução segregada das equações.
- Utilizar o esquema UDS na advecção e o CDS na difusão e na pressão.
- Usar o método de Gauss-Seidel para resolver os dois sistemas de equações algébricas.
- Para interromper o processo iterativo, usar sobre $u(1/2;1/2)$ e $v(1/2;1/2)$ o procedimento da seção 3.4.5 das notas de aula.

Dados: $N_x = N_y = 13$ (volumes de controle incluindo dois fictícios em cada direção)
$I_V = 5$ $\mu = 1$ Pa.s $\rho = 1$ kg/m^3
C.C., solução analítica e campo de pressões: p. 193-195 de Shih et al. (1989), em anexo

Resultados a apresentar:
1) Gráfico da variação de $u(1/2;1/2)$ (em escala logarítmica), em cada iteração i, *versus* número da iteração (em escala decimal). No mesmo gráfico, outra curva com a variação de $v(1/2;1/2)$ em cada iteração.
2) Para $y = ½$, tabela contendo em cada linha (incluindo as condições de contorno): x, v analítico e numérico, e o erro.
3) Gráfico de v analítico e numérico *versus* x para $y = ½$, incluindo os dois contornos.
4) Para $x = ½$, tabela contendo em cada linha (incluindo as condições de contorno): y, u analítico e numérico, e o erro.
5) Gráfico de y *versus* u analítico e numérico para $x = ½$, incluindo os dois contornos.
6) Soluções analítica e numérica do fluxo de massa, e o erro.
7) Soluções analítica e numérica da força da tampa da cavidade sobre o fluido, e o erro.
8) Listagem impressa do programa computacional implementado (sem=nota zero; com=nota obtida).

Nos itens acima, para cada variável, **erro = solução analítica – solução numérica**

DIRETRIZES OBRIGATÓRIAS
1. Usar precisão dupla e apresentar os resultados com pelo menos 10 algarismos significativos.
2. Usar papel A4 branco ou folha com pauta.
3. O texto deve ser impresso ou escrito à caneta.
4. Identificar claramente cada item dos resultados a apresentar.
5. Apresentar os resultados na seqüência solicitada no trabalho.
6. Só apresentar os resultados solicitados no trabalho.

- Haverá perda de 10 pontos (de 100) para cada um dos itens acima (das diretrizes obrigatórias) que não for satisfeito.
- **Este trabalho computacional deve ser feito individualmente ou em equipe de até dois alunos.**
- Se tiver alguma dúvida, entre em contato com o professor antes do prazo de entrega.
- **Para avaliação do trabalho, não se aceita entrega atrasada.**

RECOMENDAÇÕES:
- Usar como base o programa implementado para fazer o 8º trabalho computacional.

INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 9, 193–212 (1989)

EFFECTS OF GRID STAGGERING ON NUMERICAL SCHEMES

T. M. SHIH AND C. H. TAN

Department of Mechanical Engineering, University of Maryland, College Park, MD 20742, U.S.A.

AND

B. C. HWANG

David Taylor Research Center, Annapolis, MD 21402, U.S.A.

SUMMARY

Nine finite difference schemes using primitive variables on various grid arrangements were systematically tested on a benchmark problem of two-dimensional incompressible Navier–Stokes flows. The chosen problem is similar to the classical lid-driven cavity flow, but has a known exact solution. Also, it offers the reader an opportunity to thoroughly evaluate accuracies of various conceptual grid arrangements.

Compared to the exact solution, the non-staggered grid scheme with higher-order accuracy was found to yield an accuracy significantly better than others. In terms of 'overall performance', the so-called 4/1 staggered grid scheme proved to be the best. The simplicity of this scheme is the primary benefit. Furthermore, the scheme can be changed into a non-staggered grid if the pressure is replaced by the pressure gradient as a field variable.

Finally, the conventional staggered grid scheme developed by Harlow and Welch also yields relatively high accuracy and demonstrates satisfactory overall performance.

KEY WORDS Navier–Stokes Staggered grid Primitive variable formulation

1. INTRODUCTION

Two types of grid layout can be applied to the primitive variable finite difference method that solves incompressible Navier–Stokes flows—staggered grids[1–4] and non-staggered grids.[5,6] In finite element terminology, staggered grids are similar to mixed-order interpolation functions;[7,8] non-staggered grids resemble same-order interpolation functions.[9]

While a non-staggered grid appears simple and natural, it leads to algebraic systems with singular coefficient matrices that contain too many zero eigenvalues. Consequently, the resulting pressure solution is contaminated with pressure modes and is grossly erroneous. To avoid this problem, researchers began adopting staggered grids in which the nodal velocity components and the pressure are placed in different locations. For flows with small convection, the staggered grid solution also appears to be more accurate than the non-staggered grid result.

The computer programming for staggered grids appears to be more complex than for non-staggered grids because each velocity component requires different indexing. Furthermore, the computation of the convection terms, $\partial(uv)/\partial x$ or $\partial(uv)/\partial y$, may become inaccurate for large Reynolds numbers because the velocity components are staggered. It may be worthwhile to

0271–2091/89/020193–20$10.00

Received 15 December 1987
Revised 27 April 1988

reconsider the use of non-staggered grids, unless the accuracy and convergence rate of numerical schemes using staggered grids prove to be significantly better, or unless the pressure solution is of primary interest.

The objectives of this paper are: (1) to use nine numerical schemes (five staggered grids and four non-staggered grids) to solve a benchmark problem, and to compare the computed and exact solutions; (2) to identify the shortcomings and merits of each scheme; and (3) to recommend a scheme, based on the accuracy and the overall performance.

A well known benchmark problem is the lid-driven cavity flow originated by Burggraf.[10] Some researchers, including the authors of this paper, are unsure of the singularity at the two corners where the moving lid remains in contact with the stationary walls. They have found that specification of the velocity of either unity or zero at the two corners alters the numerical result.

Furthermore, it is difficult to compare the details of nodal values precisely, because the benchmark solution generally is presented in graphic form. Even if a tabulated benchmark solution is available, transferring it into the computer program to compute the global errors would prove laborious.

Therefore we propose a benchmark problem similar to the classical lid-driven cavity flow. The flow velocity at the two corners is now zero; the flow is driven by a specified body force (as described in the next section) in addition to a non-uniform shear. Most importantly, the exact solution to this problem exists and is known.

2. CONTINUUM EQUATIONS GOVERNING THE BENCHMARK PROBLEM

Illustrated in Figure 1, the recirculating cavity flow driven by combined shear and body forces is governed by

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\mathbf{u} \cdot \nabla u = \frac{1}{Re} \nabla^2 u - \frac{\partial p}{\partial x} \tag{2}$$

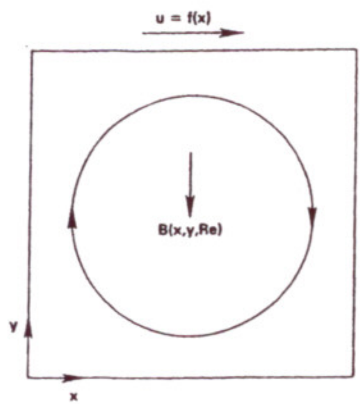

Figure 1. System schematic of the benchmark problem

and

$$\mathbf{u}\cdot\nabla v=\frac{1}{Re}\nabla^2 v-\frac{\partial p}{\partial y}-B(x, y, Re). \tag{3}$$

The boundary conditions for the velocities u and v are of Dirichlet type: zero everywhere except along the top surface where

$$u(x, 1)=16(x^4-2x^3+x^2). \tag{4}$$

Equation (4) also indicates that $u(0, 1)=0$ and $u(1, 1)=0$, which eliminates the ambiguity of specifying the top corner velocities as in the classical lid-driven flow problem.

A body force is present in the y-direction and is prescribed as

$$B(x, y, Re)=-\frac{8}{Re}[24F(x)+2f'(x)g''(y)+f'''(x)g(y)]-64[F_2(x)G_1(y)-g(y)g'(y)F_1(x)], \tag{5}$$

where

$$\begin{aligned}
f(x)&=x^4-2x^3+x^2,\\
g(y)&=y^4-y^2,\\
F(x)&=\int f(x)\,dx=0{\cdot}2x^5-0{\cdot}5x^4+x^3/3,\\
F_1(x)&=f(x)f''(x)-[f'(x)]^2=-4x^6+12x^5-14x^4+8x^3-2x^2,\\
F_2(x)&=\int f(x)f'(x)\,dx=0{\cdot}5[f(x)]^2,\\
G_1(y)&=g(y)g'''(y)-g'(y)g''(y)=-24y^5+8y^3-4y
\end{aligned}$$

$$f'=4x^3-6x^2+2x \quad | \quad f''=12x^2-12x+2 \quad | \quad f'''=24x-12$$
$$g'=4y^3-2y \qquad g''=12y^2-2 \qquad g'''=24y$$
$$\mu=\frac{1}{Re}$$

and the primes on $f(x)$ and $g(y)$ denote the differentiation with respect to x and y respectively.

The exact solution to this combined shear- and body-force-driven cavity flow exists and is known to be

$$u(x, y)=8f(x)g'(y)=8(x^4-2x^3+x^2)(4y^3-2y), \tag{6a}$$

$$v(x, y)=-8f'(x)g(y)=-8(4x^3-6x^2+2x)(y^4-y^2) \tag{6b}$$

and

$$p(x, y, Re)=\frac{8}{Re}[F(x)g'''(y)+f'(x)g'(y)]+64F_2(x)\{g(y)g''(y)-[g'(y)]^2\}. \tag{6c}$$

For convenience, the exact solution of $u(x, y)$, $v(x, y)$ and $\partial p/\partial y$ for $Re=1$ is displayed in Table I corresponding to the physical location in the flow field. The corresponding streamlines are plotted in Figure 2. It is observed that, qualitatively, the clockwise circulation is similar to the classical lid-driven recirculating flow.

An additional inconvenience of the present benchmark problem is the need to include the lengthy source term expression in the v-momentum equation. Readers who intend to solve the benchmark problem may ensure the correctness of the expression in their computer programs by ensuring that $B(0{\cdot}5, 0{\cdot}5, 1)=-3{\cdot}356250$.

3. NUMERICAL SCHEMES EXAMINED

Nine primitive variable schems are used to solve the benchmark problem. Scheme (b) is a modification of scheme (a), and scheme (e) is a modification of scheme (d); most other schemes