ISSN (Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization Volume 4, Special Issue 2, February 2015

5th International Conference in Magna on Emerging Engineering Trends 2015 [ICMEET 2015]

On 27th & 28th February, 2015

Organized by

Department of Mechanical Engineering, Magna College of Engineering, Chennai-600055, India.

Numerical Simulation of Two Dimensional Incompressible Viscous Flows in a Lid-Driven Cavity

Jagadeesh.Y.J[1], Dr. Suresh. N[2]

Assistant Professor, Dept. of Mechanical Engineering, Visvesvaraya Technological University, Belgaum

BMS Institute of Technology & Management, Avalahalli, Bengaluru, Karnataka, India[1]

Associate Professor Dept. of Mechanical Engineering, Visvesvaraya Technological University, Belgaum

BMS Institute of Technology & Management, Avalahalli, Bengaluru, Karnataka, India[2]

ABSTRACT: The present work shows the prediction in Cartesian using the stream function- vorticity formulation for the Lid Driven Cavity. The governing equations are discretised and the variation of flow is analysed by using the commercial CFD software. Contour of Variation of Local skin Friction coefficient along left and right wall of the Cavity is shown .Variation of local skin friction coefficient along top and bottom wall of the cavity is plotted. Flow simulation for Reynold number 100 is taken with a square cavity the top lid is moving and the other side is stationary

KEYWORDS: CFD, stream function, vorticity, lid driven cavity, skin friction coefficient.

I. INTRODUCTION

Due to the development in the CAD/CAE technology, now it is possible to analyse the fluid flow problem, in general any engineering problems. Now a days the CFD approach is becoming popular has come in this way. In the present work of flow analysis inside a square cavity with the top lid moving with low velocity is analyzed and postprocessor velocity contour is obtained with commercial software. Initially the geometry is created and then the fluid domain is discretized by mesh. In the solver and postprocessor its solutions and result are obtained. Basically a theory of CFD- and stream function-voritcity approach is used and new software tool are applied. This technique may be used for designing of any fluid equipment by using the advantage of CAE.

These phenomena are governed by set of partial differential equations which in most cases have no analytical solution. In addition to the governing equations, we also need the boundary and initial conditions, material properties, and geometrical details in order to completely describe the problem.

II.GOVERNING EQUATIONS AND NUMERICAL PROCEDURE

2.1Continuity equation

Physical principle:- Law of conservation of mass.

$$\frac{\partial}{\partial t}\iiint_{\upsilon} \rho \, d\upsilon + \iint_{S} \rho \, \mathrm{V}d\mathrm{S} = 0$$

2. 2 Momentum equation

Physical principle: - Newton’s second law (F=ma)

IJIRSET

ISSN (Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization Volume 4, Special Issue 2, February 2015

5th International Conference in Magna on Emerging Engineering Trends 2015 [ICMEET 2015]

On 27th & 28th February, 2015

Organized by

Department of Mechanical Engineering, Magna College of Engineering, Chennai-600055, India.

$$\frac{\partial(\rho u)}{\partial t}+\nabla.(\rho uV)=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x$$

$$\frac{\partial(\rho v)}{\partial t}+\nabla.(\rho vV)=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y$$

$$\frac{\partial(\rho w)}{\partial t}+\nabla.(\rho wV)=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z$$

Equations are the momentum equations in x, y and z direction respectively in conservation form.

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x+}\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y+}\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y$$

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z+}\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z$$

2.3 Discretisation methods

The discretisation methods i.e. the numerical methods for solving PDEs include the finite difference methods (FDM), and the finite volume method (FVM) is used.

2.4 Stream Function Formulation

Flow prediction in Cartesian using the stream function- vorticity formulation

For two dimensional steady incompressible flow in a rectangular geometry, the governing equations for stream function and vortices are the following:

(i) Definition of vorticity:

$$\frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}=-\omega$$

Where ψ is the stream function and ω is the vorticity

(ii) Convective – diffusive transport equation for vorticity

$$\left(u\frac{\partial\omega}{\partial x}+v\frac{\partial\omega}{\partial y}\right)=\nu\left\{\frac{\partial^2\omega}{\partial x^2}+\frac{\partial^2\omega}{\partial y^2}\right\}$$

Where the velocity components u and v are given by

$$u=\frac{\partial\psi}{\partial y} \qquad v=-\frac{\partial\psi}{\partial x}$$

The boundary conditions for the lid-driven cavity problem are

u = 0, v = 0 at x = 0; u = 0, v = 0 at y = 0;

u = 0, v = 0 at x = L; u = U_o , v = 0 at y = H;

Where U_o is the velocity of the lid.

$$\frac{\psi_{i+1,j}+\psi_{i-1,j}-2\psi_{i,j}}{\Delta x^2}+\frac{\psi_{i,j+1}+\psi_{i,j-1}-2\psi_{i,j}}{\Delta y^2}=-\omega_{i,j}$$

ISSN (Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization *Volume 4, Special Issue 2, February 2015*

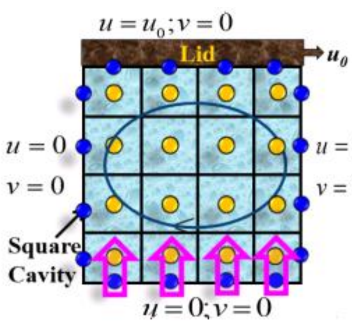
5th International Conference in Magna on Emerging Engineering Trends 2015 [ICMEET 2015]

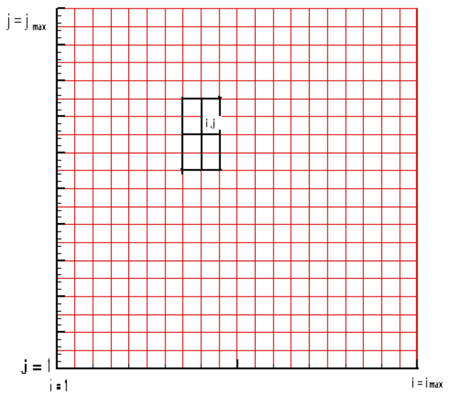
On 27th & 28th February, 2015

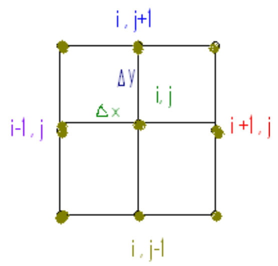
Organized by

Department of Mechanical Engineering, Magna College of Engineering, Chennai-600055, India.

$$u_{i,j}\left[\frac{\omega_{i+1,j}-\omega_{i-1,j}}{2\Delta x}\right]+v_{i,j}\left[\frac{\omega_{i,j+1}-\omega_{i,j-1}}{2\Delta y}\right]=$$
$$\nu\left\{\frac{\omega_{i+1,j}+\omega_{i-1,j}-2\omega_{i,j}}{\Delta x^2}+\frac{\omega_{i,j+1}+\omega_{i,j-1}-2\omega_{i,j}}{\Delta y^2}\right\}$$







III. RESULT

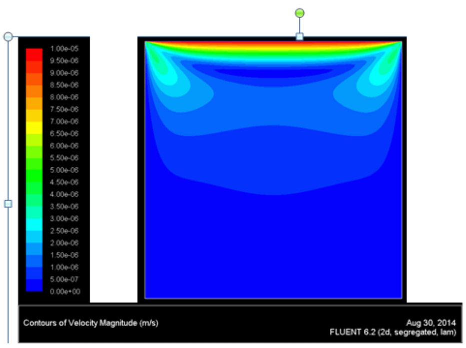


Fig.1. Velocity Contour-1

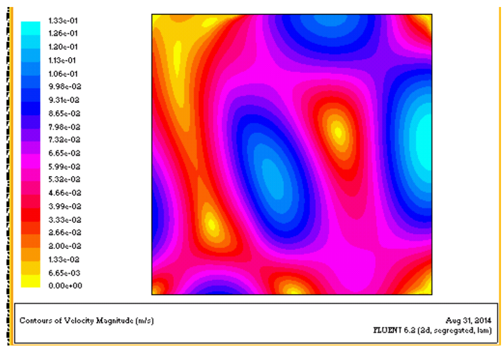


Fig.2. Velocity contour-2

IJIRSET

ISSN (Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization Volume 4, Special Issue 2, February 2015

5th International Conference in Magna on Emerging Engineering Trends 2015 [ICMEET 2015]

On 27th & 28th February, 2015

Organized by

Department of Mechanical Engineering, Magna College of Engineering, Chennai-600055, India.

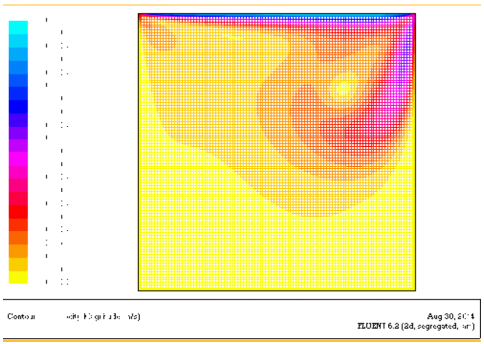


Fig.3 .Velocity contour-3

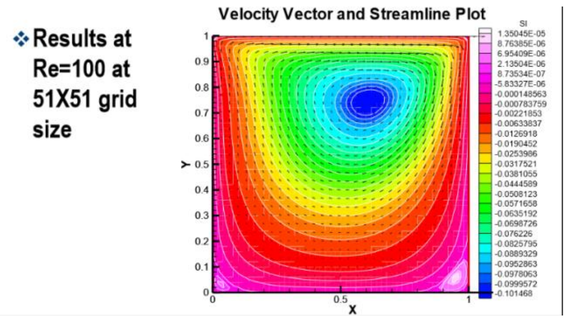


Fig.4. Velocity vector and stream line plot

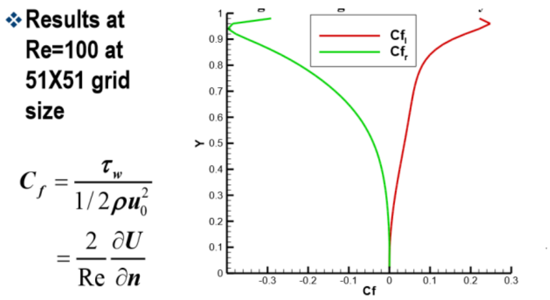


Fig.5. Variation of Local skin Friction coefficient along left and right wall of the Cavity

IV. CONCLUSION

In this present work the prediction in Cartesian using the stream function- vorticity formulation for the Lid Driven Cavity has been shown. On either side of the wall of the cavity the Local skin Friction coefficient varies in the same manner and found maximum between 0.9 and 1 on vertical and minimum at the vertically bottom.

Notations:

P-pressure

μ - viscosity

τ - viscous stress

ISSN (Online) : 2319 - 8753
ISSN (Print) : 2347 - 6710

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization Volume 4, Special Issue 2, February 2015

5th International Conference in Magna on Emerging Engineering Trends 2015 [ICMEET 2015]

On 27th & 28th February, 2015

Organized by

Department of Mechanical Engineering, Magna College of Engineering, Chennai-600055, India.

U,V,W-free stream mean velocity
u',v',w'- instantaneous velocity
ρ – density
f- body forces
ψ - stream function and
ω -vorticity

REFERENCES

1. Computational Fluid Flow and Heat Transfer by K.Muralidhar and T Sundararajan, IIT Kanpur series of advanced texts
2. Schlichting H. (1979), *Boundary Layer Theory*, Seventh Edition, McGraw Hill
3. Emmons H.W. (1951) *The Laminar-Turbulent Transition in a Boundary Layer – Part I*, Journal of Aerospace Science, vol. 18, no.7, pp. 490-498
4. Suzen Y.B., Huang P.G. (2005) *Numerical simulation of unsteady wake/blade interactions in low-pressure turbine flows using an intermittency transport equation*, Journal of Turbomachinery, ASME vol.127, pp. 431- 444
5. Majumdar S., Rodi W., and Zhu J.(1992), *Three dimensional finite volume method for incompressible flows with complex boundaries*, Journal of Fluids Engineering., ASME, pp. 496-503.
6. Majumdar S., Rajani B.N., Kulkarni D., Mohan S., (2003), *RANS computation of low speed turbulent flow in complex configuration, Proceedings, Symposium on State of the Art and Future Trends of CFD at NAL,* NAL SP 03 01
7. Patankar S. V. and Spalding D. B. (1972), *A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows*, International Journal of Heat and Mass Transfer, Vol. 15, pp. 1787-1806.
8. Patankar S. V. (1980), *Numerical Heat Transfer and Fluid flow*, McGraw-Hill, New York, N.Y.
9. Peric M. (1985), *A finite volume method for the prediction of three-dimensional fluid flow in complex ducts, PhD Thesis*, Imperial College, London University.
10. Kulkarni D.S. Rajani B.N. Majumdar S. (2001), *Studies on the temporal and spatial discretization schemes used in a Pressure-Based RANS algorithm for incompressible Flow,* Project Document CF 0108, NAL Bangalore.