International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 (2015)
© Research India Publications; http://www.ripublication.com/ijmer.htm

Numerical simulation of various flows over a square cavity

N.Senthil Kumar[1], M.Revathi[2], P.Raju[3]

Assistant Professor

Abstract

Numerical simulation of Euler equations and Navier-Stoke equation is carried out in the present work where for each simulation we took one or more example problem say Quasi-One-Dimensional nozzle flow for Euler equations and viscous incompressible flow over a square cavity, square cylinder and tall building for Navier-Stoke equation. The entire problems are solved here numerically and investigated. Here different computational methods are applied for each problem and shown the results. Finite difference method and Finite Volume method both are studied and applied here for solving the Euler and Navier-Stoke Equations respectively for the respective problem. All the problems are solved numerically in the physical plane itself without transferring it into the computational plane using structured grid. In the present investigation, we got reasonably good agreement between the numerical and the analytical result.

Keywords: Numerical simulation, Dimensional nozzle flow, CFD.

1. INTRODUCTION

Numerical simulation using computers or computational simulation has increasingly become a very important approach for solving complex practical problems in engineering and science. Numerical simulation translates important aspects of a physical problem into a discrete form of mathematical description, recreates and solves the problem on a computer, and reveals phenomena virtually according to the requirements of the analysts.

Rather than adopting the traditional theoretical practice of constructing layers of assumptions and approximation, this modern numerical approach attacks the original problem in all its detail without making too many assumptions, with the help of the increasing computer power. Numerical simulation provides an alternative tool of scientific investigation, instead of carrying out expensive, time-consuming or even dangerous experiments in laboratories or on site. Numerical simulation with computers plays a valuable role in providing a validation for theories offers insights to the experimental results and assists in the interpretation or even the discovery of new phenomena. It acts also as a bridge between the experimental models and the theoretical predictions.

The overall goal of the field of numerical simulation is the design and analysis of techniques to give approximate but accurate solutions to hard problems. The Netlib repository contains various collections of software routines for numerical problems, mostly in FORTRAN and C.

Commercial products implementing many different numerical algorithms include the IMSL and NAG libraries; a free alternative is the GNU Scientific Library. There are several popular numerical computing applications such as MATLAB, S-PLUS, Lab VIEW, and IDL as well as free and open source alternatives such as Free Mat, Scilab, GNU Octave (similar to Matlab), IT++ (a C++ library), R (similar to S-PLUS) and certain variants of Python.

Performance varies widely: while vector and matrix operations are usually fast, scalar loops may vary in speed by more than an order of magnitude. Many computer algebra systems such as Mathematic also benefit from the availability of arbitrary precision arithmetic which can provide more accurate results.

2. COMPUTATIONAL METHODS

The finite difference scheme for the numerical simulation of quasi one dimensional nozzle flow problem consists of following methods. Here, both explicit and implicit methods are used for the same problem for understanding the idea and usage.

1) Mac-Cormack's Predictor Corrector Method

Mac Cormack's technique is an explicit finite difference technique, which is second order accurate in both space and time. The time marching solution using Mac Cormack's technique for the Euler equations [1] proceeds as follows:

The value of the solution vector U at time n+1 is obtained as:

$$U^{n+1}{}_{i} = U^{n}{}_{i} + \left(\frac{\partial U}{\partial t}\right)^{n}_{i} \Delta t$$

Where $\left(\frac{\partial U}{\partial t}\right)_{av}$ is a representative mean value of $\left(\frac{\partial u}{\partial t}\right)$ between times n and n+1.This average time derivative is obtained from a predictor–corrector philosophy as Follows:

Predictor Step: In the governing equation the spatial derivatives are replaced by forward differences to give:

$$\left(\frac{\partial U}{\partial t}\right)^{n}_{i} = -\left(\frac{F^{n}{}_{i+1} - F^{n}{}_{i}}{\Delta x} - H^{n}{}_{i}\right)$$

In this equation all the values at time n are known i.e. the RHS is known.
Now, a predicted value of U is obtained from the first two terms of a Taylor series as:

$$U^{n+1}{}_{i} = U^{n}{}_{i} + \left(\frac{\partial U}{\partial t}\right)^{n}_{i} \Delta t$$

In eq. (3.3) $U^{n}{}_{i}$ is known and $\left(\frac{\partial U}{\partial t}\right)^{n}_{i}$ is known from eq. (3.2). Hence $U^{n+1}{}_{i}$ can be readily obtained. However, $U^{n+1}{}_{i}$ is only a predicted value and is only first order accurate since eq. (3.3) contains only first two terms in the Taylor series. Using $U^{n+1}{}_{i}, F^{n+1}{}_{i} = F(\mathrm{U}^{n+1}{}_{i})$ and $H^{n+1}{}_{i} = H(\mathrm{U}^{n+1}{}_{i})$ can be found out.

Corrector Step: In the corrector steps a predicted value of the time derivative at time n+1 is obtained by substituting the predicted value of U in the equation, replacing the spatial derivatives by rearward differences.

$$\left(\frac{\partial U}{\partial t}\right)^{n}_{i} = -\left(\frac{F^{n}{}_{i+1} - F^{n}{}_{i}}{\Delta x} - H^{n}{}_{i}\right) \quad \text{... (3.17)}$$

2

The average value of the time derivative of U is now obtained from the arithmetic mean of $\lambda_3 = u - a$ and $\left(\frac{\partial U}{\partial t}\right)_i^{n+1}$ obtained from eq. (3.2) and eq. (3.4) respectively.

$$\left(\frac{\partial U}{\partial t}\right)_{av} = \frac{1}{2}\left[\left|\frac{\partial U}{\partial t}\right|_i^{n+1} + \left|\frac{\partial U}{\partial t}\right|_i^{n}\right] \quad \text{... (3.18)}$$

Thus the final 'corrected' value of U can be obtained from eq. (3.1), repeated below:

$$U^{n+1}{}_i = U^n{}_i + \left(\frac{\partial U}{\partial t}\right)_i^n \Delta t$$

In Mac Cormack's technique the use of forward differences on the predictor step and rearward differences on the corrector step is not sacrosanct; the same order of accuracy is obtained by using rearward differences on the predictor and forward differences on the corrector. Indeed, a time marching solution can be carried out by alternating between these two sequences at every other time step.

2) Crank Nicolson method with Beam and Warming linearization technique

Quasi Linear formulation:

As F=F (U) by the chain rule of differentiation: $\frac{\partial F}{\partial x} = \frac{\partial F}{\partial U}\frac{\partial U}{\partial x}$

Denoting $\frac{\partial F}{\partial U} = A$, A is called the Jacobian of the flux vector U. It is obtained by differentiating each of the elements of flux vector F one by one by each of the elements of vector U.

Equation shown above is now written as:

$$\frac{\partial U}{\partial t} + A\frac{\partial U}{\partial x} - H = 0$$

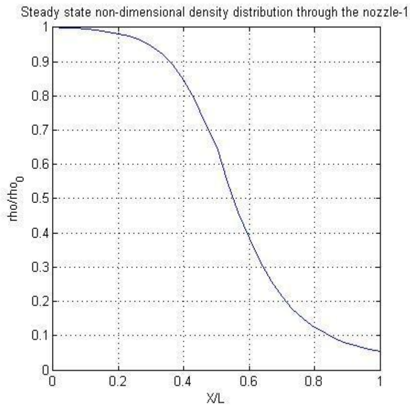
Case 1: Convergent-Divergent nozzle (Fig : Nozzle -1 shown above) domain with Purely subsonic flow

Nozzle cross section area A(x) =1+ (2.2*(x-1.5)*(x-1.5)); $0 \le x \le 3$ ft.

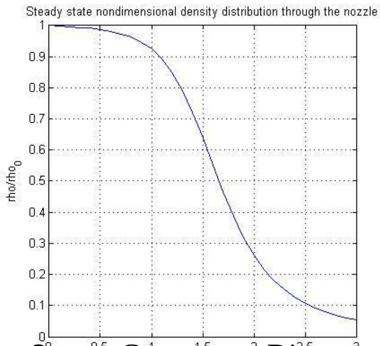
No. of grid points: 31 i.e. step size $\Delta x = 0.1$

Courant No. = 0.5

a) using Conservation form of governing equation (after 1400 time steps)

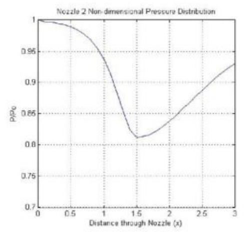


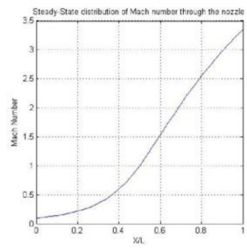
b) using non conservation form of governing equation (after 1400 time steps)



Case 2 : Convergent-Divergent nozzle (fig: Nozzle-2 shown above) domain with subsonic-supersonic isentropic flow

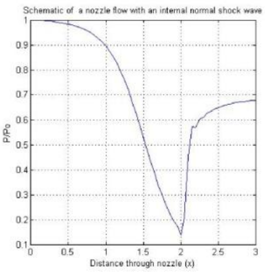
Nozzle cross section area

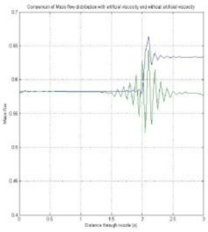




a) Using Conservation form of governing equation (after 5000 time steps)

Case 3a: Convergent-Divergent nozzle (fig: Nozzle-1 shown above) domain with subsonic-supersonic flow with normal shock inside (without Artificial viscosity)





The numerical results were obtained using Navier-Stokes equation based on Consistent Flux Reconstruction concept as described in chapter 4 for uniformly distributed grid. The convergence criterion of 0.0001 was applied.

Figure 2a). Pressure contours and Streamlines pattern for Reynolds number 75

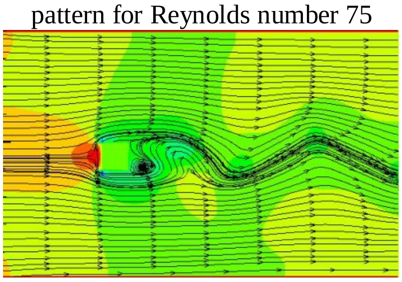


Figure 2b). Pressure contours and Streamlines pattern for Reynolds number 150

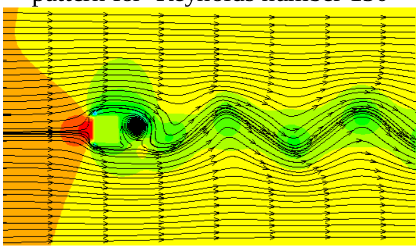
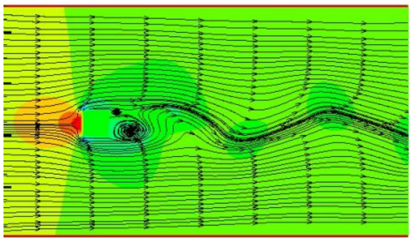


Figure 2c). Pressure contours and Streamlines pattern for Reynolds number 500



Validation of the results for Re 75,150 &150 and blockage ratio of 11%

Reynolds Number	**Grid Size**	**Drag Co efficients**	**Lift Co efficients**
75	250 x 90	1.95	0.195
150	250 x 90	1.85	0.340
500	250 x 90	2.01	0.686

Numerical calculations of laminar flow past a square cylinder for various Reynolds number (say 75,150,500) are done by solving the Navier-Stoke equation and the results are obtained. The drag and lift coefficients are calculated for all the flow cases and tabulated above. If compared with the results of some the research papers for the appropriate blockage ratio, Reynolds number, grid size and where the square cylinder was located, some results get matched.

References	**Reynolds number**	**Grid Size**	**Drag Co efficie nt**
A.K.Saha, G.Biswas,	75	178 X 80	1.76
S. Turki, H. Abbassi,	150	220 X 77	1.50
R.Franke, W.Rodi and	500	208 X 72	1.94

3. CONCLUSION

Understanding of the implementation of various CFD techniques to various problems and got the numerical results matching with analytical results. In Quasi one dimensional nozzle flow problem, the Euler equations will be solved numerically using finite difference scheme.

For understanding of the implementation of finite volume method (CFR Scheme) to develop an incompressible Navier-Stokes solver for two dimensional geometries using structured grid, flow over a square cavity problem was studied and matched the numerical result with journal paper results.

REFERENCES

1. ABDALLAH S. 1987 Numerical solution for the pressure Poisson equation with Neumann boundary condition using a non staggered grid, I. Journal of Computational Physics; 70:182– 192.
2. ABDALLAH S.1987 Numerical solutions for the incompressible Navier–Stokes equation in primitive variables using a non-staggered grid, Journal of Computational Physics;70:193– 202.
3. A.B.HARICHANDRAN et al, 2010. CFR: A finite volume approach for computing incompressible viscous flow, Journal of Applied fluid mechanics.
4. AHMAD SOHANKAR, C. NORBERG, AND L. DAVIDSON 1999, Simulation of three-dimensional flow around a square cylinder at moderate Reynolds number in Physics of fluids 11,288
5. AKHILESH K.SAHU, R.P.CHHABRA, V.ESWARAN. 2009 Two-dimensional unsteady laminar flow of a power law fluid across a square cylinder. Journal of Non-Newtonian Fluid Mechanics 160,
6. A. ROY et al., 2006 A finite volume method for viscous incompressible flows using a consistent flux reconstruction scheme, International journal for numerical methods in fluids, Volume 52, Issue 3, pages 297–319.
7. B.GALLETTI, C.H.BRUNEAU, L.ZANNETTI, A.IOLLO, 2004 Low-order modeling of laminar flow regimes past a confined square cylinder, Journal of Fluid Mech, Vol.503, pages 161-170.
8. CHENG M. TAN S. H. HUNG K.C, 2005, Linear shear flow over a square cylinder at low Reynolds number in Physics of fluids, Volume 17. Pages 078103.
9. DENG GB, PIQUET J, QUEUTEY P, VISONNEAU M. 1994 Incompressible flow calculations with a consistent physical interpolation finite volume approach. Computers and Fluids; 23(8):1029–1047.
10. FRANKE .R. RODI .W. SCHNUNG.B. 1990, Numerical calculation of laminar vortex shedding Flow past cylinders, in Journal of Wind Engineering and Industrial Aerodynamics, Volume 35. Pages 237-
11. GERA.B.PAVAN K. SHARMA ,SINGH.R.K 2010 ,CFD analysis of 2D unsteady flow around a square cylinder in International Journal of Applied Engineering Research, Dindigul Volume1,
12. GRESHO PM. 1991 Some current CFD issues relevant to the incompressible Navier–Stokes equations. Computer Methods in Applied Mechanics and Engineering; 87:201–252.
13. HARLOW FH, WELCH JE. 1965, Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. Physics of Fluids; 8(12):2182–2189.