6. Comparative study of simulation of incompressible two-dimensional laminar Duct flow in ANSYS FLUENT™ and MATLAB™

Research Article

Makrand A. Khanwale
Second Year Bachelor of Technology
Department of Oils, Oleochemicals & Surfactants Technology

Abstract

A finite difference method is described for computation of dynamics of pipe flow in two dimensions. The In-house code solves the pressure –velocity coupled momentum and the continuity equations using Semi Implicit Pressure Linked (SIMPLE) algorithm and post-processes the data. The In-house code has been written in MATLAB™. A run for the same geometry in ANSYS FLUENT™ is also put on for two-dimensional laminar flow and a systematic comparative study is presented between the two codes. The results are tabulated with their percentage accuracy with respect to the analytical solution.

1. Introduction

Traditionally the approach used for design of process equipment involved heuristic knowledge and experience. With the onset Computational Fluid Dynamics the studies of hydrodynamics of process equipment became possible. Knowledge of Fluid dynamics of the equipment enhances the understanding of the physics of the Transfer processes involved, thereby aiding in the design. Pipe flow one of the most commonly flows observed in industry. This paper deals with solutions of the governing continuity and momentum equations for the pipe geometry in two dimensions.

There are many commercially available Solvers which solve the governing partial differential equations of Fluid Dynamics like ANSYS FLUENT™. This paper presents a comparative study of In-house code developed with the commercial software FLUENT™ and the analytical solutions available for fully developed pipe flow in Laminar Regions.

The transfer of fluid through a pipe is governed by three equations vis-à-vis 'The continuity equation' and 'the momentum equations in X direction' and 'the momentum equation in Y direction'. We will be describing the code to solve these three equations in a uniform rectangular grid. Results of simulations of the In-house code for different number of grid points are also described and compared with the analytical solution. Finally, a comparison between the solution of the exactly same geometry and meshing in ANSYS FLUENT™ are compared with the In-house code.

The flexibility of the In-house code to be modified for Direct Numerical Simulations (DNS) is also discussed in Conclusions as scope for future work.

2. Numerical Analysis

2.1 Governing Equations

The governing equations for an incompressible fluid flow in a rectangular planar duct flow are namely 'the continuity equation' and 'the momentum equations.' The continuity equation arises from mass balance through a control volume. The momentum equations are a result of the Newton's 2nd law applied on a control volume. The equations aforementioned are given below:

$$\frac{\partial \rho}{\partial t} + \nabla.\mathrm{v} = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots.. (1)$$

(The continuity equation)

$$\frac{\partial \mathrm{v}}{\partial t} + (\mathrm{v}.\nabla).\mathrm{v} = \frac{1}{\rho}\{-\nabla.\mathrm{p} + \mu\nabla^2\mathrm{v} + \mathrm{g}\} ..(2)$$

(The general momentum transport equation)

Where, 'ρ' is the density of the fluid used, v is the velocity field and 'p' is the pressure.

Now for incompressible fluid flow through a 2D rectangular duct, the above mentioned equations reduce to the following equations:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial x} = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

(The continuity equation)

$$\frac{\partial \mathrm{u}}{\partial t} + u\frac{\partial u}{\partial x} + u\frac{\partial v}{\partial x} = \frac{1}{\rho}\left\{-\frac{\partial \mathrm{p}}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right]\right\} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots....(4)$$

(The momentum equation in X-direction)

$$\frac{\partial \mathrm{v}}{\partial t} + v\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial x} = \frac{1}{\rho}\left\{-\frac{\partial \mathrm{p}}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right] + g\right\} \quad \ldots\ldots\ldots\ldots\ldots\ldots .(5)$$

(The momentum equation in Y-direction)

where, 'u' is the velocity in X-direction, 'v' is the velocity in Y-direction, all the other symbols suggest the same as above. Solving these governing equations numerically is the objective of the In-house code.

A finite differencing method is used to solve these partial-differential equations and a central differencing scheme is used for discretisation of convective- diffusive terms and Simple algorithm is followed for solving pressure linked momentum equations. [1]

2.2. Grid Generation and usage

Solving the governing equations involves meshing the geometry (dividing it in smaller areas or volumes) and approximating these partial differentials by the assigning the variables at each grid point and then establishing relationships between these variables by Taylor's theorem or spectral methods. So there a matrix generation, this can be solved at each time step. The algorithm of solution is SIMPLE which requires generation of staggered grid. A staggered grid is used because if pressure and velocities are defined at the same locations a highly non-uniform pressure field can behave like a uniform field in the discretised momentum equations. The In-house code generates a grid of a rectangular shape of 0.01m x 0.6 m. Figure 1 shows a 20 x 20 grid generated by the In- house code.

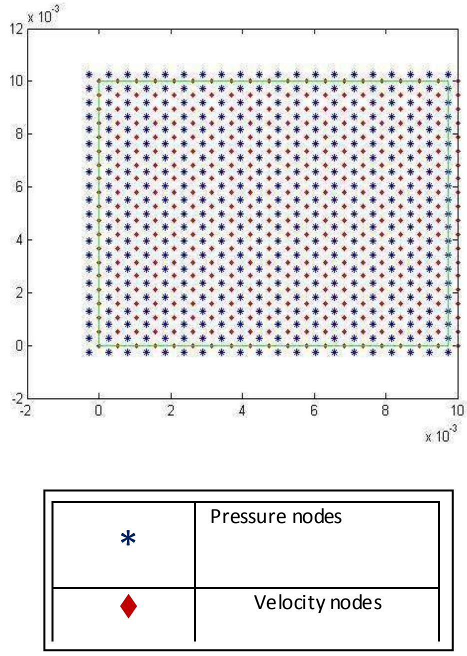

Figure 1: Staggered Grid

2.3. Discretisation

Once the grid is generated then the solvers can be used to find out parameters on respective grid points of pressure and velocity. We now develop all the discretisation involved in the solvers. The discretised parts of the Navier-Stokes equations are (without pressure term, which will be added in the pressure correction equation):

$$\frac{\partial u}{\partial x} = \frac{u_{i+1,j} - u_{i-1,j}}{2hx} \qquad \frac{\partial u}{\partial y} = \frac{u_{i,j+1} - u_{i,j-1}}{2hy} \qquad \ldots\ldots\ldots\ldots\ldots(6)$$

$$\frac{\partial v}{\partial x} = \frac{v_{i+1,j} - v_{i-1,j}}{2hx} \qquad \frac{\partial v}{\partial y} = \frac{v_{i,j+1} - v_{i,j-1}}{2hy} \qquad \ldots\ldots\ldots\ldots\ldots(7)$$

$\frac{\partial^2 u}{\partial x^2} = \frac{u_{i+1,j} - 2u_{i,j} + u_{i-1,j}}{hx^2}$,

$\frac{\partial^2 v}{\partial x^2} = \frac{v_{i+1,j} - 2v_{i,j} + v_{i-1,j}}{hx^2}$

……………...(8)

$\frac{\partial^2 u}{\partial y^2} = \frac{u_{i,j+1} - 2u_{i,j} + u_{i,j-1}}{hy^2}$

$\frac{\partial^2 v}{\partial x^2} = \frac{v_{i+1,j} - 2v_{i,j} + v_{i-1,j}}{hy^2}$

……………...(9)

Now we rewrite the transport equations in discretised form:

$$u_{ij,t} + u_{i,j}\frac{u_{i+1,j} - u_{i-1,j}}{2hx} + v_{i,j}\frac{u_{i,j+1} - u_{i,j-1}}{2hy} = \mu\left(\frac{u_{i+1,j} - 2u_{i,j} + u_{i-1,j}}{hx^2} + \frac{u_{i,j+1} - 2u_{i,j} + u_{i,j-1}}{hy^2}\right) \quad \text{…………………… (10)}$$

$$v_{ij,t} + v_{i,j}\frac{v_{i+1,j} - v_{i-1,j}}{2hx} + v_{i,j}\frac{v_{i,j+1} - v_{i,j-1}}{2hy} = \mu\left(\frac{v_{i+1,j} - 2v_{i,j} + v_{i-1,j}}{hx^2} + \frac{v_{i,j+1} - 2v_{i,j} + v_{i,j-1}}{hy^2}\right) - g \quad \text{……………........... (11)}$$

We can now write (10), (11) as

$$u_{ij,t} = f_u(u, v) \quad \text{……………… (13)}$$

$$v_{ij,t} = f_v(u, v) \quad \text{………………… (14)}$$

We will discretise these equations in time with the use of the Adam Bashforth method for time discretisation. We will first calculate a guess for the velocity and use the pressure from the old time level. After that we will update the pressure with the use of equation (6) and calculate the velocities for the new time level.

$$u^* = u^n + \frac{3}{2}f_u(u^n, v^n) - \frac{1}{2}f_u(u^{n-1}, v^{n-1})$$

$$v^* = v^n + \frac{3}{2}f_v(u^n, v^n) - \frac{1}{2}f_v(u^{n-1}, v^{n-1})$$

Or with Euler forward

$$u^* = u^n + f_u(u^n, v^n)$$

$$v^* = v^n + f_v(u^n, v^n)$$

This is the overall discretisation scheme followed, now we use velocities obtained in this level to solve the pressure correction equation to get the pressure. The code also renders a quiver plot of the Velocity Field and plots of velocity in X direction and velocity in Y-direction and the pressure field.

3. Results and Discussions

3.1. Quantitative comparison of velocities the fully developed Flow

We have run the In-house code for different set of grid points and compared the solution with the fluent solution and the analytical solution which is given in table 1. The maximum velocity in the Y-direction is obtained for different grid sizes using the solutions of in-house code. Theses velocities are tabulated with the analytical solution for the fully developed region of flow.

Table 1:- comparative study of error using different grid sizes

Mesh	Obtained Max velocity u (using In-house code)	Analytical Max velocity u	$error = \frac{u_{analytical} - u_{ccalculated}}{u_{analytical}} x100$
20 x 500	0.001136	0.00125	9.066486467
40 x 500	0.001144	0.00125	8.405576532
80 x 500	0.001146	0.00125	8.310331857

We have kept the overall Reynolds's number 500 and calculated the velocity profile using initial velocity of 8.33333x10^{-4} m/s as per standard velocity correlation of fully developed laminar flow. The standard velocity correlation is given below for

$-\frac{H}{2} \leq y \leq \frac{H}{2}$

$$v = \frac{3}{2} v_{in} \left(1 - \left(\frac{y}{\frac{H}{2}}\right)^2\right)$$

In Figure 2, we have plotted the solutions of In-house code for different grid sizes vis-à-vis 20 x 500, 40 x 500, 80 x 500 respectively. The solution of the same geometry that is length of 0.6 m and height of 0.01 m, using Gambit meshing and Ansys Fluent™ is also plotted on the same graph. The analytical solution is also plotted on the same plot.

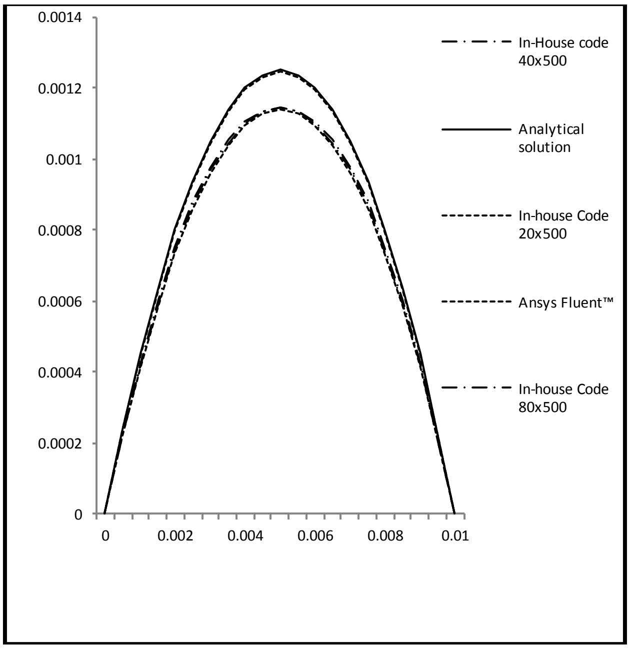

Figure 2: comparative plot for solutions of different grid sizes and analytical solution and Ansys solution

Figure two shows that the In-house code when run for different increasing grid sizes approaches the analytical solution. But after 80 x 500 grid size the solution attains grid independence. Percentage standard deviation of the solution for different grid size is given below:

We see that the error goes down as we increase the grid size from 20 x 500 to 40 x 500, whereas at grid size of 80 x 500 the error decrement is almost negligible that's why the plots in Figure 2 are very close to each other.

3.2. Contour Plots

The In- house code is capable of generating velocity contours and the quiver plots at every 100th iteration. It generates a plot of Velocity in X- direction, Velocity in Y-direction, pressure and mass residual. Figure 3 shows the In-house code generated render of fully developed flow at 65 seconds.

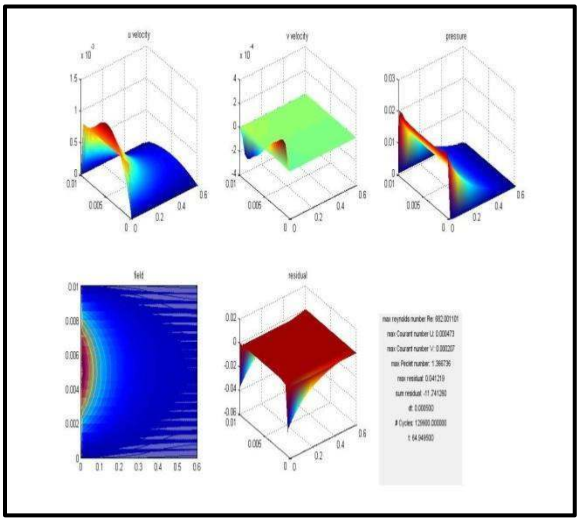

Figure 3: The above figure shows a render of fully developed flow at 65 seconds for 0.01 m x 0.6 m with a grid of 80 x 500 grid size.

4. Conclusion

We see that the solutions obtained from the In-house code in laminar fully developed regions. The velocity profile in Y-direction is indeed parabolic in nature as expected.

The comparison clearly indicates that the code is rendering sufficiently close magnitudes and the trends of flow parameter variation match with the commercial code vis-à-vis ANSYS FLUENT™. The variation in values are due to the change of discretisation scheme as our code implements central differencing scheme and ANSYS FLUENT™ implements 1st order upwind scheme in space. This code can be modified for different geometry and turbulence as well. The velocity profiles generated by the code are matching completely with ANSYS FLUENT™'s solution for incompressible unsteady laminar pipe flow in two dimensions.

This comparison throws into light that this In-house code can be modified with higher order discretisation schemes and extremely fine grid size i.e. upto Kolmogorov length scales and three dimensions to be used as a DNS(direct numerical simulations) code for analysis of Duct flow for high Reynolds numbers.

The pipe flow is a classic example of wall generated turbulence. DNS analysis tracks the turbulence till its lowest possible scales. The near wall turbulence is crucial for transport processes, understanding its structure and associated transport mechanisms is of great importance for scientific and engineering viewpoints. This In-house code can form a basis of this kind of direct numerical simulations.

References

1. Versteeg, H.K., Malalasekara, W., An Introduction to computational fluid

dynamics, The finite volume method, Second Edition, Pearson 2007.

2. Patankar S.V., Numerical Heat Transfer and Fluid Flow, 1st edition, Taylor and Francis, 1980.
3. Chung T.J., Computational Fluid Dynamics, 2nd edition, Cambridge University Press, 2010.
4. Jiyuan Tu., Yeoh G.H., Liu C., Computational Fluid Dynamics A Practical Approach, 1st edition, Butterworth-Heinemann, 2008.
5. Date, Anil., 2005, introduction to Computational Fluid Dynamics, 1st edition, Cambridge university Press.