Numerical Study of Steady One Dimensional Incompressible Flow through a Nozzle Using SIMPLE Algorithm

Arti Kaushik
Maharaja Agarsain Institute of Technology

Anil Kumar
Maharaja Agarsain Institute of Technology

International Journal of Scientific Research in Mathematical and Statistical Sciences
Research Paper Volume-3, Issue-5 E-ISSN: 2348-4519

Numerical Study of Steady One Dimensional Incompressible Flow through a Nozzle Using SIMPLE Algorithm

Arti Kaushik[1*] and Anil Kumar[2]

[1,2] Maharaja Agrasen Institute of Technology, Delhi, India

Available online at www.isroset.org

Received: 16/Sep/2016 Revised: 22/Sep/2016 Accepted: 15/Oct/2016 Published: 30/Oct/2016

***Abstract-* In the present study the problem of a steady 1-D incompressible flow through a planar two dimensional converging nozzle is investigated. Numerical solutions of the set of one dimensional governing equation are obtained by using SIMPLE algorithm. The numerical computations for velocity and pressure have been conducted using a staggered grid system. Iterative solution of the discretised momentum equation and the pressure correction equation is done to obtain the velocity and pressure field. The numerical solutions of the velocity and pressure obtained in the present study have been ensured to be stable using under relaxation factors. The accuracy of the computed solution has been checked against the well known Bernoulli equation. The significant findings from this investigation have been given under conclusion.**

***Keywords*: Navier-Stokes equations, SIMPLE algorithm, Staggered grid, Nozzle flow, Incompressible flow.**

I. Introduction

Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation. The technique is very powerful and has a wide range of industrial and non industrial application areas. It is not only economical but also time saving tool to solve flow problems without performing actual experiments. The development of high speed computers and advanced software packages has made the simulation of the flow system more convenient

A nozzle is a tube of varying cross-sectional area which is usually axisymmetric and is used to increase the speed of an outflow and to control its direction and shape. Nozzle flow always generates forces associated to the change in flow momentum. Nozzles are used to accelerate the fluid in subsonic gas streams and in liquid jets. Substantial research has been in progress to study the properties of nozzles. ([1],[2],[3],[4],[5], [6], [7], [8], [9]).

In this paper, the well-known SIMPLE algorithm of Patankar and Spalding is used to deal with a converging nozzle problem. SIMPLE stands for Semi-Implicit Method for Pressure-Linked Equations. This method is very reliable and can deal with both compressible and incompressible flow. The method has been used in [10], [11] [12] and[13] among others. In section II of this paper, mathematical formulation of the problem is done. In section III, SIMPLE algorithm is discussed. In section IV and V, numerical calculations and discussion of the result are given. The conclusions of the paper are given in section VI.

II. Mathematical Formulation

We consider a steady and frictionless flow through a planar two-dimensional nozzle shown in Figure 1. The density of the fluid is constant. The stagnation pressure is given at the inlet and the static pressure is specified at the exit. Using the backward-staggered grid using the SIMPLE algorithm we write down the discretised momentum and pressure correction equations and solve for the unknown pressures at pressure nodes and velocities at velocity nodes. We will also check the continuity of the computed velocity field and evaluate the error in the computed pressure and velocity fields by comparing with the exact solution.

Corresponding Author: Arti Kaushik, arti.kaushik@gmail.com
Maharaja Agrasen Institute of Technology, Delhi, India

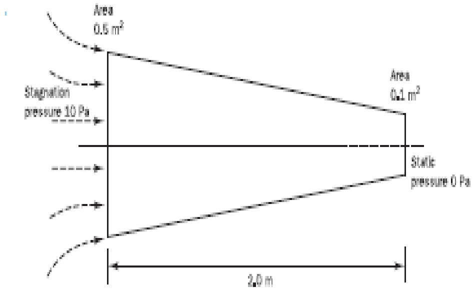

Fig 1: Geometry of planar 2D nozzle

The governing equations for steady, one-dimensional, incompressible, frictionless equations through the planar nozzle are as follows:

$$\frac{d}{dx}\left(\rho A u\right)=0 \qquad (1)$$

(Mass conservation)

$$\rho u A\frac{du}{dx}=-A\frac{dp}{dx} \qquad (2)$$

(Momentum conservation)

Discretised u-momentum equation

The discretised form of momentum equation is

$$\left(\rho uA\right)_e u_e-\left(\rho uA\right)_w u_w=\frac{\Delta p}{\Delta x}\,\Delta V \qquad (3)$$

where $\Delta p = p_w - p_e$

The discretised momentum equation for this one dimensional problem can be written in standard notation as

$$a_P u_P^* = a_W u_W^* + a_E u_E^* + S_u \qquad (4)$$

Using the upwind differencing scheme the coefficients may be obtained from

$$a_W = D_w + \max(F_w, 0)$$
$$a_E = D_e + \max(0, -F_e)$$
$$a_P = a_W + a_E + (F_e - F_w)$$

Since the flow is frictionless, there is no viscous diffusion term in the governing equation, and hence $D_w = D_e = 0$. F_w and F_e are mass flow rates through the west and east face of the *u*-control volume. We compute the face velocities needed for F_w and F_e from averages of velocity values at nodes straddling the face and use the correct values of the west and east face area. At the start of the calculations we use the initial velocity field generated from the guessed mass flow rate. For subsequent iterations we use the corrected velocity obtained after solving the pressure correction equation. The source term S_u contains the pressure gradient integrated over the control volume

$$S_u=\frac{\Delta p}{\Delta x}\times \Delta V=\frac{\Delta p}{\Delta x}\times A_{av}\Delta x=\Delta p\times\frac{1}{2}(A_w+A_e) \qquad (5)$$

Since the nozzle has a varying cross-sectional area we use an averaged area to calculate ΔV. In summary the coefficients of the discretised *u*-equations are given by

$$F_w=\rho A_w u_w \;; F_e=\rho A_e u_e$$
$$a_W = F_w$$
$$a_E = 0$$
$$a_P = a_W + a_E + (F_e - F_w)$$
$$S_P=\Delta p\times\frac{1}{2}(A_w+A_e)=\Delta p\times A_P$$

The parameter *d* required in the pressure correction equations is calculated at this stage from

$$d=\frac{A_{av}}{a_P}=\frac{(A_w+A_e)}{2a_P} \qquad (6)$$

Pressure correction equation

The discretised form of the continuity equation in this one-dimensional problem is

$$(\rho uA)_e-(\rho uA)_w=0 \qquad (7)$$

The corresponding pressure correction equation is

$$a_P p'_P = a_W p'_W + a_E p'_E + b' \qquad (8)$$

where

$$a_W=(\rho dA)_w \;;\; a_E=(\rho dA)_e$$
$$b'=(F_w^*-F_e^*)$$

Values of the parameter *d* come from discretised momentum equations.

In the SIMPLE algorithm the pressure corrections p' are used to compute the velocity corrections u' and the corrected pressure and velocity fields using

$$u'=d(p'_I - p'_{I+1})$$
$$p = p^* + p'$$
$$u = u^* + u'$$

III. SIMPLE Algorithm

SIMPLE algorithm is an iterative method to calculate pressure and velocities. To initiate the SIMPLE calculation process, a pressure field p^* and velocity field u^* are guessed. Discretised momentum Eqns. (4) are solved using the guessed pressure field to yield velocity components u* at all velocity nodes as follows:

$$a_P u_P^* = a_W u_W^* + a_E u_E^* + S_u$$

Solving Pressure correction equation (Eqn. (8))

$$a_P p_P' = a_W p_W' + a_E p_E' + b'$$

pressure correction field p' can be obtained at all pressure points. Once the pressure correction field is known, the correct pressure field may be obtained using

$$p = p* + p' \qquad (9)$$

and velocity components through correction formula

$$u' = d(p_I' - p_{I+1}')$$
$$u = u* + u' \qquad (10)$$

The pressure correction equation is susceptible to divergence unless some under-relaxation is used during the iterative process. Using under relaxation factor a new, improved, pressures p_{new} are obtained with

$$p_{new} = p* + \alpha_p p'$$
$$or \quad p_{new} = (1-\alpha_p) p* + \alpha_p p \qquad (11)$$

where α_p is the pressure under-relaxation factor. If we take α_p equal to 1 the guessed pressure field $p*$ is corrected by p'. Taking α_p equal to zero would apply no correction at all, which is also undesirable. Taking α_p between 0 and 1 allows us to add to guessed field $p*$ a fraction of the correction field p' that is large enough to move the iterative improvement process forward, but small enough to ensure stable computations. The velocities are also under-relaxed. The iteratively improved velocity components u_{new} is obtained from

$$u_{new} = \alpha_u u + (1 - \alpha_u) u*$$

where α_u is the *u*-velocity under-relaxation factors.

For cost effective simulations, a correct choice of under-relaxation factors α is essential. A very small value of α may cause extremely slow convergence, whereas very large value leads to an oscillatory or even divergent iterative solutions. The optimum values of under relaxation factors are flow dependent and must be sought on a case-by-case basis.

IV. Numerical Calculations

In order to get unknown variables u (x-component of velocity) and p (pressure), numerical computations are carried out for different number of nodes. While doing the computations the density of the fluid ρ has been chosen to be 1.0 kg/m^3.The Nozzle length L is 2.00 m. The grid is taken to be uniform for all cases. The cross-sectional area at the inlet is 0.5 m^2 and at the exit is 0.1 m^2. The area changes as a linear function of distance from the nozzle inlet. For boundary conditions we assume that at inlet the flow entering the nozzle is drawn from a large plenum chamber; the fluid has zero momentum and the stagnation pressure at inlet p_0 = 10 Pa. The static pressure at exit is 0 Pa. To generate an initial velocity field for this problem we guess a mass flow rate say $\dot{m} = 1.0\ kg/s$ and use $u = \dot{m}/(\rho A)$ along with the cross-sectional areas at velocity nodes. To generate a starting field of guessed pressures we assume a linear pressure variation between pressure nodes.

The computation of u-velocity and pressure is done by following the method which has been described under Section III. The same algorithm has been implemented in MATLAB programming language. The unknown quantities, the velocity of the fluid flow in x direction and the pressure p, obtained for different number of velocity and pressure nodes. The exact solution to this steady, one-dimensional, incompressible, frictionless flow problem can be obtained using Bernoulli's equation. The analytical values of velocity and pressure fields obtained from Bernoulli Equation are given and compared with the iterative values for different number of nodes. The comparisons are given in the Table I given below:

TABLE I: VELOCITIES AND PRESSURE FIELD AT 5 VELOCITIES NODES AND 6 PRESSURE NODES WITH $\alpha_P = 0.1$ AND $\alpha_v = 0.9$

Velocity Node	Actual Value of u	Value of u after 20 iterations	Errors (%)
1	0.9722	1.2713	-31%
2	1.1769	1.5390	-31%
3	1.4907	1.9494	-31%
4	2.0328	2.6582	-31%
5	3.1944	4.1772	-31%
Pressure Node	**Actual Value of p**	**Value of p after 20 iterations**	**Errors (%)**
1	9.6	9.7128	-1%
2	9.4331	9.1719	3%
3	9.1350	8.7442	4%
4	8.5207	7.8931	7%
5	6.9136	5.7662	17%
6	0	0	0%

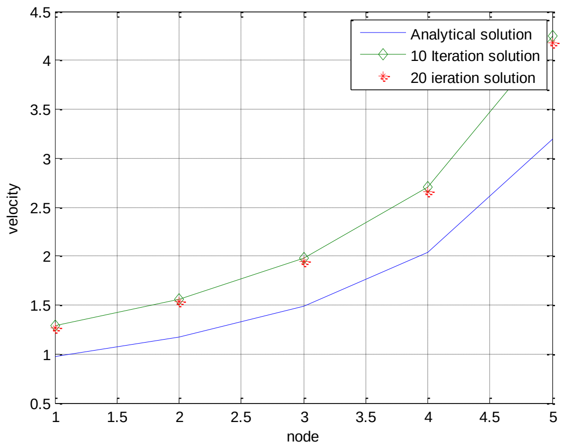

Fig.2(a)

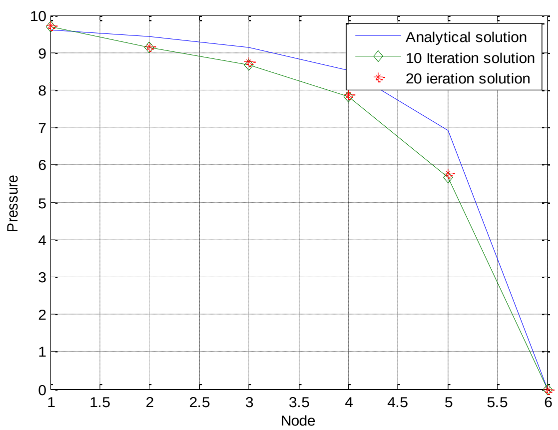

Fig. 2(b)

Figure 2(a) &2(b): Velocities and Pressure field at 5 velocities nodes and 6 pressure nodes with $\alpha_p = 0.1$ and $\alpha_u = 0.9$

TABLE II: VELOCITIES AND PRESSURE FIELD AT 10 VELOCITY NODES AND 11 PRESSURE NODES WITH $\alpha_P = 0.1$ AND $\alpha_U = 0.9$

Velocity Node	Actual Value of u	Value of u after 20 iterations	Errors (%)
1	0.9317	1.0354	-11%
3	1.1180	1.2425	-11%
5	1.3975	1.5531	-11%
7	1.8634	2.0708	-11%
9	2.7951	3.1063	-11%
10	3.7268	4.1417	-11%
Pressure nodes	Actual value of P	Value of P after 20 iterations	Errors (%)
1	9.6000	9.831	-2.4%
3	9.4331	9.3447	0.93%
5	9.1350	9.1108	0.26%
7	8.5207	8.4122	1.27%
9	6.9136	6.603	4.49%
10	4.8980	4.5947	6.19%

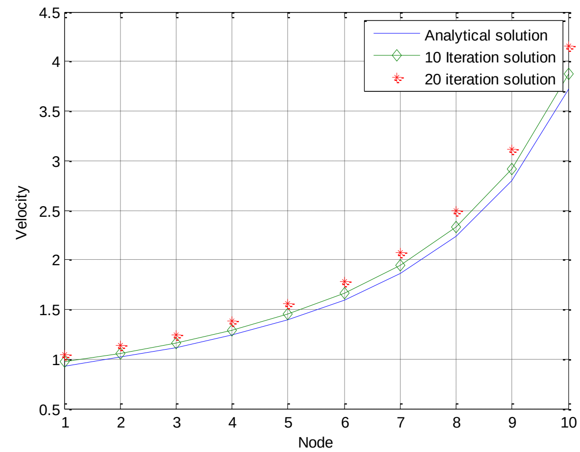

Fig.3(a)

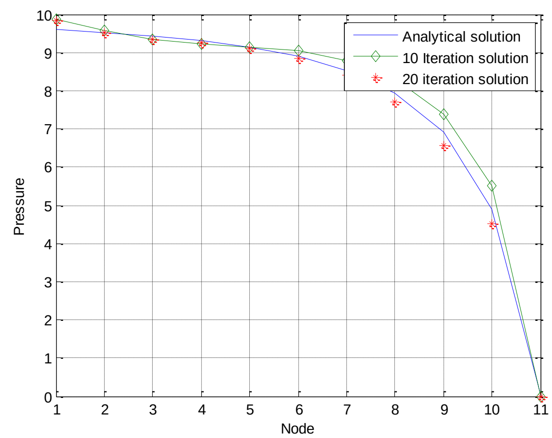

Fig. 3(b)

Fig 3(a) &3(b): Velocities and Pressure field at 10 velocity nodes and 11 pressure nodes with $\alpha_p = 0.1$ and $\alpha_u = 0.9$

TABLE III: VELOCITIES AND PRESSURE FIELD AT 20 VELOCITIES NODES AND 21 PRESSURE NODES WITH $\alpha_P = 0.1$ AND $\alpha_U = 0.9$

Velocity Node	Actual Value of u	Value of u after 20 iterations	Errors (%)
1	0.9127	0.9396	-2.9%
5	1.0908	1.1230	-2.9%
10	1.4426	1.4852	-2.9%
15	2.1296	2.1925	-2.9%
20	4.0655	4.1857	-2.9%
Pressure nodes	Actual value of P	Value of P after 20 iterations	Error (%)
1	9.6000	9.8233	-2.3%
5	9.4331	9.4361	-0.03%
10	9.0235	8.8779	1.61%
15	7.9339	8.2424	-3.8%
20	3.0557	3.2093	-5.02%
21	0	0	--

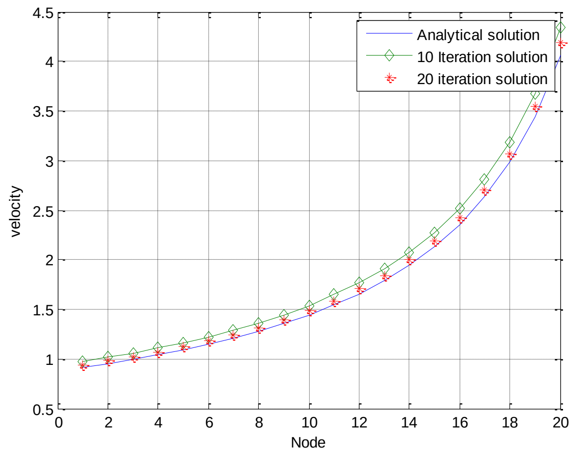

Fig 4(a)

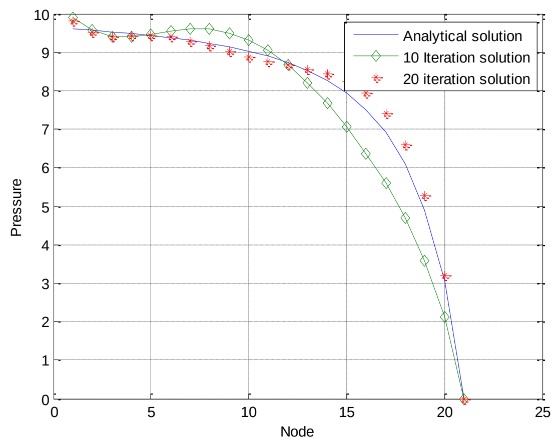

Fig 4(b)

Fig 4(a)& 4(b): Velocities and Pressure field at 20 velocity nodes and 21 pressure nodes with $\alpha_p = 0.1$ and $\alpha_u = 0.9$

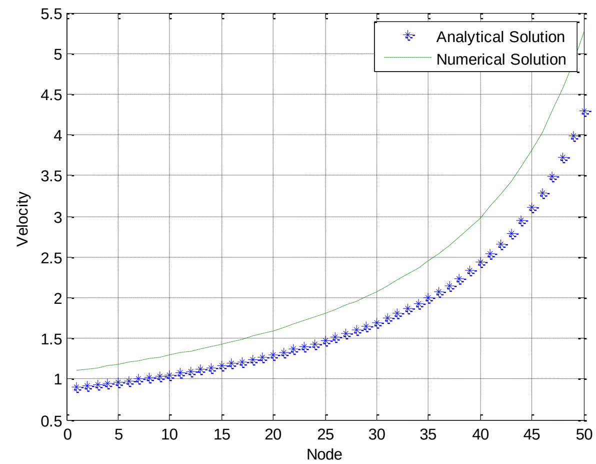

Fig 5(a)

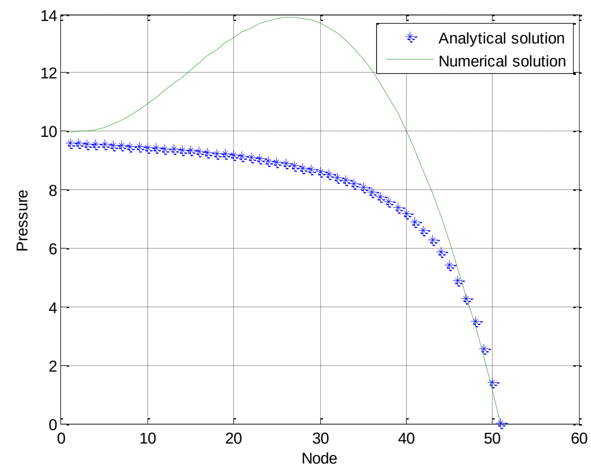

Fig 5(b)

Fig 5(a)&5(b): Velocities and Pressure field at 50 velocity nodes and 51 pressure nodes with $\alpha_p = 0.1$ and $\alpha_u = 0.9$

TABLE 4: VELOCITIES AND PRESSURE FIELD AT 50 VELOCITIES NODES AND 51 PRESSURE NODES WITH $\alpha_P = 0.1$ AND $\alpha_U = 0.9$

Velocity Node	Actual Value of u	Value of u after 5 iterations	Errors (%)
1	0.9016	1.1063	-22%
10	1.0547	1.2922	-22%
20	1.300	1.5952	-22%
30	1.6940	2.0786	-22%
40	2.4305	2.9823	-22%
50	4.3001	5.2764	-22%
Pressure nodes	Actual value of P	Value of P after 5 iterations	Error (%)
1	9.6000	9.9499	-3.6%
10	9.4541	10.9366	-15%
20	9.1743	13.2153	-44%
30	8.6077	13.6955	-59%
40	7.1707	10.0079	-39%
50	1.4267	1.1686	18%

V. Result and Discussion

The values of velocity and pressure fields are obtained using different number of velocity and pressure nodes. During the numerical process, different values of under relaxation factors α_p and α_u were tested for all cases. It was seen that for a grid with 10 velocity nodes, the solution was converging for many values of α_u ranging from 0.9 to 0.4, but the values of α_p for which the solution converges could be taken only from 0.1 to 0.3.In case of a grid with 20 velocity nodes and 50 velocity nodes ,the value of α_p for which solution converges could be only 0.1.Hence, we can say that the solution converges for many values of under relaxation factor for velocity α_u but only for few values of under relaxation factor for pressure α_p. It was also observed that as the grid gets refined i.e. number of nodes increases, the solution converges for fewer values of $\alpha_{p.}$ It was also

observed that pressure values converges faster than the velocity values when the grid is refined whereas for coarser grid velocity value converges faster. In Fig. 2(a & b) it can be seen that for a coarse grid in which number of velocity and pressure nodes are less, the solution converges to a limited extent. In such a case, increasing the number of iterations does not converge the solution any further. In Fig. 3(a & b) it can be seen that for a 10 velocity nodes and 11 pressure nodes grid a good approximation for velocities can be obtained in 10 iterations whereas pressure fields are obtained with good approximations only after 20 iterations.
From Fig. 4(a & b) it is clear that better approximation for velocity and pressure fields can be obtained by increasing the number of iterations. From fig. 5(a & b) it can be seen that the solution converges very fast in case of finer grid.
From Tables 1, 2 and 3, it can be clearly seen that by refining the grid, the error can be reduced using same number of iterations.

Using Bernoulli Equation the mass flow rate $\dot{m}$ is obtained as 0.44721 kg/s. In case of 5 node grid the converged mass flow rate is 0.5848 kg/s which is 30% higher than the exact value. After refining the grid with 10, 20 and 50 velocity grid points, the converged mass flow rates obtained are 0.4985 kg/s , 0.4604 kg/s and 0.4597 kg/s respectively .The errors in these cases are 11% , 3% and 2.79 % respectively. This demonstrates that the errors in the solution can be reduced by refining the grid. This is graphically illustrated in Fig. 6.

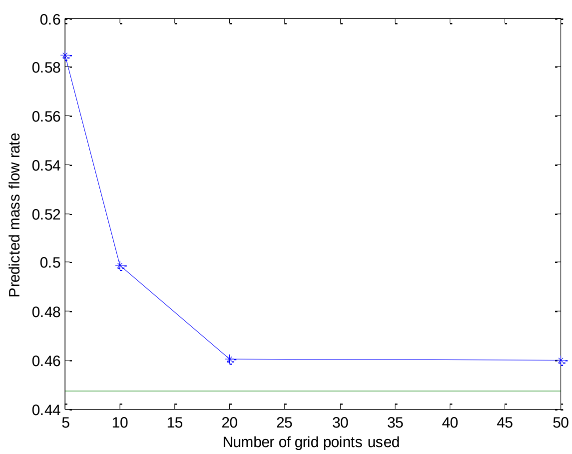

Fig 6: Predicted mass flow rate with different grids

One of the important properties of SIMPLE algorithm is that, at the end of each iteration cycle the velocity field satisfies continuity. This property of SIMPLE ALGORIHM also applies in more complex multi-dimensional problems and is a major strength of the SIMPLE algorithm and its variants. However, at the end of an iteration cycle the momentum is not conserved because the computed velocity solution is not yet in balance with the computed pressure field. This is due to the fact that the entries in the discretised momentum equations were computed on the basis of an assumed initial velocity field. Therefore, iterations are performed until both continuity and momentum equations are satisfied.

The discretised momentum equations will not satisfied by the under-relaxed values of velocity and pressure fields. The difference between the left and right hand sides of the discretised momentum equation at every velocity node is called the momentum residual. If the value of these residual decreases with increasing iterations it shows that the iteration sequence is convergent. Usually, we stop the iteration process when mass and momentum are exactly balanced in the discretised pressure correction and momentum equations.

VI. Conclusions

The problem of a steady 1-D incompressible flow through a planar two dimensional nozzle was investigated. Numerical solutions for the governing equations are obtained using SIMPLE algorithm. The numerical computations for u-velocity and pressure were conducted using staggered grid system. Conclusions of this study are as follows:

1. The solution converges for many values of under relaxation factor for velocity α_u but only for few values of under relaxation factor for pressure α_p.
2. For a coarse grid in which the number of velocity and pressure nodes are less, the solution converges to a limited extent
3. For a given grid, better approximation for velocity and pressure fields can be obtained by increasing the number of iterations.
4. The error can be reduced by refining the grid and using same number of iterations.

REFERENCES

[1]. Suhas V. Patankar, “Numerical Heat Transfer and Fluid Flow”, Hemisphere Publishing Corporation, NewYork 1980.
[2]. D.A. Anderson, R.H. Pletcher, J.C. Tenihill, “Computational Fluid Mechanics and Heat Transfer”, second Edition, Taylor and Francis, Washington D.C. 1997.
[3]. H.K. Versteeg, W. Malalsekra, “An Introduction to Computational Fluid Dynamics: The Finite Volume Method”, Second Edition, Pearson, India 2007.

[4]. Saurabh Verma, Dr. S.K. Moulick, Mr. Santosh Kumar Mishra," Nozzle Wear Parameter In Abrasive waterjet machining- The Review", International Journal of Engineering Development and Research , Vol.-2, Issue- 1, page no.(1063-1073), March 2014.

[5]. Snehil Varghese, Dushyant Kalihari & Brijesh Patel,"Critical Review Paper of Modelling of Compressible Nozzle Flow through Variable Area Duct", International Journal of Engineering Research & Technology Vol.- 4 Issue- 06, June 2015

[6]. Sriram, M.A., Rajan, N.K.S. and Kulkarni, P.S., "Computational Analysis of Flow through a Multiple Nozzle Driven Laser Cavity and Diffuser", Computational Fluid Dynamics, Vol-9, page no. (759-764), 2009.

[7]. Vincent Lijo," Numerical simulation of transient flows in a rocket propulsion nozzle", International Journal of Heat and Fluid Flow, Vol.-31, page no. (409– 417), 2010.

[8]. Osamu Tonomura , Shotaro Tanaka,Masaru Noda, Manabu Kano, Shinji Hasebe, Iori Hashimoto, "CFD-based optimal design of manifold in plate-fin microdevices", Chemical Engineering Journa,l, Vol- 101, Issues-3, , Pages (397–402) August 2004.

[9]. Gianluca Iaccarino,"Predictions of a Turbulent Separated Flow Using Commercial CFD Codes", J. Fluids Eng, Vol-123 issue-4, page - 819-828 , May 2001

[10]. P. Greyvenstein, D. P. Laurie,"A segregated CFD approach to pipe network analysis", International Journal for Numerical Methods in Engineering, Vol -37, Issue -21, Page (3685–3705), November 1994

[11]. Jong-Jin Baika, Jae-Jin Kimb, and Harindra J. S. Fernando ,"A CFD Model for Simulating Urban Flow and Dispersion", Journal of Applied Meteorology, Vol- 42 Issue- 11, November 2003.

[12]. I. E. Barton," Comparison of SIMPLE- and PISO-type algorithms for transient flows", International Journal for Numerical Methods in Fluids,Vol- 26, Issue -4, Pages (459–483),February 1998.

[13]. J.P. Van Doormaal & G.D. Raithby, "Enhancement of the SIMPLE method for predicting incompressible fluid flow", Numerical Heat Transfer, Vol-7, Issue-2 Pages (147-163)

Author Profiles

Arti Kaushik is an Assistant Professor at the Department of Mathematics, MAIT, Delhi. Her research interest is in the area of Fluid Dynamics and Compu-tational Fluid Dynamics. She has published many research papers in National and International Journals.

Anil Kumar has been associated with Department of Mathematics, MAIT, Delhi for last 4 years. He is working as an Assistant Professor. He is an ardent researcher. His research interest includes Plasma Physics and Computational Solar MHD. He has published many research papers in National and International Journals of repute.