Sindh Univ. Res. Jour. (Sci. Ser.) Vol. 50 (001) 65-72 (2018)
http://doi.org/10.26692/sujo/2018.01.0011
SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Finite Volume Simulation of Newtonian Fluids Through Combined Converging and Diverging Channel

N. A. MEMON, H. SHAIKH, B. SHAH, A. BALOCH*

Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs, Sindh

Received 11th January 2017 and Revised 19th December 2017

Abstract: A robust semi-implicit pressure-based Computational Fluid Dynamics (CFD) scheme (SIMPLE) scheme is adopted for modeling of steady and viscous compressible flows through converging and diverging channel. Second-order spatial accuracy will achieved through linear unstructured finite volume cell discretization. The SIMPLE scheme characterizes a development in mass-momentum coupled, pressure based schemes. The governing equations for this scheme are the conservative form of momentum equations (Navier-Stokes) and mass conservation equation. The grid will be developed and refined through Gambit package. Flow structure will be developed with effect of fluid inertia, primary, secondary and treasury vortex may be developed in the aspect ratio 1: 4, 1: 6 and 1: 8 of the converging and diverging channels. The small vortex observed in the ratio 1: 4 in each corner of the channel and with increasing the fluid inertia the left upper vortex in size is enhanced only due to increase the Reynolds numbers. When increased the ratio 1: 6 and 1: 8 the vortex in size is enhanced at lower and upper corner of the channel and occupied the whole domain. Due to filling the Porous material all vortices left, right, upper and lower are diminished and no recirculation flow rate observed at each Reynolds numbers and on every Darcy numbers. The numerical results achieved through finite volume technique through commercial CFD package ANSYS Fluent and will be compared with analytical approach of sheikh, et al. [2012] and also other numerical results with and without use of CFD packages.

Keywords: Open and Closed Channel flows, Finite volume Method, Porous Media.

1. INTRODUCTION

The study based upon the CFD analysis that is largely growing since the computer power and technology is developed and the CFD became a strong need in the study of the fluid flow features. The fluid flow features are controlled through mathematical equations with the computer programing with the addition of the simple and complex required domain either for channels or pipes. Here required domain is to examine the fluid flow structure through converging and diverging channel that have strong applications in the applied engineering and industries. Especially, application focused on the polymer industries such as fluid mixing and diffusing through tanks or pipes. It is also applicable in the automotive industries particularly in the vehicle industries for the development of designs of the heavy and large vehicle silencers to examine the gas flow phenomena through silencers fitted in the large racing vehicles. (Dufresne, *et al.* (2009) Abrishamchi, *et al.* 2013, and Pande, *et al.* 2015).

Various research scholars and mathematicians have contributed in the CFD particularly for the tank and pipe studies. Such as,experimental modeling was developed by Dufresne, *et al.* (2009) for the investigation of symmetric and asymmetric fluid flow features in the rectangular superficial reservoir. The different influences on the flow structure were observed as variance in the expansion ratio, dimensionless variables influences particularly Reynolds number and effect in the Froude number. The vortex intensity and reattachment length was very lightly increased in the lower silent corner of the symmetric and asymmetric channel and with growing the higher Reynolds numbers the various eddies was found in small eddies available in the lower corner, upper reattachment corner and also large eddy was found in the main core flows. The very limited experimental results was tested for the investigations of flowpattern and itself was proposed to examine the stream phenomena through the selection of advanced scientific tools.

Abrishamchi, *et al.* (2013) was studied and developed the two –dimension numerical model (combing both finite element and finite volume) for the liquid flows in the converging and diverging tanks and focused on the heat transfer also. For the numerical methodology, two CFD package COMSOL and ANSYS Fluent are chosen for the finite element and finite volume discretizations and computations of the vector velocity field and pressure with the use of governing equations like continuity and momentum equations. The different eddies were developed in the silent corner for the expansion channel due to the use of the turbulent flow phenomena with selection of k-ε model. The vortex intensity and vortex length were computed and observed the linear enhancement with increasing the Reynolds number. It was concluded that the finite

++ Correspondence Email asadnadia2014@gmail.com,hisam.shaikh@salu.edu.pk,
*ISRA University, Hyderabad

element approach was found high accuracy in the results other than the finite volume results. Also the finite element results were better than the experimental results because of the low cast in numerical lab other than the experimental setup.

In the literature, found that CFD packages are most suitable and high stable computational power. That's why selected the commercial CFD package ANSYS Fluent for the analysis the Newtonian fluid move through the combined channel of converging and diverging fitted with Pore material. The finite volume technique is applied in which semi implicit pressure linked equations (SIMPLE) scheme is chosen for the discretization of the governing equations. Through CFD package the results are observed through development of flow structures i. e. streamline patterns, recirculation flow rate and reattachment length that are the function of the various fluid inertia.

2. GOVERNING SYSTEM OF EQUATIONS

The basic equations derived from physical law i.e. Newton's 2nd law of motion are exercised for the movement of liquid fluids over domain in the occurrence of the Porous material so it is necessary to adopt the physical law relates on the pore material that is called Darcy law. This law is added into the Navier–Stokes equations and constituted the two governing equations such as continuity and momentum equation. These equations are consisted the various physical terms i.e. diffusion term, convection term, Pore material term, pressure term and time derivative term in which velocity and pressure term involved and considered as continuum and macroscopic level and assumes the fluid is isotropic without body forces. The general form of the governing equations is expressed as under:

$$\nabla.\boldsymbol{w} = 0 \quad (1)$$

$$\rho\frac{\partial \boldsymbol{w}}{\partial x} = \nabla.\left(2\mu \underline{\underline{d}}\right) - \rho(\boldsymbol{w}.\nabla)\boldsymbol{w} - \frac{\omega\mu}{k}\boldsymbol{w} - \nabla p \quad (2)$$

For the computation purpose, it is most needed to use scaling factor to convert into the non – dimension form. Therefore, introduce the highly popular non – dimensional numbers called as Reynolds number and Darcy number that are given as below:

$$\mathrm{Re} = \frac{\rho \boldsymbol{u}_c L_c}{\mu} \text{ and } D_a = \frac{k\rho \boldsymbol{u}_c}{\omega\mu L_c}$$

Now substituting the above non – dimensional numbers into (1) and (2), the governing equations are resulted as under:

$$\nabla.\boldsymbol{w} = 0 \quad (3)$$

$$\frac{\partial \boldsymbol{w}}{\partial t} = \frac{1}{\mathrm{Re}}\nabla^2\boldsymbol{w} - (\boldsymbol{w}.\nabla)\boldsymbol{w} - \frac{1}{\mathrm{Re}\,D_a}\boldsymbol{w} - \nabla p \quad (4)$$

The research problem is limited to two – dimension. Therefore, equations are limited as under:

$$\frac{\partial w_1}{\partial x} + \frac{\partial w_2}{\partial y} = 0 \quad (5a)$$

$$\frac{\partial w_1}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 w_1}{\partial x^2} + \frac{\partial^2 w_1}{\partial y^2}\right) - w_1\left(\frac{\partial w_1}{\partial x} + \frac{\partial w_1}{\partial y}\right) - \frac{1}{D_a \mathrm{Re}}w_1 - \frac{\partial \rho}{\partial x} \quad (5b)$$

$$\frac{\partial w_2}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 w_2}{\partial x^2} + \frac{\partial^2 w_2}{\partial y^2}\right) - w_2\left(\frac{\partial w_2}{\partial x} + \frac{\partial w_2}{\partial y}\right) - \frac{1}{D_a \mathrm{Re}}w_2 - \frac{\partial \rho}{\partial y} \quad (5c)$$

The problem of interest is steady state solution of (5) that is computed initially into 1D as under:

$$w(y) = w_{\max}\left[1 - \frac{\sinh\frac{y-a}{\sqrt{D_a}} + \sinh\frac{b-y}{\sqrt{D_a}}}{\sinh\frac{b-a}{\sqrt{D_a}}}\right] \quad (6)$$

Here a and b are used as boundary values of the converging and diverging channel and fitted the parabolic velocity profile that is expressed as under:

$$w = \mathrm{v}_{\mathrm{mean}} \times y \times (1 - y) \quad (7)$$

3. NUMERICAL SCHEME

Initially, in 1972 Patankar and Spalding was developed the iterative scheme for the computations of the 2nd order Navier–stokes equations known as SIMPLE scheme and it is a part of the finite volume approach. Due to quick and accurate results found in literature, chosen here same scheme through CFD package ANSYS Fluent and used for the computations of velocity and pressure for staggered grid system. This scheme is initiated through pressure component as initial approximation and will measure the vector components of momentum equation. The guessing parameters as p^*, w_1^* and w_2^* are considered through the following number of steps employed in the SIMPLE Scheme.

Step –01 In the start, initialise the pressure (p^*) to compute the velocity components (w_1 and w_2) by applying the following process.

$$a_{i,j}w_1{*}_{i,j} = \sum a_{nb}w_1{*}_{nb} + (\rho*_{i-1,j} - \rho*_{i,j})A_{i,j} + b_{i,j}$$

(8)

$$a_{i,j}w_2{*}_{i,j} = \sum a_{nb}w_2{*}_{nb} + (\rho*_{i,j-1} - \rho*_{i,j})A_{i,j} + b_{i,j}$$

Step –02 Now using the following process to computer the predicted pressure *p'*.

$$a_{i,j}\rho'_{i,j} = a_{i-1,j}\rho'_{i-1,j} + a_{i+1,j}\rho'_{i+1,j} + a_{i,j-1}\rho'_{i,j-1} + a_{i,j+1}\rho'_{i,j+1} + b'_{i,j} \quad (9)$$

Step –03 Finally need the corrected pressure (*p*) and velocity (u_1 and u_2) by the following discretization process

$$p_{i,j} = \rho*_{i,j} + \rho'_{i,j}$$
$$w_{1(i,j)} = w_1{*}_{i,j} + d_{i,j}\left(\rho'_{i-1,j} - \rho'_{i,j}\right) \quad (10)$$
$$w_{2(i,j)} = w_2{*}_{i,j} + d_{i,j}\left(\rho'_{i,j-1} - \rho'_{i,j}\right)$$

Step –04 Examine the convergence solution of SIMPLE Scheme, if satisfied otherwise change the new initial approximations of p^*, u_1^* *and* u_2^*.up to the required accuracy is achieved.

$$p^* = p, \qquad w_1{*} = w_1$$
$$w_2{*} = w_2, \qquad \Psi^* = \Psi \quad (11)$$

4. Problem Solving and Computational process

Due to great importance of converging and diverging channel in the various engineering and industries i.e. designing of tanks and tubes, selected here the combined channel of converging and diverging shape with ratio 1:6 for the investigation of liquid fluid flow behavior. (Sohu, *et al.* 2009, Solangi 2011) and Sheikh, *et al.* 2012).

Two small rectangular channel with length 5L and 15L and one large rectangular channel with length 10L are combined to develop the converging and diverging channel. Due to finite volume discretization process choose here quadrilateral elements for the complete unstructured grid of the converging and diverging channel. The mesh statistics are based upon the 8500 total elements and 8801 total nodes.

The computational process of finite volume approach is chosen for the steady state numerical solution and the most converging and quick scheme SIMPLE is adopted for the piece wise computations of the pressure and velocity component. At the last, the finite volume result is validated with analytical 1D solution of Shaikh, *et al.* (2012) and other numerical and experimental results of renowned scholar particularly (Dufresne, *et al.*(2008 and 2010)

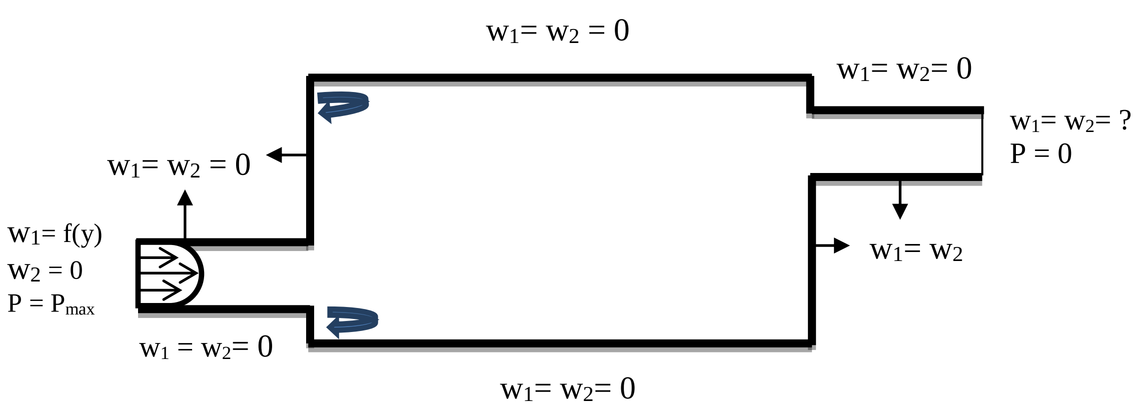

Fig. 1: Schematic Diagram of Converging and Diverging channel

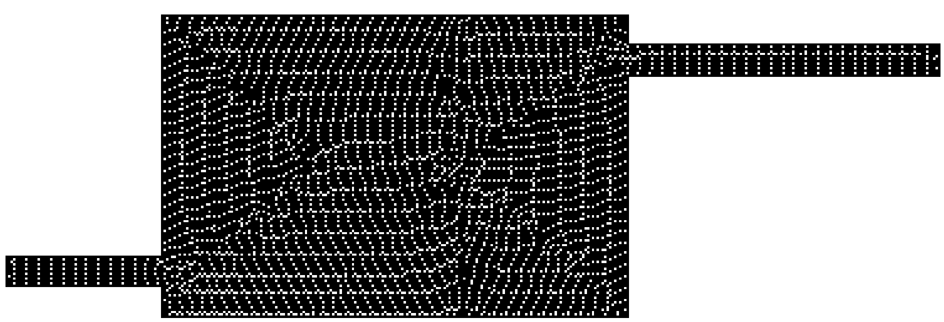

Fig. 2: Refined mesh of converging and diverging channel.

5. NUMERICAL RESULTS AND DISCUSSION

The numerical results are obtained through CFD package ANSYS Fluent by choosing the finite volume approach. The finite volume approach is based upon the various discretization procedures like SIMPLE, SIMPLEC AND PISO. Due to quick response and converge solution adopted here the SIMPLE scheme and is consisted on the three steps and also known as predictor corrector approach. The influence of fluid inertia on flow phenomena with and without Porous material is discussed as follows:

5.1 Impact of Fluid Inertia on the Flow Phenomena Through Converging and Diverging Channel

The Different streamline patterns of the velocity with increasing the fluid inertia are observed and plotted the recirculation flow rate and reattachment length of the silent corner of the converging channel. In **(Fig. 1)** displays the complete schematic diagram of the converging and diverging channel with detail initial and boundary conditions imposed separately on the stationary walls, inlet and outlet of the channel. **(Fig. 2)** displays the refined mesh designing of the channel with quadrilateral number of elements.

(Fig. 3) demonstrates the streamline laminar flow patterns of the velocity ranging from 01 to 1000 fluid inertia rate. Started from the unit fluid inertia, the very tinny eddies are observed in the all four corners of the channel, like left lower and upper and right lower and upper in the main large expansion channel. Due to increasing the fluid inertia (lower rates up to 100), the left corner eddies are very slowly enhanced and right corner eddies are diminished slowly. At higher Reynolds numbers (Re > 100), the clear enhancement is observed in the corners of the channel. With increasing the fluid inertia rate the left lower corner eddy is too slowly enhanced but the left upper corner eddy is highly enhanced in size in horizontally towards outlet of the channel and almost filled the region of the expansion channel. But the right corner eddies are remain almost same in all Reynolds numbers except higher Reynolds numbers (Re =1000), right corner lower eddy is also grows in size but slowly other than the left corner lower eddies in size.

After developing the various streamline patterns on the bases of different fluid inertia rates, found the minimum and maximum values of the recirculation flow rate. Figure–4 displays the graph of the recirculation flow rate and two trends are observed. Firstly the recirculation flow rate is linearly grows on the lower Reynolds umbers up to 1000 and with increasing the Reynolds number after 1000, the enhancement observed non–linearly patterns.

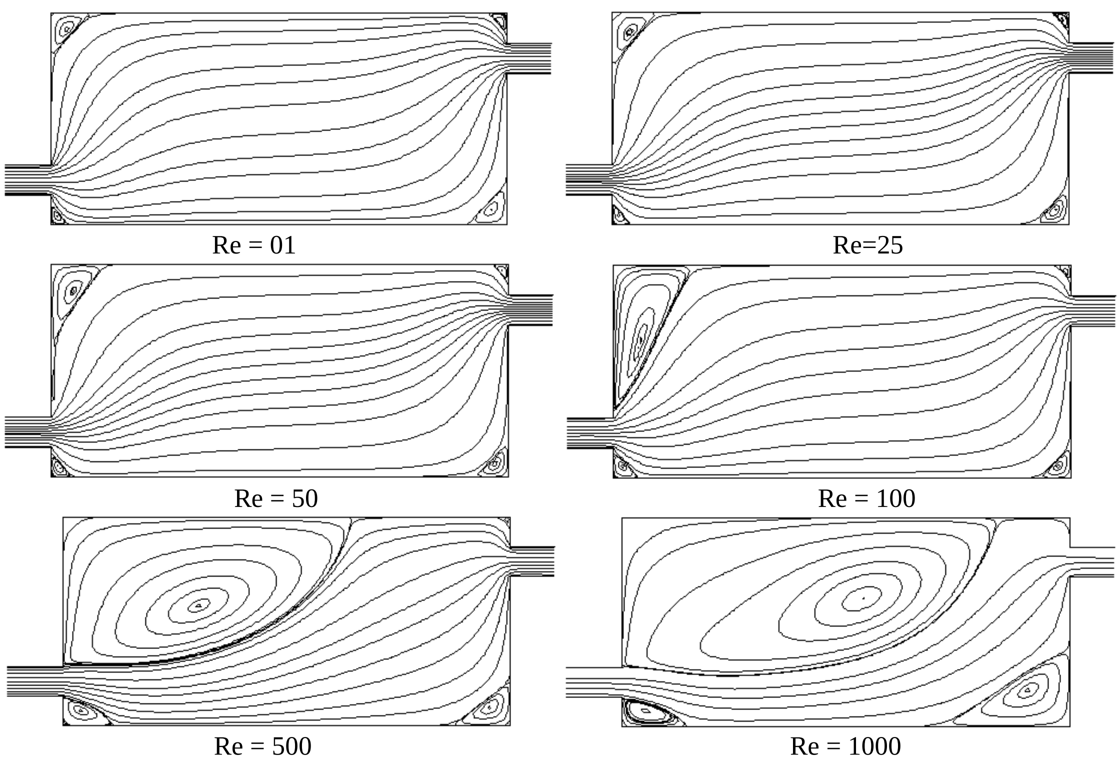

Fig. 3: Two–dimensional Streamlines flow pattern of combined Converging and Diverging channel.

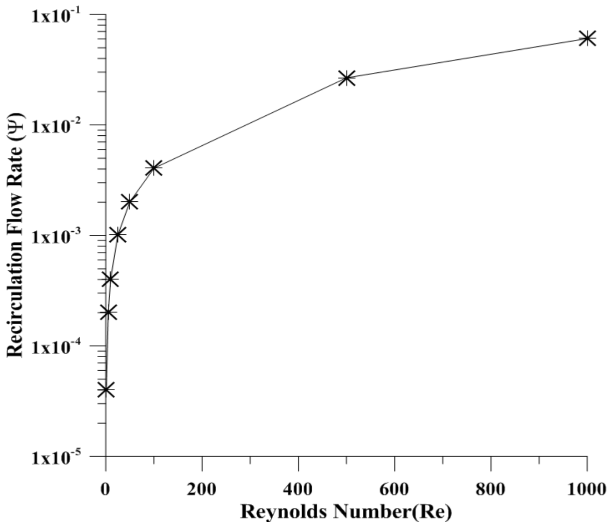

Fig. 4: Graph shows the recirculation Flow rate (RF) on the grows of fluid inertia (Re) forConverging and Diverging channel

5.2 Impact of Fluid Inertia on the Flow Phenomena Through Converging and Diverging Channel with The Presence of Porous Material

The impact of various fluid inertia from 01 to 1000 and different Darcy numbers such as 0.0001 to 0.1 are used to analyze the behavior of fluids (Newtonian) through combined channel over Porous material.

Here observed the streamline flow patterns for the combined converging and diverging channel in a ratio 1:6. The behavior of fluid observed through streamline flow patterns and found totally altered as previous streamline flow patterns in the absence of Pore material. At lower and higher fluid inertia rates, the no any eddy is appeared and completely vanished all eddies in all corners of the channel. Also various Darcy numbers lower and higher are tested but the vortex is wholly disappeared in the domain of the problem.

(Fig. 5) displays the streamline flow patterns of the velocity with filled Pore material in the channel and no any vortices appeared in all results on the impact of fluid inertia. Only flow path are observed through laminar linear paths move through inlet to the outflow of the domain. The detail values of the recirculation flow rate are listed in the table – 01 given as below. Also the main important contribution in the research developed here the empirical equations that based upon the recirculation flow rate on the impact of fluid inertia that are given as:

$$X = 0.00006\mathbf{Re} - 0.000813$$

Here Re shows the Reynolds number that is listed as under:

$$\mathbf{Re} = \frac{\rho \times w_{\max} \times L}{\mu} \qquad 01 \leq \mathbf{Re} \leq 1000$$

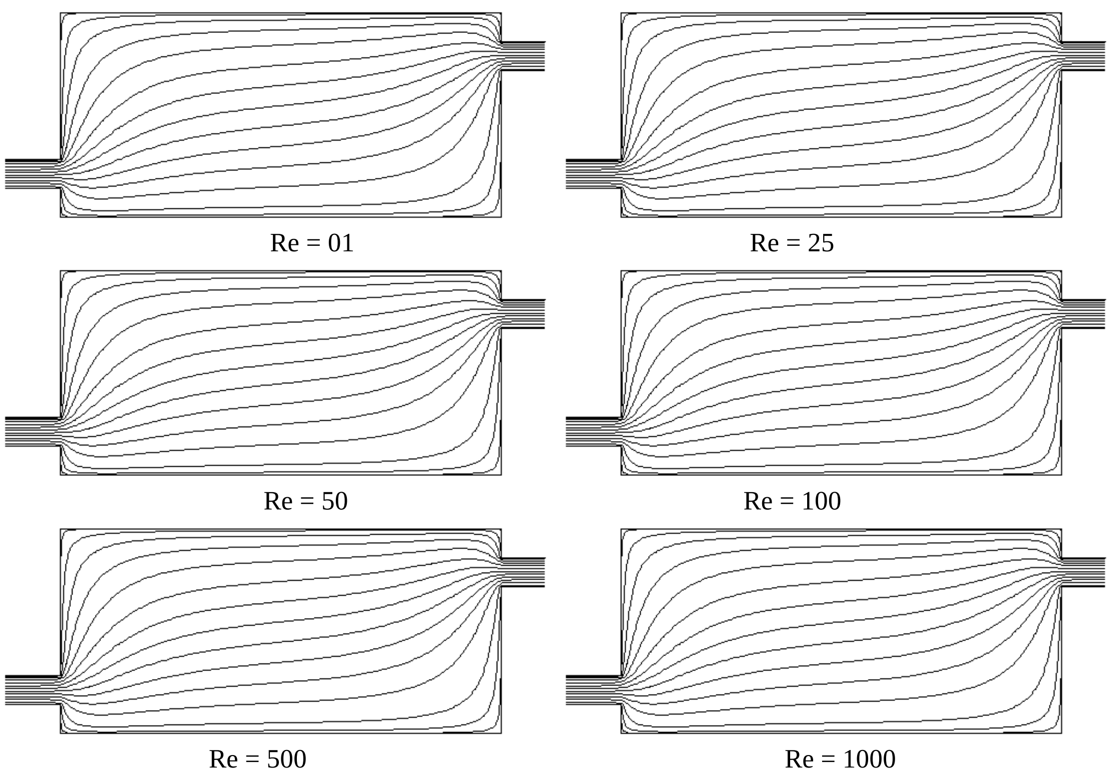

Fig. 5: Two–dimensional Streamlines flow pattern of combined Converging and Diverging channel filled with Pore material

Table –1: Table displays the all values of the Recirculation flow rate (Ψ) with different Reynolds number in both forms Porous and non – porous

WITHOUT POROUS MEDIA			WITH POROUS MEDIA	
Recirculation Flow Rate			Recirculation Flow Rate	
Re	1:6 ratio		1:6 ratio	
	Minimum flow rate	Maximum flow rate	Minimum flow rate	Maximum flow rate
01	1e-009	4.03708e-5	7e-008	4.03206e-5
05	1e-008	2.01856e-4	1e-007	2.01603e-4
10	1e-008	4.03717e-4	1e-006	4.03206e-4
25	5e-008	1.00934e-3	1e-006	1.00802e-3
50	1e-007	2.01954e-3	1e-006	2.01603e-3
100	1e-006	4.064e-3	1e-005	4.03206e-3
500	1e-005	2.67292e-2	0.0001	2.01603e-2
1000	5e-005	6.05519e-2	7e-005	4.03206e-2

6. CONCLUSION

The scientific approach SIMPLE is adopted with the use of CFD package ANSYS Fluent for the approximate solution of the governing equations (continuity and momentum equation). The approximate results are described through the various flow structures just as streamline patterns of the velocity, recirculation flow rate with different Reynolds numbers and various impacts of Darcy Numbers. Two research problems are presented through the converging and diverging channel as without Pore material and with pore material. Without Pore material, the various small eddies are observed in all left and right corners of the expansion channel at lower fluid inertia. Due to continuous grow fluid inertia up to 1000, the lower left corner eddy is very too slowly increased but left upper eddy is highly enhanced and almost filled the region of the converging channel. Also right lower eddy is too slowly enhanced and displays in figure –03. Also the continuous enhancement is also visualized in the graph of the recirculation flow rate given in Fig, 4.

Secondly, various changing in the Darcy numbers are observed due o filled pore material, all eddies are diminished either left lower and upper corners and right lower and upper corners. No any small vortex is visualized at all values of the fluid inertia. Also, developed the new empirical equations of the recirculation flow rate on the impact of fluid inertia. The good approximate results are acquired with the comparison of the 1D analytical solution and with other approximate results on the requited terminated factor 10^{-5}.

ACKNOWLEDGEMENT:

The research fellow and all contributed coauthors are greatly pleased to department of Mathematics, Shah Abdul Latif University Khairpur.

REFERENCES:

Abrishamchi, I., A. Okhovat, and S. M Nowee,.(2013) "A Comparative Numerical Investigation of Flow through Channel Expansion system, Using Finite Volume and Finite Element Methods", Petroleum & Coal research journal, www.vurup.sk/petroleum-coal, Vol. 55 (1) 20-25.

Ashhab, M. S., A. A. Salaymeth, and A. A. Muhtaseb, (2012), and Muhtaseb, M., “Hershel-bulkely Blood flow in rectangular Micro channel”, IJRRAS, Jorden.

Baloch, A. (1994), Numerical Simulation of Complex Flows of Non –Newtonian Fluids’, Ph.D. Thesis, University of Wales, Swansea, UK.

Dewals, (2008), “Experimental and numerical analysis of flow instabilities in rectangular shallow basins”, Environmental Fluid Mechanics, Vol 8.

Dufresne, M., J., Vazquez, A., Terfous, A. Ghenaim, and J. B. Poulet, (2008), “Experimental investigation and CFD modeling of flow, sedimentation, and solids separation in a combined sewer detention tank”, Computers and Fluids, article in press.

Dufresne, M., B. J. Dewals, S. Erpicum, P. Archambeau, and M. Pirotton, (2009), “Classification of flow patterns in rectangular shallow reservoirs”, University of Liege (ULg), ArGEnCo department, Hydrology, applied hydrodynamics and hydraulic constructions, Vol. 1, 2– 3.

Dufresne, (2010), "Experimental investigation of flow pattern and sediment deposition in rectangular shallow reservoirs", International Journal of Sediment Research

Gosman, A. D., E. E. Khali, and J. H Whitelaw .(1977), “The calculation of two-dimensional turbulent recirculating flows", University Park Pa., Pennsylvania State University, Vol–1, 13.35-13.45.

Kantoush, S. (2007), "Symmetric or asymmetric flow patterns in shallow rectangular basins with sediment transport", Proc., Harmonizing the Demands of Art and Nature in Hydraulics, 32nd Congress of IAHR, Venice.

Kumar, A. and S. Ram (2014), "Design Modification of Sedimentation Tank", International Journal for Scientific Research & Development, Vol. 2, Issue 09, 2321-0613.

Launder, B. and D. B. Spalding, (1974), "The Numerical Computation of Turbulent Flow Computer Methods", Computer Methods in Applied Mechanics and Engineering, Vol. 3 (2), 269-289.

Mullin, T., J. Peixinho, (2006), "Transition to Turbulence in Pipe Flow", Journal of Low Temperature Physics", 145, 75-88.

Pande, P. K. (2015), "Sediment Management in Hydropower Plants –An overview", International Conference on Hydropower for sustainable development, Dehradun.

Sahu, J. (2009), "Developed laminar flow in pipe using computational fluid dynamics", 7th International R&D Conference on development and management of water and energy resources, bhubaneswar, India.

Shaikh, H., S. B. Shah, M.A. Solangi, and A Baloch,. (2012), "A Computer Simulation of Flow of Newtonian Fluid through Backward Step Channel", Sindh Univ. Research Journal (Science Series), Vol.44 (4) 703-708.

Shaikh, K. (2016), "Influence of Local Inertia in the Forward Step Channel Filled with Porous Media", Journal of Applied Environmental and Biological Sciences, Vol 6(4) 177-184.

Shaikh, (2017), "Finite element modeling of shear–thinning flow of inelastic non–Newtonian fluid past Expansion Pipe", Sindh Univ. Res. Jour. (Sci. Ser.), Vol.49.

Solangi, M. A. (2011), "Finite element modeling of blood flow: Relevance to atherosclerosis, PhD, Mehran University of Engineering and Technology, Jamshoro.

Solangi, M. A. H. Shaikh, R. B. Khokar, and A. Baloch. (2012), "Numerical study of Newtonian blood flow through a plaque deposited artery", Sindh University Research Journal (Science Series), vol..45(01) 79-82.

Versteeg, H. K. and W. Malalasekera, (2007), An Introduction to Computational Fluid Dynamics, finite volume method, Pearson Prentice Hall.