Sindh Univ. Res. Jour. (Sci. Ser.) Vol.50 (002) 215-220 (2018)

http://doi.org/10.26692/sujo/2018.06.0038

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Least square Galerkin Finite Element study of Newtonian Fluids Flow through channel with fixed Rectangular Single Baffle

K. DHAREJO, H. SHAIKH*, B. SHAH*, A. BALOCH**

Mphil research scholar, Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs, Sindh

Received 28th July 2017 and Revised 18th Feb 2018

Abstract: A finite element approximation through commercial computational fluid dynamics (CFD) software COMSOL MultiPhysics is conducted for the incompressible constant viscosity flow of Newtonian fluid through channel with rectangular imposed baffle. The modern code of CFD and the exponential increase of computer power, it is feasible to obtain detail description of fluid flow fields by using COMSOL MultiPhysics. The geometry was simulated below the Mathematical model which governs by the conservation of mass and conservation of momentum by using finite element method. The effect of different fluid inertia is tested to examine the behavior of flow phenomena and presence of vortex in channel with fixed baffle. The vortex structure appeared in the left and right corners of the baffle of the channel and the primary vortex in size decreased with increasing Reynolds numbers and secondary vortex is slowly and continuously enhanced in the right corner of the baffle fixed in the channel. The numerical results will be validated by comparing with the other results available in literature..

Keywords: open and closed channel flows, Finite Element Method, Porous Media.

1. INTRODUCTION

The study of the solid baffles fixed in the rectangular channels is very high importance in the field of the irrigation and hydraulic open channels and baffles creates the hydraulic jumps to flow speedily. The study of these channel consisted on the physics based governing equations in the fluid dynamics problems such as continuity equation and momentum equations. The governing equations are highly complex due to non – linearity term and their analytical solution are almost impossible to find the accurate and complete solution. Conversely chosen a numerical approach but the numerical approaches are consisted a huge number of iterative process for the converge and accurate solution. For the controlling the iterative process of numerical approach to select a commercial CFD package COMSOL MultiPhysics. The Comsol package is based upon the finite element analysis and the finite element analysis is consisted two different approaches as Petrov–Galerkin scheme and least square Galerkin scheme. Due to quick and easy process of least square residuals to choose the least square Galerkin scheme for the discretization of the governing equations like partial differential equations (Benzenine, *et al.* (2010), (Arefin, *et al.* 2012 and Louhibi, *et al.*(2014)

The COMSOL package was chosen for the transient study of the 2D generator model through finite element approach by Arefin, *et al.* (2012). Through COMSOL package, the design of the generator was developed for the purpose great efficiency, productivity and small wait to ratio of the torque. High stable results in the voltage of the generator consisted on the large number of flux line segments in the stator winding and based upon the material properties used for the stator and rotor. The magnetic and non – magnetic both materials were employed and the theme of the simulation was to analyses the most suitable material collection which created significant number of output voltage includes with nethermost harmonic. The good results about 2D generator model was developed through CFD package COMSOL by Arefin and the results was justified and also verified with the other results of the AC.DC module includes with the electronic devices like motors, generators available in literature and found approximately compatible results but limited cases are tested.

Consequently, another modeling and simulation was developed for the turbulent flows in the channels with fixed various type of baffles, like tapered and transverse by Benzenine, *et al.* (2010) and employed the CFD package Gambit and Fluent. The governing equations was computed with chosen the k–ε turbulent model and discretize the model through finite volume approach such as SIMPLEC means semi implicit pressure linked equations for correction. Different flow features was observed at various rates of the fluid inertia

kashifali@ibacc.edu.pkhisam.shaikh@salu.edu.pk, baqir.shah@salu.edu.pk, csbaloch@yahoo.com.

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

**ISRA University, Hyderabad

such as upstream, downstream and coefficient in friction factor. It was concluded that due to fixed the trapezoidal type baffles in the channel the velocity rates were increased and only demerits occurred in the enhancement of the friction coefficient.

After the huge study of the literature, the CFD package COMSOL MultiPhysics is chosen for the modeling and simulation of the Newtonian fluid flows through rectangular channel fitted the rectangular type baffle. The liquid behave analyzed through governing equation of fluid dynamics with finite element discretization. Also the various flow patterns are developed such as streamline patterns of the velocity and pressure contours on the increment in the Reynolds number and plotted the vortex intensity and vortex length graphs.

.

2. GOVERNING SYSTEM OF EQUATIONS

The well-known governing equations derived from the Newton's 2^{nd} law of motion are used like continuity and momentum equation in the Cartesian coordinates. Due to various number of physical parameters like density and viscosity involve in the governing equations so that presented the non – dimension form of the equations with produced the Reynolds number. The study is limited for the incompressible fluids like water only.

For an incompressible fluid

Firstly, the continuity equation is described as given below:

$$div\ \mathbf{u} = 0 \qquad (1)$$

This equation is used for the satisfaction of continuum fluids moving in a channel. Due to two– dimensional problem, the equation is shortened as:

$$\frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial y} = 0 \qquad (2)$$

u_1 and u_2 are two components that shows the velocity vector fields.

Secondly, the momentum equation is employed for the Newtonian fluids with laminar flow patterns that are described in Cartesian coordinates form as:

x–component

$$\frac{\partial u_1}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u_1}{\partial x^2} + \frac{\partial^2 u_1}{\partial y^2}\right) - u_1\left(\frac{\partial u_1}{\partial x} + \frac{\partial u_1}{\partial y}\right) - \frac{\partial P}{\partial x} \quad (3)$$

y –component

$$\frac{\partial u_2}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u_2}{\partial x^2} + \frac{\partial^2 u_2}{\partial y^2}\right) - u_2\left(\frac{\partial u_2}{\partial x} + \frac{\partial u_2}{\partial y}\right) - \frac{\partial P}{\partial y} \quad 4)$$

For the steady state solution of the governing equations, it is necessary to fix the boundary conditions and problem of interest is rectangular channel of solid walls and fixed the solid baffles. Therefore, separate conditions are imposed for the solid walls, inlet and outlet zones in the domain. Here fixed no–slip boundary conditions on the solid walls of the rectangular channel and as well as solid baffles and imposed the zero pressure at the outlet. Whereas, the fitting the parabolic inlet profile at the inlet of the channel that is given as under:

$$\mathrm{u} = \mathrm{v}_{\mathrm{mean}} \times \mathrm{s} \times (1 - \mathrm{s}) \qquad (5)$$

Whilst, s expresses the boundary parameter that moves from 0 to 1 and the length and height of the rectangular channel is 0.m and 0.04m. Also, the length and height of the baffles in the channel are fixed as 0.01m and 0.15m.

3. NUMERICAL SCHEME

The numerical scheme least square Galerkin finite element is adopted here for the computations of the velocity vector fields and pressure component through above given governing equations (3 and 4). Due to limitations of least square approach the 2^{nd} order governing equations are splitted into the first order system of equations. Therefore, introduce the extra parameter called as vorticity that means the cross product of the vector and divergent operator and is described as under:

$$\boldsymbol{w} = \nabla \times \boldsymbol{u} \qquad (6)$$

After substituting the vorticity (6) into the governing equations (1) and 3 and 4) , the resulting system will becomes as given below:

$$\nabla .\ \boldsymbol{u} = 0 \qquad (7a)$$

$$\frac{1}{\mathrm{Re}}\nabla \times \boldsymbol{w} - \nabla P - \boldsymbol{u}.\nabla \boldsymbol{u} = 0 \qquad (7b)$$

$$\boldsymbol{w} - \nabla \times \boldsymbol{u} = 0 \qquad (7c)$$

The problem relates the incompressible fluids flow through rectangular channel fitted solid baffle and the limited into the two–dimension is expressed as:

$$\frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial x} = 0 \qquad (8a)$$

$$u_1 \frac{\partial u_1}{\partial x} + u_2 \frac{\partial u_1}{\partial y} + \frac{\partial P}{\partial x} + \frac{1}{\mathrm{Re}}\frac{\partial w}{\partial y} = 0 \qquad (8b)$$

$$u_1 \frac{\partial u_2}{\partial x} + u_2 \frac{\partial u_2}{\partial y} + \frac{\partial P}{\partial y} - \frac{1}{\mathrm{Re}}\frac{\partial w}{\partial x} = 0 \qquad (8c)$$

$$w + \frac{\partial u_1}{\partial y} - \frac{\partial u_2}{\partial x} = 0 \qquad (8d)$$

Hence the system (8) becomes as first order linear system of equations and for the Galerkin finite element analysis, the system will transport into the matrix system as:

$$A_1 = \begin{pmatrix} 1 & 0 & 0 & 0 \\ u_1 & 0 & 1 & 0 \\ 0 & u_2 & 0 & \frac{1}{\mathrm{Re}} \\ 0 & -1 & 0 & 0 \end{pmatrix} \qquad A_2 = \begin{pmatrix} 0 & 1 & 0 & 0 \\ u_2 & 0 & 0 & -\frac{1}{\mathrm{Re}} \\ 0 & u_2 & 1 & 0 \\ 1 & 0 & 0 & 0 \end{pmatrix}$$

$$A = \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix} \qquad f = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix} \qquad \mathbf{u} = \begin{pmatrix} u_1 \\ u_2 \\ p \\ w \end{pmatrix}$$

The matrix system

$$KU = f \qquad (9)$$

where, Kshows the global matrix that will found from the Galerkin finite element then the matrix system will become as algebraic system of equations. Also need to describe the boundary conditions and remove the derivatives from the boundary conditions. Here assume Γ for the side boundaries and n assume for the normal vector and assume for tangential vector then the discrete form of the complete boundary conditions are:

For solid walls foxed the no – slip boundary conditions as $u = v = 0$, and for the inlet, parabolic velocity profile is imposed with assume vorticity and for the outlet, zero pressure fixed and velocity components will obtained. After finding the global matrix K the system will be algebraic system of equations. Therefore, will adopt the Newton's method to compute all unknowns as u, v, p and w.

4. PROBLEM DEFINITION AND MATHEMATICAL

Formulation

The problem is limited to two–dimension model of Newtonian and incompressible fluid flows through rectangular channel fitted the rectangular baffle at the lower wall. The problem have large application in many engineering and industrial fields particularly irrigation flow system, groundwater flows through open and closed channel and dam flows in multiple purpose. The length of the channel is fixed as 0.2m and height of the channel is 0.04m with length and height of the rectangular baffle is 0.01m and 0.015m displays in figure – 01. The triangular elements are used for the finite element grid and the total mesh statistics is given below:

Mesh statistics	Number of elements
total mesh elements	8357
degree of freedom	76504
mesh points	4308
boundary elements	257
vertex elements	8

The best numerical results are obtained with limited comparison with other numerical and experimental results of (Tang and Tate, 1993, Erturk, (2007), Mehdi and Mushatet, 2008 and Saleel, *et al.*, 2011). The complete detail initial and boundary conditions are described into the schematic diagram of the problem **(Fig. 1)** with suitable no – slip boundary conditions on the boundary walls, parabolic velocity profile at inlet and zero pressure fitted at the outlet. Further detail is given as below:

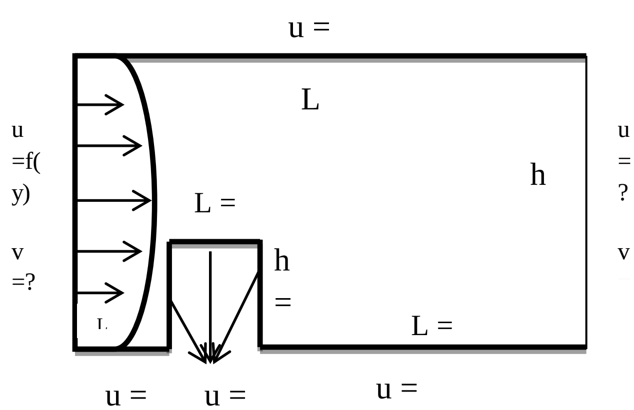


Fig.1: Schematic Diagrams of two–dimension rectangular channels filled with single baffle

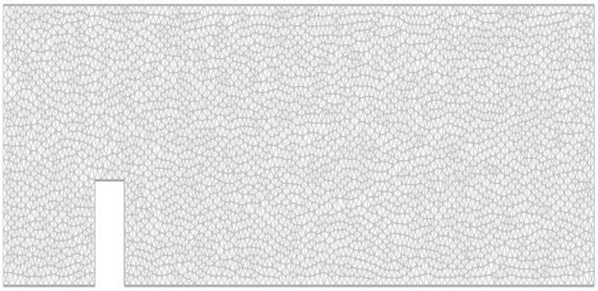


Fig. 2: Extremely fine mesh of two–dimension rectangular channel filled with single baffle

5. RESULTS AND DISCUSSION

The numerical results are based upon the CFD package COMSOL MultiPhysics and chosen here the least square Galerkin finite element approach that is very easy and quick technique to solve the 2nd order governing equations particularly partial differential equations. In COMSOL domain is chosen a rectangular channel with fitted the rectangular baffle with the effect of the fluid inertia. The extremely refined mesh is developed with chosen triangular unstructured elements. The numerical results are discussed here for development of flow structure for the velocity and pressure in the form of streamline patterns and pressure contours.

The complete domain as rectangular channel fitted the rectangular baffle at lower wall is designed with complete and well – posed initial and boundary conditions shown in **(Fig. 1)**. The finite element discretization is developed with selection of triangular elements for the purpose of extremely refined grid of the channel given ion **(Fig. 2)** Subsequently the streamline patterns of the velocity on different fluid inertia are visualized in figure – 03. Initially, the tinny eddy is observed at left and right corner of the baffle in the rectangular channel at lowest Reynolds number (Re = 01) but the eddy length in size of the right corner is higher than the left corner eddy because the direct flow hits to the left corner wall and jump to the right corner wall. Similarly, with increasing the Reynolds number (lower Re's numbers up to 100) the left corner eddy is disappeared but the right corner eddy is slowly increased and move towards horizontally. At higher Reynolds numbers (Re > 100) the primary vortex (left corner eddy) is diminished completely but the secondary vortex is largely enhanced that is clearly visualized in **(Fig. 3)** and the large eddy is observed and filled the whole right corner area of the domain at large Reynolds number (Re =1000).

The same trend of enhancement appears in the plotting of the vortex intensity and vortex length horizontally. **(Fig. 4a)** shows the graph of the vortex intensity that are the function of the Reynolds number and secondary vortex intensity is computed with different selected Reynolds numbers. The plot displays the continuum enhancement with enhancement in the Reynolds. Also, computed the secondary eddy length that observed the enhancement in streamline patterns, the **(Fig. 4b)** displays the secondary eddy length due to prove the continuum enhancement and also the plotting shows the linear enhancement with different Reynolds numbers shown in **(Fig. 4b).**

Subsequently, another flow feature known as pressure contour are developed for the rectangular channel fitted the single baffle at lower wall. In **(Fig. 5)** the pressure contour are displays at different Reynolds numbers and shows the very limited numbers of pressure difference at initial Reynolds number and with growing the pressure contours on the bases of the increasing the Reynolds numbers. Also **(Fig. 6)** shows the pressure enhancement with different Reynolds number in increasing trend and the detail of quantitative maximum and minimum values of the pressure are described in **(Table –1)**

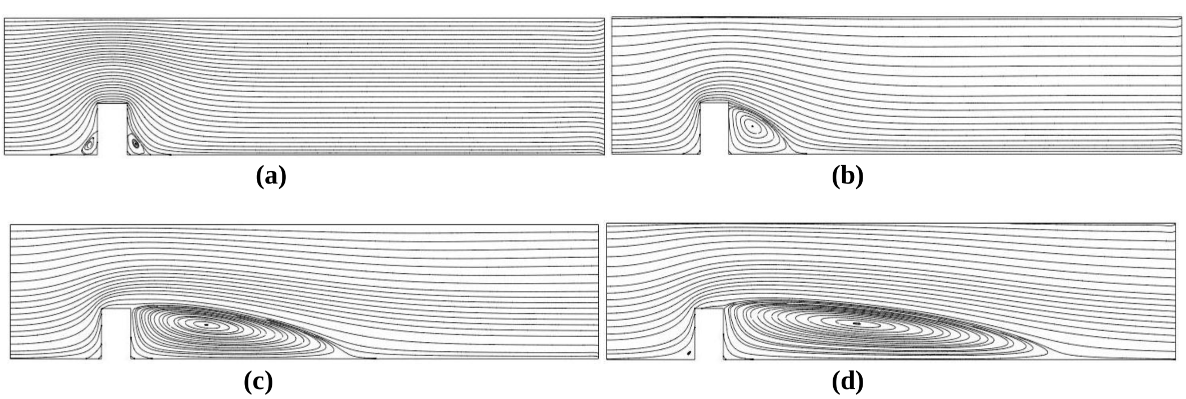
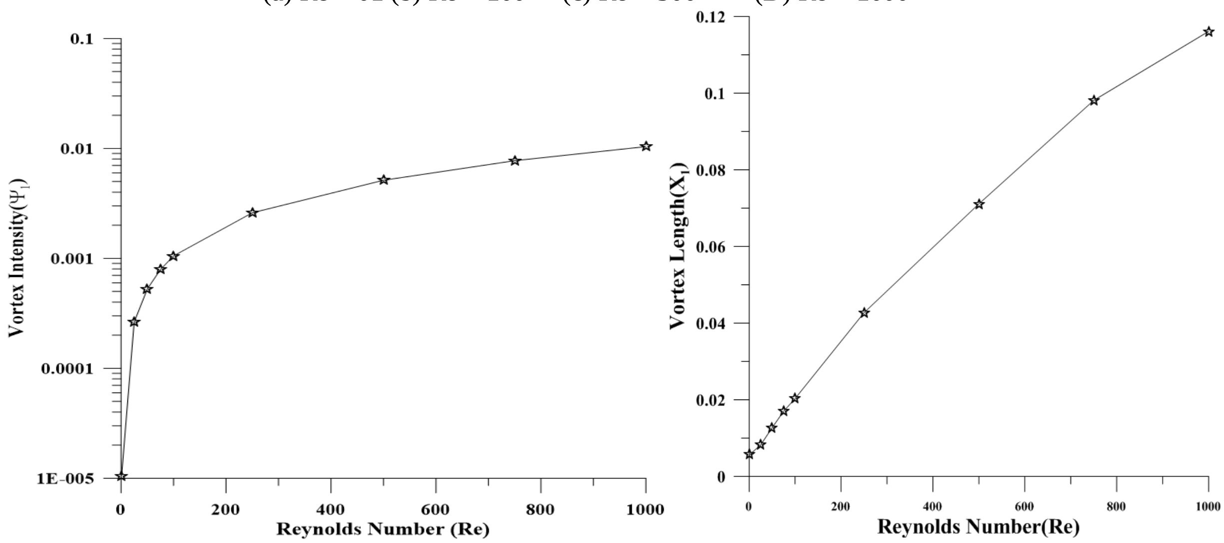


Fig. 3 (a–d):Stream lines for rectangular channel fitted single baffle at lower wall

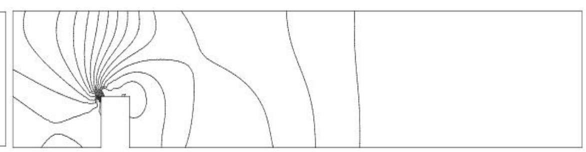
(a) Re = 01 (b) Re = 100 (c) Re = 500 (D) Re = 1000



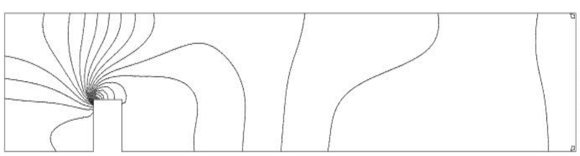
(a) **(b)**

Fig. 4(a–b): Vortex intensity and Vortex Length of two–dimension rectangular channel filled with baffles in terms of the various Reynolds Numbers .

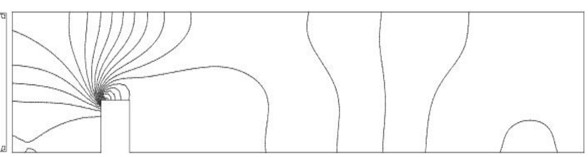
(a)



(b)



(c)



(d)

Fig. – 5(a–d): Pressure isobars of two–dimension rectangular channel filled with single baffle in the form of various Reynolds Numbers.

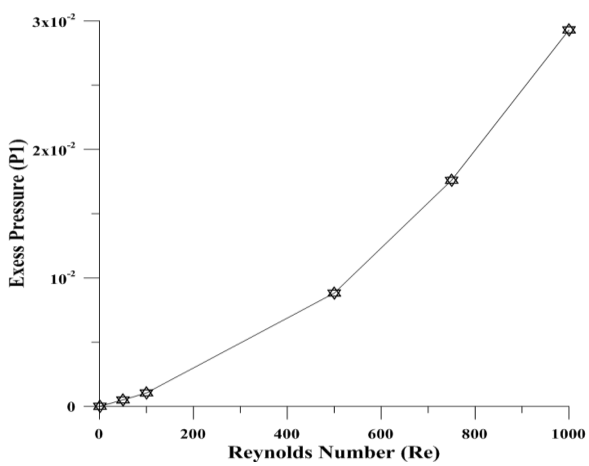


Fig. 6 Excess pressure of two–dimension rectangular channel filled with baffle in terms of the various Reynolds Numbers

Table –I: Pressure contours of 2 – Dimension rectangular channel fixed with Single baffle

	SINGLE BAFFLE	
Re	Minimum value	Maximum value
01	0.5000×10^{-5}	9.8968×10^{-6}
50	1.5896×10^{-4}	4.9262×10^{-4}
100	1.9190×10^{-4}	1.0252×10^{-3}
500	-7.4020×10^{-3}	8.4030×10^{-3}
750	-1.6000×10^{-2}	1.6800×10^{-2}
1000	-2.5600×10^{-2}	2.9300×10^{-2}

6. CONCLUSION

A numerical methodology as finite element approach with combination of least square residuals is employed to examine the flow structure of velocity and pressure. The different influence of Reynolds number ($01 \leq Re \leq 1000$) is observed to present the streamline patterns, vortex intensity and vortex length. At initial Reynolds number (= 01), the very tinny eddy is observed at the left corner of the solid baffle and secondary vortex is appeared at the right corner of the solid baffle that is larger than the left one. Due to grow in Reynolds number ($Re > 01$) the left corner eddy is disappeared due to hitting the direct flow on the wall and right corner eddy is enhanced continuously. The full large eddy is appeared in the right corner of the baffle up to thousand Reynolds number. Another flow structure is also developed as vortex intensity and vortex length on the bases of the Reynolds number and in both plotting the vortex intensity and vortex length is grows largely with increasing the Reynolds number.

ACKNOWLEDGEMENT:

The research scholar and all coauthors are highly thankful to the concerned department of Mathematics, Shah Abdul Latif University Khairpur.

REFERENCES:

Alabi, E. (2011), "Fluid Flow in Homog. and Heterog. Porous Media", Vol. 16, EJGE.

Arefin, M. (2012), " Simulation of a Time Dependent 2d Generator Model using Comsol MultiPhysics", International Journal of Advances in Engineering & Technology, Bangladesh, Vol. 4, Issue 1, 85-93.

Anderson, J. D. (2009), "An introduction of computational fluid dynamics" ISBN: 978-3-540-85055-7, http://www.springer.com.

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

Benzenine, (2010), "Numerical simulation of the dynamic turbulent flow field through a channel provided with baffles: comparative study between two models of baffles: transverse plane and trapezoidal", France, Vol. 13 No.4, 639 – 651.

Bhargava, D. S. and C. S. P. Ojha, (1993), "Models for the design of flocculating baffled channels", Journal of Water Resources, Vol. 27 (3), 465-475.

Erturk, H. (2007), "Numerical solutions of steady incompressible flow over a backward-facing step, Part I: High Reynolds number solutions" Computers & Fluids, Vol. 37, 633–655.

Khatri, R. and P. Agarwal. (2012), "Two-dimensional finite element analysis of forced convection flow and heat transfer in a laminar channel flow" ISSN: 0975-5462, International Journal of Engineering Science and Technology (IJEST), Vol. 4, No.03.

Louhibi, B. (2014), "Numerical Analysis of heat transfer in a channel Wit inclined baffles", Research Invents: International Journal of Engineering and Science, Vol. 4, Issue 8, 12-23. www.researchinventy.com.

Mehdi, S. and S Mushatet, (2008), "Simulation of Turbulent Flow and Heat Transfer Through a Duct with Baffle Plates" Journal of Engineering and Development, Vol. 12, No. 3, ISSN 1813-7822.

Qureshi, A. L., A Mahessar and A. Baloch. (2014), "Finite element model for unsteady open channel flows with sudden contraction/expansion", ISSN: 2278-1684,-ISSN: 2320-334X, IOSR Journal of Mechanical and Civil, Vol.11,10-17.

Ray, S., N. Biswas and C. P. Roy (2012), "Investigation of Newtonian Fluid Flow through a Two-dimensional Sudden Expansion and Sudden Contraction Flow Passage" International Journal of Engineering Research and Development. Vol. 1, Issue 12, 01-09.

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

Shaikh, H. (2013), "Effects of Local Inertia in the Expansion Channel Filled with Porous Media: A Finite Element Analysis", Sindh University research Journal (Science Series) Vol. 45 (2), 207-212.

Shaikh, H (2016), "Finite Element Modelling of Shear – Thinning Flow of Inelastic Non–Newtonian Fluid Past Expansion Pipe "Sindh Univ. Res. Jour. (Sci. Ser.) Vol.49 (1), 69-74.

Solangi, D. (2012), "Numerical study of Newtonian blood flow through a plaque deposited artery", Sindh University Research Journal (Science Series), Vol. 45 (01), 79-82.

Tang, T. and T. H. Tate, (1995), "Transient solutions for three-dimensional lid-driven cavity flows by a least-squares finite element method", international journal for numerical methods in fluids, Vol. 21, 413-432.

Versteeg, H. K. and W. Malalasekera, (1996), "An Introduction to Computational Fluid Dynamics: The finite Vol. Method", Longman Scientific & Technical.

Xu, L. and J. Zhu, (2014), "Simulation of Recirculating Flows", ANSYS Fluent Project Report, www.fluent.com.