

Effect of Fluid Inertia on the Liquid Flow Phenomena through a Two-Step Tapered Backward Channel

Abdul Sattar Chang

Research Scholar, Junior Elementary School Teacher (JEST), Education & Literacy Department, Government of Sindh. Corresponding Author Email: sattarchang7@gmail.com

Abdul Hameed Wahocho

Vice Principal Cadet College Khairpur, Sindh, Pakistan. abdul116778@gmail.com

Hissam Uddin Shaikh

Chairman, Mathematics Department, Shah Abdul Latif University, Khairpur, Sindh, Pakistan. hisam.shaikh@salu.edu.pk

Farzand Ali Chang

M.Phil Research Scholar, Department of Mathematics, Shah Abdul Latif University, Khairpur Mir's, Sindh.

ABSTRACT

The computational model is developed through the computational fluid dynamics (CFD) package COMSOL Multiphysics and adopts the least square residual approach with the combination of the Galerkin finite element technique. The scheme is applied to the steady state solution of the 2nd order momentum equation in Cartesian coordinates form to satisfy the continuity equations, and completes two-parameter velocity and pressure separately. Due to the least square residual approach, an extra parameter, vorticity, is assumed to decrease the 2nd order diffusion term in the momentum equation into the system of first-order governing equations. Two cases of fluid flow features are examined as Newtonian, shear thinning ($n=0.95$ and 0.90), and shear thickening ($n=1.1$ and 1.2), with and without the availability of Porous material. The flow patterns are visualized into the streamline patterns of velocity and pressure with the differences of non-dimensional Reynolds number. Also, various plots are developed for the recirculation flow rate and vortex length with the newly established empirical equations for both fluids.

The small eddies are developed into the left (primary) and right (secondary) silent corners of the cavity at lower Reynolds numbers, and as the Reynolds number increases, the primary eddy is enlarged in Newtonian fluids, very slowly enhanced in the shear thickening fluids, and greatly grows the primary vortex in the shear thinning fluids. Conversely, all eddies vanish to fill the Porous material. The finite element results are compared with the other predicted results, and a good approximation is achieved on the basis of available literature.

Keywords: Fluid Inertia, Liquid Flow Phenomena, Curved Converging Channel

Introduction

The flow phenomena through rectangular tanks is an interesting research problem due to sudden expansion and contraction geometry, and the problem is related to hydraulic engineering, such as canals, reservoirs, and tubes in rectangular channel or circular tube geometry. The numerical approaches, such as finite differences, finite volume, and finite elements, are highly applicable in every research problem related to the channels, ducts, and rectangular tube-type geometries. The detailed study was examined and investigated by Wang et al. [2014], Tinney and Ukeiley [2009], Jayaraj et al. [2013], and Mustafa et al. [2011].

Various researchers contributed to the vortex phenomena through single-step and double-step backward channel with the implementation of numerical schemes such as finite volume and finite element methods. The researchers validated that the double backward-facing step problem is a benchmark problem in the applied sciences, engineering, and various industries. The research problem is chosen here to adopt the most advanced numerical approach, i. e, Galerkin least square finite element scheme with the use of commercial computational package COMSOL MultiPhysics. The behavior of transport phenomena of Newtonian fluids is examined to visualize the flow phenomena through streamline patterns, recirculation flow rate, and vortex length in terms of fluid inertia.

Is an attempt to describe the concise orientation to the basic equations related to the Navier–Stokes equations derived from the most advanced physical law called Newton’s 2nd law of motion. Mathematical depiction of the Navier–Stokes equations is focused on the two –dimensional model for the Newtonian with

constant viscosity fluids of only laminar flows through the rectangular channel with imposed triangular obstacles at the lower and upper stream. The well-suited initial and boundary conditions are provided according to the classification of mathematical equations, particularly partial differential equations. The partial differential equations are mostly studied in physics-based real problems in the field of computational fluid dynamics (CFD), known as conservation laws of the material, like mass, momentum, and energy, if needed. These laws focus on the continuity of the fluids and the dynamics of the fluid flow phenomena. Due to the choice of the rectangular domain. The physical laws are described in the curvilinear coordinates form and deduced into the two dimensions for incompressible fluids only

MOMENTUM EQUATION

EQUATIONS OF CONTINUITY FOR LIQUID FLOWS

The equation of continuity is also known as the law of conservation of mass and describes the liquid flows so that the time derivative of the mass with respect to time. The remaining equation of continuity is defined as the volume of liquid that flows through the inlet of the body is equal to the volume of the liquid at the outlet of the body.

The equation of continuity is articulated as follows;

$$\text{grad. } u = 0 \quad (3-1)$$

Though the problem is limited to the two-dimensional, so the deduce into the following component form of the velocity as u_1 and u_2 ;

$$\frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial y} = 0 \quad (3-2)$$

EQUATION OF MOMENTUM FOR LIQUID FLOWS

The equation of momentum is also known as the law of conservation of momentum and is limited to Newtonian and incompressible fluids. The equation of momentum is consisted on the various physical terms, such as the time derivative of velocity to describe the solution as steady state or transient, the diffusion term in 2nd order to describe the movement of fluid flows in a particular direction, the convection term to describe the transport of the fluids, and the last one is the pressure term to describe the pressure horizontally and vertically.

$$\rho \frac{Du}{Dt} = \nabla P + \rho g + \mu \nabla^2 u \quad (3-3)$$

Whilst ρ stated as density of the fluids, μ stated as viscosity of the fluids, limited to the constant, g , stated as the gravitational force, and the mathematical operators like ∇ and ∇^2 stated as gradient and Laplacian that are described as under;

$$\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y} \right) \quad \text{and} \quad \nabla^2 = \left(\frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} \right) \quad (3-4)$$

The above equation, named as (3-3), must be deduced because of the Newtonian and employed for the incompressible fluids, whereas, viscosity remains as constant. The equation followed as;

$$\rho \left(\frac{\partial u}{\partial t} + (u \cdot \nabla)u \right) = \mu \nabla^2 u - \nabla p \quad (3-5)$$

NON – DIMENSION FORM OF THE GOVERNING EQUATION

For the easy and quick predicted solution, it is necessary to convert the dimension form of governing equations into the non-dimensional form. Therefore, presuming some selected variables such as velocity vector, $u^* = U/u$, displacement vector $x^* = L/x$ time variable $t^* = T/t$ and in last pressure term, $p^* = p/p$. Here used some arbitrary variable like U and T_0 , L_c shows the total specific length of the channel and u shows the velocity vector field imposed at the inlet of the channel. Consequently, the resulted equation (3-5) is formed as follows;

$$\frac{\partial u^*}{\partial t^*} + (u^* \cdot \text{grad})u^* = \frac{1}{\text{Re}} (\text{grad})^2 u^* - \text{grad } p^* \quad (3-6)$$

For the removing the physical properties of the fluids like density and viscosity is transferred into the non – dimensional number known as Reynolds number. It is stated as follows;

$$\text{Re} = \frac{\rho u L_c}{\eta} \quad (3-7)$$

The equation (3-6) is called the general form of the Navier – Stokes equation and it

is used for the Newtonian and incompressible fluids. Due to required problem as two – dimensional form of the equations in the absence of external forces like gravity. The resultant momentum equations are stated as follows in the Cartesian Coordinates form because of the research problem ,chosen as rectangular channel with imposed triangular baffles at lower and upper wall;

X-element

$$\frac{\partial u}{\partial t} + \frac{1}{\text{Re}} \left(\frac{\partial^2 u}{\partial x^2} \right) - \left(\frac{\partial u}{\partial x} \right) \frac{\partial p}{\partial x} = 0 \quad (3-8a)$$

$$\frac{\partial u}{\partial t} + \frac{1}{\text{Re}} \left(\frac{\partial^2 u}{\partial x^2} \right) - \left(\frac{\partial u}{\partial x} \right) \frac{\partial p}{\partial x} = 0$$

Y-component

$$\frac{\partial u}{\partial t} + \frac{1}{\text{Re}} \left(\frac{\partial^2 u}{\partial x^2} \right) - \left(\frac{\partial u}{\partial x} \right) \frac{\partial p}{\partial x} = 0 \quad (3-8b)$$

Whilst, a shows the dimension as (= 1, 2) and limit of the Reynolds number is fixed as $0 \leq \text{Re} \leq 2200$ because of the laminar flow patterns.

NUMERICAL SCHEME

The problem of interest to compute the steady state solution of Newtonian fluids through a rectangular channel with imposed the triangular obstacles. Therefore, discussed the approximate procedure in the Cartesian coordinates form and focused the two– dimensional problem. The procedure is followed as under;

$$\text{grad} \cdot \mathbf{u}^* = 0 \quad (3.9)$$

$$(\mathbf{u}^* \cdot \text{grad}) \mathbf{u}^* - \frac{1}{\text{Re}} (\text{grad})^2 \mathbf{u}^* + \text{grad} p^* = 0 \quad (3.10)$$

As given in the equation (3.10), double grad is specified so that is is called as higher order partial differential equation, but adopted the LSGFEM in this chapter and is limited to solve only the first order systems. Therefore, select another parameter called as vorticity and it is known as the curl of the velocity and is followed as under;

$$\psi = \nabla \times \mathbf{u} \quad (3.11)$$

$$\text{grad} \cdot \mathbf{u}^* = 0 \quad (3.12a)$$

$$(\mathbf{u}^* \cdot \text{grad}) \mathbf{u}^* - \frac{1}{\text{Re}} (\text{grad} \times \boldsymbol{\psi}) + \text{grad} p^* = 0 \quad (3.12b)$$

$$\boldsymbol{\psi} - \text{grad} \times \mathbf{u} = 0 \quad (3.12c)$$

Due to need of the research problem, the system (3.12) can be transoorted into the two – dimensional steady state and limited for the constant viscosity fluid models and flow assumed as laminar. Therefore, the system is discretised as under;

$$\frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial y} = 0 \quad (3.13a)$$

$$\frac{1}{\text{Re}} \left(\frac{\partial \psi}{\partial y} \right) + u_1 \left(\frac{\partial u_1}{\partial x} \right) + \frac{\partial P}{\partial x} = 0 \quad (3.13b)$$

$$- \frac{1}{\text{Re}} \left(\frac{\partial \psi}{\partial x} \right) + u_2 \left(\frac{\partial u_2}{\partial y} \right) + \frac{\partial P}{\partial y} = 0 \quad (3.13b)$$

$$\frac{\partial u_1}{\partial y} - \frac{\partial u_2}{\partial x} + \psi = 0 \quad (3.13c)$$

Here the system is converted into the first order system so easily convert into the matrix form as;

$$\lambda = \begin{pmatrix} 1 & 0 & 0 & 0 \\ u & 0 & 1 & 0 \\ 1 & & & \end{pmatrix} \quad \lambda = \begin{pmatrix} 0 & 1 & 0 & 0 \\ u_2 & 0 & 0 & 1 \\ & & & \end{pmatrix}$$

$$\lambda = \begin{pmatrix} 1 & 0 & u & 0 \\ 0 & 1 & & -1 \\ & & & \end{pmatrix} \quad \lambda = \begin{pmatrix} 2 & & & \\ 0 & u & 1 & 0 \\ & & & \end{pmatrix}$$

$$\lambda = \begin{pmatrix} 0 & -1 & 0 & 0 \\ & & & \end{pmatrix} \quad \lambda = \begin{pmatrix} 2 & & & \\ 1 & 0 & 0 & 0 \\ & & & \end{pmatrix}$$

$$\lambda = \begin{pmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{pmatrix} \quad f = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix} \quad u = \begin{pmatrix} u_1 \\ u_2 \\ p \\ \psi \end{pmatrix}$$

After the first order system (3.13), it is necessary to require initial and boundary condition so vanished the Neumann boundary conditions and converted into the Dirchlet boundary conditions. Also, the no-slip boundary conditions fixed at the stationary walls of the domain and due to the parabolic governing equations, fixed the parabolic velocity profile at the inlet of the domain and no pressure is fixed at the outlet of the channel and the solution must be converging at the outlet of the channel to compute the all unknowns like velocity field, pressure and vorticity. Furthermore, employed the linear combination to discretization into the finite elements as follows;

$$\psi_1 = \psi_{1,x} A_1 + \psi_{1,y} A_2 + \psi_1 A \quad (3.14)$$

And then find the stiffness matrix of each element given as under;

$$K = \begin{bmatrix} k_{11} & k_{12}+k_{21} & k_{13}+k_{31} \\ k & k & k \\ e_1 & 21 & 12 & 22 & 23 & 32 \\ k_{31}+k_{13} & k_{32}+k_{23} & k_{33} \end{bmatrix} \quad (3.15)$$

$$K_{11} = \int_{\Omega_{e1}} \mathbf{L} \psi_1^* \mathbf{L} \psi_1 d\Omega_{e1} \quad (3.16)$$

Similarly applied (3.16) for next K's to complete the stiffness matrix (3.15) and then convert the matrix into the algebraic form as, $KU = f$, After applying the least squares Galerkin finite element method, the governing equations are transported into the algebraic system of equations and followed the above given procedure of finite element method.

Now, need to solve algebraic system for the final solution of unknowns, so choose the best convergence rate Newton's method. Already familiar about the general form of the Newton's method given as under;

$$U^{n+1} = U^n + H^{-1} * U^n \quad (3.17)$$

Whilst H defined as the Jacobean matrix that is computed as follows;

$$\begin{bmatrix} \frac{\partial h_1}{\partial u_1} & \frac{\partial h_1}{\partial u_2} & \frac{\partial h_1}{\partial p} & \frac{\partial h_1}{\partial \Psi} & \frac{\partial h_1}{\partial u_m} & \frac{\partial h_1}{\partial u_{2m}} & \frac{\partial h_1}{\partial p_m} & \frac{\partial h_1}{\partial \Psi_m} \\ \frac{\partial h_2}{\partial u_1} & \frac{\partial h_2}{\partial u_2} & \frac{\partial h_2}{\partial p} & \frac{\partial h_2}{\partial \Psi} & \frac{\partial h_2}{\partial u_m} & \frac{\partial h_2}{\partial u_{2m}} & \frac{\partial h_2}{\partial p_m} & \frac{\partial h_2}{\partial \Psi_m} \\ \frac{\partial h_3}{\partial u_1} & \frac{\partial h_3}{\partial u_2} & \frac{\partial h_3}{\partial p} & \frac{\partial h_3}{\partial \Psi} & \frac{\partial h_3}{\partial u_m} & \frac{\partial h_3}{\partial u_{2m}} & \frac{\partial h_3}{\partial p_m} & \frac{\partial h_3}{\partial \Psi_m} \end{bmatrix}$$

The lowest size shows the maximum accuracy. The all results of FE scheme is compared with the other predicted results of Shima, [2016] and Tinney and Ukeily, [2009].

RESULT

Effect of Reynolds Number on streamline flow patterns of velocity

These are called streamline flow patterns of velocity for the double step backward step channel at very lowest Re's no any primary and secondary eddy is observed at the corner of the step below 300. The secondary eddy initiated at $Re = 300$.

These are same streamline flow patterns at higher Reynolds numbers. Here also, initiated the Primary eddy in the start of the $Re = 600$ at the silent corner of the step and very slowly increased in size but the secondary eddy is fastly enhanced in size gradually up to $Re = 900$ due to complete fluid transport into the silent corner of the 2nd step of the channel.

In these flow patterns, the primary eddy is enhanced and remained same up to large Re's ($=1500$). The Secondary vortex is highly enhanced and filled the whole region of silent corner of the 2nd step of the backward channel.

This graphs shows the recirculation flow rate and expressed that the rate is increased non- linearly due to increase Reynolds number that enhancement is already observed in the streamline patterns

This graphs shows the length of the primary and secondary vortex that validated that the primary and secondary vortex is increased due to Reynolds number grows. Also, observed that the secondary vortex is higher than the primary in length that is also visualized in the streamline patterns

This is my contribution in research to develop the new empirical equation based upon the recirculation flow rate on various Re's

Velocity

$$X = 1.0023541E - 7Re + 4.662988E - 4$$

Pressure

$$X = 8.04822E - 8Re + 2.970373E - 4$$

Figure List

Figure – 4.1(a and b): Schematic diagram of rectangular duct with imposed semi-circular

Figure-4.2: Extremely fine mesh of rectangular duct with imposed semi-circular obstacles.

Figure-4.3 (a-d): Streamlines flow through of rectangular duct with imposed semi-circular obstacles with Reynolds number from 1 to 400.

Figure-4.3 (e-h): Streamlines flow through of rectangular duct with imposed semi-circular obstacles with Reynolds number from 600 to 1200

Figure-4.4: Recirculation flow rate is the function of Reynolds number for the rectangular duct with imposed single semi-circular obstacles

Figure-4.5: Vortex length is the function of Reynolds Number (Re) for the rectangular duct with imposed single semi-circular obstacles

Figure-4.6 (a-c): Streamlines flow through of rectangular duct with imposed more semi circular obstacles with Reynolds number from 1 to 200

Figure-4.6 (d-f): Streamlines flow through of rectangular duct with imposed more semi circular obstacles with Reynolds number from 400 to 800

Figure-4.7: Recirculation flow rate is the function of Reynolds number for the rectangular duct with imposed more semi-circular obstacles

Figure-4.8: Vortex length is the function of Reynolds Number (Re) for the rectangular duct with imposed more semi-circular obstacles.

Table 4.9: Maximum and Minimum Recirculation Flow rate (Γ) is a function of Reynolds numbers (Re) for single and more obstacles imposed in a channel

Table 4.10: Maximum and Minimum values of vortex lengths (X) is a function of Reynolds numbers (Re) for single and more obstacles imposed in a channel.

$$u = f(y)$$

$$v = 0$$

$$P = P_{max}$$

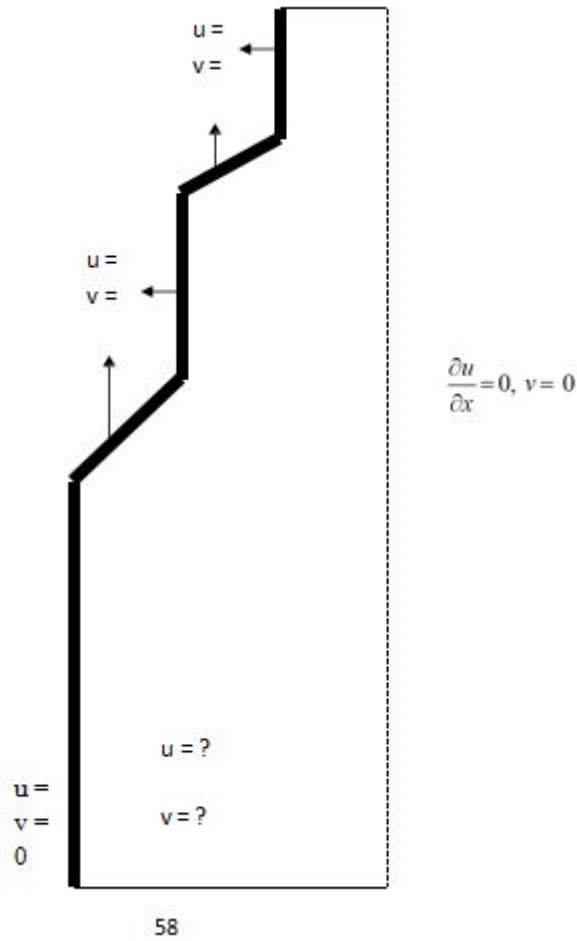
Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>



Online ISSN

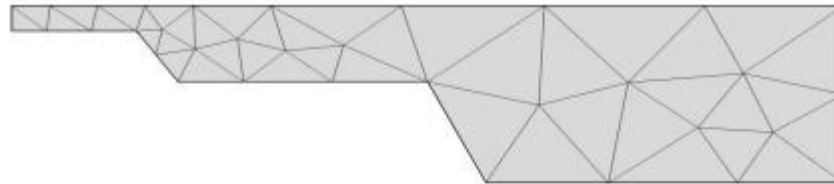
3007-3197

Print ISSN

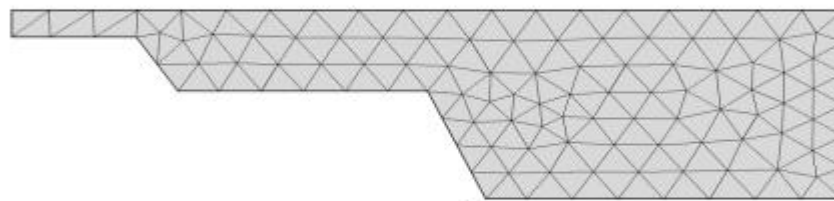
3007-3189

<http://amresearchreview.com/index.php/Journal/about>

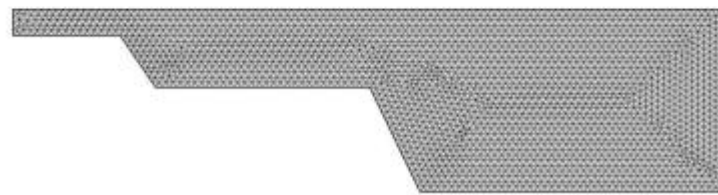
Feature – S#1: ***Graphic Structure of tapered double step backward channel.***



(a)



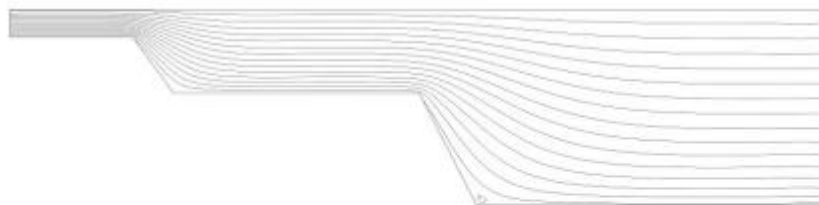
(b)



(c)

Feature – S#2: ***Extra corser, fine and extremely fine meshes of tapered double step backward channel.***

(i)



(ii)



(iii)

Online ISSN

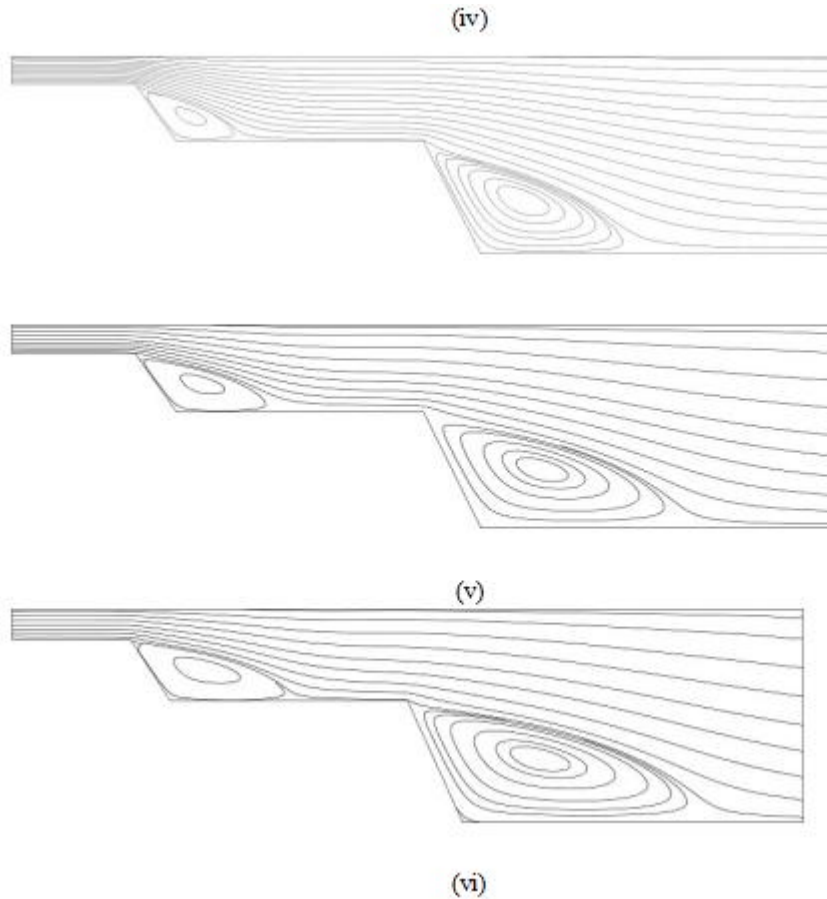
3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>

Feature – S# 3(i-iii):Streamline of transported fluids of tapered double step backward Channel initiated from Re 01 to 300.



Online ISSN

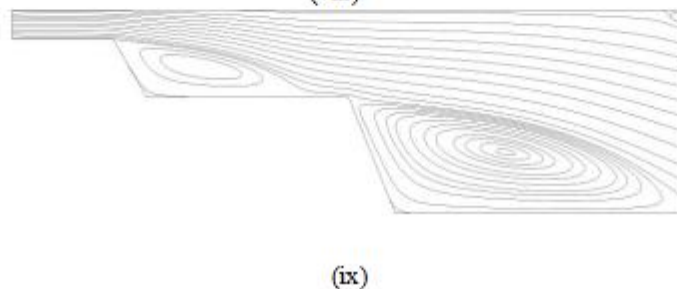
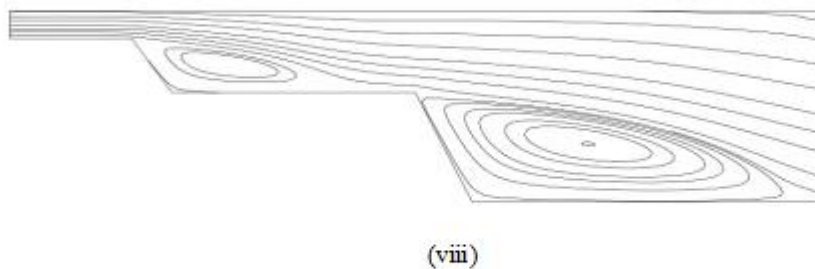
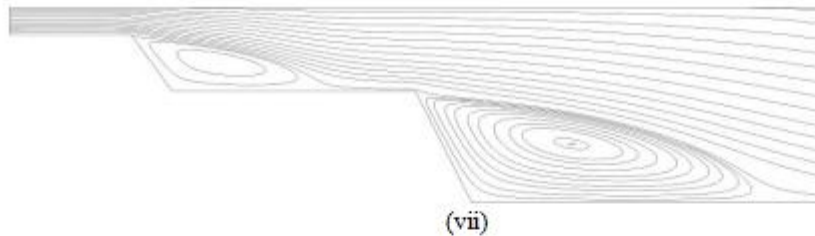
3007-3197

Print ISSN

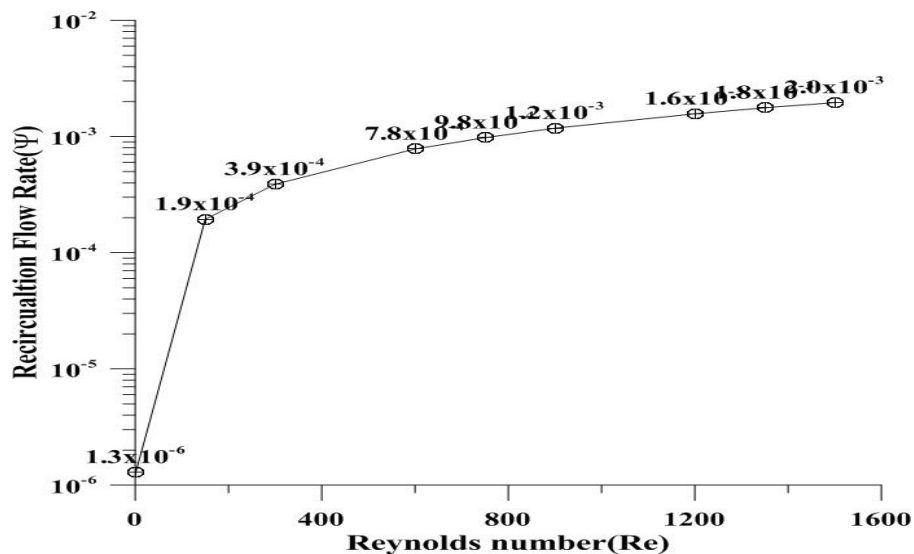
3007-3189

<http://amresearchreview.com/index.php/Journal/about>

Feature – S# 3(iv-vi): Streamline of transported fluids of tapered double step backward Channel initiated from Re 600 to 1000.



Feature – S# 3(vii-ix): Streamline of transported fluids of tapered double step backward channel initiated from Re = 1200 to 1500.



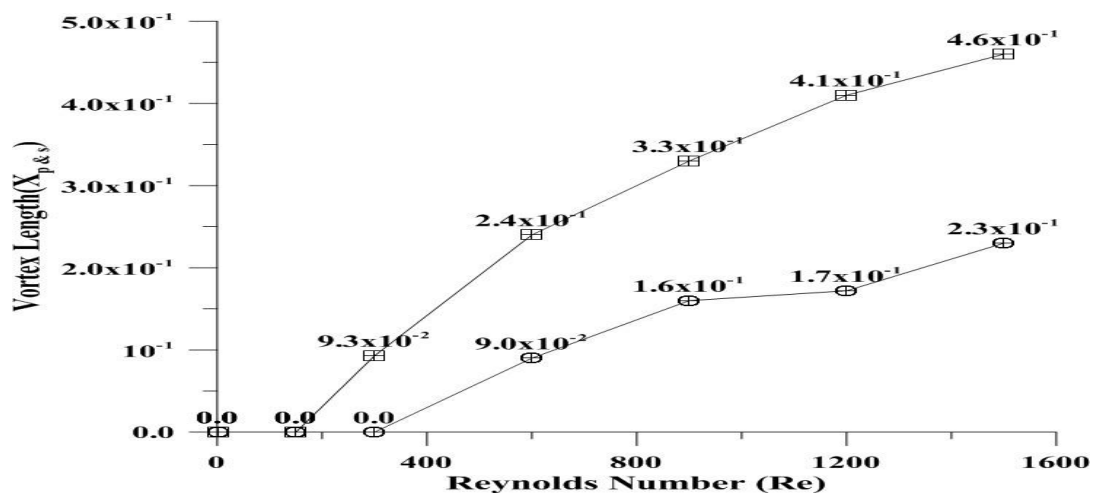
Online ISSN

3007-3197

Print ISSN

3007-3189

<http://amresearchreview.com/index.php/Journal/about>



Feature – S# 5: Primary and Secondary vortex length is the function of Reynolds number

Table – S#6: Maximum and Minimum Recirculation Flow rate () is a function of Reynolds numbers (Re) with two step angles.

RECIRCULATION FLOW RATE ()		
Reynolds number(Re)	VELOCITY	
	(1 min)	(1 max)
01	1.00e-11	1.29e-06
150	6.36e-09	1.94e-04
300 Cr	3.29e-07	3.89e-04
600	1.56e-07	7.82e-04
750	1.57e-07	9.78e-04
900	1.42e-07	1.18e-03
1200	7.00e-08	1.57e-03
1350	4.33e-09	1.77e-03
1500	4.76e-09	1.96e-03

Conclusion

The all objectives are fully obtained. The best and quick numerical method is developed known as Galerkin Least square finite element method. For the transport of the 2nd order equations into first order system of PDE's to introduce the extra variable Vorticity. The predicted results are compared with the other flow patterns obtained through various numerical schemes. Two parameters are discussed as

velocity flow patterns and pressure isobars with increasing Re 's. The effect of Re 's on the flow features for velocity and pressure are visualized. Various Re 's ranging from 01 to 1500 are discussed. It was observed that vortex phenomena at each silent corner of the expansion plane at each step. At $Re = 01$, the very small vortex observed at each silent corner in the expansion channel and due to grow Re 's, the primary vortex is slowly enhanced but the secondary vortex is fastly enhanced due to large expansion part of the channel. Due to increase the Re 's, the linear pressure isobars are increased moved towards non – linear patterns. The enhancement observed in streamline patterns that is also visualized in the plots of the recirculation flow patterns, vortex length and excess pressure drop. Newly developed the empirical equations on the recirculation flow patterns with increasing Re 's.

Acknowledgement:

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

References:

- Baloch, Z. A. K., Baloch, M. E., Shaikh, H. and Shar, M. A., (2013), “ Computer Simulation of Lid Driven shallow cavity flows: Effect of fluid inertia and Aspects Ratios”, Sindh Univ. Res. Jour.(Sci. Ser). Vol.45 (2) 277-280.
- Beaudoin, J. F., Cadot, O., Aider, J. L. and Wesfreid, J. E. (2004), “Three- dimensional stationary flow over a backward-facing step”, pp 147-155, European Journal of Mechanics B/Fluids, Elsevier.
- Biswas, G., Breuer, M., Durst, F., (2004), “Backward-Facing Step Flows for Various Expansion Ratios at Low and Moderate Reynolds Number”, pp 362- 374, Vol- 126, Journal of Fluids Engineering.
- Chiang, T. P. and Sheua. T. W. H., (1998), “A numerical revisit of backward- facing step flow problem” , v- 11, Department of Naval Architecture and Ocean Engineering, National Taiwan University, 73 Chou-Shan Rd., Taipei, Taiwan, Republic of China.
- Eesa, M., (2008) , Cfd study of Complex fluid Flows in pipes, Ph.D. thesis University of Birmingham
- Jayaraj, S., Shaija, A., and Saleel, C.A., (2013), “on Simulation of Backward Facing

- Step Flow Using Immersed Boundary Method”, pp 164-174, Mechanical engineering department, National Institute of Technology, Calicut, India.
- Jongsoo, H. A., Shigeru, O., And Yasuaki, K., (2009), “Drag Characteristics of a Pickup Truck according to the Bed Geometry”, pp 122-127, Institute of Fluid Science, Tohoku University 2-1-1, Katahira, Aoba-Ku, Sendai, Japan.
- Kopera, M. A., Kerr, R. M., Blackburn, H. M., and Barkley, D., (2014), “Direct numerical simulation of turbulent flow over a backward-facing step”, pp 1-23, Journal of Fluid Mechanics.
- Mohammad, A. H., Md, T. R., Shakerur, R., (2013), “Numerical Investigation of Fluid flow through a 2D backward facing step channel”, pp 3700-3708, Department of Mechanical Engineering, University of Texas at El Paso El Paso, TX, USA.
- Mustafa, A. W., (2011), “ Numerical study of separated flows in backward facing step by using collocated grid Arrangement”, pp 1-9, vol-18, Tikrit Journal of Engineering Sciences
- Poole, R. J., Escudier, M. P., (2002) , “Turbulent flow of non-Newtonian liquids over a backward-facing step,” Part I. A thixotropic and shear-thinning liquid.” pp 177-191, Journal of Non-Newtonian Fluid Mechanics
- Pozrikidis, C., (2009), Fluid Dynamics: Theory, Computation, and Numerical Simulation, Second Edition. Springer.
- Qureshi, A. L., Mahesar, A. A., baloch, A., “Finite element model for unsteady channel flows with sudden contraction/expansion,” Volume 11, Issue 3 Ver.V, IOSR Journal of mechanical and Civil Engineering(IOSR-JMCE).Saleel, C. A., Shaija, A., and Jayaraj, S., (2011), “ Numerical Simulation of fluid flow over a forward-backward facing steps using immersed boundary method”, pp 7714-7729 vol-3, International Journal of Engineering Science and Technology
- Selvadurai, A. P. S., (2013), Transport Phenomena in porous Media, Springer-Verlag Berlin Heidelberg
- Wang, S., Burton, D., Sheridan, J. and Thompson, M.C. (2014), “Characteristics of Flow over a Double Backward-Facing Step”, pp8-11, 19th Australasian Fluid Mechanics Conference Melbourne, Australia.