

Online ISSN

3007-3197

Print ISSN

3007-3189

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

Effect of sector-shaped obstacle with 90 degree angle on incompressible fluid transported through one inlet and two outlets

Akhtiar Ali Mirbahar

Assistant Professor Mathematics, Government Degree College, Pano Akil,
Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs, Sindh.
akhtiararab1981@gmail.com

Rafique Ahmed Soomro

Associate Professor Government Islamia Science College Sukkur.
rafiquesoomro05@gmail.com

Hisamuddin Shaikh

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

Bakhtiar Ali Mirbahar

Lecturer Zoology Govt. Degree College, Pano Akil. bakhtiar.m.arab@gmail.com

Muhammad Bachal Korai

Lecturer Chemistry, Government Degree College, Pano Akil.
m.bachalkorai@gmail.com

Asif Ali Lashari

Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs, Sindh.
asimazad98@salu.edu.pk

ABSTRACT

Finite element simulation is adopted here with the selection of least square approach for the complete discretization of basic governing equations for the transportation of incompressible fluids and then compute the fluids parameters as velocity and pressure for the steady state solution through Newton's technique. It is shown that the flow pattern is changed to two outlets of a channel through the sector-shaped obstacle. The flow features consisted on the different streamline patterns of velocity, reattachment length, vortex intensity and as well as pressure isobars. Two types of impacts on flow features are described her, either change in the fluid inertia and sector type angle. The small primary eddy is developed at right corner of the obstacle on the small Reynolds numbers and by enhancing the Reynolds number, the primary eddy is fully enhanced and filled the whole region. And no any secondary eddy is developed at lower Re's and at large Reynolds number the slowly grows towards the outlets of the channel. Due to

decrease the angle of sectors, the eddies are vanished and primary (PV) and Secondary Vortex (SV) are developed at large critical numbers. Other flow phenomenon's are also plotted like vortex intensity and computed the PV length and established newly empirical equations on the bases of vortex intensity with Various Reynolds numbers (Re). The obtained results are compared with the other results that were predicted (flow patterns) with the Shamloo, et al. [2008] and Jothiappan, et al. [2015].

Keywords— Incompressible Fluid Flow, Finite Element Simulation, Sector-Shaped Obstacle, Reynolds Number, Streamline Patterns, Vortex Intensity, Pressure Distribution, Newton's Technique, Fluid Dynamics, Reattachment Length

Introduction

After 19th century, applied sciences, industries and engineering had made computer simulation as an integral part & with help of this provide an environment that the real life Physics problems are modified into a related picture. Important things are that we require high rate of convergence and the compatible accuracy of results that are estimated for physical problems. It would be better to use any computer package when working for numerical solution of real life physical problems and then illustrate the material in pictorial forms. For instance, COMSOL MultiPhysics is a computational tool that serves as a unique plate form which is specialized for solving problem through numerical and Mathematical calculation. The COMSOL MultiPhysics solve the problems related to fluid dynamic and it uses to equation like momentum, energy and continuity equation, these only transfer heat and mass through any domain.it uses the finite element model which solve fluid problems.

Likewise it has various applications in fields like; Bioscience, irrigation, engineering etc., and it solves the problems very easily. It's specially solves problems of Newtonian and Non- Newtonian fluids. We are using finite element model and obtained the pattern of velocity and pressure with help of least square Galerkin approach.

Governing System of Equations

The basic equations of fluid; continuity and momentum equations are derived from Newton's second law of motions. The governing equations converted into non-dimensions form which produced the Reynolds number and various physical parameters such as viscosity, density and pressure which is present in basic governing equations.

For an incompressible fluid

Initially, the fluid –conservation equation is described as given below

$div(\mathbf{u}_1) = 0$ (1) This equation is employed for the fulfillment of continuous fluid moving in a duct, the equation is simplified as:

$$\frac{\partial(u_1)}{\partial x_1} + \frac{\partial(u_2)}{\partial y_1} = 0$$

(2)

The velocity vector fields are shown by two components u_1 & u_2 . Next, the Newtonian

fluids with laminar flow pattern that are illustrated in Cartesian coordinates use the momentum equation form as:

X – Direction

$$\frac{\partial u_1}{\partial t_e} = \frac{1}{\text{Re}} \left(\frac{\partial^2 u_1}{\partial x_1^2} + \frac{\partial^2 u_1}{\partial y_1^2} \right) - u_1 \left(\frac{\partial u_1}{\partial x_1} + \frac{\partial u_1}{\partial y_1} \right) - \frac{\partial P_1}{\partial x_1} \quad (3)$$

Y – Direction

$$\frac{\partial u_2}{\partial t_e} = \frac{1}{\text{Re}} \left(\frac{\partial^2 u_2}{\partial x_1^2} + \frac{\partial^2 u_2}{\partial y_1^2} \right) - u_2 \left(\frac{\partial u_2}{\partial x_1} + \frac{\partial u_2}{\partial y_1} \right) - \frac{\partial P_1}{\partial y_1} \quad (4)$$

$$u_1 = v_{*(\text{mean})} \times (s - s^2) \quad (5)$$

With the help of the governing equations; continuity and momentum equations with boundary condition of the T-type channel fixed with sector type obstacle present in the center of the T-type channel with two outlets, one is primary outlet (PV) and other is secondary outlet (SV). At the inlet of the channel velocity profile is fixed. No-slip boundary condition applied at the rigid wall of the T-shaped channel, at secondary and Primary outlet pressure is zero and length along height of the T-type channel is 0.01m and 0.04m. and 0.07 is the highest triangular element's size. After analysis, the good results of numerical technique are obtained; they are comparing with the ID analytical

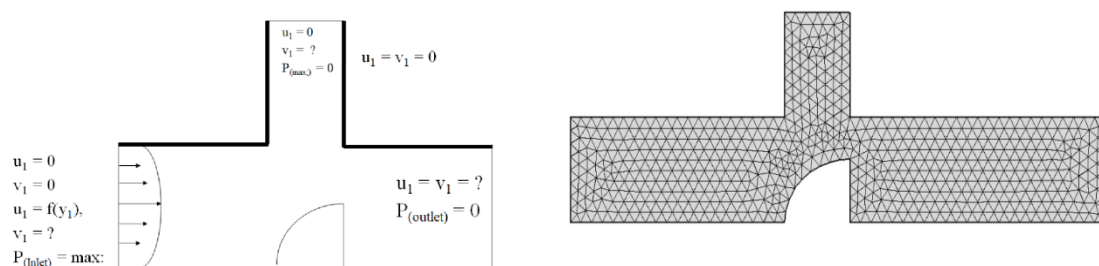


Figure –1: Above diagram represents T-shaped channel in which, sector shaped blockage fixed at the center.

PROBLEM SPECIFICATION

We have inserted a sector at varied angles i-e; 45° & 90° , by which laminar flow crosses through the sector obstacle with solid of sector and T-type channel. Neumann and Dirchlet boundary conditions are applied at T-type channel with parabolic profile is fixed at inlet of the channel. Length and height of the channel has described in Schematic Diagram, fig-01. The sector type obstacle is fixed at the lower wall, as well as having two outlets such that primary and secondary.

$$u_1 = f(y_1) = U_{(\text{max})} \times (y - y^2)$$

Multiple triangular elements are used which covered whole domain have an amount of

Online ISSN

3007-3197

Print ISSN

3007-3189

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

4044, it has minimum size 0.000015 which shows more accuracy and maximum size of triangular elements is 0.007. It gives us a perfect result after the use of finite element model and compare numerical and analytical obtained results with Shaikh[2016], also compare with solution of Jothiappan, et al[2025] and Shamloo,[2008].

Figure –2: T-type channel applied with finite element triangular mesh.

NUMERICAL PREDICTIONS AND DELIBERATIONS

COMSOL MultiPhysics is a package where we use a model known as least square approach to measure, numerical predictions are completely deals on the finite element. We have used case of 90 degree angle of sector obstacle in T-type channel, obtained results are discussed below:

IMPACT OF DIFFERENT REYNOLDS NUMBERS ON THE T-TYPE CHANNEL FIXED WITH SECTOR OBSTACLE AT 90°

Varied range of Reynolds number ($01 \leq Re \leq 2000$), is used with sector shaped obstacle, which shows stream line patterns of velocity at 90 degree angle. The stream line pattern of the velocity with angle 90 degree is displaced in fig-(a), fig-(b), fig-(c) and fig-(d). Results are obtained by the range of Reynolds number $Re=(01 \text{ to } 2000)$, are computed in the stream line pattern of the velocity. The two eddies are developed primary and secondary because of change in Reynolds numbers as the Reynolds number are increased, primary eddy appear at the right corner of the obstacle and with increase Re 's it slide's towards the primary outlet. Secondary eddy start at the left of the secondary outlet with $Re=900$ and increases towards the secondary outlet with $Re=2000$, with the moves to the direction of the first outlet with a large Reynolds number and occupied the whole silent corner.

Table – 1: Maximum and Minimum Recirculation Flow pattern (Ψ_1) at 90 degree T-type channel fixed with sector obstacle.

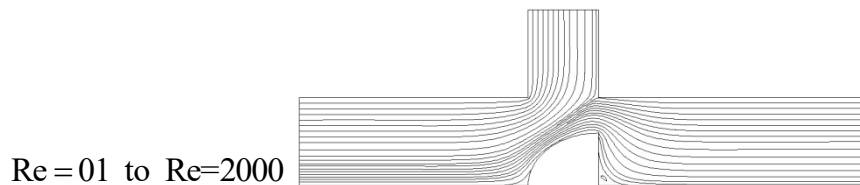
Rotational flow rate (Ψ_1)		
Re	Obstacle's Angle (90°)	
	($\Psi_{1 \text{ min:}}$)	($\Psi_{1 \text{ max:}}$)
01 (P.V)	3.27e-12	5.14e-7
200	3.29e-9	2.03e-4
500	1.32e-8	2.64e-4
800	8.66e-10	3.00e-4
890 (S.V)	3.77e-9	4.25e-4
1000	1.30e-8	4.75e-4
1100	1.88e-8	5.63e-4
1400	9.32e-9	6.82e-4
1700	2.76e-9	8.22e-4
2000	2.93e-9	10.66e-4

$$X_1 = 1.318e-5 \text{ Re} - 1.276e-5$$

Sector type obstacle at 90° in Domain

$$X_1 = 1.3456e-6 \text{ Re} - 10.8372e-7$$

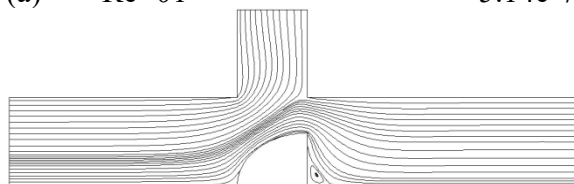
$$\text{Re} = \frac{\rho \times U_{(\text{max:})} \times L_{(\text{t})}}{\mu}$$



Re = 01 to Re = 2000

(a) Re = 01

$$R_{*f} = 5.14e-7$$



(b) Re = 500

$$R_{*f} = 2.64e-4$$

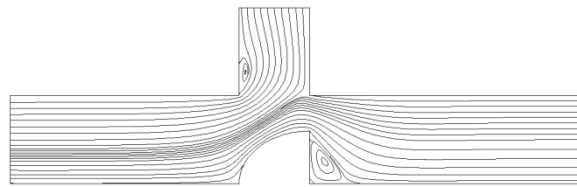
Online ISSN

3007-3197

Print ISSN

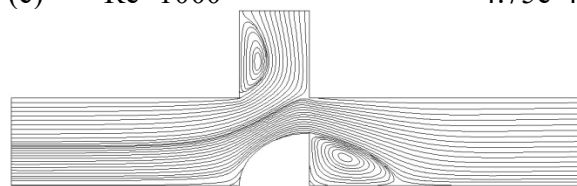
3007-3189

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



(c) Re=1000

$$R_{*f} = 4.75e-4$$



(d) Re=2000

$$R_{*f} = 10.66e-4$$

Fig: (a – d)- Velocity magnitude with 90 degree at different Reynolds numbers (Re=01 to Re=2000)

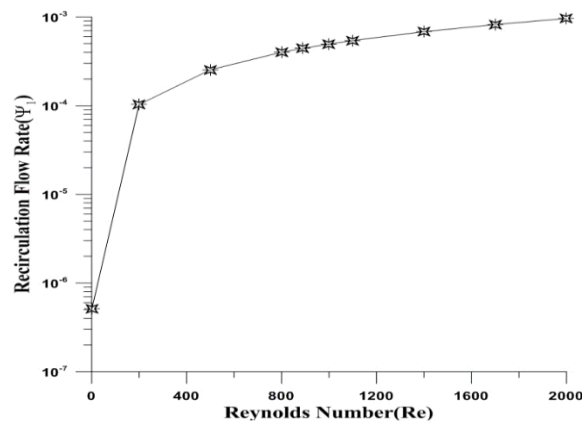


Figure –3: Recirculation Flow Rate (Ψ_1) is a function of (Re) with angle 90° .

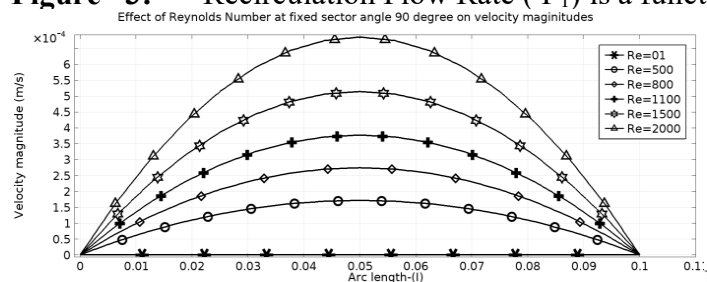


Figure –4: Effect of Different Reynolds numbers at outlet –I, fixed with sector type obstacle

Online ISSN

3007-3197

Print ISSN

3007-3189

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

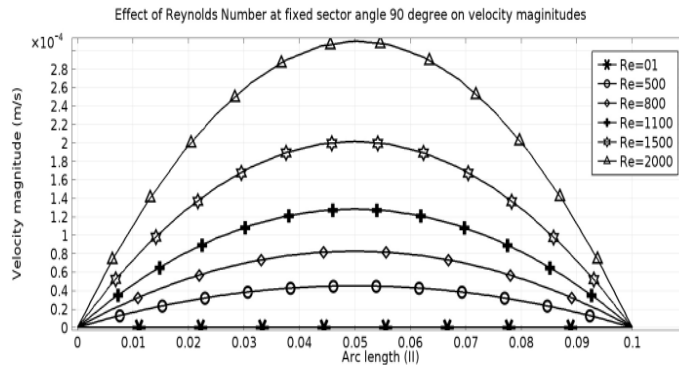


Figure –5: Effect of Reynolds numbers at outlet-II with 90 degree sector blockages at center of the domain

Conclusion

All the results are similar to the objectives discussed earlier with help of Galerkin least approach, design the finite element model. Second order of basic is modified into first ordered governing equations. This alteration is governing equation uses VORTICITY. Finite element method is used to obtained the results, the domain is fixed with a sector and different Reynolds numbers. Sector type obstacle fixed in domain and angle of 90 degree so, the primary and secondary vortex initiates. The condition for appearance of primary vortex is ($Re=10$) and secondary vortex is ($Re=215$). The increase in Reynolds numbers ($Re=2000$) so, the secondary vortex appears in large size. In obtain result , the increase Reynolds number $Re=2000$ so the primary and secondary vortex covered the whole area of the domain.

References

1. **Afshar** H, Hoseini S. H., (2013), “Experimental and 3-D Numerical Simulation of Flow over a Rectangular Broad-Crested Weir”, International Journal of Engineering and Advanced Technology (IJEAT), Volume-2, Issue-6, pp- 2249 – 8958.
2. **Baloch**, A, (1994), Numerical Simulation of Complex Flows of Non-Newtonian Fluids, Ph.D. Thesis, University of Wales, Swansea, UK.
3. **Cehidro** O. B, Guillen S., Cardoso A.H., (2014), Taylor & Francis Group “Three dimensional flow field at confluent fixed-bed open channels”, River Flow–Schleiss., London, pp-1-138.
4. **Gosselin** D., Berthier J., Chaussy D., and Belgacem N., (2015). “Capillary Flows: Dynamics and Geometry Effects”, Excerpt from the Proceedings of the 2015 COMSOL Conference in Grenoble.
5. **Goudarzizadeh** R., Jahromi S.H.M, and Hedayat N., (2010), “Simulation of 3D Flow using Numerical Model at Open-channel Confluences”, International Journal of Mathematical, Computational, Physical, Electrical Computer Engineering Vol:4, No:11.

Online ISSN

3007-3197

Print ISSN

3007-3189

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

6. **Islam Z. Md.**, and Tsui Y.Y., (2016), “Quasi-3D Modeling and Efficient Simulation of Laminar Flows in Microfluidic Devices”, Sensors, 6.
7. **Jothiappan P.**, (2015), Jothiappan, J Phys Math ,“Control of Laminar Fluid Flow and Heat Transfer in a Planar T-Channel with Rotating Obstacle”.
8. **Loth F.**, Paul F. F., and Bassiouny H. S., (2007), “Blood Flow in End-to-Side Anastomoses”. Advanced Scientific Computing Research, Department of Energy, US.
9. **Memon, R. A.**, Baloch, A., Shaikh, H., Farooq. A. and Sajjad, H., “Numerical Simulation of Rotating Mixing in Cylindrical Vessel with Couple of Mix Rotating Stirrers”, J. Basic. Appl. Sci. Res., 4(4)169-173.
10. **Mohammadiun S.**, Neyshabouris S.A.A., Naser G., Vahabi H. P.H., (2015), “Effects Of Open-Channel Geometry On Flow Pattern In A 90 Junction”, IJST, Transactions of Civil Engineering, Vol. 39, pp 559-573.
11. **MacCormick I. J. C.** , Beare N.A. V. B., Taylor T.E. T., Barrera V., White V.A., Baljean M. D., and Harding S.P., (2014), “Cerebral malaria in children: using the retina to study the brain”, Brain 2014: pp- 2119–2142.
12. **Mussawi W. H. A.**, Shammery M. H. A., Alwan H. H., (2009),“Three-dimensional numerical investigation of flow at 90° open channel junction” Journal of Scientific. Kerbala University , Vol. 7 No.4 .
13. **Rudyak V.**, and Minakov A., (2014), “Modeling and Optimization of Y-Type Micromixers”, ISSN 2072-666X.
14. **Ready, J. N.**, (2006), An introduction to the finite element methods, McGraw Hill Publishing.
15. **Shaikh, H.**, Shah, S.B., Solangi, M.A. and Baloch, A., (2012).,“A Computer Simulation of Flow of Newtonian Fluid through Backward Step Channel, Sindh Univ. Res. Jour. (Sci. Ser.) Vol.44 (4), pp - 703-708.
16. **Shaikh, H.**, SB., Solangi, M.A. Baloch, A., “Effects of Local Inertia in the Expansion Channel Filled with Porous Media: A Finite Element Analysis”, Sindh Univ. Res. Jour. (Sci. Ser), Vol. 45 (02) 207-212.
17. **Shaikh, H.** (2016), Numerical Simulation of Expansion and Contraction Flow in Channel Filled with Porous Media, Ph.D. Thesis, Shah Abdul Latif University Khairpur.