ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 4, August 2016

# Experimental Techniques to Control Flow Structure in Sudden Expansion Geometry

[1]K.Ashokachary, [2]S.Manikanthachary, [3]V.Rajarao
[1] Department of Mechanical Engineering, Vijaya Engineering College, Khammam – 507305,
[2]Department of Mechanical Engineering, Laqshya institute of science and technology, Khammam – 507305,
[3]Department of Mechanical Engineering, Jayamukhi institute of technological sciences, Narsampet, Warangal - 506332,
[1]ashokacharyideal@gmail.com, [2]manisajjanapu@gmail.com, [3]rajaraovanguri@gmail.com

***Abstract-*** **Reduction in recirculation zone size in suddenly expanded geometries is one of the major issues to deal with. After the flow encountering the sudden expansion geometry it gets many complex features like separation, recirculation, attachment and reverse flow. In the present study, flow over such geometries with blowing and suction scenario are studied for different$L_{max}/L_{min}$models namely 3, 4 and 5.It is found that for $L_{max}/L_{min}$= 5 the re-circulation and loss zones are more, for further studying $L_{max}/L_{min}$= 5 is subjected to different operating conditions like with blowing and suction. For further improvement the blowing is studied at different heads of pressure like 200 mm and 300 mm of water. With suction flow scenario is completely changed and attachment point is observed at the base of the model itself. Results thus identify the improved flow structure.**

***Keywords:*** **sudden expansion geometries, flow visualization, open channel, experimentation.**

## I. INTRODUCTION

Fluid flow past obstacles like steps is an interesting problem with wide range of applications. Flow over steps plays a dominant role in many practical situations. Most of the sequences involving flow past over various structures regard for the analysis of characteristics such as flow pattern, pressure distribution at various points etc. In order to meet the requirements of the working environment with accuracy and efficiency, most of these obstacles are comprised of cubical or cylindrical geometries. Hence researchers are showing much interest in those areas involving flow over such objects. Out of the geometries involving cubical structures, flow past steps has been proved as an interesting problem with wide range of practical applications. In the open literature, various geometric parameters for this study are hardly available. Though there is large amount of data both experimental and theoretical available in the literature, they are applicable mostly for high-speed flows. Only limited information is available for the case of low subsonic flow past steps. Even the available information was confined to only steps with faired entrance. The subsonic flow past the steps is an important area which needs further investigation. Main reason to study flow over stepped pattern is that it is being used most frequently than any of the other models. There are number of practical applications for a stepped pattern. Though every application may not be an exact replica of the pattern, there may be slight modifications, which have their own specific considerations. For example the most real time application is the body shape of any automobile (cars). A closer view at its shape gives an idea that it has been derived from the stepped pattern. The body shape of a car combines both forward facing step and backward facing step. However for drag reduction and practical utility point of view, various combinations of step patterns have to be studied thoroughly.

Change in flow velocity due to change (increase) in the geometry of a pipe system i.e., change (increase) in cross-section sets up eddies in the flow. These eddies do not follow a straight path to the center. It follows a spiraling, whirling path called vortex. Characteristic feature of a vortex is that outside of the vortex moves slowly and the center moves rapidly. If the cross-section of a pipe with fluid flowing through it is abruptly enlarged at certain place, fluid emerging from the smaller pipe is unable to follow the sudden deviation of the boundary. Then the streamlines take a typical diverging pattern. This creates pockets of turbulent eddies in the corners, resulting in the dissipation of mechanical energy into intermolecular energy and the fluid flows against an adverse pressure gradient. The upstream pressure $p_1$ is lower than the downstream pressure $p_2$ since the upstream velocity, $V_1$ is higher than the downstream velocity, $V_2$ as a consequence of continuity. The fluid particles near the wall due to their low kinetic energy cannot overcome the adverse pressure hill in the direction of flow and hence follow reverse path under the favorable pressure gradient. This creates a zone of re-circulating flow with turbulent eddies near the wall of larger tube at the abrupt change of cross-section, resulting in a loss of total mechanical energy.

After the flow encountering the sudden expansion, it separates from the wall and attached to the wall far away from the vertical wall. After the flow attaches to the wall, it moves towards the face of the model because of the presence of low pressure at the vertical wall. This re-circulated flow rejoins with the free stream flow and moves downstream. As the flow moves over a wall, loss in pressure takes place due to friction and after sudden expansion, velocity decreases and pressure increases Fluid particles near the wall due to their low kinetic energy cannot overcome the adverse pressure hill in the direction of flow and hence follow up the reverse path under the favorable pressure gradient as shown in Fig 1.

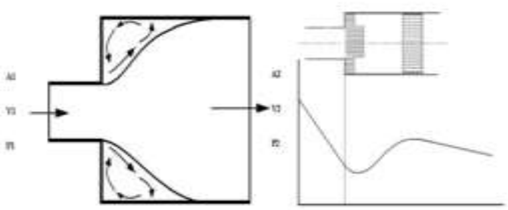

***Figure 1: Geometrical representation of sudden expansion geometry and pressure variation***

## II. LITERATURE REVIEW

In this section, previous works related to the studies on expansion geometries are presented. Dales. [1] studied asymmetry in the turbulent flow of a visco-elastic liquid through an axi-symmetric sudden expansion. Conclusion is drawn that the asymmetry is purely a physical feature of such flow and not the product of upstream or downstream flow conditions derived from the flow facility or the result of geometrical imperfections in the axi-symmetric sudden expansion. Layek. [2] studied two-dimensional laminar flow of an incompressible Newtonian fluid in a symmetric sudden expanded channel with moderate expansion ratio. When the Reynolds number is relatively low, flow and the re-circulating regions at the two channel walls are symmetric. With the increase of Reynolds number, flow remains two-dimensional, but asymmetry of the flow occurs and additional recirculation zones appear along the channel walls. Dağtekin and Ünsal [3] conducted numerical analysis of axi-symmetric and planar sudden expansion flows for laminar regime. In this study, Navier–Stokes equations are solved numerically for axi-symmetric and planar sudden expansion flows. Flow is considered laminar and steady state and fluid is incompressible. Kaushik. [4] performed CFD simulation to investigate core annular flow through sudden contraction and expansion. Flow of lubricating oil and water are simulated using VOF technique.

Detailed study is performed to generate the profiles of velocity, pressure and volume fraction over a wide range of oil and water velocities for an abrupt expansion and contraction. Asymmetric nature of velocity across the radial plane is observed in both the cases. Tsai. [5] were involved in both computational and experimental investigations into the flow of a Newtonian fluid through a sudden expansion micro channel consisting of a rectangular block. It was found that both Reynolds number and aspect ratio have significant impact on the sequence of vortex growth, downstream of the expansion channel. Recently, laminar flow of inelastic non- Newtonian fluids, obeying the power-law model, through a planar sudden expansion with a 1:3 expansion ratio is investigated numerically using a finite volume method by Dhinakaran. [6]. Shear-thinning, Newtonian and shear-thickening fluids are analyzed, with emphasis on the flow pattern and bifurcation phenomenon occurring at high Reynolds number laminar flows. The effect of generalized Reynolds number, based on power-law index and the inflow channel height on the main vortex characteristics and Couette correction are examined in detail. In this context, flow over such geometries with blowing and suction scenario are studied for different $L_{max}/L_{min}$ models namely 3, 4 and 5.It is found that for $L_{max}/L_{min}$= 5 the re-circulation and loss zones are more, for further studying $L_{max}/L_{min}$= 5 is subjected to different operating conditions like with blowing and suction. For further improvement the blowing is studied at different heads of pressure like 200 mm and 300 mm of water. With suction flow scenario is completely changed and attachment point is observed at the base of the model itself. Results thus identify the improved flow structure.

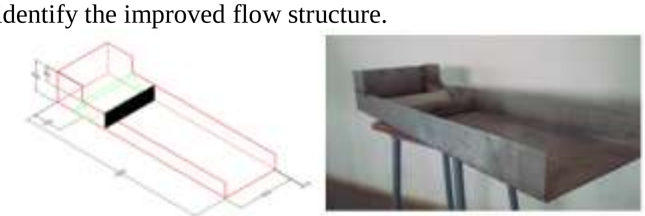

***Figure 2: Open channel design and fabricated work piece***

## III. EXPERIMENTAL SETUP

Flow visualization is proved as one of the best techniques for describing and evaluating the flow features of all realistic problems in both subsonic and supersonic speeds. Previously, researchers developed many techniques such as smoke flow visualization, tufts and chemical coating, shadowgraph and schlieren techniquesto study the flow motion over the objects. When compared to these,

IFERP
connecting engineers... developing research
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 1, Issue 4, August 2016

open channel is the cheapest technique used to visualize this scenario. The experimental system considered in the present work mainly consists of four parts, viz., water chamber, wedge, wire mesh and test section as shown in Fig.2. Water chamber is used to store water from any external source. Function of wedge is to allow the flow uniformly into the channel. In other words, it acts as a spillway to the water chamber. By using the wedge, we can also reduce the effect of turbulence in water. Function of the wire mesh is to condition the flow to fairly uniform and laminar before it goes to the test section. The area of the channel next to the wire mesh is treated as the test section where the objects are tested. The test section is calibrated with the scale and the angles for direct measurement of different parameters.

### *3.1. EXPERIMENTATION PROCEDURE*

Experimentation procedure adapted in the present work is described here.

1) In the first step, the scale and angles are marked on the test section portion of the channel, which will be used to measure some flow parameters. Channel is placed straight with no inclination.
2) Water supply is given to the channel in such a way that fairly uniform flow enters the test section. This is ensured by edge and screens. Thus, uniformity of flow from starting to the end of the channel is checked.
3) Flow velocity is set in the channel, according to the requirement. Velocity of the flow can be calculated by using the floating particle method.
4) After setting the required velocity, model to be tested is placed in the test section according to the requirements. Sufficient amount of dye is injected at the second screen that helps in getting the streamline flow and video is taken to capture the flow pattern over the object for further analysis.
5) From the video, necessary frames at regular intervals are extracted and used in the present investigation.

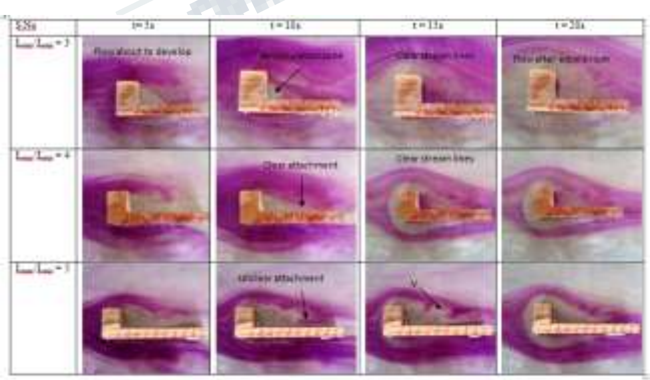

***Figure 3 : Flow development over the models without blowing and suction***

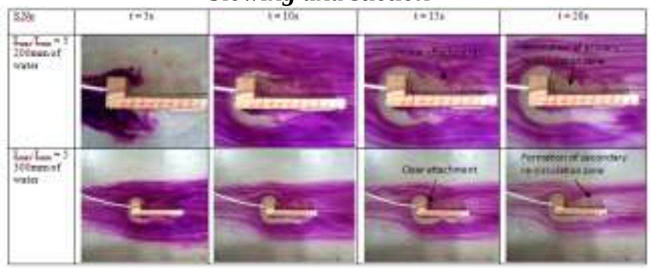

***Figure 4 Flow development over the model $L_{max}/L_{min} = 5$ with blowing at different heads***

## IV. FLOW STUDIES

The present work aims at understanding flow separation, recirculation, attachment and reverse flow phenomenon. These flow features are highly complex to deal with. An attempt has been made to understand them in case of sudden expanded geometries. Usually, the models are designed to understand the effect of length and height on the complex flow phenomenon. In this work, only one case of $L_{max}/L_{min}$ = 5 is presented with both suction and blowing effect. Flow scenarios are explained in both the cases and comparison is made without blowing and suction.

### *4.1. FLOW DEVELOPMENT OVER THE MODELS WITHOUT BLOWING AND SUCTION*

Figure 3 represents the flow structure over the models of $L_{max}/L_{min}$= 3, 4 and 5, with no blowing and suction. From this all the complex flow features like separation, attachment point, vortex size and recirculation regions t = 15s can be identified. For $L_{max}/L_{min}$= 3, flow attaches to the wall at 2.25 (attachment point / step height) at time t = 15s. For $L_{max}/L_{min}$= 4, flow attaches to the wall at 2.5 and for $L_{max}/L_{min}$= 5, the flow attaches to the wall at 3 from the vertical wall as shown in the figure. As the time passes, the attachment point shifts towards the base of the model leading to the formation of vertical streamline flow (vortex) and re-circulating zone. $L_{max}/L_{min}$= 3 is showing small re-circulation zone than $L_{max}/L_{min}$= 5. At time t = 15s, flow becomes clearly streamlined with $L_{max}/L_{min}$= 3. For $L_{max}/L_{min}$= 4, the stream lines are not so clear and the streamlines become subsequent slightly unclear. For $L_{max}/L_{min}$= 5, rotational flow is created in the presence of large vortex and more re-circulation. The following sections are used to analyze the methods to reduce the vortex and re-circulation region in order to improve the flow scenario. To meet this, different operating conditions are considered.

### *4.2. FLOW DEVELOPMENT OVER THE MODEL $L_{MAX}/L_{MIN}$ = 5 WITH BLOWING AT DIFFERENT HEADS*

 

Figure 4 represents flow structure over the model of $L_{max}/L_{min}$= 5 for blowing at different heads of 200 and 300mm of water respectively. With blowing at 200mm of water, attachment point occurs at 0.25in the beginning of the flow. Because of pressurization in the corner, the flow is pushed to the downstream direction without re-circulation that leads to elimination of vortex and re-circulation. For blowing at 300 mm of water, flow experiences different structure than the earlier one. Initially, the flow attaches at 0.25 from the vertical wall. As the time passes, flow gets reversed and moves as per the geometry change. After that flow again detaches from the wall at a distance of 1 from vertical wall, which is known as detachment point. Again, flow gets attached to the wall far away from the vertical wall and forms secondary recirculation zone. In this,primary recirculation zone is eliminated and only secondary recirculation is appearing. By comparing these, it is identified that vortex and re-circulation zones are eliminated in case of blowing at 200 mm of water when t = 15s. But in the case of blowing at 300 mm of water, flow attaches to the vertical wall itself and moves downstream by attaching the wall up to 1 from the vertical wall. After that as well, flow experiences complex phenomena, where flow is detached from the wall and forms secondary re-circulation zone.

## REFERENCES

[1] C. Dales, M.P. Escudier, R.J. Poole, "Asymmetry in the turbulent flow of a viscoelastic liquid through an axisymmetric sudden expansion", J. Non-Newtonian Fluid Mech. 125 (2005) 61–70.

[2] G.C. Layek, C. Midya, S. Mukhopadhyay, "Effects of Suction and Blowing on Flow Separation in a Symmetric Sudden Expanded Channel", Modelling and Control, 2008, Vol. 13, No. 4, 451–465.

[3] İhsanDağtekin and MazharÜnsal, 'Numerical analysis of axisymmetric and planar sudden expansion flows for laminar regime', International Journal for Numerical Methods in Fluids, Vol. 65, No. 9, pp. 1133-1144, 2011.

[4] Kaushik V.V.R, GhoshSumana, D.A.S.Gargi, Kumar Das Prasanta, CFD simulation of core annular flow through sudden contraction and expansion, Journal of petroleum science & engineering, Vol. 86-87, pp. 153-164, 2012.

[5] Chien-Hsiung Tsai, Cheng-PengYeh,Che-Hsin Lin, Ruey-Jen Yang, Lung-Ming Fu, Formation of recirculation zones in a sudden expansion microchannel with a rectangular block structure over a wide Reynolds number range, Micro fluid Nan fluid (2012) 12:213–220.

[6] S.Dhinakaran, M.S.N. Oliveira, F.T. Pinho, M.A. Alves, Steady flow of power-law fluids in a 1:3 planar sudden expansion 2013 Journal of Non-Newtonian Fluid Mechanics 198 (2013) 48–58

 