Improving Flow Pattern in a Sudden Expansion Geometry Using Blowing and Suction

[1] K.Ashokachary, [2] V.Ramesh [3] V.Rajarao

[1][2] Department of Mechanical Engineering, Vijaya Engineering College, Khammam – 507305,

[3] Department of Mechanical Engineering, Jayamukhi Institute Of Technological Sciences, Narsampet, Warangal - 506332

[1] Ashokacharyideal@gmail.com. [2] rameshvellanki0342@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 $L_{max}/L_{min} = 5$.The operating conditions for each model vary with water being blown at different levels of water and single and double injections as well. Flow patterns for all the models under consideration are captured. Variation in the position of separation point is observed, i.e. the separation point shifts towards downstream of the channel and attachment point shifts towards upstream direction, which is more desirable for sudden expanded geometries.By using blowing technique, recirculation zone reduces about 12 times. Strength of the vortex also changed with the configurations considered. Such findings with different height ratios and blowing techniques provide researchers with valuable guidelines for improving the design and operation of various fluidics used to study flow in channels.

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 pathcalled 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 takea 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 freestream 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.
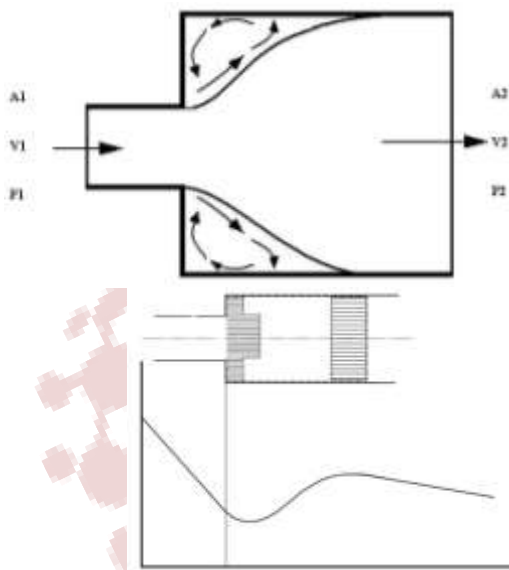

Figure 1: Geometrical representation of sudden expansion geometry and pressure variation in that.

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 anaxi-symmetric sudden expansion. Conclusion is drawn that the asymmetry is purely a physical feature of such flowand not the product of upstream ordownstream flow conditions derived from the flow facilityor 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 are examined in detail. In this context, interest of the present work is to investigate the nature of primary and secondary recirculation regions formed near the wall when flow encounters sudden change (increase) in geometry in its direction. By using blowing and suction at different step heights, flow scenario is studied and the effects of on recirculation regions aredetailed.

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 techniques tostudy the flow motion over the objects. When compared to these, 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 testsection is calibrated with the scale and the angles for direct measurement of different parameters.

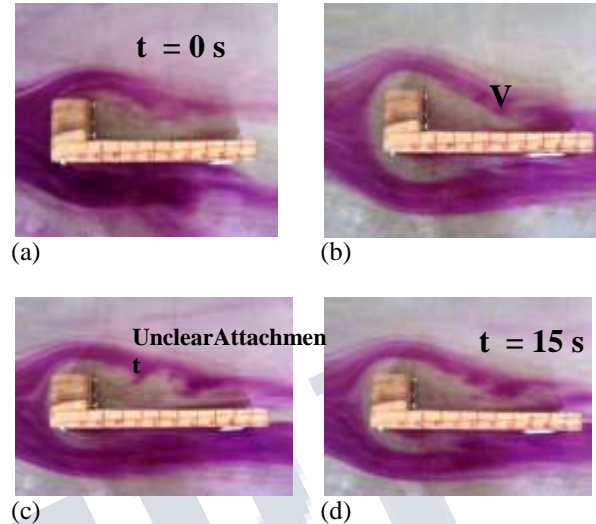*Figure 2: Open channel design and fabricated work piece.*

(a) (b) (c) (d)

Figure 3:Flow development over the model without blowing and suction.

3.1. EXPERIMENTATION PROCEDURE

1) Experimentation procedure adapted in the present work is described here 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.

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 MODEL WITHOUT BLOWING AND SUCTION

Figure 3 represents the flow visualization study of $L_{max}/L_{min} = 5$ at without blowing and suction. Dye is injected and observations are made. The attachment point is not clear and occurs at a farther point from the vertical wall. Also, the formation of vortex is not observed. Flow is not at all along the geometry and is far away from it. For improved flow scenario, attachment should occur nearer to the wall, which is not observed in this case of no blowing and suction.
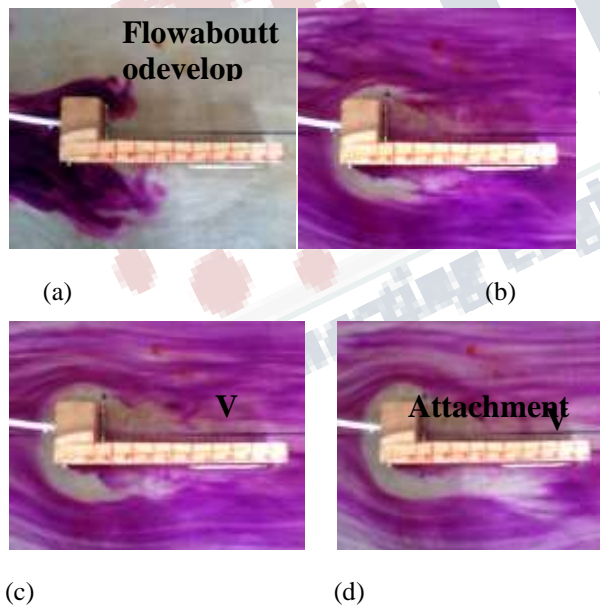

(a)   (b)

(c)   (d)

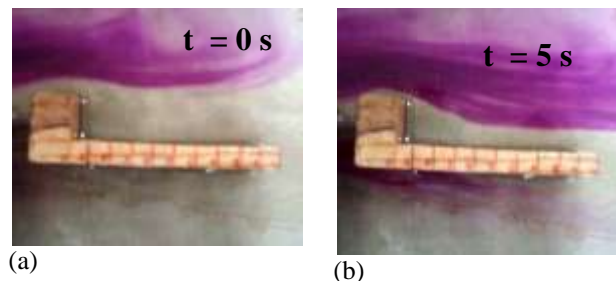*Figure 4: Flow development over the model with blowing.*

4.2. FLOW DEVELOPMENT OVER THE MODEL WITH

BLOWING

Figure 4 represents the flow visualization study of $L_{max}/L_{min} = 5$ when blowing at 200 mm of water with single injection. As shown in Fig. 4(a), the flow just tends develop and flows over the step. After that flow finds sudden increase in area as it moves gradually and slightly attached to the vertical wall. At a distance 0.25 (ratio of attachment point/step height) from the vertical wall, flow again gets attached to the obstacle. Then flow reversal takes place because of presence of low pressure at the vertical wall and this reverse flow recalculates at corner and rejoins with free stream flow. This makes primary recirculation zone and the fluid particles move smoothly after the attachment point. Vortices are clear as shown in Fig. 4(c) and (d) when compared to the previous instances.

4.3. FLOW DEVELOPMENT OVER THE MODEL WITH SUCTION

Figure 5 represents the flow visualization study of $L_{max}/L_{min} = 5$ with suction. In this, the effect of suction on recirculation is observed clearly. As the flow finds sudden change in geometry, flow separation takes place and tries to attach to the wall on the downstream. Because of suction affect, the flow moves towards the corner, by influencing in recirculation zone and attachment point. The attachment point moves towards the base of the model. Attachment point length is decreased compared to the without suction case. The size of the recirculation zone is reduced. This is very much desirable for suddenly expanded geometries. The attachment point is 1 (ratio of attachment point/step height) from the vertical wall, this value is 3 for without blowing and suction effect.
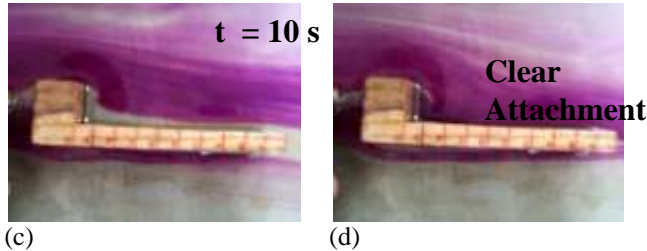

(a)   (b)

t = 15 s

t = 10 s

(c)

Clear Attachment

(d)

Figure 5: Flow Development Over The Model With Suction.

4.3. COMPARISON OF ATTACHMENT POINT/STEP HEIGHT AT STEP HEIGHT (H) = 20 MM FOR DIFFERENT CONDITIONS

Figure 6 represents variation of attachment point/step height for $L_{max}/L_{min} = 5$ and step height 20 mm at various operating conditions. The attachment point/step height occurs at 3 from the vertical wall and this value is reduced to 0.25 for the case of blowing at 200 and 300 mm of water with single and double injections.Without blowing and suction, the attachment is away from the geometry as shown by large bar in the figure. With blowing at 200 and 300 mm head in case of single and double injections showed similar locations of the attachment point. But, the attachment point showed drastic change in location. If the head is reduced to 100 mm, the attachment shifted towards right, i.e., away from the wall. Thus, it can be concluded that by using blowing at 200 and 300 mm of water, with single and double injections we can reduce the recirculation zone size nearly about 12 times.
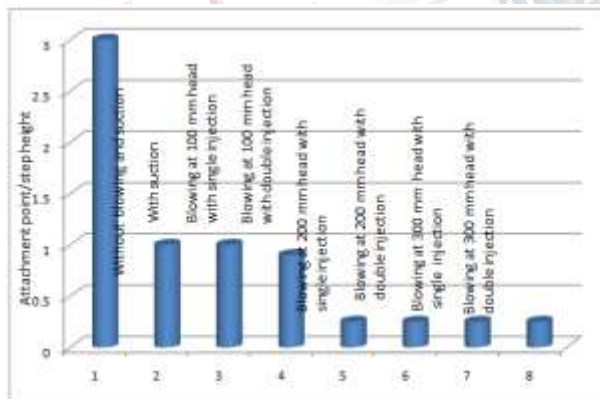

Figure 6: variation of attachment point/step height for various operating conditions.

V. CONCLUSIONS

Flow pattern in an open channel is experimentally studied. The case of $L_{max}/L_{min} = 5$for different conditions like blowing, suction and without blowing and suction are considered. It is observed that flow pattern is not as required without blowing and suction. With blowing, the region of vortex is observed, confirming the effect of blowing. The attachment point shifts towards the wall with blowing. Further, when suction is applied, the flow follows the geometry and vortex is not observed near the walls. Also, the attachment is nullified and flow pattern is smooth when compared to the other two considerations. Similar experiments can be done for different heights and configurations for better understanding of the flow scenario. Suggestion for reducing re-circulation regions and thus flow losses would help the designers in a big way.

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]   İhsan Dağtekin and Mazhar Ünsal, '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, Ghosh Sumana, 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.