

Prediction of pressure drop across orifice using MATLAB and 2D analysis of flow through it using CFD

Mr. Krishna Sharma*, Mr. Pranav Salunke

Department Of Mechanical Engineering, Marathwada Mitra Mandal's College Of Engineering, Pune

*Corresponding Author: krishnabsharmas@gmail.com

Abstract—Pressure drop is predicted across the orifice when atmospheric air is flowing through it at different flow rate. For this purpose MATLAB code is created. Experimental data of pressure drop is compared with the output of MATLAB code. Values of experimental and calculated pressure drop are nearly same. This MATLAB code is useful to predict the pressure drop. To analyze the flow through orifice ANSYS FLUENT is used for CFD analysis. k-epsilon model is used to simulate turbulence flow in axisymmetric orifice. CFD results demonstrate the following parameters: axial velocity profile, static pressure, and contour of velocity inside the orifice. Numerical result matches with experimental one.

Keywords— Orifice, atmospheric air, pressure drop, MATLAB code, CFD analysis, k-epsilon model, turbulence intensity, velocity contour

INTRODUCTION

Orifice is fundamental geometry which is used in many engineering applications such as air filter assembly, gas pipelines etc. It can be also used for calibration of test setup. Experiment is conducted on sudden expansion geometry which is made up of aluminum at various flow rate.

GEOMETRY OF ORIFICE

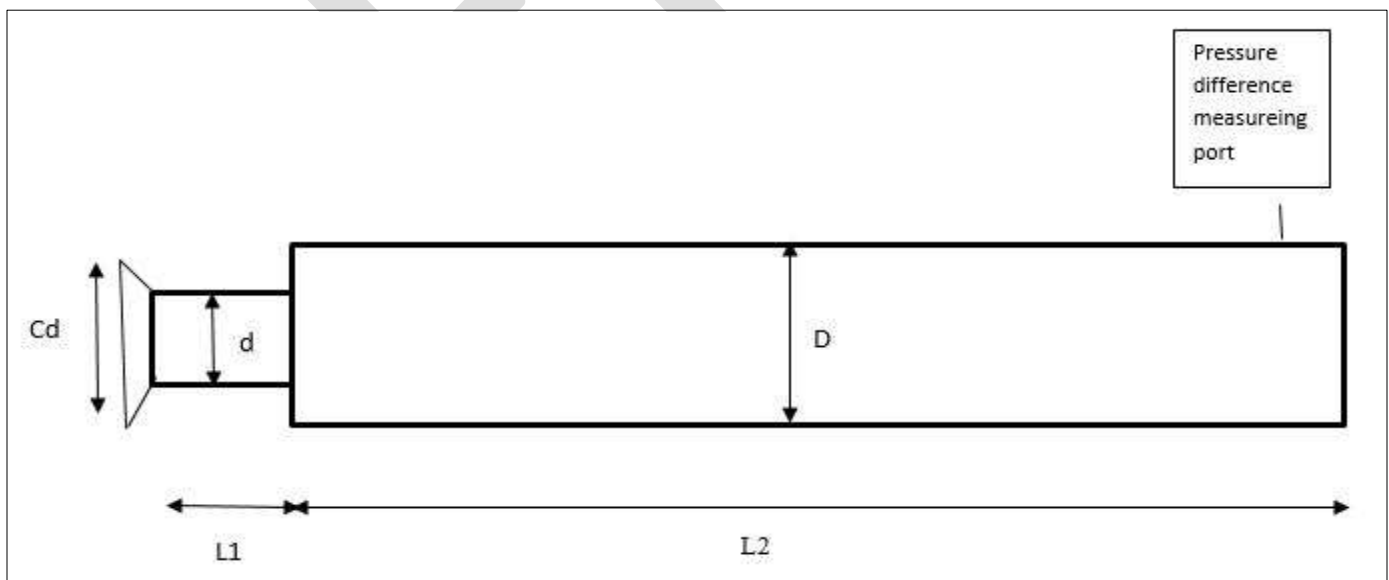


Fig. 1. Cross section of orifice

Fig. shows orifice cross section. The orifice geometry consist of a cylindrical tube of length $L_1=0.96''$ with diameter $d=1.5''$ and $L_2=16.58''$ with diameter $D=4''$. Value of diameter at upstream is $2.34''$ with cone angle 51.74 degree.

TEST CONDITIONS

Test are conducted at different flow rate i.e. 100 cfm, 150.1 cfm and 201.2 cfm. While testing temperature was 29.2°C with barometric pressure 953 milibar. Value of relative humidity was varying between 60.7% - 60.2%.

CALCULATION OF PRESSURE DROP

To calculate pressure drop across the orifice MATLAB code is created. It includes calculation of density, velocity at different sections, Reynolds number, and loss due to conical entrance, sudden expansion loss and finally pressure drop calculation. Friction losses are neglected. Generated MATLAB code is given below

```
% upstream & downstream diameter, flow rate, temperature, Relative
% humidity, Restriction at downstream, Barometric pressure, Inlet pressure
% are required as input parameter.
clc
d=input('Enter the value of orifice diameter [inch]      :');
D=input('Enter the value of pipe diameter [inch]        :');
teta=input('Cone angle in degree                        :');
cd=input ('Cone diameter [inch]                        :');
q=input('Flow rate [cfm]                                :');
T= input ('Enter the value of temperature [degree Celsius]d :');
F=input('restriction at downstream [millimeter of water column :');
RH=input('Relative humidity %                          :');
BP=input('Barometric pressure [milibar]                 :');
d=(d*0.0254);% inch to meter
D=(D*0.0254);
cd=(cd*0.0254);
A=((pi/4)*D*D);%area calculation
Ad=((pi/4)*d*d);
Acd=((pi/4)*cd*cd);
g=9.81;
b=d/D;% expansion ratio
K=((1-b*b)^2)/((b^4));
b1=d/cd;
teta=teta*(pi/180);% degree to rad
K1=((1-b1*b1)^2)/((b1^4))*0.5*((sin(teta/2))^0.5);% conical loss coefficient
q=((q*1.699)/3600); % cfm to m^3/s
RH=RH/100;
BP=BP*100; % millimeter to Pa
Vcd=q/Acd;% velocity calculation
V1=q/Ad;
V2=q/A;
m=0.017362065;
a=393;
B=(T+273)+120;
mi=m*(a/B)*((T+273)/273)^1.5;% dynamic viscosity calculation
mi=mi*10^(-3);% cP to kg/ms
Recd=(cd*Vcd*rho)/mi;
fprintf('\n Reynolds number based on cd                = %f',Recd);
Re1=(d*V1*rho)/mi;
fprintf('\n Reynolds number based on d                    = %f',Re1);
Re2=(D*V2*rho)/mi;
fprintf('\n Reynolds number bases on D                      = %f',Re2);
```

```

hteta=(K1*V1*V1)/(2*g); % pressure loss due to conical section
F=(F*100)/(10.19744);
actualp=BP-F;
rho=(actualp-(4260.96*RH))/((287.05*(273.15+T)))+(2344.217*RH)/((461.49*(273.15+T))); %
density calculation
hl=(K*V2*V2)/(2*g); % loss due to sudden expansion
p1=BP-hteta;
p2=((p1/(rho*g))+((V1*V1)-(V2*V2))/(2*g))*rho*g)-hl;
P=p2-p1; % pressure drop
cf=(293/(273+T))*(BP/101325); % correction factor
hteta=hteta/(25.4*9.8);
P=P/(25.4*9.8); % Pa to inches of water column
P=(P*cf)+hteta;
fprintf('\n Pressure difference [inches of H20]                =%f', P);
    
```

COMPARISION BETWEEN EXPERIMENTAL AND CALCULATED VALUE

Experimental pressure drop value is compared with calculated value which is obtained from MATLAB code. Comparison table is given below

| Flow rate (cfm) | Pressure difference experimental (inches of water) | Pressure difference MATLAB (inches of water) |
|-----------------|--|--|
| 100 | 3.31 | 3.2851 |
| 150.1 | 7.41 | 7.3059 |
| 201.2 | 13.57 | 12.87 |

Table 1. Comparison between Experimental and calculated value of pressure drop

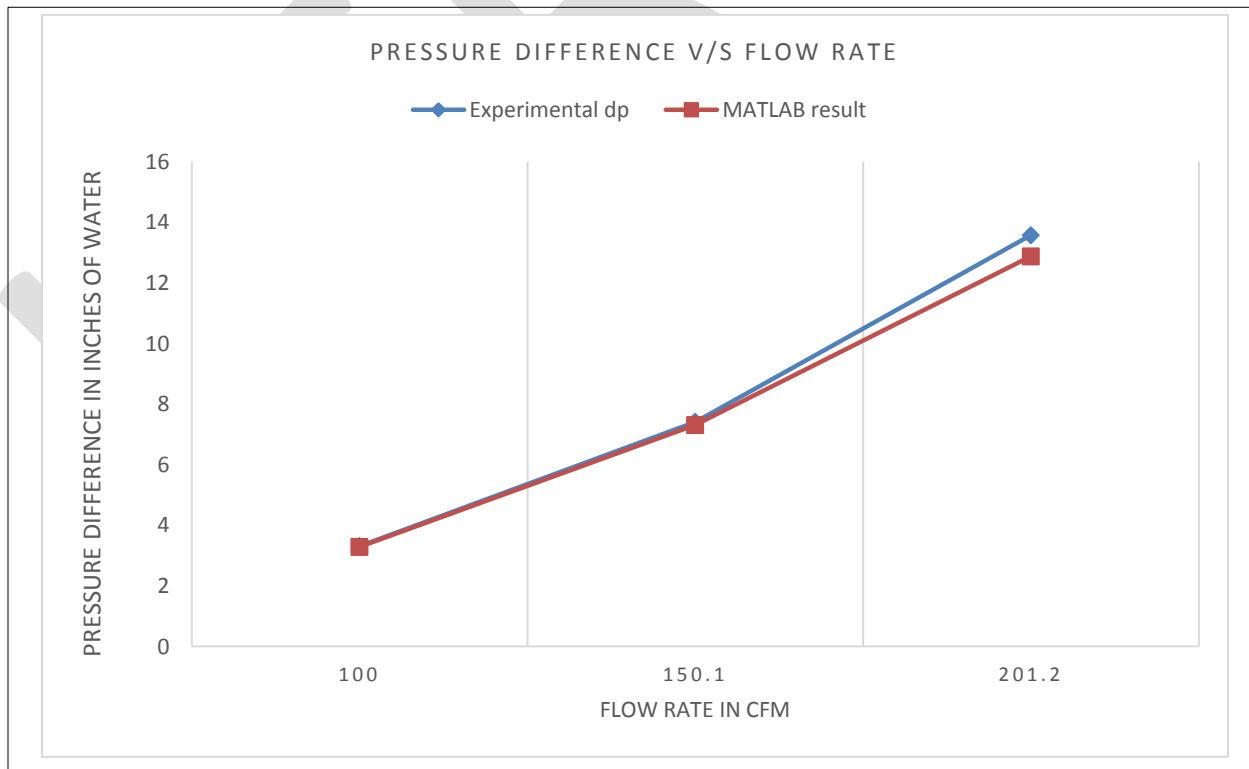


Fig.2. Graphical representation of pressure drop

From the comparison table and graph it can be observed that experimental and calculated value of pressure drop approximately equal. This means generated MATLAB code can be used to predict the pressure drop across orifice for given conditions. To understand the

behavior of the flow inside orifice CFD analysis is done.

CFD ANALYSIS OF FLOW THROUGH ORIFICE

Computational fluid dynamics had been spear heading science and technology progression by being cost effective, determining solutions for complex design challenges, incorporation of advanced. For orifice, 2D CFD analysis is done by using ANSYS FLUENT 14.0. Design Modeler is used to create 2D geometry of orifice. Proper meshing is important to analyze flow correctly. To capture the two-dimensional flow inside the domain with reasonable accuracy, one needs good quality mesh. ANSYS ICME is used for this purpose. Also while meshing named creation is done to ease boundary conditions.

Mesh details: Domain- surface_body, Nodes -38460, Elements-18520. Generally orthogonal quality ranges from 0 to 1, where value close to zero corresponds to low quality. Generated mesh quality is 0.81 which is good.

Turbulence Model: Standard (k- ϵ) turbulence model is used over than other turbulence model as it can predict boundary layer under strong adverse pressure gradients or separation. Rotational flow and recirculation can be exactly model by this model. Standard (k- ϵ) turbulence uses two partial differential equations to estimate the velocity and length scales and hence it is commonly known as two-equation model of turbulence.

Boundary conditions: For inlet, velocity boundary condition is used with gauge pressure value is equal to zero and velocity is taken from output of MATLAB code. Turbulence specifications are given with turbulence intensity and hydraulic diameter. Turbulence intensity is calculated Reynolds number i. e. $0.16 (Re)^{1/8}$. For center axis boundary condition is suitable. At outlet, pressure outlet is used with gauge pressure value equal to pressure drop value which has been derived from experiment.

RESULTS & DISCUSSION

CFD Analysis was carried out with different flow rate CFD Simulation shows pressure drop across the orifice system increases as the engine rpm increase. CFD results show very good correlation with testing result. Pressure plots, streamlines, velocity vectors are created graphically.

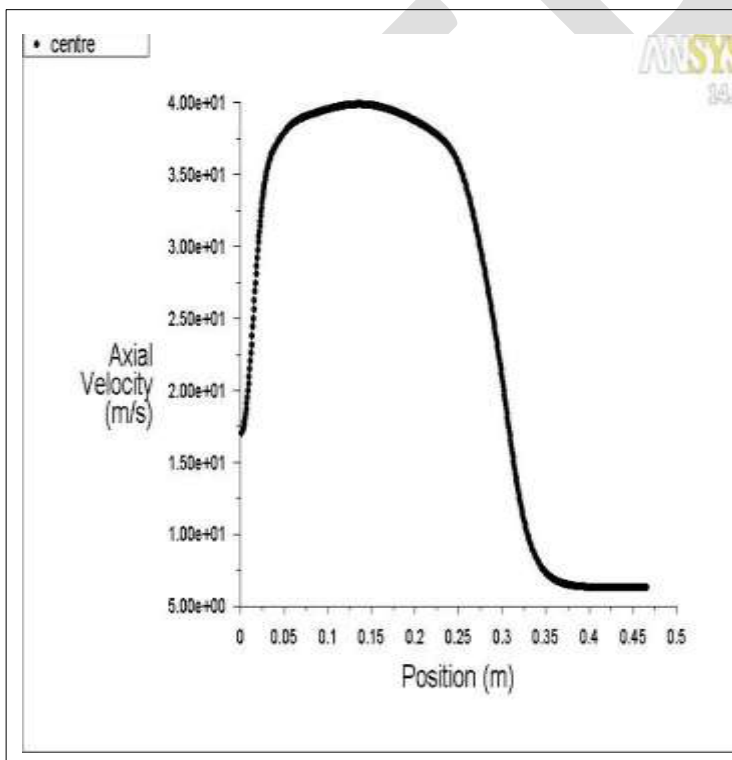


Fig.3 a) Axial velocity at flow rate 100 cfm

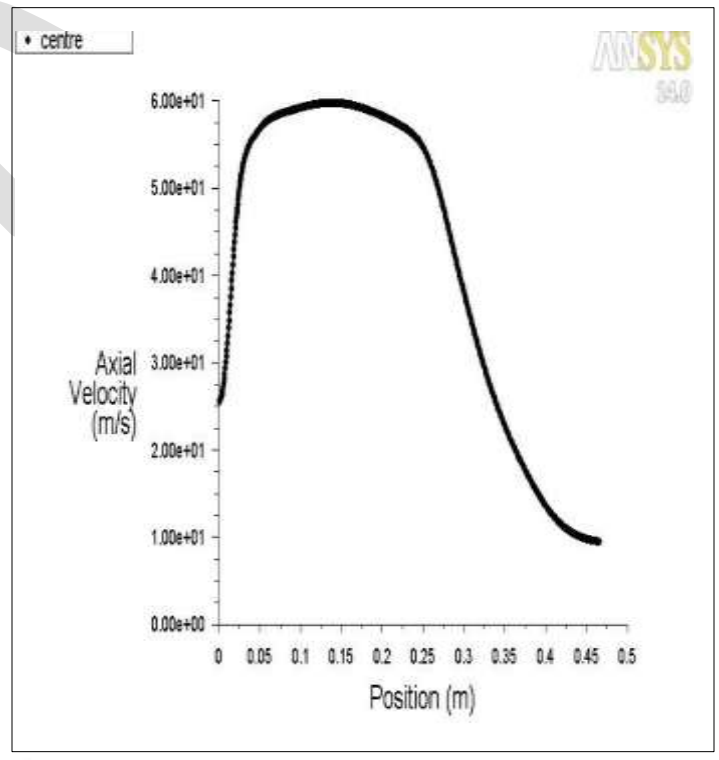


Fig 3 b) Axial velocity at flow rate 150.1 cfm

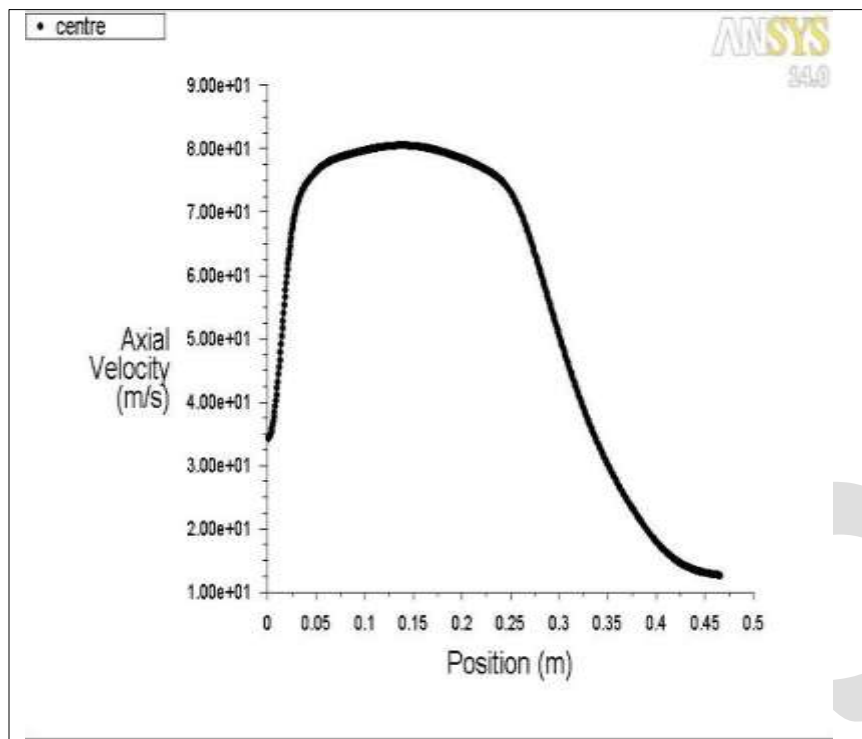


Fig 3 c) Axial velocity at flow rate 200.1 cfm

From these graphs, analysis of axial velocity inside the orifice can be done properly. It can be observed that axial velocity first increases from conical section to orifice diameter. Then due to sudden expansion velocity decreases. Axial velocity is minimum at the section of large diameter. Velocity calculated at the output from MATLAB code matches with CFD results.

Value of static pressure:

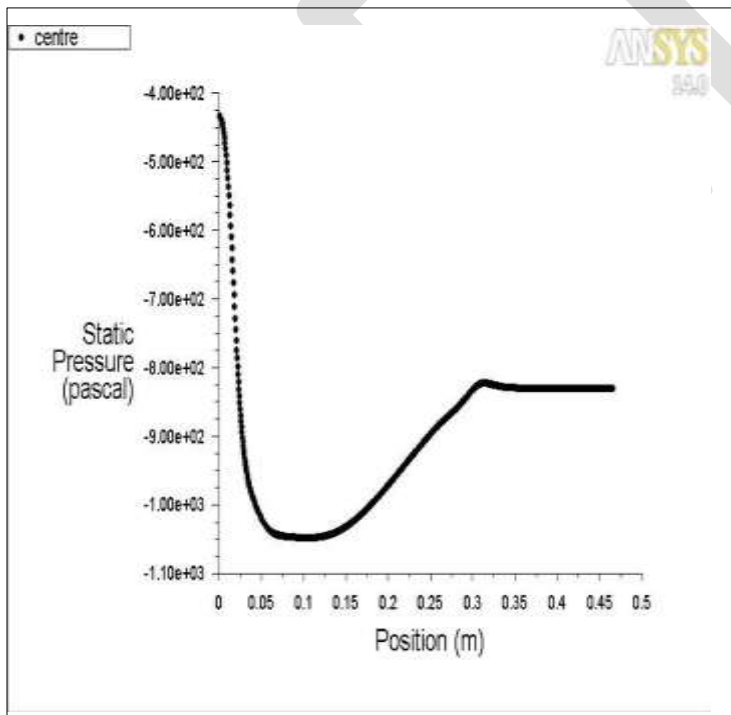


Fig. 4 a) Static pressure at 100 cfm

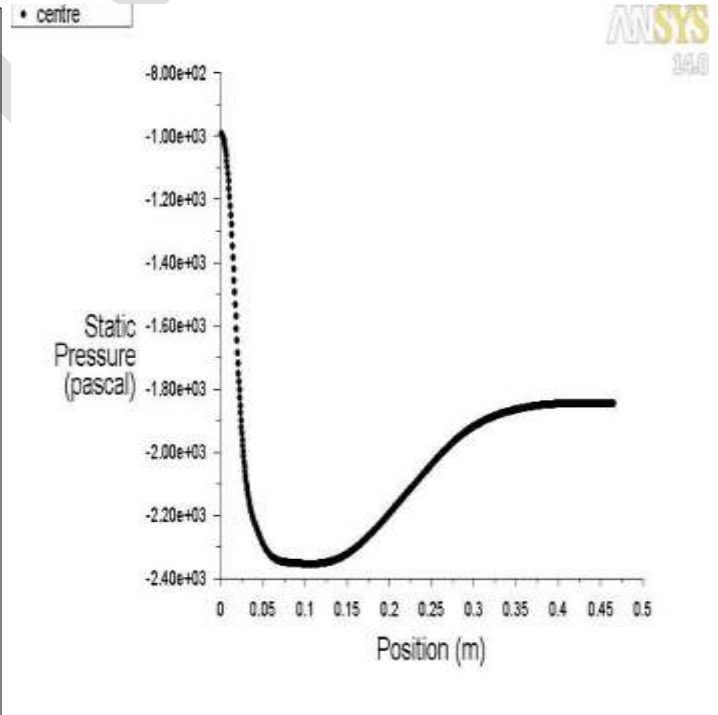


Fig. 4 b) Static pressure at 150.1 cfm

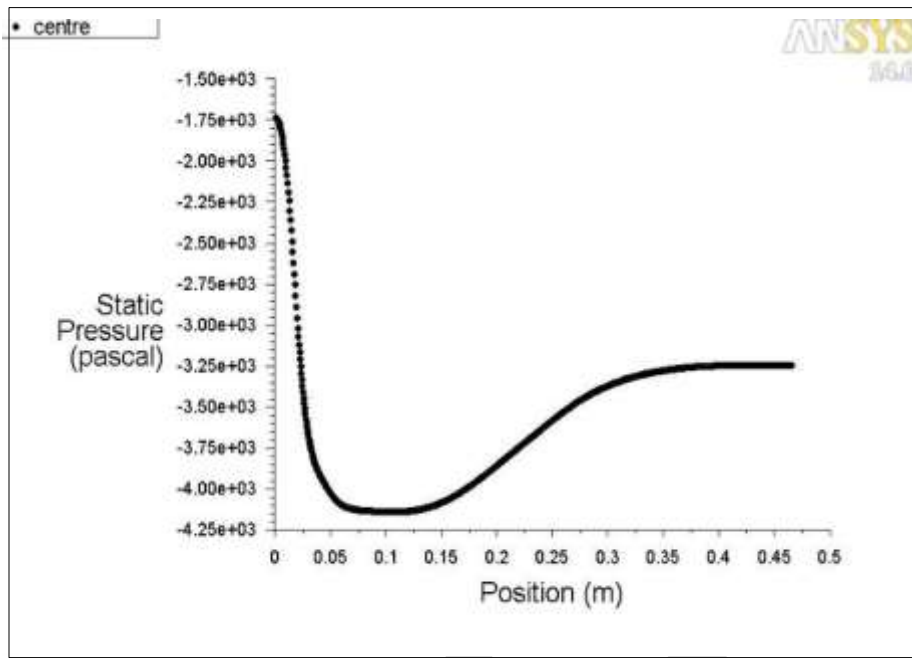


Fig. 4 c) Static pressure at 200.1 cfm

Behavior of static pressure can be observed clearly. Maximum pressure is at inlet and it reduces due to decrease in diameter. Again due to sudden expansion value of static pressure increases.

Contour of velocity streamline is generated for the flow rate of 100 cfm. It is maximum just after expansion.

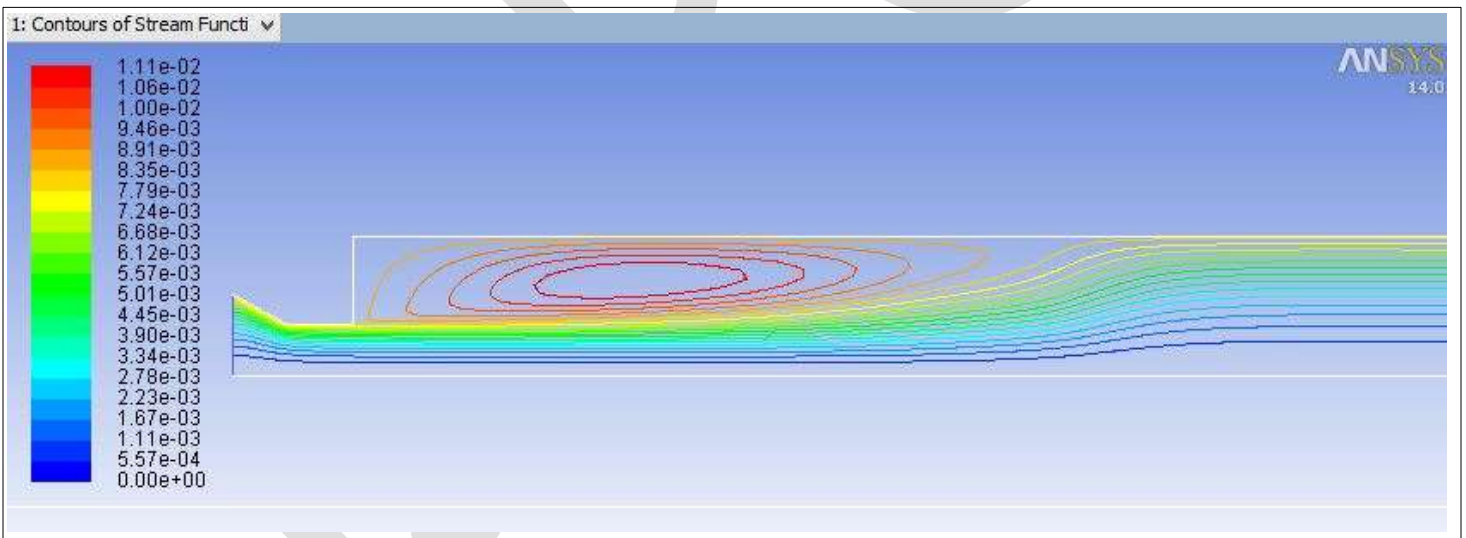


Fig.5 Contour of velocity streamline

It also matches with PIV experiment (Hammad et al., Experiments in Fluids, 1999, 26:266-271). It can be seen that the numerical result agree with the experiment one.

ACKNOWLEDGMENT

We wish to thank and express deep sense of gratitude to our college Marathwada Mitra Mandal's College Of Engineering, Pune and Dr. Ashok Kumar Vaikuntam for his consistent guidance and sympathetic attitude throughout the work.

CONCLUSION

As stated earlier, the objective of this study is to predict the pressure drop across orifice which is achieved by creating the MATLAB code. Numerical result agree with experiment one. Analysis of flow is done by ANSYS FLUENT. Axial velocity at various section and value of static pressure at center is studied. Contour of velocity streamline matches with PIV experiment (Hammad et al., Experiments in Fluids, 1999, 26:266-271).

REFERENCES:

- [1] J. Mamizadeh abd S.A. Ayyoubzadeh “ Simulation of flow pattern in open channel with sudden expansions ” Research Journal Of Applied Science and Technology, ISSN: 2040-7467 , October 01, 2012
- [2] Baoyu Guo, Tim A.G. Langrish and David F. Fletcher “ Simulation of precession in axisymmetric sudden expansion flows” Second International Conference on CFD in Minerals and Process industries CSIRO, Melbourne, Australia, 6-8 December 1999
- [3] Mohommad Reza Bazargan-Lari and Abbas Mansoori “ CFD based location prediction of maximum negative pressure in pipe sudden expansions ”5th WSEAS International Conference Of Fluid Mechanics Acapulco, Mexico, January 25-27,2008
- [4] Monomed A. Siba, Wan-Mohd Faizal Wan Mohomood and Mohamed H. Nassir “ Modelling and application of 3D flow in orifice plate at low turbulence Reynolds numbers” International Journal Of Mechanical and Mechatronics Engineering IJMME-IJENS, Vol:15 No:4
- [5] Abbas Mansoori and Mohommad Reza Bazargan-Lari “ Evaluation of turbulence model in sudden expansion analysis at high Reynolds number” Proceeding of 5th IASME/ WSEAS Internatinal Conference Of Fluid Mechanics and Aerodynamics,Athens, Greece, August 25-27,2007
- [6] L. Casarsa and P. Giannattasio “Three-dimensional features of the turbulent flow through a planar sudden expansion ” PHYSICS OF FLUIDS 20, 015103 (2008), 28 January 2008
- [7] A.S.Phulpagar,and N.S Gohel “ CFD Analysis of Air Intake System” Ird India
- [8] John D Anderson Jr, Computational Fluid Dynamics: The Basics with Applications, Fourth Edition, New York: McGraw Hill International Editions, 1995, pp. 4-82.
- [9] Graber, D., 1982. Asymmetric flow in symmetric expansions. J. Hydr. Div., 108: 133-140.
- [10] Poole, R.J., Escudier , M.P. (2004) “ Turbulent flow of viscoelastic fluids through an axisymmetric sudden expansion” J.Non-Newtonian Fluid Mech.
- [11] Chaturvedi, M.C. (1993) “Flow characteristics of axisymmetric expansions” Journal of hydraulic division, ASCE Vol89,No HY3.
- [12] Ansys Inc, Ansys Fluent User Manual