Transient flow characteristics through sudden expansion and contraction

C. Lisa[1], Riya kumar[1], Tathagata Ray[1], Sagnik Dutta[1], Triparna Datta[1,]*

[1]Department of Basic Science & Humanities, Institute of Engineering &Management, Kolkata-700091

Email: triparna.datta@iemcal.com

Abstract

The transient flow characteristics have significant applications in the engineering industry, especially in the domains of high energy fluid physics and compressible flow. The present research work provides an investigation of the laminar-to-turbulent transition fluid flow when air passes through successive contractions and expansions. The study has been performed through experimental runs in a fiberglass wind tunnel at different check points and then validated through simulation in the ANSYS workbench by keeping the input parameters constant. The simulation reveals the flow detachment due to turbulence and sudden increase in area. The research also plays a pivot roll to depicts that the velocity has been changing over the different sections of the flow conduit. The validation has resulted in an accuracy reach of almost 99% and has therefore, provided a lucid understanding of the velocity and pressure profiles of the flow through the conduit.

Keywords – *Transient; Laminar; Turbulent;compressible flow, Flow Profiles; ANSYS*

1. Introduction

Continuity of state is the property of a transition between two states of matter, as between gas and liquid, during which there are no abrupt changes in physical properties It states that the rate at which mass enters a system **is** equal to the rate at which mass leaves the system plus the accumulation of mass within the system in this research paper we have used the continuity to study the transient flow characteristics. Tusar et al. [1] 2019 has studied a 3D computational fluid dynamics analysis to investigate the heat transfer and fluid flow characteristics using a helical screw tape. Also, in another research paper kim et al [2]2018 has studied the flow field in the liquid phase of high viscous oil and air the phase slug flow were studied in laminar flow conditions. Many researches were done on pressure drop laminar flow, Ihsak et al. [3] 2019 has investigated the pressure drop in laminar flow in circular pipe with perforated plate. Dehkordi et al [4] 2018 has written this paper and the main purpose was to study the hydro dynamic behavior of high viscous oil water flow within the horizontal ducts undergoing sudden expansion. Many studies were done on laminar turbulent transmissions one of them was Nguyen et al [5] 2019 who has studied the laminar-turbulent transition in a sudden expansion with expansion ratio of 1:2 subjected to an inlet parabolic velocity profile. Naeyyf et al [6] 2019 has studied numerically unsteady laminar flow through sudden expansion channel. CFD software was developed top Navier-Stoke’s equation. Lebon et al [7] 2018 has studied the results of experiments on the flow through a circular sudden expansion pipe at moderate reynold’s number.

.

2. Theoretical Background

The continuity equation has been used primarily to compute the area and velocities of the three fluid domains in the ANSYS workbench, where A = Area of Cross Section and V = Velocity.

$$A * V = C \qquad \text{………………..1}$$

Next, the Navier Stokes equation has been used to study the changes in the flow such as limar-to-turbulent or vice versa, where, p = pressure, ρ = density and μ = viscosity.

$$\rho\left[\frac{\delta V}{\delta T} + (V.\nabla V)V\right] = -\nabla V p + \rho * g + \mu\nabla^2 V \qquad \text{………………..2}$$

For validation, the individual error percentage and the total averaged error percentage have been formulated and calculated using the following equations 3-4, where, N = the total number of observations recorded.

$$Error(\%) = \frac{ActualValue - ExpectedValue}{ExpectedValue} * 100\% \qquad \text{……..…3}$$

$$TotalAveragedError(\%) = \frac{\sum Individual\,Error(\%)}{N} \qquad \text{………..4}$$

3. Experiment Setup & Methodologies

In this section, a detailed description of the experimental setup, initial experimentation parameters and the methodologies used while investigating the flow parameters have been explained. Firstly, the real-time experimentation shall be discussed in details along with the real-time results and then a detailed simulation study in the ANSYS workbench shall be presented in the study. The simulation study has been performed to compute the *degree of validation* with the experimental results. The change in the results shall be notified in the form of a percentage error.

3.1 Experiment Design

The experiment has been conducted in the apparatus as shown in the FIG 1. The setup shown in the figure primarily consists of an electric blower along with attached transparent fiber glass pipes of different diameter and cross sectional areas. Air is used as a moving fluid inside the apparatus and a detailed study of velocity and pressure profiles have been studied here.

FIG1: Experimental Setup

The diameter for the three different fluid domains can be demonstrated as in table 1. Figure 2 is an ANSYS workbench created diagram of the apparatus – the three zones have been marked separately for better understanding. The *inlet* of the apparatus has been attached with the electric blower's *outlet* and the *outlet* zone of the setup has been exposed to atmosphere for *free air discharge*.

Table1: Cross Sectional Diameters

Fluid Domain	Diameter
Domain 1	12 cm
Domain 2	8 cm
Domain 3	10 cm

The electric blower has been given an input speed of around 40.1 m/s which is generally in the *laminar-turbulent transition zone*. However, when the flow has passed from an expanded region to a constricted passage and then again into an expanded conduit, there were severe changes in the flow parameters which we refer to as *turbulence.* The flow velocities and pressure difference in the three different zones have been recorded to be discrete in magnitude. The initial parameters of the experiment can be demonstrated in table 2 as:

Table 2: Initial Experimental Parameters

Inlet Velocity	Density of Air	Diameter	Viscosity of Air
40 m/s	1.164 kg/m^3	12 cm	1.868 x 10^{-5} kg/ms

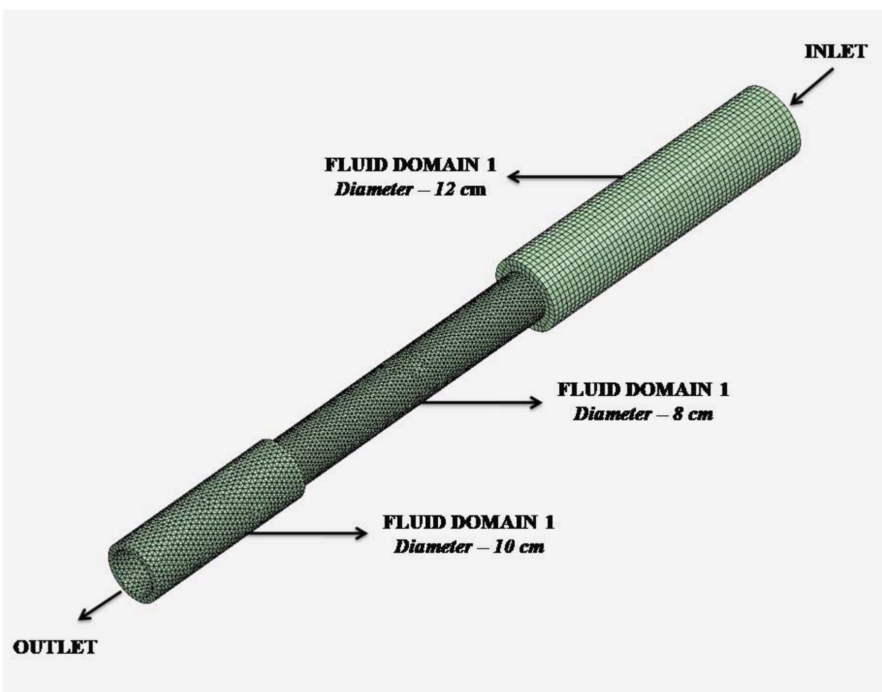

FIG 2: ANSYS Workbench Diagram of Setup

3.2 Methods

The experiment has been conducted to measure the velocity profiles at different points in the flow conduit – all the three fluid domains as mentioned in table 1. Initially, while setting up the apparatus, the different experimental zones or stations or check points have been marked and suitable provisions have been made for the study. For experimentation purpose, the velocities have been recorded in three different zones or check points for the three different fluid domains. Then, a pitot tube has been inserted at the different check points along the flow to measure the air flow velocities. Significant changes in the flow have been observed in the diverging and converging sections of the conduit. Assumptions were made while studying the flow – (1) Increase or decrease in the cross-section area shall lead to decrease or increase in pressure respectively *(ref. equation 1)* & (2) With reference to the continuity equation, we can infer that the velocity of flow at the converging sections shall be higher than the divergent sections. These assumptions lead to the set of conditions as discussed below:

$$A_1 > A_2 > A_3 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 1$$

$$A_2 < A_3 < A_1 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2$$

$$P_2 < P_3 < P_1 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 3$$

$$V_1 < V_3 < V_2 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 4$$

Where, A_1, A_2 and A_3, P_1, P_2 and P_3 & V_1, V_2 and V_3 are the area, pressure and velocities respectively at the different fluid domains of the experimental setup.

4. Experimental Results & Plots

This section would provide the detailed experimental results obtained through the real-time experiment performed through the setup as discussed above. Then, the experimental simulation and the related results obtained through ANSYS workbench shall be compared with the real time experimental data.

The velocities have been measured in the upstream flow direction of the conduit by the insertion of the Pitot tube and can further be demonstrated in the table 3:

Table 3: Velocities of Different Fluid Domains

Fluid Domain	Check Points	Velocity (m/s)
Domain 1	1	40
	2	42.04
	3	48.06
Domain 2	1	96.66
	2	100.12
	3	111.23
Domain 3	1	127.98
	2	143.21
	3	150.34

It is to be noted that these readings are being taken as a reference while validating with the ANSYS simulation study, as discussed in further sections. The figures 3-5 illustrate the velocity profiles at different fluid domains as obtained from table 3.

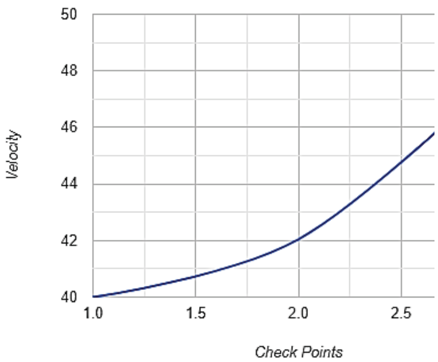

FIG 3: Fluid Domain 1

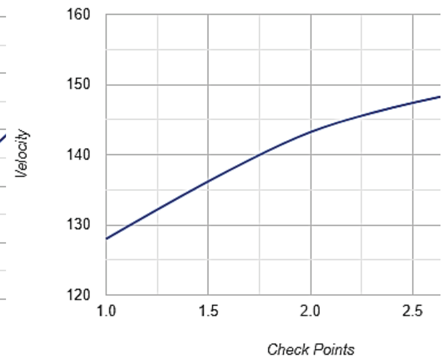

FIG 4: Fluid Domain 2

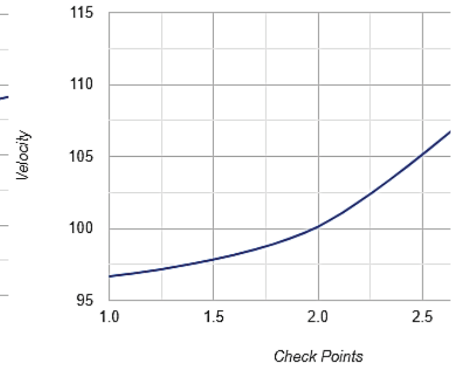

FIG 5: Fluid Domain 3

5 ANSYS Simulation

ANSYS is software where the fluid properties of a fluid flow in a conduit of any cross section can be studied in details with the highest precision. The ANSYS solves the flow through *Finite Element Method (FEM)* by considering each part of the device as an element in the form of dx. The area encompassing the device or fluid domain is then solved with the initial parameters taken as input.

5.1 ANSYS Design

Figure 6 depicts the exact design of the experimental setup in the workbench of ANSYS. The diameters have been taken as shown in table 1 and the material of construction has been specified to be of *fiberglass*.

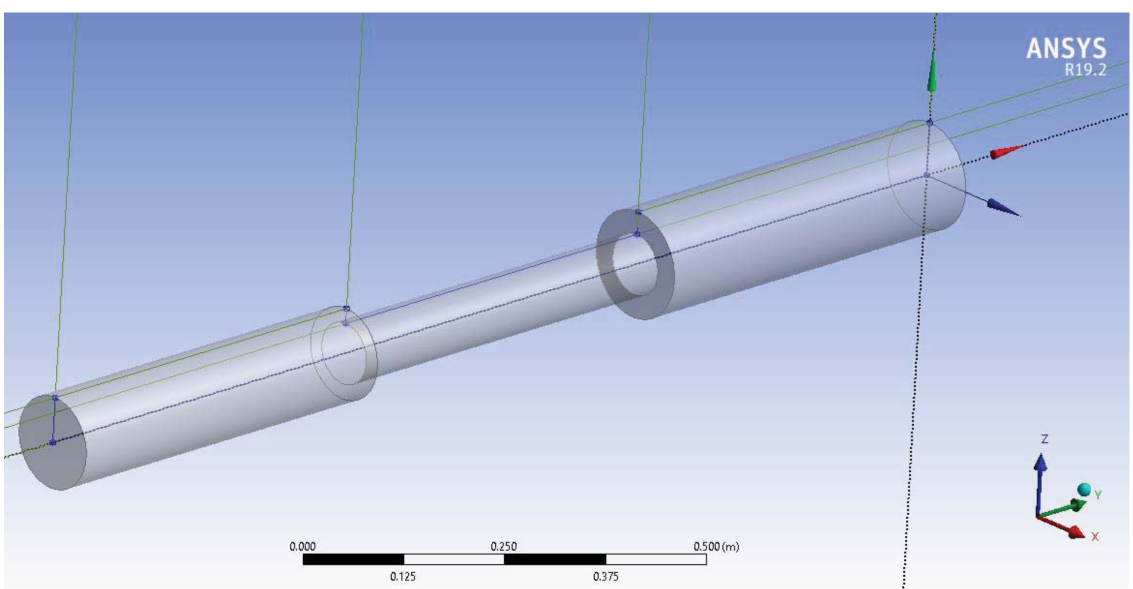

FIG 6: Design

5.2 Simulation Methodology

The simulation methodology can be divided into 3 steps and can be discussed in details – (1) Meshing – This is the process where we consider each part of the conduit as single elements. The more microscopic the meshing structure is, the better solutions would be generated. For this study, a meshing constraint of 2 e-3 has been considered with hex-dominant feature. Figure 7 demonstrates the meshing of the flow conduit. Here, the meshing has been performed by considering each element to be in the shape of a hexagon having 6 different nodes. Higher is the number of nodes; better shall be the meshing and corresponding solutions.

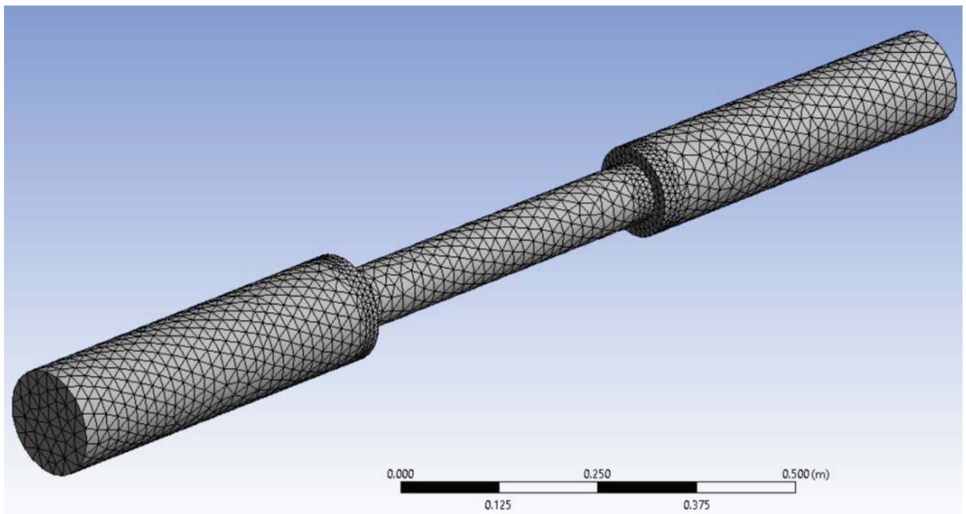

FIG 7: Mesh Generation

(2) Once the meshing is complete, the initial boundary parameters are set which are well discussed in table 1 and 2. In addition to that, the flow direction has been specified – that is, flow through the inlet as demonstrated in figure 2 and lastly in (3) simulation run, the entire flow analysis has been performed. Different velocity and pressure profiles have been studied in the flow conduit by varying parameters such as distance travelled by the fluid, density and other flow parameters.

5.3 Simulation Results

The primary simulation results obtained from ANSYS (equation 2) can be discussed as in figures 8-9 – velocity and pressure contours. From figure 8, the velocity profile can be well understood, where, the *Deep Blue* zones are referred as minimum velocity zones having a velocity range of 0-10 m/s, *Light Blue* zones which depict a slightly higher velocity with range of 35-55 m/s, *Green* zones which depict the transition zone from laminar to turbulent with a velocity range of 70-110 m/s and finally the *Red* zones, where the velocity is maximum at around 120-150 m/s. The red zones indicate the sudden increase in velocity due to sudden contraction in area. Due to sudden contraction in area, there will be a significant pressure drop due to which the velocities shall increase drastically. As soon as the air flow crosses the second fluid domain and enters the third fluid domain, the velocity of air is demonstrated to decrease as the resultant area has increased gradually. From the figure it can be inferred that in the third fluid domain, there are portions in the flow where the slip velocity or the velocity close to the surface is 0. This phenomenon is called the flow detachment due to turbulence and sudden increase in area.

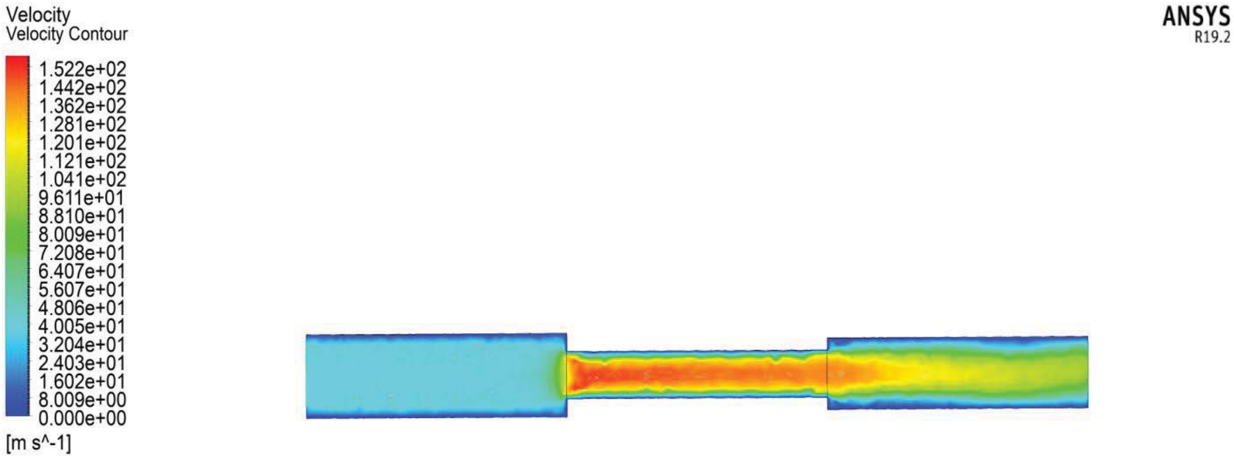

FIG 8: Velocity Contour

The figure 9 shows the pressure contour obtained through the study in ANSYS. This figure shall provide a better understanding of the reasons why the velocity has been changing over the different sections of the flow conduit. From the figure, it can be well-observed that the pressure seems to be in the *Red* zone when the air flow is entering into the conduit at a given velocity and then decreases over time. The high pressure zone is therefore having a range from -2 KPa to 1.8 KPa. The zones succeeding the high pressure are: *Yellow, Green* and *Light Blue*. These zones are having pressure ranges from -5 KPa to -3 KPa, -10 KPa to -6 KPa and -17 KPa to -13 KPa respectively. It can be inferred, that, as the velocity is decreasing, the pressure transition occurs from a higher value to a smaller value, thereby imposing significantly lower wall stress than in the previous fluid domains.

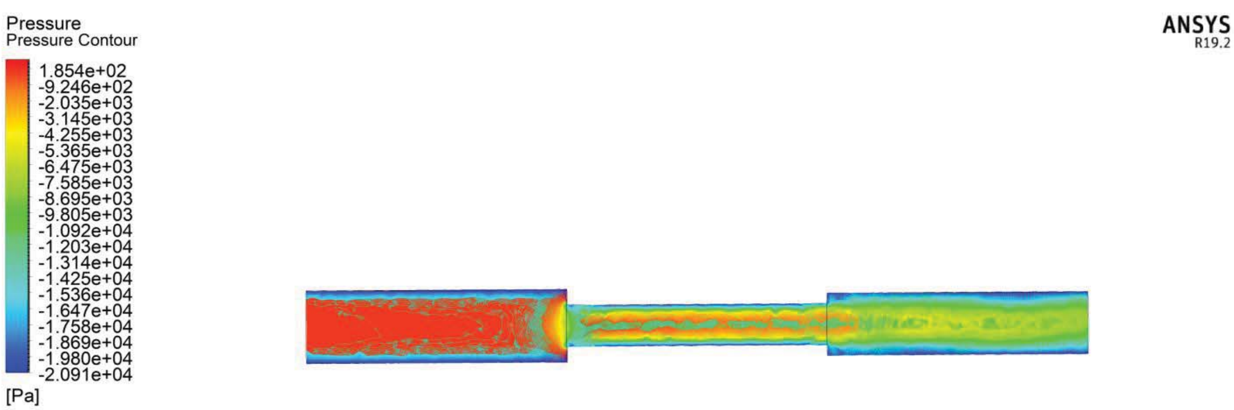

FIG 9: Pressure Contour

5.4 ANSYS Graphical study

In this section a detailed graphical study of flow characteristics have been demonstrated. The graphical study can again be divided into 3 categories - (1) Velocity vs Pressure, (2) Velocity vs Z (Z = distance travelled by the air) and (3) Pressure vs Z. The results as shown in the figures 10-12 are all software-generated results with the highest precision of 99.99% and with a tolerance level of around 1e-6.

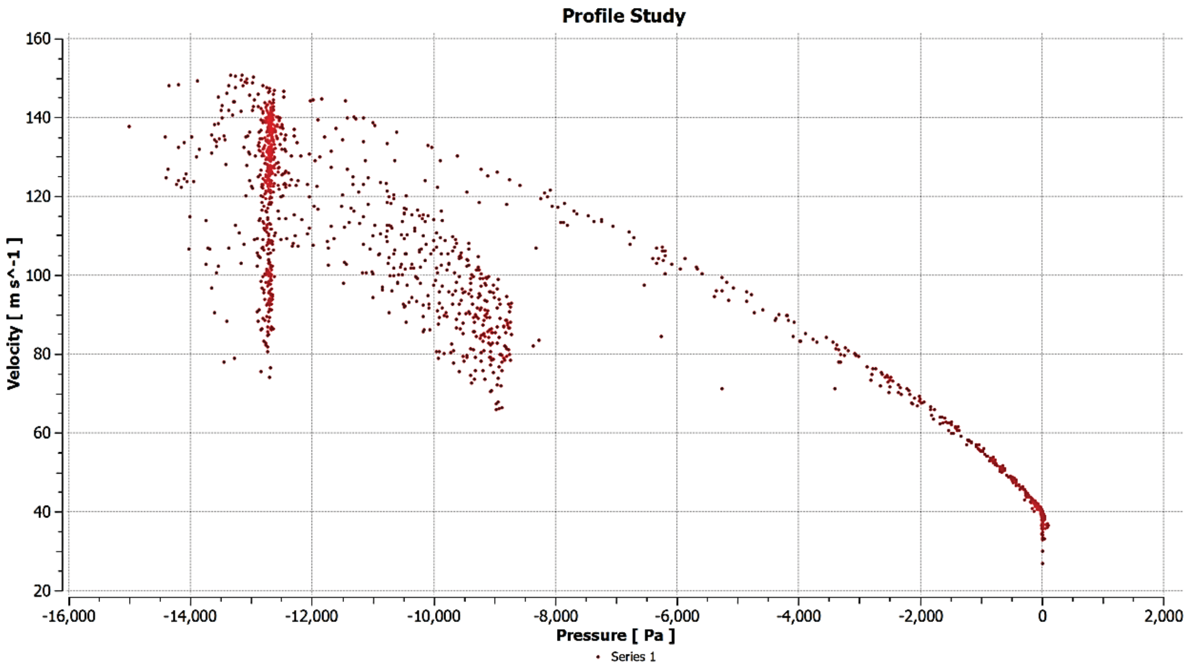

FIG 10: Velocity vs Pressure

As already discussed in previous sections – the air shall undergo changes in the fluid flow profiles as it travels along conduit. Figure 10-11 show the variation in the velocity with changes in the pressure inside the apparatus at different locations of the apparatus. It can be inferred that the velocity is highest in regions of lower pressure such as 150 m/s in -14 KPa and so on. The graph then shows a steep decrease in the velocity as the pressure is increased from -14 KPa to -2 KPa and the lowest velocity in regions of high pressure of around 1-2 KPa.

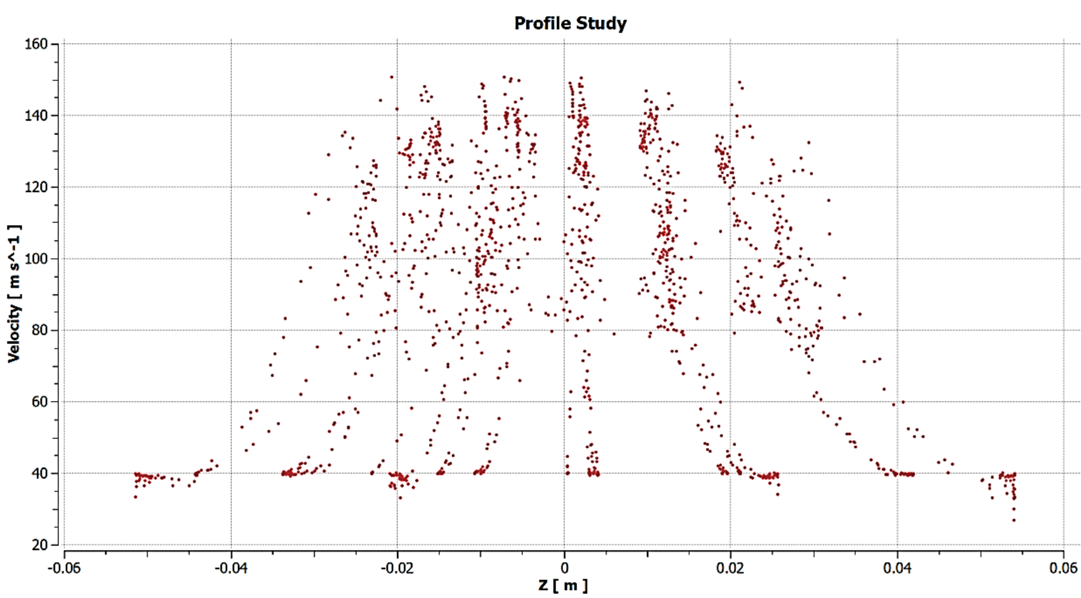

FIG 11: Velocity vs Z

Figure 12 shows the relation between the flow pressure and the distance travelled by the fluid (air) in the conduit. It can be inferred that along the flow conduit the pressure first increases (due to increase in pressure), then decreases (as the area decreases) and then again increases by a considerable amount (due to sudden expansion).

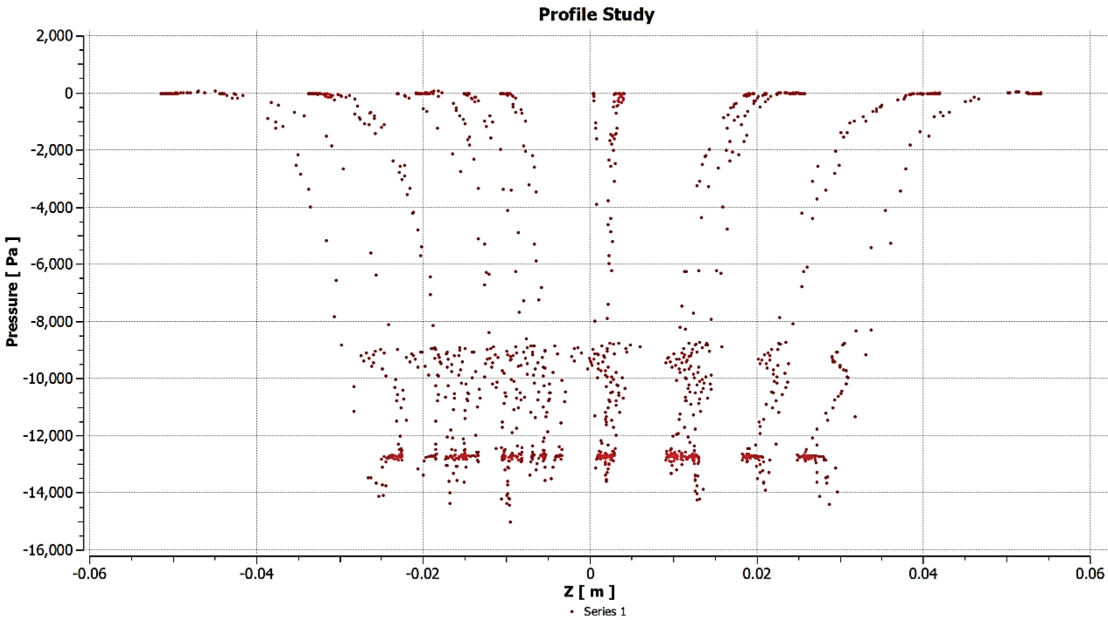

FIG 12: Pressure vs Z

The pressure change at each and every point has been calculated and mapped through the mathematical simulation runs by ANSYS. The figure 12 shows that the pressure change in the transition zones (0.02m to 0.04 m) have been significant from around -14 KPa to as high as 2 KPa.

6. Validation

The results obtained through the real-time experiment have been compared and validated with the ANSYS simulation results and can be viewed in the table 4. The validation has been demonstrated in the form of error percentages (ref. equation 3) to better understand the degree of compliance to the actual standard values.

Table 4: Estimation of Error

Domain	Experimental Results	Simulated Results	Error	Total Average Error
(no.)	(m/s)	(m/s)	(%)	(%)
1	40	40	0	1.01
	42.04	43.23	2.7	
	48.06	48	0.12	
2	96.66	97	0.35	
	100.12	100.52	0.39	
	111.23	112.45	1.1	
3	127.98	130.21	1.7	
	143.21	145.46	1.54	
	150.34	152.31	1.2	

The validation has been performed by keeping all the process parameters constant in simulation run as in the real-time experiment. The error percentages have been calculated and then averaged, bearing the *Total Averaged Error* (ref. equation 4). It can be inferred that as the error percentage is extremely low, around 1.01 %, the experimental results are perfectly in compliance with an accuracy reach of 98.99%. The minor losses encountered are mainly due to the presence of the wall friction factor or any potential physical impedance to the flow in the conduit.

7. Conclusion

The experimental results are in-line with the simulation results performed in ANSYS bearing an accuracy of around 98.99%. The research has demonstrated the different flow characteristics of an air travelling at a speed of 40 m/s through the inlet of the apparatus. The continuity phenomenon and the assumptions made have been proved to be true in concurrence with the different contours developed, as shown in the figures. The instances of flow separation have also been observed in the sudden expansion zones where the flow velocity at the wall surface is 0 and highest in the constricted passages. This study, therefore, allows us to investigate the different transient flow characteristics when air passes through different cross sectional areas having sudden expansions or contraction.

8. Acknowledgements

The authors would also like to extend their sincere and profound gratitude to the Jadavpur University for providing the laboratory facilities required for implementation of this work.

REFERENCES

[1] M. Tusar, K.Ahmed, M. Bhuiya, P.Bhowmik, M.Rasul, N.Ashwath,. . Energy Procedia, 160, 699 (2019).

[2] T.Kim, W. Aydin, T. B.Pereyra, E., & C. Sarica, International Journal of Multiphase Flow, 106, 75 (2018).

[3] N.Ihsak, M. E. M. A. Shaffie, M. A. A. M. Azam , M. A. Razali & amp, Journal of Complex Flow, 1(1) (2019).

[4] P. B. Dehkordi, A. Azdarpour, & E. Mohammadian , Chemical Engineering Research and Design, 139, 144(2018).

[5] M. Q Nguyen, M. S. Shadloo, A. Hadjadj , B. Lebon, & J. Peixinho, International Journal of Heat and Fluid Flow, 76, 187(2019).

[6] A.Naeyyf, Q.A Rishack, International Journal of Mechanical Engineering and Technology, 10(3) (2019).

[7] B.Lebon , J. Peixinho , S.Ishizaka, & Y.Tasaka, Journal of Fluid Mechanics, 849, 340(2018).