CFD Study of Elbow Draft Tube to Increase the Pressure Energy at the Outlet of the Reaction Turbine using Different Geometrical Configurations

Subhadip Bhattacharjee [1]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
Kalyani, India

Saroj Karmakar [2]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
Kalyani, India

Shuvam Gupta [3]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
Kalyani, India

Sudeb Saha[4]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
Kalyani, India

Prodyut Das [5]
Undergraduate Student,
Department of Mechanical Engineering
JIS College of Engineering
Kalyani, India

Dhiraj Mondal [6]
Assistant Professor,
Department of Mechanical Engineering,
JIS College of Engineering
Kalyani, India

Sudip Chakraborty [7]
Assistant Professor, Department
Of Mechanical Engineering
ADAMAS University
Barasat, India

***Abstract*— The purpose of this paper is to investigate the kinetic energy at turbine runner level of a novel draft tube by varying different geometrical configurations. On the basis of the study a suitable design of draft tube to produce maximum pressure has been predicted. Six different novel geometries have been taken for this analysis. Pressure, Velocity and Turbulence are calculated for the chosen profiles. The best profile based on the analysis is identified and increase in pressure is shown in this paper. The CFD analysis has been carried out by Fluid Flow (Fluent) Analysis in ANSYS'19.2.**

Keywords*— *Draft Tube, CFD, Modal Analysis, ANSYS.

I. INTRODUCTION

Draft Tube is a diverging tube which is fitted at the exit of turbine runner to efficiently utilize the available kinetic energy of fluid at the exit of the runner. It is an integral part of a turbine. It discharges the water smoothly from the turbine exit. It connects the exit of the turbine to the tailrace. As the draft tube is handling high pressure so it requires a robust structure. In industry cast steel and cemented concrete are used to make it.

The main purpose of a draft tube is to increase the pressure of fluid before joining it to the tailrace. It increases the pressure of leaving fluid at the expense of its velocity. It also resists the backflow of fluid into the turbine. In general a draft tube raises the fluid pressure upto atmospheric pressure.

So, here in this study we had change the geometrical configuration of a draft tube in terms of bend diameter and diffuser length to observe the pressure change. After the observation best suited profile is identified. As the draft tube is use to increase the pressure at turbine outlet in expense of velocity, for that the basis of this study is to find a suitable design for draft tube to produce maximum pressure at turbine outlet. Six different novel geometries are taken for the analysis. As an input, mass flow rate is consider as 20000 kg/s at the inlet of the Draft tube and Pressure, Velocity and Turbulence are calculated for the each profiles. On basis of the study the best profile is identified. For modelling ANSYS 19.2 Design modeller Geometry is used and the CFD analysis has been carried out by Fluid Flow (Fluent) Analysis in ANSYS 19.2. After the simulation the suitable draft tube geometry is predicted by the considering the output results.

Gunjan B Bhat et. al.," Design Automation and CFD Analysis of Draft Tube for Hydro Power Plant***",*** they had investigated that the efficiency of hydraulic turbine is significantly affected by its draft tube.[1] ***Tarang Agarwal et. al.,"*** Numerical and Experimental Analysis of Draft Tubes for Francis Turbine**",** they had observed that the efficiency of the turbine can be increased by increasing the overall-efficiency of the draft tube.[2] ***Spandan Chakrabarty et. al,"*** Numerical and Experimental Analysis of Draft Tubes for Francis Turbine, Indian Journal of Science and Technology***",*** they had observed that the efficiency of the draft tube is mainly depends upon the energy recovery in it and the energy recovery depends upon the design of the draft tube.[3] ***Jitendra Gupta et. al*.,"** A Review Paper on Design of Elbow Draft Tube for Unsteady Flow**",** they had find the optimum geometry by varying diffuser angle to increase the overall efficiency of elbow draft tube.[4] ***Mun Chol Nam et. al.,"*** Design optimization of hydraulic turbine draft tube based on CFD and DOE method**",** they had observed that the performance optimization of draft tube can be achieved by varying the size and shape of it.[5] ***Vishnu Prasad et. al.,"*** Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations using CFD***",*** they find optimum design of draft tube by varying different parameters like length and height at different mass flow rate.[6] ***Vishal Soni et. al.,"*** Design Development of Optimum Draft Tube for High Head Francis Turbine using CFD***",***They created various designs of bend type curved draft tube using conventional approach and their CFD simulations were carried out at Best Efficiency Point.[7] ***Umashankar Nema et. al.,"*** Design and Evaluation of Performance of Conical type Draft Tube with Variation in Length to Diameter Ratio***",***they varied the length of the draft tube for finding the best optimum length keeping the inlet diameter constant.[8]

II. PROBLEM STATEMENT

The draft tube is use to increase the pressure at turbine outlet in expense of velocity, thus in this study we will find a suitable design for elbow draft tube to produce maximum pressure at turbine outlet by varying the bend diameter and the diffuser length of six different novel geometries.

III. MATHEMATICAL MODELLING

The performance of the draft tube can be can be specified by the pressure recovery and the loss coefficient. The average pressure recovery factor can be represent as,

$$C_p = \frac{\frac{1}{A_3}\int_{A_3} pdA - \frac{1}{A_1}\int_{A_1} pdA}{\frac{1}{2}\rho V_{1av}^2}$$

Where $\boldsymbol{A}_1$ and $\boldsymbol{A}_3$ are the inlet and exit cross section area of the draft tube respectively, $\boldsymbol{V}_{1av}$ is the average flow velocity of the inlet of the draft tube and $\boldsymbol{\rho}$ is the density of the flowing water.

The draft tube efficiency can be represent as,

$$\eta_{dt} = \frac{(V_2^2 - V_3^2) - 2gh_d}{V_2^2}$$

V_2 = Fluids velocity at inlet of draft tube or at the outlet of turbine

V_3 = Fluids velocity at outlet of draft tube

g= gravitational acceleration

h_d = head losses in draft tube

IV. VALIDATION OF NUMERICAL SIMULATION

Mesh independence study has been the practice in this study to know the exact number of elements for which the output parameter will not change. By this verification, it can be concluded that our simulation outputs are stable and accurate.

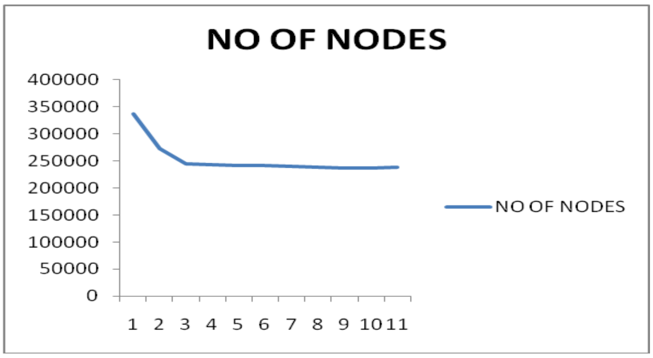

Fig 1. Mesh Independence Graph

V. ANALYSIS

For carrying out the simulation ANSYS '19.2 has been used and the steps involving in the research work are as follows,

A. Selection of Dimensions

At first, the dimensions showed below in the table are selected.

Table 1. Dimensions

CASES	BEND DIAMETER (IN METER)	DIFFUSER LENGTH (IN METER)
1	2	8
2	2	10
3	2	12
4	3.5	8
5	3.5	10
6	3.5	12

B. Modelling

Next, the Draft Tube is modelled. The modelling is done in "ANSYS 19.2 Design Modeller Geometry" software. The dimensions are as shown,

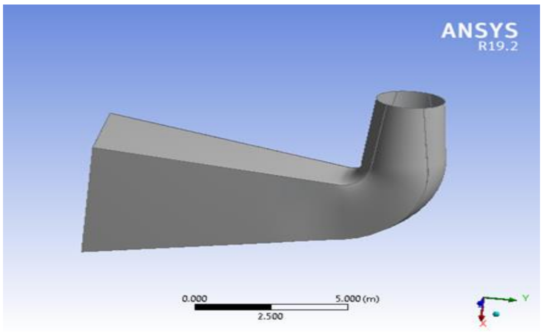

Fig. 2. Bending diameter 2m, Diffuser length 8m

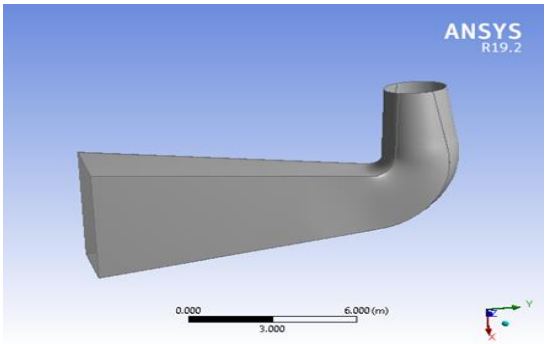

Fig. 3. Bending diameter 2m, Diffuser length 10m

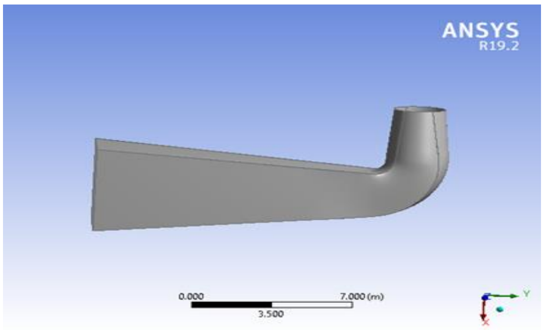

Fig. 4. Bending diameter 2m, Diffuser length 12m

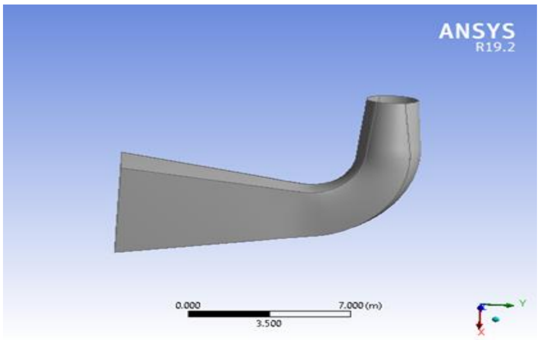

Fig. 5. Bending diameter 3.5m, Diffuser length 8m

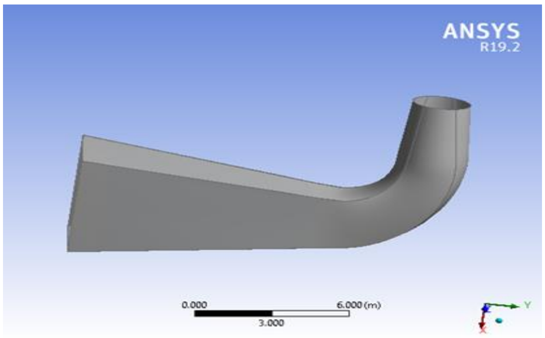

Fig. 6. Bending diameter 3.5m, Diffuser length 10m

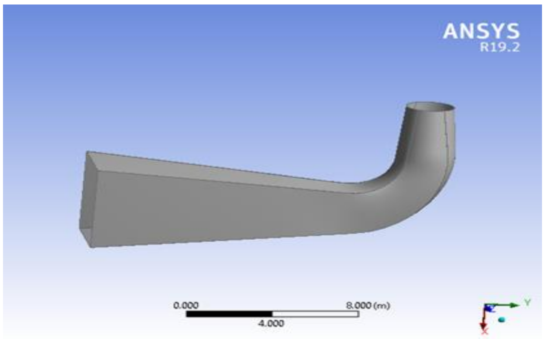

Fig. 7. Bending diameter 3.5m, Diffuser length 12m

C. Meshing

Now, the meshing is done on the Draft Tube geometries. Body Sizing Method and Face Sizing Method are used here for meshing.

Table 2. Mesh results

Cases	No. of Nodes	No. of Element	Body Sizing Element Size	Face Sizing Element Size	Edge Sizing no. of Division
1	258584	128551	0.1	0.1	500
2	107363	53647	0.1	0.1	500
3	111730	56064	0.1	0.1	500
4	103866	51886	0.1	0.1	500
5	111042	55534	0.1	0.1	500
6	111987	56066	0.1	0.1	500

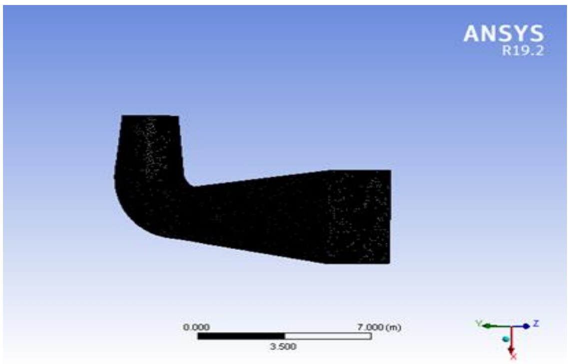

Fig. 8. Bending diameter 2m, Diffuser length 8m

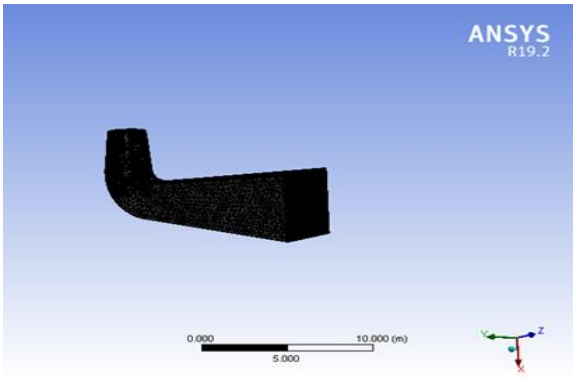

Fig. 9. Bending diameter 2m, Diffuser length 10m

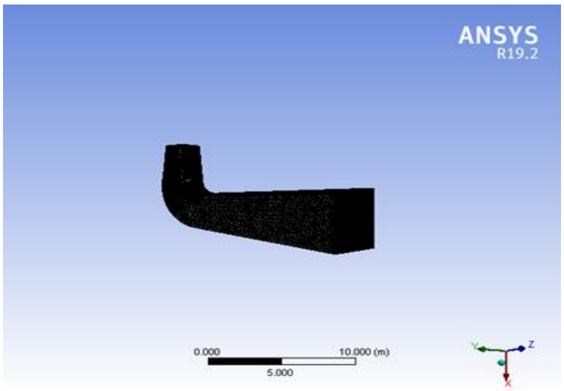

Fig. 10. Bending diameter 2m, Diffuser length 12m

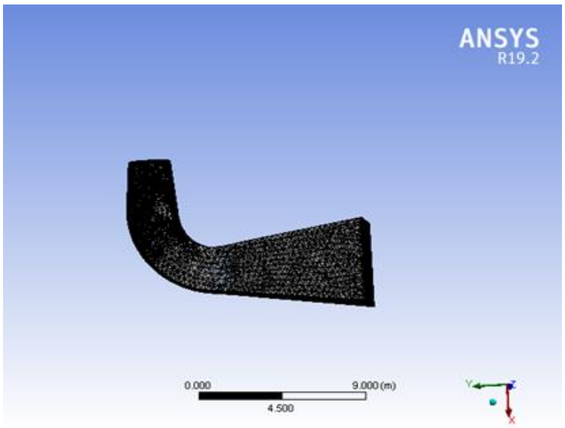

Fig. 11. Bending diameter 3.5m, Diffuser length 8m

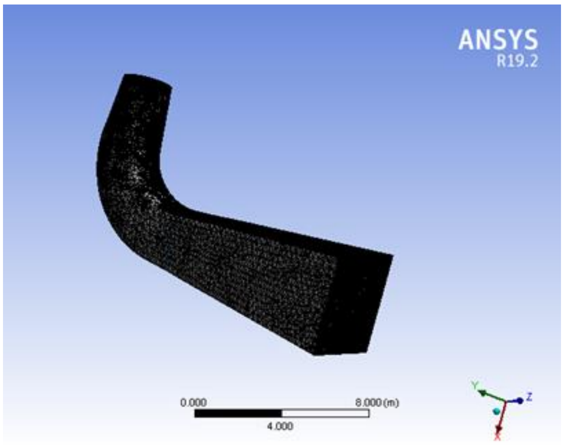

Fig. 12. Bending diameter 3.5m, Diffuser length 10m

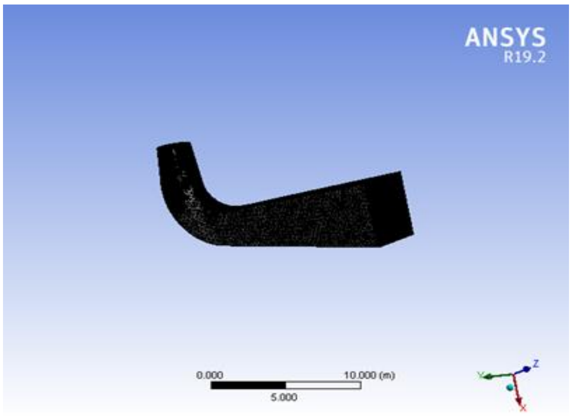

Fig. 13. Bending diameter 3.5m, Diffuser length 12m

D. Material Selection

Steel is chosen for draft tube material and water is considered as fluid for this study.

E. Simulation

After material selection, simulation is done on draft tube geometries (Fluid Flow Fluent Analysis**).** We take Mass flow rate at inlet as 20000 kg/s.

1) Simulation Results for Case 1

After meshing the draft tube geometry profile (bend diameter 2m and diffuser length 8m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below,

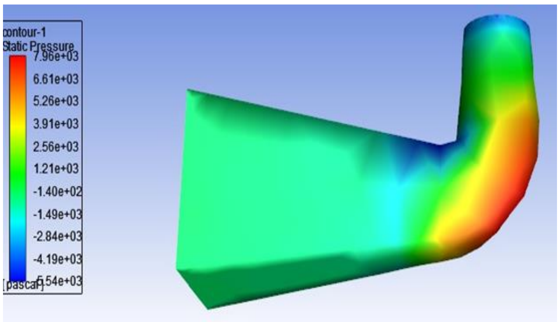

Fig. 14. Contour plot of Pressure

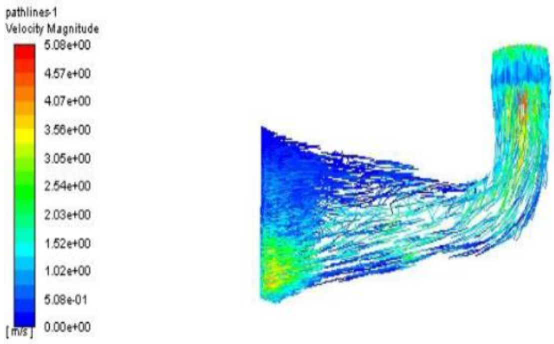

Fig. 15. Pathline plot of Velocity

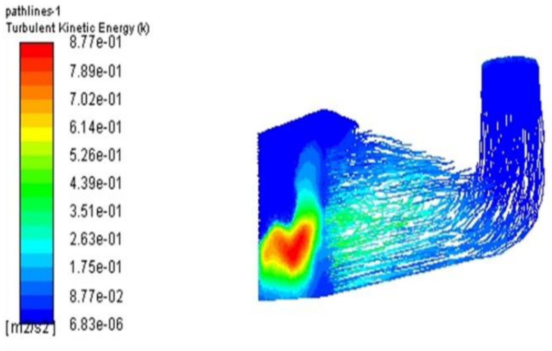

Fig. 16. Pathline plot of Turbulence

2) *Simulation Results for Case 2*

After meshing the draft tube geometry profile (bend diameter 2m and diffuser length 10m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below

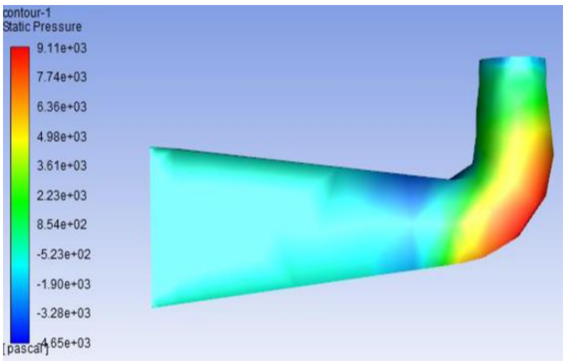

Fig. 17. Contour plot of Pressure

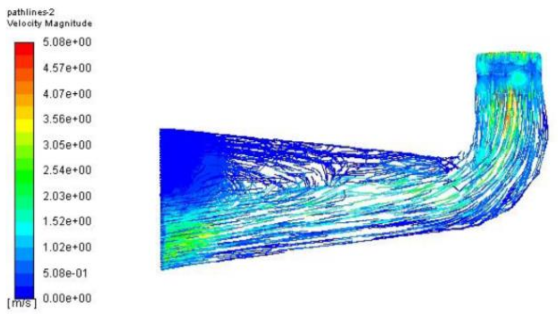

Fig. 18. Pathline plot of Velocity

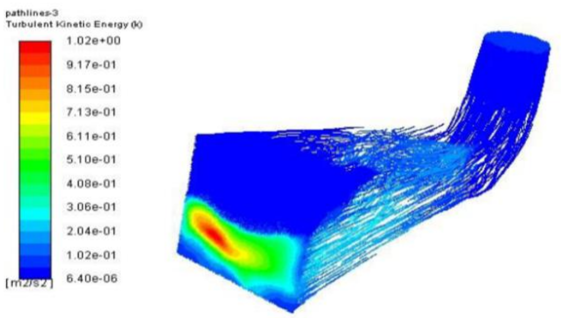

Fig. 19. Pathline plot of Turbulence

3) *Simulation Results for Case 3*

After meshing the draft tube geometry profile (bend diameter 2m and diffuser length 12m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below,

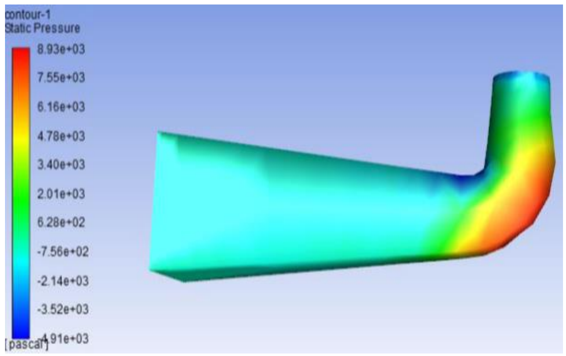

Fig. 20. Contour plot of Pressure

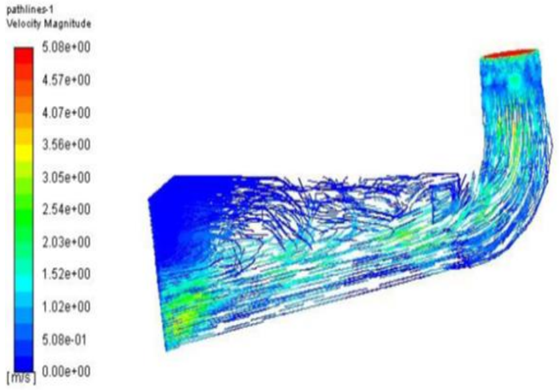

Fig. 21. Pathline plot of Velocity

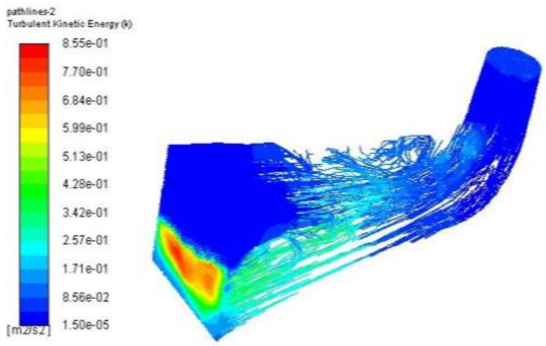

Fig. 22. Pathline plot of Turbulence

4) *Simulation Results for Case 4*

After meshing the draft tube geometry profile (bend diameter 3.5m and diffuser length 8m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below,

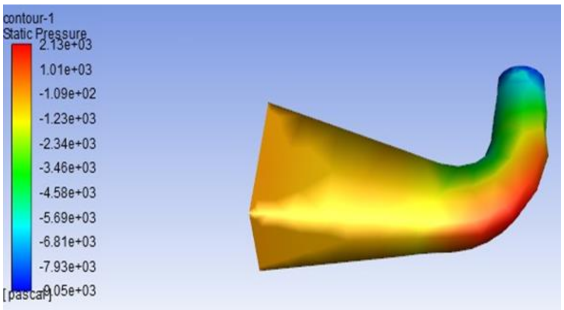

Fig. 23. Contour plot of Pressure

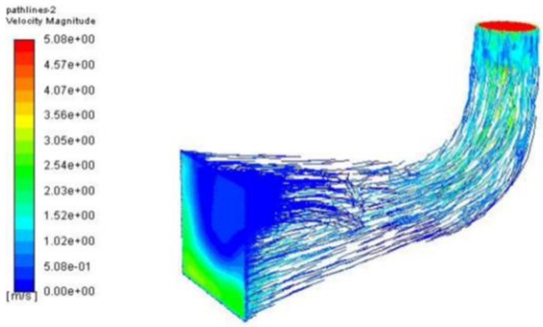

Fig. 24. Pathline plot of Velocity

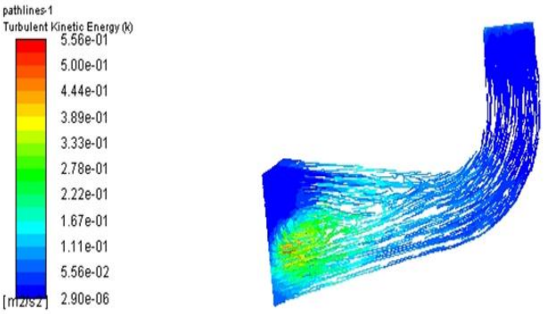

Fig. 25. Pathline plot of Turbulence

5) Simulation Results for Case 5

After meshing the draft tube geometry profile (bend diameter 3.5m and diffuser length 10m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below

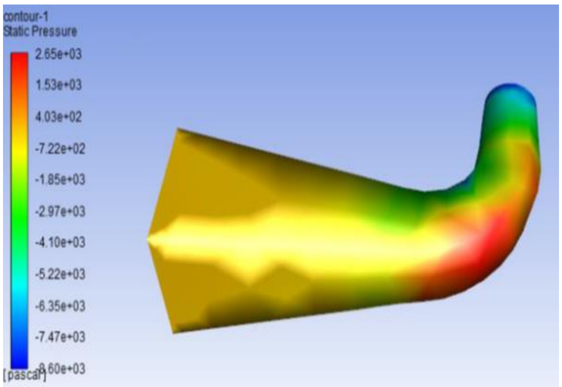

Fig. 26. Contour plot of Pressure

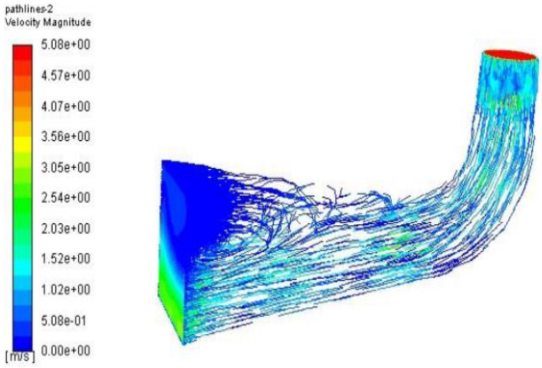

Fig. 27. Pathline plot of Velocity

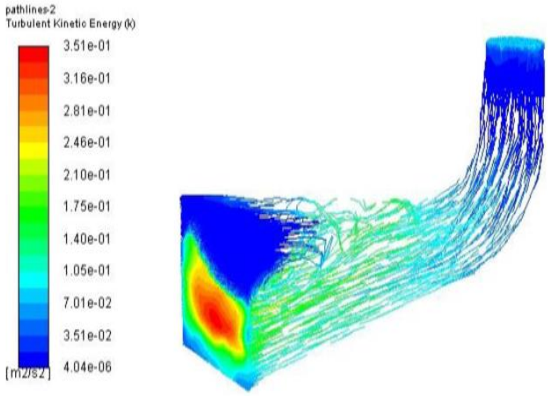

Fig. 28. Pathline plot of Turbulence

6) Simulation Results for Case 6

After meshing the draft tube geometry profile (bend diameter 3.5m and diffuser length 12m) is simulated. The simulation result (Pressure, Velocity and Turbulence) of the Draft Tube are given below,

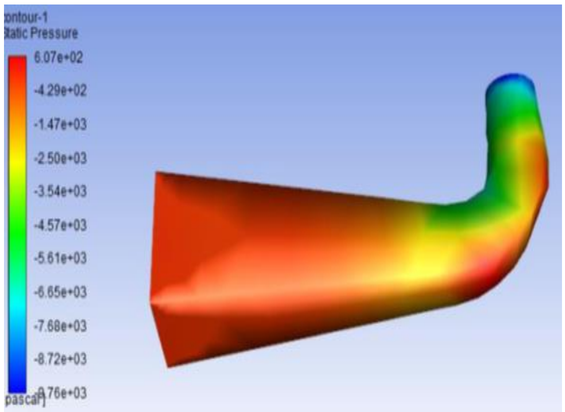

Fig. 29. Contour plot of Pressure

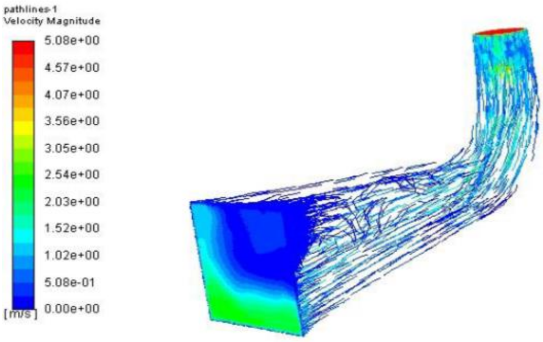

Fig. 30. Pathline plot of Velocity

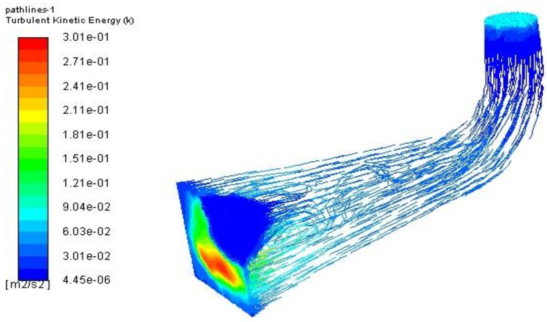

Fig. 31. Pathline plot of Turbulence

VI. RESULT AND CONCLUSION

After simulation of different geometrical profiles of Elbow draft tube, the simulation result (Pressure , Velocity and Turbulence) are given below:

Table 3. Simulation results

CASES	PRESSURE (in pascal)	VELOCITY (in m/s)	TURBULENCE (in m2/s2)
1	7.96E+03	5.08	8.77E-01
2	9.11E+03	5.08	1.02
3	8.93E+03	5.08	8.55E-01
4	2.13E+03	5.08	5.56E-01
5	2.65E+03	5.08	3.51E-01
6	6.07E+02	5.08	3.01E-01

Variation of pressure and turbulence with the geometrical cases of Draft Tube are represented in graphs:-

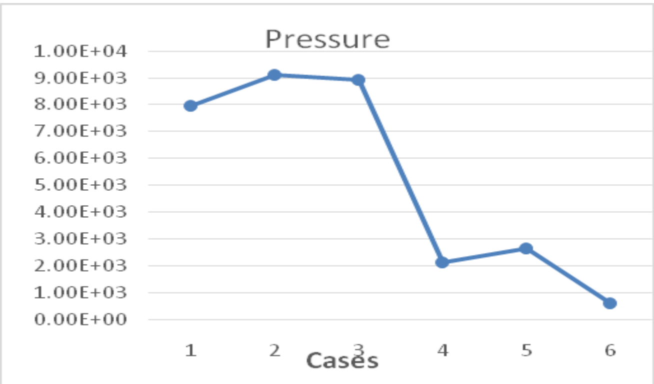

Fig. 32. Pressure (pascal) – cases

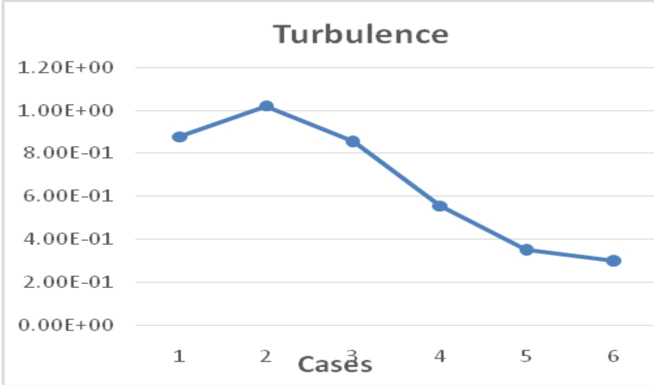

Fig. 33. Turbulence (m2/s2) – cases

- It has been found from the study that the pressure and turbulence of the draft tube depends on the bend diameter and the diffuser length.
- It is observed that between six geometry, geometry of case 2 achieved maximum pressure (bend diameter- 2m and diffuser length – 10m).

ACKNOWLEDGMENT

Apart from the efforts of us, the success of any project depends largely on the encouragement and guidelines of many others. We take this opportunity to express our gratitude to the people who have been instrumental in the successful completion of this project.

We would like to show our greatest appreciation to **Mr. Dhiraj Mondal** (Assistant Professor, Department of Mechanical Engineering, JIS College of Engineering) and **Mr. Sudip Chakraborty** (Assistant Professor, Department of Mechanical Engineering, ADAMAS University). We can't say thank you enough for their tremendous support and help. We feel motivated and encouraged every time we attend his classes. Without their encouragement and guidance, this project would not have materialized.

The guidance and support received from all the members who contributed and who are contributing to this project, was vital for the success of the project. We are grateful for their constant support and help.

This report has been prepared based on our own work. Where other published and unpublished source materials have been used, these have been acknowledged.

REFERENCES

[1] Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel, Design Automation and CFD Analysis of Draft Tube for Hydro Power Plant, Nirma University International Conference on Engineering(2015)
[2] Tarang Agarwal, Shreyash Chaudhary and Shivank Verma, Numerical and Experimental Analysis of Draft Tubes for Francis Turbine, Indian Journal of Science and Technology,(2017)
[3] Spandan Chakrabarty, Bikash Kr. Sarkar, Subhendu Maity, CFD Analysis of The Hydraulic Turbine Draft Tube to Improve System Efficiency, MATEC Web of Conferences (2016).
[4] Jitendra Gupta, Santosh Sahu, A Review Paper on Design of Elbow Draft Tube for Unsteady Flow, International Research Journal of Advanced Engineering and Science ISSN (Online): 2455-9024

[5] Mun Chol Nam, Ba Dechun, Yue Xiangji, Jin Mingri, Design optimization of hydraulic turbine draft tube based on CFD and DOE method, 6th International Conference on Power Science and Engineering (2017)
[6] Vishnu Prasad, Ruchi Khare, Abhas Chincholikar, Hydraulic Performance of Elbow Draft Tube for Different Geometric Configurations using CFD, AHEC, IIT Roorkee, India, IGHEM-2010
[7] Vishal Soni, Amit Roghelia, Jaymin Desai, Vishal Chauhan , Design Development of Optimum Draft Tube for High Head Francis Turbine using CFD, Proceedings of the 37th International & 4th National Conference on Fluid Mechanics and Fluid Power(2010)
[8] Umashankar Nema, Dr. Rohit Rajvaidya, Design and Evaluation of Performance of Conical type Draft Tube with Variation in Length to Diameter Ratio, Indian Journal of Engineering Sciences & Research Technology ISSN: 2277-9655 (2017)