iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 8 **Issue:** VIII **Month of publication:** August 2020

DOI: https://doi.org/10.22214/ijraset.2020.31086

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

Modeling and Analysis of Elbow Draft Tube for Different Geometric Configuration using CFD Simulation

Dikeshwar Patel[1], Dr. Devesh Shrivastava[2]
[1]M Tech, Mechanical Engineering Dept., BIT, Durg, India
[2]Associate Professor, Mechanical Engineering Dept., BIT, Durg, India

Abstract: The proficiency of a hydraulic reaction turbine is essentially influenced by the exhibition of its draft tube. The shape and speed dispersion at the bay are fundamental components which influences the exhibition of the draft tube. Customarily, the structure of this part has been based on improved analytical methods, test thumb rules and model tests. In the last decade or two, the use of computational liquid elements (CFD) has significantly expanded in the plan procedure and will keep on becoming due to is adaptability and cost-viability. A CFD-based structure search can additionally be supported with a hearty and easy to use advancement outline work hypothesis and building. In this paper, parametric modeling has been executed to adjust the elements of parts by methods for expert program to diminish geometric modeling time in the overhaul. The CFD examination has additionally been done for finding the pressure and velocity distribution, which are coordinating with test readings. The draft tube configuration has been adjusted dependent on writing and assembling potential outcomes.
Keywords: Elbow Draft tube, CFD,

I. INTRODUCTION

The inspiration behind the draft tube of a water turbine is to scale back the exit velocity with a minimum loss of energy. The draft tube changes over the dynamic pressure (kinetic energy) into static pressure. Not all energy will be recouped; the all out weight is diminishes through the diffuser because of misfortunes. Geometrically the draft tube is a genuinely basic gadget, a bowing funnel wandering in the stream astute bearing. Anyway, the dynamical procedures of the stream in a draft tube are exceptionally mind boggling and numerous shaky impacts have been watched.

All the structure of a hydropower framework, the draft tube is a significant segment that altogether influences both the effectiveness and cost, particularly in low-head frameworks. In view of the impacts on generally effectiveness, even a slight increment in execution could bring about a considerable vitality investment funds. Draft cylinders can be huge and costly; in this manner increasingly reduced plans offer the capability of lower cost. The ideal exchange off among productivity and cost requires an intensive information on diffuser execution. For customary frameworks, creators have a lot of understanding, however the opportunities for development is still there.

The elements of numerous huge frameworks just as normalized units have been distributed. These last structures are commonly the aftereffects of investigations, yet complete outcomes and execution of independent parts are restricted. The aftereffects of some methodical trials with the turbines have been distributed. Draft cylinders can be design in somewhat various jobs; the plan of the draft tube was represented by a couple hydro-mechanical standards with extraordinary thought of basic and constructional application.

A smoothed out shape with smooth ebb and flow in the elbow was excessively costly and tedious to assemble. It has been utilized in more current hydropower framework yet at the same time the plan is essentially founded on different contemplations than stream streamlining. Likewise sans cavitation activity is liked, and the sprinter has regularly a profound setting corresponding to tail-water. This is the reason the draft tube entrance is frequently beneath the tail-water, which is particularly valid for old structure.

There are various sorts of draft tube utilized in hydro power plants. Some of draft tubes ordinarily utilized are appeared in fig.1.

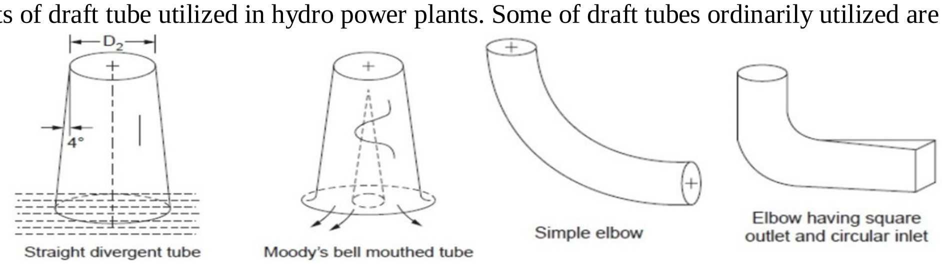


Figure 1: Various shapes of draft tubes

A. Draft Tube of work
1) The draft tube assumes a significant job on by and large execution of response turbine.
2) The effectiveness of draft tube relies upon its shape and other geometric boundary the most usually utilized draft tube is elbow draft tube with rectangular outlet.
3) The shape and region of draft tube at outlet just as along its length may improve the productivity of turbine.

B. Draft Tube's principle.
Bernoulli's Equation used for determining the principle of draft tube between section inlet 1-1 and outlet 2-2

$$\frac{p_1}{\rho g} + z_1 + \frac{V_1{}^2}{2g} = \frac{p_2}{\rho g} + z_2 + \frac{V_2{}^2}{2g} + h_f. \tag{1.1}$$

Where p is the absolute prressure, z is the height, V is the mean velocity and h_f is the hydraulic losses in the draft tube. The absolute pressure at section 2-2 can also be defined as

$$p_2/\rho g = z_2 + p_{atm}/\rho g, \tag{1.2}$$

Where p_{atm} is the atmospheric pressure. Assuming that the turbine is installed at height, H_s which is approximately equal to z_1, hence equation becomes,

$$\frac{p_1}{\rho g} = \frac{p_{atm}}{\rho g} - \left(H_s + \left(\frac{V_1{}^2}{2g} - \frac{V_2{}^2}{2g} - h_f\right)\right). \tag{1.3}$$

The equation 1.shows, under the runner, the draft tube generates a low pressure region, which can be utilized by the turbine. There are two terms associate lower pressure; static fall of pressure and dynamic fall of pressure, H_s and $V_1{}^2/2g$-$V_2{}^2/2g$-h_f respectively.

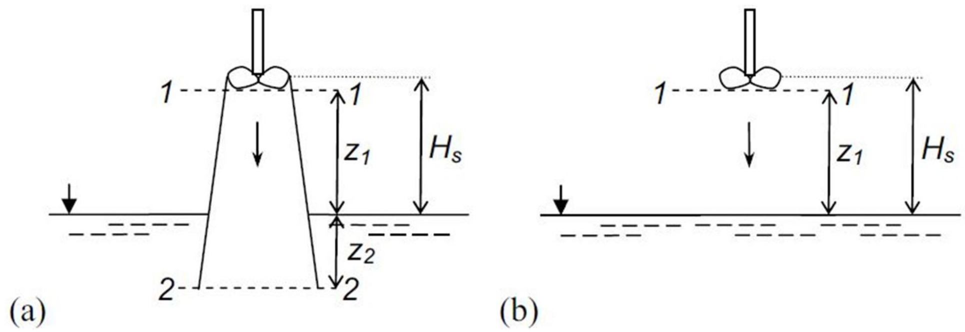


Figure 2: Hydraulic principles of draft tube (a) with; (b) without

The static fall of weight is autonomous of the release while the dynamic fall of weight increments with the stream rate. The dynamic fall of weight can without much of a stretch be limited by expanding the diffuser point or potentially by augmenting the length of draft tube. For keeping up the weight decrease beneath the sprinter maximal, an effective draft tube has ideal diffuser edge and length.

C. Types of Draft Tube
1) Straight conical
2) Curved draft Tube.
3) Bell Mouth with or without cone.

D. Elbow of Draft Tube
It is one of the most mind boggling components of the draft tubes where there is greatest event of water driven misfortunes. The misfortunes in the elbow of bended draft tube represent about 20 % or a greater amount of the all out water powered misfortunes in the draft tube. These misfortunes are brought about by the turning of the stream just as because of complex state of the elbow. The cross-area as a rule changes from roundabout to rectangular along the length. In hydroelectric stations we frequently run over circumstances where the outpouring diffuser is slanted towards the tail race of the station. In these cases the upper surface of the diffuser is flat or may even have a positive incline towards the tail race; the base of the diffuser likewise slants towards the tail race

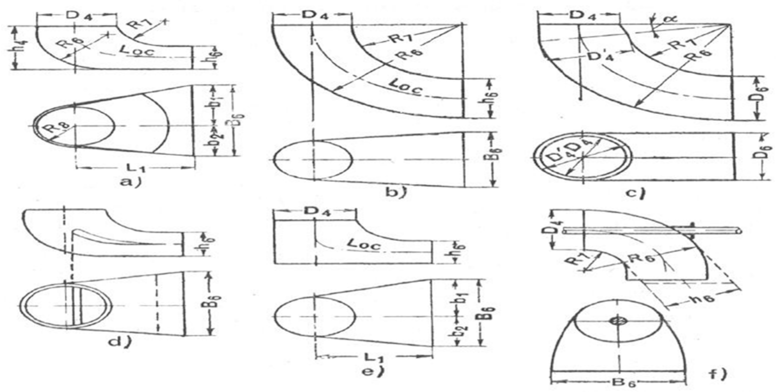


Figure 3: Various types of elbows used in draft tubes

E. Advantages of CFD

1) A test liquid element has assumed a significant job in approving the restrictions of different approximations to the administering conditions. The test rig is an exploratory arrangement to approve and reenact genuine streams yet now daily computational liquid elements praises trial and hypothetical liquid elements by giving a substitute financially savvy method for mimicking genuine stream. CFD is essentially less expensive and helpful than trial testing. It allows the numerical reproduction of hypothetical models and appropriate adjustments in it. CFD has following points of interest: Time required in design and development is significantly reduced.
2) CFD can simulate flow conditions which are not reproducible in experimental model tests.
3) CFD provides more detailed and comprehensive information.
4) CFD involves low energy consumption.
5) No scale up of problem required.
6) Less dependent on expert know how.

Traditionally, large lead times have been caused by necessary sequence of design, model construction, testing and redesign. Model constructions are often quite slow and costly. Using a well-developed CFD codes provides alternative designs to be run over a range of parameter values, e.g. Reynolds number, Mach number, Flow direction etc. CFD is very useful and effective in the early elimination of competing design and its configurations.

F. Applications of CFD

CFD can be utilized to deliver forecasts for a wide assortment of issues identified with liquid mechanics. In the event that we take a gander at an assortment of mechanical divisions, for example, aviation, protection, power, process, car, electrical and structural designing regions, CFD is presently being utilized. For instance, forecasts can be made:

In lift and drag of airplane. Here, as we have said engineers need the information for execution forecast.

1) Jet Flows inside atomic reactor corridors. Such issues include the reenactment of issue conditions, thus designs have incredible trouble in performing real tests, for wellbeing reasons. Consequently, calculation is the main method of attempting to see such streams.
2) Flows over Missiles: This is a region where there is requirement for lift, drag and side power information, with the goal that reenactment of execution can be made. Similarly as with airplane, CFD and air stream tests are utilized, but since of the wide scope of streams that must be mimicked for a given design, use is additionally made of semi – observational strategies, which are gotten from enormous measure of trial information.
3) Fame in Burners. There is have to comprehend the unpredictable communications between liquid stream and substance responses on fire. This can aid the creation of increasingly effective plans for burners in boilers, heaters and other warming gadgets
4) Air Flow inside interior burning motors. At the point when air is utilized to consume fuel inside an IC motor, be it a gas turbine motor or a petroleum motor or a diesel motor the air must be brought into the chamber with the base measure of exertion and the progression of air once it is in chamber must have the option to advance great consuming. Consequently, engineers need to realize the weight drop through a framework and the speed dissemination in the ignition chamber.

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

II. PROBLEM IDENTIFICATION

In this exploration our fundamental goal is to plan an elbow draft tube with changing diffuser point and to discover the upgraded model for the equivalent or with improved diffuser edge. So as to upgrade the structure philosophy as contrast with the ordinary draft tube plan, we are embracing the reproduction technique to determine the issue of draft tube structure and to lessen the cost of prototyping and to get the better outcomes by utilizing computational liquid examination in ANSYS 14.0 by CFX solver.

A. Computational Fluid Dynamics Analysis

CFD is a computational innovation that empowers to contemplate the elements of issues that streams. CFD is foreseeing what will occur, quantitatively, when liquids stream even with the inconveniences of synchronous progression of warmth, mass exchange, stage change, concoction response, mechanical development, and worries in and uprooting of submerged or encompassing solids. CFD incorporate articulations for the preservation of mass, force, weight, species and choppiness. Navier-Stokes condition given by Claude Louis Marie Henry Navier and the George Gabriel Stokes. Which characterizes any single-phase liquid stream is the key bases of all CFD issues. CFD programming depends on sets of complex non-linear numerical articulations that characterize the key conditions of liquid stream, warmth and materials transport. These conditions are comprehended iteratively utilizing complex PC calculations implanted inside CFD programming. Yields from CFD programming can be seen graphically in shading plots of speed vectors, forms of weight, lines of consistent stream field properties, or as "hard" numerical information and X-Y plots.

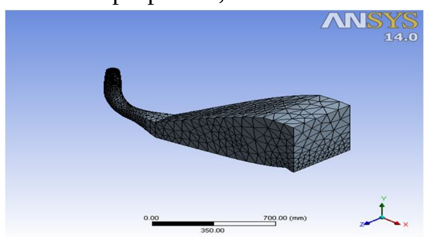


Figure 4**:** Elbow Draft Tube Meshing

For elbow draft tube, analysis have been applied in boundary condition inlet mass flow rate is given 80 kg/s and outlet boundary condition is given 1 atm as shown in figure-5 and figure-6.

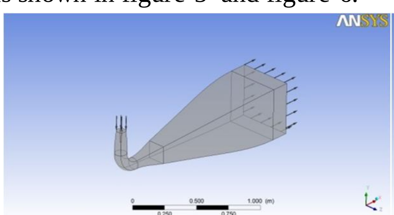


Figure 5: Inlet Boundary Conditions

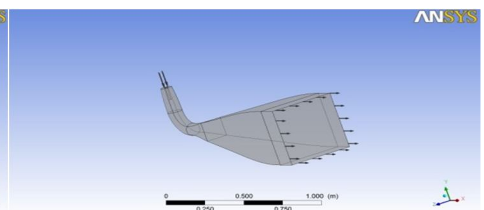


Figure 6: Outlet Boundary Condition

To get the result, next step is run the CFX solver. After the last step is analysis and visualize the result in post processor. the post processor, the pressure and velocity contours are visualized and process will be done.

III. RESULTS AND DISCUSSION

A. Validation with [Gunjan B. Bhatt et al's][1] ANSYS work and Experimental Reading

The pressure and velocity distribution are determined by ANSYS 14.0 CFX solver in the postprocessor stage. The outcomes for the velocity and pressure contour for the Base model (diffuser angle 20^0) Elbow draft tube as shown in figures 7 and 8 respectively.

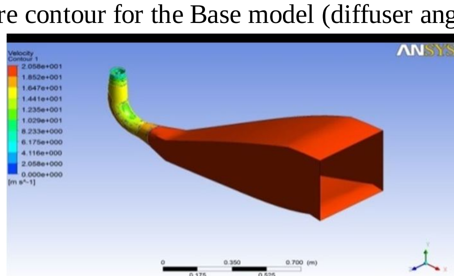


Figure 7

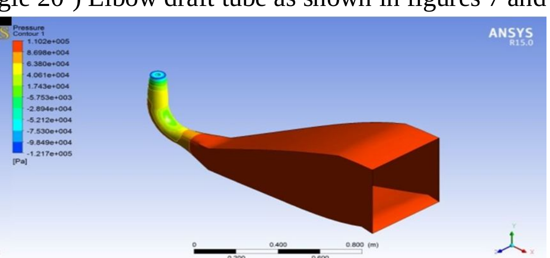


Figure 8

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

Present work in ANSYS (CFX) With Elbow and diffuser angle 20^0 is to be selected as Base model for our study and, For Elbow Draft Tube minimum Inlet Pressure-1.21×10^5 Pa and maximum Outlet Pressure1.10×10^5Pa are obtained by pressure Contour and the value of maximum outlet velocity 20.58 m/s is obtained by Velocity Contour in Base model for elbow draft tube with diffuser angle 20^0.

The pressure distribution at inlet and outlet of draft tube has been taken by experimental procedure and ANSYS work by referring [Gunjan B. Bhatt et.al]. The same results have been compared with Present work in ANSYS (CFX) by using Elbow draft tube and % Deviation in between Present and Experimental Reading has been originated as given in Table 1, which shows both results are in good agreement and satisfactory range with each other hence the design of elbow draft tube is state as given in Table 1.

Table 1.Comparison between the Present reading, ANSYS (CFX) & Practical Reading [Gunjan B. Bhatt et.al]

Comparison	Inlet Pressure	Outlet Pressure
Present work in ANSYS (CFX) With Elbow	-1.21×10^5 Pa	1.10×10^5 Pa
Gunjan B. Bhatt et.al ANSYS work (Without Elbow)	-2.100×10^5Pa	1.071×10^5Pa
Experimental Reading [Gunjan B. Bhatt et.al]	-1.99×10^5 Pa	1.12×10^5 Pa
% Deviation in between Present and Experimental Reading	2.51	1.78

B. Various Cases of Diffuser angle for Optimized Model of Elbow Draft Tube

The original shape of the draft tube is described by a number of traditional design parameters .A parametric study of this original shape is carried out by changing the traditional design parameters and using some powerful CFX geometry tools. The elbow geometry (alteration) i.e. the various diffuser angles with horizontal can easily be implemented by typical CFX geometry action.

(7) different cases are proposed for design Optimization of elbow draft tube with various diffuser angle, In which 20 degree diffuser angle is selected as a base model and it is termed as Case 3.2.1, the maximum outlet pressure 1.10×10^5 Pa and maximum outlet velocity 20.58 m/s are found respectively.

In Case- 3.2.2 design, with elbow draft tube (horizontal section) diffuser angle is modified by giving 0^0 diffuser angle as shown in figure. For this Case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure5.3 below.

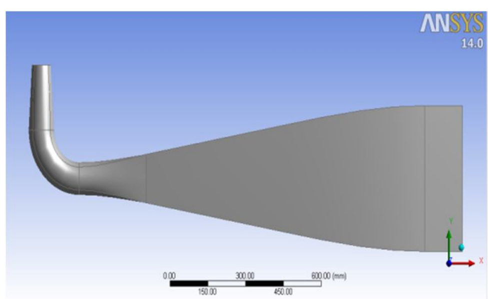


Figure 9: CFX Model for Case 3.2.2 (diffuser angle 0^0) Elbow Draft Tube

The below two figure 10 & 11 shows that the pressure and velocity distribution contour for Case- 3.2.2, in which the maximum outlet pressure 8.38×10^5 Pa and maximum outlet velocity 49.99 m/s are found respectively.

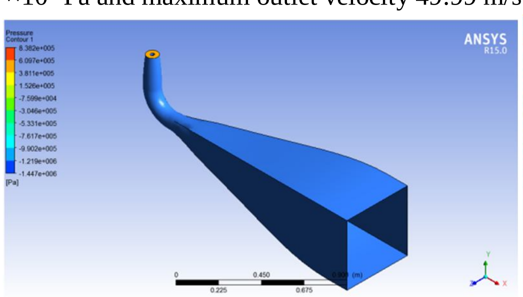


Figure 10:

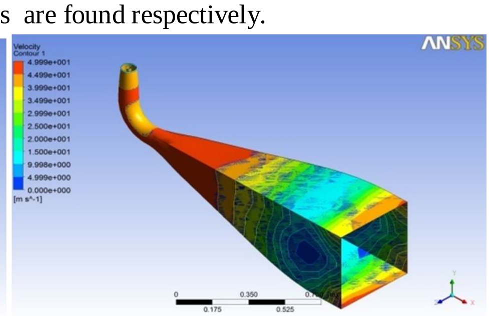


Figure 11

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 8 Issue VIII Aug 2020- Available at www.ijraset.com

In Case- 3.2.3 design, for elbow draft tube (horizontal section) diffuser angle is changed by giving 5 ^{0}.diffuser angle as shown in figure. For this case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure 12 and figure 13 below.

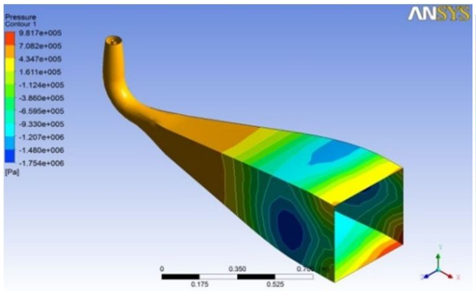


Figure 12

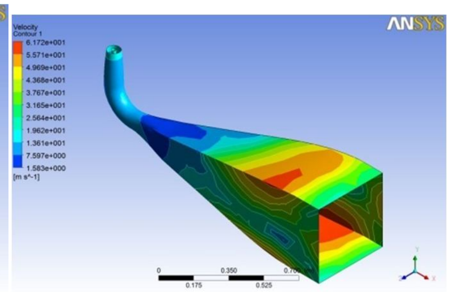


Figure 13

The above two figure shows that the pressure and velocity distribution contour for Case- 3.2.3, the maximum outlet pressure 9.82 $\times 10^5$ Pa and maximum outlet velocity 61.72 m/s are achieved respectively.

In Case-3.2.4 design, figure shows 10^0 diffuser angles. For this case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure14 and figure15 below.

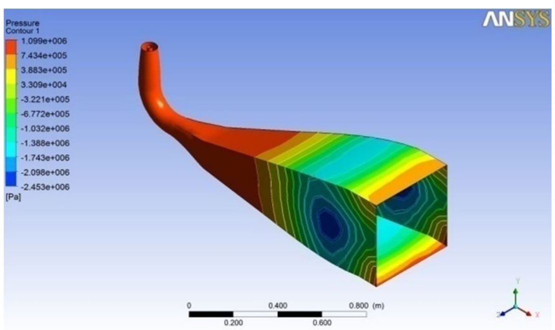


Figure 14

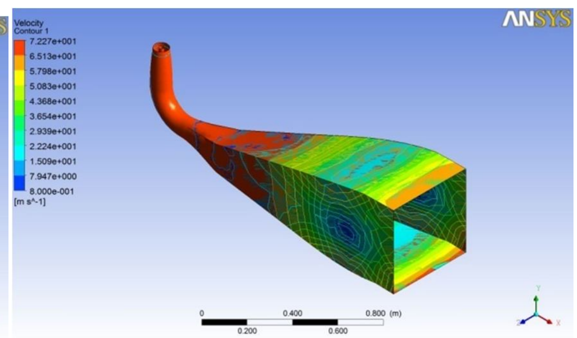


Figure 15

In figure 14 & 15 shows that the pressure and velocity distribution contour for Case-3.2.4, the outlet pressure 1.09×10^6 Pa and outlet velocity 72.27 m/s are achieved respectively.

In Case-3.2.5 design, diffuser angle is changed by giving 15 ^{0}. diffuser angle as shown in figure. For this case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure 16 and figure 17 below.

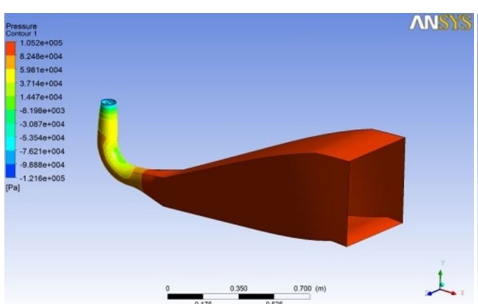


Figure 16

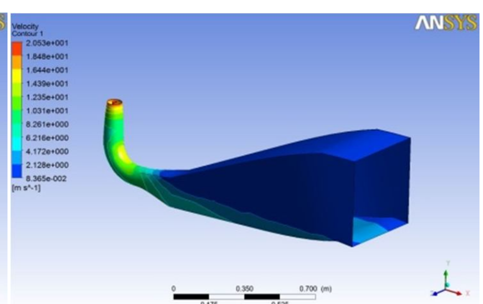


Figure 17

In figure 16 & 17 represents that the pressure and velocity distribution contours for Case-3.2.5, the maximum outlet pressure 1.05×10^5 Pa and maximum outlet velocity 20.53 m/s are achieved respectively.

In Case-3.2.6 design, with horizontal section of elbow draft tube, diffuser angle is changed by giving 25 ^{0} diffuser angle as shown in figure. For this case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure 18 and figure 19 below.

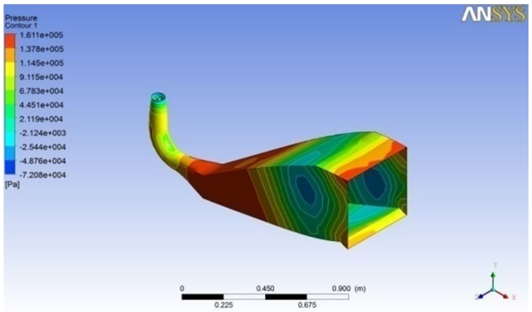


Figure 18:

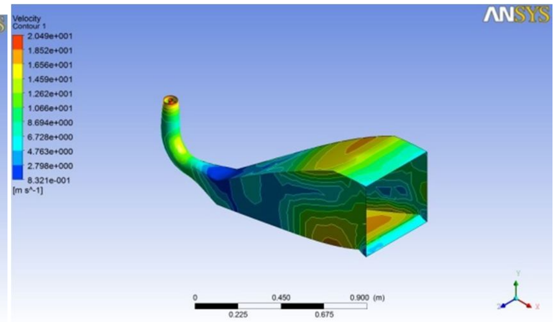


Figure 19:

In figure 18 & 19 shows that the pressure and velocity distribution contour for Case-3.2.6 , the maximum outlet pressure 1.61×10^5 Pa and maximum outlet velocity 1.61×10^5 Pa and 20.49 m/s are achieved respectively.

In Case- 3.2.7 design, with horizontal section of elbow draft tube, diffuser angle is greater by giving 30^0 diffuser angles as shown in figure. For this case CFD analysis (ANSYS 14.0, CFX) is carried out by same procedure to determine pressure and velocity distribution as given in figure below.

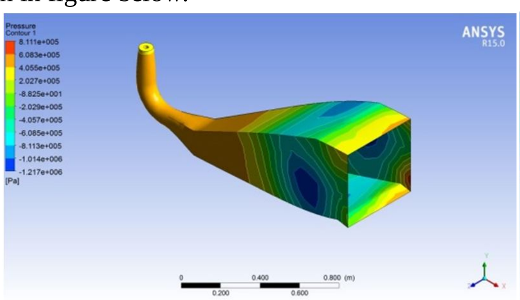


Figure 20

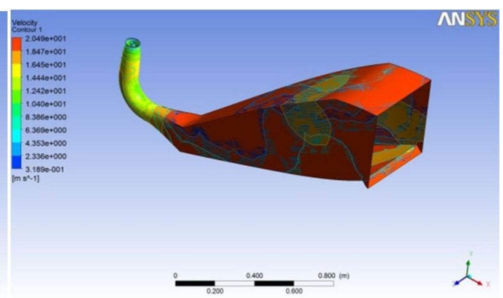


Figure 21

In figure 20 & 21 shows that the pressure and velocity distribution contour for Case-3.2.7, the maximum outlet pressure 1.17×10^5 Pa and maximum outlet velocity 20.49 m/s are achieved respectively.

To enhance maximum pressure at outlet Using CFD analysis different Cases (seven case) of elbow draft tube are suggested as a part of design optimization in ANSYS 14.0 CFX, for the same boundary conditions pressure distribution of seven case are determined. The maximum and minimum outlet and inlet pressure and velocity for each cases are given in Table 2 and Table 3 resp.

Table 2: Pressure at inlet for Different Cases

Pressure	Case 3.2.2 (0^0)	Case 3.2.3 (5^0)	Case 3.2.4 (10^0)	Case 3.2.5 (15^0)	Case 3.2.1 (20^0) BASEMODEL	Case 3.2.6 (25^0)	Case 3.2.7 (30^0)
Maximum (inlet) in Pa	8.38×10^5	9.82×10^5	1.09×10^6	1.05×10^5	1.102×10^5	1.61×10^5	8.11×10^5

Pressure	Case 3.2.2 (0^0)	Case 3.2.3 (5^0)	Case 3.2.4 (10^0)	Case 3.2.5 (15^0)	Case 3.2.1 (20^0) BASEMODEL	Case3.2.6 (25^0)	Case 3.2.7 (30^0)
Minimum (inlet) in Pa	-1.44×10^6	-1.75×10^6	-2.45×10^6	-1.21×10^5	-1.217×10^5	-7.2×10^4	-1.217×10^5

.Table 3: Velocity at Outlet and Inlet for Different Cases

Velocity	Case 3.2.2 (0^0)	Case3.2.3 (5^0)	Case 3.2.4 (10^0)	Case 3.2.5 (15^0)	Case 3.2.1 (20^0) BASEMODEL	Case 3.2.6 (25^0)	Case 3.2.7 (30^0)
Maximum (Outlet) in (m/s)	49.99	61.72	72.27	20.53	20.58	20.49	20.49

IV. CONCLUSION

A. To determine pressure & velocity profile at inlet and outlet conditions on elbow draft tube, the optimization and Computational Fluid Dynamics CFD (ANSYS 14.0, CFX) analysis has been performed. This analysis can be help to reduce the higher cost experimentation.

B. For draft tube to improve efficiency and pressure at outlet, seven cases have been proposed.

C. With the same boundary conditions, the same analysis has been performed for different cases. The improved value data of maximum pressure and velocity is achieved in Case- 5.2.4 gives maximum outlet pressure as per analysis results compared to all other cases like base model (Case 5.2.1.)

D. Percentage deviation in between present work (ANSYS 18.1 CFX) and experimental reading is inlet pressure 2.51% and outlet pressure 1.78 % achieved, which shows both results are in good agreement and acceptable range with each other.

E. The value of optimized maximum outlet pressure and velocity is 1.09×10^6 Pa and 72.27 m/s respectively.

F. The optimized model result has been found to obtained for maximum outlet pressure and velocity with Case-5.2.4, 10^{0} diffuser angle with horizontal is achieved as compare to base model (Case-5.2.1, 20^0 diffuser angle.)

G. In comparison of velocity contours for different cases up to certain angle exit velocity increases and suddenly it decreases and retains its velocity for all the next cases considered therefore the exit velocity will be consistent and very low with further angle of bend if considered.

H. At inlet of draft tube pressure should be negative so that loss of head can be used due to the height of the draft tube but with respect to temperature of water more vacuum will lead to cavitation which is not desirable for any draft tube so the above cases of draft tube bend angle will prevent cavitation..

I. Hence it may be concluded that CFD analysis is very effective tool for numerical flow simulation in complex flow fields with equitable accuracy.

V. ACKNOWLEDGEMENT

It is pleasant task to express my gratitude and respect to all those who contributed in many ways to the success of this research work and made it an unforgettable experience for me. First of all, I indebted to Dr. Devesh Shrivastava, Associate Prof. of Mechanical Engineering Department, Bhilai Institute of Technology, Durg (C.G.) for her expertise, painstaking guidance, farsightedness, strong will and her positive and simple approach to solve every kind of problem.

REFERENCES

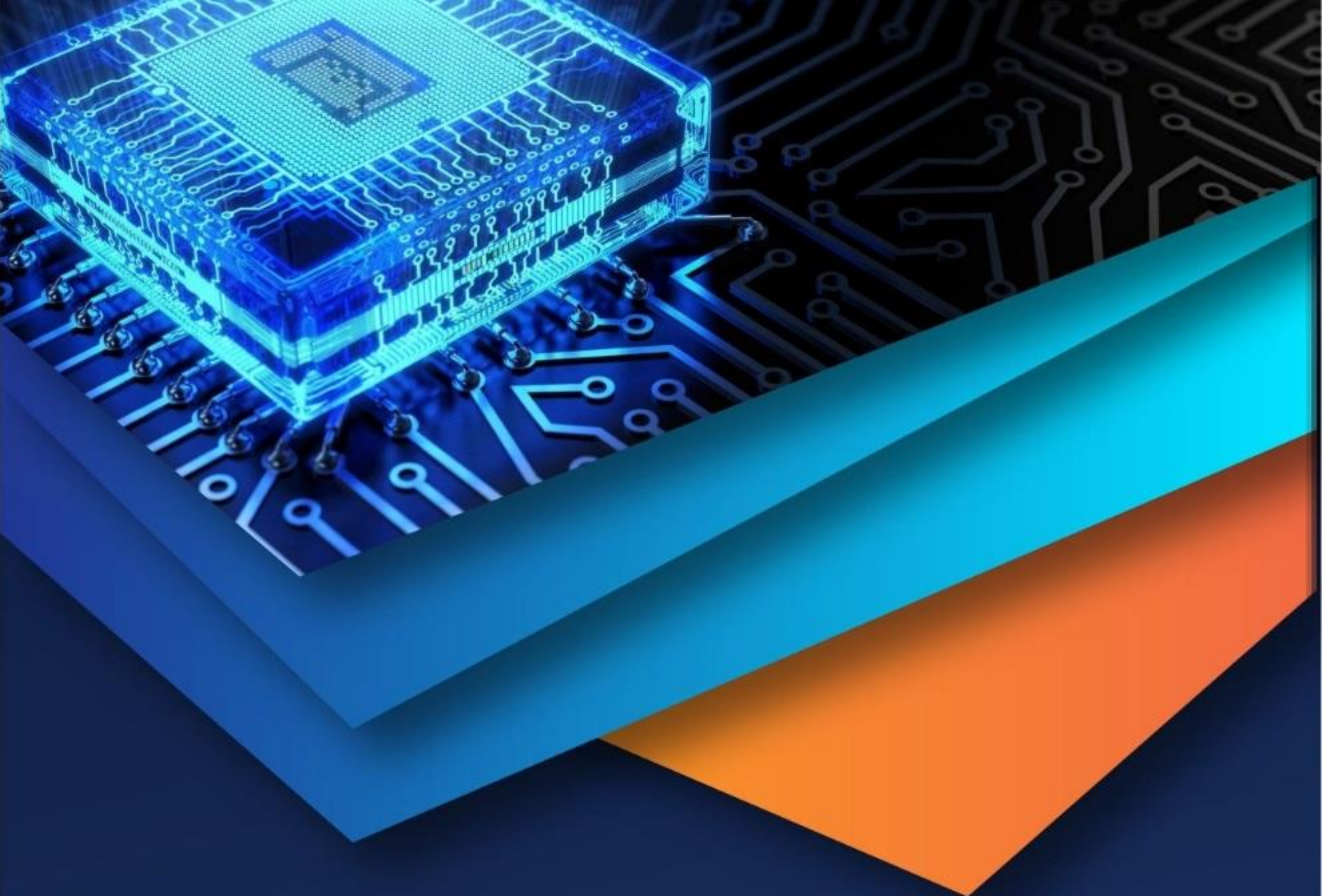
[1] Gunjan Bhatt, Dhaval B. Shah, Kaushik M. Patel, “Design Automation and CFD Analysis of Draft Tube For Hydro Plant.” International Journal Of Mechanical And Production Engineering, ISSN-2320-2092 ,Volume 3 Issue-6 2015.
[2] [2] Anderson U., 2009. “Experimental study of Sharp Heel Draft Tube.” Thesis (PhD), Lulea University of Technology, ISBN: 978-91-86233-68-6, Sweden.
[3] [3]RuchiKhare, Vishnu Prasad, Sushil Kumar Mittal, 2012. “Effect of runner solidity on performance of elbow draft tube” Proceedings of the 2nd International conference on advances in energy engineering (ICAEE), Energy procedia 14, page 2054-2059.
[4] [4] Vishnu Prasad, RuchiKhare, AbhasChincholikar , 2010. “Hydraulic Perfomance of Elbow Draft Tube for Different Geometric Configurations Using Cfd”. IGHEM -2010,Oct.21 -23,AHEC,IIT Roorkee India.
[5] [5] UpendraRajak , Vishnu Prasad ,Ruchi Khare,2013.”Numerical flow simulation Using Star CCM+.“ IISTE Vol.3 ,No.-6,2013.
[6] [6] Siake A , Koueni-Toko C ,Djemako B.,Tcheukam-Toko,2014.Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine.IJHE-2014:103-114.
[7] [7] Jiri Obrovsk, 2013. Development of high specific speed Francis turbine for low head,
[8] Engineering Mechanics, Vol. 20, No. 2, p. 139–148.
[9] [8] Christopher, B. Cook, Marshall C. Richmond, John A. Serkowski, (2007),"Observations of Velocity Conditions Near a Hydroelectric Turbine Draft Tube Exit Using ADCP Measurements", Flow Measurement and Instrumentation, Vol. 18 , pp.148–155.
[10] [9] B.G. Mulu, P.P.Jonsson, M.J. Cervantes, (2012), "Experimental Investigation of a Kaplan Draft Tube-Part I Best Efficiency Point", Applied Energy, Vol. 93, pp.695 - 706.
[11] [10]Michihiro Nishi and Shuhong Liu “An Outlook on the Draft-Tube-Surge Study” ISSN (Online): 1882-9554International Journal of Fluid Machinery and Systems Vol. 6, No. 1, January-March 2013.

[12] [11]Thi C. Vu, Christophe Devals, Ying Zhang, Bernd Nennemannand François Guibault
[13] "Steady and unsteady flow computation in an elbow draft tube with experimental validation."
[14] International Journal of Fluid Machinery and Systems Vol. 4, No. 1, January-March 2011 ISSN (Online): 1882-9554
[15] [12] I.Gunnar , J. Hollestrom,"Redesign of Existing Hydro Power Draft Tube.ISSN 1402-1617,Sweden.
[16] [13] Coelho J. G., Brasil Junior A. C. P.,"Numerical simulation of draft tube flow of a bulb turbine."International Journal of Energy and Environment Vol.4, Issue 4, 2013 pp.539-548.
[17] [14] Dr. RuchiKhare, Dr. Vishnu Prasad, MitrasenVerma, Design Optimization of Conical Draft Tube of Hydraulic Turbine. International Journal of Advances in Engineering, Science and Technology (IJAEST) Vol. 2 No. 1 Mar-May 2012.
[18] [15]DheerajSagar, TarangAgarwal, ShubhamBhatnagar, Recapitulation of Draft Tubes. International Journal of Advance Research In Science And Engineering IJARSE, Vol. No.4, Special Issue (01), February 2015 ISSN-2319-8354(E)
[19] [16]Marjavaara B. D., 2006."CFD driven Optimization of Hydraulic Turbine Draft tubes Using
[20] Surrogate models." Thesis (PhD), Lulea University of Technology ISSN: 1402-1544, Sweden.

A Brief Author Biography

***Dikeshwar Patel* –** I am a student of M. Tech in BIT, Durg. I have completed my B.E. in Mechanical Engineering in 2014 from BCET, Bhilai, India. My interested topics are production engineering, designing and automobile.

***Dr.Devesh Shrivastava* –** I am currently working as Associate Professor, Mechanical Engineering Department, BIT, Durg, Chhattisgarh, India. I have completed BE in Mechanical Engineering in 2006, PT.R.S.U, Raipur and ME in Production Engg. 2011, C.S.V.T.U., Bhilai, Chhattisgarh, India. Have awarded PhD in Mechanical Engineering from Dr. C V Raman University, Bilaspur, 2019 . I have 12 years of teaching experience and research experience and has published more than 13 research papers in leading National and International conferences. My areas of research interest include industrial engineering & management, production & materials management, production engineering, manufacturing processes.



10.22214/IJRASET

INDEX COPERNICUS

45.98

ISRA JIF

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

ISI