# Design Modification of Draft Tube for Hydro Power Plant using CFD Analysis

**Gunjan B. Bhatt[1] , Dhaval B. Shah[2] and Kaushik M. Patel[3]**

## Abstract

The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. The shape and velocity distribution at the inlet are main factors which affects the performance of the draft tube. Traditionally, the design of this component has been based on simplified analytical methods, experimental thumb rules and model tests. In the last decade or two, the usage of computational fluid dynamics (CFD) has dramatically increased in the design process and will continue to grow due to is flexibility and cost-effectiveness. A CFD-based design search can further be aided with a robust and user-friendly optimization frame work theory and engineering. In this paper, parametric modeling has been implemented to modify the dimensions of components by means of pro-program to reduce geometric modeling time in the redesign. The CFD analysis has also been carried out for finding the pressure and velocity distribution, which are matching with experimental readings. The draft tube design has been modified based on literature and manufacturing possibilities. Five different concepts for draft tube model are proposed and compared to achieve maximum efficiency and pressure at outlet based on CFD analysis.

**Keywords:** CFD analysis, draft tube, pressure distribution, streamline.

## Introduction

The utilization of the hydraulic force in the domain of the electricity production has long been majority; it has started in antiquity with mills of water. The techniques permitting the exploitation of hydroelectric resources have benefited important progress during the XX century, in the scope of projects construction of the hydroelectric center of great speed. Through their size, their precision and their efficiency, the equipment's of these hydroelectric center and especially hydraulics turbines arrived in first plan of realization. The hydraulic turbine is a mechanic dispositive which is used to transform potential energy and the kinetic energy of water, in mechanic energy. This will then be transformed into electric energy by an alternator. There exist two categories of hydraulics turbine. The turbine of action, which do not constitute a draft tube and function with the kinetic energy of water, and the turbine of reaction, which function with the pressure difference and the energy pressure.

With the increasing coast of energy and the high demand of green energy, hydraulic turbine of thin height of falls such as Francis and Kaplan turbines, are those targeted as being economically profitable which are constituted of distributor, of volute, of runner and draft

[1] Post Graduate Student, Mechanical Engineering Department, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India. E-mail: gunjanbhatt23@gmail.com.

[2] Assistant Professor, Mechanical Engineering Department, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India. E-mail: dbshah@nirmauni.ac.in.

[3] Professor, Mechanical Engineering Department, Institute of Technology, Nirma University, Ahmedabad, Gujarat, India. E-mail: kaushik.patel@nirmauni.ac.in.

tube. The draft tube permits the recuperation of excess water kinetic energy coming from the runner and converts it into energy of static pressure.

Many studies on the draft tube flow have been done. Marjavaara (2006) carried out a numerical study to show that the draft tube have an important rule on the global efficiency of a hydraulic turbine. According to Cervantes et al. (2010), draft tubes are of great interest for turbines of thin height of fall like Kaplan turbines, since the draft tube efficiency increases with the decreasing of the height of fall. Anderson (2009) done an experimental study in the draft tube cone and demonstrated that for small height of fall and high output, loses in draft tube are considerably high and can go up to 50%. Labrecque (1993) did a study on the conception of the axial turbine and demonstrated that the augmentation of the performance of hydraulic turbine pass by a good knowledge of the flow in the turbine. Siake et al. (2014) performed numerical analysis for a draft tube flow of a hydraulic turbine. Ruchi Khare et al. (2012) performed 3D viscous turbulent flow simulation in the complete flow passage of Francis turbine using commercial Computational Fluid Dynamics (CFD) code for three runner solidities at different rotational speeds. Jiri Obrovsk et al. (2013) has been studied development of high specific speed Francis turbine for low head using CFD analysis.

## Design of Draft Tube

As per the output requirement, design of draft tube has been carried out using analytical approach. Based on determined dimensions, 2D drawing has been prepared as shown in figure 1.

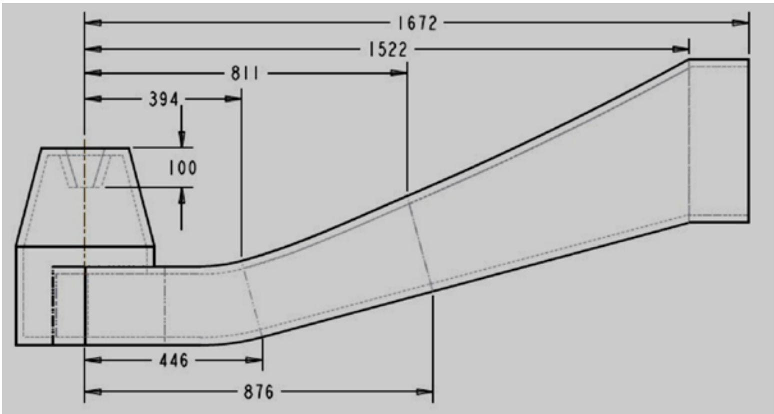


Figure 1. 2D Drawing of Draft Tube

Pro/Program is a powerful secondary generation tool to validate parametric design of the component. Pro/Program reflects all parameters and geometric data of the part in a text data form. This data can be modified to new feature, deleting existing feature, suppress the feature and change the dimension of the feature. A proper user interface (API) can directly modify the Pro/Program and the part modal can be driven according to the user input in user interfacing. Here Microsoft Excel and Creo are integrated by Excel Analysis tool which acts as an interfacing media. Excel analysis tool transfers the spreadsheet data to the Creo database. An excel spreadsheet which contains feature names and respective dimensions of the draft tube has been prepared as shown in figure 2.

| | A | B | C |
|---|---|---|---|
| 1 | | | |
| 2 | | FEATURE NAME | DIMENSION |
| 3 | | CENTRE DIST_1 | 1600 |
| 4 | | CENTRE DIST_ 2 | 850 |
| 5 | | CENTRE DIST_3 | 420 |
| 6 | | FILLET_1 | 500 |
| 7 | | | |

Figure 2. Excel Spreadsheet

An Excel Analysis from Analysis tab have been selected in Creo as shown in Figure 3 to transfer data from Excel Spreadsheet to Creo.

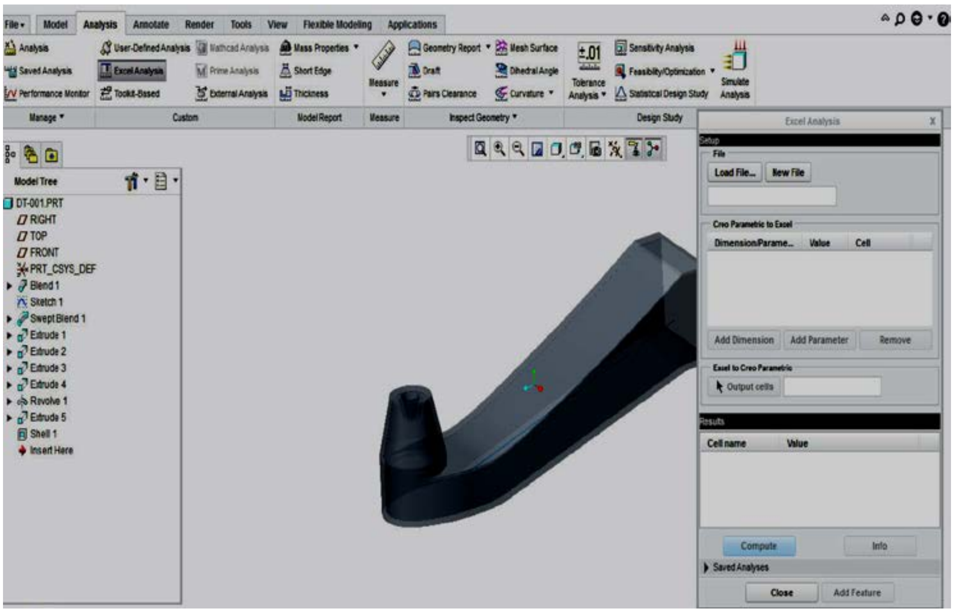


Figure 3. Excel Analysis in Creo

## CFD Analysis

Here, ANSYS workbench is used for CFD analysis of draft tube. For CFD analysis, draft tube model prepared in creo is converted in to STEP file which is further imported in ANSYS. After importing model in ANSYS, cavity model of draft tube is generated using fill command shown in figure 4.

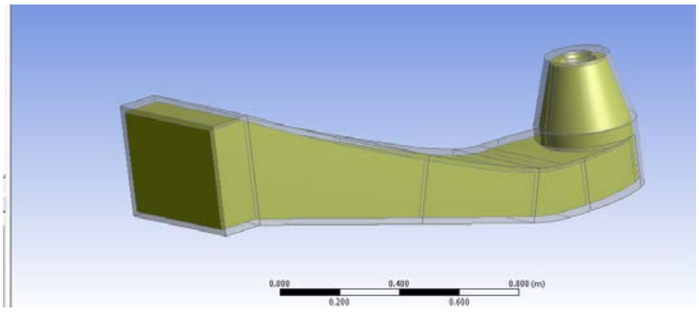


Figure 4. Cavity Model of Draft Tube

The meshing of the draft tube model is carried out with fine size using CFD mesh type.

The ten node tetrahedral elements have been selected because of to achieve good meshing on curvature parts. Meshing model of draft tube is shown in figure 5. Total 12330 nodes and 66821 elements with water domain are created for CFD analysis.

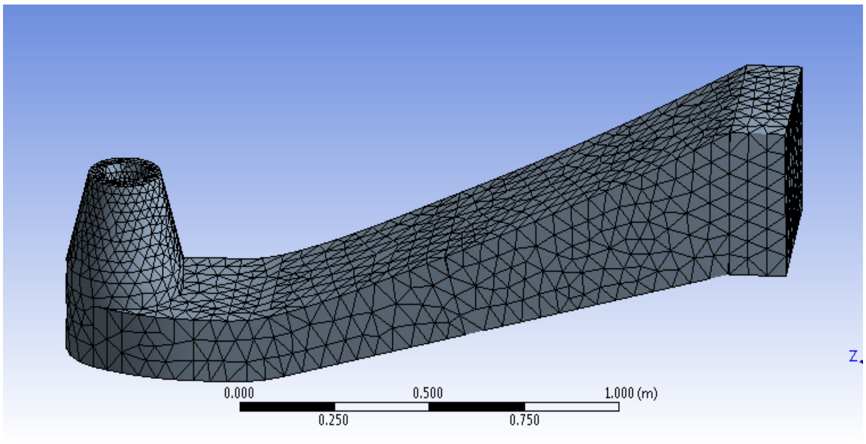


Figure 5. Meshing of Draft Tube

The boundary conditions i.e. inlet mass flow rate is 1000 kg/sec and outlet boundary condition is 1 atm are applied for draft tube CFD analysis as shown in figure 6 and 7 respectively.

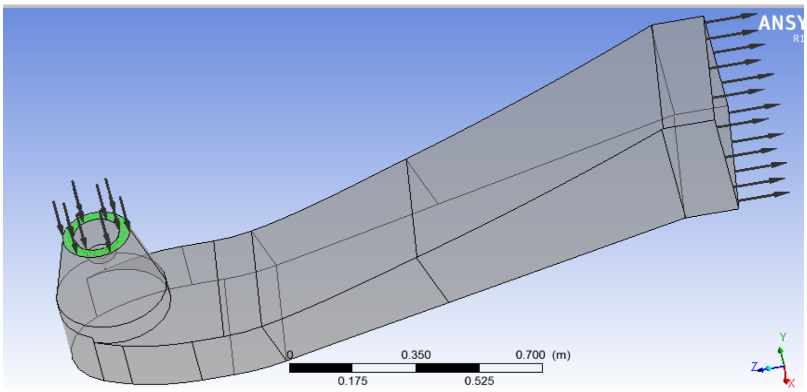


Figure 6. Inlet Boundary Condition

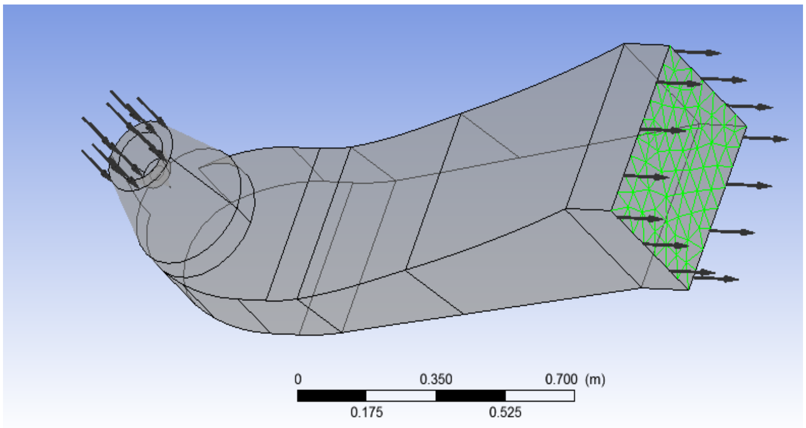


Figure 7. Outlet Boundary Condition

The velocity and pressure distribution are determined using CFX solver in the post processor stage. The results for the velocity and pressure contour for the draft tube as shown in figures 8 and 9 respectively.

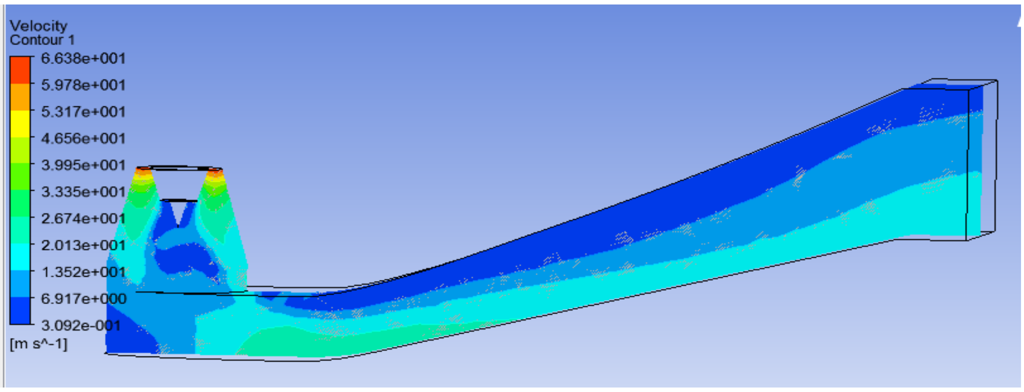


Figure 8. Velocity Contour of Draft Tube

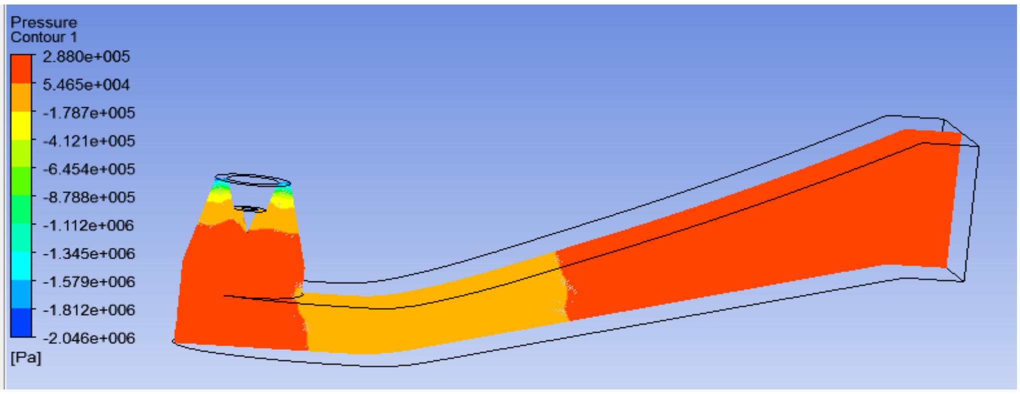


Figure 9. Pressure Contour of Draft Tube

### Comparison with Experimental Reading

The pressure distribution at inlet and outlet of draft tube has been measured by experimental procedure. The same results have been compared with ANSYS analysis results and % difference has been found as given in table 1, which shows both results are in good agreement with each other.

Table 1. Comparison between ANSYS and Practical Reading

| | Inlet Pressure | Outlet Pressure |
|---|---|---|
| ANSYS Result | $-2.100\times 10^5$ Pa | $1.071\times 10^5$ Pa |
| Practical Reading | $-1.99 \times 10^5$ Pa | $1.12 \times 10^5$ Pa |
| % Difference | 5.23 | 4.38 |

## Concepts for Design Modification of Draft Tube

Draft tube design has to a large extent been based on the intuition and on the experience of the design engineer. Recent studies have shown that efficiency improvement can also been realized by minor modification to the geometry of the draft tube at waterway. 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 CAD tools. The alteration of the elbow geometry i.e. cutting the geometry or providing sharp heel by radius at elbow section can easily be implemented by typical CAD action. Five different concepts are proposed for design modification of draft tube.

In concept-1 design, the sharp heel at elbow section of draft tube is modified by giving radius 0.180 m as shown in figure 10. For this concept CFD analysis is carried out by same procedure to determine pressure distribution as given in figure 11.

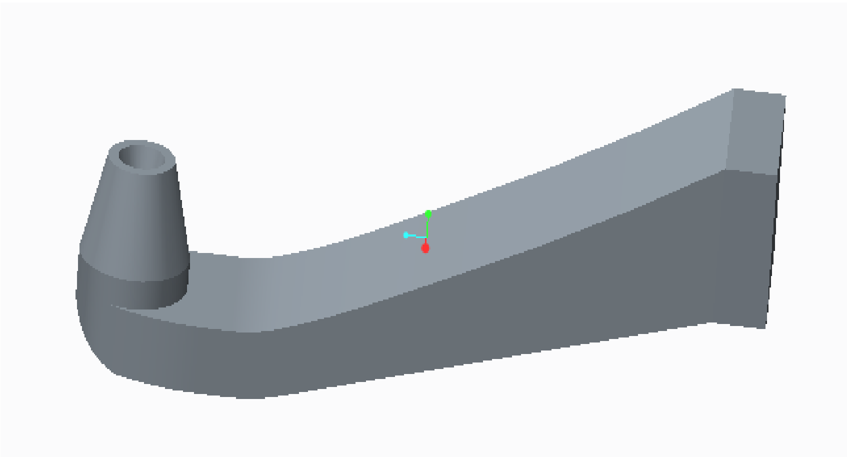

Figure 10. CAD Model for Concept 1

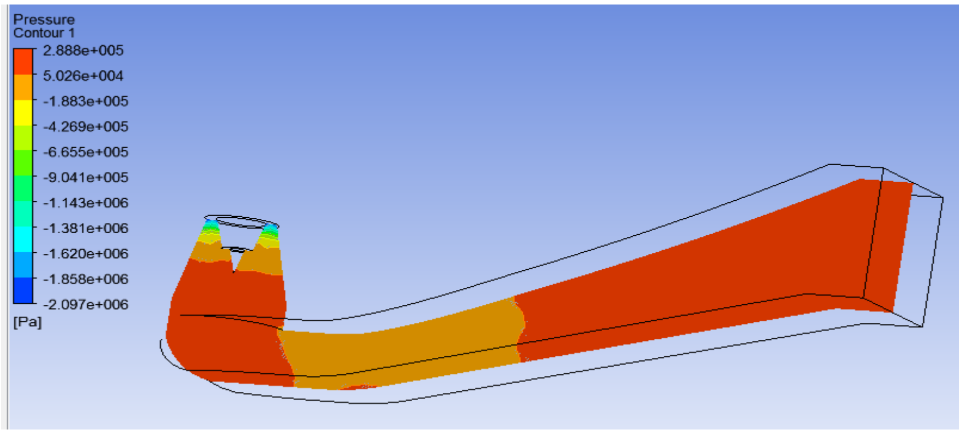


Figure 11. Pressure Distribution for Concept 1

In concept-2 design, the sharp heel at elbow section of draft tube is modified by giving radius 0.128 m and CFD analysis is carried out by same procedure to determine pressure distribution as given in figure 12.

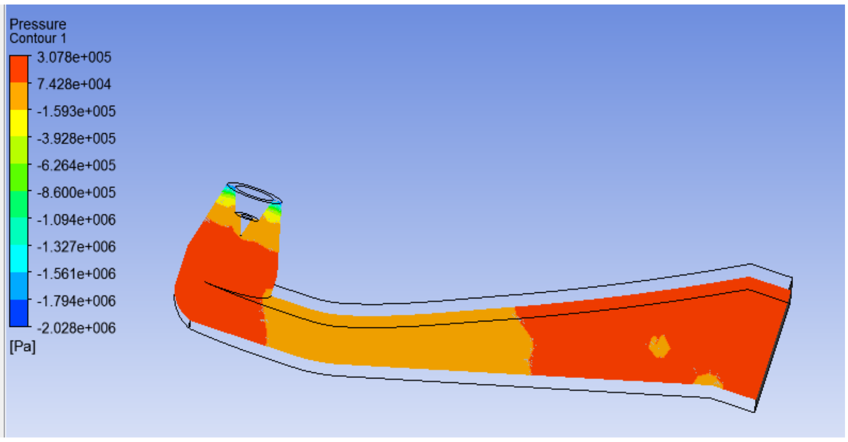


Figure 12. Pressure Distribution for Concept 2

The sharp heel and flow domain of the draft tube is extended up to 0.6 m further downstream to avoid recirculation at the outlet boundary in concept 3. The pressure distribution plot after CFD analysis is shown in figure 13.

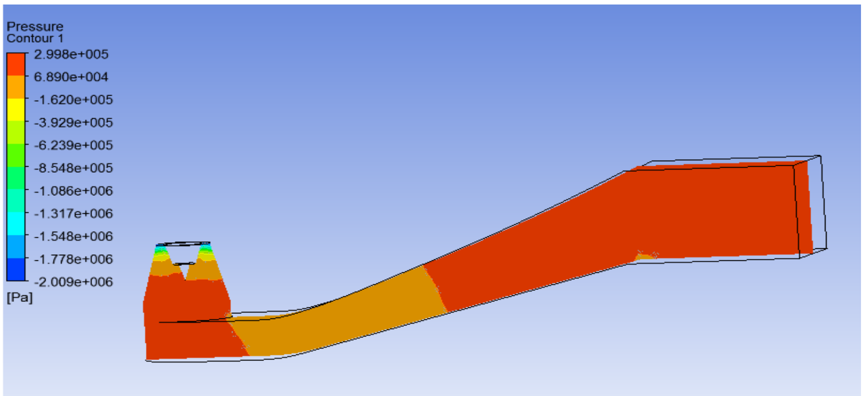


Figure 13. Pressure Distribution for Concept 3

In concept 4 design, the sharp heel draft tube is modified by giving radius of 0.180 m at bottom and 0.612 m at top side. Whereas the sharp heel draft tube is extended with radius at both sides in concept 5 design. The pressure distribution plot after CFD analysis for the concept 4 and 5 are shown in figure 14 and 15 respectively.

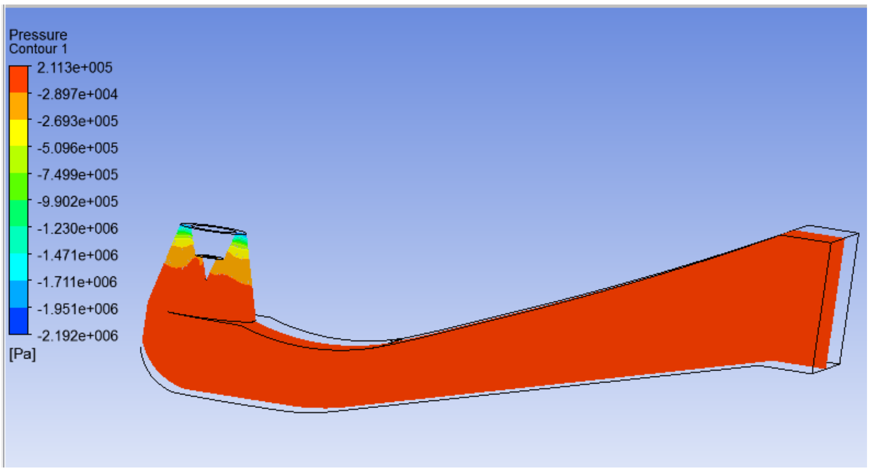

Figure 14. Pressure Distribution for Concept 4

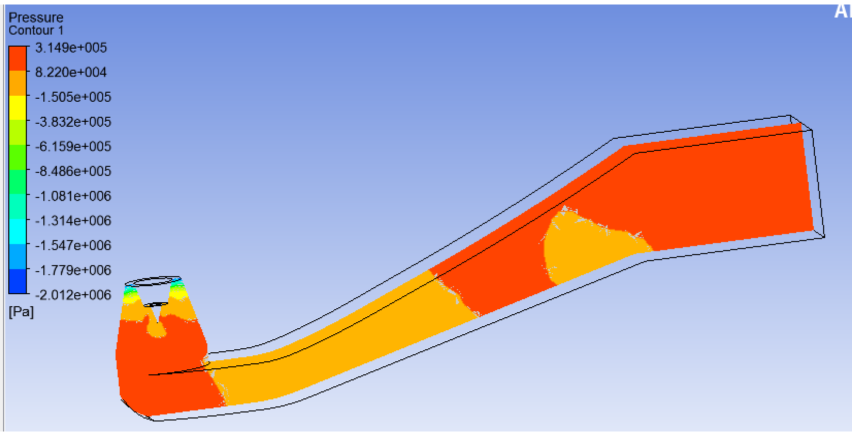


Figure 15. Pressure Distribution for Concept 5

## Result and Discussion

For five different concepts of draft tube are proposed as a part of design modifications to improve efficiency and pressure at outlet. Using CFD analysis, for the same boundary conditions pressure distribution of each concepts are determined. The maximum and minimum outlet and inlet pressure for each concepts are given in table 2. For maximum outlet pressure, graph has been generated to compare all concepts as shown in figure 16.

Table 2. Pressure at Outlet and Inlet for Different Concepts

| Pressure | Original Design | Concept 1 Design | Concept 2 Design | Concept 3 Design | Concept 4 Design | Concept 5 Design |
|---|---|---|---|---|---|---|
| Maximum (Outlet) in Pa | $2.880\times10^5$ | $2.888\times10^5$ | $3.078\times10^5$ | $2.998\times10^5$ | $2.113\times10^5$ | $3.149\times10^5$ |
| Minimum (Inlet) in Pa | $-2.046\times10^6$ | $-2.097\times10^6$ | $-2.028\times10^6$ | $-2.009\times10^6$ | $-2.192\times10^6$ | $-2.012\times10^6$ |

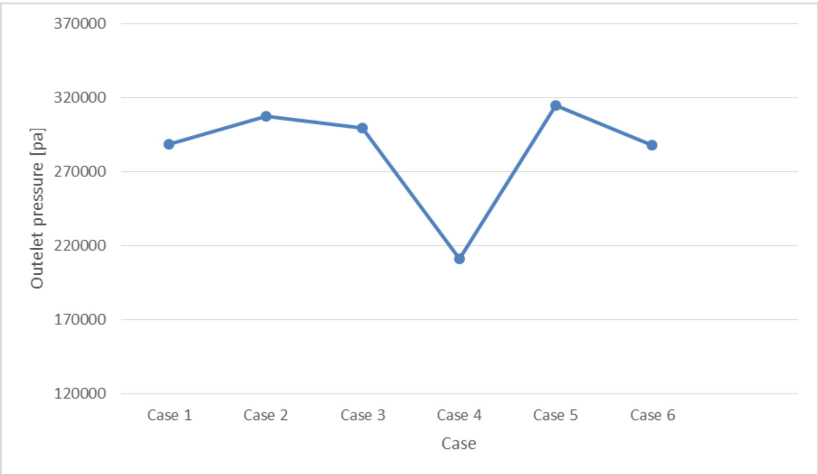


Figure 16. Line Chart for Outlet Pressure

## Conclusion

The design and CFD analysis of draft tube for hydro power plant has been carried out. The design automation has been successfully implemented between Excel spread sheet and solid modeling software, which reduce the time and cost of 3D modeling and 2D drafting. CFD analysis has been performed for draft tube to determine pressure and velocity profile at inlet and outlet condition. The analysis results give good agreement with experimental readings, which reduce higher cost experimentation. Various design concepts has been proposed for draft tube to improve efficiency and pressure at outlet. The same analysis has been performed for each concepts with same boundary conditions. Concept 5 design i.e. sharp heel draft tube is extended with radius at both sides, gives maximum outlet pressure as per analysis results compared to all other concepts.

## References


Marjavaara B. D., 2006. *CFD driven optimization of hydraulic turbine draft tubes using surrogate models*. Thesis (PhD), Lulea University of Technology ISSN: 1402-1544, Sweden.

Cervantes M., Andersson U., and Löfgren H., 2010. Turbine-99, unsteady simulations-validation, *25th IAHR Symposium on Hydraulic Machinery and Systems*. Timisoara Romania.

Anderson U., 2009. *Experimental study of sharp-heel Kaplan draft tube*. Thesis (PhD), Lulea University of Technology, ISBN: 978-91-86233-68-6, Sweden.

Labrecque Y., 1993. *Conception de turbines axiales*, Projet de fin d'Etudes, Université de Laval, France.

Siake A, Koueni-Toko C, 2014. Hydrodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine. *International Journal of Hydraulic Engineering*.

Ruchi Khare, 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.

Jiri Obrovsk, 2013. Development of high specific speed Francis turbine for low head, *Engineering Mechanics*, Vol. 20, No. 2, p. 139–148.