Impact Factor (SJIF): 4.542

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 4, Issue 4, April-2017

Computational fluid dynamic analysis of draft tube used in hydro power plant

Parmar Hardik C.[1]

Falgun patel[2]

[1]Student, ME (IC/AUTO), Mechanical Engineering Department, Hasmukh Goswami college of engineering

[2]Asst. Prof., Mechanical Engineering Department, Hasmukh Goswami college of engineering

***Abstract*—**At present scenario the cost of hydroelectricity is relatively low, making it a competitive source of renewable electricity. It is also a flexible source of electricity since amount produced by the plant can be changed up or down very quickly to adapt to changing energy demands. Main components of hydropower plant are dam, penstock, turbine, and generator and draft tube. This paper contains modeling and analysis of existing draft tube for hydropower plant. Computation fluid dynamics result shows the velocity outlet. For future work efficiency of the draft tube will be increased by modifying the dimensions of draft tube**.**

Keywords: CFD, draft tube, hydrop power plant

I. INTRODUCTION

The cost of hydroelectricity is relatively low, making it a competitive source of renewable electricity. It is also a flexible source of electricity since amount produced by the plant can be changed up or down very quickly to adapt to changing energy demands. Main components of hydropower plant are dam, penstock, turbine, and generator and draft tube.

At the beginning of the 20th century, many small hydroelectric power stations were being constructed by commercial companies in mountains near metropolitan areas. As the power stations became larger, their associated dams developed additional purposes to include flood control, irrigation and navigation. Hydroelectric power stations continued to become larger throughout the 20th century. Hydropower was referred to as white coal for its power and plenty. Hoover Dam's initial 1,345 MW power station was the world's largest hydroelectric power station in 1936The Itaipu Dam opened in 1984 in South America as the largest, producing 14,000 MW but was surpassed in 2008 by the Three Gorges Dam in China at 22,500 MW. Hydroelectricity would eventually supply some countries, including Norway, Democratic Republic of the Congo, Paraguay and Brazil, with over 85% of their electricity. The United States currently has over 2,000 hydroelectric power stations that supply 6.4% of its total electrical production output, which is 49% of its renewable electricity[1]

.

II. Components of hydro power plant

Following are some of the main components of the hydroelectric power plant.[3]

(1) Reservoir and Dam

Water harvested from the catchment area is stored in the reservoir which is then used to generate the electricity. Dams are structures built over rivers to stop the water flow and form a reservoir. The reservoir stores the water flowing down the river. This water is diverted to turbines in power stations. The dams collect water during the rainy season and stores it, thus allowing for a steady flow through the turbines throughout the year. Dams are also used for controlling floods and irrigation. The dams should be water-tight and should be able to withstand the pressure exerted by the water on it. There are different types of dams such as arch dams, gravity dams and buttress dams. The height of water in the dam is called head race.

(2) Control Gate

A spillway as the name suggests could be called as a way for spilling of water from dams. It is used to provide for the release of flood water from a dam. It is used to prevent over toping of the dams which could result in damage or failure of dams. Spillways could be controlled type or uncontrolled type. The uncontrolled types start releasing water upon water rising above a particular level. But in case of the controlled type, regulation of flow is possible.

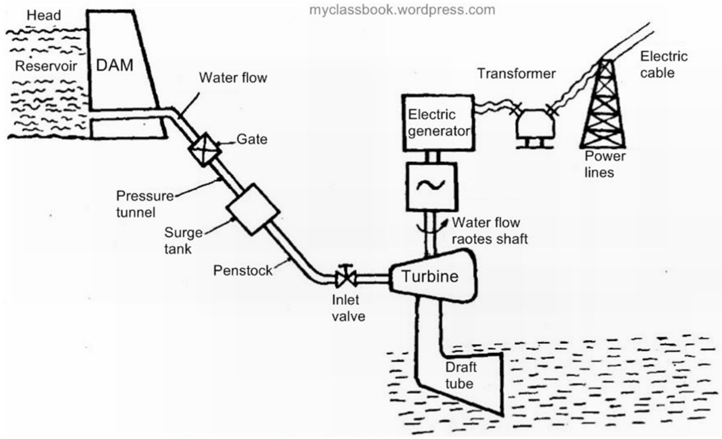

Figure: 1 Layout hydroelectric power plant

(3) Penstock

A penstock is a huge steel pipe which carries water from the reservoir to the turbine. Potential energy of the water is converted into kinetic energy as it flows down through the penstock due to gravity.

(4) Surge tank

Surge tank is build in between dam and the valve house. It is used to take care of the system load fluctuations. The sudden surges of water in penstock is taken by the surge tank, and when the water requirements increase, it supplies the collected water thereby regulating water flow and pressure inside the penstock.

(5) Hydraulic Turbine

Hydraulic turbines are used to extract energy from the continuously flowing water by dynamic action of water on the blades on runner. The flow passage has stationary and moving blades, thus the flow inside the turbine is highly complicated and a well-designed water path can only give the best performance. In general, one of three types of turbines is chosen depending on the head and discharge of the hydropower plant: either a Pelton, a Francis or a Kaplan turbine. A Pelton impulse turbine is suited for high heads and low discharges, while Francis and Kaplan reaction turbines are more appropriate for medium and low heads, respectively, and high discharges.

(6) Draft tube

The water after working on the turbine imparts its energy to the vanes and runner, thereby reducing its pressure less than that of atmospheric pressure (Vacuum). As the water flows from higher pressure to lower pressure, it cannot come out of the turbine and hence a divergent tube is connected to the end of the turbine. Draft tube is a divergent tube one end of which is connected to the outlet of the turbine and other end is immersed well below the tailrace (Water level). The major function of the draft tube is to increase the pressure from the inlet to outlet of the draft tube as it flows through it and hence increase it more than atmospheric pressure. The other function is to safely discharge the water that has worked on the turbine to tailrace.

(7) Power station

Power station contains a turbine coupled to a generator (see the cross section of a power house on the left). The water brought to the power station rotates the vanes of the turbine producing torque and rotation of turbine shaft. This rotational torque is transferred to the generator and is converted into electricity. The used water is released through the tail race. The difference between head race and tail race is called gross head and by subtracting the frictional losses we get the net head available to the turbine for generation of electricity.

III. Objective of the Work

(i) To prepare the 3D CAD model and 2D drawing for present draft tube.

(ii) To develop program for design automation for generation of CAD model and drawing with short time.

(iii) To perform CFD analysis of draft tube

IV. Summary of reviewed Literature

Marjavaara B. D., Lundstrom T. S., Automatic shape optimization of a hy-dropower draft tube, Proceedings of the 4th ASME/JSME joint fluids engineering conference (FEDSM'03), Vol 1C, page 1819-1824, (2003).[1]

In this study a shape optimisation technique to redesign an existing draft tube is presented in this paper. By this method, a design can be predicted in terms of a predefined objective function, here the pressure recovery factor. The optimisation is performed with the Response Surface Method (RSM) implemented on the commercial code isight7.0, while the CFD simulations are carried out with the commercial code CFX4.4.

Marjavaara B. D., Lundstrom T. S., Redesign of a sharp heel draft tube by a validated CFD-optimization, International journal for numerical methods in fluids, Vol 50, page 911-924, (2006).[2]
In this study Numerical optimization techniques in flow design are often used to find optimal shape solutions, regarding, for instance, performance, flow behaviour, construction considerations and economical aspects. The present paper investigates the possibilities of using these techniques in the design process of a hydropower plant. The actual shape optimization is carried out with the response surface methodology, by maximizing the average pressure recovery factor and minimizing the energy loss factor.

Ruchi Khare, Dr. Vishnu Prasad, Dr. Sushil Kumar Mittal, E□ect of runner so-lidity on performance of elbow draft tube, Proceedings of the 2nd International conference on advances in energy engineering (ICAEE), Energy procedia 14, page 2054-2059, (2012).[3]
In this study 3D viscous turbulent flow simulation has been done in the complete flow passage of Francis turbine using commercial Computational Fluid Dynamics (CFD) code for three runner solidities at different rotational speeds. The draft tube performance parameters in non-dimensional form are computed from simulation results and the effects of runner solidity and operating speed on draft tube performance are discussed. It is found that the draft tube loss and efficiency have parabolic variation with speed factor and the point of maximum efficiency or minimum loss shifts towards higher speed factor with decrease in solidity. The loss, efficiency and recovery characteristics of draft tube using CFD will be useful for design optimization of draft tube geometry to give the best performance.

Sumeet J. Wadibhasme1, Shubham Peshne2, Pravin Barapatre3, Santosh Barade4, Saurabh Dangore5, Shubham Harde6, Prof. Shailendra Daf Hydraulic turbine draft tube: literature review International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016[4]
In this study the principle of draft tube and its types through literature review. The parameter affecting the performance of draft tube are also discussed with the help of researches carried out earlier. The reviews it is found that the CFD can be used as a tool for analysing the performance and the flow pattern inside the draft tube.

Siake , Koueni-Toko , Djeumako , Tcheukam-Toko , Soh-Fotsing , Kuitche, Hy-drodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine, Inter-national Journal of Hydraulic Engineering 2014, 3(4): 103-110.
This study is performed for a draft tube flow of a hydraulic turbine. Special attention has been paid for the friction effect through the flow inside the complex geometry of the draft tube, and for the interaction between the vortex structures and the draft tube volute. A draft tube affected could have a large significance on the performance prediction of hydraulic turbines, even on the efficiency of a hydroelectric center. The turbulent model has been applied a standard two equations model and the two-dimensional Reynolds Averaged Navier–Stokes (RANS) equations, are discredited with the second order upwind scheme. The SIMPLE algorithm, which is developed using control volumes, is adopted as the numerical procedure. Calculations were performed for a wide variation of runner velocities. The results reveal that with increasing of the runner velocity, the velocity decreases and the static pressure increases, justifying the total recuperation of kinetic energy at the draft tube outlet. Comparison of Numerical results with the experimental data available in the literature is satisfactory.

Ruchi Khare and Vishnu Prasad, Numerical Study on Performance Characteristics of Draft Tube of Mixed Flow Hydraulic Turbine, Hydro Nepal issue no. 10 January, 2012 49.
In this study the computational fluid dynamics (CFD) has been used for flow simulation in complete mixed flow Francis turbine for performance analysis for energy recovery, losses and flow pattern in an elbow draft tube used in Francis turbine at different operating conditions. The overall performance of the turbine at some typical operating regimes is validated with the experimental results and found to be in close comparison. It is found that the draft tube loss and efficiency have parabolic variation with speed factor and the point of maximum efficiency or minimum loss shifts towards higher speed factor with increase in guide vane opening. The performance characteristics of draft tube and flow pattern within it will be useful for the design optimization of draft tube geometry to improve the performance of draft tube as well as turbine.

Z. Carija, Z. Mrsa and L.Dragovic, Turbulent Flow Simulation In Kaplan Draft Tube, 5th International Congress of Croatian Society of MechanicsSeptember, 21-23, 2006 Trogir/Split, Croatia.
Draft tube fluid flow is difficult to predict numerically due the highly swirled, fluctuating and non uniform flow that leaves the rotating runner and enters the draft tube. Furthermore, draft tube fluid flow is characterized by adverse pressure driven fluid flow, with several zones of fluid flow separation. Fluid flow complexity in draft tube arises when working out from optimal point of operation, where higher quantity of kinetic fluid flow energy enters the draft tube. Without visualization of

the fluid flow simulation results, only pure intuition existed of what the fluid flow in hydraulic turbines looks like. From the flow simulations results the fluid flow behaviour inside draft tube can be understood in sufficient detail and this knowledge can be used to design new generation of the draft tubes or revitalize old ones.

Manoj kumar, Rajeev jain, S.N Shukla, CFD Analysis of 3-D flow francis turbine, ISSN No. 2230 – 7699 MIT Publications.
In this study 3 D real Flow analysis is done for experimentally tested Turbine and the Characteristics of prototype Turbine were predicted in actual operating regimes. The operating conditions considered are in accordance with that, where actual prototype turbine is to be installed. Flow structure inside the machine is analysed and it showed the scope of improvement in the design. Results obtained by computational tool were very close to experimental results. Present paper elaborates model selection for prototype turbine, details of methodology used, visualization of results in CFX-post & then validation of computational results.

Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel," Design Automation and CFD Analysis Of Draft Tube For Hydro Power Plant", International Journal Of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-6, June-2015.
The efficiency of a hydraulic reaction turbine is significantly affected by the performance of its draft tube. In this paper, an attempt has been made for design automation of modeling of draft tube using Excel spreadsheet and Creo parametric software. 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, the CFD analysis of draft tube has been performed and results for the same are compared with experimental reading and which are found within the limit. The application of ANSYS is wide in engineering field. From above analysis it has been found that ANSYS result give the good agreement with experimental reading. By design automation one can reduce the time and cost of modeling and drafting. By analysis results one can predict almost nearer results of pressure and velocity profile without carrying higher cost experimental work.

V. Problem Definition

Most of the past studies have focused on the runner for increasing the e□ciency of the plant. But a good runner design is not enough. Recent studies have shown that e□ciency improvement can also be realized by minor modification on the older design in the rest of the waterway i.e., in the draft tube and spiral casing. Approximately half of all operating costs in most manufacturing and processing operations can be relevant to maintenance.

Recent studies have shown that e□ciency improvements can also be realised by minor modifications to the geometry of the waterways. An important part of the waterways is the draft tube, often being a curved di□user connecting the runner to the outlet.

VI. Calculations

Velocity at inlet V [10]
From Bernoulli's equation

$V = \sqrt{2\,gz}$

$V = \sqrt{2 \times 9.81 \times 25}$

$\mathbf{V = 22.1472\ m/s^2}$

Net head at the inlet of the penstock
[For laminar flow]
Frictional factor

f= Frictional factor = $\frac{64}{R_e} = \frac{64}{60\ X\ 10^6}$ = 1.066 X 10^{-6}[for laminar flow]

Length of the pipe connected to the water source = 7m
Total length of the penstock L = 7 = 7 m

$$H_f = f\frac{L}{D}\frac{V^2{}_{avg}}{2g}$$

$$H_f = 1.066\ X\ 10^{-6}\ X\frac{7}{2.714}X\ \frac{22.1472^2}{2\ X\ 9.81}$$

$$H_f = 2.269\ X\ 10^{-4}\ m$$

Now,

$H_{Net} = H_g - H_f - Hb$

Where

Hg = gross head

H_f= head loss

Hb =Loss due to bend.

There is no bend in the penstock Hb = 0 m.

$H_{Net} = 25 - 0.000269$

$\boldsymbol{H_{Net} = 24.999731 = 25m}$

For turbulent flow

Roughness of the pipe ε = 45 X 10^{-6} m

Relative roughness= $\frac{\varepsilon}{D_H}$ = $\frac{45 \, X \, 10^{-6}}{2.714}$ = 0.0000166

From moody diagram

$f = 0.0105$

So,

$$H_f = f \frac{L}{D} \frac{V^2{}_{avg}}{2g}$$

$$H_f = 0.0105 \, X \frac{7}{2.714} X \frac{22.1472^2}{2 \, X \, 9.81}$$

$H_f = 2.3 \, m$

Now

$H_{Net} = H_g - H_f$

Where

Hg = gross head

H_f= head loss

$H_{Net} = 25 - 2.3$

$\boldsymbol{H_{Net} = 22.70m}$

Pressure at inlet of the penstock

$p = \rho g H_{net}$

Where

ρ = Density of water (1000kg/m^3)

g = Gravitational acceleration (9.81m/s^2)

$p = 1000 \times 9.81 \times 22.7$

VII. Modeling

Modeling is the process of producing the model is a representation of the construction and working of some system of interest. One purpose of a model is to enable the analyst to predict the effect of changes to the system. A model should be a close approximation to the real system and incorporate most of its salient features. It should not be so complex that it is impossible to understand and experiment with it.

2D drawing is given by the client in pdf format. Objective is to create 3D model from given drawing.

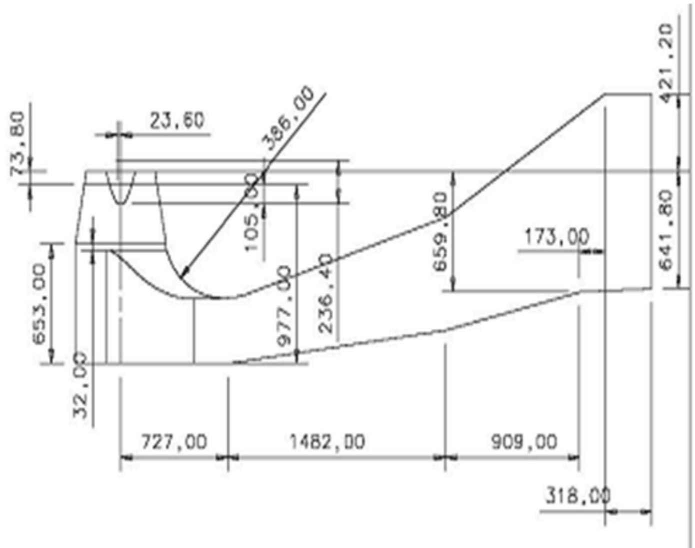

Figure 2 2D drawing

--- 3D Modelling

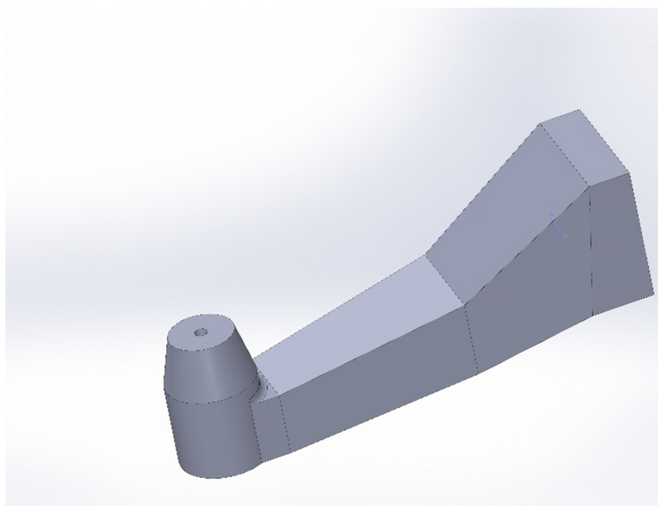

Figure 3 3D Modelling

VIII. CFD analysis

(i) Boundary conditions

When solving the Navier-Stokes equation and continuity equation, appropriate initial conditions and boundary conditions need to be applied.
Boundary conditions are a required component of the mathematical model. Boundaries are direct the motion of flow.
Boundary conditions applied on model are

Inlet velocity: 22 m/s
Inlet pressure: 2, 22,687 Pa

(ii) **Initialization** and Calculation

Run the calculation. Initially the calculation starts with the 50 iteration. And number of integration is gone on increasing till the all adjacent output value remains nearly same. For that number of iteration set is 200.

(iii) Post **processing**

After the calculation it needs to interpret the result as per requirement. In this case the required data is the pressure on the inner. This pressure is used for the FEA analysis.

After preparation of model mesh is been generated on model.

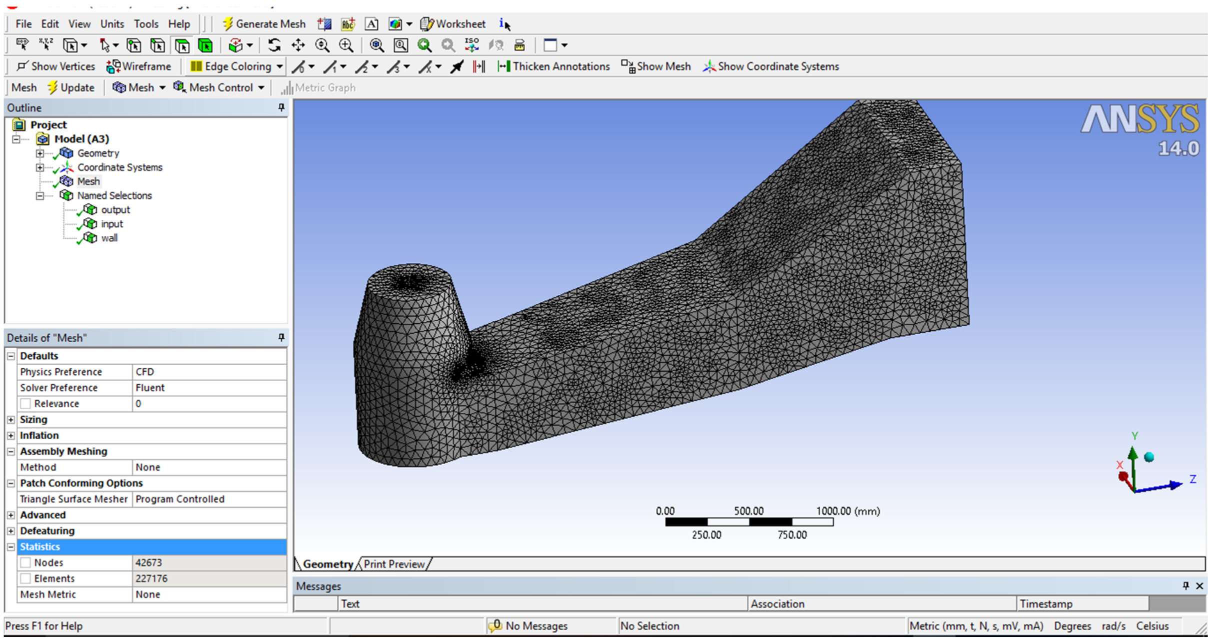

Figure 4 Process for the mesh

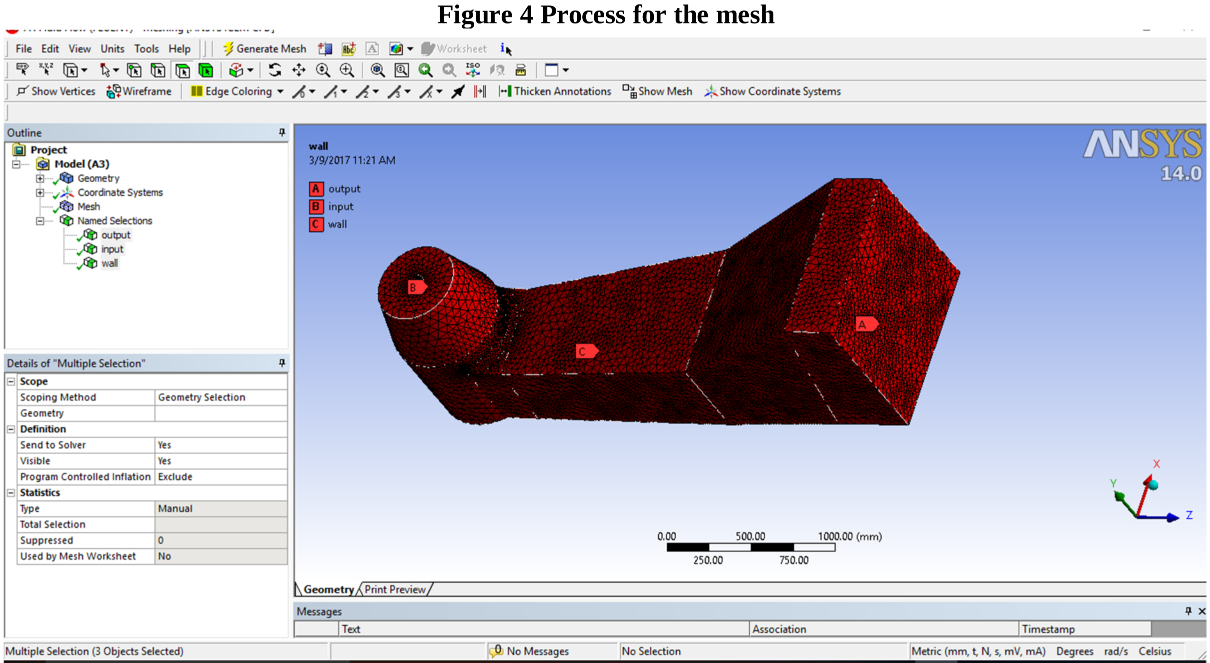

Figure 5 boundary condition

Boundary conditions applied on model are

Inlet velocity: 22 m/s

Inlet pressure: 2, 22,687 Pa

Solution

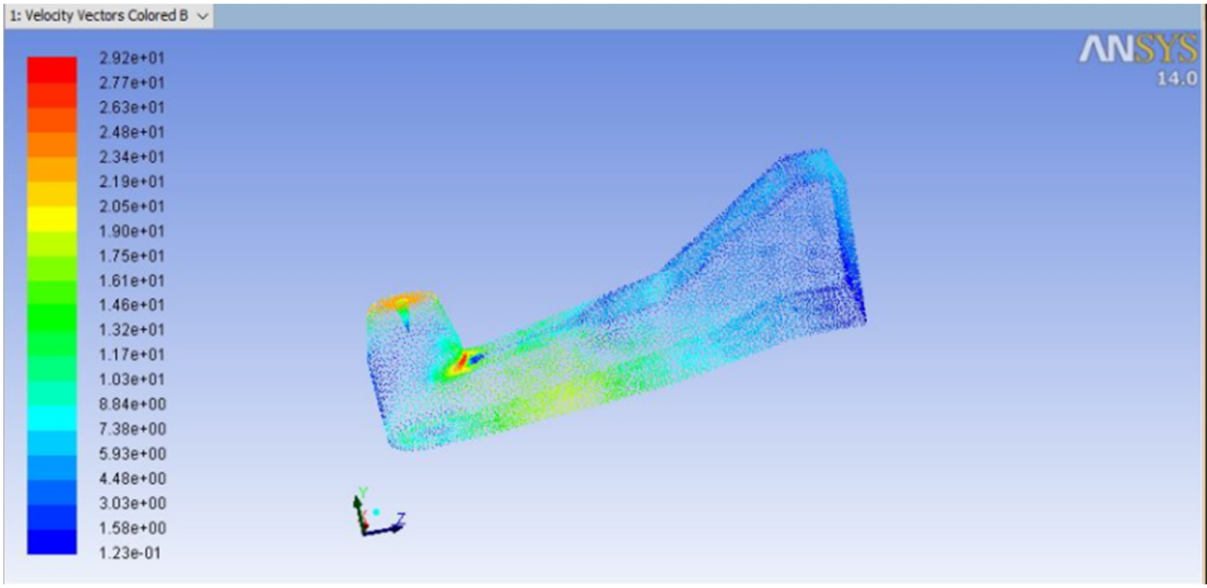

Figure 6 velocity

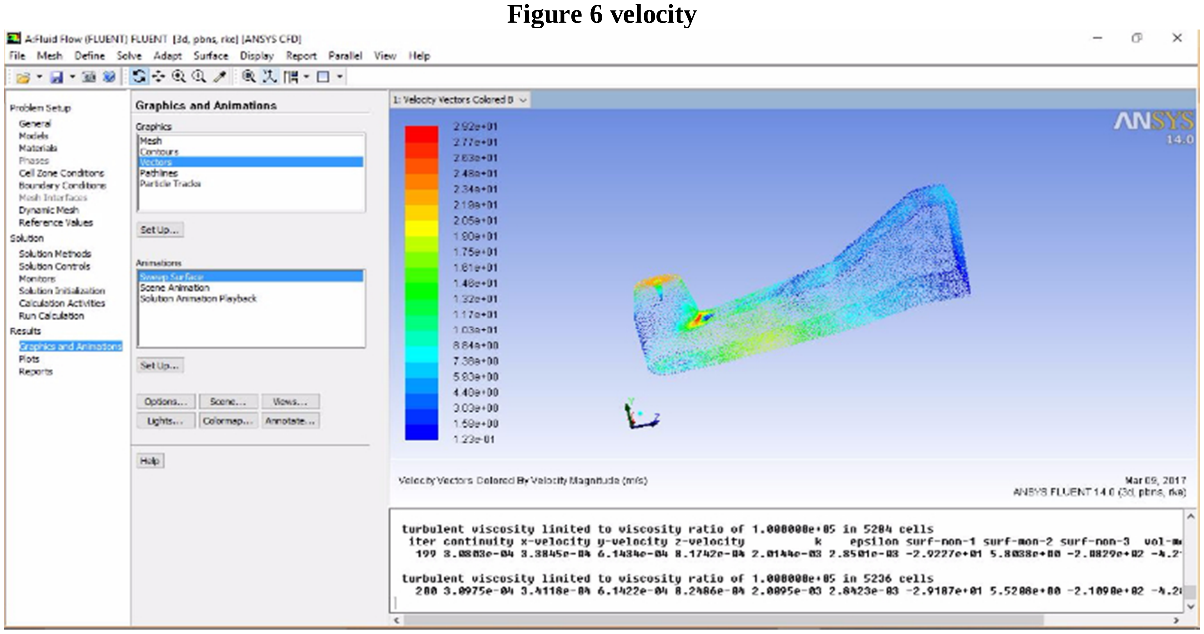

Figure 7 result

CFD analysis shows outlet velocity is 0.1 m/s.

IX. Conclusion & future work

In this, study carried out to understand the working of draft tube. The design and modelling of draft tube for hydro power plant is carried out. CFD analysis is performed for draft tube to determine pressure and velocity profile at inlet and outlet condition. And from result outlet velocity is 0.1m/s.

The efficiency of Draft Tube can be increased by decreasing outlet velocity and Head loss in Draft Tube so in this project this two parameter will be change for best efficiency

X. References

1. Marjavaara B. D., Lundstrom T. S., Automatic shape optimization of a hy-dropower draft tube, Proceedings of the 4th ASME/JSME joint fluids engineering conference (FEDSM'03), Vol 1C, page 1819-1824, (2003).
2. Marjavaara B. D., Lundstrom T. S., Redesign of a sharp heel draft tube by a validated CFD-optimization, International

journal for numerical methods in fluids, Vol 50, page 911-924, (2006).

3. Ruchi Khare, Dr. Vishnu Prasad, Dr. Sushil Kumar Mittal, Eect 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, (2012).
4. Sumeet J. Wadibhasme1, Shubham Peshne2, Pravin Barapatre3, Santosh Barade4, Saurabh Dangore5, Shubham Harde6, Prof. Shailendra Daf Hydraulic turbine draft tube: literature review International Journal of Science, Engineering and Technology Research (IJSETR), Volume 5, Issue 3, March 2016.
5. Siake , Koueni-Toko , Djeumako , Tcheukam-Toko , Soh-Fotsing , Kuitche, Hy-drodynamic Characterization of Draft Tube Flow of a Hydraulic Turbine, Inter-national Journal of Hydraulic Engineering 2014, 3(4): 103-110
6. Ruchi Khare and Vishnu Prasad, Numerical Study on Performance Characteristics of Draft Tube of Mixed Flow Hydraulic Turbine, Hydro Nepal issue no. 10 January, 2012 49.
7. Z. Carija, Z. Mrsa and L.Dragovic, Turbulent Flow Simulation In Kaplan Draft Tube, 5th International Congress of Croatian Society of MechanicsSeptember, 21-23, 2006 Trogir/Split, Croatia
8. Manoj kumar, Rajeev jain, S.N Shukla, CFD Analysis of 3-D flow francis turbine, ISSN No. 2230 – 7699 MIT Publications.
9. Marjavaara, B. D., CFD driven optimization of hydraulic turbine draft tubes using surrogate models, PhD thesis, Lulea University of technology, Sweden, (2006).
10. Gunjan B. Bhatt, Dhaval B. Shah, Kaushik M. Patel," Design Automation and CFD Analysis Of Draft Tube For Hydro Power Plant", International Journal Of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 3, Issue-6, June-2015.