ISSN: 2278-0181
RDMEl - 2018 Conference Proceedings

Fluid Flow Analysis of Hydroelectric Turbine System for Treated Waste Water

Hydroelectric Turbine System

A. K. Gupta, Manoj Kumar*, P. Kumar, D. Panda
Ph.D. Scholar, Cryogenics Engineering Laboratory,
Department of Mechanical Engineering
National Institute of Technology, Rourkela
Odisha, India, 769008

R. K. Sahoo
Professor, Cryogenics Engineering Laboratory,
Department of Mechanical Engineering
National Institute of Technology, Rourkela
Odisha, India

Abstract*—This paper explored the feasibility of implementing hydroelectric turbine systems in wastewater treatment facilities. This paper introduces the technical and economic viability of a new hydro installation solution designed to reduce the power consumption of a medium-sized wastewater treatment plant. The work analyses the hydroelectric potential of the plant and summarizes the turbine-generator design procedure performed to optimize the production. Results demonstrate the viability when energy produced is used for self-consumption. Hydroelectric turbine can be used to partly meet the high demands of electricity. Hydroelectric turbine in wastewater treatment plant is no new concept and has been practiced in many parts of the world. The research outlines some of the advantages and disadvantages of installing a hydro power plant in between the flow of wastewater treatment plant along and can be implemented in overhead tank at various locations. The research showcases a Computational flow dynamics of a fluid flow through a hydro turbine using fluent for two geometry and three varying flow rates. Computational fluid dynamic (CFD) techniques have played an important role in the design and development of such turbines. The simulation of a complete turbine requires substantial computational resources. This research discusses the presented numerical results and the important outcome of the extensive numerical studies on the spiral case Francis turbine. The use of a wall function assuming equilibrium between the production and dissipation of turbulence is widely used in the simulation of hydraulic turbines. We have been undertaking three-dimension steady state simulation using ANSYS Fluent in order to compute the average flow features and investigate the overall turbine performance the simplest model, the homogenous model was used in view of its lower computational time and satisfactory results for such flows. The K-E model of turbulence was for this model has been shown to have batter performance for turbo m/c close compared two equation models. Over a wide range of operating condition, we have obtained.

***Keywords*—Turbine, CFD, Volute, Guide vanes**

I. INTRODUCTION

The importance of water throughout history, not only as a source of life but also as an energy source, has been a constant since the beginning of civilizations. In addition, the consideration of the water as a fundamental resource associated with energy is a matter of particular importance in recent years [1]. So much so that the "International Water Summit", presents as its motto "No water, no energy. No energy, no water", summing up perfectly the two directions of the water-energy relationship. The Iberian Peninsula has not been out to this reality and, because of its scarce water resources, especially in the south; it has historically developed a strong water policy. From the Romans with their large infrastructure to the current “Agua” Plan, through the large development of ditches and irrigated areas in the Islamic era, the multiple river basins utilization during the Middle Age, the massive development of marshes and large pumping systems made along the 20th century, have been oblivious to the need for water use. Therefore, despite not having great rivers in the region, one can find some interesting facilities such as: the old water mills of Ares del Maestre or different water reservoirs with a combined storage capacity of 185 hm3. In this context, the importance of an optimized treatment of wastewater from both energetic and public health standpoints is crucial. This task is clearly internalized in all first world countries. Note that the amount of energy used for example in Spain to treat the 3,000 hm3/year of urban wastewater represents the 1% of the country's total energy consumption [2]. However, there are huge differences in their technological conception between small and large size Wastewater Treatment Plants (WTP). While small and medium-sized WTP sometimes lack of aeration controls and their design is based on the mechanical strength and liability (which implies higher unit costs and consumption), in large WTP, the design is optimized to achieve energy consumptions well adjusted.

In this sense, while some large WTP are currently operating with consumptions rounding 20-30 kWh/hm3 throughout the year, the average consumption for the wastewater purification park in Spain is around 50 kWh [2]. Therefore, although efforts have been done in this sense [3, 4], there is stillroom for improvement in the efficiency operation of WTP.

Computational fluid dynamic (CFD) techniques have been used to study the flow conditions inside hydraulic turbines over the past three decades [5]. The numerical modeling of hydraulic turbines is a challenging task because a specific modeling approach applied to investigate a certain operating condition does not necessarily work for another operating condition. Small changes in discharges and/or head significantly affect the flow conditions inside the turbine. There are several challenges for hydraulic turbine modeling in terms of obtaining useful results. An open test case allowing

Researchers to interact to address such questions is, thus, necessary to develop the numerical capacity for the study of hydraulic turbines. The main objective of the Francis test case is to provide an open platform to industrial and academic researchers to explore and develop the capabilities of CFD techniques within the field of hydropower research. The Francis test case consists of a high-head Francis turbine model, whose geometry, together with meshing and measurement (pressure and velocity) data are available for academic research purpose [6].

Today, hydropower is the most important renewable energy source among the other sources for Turkey and all other countries around the world. Hydraulic turbines are used for hydropower generation. Hydraulic turbines produce approximately one fifth of the total electricity in the world (World Energy Council, 2006). Their efficiencies can rise up to 95% and hydraulic turbines generate electricity with a minimum amount of pollution. In addition, they have a great energy storing capability and are able to meet the daily changing electricity demand. [7]

Hydraulic turbines are basically classified in two groups; impulse and reaction turbines. Impulse turbines work based on the momentum principle. Water hits the runner blades in the form of a water jet and this impact causes a force on the runner, which causes the runner to turn [8]. Pelton turbine is an example of impulse turbines. In reaction turbines, the flow is fully pressurized through the turbine. The potential energy of water is converted to kinetic energy by a velocity rise. It uses the action-reaction principle. Examples of reaction turbines are Francis and Kaplan type turbines [9].

Francis turbines are applicable to a wide range of head (from 64 m to 700 m) and specific speed (from 51 rpm to 250 rpm) values. Their wide range of applicability and easier structural design makes Francis turbines more advantageous than other hydraulic turbines [10].

The aim of this thesis is to develop a method to automate the set up process of a CFD simulation and study the flow of a generic fluid radial turbine design. The method can also be used as a conceptual design tool for radial turbines and supports most fluids deigned. To show the capability of the method developed, without going into full analysis with mesh dependent study and as high quality mesh as possible, a comparison is made between the performance two different geometry 3D CFD simulations.

A. Preparation of model

The sketch model explains the geometry and the working of numerical method. In this model the velocity at the inlet of the turbine blade is consider as the outcome result. The guide vanes are provided to guide the flow pattern and accrue the maximum velocity at the inlet of the turbine.

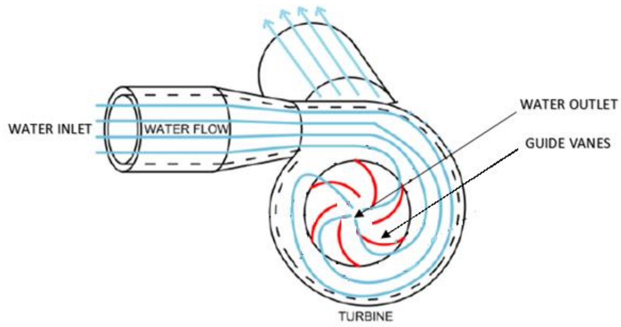


FIG: 1. SKETCH OF MODEL

1.1. Geometry

A two-dimensional drawing of the turbine was also provided for researchers interested in preparing a geometry using their own software and techniques.

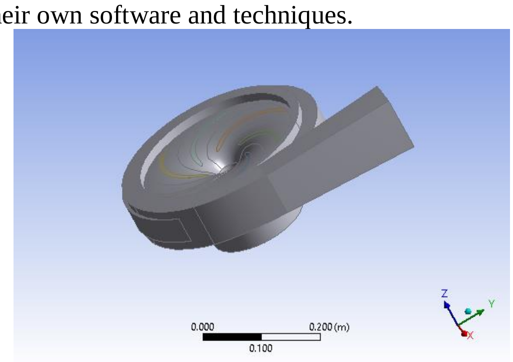


FIG: 2 GEOMETRY OF RECTANGULAR MODEL

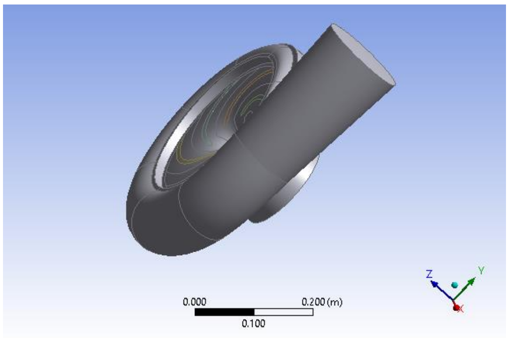


FIG: 3 GEOMETRY OF CIRCULAR MODEL

1.2. Mesh Creation

The turbine model was divided in solid and fluid domains the guide wanes is of solid domains were as the volute profile is of fluid domains the mesh was independently created in all domains.

1.3. Interface Modeling Techniques.

Hydraulic turbines are operated from part load to full load conditions. The consequences of rotor-stator interactions are one of the key concerns for the life expectancy of high-head turbines. It is expensive to model and simulate a complete turbine to investigate the rotor-stator pressure amplitudes. A conventional approach is to model a guide vane and a volute passage. This approach has some limitations and introduces errors at the interface locations due to the averaging of the flow variables, i.e., flow unsteadiness is not resolved. The Problem has been taken from the Manual on Sewerage and sewage Treatment.

1.4 Governing equations

Governing equations, which is used to solve the computational domain, are as follows:

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho U\right) = 0 \tag{1}$$

Momentum Equations:

$$\frac{\partial\left(\rho U\right)}{\partial t} + \nabla.\left(\rho U \times U\right) = -\nabla p + \nabla.\tau + S_M \tag{2}$$

Where the stress tensor τ is related to the strain rate as:

$$\tau = \mu\left(\nabla U + \left(\nabla U\right)^T - \frac{2}{3}\delta\nabla.U\right) \tag{3}$$

Total Energy Equation:

$$\frac{\partial\left(\rho h_{tot}\right)}{\partial t} - \frac{\partial p}{\partial t} + \nabla.\left(\rho U h_{tot}\right) = \nabla.\left(\lambda \nabla T\right) + \nabla.\left(U.\tau\right) + U.S_M + S_E \tag{4}$$

Where h_{tot} is the total enthalpy, which is related to static enthalpy:

$$h_{tot} = h + \frac{1}{2}U^2 \tag{5}$$

1.4. Turbine Power

The power available from water can be calculated for a specified net head and rate of flow of water, as follows

$$P_a = \frac{\gamma.Q.H}{75} \tag{6}$$

Pa = Power available in HP

γ = Specific weight of water in kg/m3

Q = Rate of flow of water in m3/sec

H = Head in m, acting at the turbine inlet

1.5. Mannings Formula

$$V = \frac{1}{n} R^{\frac{2}{3}} S^{\frac{1}{2}} \tag{7}$$

And

$$Q = A \times V \tag{8}$$

Where,

Q	=	Discharge
S	=	Slope of hydraulic gradient
D	=	Internal dia of pipe line
R	=	Hydraulic radius in m
V	=	Velocity in m/s
n	=	Manning's roughness coefficient

TABLE 1. ASSUMPTIONS

Assumptions	Data
➢ Sewage Generated	80 % of Total Water supply
➢ Flow Assumptions	Minimum Flow Varies from 1/3 to ½ of
➢ Ground Water Infiltration	500 to 5,000 litres/km/day
➢ Manning's Roughness Coefficient	0.013
➢ Velocity of Flow	0.60 m/s to 6.0 m/s
➢ Rate of Flow	80% Running Full
➢ Minimum Size of Sewer	150 mm to 250 mm
➢ Flow Discharge	150 m^3/hr to 250 m^3/hr

II. RESULTS AND DISCUSSIONS

In this paper, the main results of the project will be presented. The flow of the turbine is studied with the two model using commercially available ANSYS Fluent 15.0. The difference between two model the rectangular, the Circular profile is investigated, and the two models compared to with different inlet velocity studies.

To compare the model and the performance of the models, the performance is evaluated only over the volute and guide vanes. The simulation of the full turbine gives the best result to study the flow. Some interesting phenomena such as a vortex appear. It is better to analyze the results of both the model to draw conclusions from the flow.

The volute distributes the flow and should produce uniform outlet flow. At this stage of the design, the volute could be improved to avoid flow separation. Separation at entry to Guide vanes. In Figure 4, a cross section of the volute and the volute Guide vanes can be seen. At the sharp corners of the volute outlet to the duct, the flow separates. The separation extends to the leading edge and over the Gide vanes.

The equal distribution of the water around the runner is significant for a balanced operation of the turbine. As plotted in Fig. 3, the radial flow velocity has a uniform distribution at the outlet of the spiral case. The same behavior is observed in the pressure distribution on the spiral mid-plane, shown in Fig 5.

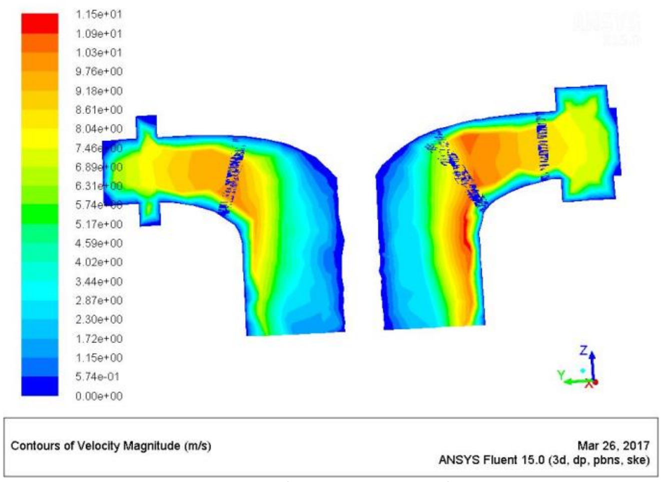
Distribution of the Total pressure in the Guide vane passage is as shown in Fig. 5. The gradual pressure values between the stay vanes indicate the use of correct inflow and outflow angles. As shown in Fig. 6, the flow is distributed uniformly between stay vanes.

The velocity variation between the guide vanes which are assigned a symmetric profile is shown in Fig. 7. The flow is guided with correct angle and minimum hydraulic loss to the runner. Any backflow or flow separation is not observed in flow area between the guide vanes, as shown in Fig. 8.

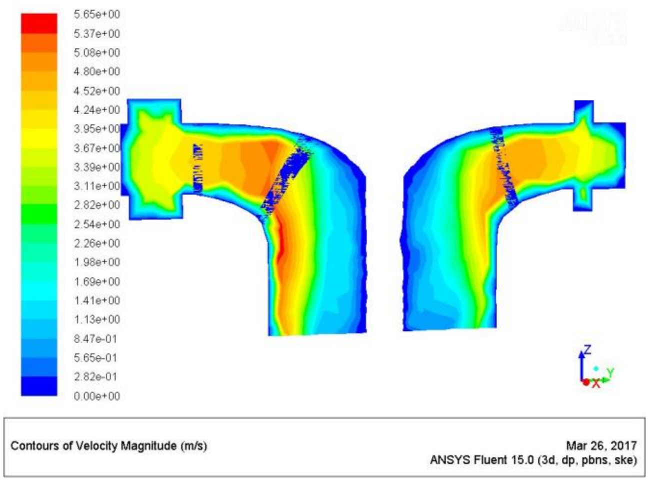
A. Comparison of Rectangular profile and Circular profile volute

The difference in rectangular profile and Circular profile volute can be identify from the Fig.6. The flow velocity distribution and the total pressure at the outlet of the guide vane in circular profile volute is more as compared to rectangular profile volute, which can be verified by the Fig. 4 & 5.

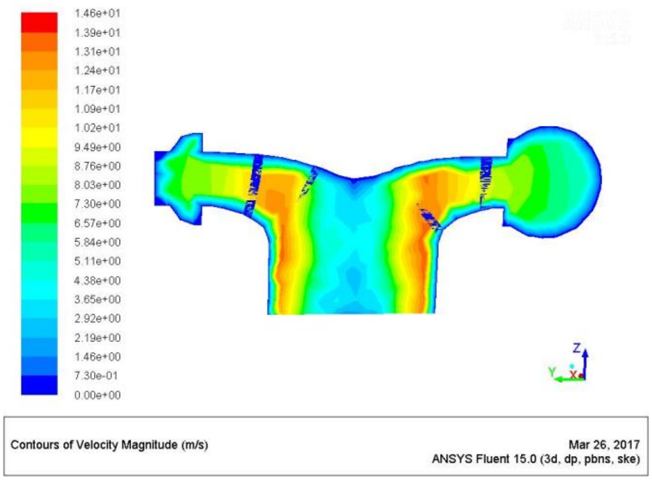
The reason behind this is that the eddy formation is occurred due to the Sharpe edges at the corner of the rectangular profile.



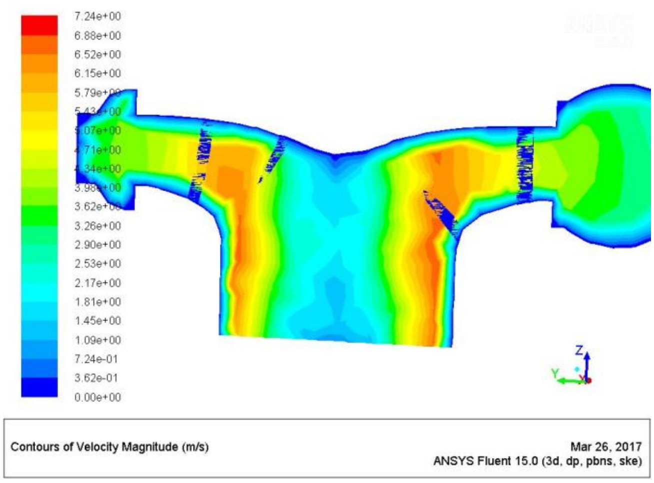
Velocity contours for rectangular profile at 4m/s



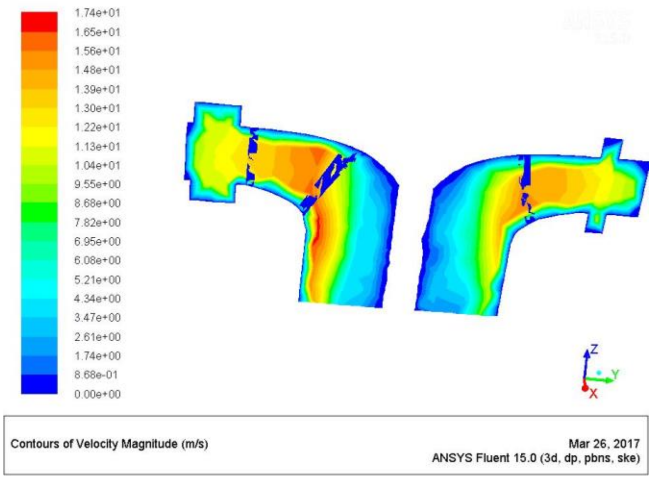
Velocity contours for rectangular profile at 2 m/s



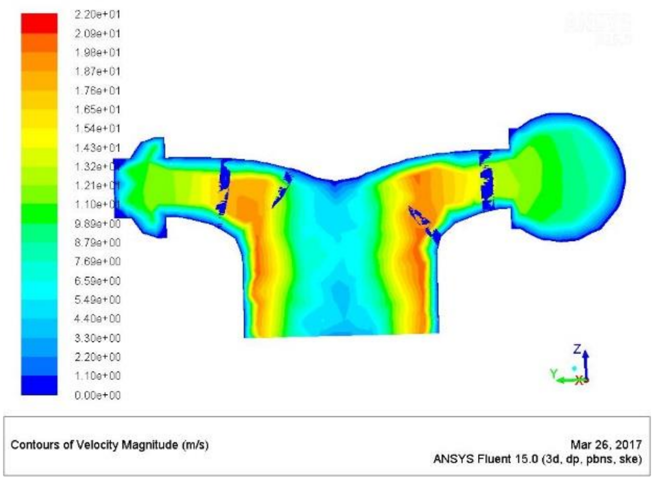
Velocity contours for circular profile at 4m/s



Velocity contours for circular profile at 2m/s

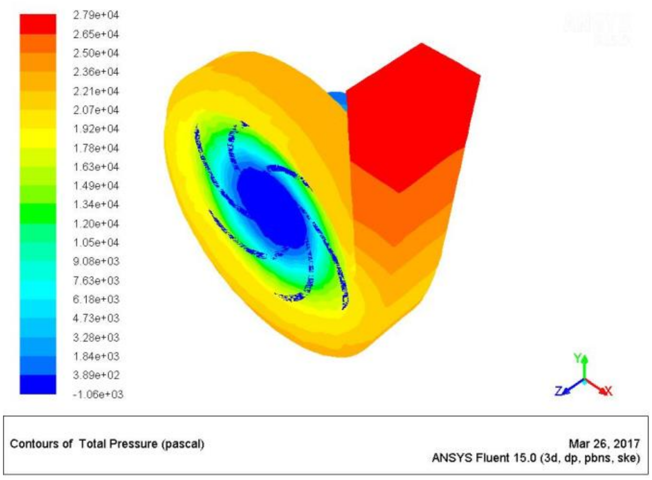


Velocity contours for rectangular profile at 6m/s

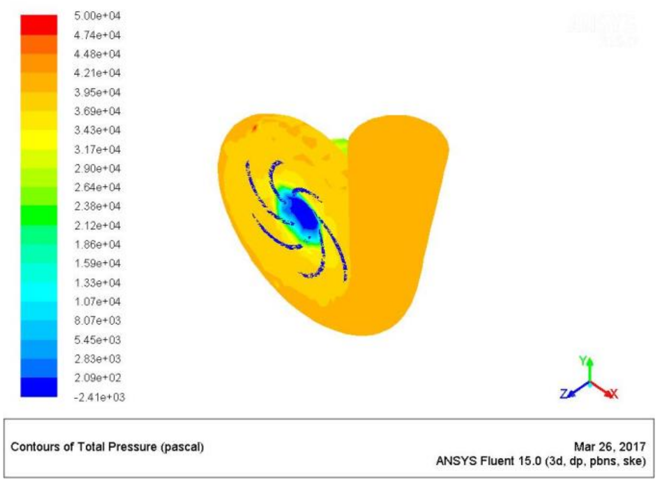


Velocity contours for circular profile at 6m/s

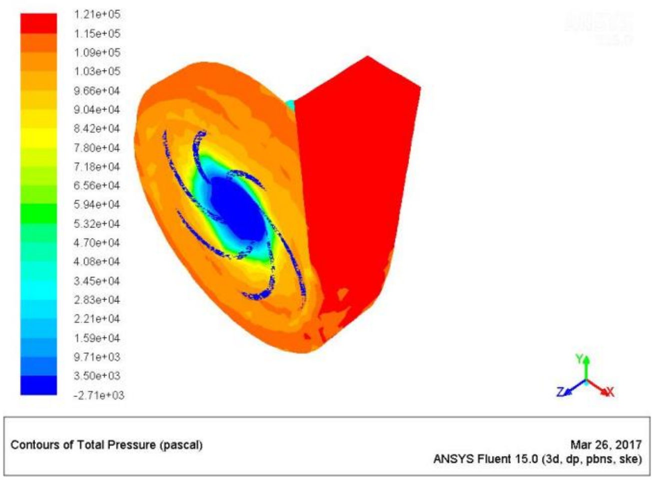
FIG: 4 Velocity contours at mid-section



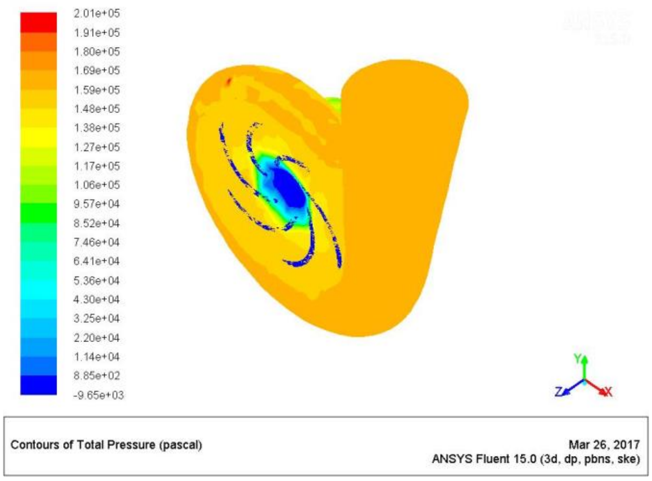
Pressure contours for rectangular profile at 2m/s



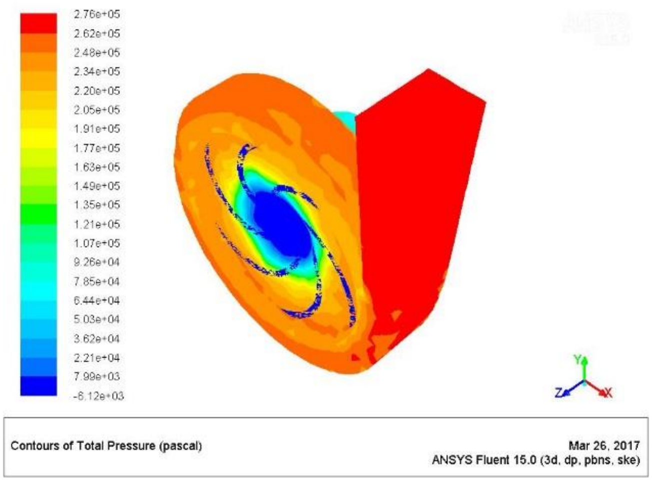
Pressure contours for circular profile at 2m/s



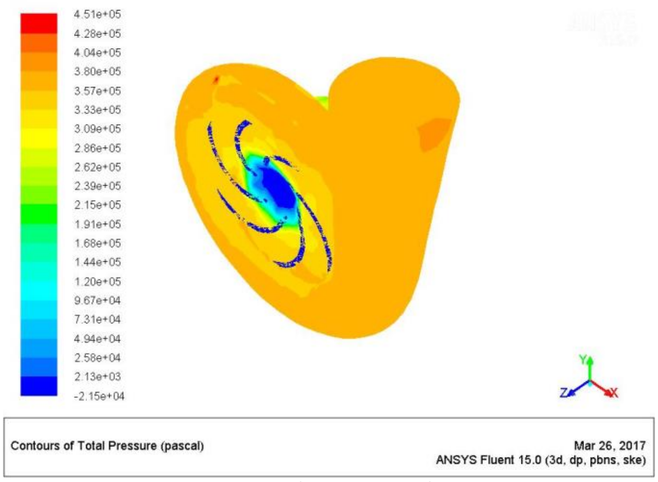
Pressure contours for rectangular profile at 4m/s



Pressure contours for circular profile at 4m/s



Pressure contours for rectangular profile at 6m/s



Pressure contours for circular profile at 6m/s

Fig: 5 Pressure Contours at mid-section

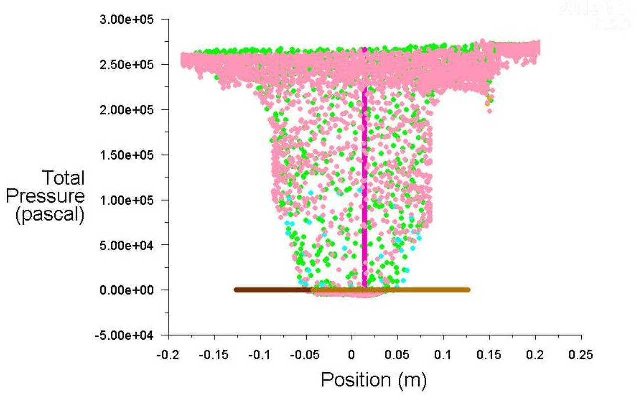


Fig. 6. Pressure Graph for Rectangular Model

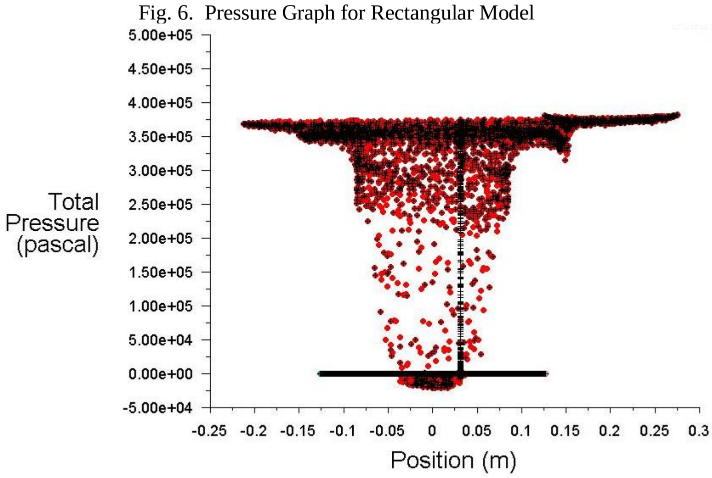


Fig. 7. Pressure Graph for Circular Model

CONCLUSIONS

The objective of the thesis was automating the process of setting up a CFD simulations. It can be concluded from the thesis work that the process can be simplified to a large extent. The performance reduces the amount of time needed to set up the geometry and mesh. For the model, setting up the geometry and the blocks for the mesh was very time consuming. To create a high quality, mesh some modifications is most likely needed. The same applies to the rectangular profile. The methods developed for the wastewater treatment flow passages have shown good results for most geometries. When the geometry is more complex, e.g. more twist on the guide vanes, some manual modifications might be necessary. The process can possibly be further developed with more knowledge and experience in CFD. The set up time is also highly dependent on mesh density that depends on flow velocity. Comparing the Circular and the Rectangular model it can be concluded that the circular can provide a good estimate of the performance, e.g. outlet velocity at same inlet loading. To ensure the quality of the 3D geometry the two model has to be used to simulate the volute and as well as the Guide vanes. It can be concluded that the CFD is a promising tool but could be improved further to generate even more accurate representation of radial turbine that could finally be manufactured. The future developments involve better geometrical representation, including the volute inlet and outlet, trailing edge of the stator and leading edge of the rotor. From the CFD results it can be concluded that sometimes over predicts the efficiency and could give a better estimate.

REFERENCES

[1] K. Galbraith, “How Energy Drains Water Supplies”, New York Times, September, 2011.
[2] IDAE, Fundación OPTI, “Consumo Energético en el sector del agua”, Estudio de rospectiva. Tecnologías del agua, 2010.
[3] P. Caldwell, "Energy efficient sewage treatment can energy positive sewage treatment works become the standard design?" in Proceedings of the 3rd European Water and Wastewater Management Conference,22nd-23rd September, 2009.
[4] S. Gillot, B. De Clercq, D. Defour, F. Simoens, K. Gernaey and P. Vanrolleghem, "Optimization of wastewater treatment plant design and operation using simulation and cost analysis," in Proceedings of 72nd Annual WEF Conference and Exposition, New Orleans, USA, 1999, pp. 9-13
[5] Keck, H.; Sick, M. Thirty years of numerical flow simulation in hydraulic turbomachines. Acta Mech. 2008, 201, 211–229.
[6] Cervantes, M.J.; Trivedi, C.; Dahlhaug, O.G.; Nielsen, T. Francis-99 Workshop 1: Steady operation of Francis turbines. J. Phys. Conf. Ser. 2015, 579, 011001.
[7] B. Daniel Marjavaara, “CFD driven optimization of hydraulic turbine draft tubes using surrogate models”,Doctoral Thesis, Depertmant of Applied Physics and Mechanical Engineering, Lulea University of Technology, Porsön, Sweeden, 2006.
[8] Munson, Young, Okiishi and Huebsch, Fundementals of Fluid Mechanics, 6th ed. Asia: Wiley, 2010, ch.12.
[9] G.I. Krivchenko, Hydraulic Machines: Turbines and Pumps; Moscow: Mir Publishers, 1986.
[10] J. Raabe, “Hydropower: The Design, Use, and Function of Hydromechanical, Hydraulic, and Electrical Equipment,” VDI-Verlag, Verlag des Vereins Deutscher Ingenieure, Düsseldorf, 1985.