International Journal of Latest Research in Engineering and Technology (IJLRET)
ISSN: 2454-5031
www.ijlret.com || Volume 02 - Issue 08 || August 2016 || PP. 64-75

NUMERICAL ANALYSIS AND PARAMETRIC STUDY OF DUCT CROSS FLOW TURBINE FOR IMPROVED EFFICIENCY

Anuj Agarwal[a] , Dr.Vilas Warudkar[b],
[a]*Department of mechanical engineering ,M.tech scholar, MANIT Bhopal,462003,Madhya Pradesh.*
[b]*Department of mechanical engineering ,Asst.Professor ,MANIT,Bhopal, 462003,Madhya Pradesh.*

Abstract: There is a growing interest in utilizing tidal current energy for power generation which has led to extensive research on this source of renewable energy. The work presented in this project aims to study the tidal current energy extraction using a cross-flow turbine. The bi-directional flow of tidal currents is used to drive a uni-directional cross-flow turbine. Thus in present study cross-flow turbine studied is placed in an augmentation channel, having a nozzle and a diffuser. The performance of the device is studied numerically using the commercial ANSYS Fluent code while the model is being build using SOLIDWORKS.
The internal flow characteristics of the turbine are studied for various cases. Results of the numerical analysis are presented in terms of pressure contours, streamlines and velocity vectors. A good agreement was observed between literature and present study .A comparative study has been performed by changing the turbine inlet profile from parabolic to invert parabolic in order to study the impact on pressure and velocity change. Moreover a parametric study of variations in TSR ratio and radius of blades is done for their effects on Power coefficient and Turbine Torque. The tip speed ratio was varied from 0.8- 1.4.The analysis from above study can be used for enhancement of performance of cross flow Darrius turbine.

Keywords: Cross flow turbine, SOLIDWORKS, ANSYS Fluent, Parametric Study, Power coefficient , Torque, Performance enhancement

1. Introduction

The urgent need to establish a clean, safe and affordable energy supply has placed increased emphasis on the exploitation of new renewable energy sources. The ocean offers immense potential for clean energy extraction and besides wave power and tidal barrage technologies, tidal stream turbines have been identified as prospective marine energy converters. This presents numerical investigations of the hydrodynamics of generic marine cross-flow turbines.

Evidence suggests that global energy generation, currently dominated by fossil fuels, is one of the prime sources for causing or at least accelerating climate change. A strong correlation between the earth's average temperature and the global concentration of carbon dioxide has been demonstrated. Another key factor is energy security that is how to reliably match demand and supply in years to come.

These aspects underline why the renewable energy sector has received renewed attention over the last few decades; renewable energy is expected to play a pivotal role with regard to solving the challenges of climate change and establishing a safe and affordable energy supply. The UK has manifested its support of the renewable energy sector by committing to ambitious goals, such as 20% of the country's electricity come from renewable sources by 2020, and reducing carbon dioxide emissions based on 1990 levels by 60% by 2050, as outlined in DTI (2003). While the electricity generated from renewable sources in the India has increased from 10 KWh/yr in 2000 to 25 KWh/yr in 2010, it is still only about 7% of the India's electricity generation that came from renewable sources in 2010. In order to achieve its goals the India is required to significantly increase the contribution from renewable sources to the energy supply over the next 10 years. So far, the focus of investors, developers as well as the media has primarily laid on biofuels, wind and solar energy. More recently, however, the vast untapped energy source of the ocean has also caught the interest of the researchers. It was seen that previous researches were done on bi-directional cross flow turbine and augmentation channel strongly influenced the flow and the turbine performance. In addition, the flow resistance by the augmentation channel and the blades forced the flow to divert away from the augmentation channel, reducing the flow velocity and turbine efficiency. Most of the literatures focused on experimental study which were restricted to many constraints like high cost and constant parameters turbine model. Not much literature were focused on 3-D simulation of cross flow turbine and they ignored effects of various parameters such as TSR, blade radius and channel shape on the performance of turbine. Also parametric study for various parameters is missing in many literatures. In this study, performing of CFD analysis of turbine using ANSYS Fluent and studying various parameters like TSR ratio, radius of blades and turbine channel profile and their effects on Power coefficient and Turbine Torque is done.

1.1 Objectives of this study

- To generate numerical model of cross flow turbine using Solid works and ANSYS Design Modular.
- To perform CFD Analysis of Cross flow turbine using ANSYS Fluent.
- To Study the pressure, velocity and streamline contours of turbine.
- To perform comparative study by varying inlet channel profile of turbine.
- To perform parametric study (varying parameter like TSR and blade radius) and studying the effects on performance of turbine.

2. Numerical analysis and Parametric study using ANSYS Fluen

Here it is chosen to use the CFD software ANSYS FLUENT 14.5.0, which solves the Reynolds-Averaged Navier-Stokes (RANS) equations, using a finite volume approach. For this study, ANSYS FLUENT is used as a three-dimensional, pressure-based, segregated, implicit, incompressible flow solver. In the pressure-based approach the velocity field is computed from the momentum equations, while the pressure field is computed by solving a pressure correction equation, which is obtained by manipulating continuity and momentum equations. The solver employed for this study used a segregated solution algorithm, in which the governing equations for the solution variables are solved sequentially from one another. When being solved, the individual governing equations are segregated from all other equations. In order to apply cross flow turbines for tidal power generation, the necessary parameters are fluid direction, angle to develop the optimum angle of attack on the blade and the outlet angle of the runner blade. To satisfy these necessary conditions, the nozzle was installed into turbine inlet area. Also, to produce maximum torque, the design of the blade and the angle of the nozzle to turbine inlet area were considered. The blade length is 464 mm, the axial length is 3,200 mm, the number of blade is 30 and the thickness of the blade is 26.24mm.There were 2 bodies in the overall CAD model that has been developed in Solid Works.

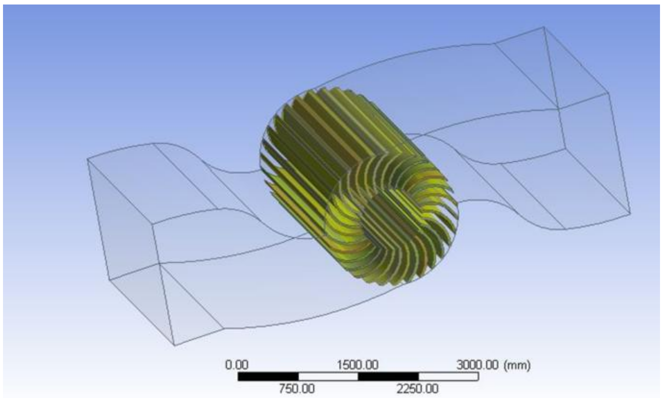

Figure 1: Ductless Model of Cross flow turbine

2.1 Mesh Generation

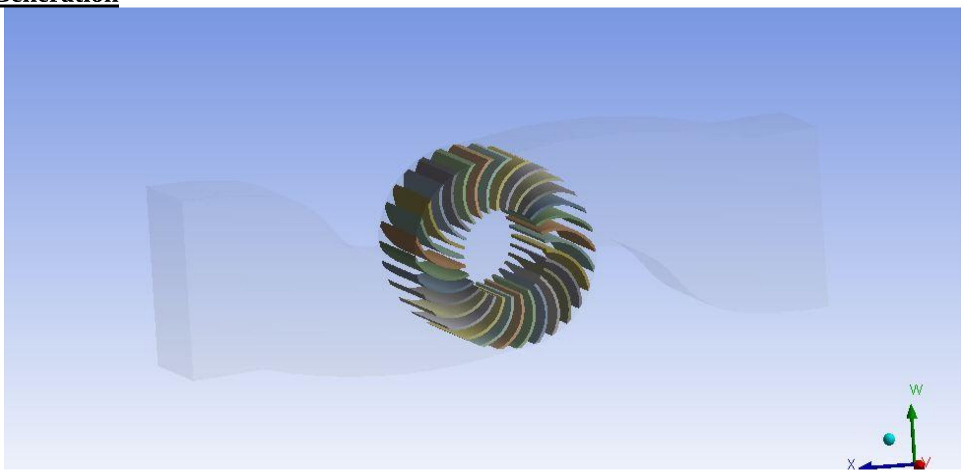

Figure 2: Meshing domain for Turbine

There are two domains in fig. 2 and these are:
1. Runner Blades ---- Solid Domain
2. Flow Channel ----- Fluid Domain

Model was imported in ANSYS Meshing after the initial CAD model import was completed over ANSYS Design Modeler.

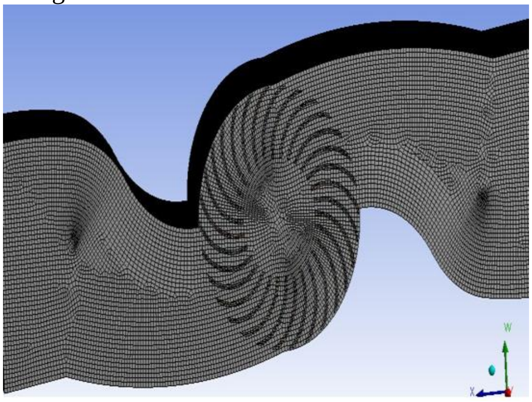
Figure 3: Meshed model of turbine

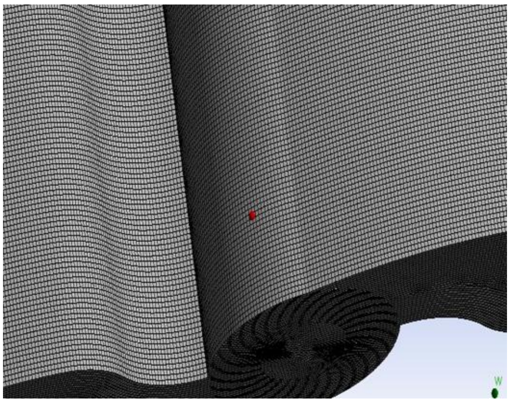
Figure 4: Zoomed view of mesh from top

Meshing was based on CFD Mesh with relevance 40 (more the relevance more finely is the mesh, it should be optimum to avoid large number of nodes). Hex dominant meshing was performed with maximum face size of 30mm and max size of 70mm in order to finer the mesh according to model geometry.
After generation of mesh it is found that--
Total no of nodes- 1429596
Total no of elements- 1316956
The number of nodes and elements were considerably nearby to number of elements considered in the literature that has been followed.

2.2 Import Mesh to ANSYS Fluent

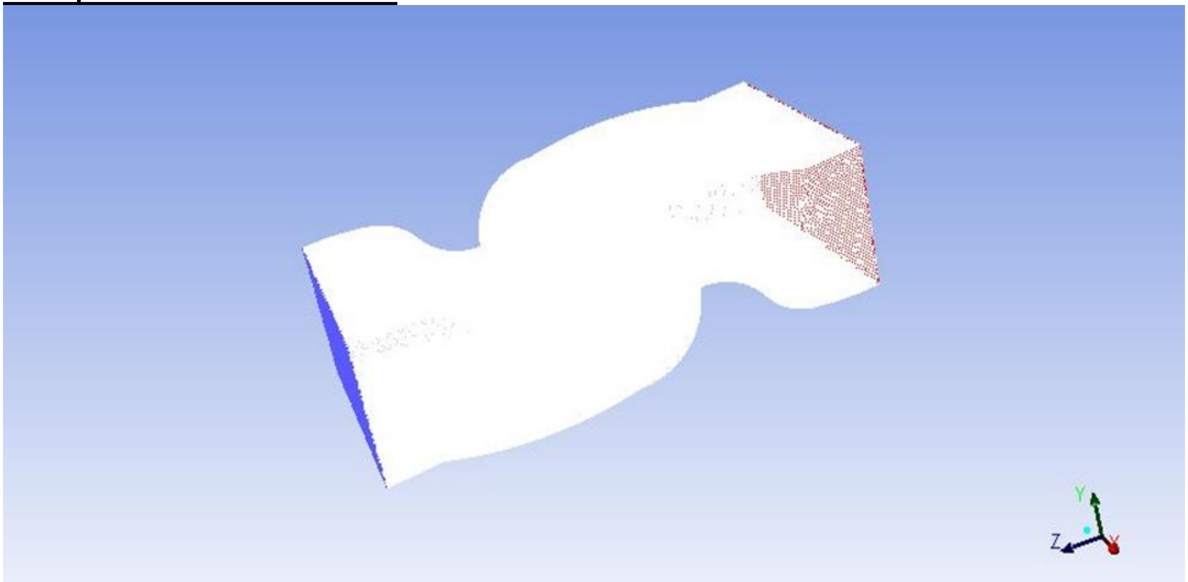
Figure 5: Imported model in ANSYS Fluent

Material used
- Water as the fluid Medium
- Turbine blades made of Aluminum as the solid Bodies

Standard k-ϵ model

In this model, the dissipation rate equation is derived from the mean square vorticity fluctuation which is fundamentally different from the Standard k-ϵ model. Several realizability conditions are enforced to solve the Reynolds stresses. The different benefits of using this model are: It accurately predicts the spreading rate of both planar as well as round jets. This model is also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation and recirculation**.**

2.3 Boundary Conditions

Velocity inlet-2.5m/s, Pressure outlet- atmospheric and all other conditions same as literature (K-e turbulent model), Transient flow (varying with time). Since there were 1.4 million elements so it was very difficult to run the simulation with fewer ram so 120 iterations were performed.

2.4 Convergence Criteria

Calculations reported from now onwards have been done with the general-purpose commercial package FLUENT, based on the finite-volume method. UPWIND scheme of Second-order is used in order to discretize energy, momentum and PRESTO for pressure correction equation was used while the SIMPLE algorithm is incorporated for the treatment of the velocity–pressure coupling. The solution gets converged after iterations as shown in figure 6 below. Conservation equations, which were solved for the control volume to yield the pressure and velocity fields for the water flow in the temperature fields for the absorber plate and the absorber tube. Total of 192 iterations were performed for the solution to converge and drop residuals to 10-4 as can be seen below.

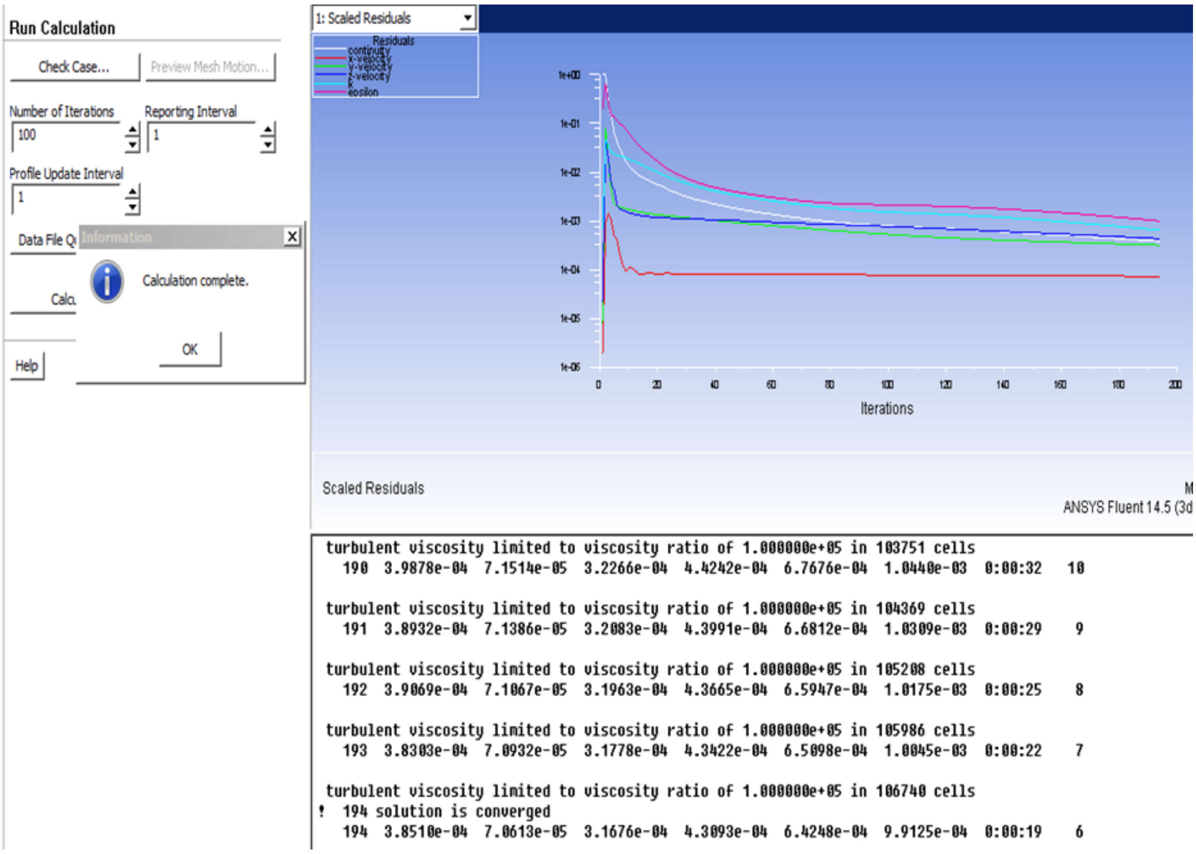

Figure 6: Convergence Plot

3. Results and Discussions

The CFD analysis of cross flow turbine was performed using ANSYS Fluent and thus obtaining various plots (Velocity, pressure and streamline) using ANSYS CFD post. After running the solution velocity vectors was plotted to verify with literature

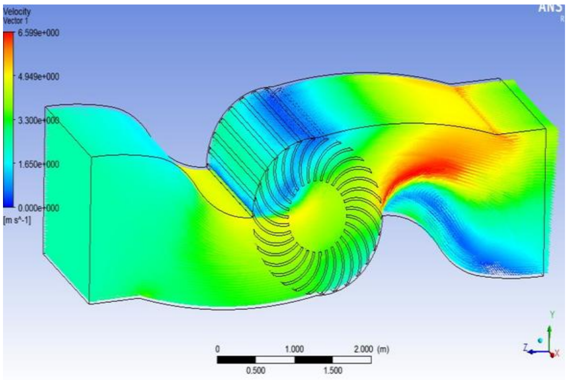

Figure 7: Velocity vector of full model

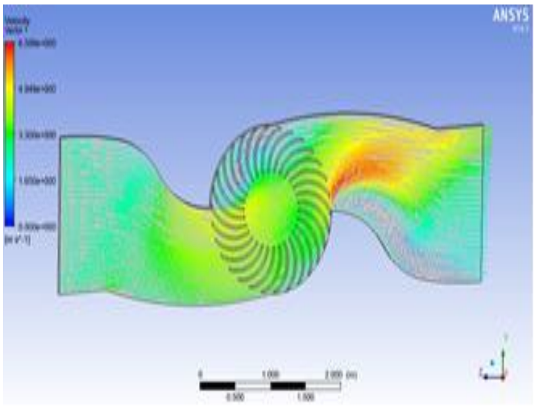

Figure 8: Velocity vector around YZ plane

Thus here it can be seen overall variation of velocity within the whole turbine model, it can be observed that velocity is maximum at outlet of turbine.

It was observed from above graph that maximum velocity **was 6.59m/s**.While from literature it was observed that max velocity was **6.26m/s**. Thus a good agreement between velocity vectors could be observed between present simulation and literature study.

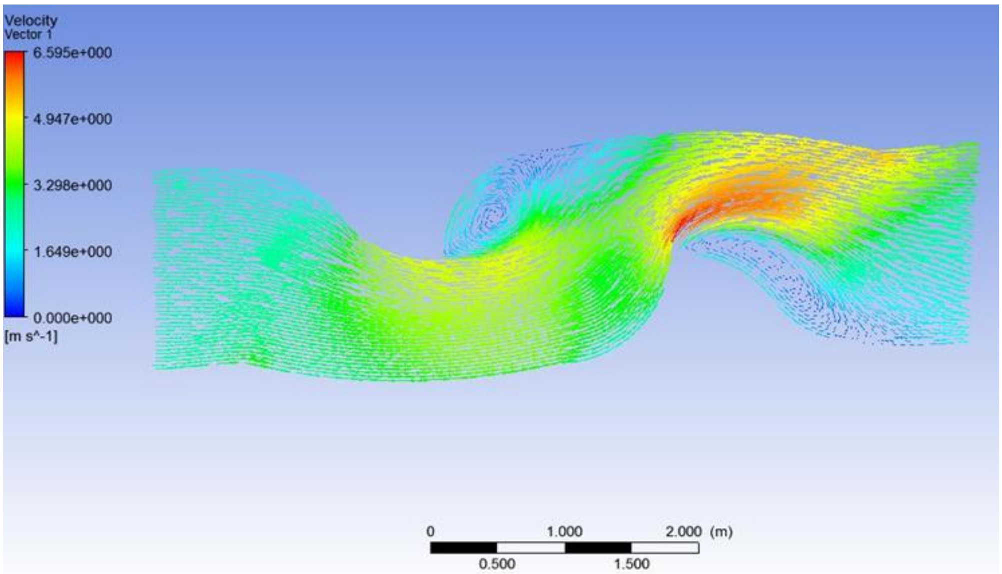

Figure 9: Velocity vector without wireframe

Thus from the above velocity plots and contours it can be observed that velocity increases after passing through turbine in the diffuser section along the edges. It can be explained by the fact that there is a sudden decrease in area in diffuser after water flows through turbine thus increasing velocity.

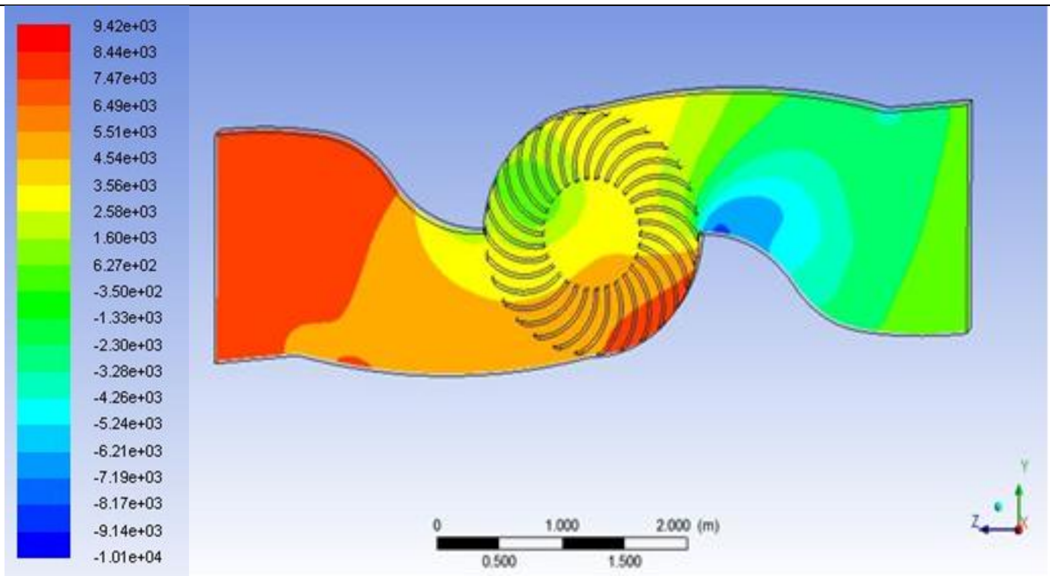

Figure 10: Pressure Contours

The maximum pressure was observed to be **9420Pa** in the above contour which is almost similar to that of Literature. From Pressure contours it can be observed that pressure on left side is greater than right side of turbine, since velocity at the inlet is less compared to that of velocity at exit of turbine. Moreover it can be seen that Velocity of water after passing through turbine increases to 6.6m/sec on the red zone of velocity vector, at that point only the pressure becomes almost negative(blue zone in above figure).

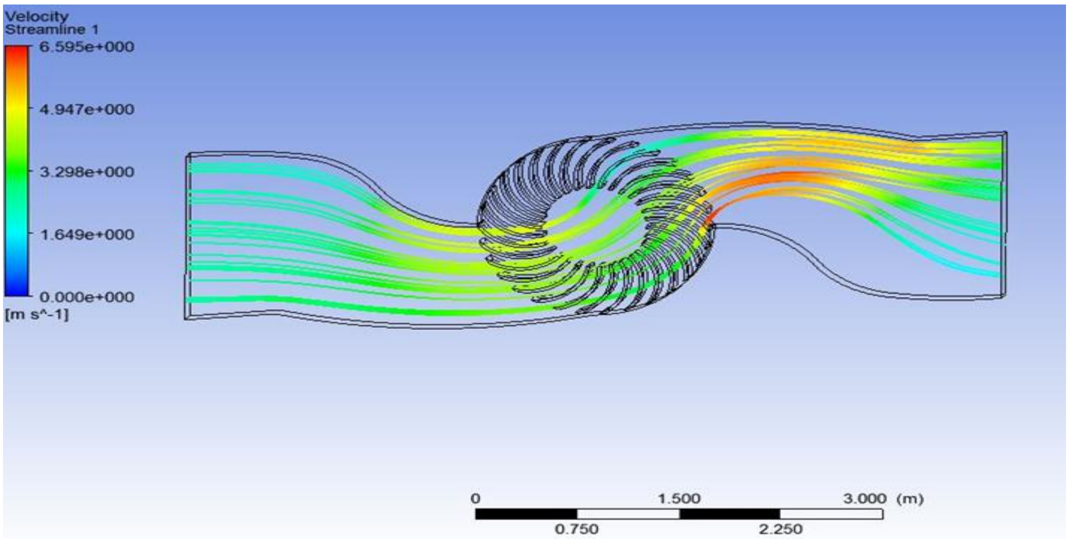

Figure 11: Streamline flow

The above figure shows streamline of how the water flow proceeds from inlet to exit of turbine.
The stream line is plotted to observe the full flow of water passing through turbine blades.

3.1 Numerical Analysis

The entire study is based on Modelling the cross flow turbine at different Tip Speed Ratios and the following equations can be used to calculate the Torque generated from the turbine, Power Developed and at last the Coefficient of Pressure Developed at the turbine blades. From this simulation the best case was selected to be analysed further.

The tip speed ratio was varied from 0.8-1.4 and the turbine performance characteristics were further analysed. The Torque and angular velocity calculated from this simulation was used to calculate the Cp by using the equation. From literature we get the following parameters
R=1696-768 mm= 928mm or 0.928m, TSR= 0.85-1.4(0.85, 1, 1.1, 1.3, 1.4), v= 1.38m/s. Thus using above parameters we can calculate the power and Cp.

Case 1 (when TSR=0.85)
ω = 0.85*(1.38)/0.928, ω = 1.257 rad/s
A=3.2 * 1.6 m2 = 5.12m2, ρ = 1000 kg/m3,
Torque from simulation was found to be = 17102 Nm,
Thus Cp = 0.5802, P= T ω= **P = 21.49 KW**

Case 2 (when TSR = 1)
1*(1.38) /0.928 = ω, ω = 1.5 rad/s
Torque from Simulation was 14920 Nm, so Cp = 0.59, **P = 22.38KW**

Case 3 (when TSR = 1.1)
ω = 1.72 rad/s, Torque from Simulation was 12918 Nm, so Cp = 0.589, **P = 22.21KW**

Case 4(When TSR=1.3)
ω = 2.01 rad/s, Torque from Simulation was 10880, so Cp = 0.585, **P = 21.86KW**

Case 5 (When TSR= 1.4)
ω = 2.4 rad/s, Torque from Simulation was 8948 Nm, so Cp = 0.579, **P = 21.48KW**

3.2 Validation of work

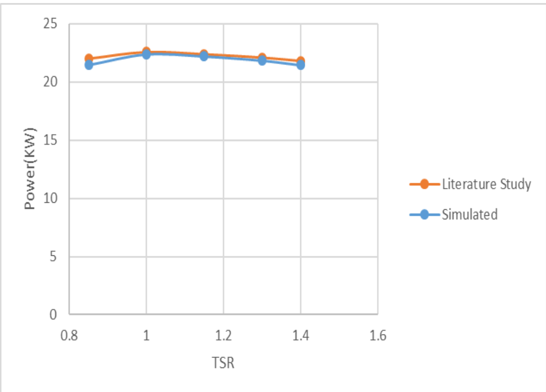

Figure 12: Power versus TSR

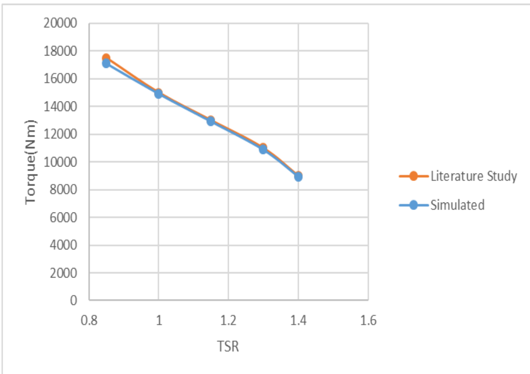

Figure 13: Torque versus TSR

From the above figures it can be observed that there is a good agreement between literature study the error is less than 7%. It can be observed that as TSR increases there is little effect on Power of Turbine but it can be observed from the graph that power is maximum at 1 TSR. The good agreement between literature and simulated study is observed as can be seen from the above figure. The overall Torque is decreasing with increase in TIP speed ratio due to fact that tip blade speed is increasing with respect to Tidal current speed leading to more energy losses.

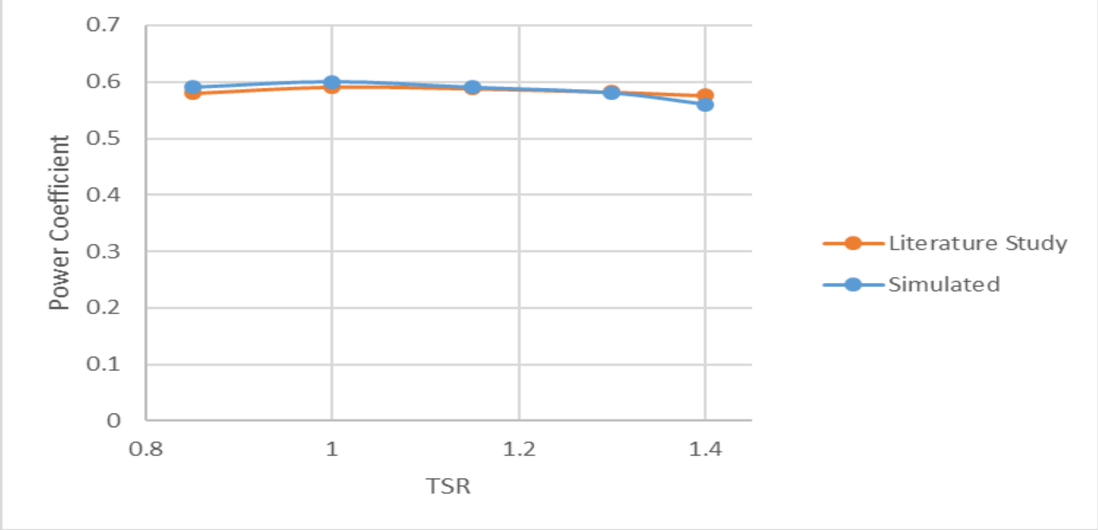

Figure 14: Power coefficient versus TSR

Moreover Power coefficient was also varied with TSR and compared with literature and good agreement can be observed from the above figure 14 since Cp is proportional to power it is maximum at TSR 1.

Table 1: Study from literature

Literature			
TSR	T	Cp	P(KW)
0.85	17500	0.58	22
1	15000	0.591	22.6
1.15	13000	0.588	22.4
1.3	11000	0.582	22.1
1.4	9000	0.576	21.8

Table 2: Study from present study

Simulated Results			
TSR	T	Cp	P(KW)
0.85	17102	0.59	21.49
1	14920	0.6	22.38
1.15	12918	0.59	22.21
1.3	10880	0.58	21.86
1.4	8948	0.56	21.48

3.3 Comparative study

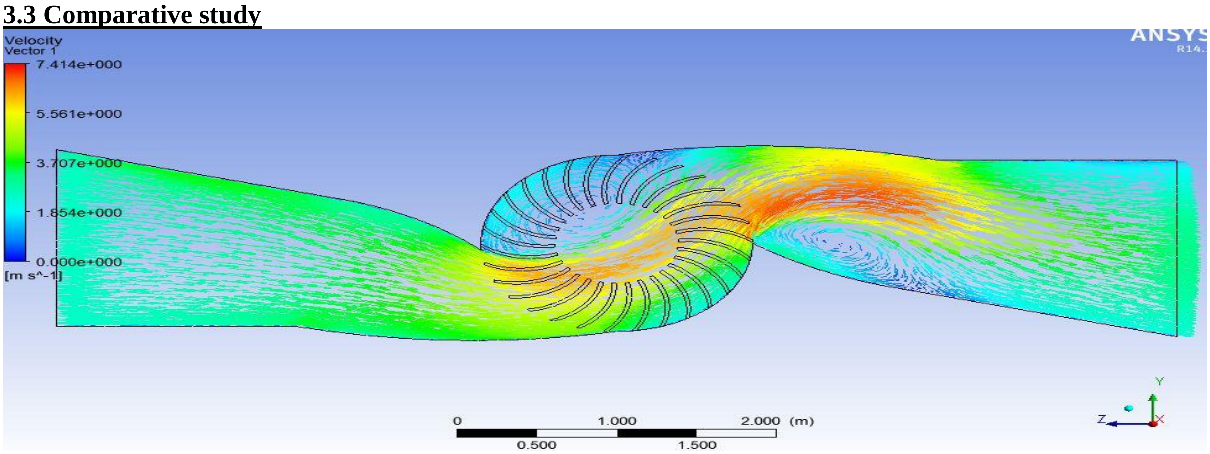

Figure 15: Velocity Contours for inverted parabolic channel flow

For comparative study the flow to turbine is being diverted through parabolic channel as compared to inverted parabolic in study. In order to study the impact on pressure and velocity change.

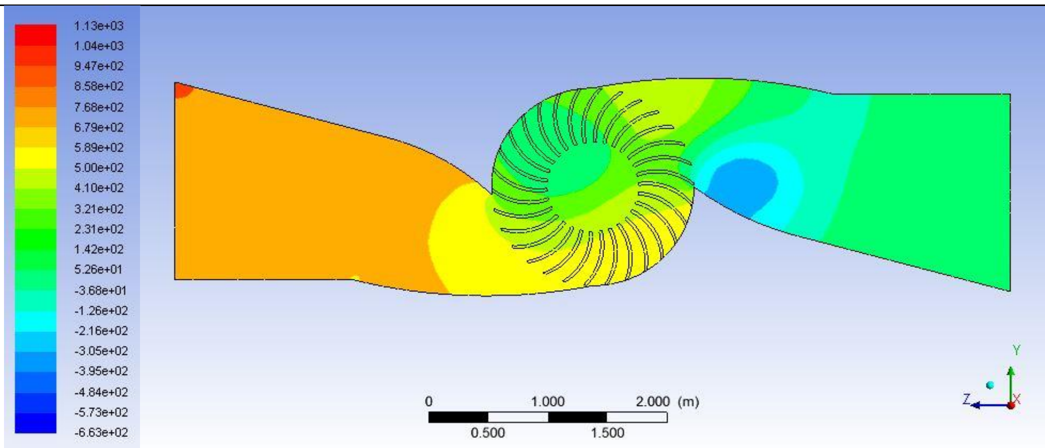

Figure 16: Pressure Contour for inverted parabolic channel

From the above pressure plot it can be observed that contours of pressure have become smooth as compared to previous study. But it can also be observed that the maximum pressure reaches to **1130 Pa** as compared to **9420 Pa** as was observed previously. Moreover maximum velocity at exit of runner blades reaches to **7.41m/sec**. Thus it can be observed that velocity is increasing at the outlet of turbine that is in the diffuser section and thus as a result pressure is decreasing.

3.4 Parametric Study

	A	B	C
1	ID	Parameter Name	Value
2	⊟ Input Parameters		
3	⊟ Geometry (A1)		
4	P1	Internal_diameter	768
*	New input parameter	New name	New expression

Figure 17: Setting internal diameter as input parameter

From the above figure it can be seen that internal diameter is kept as input. There could be a try to vary internal diameter and see for any changes in the model. The internal diameter of cross flow turbine is being varied and a parametric study is being performed to study its effects on turbine performance.

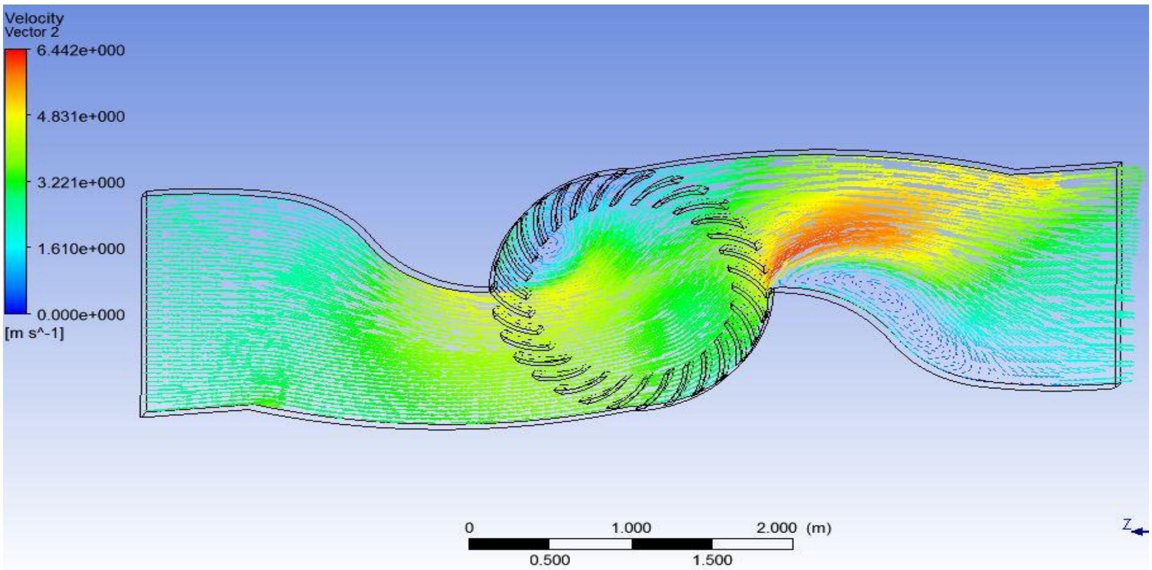

Figure 18: Velocity vector for increased internal runner diameter and decreased blade size

It can be observed that there is very little fluctuation in the flow from inlet to channel outlet. But some fluctuations can be for outlet velocity from the runner blade.

Table 3: Variation of inlet diameter and its effects on exit Velocity

Internal Diameter(mm)	**Runner Outlet Velocity(m/s)**
775	5.057681
871	5.33958
970	5.241418
1070	5.58
1170	5.648
1270	6.4

Thus from the above table the effects on velocity with varying internal diameter can be observed. As internal diameter is increasing the runner outlet velocity is increasing. So this leads to less pressure at the outlet thus internal diameter should be kept minimum to avoid pressure loss.

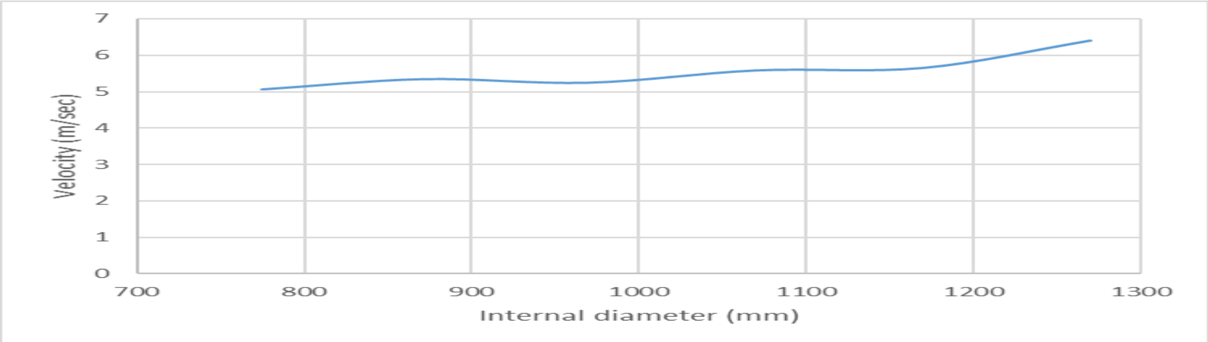

Figure 19: Runner Internal diameter versus runner outlet velocity

Thus it can be observed that velocity increases from 5m/sec to 6.5 m/sec at runner outlet. Thus it can be seen that turbulence increases due to increased velocity on increasing internal runner diameter thus reducing pressure. But velocity should be optimum so the internal diameter would range from 1000-1200 so that velocity do not increases beyond certain limit.

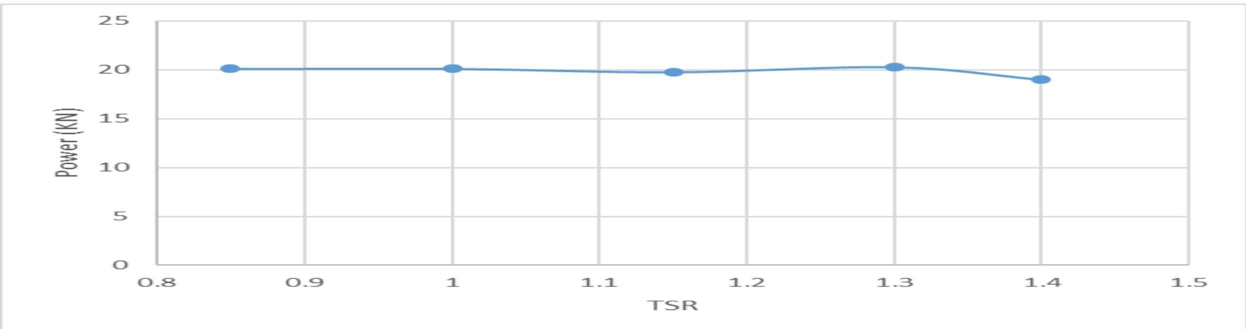

Figure 20: Power versus TSR of varying internal diameter of turbine'

It can be observed from above figure that as internal diameter is varied the power is changing with different TSR and can be found to be maximum at around 1.3 TSR.

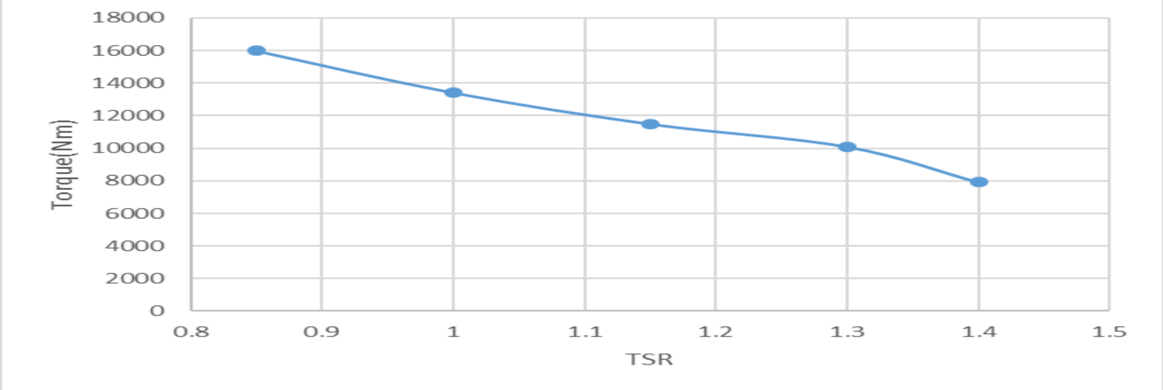

Figure 21: Torque versus TSR of varying internal diameter of turbine

The Torque is decreasing with increased TSR but the decrease is much more as compared to to our previous study.

Table 4: Torque and Power variation with varying internal diameter of turbine

TSR	T(Nm)	P(KW)
0.85	16000	20.112
1	13426	20.13
1.15	11500	19.78
1.3	10100	20.3
1.4	7920	19.008

Reducing in radius leads to reduction in ω thus leading to less torque and Power Output of Turbine.

4. Conclusion

- The numerical study and CFD analysis of Cross flow Darrius turbine is performed using Finite Volume method in ANSYS Fluent by solving Reynolds averaged Navier-stoke equation along with continuity equation.
- The turbulence model used is K-epsilon model.
- From analysis it was concluded that velocity is maximum at exit of turbine runner due to sudden decrease in area and as a result pressure reduces at outlet. Also the results obtained from present study were in good agreement with the literature.
- The turbine performance was measured based on Power, Torque and Power coefficient variation with Tip speed ratio and it was observed that with current turbine parameters used in present study maximum power is obtained at Tip speed ratio of 1. While it was observed that torque was decreasing with increasing TIP speed ratio.
- A comparative study is performed by varying shape of turbine inlet channel to inverted parabolic profile and it was observed that the losses along inlet due to inverted parabola decreases but there is slight velocity increase at the outlet of turbine.
- A parametric study is performed by varying turbine internal diameter and its effects on turbine performance were observed. It was found that the diameter should be kept minimum in order to avoid pressure losses at exit of turbine.

5. Future scope

- In present study Torque, power and Cp of turbine is found out for overall turbine performance, so in future there can also be a study of the frictional and channel losses for investigating the overall losses and its effects on turbine.
- In present study the duct was not considered in order to avoid more complex mesh and save computation time, in future we can include the duct also in the study.
- Also there can be variation of blade profile of turbine to enhance the turbine performance further.

6. Reference

[1]. Garrett, C. and Cummins, P., 2007. The efficiency of a turbine in a tidal channel. Journal of fluid mechanics, 588, pp.243-251.

[2]. Khalid, S.S., Liang, Z. and Shah, N., 2012. Harnessing tidal energy using vertical axis tidal turbine. Research Journal of Applied Sciences, Engineering and Technology, 5(1), pp.239-252.

[3]. Niblick, A.L., 2012. Experimental and analytical study of helical cross-flow turbines for a tidal micropower generation system (Doctoral dissertation, University of Washington).

[4]. Khosrowpanah, S., Fiuzat, A.A. and Albertson, M.L., 1988. Experimental study of cross-flow turbine. Journal of Hydraulic Engineering, 114(3), pp.299-314.

[5]. Fiuzat, A.A. and Akerkar, B.P., 1991. Power outputs of two stages of cross-flow turbine. Journal of energy engineering, 117(2), pp.57-70.

[6]. Olgun, H., 1998. Investigation of the performance of a cross-flow turbine.International Journal of Energy Research, 22(11), pp.953-964.

[7]. Pereira, N.C. and Borges, J.E., 1996. Study of the nozzle flow in a cross-flow turbine. International journal of mechanical sciences, 38(3), pp.283-302.

[8]. FUKUTOMI, J., NAKASE, Y. and Watanabe, T., 1985. A numerical method of free jet from a cross-flow turbine nozzle. Bulletin of JSME, 28(241), pp.1436-1440.

[9]. Kelso, R.M., Lim, T.T. and Perry, A.E., 1996. An experimental study of round jets in cross-flow. Journal of fluid mechanics, 306, pp.111-144.

[10]. Hee Jo, C., young Yim, J., hee Lee, K. and ho Rho, Y., 2012. Performance of horizontal axis tidal current turbine by blade configuration. Renewable Energy, 42, pp.195-206.

[11]. Lain, S. and Osorio, C., 2010. Simulation and evaluation of a straight-bladed Darrieus-type cross flow marine turbine. Journal of Scientific & Industrial Research, 69(12), pp.906-912.

[12]. Schlüter, J.U. and Schönfeld, T., 2000. LES of jets in cross flow and its application to a gas turbine burner. Flow, Turbulence and Combustion, 65(2), pp.177-203.

[13]. Kim, I.C., Wata, J., Ahmed, M.R. and Lee, Y.H., 2012. CFD study of a ducted cross flow turbine concept for high efficiency tidal current energy extraction. In Proceedings of Asian Wave and Tidal Energy Conference 2012(pp. 400-405).

[14]. Polagye, B., Cavagnaro, R., Niblick, A., Hall, T., Thomson, J. and Aliseda, A., 2013. Cross-flow turbine performance and wake characterization. InProceedings of the 1st Marine Energy Technology Symposium, Washington, DC.