Turbine Design

Abstract

Big water turbines are some of the machines that produce power and we can use computational fluid dynamics method to design and analyze them. Nowadays considering finances and also the need of energy in industries, using the whole potential of a source and getting the most possible energy out of it is considered something important. After many years of experimenting and experiencing different turbines and considering the head and fluid rate based on increasing specific speed, the turbine type for the best performance are respectively impulse turbines, radial turbines, mixed flow turbines, and at last axial turbines. For heads lower than 5.4 meters, VLH turbines (very low head)[1] are used which besides their working conditions, we must consider their environmental effects and the development plans, and also having in mind that they are installed on rivers. In this research first we extract geometrical data regarding the head, output power and desired efficiency and then it will be designed using Solidworks software and then using numerical analysis in ansys cfx software, we determine the accuracy of produced geometry. After reaching the desired geometry, cross section of blades made of NACA standard airfoil which are two dimensional airfoils will be created. Initial designed geometry of the head is 2 meters, the power is 190 KW and efficiency is 90%. Also any change in geometrical parameters like Stagger angle (which determines the blade rotation) and external diameter and their effects on turbine performance are studied.

Keywords: VLH Turbine, Very Low Head, Design, Airfoil, Stagger Angle

---

[1]- Very Low Head(VLH)

Preface

Advances and developments of a country are critically dependant on power industry. So paying attention to power industry as one of the infrastructure services in a country is very important. World wide statistics show an increase in demand to use electricity. Currently because of increasing oil price in the world and current situation of our area, the importance of hydroelectric power as a clean energy has increased. Attention and investment in hydroelectric power plants can help economical development, and also play a major role in energy development too. Today about one-fifth of produced electricity in the world is from hydroelectric turbines. This kind of turbine has the lowest pollution and a high efficiency (up to about 95%). The most common types are Pelton, Francis and Kaplan [25]. Water potential since old times has been one of the noteworthy energy sources. Today with the significant growth in industry, the need to this energy has increased. In addition, all big industries use electrical power in their machinery and tools. Knowing that electrical energy mainly is produced from fossil fuels which are irreplaceable sources, and the environmental problems that are caused by these sources, has increased enthusiasm in using water turbines to produce electricity. On the other hand, limited water potential and also the cost of power plant construction and delivering the electricity from the plant to consumers has made people turn to local power production in smaller scales. Using small water turbines can be a solution to achieve this goal. First step in designing turbine wheels is to determine the original dimensions which in addition to showing total geometry and size of the turbine, will give a cost estimation of mechanical and structural devices which are needed. Entry and exit diameters of the design and also the blade profile from hub to tip are some of the most important dimensions of Francis turbine wheel. To determine the geometry of the wheel first these dimensions are specified, and then total solitary shape of the wheel is determined and the layout of the blade will be given. Next this layout will be the base to fully design the blade profile.

Energy conversion depends on all different parts of turbine consisting of runner, fixed and moving blades, and wheel. Yet it’s the turbine wheel which has a primary role. The wheel blades design is important in energy conversion and it must be designed as good as possible. Although in designing turbines, study of distributor parts is important, knowing primary characteristics of the wheel is the most important part of the study. Determining dimensions in old turbines were done using hydraulic calculations and trial and error method, and then necessary corrections were made in order to optimize the design. Today main dimensions are extracted based on experimental relations from working turbines, and then necessary changes will be made to achieve intended results. So because of different methods of determining the dimensions, different producers use different sizes and forms of turbines even if they have the same application. The number of unknown parameters in designing turbines is bigger than the number of known parameters, and two quantities of height and flow will give many choices for the design and each design has a primary role in efficiency and output, so we can say that design process can not be unique. Different methods have common concepts and basically their difference is because of the experiences that the design is based on.

Water turbines are normally big systems and they’re designed for specific conditions. So they can’t be mass produced. If these systems are only tested after their production, then:

- Performing many testes on prototypes would be expensive and time consuming.

- Changing dimensions and turbine size, if test results don’t match the analysis data, would be very expensive.
- Also prototype tests, have limits like impossibility of changing the head, the need to apply static speed to the system, possibility of unstable available load based on correct and precise situation [21].

Because of the reasons mentioned, doing model tests is a definite necessity in order to predict the main system behavior. These tests have relatively low costs, acceptable accuracy and ease, and they’re widely and comprehensively achievable and their results can be extended to the main model using dynamic similarities.

It must be considered that in some cases in which there are many dimensionless parameters, it would be very difficult to make the system and the model completely similar. Because it’s necessary that many dimensionless parameters become equal simultaneously.

In this case, usually they remove some of the dimensionless parameters which have little effect on the results and only important dimensionless parameters are analyzed. Then they study the effects of removed parameters using theories or separate experiments.

Not involving Reynolds effects in relations regarding the behavior and the performance of turbo machines gives a relatively good estimation; because Reynolds number in most turbo machines is relatively high and the flow is completely disarranged. So changes in friction coefficient as a result of changes in Reynolds number are insignificant. Despite this, to study turbo machines behavior more exactly and to calculate their efficiency it’s necessary to consider changes in Reynolds number. In cases of high changes in Reynolds number (as in comparison between model and prototype, which because of big changes in dimensions and fluid velocity, Reynolds number can be very different), the efficiency can have relatively big changes and so Reynolds changes must be checked. There are many relations in order to calculate efficiency changes in turbo machines as a result of friction and we will mention some of them in next chapters. On the other hand using numerical analysis of full geometry can significantly help us avoid expensive and time-consuming tests.

In the past, different parts of turbines were studies separately; and there can be many reasons for this separate analysis of different parts of a turbine. First, for some time the codes were not able to solve two fixed and moving parts simultaneously; but now this problem is solved and we can solve the flow of rotating and non-rotating parts simultaneously. Second, it's because of limited capacity of computer in solving the full geometry simultaneously; and despite science advances, this issue still exists. There is no doubt that there is great benefit in simultaneously solving rotor, stator and suction pipe because this will avoid applying non-exact boundary conditions on every separately solving part [19].

Because of very small heights in lateral structure of many rivers, they can't be used to produce energy. So VLH turbines are used in order to produce hydroelectric energy in such places.

VLH turbine has standard impeller diameter of 3.15 - 5.0 meters. The maximum water that the turbine can store is between 8.2 m^3 to 27.1 m^3.

Because of VLH turbines being submerged, they have very little noise and they have very small effects on the environment around them. Also because of placing these turbines in rivers, the location of their equipments would be near urban areas and these areas wouldn't be inaccessible [21].

It's obvious that despite small height for falling water, if the required water flow is achieved by a suitable structure, using water turbines to produce electricity would be possible. But using contemporary common turbines in such conditions require infrastructures to produce this water flow and because of high costs of construction research, practically using common turbines is not beneficial.

So practically what's important is to use the turbine which we don't need significant infrastructures for it. In recent decade fluid dynamics method has been used to design many turbines and energy conversion systems. This method has been used in modeling Viscous flow around blades. Every method has its own accuracy and costs. To simulate computational fluid dynamics model we could use a software like Ansys CFX.

If turbine design data are as below, then we would have:

$P = 190\ Kw$

$H = 2.m$

$\eta = 90\ \%$

$P = \eta \rho g H Q \rightarrow Q = 10.8\ m^3/s$

$d_{tip} = 1.9\ m$

$d_{hub} = 0.42 d_{tip} \rightarrow d_{hub} = 0.798\ m$

Considering these calculations and given steps above, full data for each section would be as the table below:

	r	$\beta 1$	$\beta 2$	α	*Chord*	*stagger*	*L / D*	Re*10^6	*Mach*	*Solidity*
1	0.399	-32.16	58.24	51.28	0.329	16.96	9.75	1.8372	0.0039	1.0500
2	0.4833	-15.68	53.15	45.85	0.4089	24.36	12.3	2.0073	0.0035	1.0773
3	0.5549	-2.1	49.3	41.9	0.4754	30.67	14.68	2.2486	0.0033	1.0909
4	0.6182	8.73	46.21	38.84	0.5363	35.71	16.95	2.5647	0.0034	1.1045
5	0.6757	17.28	43.66	36.38	0.5934	39.61	19.15	2.9374	0.0035	1.1182
6	0.728	24.0	41.5	34.3	0.6477	42.63	21.29	3.3530	0.0036	1.1318

	6	7	1	4						
7	0.7779	29.53	39.66	32.61	0.6999	44.97	23.38	3.8021	0.0038	1.1455
8	0.8243	34	38.03	31.12	0.7504	46.82	25.44	4.2786	0.0040	1.1591
9	0.8682	37.71	36.6	29.83	0.7997	48.3	27.47	4.7781	0.0042	1.1727
10	0.910	40.84	35.32	28.68	0.8479	49.51	29.47	5.2977	0.0044	1.1864
11	0.950	43.51	34.17	27.65	0.8954	50.49	31.45	5.8354	0.0046	1.2000

Table 3-1 design data of turbine sections

To make it easy to create a 3D geometry and to decrease the number of sections to be drawn, the design would be done in 4 sections as below:

	r	$\beta 1$	$\beta 2$	α	*Chord*	*stagger*	L / D	Re*10^6	*Mach*	*Solidity*
1	0.3990	-32.16	58.25	51.29	0.3290	16.96	9.75	1.8373	0.0039	1.0500
2	0.6379	11.806	45.31	37.96	0.5637	37.12	17.691	2.7919	0.0034	1.1250
3	0.8092	32.605	38.55	31.60	0.7388	46.25	24.76	4.1454	0.0040	1.1625
4	0.9500	43.51	34.17	27.65	0.8954	50.49	31.45	5.8354	0.0046	1.2000

Table 3-2 design data for decreased sections of turbine.

Drawing blade sections

Several standard airfoils with different numbers and widths were studied and analyzed and at last they were chosen respectively as *NACA4404*, *NACA4405*, *NACA4407* and

NACA4412 sections considering NACA width and chord length of each section. Data for each airfoil was extracted from *XFoil* software, and after creating Stagger angle for the section, it will be used in Solidworks software to create 3D geometry. Below you see some steps of designing a 3D turbine blade:

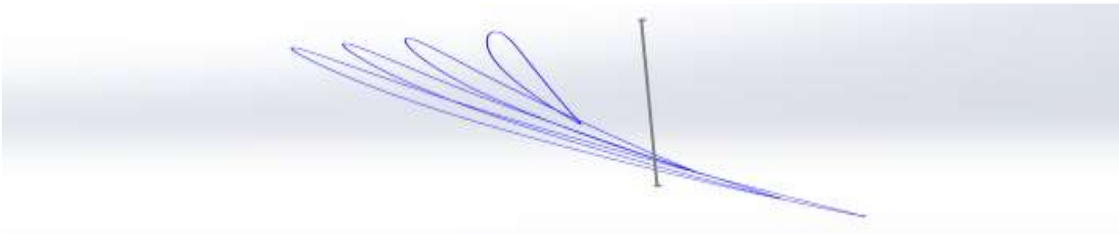


Figure 3-9 airfoils placed with desired Stagger angle and chord length and their position in relation to rotation axis.

Figure 3-10 creating 3D blade geometry.

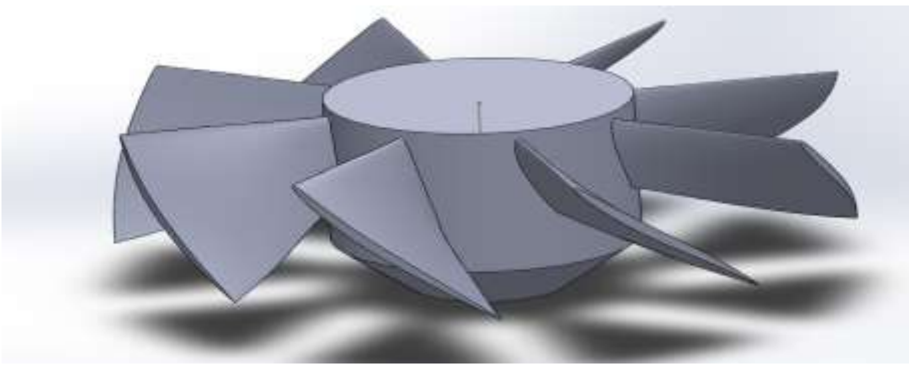


Figure 3-11 complete turbine rotor with 8 blades.

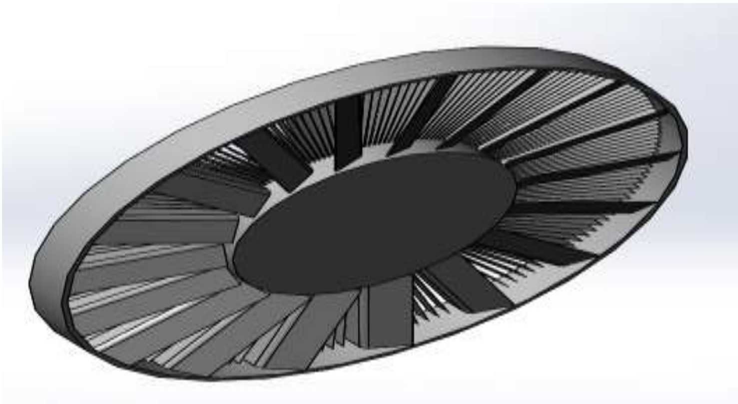


Figure 3-12 entry runner blades.

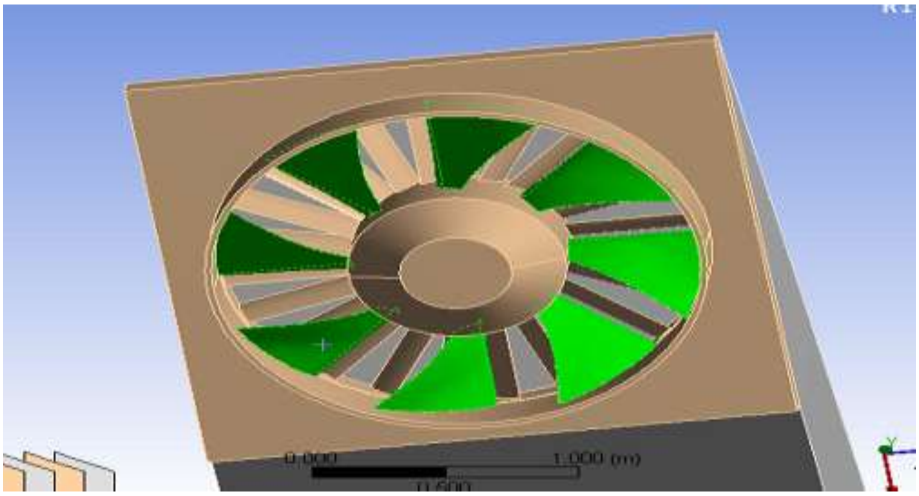


Figure 3-13 runner blades and their position in relation to fixed turbine chamber.

Numerical analysis and results

Software analysis

Due to current limitations in software analysis of the designed turbine, only one turbine blade with its flow direction is simulated and different conditions will be applied periodically to this blade and its surrounding geometry. This geometry is shown in the figure below:

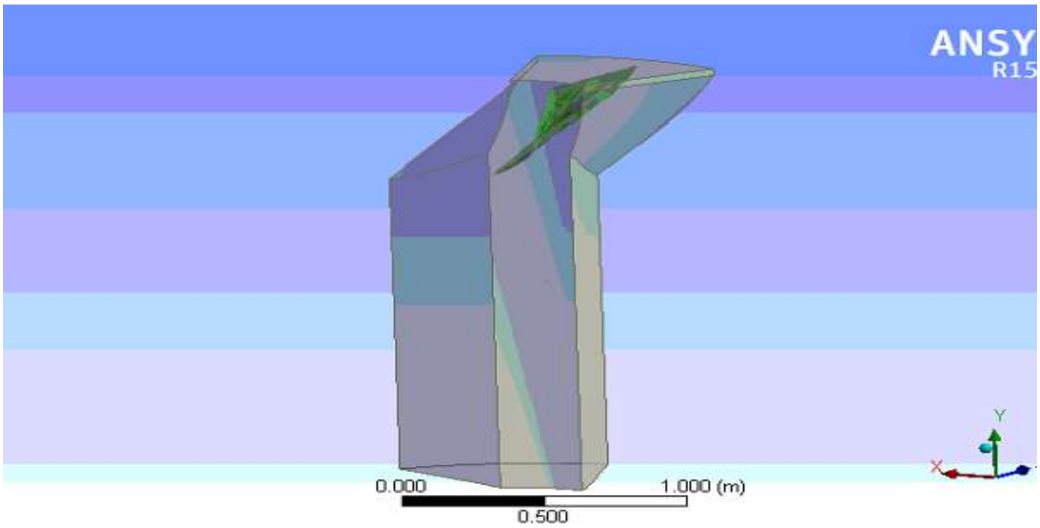


Figure 4-1 one-blade view and periodic conditions.

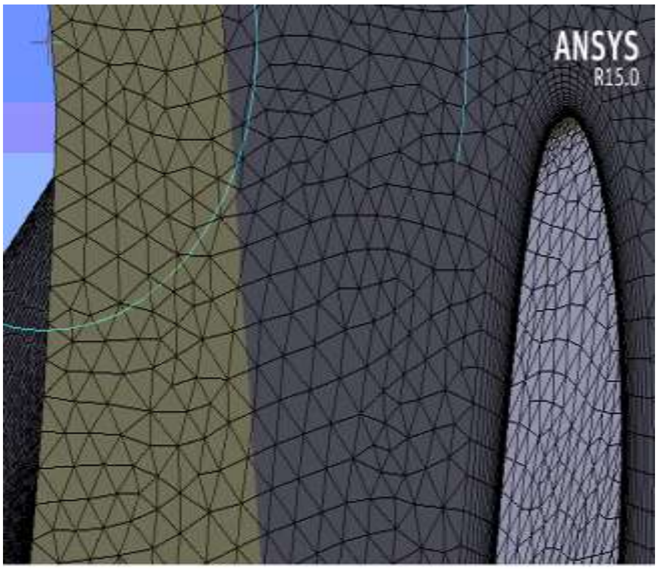
Producing mesh and its related settings

After creating intended geometry, it will be analyzed in Ansys15-CFX software. In the next sections we will see the produced mesh on the geometry and its settings:

Details of "Mesh"

	Physics Preference	CFD
	Solver Preference	CFX
	Relevance	0
⊞	**Sizing**	
⊞	**Inflation**	
⊟	**Patch Conforming Options**	
	Triangle Surface Mesher	Program Controlled
⊟	**Patch Independent Options**	
	Topology Checking	Yes
⊞	**Advanced**	
⊞	**Defeaturing**	
⊟	**Statistics**	
	Nodes	2527941
	Elements	12860102
	Mesh Metric	None

Figure 4-2 the produced mesh and its settings.

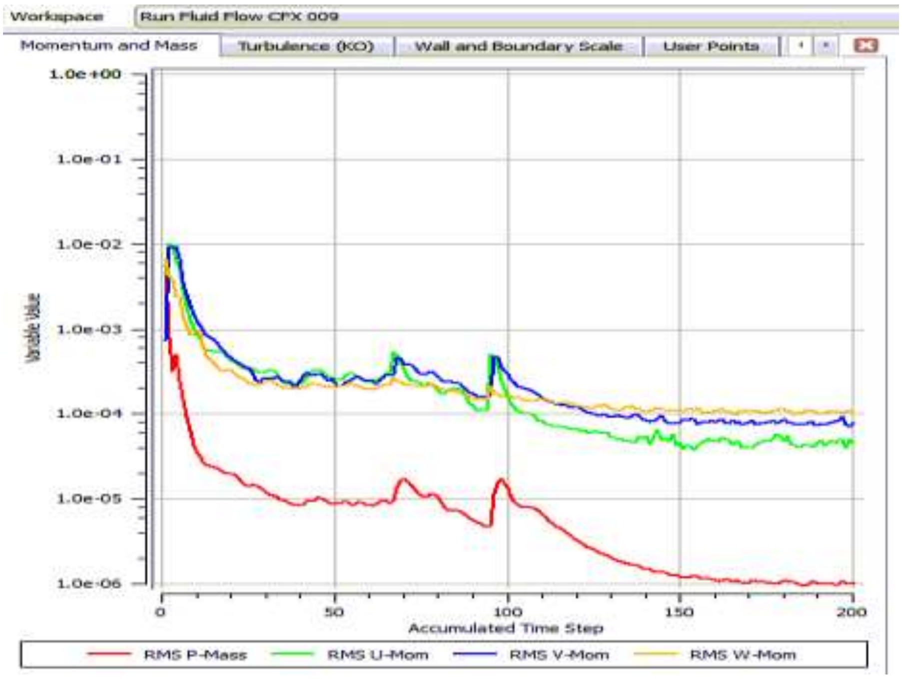


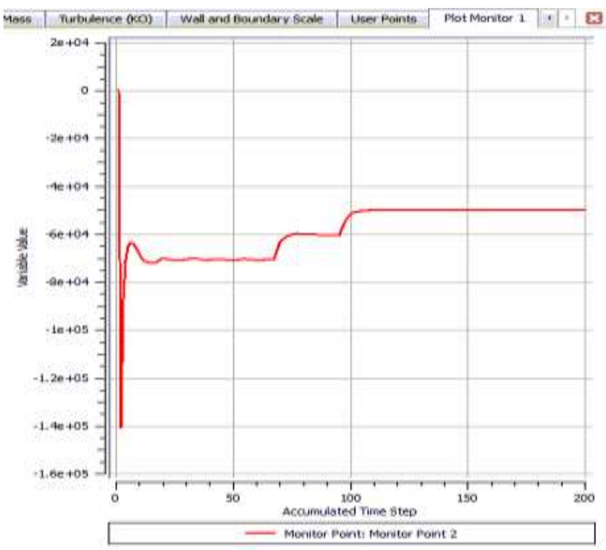
Details of "Inflation" - Inflation

⊟	**Scope**	
	Scoping Method	Geometry Selection
	Geometry	1 Body
⊟	**Definition**	
	Suppressed	No
	Boundary Scoping Method	Named Selections
	Boundary	blade
	Inflation Option	First Layer Thickness
	First Layer Height	2.e-002 mm
	Maximum Layers	25
	Growth Rate	1.21
	Inflation Algorithm	Pre

Figure 4-3 boundary layer mesh and its settings

To create boundary layer mesh on the blade wall, the first layer width is considered as 2e-5. This value affects the accuracy of the results and then *Yplus* value on the wall must be analyzed and its accuracy must be determined. In the following figures you see the solutions convergence and convergence of the head and the calculated torque:





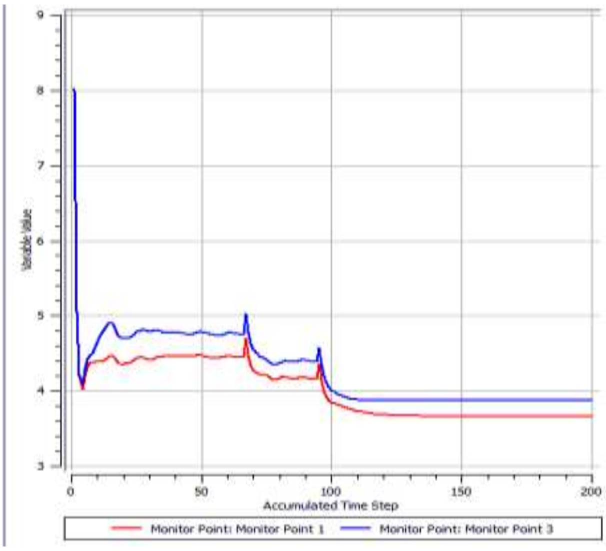


Figure 4-4 convergence charts for numerical solutions.

Drawing the results from the analysis

In this figure you see pressure contours on *pressure side* and *suction side* of the blade:

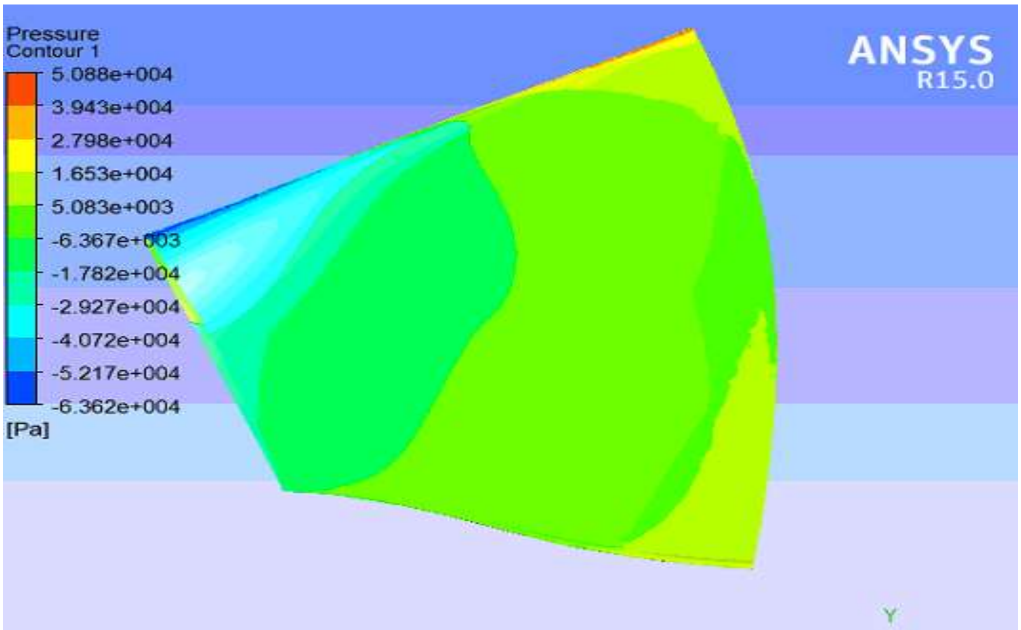


Figure 4-5 pressure contour for suction side

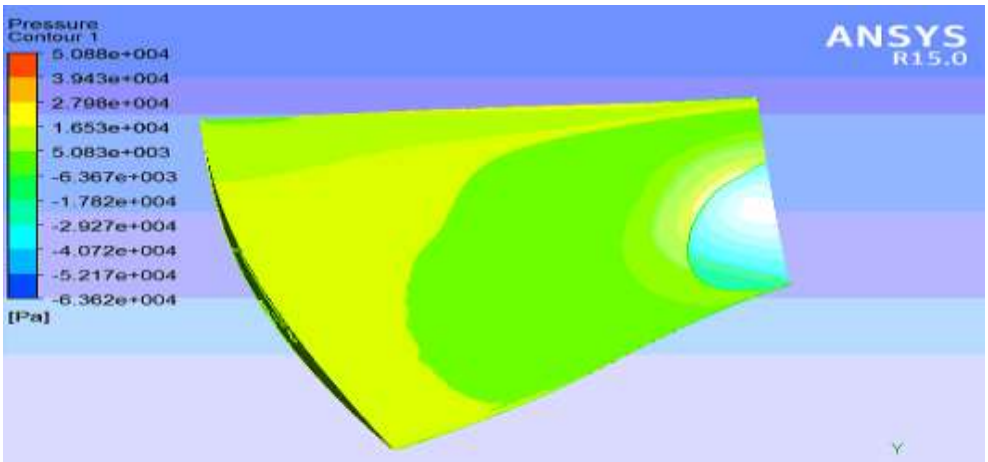


Figure 4-6 pressure contour for pressure side

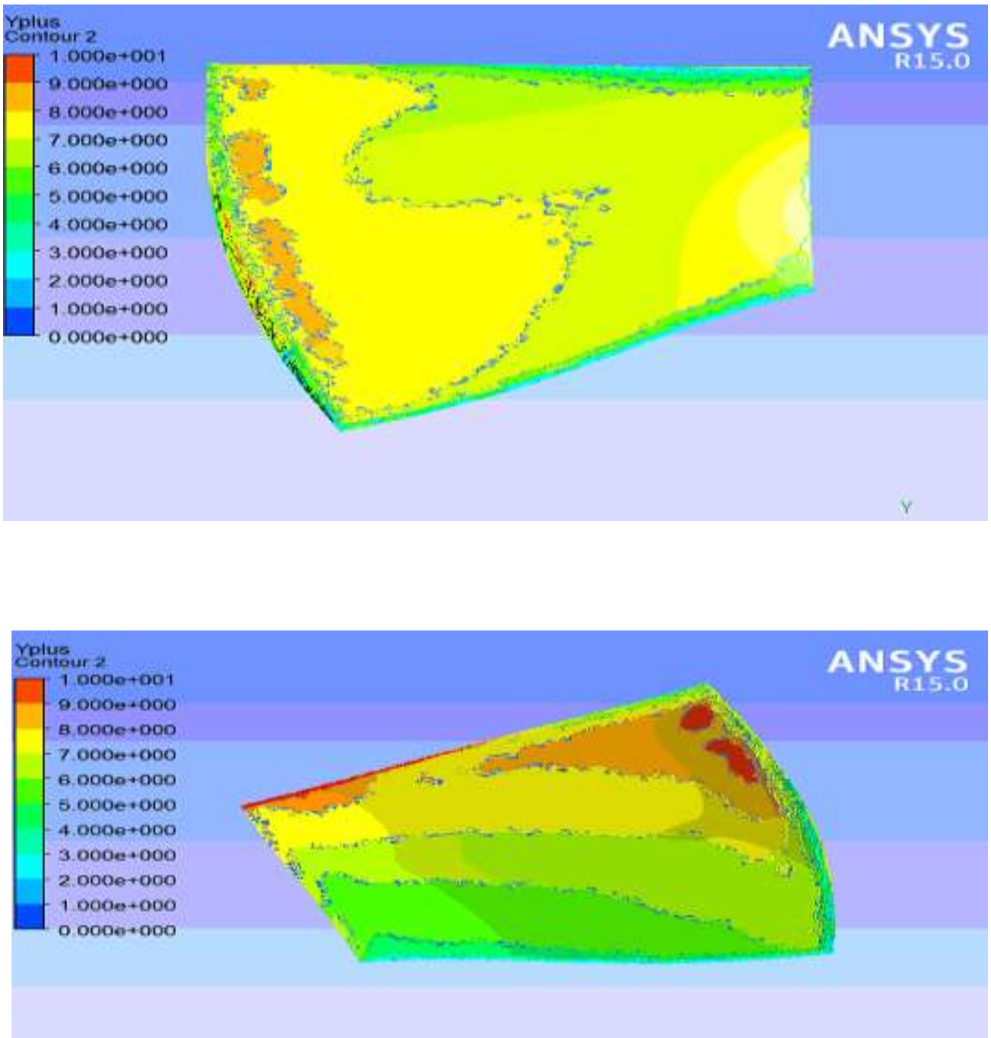


Figure 4-7 Y^+ contour on the blade.

Calculations sample

All steps in designing turbine blades will be repeated considering Stagger change in a similar way and the change in tip diameters in two different ways. Then the designed blade will be analyzed. Below you see a sample of extracted calculations from the software for the main turbine and other designed turbines (in the software only one blade is analyzed and it must be considered that we have 8 blades in a turbine, so for calculating output power and intended flow we must consider all these blades):

Head(m): *H=(Total Pressure in Stn frame@inlet- Total Pressure in Stn frame@outlet) / (ρ*g)*

Power(kw): *P=(torque_y()@blade* 8) ** $\omega(rad.s\wedge -1)$

Q(m^3/s): *massFlow@ inlet) * 8 / (ρ)*

P Total(kw)= ρ .g.Q.H $\longrightarrow$ *η = P Total(kw) / Power(kw)*

After calculating the above equations and considering $\rho = 998$, $g = 9.8$ and $\omega = 70rpm = 7.33rad/s$, we would have:

	η(%)	Q(m^3/s)	Power(kw)	Head(m)
Initial turbine	89.4	11.17	194.6	1.992
Turbine with higher Stagger	86.9	11.17	228	2.406

Table 4-1 comparison between initial turbine and 4 degree Stagger turn.

Now characteristic curve for this turbine at different rotational speeds is extracted. Because in these turbines - unlike Kaplan turbines which have adjustable angle for entry runner blades – angle of incidence for entry runner blades is fixed, synchronous equipments are used in these turbines in order to connect to electricity network and the rotational speed can change. In the following sections these charts are shown:

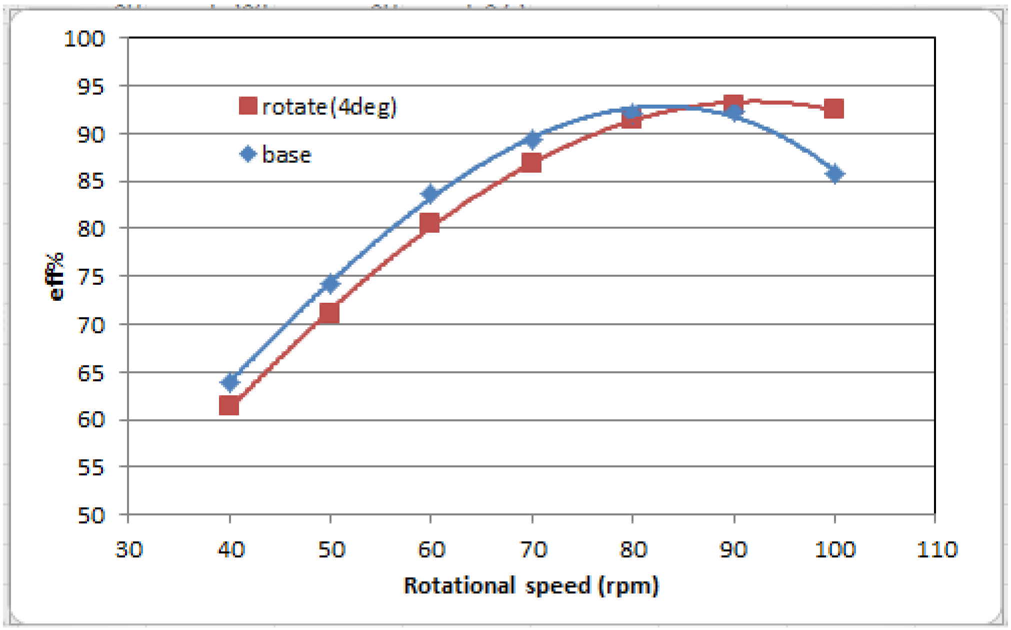


Figure 4-8 efficiency curve for 11.17 m^3/s flow

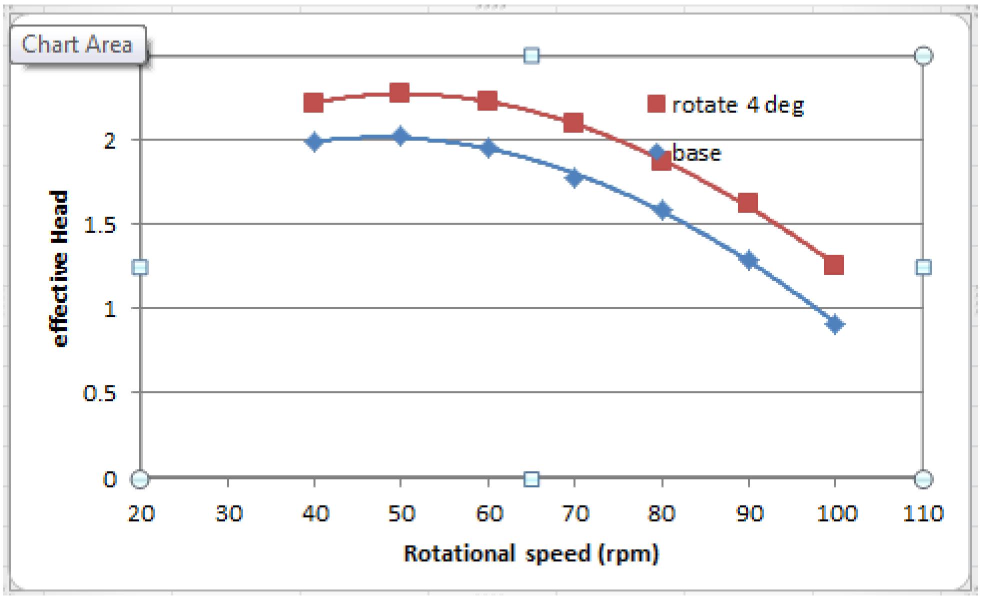


Figure 4-9 efficient head curve based on rotational speed.

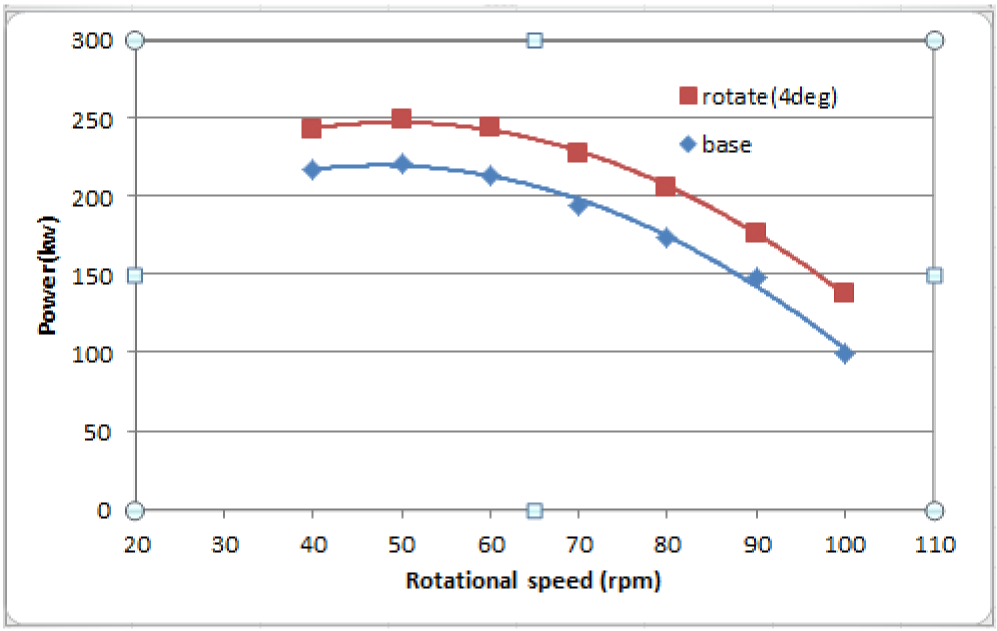


Figure 4-10 power curve based on rotational speed.

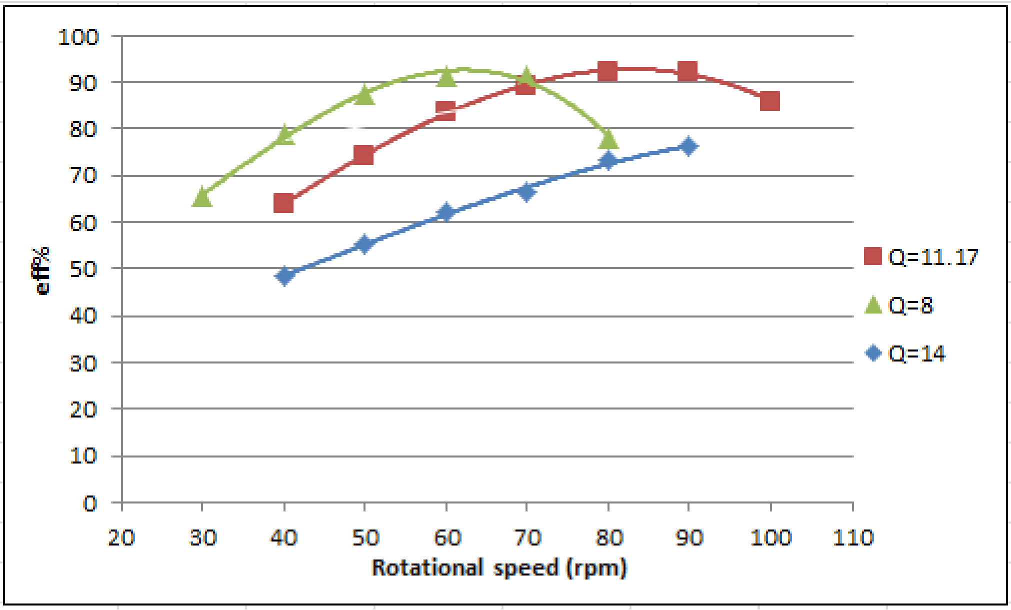


Figure 4-11 efficiency curve based on rotational speed at different flows for the initial turbine

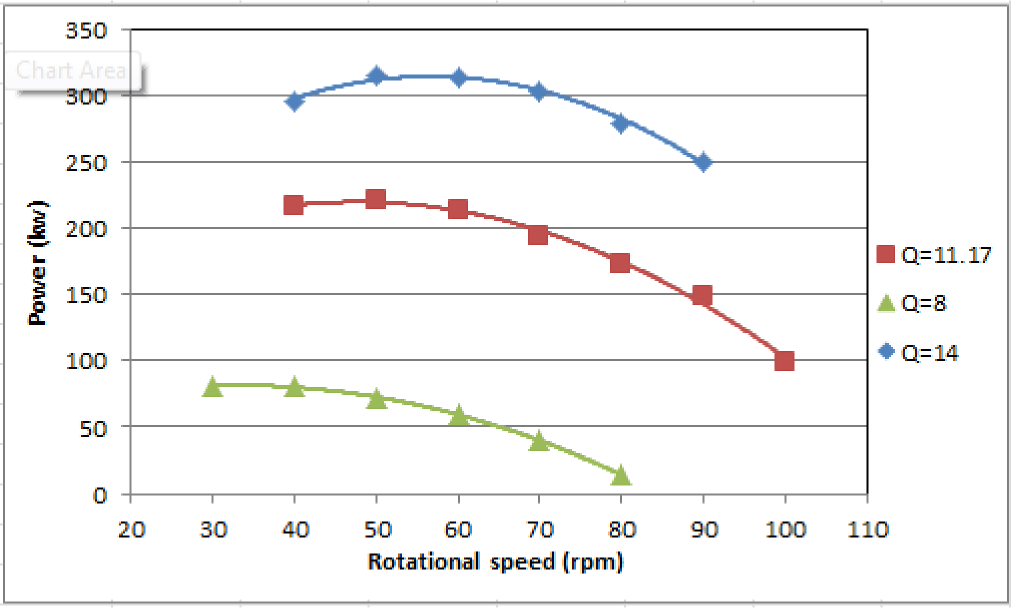


Figure 4-12 power curve based on rotational speed at different flows for the initial turbine.

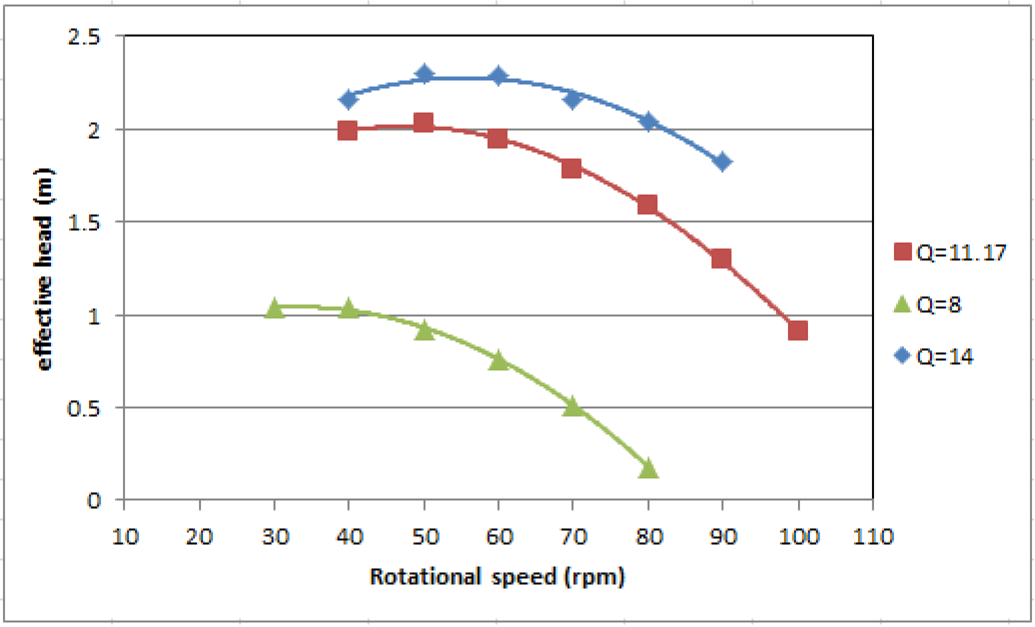


Figure 4-13 efficient head curve based on rotational speed at different flows for the initial turbine.

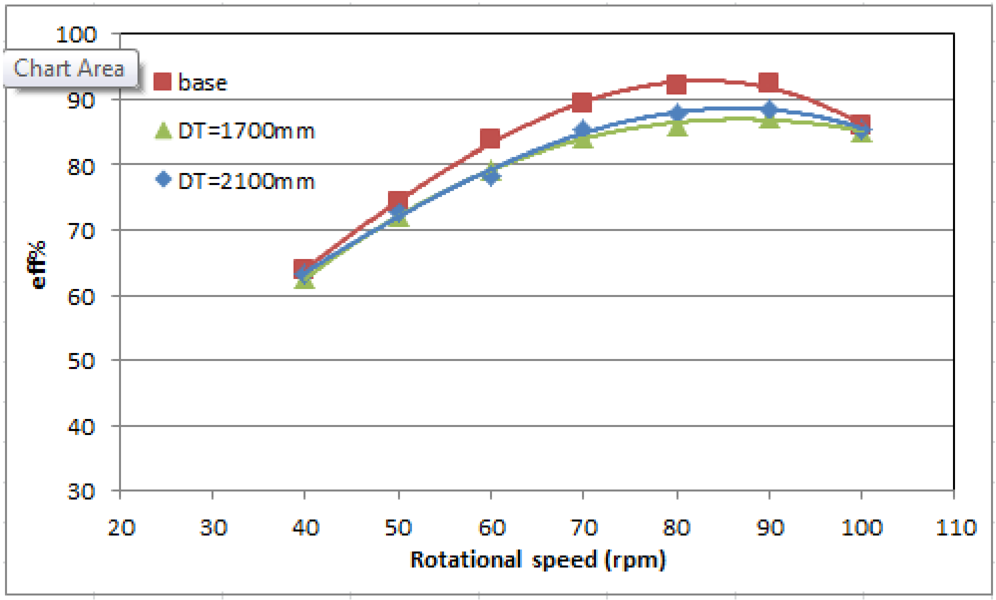


Figure 4-14 efficiency curve based on rotational speed with different tips.

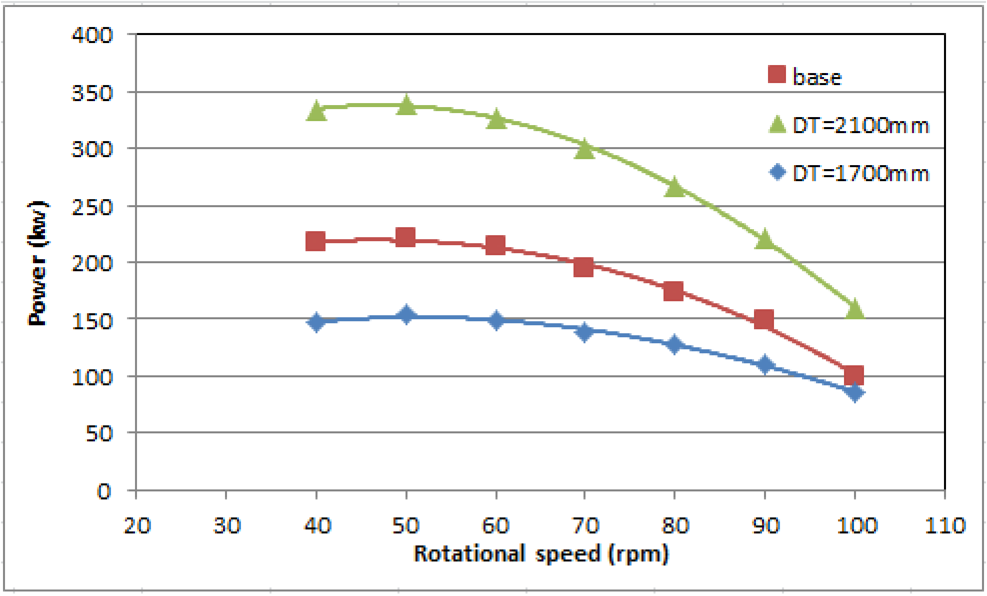


Figure 4-15 power curve based on rotational speed with different tips.

Figure (4-8) show flow efficiency for the initial turbine and the turbine with 4 degree turning angle. With 11.17m^{3}s^{-1} flow and 70rpm rotation speed, initial and secondary turbines respectively have efficiency of 89.4% and 86.9% which shows 2.80% relative decrease in efficiency. As shown in figure (4-11), similar results are obtained in initial turbine for $14m^3s^{-1}$ and $8m^3s^{-1}$ flows which, for example, in initial turbine shows maximum efficiency of 91.4% at $8m^3s^{-1}$ flow with 60rpm rotational speed. As shown in the figure it's completely clear that with increasing flow, we would have optimal efficiency at higher rotational speeds. Figures (4-9) and (4-13) also show efficient head (multiplication of head and efficiency) for the initial turbine and the rotated turbine and also efficient head for the initial turbine at different flows. In the initial turbine with flow of 11.17, 14 and 8 m^3/s we would have maximum efficient head of respectively 2.026, 2.303 and 1.035 meters, and also for the rotated turbine at the design flow it would be 2.275 meters which happens at 50rpm rotational speed, and this shows 12.23% relative increase. Also in initial turbine, with increasing flow, efficient head will increase. It must be mentioned that after flow increase, the rotational speed at which efficient head reaches 0 increases too. Figure (4-10) and (4-14) also show the produced power for the turbines. As the rotated turbine works at higher efficient head compared to the initial turbine, its power will be higher than the initial turbine. For the initial turbine at 11.17, 14 and 8 m^3/s flows, optimal power would be respectively 221.4, 315.4 and 81 KW, and for the rotated turbine at the design flow it would be 248.76 KW which shows a 12.357% relative increase compared to the initial turbine with similar conditions. Maximum power also increases with the increase in flow, as we saw in maximum efficient head. Also it's apparent that maximum power and maximum efficient head happen when flow and rotation speed are fixed, and this rotational speed may not be equal to the rotational speed at which we have maximum efficiency. Figures (4-14) and (4-15) respectively show efficiency and power for the turbines which are compared based on changes in external diameter. In figure (4-14) it's obvious that with changes in tip, maximum efficiency decreases but considering figure (4-15) we can show that with increasing tip, we would acquire higher power.

conclusion

Based on the extracted charts and the result, we can see that:

- With an increase of Stagger angle in the design, efficient head and output power increase;
- With an increase of Stagger angle in the design, efficiency is lowered and maximum efficiency happens at higher rotational speed;
- With a decrease of flow, the maximum efficiency point will move to the left side;
- Always with a specific tip diameter, we would have the best performance and the design must be edited to achieve this number;
- With an increased change of the flow, the efficiency would be lower than the design;
- With increasing flow, the efficient head of the turbine would increase;
- With increasing the tip diameter, power and efficient head increase.

References:

1. Abdul Muis, et al., Priyono S and Fiiman H. Design and Simulation of Very Low Head Axial Hydraulic Turbine with Variation of Swirl Velocity Criterion. The 12 th Asian Internatinal Conference on Fluid Machinery (AICFM12). 2013.
2. A.Muis, et al., Design optimization of axial hydraulic turbine for very low head application, 2nd International Conference on Sustainable Energy Engineering and Application, ICSEEA 2014, Energy Procedia 68(2015) 263-273.
3. Anton, I. (2006). Can be Avoided the Helical Vortex of the Hydraulic Turbines Draft Tube? Part I. Kaplan Turbing, 2^{nd} Workshopon Vortex Dominated Flows Bucharest, Romnia,June 30-1.
4. Andersson, U., (2009) . An Expernental Study of the Flow in a Sharp Hill Kaplan Draft Tubes. Doctoral Thesis, Lulea University of Teach., Dep. Of Applied Physics and Mech.Eng., Division of Fluid Mechanics.
5. Arasu, A.T., (2001). Turbo Machines, Vikas Publishing House PVT LTD., Dep. Mech.Eng., Thiagar Colleg of Eng., Madurai.
6. Anton, L (2006). Can be Avoided the Helical Vortex of the Hydraulic TurbinesDraft Tube? Part I. Kaplan Turbing, 2^{nd} Workshop on Vortex Dominated Flows Bucharest, Romania, Jun 30-1.
7. Balint, D., Resiga, R. and Montean, S. (2004). A Numerical Investigation of the Full 3D Turbulent Flow in kaplan Hydralic Turbines. 6^{th} International Conference on Hydraulic and Hydrodynamics Timisoara, Romania, Oct.
8. Balint, D., Susane-Resiga, R., Montean, S., Anton, I. (2005). A Numerical Study of the 3D Swirling Flow Upstream the Kaplan Turbine Runner at Off-Design Operating

Conditions. Workshop on Vortex Dominated Flows-Achievements and Open Problems Timisoara, Romania, June 10-11.

9. Bansal, R.K., (1998). A Text book of Fluid Machines and Hydraulic Machines. LAXMI Publication (p) LTD, New Delhi.

10. Djelic, V., Kercan, V., Vujunic, V. (2000). Efficiency Scale-up in RefurbishedLow Head Kapian Turbines, the 20^{th} IHAR Symposium, Charlotte, USA, August 6-9.

11. Fay, A. (2004). The Prenciple of Neglecting Upstream Reactions. The 6^{th} International Conference on Hydraulic Machinery and Hydrodynamics. Timisoara, Romania, Oct.

12. FLUENT 6.3 Documentation.

13. Gagnon, J-M., Deschenes, C., Ciocan, G.D., Iliescu, M. (2008). Numerical Simulation and Experimental Investigation of the Flow in an Axial Turbine. IAHR 24 TH Symposium on Hydraulic Machinery and Systems, Oct 27-31.

14. Geberkiden, B.M. (2009), Experimental and Numerical Investigation of Axial Turbine Models. Licentiate Thesis, Lulea University of Teach., Dep. Of Applied Physics and Mech. Eng., Division of Fluid Mechanics.

15. Gyllenram, W. (2008). Analytical and Numerical Studies of Internal Swirting Flows. Thesis for the Doctor of Philosophy in Thermo and Dynamic, Chalmers Uni. Of Tech, Goteborg, Sweden, 2008.

16. Hothersall R. (2004). Hyrodynamic design guide for small Francis andPropeller Turbines. United Nation Industrial Development Org., Veena, Austria.

17. Ida, T., Kurokawa, Tanaka, H., (1996), Recent Development of Studies on Scale Effect Hydraulic Machinery and Cavitation , Kluwer Academic Publishers, Printed in Netherlands.

18. Isbasoiu, E., Constantinescu, M., Otomega, D., Safta, C., Neagoie, c., Ghergu, C. (2004). Indirect Method to Calculate the Operating Flow on a Kaplan turbine.The 6^{th} International Conference on Hydraulic Machinary and Hydrodynamics, Timisoara, Romania, Oct.

19. Jost, D., lipej, A., Sirok , B. (1998). Comparision between Results of the Components and Complete Turbine CFD Simulation, J. Hydropower & Dams.0952264277.

20. Karlsson, M., Nilsson, H. and Aidanpaa, J-O. (2008). Influnce of Inlet boundary conditions in the prediction of rotor dynamic forces and moments for a hydraulic turbine using CFD. The 12^{th} International Symposium on Transport Phenomena and Dynamics of Rotating Machinery Honolulu, Hawaii, Feb. 17-22.

21. Lal, J. (1998). Hydraulic Machines, 6^{th} ed., Metropoliten Book Co., Netaji Subhash Marg, NEW DELHI.

22. Liu, sh., Li, sh, Wu, Y. (2009). Pressure Fluctuation Prediction of a Model Kaplan Turbine by Unsteady Turbulent Flow Simulation, J. Fluid Engineering. Vol. 131/101102-1.

23. Marjavaara, B.D., etal (2003). Automatic Shape Optimization of a Hydropower Draft Tube, 4^{th} ASME _JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, USA, July 6-11.

24. Marjavaara, B.D., Lundstrom, T.S., (2005). Steady and Unsteady CFD Simulation of the Turbine-99 Draft Tube using CFX and STREAM, 3rd LAHRERCOFTAC Workshop on Draft tube flow, Porjus, Sweden, USA, Dec, 8-9.

25. Marjavaara, B.D., (2006). CFD Driven Optimization of Hydraulic Turbine Draft Tubes using Surrogate Model. Doctoral Thesis, Lulea University of Teach., Dep. Of Applied Physics and Mech Eng., Division of Fluid Mechanics.

26. Nilsson, H. and Davidson, L. (2003). Validations of finite volume CFD against detailed velocity and pressure measure ements in water turbine runner flow, Int J.Numer. Meth. FLUIDS, 41: 863-879.

27. Nilsson, H., Page, M. (2005). Open FOAM Simulation of the Flow in the Holleforsen Draft Tube Model, Proceedings of the third IAHR/ERCOFTAC Workshop on draft tube flow, Porjus, Sweden, Dec. 8-9.

28. Nilsson, H. (2006). Evaluation of Open FOAM for CFD of turbulent flow in water turbines, 23rd LAHR Symposium- Yokohama, Oct.

29. P. Sutikno, I. K. Adam, Designg, Simulation and Experimental of the Very Low Head Turbine with Minimum Pressure and Free Vortex criterions, International Journal of Mechanical & Mechatronics Engineering (IJMME-IJENS), vol. 11, no. 01, pp.9-16, 2013.

30. Prasad, V., Khare, R., Chincholikar, Abbas. (2008). Numerical Simulation for Performance of Elbow Draft Tube at different Geometric Configuration. J. Continuum Mechanics, Fluids, Heat, ISSN: 1790-5095, ISBN: 978-960-474-158-8.

31. Roth, M. (1996). A Visualization System for Turbomachinery Flow. Technical Report, Swiss Center for Scientific Computing, April.

32. Sayers, A.T., (1990). Hydraulic and Compressible Flow Turbomachines. McGraw Hill Book Company.

33. Skotak, a., Obrovsky, J. (2007). Low Swirl Flow Separation in a Kaplan Turbine Draft Tube, 2rd LAHR International Meeting of the Workgroup Caviation and Dynamic Problems, Timisoara, Romania, Oct. 24-26.

34. S. L. Dixon, B. Eng., Fluid Mechanics and Thermodynamics of Turbomachinery, 7th ed, Elsevier, 2014.

35. Wilcox, D.C. (1993). Turbulence Modeling for CFD. DCW Industries, Inc. La Canada, Califomia.

36. A Water Resources Technical Publication Engineering Monograph No. 20, (1976). Selecting Hydraulic Reaction Turbines. U.S. Government, Kluwer Academic Publishers, Printed in Netherlands.

37. WWW. Coastalhydropower.com

38. Shirani, Ebrahim, (2008). Turbo Machines. Third Edition. Isfahan Industrial University Publications.

39. Ziei Nejad, Mahdi, (2004). A Brief Introduction of Turbulent Flows and Modeling Them, One of the Booklets for Fluid Dynamics, Third Edition.