Investigation into Kaplan Turbines

RQ: How does the changing flow rate on a Kaplan turbine affect its power output.

Word count: 3380

Contents

Introduction

With global warming causing massive changes in the climate and the ecosystems of the world. To combat this problem many countries are turning into renewable energy sources to reduce the pollution caused by fossil fuels and other harmful energy sources like coal, oil and gasses. From this rapid approach to renewable energy sources the interest in hydro electricity power plants peaked in the current years as with improving technology it is way easier and much more efficient to use hydro power. In this investigation I choose Kaplan turbines for the type of hydraulic turbine as it is the most efficient one. Also, the characteristics of the Kaplan turbine fits my needs as it doesn't need certain moving parts like the guide vanes in turbines like Francis which makes the modelling and simulating part easier.

As an aspiring engineering with a curiosity in hydraulics I choose this topic to learn more about hydro electric power plants and how they function however as this is a highly complex topic, I tried to simple it down with the usage of simulations and other investigations into Kaplan turbines.

This essay's main purpose is to see how the potential energy changes into kinetic energy and then how it turns into electricity inside of a turbine and with increasing volumes of water how does this energy output increases or decreases then finally check if with increasing power output how does efficiency change with it.

After choosing the type of turbine and finding research on it was time to pick a research question as the dams in my country losing their reservoirs with increasing speeds, I chose the research question of How does the power output of a Kaplan turbine is affected from changing flow rates.

The reasoning behind choosing power over efficiency and other important factors such as cavities and cavity chances is that with the usage of simulations it is the easiest factor to find, and it requires less engineering physics knowledge.

Background Information:

Kaplan turbines: A Kaplan turbine is a hydraulic turbine with an axial flow (the flow of fluid is downwards) that is specialized in low head (the height difference between where the water enters the system and where it leaves) and high flow rates. Kaplan turbines are special with their adjustable rotating blades to maximize the efficiency with changing flow rate. Kaplan turbines consist of 4 main parts which are.

1.Scroll Casing: It is a spiral type of casing that has decreasing cross section area. The water from the penstocks enters the scroll casing and then moves to the guide vanes where the water turns through 90° and flows axially through the runner. It protects the runner, runner blades guide vanes and other internal parts of the turbine from an external damage.

2. Guide Vane Mechanism: It is the only controlling part of the whole turbine, which opens and closes depending upon the demand of power requirement. In case of more power output requirements, it opens wider to allow more water to hit the blades of the rotor and when low power output requires it closes itself to cease the flow of water. If guide vanes are absent than the turbine cannot work efficiently and its efficiency decreases.

3. Draft Tube: The pressure at the exit of the runner of Reaction Turbine is generally less than atmospheric pressure. The water at exit cannot be directly discharged to the tail race. A tube or pipe of gradually increasing area is used for discharging water from the exit of turbine to the tail

race. This tube of increasing area is called Draft Tube. One end of the tube is connected to the outlet of runner while the other end is sub-merged below the level of water in the tailrace.

4. Runner Blades: The heart of the component in Kaplan turbine are its runner blades, as it the rotating part which helps in production of electricity. Its shaft is connected to the shaft of the generator. The runner of this turbine has a large boss on which its blades are attached, and the blade of the runner is adjustable to an optimum angle of attack for maximum power output. The blades of the Kaplan turbine have twist along its length. *(8)*

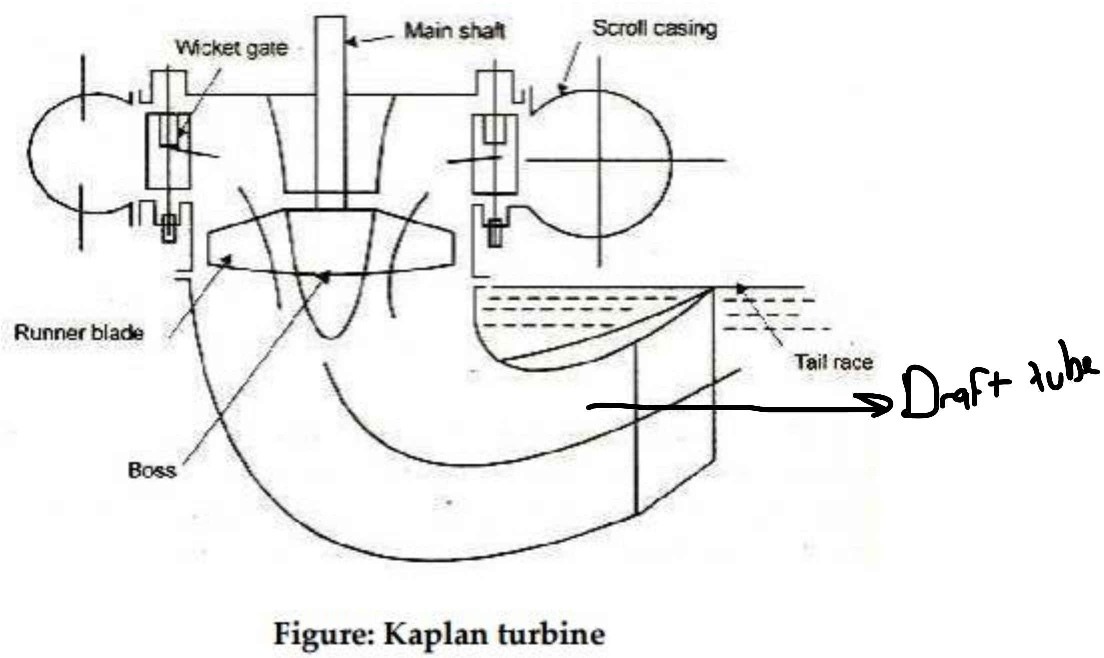


Figure 1. Kaplan Turbine with its components

<u>CFD simulations and CAD:</u> Computational Fluid Dynamics (CFD) is the analysis of fluid flows using numerical solution methods. Using (CFD) one can analyze and investigate complex problems and mechanisms involving fluid to fluid, fluid to gas and fluid to solid interactions. By using simulations CFD programs can calculate differential equations that are very hard and complex to understand with ease and help with simplifying the investigation. Also, with the current technology the calculations done with CFD are accurate enough to be used in engineering reports and project designs. CAD is needed for CFD as CAD stands for Computer aided design and everything needs to be designed in CAD before starting the simulation.

Formulas: While working on Kaplan turbines certain formulas are used in the simulations as These formulas got standardized within the engineering community

Power: A turbine is a rotary machine that converts kinetic energy and potential energy of water into mechanical work. Power of the turbine can be calculated using this equation.

$$P = QH\eta_h\rho g[1]$$

Where:

Q = Discharge

H= Gross head (head)

η_h= Hydraulic efficiency

ρ= Water density

G= Gravitational acceleration

The Kaplan turbines Hydraulic Efficiency formulas uses the Inlet and Outlet velocity triangles however with the usage of simulations there is no need to do the advanced calculations needed.

The simulation uses the formula given bellow.

$$\eta_h = \frac{v_{w_1}u_1}{gH}[2]$$

(7)

Where:

v_{w_1}= Inlet velocity triangle

u_1= Outlet velocity triangle

H= Head of the Kaplan turbine

With the overall efficiency formula being

$$\eta_o = \frac{shaft\ power}{water\ power} [3]$$

waterpower =$\frac{\rho g Q H}{1000}$

(7)

Inlet and outlet velocity triangles can be found using the figure bellow with the formulas listed.

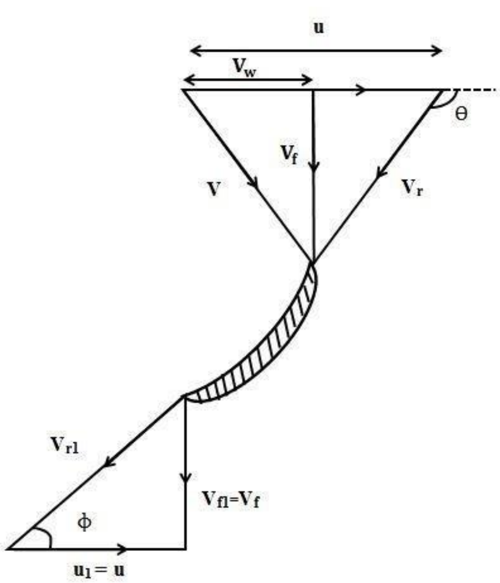


Figure 2. Kaplan turbine inlet and outlet velocity triangles

$$\text{Where: } Huboutlet(\theta) = \tan^{-1}\left(\frac{vf}{U_h - v_{wh}}\right) + 180[4]$$

$$\phi_h = tan^{-1}\left(\frac{v_f}{U_h}\right)[5]$$

$$U_h = \frac{\pi}{60} D_1 N[6]$$

(6)

Finding the discharge (Q) using the formula

Area of flow x velocity of flow= $\frac{\pi}{4}(D_0^2 - D_b^2)v_F[7]$

Where: $D_0 =$ outside diameter of the runner

D_b= Diameter of the hub

v_F = Velocity of flow

And one can find flow rate using the formula $Q = \frac{v}{t}[8]$

Where:

V= velocity

T= time

This formula can be rewritten as Av

A= area of the opening in which fluid is moving through

V= velocity of the fluid

There are many more formulas however by relying on a simulation those calculations are being handled by a computer.

Methodology:

In this investigation a simulation was used as it is nearly impossible to make a real turbine because of its sheer size and production cost. Making a small-scale version is an option but with the turbine parts being fragile and complex this option isn't viable too as the parts cant be printed because they are complex and machines and it out or cant withstand the constant pressure coming from the stream of water and cave in resulting in a system collapse. Even though making a small scale version is an option by conducting a preliminary experiment in which I printed out a little Kaplan turbine which was scaled to the flow rate of an 1 L/s the turbine couldn't work as the pieces weren't made for this small of a scale and in the end the turbine collapsed as it couldn't handle the constant pressure. These limiting factors forced me to use simulations and rely on numerical analysis alone. To do this simulation I found a simulation program that was easy enough to use and a Kaplan turbine design from the simulation's tutorial builds and edited the design for my own personal usage. To edit the design and also check certain parts of it (runner diameter, blade angles, number of wicked gates, number of blades) a CAD program is also essential. In this simulation certain aspects of the turbine are kept as a constant as changing these variables directly impact power output and efficiency.

Variables:

Controlled: the density of the water, the turbine design, number of wicked gates and their angels, number of turbine blades and their angles, gravitational acceleration, the fluid used, the physical model of the fluid(incompressible), number of runs in each simulation, Gross head of the turbine , temperature of the fluid

Independent: The flow rate going into the turbine

Dependent: The power output, efficiency, cavities

Material list:

- CAD program where you can edit and create the turbine.
- Kaplan turbine model that can be simulated without errors.
- Simulation capable of doing CFD simulations.
- Computer

Procedure:

1. Either find a Kaplan turbine design which can be simulated from resources such as grabhub.com or design a turbine design
2. Check the design for its runner diameter, blade angles , number of blades, number of wicked gates
3. Use the simulation program to check the mesh of the design(with mesh errors simulation can give errors and have fluctuating results or no result at all)
4. After checking the mesh follow the instructions in the simulation training booklet to combine each part of the turbine and designate where water will enter and leave

5. By using the simulation training booklet make the turbine part of the design rotatable as it needs to be able to turn to finish the simulation
6. After making sure the simulation boundaries are set choose water as the fluid and choose the incompressible setting
7. Enter the desired flow rates into the simulation to get results.
8. Repeat steps 1-7 2 more times to check if the values in each simulation match each other this is for finding the uncertainty of the simulation.

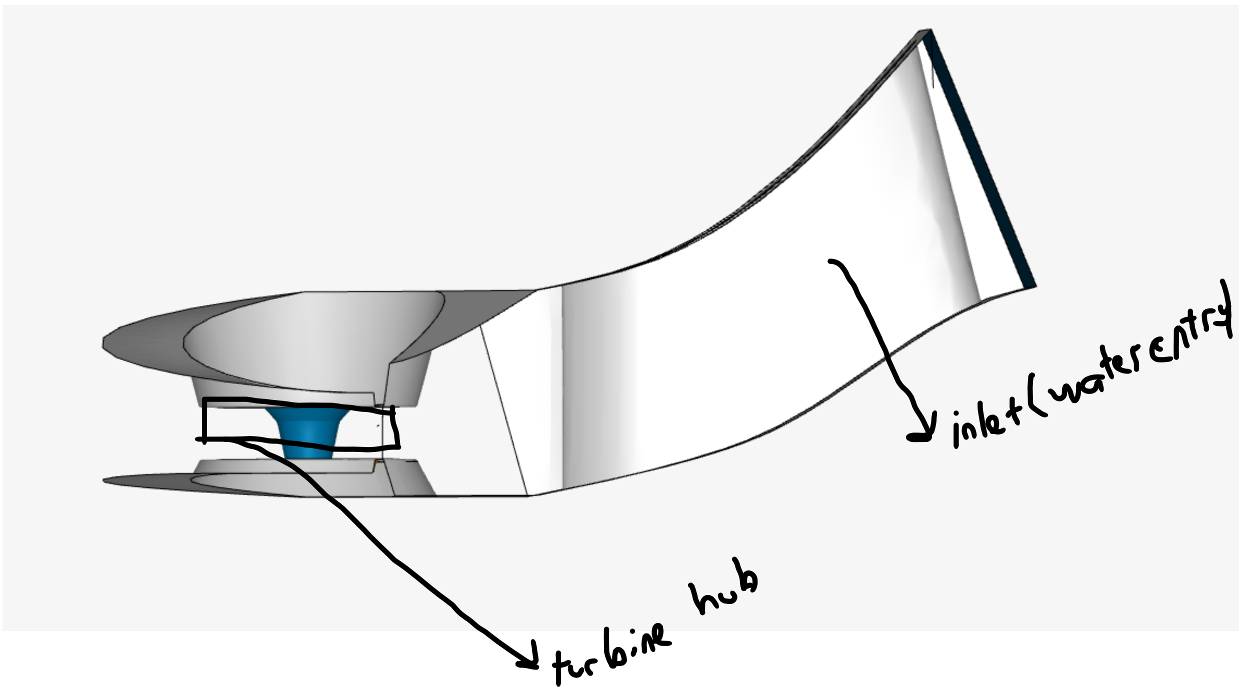


Figure 3. Design of inflow-inlet

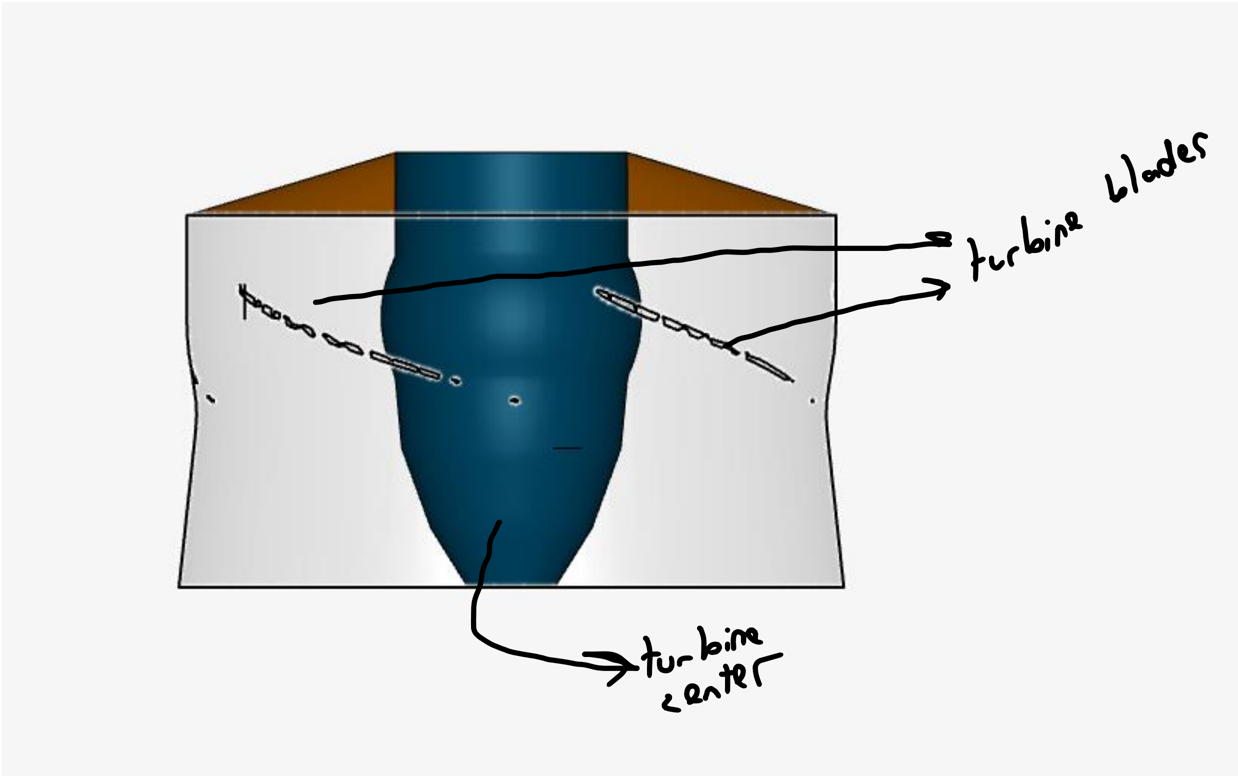


Figure 4. Design of the Wheel

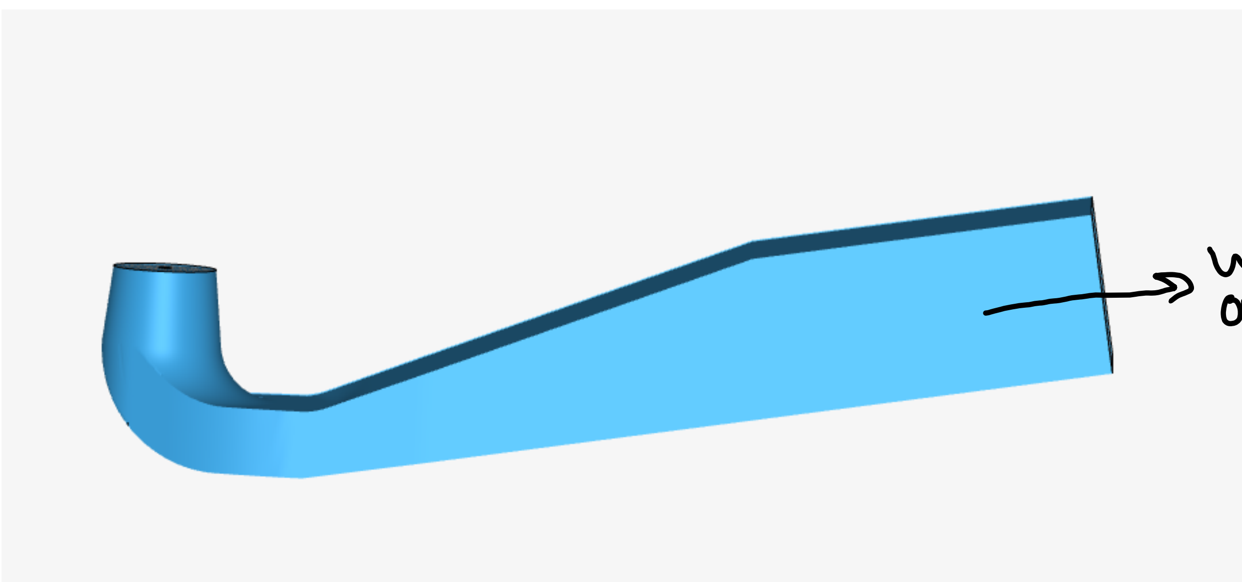


Figure 5. design of outflow-outlet

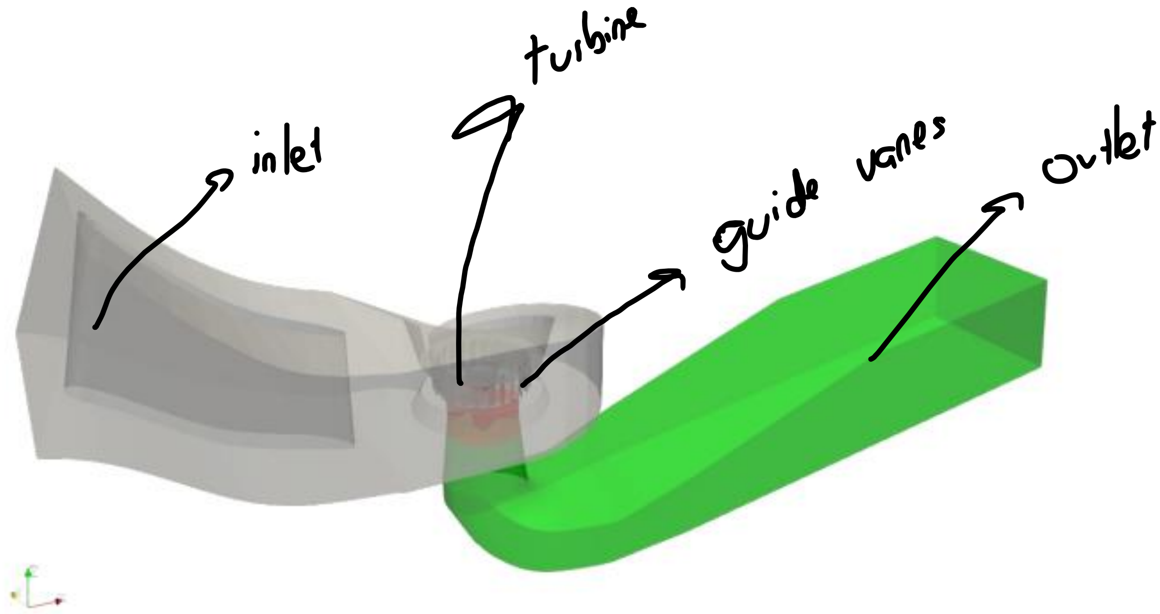


Figure 6. Complete design of the turbine

Hypothesis:

Going from my physics knowledge and analyzing the formulas by increasing the flow rate the power output will also increase as they are directly positively correlated. However, there are no predictions made for efficiency as there is too many factors that affect it.

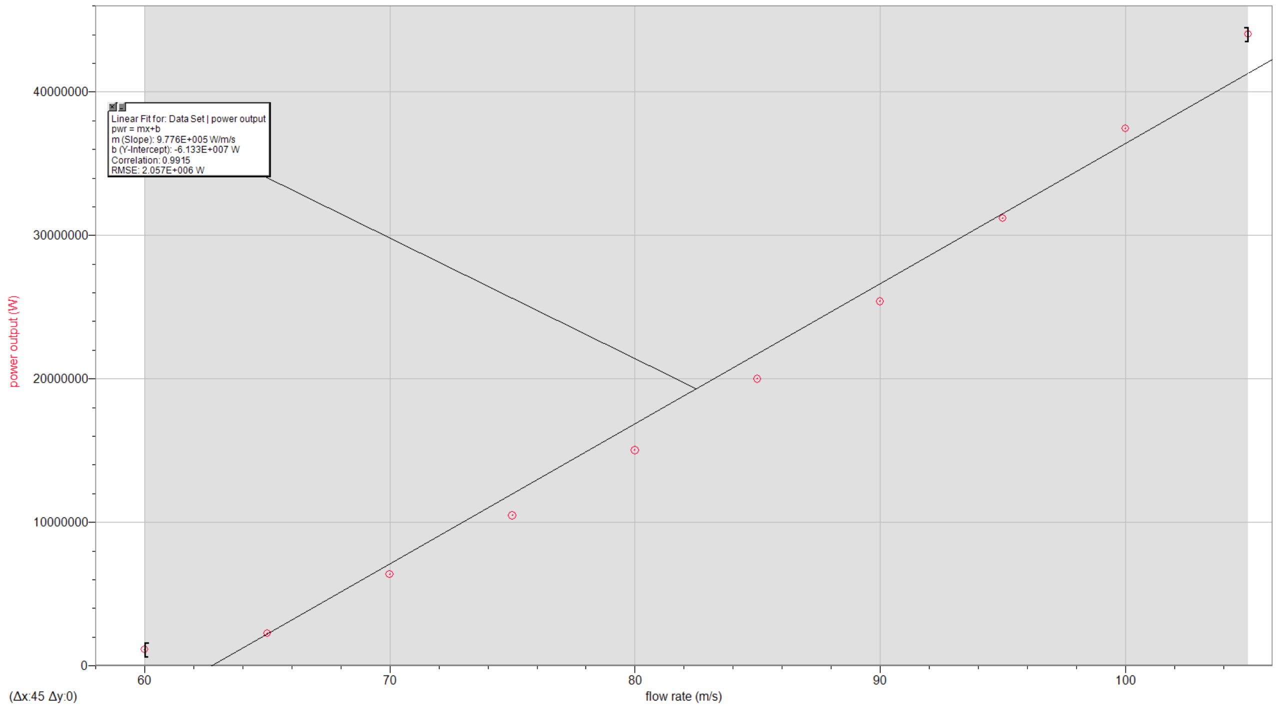
Going from this hypothesis by using the power output and efficiency graph it is possible to find the ideal flow rate for this turbine with constant gross head and other constant variables listed above.

The ideal flow rate will be the highest efficiency one as wasting power isn't something engineers are looking for while designing turbines.

Data collection:

Flow Rate (m^3 / s)	Power output (W)
105	44018000
100	37443100
95	31199000
90	25392600
85	19990000
80	15012000
75	10467400
70	6362870
65	2244520
60	1110020

Table 1. Flow rate vs Power output



Graph 1. Flow rate Against Power output

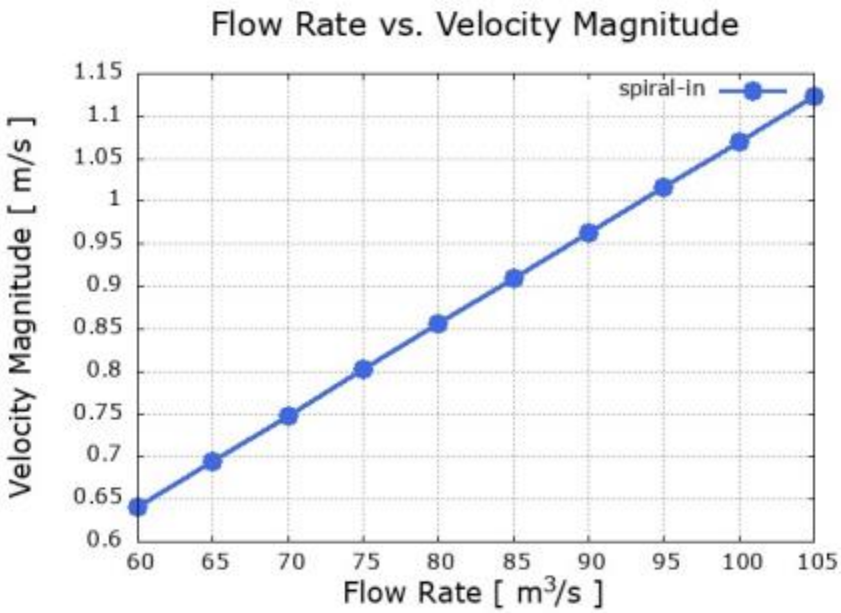
After checking the values that the simulation calculated it can be clearly seen that power output has a strong positive correlation with flow rate . The results of the simulation follows my hypothesis as it was stated before flow rate should have a positive and directly proportional relationship with power out. The linear trend of the data supports this verdict and going from the graph. By checking the equations [1,8] the volume of the flow is directly proportional to power output. The values given by the simulation can be used to reverse engineering to find missing parts of certain equations and this can be used to figure out nearly every aspect of the turbine.

Further analysis of the graph and the best fit line shows a correlation of 0.9915 which is an indicator of the positive connection

.

Flow Rate (m^3 / s)	**Velocity Magnitude (Inlet) (m/s)**
105	1.122730
100	1.069260
95	1.015800
90	0.962337
85	0.908874
80	0.855410
75	0.801947
70	0.748484
65	0.695021
60	0.641558

Table 2. Flow rate vs Velocity Magnitude

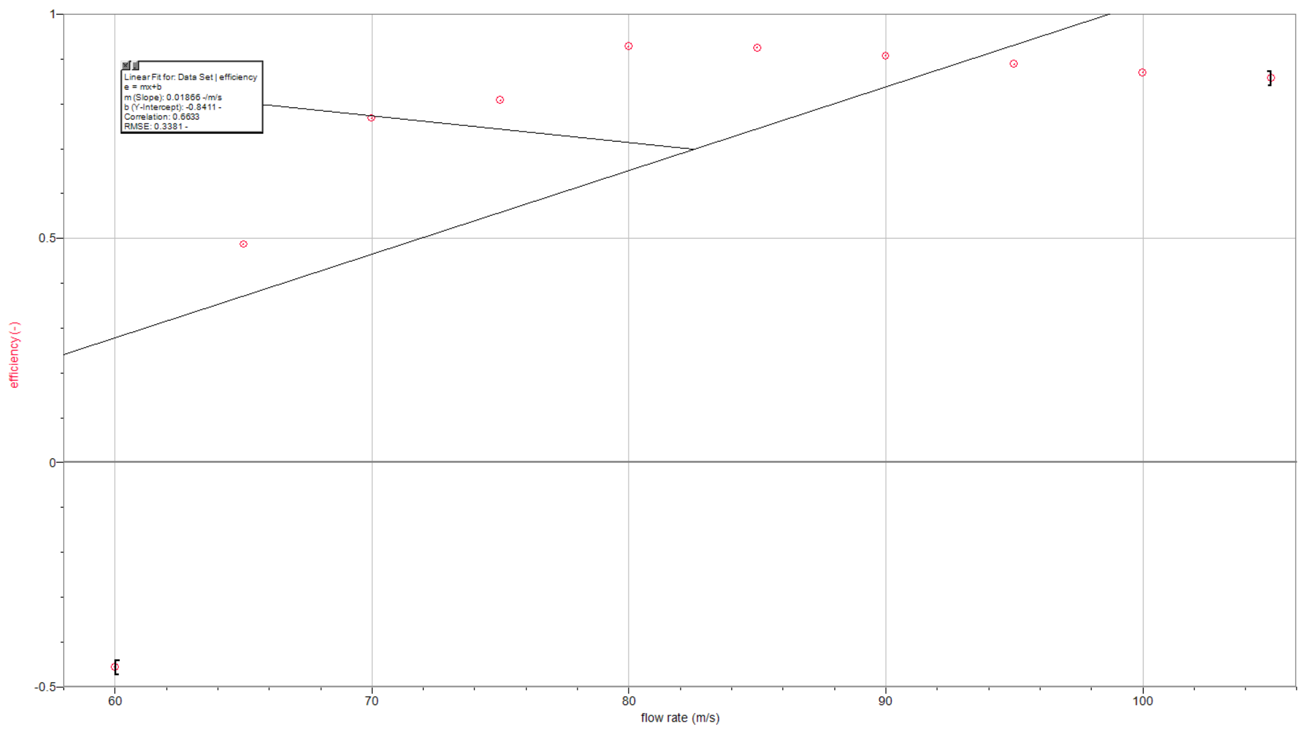


Graph 2. Flow rate Against velocity magnitude (Inlet)

By checking the values, we get from Flow rate against velocity magnitude we can also see a positive correlation between each other as this was expected. By using the equation [8] we can see that there is a positive correlation between velocity magnitude and flowrate as well. This further supports my thesis as increasing velocity magnitude means increasing discharge and increasing discharge meaning increasing.

Flow Rate (m^3 / s)	**Efficiency**
105	0.856753
100	0.870055
95	0.888706
90	0.906776
85	0.925090
80	0.927862
75	0.807938
70	0.768175
65	0.486739
60	-0.455908

Table 4. Flow rate vs Efficiency



Graph 3. Flow rate against Efficiency

the Efficiency curve gives unexpected results as going from other trends like power output, velocity magnitude there is simple formulas and already expected results as there is enough knowledge in how kinetic and potential energy works and with simple knowledge in fluid dynamics these values can be reached . However, in the case of efficiency there is a lot of factors effecting the results with the main one being the velocity triangles and cavities (Cavitations are the process of formation of vapor of a liquid when it has its pressure dropped at a constant temperature. Which in the end damages the turbine lowering the efficiency and creating a need for maintenance or just simply disturbs the flow of the liquid). Because these topics are quite complex no correlation can be found using or comment can be found using the current information available from the simulation

Error Calculation:

One of the strong suits of this investigation is that there is no need for error calculation as the simulation does every calculation on its own leaving no room for human error. However just to be safe I used the simulation with the same inputs 2 more times to see if any value changed and after checking it there was no change in any value which makes error calculations. Also, with the simulation running 2000 iterations for each value and the mesh passing the mesh test in every variable between 105-65(including these) there is no random or systemic errors in these values. For the flow rate 60(m^3 / s) there is a problem with efficiency as it shows -%40 efficiency which is impossible. Because of this error flow rate of 60 (m^3 / s) won't be considered while doing the evaluation of the data.

Theoretical Data

For this investigation only the power output and efficiency formula needed is the power output as the power output is the independent variable. Using the simulation values and founding certain values such as hydraulic head and gross head from reverse engineering a sample calculation can be made for certain conditions. However these calculations aren't perfectly accurate as there are many factors to affect the power output and efficiency in each iteration so the simulation takes the average of each iteration.

The sample calculation for 105(m^3/s) flow rate will use formulas [1]

$$P = QH\eta_h \rho g[1]$$

Where:

Q = Discharge

H= Gross head (head)

η_h= Hydraulic efficiency

ρ= Water density (997)

G= Gravitational acceleration (9.81)

With a head of 50 meters and a discharge of 105 to get

50x105x0.857x997x9.81= 44005230 (W)

There is a small difference between the theoretical power output and the simulations output as the simulation accounts for the chaotic flow of the water and the random movement of the fluid.

After dividing the values 0.1% of a difference is found which shows the simulation is almost perfect in finding the power output.

Conclusion and Evaluation

In this investigation about different flow rates and how they affect the power output of an Kaplan turbine usage of Continuity principle, Potential energy of fluids (fluid dynamics) and kinetic energy leads to conclusion that was expected from the hypothesis as the hypothesis was written using basic knowledge in engineering physics this shows that the correlation between flow rate and power output is directly proportional as the flow rate increases the volume of the water increases . With higher volumes of water there is more potential energy which turns into kinetic energy then mechanical power. This conclusion is supported by the data gathered from the simulation and the theoretical data as well.

Secondly, it can be noticed that efficiency doesn't fit the same trend as power output and other velocity, speed, force graphs as the efficiency curve shown in Graph 4 has a bell curve and it maxes out in the flow rate of 80 $m^3/_s$. No comments were made about efficiency as it required further knowledge in the subject and with the inclusion of cavities to the equations it isn't something that can be explained an investigation of this level. Simply going from the data given by the graph for this specific turbine the ideal flow rate is 80 $m^3/_s$. Even though 105 $m^3/_s$ has greater power output with having lower efficiency it is the worse choice in these conditions.

Lastly with the usage of formulas listed in the background information ,the data gathered from the simulation and by reverse engineering the turbine in a CAD program certain unknown aspects of the turbine can be learnt . Also, by investigating the turbine in a CAD program its blade angles, runner diameter, hub diameter and the distance between blades can be found. These can provide additional information about the turbine and help with the calculations as one doesn't need to find the values of these parts by hand.

Overall, usage of simulation program aided this investigation drastically it doesn't mean there are no problems with using a simulation. During the meshing of the turbine, I encountered an error which messed with the 60 $m^3/_s$ flow rate and caused it to have an efficiency of -%40. Additionally by not having a real life experiment no real value can be reached as it is all numerical analysis even though the calculations done by the simulation is accurate . By having real life examples and values from a real example can better support the investigation. On the contrast by having simulation for my experiment there is no random errors and 1 systematical error in the system. Additionally by using simulations it removes the need to calculate hard differential equations and made the investigation exceptionally easier making it understandable by an Highschool student.

This investigation can be improved by having better knowledge in engineering physics (fluid dynamics) and better knowledge in CAD, CFD. Because I don't have much experience with these kinds of soft wares I relied on premade designs with little adjustments made by me which messed up the mesh of the turbine. By following the instructions on the tutorials of these programs and experimenting on my own I tried to design and simulate a turbine which wasn't that efficient and has no room for error. Someone with experience can use these tools better can simulate the turbine better and design his/her own turbine.

The aim of this investigation was to find the effect of changing flow rates on a Kaplan turbine. Simple comments were made about their relation and the data gathered supported my hypothesis. However, a detailed investigation requiring knowledge on a higher level this investigation can be extended excessively. Because of the limitations caused by my knowledge efficiency, cavities and forces weren't investigated and was briefly mentioned which leaves this investigation incomplete and beyond my level to complete it myself.

Extension:

In this investigation power output was the main variable and it was focused on it however Kaplan turbines are complicated designs with many aspects with the main one being able to change its blade angles according to the flow rate going into it. Someone with enough experience and knowledge in CAD can adjust these blades to see how in constant flow rate the blade angles affect the efficiency as they are the main component changing the efficiency and also try to work out how the blades move according to the changing flow rate and find a function that can be used to adjust the blades according to the flow rate. Also by moving this investigation to the real world and inspecting real life turbines can aid with better understanding the mechanism of the turbine. With having a real turbine inspected monetary costs can be introduced into the investigation. Finally with the usage of certain programs turbines and other designs can be optimized. Using those programs on this turbine to find its maximum efficiency in its optimal blade angle, flow rate.

By introducing these components into the investigation unfinished parts of this paper can be filled in and explain every principle of the turbine better.

Works Cited:

1.Chan, Zin Mar, and Zar Ni Aung. “Design Calculation of Kaplan Turbine Runner Blade for 15kw Micro Hydropower Plant.” *IJARnD*, IJARnD, 1 Jan. 1970, https://www.ijarnd.com/manuscript/design-calculation-of-kaplan-turbine-runner-blade-for-15kw-micro-hydropower-plant/.

2.Design and Analysis of a Kaplan Turbine Runner Wheel. https://pure.manchester.ac.uk/ws/portalfiles/portal/56597486/Design_and_Analysis_of_a_Kaplan_Turbine_Runner_Wheel.pdf.

3.Design of 50 KW Kaplan Turbine for Micro Hydro Power Plant - IRE Journals. https://www.irejournals.com/formatedpaper/1701517.pdf.

4.“Design of Kaplan Turbine | Part-1: Introduction to Blade Profile & Velocity Diagrams | Xdynemo.” *YouTube*, YouTube, 25 May 2021, https://www.youtube.com/watch?v=nUD-ZAOham8.

5.Diva Portal. https://www.diva-portal.org/smash/get/diva2:994607/FULLTEXT01.pdf.

6.IJSTR. https://www.ijstr.org/final-print/jun2021/Modeling-Simulation-And-Fabrication-Of-Micro-Kaplan-Turbine.pdf.

7.“International Journal for Advance Research and Development.” *IJARnD*, https://www.ijarnd.com/.

8.“Kaplan Turbine - Its Components, Working and Application.” *The Constructor*, 22 Sept. 2018, https://theconstructor.org/practical-guide/kaplan-turbine-component-working/2904/.

9.“Kaplan Turbine.” *BrainKart*, https://www.brainkart.com/article/Kaplan-turbine_5578.

10.Kumar, Ravi, et al. “Design and Cost Analysis of Fish Friendly Kaplan Tubine: A Case Study.” *Materials Today: Proceedings*, vol. 46, 2021, pp. 5384–5388., https://doi.org/10.1016/j.matpr.2020.08.805.

11.Numerical Simulation of Turbulence Flow in a Kaplan Turbine -Evaluation ... https://iopscience.iop.org/article/10.1088/1755-1315/22/2/022006.

12.OpenStax. “12.1 Flow Rate and Its Relation to Velocity.” *College Physics OpenStax*, https://pressbooks.bccampus.ca/collegephysics/chapter/flow-rate-and-its-relation-to-velocity/.

13.Satish Taji Follow. “Kaplan Turbine - Design and Numerical.” *Share and Discover Knowledge on SlideShare*, https://www.slideshare.net/SatishTaji/kaplan-turbine-design-and-numerical.

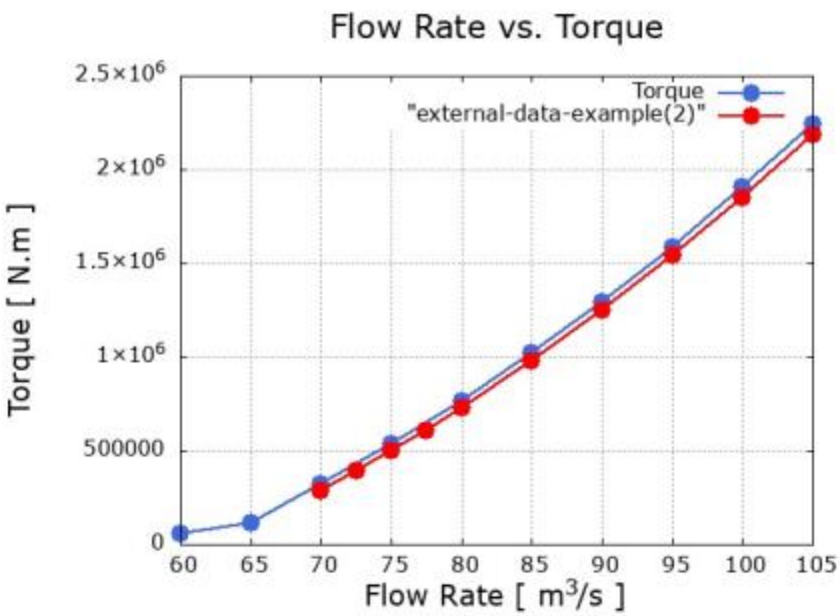
14. https://www.shobhituniversity.ac.in/pdf/econtent/Hydraulic-Machines-Mr-Raman-Gahlaut.pdf

15.“UGC Approved Journal.” *IJARSE*, http://ijarse.com/.

16“What Is Head? - Renewables First.” *Renewables First - The Hydro and Wind Company*, 24 Mar. 2016, https://www.renewablesfirst.co.uk/hydropower/hydropower-learning-centre/what-is-head/#:~:text=it%20leaves%20it-,What%20is%20head%3F,leaves%20it%2C%20measured%20in%20metres.

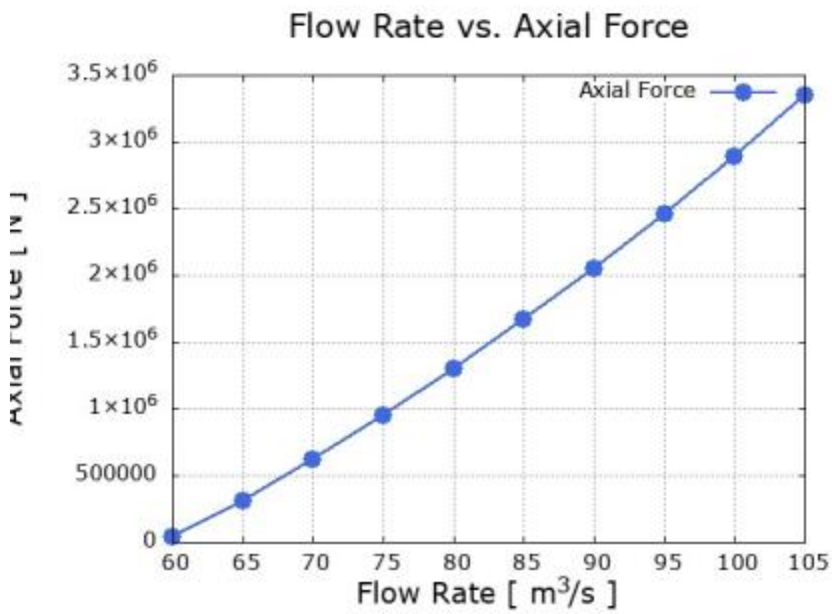
Appendix

Point Nr.	Torque [N.m]
1	2.24181×10^6
2	1.90696×10^6
3	1.58895×10^6
4	1.29323×10^6
5	1.01808×10^6
6	764552.
7	533097.
8	324058.
9	114312.
10	56532.7


Flow Rate vs. Torque
Torque
"external-data-example(2)"
Torque [N.m]
Flow Rate [m3/s]
2.5×10^6
2×10^6
1.5×10^6
1×10^6
500000
0
60
65
70
75
80
85
90
95
100
105

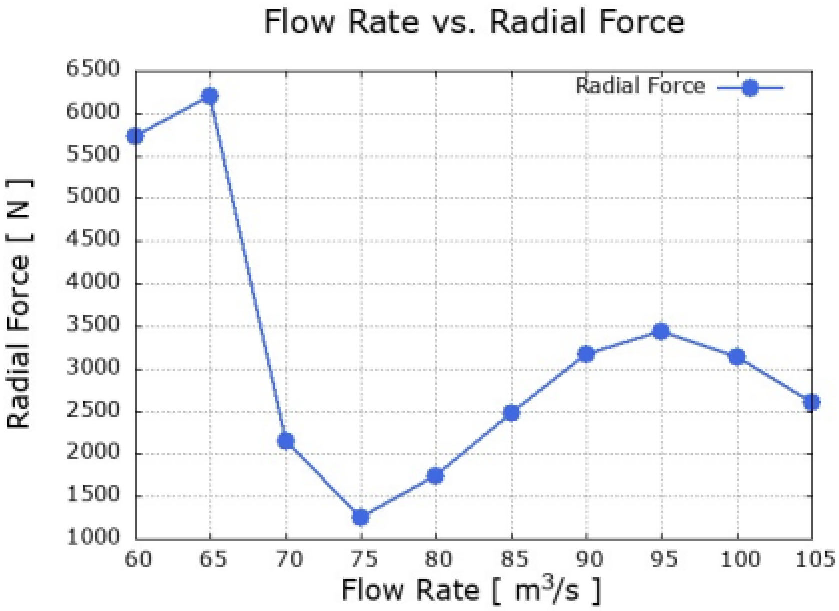
Axial Force

Point Nr.	Axial Force [N]
1	3.34751×10^6
2	2.88882×10^6
3	2.45917×10^6
4	2.05337×10^6
5	1.66670×10^6
6	1.29685×10^6
7	945738.
8	618541.
9	313674.
10	39411.5


Flow Rate vs. Axial Force
Axial Force
3.5×10^6
3×10^6
2.5×10^6
2×10^6
1.5×10^6
1×10^6
500000
0
60
65
70
75
80
85
90
95
100
105
Flow Rate [m^3/s]

Radial Force

Point Nr.	Radial Force [N]
1	2601.15
2	3140.42
3	3434.49
4	3175.55
5	2479.71
6	1735.60
7	1249.90
8	2139.35
9	6198.15
10	5724.27


Flow Rate vs. Radial Force
Radial Force
Radial Force [N]
6500
6000
5500
5000
4500
4000
3500
3000
2500
2000
1500
1000
60
65
70
75
80
85
90
95
100
105
Flow Rate [m3/s]

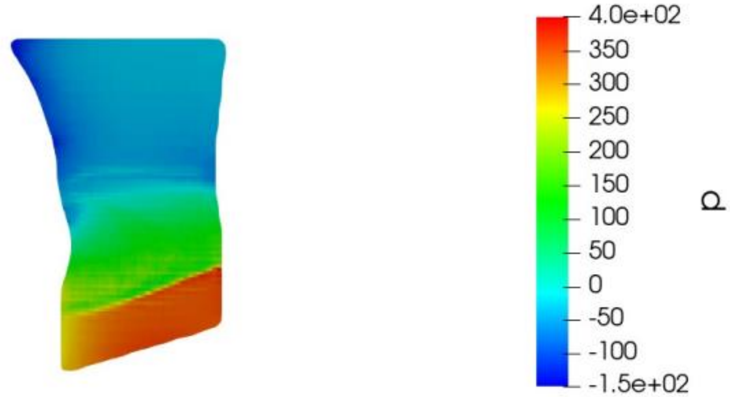
Cavitation Risk

Point Nr.	Flow Rate [m^3/s]	Cavitating Volume [m^3]	Cavitating Volume [%]
1	105.000	1.85658	0.0703788
2	100.000	0.738071	0.0279786
3	95.0000	0.157802	0.00598192
4	90.0000	0.0435993	0.00165275
5	85.0000	0.00757655	2.87209×10^{-4}
6	80.0000	8.77952×10^{-4}	3.32811×10^{-5}
7	75.0000	8.66285×10^{-4}	3.28388×10^{-5}
8	70.0000	0.00279713	1.06033×10^{-4}
9	65.0000	0.0122798	4.65498×10^{-4}
10	60.0000	0.0346627	0.00131398

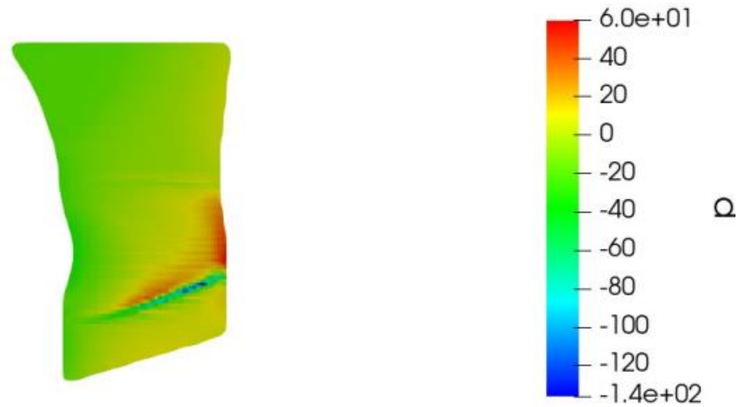
Cavitation Index

Point Nr.	Flow Rate [m^3/s]	Cavitating Volume [–]
1	105.000	476.950
2	100.000	468.261
3	95.0000	457.659
4	90.0000	448.920
5	85.0000	434.799
6	80.0000	416.838
7	75.0000	409.744
8	70.0000	391.616
9	65.0000	368.018
10	60.0000	382.795

Point 1



Point 10



Simulation Stats

Test Case Name	Kaplan-hydro-turbine
Number of Points [–]	10
Machine Type	hydroTurbine
Rotation Speed [rad/s]	-19.6350
Number of Components [–] (topology)	3
Mesh Size [cells] (details)	spiral : 282015 wheel : 286966 draftube : 84559 Total : 653540

Mesh details [mm]:

Component	Property	Min	Max	Avg
1	cellOpenness	N/A	3.83936×10^{-16}	N/A
1	cellVolume	1063.19	8.51669×10^{7}	N/A
1	faceArea	11.3555	205010.	N/A
1	aspectRatio	N/A	9.05810	N/A
1	nonOrthogonality	N/A	59.4074	13.6429
1	skewness	N/A	7.14633	N/A

1	edgeLength	0	454.180	N/A
1	concaveAngle	N/A	69.9834	N/A
1	faceFlatness	0.515073	N/A	0.999104
1	cellDeterminant	0.0489233	N/A	20.6423
1	faceInterpolationWeight	0.0771941	N/A	0.441974
1	faceVolumeRatio	0.0149803	0.704471	N/A
1	*Warnings:*	4 skew faces 171 faces with low quality or negative volume decomposition tets 16 points on short edges 3314 faces with concave angles 7 warped faces 25274 concave cells		
2	cellOpenness	N/A	3.44695×10^{-16}	N/A
2	cellVolume	158.768	1.20339×10^{6}	N/A
2	faceArea	3.46884	11613.2	N/A
2	aspectRatio	N/A	7.62437	N/A
2	nonOrthogonality	N/A	64.5863	15.8781
2	skewness	N/A	5.52237	N/A
2	edgeLength	0	107.767	N/A
2	concaveAngle	N/A	79.7310	N/A
2	faceFlatness	0.373796	N/A	0.996270

2	cellDeterminant	0.0120155	N/A	24.9681
2	faceInterpolationWeight	0.0755412	N/A	0.428771
2	faceVolumeRatio	0.0112594	0.632688	N/A
2	*Warnings:*	3 skew faces 354 faces with low quality or negative volume decomposition tets 10 points on short edges 7121 faces with concave angles 66 warped faces 31047 concave cells		
3	cellOpenness	N/A	3.49790×10^{-16}	N/A
3	cellVolume	892.775	8.04288×10^{7}	N/A
3	faceArea	56.6646	252451.	N/A
3	aspectRatio	N/A	5.96590	N/A
3	nonOrthogonality	N/A	53.6855	9.18775
3	skewness	N/A	1.91554	N/A
3	edgeLength	N/A	N/A	N/A
3	concaveAngle	N/A	43.1235	N/A
3	faceFlatness	0.885568	N/A	0.999738
3	cellDeterminant	0.0793814	N/A	11.9414
3	faceInterpolationWeight	0.117241	N/A	0.470432
3	faceVolumeRatio	0.0319623	0.850656	N/A
3	*Warnings:*	4 faces with low quality or negative volume decomposition tets		

Simulation score [–] details

Point	flow rate	bounding	residuals
1	✓	✓	✓
2	✓	✓	✓
3	✓	✓	✓
4	✓	✓	✓
5	✓	✓	✓
6	✓	✓	✓
7	✓	✓	✓
8	✓	✓	✓
9	✓	✓	!
10	✓	✓	!

Wall-clock time [hh:mm:ss]	00:02:07 (meshing) 02:56:50 (calculation)
Parallel Processors [–]	6
Fluid Name	water
Physical Model	incompressible
Numerical order	second
Turbulence model	kOmegaSST