Universitat de Girona
Escola Politècnica Superior

Incoming Exchange Student - Master Thesis

Erasmus □ Techno □ Other (specify):

Title of master course: Master in Industrial Engineering

Title of master thesis: Parametric study of the performance of an impulse-type turbine with CFD

Document: Master Thesis

Student (Name & Surname)**:** Sven Cornelis

EPS Advisor: Toni Pujol
Department: Eng. Mecànica i de la Construcció Industrial

Delivered on (month/year): June 2016

Foreword

This Master thesis is a part of my Erasmus study at the University of Girona. The essence of this thesis is to complete my study ‘Master of Electro-mechanics at the University of Antwerp. A special word for dr. Toni Pujol, who guides me through the project and helps me understand everything very well.

Nomenclature

ω	Angular velocity of the turbine
h	Height of the center of the nozzle
d	Diameter of the nozzle
n	Rotation speed
P	Power of the turbine
E_t	Total energy of the water jet
E_k	Kinetic energy of the water jet
E_p	Potential energy of the water jet
m	Mass of the water jet
v	Velocity
g	Gravity
t	Time
$\dot{m}$	Mass flow of the water jet
V	Volume of the water jet
ρ	Material density of the water jet
Q	Volume flow of the water jet
A	Area of the inlet of the nozzle
P_k	Kinetic power of the water jet
P_p	Potential power of the water jet
T	Torque
F	Force
r	Radius
F_n	Force in normal direction
F_t	Force in tangential direction
α	Angle of the impact
η_{th}	Theoretical efficiency
η_{sim}	Efficiency after simulation
P_{avg}	Average theoretical power

Table of Contents

1 Introduction

A water wheel is an old machine, used ages ago to provide mechanical energy from hydraulic energy [1]. First, ancient societies used this energy directly to grind in order to get flour and other products. Later, when generators existed, they were employed to provide electrical energy. Today, there are still some isolated communities in rural areas whose main source of energy comes from water wheels [2].

There have been many studies about water wheels and the designs that provide a maximum available installed power [3]. This thesis will do a research on an already existing water wheel located at the Hydraulics Laboratory of the Department of Mechanical Engineering and Industrial Construction of the University of Girona. The water wheel-type is a horizontal axle with a very simple design of the blades. Although these kind of water wheels are known of not being the best designs, they can be very useful in developing countries because of their simplicity. The purpose of the research is to determine the efficiency of this water wheel with simulation techniques and to obtain the geometrical conditions that may provide high enough efficiency values for being a feasible design as an element for producing electricity in isolated communities of developing countries with high resources of water.

For doing so, we will simulate the water wheel employing the ANSYS 15.0 software package and choosing the commercial code CFX, which is a general purpose Computational Fluid Dynamics CFD model. After setting up the model, we will do a parametric study to optimize the efficiency of the water wheel.

Firstly, a geometrical model of the water wheel will be designed to be as similar as possible to the actual water wheel, being a copy of the current water wheel of the laboratory. This includes every important piece of the water wheel and also the holes in the plates designed to lower the weight. We have to keep in mind, however, that a good geometrical design has to be accurate enough to reproduce the geometry of the real water wheel, but simple enough to avoid excessive unimportant details so we may have the results using reasonable computational resources.

Secondly, a study on the suitable mesh is required. Because of the limit of 512.000 elements in the educational version of ANSYS 15.0, we have to decide which part is more important in the contribution to the hydraulic power in order to apply a finer mesh there. The blades of the water wheel will receive the water jet impact and will provide the reaction forces that make the water wheel rotate. It is reasonable that a smaller mesh size at the blade will allow to obtain better results. Also the accuracy of reproducing the path of the water jet is important because the reaction forces come from the impact of the water.

Once the discretization of the volume (i.e., the mesh) has been defined, the setup has been created. A parametric study that takes into account the rotation speed of the waterwheel will be carried out in order to determine at which point the power reach the maximum value. The formula of power is equal to the torque multiplied by the angular speed. If we fix the water wheel to a speed of zero (at rest), there will be a maximum of torque but a minimum of power. On the other hand, the water wheel turning at a constant speed equal to the speed of the incoming waterjet will not provide torque although it will give to the water wheel a maximum value of the rotational speed but, again, a minimum of power. These two points will be the starting and ending points of the power curve. After determining the maximum power as a function of the rotational speed, a second parameter will be modified. This parameter will be the position of the inlet with respect to the water wheel location, where a perpendicular impact to the water wheel is required.

Afterwards, we can compare the results with the experimental data to see how accurate the simulations are and how much the efficiency is improved.

2 Methodology

The first approach of the analysis will be a theoretical calculation of the model, which will be done in chapter 4. The meaning of this analysis is to calculate the theoretical efficiency of the turbine so afterwards the efficiency of the simulations can be compared with the theoretical efficiency.

The second approach are simulations of the turbine with different setups where three parameters will be variable. Every single simulation will have a fixed rotation speed. The simulations will calculate the torques of the different setups. The power of the turbine can be calculated with these two values. The three parameters that we will change in this study are the rotation speed ω, the height of the inlet h and the diameter of the nozzle d, as shown in the schematics of Figure 1.

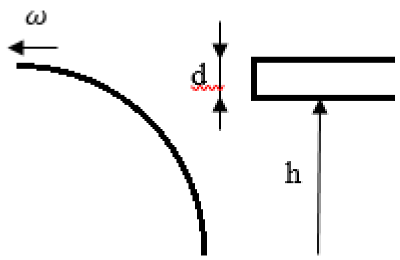


Figure 1. Schematics of the three parameters analyzed in this study. The nozzle has an inner diameter d at a height h and the water wheel turns at ω.

Initially, several simulations with fixed rotation speeds will be done on a setup with the standard height and constant value for the diameter. After a first observation of the results, a few more simulations around the point of maximum power will show the best solution. Then the second parameter, the height of the inlet, will be changed. Values bigger and smaller than the standard height are going to show the change of the power as a function of height. The first type of simulations with another height will be done with the best rotating speed coming from previous observation. This will give us the point with maximum power as a function of the height and at a rotation speed where the power is maximum for a standard height (see *Table 1*). To check if this point is indeed the maximum point of power, the same procedure will be done with rotation speeds near the rotation speed that was obtained at the beginning of the case of the standard height (see *Table 2*). A matrix will be filled where one axis corresponds to the rotation speed and the other axis to the height.

d=20,7	n [rpm]				
	50	100	200	300	400
P [W]	×	×	×	×	×

Table 1. List of simulations with inner nozzle diameter d = 20,7 mm at height h = mm

d=20,7	n [rpm]							
h [mm]	50	100	200	250	260	270	280	300
631	×	×	×	×	×	×	×	×
633	×	×	×	×	×	×	×	×
634	×	×	×	×	×	×	×	×
635	×	×	×	×	×	×	×	×
640	×	×	×	×	×	×	×	×
646	×	×	×	×	×	×	×	×
653	×	×	×	×	×	×	×	×

Table 2. List of simulations carried out with with inner nozzle diameter d = 20,7 mm and variable height.

This matrix will give us a three dimensional graph of the power as a function of the rotation speed and the height of the inlet. The maximum point of power for a constant value of the diameter is found.
The same procedure can be followed for the analyses with a different diameter. Before doing that, the results from the previous analysis will be compared with the theoretical calculations. This comparison will give us a better view on the setup and proper starting point for the matrix to be used. It is ideal that the maximum point of power lies within the dimension of the matrices and that the dimensions are as small as possible so we can do a quicker refinement.

d=20	n [rpm]		
h [mm]	270	300	330
631	×	×	×
634	×	×	×
646	×	×	×

Table 3. List of simulations with inner nozzle diameter d = 20 mm and variable height h.

d=18	n [rpm]		
h [mm]	300	350	400
631	×	×	×
634	×	×	×
646	×	×	×

Table 4 . List of simulations with inner nozzle diameter d = 18 mm and variable height h.

Every single simulation run is a transient state, so the timesteps value must be defined. In this thesis one degree per timestep is used for every simulation. This leads to smooth enough simulations with steps not too small. For every step, the solver will use ten internal iterations to be sure that the results are correct. During a simulation, the values of the torques of the turbine are monitored, as shown in *Figure 2*. The curves are the torques for one simulation with constant values for the rotation speed, the height of the inlet and the diameter of the nozzle. The curves can be exported to Microsoft Excel. With Excel the average value of the torque around the normal axis of the turbine can be calculated and also the power the turbine provides with this setup.

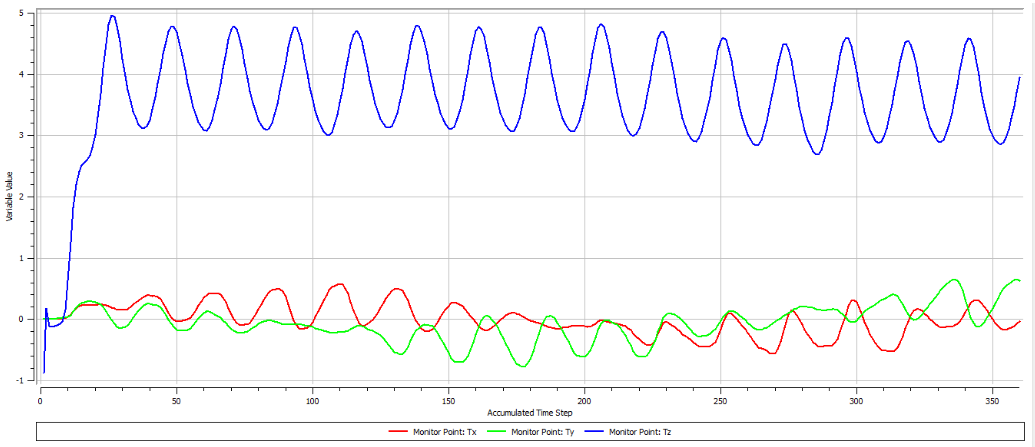


Figure 2. Example of the torque (x, y, z) calculated from the CFX model. Blue line: Torque z (main torque value). The periodicity of the signal corresponds to the time interval between two consecutive blades.

3 Laboratory water wheel

Before starting with the simulations, the whole geometry of the existing water wheel has to be measured. The water jet velocity will be calculated by taking three repetitions of an experiment that measures the time for having a volume flow of 100 liters.

3.1 Dimensions

Box			**Wheel plates**		
Length	553	mm	Width	50	mm
Height	650	mm	Length	50	mm
Width	175	mm	Height	1,5	mm
			Number of plates	16	
			Angle	0	°
			Position of plate to wheel	15	mm
Plate			**Inlet tube**		
Diameter	418	mm	Inner diameter	20	mm
Space between two plates	50	mm	Length	73	mm
Height of center	280	mm	Position of center	496	mm
Width of center	87,5	mm	outer diameter	25,2	mm
Length of center	275	mm			

Table 5. Main dimensions of the water wheel

The water flow of 100 liters was measured within a time of 20.61 s, 20.69 s and 20.66 s. This gives us an average time of 20.65 s and a volume flow of 4.84 l/s.

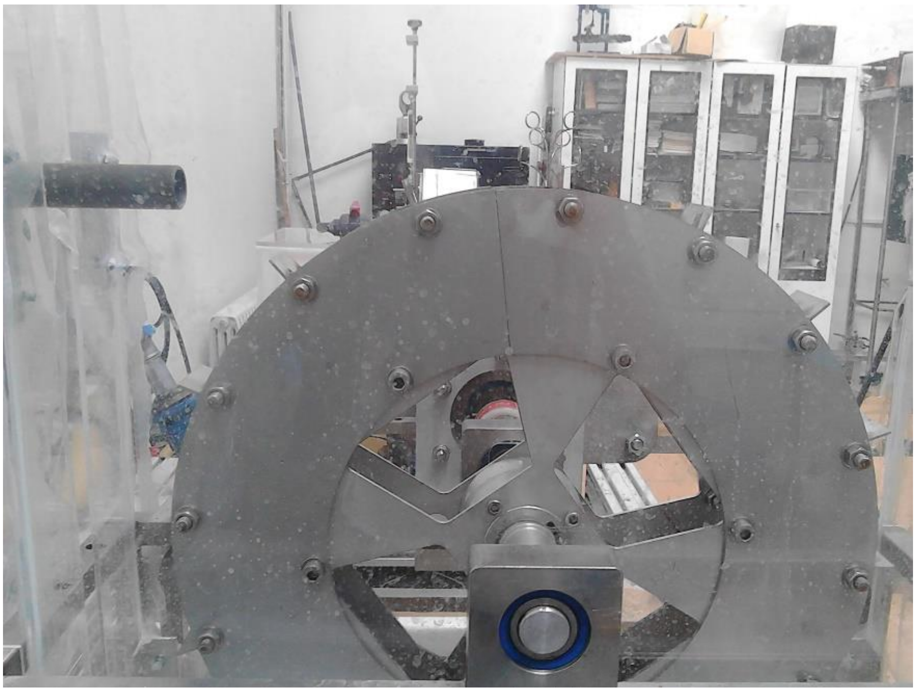


Figure 3. Image of the laboratory water wheel.

4 Theoretical approach.

Before modelling and simulating the problem, it is handful to carry out a small theoretical study to determine what the maximum energy of the system is, so we can compare the theoretical results with the simulation ones.

4.1 Power of the water flow

We can use the formulas of kinetic and potential energy to determine the maximum energy of the whole system. The existing water wheel uses a constant value of the incoming water flow, as stated above, which is measured with a volumetric flow counter and a chronometer giving a value of 100 liters each 20.65 seconds (mean average value of repeating the measure three times). The inner diameter of the inlet is also a constant value. As a difference in height, in order to calculate the available potential energy, we will use the radius of the water wheel and assume that the maximum potential energy used by the water wheel starts at the top of the wheel and ends 90° further where the water will leave the wheel due to the effect of gravity.

To calculate the efficiency of the system, the total energy Et has to be known.

$$E_t = E_k + E_p$$

with E_k equal to the kinetic energy and E_p to the potential energy,

$$E_k = \frac{mv^2}{2}$$

$$E_p = mgh$$

In our case, we work with a mass flow instead of a mass. Therefore, we divided the previous expressions by the time, so the formula is equal to the power.

$$P = \frac{E}{t}$$
$$P_k = \frac{mv^2}{2}$$
$$P_p = mgh$$

As explained above, the mass flow can be experimentally measured and, it is in this case a constant value,

$$m = \frac{V\rho}{t}$$

$$m = \frac{100l \cdot 1\,dm/l \cdot 998\,kg/m^3}{20.65\ s} = 4.84\frac{kg}{s}$$

The speed of the water jet is equal to the volume of flow divided by the cross-sectional area of the inlet tube,

$$v = \frac{Q}{A} = \frac{Q}{\frac{\pi d^2}{4}} = \frac{4.84\,dm^3/s}{\frac{\pi \cdot \ 20.7\ mm\ ^2}{4}} = 14.39\frac{m}{s}$$

With these constant values, the total power of the incoming water flow can be calculated as

$$P_k = \frac{mv^2}{2} = \frac{4.84\,kg/s \cdot \ 14.39\,m/s\ ^2}{2} = 499.08\frac{J}{s} = 499.08\ W$$

$$P_p = mgh = 4.84\,kg/s \cdot 9.81\,m/s^2 \cdot 0.209\ m = 9.92\frac{J}{s} = 9.92\ W$$

$$P = P_k + P_p = 499.08\ W + 9.92\ W = 509\ W$$

4.2 Maximum power of the turbine

The total power of the turbine P depends on the torque $\boldsymbol{T}$, which is delivered by the impact of the water, and the angular speed of the turbine $\boldsymbol{\omega}$

$$P = \int dP = \int \boldsymbol{dT} \cdot \boldsymbol{\omega}$$

so, as pointed out in the introduction section, the power of the wheel is equal to the torque multiplied by the angular velocity of the wheel. Note that in the previous equations, $\boldsymbol{dT}$ is the torque in a differential area of the turbine dA and the integration covers all the surface area of the turbine. In this theoretical analysis, we assume, by simplicity, a mean value of the torque,

$$P = \boldsymbol{T} \cdot \boldsymbol{\omega}$$

which is equal to a mean force multiplied by the radius of the wheel, with the angular velocity being the peripheral velocity of the wheel divided by its radius. If we substitute these equations into the simplified power equation, we obtain that the power is equal to the force multiplied by the peripheral velocity.

$$\boldsymbol{T} = \boldsymbol{F} \cdot \boldsymbol{r}$$

$$\omega = v/r$$

$$P = \boldsymbol{F} \cdot \boldsymbol{v}$$

The force of the impact of the water jet over a flat blade depends on the mass flow of the water, the velocity of the impact and the angle of impact. The general equation of the force is in case of a static perpendicular surface,

$$F = \rho A v^2$$

In case of a moving surface, there will be a difference in the velocity of impact since this will be equal to the relative to the blade velocity $\Delta\ v$. Then, the equation can also be written as the force being equal to the mass flow multiplied by the velocity of impact or a difference in impact for a moving surface.

$$m = \rho A v$$

$$\boldsymbol{F} = m\boldsymbol{\Delta v}$$

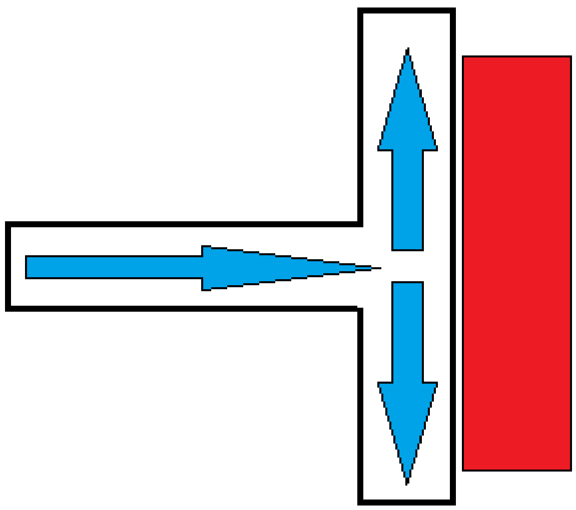


Figure 4. Schematics of a control volume corresponding to a perpendicular impact on the blade.

Using the impulse theorem, we can define a control volume as that shown in *Figure 4* and express the forces with respect to a coordinate system attached to the turbine blade perpendicular to the incoming waterjet (normal and tangential direction axis).[4] Thus, for a perpendicular impact, the equations are:

$$m_{water}\Delta v - F_n = 0$$

$$F_n = m_{water}\Delta v$$

$$\sum F_t = 0\text{: } m_1 \cdot v_1 - m_2 \cdot v_2 = 0$$

In case of an impact on an inclined surface, the equations change since the force tangential to the blade surface is not zero anymore. (see *Figure 5*). The contributions to the normal and tangential components to the blade surface are listed in *Table 6*.

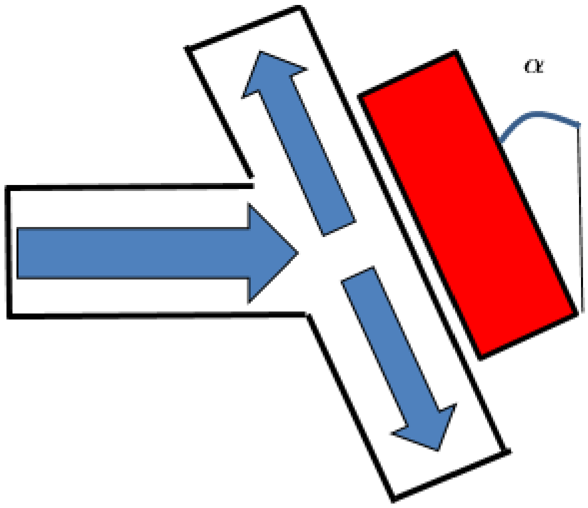


Figure 5. Schematics of a control volume corresponding to an inclined impact on the blade.

	F_n	F_t
Water jet	$m_{water} \cdot \Delta v \cdot \cos\alpha$	$-m_{water} \cdot \Delta v \cdot \sin\alpha$
Flow 1	0	$m_1 \cdot v_1$
Flow 2	0	$-m_2 \cdot v_2$

Table 6. Summary of the components of the impulse theorem applied in an inclined blade.

The impact velocity of the water jet on the blades depends on the inclined angle of the blade.

$$\Delta v = v_{impact} = v_{water} - v_{wheel}$$

Where the velocity can be divided into the normal velocity and the tangential velocity (attached to the blade surface)

$$\Delta v_n = v_{water} \cos\alpha - v_{wheel}$$
$$\Delta v_t = v_{water} \sin\alpha$$

To calculate the power, the force is multiplied by the peripheral velocity of the turbine, which is in the same direction to the normal force. The tangential force is always perpendicular to the peripheral velocity, which makes that the tangential contribution does not take into account for calculating the power.

When we substitute all the sub equations into the main one, we find the theoretical solution for the turbine in case of an inclined blade,

$$P = m_{water} \; v_{water} \cos\alpha - v_{wheel} \; \cos\alpha \, v_{wheel}$$

Note, that for $\alpha = 0$, we obtain the equation corresponding to the perpendicular case.

The water wheel has 16 blades. This makes that every blade has a contact with the water flow over 22.5 °, with $-11.25° \leq \theta \leq 11.25°$. If the turbine has less blades, the contact angle will be bigger. Theoretically the contact angle stops when the contact angle of the next blade starts. With this assumption we can calculate the average power of the turbine.

$$P_{avg} = \frac{1}{T} \int_{t_1}^{t_2} m_{water} \cdot \; v_{water} \cdot \cos\alpha - v_{wheel} \; \cdot \cos\alpha \cdot v_{wheel} \; dv_{wheel}$$

The only variable is the velocity of the wheel. To calculate the optimal peripheral speed, we take the first derivative of the power with respect to the speed.

$$\frac{dP}{dv_{wheel}} = 0\colon \; v_{wheel\ Pmax} = \frac{v_{water} \cdot \cos\alpha}{2}$$

The function $cos\,\alpha$ is the same for every velocity, the peripheral velocity of the wheel at the maximum power point is equal to half the value of the water velocity.

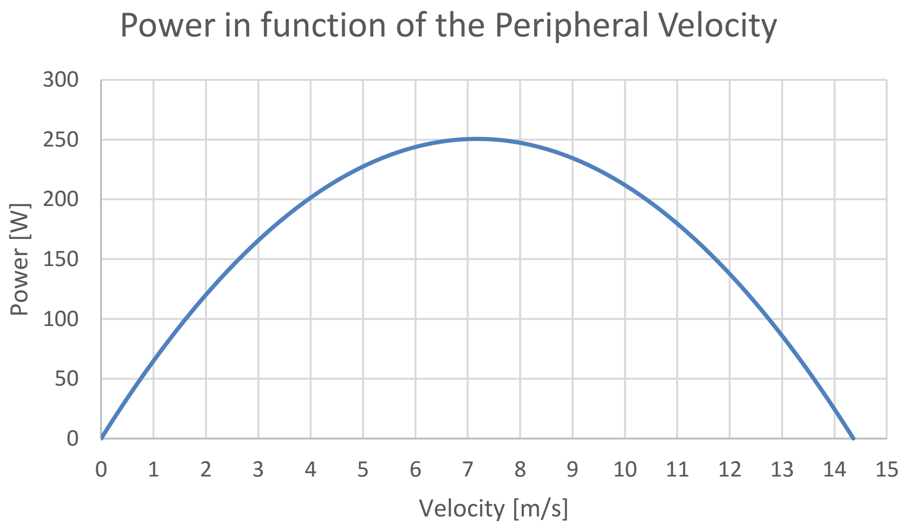


Figure 6. Power as a function of the peripheral velocity.

With a water velocity of 14.39 m/s the maximum power of the water turbine is 250.54 W at a peripheral speed of 7.20 m/s or 328.06 rpm.

5 CFD-model

The purpose of this thesis is to find the point of maximum power of the water turbine while varying three parameters. These parameters are the diameter of the nozzle, the vertical position of the nozzle and the rotation speed of the turbine. By doing the previous theoretical study, it is easier to guess a good first point of the rotation speed. Firstly, we will search the maximum point of power as a function of the rotation speed for a system with the same geometrical dimensions than the existing turbine, so we can do a comparison with experimental data obtained in previous studies. Secondly, we change the other two parameters and search for a relationship between these parameters and the power.
As point out previously, the simulations will be solved on ANSYS 15.0 with Fluid Flow CFX software code. The simulations are done on computers with operating system Windows 7, processor Intel Core 2 Quad Core Q8200 @ 2.33 GHz (4CPU) with a RAM memory of 4096 MB.

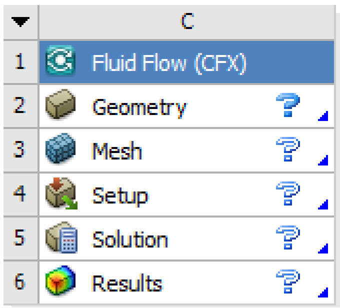


Figure 7. Schematics of the block system of the CFX code.

Inside the block of Fluid Flow (CFX), shown in *Figure 7*, there are five different steps. In the Geometry step, the geometry of the turbine will be defined and the input parameters will be declared. The Mesh step is limited to 512.000 elements in the academic version. Here every single part will have their own mesh size to calculate the simulation as correct as possible. Inside the Setup step we will define the boundary conditions of every part of the turbine and the type of the simulation in detail, including the chosen models. We can monitor the simulation inside the solution step and after the simulation, we will check the results in the Result step.
In order to obtain the total power of the turbine from the simulation data, we will use the formula where the power P is equal to the mean torque T

multiplied by the angular velocity ω. In every simulation, the angular velocity will be a constant value. The water jet will impact on the blades and reaction forces appear. These reaction forces will produce the turbine torque that is calculated as an output parameter by the CFX model.

5.1 Geometry

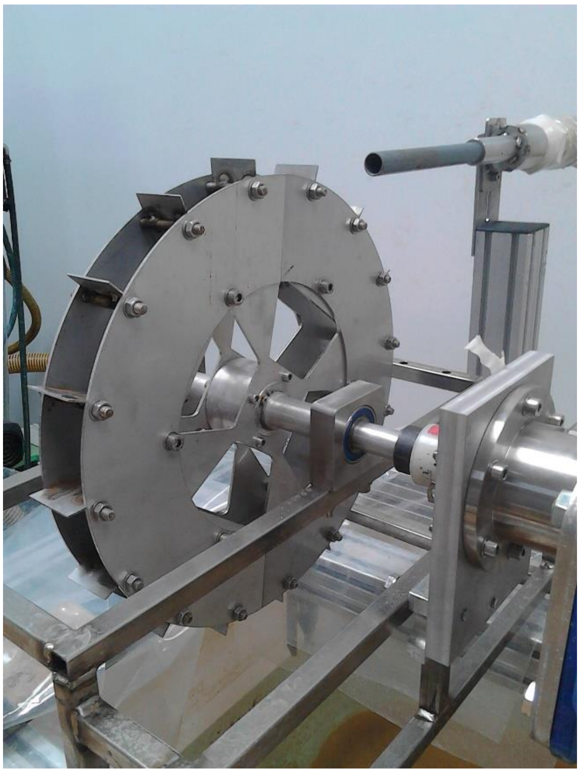


Figure 8. Laboratory turbine

The geometry of the simulation model has to be a copy of the existing turbine without the elements that are not relevant for the simulation (see *Figure 8*). Here, the geometry is split in different parts. The box (or external casing) is one part, which is almost exactly the same than the existing casing of the turbine whose main function is to collect the water and redirect it through the exit (downwards). For improving the simulation convergence, the gap between the turbine and the bottom of the box is bigger than the actual case. It does not change the value of the simulations but it is needed to avoid errors. The second part is the turbine. We are interested in the flow of the water so after drawing the turbine, it has to be substracted from the

rotating region. This rotating region is a cylinder that includes every part that rotates. Later we will assign an angular velocity to this part. The inlet is a tube starting at the side of the box. Note that we are not interested in the flow of the water inside the tube, only the outlet of the tube is important. The tube contains two parameters very relevant for the simulations: the relative height of the tube with respect to the bottom of the box and the inner diameter of the tube. This tube will be substracted from the external box. The last part is the body of influence (see *Figure 9*). This part is a region inside the box. It starts from the waterjet inlet and ends before the rotating region. It is the part where the water jet will reach the rotating region. It is defined in order to have a finer mesh than in the box. This refined region within the body of influence will provide us with element sizes small enough for being suitable for having a correct simulation of the water jet trajectory. In case we use them for the whole box, we would obtain a very large numbers of element which would extend the time of simulation.
An important point is that we have taken advantage of the symmetry properties. Thus, the geometry is cut in the longitudinal dimension. The geometry is symmetric and it will reduce the element size and the simulation time.

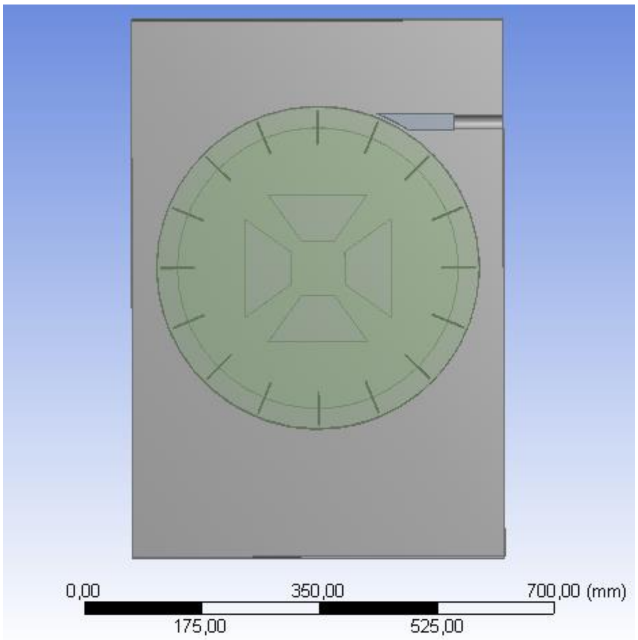


Figure 9. Parts in which we have divided the whole domain. Static external one or box (pale gray), rotating one (green) and body of influence (gray).

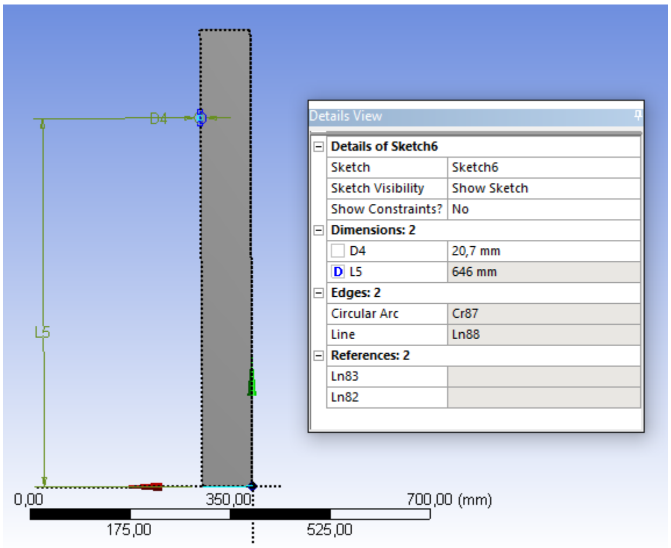


Figure 10. Side view. Position of the center of the nozzle.

The vertical position of the inlet surface is a parameter that will have upper and lower bounds. The upper bound will correspond to a vertical height of 661 mm, where the top of the inlet has the same height as the top of the 0° blade (normal to the waterjet inlet). The lower bound is where the bottom of the inlet waterjet reaches the same height as the bottom of the 0° blade, which is 631 mm (see *Figure 10*).

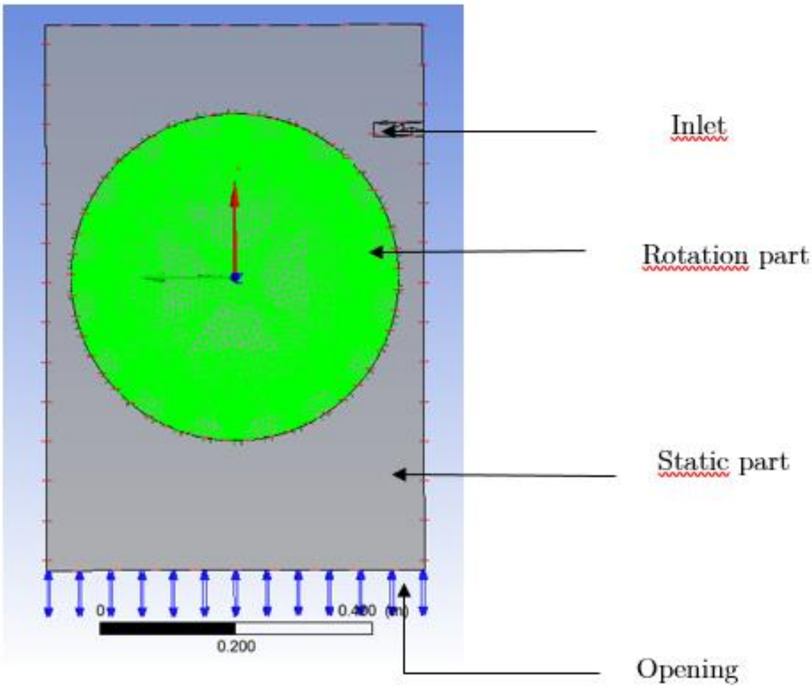


Figure 11. The main parts of the setup

5.2 Mesh

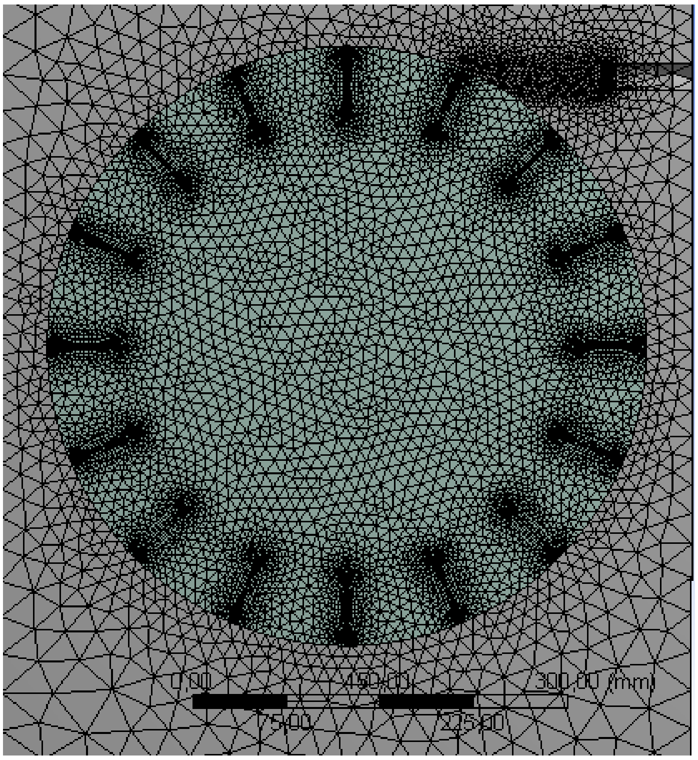


Figure 12. Mesh at the symmetry region.

To mesh the geometry, we are limited to a maximum of 512.000 elements. Firstly, a decision has to be made what the most important parts are and where the mesh has to be much finer than other regions Before giving every part an element size, it is easier to identify it with a name selection at the meshing software every single type of faces.

The box itself is meshed with a maximum element size of 30 mm. The face of the inlet has an element size of 2 mm and the faces touching the rotating part have element sizes of 10 mm. Because of the importance of a fine mesh on the path of the water jet, the body of influence has an element size of 3 mm. To use this finer mesh inside the box, the command ‘Body Sizing’ is used. The mesh of the static part of the geometry has a total amount of elements equal to 101.959. The faces of the rotating part, which touch the static body must have the same element size as the faces of the static part, which is 10 mm. The side plate of the turbine can have a bigger element size because it does not have a big influence in the results of the torque. Here it will have an element size of 20 mm. The most important parts of the turbine are the blades. A fine mesh is important in order to have a correct value of the torque. We use an element size of 3 mm in the blades. In addition, a layer of prisms has been used around this geometry. An inflation with four layers of prisms is used. The growth rate of the geometry is 1.20. This means that every next element can be a maximum of 20% bigger than the smallest neighbor element. The rotating part of the geometry have a total amount of 402.077 elements. Thus, the total of elements is 504.036, which is under the maximum of 512.000 elements.

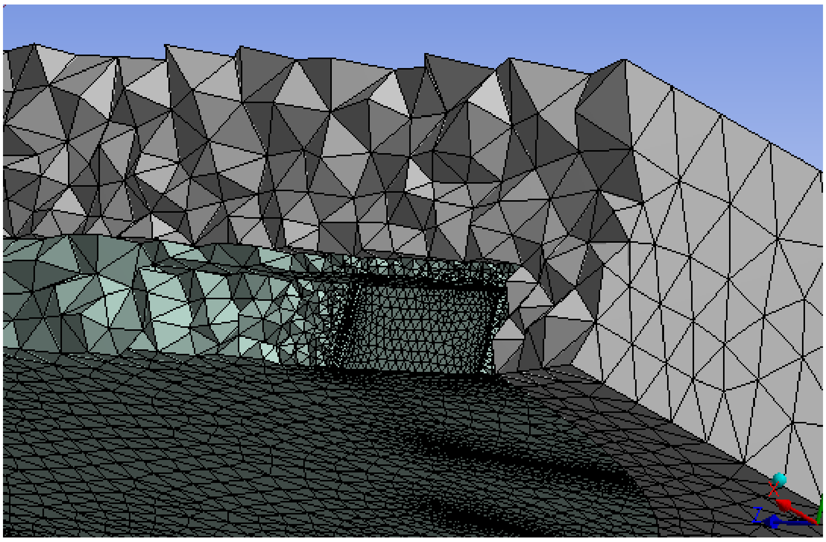


Figure 13. Detail of the mesh around one blade within the rotating domain (green part)

5.3 Setup

In the setup we define two domains, one rotating domain and one static domain. Inside the rotating domain we assign the blades and the plate as a boundary condition type 'Wall'. The face of the rotating part that is on the symmetry axis is assigned as boundary condition type 'Symmetry'.
Within the rotating domain, the fluids water and air are defined since the simulation is multiphase. Both have a zero velocity initialization. In the beginning both static and rotating domains are filled with 100% of air. The domain has a gravity of -9.81 m/s^2 with respect to the suitable coordinate

system. Inside the domain motion, the domain will be rotating with a fixed rotation speed, according to the right hand rule. The axis definition of this rotating motion will be a coordinate system with an origin in the middle of the plate of the turbine and with one axis on the normal of this plate. The theoretical approach showed the theoretical maximum point of power as a function of the angular velocity. These values are used to make smart choices as beginning points for the simulations.

Inside the static domain the walls correspond to the whole box and the tube with exception of the bottom of the box, the symmetric face and the inlet face of the tube. The inlet face is defined as an inlet boundary. The normal speed of the water jet is related to the diameter of the inlet. Within these simulations, the volume flow of the water jet is a constant value, being 4.84 l/s. Depending on which diameter we use (18 mm, 20 mm or 20.7 mm) the normal speed of the inlet is equal to 19.02 m/s, 15.41 m/s or 14.39 m/s. The fluid value of the inlet is 100 % of water. The bottom of the box is an opening boundary, so the water can go out of the box. The pressure is equal to the pressure of the atmosphere and the fluid value is 100 % of air. The symmetric face is defined as a symmetric boundary.
The analysis type has two interfaces. One for the lateral contact surface of the rotating region with the static part and one for the circle surface of the contact surfaces. Both have a frame change defined as 'Transient Rotor Stator' since the simulation is transient. The lateral interface has a pitch change option 'none' and the circle interface have the option 'Automatic'.
As analysis type, transient is used. The power will never converge to a constant value because it is a function of the position of the blades and so a function of time. To be sure that the water jet reaches the turbine and the analysis is out the range of switching-on errors, the analysis is taken for a 360 degrees rotation of the wheel. To have a proper simulation, we will carry out iterations with a time step equal to one degree of rotation. This means that the timestep duration depends on the speed of revolution per minute.

$$Time = \frac{1}{\frac{\left(\frac{rev}{min} \cdot \frac{360°}{rev}\right)}{\frac{60\ s}{min}}}$$

In addition, we need to know the torque for every single simulation. The angular speed is a constant value for the whole analysis so the power is equal to the torque multiplied by the angular speed. To calculate the torque, an expression defined in CFX with CEL code has to be made. Firstly, we define a new coordinate system with the origin in the middle of the symmetric circle face, as we said before. Then we can make the expression.

torque_x_Coord()@Rotating_Blades
torque_y_Coord()@Rotating_Blades
torque_z_Coord()@Rotating_Blades

The expressions mean that we ask the torque around the x, y, and z axis of coordinate system 'Coord' of the part called 'Rotating_Blades', which contains the blades and the plate of the turbine. These expressions allow us to monitor these values as a function of the timesteps.[5]

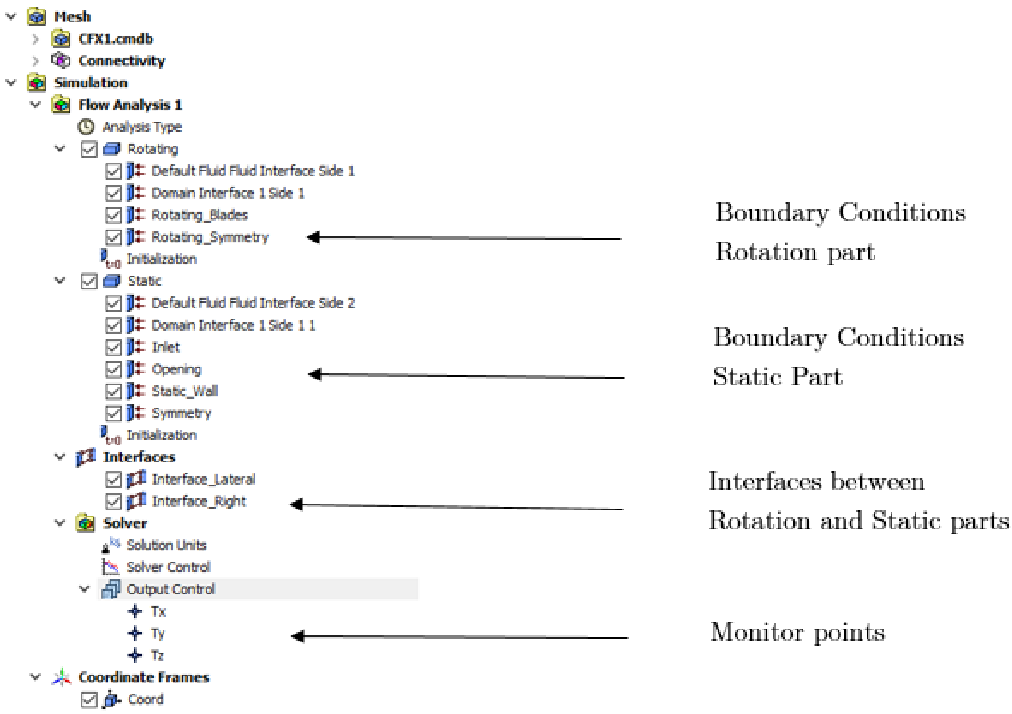


Figure 14. An overview of the Setup contents

6 Results

6.1 Diameter equal to 20,7 mm

The first analysis was done with a diameter of 20.7 mm for the inlet nozzle. This means a water jet velocity of 14.39 m/s. The theoretical equation showed a maximum point of power at a peripheral velocity half of the water jet velocity, which is now 7.20 m/s or 328.06 rpm. After doing several simulations with different rotation speeds between 50 rpm and 400 rpm, a curve can be drawn to observe the best rotation speed and to refine the step size between the different rotation speeds. (see *Figure 15* and Table 9)

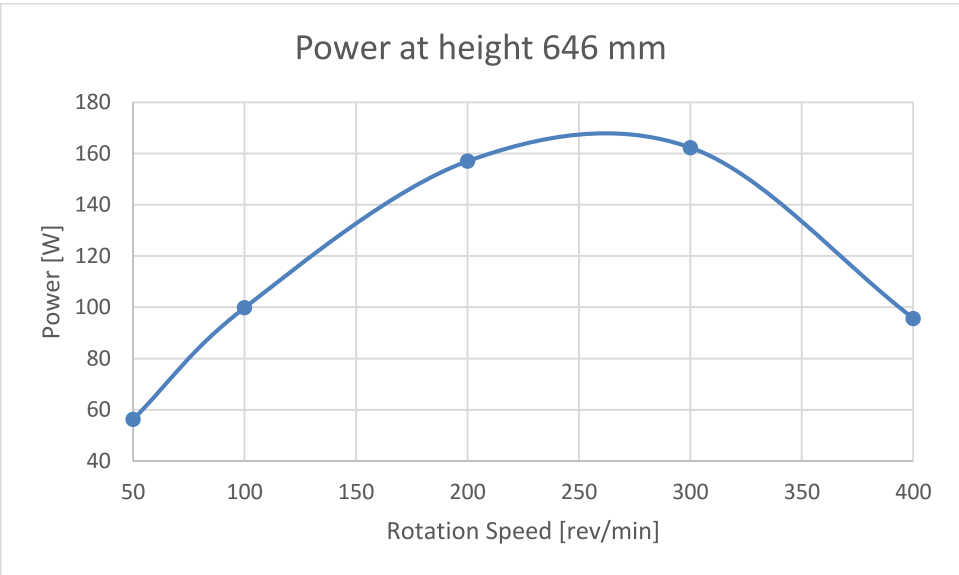


Figure 15. Power as a function of the rotation speed. Case d = 20.7 mm, h = 646 mm.

d=20,7	n [rpm]				
	50	100	200	300	400
P [W]	56,24	99,79	156,96	162,28	95,61

Tabel 7 . Power as a function of the rotation speed. Case d = 20.7 mm, h = 646 mm.

The maximum point of power is located around the speed of 250 rpm. Three extra simulation with revolution per minute of 250, 260 and 270 were analyzed for showing the maximum point of power. Observing both graphs let see us that 250 revolutions per minute is the point of maximum power among those three for a height of 646 mm, being equal to 167 W. We have not investigated the power between 200 and 250 rpm.

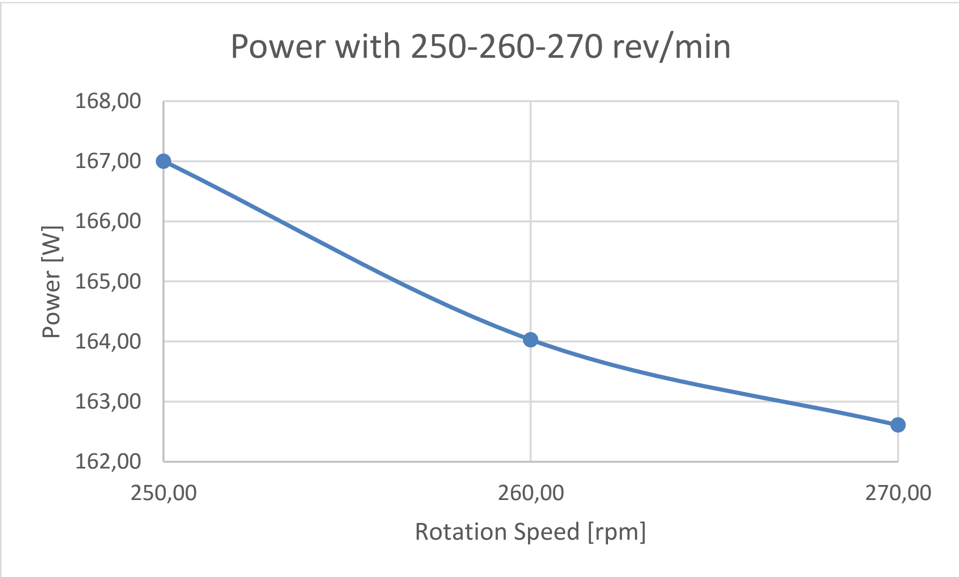


Figure 16. Power as a function of the rotation speed. Case d = 20.7 mm, h = 646 mm (detail)

After analyzing the water jet flow and the impact on the turbine, the height of the nozzle is changed. Following the formula of the torque, the higher the impact is, the more time we use the blade close to the normal position since the interference of the waterjet with the following blade is reduced and, therefore, the higher the total power. An analyze of the water jet flow shows that with the standard height a percentage of the water jet does not impact on the turbine. The water in the middle of the flow push the water at the top of the flow over the turbine (see Figure 17). When doing several simulations with higher and lower values of the nozzle height than the standard height, we will observe that the percentage of water reaching the turbine is a bigger influence than having a bigger radius.

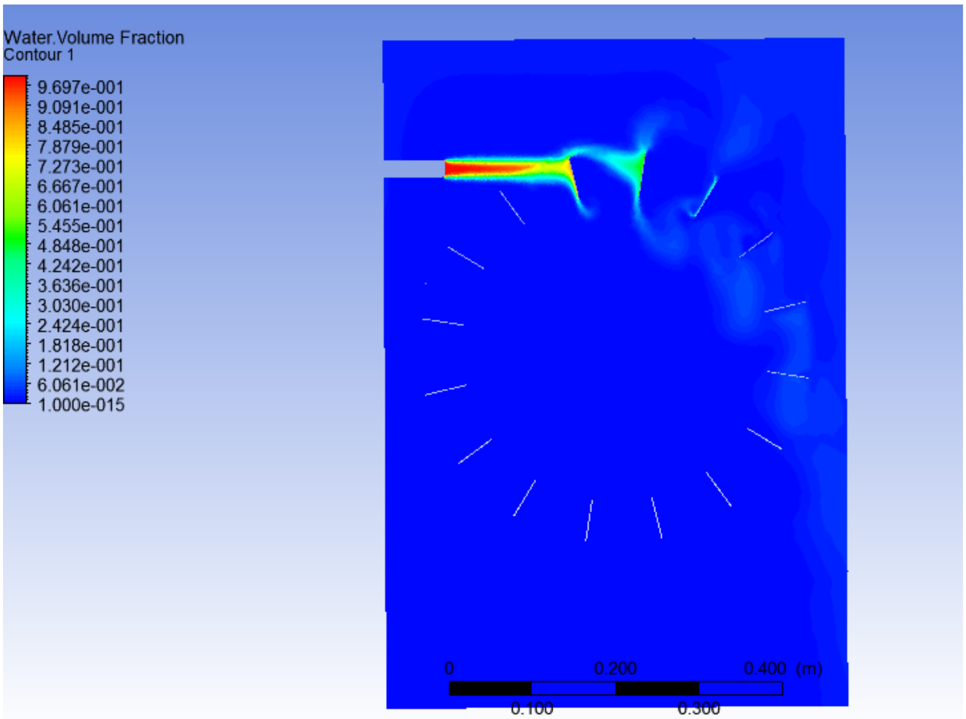


Figure 17 . Contour plot of the water trajectory. Case d = 20.7 mm, h = 646 mm, n=260 rpm.

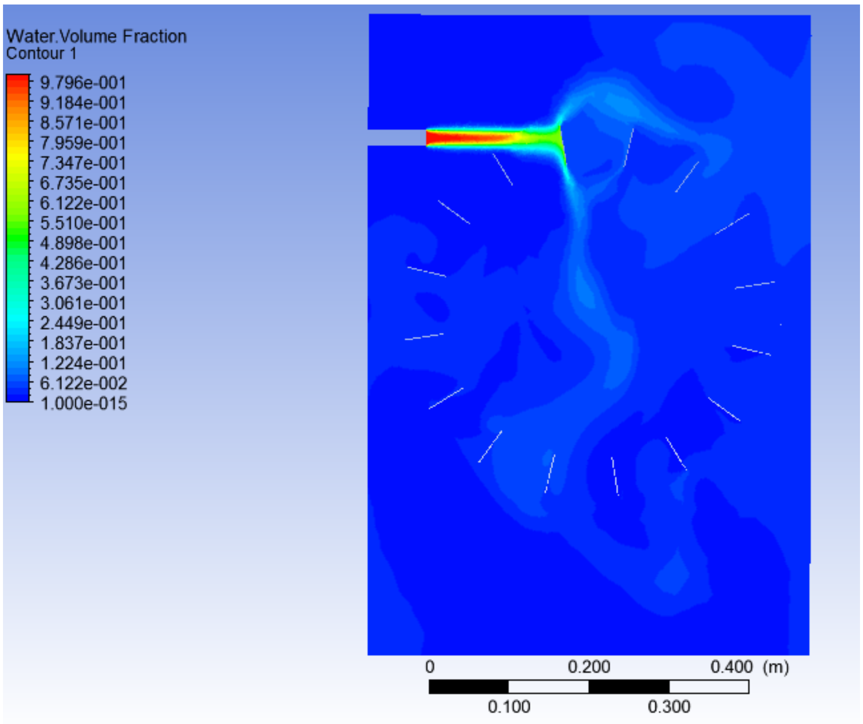


Figure 18 . Contour plot of the water trajectory. Case d = 20.7 mm, h = 646 mm, n=50 rpm.

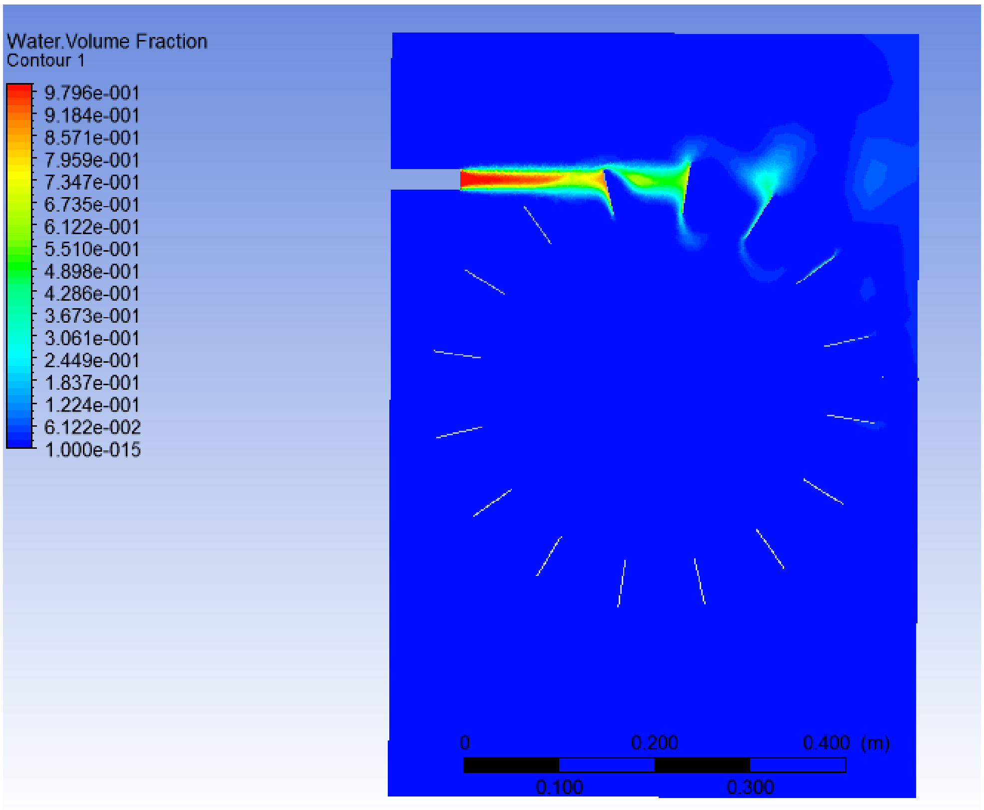
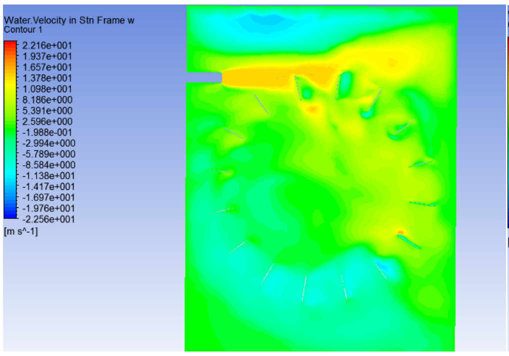


Figure 19 . Contour plot of the water trajectory. Case d = 20.7 mm, h = 646 mm, n=400 rpm.

Before changing the next parameter, which is the height of the inlet nozzle, lower and upper bounds of its value have to be determined. As a higher bound we have chosen that when the top of the turbine will be of the same height as the top of the nozzle. The standard height is 646 mm measured from the bottom of the box. Here, this standard height is the vertical distance that satisfies the condition of having a middle of the nozzle at the same height than the middle of the vertical blade on the turbine (when located normal to the waterjet). The upper bound will be the standard height plus half the length of the blade and minus the radius of the nozzle. The lower bound is the opposite of the calculation. This leads us to an upper bound of 661 mm and a lower bound of 631 mm.
The higher the position of the nozzle, the more water will be pushed over the turbine without reaching the blades, which is showed in *Figure 20. Upper bound position* *Figure 21.* The maximum point of the power as a function of the height will be that point where all of the water reach the turbine and where the angle of the blade, at first contact with the water, is as small as possible.



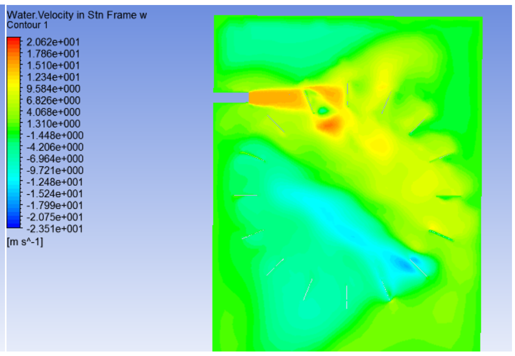


Figure 20. Upper bound position *Figure 21. Optimal height of 633 mm.*

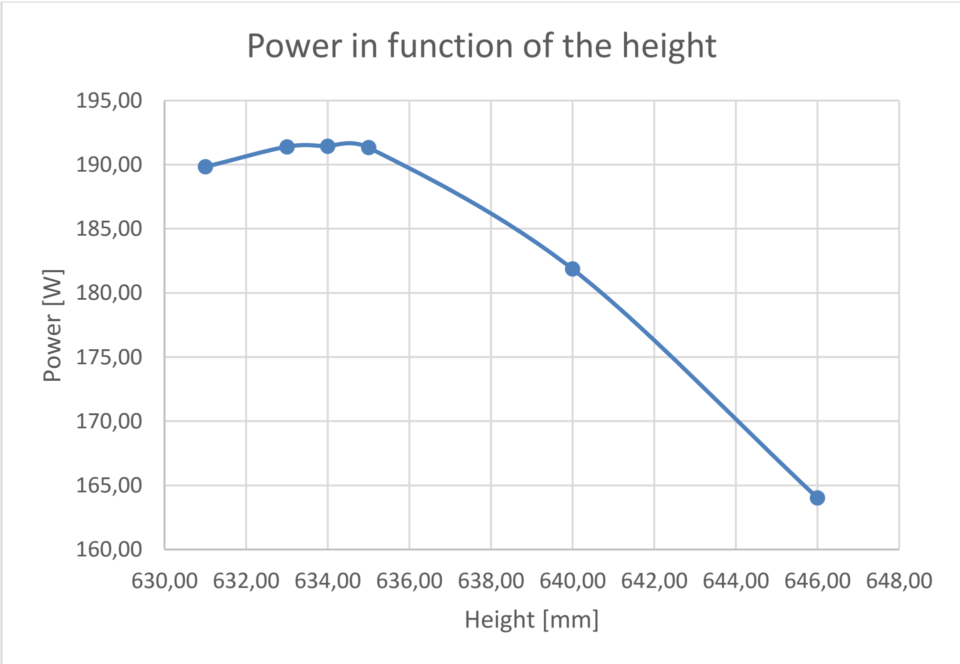


Figure 22. Power as a function of height for a nozzle diameter d = 20.7 mm and rotating speed of 260 rpm.

We can clearly see that the maximum point of power is at a lower height than the standard one of 646 mm. If we combine these two analyses of the two parameters, the rotation speed and the height, we have done the full analysis for one diameter. The power at a height of 646 mm is maximum for a rotation speed of 260 rpm. If we do an analysis of the height parameter for other rotating speeds, the maximum point of power as a function of the rotating speed shifts. As the graph shows, doing an analysis of the power as a function of the rotating speed on the maximum point of power as a function of the height, we can see that the value of the power still grows with a higher rotation speed. It is important to do a two dimensional analysis and fill a matrix in order to observe the optimal point (see values in Tabel 8 and *Figure 23*).

d=20,7	n [rpm]								
h [mm]	50,00	100,00	200,00	250,00	260,00	270,00	280,00	300,00	400,00
631,00				187,30	189,82	190,33			
633,00				188,67	191,39	192,31	192,01		
634,00				188,80	191,43	192,43	191,23		
635,00			175,00	189,50	191,32	190,84		185,00	
640,00			170,34	179,00	181,86	181,05		181,00	
646,00	56,24	99,79	156,96	167,00	164,03	162,61		162,28	95,61
653,00				132,96		131,22			

Tabel 8. Power values obtained for different values of rotating speed n and height h with the same nozzle diameter d = 20,7 mm.

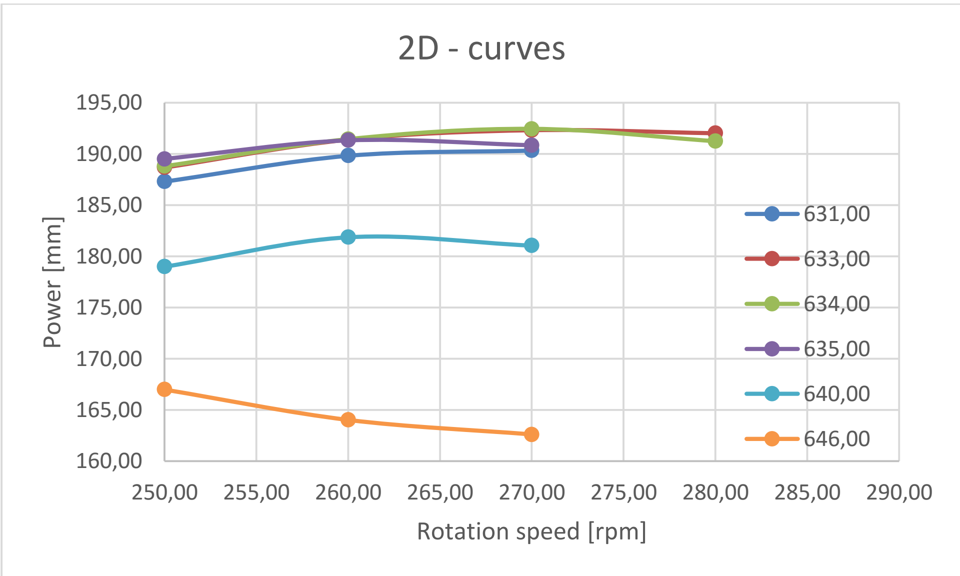


Figure 23. Power values obtained for different values of rotating speed and height h (in mm) with nozzle diameter d = 20,7 mm.

Efficiency

The total power of the incoming water jet was calculated before, and it was equal to 509 W. The theoretical power of the turbine can be calculated by the equation from chapter 4. For the parameter of the angle, we will use the average value of the formula

$$P_{avg} = \frac{1}{T}\int_{t_1}^{t_2} m_{water} \cdot \; v_{water} \cdot \cos\alpha - v_{wheel} \;\; \cdot \cos\alpha \cdot v_{wheel} \;\; dv_{wheel}$$
$$P_{avg} = 244.573\ W$$

With the velocity of the water jet equal to 14.39 m/s, the peripheral velocity of the wheel equal to half the velocity of the water jet, which is 7.20 m/s. The water jet mass flow is 4.84 kg/s. The period is $-11.25° \leq \theta \leq 11.25°$.

The theoretical efficiency of the water turbine is equal to the average power divided by the total power of the water flow.

$$\eta_{th} = \frac{P_{th-turbine}}{P_{water}}$$

$$\eta_{th} = \frac{244.573\ W}{509\ W} \;=\; 48.05\%$$

The point with maximum power from the simulations is with a rotation speed of 270 rpm and a height of the nozzle of 634 mm. The power at this point is 192.43 W

$$\eta_{sim} = \frac{P_{sim-turbine}}{P_{water}}$$

$$\eta_{sim} = \frac{192.43\ W}{509\ W} \;=\; 37.81\%$$

6.2 Diameter equal to 20 mm

The second analysis was done with the same diameter than the existing water wheel. A comparison with the theoretical calculation and with the laboratory data can be done. For every analysis, the flow of the water jet is fixed. So by changing the diameter of the nozzle, the velocity of the water jet will also change.

Firstly, some starting points has to be chosen to start the analysis and to search for the optimal point. The theoretical analysis says that the optimal point corresponds to a rotation speed where the peripheral velocity is equal to half the water jet velocity.

$$v_{water} = \frac{Q}{A} = \frac{4.84 \frac{dm^3}{s}}{\frac{\pi \cdot (20mm)^2}{2}} = 15.406\ m/s$$
$$n_{th} = \frac{v_{water}}{2 \cdot \pi \cdot d} = \frac{15.406\, m/s}{2 \cdot \pi \cdot 0.418\ m} = 351.95\ rev/min$$

The previous analysis shows an optimal rotation speed below the theoretical rotation speed. The optimal simulation rotation speed was 79.25% of the theoretical rotation speed. It is handful to use this value to make the first guess.

$$n_1 = n_{th} \cdot 79.25\% = 278.93\ rev/min$$

With the rotation speed of 279 revolutions per minute we set three rotation speeds: 270, 300 and 330 revolutions per minute. Two around 279 revolutions per minute to observe the simulation and one close to the theoretical rotation speed.

We also know from the previous analysis that a two dimensional analysis is required if we also will change the height of the nozzle. The optimal height of the previous analysis was at a height of 634 mm. We will use a height of 631 mm, which is the lower bound of the height of the nozzle, a height of 634 mm and a height of 646 mm. The height of 646 mm is the same height as the existing water wheel.

We have now three values for the input parameter of the rotation speed and three value for the height. This will give nine simulations which can be brought together into one matrix (see Table 9). The power is displayed in the matrix.

d=20	n [rpm]		
h [mm]	270	300	330
631,00	215,73	219,18	207,70
634,00	217,83	220,10	203,42
646,00	187,34	182,80	175,62

Table 9. Power output for different rotating speeds and nozzle height for inner nozzle diameter d = 20 mm.

To observe the simulations and refine the two input parameters, we observe Figure 24 and Figure 25 of the matrix shown in Table 9.

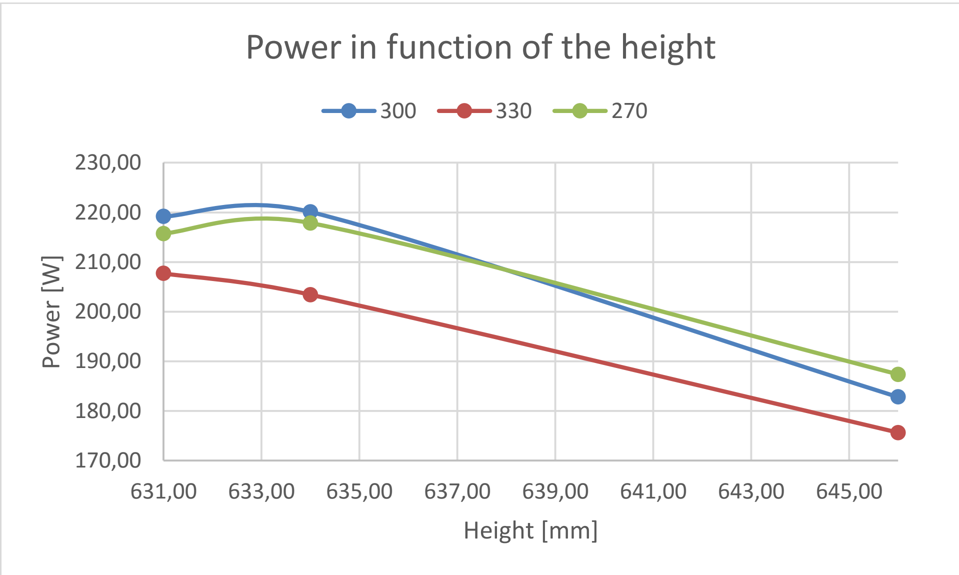


Figure 24. Power as a function of the height of the nozzle at different rotation speeds for nozzle inlet diameter equal to 20 mm.

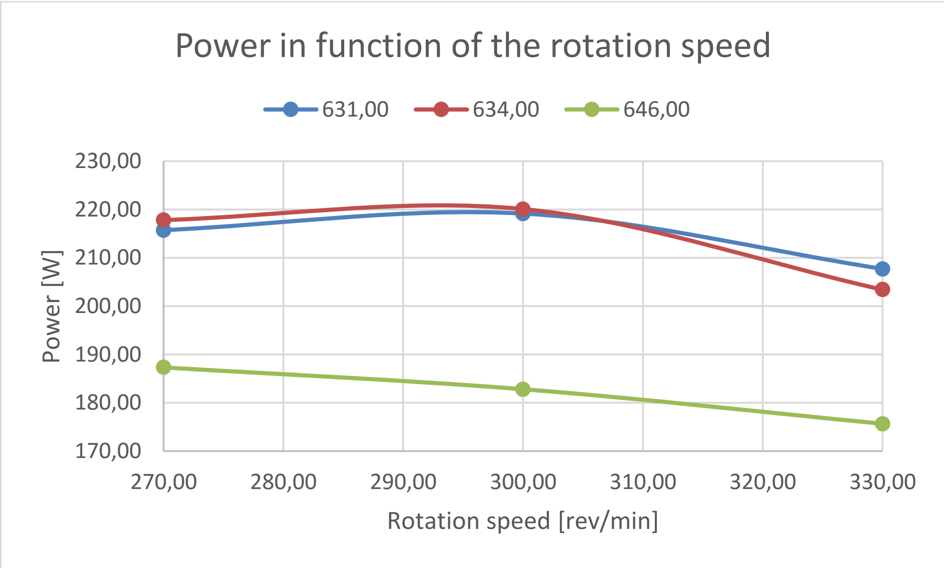


Figure 25. Power as a function of the rotation speed for different nozzle heights and for a nozzle inlet diameter equal to 20 mm.

The optimal rotation speed is around 290 revolutions per minute while the optimal height is around 633 mm. We will do three more simulations with 290 revolutions per minute and heights of 631, 632 and 633 mm (Table 10 and Figure 26).

d=20	n [rpm]
h [mm]	290,00
631,00	219,16
632,00	221,06
633,00	220,60

Table 10. Power output for inner nozzle diameter equal to 20 mm at rotation speed equal to 290 rpm and different heights of the nozzle.

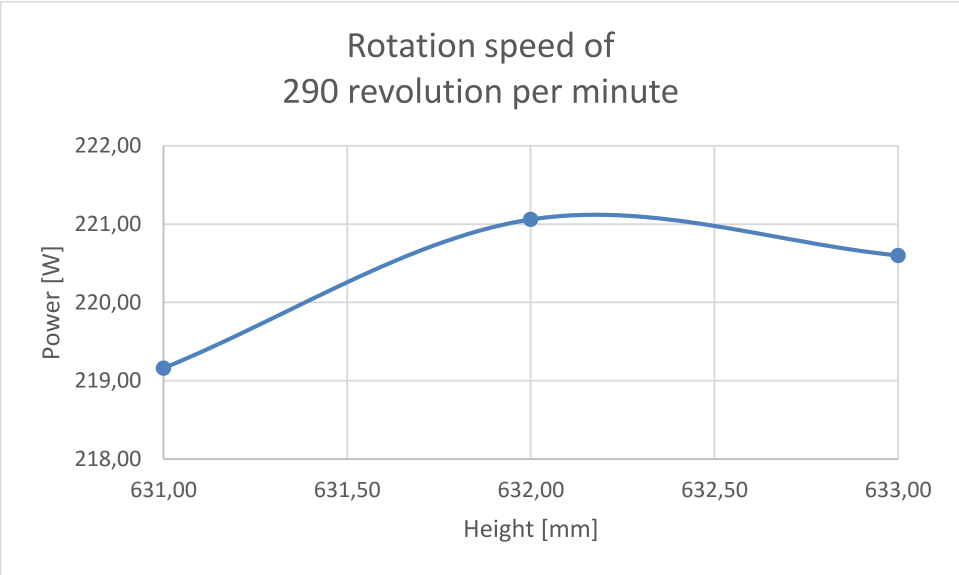


Figure 26. Power output for inner nozzle diameter equal to 20 mm at rotation speed equal to 290 rpm and different heights of the nozzle

We see that the optimal point for a maximum power is at a height between 632 mm and 632.5 mm, with a rotation speed of 290 revolutions per minute. The power is equal to 221.06 W.

Efficiency

The total power of the water jet is calculated with to formula of kinetic power and the potential power chapter 4.

$$P_{water} = \frac{m \cdot v_{water}^2}{2} + m \cdot g \cdot h = 584.31\ W$$

The efficiency of the theoretical power should be the same as the value of the first analysis, which was 48.84%.

$$P_{avg} = \frac{1}{T} \int_{t_1}^{t_2} m_{water} \cdot \ v_{water} \cdot \cos\alpha - v_{wheel} \ \cdot \cos\alpha \cdot v_{wheel}\ dv_{wheel}$$

$$P_{avg} = 281.501\ W$$

$$\eta_{th} = \frac{P_{avg}}{P_{water}} = \frac{281.501\ W}{584.31\ W} = 48.18\ \%$$

The efficiency calculated with the results of the simulations is done with the maximum point of power, which is 221.06 W.

$$\eta_{sim} = \frac{P_{sim}}{P_{water}} = \frac{221.06\ W}{584.31\ W} = 37.83\ \%$$

6.3 Diameter equal to 18 mm

The last simulation is done with a smaller diameter, which is now 18 mm. This will directly increase the velocity of the water jet and so the power of the turbine because of the linearity between both.

Just like the second analysis, we will take 79.25 % of the theoretical optimal velocity and rotation speed.

$$v_{water} = \frac{Q}{A} = \frac{4.84\ \frac{dm^3}{s}}{\frac{\pi \cdot (18\ mm)^2}{2}} = 19.02\ m/s$$

$$n_{th} = \frac{v_{water}}{2 \cdot \pi \cdot d} = \frac{19.02\,m/s}{2 \cdot \pi \cdot 0.418\ m} = 434.52\,\frac{rev}{min}$$

$$n_2 = n_{th} \cdot 79.25\% = 344.35\ rev/min$$

According to the percentage of the theoretical optimal rotation speed, the expected optimal rotation speed will be around 344.35 rpm. We can conclude that the chosen heights for the analysis for d=20 mm are good to start the matrix, which are 646 mm, 634 mm and 631 mm. The rotation speeds will be 300 rpm, 350 rpm and 400 rpm.

d=18	n [rpm]		
h [mm]	300	350	400
631	327,08	338,26	326,86
634	328,14	342,75	326,25
646	282,90	290,06	281,07

Table 11. Power output for inner nozzle diameter equal to 18 mm at different rotation speeds and different heights of the nozzle.

The highest value for the power is in the middle of the matrix with almost the same values for 300 rpm and 400 rpm. We can conclude that the rotation speed will be close to 350 rpm, as we can see on Figure 27. For the height as parameter, 634 mm gives the highest values but after observing Figure 28 we can guess that a lower height of 633 mm may be better.

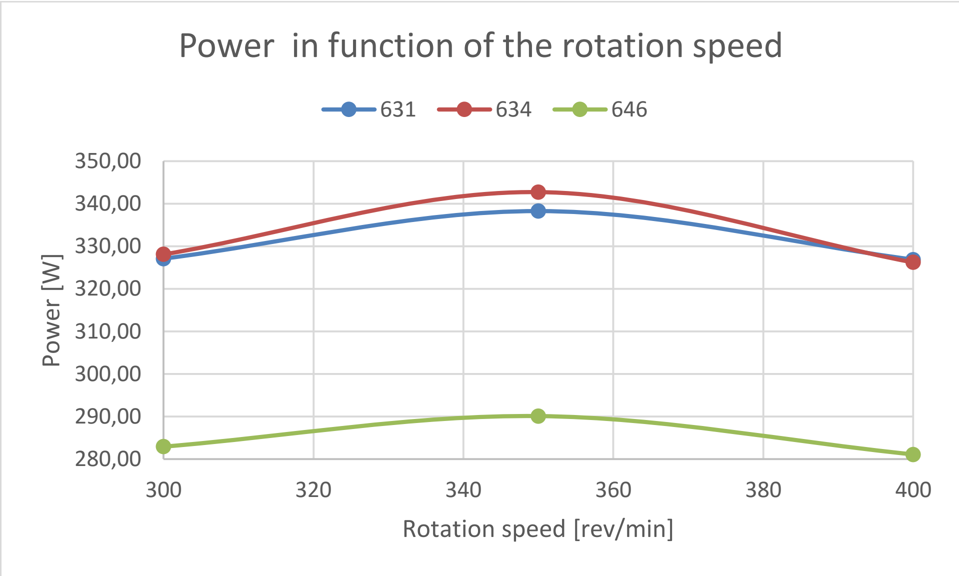


Figure 27. Power output for inner nozzle diameter equal to 18 mm at different rotation speeds and different heights of the nozzle.

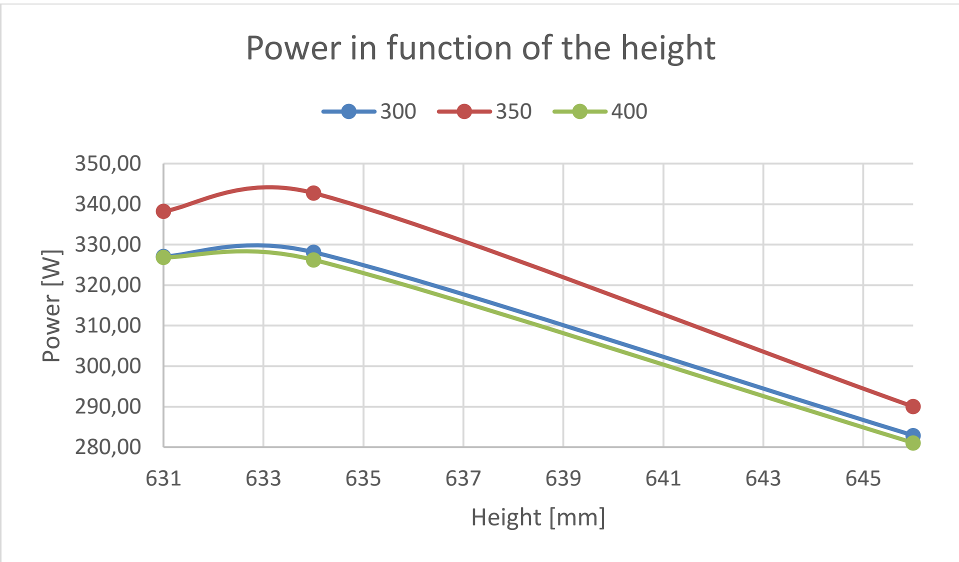


Figure 28. Power output for inner nozzle diameter equal to 18 mm at different rotation speeds and different heights of the nozzle.

To be sure that 350 rpm gives the highest values of the maximum power, an extra simulation will be done with a slightly lower and higher rotation speed, respectivily 340 rpm and 360 rpm. We will do the same with the height were we will do an extra simulation with heights of 633 mm and 636 mm.

d=18	n [rpm]	
h [mm]	340	360
633	341,03	341,46
636	337,39	335,53

Table 12. Power output for inner nozzle diameter equal to 18 mm at different rotation speeds and different heights of the nozzle.

We can see that for the four extra simulations the results are lower than the one from 350 rpm and a height of 634 mm.

Efficiency

The efficiency is calculated and compared with the power of the water jet, which is calculated with the kinetic energy like in the other cases.

$$P_{water} = \frac{m \cdot v_{water}^2}{2} + mgh = 892.76\ W$$

The theoretical power of the turbine will be calculated with the same formula as before, where the velocity of the wheel is half the water speed, within the bounds of -11.25 ° and 11.25 °.

$$P_{avg} = \frac{1}{T} \int_{t_1}^{t_2} m_{water} \cdot\ v_{water} \cdot \cos\alpha - v_{wheel}\ \cdot \cos\alpha \cdot v_{wheel}\ dv_{wheel}$$

$$P_{avg} = 429.06\ W$$

$$\eta_{th} = \frac{P_{avg}}{P_{water}} = \frac{429.06\ W}{892.76\ W} = 48.06\ \%$$

The point of maximum power is 342.75 W, with a rotation speed of 350 rpm and a height of 634 mm.

$$\eta_{sim} = \frac{P_{sim}}{P_{water}} = \frac{342.75\ W}{892.76\ W} = 38.39\ \%$$

7 Comparison

Points of maximum power			
D [mm]	n [rpm]	h [mm]	P [W]
20,7	270	633	192,43
20,0	290	632	221,06
18,0	350	634	342,75

Table 13. Overview of the optimal points for the three cases

Table 13 shows the three points of maximum power for the three cases. For the first case, with a diameter of 20.7 mm, we already calculated that the optimal rotation speed after the simulations was at a ratio of 79.25 % compared to the theoretical optimal rotation speed. For case two and three, with diameter of 20 mm and 18 mm, are these ratios' respectively 82.40 % and 80.55 %. We can see that for the height as parameter, in the three cases the maximum point of power is around 633 mm. There is no big difference in this value.

The best value to do a comparison of the three analysis is to compare is the efficiency of the turbine.

D [mm]	η [%]
20,7	37,81
20,0	37,38
18,0	38,39

Table 14. Overview of the maximum efficiency of the three cases

Table 14 shows that there is no significant difference in efficiency between the points of maximum power in the three cases with the diameter as parameter. The efficiency of the turbine is independent of the diameter.

Before this thesis, the existing turbine has been evaluated and tested. In Table 15 the experimental data is shown. The existing turbine has a diameter of 20 mm.

N [rpm]	T [Nm]	P [W]	Efficiency [%]
460	0,00	0,00	0,00
440	2,58	118,91	20,70
380	4,99	198,56	34,57
340	6,55	233,17	40,60
320	7,09	237,65	41,37
300	7,65	240,33	41,84
280	8,11	237,67	41,38
240	8,78	220,57	38,40
200	9,92	207,67	36,16
170	10,36	184,50	32,12

Table 15 Laboratory data

The experimental data shows a slightly higher maximum power and efficiency (240.33 W and 41.84 %) than the simulations (221.06 W and 38.48%). The optimal rotation speed is almost the same, with 300 rpm for the experimental data and 290 rpm for the simulations.

8 Conclusion

The theoretical approach to calculate the best rotation speed of the turbine use a point mass flow instead of an area where the water jet has the impact. There is also no losses on the water speed. After doing three different cases we can conclude that the true optimal rotation speed is around 80 % of the theoretical rotation speed.

The height of the nozzle has a big impact on the efficiency of the turbine. The existing turbine has a height of 646 mm, which is the position where the nozzle is at the same height as the middle of the zero-degree blade. The optimal height coming from the simulations is lower. For the case with diameter of 20.7 mm, the optimal height is 633 mm. This change leads to a power increase of 18.34 % compared to the standard height. It is hard to say what to optimal height is for a turbine. In this thesis, several simulations were done and the results were analyzed. At a height equal to the existing turbine or even higher, the top part of the water jet is pushed over the turbine. The water jet will not have been used for 100 %. This effect has a big impact on the efficiency. If the nozzle goes below the optimal height, the angle of the impact of the water jet with the blades is too high which has an influence on the force and thus on the power and efficiency. The optimal height is as high as possible where the water wouldn't have pushed over the blades and where the angle is as low as possible.

The final conclusion of this thesis is that when a water wheel turbine is used with straight blades, the maximum efficiency of the turbine is around 38 % and that the vertical position of the nozzle has a big impact on the total efficiency of the turbine.

9 Bibliography

[1] R. Tony, "L. A. Moritz, Grain-Mills and Flour in Classical Antiquity," *Antiq. Class.*, p. 223, 1959.

[2] A. A. Lahimer, M. A. Alghoul, K. Sopian, N. Amin, N. Asim, and M. I. Fadhel, "Research and development aspects of pico-hydro power," *Renewable and Sustainable Energy Reviews*, vol. 16, no. 8. pp. 5861–5878, 2012.

[3] A. K. Vashisht, "Current status of the traditional watermills of the Himalayan region and the need of technical improvements for increasing their energy efficiency," *Appl. Energy*, vol. 98, pp. 307–315, Oct. 2012.

[4] F. White, "Fluid Mechanics," *McGraw-Hill,New York*, p. 161, 2010.

[5] T. D. Canonsburg, "ANSYS CFX Reference Guide," *Elements*, vol. 15317, no. November, pp. 724–746, 2011.