International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 09, September 2022

Microgravitational Hydro-Turbine for Rural Areas as Renewable Energy Resource: Numerical Simulation

Sunil Gurav[1]
M. Tech. Scholar
Mechanical Engineering Department
Sardar Patel College of Engineering,
Andheri (west)
Mumbai, Maharashtra, India

Sudhakar Umale[2]
Former Associate Professor,
Sardar Patel College of Engineering,
Andheri (west)
Mumbai, Maharashtra, India

***Abstract*— One of the most dependable and effective methods for producing clean, renewable energy is hydropower. The vast expansion of hydropower development results from rising energy demand in developing nations. With this in mind, using low-head hydro vortex turbines has been suggested as a potential remedy for the lack of electricity supply in rural areas. It is crucial to create a Micro Gravitational Hydro-turbine, which enables the conversion of energy in a moving fluid to rotational energy through a low head and low flow rate with a reasonably simple structure in order to alleviate the energy supply shortfall. So, try to analyze the most output possible. The current project uses the commercial ANSYS Fluent 2021 R1 software to perform CFD analysis on a gravitational hydro-turbine system while varying the number of blades to compare the power output. Based on the Micro Gravitational Hydro-turbine speed range of 600-1000 RPM, the rotational velocity of 1000RPM is also considered. 20, 10, and 15 blades were removed. It is investigated how many blades will affect the turbine's ability to produce the most torque. The output torque measured with 20 blades was 6565 N.m, which is higher than designs with 15 and 10 blades. The 20-blades design is considered superior to the 15 and 10 blades designs.**

Keywords: Hydro-Turbine, Torque, Performance, Ansys Fluent, Number of Blades.

Nomenclature

GWVPP	Gravitational Water Vortex Power Plant
V_θ	Tangential (m/s)
V_r	Radial (m/s)
V_z	Axial velocity (m/s)
V	kinematic viscosity (m^2/s)
ρ	Fluid density (kg/m^3)
g	Gravitational acceleration (m/s^2)

I. INTRODUCTION

(GWVPP) is a hydroelectric facility with a low hydraulic head. An appropriate turbine with an optimal form and blade profile should be built in order to achieve maximum efficiency [1][2]. In order to get the maximum performance out of GWVPP [3], use lighter and stronger materials to minimize the weight of the turbines. The overall efficiency of the vortex power plant varied from 15.1% to 25.36% depending on the geometry of the basin, volumetric flow rate, turbine position and blade geometry, and number [4]. Christine et al. [4] discovered that the power plant's efficiency improved when the number of turbine blades was raised from two to four. The result of the turbine mentioned above [4] is the opposite of that of GWVPP [3]; efficiency decreased as turbine blades increased. The optimal number of turbine blades and ratio between blade size and basin diameter may exist, according to literature with opposing conclusions. A gravitational water vortex turbine is an ultra-low head turbine that can operate in a low head range of 0.7-2 m [2]. Dhakal et al. [2] focused on the effect of runner position on the cylinder and conical geometry basin structure of Gravitational water vortex power plant.

Micro-hydro power plants are a subset of the gravitational water vortex power plant, which is a green technology. Because the highest reported power generation did not surpass 100 kW [5], it is now designated as micro hydropower. The water in this plant enters through a sizable, straight intake and exits tangentially into a circular basin. The water will create a strong vortex that exits the outlet at the shallow basin's center bottom. The plant uses the dynamic force produced by the vortex rather than the pressure differential since it only needs an extremely small number of the hydraulic head [6][7]. As a result, the GWVPP has relatively cheap development and production costs compared to other hydropower technologies.

Fig. 1. A gravitation water vortex plant with a Zotlöterer turbine near Ober-Grafendorf, Austria.

According to the literature, geometrical factors influence how much power can be extracted. In order to compare power production and attempt to analyze a gravity hydro-turbine system in this study, commercial software ANSYS Fluent 2021 R1 was used.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 09, September 2022

II. METHODOLOGY AND RESULTS

A. The Mathematics of CFD

The Navier-Stokes equations are described as used for describing the processes of momentum, heat, and mass transport. Although there was no universal analytical solution for these partial differential equations, which were developed in the early nineteenth century, they can be discretized and solved numerically. These additional equations are frequently derived using an approximation model, with turbulence models serving as a key illustration. In CFD codes, a variety of different solution techniques are employed. The finite volume technique is the most widely used and the foundation for CFX.

The most common, and the one on which CFX is based, is known as the finite volume technique. Using this method, the area of interest is split up into smaller subdivisions known as control volumes. For each control volume, the equations are discretized and solved iteratively. As a result, it is possible to estimate each estimating each variable value throughout the domain at particular locations is possible variable's value throughout the domain at particular locations. This provides a complete picture of the flow's behaviour.

Governing Equations

It was considered that the river flow across the basin was constant, incompressible, and turbulent. Equations (1), (2) and (5) are the continuity equation, Conservation of momentum and conservation of energy equation used.

Continuity Equation

$$\frac{\partial V_r}{\partial r} + \frac{\partial V_z}{\partial r} + \frac{V_r}{r} = 0 \qquad (1)$$

Conservation of momentum:

R-momentum equation:

$$V_r \frac{\partial V_\theta}{\partial r} + V_z \frac{\partial V_\theta}{\partial z} - \frac{V_r V_\theta}{r} = V\left(\frac{\partial^2 V_\theta}{\partial r^2} + \frac{\partial V_\theta}{r \partial r} - \frac{V_\theta}{r^2} + \frac{\partial^2 V_\theta}{\partial z^2}\right) \qquad (2)$$

$\boldsymbol{\theta}$-momentum equation:

$$V_r \frac{\partial V_r}{\partial r} + V_z \frac{\partial V_r}{\partial z} - \frac{{V_\theta}^2}{r} + \frac{\partial \rho}{\rho \partial r} = V\left(\frac{\partial^2 V_r}{\partial r^2} + \frac{\partial V_r}{r \partial r} - \frac{V_r}{r^2} + \frac{\partial^2 V_r}{\partial z^2}\right) \qquad (3)$$

Z-momentum equation:

$$V_r \frac{\partial V_z}{\partial r} + V_z \frac{\partial V_z}{\partial z} + \frac{\partial p}{p \partial z} = g + V\left(\frac{\partial^2 V_z}{\partial r^2} + \frac{\partial V_z}{r \partial r} + \frac{\partial^2 V_r}{\partial z^2}\right) \qquad (4)$$

Conservation of energy

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} + \frac{\dot{q}}{k} = \frac{1}{\alpha}\left(\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z}\right) \qquad (5)$$

Geometry Detail

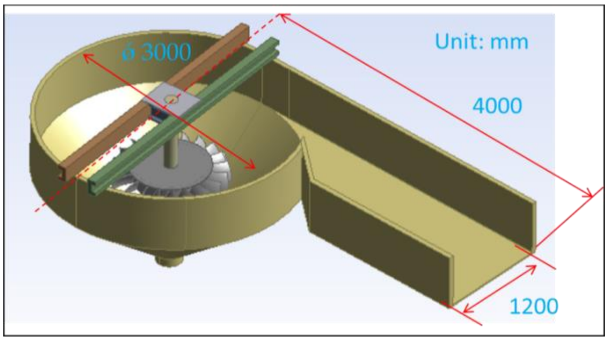

Fig. 2. Structural Geometry of Hydro Turbine.[5]

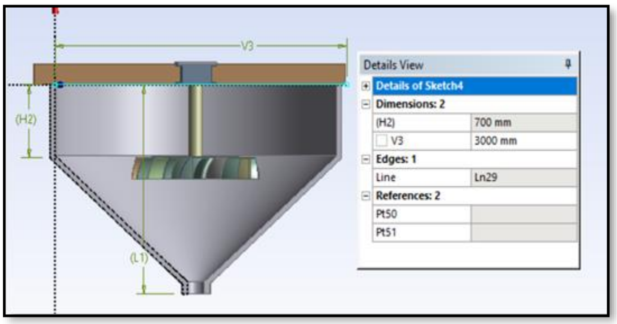

Fig. 3. Top View of Structural Geometry of Hydro Turbine.

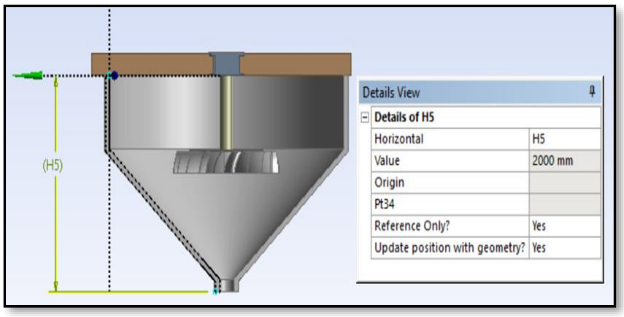

Fig. 4. Cross Sectional View of Structural Geometry of hydro turbine.

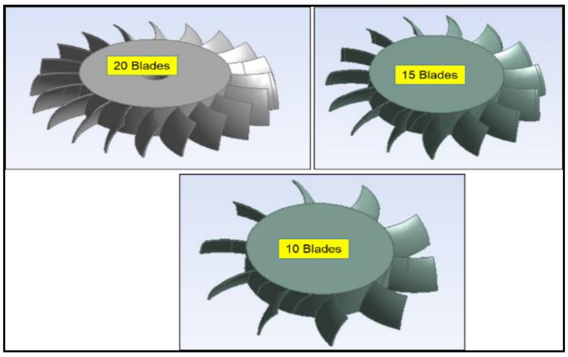

Fig. 5. Micro gravitational hydro turbine blade design with 20, 15 and 10 Blades respectively

The values of parameters of micro-hydro turbine:

- Number of blades = 20, 10 & 15
- Diameter of turbine = 300 mm
- Inlet area of turbine = $1200 \times 700\ mm^2$
- Thickness of blades = 10 mm
- Diameter of outlet = 150 mm
- Angle of blade = 80°

Based on 3D geometry, fluid domain has been prepared in ANSYS Design Moduler.

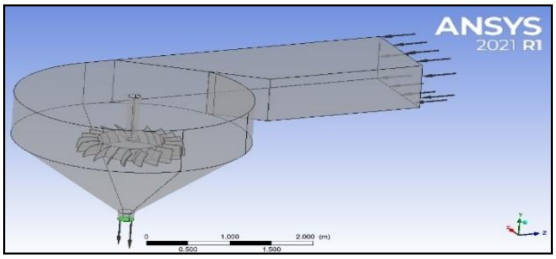

Fig. 6 Fluid Domain of the micro gravitational Hydro turbine with 20 number of blades.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 09, September 2022

Meshing

It is convenient to select the free mesh because it has sharp curves, so that shape of the object will not alter. To mesh the fluid domain the fluid element is used. The final model from design modeler was then imported in Ansys Workbench. At all wall surfaces inflation layers were used for proper flow and very fine mesh was used to capture the accuracy of solution. Body size of 1mm used for fluid domain and 0.1 mm element size for blade surface.

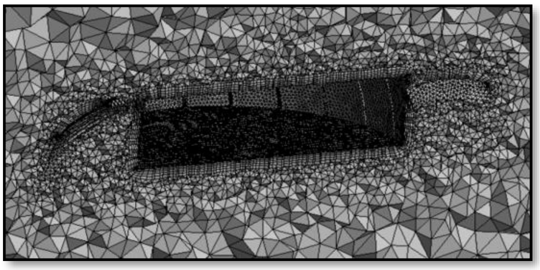

Fig.4.7 Meshing Fluid Domain of micro gravitational Hydro turbine with different number of blades are 20, 15 and 10.

The K-epsilon (k-ε) turbulence model is used to simulate the flow characteristics of turbulent flow conditions. It is two-equations model that provides a broad definition of turbulence with two equations of transport. The real purpose of this k-ε model was to improve the mixing length model. It is used to find an alternative to the algebraically defined turbulent length on the scales in a fluid simple to complex fluid flow. The k-ε model does not focus on processes that affect kinetic energy as in the previous turbulence model. The basic assumption of this model is that, a turbulent dynamic viscosity means that the ratio between shear stress and shear strain is the same in all fluid flow directions.

Details Solver Control in Flow analysis

- Continuity
- Momentum
- Turbulence eddy dissipation
- Turbulence Kinetic Energy

Fluid Properties: Water as fluid was considered.

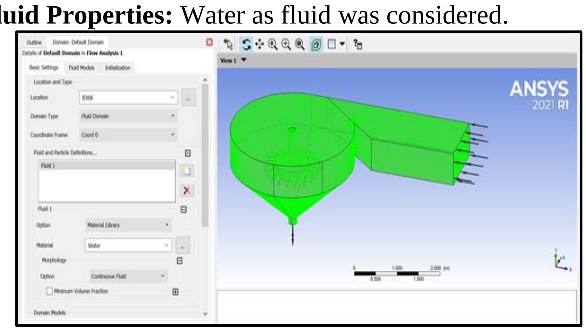

Fig. 8. Fluid domain defined with water as material

Boundary Condition:

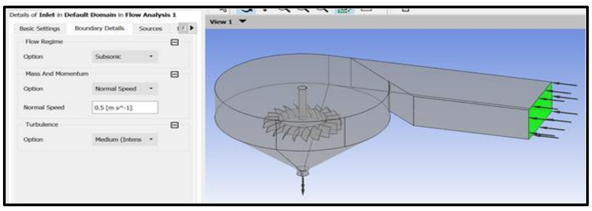

Fig. 9. Inlet: Velocity of 0.5M/s is defined at inlet

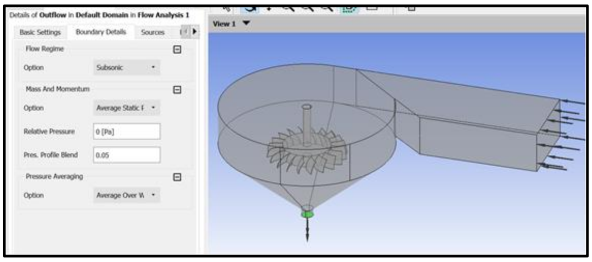

Fig. 10. Outlet: Outlet defined with outflow boundary.

Solver Settings: 100 iterations with convergence criteria of residual were below 10−4.

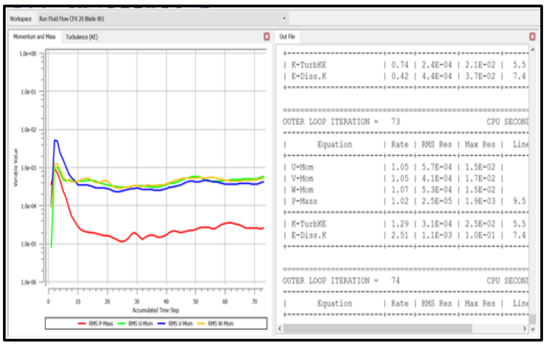

Fig. 11. Convergence graph

The velocity plots

Fig. 12 shows that the flows at a high speed at the inlet, supporting the assumed turbulent flow, the plot is helpful to understand the development of flow inside the domain.

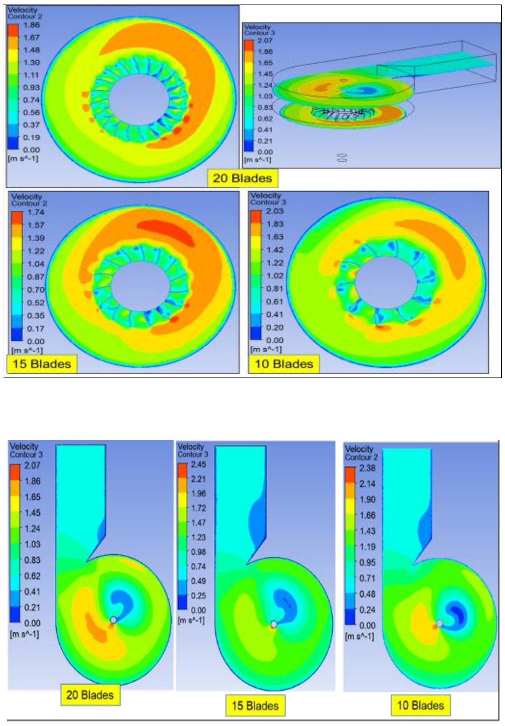

Fig. 12. Velocity plot CFD model of micro gravitational hydro turbine with number of blades 20, 15 and 10.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 11 Issue 09, September 2022

It is found that during 15 number of blades, noticeable decrease in velocity was seen at the throat area. The velocity increases at the end of throat region. In the same plane, there were alternate high- and low-velocity zones.

Pressure Plot:

Fig. 13 shows that the maximum pressure location for diameter d and its distribution to calculate reacting torque and power output.

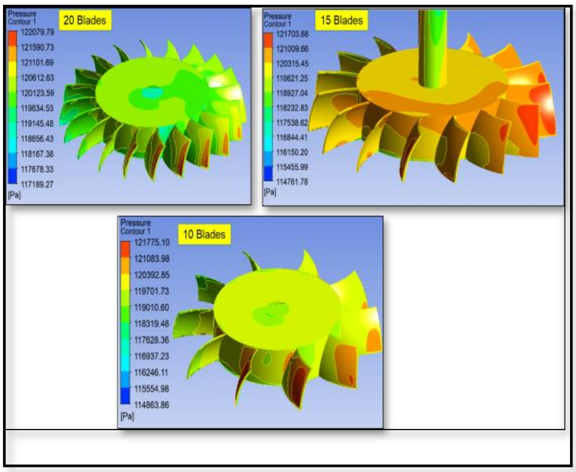

Fig. 13. Pressure plot CFD model of micro gravitational hydro turbine with different number of blades are 20, 15 and 10.

The largest output torque was detected because of the pressure distribution from CFD study, which showed a maximum pressure of 0.12MPa for 20 blades. The distribution of pressure also reveals the maximum pressure near the blade's edge.

Torque:

The function calculator is available in CFX post and used to calculate the output torque on the blade due to pressure distribution.

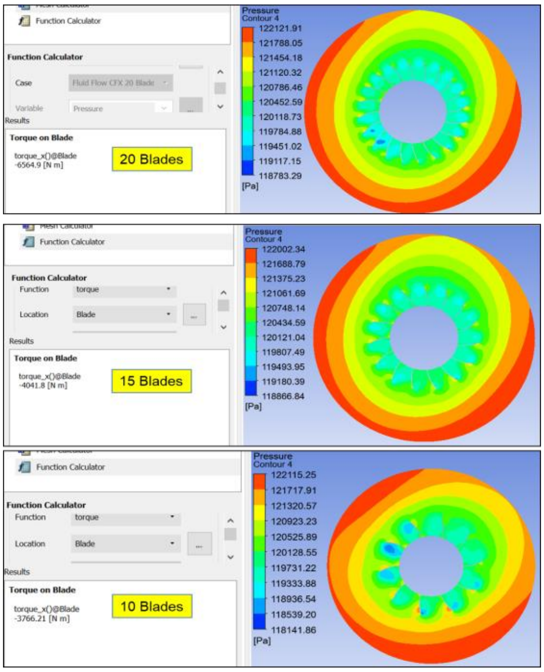

Fig. 15 Torque Plot of CFD model of micro gravitational Hydro turbine with different number of blades are 20, 15 and 10.

The micro gravitational hydro-turbine with the maximum number of blades 20 produces the maximum torque, followed by 15 and 10 blades. The maximum torque achieved is 6564.9 Nm, which is approximately twice as much as torque, when number of blades are 10.

Table I. Output of Torque results from CFD mode

No Of Blade	*Output Torque (N. m)*
20	6564.9
15	4041.8
10	3766.2

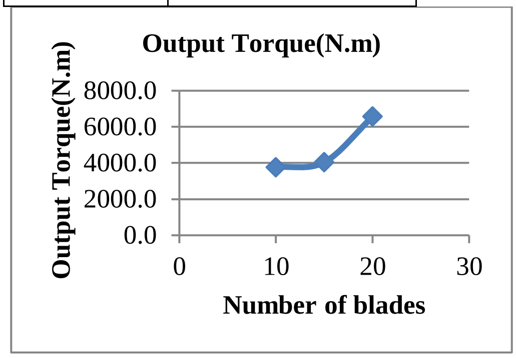

Fig. 16. Number of blades v/s output Torque (N. m)

- CFD analysis is performed in ANSYS CFX with water flow input velocity of 0.5 m/s to calculate pressure and torque developed on blade.
- In case of 20 number of blades, the output torque is observed 6565N.m which is maximum than 15 and 10 number of blades design.
- Pressure distribution from CFD analysis shows maximum pressure of 0.12 MPa for 20 blades and hence the output torque observed highest. The distribution of pressure shows maximum pressure at the edge of blade. This pressure is used to add pressure loading in static analysis to calculate its impact on the structural integrity of blades.
- It is observed that 20 number of blades produced maximum torque followed by static analysis and dynamic analysis for safer design.

B. STATIC ANALYSIS

The displacements, stresses, strains, and forces in the structures or components are determined by a static structural analysis to be caused by loads that did not cause significant inertia and damping effects. Steady loading and response conditions are assumed such as, the loads and the structure's response are assumed to vary slowly with respect to time. A static structural load can be performed using the ANSYS solver. The types of loading that can be applied in a static analysis include:

- Externally applied forces and pressures
- Steady-state inertial forces (such as gravity or rotational velocity)
- Imposed (nonzero) displacements
- Temperatures (for thermal strain)

Geometry Modelling

CAD Modelling of any project is one of the most time-consuming processes one cannot shoot directly from the form sketches to Finite Element Model. CAD (Geometry) modelling is the base of any project. Finite Element software will consider shapes, whatever is made in CAD model. CAD modelling of the Blade design is performed by using Ansys Design Modeller software.

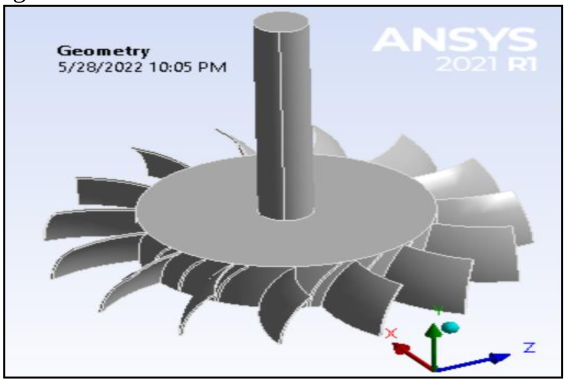

Fig. 17. FEA geometry (20 number of blades)

Material Properties: Mild Steel material is considered for blades.

Table II. Material Strength (MPA)

Material	*Yield Strength*	*Ultimate Tensile Strength*	*Ultimate Compressive Stress*
Steel	250	250	460

Meshing

It is convenient to select the free mesh because it has sharp curves, so that shape of the object will not alter. To mesh the plate the element type must be decided first. SOLID187 was used for meshing plates.

SOLID187 Element Description:

SOLID187 element was a higher order 3-D, 10-node element. SOLID187 has a quadratic displacement behaviour and is well suited to modelling irregular meshes (such as those produced from various CAD/CAM systems). The element is defined by 10 nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities.

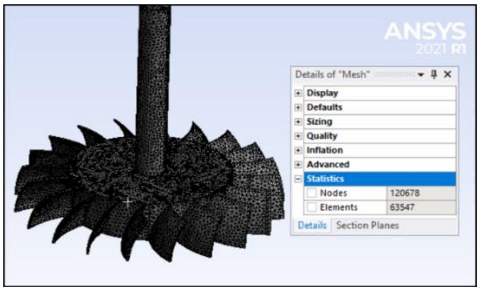

Fig. 18. Meshing of Static Model (micro gravitational Hydro turbine with 20 number of blades)

Loads and Boundary Conditions:

Shaft faces were constrained with fixed support. The pressure developed due to water impact from CFD analysis was applied. Rotation velocity of 1000 RPM was applied based on Micro gravitational hydro-turbine speed range of 600-1000RPM.

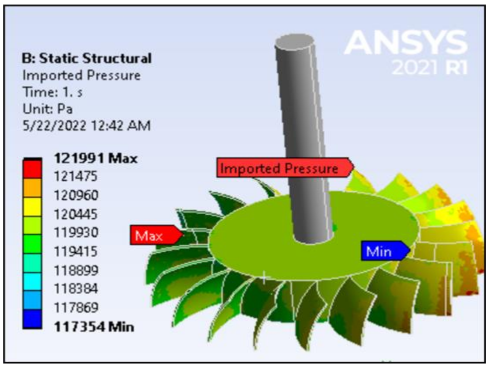

Fig. 19. Imported Pressure due to water impact from CFD analysis (20 Number of Blades)

Deformation Plot

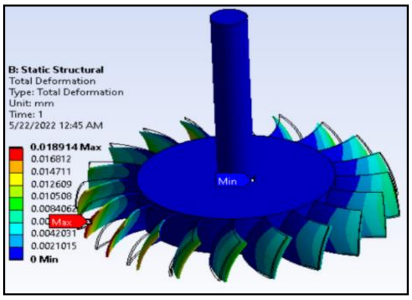

Fig. 20. Deformation plot for the pressure loading

It was observed that in case of micro gravitational hydro-turbine with 20 number of blades experiences less deformation. It is found that the tendency of the deformation is more at the edges of the blades.

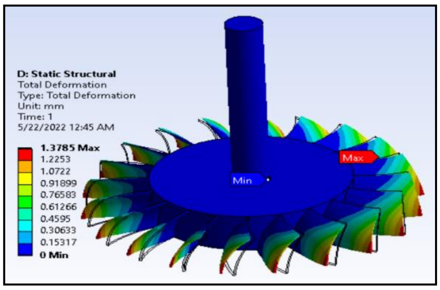

Fig. 21. Deformation plot for the Rotation loading

The variation of the deformation is due to the rotation of the shaft which is negligible in other cases. It has been observed that there is a significant deformation due to revolution as compared to the deformation due to water impact on the blades.

Von Mises Plot

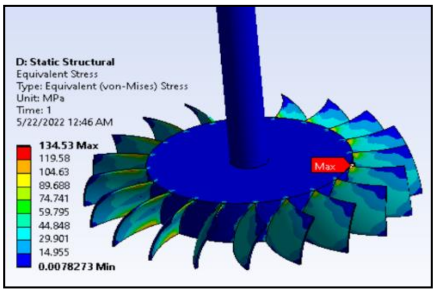

Fig. 22. Von Mises Stress Plot for the Pressure Loading

The von mises stress is 40 percent more in case of turbine with 10 number of blades as compared to turbine with 20 number of blades. Von mises stress is more on the root of the blades as compaed to the edges.

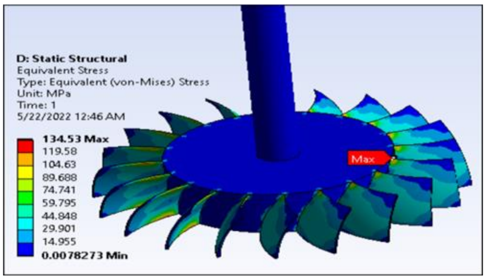

Fig. 23. Von Mises Stress Plot for the Rotation Loading

It was observed that the stress due to the rotational velocity is 60 times more as compared to the stress due to pressure loading, which is very significant pressure rise. So, the turbine should be designed for the rotational speed stress.

Table III. CFD Pressure and Rotational Velocity Von Misses Stress (MPa)

No Of Blades	*Von Misses Stress (MPa)*	
	CFD Pressure Loading	*Rotational velocity*
20	2.2405	134.53

Von misses stress with both pressure and rotation velocity loading is observed to be lower than material with yield strength of 250 MPa. Hence, current design is safe.

C. Dynamic Analysis (Modal Analysis)

The goal of modal analysis in structural mechanics is to determine the natural mode shapes and frequencies of an object or structure during free vibration.

The specific speed of Kaplan turbine ranges from 600 to 1000 rpm. It is a low head axial flow turbine. From the Table IV, it can be concluded that the Kaplan turbine has the maximum specific speed. Considering worst case of 1000 RPM, the dynamic response of turbine blade is assessed to check the resonance condition. The force frequency of the current engine considering first critical order and 1000RPM is 17Hz.

$$Force\ Frequency\ = Crtical\ order \times \frac{RPM}{60} \times 1.1(10\%\ higher\ for\ safer\ side)$$

$$Force\ Frequency = 1 \times \frac{1000}{60} \times 1.1 = 16.66 \sim 17Hz$$

Table IV. Specific Speed Table

Flow	*Enrgy*	*Head*	*Specific speed*	*Example*
Tangential	Impulse	High (300 m & above)	Low (0-60 RPM)	Pelton wheel turbine
Radial	Reaction	Medium (30 m to 300 m)	Medium (60-300 RPM)	Francis turbine
Axial	Reaction	Low (less than 30 m)	High (300-600 RPM)	Propeller turbine
			(600-1000 RPM)	Kaplan turbine

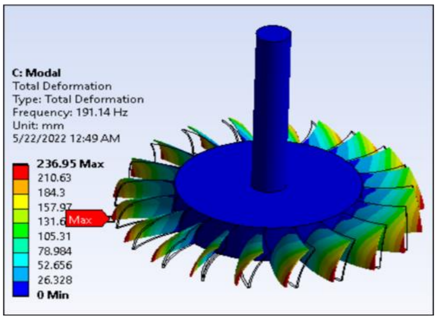

Fig. 24. 1st Natural frequency

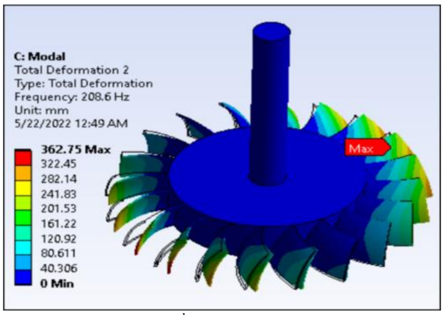

Fig. 25 2nd Natural Frequency

Table V Natural frequency

No Of Blade	*1st Natural Frequency (Hz)*	*2nd Natural Frequency (Hz)*
20	*191.14*	*208.6*

First natural frequency of blade is 191 Hz which is higher than excitation frequency of 17 Hz and hence the resonance conditions will never be achieved.

III. CONCLUSIONS

- The output torque observed for turbine with 20 number of blades is 6565 N.m., which is 38.43 % and 42.63 % more compared to turbine with 15 and 10 number of blades.
- Pressure distribution from CFD analysis shows maximum pressure of 0.12 MPa for turbine with 20 number of blades. The distribution of pressure shows maximum pressure at the edge of blade.
- Current design is safe because, von misses stress with both pressure and rotation velocity loading are

lower as compared to material with yield strength of 250 MPa.

- As per the dynamic analysis excitation frequency of blade is 17 Hz, which is very less as compared to first natural frequency of blade, 191 Hz. Hence resonance condition will never occur. Hence based on CFD, FEA (Static and Dynamic) Analysis, it is concluded that the micro gravitational hydro-turbine with 20 number of blades design is best compared to turbine with 15 and 10 number of blades design.

REFERENCES

[1] Marian G-M, Sajin T and Azzouz “A Applied Mechanics and Materials”, 371 601-605, 2013.

[2] Dhakal S, Timilsina A B, Dhakal R, Fuyal D, Bajracharya T R, Pandit H P, Amatya N and Nakarmi A M “Renewable and Sustainable Energy Reviews”, 48 662-669, 2015.

[3] Dhakal S, Nakarmi S, Pun P, Thapa A B and Bajracharya T R Journal of the Institute of Engineering 10 140-148, 2014.

[4] Power C, McNabola A and Coughlan P “Journal of Clean Energy Technologies”, 4 112-119, 2016.

[5] Pimnapat L, Patib T and Bhumkittipichc K Energy Procedia 34 235-242, 2013.

[6] Zotlöeterer F. Energy Systems [updated 09 August 2014; cited 12th September 2016]. Available from: http://www.zotloeterer.com/welcome/gravitation-water-vortex-power-plants/zotloe terer-turbine/

[7] Systems P E. PESWiki [updated March 2014; cited 13th September 2016]. Available from: http://www.pureenergysystems.com/NEC/conferences/2008/EnvironmentalHallofFame-Chicago/displays/Zotloterer,pdf