Design, Construction and Simulation of Tesla Turbine

SMG Akele[1], M. Ejededawe[2], G. A.Udoekong[3], A. Uwadiae[4], M. E. Oviawe[5] and B. O. Ayewa[6]
[1]Lecturer, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA
[2]Student, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA
[3]Student, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA
[4]Student, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA
[5]Student, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA
[6]Student, Mechanical Engineering Department, Auchi Polytechnic, Auchi, NIGERIA

[1]Corresponding Author: smg_sam2009@yahoo.com

ABSTRACT

Investigations of laminar fluid flow between two moving or stationary plates, and two rotating discs, over the years were geared toward how to increase Tesla-based turbine efficiency. Therefore, this research entails the construction, design and simulation of a Tesla turbine in order to investigate the potential of Tesla turbine for energy generation. Method of solution entails the design and construction of a physical model Tesla turbine from locally sourced materials. The physical model geometry and design parameters were then used to conduct numerical simulation. Performance evaluation was then carried on the physical model and the simulation model. The result showed that voltage, current and power all increase with increase in rev. per minute. The result obtained indicates that for higher power generation, a Tesla turbine design with higher revolution per minute capability will be required. Turbine model simulation showed that radial velocity vector to be concentrated at the discs periphery and outlet.

The research results are good references for design of larger Tesla turbine for community use.

Keywords**--** Physical Model, CFD Simulation, Radial Velocity, Static Pressure, Swirling Flow

I. INTRODUCTION

Tesla turbine was first patented by Nikola Tesla in 1913. Tesla turbine consists of a set of smooth closely-spaced discs that are arranged parallel to each other and rigidly fixed along a shaft. Tesla turbine operates with fluid entering the space between any two discs and swirling inwards towards the center of the discs. Around the shaft, at the discs center, are small holes which act as outlet for the fluid to exit. In Tesla turbine flow field, the fluid and discs interfaces are known to be governed by viscous and adhesion forces. (Sengupta and Guha, 2012). Tesla turbines are widely used for power generation (Zahid et al., 2016), as micro-pump, and in vehicle technology. In most previous studies of Tesla turbine, it is easily seen that each study used different dimension for discs geometry, discs space, or the number of discs used. This may be due to the fact that they few on the factors on which Tesla turbine efficiency depends. Gupta and Kodali (2013) presented previous research modeling, simulation, and experimental results on Tesla turbine over the last forty years. The work revealed that the performance of Tesla machines is influenced by a number of parameters such as width of disks, number of disks used, discs gap, inlet jet angle and so on. Kumhar and Dubey (2017) performed numerical simulation on several bladeless turbine models for a range of design parameters. Results of the investigations showed that the efficiency of the bladeless turbine depends on parameters such as pressure, temperature, velocity, rotational speed of the rotor, number of discs, disc diameter, disc gap, disc surface finish, and the number and arrangement of the inlet nozzles. The calculated flow efficiencies of the investigated bladeless turbines compared competitively with conventional small bladed turbines. Akpobi and Akele (2016) carried out 2D finite element method analysis to predict velocity components and pressure distributions. The results showed radial velocity to increase from 0 at discs walls to maximum of 260 at domain centerline and pressure increasing from 0 at the domain centerline inlet to 32.00 at the disc outlet. Eshita (2014) compared flow fields using standard k-epsilon and RNG k-epsilon turbulence models as well as comparing experimental data with numerical solution for sink flow. It was observed that the numerical solution showed good agreement with the experimental data. However, for standard k-epsilon model gave poor predictions for swirling and rotating flows. Tao (2014) used direct numerical simulations to investigate the Open Von Kármán swirling flow. Monotonic convergence was achieved for average pressure at disk periphery with small grid uncertainty at 3.5%. This study reveals strong three-dimensional flow structures, which undermines the use of axisymmetric model with a two-dimensional grid to approximate the flow field in most previous studies of similar geometry and flow conditions. The maximum pressure was observed near the wall chamber and minimum was near the axial suction. Akpobi and Akele (2019) performed numerical simulation to investigate axisymmetric swirling flow between two parallel co-rotating discs in a Tesla turbine. Results obtained showed that the dimensional and dimensionless radial velocity distribution both increased proportionately from the disc surface toward the centre.

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)

https://doi.org/10.31033/ijemr.11.1.13

For turbulent flow, the swirl velocity profile was observed to be smoother than for laminar flow around the discs centre. Lampart and Jędrzejewski (2011) analyzed Tesla bladeless turbine for a co-generating micro-power plant of 20kW. Numerical calculations of the flow in several Tesla turbine models were performed for a range of design parameters. The results of analysis showed that for 100mm model efficiency of 30% was obtained. The turbine was observed to improve when 300mm model was investigated. However, efficiency of 50% was obtained by using a model with two nozzles, at inlet angle of 10^o and 9000rpm, for 100mm model.A prototype of Tesla disk turbine was constructed and different experiments were performed with various pressure ranges of an incompressible fluid (water). (Zahid et al., 2016)

II. NUMERICAL METHOD

The partial differential equations governing fluid flows of this study are the known non-linear continuity and Navier-Stokes equations that completely describe the flow of Newtonian fluids. Finite element method (FEM), finite volume method (FVM) and finite difference method (FDM) and few of the numerical technique for solving partial differential equations (PDEs) with the ability to discretized and approximate the governing equations and physical geometry model of a physical phenomenon. These coupled and nonlinear continuity and Navier-Stokes equations solved using finite volume numerical method CFD software by applying relevant assumptions and appropriate boundary conditions. (Akpobi and Akele, 2016).

III. METHODOLOGY

The turbine configuration is basically a swirling inflow configuration with the fluid entering at the disc periphery, and swirling inwards towards the discs centerline (outlet). The fluid flow domain geometry is modeled in 3D Cartesian coordinates with origin along shaft axis.

3.1 Physical Model Design

The physical design involved the purchase of cylindrical plastic for housing, plexiglass for body panels, 12mm by 80mm aluminum rod for shaft, 25 plastic discs, two bearings, 11 bolts and nuts, washers, pvc pipe, copper pipe, electric generator, super glue, silicone glue shown in Fig. 1.

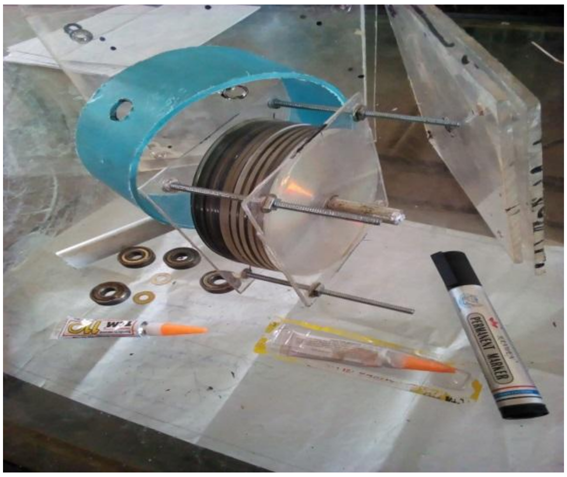


Figure 1: Physical model parts

The plastic and plexiglass were cut to design sizes; the aluminum rod was machined according to design requirement to carry the discs and the bearings at both ends; holes were drilled around the discs, with the disc center hole as fluid outlet.

Table 1: Physical model parameters

Parameters	Symbols	model value
Outer radius	r_o	120mm
Inner radius	r_i	30mm
Disc gap	b	1mm
Disc thickness	t	2mm
Casing (stator)inner diameter	D_i	140mm
Nozzle angle	α	60^o
Nozzle length	l_n	40mm
Nozzle diameter	d_n	20mm
No of spacers	S	32units

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13

No of disc	C	31units
Angular velocity	ω (average)	1800rpm
Height	h	500mm
Width	w	500mm
Length	l	500mm
Mass	m	12kg

3.2 Simulation Model Design

The simulation design is modeled in 3D Cartesian coordinates with origin along shaft axis, Fig. 2.

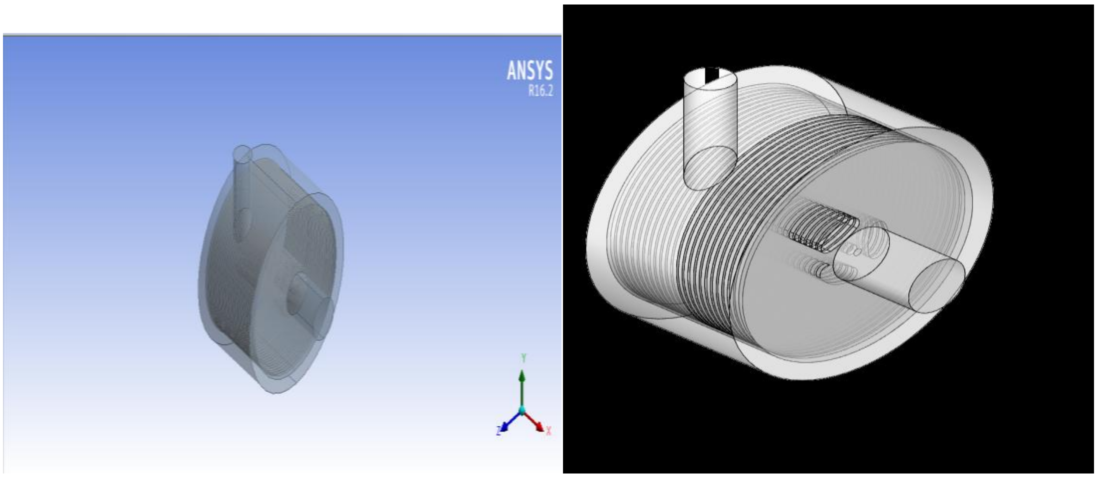


Figure 2: 3D model geometry

(I) Domain Discretization

The fluid flow domain was discretized using quadrilateral elements. The domain is subdivided into quadrilateral elements mesh along the x- and y-axes respectively, Fig. 3.

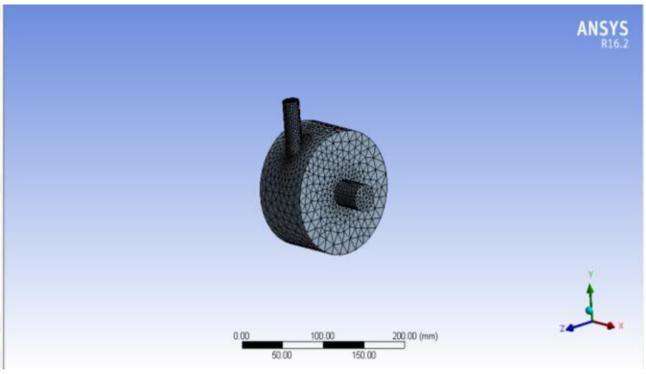


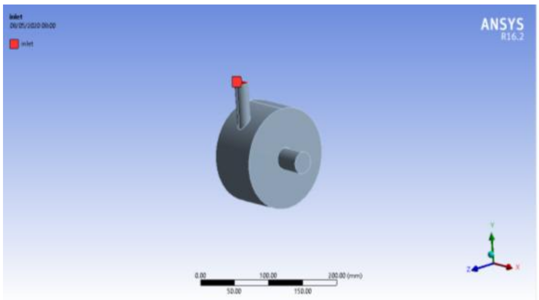
Figure 3: 3D model discretization

Mesh: Nodes, 64884; elements, 244179

(II) Boundaries Geometry and Boundary Conditions

Considering the geometry of Fig.4, the fluid entering at the inlet, comes in contact with both discs surface. Thus, velocity gradient is set up in the boundary layer with no-slip boundary condition between disc and fluid interface with the viscous drag in the flow domain also setting up a swirling radial inward flow in the fluid.

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13



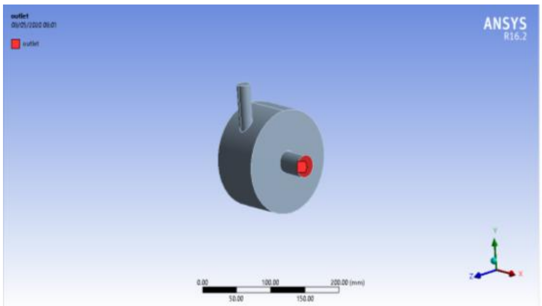


Figure 4: Boundary conditions

(III) Model Simulation

This CFD design model was simulated by using Fluent Fluid Flow Design Modeler bench to generate the model quadrilateral meshes; while the Fluent Solver was used to analyzed the flow to obtain results. The 3D analysis was set up on 3D space, pressure-based, absolute velocity formulation, steady and axisymmetric swirling flow with air as the working medium.

Boundary conditions were set as: inlet - 0.001kg/s; outlet – gauge pressure 0 pascal; moving wall, no-slip; rotational velocity – 1800 rpm.

Table 2: Simulation model parameters

Parameters		Symbols	Simulation value
Outer radius		r_o	120mm
Inner radius		r_i	30mm
Disc gap		b	1mm
Disc thickness		t	2mm
Casing inner diameter		D_i	140mm
Boundary conditions:	inlet	m_i	0.001kg/s
	outlet	p_g	0pascal
Turbulence model		-	K-epsilon
Mass flow rate		m_o	7.2kg/s
Nozzle angle		α	60^o
Nozzle length		l_n	40mm
Nozzle diameter		d_n	20mm
No of spacers		S	32units
No of disc		C	31units
Angular velocity		ω	1800rpm
Height		h	140mm
Width		w	140mm
Length		l	140mm

IV. RESULTS AND DISCUSSION

Table 3: Physical model output result

Inlet pressure(pa)	Angular speed(rpm)	Voltage (Volts)	Current (amps)	Electric Power (Watts)	Efficiency (%)
190	1780	10	1.27	12.70	62%
190	1800	11	1.30	14.30	
190	1850	12	1.35	16.20	
190	1750	10	1.20	12.00	
190	1820	11	1.33	14.63	

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13

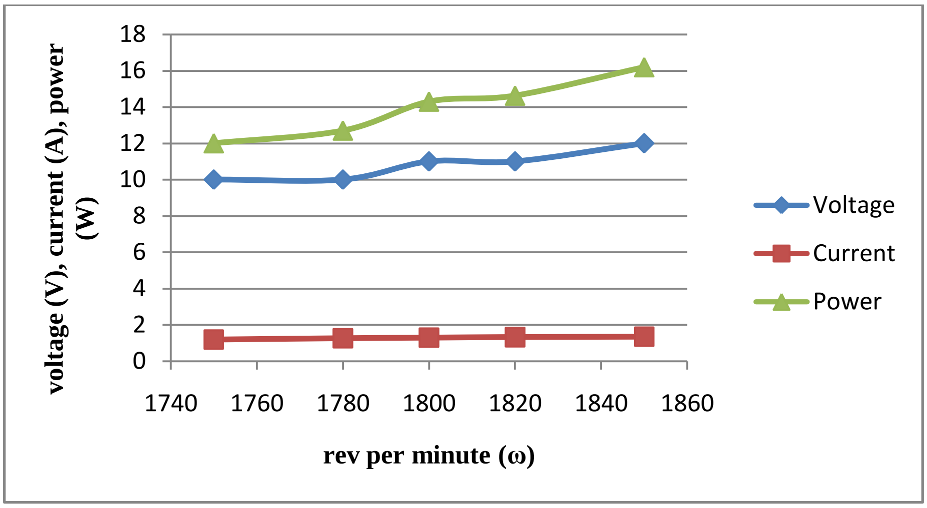


Figure 5: Plot of voltage, current, power against rev per minute

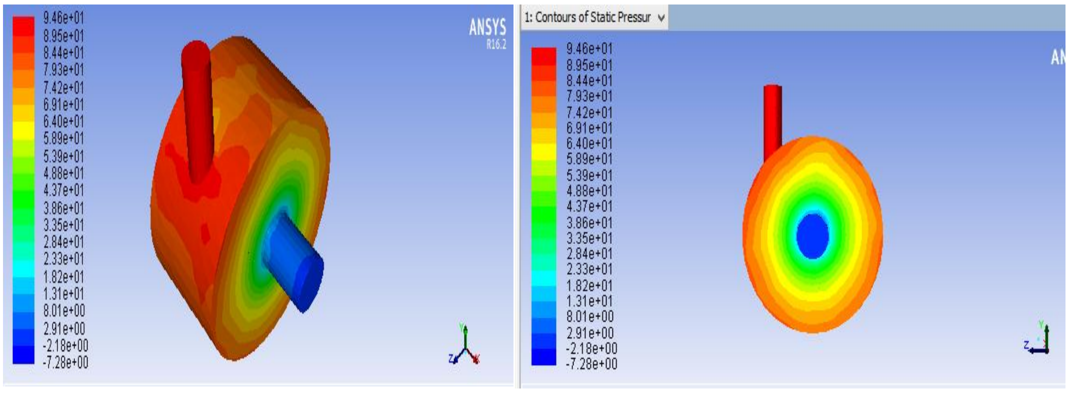


Figure 6: Contour of static pressure

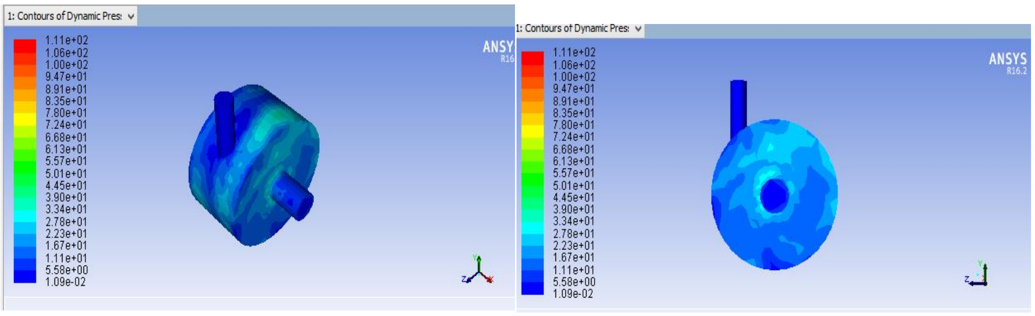


Figure 7: Contour of dynamic pressure

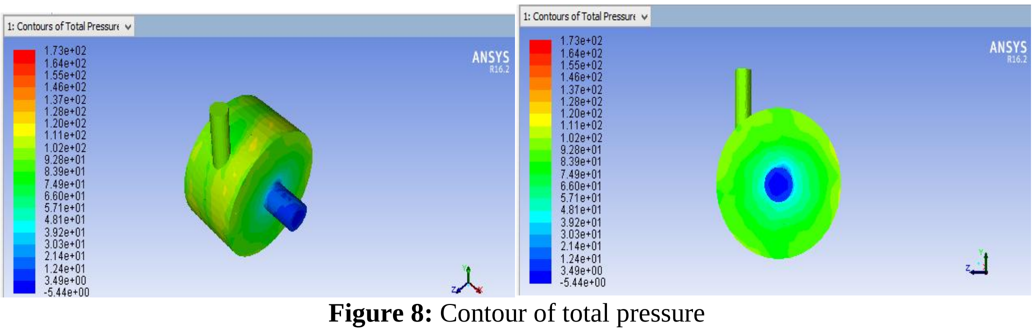


Figure 8: Contour of total pressure

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13

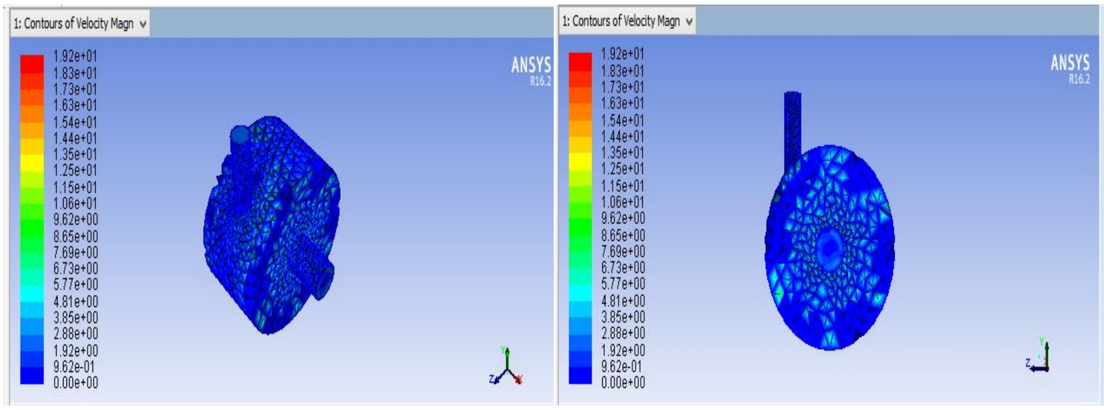


Figure 9: Contour of velocity magnitude

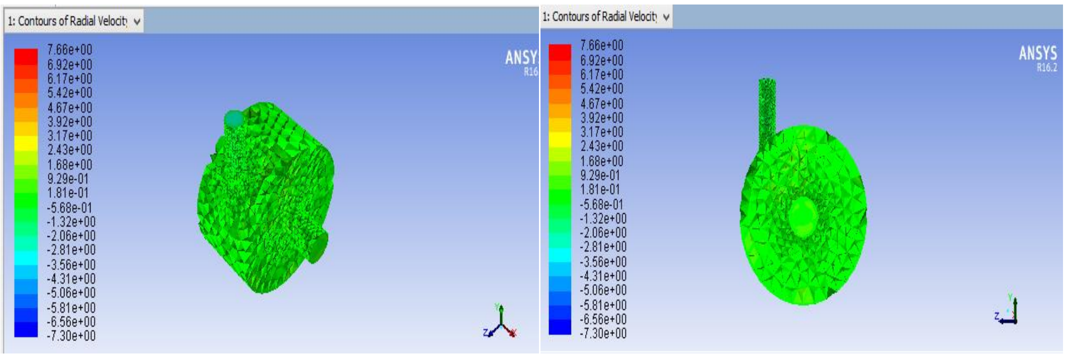


Figure 10: Contour of radial velocity

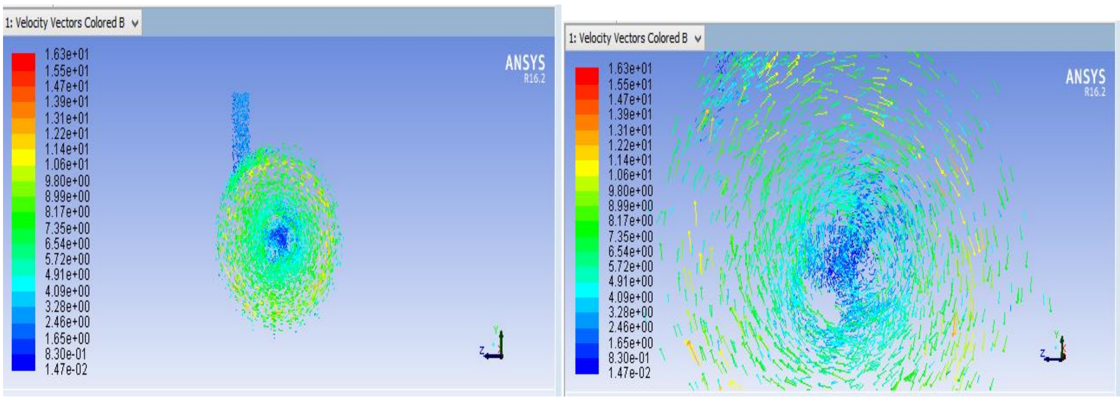


Figure 11: Velocity vector

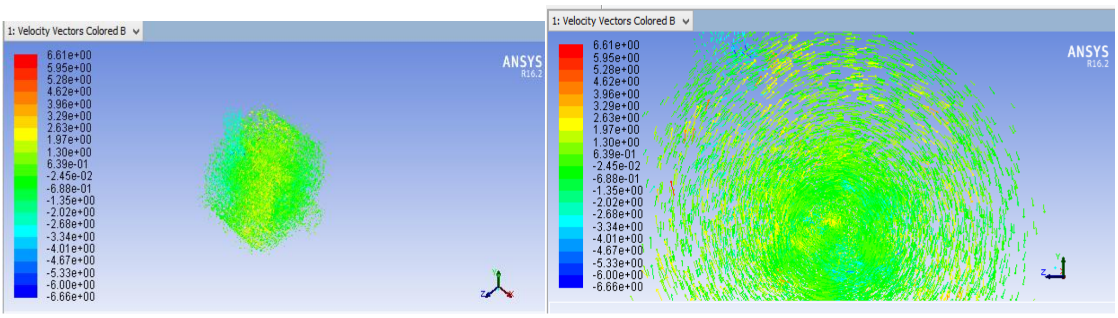


Figure 12: Radial velocity vector

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13

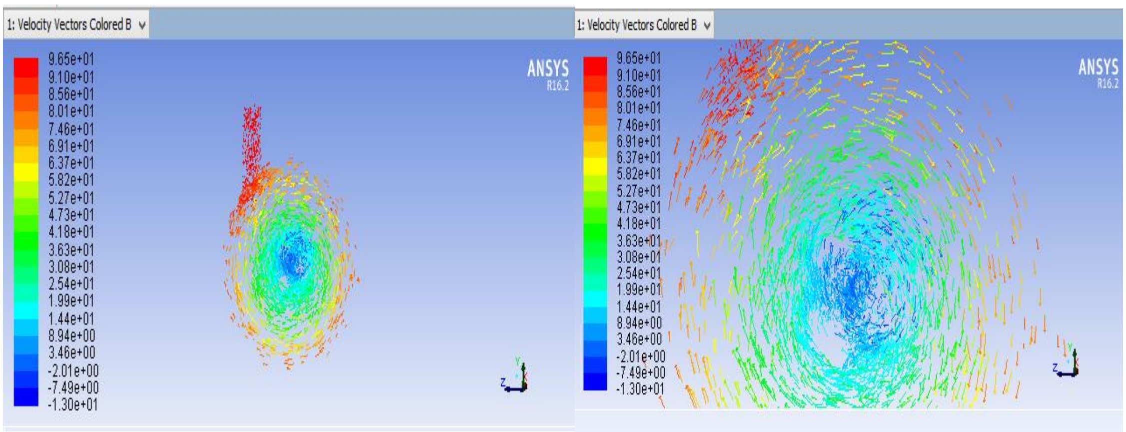


Figure 13: Pressure vector

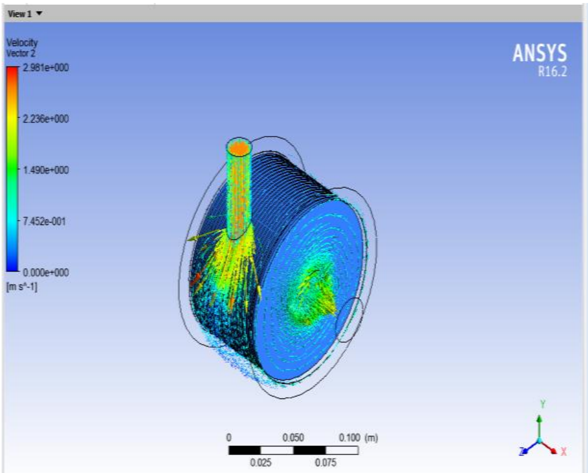


Figure 14: 3D view of model

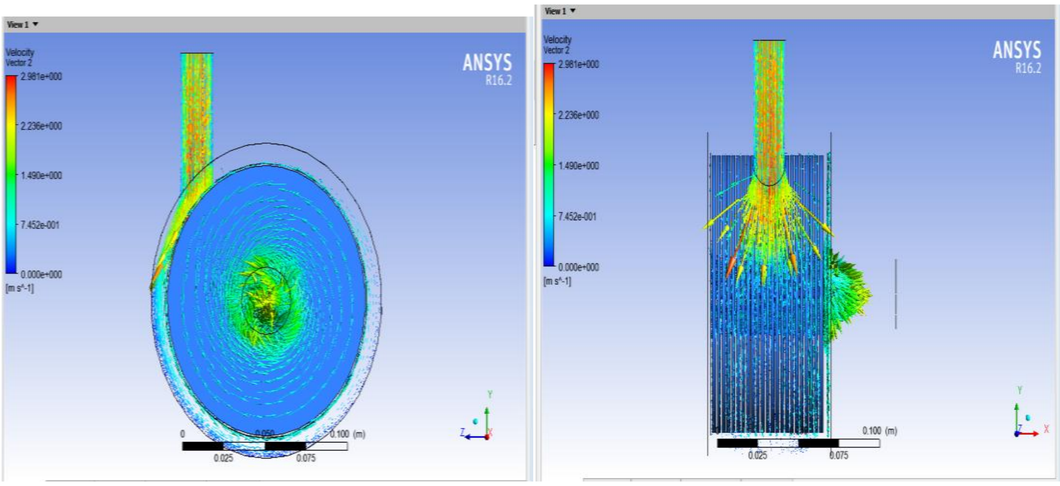


Figure 15: Velocity vector view 1

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)
https://doi.org/10.31033/ijemr.11.1.13

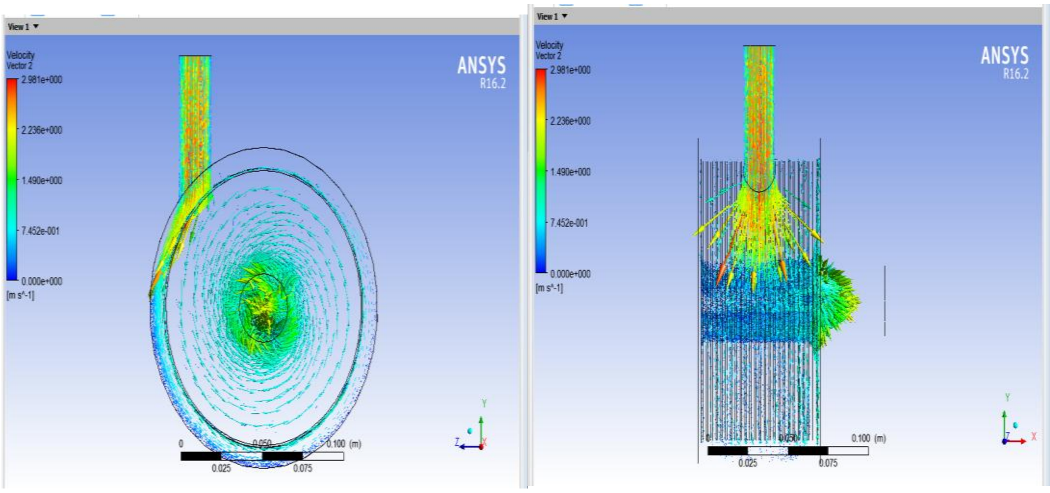


Figure 16: Velocity vector view 2

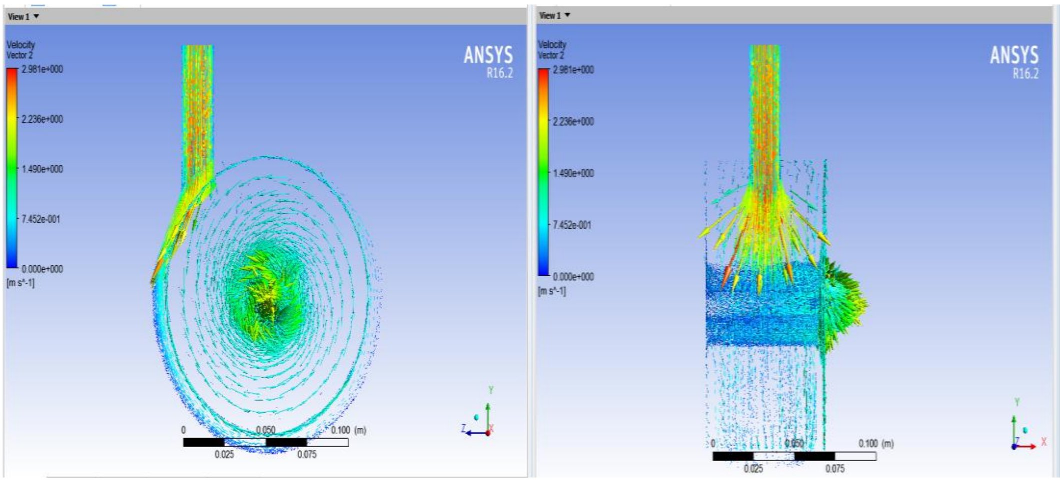


Figure 17: Velocity vector view 3

Table 3 shows the results for five different readings obtained when the physical model was operated to evaluate its performance.

Fig.5 shows that voltage, current and power all increase with increase in rev. per minute. This result indicates that for higher power generation, a Tesla turbine design with higher revolution per minute capability will be required. Fig. 6 shows 3D view for contour of static pressure, which is seen to have minimum value of -7.276Pa at the outlet and 94.605Pa near the turbine inlet. Fig 6 reveals how pressure drops from the turbine inlet t0owrads the outlet. Fig. 7 shows the dynamic pressure of the simulated turbine occurring in the inner part of the turbine with minimum pressure value of 0.011Pa and maximum of 111.355Pa. Fig. 8 indicates minimum and maximum total pressure of 5.435Pa and 173.138Pa respectively.

Fig. 9 shows contour of velocity magnitude with minimum velocity of 0.000m/s occurring close to the turbine inlet and maximum velocity of 19.232m/s occurring near outlet the outlet. The strength of the contour is seen to be very high where fluid is far away from the disc surface. Fig 10 shows contour of radial velocity with minimum and maximum values of -7.304m/s and 7.665m/s respectively. Fig. 11 depicts turbine vector of velocity (magnified view) with minimum velocity of 0.015m/s and maximum velocity of 16.325m/s; the velocity vector is seen to be concentrated at the discs periphery and outlet. Fig. 12 shows turbine simulation radial velocity vector(magnified view) with minimum value of -6.661m/s and maximum value of 6.612m/s; the radial velocity vector is seen to be concentrated at the discs periphery and outlet. Fig. 13 shows pressure vector of flow domain (magnified view) with minimum and maximum values of -12.960Pa and 96.519Pa. Fig 11, Fig. 12 and Fig 13 all show the swirling effect of the simulated turbine.

For post-processing results, Fig. 14 shows a 3D view of the fluid entering and making contact with the discs, at the disc periphery, and swirling inwards towards the discs outlet at the center of the discs. Fig. 15 through Fig.17 show side and end views of the turbine swirling

e-ISSN: 2250-0758 | p-ISSN: 2394-6962
Volume-11, Issue-1 (February 2021)

https://doi.org/10.31033/ijemr.11.1.13

motion having post-processing minimum and maximum velocity vectors of 0.00m/s and 2.981m/s respectively.

V. CONCLUSION

A physical Tesla turbine was designed and CFD simulation performed based on the parameters. Model results and simulation results both exhibited approximately the result that the higher the rpm, the higher power electric power generated. With the 3D simulation, tangential velocities alongside radial velocities are able to be predicted.

REFERENCES

[1] Akpobi, J. A. & Akele, SMG. (2016). Development of 2D models for velocities and pressure distribution in viscous flow between two parallel co-rotating discs. *Transactions of the Nigerian Association of Mathematical Physics, 2*, 233–290.

[2] Akele, SMG. & Akpobi, J. A. (2019). Numerical simulation of flow between two parallel co-rotating discs. *International Journal of Engineering and Management Research, 9*(6), 13–22.

[3] Eshita, I. J. (2014). Turbulence modeling of source and sink flows. *World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, 8*(5), 821–826.

[4] Ho-Yan, B.P. (2011). Tesla turbine for pico hydro applications. *Guelph Engineering Journal, 4*, 1-8.

[5] Gupta, H. & Kodali, S. P. (2013). Design and operation of Tesla turbo-machine – A state of the art view. *International Journal of Advanced Transport Phenomena, 2*(1), 7-14.

[6] Sengupta, S. & Guha, A. (2012). A theory of Tesla disc turbines. *Journal of Power and Energy, 266*(5), 650-663.

[7] Lampart, P. & Jędrzejewski, L. (2011). Investigations of aerodynamics of tesla bladeless microturbines. *Journal of Theoretical and Applied Mechanics, 49*(2), 477- 499.

[8] Tao, X. (2014). Direct numerical simulation of open von kármán swirling flow. *Journal of Hydrodynamics, 26*(2), 165–177.

[9] Kumhar, V. & Dubey, A. (2017). Performance analysis of bladeless turbine using computational fluid dynamics. International Journal of Mechanical and Production Engineering, 5(11), 81–83.

[10] Zahid, I., Qadir, A., Farooq, M., Zaheer, M. A., Qamar, A., & Zeeshan, H. M. A. (2016). Design and analysis of prototype tesla turbine for power generation applications. *Technical Journal, University of Engineering and Technology (UET) Taxila, Pakistan. 21*(II), 1–5.