Influence of Outlets Port Design on The Tesla Turbine Performance

Sufi Halim¹, Md Tasyrif Abdul Rahman¹, Anas Abdul Rahman¹, Muhammad Adi Hilmi Adnan¹, Muhammad Hafizi Azuzin¹ and Gisrina Elin Suhri¹ and Mohd Fathullah Ghazli¹²

¹Faculty of Mechanical Engineering, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia.
²Green Design and Manufacture Research Group, Geopolymer & Green Technology, Centre of Excellence (CEGeoGTech), Universiti Malaysia Perlis (UniMAP), Perlis, Malaysia.

Email: tasyrif@unimap.edu.my

Abstract. The boundary layer turbine known as Tesla Turbine invented long ago but has failed to be commercialized and replaced by bladed turbines. In this paper, two new techniques for improving the turbine have been proposed. A test model of the proposed boundary layer turbine has been fabricated made and tested under different conditions. The design process includes producing a virtual design and simulation of the turbine using computer software. The proposed designs were fabricated and then tested to analyse results such as speed produced, power produced, and the turbine efficiency. From this study, the proposed turbine designs manage to achieve 18% and 69% efficiency. The findings of this study will serve as a reference for future studies in the generation of power through an alternative powered driven turbine.

1. Introduction
Turbines are a machine that produces energy from flowing fluids [1]. The geometry of the turbine is such that the fluids applied torque on the rotor in the direction of the rotation. The energy is transmitted from the rotor to the shaft and making it available to drive generators or other devices. The turbine is widely used in power generation and uses elements such as water, wind, and tidal wave as working fluids. These elements have kinetic and potential energies to propel the turbine sufficiently to allow the generator attached to produce power [2]. Some turbine uses compressible gasses as working fluid and one is widely known as Tesla turbine. Work is produced when the working fluid is introduced tangentially at the outer edge of the plates and exits the turbine at exhaust ports located near the centre of the discs [3]. The wall shear stress will happen at the surface of the turbine disc, and a recent study had indicated that the shear stress is proportional to the velocity gradient [4].

Tesla turbine was first patented in 1913, but did not find any commercial success, mainly due to its characteristic of lower efficiency than conventional turbines [5]. Despite that, Tesla turbine also offered several advantages like the simplicity of construction and higher temperature application [5]. The cost would be much lower compared to the cost of manufacturing an impinging turbine. The Tesla turbine also has a greater resistance towards high temperatures [6].

Studies were carried out to define the analytical model of the flow and improve turbine efficiency. Various parameters and characteristics of the turbine are being modified to test its performance. So, in...
this study, we will design a Tesla turbine by determining the variables and design considerations to construct the turbine disc and to analyse the power produced and efficiency from the design. It is mainly purposed to identify whether this turbine will have good power optimization and efficiency. This study process will start by making a virtual design and analysis before fabricating the prototype. Then, it will proceed by running the actual experiment on the prototype.

2. Methodology

The research methodology mainly consists of design establishing, virtual analysis, calculations based on theory, and verified experimental procedure using the turbine torque test and turbine test to get the data values. An appropriate design study for the disc mechanism was calculated to suit the turbine concept.

In this research, the methodology was divided into three sections where each section focuses on specific analysis. In the first section, the designs of the disc were generated using Computer-Aided Engineering Software (CAE). Then the designs are run for the virtual analysis. Finally, the design is fabricated and run for the physical experiment. All data and outcomes for each section were presented in the results section.

Initially, we have designed a disc with circular exhaust ports, and it is designed with 2 degrees offset to create a cascading angle when the discs stacked together. Since the design is developed on the Solidworks software, the design undergoes virtual analysis under the same software. The design then fabricated and run for the physical experiment. However, the data collected from the experimental results come out to be unsteady. This happens because the shaft was improperly built and mounted on the turbine. It has caused a frictional problem between the shaft and its mounting as the shaft vibrated heavily at a speed higher. Thus, we came out again with another disc design. This second design was also being virtually and experimentally tested. Both design specifications and details working as explained further through this paper.

2.1 Design Generating

The design of the disc mainly focused on its exhaust port. Due to the fluid’s spiral movement, it will flow towards the centre of the disc and exits. Thus, the exhaust ports are purposely designed to let the fluid flows more streamlined. As mentioned before, the first disc was designed with circular exhaust ports with 2 degrees offset, while for the second design, the disc was designed with an oblong shape exhaust port. The details of the first and seconds design areas figure 1 and table 1.

![Design 1](a) Design 1
![Design 2](b) Design 2

**Figure 1.** Disc design.
Table 1. Disc design specification.

| Parameters          | Design 1            | Design 2            |
|---------------------|---------------------|---------------------|
| Number of discs     | 4 discs             | 4 discs             |
| Disc diameter       | 120mm               | 126mm               |
| Disc thickness      | 1.5mm               | 1.5mm               |
| Number of outlets   | 10                  | 4                   |
| Outlet diameter     | 10mm                | 10mm                |
| Outlet shape        | Oblong              | Circular with 2° offset |
| Gap spacing         | 1.5mm fixed         | 1.5mm fixed         |

2.2 Virtual Analysis
Design 1 was generated by using Solidworks software. Since the software provides the fluid flow analysis simulation which integrated inside it thus the virtual analysis was performed directly via the Solidworks software. From the analysis, a variety of results can be directly generated such as pressure, temperature, flow rate, mass, and other variables needed for this research. To run the simulation, the boundary condition was set up in table 2 and figure 2.

![Boundary layer setup](image)

**Figure 2.** Boundary layer setup.
Table 2. Disc design specification.

| Boundary conditions                |
|-----------------------------------|
| Velocity inlet 20.26m/s           |
| Pressure outlet Environmental     |
| Fluid Air Incompressible          |
| Analysis type Exclude cavities    |
| Calculations Pressure inlet Pa   |
| Pressure outlet Pa                |
| Pressure drop Pa                  |

Design 2 analysis was performed via ANSYS software using fluid flow analysis. In generating the analytical results, the calculation was carried out using Computational Fluid Dynamic (CFD) approach. To use this, firstly the geometry of the design was set to the cavity inside the turbine as shown in figure 3. By referring to the geometry set, the meshing process will take place and the simulation by obeying the boundary condition such as table 3.

![Figure 3. Boundary condition for Design 2.](image)

Table 3. Parameters definition in fluid flow analysis,

| Cell zone condition               |
|-----------------------------------|
| Angular velocity 1000RPM          |
| Material Air                       |
| Boundary condition Pressure inlet 1 Bar |
| Pressure outlet Normal to boundary|
| Solution Method Second order       |
| Calculations Time step size 0.0005s |
| Number of time step 100           |
| Max iteration time step 50        |
| Reporting interval 1              |
| Profile update interval 1          |
2.3 Experimental analysis

For the experimental analysis, the procedure was divided into two parts. The first part of the experiment is to observe the turbine speed while the other part of the experiment is to calculate the torque produced by the turbine via the central.

For the turbine speed analysis, generally, the compressor was firstly run with the air regulator set to a pressure. When the compressor is steady, the rotation speed is recorded. The procedure is continued by increasing the pressure with a certain value of increment until it reaches a pressure limit. Both Design 1 and Design 2 are having a different set of values for pressure regulation where Design 2 pressure regulations are lowered to avoid the same problem happen as Design 1. The pressure regulations data are presented in Table 4.

Table 4. Pressure regulation for turbine speed test.

| Pressure regulation | Design 1 | Design 2 |
|---------------------|----------|----------|
| Initial pressure    | 69 kPa   | 25 kPa   |
| Pressure increment   | 34.5 kPa | 25 kPa   |
| Pressure limits      | 551.5 kPa| 100 kPa  |

For the turbine torque analysis, like the previous test, both Design 1 and 2 have a different set of pressure regulations where Design 2 pressure values are being lowered. The pressure regulations are presented in Table 5. To run the analysis, the air compressor was run with an initial pressure and the arm grip was tightened at the shaft while the inlet valve was opened. The turbine rotates and allows the shaft and blade to start rotating too. As the movement is more stable, the arm grip was loosened a little bit to slow down the shaft. At this moment, the weight scale and tachometer value were recorded. Then the procedure is continued by loosening the arm grip. For Design 1 this step was repeated with a pressure increment of 34.5 kPa while for Design 2, the step was repeated until the weight scale drop to zero.

Table 5. Pressure regulation for turbine torque test.

| Pressure regulation | Design 1 | Design 2 |
|---------------------|----------|----------|
| Initial pressure    | 69 kPa   | 100 kPa  |
| Pressure increment   | 34.5 kPa | Unfixed  |
| Pressure limits      | 551.5 kPa| 200 kPa  |

All the data and results from this physical experiment are collected and be used in further calculation to find power and efficiency using formula from [8-10].

3 Results and Discussion

3.1 Virtual analysis

For the virtual analysis results, since both designs used different software to carry out the simulation, thus the results collected are in different characteristics and categories. This is due to each software having a different working system, and the requirement for defining the parameters to set the boundary layers, and a different way to represent the results.
3.1.1 Design
From figure 4, the relationship between the inlet velocity with the pressure drop can be seen. We can see that higher inlet velocity will lead to a higher pressure drop. Since the pressure drop is inversely proportional to the turbine efficiency, thus high-pressure drops will yield a low efficiency.

![Figure 4. Relationship between inlet velocity and pressure drop.](image)

Figure 4. Relationship between inlet velocity and pressure drop.

Figure 5 shows that the streamline result for Design 2. From the figure, we can see that there are circular lines around the disc and those lines represent the fluids flow on the disc. The more the spiral lines on the disc means that the environment is more packed and the interval between the lines is minimal. This shows that the fluids conserve the space maximumly and producing more shear stress due to high velocity.

![Figure 5. Streamline analysis result.](image)

Figure 5. Streamline analysis result.

Meanwhile, figure 6 shows the pressure contour results for Design 2. It shows that the pressure drops from the inlet to the exhaust port. These pressure contour result achieved is the normal behaviours of the turbine where the air pressure will drop as soon as the fluid flow exits from the inlet nozzle [11].

![Figure 6. Pressure contour result.](image)
3.2 Experiment

3.2.1 Effect of rotor speed on the turbine efficiency and turbine pressure

Figure 7 shows the effects of turbine speed to its efficiency. From the figure, both designs are showing a different pattern result. While the same experimental procedure was performed for both designs, different sets of properties been defined for each design such as the pressure regulator values while performing the analysis. Moreover, during the building and mounting of the turbine shaft, the turbine could not run with a higher velocity. This action has caused the RPM values for each design to differ from each other. For Design 1, the RPM values were in the range of 0 - 525 RPM meanwhile for Design 2 the RPM values are in the range of 600 - 3900 RPM. Not only for this figure, but this dissimilar RPM range also affected the rest of the experimental results.

From the experimental results, the highest efficiency achieved is 18.64% at the maximum rotor speed of 522.53 RPM for Design 1 and 69.81% at speed of 3070 RPM for Design 2. Both figures show that the efficiency increases as the RPM increase. This happens due to the increasing value of inlet pressure which can be seen in figure 8.
A higher inlet pressure will lead to a higher RPM. This is because when the rotor is not under load, the relative speeds between the fluid and the disc are minimal and the disc can move freely. As the disc rotates faster, the air starts to spiral around the disc towards the centre. When the angular velocity of the disc is greater, a tight spiral line is produced, and it will directly enter the disc centre, generating higher efficiency. The turbine efficiency starts to drop because the rotor speed had reached its maximum speed of 522 RPM for Design 1 and 3070 RPM for Design 2. For Design 1, the line is not smooth due to improper fabrication on the shaft of the turbine.

3.2.2 Relationship of rotor speed on torque and power

Figure 9 below shows the relationship between rotor speed and the torque produces. From the figure, it shows the maximum torque calculated for Design 2 is 0.013 Nm at 619 RPM and 0.057 Nm at 552.53 RPM for Design 1. The torque of Design 1 cannot exceed this maximum point as it reaches its maximum angular speed. While for Design 2, after reaching the maximum torque, we can observe that, the torque value drops as the RPM increase. This is because after a certain point the airflow cannot overcome the restriction of the disc thus lowering the torque produces [12].

From the graph, it can be observed that before reaching the maximum torque, the torque increases along with RPM. This happens when the fluid starts to develop its momentum due to the increasing value of the relative disc speed. The fluid will be acting as a solid body and drags the disc surface. The fluid will expand and increase the pressure perpendicular and radially to the disc on the axis of rotation. These positive behaviours are transmitting a stronger dragging force on the disc and increases the torque. Thus, this will also increase the power generated by the turbine [12] because power is the ability to apply torque in each amount of time. This explains the graph pattern as shown in figure 10.
4. Conclusion

Both Design 1 and 2 have a different set of data. With 500 RPM speed as the benchmark, the efficiency of the turbine for Design 1 is slightly lower than Design 2. Same goes for other parameters value, it seems that the results data for Design 2 are higher than Design 1. This distinct value of results is mainly due to different setup use for the virtual analysis. Not only that, in fabricating the Design 1, the shaft is built and mounted on the turbine improperly and causing a frictional problem between the shaft and its mounting as the shaft vibrated heavily at speed higher than 25 m/s. This has made the rotor shaft to slow down after some time, which will be resulting in unsteady measurement readings. Moreover, this distinct value may also be due to losses such as the expansion of the inlet and the sudden change in flow state, from radial to a direction parallel to the shaft at the exits.

Despite this error, the methods stated in this thesis had been followed carefully. The result of this experiment brought a promising turbine efficiency for the Tesla turbine. The Tesla turbine which used the cascading outlets on the turbine blades has achieved 18% efficiency. Whereas the Tesla turbine which uses the oblong outlets, has achieved 69.81% efficiency. Overall, this result seems to be in logical value. Although we are facing some limitations on conducting the study, however, this study is still beneficial as it uses virtual design and analysis software that saves cost and time during the design process thus lessen the overall cost for the development process. Nevertheless, the efficiency is far less than the practical conventional blade turbine. We realized that results can be improved by considering better equipment and properly fabricate the Tesla turbine. Further research and improvement are needed to optimize the design. Some suggestions are to include an equation to calculate the turbine safety factor, the disc spacing effect, disc thickness effect, and several discs in the turbine to produces a better turbine.

References

[1] Okiishi M, Rothmayer H 2009 Fundamentals of Fluid Mechanics (USA: John Wiley & Sons)
[2] Guha A, Sengupta S 2013 Eur. J. Mech. 37 112
[3] Çengel Y A 2006 Fluid Mechanics (USA: McGraw-Hill)
[4] Pandey R J, Pudasaini S, Neopane H P Int. J. of Sci. and Research Pub. 11 4
[5] Manfrida G, Pacini L, Talluri L 2017 Energy Procedia 129 1055
[6] Peshlakai A, Phelan P, and Trimble S 2014 IGARSS 1 1
[7] Rusin K 2017 Acta Innovations 22 58
[8] Holland K 2015 Appl. Sci. 1 89
[9] Heywood J B 1988 Internal combustion engine fundamentals (USA: McGraw-Hill)
[10] Çengel Y A, Boles M A 2016 Thermodynamics An Engineering Approach (USA: McGraw-Hill)
[11] Hassan H F, El-Shafie A and Karim O A 2012 Renew. Sust. Energ. Rev. 16 5707
[12] Zuber M, Ramesh A, Bansal D 2019 Adv. Res. Fluid Mech. Therm. Sci. 62 1 122