Abstract—This paper reports a wind-powered water pumping system implemented in rural side of Pakistan. The design methodology presented in this paper shall enhance the efficiency of the existing Savonius turbine’s performance by modifying it aerodynamically. Blades with different twist angles are designed in Solid Edge and analyzed using computational fluid dynamics (CFD) with ANSYS FLUENT software. Static and rotational analyses are performed to get optimized twist angle and results are highlighted. The performances of the turbine in both static and rotational analyses are compared.

Index Terms—Computational fluid dynamics, rotational, static, turbine.

1. Introduction

Generating power is of great importance in today’s world. Due to the pending exhaustion of fossil fuels, it is crucial to develop alternative energy sources. Wind energy is a source of renewable power which comes from air current flowing across the earth’s surface. Wind turbine is a device that converts wind’s kinetic energy into mechanical energy. If the mechanical energy is used to produce electricity, the device may be called a wind generator or if it is used to drive machinery, such as for grinding grain or pumping water, the device is called a windmill or a wind pump[1].

In Pakistan the province of Sindh and Punjab lead in the field of wind energy. At the end of March 2002 Pakistan had 1080 MW capacity wind farms, of which Sindh contributed 770 MW capacity. Punjab has 167 MW followed by NWFP, which has 88 MW installed wind farms. There are about a dozen wind pumps of various designs providing water for agriculture, irrigation, for shredding crops and domestic purposes, all scattered over the country[2]. The application of the system employed is to pump water for irrigation purpose and it comprises of a two bladed Savonius wind turbine. Currently, the system does not work to its required efficiency due to its wind turbine performance.

The objective of this paper is to enhance the performance by optimizing the Savonius’s rotor. Improvement to the aerodynamic characteristics of the rotor is achieved by changing the geometry. Computational fluid dynamics analysis is carried out to assess the improvements and the results are compared to select the optimum design.

Numerous investigations have been carried out in the past to study the performance characteristics of two and three bucket Savonius rotor. These investigations included wind tunnel tests, field experiments and numerical studies. Researches are carried out by analyzing the blades in computational software with varying geometrical parameters like overlap ratio, speed tip ratio[3]-[5].

2. Existing System

The system comprises of a two bladed Savonius wind turbine, pipes, pump and a generator. Solar PV panels are also installed as a backup system, as wind is not always available. Fig. 1 shows the existing turbine blade with 0° twist angle.

Fig. 1. Existing turbine blade.

Savonius rotor was chosen for this application because of the following reasons:

- It is not affected by the direction of wind which is useful in areas where the wind changes its direction frequently.
- It is easy to install and maintain.
They are less expensive to build. Currently, the system does not work to its required efficiency. The novelty of this paper is to enhance the performance of the system by improving the aerodynamic characteristics of rotor.

2.1 Wind Speed Data of the Site

Monthly average wind speed data at different altitude has been obtained from the nearest meteorological station and is shown in Fig. 2.

2.2 CAD Model of Existing System

Two bladed Savonius rotor (semicircular blades) was designed using Solid Edge, to analyze its performance. The 3-D design views of the Savonius wind turbine are shown in Fig. 3.

There are three basic dimensions of the turbine:
- Height of the Turbine \( H = 1500 \text{ mm} \)
- Diameter of the Turbine \( D = 1090 \text{ mm} \)
- Diameter of the Blade \( d = 545 \text{ mm} \)

3. Proposed Methodology and CAD Models

Twisted blade, in general, would possess the following characteristics:

Negative projected area is decreased. The air is swept inward and outward, creating a couple, to ensure smooth rotation of the rotor with high rotational velocity and torque. Having this, the twisted turbine does not need to be in the direction of wind flow and it will self-start.

3.1 Static CFD Analysis

A CFD analysis is used to determine the performance of the rotor blades, flow patterns, pressure distribution, velocity counters around the blades and possible improvements to the design. The CFD analysis is carried out using the ANSYS 14 software package.

To carry out the static CFD analysis, the required sets of steps were followed. Fig. 6 shows the blade in the domain and the coarse mesh generated.

Several CAD models were designed with different twist angles in a selected range with appropriate intervals to find the angles which gives substantial torque. Fig. 4 and Fig. 5 show two such CAD- models which are designed with twist angles of 40° and 45°, respectively.

### Table 1: Boundary conditions for CFD analysis

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td>Solid aluminum</td>
</tr>
<tr>
<td>Enclosure</td>
<td>Fluid (air)</td>
</tr>
<tr>
<td>Operating velocity</td>
<td>5m/s</td>
</tr>
<tr>
<td>Operating pressure</td>
<td>Atmospheric</td>
</tr>
<tr>
<td>Viscous model</td>
<td>Laminar</td>
</tr>
<tr>
<td>Near-wall treatment</td>
<td>Standard wall function</td>
</tr>
<tr>
<td>Velocity formation</td>
<td>Absolute</td>
</tr>
<tr>
<td>Solver type</td>
<td>Pressure-based</td>
</tr>
<tr>
<td>Time</td>
<td>Steady</td>
</tr>
</tbody>
</table>

The rest of the boundaries are considered to be walls.
Table 1 shows the boundary conditions used for the analysis.

The results are obtained for various twist angles ranging from 0 to 45° and summarized in Table 2. Pressure and Velocity contours for two such twist angles (40° and 45°) are shown in Fig. 7 and Fig. 8, respectively.

Table 2: Results extracted for various twist angles from ANSYS

<table>
<thead>
<tr>
<th>Angle (°)</th>
<th>Inlet Velocity $V_1$ (m/s)</th>
<th>Outlet Velocity $V_2$ (m/s)</th>
<th>Wind Power $P_w$ (W)</th>
<th>Mechanical Power $P_m$ (W)</th>
<th>Power Coefficient $C_p$ ($P_m/P_w$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>5.08</td>
<td>2.738</td>
<td>130.749</td>
<td>71.1648</td>
<td>0.544</td>
</tr>
<tr>
<td>10</td>
<td>5.05</td>
<td>2.25</td>
<td>128.446</td>
<td>74.1807</td>
<td>0.577</td>
</tr>
<tr>
<td>20</td>
<td>5.05</td>
<td>2</td>
<td>128.446</td>
<td>75.3642</td>
<td>0.586</td>
</tr>
<tr>
<td>30</td>
<td>5.05</td>
<td>1.95</td>
<td>128.446</td>
<td>75.5170</td>
<td>0.587</td>
</tr>
<tr>
<td>40</td>
<td>5.08</td>
<td>1.907</td>
<td>130.749</td>
<td>77.0086</td>
<td>0.589</td>
</tr>
<tr>
<td>45</td>
<td>5.07</td>
<td>2.02</td>
<td>129.978</td>
<td>76.2219</td>
<td>0.586</td>
</tr>
</tbody>
</table>

The values of Table 2 are calculated from general expressions for turbines. It is seen that considerable improvement in mechanical power and power coefficient is achieved at twist angle of 40°. Fig. 9 shows velocity versus twist angle graph.

It can be observed from Fig. 9 that as twist angle increases the velocity drop across the blades increases till it reaches an angle 40°. At this angle, the drop in the velocity is maximum due to aerodynamic effects beyond which there is a sudden decrease in velocity drop. Hence it can be concluded that the optimum twist angle is 40° and the blade design is most efficient as most of the kinetic energy is converted into work.

3.2 Rotational CFD Analysis

Rotational analysis is carried out incorporating the rotation of the blades to compare the performance with the static case. In this case, three models are selected the existing one (twist angle = 0°), the optimum one (twist angle = 40°) selected from the static analysis and blade with twist angle=45°. CFD analysis carried out and pressure and velocity profiles are obtained. Pressure and Velocity contours for two such twist angles (40° and 45°) are shown in Fig. 10 and Fig. 11, respectively.

Velocity at entry and exit of the blades are taken and the results are tabulated as in Table 3.
Table 3: Results extracted for selected twist angles for rotational analysis

<table>
<thead>
<tr>
<th>Angle (°)</th>
<th>Inlet velocity $V_1$ (m/s)</th>
<th>Outlet velocity $V_2$ (m/s)</th>
<th>Wind power $P_w$ (W)</th>
<th>Mechanical power $P_m$ (W)</th>
<th>Power coefficient $C_{p} = (P_m/P_w)$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>5</td>
<td>3.95</td>
<td>124.668</td>
<td>41.8141</td>
<td>0.335</td>
</tr>
<tr>
<td>40</td>
<td>5</td>
<td>2.7</td>
<td>124.668</td>
<td>67.79485</td>
<td>0.544</td>
</tr>
<tr>
<td>45</td>
<td>5</td>
<td>3.09</td>
<td>124.668</td>
<td>62.1466</td>
<td>0.498</td>
</tr>
</tbody>
</table>

Fig. 10. Pressure and velocity contours for 40°.

Fig. 11. Pressure and velocity contours for 45°.

Fig. 12. Pressure coefficient versus twist angle.

A reduced negative wetted area is the primary reason for a marked improvement in the performance of twisted blades. This leads to a realization that at some optimal angle, there reaches a minimum negative wetted area at which a high torque as well as high RPM is achievable. This can be viewed from power coefficient versus twist angle graph as in Fig. 12.

The objective of this work is to obtain an optimum geometry capable of delivering high torque for the vertical axis wind turbines. Performance determination of the turbine has been made on the basis of coefficient of power. The parameters kept constant in this investigation are inlet velocity and aspect ratio. Initially, the existing wind turbine’s semi-circular blades, which serves as a baseline geometric configuration is modeled and its flow field is evaluated with the help of CFD. It is pertinent to note that the blade with a twist of 0° is a semi-circular blade, and then an optimization strategy is selected to set the roadmap for the optimum twist angle in the blade. Parameterization of an array of twist angles (from 0° to 50°) is performed along with their subsequent CAD modeling and their flow characteristics thoroughly analyzed in ANSYS. Static and Rotational analysis were carried out and an increase in velocity drop is observed from inlet to outlet and this is evident in the contour plots. Likewise, the contour plots of static pressure show a decrease in static pressure from the upstream to downstream side of the rotor, resulting in useful drag and torque for the rotor. The maximum drop in velocity was observed at twist angle 40° which gives the maximum power coefficient in both of the analysis.

4. Conclusions

For the application selected in this work the optimum twist angle for maximizing the output is determined to be 40°.

Although, the coefficient of power and torque of the wind turbine is affected by the tip speed ratio and blade shape, theoretically the rotor height only affects the torque. The negative wetted area determines the speed and torque which can be controlled to achieve desired results. The forces to which the blades are subjected vary on flow conditions, namely the relative velocity of the blade to the
wind, or the tip speed ratio. Further a general optimization procedure could be investigated considering all these factors.

References


Authors' photographs and biographies are not available at the time of publication.