CFD modeling and multi-objective optimization of the axial fan parameters

Hasan Koten
Istanbul Medeniyet University, Istanbul, Turkey, hasan.koten@medeniyet.edu.tr
ORCID: 0000-0002-1907-9420

Arrived: 16.08.2018  Accepted: 23.09.2018  Published: 31.12.2018

Abstract: Axial fans are widely used in many areas to provide necessary air and heat and mass transfer, especially in HVAC industry. These fans are designed according to their cooling, heating and drying needs. In this optimization study, the performance of the 16-blade novel design axial fan CVS-AJ-R 1000 is analyzed in 3D using Computational Fluid Dynamics (CFD) technique. First, forward skewed 3D CAD geometry of axial fan is drawn and normalized for CFD pre-processing step. Then volumetric flow rates, pressures, energy consumptions and air velocities are evaluated in inlet and outlet region optimizing blade angle, shroud angle and other design parameters. Results are agreed with experimental studies. Optimized results are discussed and reported comparing with literature studies. As a results, the static pressure in the best-case decreases from 87.79 Pa to 54.59 Pa, the flow rate increases from 1.3 m3/sec to 2.78 m3/sec at 1550 rpm.

Keywords: Axial fan, HVAC, Optimization, Energy.

Cite this paper as: Koten H, CFD modeling and multi-objective optimization of the axial fan parameters. Journal of Energy Systems 2018; 2(4): 137-144, DOI: 10.30521/jes.454146

© 2018 Published by peer-reviewed open access scientific journal, JES at DergiPark (www.dergipark.gov.tr/jes)
1. INTRODUCTION

Generally, fans as a mechanical device provide motion of air, vapor and different gases in a given system at the desired velocity [1-8]. To do this, it is utilized from difference across the fan rotor shown in Fig. 1. The axial fans are mostly used for the required gas flow for transfer operations in many industrial processes. As applications, HVAC systems, humidifiers processes, heat exchangers, mining ventilation systems, cooling electric equipment, exhausting systems, in steam boilers, cooling of heavy-duty vehicle engines can be evaluated. Axial fans can be considered as propeller fan, tube type axial fan and vane type axial fans. They generate high pressures makes them proper for difficult operating conditions, in high working conditions [9-11].

In literature, many studies were done about performance characteristics of fans. During a fire, it may be desirable to reverse flow in order to provide an escape way or to isolate a fire [10]. Also, in colder areas, the gas flow may be reversed to prevent ice buildup. When reversing main special designed fans, the mine operator usually does not know about operating characteristics of flow and pressure to expect. Laboratory and field tests of vane axial main special designed fans were conducted to establish forward and reverse performance characteristics under controlled conditions and in typical mine installations [12-19]. All fans tested were between 81 and 96 in. (2.1 and 2.4 m) in diameter. The vane axial fan is shown in Fig. 2. The data obtained suggest that reverse performance characteristics are dependent upon the blade angle and the hub to tip ratio [20]. The special fan performance curve is shown in Fig. 2 there is also evidence that reverse performance can be predicted for a family of blade angles for a given hub to tip ratio. Generally, fan quantity is~30-60 % less in reverse than when operating in the normal forward mode, with the static pressure equal to the square of the volume percentage change [20].
It is studied the flow mechanism analysis and experimental study of a forward-skewed impeller and a radial impeller in the low-pressure axial fan [20]. The forward-skewed blade was obtained by the optimization design of the radial blade and CFD technique. Measurement of the two blades was carried out in aerodynamic and aeroacoustics performance. Compared to the radial blade, the forward-skewed blade has demonstrated the improvements in efficiency. The aerodynamic curve is shown in Fig. 3.

Detailed flow measurement and computation were performed for outlet flow field for investigating the responsible flow mechanisms. The results show the forward-skewed blade can cause a span wise redistribution of flow toward the blade mid-span and reduce tip loading [21-32].
2. CFD STUDY

Solid geometry of axial fan, computational domain, boundary conditions, and numerical methods are prepared in CFD code to figure out the performance of axial flow fan. The geometry is created in CAD software and it is processed according to CFD methodology. The CAD geometry also is checked and repaired before the analyze. The analysis is carried out in the 3D CFD code. The CAD geometry of the axial fan and mesh structures were shown in Fig. 4. In the CFD calculation, commercial CFD code and multi-objective optimization code are used and RANS based study is applied in the analyze.

In this novel model, the axial fan designed at CVSair Co. is considered as base work. The setup consists of the axial fan having a diameter of 1000 mm. The rotor consists of five blades, with a stagger angle of 27°. In the experimental study, generally, hot-wire anemometers are used to measure the speed of air. As an experimental study, in literature, the manometer is provided to measure the static pressure generated by the axial fan, Pitot tube is used to measure fluid flow velocity. The thermometer is provided to measure the temperature of incoming air; the barometer is provided to measure the atmospheric pressure. The electric motor is used as a power source for the axial fan, and then the streamline lattice used. The parameters studied with this system were flow rate, static pressure, velocity, pressure coefficient and flow coefficient [21].

![Figure 4. (a) CVS-AJ-R 1000 rotor shape and (b) mesh structure.](image)

3. RESULTS AND DISCUSSION

In this optimization study, mesh geometry, the variation of temperature and velocity along the axial fan and along the slope of the case, and different values were investigated. Figure 4(a,b) shows the mesh geometry of the examined fan. Tetrahedral elements were used as a mesh element. Since cases examined during the study, the mesh sizes of the models were approximately between 800000 and 2000000. Before the optimization study, the mesh dependency was analyzed and different cases were tried to get more reliable mesh structure. Boundary conditions were defined in commercial CFD code. At the exit of the fan pressure, outlet boundary condition was assumed with a static pressure due to the wind velocity over the fan exit. At the fan sidewalls, constant temperature and no-slip boundary conditions were assumed. Housing material was defined as steel. At the air inlet, standard atmospheric conditions were defined as the boundary condition.
Table 1. Used cases in optimization study.

<table>
<thead>
<tr>
<th>Name</th>
<th>Units</th>
<th>P17 – flow rate ( \text{m}^3/\text{s} )</th>
<th>P7 - Pressure inlet ( \text{Pa} )</th>
<th>P9 - Energy ( \text{J} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>DP 0 (Base)</td>
<td></td>
<td>1.56</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 1</td>
<td></td>
<td>2.23</td>
<td>-72,1907</td>
<td>-166,474</td>
</tr>
<tr>
<td>DP 2</td>
<td></td>
<td>2.26</td>
<td>-72,6749</td>
<td>-168,178</td>
</tr>
<tr>
<td>DP 3</td>
<td></td>
<td>2.67</td>
<td>-80,0479</td>
<td>-156,914</td>
</tr>
<tr>
<td>DP 5</td>
<td></td>
<td>3.34</td>
<td>-78,8074</td>
<td>-149,285</td>
</tr>
<tr>
<td>DP 6</td>
<td></td>
<td>3.56</td>
<td>-65,101</td>
<td>-186,154</td>
</tr>
<tr>
<td>DP 7</td>
<td></td>
<td>3.78</td>
<td>-82,9685</td>
<td>-138,948</td>
</tr>
<tr>
<td>DP 8</td>
<td></td>
<td>4.32</td>
<td>-56,9191</td>
<td>-203,982</td>
</tr>
<tr>
<td>DP 9</td>
<td></td>
<td>4.34</td>
<td>-63,1881</td>
<td>-178,719</td>
</tr>
<tr>
<td>DP 10</td>
<td></td>
<td>4.45</td>
<td>-82,339</td>
<td>-151,564</td>
</tr>
<tr>
<td>DP 11</td>
<td></td>
<td>4.56</td>
<td>-78,031</td>
<td>-113,014</td>
</tr>
<tr>
<td>DP 12</td>
<td></td>
<td>4.78</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 13</td>
<td></td>
<td>5.97</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 14</td>
<td></td>
<td>9.87</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 15</td>
<td></td>
<td>15.78</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 16</td>
<td></td>
<td>18.99</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 17</td>
<td></td>
<td>23.54</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
<tr>
<td>DP 18</td>
<td></td>
<td>27.85</td>
<td>-72,6336</td>
<td>-165,938</td>
</tr>
</tbody>
</table>

Fig. 5 shows the velocity magnitudes along the axial fan from front and side zoomed view. The velocity vectors were distributed along the fan which was located at the center. In Fig. 6, streamlines were illustrated in duct.

After examining these results, investigation of the axial fan has been performed. Results were discussed on selected 19 cases. Volumetric flow rate, rotor diameter, blade height and housing height were
changed parametrically to get optimum axial fan geometry. These results were discussed in axial fan characteristics.

In the multi-objective optimization study, novel design 16 bladed axial fan design points were investigated for blade angles, shroud angles and pressure inlet values as shown in Figs. 7 and 8. Thanks to the CFD model, several results were evaluated, such as air flow, volumetric flow rates and velocities on the blades. Selected 19 cases were examined and evaluated using CFD code via parallel processing. The volumetric flow rate is analyzed taking into consideration housing height, and fan diameter. This study has shown that the CFD code provides ease for the prediction of airflow in fan cases.

![Figure 6: Stream lines around the fan blades, full and zoom view.](image1)

![Figure 7: Design points in pressure inlets versus blade angles.](image2)
4. CONCLUSION

In the first part of the study, by using the CFD model, several results were evaluated, such as air flow, volumetric flow rates and velocities on the blades. Selected cases were examined and evaluated using CFD code via parallel processing. The mass flow rate is analyzed taking into consideration housing height, and fan diameter. This study has shown that the CFD code provides ease for the prediction of airflow in fan cases. To evaluate the effect of height and diameter on the ventilation flow rate, several dimensions were simulated. To evaluate the effect of height and diameter on the ventilation flow rate, several dimensions were also simulated.

Average volumetric flow rates for an axial fan, for different values, were showed in the result section. The pressure difference between fan back and forward side were evaluated. The magnitude in airflow rate difference increases alongside the fan. Also in this study, in axial flow fan within a multidimensional framework blade profiles computationally were modeled using k-ε turbulence model. Different cases were investigated to get the proper boundary conditions using 3D CFD and multi-objective optimization code. As shown in Table 1, flow rates are changing between 1.56 m³/s and 27.85 m³/s. The axial fan performance is analyzed at 1550 rpm, the studied parameters are the flow rate, static pressure, flow coefficient and pressure coefficient. The static pressure in the best case decreases from 87.79 Pa to 54.59 Pa, the flow rate increases from 1.3 m³/sec to 2.78 m³/sec at 1550 rpm.

Acknowledgements

This work was supported by CVS air Co. and Istanbul Medeniyet University. The authors thank to Muharrem Ozturk, CVS air Co. and Istanbul Medeniyet University for their support.

REFERENCES