Performance Analysis of Centrifugal Fan by CFD

Jaykumar Gajjar #1, Prof. Chetan Vora*2, Prof. Vipulkumar Rokad$3

#PG student Mechanical Department, Kalol Institute of Technology and Research Centre
*Asso. Professor Mechanical Department, KITRC, Kalol, Gujarat, India
$Assist.Professor, KITRC, Kalol, Gujarat, India
jkmar.gajjarl@gmail.com
vorachetan1@gmail.com
vipulrokad@gmail.com

Abstract- Our research paper is based on performance improvement of centrifugal fan. Centrifugal fans are consuming very high energy and require frequent maintenance. The Fan having low capacity, Low Static Pressure, Low Static efficiency during the performance testing by manufacturer. We will carried out analysis for existing design of the fan by ANSYS and modify the design of the impeller.

Keywords- Centrifugal Fan casing, Impeller and Inlet &Outlet section etc.

1. INTRODUCTION

A centrifugal fan is a mechanical device for moving air or other gases. They convert the kinetic energy of the impeller for increment of the pressure of the air or gas which is moves against the resistance caused by casing, dampers and other components. Centrifugal fans increase air velocity radially, by changing the direction (typically by 90°) of the airflow.

Centrifugal fans are continuous displacement devices or constant volume devices, at a constant speed, a centrifugal fan will flow at a constant rate volume of air rather than a constant mass. This means that the air velocity of the system is fixed even though the mass flow rate through the fan is not.

![Centrifugal Fan](image)

Figure 1 Centrifugal Fan

Main component of a centrifugal fan are:

1. Fan casing
2. Impeller
3. Inlet and outlet Vents
4. Drive shaft
5. Drive mechanism

<table>
<thead>
<tr>
<th>Sl No.</th>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Fan Make</td>
<td>Suvidha Air Engineers</td>
</tr>
<tr>
<td>2</td>
<td>Type of Fan</td>
<td>Centrifugal Fan DWDI</td>
</tr>
<tr>
<td>3</td>
<td>Fan Model/Size</td>
<td>SB-550</td>
</tr>
<tr>
<td>4</td>
<td>Blade Type</td>
<td>Backward Inclined/20 Nos/62.0°</td>
</tr>
<tr>
<td>5</td>
<td>Impeller Diameter</td>
<td>Ø 470 mm</td>
</tr>
<tr>
<td>6</td>
<td>Capacity (M³/Hr) /CFM</td>
<td>13803/8124</td>
</tr>
<tr>
<td>7</td>
<td>Static Pressure (mm WC)</td>
<td>50 at 50°C</td>
</tr>
<tr>
<td>8</td>
<td>Total Pressure (mm WC)</td>
<td>85 at 50°C</td>
</tr>
<tr>
<td>9</td>
<td>Fan Design Temperature</td>
<td>50°C</td>
</tr>
<tr>
<td>10</td>
<td>Type Drive/Arrangement</td>
<td>V Belt/ #3</td>
</tr>
<tr>
<td>11</td>
<td>Fan Speed (RPM)</td>
<td>1165</td>
</tr>
<tr>
<td>12</td>
<td>Critical Speed</td>
<td>2018</td>
</tr>
</tbody>
</table>

Other parts used are bearings, couplings, impeller locking device, fan discharge casing, shaft seal plates etc.
### 3. DESIGN CALCULATIONS

The input parameters for the design of backward curved inclined blade centrifugal fan are summarized below.

Flow Capacity $Q = 3.83 \text{ m}^3/\text{s}$
Static Pressure $= 490.32 \text{ N/m}^2$
Inlet Diameter of impeller $D_1 = 0.428 \text{ m}$
Outlet Diameter of impeller $D_2 = 0.470 \text{ m}$
Impeller Speed $N = 1165 \text{ rpm}$
Air Density $= 1.09486 \text{ kg/m}^3$
Number of blades $z = 10$
Suction Temperature $T_s = 50 \degree \text{C} = 323 \text{ K}$
Atmospheric Pressure $P_{atm} = 1.01325 \times 10^5 \text{ Pa}$
Atmospheric Temperature $T_{atm} = 50 \degree \text{C} = 323 \degree \text{ K}$

These parameters are kept ideal for existing design methodology defined here in.

Let consider the tangential velocity component 10% more than axial velocity component for greater suction of flow.

$$U_1 = 1.1 \times V_1$$

Discharge, $Q = \frac{\pi}{4} \times D_1^2 \times V_1$

$$V_1 = \frac{4 \times Q}{\pi \times D_1^2}$$

$$U_1 = \frac{\pi D_1 N}{60} = 1.1 \times V_1$$

$$\Rightarrow \frac{\pi D_1 N}{60} = 1.1 \times \frac{4 \times Q}{\pi \times D_1^2}$$

$$\Rightarrow Q = \frac{\pi^2 \times D_1^3 \times N}{60 \times 1.1 \times 4}$$

$$\Rightarrow Q = \frac{3.41 m^3}{s} = 7220 \text{ ft}^3/\text{m}$$

So, it is clear that the required capacity is less than existing from the fundamental calculation as per the given speed of the fan and inlet diameter.

Now, for the required flow discharge the speed of the fan is
\[ \frac{\text{CFM}_{\text{new}}}{\text{CFM}_{\text{old}}} = \frac{\text{RPM}_{\text{new}}}{\text{RPM}_{\text{old}}} \]
\[ \Rightarrow \frac{8124}{7220} = \frac{\text{RPM}_{\text{new}}}{1165} \]
\[ \text{RPM}_{\text{new}} = 1310.86 \approx 1311 \]
\[ \Rightarrow \frac{\pi D_1 N}{60} = 1.1 \times \frac{4Q}{\pi \times D_1^2} \]
\[ \Rightarrow D_1 = \frac{1.1 \times 4 \times Q \times 60}{\pi^2 \times N} \]
\[ \Rightarrow D_1 = 0.427 \text{m} \]
\[ \Rightarrow V_1 = 26.62 \text{ m/s} \]

So, that Inlet tip velocity
\[ U_1 = \frac{\pi D_1 N}{60} = 1.1V_1 \]
\[ \Rightarrow U_1 = 29.29 \text{ m/s} \]
\[ \Rightarrow V_1 = 26.62 \text{ m/s} \]

\[ V_{r1} = \sqrt{U_1^2 + V_1^2} = 39.57 \text{ m/s} \]

Impeller inlet blade angle:
\[ \tan \beta_1 = \frac{V_1}{U_1} = \frac{26.62}{29.29} \]
\[ \beta_1 = 42.26^\circ \]

Impeller width at inlet:
Here Z=10 and blade thickness t=2.5 mm
\[ Q = [\pi D_1 - Zt] \times b_1 \times V_1 \]
\[ b_1 = 0.109 \text{ m} = 109 \text{ mm} \]

Impeller outlet parameters:
The Fan power = \[ Q \times \text{Total Pressure} \]
\[ = 3.33 \times 833 \]
\[ = 3190.39 \text{ W} \]

Considering 10% extra to accommodate flow recirculation and impeller exit hydraulic losses
So,
\[ 1.1 \times \text{The Fan Power} = 3509.429 \text{ W} \]
\[ \text{Power, } P = \dot{m} \times W_a \]

Specific Workdone,\[ W_a = 837.56 \text{ W/(kg/s)} \]

\[ \text{Eulerpower} = \dot{m} V_{w2} U_2 \]

Taking, \[ V_{w2} = 0.8 U_2 \]
\[ 3509.429 = 1.09486 \times 3.83 \times \dot{m} U_2 \]
\[ U_2 = 32.35 \text{ m/s} \]
\[ V_{w2} = 0.8 \times 32.35 = 25.88 \text{ m/s} \]

\[ \text{And, } U_2 = \frac{\pi D_2 N}{60} \]
\[ D_2 = 0.471 \text{ m} \]

Taking width of blade at inlet= outlet blade width
\[ \Rightarrow b_1 = b_2 \]
\[ Q = (\pi D_2 - Zt) \times b_2 \times V_{f2} \]
\[ V_{f2} = 24.16 \text{ m/s} \]
\[ W_2 = 0.2 U_2 = 6.47 \text{ m/s} \]
\[ V_{r2} = \sqrt{W_{w2}^2 + V_{f2}^2} \]
\[ V_{r2} = 25.01 \text{ m/s} \]
\[ V_2 = \sqrt{V_{r2}^2 + V_{f2}^2} \]
\[ V_2 = 35.40 \text{ m/s} \]

\[ \tan \alpha_2 = \frac{V_{f2}}{V_{w2}} = \frac{24.16}{25.88} = 0.9335 \]
\[ \alpha_2 = 43.03^\circ \]
4. CFD Analysis of centrifugal fan.

Basic Steps for CFD Analysis

(1) Preprocessing: defining the problem
Import the geometric model in the ansys and the geometry is checked for any free edges, copy surfaces, and little gaps/filets. Crossing point and globules so it doesn't trade off the outline result.
(i) define element type and material/geometric properties,
(ii) Mesh lines/areas/ volumes as required. The amount of detail required will depend on the dimensionality of the analysis, i.e., 1D, 2D, ax symmetric, and 3D.

(2) Solution: assigning cellzone condition , boudry conditions, solution model, solution initialization etc.
Here, it is necessary to specify the inlet conditions (velocity or pressure), rotational speed provide to rotational frame of the fan and finally solve the resulting set of equations.

(3) Post processing: further processing and viewing of the results
In this stage one may wish to see various plot of the outlet of the fan,
(i) Velocity stream lines,
(ii) Velocity contours,
(iii) Pressure contours
Meshing and Its Quality

Figure 2 Meshing Quality of fan

5. Comparison between Existing Design and Modified design:

Figure 3 Velocity stream lines of existing design

Figure 4 Velocity stream lines of modified design

Figure 5 Velocity Contour of existing design

Figure 7 Pressure Contour of existing design
6. Comparison of Centrifugal Fan Outlet

<table>
<thead>
<tr>
<th>Existing Design Analytical outlet velocity</th>
<th>Modified Design Analytical outlet velocity</th>
<th>Existing Design Fluent Analysis outlet velocity</th>
<th>Modified Design Fluent Analysis outlet velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>25.23 m/s</td>
<td>35.40 m/s</td>
<td>28.39 m/s</td>
<td>38.75 m/s</td>
</tr>
</tbody>
</table>

7. CONCLUSION

We have compared the existing design and modified design outlet velocity of the fan. After the modified design by changing the speed of the impeller at 1311rpm and width of the blade 109mm by fundamental calculations it is understandable from the results that the maximum velocity for the flow by CFD simulation is 38.75 m/s and 35.40 m/s with the fundamental calculation value. Thus the results are clear enough for comparison with new design and finding out what effects will be there among both the impeller designs.

8. REFERENCES

[6] Young-Tae Lee and Hee – Chang Lim: “Performance assessment of various fan ribs inside a centrifugal blower”