IOP Conf. Series: Materials Science and Engineering **297** (2018) 012026 doi:10.1088/1757-899X/297/1/012026

Comparison of two methods to determine fan performance curves using computational fluid dynamics

Patinya Onma[*] and Tonkid Chantrasmi[*]

Department of Mechanical and Aerospace Engineering, King Mongkut's University of Technology North Bangkok, Bangkok 10800, Thailand

*Corresponding Author: Patinya@artith.com, tonkid.c@eng.kmutnb.ac.th

Abstract. This work investigates a systematic numerical approach that employs Computational Fluid Dynamics (CFD) to obtain performance curves of a backward-curved centrifugal fan. Generating the performance curves requires a number of three-dimensional simulations with varying system loads at a fixed rotational speed. Two methods were used and their results compared to experimental data. The first method incrementally changes the mass flow late through the inlet boundary condition while the second method utilizes a series of meshes representing the physical damper blade at various angles. The generated performance curves from both methods are compared with an experiment setup in accordance with the AMCA fan performance testing standard.

1. Introduction

Centrifugal fans or blowers are a type of turbo machinery crucial to many applications in a wide variety of industries as well as residential areas. All of them are used to move air or gas from one place to another, but it is important to design and/or select the fans that are appropriate to the intended applications. For instances, a majority of fans in chemical industries must be made of special materials to withstand high-temperature, corrosive or hazardous gases [1]. Some fans will be used to transport air with particles such as grain in rice mills, kernels in corn factory farms, and sawdust in wood factories. Besides the above specifications, one has to consider the capacity and efficiency of the centrifugal fans according to the system load in the desired applications. As a selection guideline, one can consider what type of blade shapes will be likely appropriate first. In general there are six blade types: airfoil (AF), backward-curved (BC), backward-inclined (BI), radial-tip (RT), forward-curved (FC) and radial-blade (RB) [2].

The first three categories generally have high efficiency (around 80 percent) and their performance curves are stable over a relatively wide range; however, they are primarily suitable only for clean air or clean gas or perhaps air with small non-sticky dust particles. Centrifugal fans with FC blades are well suited for handling high flow rates and low heads. Their performance curves are typically stable from free-delivery point to peak-pressure point, but there is usually a pressure dip slightly to the left of the peak. As for centrifugal fans with RB and RT blades, they have moderate to low efficiency (around 60-75 percent) but can generate a high head than other types. They can be made to handle a wide range of contaminants and can be easily modified or designed to accommodate high temperature.

In this work, a centrifugal fan with BC blades is considered with the main focus on the two primary performance variables, namely volume flow rate and head (or fan pressure). Other variables such as power output and mechanical efficiency can, of course, be calculated afterward for comparison. These two variables at a series of operating points will be obtained both by simulations using Computational Fluid Dynamics (CFD) as well as by experiments according to industrial fan testing standards [3-5]. In this case, AMCA fan testing standard was chosen for the experiment but other standards are quite similar and comparable results can be expected. The goal is to validate some simulation techniques, establish a trustworthy range of operating conditions for each of them, and recommend an appropriate and efficient simulation technique for generating a fan performance curve.

Recently, many research works employed CFD to simulate centrifugal fans usually with a goal to optimize certain design aspects of the fan such as the number of the blades and their physical dimensions [6-10]. Some researches focused on varying the rotational speed of the fan [7-8, 11] while other may study the effects of fan blade types in some specific applications [11-12]. Many of these works performed the simulations only at the design points [9, 13-15], while a few generated an entire performance curve [6, 10, 16-17] with or without comparing to experiment.

In this work, two simulation techniques were used to compute the same performance curve. The first method was similar to many researches in the past. It was done by enforcing varying inlet air velocities, or effectively the volume flow rate, and extracting the fan pressure values at different inlet velocities to generate the performance curve. The second method used a series of computational domains resembling the throttling device in the experiment (a butterfly damper at different opening angles) and imposed the pressure at the boundaries to extract the corresponding volume flow rate. The second method was not involved as it required a different mesh at each condition, but it was more intuitive and could be extended to study the transient behaviors of the flow.

In the next section, the centrifugal fan in this study was described along with the experimental setup that was done according to the AMCA fan testing standard to be used as a reference for comparing the simulation results from the two techniques. Next, in the following section, the two computational techniques were described in details. Then, the simulations from both methods as well as from the experiment were compared and discussed. Finally, in the last section, the conclusion and recommendation were made.

2. Problem Description and Experimental Setup

Figure 1 shows an actual photograph and cross-sectional drawings of the centrifugal fan used in this work. The fan consists of twelve BC blades. The radius of the blade assembly is 762 mm with the diameter at the inlet of 450 mm. The height of blade assembly at the outlet is 165 mm giving the fan outlet of 0.746 m^2. The casing is 1,340 mm by 1,265 mm with geometrical shape as shown in the figure 1. The outlet from the casing to the system is 785 mm tall and 590 mm wide.

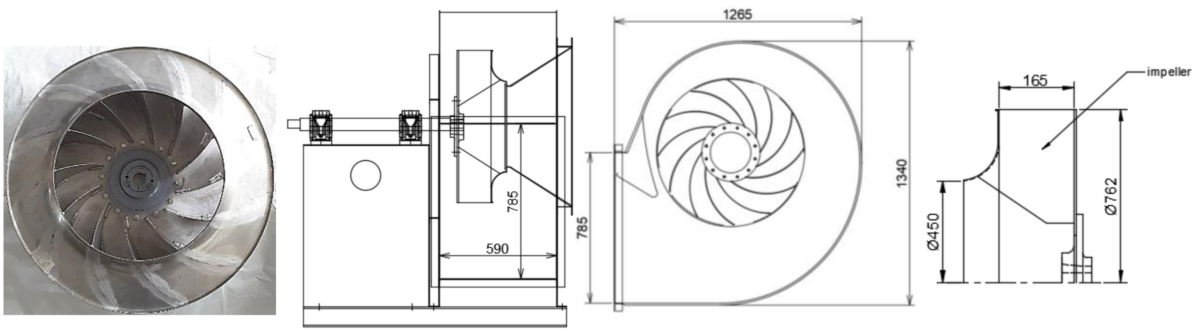

Figure 1. A frontal photograph and cross-sectional drawings of the fan used in this study.

Figure 2 shows the fan performance curves at 1,080-1,755 RPM as provided by the designers (not from an actual testing). In this work, the fan will be run at fan speed of 1,220 RPM in all cases and in the experiment. Note that once a performance curve at one rotational speed is known, one can use Fan Affinity Laws to create performance curves at other speeds over a relatively wide range. According to the provided performance curves in the below figure, the maximum volume flow rates is approximately 325 m^3/min and the peak (static) pressure is about 1,130 Pa (shown as 115 mmAq in the figure).

Note that the performance curve given by the designers does not include the points near the shut-off nor the free-delivery point. It only provides the data around the design point.

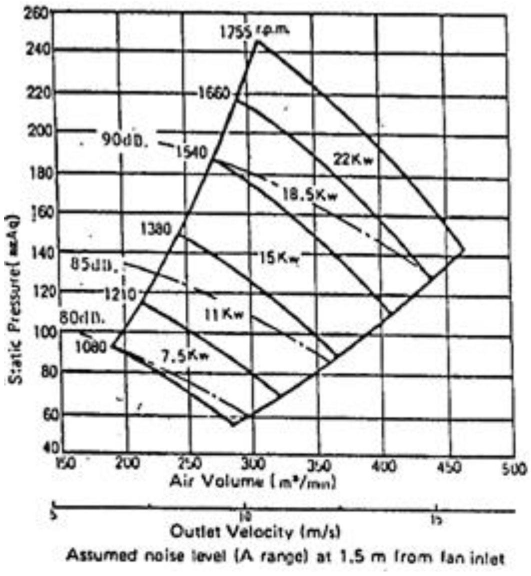

Figure 2. Fan performance curves from design [18]

The outlet of the centrifugal fan was connected to a straight horizontal duct with an adjustable throttling device at the end to simulate various system loads when the fan was installed in the intended application. The dimensions of the duct as well as various components such as the transformation connecting piece, the flow straightener, and the measuring point location are in accordance with the AMCA 210-99 international fan testing standard [3]. Figure 3 shows the schematics of this experimental setup. The length of the duct is 10 times the duct diameter (D), which is the minimum length required by the testing standard.

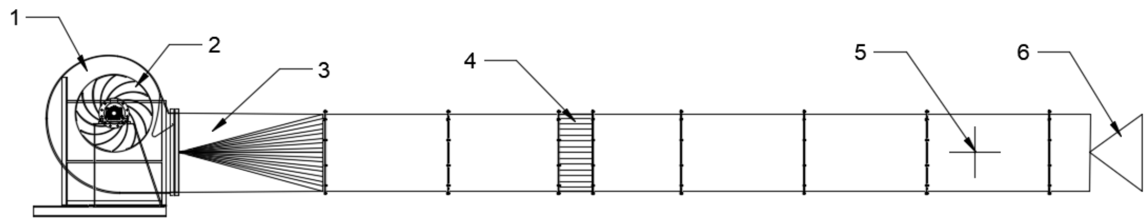

Figure 3. Schematics of AMCA fan testing standard used in this work. The numbers in the diagram indicate each component (1) fan casing (2) fan blades (3) transformation connecting piece (4) flow straightener (5) measuring point, and (6) throttling device.

The data measurement is done using 24 pitot-static tubes at the positions given in the standard (6 radii and 4 positions per radius) with tolerance of no more than ± 0.0025D. From the (averaged) pressure readings of the pitot tubes, the volume flow rate as well as pressure at the measurement can be accurately calculated. The results from the test are shown in table 1.

Table 1. Experimental data showing volume flow rate (Q), fan total pressure (P_t) and fan overall efficient (η) at various operating points at 1,220 RPM

Q [m^3/s]	P_t [Pa]	η [%]
6.71	149	14
6.61	194	18
6.36	232	21
5.81	553	45
4.99	828	59
4.11	1,155	79
2.53	1,395	68
1.86	1,463	65
1.15	1,545	42
0.00	1,521	0

According to the experimental data above, the free-delivery point should be slightly more than 6.71 m^3/s (it is not exact because it is impossible to have zero pressure loss), the shut-off pressure is 1,521 Pa, and the peak efficiency is approximately 79% at volume flow rate of 4.11 m^3/s and total fan pressure of 1,155 Pa.

3. Computational Techniques

Three-dimensional CFD simulations were setup to compute the desired quantities at various operating points along the performance curve at the given rotational speed of 1,220 RPM. ANSYS Fluent commercial software was used to solve for the solution based on finite volume formulation [19 -21]. The simulation settings were steady, pressure-based, and incompressible. The turbulence model was the realizable k-epsilon with standard parameters.

Two different approaches were employed to generate the same performance curve (volume flow rate vs. fan total pressure). They will be discussed in details in the subsections below. For each method, eight simulations were performed to generate eight different points on the performance curve.

All the simulation was done on a dedicated workstation with Intel Xeon Processor E5 1660 v3 (8-core, 20MB cache, 3.0GHz) with 32GB of DDR RAM.

3.1. Method 1: Varying Inlet Velocity

In the first method, the main idea is to use the same geometry and mesh files for all eight simulations. In each of the eight simulations, the volume flow rate was fixed by prescribing a uniform inlet velocity profile at the inlet. The magnitude of the velocity will be varied across each individual simulation. Once the solution had converged, (area-averaged) total pressure difference across the fan was extracted to be plotted on the performance curve.

To avoid the effect of the prescribed uniform velocity profile, a straight section upstream of the inlet was added to give some additional length for the flow to develop. Figure 4 shows the overall geometry used in this method. Since the desired pressure output was measured across the fan only, this straight section should not introduce additional pressure loss to the output. Figure 5 shows a contour of velocity magnitude and that of static pressure near the blades from the simulation with the inlet velocity of 25.4 m/s.

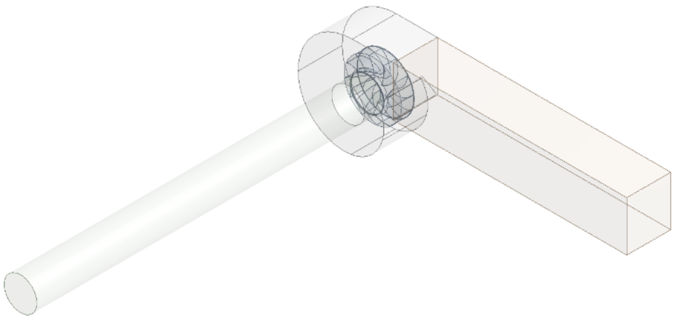

Figure 4. Overall geometry used in Method 1

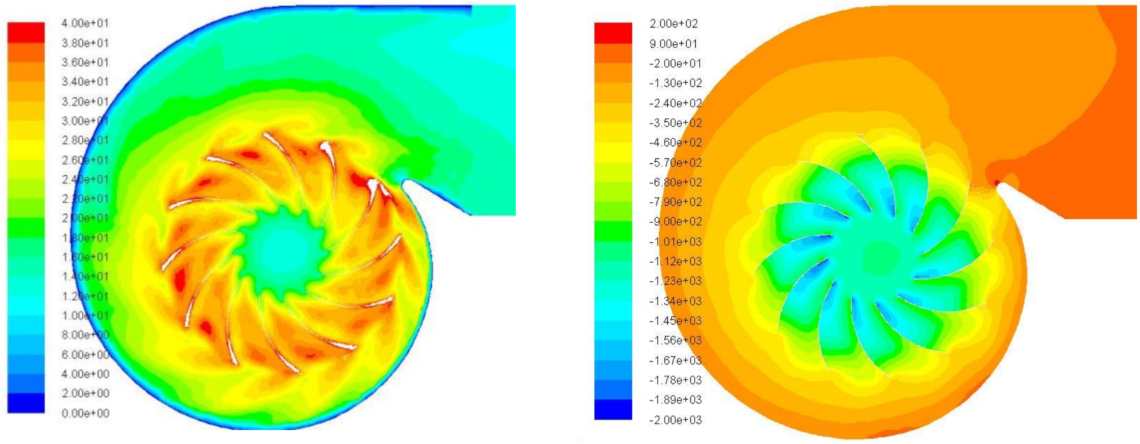

Figure 5. Example of contours of velocity magnitude (left) and static pressure (right) near the blades

The total number of cells in the mesh used was around 500,000 cells. Figure 6 shows 4-point coarser performance curves calculated with these different meshes. The current mesh was deemed to be sufficiently accurate for the purpose of generating the performance curve.

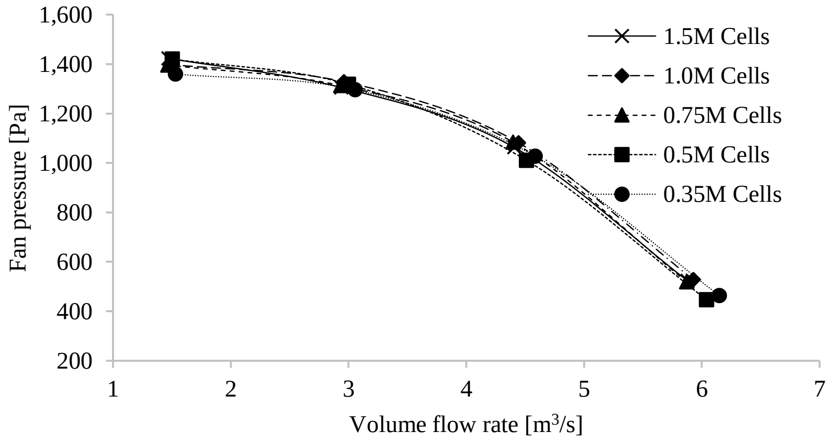

Figure 6. Performance curves generated with different mesh resolutions

3.2 Method 2: Modeling the Throttling Device

In the second method, the geometries of the throttling device at different angle were modeled as an angled flat plate. Since the geometry changes from one simulation to another, one needs to create a new mesh for each simulation at a new point on the performance curve; however, since it more closely resembles the real world, it has a potential to match the experimental data better. Figure 7 shows an example of the overall geometry for this method as well as a side view of the geometry of the throttling device at 50° angle.

The boundary conditions were set to be atmospheric at both inlet and outlet boundaries without prescribing any velocity or mass flow rate. This is also realistic and more intuitive than the first method. Also, there was no additional straight section added upstream of the inlet. Once the solution had converged, both the (area-averaged) total pressure difference across the fan *and* the (integral) volume flow rate were extracted to be plotted on the performance curve.

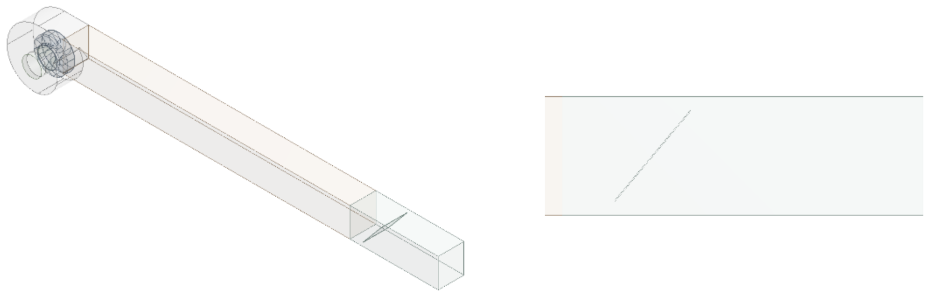

Figure 7. Overall geometry used in Method 2 (left) and geometry of the throttling device at 50° angle

4. Results and Discussions

Table 2 shows the computational from the two methods mentioned above. Note that even though the results from both methods contain eight data points, they are not at the same volume flow rates (nor pressure). In the first method, the volume flow rate can be effectively chosen precisely. In the second method, the input is the angle of the throttling device, and both the flow rate and the fan pressure are not known beforehand.

Table 2. Computational results from Methods 1 and 2

Method 1		*Method 2*	
Q [m^3/s]	P_t [Pa]	Q [m^3/s]	P_t [Pa]
6.0	446	6.4	212
5.3	762	6.3	261
4.5	1,011	6.1	387
3.8	1,187	5.5	689
3.0	1,319	4.5	1,073
2.3	1,415	3.1	1,426
1.5	1,421	1.7	1,580
0.8	1,429	0.8	1,589

Figure 8 compares the computational results from the two methods with the experimental data. For the high volume flow rates (more than 4 m^3/s or 240 m^3/min), the data from both computational methods and the experiment are in good agreement. The experimental results fluctuate from point to point was likely due to some experimental errors, but the magnitude of this fluctuation is small enough to be negligible (relative to the larger differences elsewhere). It is worth noting that this region with high volume flow rates is unsurprisingly the region shown in the fan performance curve from designers (figure 2). Both computational methods can predict the performance curve in this region quite accurately.

At a volume flow rate lower than 4 m^3/s, a clear discrepancy between the two computational methods can be clearly observed. At the same flow rate, the first method predicts lower fan pressure than the second method and the difference increases closer to the shut-off point. The fan pressure from the experiment is between those from the two methods, but given the fluctuation in the experimental data it is difficult to decisively judge which method is more accurate in this case. The authors suspect that the difference is due to the different velocity profiles near the front of the centrifugal fan. This could potentially create a different performance as the flow interacts with the rotating fan blades; however, further investigation is needed to definitively explain this behavior.

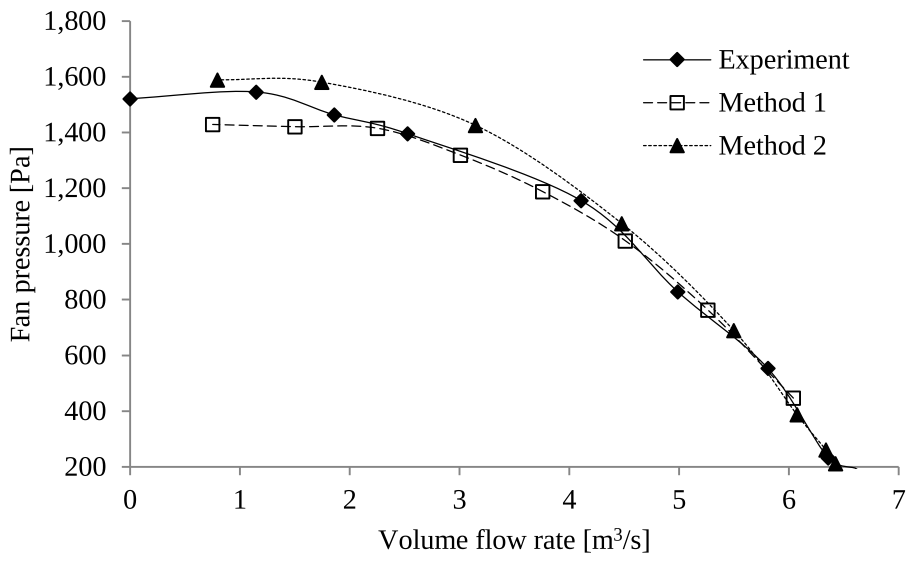

Figure 8. Comparison of the performance curves from the two methods and the experiment

5. Conclusions

In conclusions, this work compares two computational techniques using CFD to construct a fan performance curve. A centrifugal fan with twelve backward-curved blades was used as a model and the results were compared with the experiment following the AMCA fan testing standard. For each technique, eight simulations were performed at different conditions, each of which resulted in a data point along the performance curve. The first method fixed the geometry, added the extra straight section upstream of the fan, varied the flow rate, and extracted the fan pressure. The second method included the throttling device at different angles into the computational domains, which needed changing for each simulation. For this second method, both flow rate and fan pressure were extracted at each simulation. At a flow rate higher than 4 m^3/s, all results were in quite good agreement; however at a flow rate lower than 4 m^3/s, there was a clear difference in fan pressure at the same flow rate. The first method consistently gave a lower fan pressure than the second method. Also, it was inconclusive from the experimental data which of the two methods was more accurate.

Acknowledgment

The authors would like to thank Artith Machinery Co.,Ltd. and Artith Ventilators Ltd., Part. for providing the centrifugal fan and supporting the fan testing experiments in accordance with the AMCA testing standard.

IOP Conf. Series: Materials Science and Engineering **297** (2018) 012026 doi:10.1088/1757-899X/297/1/012026

References

[1] Fran P B 1997 *Fan Handbook* (New York: McGraw-Hill)
[2] CEATI International Inc. 2008 *Fans & Blowers: Energy efficiency reference guide Online*
[3] Laboratory Methods of Testing Fans for Rating *ASHRAE Standard (51-1985), AMCA Standard 210-85*
[4] ISO 5801:2007 *Industrial Fans - Performance Testing using Standardized Airways*
[5] Japanese Industrial Standard *Testing Methods for Turbo-Fans and Blowers (JIS B 8330)*
[6] Chunxi L, Song Ling W and J Yakui J 2011 *J. Energy Conversion and Management* **52** p 2902
[7] Keyur K and Prajesh M 2013 *Int. J. of Advanced Engineering Research and Studies* **2** p 1
[8] Jayapragasan C N, Sumedh J and Reddy K, 2016 *APPN Journal of Engineering and Applied Science* **9** p 1637
[9] Lin S and Huang C 2002 *Experimental Thermal and Fluid Science* **26** 421
[10] Amjadimanesh A, Ajam H and Hossein Nezhad A 2015 *Int. J. of Mechatronics, Electrical and Computer Technology (IJMEC)* **5** p 2109
[11] Singh O P, Khilwani R, Sreenivasulu T and Kannan M 2011. *Int. J. of Advances in Engineering & Technology* **1** p 33
[12] Jayapragasan C N and Reddy K J 2016 *Scientific and Industrial Research* **75** p 638
[13] Chaudhari D R and Patal HN 2015 *Int. J. of Advanced Technology in Engineering and Science* **3** p 134
[14] Lin S and Huang C 2002 *Experimental Thermal and Fluid Science* **26** p 421
[15] Atre Pranav C and Thundil Karuppa Raj R 2012 *Research Journal of Recent Sciences* **1** p 7
[16] Gholamian M, Rao G.K.M and Panitapu B 2013*Case Studies in Thermal Engineering* **1** p 26
[17] Siwek T, Gorski J and Fortuna S 2014 *Environ. Stud* **23** p 2359
[18] ARTITH Ventilator 2016 *SD-EN-01.* **1** p 26
[19] Moukalled F, Mangani L and Darwish M 2016 *The Finite Volume Method in Computational Fluid*
[20] ANSYS Fluent Theory Guide 2013 (ANSYS, Inc)
[21] Stephen B P, Turbulent Flows, (Cambridge: Cambridge University Press)

.