Journal of Energy Technology

JET Volume 7 (2014) p.p. 11-20
Issue 4, November 2014

www.fe.um.si/en/jet.html

NUMERICAL ANALYSIS OF AN AXIAL FLOW FAN: ANSYS VS SOLIDWORKS

NUMERIČNA ANALIZA AKSIALNEGA VENTILATORJA: ANSYS IN SOLIDWORKS

Igor Ščuri ℜ, Marko Pezdevšek, Igor Spaseski, Matej Fike, Gorazd Hren

Keywords: numerical analysis, axial fan, CFD, Ansys CFX, SolidWorks

Abstract

Axial flow fans are designed to operate in stable parts of the characteristic curves for the axial fan. However, it is possible that the operation regime becomes unstable due to changing conditions, resulting in a decrease of the fan's operational characteristics. In this article, a simulation is focused on the stable part of characteristic curves for the axial fan at various mass flow rates, which were acquired with numerical simulation software packages: Ansys CFX and SolidWorks Flow Simulation. The analyses were performed on designed structured meshes with similar numbers of elements, for different mass flow rates. In order to validate the numerically obtained results from both software packages, we compared them to experimental values from a reliable source.

Povzetek

Aksialni ventilatorji so zasnovani tako, da delujejo v stabilnem področju dušilne krivulje. Določene omejitve lahko povzročijo, da delovanje ventilatorja preide iz stabilnega v nestabilno področje dušilne krivulje. Slednje negativno vplivajo na karakteristike ventilatorja. V članku so predstavljeni rezultati numerične analize dušilne krivulje aksialnega ventilatorja s poudarkom na stabilni del pri različnih masnih pretokih. Za izvedbo numeričnih simulacij sta bila uporabljena programska paketa Ansys CFX ter SolidWorks Flow Simulation. Analiza je bila izvedena s strukturirano mrežo za več masnih pretokov. Z namenom, da ovrednotimo rezultate numeričnih simulacij, smo le te primerjali z eksperimentalno pridobljenimi vrednostmi iz literature.

ℜ Corresponding author: Igor Ščuri, Tel.: +386 40 783 926, Mailing address: Brezina 82b, 8250 Brežice, Slovenia, E-mail address: igor.scuri@student.um.si

1 INTRODUCTION

A fan is typically a mechanical device that causes the movement of air, vapour or other gases in a given system. The basic purpose of the fan is to move a mass with a desired velocity. In order to achieve this objective, there is a slight increase of pressure across the fan rotor. However, the main aim remains to move the mass without any appreciable increase in pressure.

Axial flow fans are mainly used for ventilating and air conditioning applications for buildings, mines, vehicles, underground transportation systems, etc. Each application requires a different type of fan. Within a system, particular fans are more appropriate than others in terms of capacity and pressure increase capabilities. In this sense, axial flow fans are generally categorized into four types: propeller fans, tube-axial fans, vane axial fans, and two-stage axial fans, [1].

Axial flow fans are the type for which the fluid flow is predominantly axial, i.e. parallel to the axis of rotation. Axial flow fans usually use air as the working fluid, which operates in an incompressible range, at low speeds and moderate pressures. The flow is treated as axial, with no radial component. The pressure rises with the tangential velocity component increasing, due to the rotation of the impeller and an aerodynamic diffusion process afterwards, [2].

The airflow around the blades profile affects the general shape of the characteristic curve. Furthermore, the airflow through the axial fan causes changes in the angle of attack and the velocity of the airflow around the blades. Due to the aforementioned effects, the axial fan operates in a stable or an unstable part of the characteristic curve, [3].

Axial fans are designed to operate in stable parts of characteristic curve. However, due to spatial or other limitations, it is possible that the fan operation regime becomes unstable, resulting in a decreasing of the fan's operational characteristics. This paper presents a numerical analysis resulting in the characteristic curve of the axial fan with a focus on the stable operating regime.

2 GOAL DEFINITIONS

In order to compare different numerical software packages, we decided to perform numerical analyses and compare the computed results to existing experimental data of characteristic curves, [3]. The comparison was performed at different mass flow rates, declining from 0.7 kg/s to 0.4 kg/s with a step of 0.05 kg/s and from 0.45 kg/s to 0.40 kg/s with a step of 0.01 kg/s. In addition, the comparison was also made at mass flow rate of 0.475 kg/s.

Steady-state simulations were made with Ansys CFX 15.0 and SolidWorks Flow Simulation (SWFS) 2014 software packages in order to obtain the characteristic curve of the axial fan. We have to emphasize the differences in the purposes of the software packages and their limitations. Ansys CFX is a well-known commercial standalone numerical software for numerical analyses, and SolidWorks Flow Simulation is part of a CAD (computer-aided design) package. We attempted to create corresponding meshes and boundary conditions in order to compare the results of analyses, the definitions of meshes, and the user-friendliness of software.

3 NUMERICAL MODEL

In order to obtain reliable results, we created meshes with a comparable number of linear elements in both software packages. The axial fan model was simulated assuming steady-state conditions using various Reynolds-averaged Navier-Stokes (RANS) turbulence models. SWFS provides only the *k-ε* turbulence model, while Ansys CFX offers various options. We performed analyses with the *k-ε* turbulence model and the shear stress transport (SST) turbulence model in Ansys CFX.

3.1 Geometry

The geometry of the axial fan used in the experimental measurements is highly complex. The fan rotor contained small parts, e.g. screws and mounting brackets, which are difficult to survey with the computational mesh. Therefore, the small parts that we assume are not necessary for simulation results have been suppressed from the numerical model, thereby reducing the number of elements of the mesh, and decreasing the required computer resources and time for solving governing equations.

The three-dimensional geometry of the axial fan was modelled in SolidWorks. The numerical model of the axial fan consists of three main parts: inner hub, axial fan rotor with blades and outer tube. The inner hub was divided into two parts. The hub ahead of the axial fan rotor was 1000 mm long and behind it was 2000 mm long. The diameter of the hub is constant with the length of the whole model. With a diameter of 285 mm, the axial fan rotor includes ten aerodynamic blades. The blades have a NACA 6508 profile with a chord length of 80 mm. The blade shape is constant in the radial direction of the blade. The blade angle, i.e. the angle between the chord of the blade and the circumferential direction of the fan, is 45°. The blades have been designed with 2.5 mm as the gap between the top of the blade and outer tube. Therefore, the gap between the top of the blade and the outer tube was constant through the entire length of the model. The geometry of the fan is presented in Figure 1.

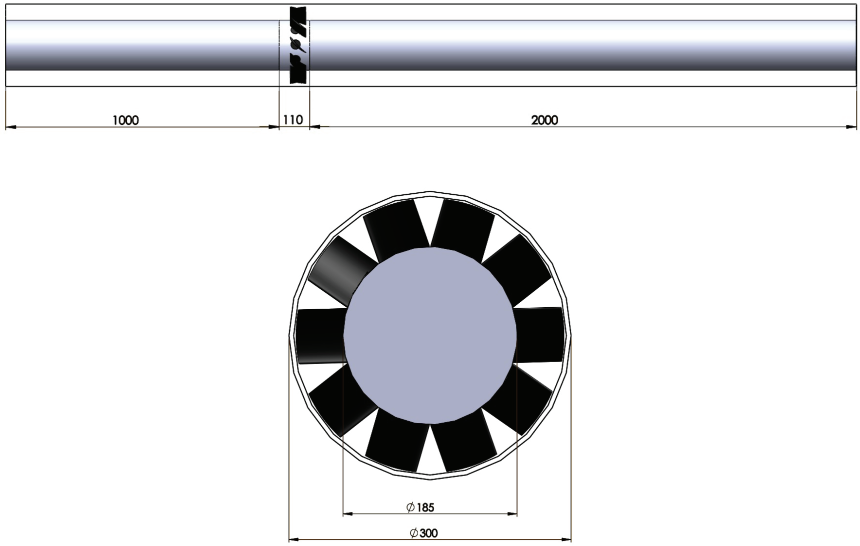


Figure 1: *Axial fan geometry.*

3.2 Mesh creation

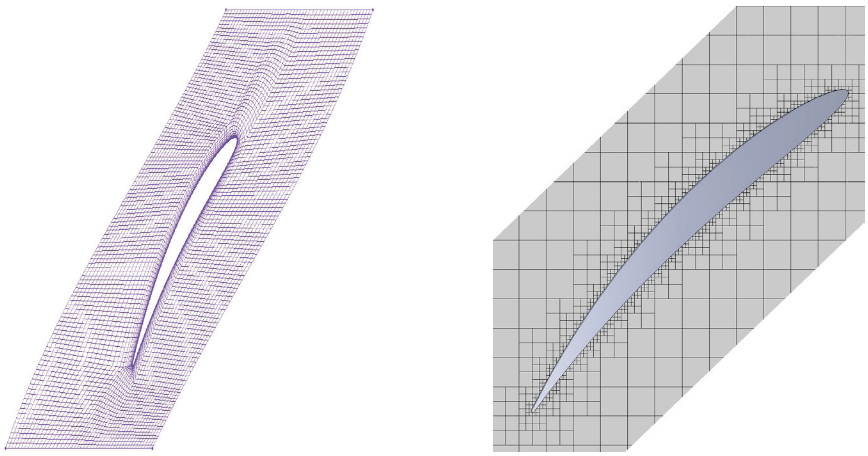
In CFD, the numerical mesh has two basic functions: defining the geometry and the discretization of computational domain. The geometry of the numerical mesh has to fulfil the geometry to the greatest extent possible. The complexity of the modelled object affects the final mesh size and, consequently, the required design time. The amount of computer resources for the implementation of the numerical simulations is directly proportionate to the mesh resolution. The accuracy for solving the discrete equations depends on the number of discrete elements and the nodes of the mesh. Generally, a numerical solution becomes more accurate when a mesh with greater resolution is used. Furthermore, a mesh with a higher density is commonly used in areas where high spatial and temporal gradients of critical quantities accrue. In numerical simulations, we tended to optimise mesh size for greater accuracy of results, which are usually limited by available computer resources.

Structural computational meshes were designed with a pre-processor ICEM CFD 15.0 for the Ansys CFX software, with which we designed separate meshes for the domain ahead of the rotor, behind the rotor and the rotor itself. Once the three meshes were created, they were merged in Ansys CFX to form a single mesh, which represents the full computational domain. In SWFS, the mesh was created with the built-in mesh manager. Table 1 shows comparable number of elements in both software packages.

Table 1: *Mesh data*

Ansys CFX Number of elements	SWFS Number of elements
3,030,110	2,929,807

Figure 2, shows a part of the structural mesh created in ICEM CFD and SolidWorks. The resolution of the mesh is greater in regions where greater computational accuracy is needed, e.g. the region close to the blade. The close-ups of the mesh are shown, which clearly represent a different type of mesh creation. Typical structured meshes from Ansys and the use of a Cartesian-based mesh generation in SolidWorks, [4], are presented.



a) b)

Figure 2: *Numerical mesh designed with a) ICEM CFD software and b) SWFS software in close-up view.*

3.3 Boundary conditions and convergence criteria

Boundary conditions fulfil an important role in all simulations because they govern computational stability and numerical convergence. The initial conditions of the simulation are also specified at these boundaries.

The boundary conditions inlet and outlet were defined in both software packages. Different mass flow values were defined for the inlet boundary condition, and the static pressure of 100 kPa was applied to the outlet boundary condition. The mass flow inlet boundary condition requires the specification of the turbulent inlet flow conditions, so the turbulence intensity was set to 1.5% and the turbulence length to 0.01 m. The air density was set to 1.185 kg/m³ and the dynamic viscosity to $1.79 \cdot 10^{-5}$ kg/ms. The axial fan rotor's angular velocity was set to 1440 rpm, [3].

For SWFS, the outer tube and inner hub surfaces were defined as a real wall and then selected to be stationary (stator). For the computation in Ansys CFX, the domains ahead and behind the rotor were defined as stationary domains. The rotor was defined as a rotating domain with the abovementioned angular velocity. In the stationary domain, the top and bottom surfaces were defined as walls. In the rotational domain, the inner surface of the blade and the bottom surface were defined as walls. The boundary condition stage was applied to the surfaces between the rotating and stationary domains, and the sides were defined as rotational periodicity.

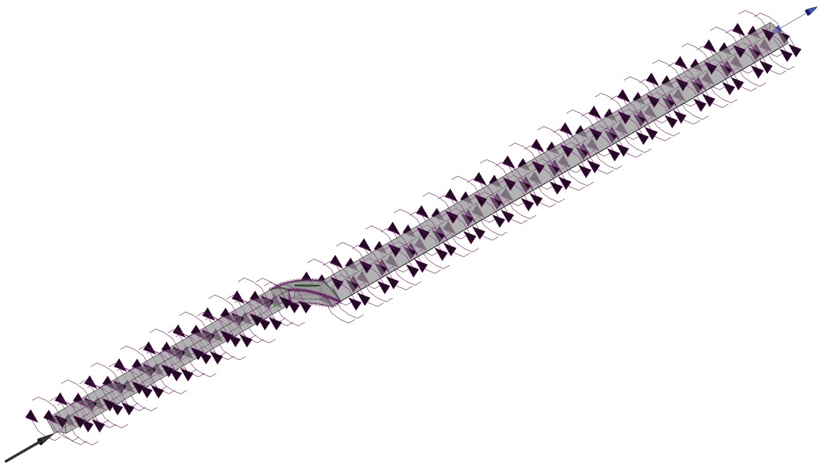


Figure 3: *Boundary conditions defined in Ansys CFX.*

To satisfy the convergence criteria, all the RMS (root mean square) leftovers from solving equations must be under $1 \cdot 10^{-5}$. We also set number of maximum iterations to 500 and used automatic timescale control. Both software packages have an automatic system for stopping the analysis when it reaches predefined convergence criteria.

4 RESULTS

4.1 Characteristic curve

The computed results of the axial fan characteristic curves at various mass flow rates were compared to existing experimental data from [3].

The characteristic curve of the axial fan is defined with the relationship of integral parameters, which are presented by dimensionless numbers. The flow number (coefficient) φ is calculated with the equation:

$$\varphi = \frac{4 \cdot q_v}{\pi \cdot D^2 \cdot u}, \qquad (4.1)$$

where:

q_v - mass flow rate;

D - rotor diameter;

u - tangential velocity.

The pressure number (coefficient) ψ is defined with the equation:

$$\psi = \frac{2 \cdot \Delta p_s}{\rho \cdot u^2}, \tag{4.2}$$

where:

Δp_s - simulated (measured) static pressure increment;

ρ - air density;

u - tangential velocity.

Figure 4 shows the comparison between experimental data and numerical analysis results for characteristic curves at different mass flow rates.

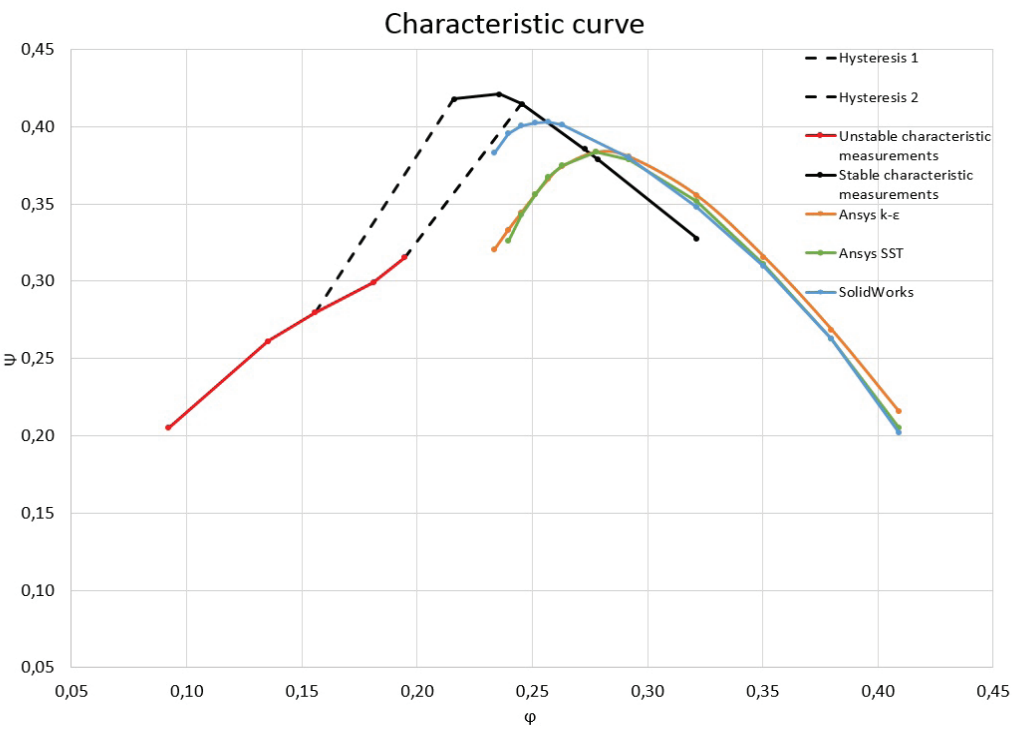


Figure 4: *Comparison between experimental data and the data obtained from simulation software packages of the characteristic curves at different mass flow rates.*

4.2 Computation time

Table 2 shows the average computational times needed for computing the characteristic curve parameters. Computational times in Ansys CFX software using the *k-ε* turbulence model were slightly longer than those using the SST model when one tenth of the axial fan model was simulated. Furthermore, the SolidWorks software took about twice as much time to finish the simulation on the entire axial fan model.

Table 2: *Average computational times needed for computing characteristic curve parameters*

Simulated model (ca. 3,000,000 elements)	Computational time [hh]:[mm]:[ss]		
	SolidWorks	Ansys CFX *k-ε* model	Ansys CFX SST model
1/10 of the axial fan model	/	0:40:03	0:38:52
Whole axial fan model	10:11:42	6:40:30	6:28:40

5 CONCLUSIONS

Simulations of the characteristic curve parameters for various mass flow rates were conducted in order to compare the results from different software packages and then validate them with experimental data.

In Ansys CFX, at mass flow rates lower than 0.4 kg/s, convergence was found to be problematic. Therefore, the comparison between the software was made at mass flow rates ranging from 0.7 kg/s to 0.4 kg/s.

Results obtained from SolidWorks correlate quite well with the experimental results within the normal (stable) operating range of the axial fan. Generally, SolidWorks produced, in our case, better results than Ansys CFX, when using both turbulence models. The correlation between the numerical and experimental values for both turbulence models in Ansys CFX was found to be adequate.

Computational times in Ansys CFX software using *k-ε* turbulence model were slightly longer than those using the SST model when one-tenth of the domain was simulated. With the lack of a rotating periodicy feature in SolidWorks, the results could not be obtained and consequently compared. In this case, when the whole axial fan model was simulated, SolidWorks software took about twice as much time for performing the simulation.

Based on the computed results and computational times, Ansys CFX would be the better pick for the example used in this paper, regardless of the turbulence model. Nevertheless, SolidWorks' computed results were found to be sufficient, but the lack of rotational periodicy means that a large amount of computational resources and time was required to compute the results.

References

[1] **D. Dwivedi, D. S. Dandotiya:** *CFD Analysis of Axial Flow Fans with Skewed Blades*, International Journal of Emerging Technology and Advanced Engineering, Volume 3, Issue 10, October 2013,

[2] Available at: http://www.ijetae.com/files/Volume3Issue10/IJETAE_1013_121.pdf,

[3] Accessed on November 6, 2014.

[4] **T. Köktürk:** *Design and performance analysis of a reversible axial flow fan*, Master's thesis, Middle East Technical University, Graduate School of Natural and Applied Sciences, 2005

[5] **M. Fike:** *Experimental and numerical analysis of fluid flow in an axial fan*, Doctoral thesis, University of Maribor, Faculty of Mechanical Engineering, 2013

[6] **A.Sobachkin, G.Dumnov:** *Numerical Basis of CAD-Embedded CFD, February 2014,* Available at: http://www.solidworks.com/sw/docs/Flow_Basis_of_CAD_Embedded_CFD_Whitepaper.pdf, Accessed on November 13, 2014