Effect of the Computational Domain Shape on the Performance Characteristics of an Axial Flow Fan

Ayhan Nazmi İlikan, Department of Mechanical Engineering, Isik University, Istanbul, Turkey

Erkan Ayder, Department of Mechanical Engineering, Istanbul Technical University, Istanbul, Turkey

Long Abstract

Introduction

In this study, the performance characteristics of a propeller fan (also known as free-inlet free-outlet axial fan which does not contain any duct at the upstream or downstream) is obtained by using CFD (Computational Fluid Dynamic) method. The numerical calculations are performed by using the commercial software ANSYS CFX. Three different inlet and outlet computational domain shapes that are widely used for simulations of axial flow fans are tested. In the first method, the fan is modeled inside a circular pipe where the hub elongates in axial direction. In the second method, the domain represents a real fan inside a pipe (the hub does not elongate in axial direction). The third approach does not contain the domain of a pipe at the inlet and the outlet but takes into account large inlet and outlet domains connected directly to the fan rotor domain. The performance curves obtained from three approaches are compared to each other and to the experimental results obtained from the manufacturer of the fan. The results show the error of each approach and the effect of the upstream flow conditions on the flow at the inlet of the fan rotor.

1. Methods

Three popular computational domain shapes are chosen to model the flow inside the axial flow fan. All the three methods consist of one rotational domain that contains the fan rotor and two additional stationary domains to model the flow at the inlet and the outlet of the fan. In the first method, the inlet and the outlet domain shapes are formed by extension of rotor shroud and hub in axial direction that results in the upstream and downstream flows to occur in a circular annulus. This approach is the most popular one among the three approaches. In the second method, the shroud surfaces of the stationary domains are the same as the first approach; however the hub surfaces of the same domains do not exist since in real conditions, in propeller fans, the flow does not happen in an annulus at the inlet and the outlet domains. In the third approach, the rotational domain is connected to very large inlet and outlet domains which simulate direct opening to atmosphere.

In all the cases, the computational grid is generated by using the commercial software ANSYS Turbogrid and ANSYS Mesh. The finite volume code ANSYS CFX is used to simulate the 3D flow through the fan. The performance characteristics of the fan are predicted for 3D, steady, incompressible flow conditions. The interfaces between the stationary and the rotational domains are modeled by MRF (Moving Reference Frame) method. The turbulence is modeled by k-ω SST approach. In addition, a laminar-turbulent transition model (γ-θ) provided by the code is used.

The results are compared to each other and to experimental ones provided by the manufacturer of the fan. These experimental results are obtained in a standard test chamber to test commercial axial flow fans. The comparisons show that there are some differences between the results of the three approaches that are widely used to obtain numerically the performance characteristics of axial flow fans. It should be indicated that this study is performed for a propeller fan; thus for an axial fan working in a long duct, large rotational domains for the simulations may be unnecessary.