CFD ANALYSIS OF AN INDUSTRIAL AXIAL FAN

MAK 492E Senior Design Project – İ.T.Ü. Mechanical Engineering 2009-2010 Spring Term

Adem CANDAŞ, Hasan Erkan ÇELIKER, Ersin ÖZBALABAN

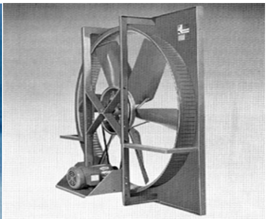
İstanbul Teknik üniversitesi Makina Mühendisliği , Gümüşsuyu, İstanbul, 34437, Türkiye

Summary: In this study, performance of an axial flow fan is numerically examined by using LES and k-ε turbulence models. After creation of solid model out of point cloud, computational flow field is discretized using unstructured grids for both rotating and stationary domains by a commercial CFD code. Rotation is defined between two domains by sliding mesh model. In order to observe effect of different pressure rises on flow rate, simulations are carried out for five pressure values: 80 Pa, 90 Pa, 100 Pa, 110 Pa and 120 Pa. Using flow rates corresponding to these pressure rises, performance curve of the fan is created. Computations are done for two rotational speeds, 720 rpm and 1080 rpm, by using a commercial code, FLUENT. Thus, effect of rotational speed on flow rate is investigated for each pressure rise. Finally, structural analysis is performed to examine the mechanical endurance of the fan. Static analysis is carried out on one blade under operating conditions 100 Pa and 720 rpm.

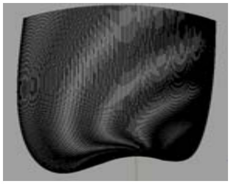
1. INTRODUCTION

Axial flow fans have wide spread usage from heavy industry applications to computer cooling systems. Hence design of axial fans became more important in recent years due to performance and noise issues. In general, high pressure rises or high flow rates are desired depending on the application. Therefore while high pressure with a reasonable flow rate is provided, noise level must be considered. Main source of noise generated in an axial flow fan is mainly on aerodynamic loads on fan blades. Fans with improper intake geometry suffer from poor inflow conditions such as asymmetric velocity profiles and fluctuations leading increase in unsteady forces acting on blades. In other words, with proper blade profile, low-noise conditions can be obtained without giving up high aerodynamic performance.

In this study, point cloud created by Özdemir [1] is converted into solid model in CAD format and then flow field is defined by computational grids. Using k-ε and LES are used as turbulence models, simulations are carried out for different rotational speeds and pressure rises in order to determine fan performance.

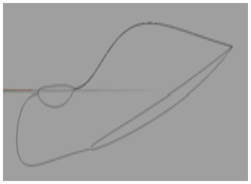


2. CAD OBJECT AND GENERATION



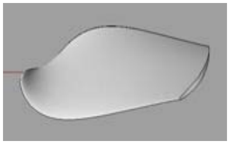
The cloud points of blade

The point cloud of a single blade given by Özdemir [1] is imported into a commercial software which is used to create 3D geometry.



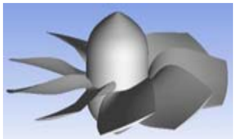
Edges curve of the blade

Then the surface of the area between the curves is generated.



Blade 3D geometry

The edge curves are created by combining points with line segments.



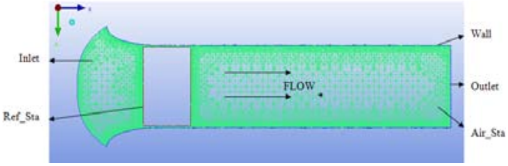
Rotor 3D geometry

The surfaces are constructed, rotor part is formed.

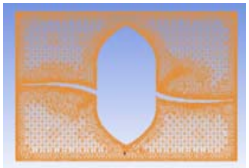
To conduct numerical analysis, the rotor is surrounded by a round duct with the dimensions given in table below.

Inlet Diameter	152 mm
Outlet Diameter	103 mm
Length	5000 mm

After CAD object is imported in ".igs" file format, a topology check is performed to check gaps in the geometry. This operation is necessary in order to have solid geometry fully enclosed so that the flow domain can be properly meshed separate from solid geometry. Once topology check is accomplished, geometry is then partitioned into related families. Since sliding mesh technique is used in this study, stationary and rotating domains are meshed separately. While AIR_ROT, ROTOR and REF_ROT families are defined for rotating domain, INLET, OUTLET, REF_STA, WALL and AIR_STA families are defined for stationary domain.



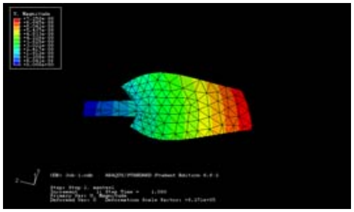
The grid of stationary domain



The grid of rotating domain

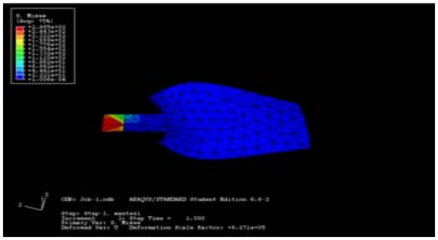
4. ANALYSIS, RESULT AND DISCUSSION

Structural Analysis



Magnitude of displacement distribution on blade

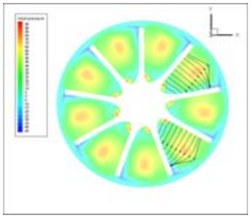
As a result, since no friction and deformation is noted, it can be said that Etal171 is suitable for fan's endurance.



Magnitude of stress distribution on blade

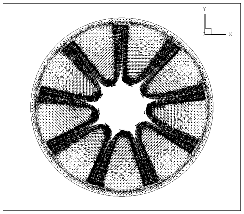
CFD Analysis

All the velocity and pressure distributions are calculated on the axial fan for the given boundary conditions. Simulations are carried out for five pressure values: 80 Pa, 90 Pa, 100 Pa, 110 Pa and 120 Pa at two rotational speeds: 720 rpm and 1080 rpm. Output figures are exported for 100 Pa.



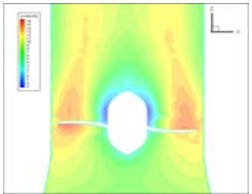
Total pressure distribution on XY plane at 720 rpm

Total pressure distribution can be seen on left and it is observed that streamlines are moving through –z. Furthermore, below, it is seen that flow vectors are uniform in the radial direction.

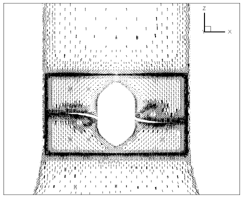


Flow vectors on XY plane at 720 rpm

Below, z component of velocity distribution is seen and as expected, velocity values reach their maximum at the tip of the fan blades due to centrifugal forces.

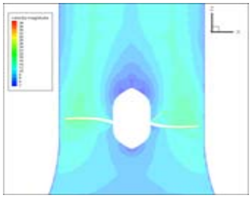


z-velocity distribution on XZ plane at 720 rpm

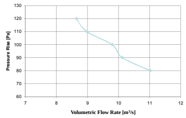


Flow vectors on XZ plane at 720 rpm

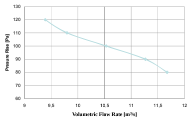
As mentioned below, velocity magnitude is maximum at the tip. After leaving the blade, velocity of the flow decreases.



Velocity magnitude distribution on XZ plane at 720 rpm



Performance curve of the fan at 720 rpm



Performance curve of the fan at1080 rpm

6. REFERENCES

[1] **I. B. Özdemir,** 2010. Private Communication.
[2] **F. Bleier,** 1998. Fan Handbook, McGraw Hill.
[3] **T. Köktürk,** 2005. Design and Performance Analysis of a Reversible Axial Flow Fan, *M.Sc. Thesis*, The Graduate School of Natural and Applied Sciences of Middle East Technical University.
[4] **S.H. Liu, R.F. Huang, C.A. Lin,** 2010. Computational and Experimental Investigations of Performance Curve of an Axial Flow Fan Using Downstream Flow Resistance Method, *Experimental Thermal and Fluid Science* (2010).
[5] **A. Maaloum, S. Kouidri, R. Rey,** 2004. Aeroacoustic Performance Evaluation of Axial Flow Fans Based on the Unsteady Pressure Field on the Blade Surface, *Applied Acoustics*, 65, 367-384.
[6] **B. Eck,** 1973. Fans, Pergamon Press.
[7] **K. Küçükcoşkun,** 2009. Calculations of Flow and Noise Propagation in Centrifugal Fans, *M.Sc. Thesis*, Institute of Science and Technology, Istanbul Technical University.
[8] **E. L. Blades, D. L. Marcum,** 2007. A Sliding Interface Method for Unsteady Unstructured Flow Simulations, *International Journal for Numerical Methods in Fluids,* 53, 507–529.
[9] **G. Constantinesco,** Large Eddy Simulation II 058:0268 Turbulent Flows, 1-5.
[10] **L. Davidson,** 2010. Mechanics of Solids & Fluids. Part II: Fluids Mechanics, Göteborg: Chalmers University of Technology, 43-45.
[11] **S. Pope,** 2000. Turbulent Flows, Cambridge: Cambridge University Press, 110-145.
[12] **M. Germano, et al.,** 1991. A Dynamic Subgrid-Scale Eddy Viscosity Model, *Phys. Fluids* **A, 3,** 1760-1765.
[13] **K.A. Hoffman,Chiang S.T.,** 2000. Computational Fluid Dynamics Vol III, 146–152.
[14] **M. Uygun,** 2004. Turbulence Modeling for Computational Fluid Dynamics, Part 1 Conceptual Outlook, *Journal of Aeronautics and Space Technologies*, **Volume I Number 4**, 20-25.
[15] **S. Kahraman,** 2010. Calculation of Turbulent Flow in a Curved Duct, *M.Sc. Thesis*, Institute of Science and Technology, Istanbul Technical University.
[16] **D. Gizli,** 2009. Calculations of Flow and Noise Propagation in Axial Fans, *M.Sc. Thesis*, Institute of Science and Technology, Istanbul Technical University.