Computational Analysis of an Axial Fan Using Large Eddy Simulation

Selahattin Doğramacı

I.T.U. School of Mechanical Engineering Mechanics Division, Fluids Laboratories

Abstract

It is well known that the aero-acoustic noise of low-Mach-number axial fans mainly occurs due to inflow disturbances and the turbulent flow and separation developed over the blades. Fans with improper intake geometry often suffer from poor inflow conditions which varies from spatially asymmetric velocity profiles to ingested eddies that fluctuate with respect to time and lead to random forces acting on the blades. This produces broadband gust noise and, in most cases, trigger selectively organized structures over the blades causing tonal self-noise.

The Unsteady-Reynolds-Averaged-Navier-Stokes (URANS) is one of the methods used to predict noise radiation from fans. However, URANS often fails to predict noise spectra in the case where the broadband components dominate the upstream noise level, unless the turbulence model is properly tuned. Large Eddy Simulation (LES) is, however, much more suitable for predicting broadband noise as it directly computes interactions of eddies larger than the numerical grid, from which turbulence essentially originates. The work proposed here is concerned with performing the LES prediction of the sound sources of these fans.

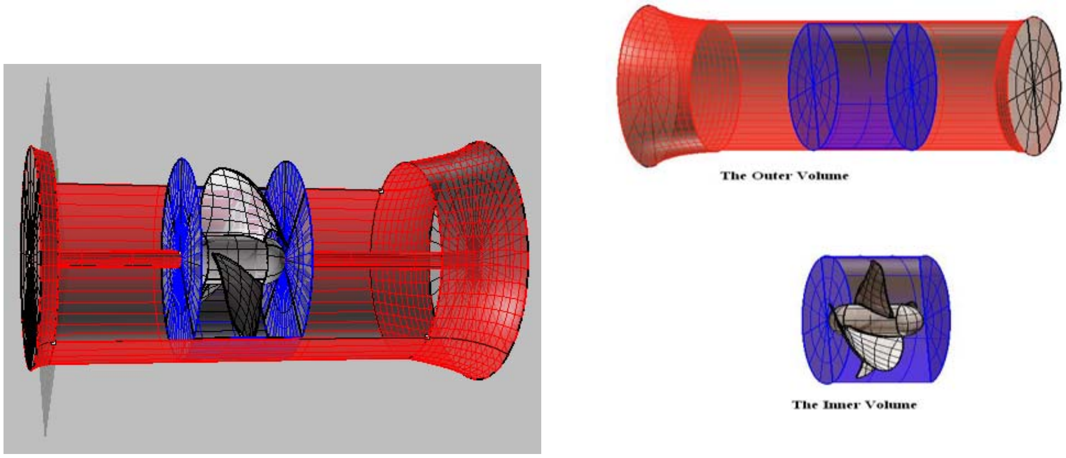
1. Introduction

Axial fans give out air parallel to the axis of rotation, which means air enters into the fan in the axial direction and leaves it in a path on which its axis is the same as the axis of the fan. However, although fluid particles move along the axis, they also have some radial velocity. Thus, as it leaves the blade, fluid particle moves on a rotational path. This rotational motion diminishes in time and after some time, radial velocity component vanishes. Because of this rotational motion, any equipment that will disturb the flow in the fan outlet zone is not used, which can be concluded to the use of fan motor in the inlet side of the fan.

Axial fans are usually made of one block unit but sometimes can consist of more units which are merged in order to make the fan. These units can be listed as; in the center, mounted on the electrical motor shaft, there is a cylindrical shaped 'hub', 'blades' which are attached to the hub and inlet/ outlet zones which condition the flow coming in and going out.

2. The Meshing Process

The crucial point in the meshing procedure of the fan geometry is to identify whether the surface of the geometry is critical to numerical solution or not. For instance, the pipe, which the fan model is put in, just closes the area and acts as a stationary wall during the analysis, so it does not require a detailed mesh. However, the rotating parts (blades for instance), inlet and outlet zones are highly influential on the result of the analysis and they are meshed with more detailed mesh parameters. Moreover, as the sliding mesh procedure requires two different mesh families, the model is divided into two parts in the meshing: one is the inner volume which involves the fan and its vicinity zone as closed cylindrical shape, and the other one is the rest of the pipe geometry which will be called as the outer volume. ANSYS ICEM CFD 10.0 is used as software for meshing both the inner and outer volumes.



(a) **(b)**

Figure 2.1 Meshed Model ;

a) Merged, b) Separately

2.1 Meshing the Inner Volume

Unstructured meshes are used in the inner volume due to their ease to apply. Trigonal elements are used on surfaces and tetrahedrals in the fluid zone of the volume. There are 38506 trigonal 846445 tetrahedral elements in the inner volume.

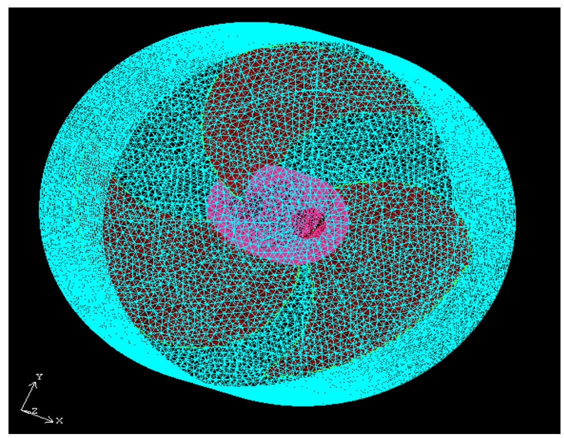


Figure 2.2 Mesh of the inner Volume

2.2 Meshing the Outer Volume

Unstructured meshes are used in this volume, too. There are 108144 trigonal and 3420134 tetrahedral elements in the outer volume. The total mesh number of the two volume is 4266579.

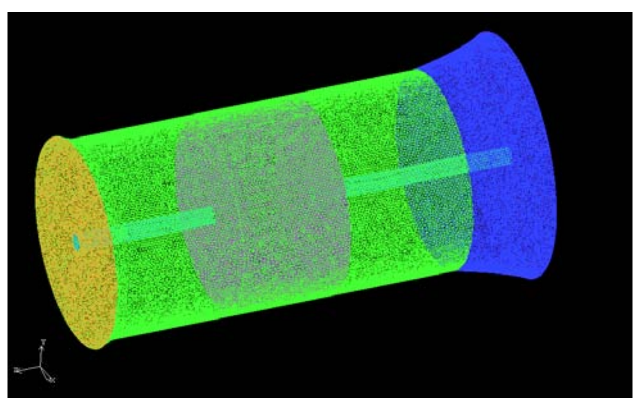


Figure 2.3 Mesh of the Outer Volume

2.3 The Sliding Mesh

Fans include rotating parts and fluid zones. While the moving parts like hub and blades are rotating with some angular velocities, some other parts like walls are stationary. As the different parts have different velocities, in these kinds of applications sliding meshes are used as boundaries between rotating and stationary parts. As the two inner and outer volumes are meshed separately, they are merged with the TGRID software which runs under FLUENT.

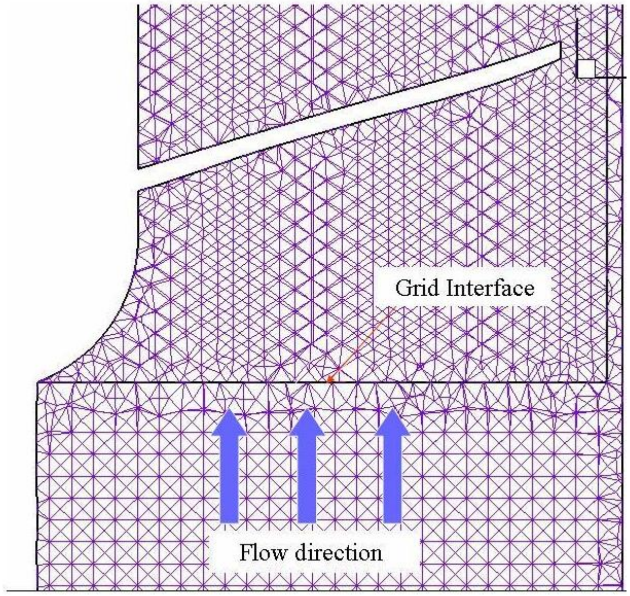


Figure 2.4 Sliding Mesh

3. Computational Fluid Dynamics (CFD)

Fluent V 6.2 is used as CFD solver. Large Eddy Simulation (LES) is chosen as viscous model. LES is used because we need time dependent solution for aero-acoustic solution and LES is not highly dependent to geometrical conditions.

Segregated solution method is used FLUENT in which mass, momentum and energy equations are solved in order. 2nd-Order Implicit method is used for time discretization and spatial derivatives are calculated Node-Based.

3.1 Boundary Conditions

In CFD one of the major difficulties and property that determines the accuracy of the solution is definition of boundary conditions. In this paper, surfaces that rotates relatively are defined as “moving wall”. Moreover, as they are dependent to the fluid around them and as they rotate, they are defined as “Relative to Adjacent Cell Zone” and “Rotational Motion”.

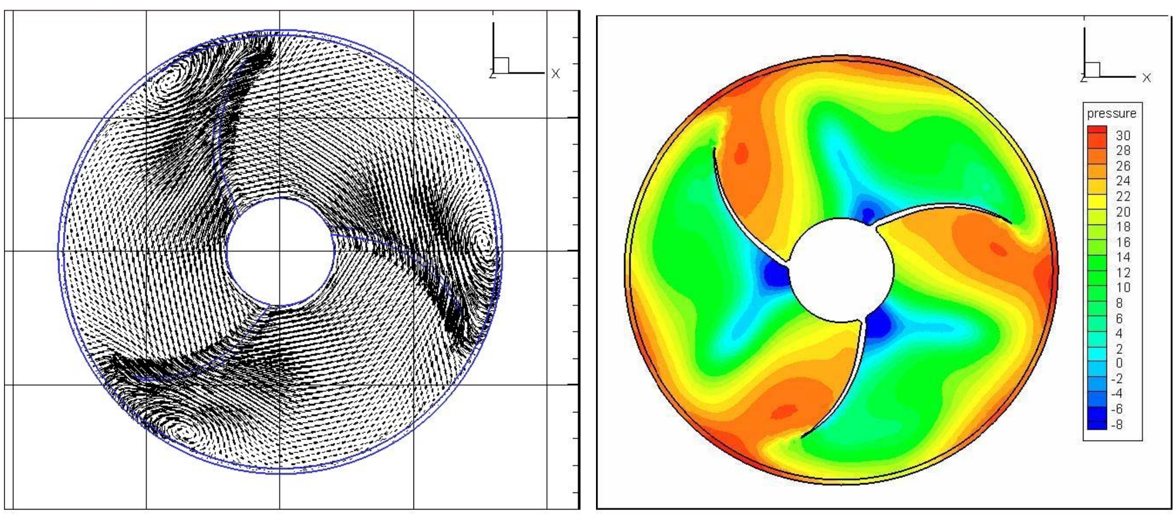
Pipe walls are defined as “Stationary wall” and the inlet and outlet are defined as “pressure inlet” and “pressure outlet”. In the end, different flow rate values at the outlet for different pressure outlets are obtained and by using them, fan performance curve is evaluated.

Fluid zone in the inner volume is defined as “moving mesh” and -1200 rpm angular velocity in z-direction. However, fluid in the outer volume is defined as “stationary”. In order to give sliding mesh property, the same surfaces in the inner and outer volume families are defined as “interface”, and then merged with “grid interface”.

4. Results and Discussion

4.1 Results from the Solver

As the solution iterated 100 times and the fan completed a full turn, following results are taken from different axis and cross-sections.



(a) **(b)**

Figure 4.1 Cross-section in z-plane

a) velocity vectors, b)pressure contours

As it can be seen from Figure 4.1 (a), flow is organized in radial direction and there occurs only three vortices behind every blade near the connection of hub. Moreover, Figure 4.1 (b) suggests that blades are exposed to positive static pressure from their flow direction sides.

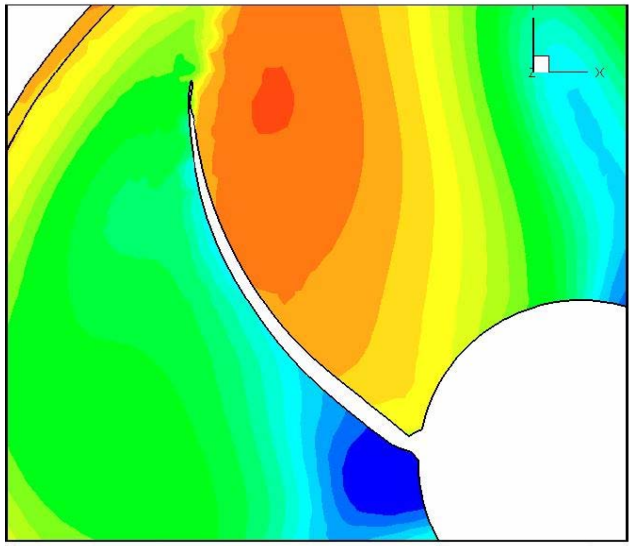
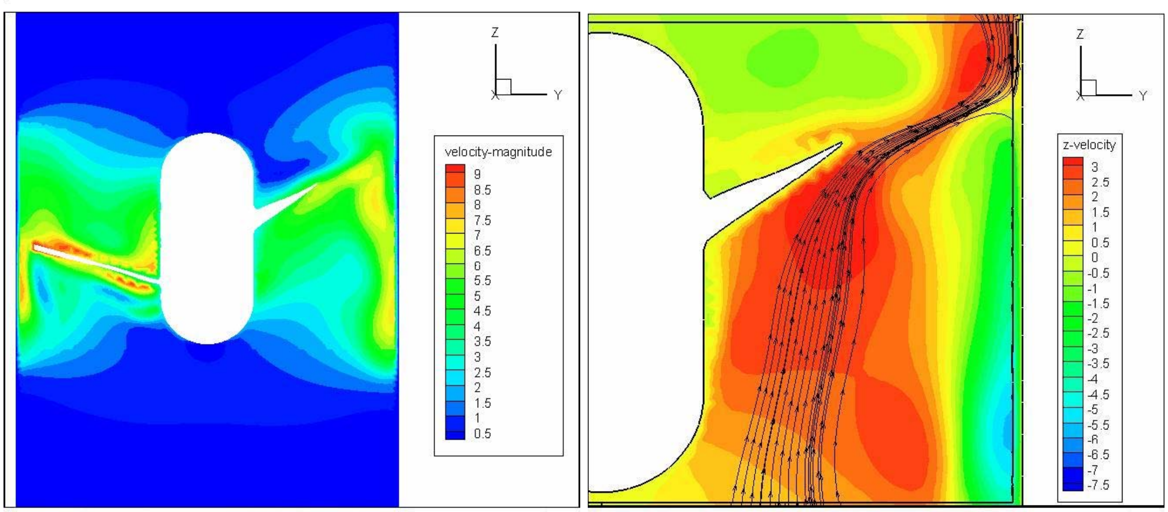


Figure 4.2 Closer view of the pressure contours

It is obvious from the Figure 4.2 that there is negative pressure on the back side of the blade. Moreover, the lowest pressure occurs at the connection of the back side of the blade and the hub, and there is a uniform pressure distribution on the blade which can be concluded that there is no stall on the blade.

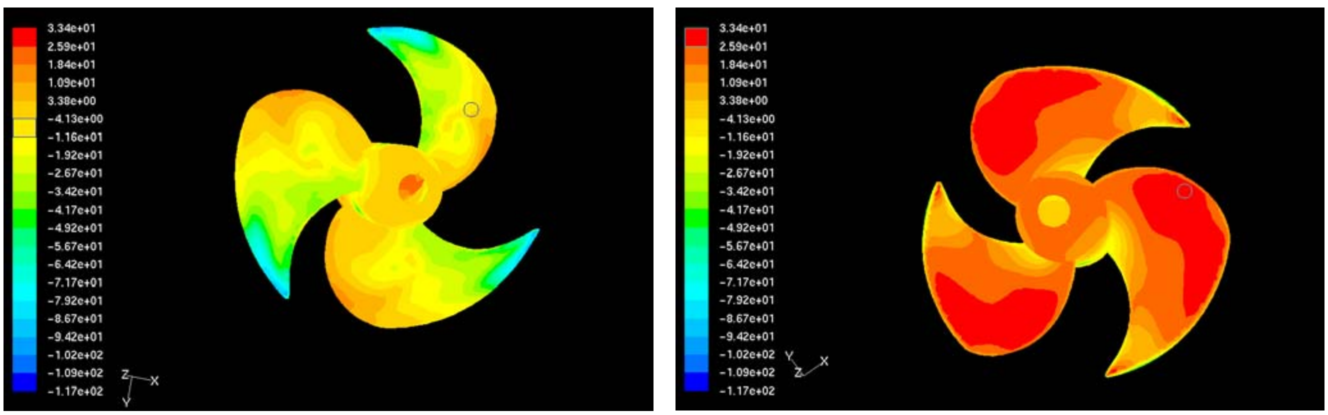


(a) **(b)**

Figure 4.3 Velocity contours on x-plane

a) velocity magnitude, b) z-velocity and streamlines

Figure 4.3 (a) shows the velocity magnitude distribution x plane at x=0. Fluid particles get faster on their way from inlet region to blades and they have their maximum around the blades. Moreover, after leaving blades, they slow down as they go to the outlet region. In Figure 4.3 (b), z component of velocity of the fluid particles are displayed. It can be concluded from the figure that as fluid enters blades, particles are scattered towards the wall due to the centrifugal force. Moreover, because of this centrifugal effect, they become faster and faster as they get nearer to the wall.



(a) **(b)**

Figure 4.4 Pressure distributions on blades

a) Suction side, b) Blowing side

Figures 4.4 (a) and (b) shows the pressure distribution on the front and back surfaces of blades. As it is obvious from figures, on the front side, pressure is rising from hub to the tips of the blades, while there is negative pressure on the back side of the blade surfaces.

4.2 The Fan Performance Curve

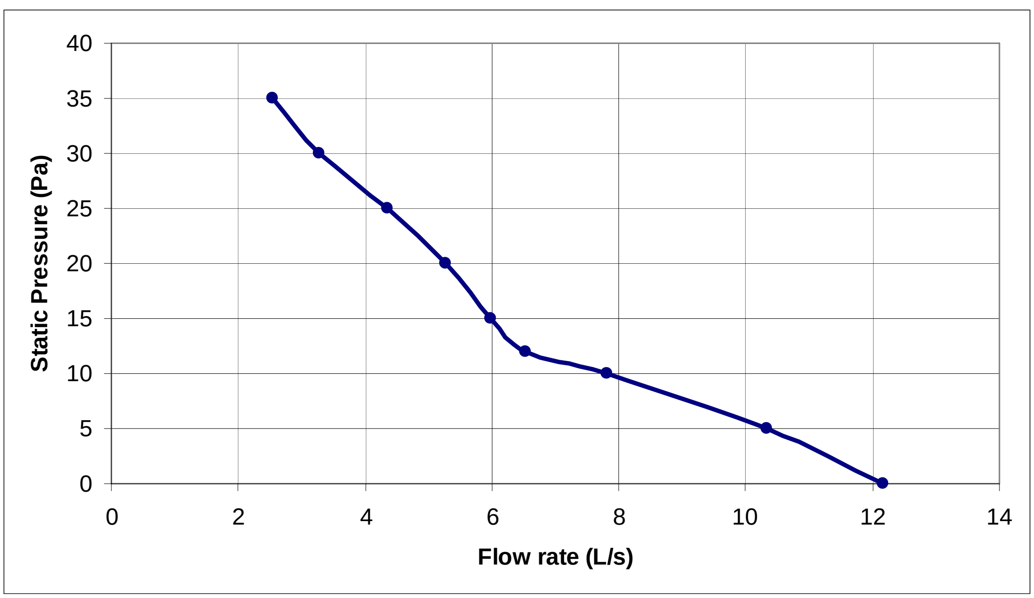
For the performance curve, inlet total pressure is set constant to 0 Pa and the outlet static pressure is changed from 0 to 40 Pa. Following table shows the results of these values:

Outlet Pressure P=0		Outlet Pressure P=5		Outlet Pressure P=10	
Inlet Velocity	0	**Inlet Velocity**	0,01	**Inlet Velocity**	0,04
Outlet Velocity	0,67	**Outlet Velocity**	0,57	**Outlet Velocity**	0,43
Area	0,018145	**Area**	0,018145	**Area**	0,018145
Flowrate (L/s)	12,15771	**Flowrate (L/s)**	10,34312	**Flowrate (L/s)**	7,802710

Outlet Pressure P=12		Outlet Pressure P=15		Outlet Pressure P=20	
Inlet Velocity	0,05	**Inlet Velocity**	0,05	**Inlet Velocity**	0,05
Outlet Velocity	0,36	**Outlet Velocity**	0,33	**Outlet Velocity**	0,29
Area	0,018145	**Area**	0,018145	**Area**	0,018145
Flowrate (L/s)	6,532502	**Flowrate (L/s)**	5,988126	**Flowrate (L/s)**	5,262293

Outlet Pressure P=25		Outlet Pressure P=30		Outlet Pressure P=35	
Inlet Velocity	0,05	**Inlet Velocity**	0,02	**Inlet Velocity**	0,02
Outlet Velocity	0,24	**Outlet Velocity**	0,18	**Outlet Velocity**	0,14
Area	0,018145	**Area**	0,018145	**Area**	0,018145
Flowrate (L/s)	4,355001	**Flowrate (L/s)**	3,266251	**Flowrate (L/s)**	2,540417

As the outlet pressure exceeds 35 Pa, the fan is not able to blow out air but instead, fluid comes in from the outlet.



5. Conclusion

Although URANS is a model that can be used for aero-acoustical analysis of an axial, LES is used in this paper work, due to its superiorities. By using the Fluent Software, an aero-dynamical analysis has been conducted and results are taken. It can be concluded that, as the highest pressure on blades occur on the tip of them. Moreover, the highest velocities of the fluid in the pipe are reached around the blades, between blade tips and wall. Furthermore, nearly uniform pressure distribution on the blades suggests that there is nearly no stall from the blades. Finally, with different outlet pressures, fan performance curve is built and static pressure – flow rate relation is gained.

References

1. **Doğramacı, S.,** 2007. Derin Dondurucularda Kullanılan Eksenel Hava Üfleyicisinin Aero-Akustik Önceliklerle Tasarlanması, *Lisans Tezi*, İ.T.Ü Makine Fakültesi, İstanbul. Haziran

2. **Köktürk, T.,** 2005. Design And Performance Analysıs Of A Reversible Axial Flow Fan, *Yüksek Lisans Tezi,* O.D.T.Ü Fen Bilimleri Enstitüsü, Ankara.

3. **Reese, H. Kato, C. Carolus, T. H.,** 2007. Large Eddy Simulation of Acoustial Source in a Low Pressure Axial-Flow Fan encountering Highly Turbulent Inflow. *Journal of Fluids Engineering.* [FE-05-1311] 003703JFG.

4. **CFD Online.,** 2007. *http://www.cfd-online.com/ W/index.php? title=Fluent_FAQ & printable=yes*. Mart 22.

5. **Colonius, T. ve Lele, S. K.** 2004. Computational aeroacoustics: progress on nonlinear problems of sound generation, *Progress in Aerospace Sciences*, **40**, 345-416.

6. **Envia, E.** 2001 Fan noise reduction: An overview, *the39th Aerospace Sciences Meeting and Exhibit*, Nevada, January 8–11, 2001.9