ENINEERING PROJECTS IN AERO-DYNAMICS
Eng. Pro. in Aero-Dyn. 2007; June

Aero-dynamical Analysis of a Radial Fan with Large Eddy Simulation

Korcan Kucukcoskun

Dinçer Gizli

Department of Advanced Technologies in Engineering, Istanbul Technical University, Turkey

Abstract

Large Eddy Simulation (LES) has been performed in a centrifugal fan to determine the aero-dynamical problems on the flow to find out aero-acoustic noise sources. The flow domain has 4 million unstructured grids with a sliding interface. Results are taken at blade – wedge interactions and at sharp corners in the suction side at certain time steps and all are compared.

1. Introduction

Centrifugal turbomachines are common devices used in many flow control applications due to their ability to achieve relatively high-pressure ratios in a short axial distance compared with axial fans. They are often found in gas turbine engines, heating ventilation and air-conditioning systems (HVAC) and pumps. Noise generated by these machines causes one of the serious problems, related to their widespread use.

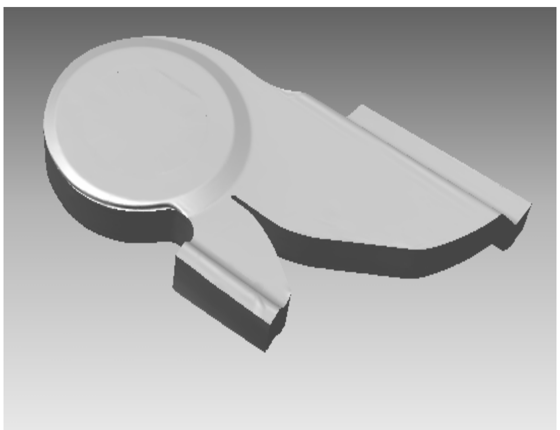
The noise is often dominated by tones at the strong interactions between the flow discharged from the impeller and the casing. As Jeon stated, the unsteady force fluctuation is generated by the periodic interaction of the impeller blade with the wedge of the casing where force fluctuation is the main source of acoustic pressure [1].

In order to understand the acoustic source of a centrifugal fan and the casing, detailed information of the flow field is needed. Therefore, the numerical method to analyze the flow field of the centrifugal fan is necessary.

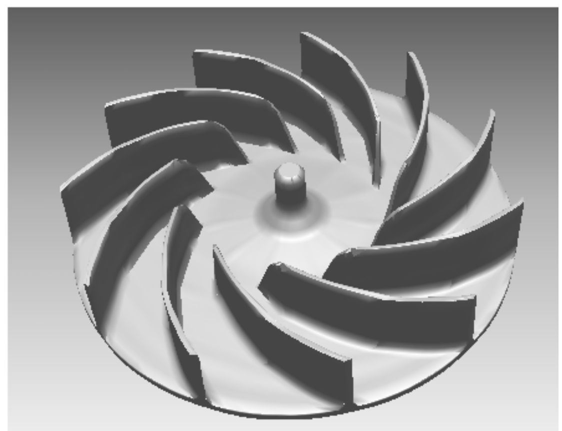
Since there is both a rotating and stationary part in the time dependent flow field, a special meshing process is required, Blades concluded sliding interface method relieves computationally expensive grid deformation, remeshing, and hole cutting procedures which method has been developed for simulations involving relative rotational grid motion [2].

2. Model Description

The fan model is selected as freezer fan which is located at the evaporator side with its original casing. The original fan and case are scanned in ROTAM, ITU to provide the point cloud of the model which is the fundamental of the numerical grid. The point cloud is converted to a 3-D geometry with Rhinocerous and RapidForm which are both commercial programs by building surfaces on the point cloud. Figure 1 shows the case and the impeller in 3-D generated from point cloud.



(a)

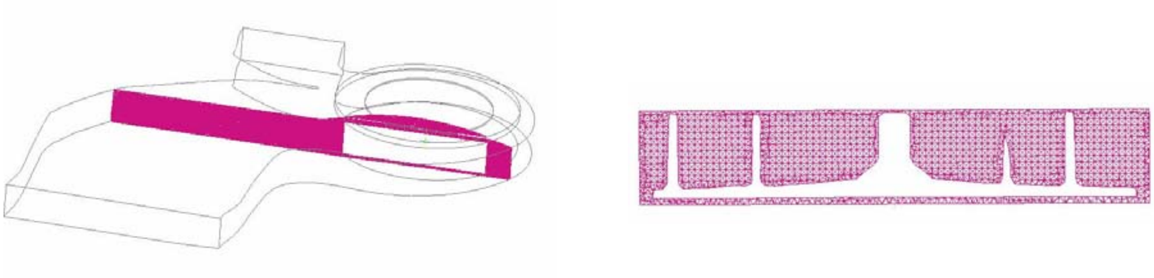


(b)

Figure 1: 3-D model (a: The case b: The impeller)

ANSYS-ICEM CFD is used to the meshing process of the domain.

Because of the sliding mesh is used, the fan and the case are both meshed separately to create "grid interface" between two separate domains. A simple cylinder surrounding the impeller is formed which fits entirely inside the case. Since the fan and case geometry are complicated unstructured mesh is used both for in the case and surrounding the impeller.



(a) (b)

Figure 2: Unstructured domains (a: Stationary b: Rotating)

Figure 2 shows the domains of the flow in which 3,25 million elements in the case and 800 thousands in the rotating part. The domain is formed from total 4 million unstructured elements which is merged in T-Grid.

3. Numerical Method

The numerical simulations are conducted using FLUENT 6.2 with LES as the viscous model. Since the domain is unstructured, node-based scheme is used. The analysis is performed by using forward differencing for spatial progress of the momentum equations, and second-order accurate implicit scheme for the time auction.

$$\eta / l_o \approx \mathrm{Re}^{-3/4} \qquad \text{(Eq. 1.a)}$$

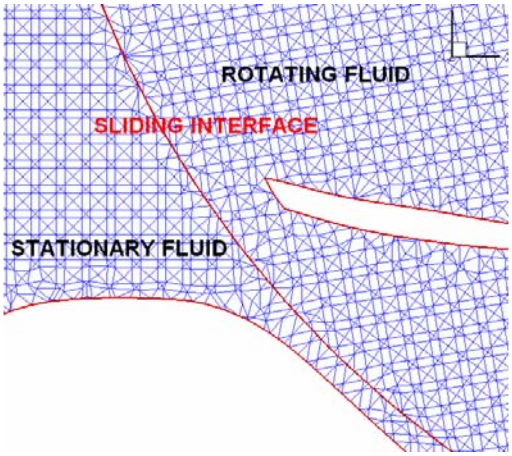
$$u_\eta / u_o \approx \mathrm{Re}^{-1/4} \qquad \text{(Eq. 1.b)}$$

$$\tau_\eta / \tau_o \approx \mathrm{Re}^{-1/2} \qquad \text{(Eq. 1.c)}$$

As the Reynolds number is 42000, the time step size is calculated from Kolmogorof Scales from Eq. 1.a , 1.b and 1.c for high Reynolds numbers where η is the smallest eddy scale and u_η and τ_n are smallest velocity and time scales where t_o, u_o and τ_o are the largest eddies as Pope explained [3]. The time step is estimated 2x10^{-4} s.

4. Boundary Conditions

Since the domain contains both a rotating and a stationary part, two different fluid zones are created and both are marked as air. The inside fluid zone is rotating one and the boundary condition is seleceted rotationally moving zone with 1990 RPM where as the outer one is selected as stationary fluid.



However the rotating is produced by fan which is described as "moving wall", the rotation of the fan is defined with a relative to the adjacent zone coommand with "0 relative velocity" which means a 1990 RPM absolute rotational velocity. Grid interface is created between two fluid zones for the sliding mesh technique. Figure 3 shows the rotating fluidzone, stationary fluid zone and the sliding interface.

Figure 3: Sliding interface

Pressure boundary conditions are set at both inlet and outlet sections as Eralp and Gokturk are introduced [4].

For the inlet condition, a stationary fluid zone is defined with a semisphere region to provide a distinctive suction and also avoid the adjacency of the inlet zone and rotating fan. The boundary condition is selected as pressure inlet with both static and dynamic pressures are zero which defines a stationary ambient at the suction side.

Outlet condition is selected as pressure outlet with 5,10, 20 and 30 pa values which condition fixes the static pressure where as the dynamic pressure is provided by the fan.

Figure 4 shows the fan characteristics for given pressure values.

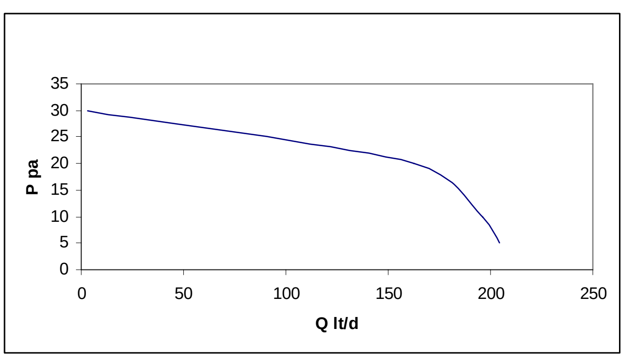
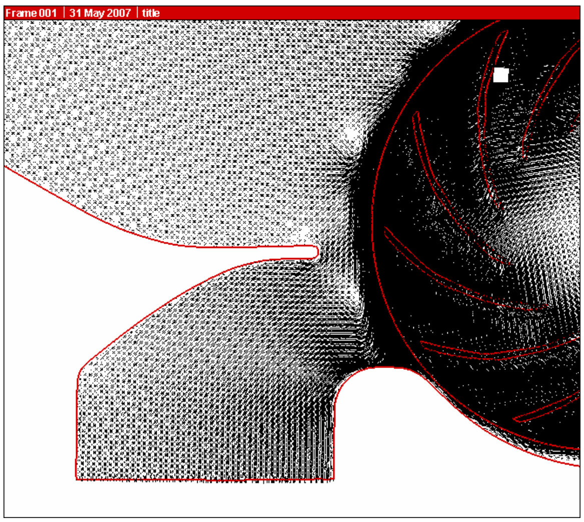


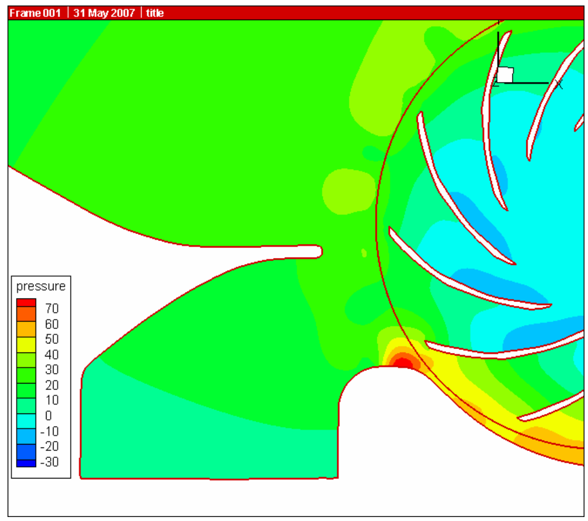
Figure 4: Fan Characteristics

5. Results

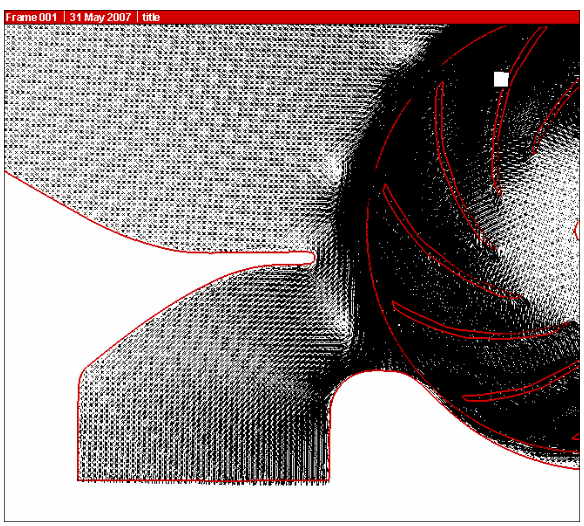
The unsteady pressure fluctuation in the domain between the impeller blade and the wedge on the case is analyzed. Analyze should run at least 5ms to reach a periodic steady state condition as Park's results [5], which is equal to at least 25 time steps with a $2x10^{-4}$ s time step size.

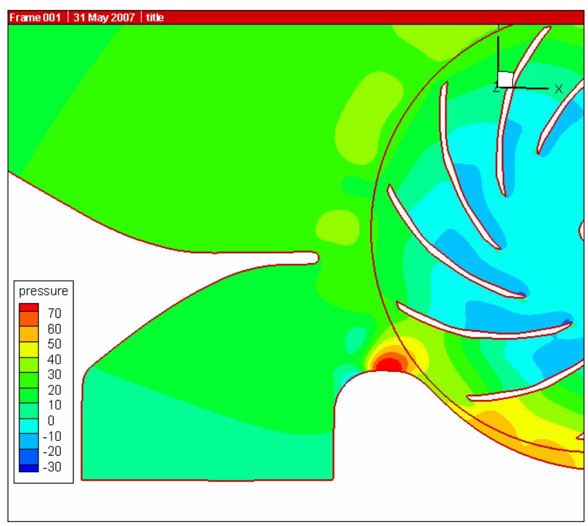
The one blade period is 14 time steps for the impeller rotating with 1990 RPM. The results are taken at 32^{nd}, 40^{th} and 45^{th} time steps which cover a blade pass time.



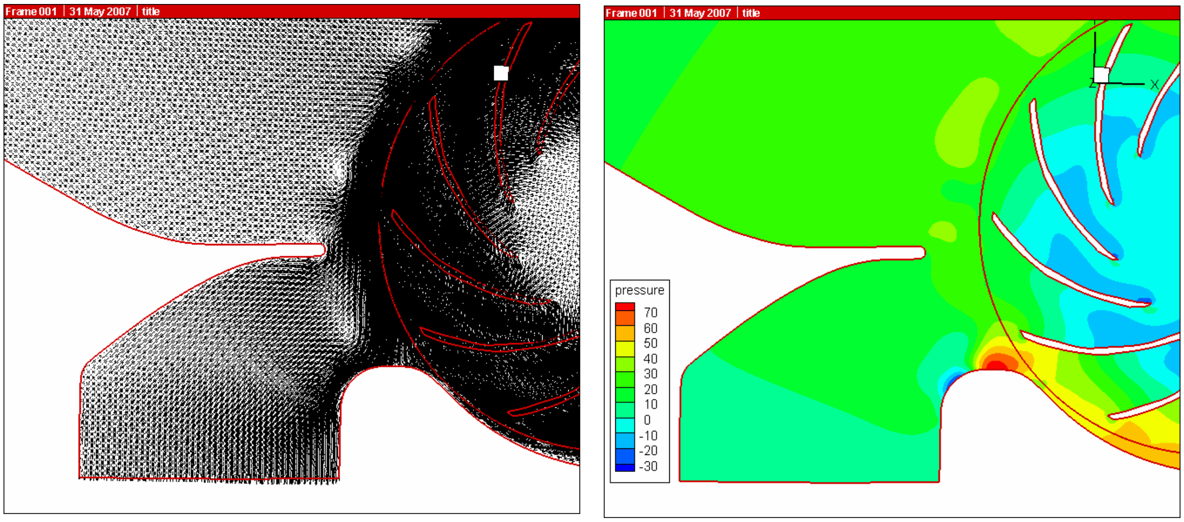


(a) (b)





(c) (d)



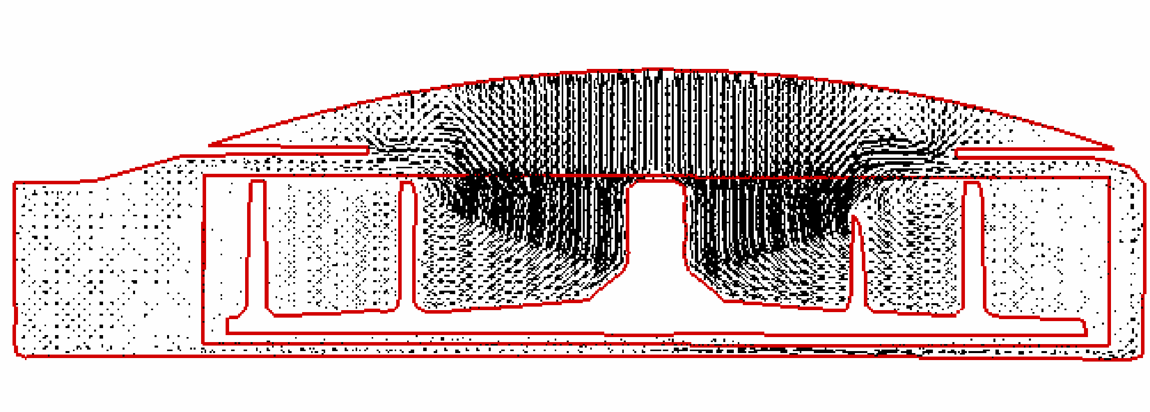
(e) (f)

Figure 5:Velocity vectors and Pressure contours at the blade-wedge intersection
(view on z-cut plane)
(a: Velocity vectors at 6.2 ms b: Pressure contours at 6.2 ms
c: Velocity vectors at 8.0 ms d: Pressure contours at 8.0 ms
e: Velocity vectors at 9.0 ms f: Pressure contours at 9.0 ms)

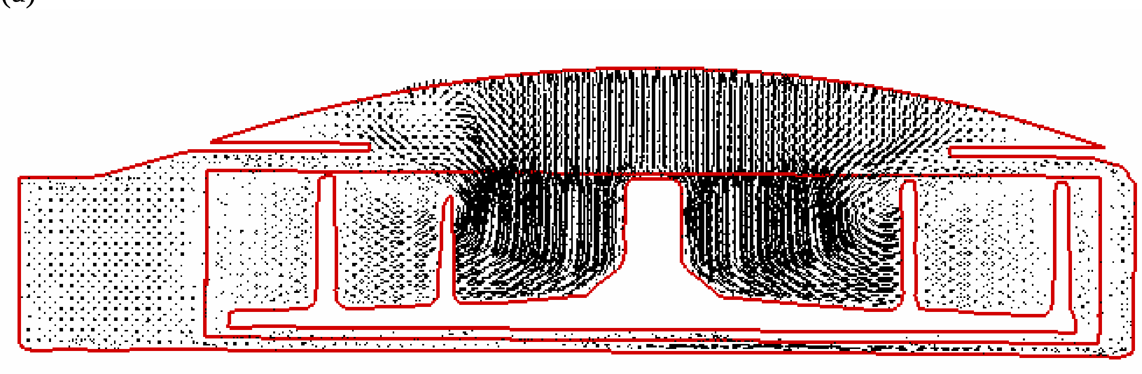
The stagnation and stalling is clearly seen in the velocity vector scheme at the wedge in figure 5.a, 5.c, 5.d. Also vortices around the blade tips are distinctive on both velocity and pressure scheme.

Figure 5.b, 5.d, 5.e contours show that the common location for microphone for an experimental study is in the first centimeter above the wedge.

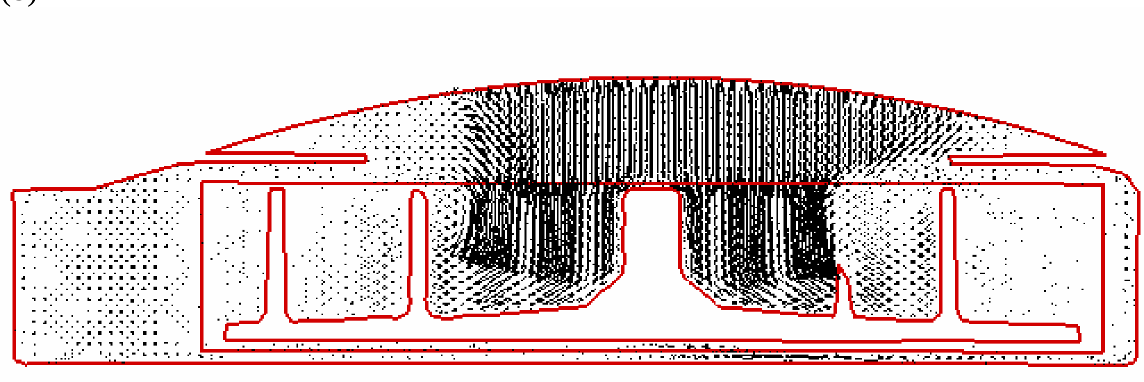
Another problem for the velocity fluctuation occurs just above the impeller at the suction side where the shape of the inlet section acts a wedge role in the suction. The vortices in the flow are aero-acoustic noise sources.



(a)



(b)



(c)

Figure 6: Velocity vectors in the suction side (view on x-cut plane)
(a: at 6.2 ms b: at 8.0 ms c: at 9.0 ms)

The vortices can be seen at the suction, just adjacent of the intake wedge. In figure 6.a the vortices are inside the domain which the fluctuation occurred by vortices is an aero-acoustic source, however in figure 6.b and 6.c the vortices are caused backflow at the suction side. All affects the fan performance and noise level and all are aerodynamic based aero-acoustics problems.

7. Conclussion

A LES analyze is performed in a centrifugal fan with an unstructured domain containing sliding interface between rotating and stationary fluids. Vortices and stagnation points are detected in the flow which are common locations for microphones in experimental studies. The sharp edge at the suction side and stagnation point at the wedge should be avoided with geometric design to prevent noise of the fan by keeping the performance of the fan without decreasing.

The analyze is run with one processor and unstructured domain which takes 20 minutes for one processor. Structured mesh and parallel processing can decrease the analyze time and periodic flow values can be taken for a better approach in aero-acoustic problems.

References

[1] Wan-Ho Jeon, Duck-Joo Lee, A numerical study on the flow and sound fields of centrifugal impeller located near a wedge, Journal of Sound and Vibration 265 (2003) 221–230.

[2] Eric L. Blades, David L. Marcum, A sliding interface method for unsteady unstructured flow simulations, International Journal for Numerical Methods in Fluids 53 (2007) 507–529.

[3] Stephen B. Pope, Turbulent Flows, Cambridge, 2000, pp. 185-186

[4] Cahit O. Eralp and Tolga Gokturk, Design and performance analysis of a reversible axial flow fan, Ankara, 2005, pp. 93-94.

[5] Jonghyun Park, A sound method for fan modeling, Fluent News, Summer (2005) 6-7.