Numerical Modeling of Turbulent Flow Inside the Centrifugal Blower

Majid Almas[1,2,*]

[1]Department of Mechanical and Materials Engineering, Florida International University, Miami, FL 33199, USA

[2]Department of Marine Engineering, King AbdulAziz University, 21589, Saudi Arabia

Abstract: In this paper, turbulent flow inside the centrifugal blower has been studied numerically. The computational simulation is conducted by employing the Reynolds Averaged Navier-Stokes (RANS) approach. The standard $k-\varepsilon$ model with enhanced wall treatment has been implemented for modeling the turbulent flow. Effects of different angular velocities have been studied on static pressure and mass flow rate. The numerical results shows that static pressure and mass flow rate significantly increase as the angular velocity of blades increases.

Keywords: Centrifugal blower, turbulent flow, RANS model, angular velocity, hybrid mesh.

1. INTRODUCTION

Many engineering problems involve rotating flow domains. One example is the centrifugal blower unit that is typically used in automotive climate control systems. For problems where all the moving parts (fan blades, hub and shaft surfaces, and so on) are rotating at a prescribed angular velocity, and the stationary walls (for example, shrouds, duct walls) are surfaces of revolution with respect to the axis of rotation, the entire domain can be referred to as a single rotating frame of reference. Lee and Lim [1] studied the development of an optimised design of a centrifugal blower consisting of various fan ribs, based on performance assessments following changes in the shape of its internal components. They evaluated the various components, such as the external cases and the rotating fan ribs placed in a variety of operating conditions, numerically and experimentally. Xu and Mao investigated the investigation of metal foam for controlling centrifugal fan noise experimentally [2]. Baloni *et al*. [3] studied the pressure recovery and loss coefficient variations in the two different centrifugal blower. Engeda *et al*. [4] studied the centrifugal pump with the sensitivity factor of tip clearance loss depending upon pump specific speed. They observed in the experiment that the pump with a specific speed was highly insensitive to the varying tip clearance and could be regarded as an ideal design for semi-open centrifugal pump impeller. Cao and Chu [5] reported that volute width and hub-volute clearance had a significant effect to the performance of centrifugal fan. In order to control the flow rate and the noise level, Bayomi *et al*. [6] reported that the straightener at the inlet of centrifugal blower increased the flow rate and reduced noise revel for the radial and backward impellers. There are more works [7-10] that investigated the different aspects of the centrifugal blower. In this study turbulent flow inside the centrifugal blower has been studied numerically using the standard $k-\varepsilon$ model. Effects of different angular velocities have been studied on static pressure and mass flow rate.

2. PROBLEM DEFINITION

This problem considers a 2D section of a generic centrifugal blower. A schematic of the problem and also the grid generation are shown in Figure **1**. The blower consists of 32 blades, each with a chord length of 13.5mm. The blades are located approximately 56.5mm (measured from the leading edge) from the center of rotation. The radius of the outer wall varies logarithmically from 80mm to146.5mm. The flow is simulated under no load, or free-delivery conditions when inlet and outlet pressures are at ambient conditions (0 Pa gauge). This corresponds to the maximum flow-rate of the blower when sitting in free air. The blades are rotating with angular velocities of 1500, 2000, 2500 and 3000rpm. The flow is assumed to be turbulent. Hybrid mesh including structured mesh for the outlet channel and hexahedral mesh for blade and suction regions. About 800000 mesh have been used for modeling the problem. Second order upwind scheme for turbulence kinetic and dissipation and that, the coupled scheme for advection have been applied. The convergence criteria for continuity is 10e-5 and for energy and turbulence equations is 10e-3.

*Address correspondence to this author at the Department of Mechanical and Materials Engineering, Florida International University, Miami, FL 33199, USA; Tel: (305) 348-1932; Fax: (305) 348-2569; E-mail: malma016@fiu.edu

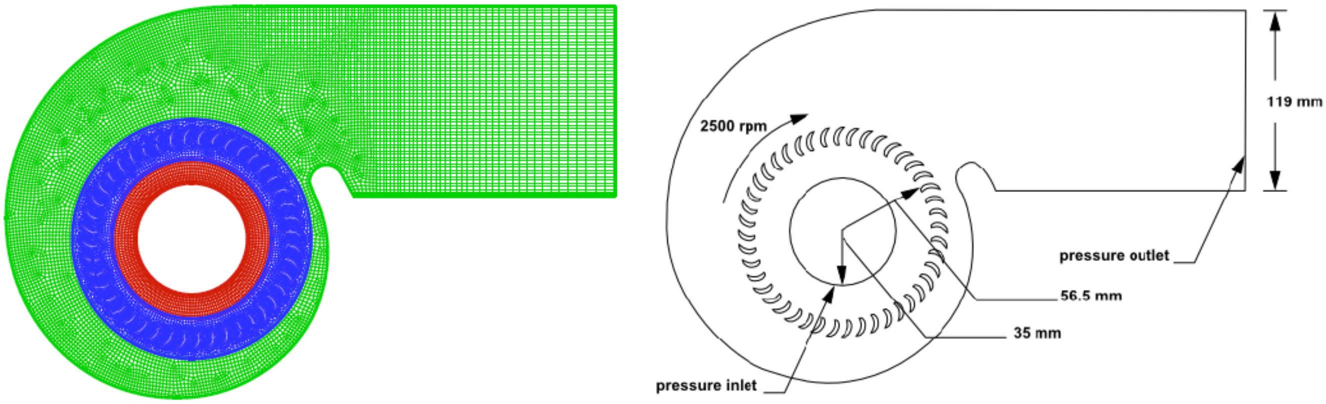

Figure 1: Schematic of the centrifugal blower and grid generation.

3. RESULTS AND DISCUSSIONS

Figure **2** shows the variation of the static pressure contour for the centrifugal blower for different angular velocities. As figures show the value of static pressures increase as the value of angular velocity increases. As seen at angular velocity of 1500rpm the maximum value of pressure is 69.56Pa. And this value increases to 125.3, 197.9 and 287.4Pa for angular velocities of 2000, 2500 and 3000(rpm) respectively. Figure **3** is also depicts the static pressure profile of centrifugal blower for different angular velocities. As seen in the figure there is direct relation between the angular velocity and static pressure. The values of static pressures increase significantly. As mentioned above the values of pressures rise from 69.56Pa to 287.4Pa as the angular velocities enhance from 1500rpm to 3000rpm.

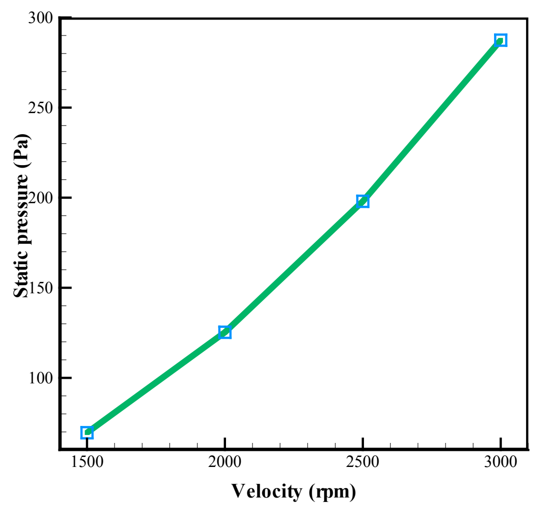

Figure 3: Static pressure profile of centrifugal blower for different angular velocities of 1500, 2000, 2500 and 3000 (rpm).

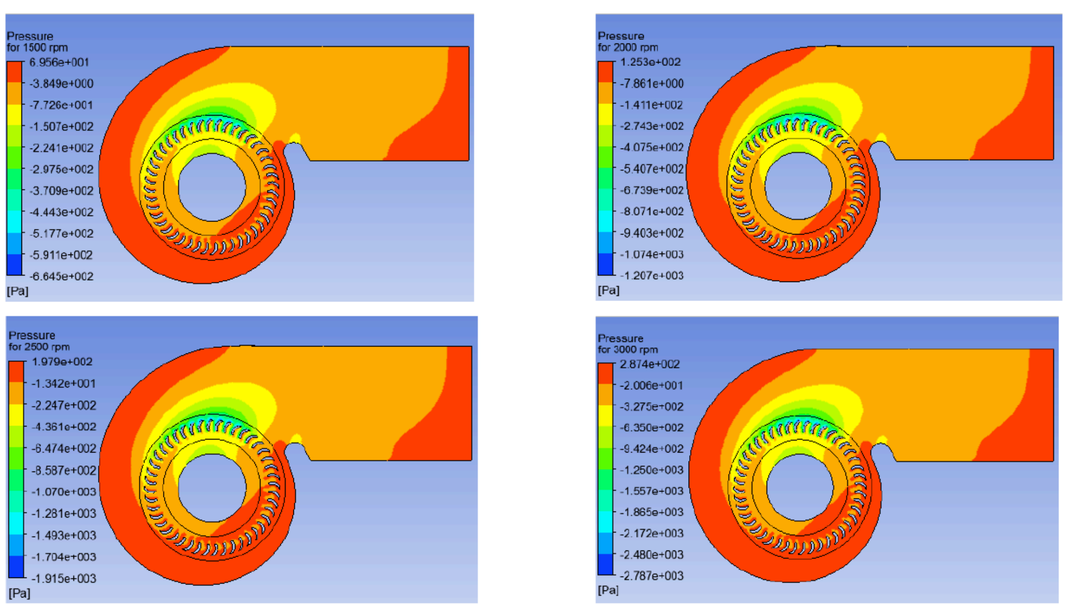

Figure 2: Contours of static pressure of the centrifugal blower for different angular velocities of 1500, 2000, 2500 and 3000(rpm).

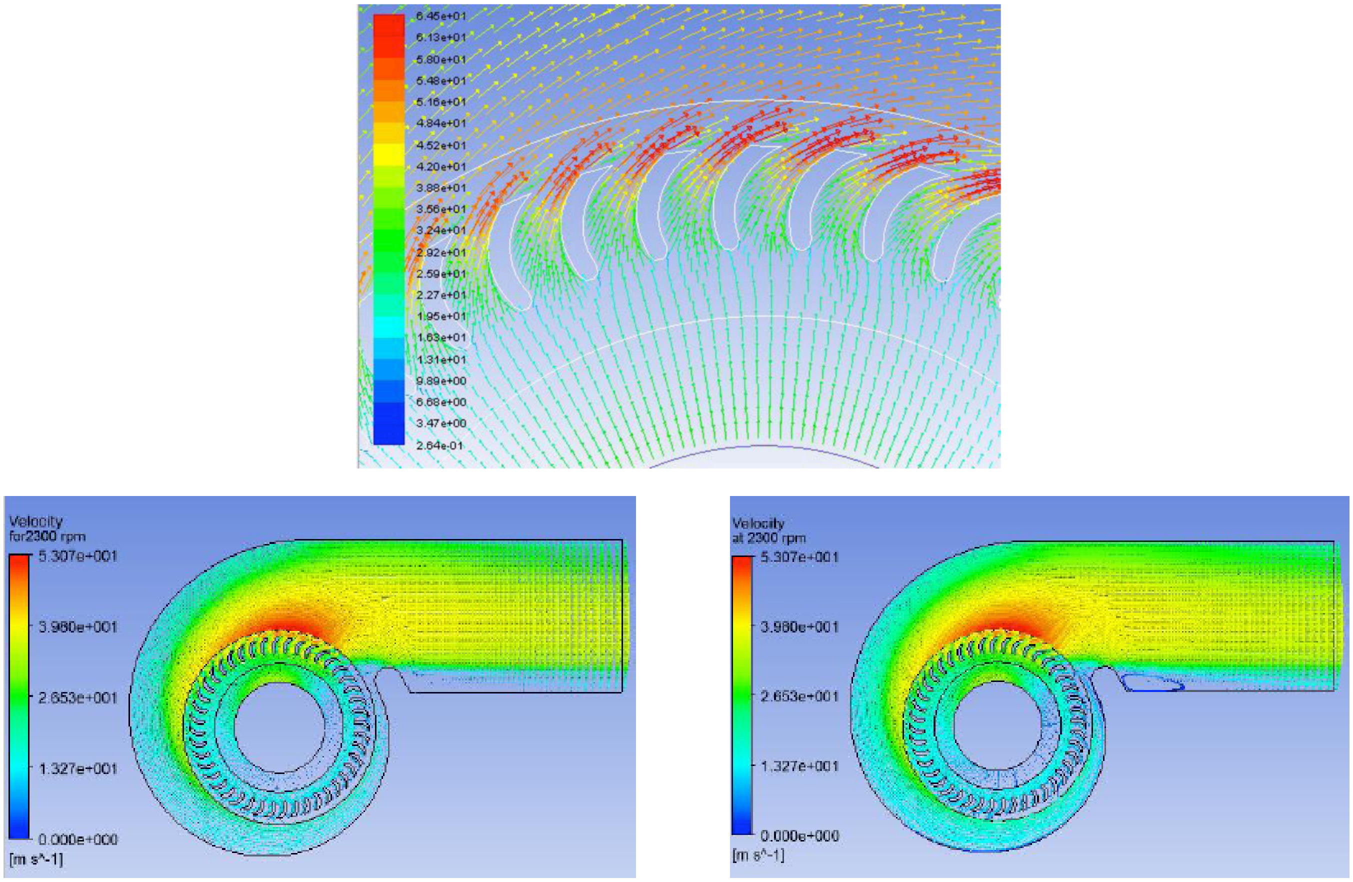

Figure 4: Velocity vector (bottom left) and streamline (bottom right) for angular velocity of 2500 (rpm).

Figure **4** displays the velocity vector and streamline for angular velocity of 2500rpm. As can be seen the velocity has its maximum values at the top tip of the blades as the flow discharges toward the outlet. This value has its lowest in the separation region which is clearly shown in velocity streamline. As shown the velocity vectors show an area of flow separation near the bottom of the duct. As can be seen there is a flow circulation on this area. In this figure the zoomed rotor blade region has also been displayed.

Figure **5** demonstrates the mass flow rate profile of centrifugal blower for different angular velocities of 1500, 2000, 2500 and 3000rpm. Again as seen in the figure the angular velocity variation has an important effect on mass flow rate at the outlet of the centrifugal blower. As results show the value of mass flow rate enhances significantly as the values of velocities increase. At angular velocity of 1500rpm the mass flow rate is 2.53kg/s and this value increases to 3.39, 4.27 and 5.15kg/s for angular velocities of 2000, 2500 and 3000(rpm) respectively. Results clearly show that inlet angular velocity of the blade has a significant effect on outlet mass flow rate.

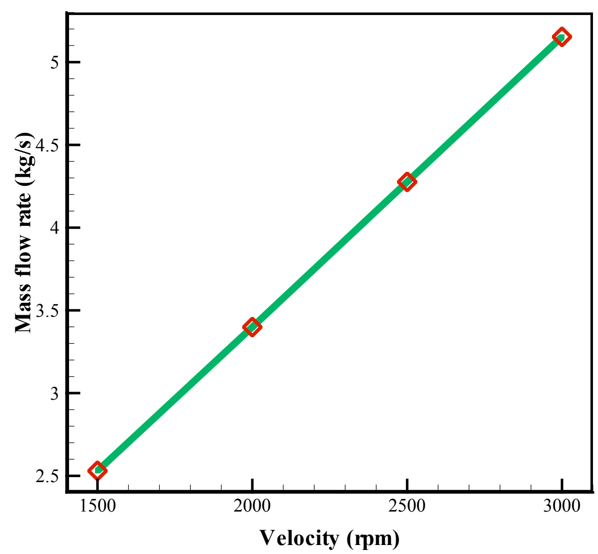

Figure 5: Mass flow rate profile of centrifugal blower for different angular velocities of 1500, 2000, 2500 and 3000 (rpm).

CONCLUSIONS

In this study a numerical modeling of a turbulent flow inside the centrifugal blower has been studied. The computational simulation is performed using standard $k-\varepsilon$ model RANS model [11]. Various angular velocities have been applied to investigate the complex turbulent flow inside the centrifugal blower. Numerical results revealed that inlet angular velocity has a significant effect on static pressure and mass flow rate. As results shown the values of mass flow rate and static pressure increase noticeably as the angular velocity increases.

ACKNOWLEDGMENTS

The author would like to thank both the Saudi Arabian Cultural Mission in Washington D.C. and King Abdulaziz University in Jeddah, KSA for their support.

REFERENCES

[1] Young-Tae Lee and Hee-Chang Lim. Performance assessment of various fan ribs inside a centrifugal blower. Energy 2016; 94: 609-622. http://dx.doi.org/10.1016/j.energy.2015.11.007

[2] Chen Xu and Yijun Mao. Experimental investigation of metal foam for controlling centrifugal fan noise. Applied Acoustics 2016; 104:182-192. http://dx.doi.org/10.1016/j.apacoust.2015.11.014

[3] Beena D, Baloni SA and Channiwala VK. Mayavanshi, Pressure recovery and loss coefficient variations in the two different centrifugal blower. Appl Energy 2012; 90: 335-343 http://dx.doi.org/10.1016/j.apenergy.2011.02.016

[4] Engeda A, Strate WP and Rautenberg M. Correlation of tip clearance effects to impeller geometry and fluid dynamics. ASME 1988 International Gas Turbine and Aeroengine Congress. Paper 88-GT-92 (1988) V001T01A040. http://dx.doi.org/10.1115/88-gt-92

[5] Cao S and Chu L. Experimental study on the matching between centrifugal impeller and volute. Chin Fluid Mach 1991; 10: 2-4.

[6] Bayomi NN, Hafiz AA and Osman AM. Effect of inlet straighteners on centrifugal fan performance, Energy Convers Manag 2006; 18: 3307-3318. http://dx.doi.org/10.1016/j.enconman.2006.01.003

[7] Li Chunxi, Wang Song Ling and Jia Yakui. The performance of a centrifugal fan with enlarged impeller. Energ Convers Manage 2011; 52: 2902-2910. http://dx.doi.org/10.1016/j.enconman.2011.02.026

[8] Qi Datong, Yijun, Liu Xiaoliang and Minjian Y. Experimental study on the noise reduction of an industrial forward-curved blades centrifugal fan. Appl Acoust 2009; 70: 1041-1050. http://dx.doi.org/10.1016/j.apacoust.2009.03.002

[9] Majidi Kitano APRIL. Numerical study of unsteady flow in a centrifugal pump. J Turbomach 2005; 127: 363-371. http://dx.doi.org/10.1115/1.1776587

[10] Sandra VS, Rafael BT, Carlos SM and Bruno PG. Reduction of the aerodynamic total noise of a forward curved centrifugal fan by modification of the volute tongue geometry. Appl Acoust 2008; 69: 225-232. http://dx.doi.org/10.1016/j.apacoust.2006.10.009

[11] Ghasemi E, McEligot DM, Nolan KP, Crepeau J, Tokuhiro A and Budwig RS. Entropy generation in a transitional boundary layer region under the influence of freestream turbulence using transitional RANS models and DNS. Int Commu heat Mass transfer 2013; 41: 10-16. http://dx.doi.org/10.1016/j.icheatmasstransfer.2012.11.005

Received on 26-05-2016 Accepted on 06-06-2016 Published on 31-07-2016

DOI: http://dx.doi.org/10.15377/2409-5761.2016.03.01.8