FLOW RATE INCREASE IN THE FORWARD CURVED CENTRIFUGAL FAN USING CFD SIMULATIONS

Kağan Özerk, Ali Pınarbaşı

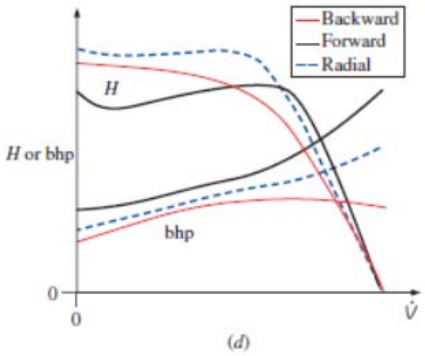
ABSTRACT

In this study, it is tried to develop methods that will provide high flow rate of a forward curved centrifugal fan without increasing the width - length - height values via cfd simulations. In order to verify the results of the analysis, the pressure difference produced by the fan at a certain flow rate was compared and the speed profiles at the fan output were compared. After validating the CFD model with the help of experimental results, the improvements were tried to be made through the CFD model with the preferred mesh type and turbulence model. The parameters affecting to the flow rate of the centrifugal fan have been determined and these parameters which is recommended in the literature have been analysed in the CFD model. According to the results of CFD analysis, design improvement is proposed.

Keywords: CFD, Centrifugal Fan, Forward-curved blades, Flow Rate Increase

INTRODUCTION

Centrifugal fans are frequently used in the industry for pressurizing air. Centrifugal fans are divided into three types as radial, forward curved and backward curved according to the blade type. Centrifugal fans with radial blades have a simple geometry and are the type that creates the most pressure difference, but sudden pressure drop occurs after the maximum efficiency point. Centrifugal fans with forward curved blades create an almost constant pressure difference as the flow rate increases from zero. These fans are designed with more but smaller blades. Also, these fans have lower efficiency than other types. The use of forward curved fans is recommended when a high-pressure differential fan is required over a wide flow range. [1]



Studies were carried out to increase the flow rate of the centrifugal fan used in a closed loop system. The main limitation of the operation is that the fan volume is not increased.

There are studies aimed at optimizing centrifugal fans for various purposes, in the literature. Some of these objectives; to reach the desired flow - pressure values, to reduce the noise level, to increase the efficiency of the fan. The recommended fan parameters are available in these studies.

Roth [2] proposed that the fan inlet angle is 80 degrees and the outlet angle is greater than 150 degrees and the inner and outer diameters of the fan blades are selected as 0.8 percent in his study.

In other article, the methods that can be used in cfd simulation of centrifugal fans are examined. Realizable k-ε, SST k-ω and Spalart-Allmaras turbulence models have been confirmed by experimental studies. They found that the most effective model in the comparison of turbulence models is Realizable k-ε. The adaptation of the model to the rotational flow character pointed out its success in flow separation. According to the turbulence models, the solution time was found to be 10% short in the Spalart-Allmaras model, while the other two models showed insignificant differences in terms of time. The effect of different mesh types on the results was investigated. The use of structured grid has positive properties, such as the reduction of computational errors, as well as the difficulty of applying it to complex geometries and the excess of time spent on meshing. [3]

ANSYS 15.0 Turbulence modeling document published by Ansys in 2014 on Realizable k-ε turbulence model: "Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g. boundary layer separation, massive separation, and vortex shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation)."

Skewness is one of the most important numerical values for mesh quality. "0" is the best value for skewness (proximity to the ideal tetra or quad cell) and 1 is the worst. It is stated that Skewness value should be kept below 0.9 for a good solution. [4]

Ansys customer training document recommended that the Skewness value should be kept below 0.95, while orthogonal quality should be kept above 0.1. [5]

Steady analysis was performed and frozen rotor approach was selected for rotating regions in models in this study. The MRF approach is the simplest approach for multiple zones. It is a steady-state approximation and it can be use for rotational or translational regions. While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. For example, the MRF model can be used for turbomachinery applications in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. [6]

CFD ANALYSIS AND VERIFICATION OF ANALYSIS RESULTS

The original forward curved centrifugal fan's flow - pressure curve is firstly generated in the wind tunnel, which is required to be developed. Pressure and flow values at the operating point of the fan are determined. Then, while the fan was operating at the operating point, the speed profile at the fan outlet was determined by means of a pitot tube. After obtaining these experimental data, the cfd model was established and the model was verified.

The values given as input condition in CFD analysis are as follows.

Fan rotation speed: 2280 rpm

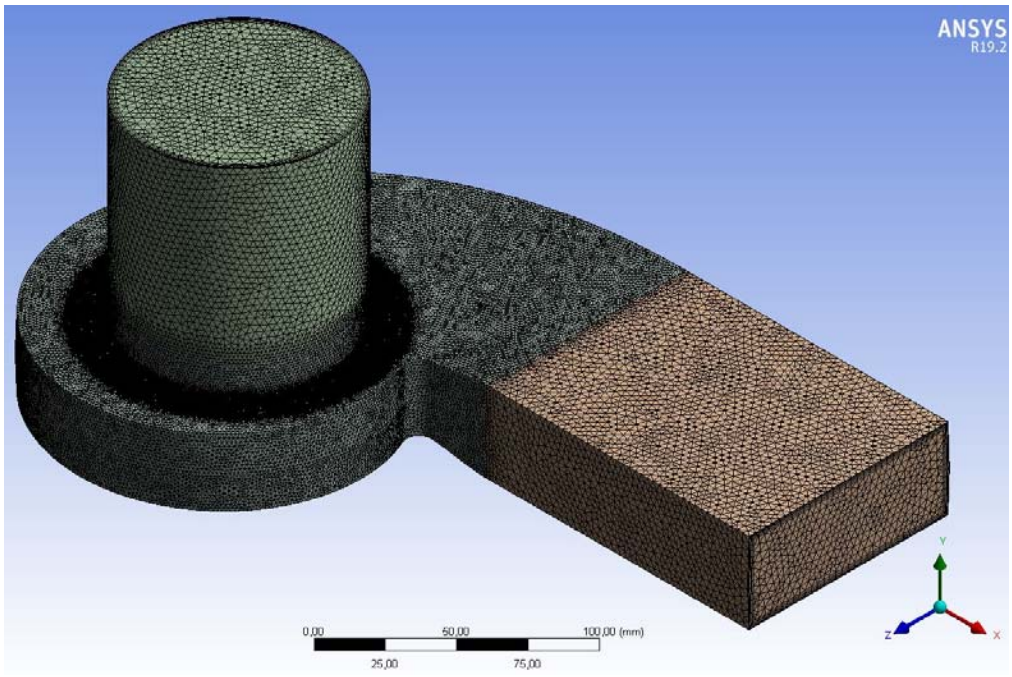
Inlet velocity: 4.1594 m / s

Pressure outlet: 0 Pa.

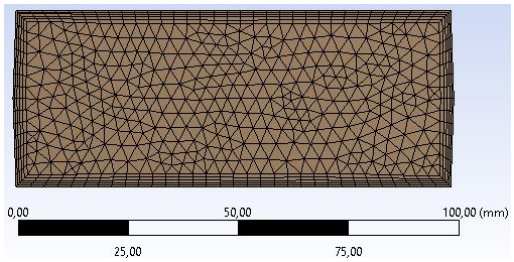
Turbulence model: Realizable k-ε, standard Wall function

It was found that mesh quality and quantity had a significant effect on the results of the cfd solution. The normal angle was set as 10^{0} degrees, also inflation was set on walls so calculation around walls can be done with precision. Mesh was laid on the other surfaces of the wing at maximum 0.5 mm intervals. This value was reduced to 0.3 mm to prevent the formation of thin - long and high - angle cells which adversely affect the mesh quality in the hard corner on the outside of the blade. The Skewness value of the analyzes was kept below 0.9. and orthogonal quality is above 0.1.

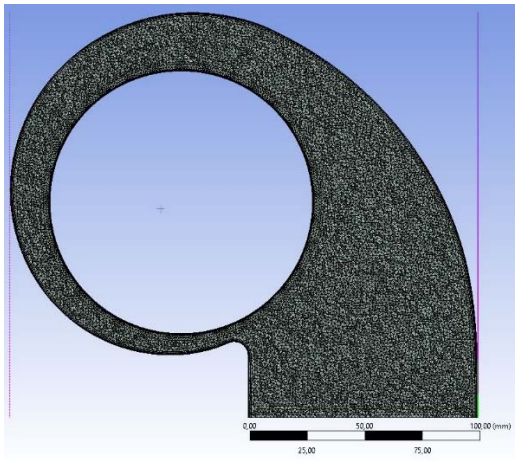
The mesh images of the original fan are as follows.



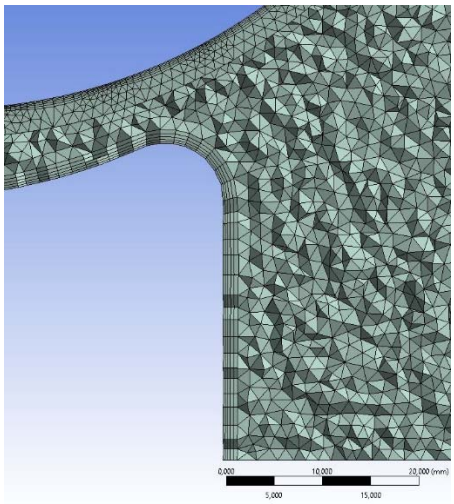
Picture 1. Isometric view



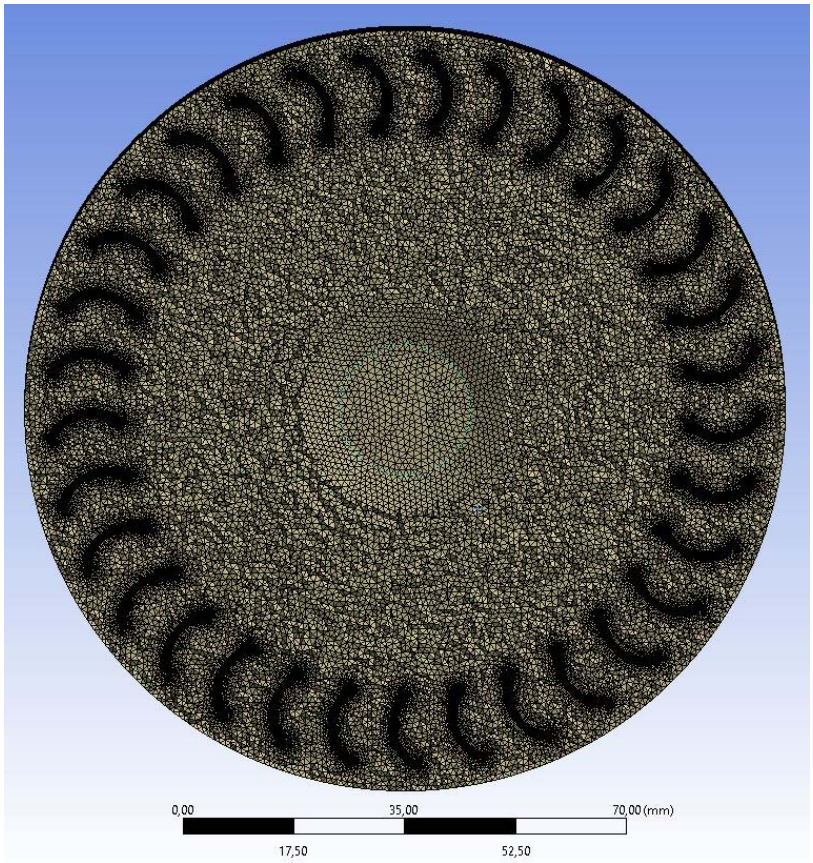
Picture 2. Fan pressure outlet



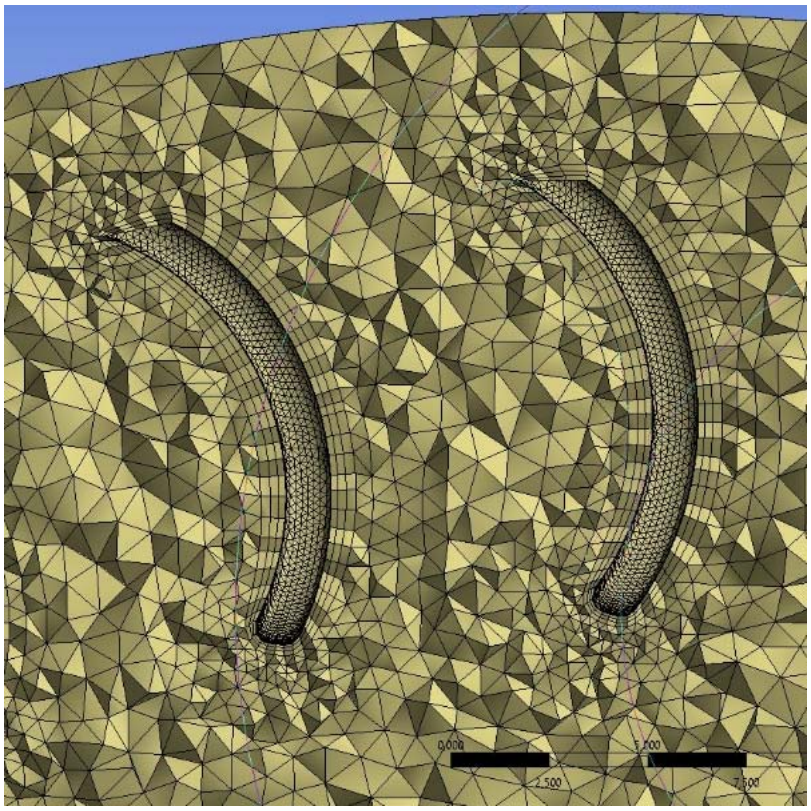
Picture 3. Fan volute section view



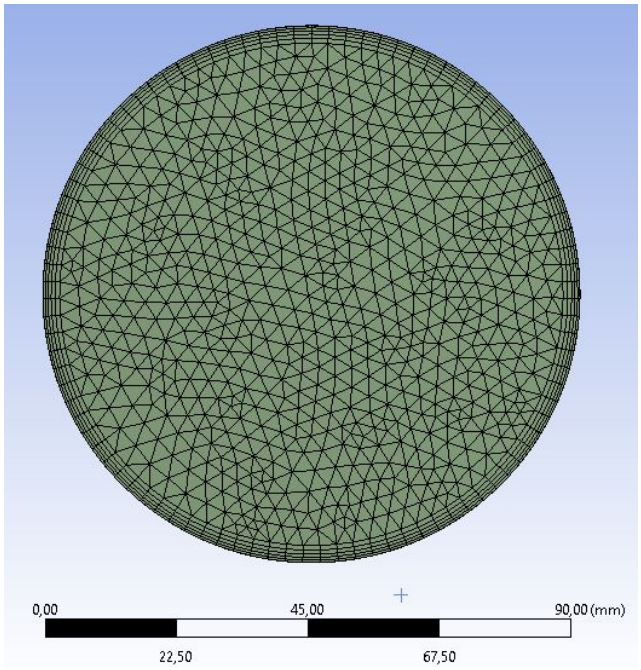
Picture 4. Fan volute mesh details.



Picture 5. Rotating domain



Picture 6. Mesh details around blades

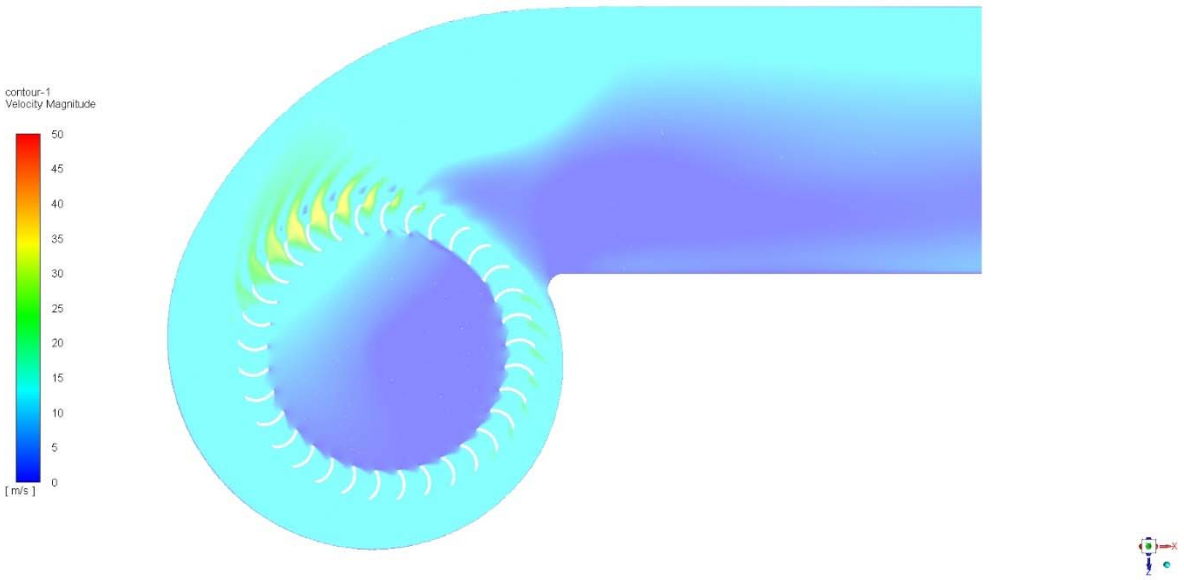


Picture 7. Velocity inlet

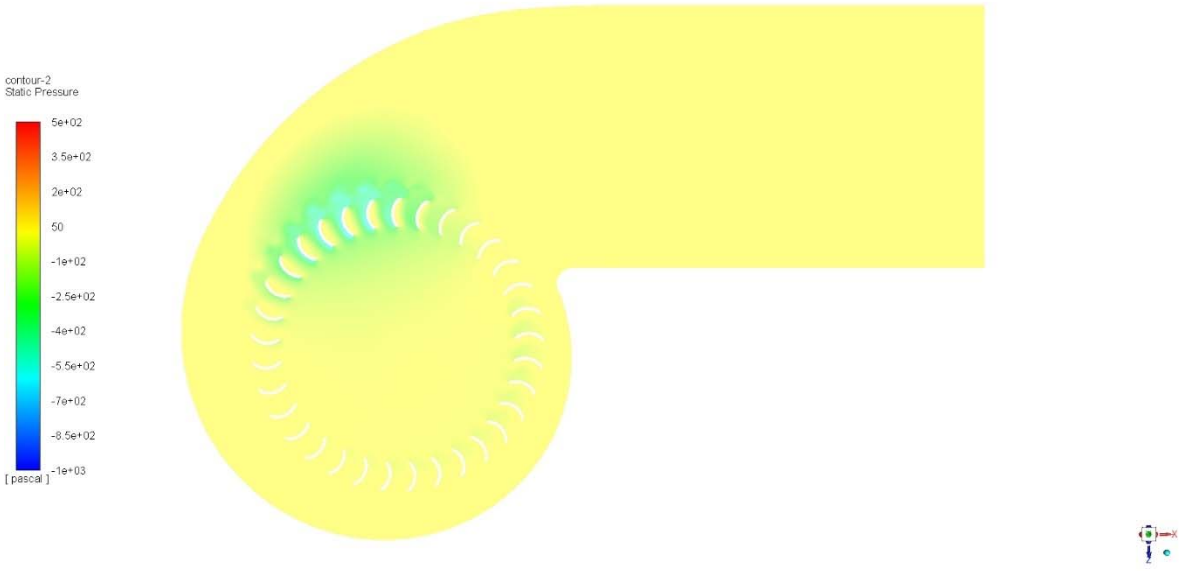
Mesh quality values:

Skewness 0.85

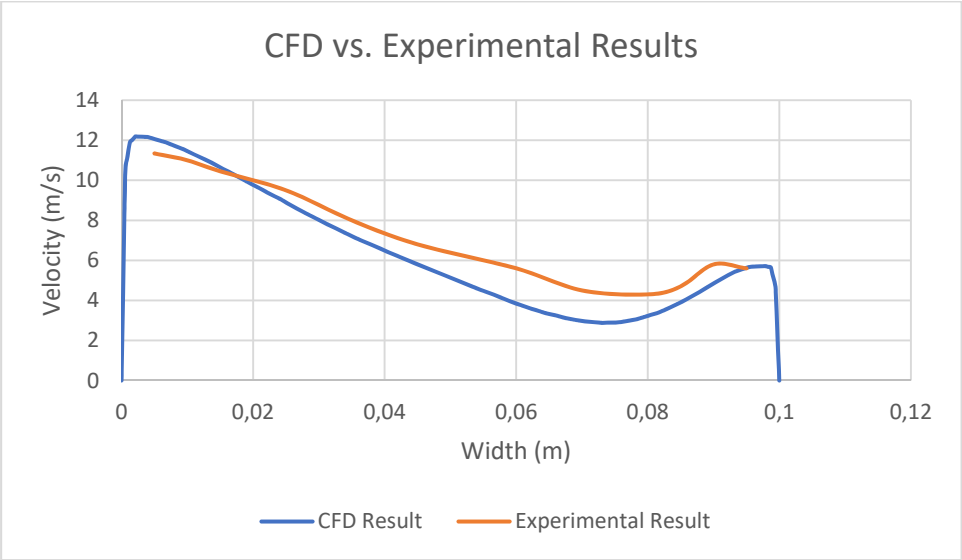
Orthogonal Quality 0.16



Picture 8. Velocity Profile of the original fan



Picture 9. Pressure Profile of the original fan



It was found that the pressure value measured in the wind tunnel was in agreement with the cfd result. Speed profiles at the fan outlet are also compatible.

USE OF DIFFERENT WING GEOMETRIES

The inner and outer diameters of the fan blades have been changed without changing the number of fan blades. 2 different inner diameters and 2 different outer diameters were tested. These angles are as follows.

B_1= 90º & 75º

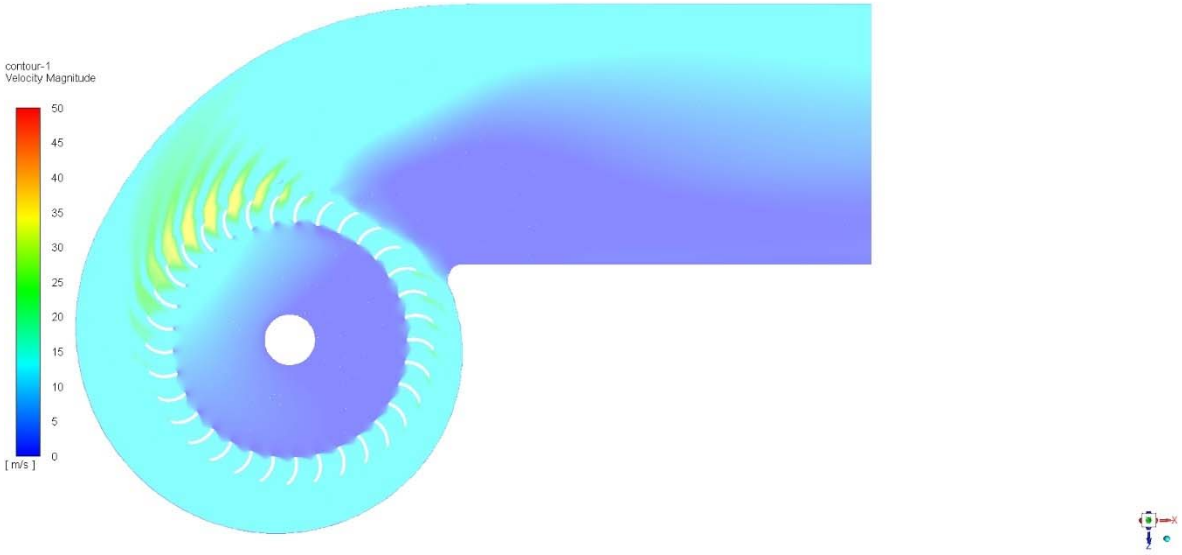
B_2= 140º & 160º

Blade outer angles: 140º & 160º

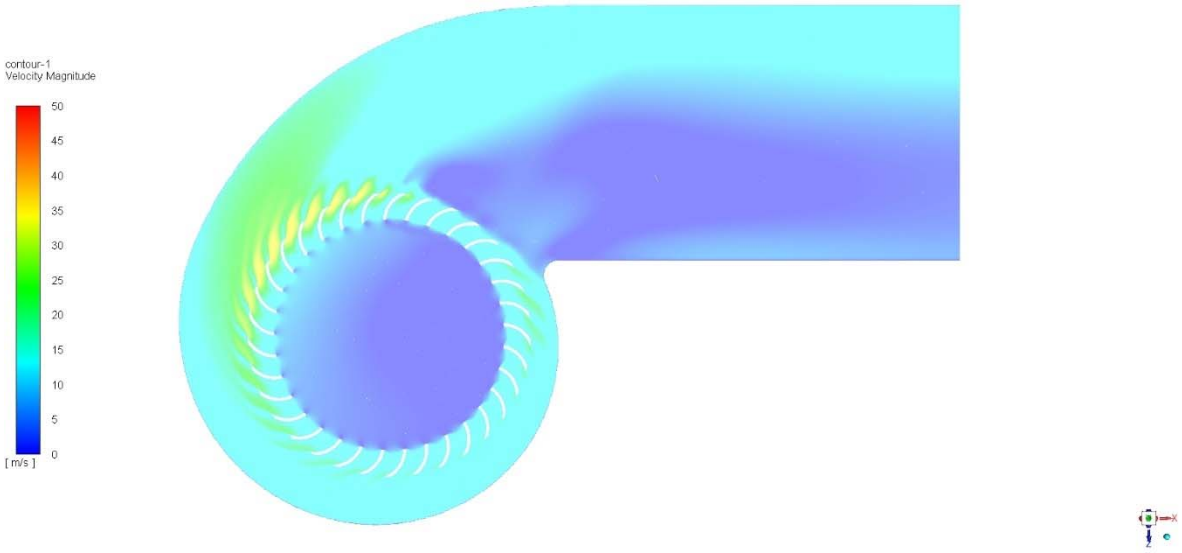
RESULTS AND DISCUSSION

The analysis results for a total of 4 different cases are as follows. The flow rate was kept constant in the analyzes. Pressure differences were compared.

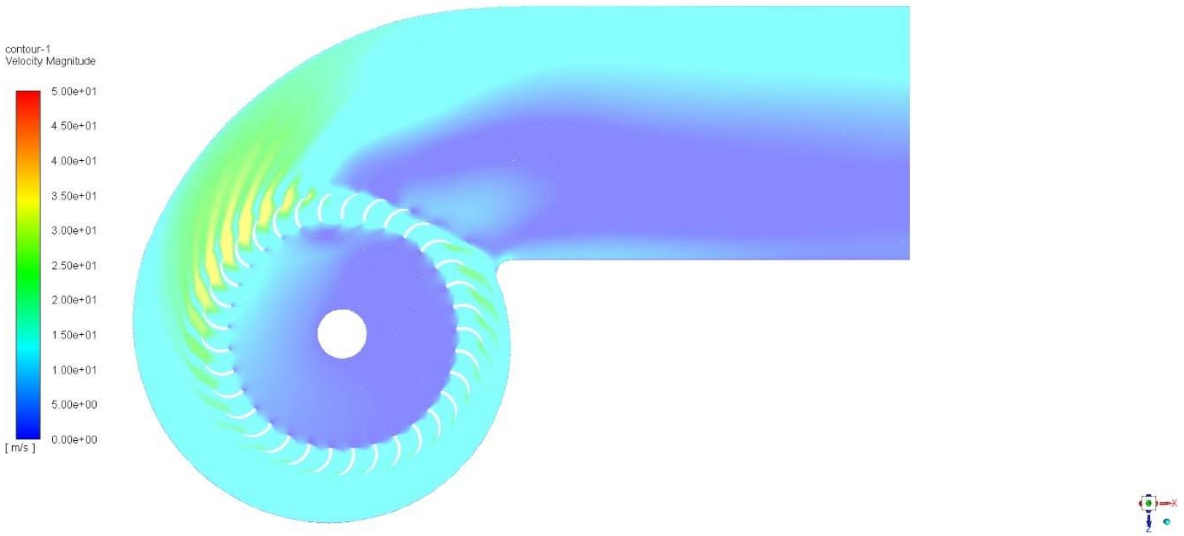
Pressure Differences for B1 & B2 Combinations				
	140 - 75	140 - 90	160 - 75	160 - 90
Pressure Difference (Pa.)	98	95	108	106



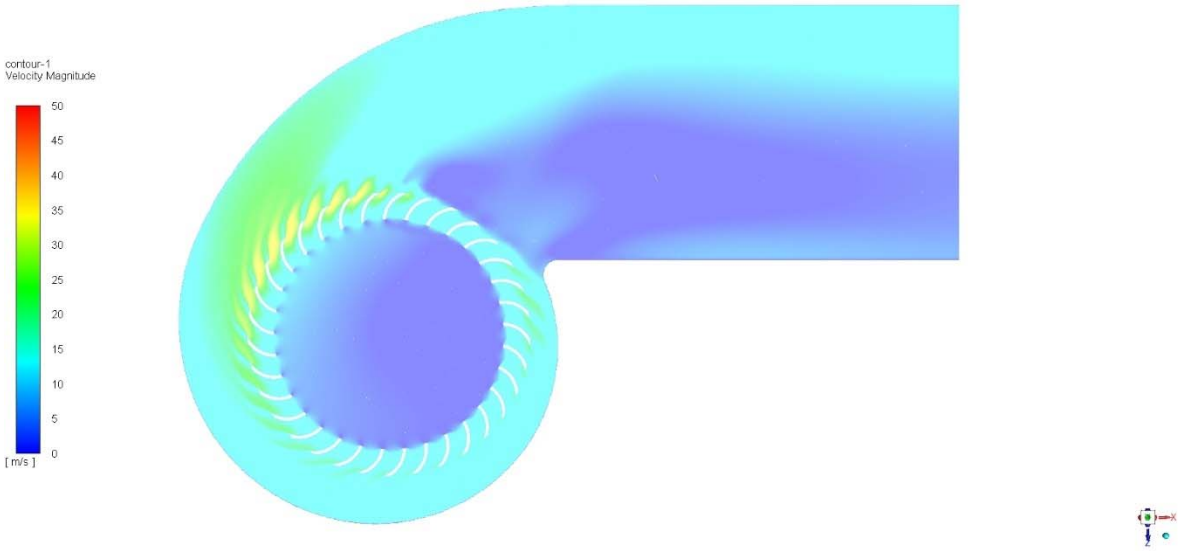
Picture 11. Velocity profile of 140-75



Picture 12. Velocity profile of 140-90



Picture 13. Velocity profile of 160-75



Picture 14. Velocity profile of 160-90

Backflows were detected in the fan tongue region. Subsequent studies will focus on narrowing the tongue region and preventing backflow. In addition, larger blades with bigger outer diameter will be used without changing fan sizes.

CONCLUSION

In this study, the effect of change in blade inlet and outlet angles on pressure difference was investigated. Increasing the output angle increases the pressure difference, but also causes the required torque to increase.

NOMENCLATURE

B_1 = Blade entrance angle

B_2= Blade exit angle

H = Fan Pressure

Bhp = Horse power

V = Volume Flow Rate

REFERENCES

[1] Yunus Cengel, John Cimbala - Fluid Mechanics Fundamentals and Applications-McGraw-Hill Science_Engineering_Math (2013)

[2] H. W. Roth, H. Leist, and J. Zierep. Untersuchungen an Trommell ¨aufer-Ventilatoren. Technical report, Institut f¨ur Str¨omungslehre und Str¨omungsmaschinen der Technischen Hochschule Karlsruhe, October 1979.

[3] Manoochehr DARVISH, Stefan FRANK, Toward the CFD Simulatiın Of Sirocco Fans: From Selecting A Turbulence Model to The Role Of Cell Shapes

[4] ANSYS Meshing User's Guide, Release 13.0

[5] Ansys Numesys, 19.2 Version Customer Training Document

[6] ANSYS FLUENT 12.0 Theory Guide