Journal of Engineering and Development
www.jead.org
Vol. 20, No.02, March 2016
ISSN 1813-7822

EFFECT OF MODE OF ROTATION AND SPEED ON THE PRESSURE FIELD AND AXIAL THRUST OF INDUSTRIAL CENTRIFUGAL FAN

Dr. Mauwafak A. Tawfik[1],Dr. Mohammed I. Abu-Tabikh[2],Adil A. Nayeeif[3]

1) Assist Prof, Mechanical Engineering Department, University of Technology / Baghdad, Iraq.
2) Assist Prof, Mechanical Engineering Department, University of Technology / Baghdad, Iraq.
3) Assist lecturer, Mechanical Engineering Department, Al-Mustansiryah University, Baghdad, Iraq.

(Received: 21/05/2015; Accepted: 23/09/2015)

Abstract: Centrifugal fans are used widely in most of the manufacturing processes for ventilating and/or air conditioning the working areas. Because of relatively limited documentation on the design of this type of fans, experimental and theoretical works are investigated in this paper for forward backward centrifugal fan to calculate the average dynamic pressure and axial thrust forces at different speed conditions. The flow in centrifugal fan is highly complex with taking place on the impeller .The centrifugal fan simulated using computational fluid dynamic (CFD) approach. Flow in centrifugal fan with duct is simulated by Navier – Stokes equation and performance curved are obtained. Most of companies creates in 3D model inSolid works version 2013 and the conduct analyses by the CFD programs. At first step , an experimental step was developed and prototypes of impeller was made to carry out at the south Baghdad power plant for the sake of flow measurements and next , computational fluid dynamics model was developed for the above step and the result are validated with experimental measurement. Fine mesh is generated for centrifugal fan with duct in GAMBIT programs to carry out analyses for whole computational domain in fluent programs. Further parametric studies were carried out to calculate the average dynamic pressure when rotating the impeller backward and forward rotation modes. The parameter considered in this study are changing in dimension of outlet duct at different speeds. Finally, the result shows Increasing the speed of motor caused increasing in the average dynamic pressure is(98.43%)at outlet and on the blade of the impellers .The average of dynamic pressure for forward rotation mode is (61.83%) greater than that of backward direction mode .The results exhibited that the maximum discrepancy between the experimental and numerical is (9.8%) for backward rotating and (10.44%) for forward directionat different speed. The magnitude of dynamic axial thrust forces in the z – direction are maximum for forward rotation mode compared with another rotation mode because the differences in pressure on the impeller fan. The experimental and theoretical results are in well agreement, with the (CFD) model developing .Benefit from the results of this research in the influence guess bearings on the vibration characteristics of the system .

Keywords*: impeller, forward, backward, dynamic pressure, centrifugal fan*

تأثير حالة الدوران على مجال الضغط وقوة الدفع المحوري لمروحه طرد مركزي صناعي

الخلاصة: تمت مناقشه الجانب العملي والنظري لمروحه الطرد المركزي بالاتجاهين الامامي والخلفي لحساب معدلالضغط الديناميكي والقوة المحوريه ولسرع دورانيه مختلفه . يكون الجريان في مروحه الطرد المركزي معقده جداً في حاله اخذ المروحه (Impeller) . تمت المحكاه مروحه الطرد المركزي باستخدام برنامج (CFD) ، في حين تم محكاه الجريان في مجري المنظومه باستخدام معادلات

Corresponding Author:Adilnayeeif@yahoo.com

(Navier – Stockes) وتم الحصول على منحنيات الكفاءة . تم بناء معظم الاجزاء باستخدام برنامج (Solid works 2013) . تم البدء بالعمل في ورشه تقع في محطه جنوب بغداد البخاريه لقياس جريان الهواء عبر المجرى (Duct) ومقارنته مع النتائج النظريه الحاصله من برنامج (CFD) والنتائج كانت متطابقه . تم بناء المنظومه وتكوين الشبكه الدقيقه (Fine mesh) باستخدام برنامج (Gambit) لكافه المجالات في برنامج (Fluent) . اجريت المزيد من الدراسه وتنفيذها لحساب معدل الضغط الديناميكي عندما تكون حاله الدوران بالاتجاهين الامامي والخلفي . واخيراً اظهرت نتائج البحث على ان معدل الضغط الينامكي يزداد مع زياده السرع الدورانيه وتكون قيمته في حاله الدوران الامامي اكبر مما لو كانت السرع بالاتجاه الخلفي ، وكذلك اظهرت تطابق بين النتائج الجانب العملي والجانب التحليلي اذا تصل اعلى قيمه فرق بين النتائج الى (9.8%) في حاله الدوران الامامي بينما تصل قيمه الانحراف الى (10.44%) في حاله الدوران الخلفي .اما فيما يخص القوة المحوريه باتجاه (Z) اعلى ما يمكن في حاله الدوران الامامي مقارنته بالدوران الخلفي .

1. Introduction

Centrifugal fans are being widely used in many industrial applications. Those fans are subjected to dynamic forces from many different sources these include mechanical forces such as unbalance and misalignment and aerodynamic forces produced as a result of rotating impeller. The flows in centrifugal fan are of a very great complex in fluid dynamics. They are characterized by their three-dimensional dependent on geometrical characteristics as well as the operating conditions. The aerodynamic loads applied to the blades of fan are highly unsteady, this loading forms a source of axial thrust on the fan rotor bearings. Several studies based on both experimental and computational fluid dynamics (CFD) models had been carried out for the analysis and design of centrifugal fans .Roberts (2004) [1] studied the geometric parameters of a centrifugal fan with backward- and forward-curved blades, an experimental setup was developed and prototypes of fans were made to carry out measurements of flow and power consumed by the fan. The fan mounting setup was such that fan with uniform blades can be tested. (CFD) model was developed for the above setup and the results were validated with the experimental measurement.

Younsietal., (2007)[2] studied numerically and experimentally four centrifugal impellers designed with various geometrical parameters. The numerical simulations of the unsteady flow had been carried out using (CFD) tools based on the unsteady Reynolds averaged Navier – Stokes (URANS) approach. The experimental part of this work was concerned with the measurement of aerodynamic performance of the fans using a test bench builted according to ISO 5801 (1997) standard. The numerical results had been compared with the experimental measurements and a correlation between the wall pressure fluctuations and the far field noise signals had been found. Pathak etal., (2012)[3] studied the numerical analysis of the single stage centrifugal blower which was carried out for different flow coefficient, to analyze the 3-D flow field. Fluid domain was created and simulation was done with the CFD software code ANSYS CFX. Three dimensional Navier-Stokes equations were used to analyze the flow. Standard k-ε turbulence model and unstructured grid was adapted to solve the Navier-Stokes equations. Results of numerical analysis suggested that the pressure coefficient at an outlet of fluid domain was continuously decreased with increasing in flow coefficient.

Patel etal., (2013)[4] investigated the centrifugal fan using (CFD) approach, flow in centrifugal impeller was simulated by Navier-Stokes equation and performance curves were obtained. An experimental setup was also developed and prototypes of fans were made to carry out measurements of flow. The theoretical results were validated with the experimental measurements, the result of this numerical analysis improved the performance of the centrifugal fan. Pranav and Raj (2012) [5] presented the design methodology for the centrifugal fan system with impellers having airfoil blades. The numerical design procedure was developed for it and the CFD optimization had been carried out for volute casing to improve the results which had got from the numerical procedure only. Thus the design methodology which includes the assistance of CFD optimization has been developed successfully.

In the present work, an experimental and numerical study is carried out to investigate the effect of impeller rotational speed, and rotational mode (Backward / Forward) on the pressure field and axial thrust force of a centrifugal fan. It is hoped that this work be the first step toward more sophisticated studies about the effects of fan rotor bearing positions on the vibration characteristics of centrifugal fans.

2. Experimental Techniques

An experimental work was carried out at the south Baghdad power plant in order to measure the main flow parameter of the centrifugal fan at different operating speed conditions. Figure (1) shows the main components of the test rig.

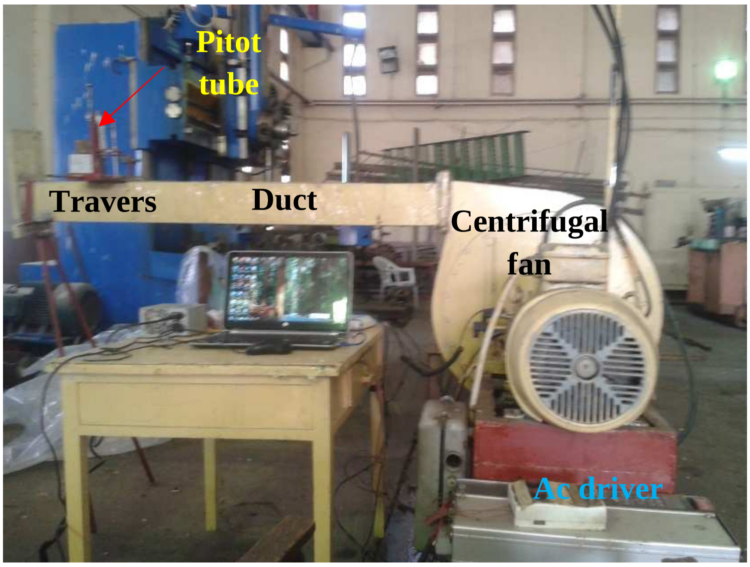


Figure (1) Test rig

2.1. Centrifugal fan

It is composed of a number of fan blades, or ribs, mounted around a hub. As shown in Figure (2), the hub turns on a driveshaft that passes through the fan casing. The air enters from the side of the fan wheel, turns 90 degrees and accelerates due to centrifugal force as it flows over the fan blades and exits the fan casing.

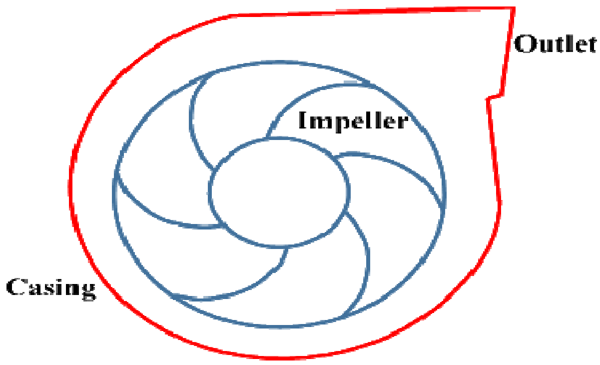
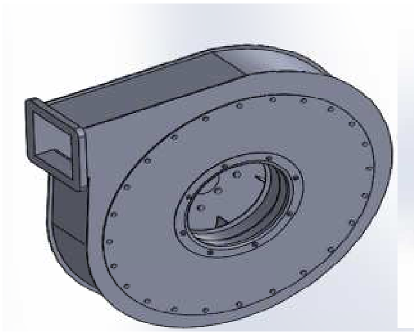
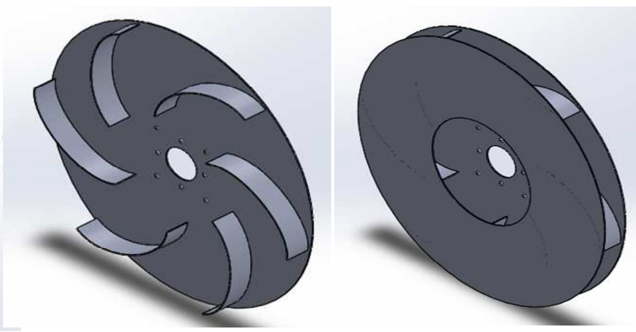


Figure (2) Schematic diagram of centrifugal fan.

The centrifugal fan used in the present work is a single volute casing which is designed for industrial applications as shown in figure (3) and it has an open radial impeller with six curved non – twisted blades as shown in figure (4). The fan is provided with three – phase electric induction motor .The specifications of the centrifugal fan and the geometry of the impeller are listed in tables (1) [6],[7].





Figure(3) centrifugal fan

Figure(4) Fan impeller

Table(1) Specification and geometrical parameters for fan impeller

Type	Centrifugal (Backward/Forward) fan
Shaft length	32 mm
Fan weight	12 kg
Number of impeller blades	6
Motor speed	2820 rpm
Power	7.5 hp
Voltage/current/frequency	380 volt / 13 amp / 47 Hz
Impeller inlet diameter (mm)	275
Impeller outlet diameter (mm)	637
Blade thickness(mm)	2
Blade inlet width (mm)	50
Blade outlet width (mm)	50

2.2. Duct

A duct was designed and fabricated according to ASME standards **[8],** to incorporated in the workshop setup to provide a measuring plane and simulate the condition. The fan is expected to encounter in service. In experimental test the air flow determination shall be straight and have uniform square cross – section duct with dimension shown in figure (5).

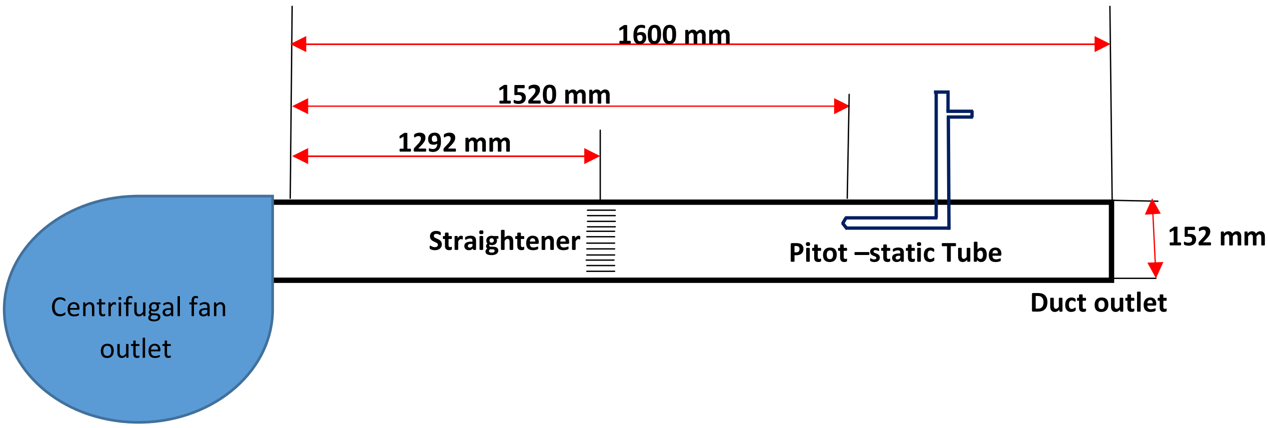


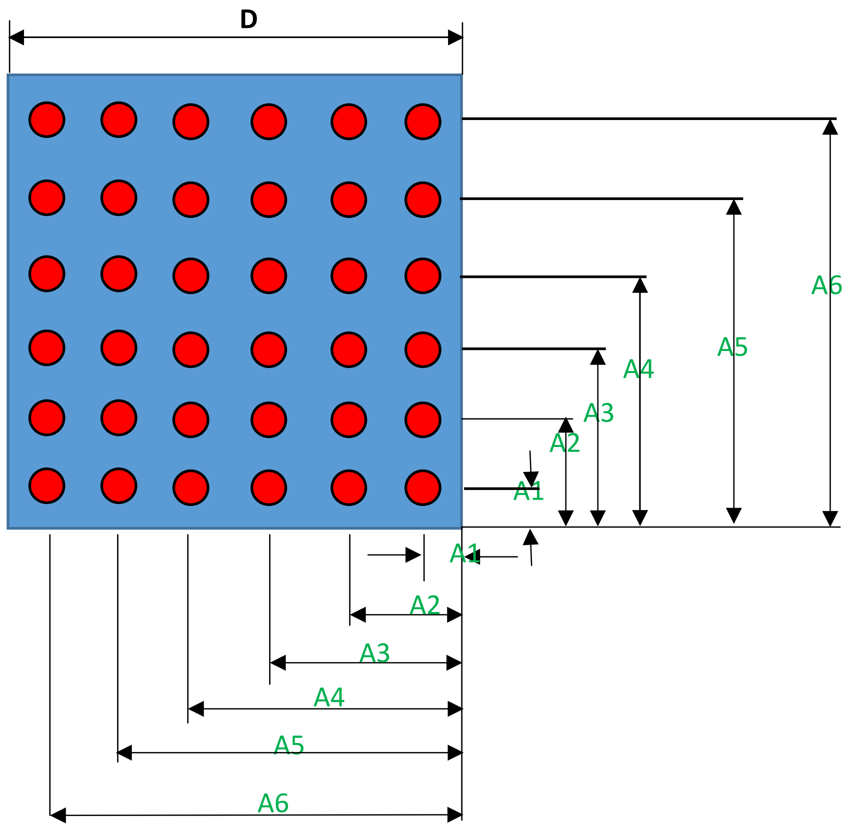
Figure (5) Test Duct [8]

2.3. Air flow measurements

Figure (6) illustrate the pitot – static tube traversing mechanism mounted at the plane of traverse. A sensitive inclined monometer was used for the measurement of differential pressure [9].

Figure (6) Monometer with traverse

The fan airflow rate is calculated from dynamic pressure measurements taken by pitot – static tube traverse at the location of pitot – static tube, the plane of traverse is divided in several numbers of points shown in figure (7).



A1	A2	A3	A4	A5	A6
0.061D	0.235D	0.437D	0.563D	0.765D	0.939D

Figure (7) Traverse points for rectangular duct [9]

The dynamic pressure (P) corresponding to the average velocity in the duct is obtained by:

$$P = \left(\frac{\sum\sqrt{P_n}}{n}\right)^2 \quad (1)$$

Where P_n is the individual dynamic pressure measurements taken at the plane of traverse.

The average velocity (V) is obtained from the density at the plane of travers(ρ) and the corresponding dynamic pressure (P) using:

$$V = \sqrt{\frac{2P}{\rho}} \quad (2)$$

Finally the airflow rate (Q) at the plane of traverse is obtained from the velocity (V) and the area (A) using:

$$Q = A*V \quad (3)$$

2.4. Test procedure

The following producer were followed during the experimental test:

1- The backward rotational mode was selected.
2- The fan was operated at the desired speed.
3- Three minutes were allowed for the rig to reach a steady state.
4- The dynamic pressure measurements were performed by means of pitot – static tube, inclined monometer, and the traversing mechanism .Equations (1) to (3) were used for calculating the average dynamic pressure, velocity,and air flow rate.
5- Steps (2-4) were repeated for the forward rotational mode.

3. Modeling and Numerical Simulation

3.1. Modeling of centrifugal fan

In the present work, a three dimension model of the centrifugal fan including a volute casing and curved blades impeller was implemented. The geometry of the system was created by using professional “Solid Works“software, 2013 [10]. Dimensions were fitted define the size and location of the fan geometry. The model (volute casing,wheel impeller) is assembled for all parts in solid works software as shown the figure (8).

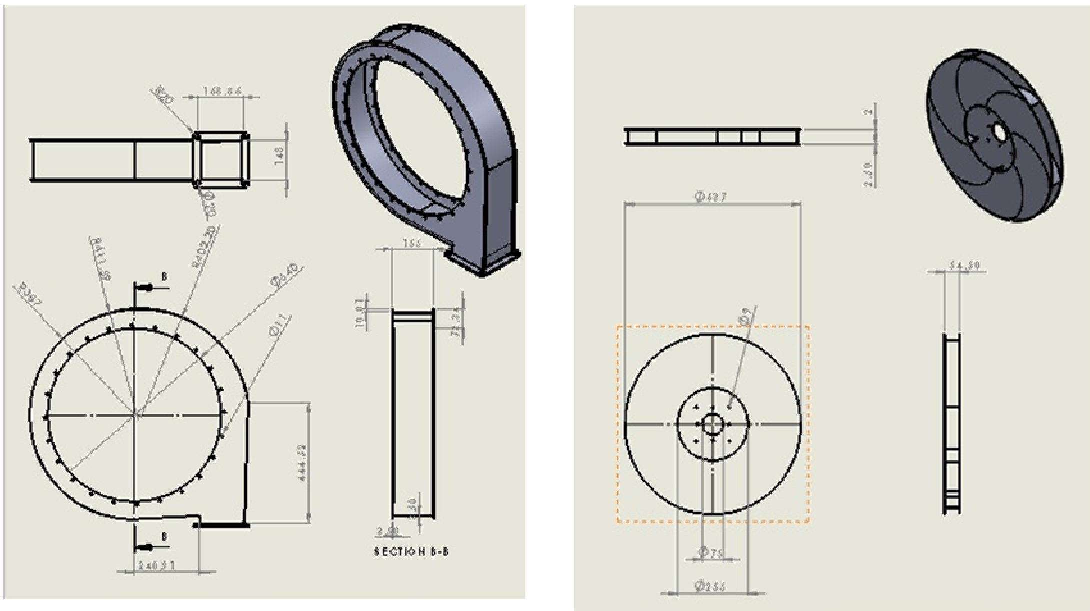
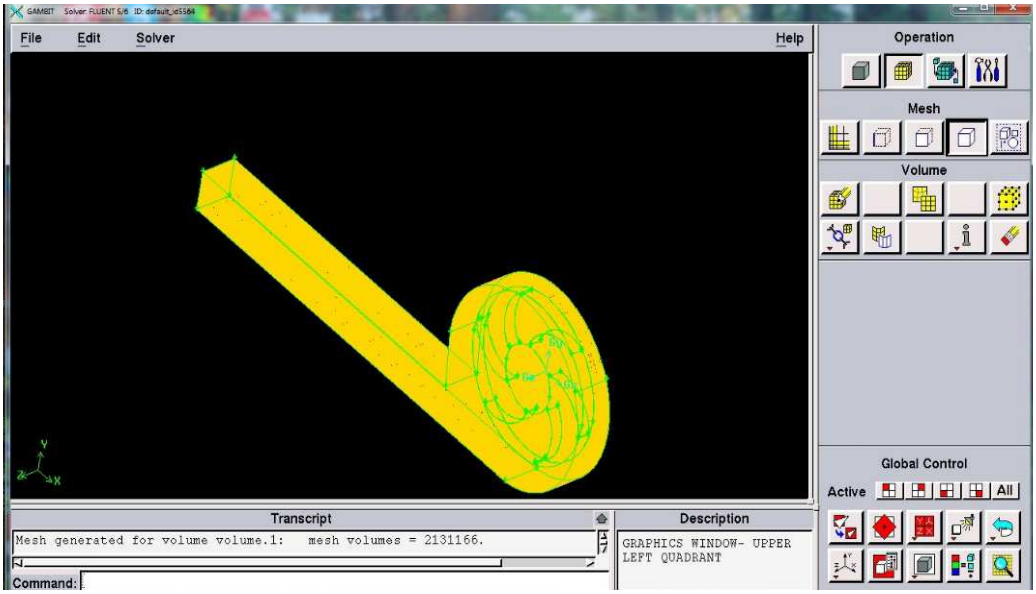


Figure (8) Model of casing and wheel impeller in solid works (dimension in (mm))

3.2. Fan Geometry and Mesh Generation

The centrifugal fan “Solid works” model was then exported into GAMBIT in order to form the air volume in the fan and to mesh of the whole geometry as seen in Figure (9). The required meshes were generated by using the CFD pre – processing software, “GAMBIT“, version (2.4.6). An unstructured 3 – D tetrahedral mesh was used for the generation on the computational domain surface as well as a boundary conditions. Whole domain of centrifugal fan is meshed using the unstructured type of grid.



Figure(9) Meshed centrifugal fan with wheel impeller

The next step is to check the model geometry. This gives the possibility to spot any discontinuities at the edges and corners of the geometries, which may have resulted during transfer of models created using different software. This process is important because it prevents these discontinuities to cause larger problems during further steps of the analysis. After the geometry checking and cleaning processes have been performed on the model, meshing that is the discretization of the geometry into a numerical grid(grid size that adopted in the geometry is 0.5) is carried out. In the present study more elements were used near the volume swept by the impeller, and the element size increases away from this volume.

Therefore, both the solution accuracy and time as well as processor capacity were taken into consideration. In order to achieve grid independence, a total number of (2373738) elements was used in the mesh. In GAMBIT the quality of these elements is characterized by a factor called “skewness factor” that has values between 0 and 1. It is best to have this factor as close as possible to zero. GAMBIT determines the mesh type and boundary conditions to be defined according to the chosen solver. In the present study the FLUENT/ANSYS solver was used [11].

3.3. CFD simulation

The Conservation of Mass and Navier–Stokes equations related to fluid dynamics involved have been solved with appropriate boundary conditions similar to those used in the experimental test.

The finite volume method and grid generation in the CFD environment provided by the FLUENT/ANSYSsoftware was used. Following is the brief outline of the method adopted [12].

- Creating working fluid of air and selecting material of the impeller
- Assigning a rotating region material to the volume surrounding the fan. The main construction of model is rotating region. Rotating region is envelope the impeller of fan. This is not torching the surface of fan.
- Creating the boundary conditions at the inlet and outlet. At the inlet, the velocity depended on operating speed and pressure is zero at outlet.
- Setting the known rotational speed.
- Defining the mesh distribution using automatic-sizingwas gives in this study.
- Solving the application to obtain the results.

4. Results and discussion

Figure (9) and (10) show the theoretical dynamic pressure distribution inside the centrifugal fan at different speeds and rotation modes. Air is drawn in through the inlet casing into eye of the impeller. The impeller eye is a low pressure region as shown in these figures .The function of the impeller is to increase the energy level of the air by whirling it outwards. Both the static pressure and velocity and, hence, the dynamic pressure are increased within the impeller.

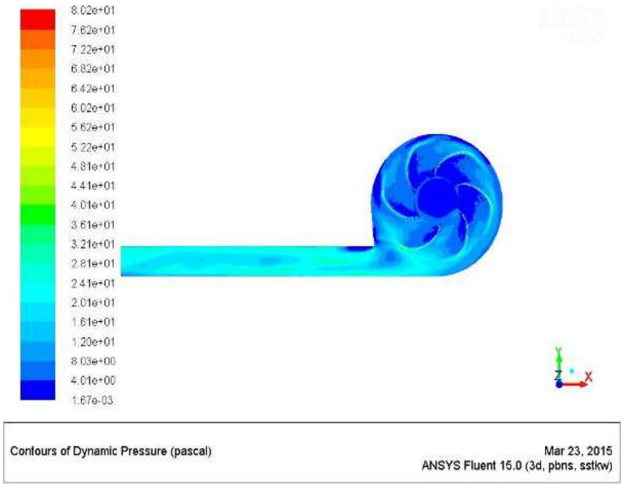
For the backward rotation mode, figure (9), the pressure at the convex faces of the blades is higher than the pressure at the concave faces, since the former are considered as the leading faces at the backward rotation mode.

At the forward rotation mode, figure (10), it is obvious a high level of pressure inside the fan casing in comparison with the backward mode.

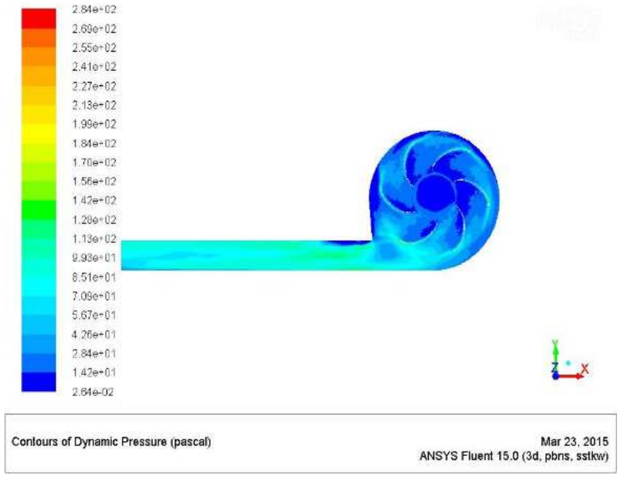
Figure (11) and (12) show the contour of pressure distribution at location on the centrifugal fan, noting the pressure was maximum at the impeller due to kinetic energy and load from pressure is rise at this location, the experimental and theoretical variation of average dynamic pressure with fan speed at backward and forward mode of rotation respectively. In general, the experimental and theoretical results are in well agreement, with the (CFD) model developing slightly lower pressure with the increase of fan speed. Figure (13) shows the effect of rotational mode on the experimental dynamic pressure and discharge at different fan speed.

The forward rotation mode is usually employed for higher pressure rise and higher discharge. The backward mode is employed for lower pressure and discharge, therefore it must run at higher speed to develop the same pressure rise as that for forward mode.

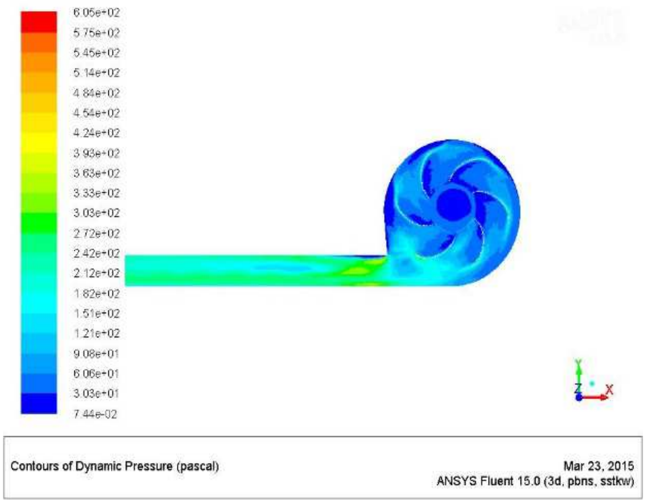
Finally, the axial thrust force (F_Z) developed by the fan impeller is shown in figure (14), It is obvious that the axial thrust developed by the fan operating at forward mode is higher (94.2643%). For example, at 2400 rpm, the axial thrust at forward mode of rotation is higher by a bout (0.815 %) in comparison with backward mode of rotation.



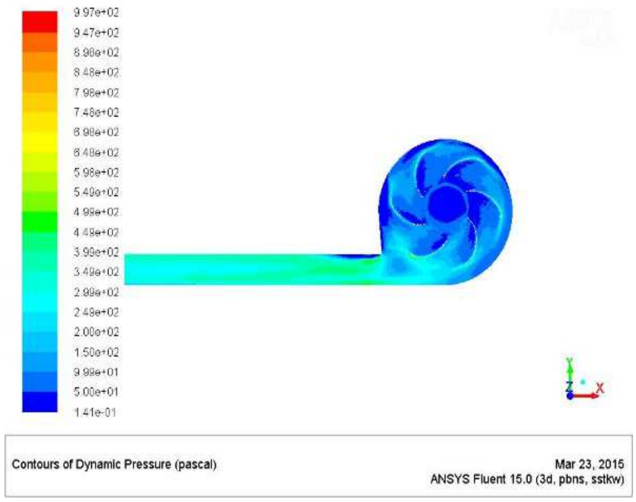
a-600 rpm (Re =1.08$\times 10^5$)



b- 1200 rpm(Re =2.18$\times 10^5$)

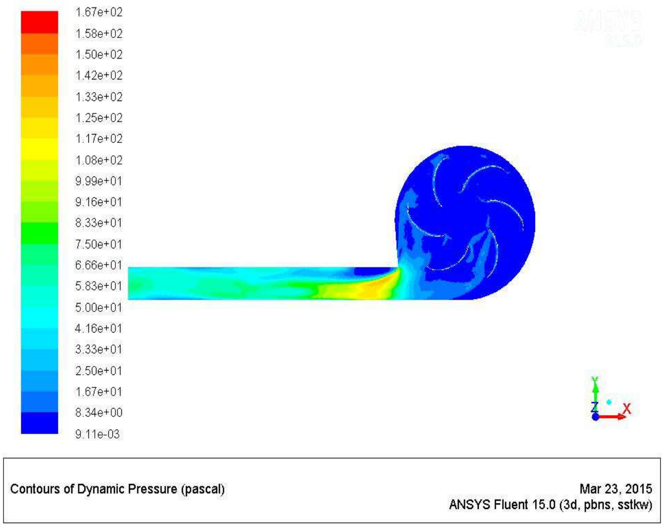


d-1800 rpm (Re =3.54$\times 10^5$)

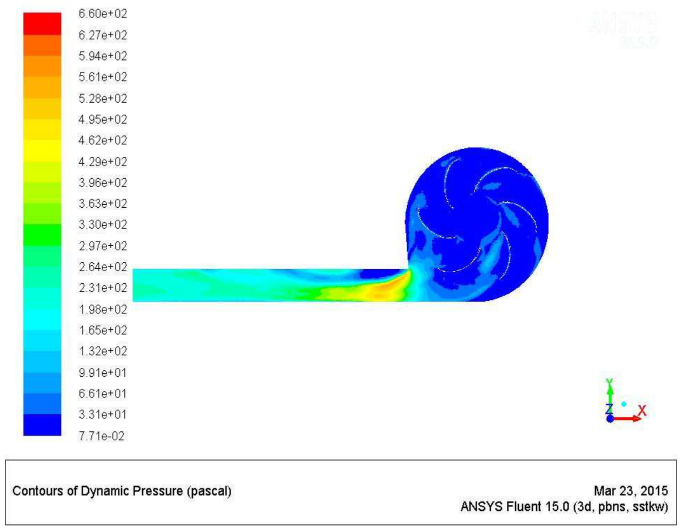


e- 2400 rpm(Re =4.25$\times 10^5$)

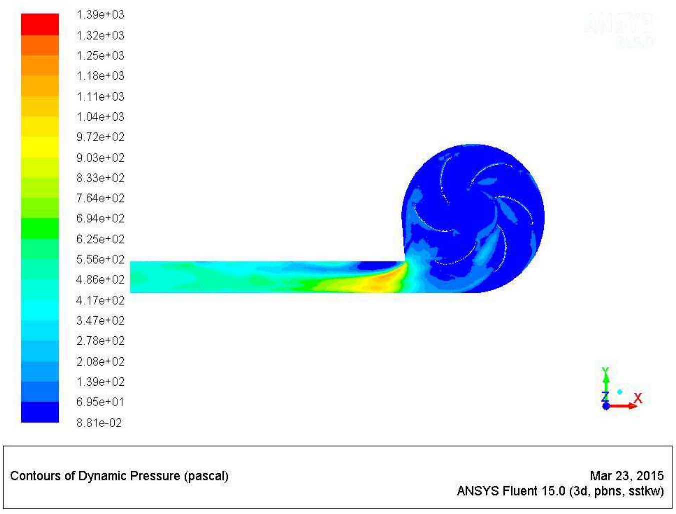
Figure (9) Pressure distribution for centrifugal fan at different speeds (Backward rotation mode)



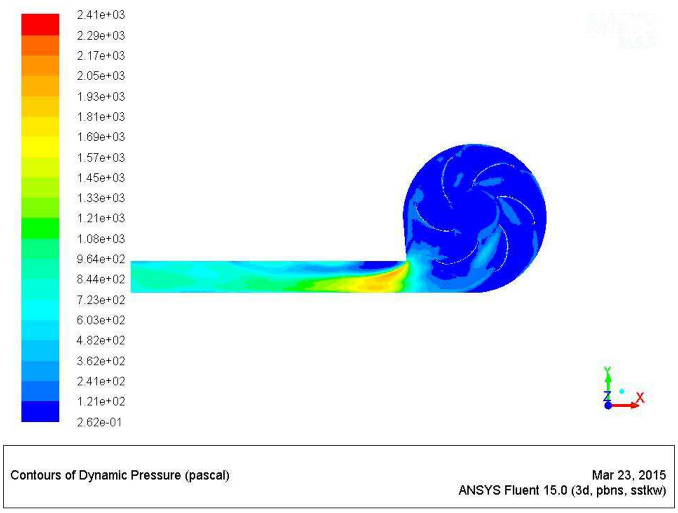
a- 600rpm (Re =1.76$\times10^5$)



b- 1200 rpm(Re =3.59$\times10^5$)



c-1800 rpm (Re =5.19$\times10^5$)



d- 2400 rpm(Re =6.89$\times10^5$)

Figure (10) Pressure distribution for centrifugal fan at different speeds (Forward rotation mode)

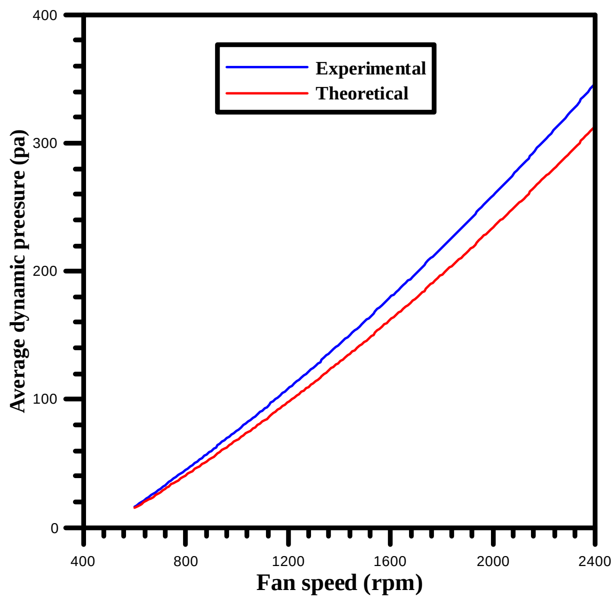


Figure (11) Variation of average dynamic pressure with fan speed (Backward rotation mode)

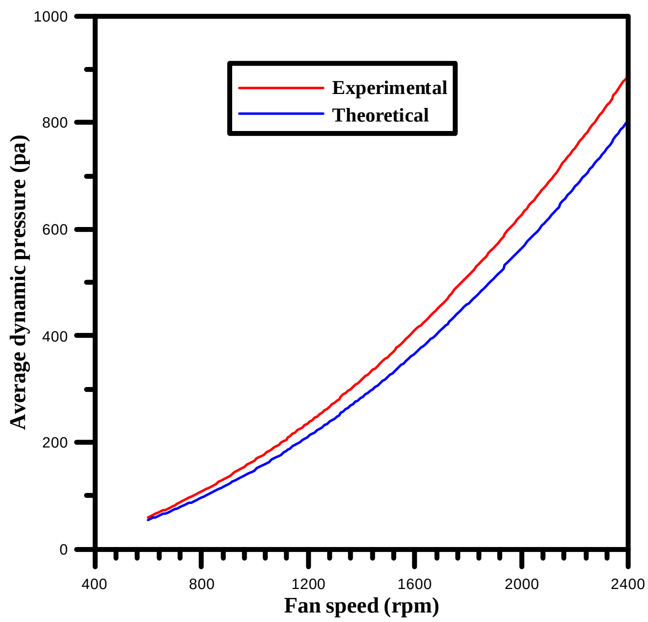
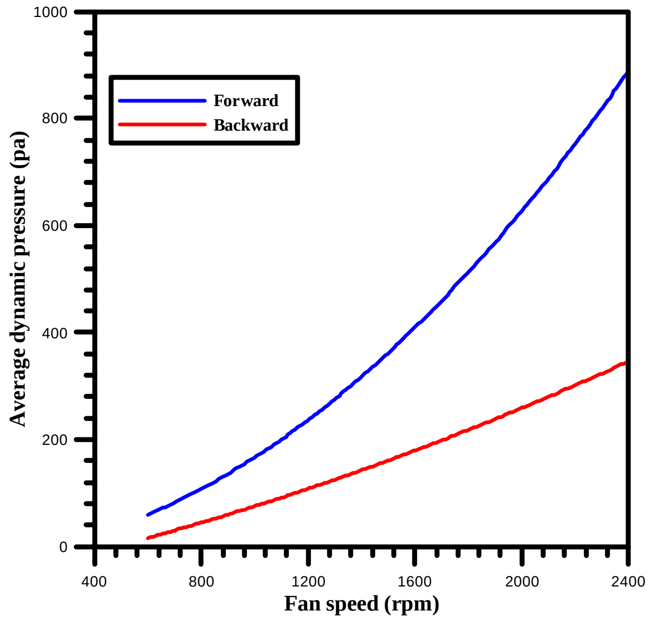
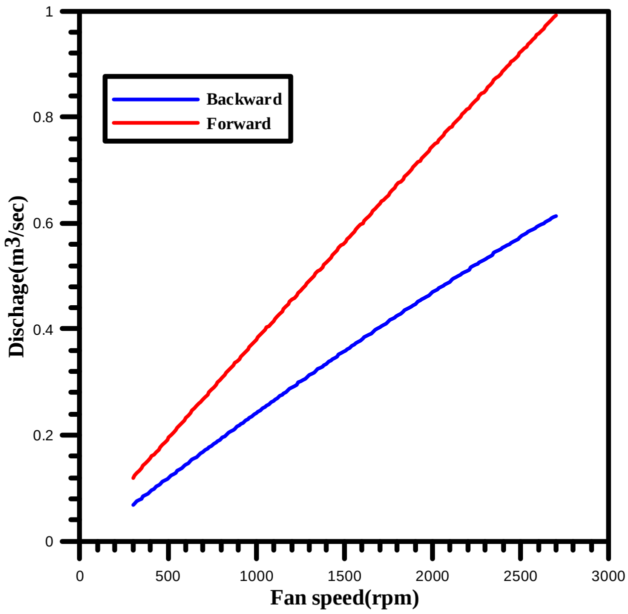


Figure (12) Variation of average dynamic pressure with fan speed (forwardrotation mode)





a-Average dynamic pressure b- Discharge

Figure(13) Comparison between forward and backward rotational modes

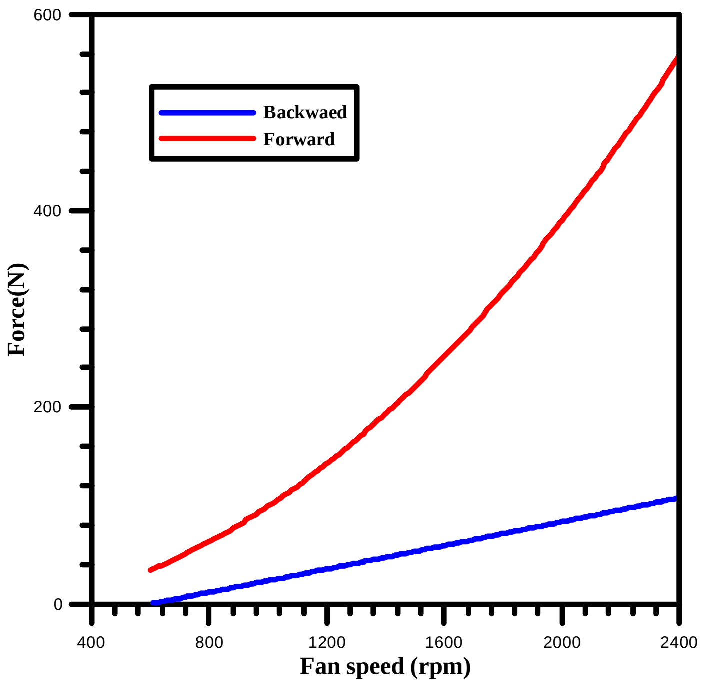


Figure (14) axial thrust force developed by the fan impeller

5- Conclusions

1- Increasing the speed of motor caused increasing is (98.4385%) in the average dynamic pressure at outlet and on the blade of the impellers.
2- The average of dynamic pressure for forward rotation mode is (61.82933%) greater than that of backward direction mode.
3- The results exhibited that the maximum discrepancy between the experimental and numerical is (9.8%) for backward rotating and (10.44%) for forward directionat different speeds.
4- The magnitude of dynamic axial thrust forces in the z – direction are maximum for forward rotation mode compared with another rotation mode because the differences in pressure on the impeller fan.
5- The experimental and theoretical results are in well agreement, with the (CFD) model developing.

Nomenclature

Pn Individual dynamic pressure measurements (N/m^2)
P Dynamic pressure (N/m^2)
ρ Density of air (kg/m^3)
V Average velocity (m/sec)
Q Airflow rate (m^3/sec)
A Cross – sectional area (m^2)

6- References

1- Douglas A. Roberts, 2004"A Comparison of Steady-State Centrifugal Stage CFD Analysis to Experimental Rig Data "Pratt & Whitney Canada, ANSYS/CFX Canada.
2- M. Younsi, F. Bakir, S. Kouidri, and R. Rey, 2007"Influence of Impeller geometry on the unsteady flow in a centrifugal fan: Numerical and Experimental analysis " International Journal of rotating Machinery , Article ID 34901, 10 pages.
3- Yogesh R. Pathak, Beena D. Baloni, Dr.S.A.Channiwala, Apr-2012" Numerical simulation of centrifugal blower using CFX" ,International Journal of Electronics, Communication & Soft Computing Science and Engineering,ISSN: 2277-9477.
4- 1Keyur K. Patel, 2Prajesh M. Patel, Jan 2013, "Performance Improvement of centrifugal fan by using CFD",International Journal of Advanced Engineering Research and Studies, Review Article, IJAERS, vol. (2), ISSU (2).
5- Atre Pranav C. and Thundil Karuppa Raj R., October (2012),"Numerical Design and Parametric Optimization of centrifugal Fans with airfoil blade impeller "Research Journal of Recent Sciences, vol.1 (10).
6- Kamal, N.A. ,20013," Experimental and Theoretical study on cavitation effect in centrifugal pumps" M.Sc. thesis university of Baghdad.
7- S.M.Yahya,1983, " Turbines compressor and fans " ,Indian Institute of Technology.

8- AMCA 210, 1999, “LaboratoryMethodof testing fans for Aerodynamic performance rating “ Air Movement and control Association , 10 CFR 430 Subpart B , App.M ,Approved by ANSI on December.

9- Air flow , “Type 4 test set “ , operating instruction 900449 / D/ 196.

10- C. N. Jayapragasan, Sumedh J. Suryawanshi and K. Janardhan Reddy, 2014,”Design Optimization of centrifugal fan of travelling cleaner “ , ARPN Journal of Engineering and Applied Sciences, VOL. 9, NO. 9.

11- K. Turgut GÜRSEL, Onur GÜRSEL, Mehmet ERKEK,“Developing an industrial centrifugal fan as prototype using an experiment series and finite volume method “ , International Journal of Engineering Research and Development, Vol.4, No1, June

12- Javad Alinejad and Farhad Hosseinnejad ,(2012)”Aerodynamic Optimization In the Rotor of Centrifugal Fan Using Combined Laser Doppler Anemometry and CFD Modeling “,World Applied Sciences Journal 17 (10) .