Journal of Engineering and Sustainable Development

Vol. 23, No.04, July 2019
ISSN 2520-0917
https://doi.org/10.31272/jeasd.23.4.12

INVESTIGATION THE EFFECT OF BLADE'S SLOT ON ROTATING STALL OF HIGH SPEED BLOWER

Dr. Muna Sabah Kassim [1], Dr. Fouad Aluan Saleh [2], *Alaa Thayer Aliwi [3]

1) Assistant Prof., Mechanical Engineering Department, Mustansiriyah University, Baghdad, Iraq.
2) Assistant Prof., Mechanical Engineering Department, Mustansiriyah University, Baghdad, Iraq.
3) MSc.Student,Mechanical Engineering Department, Mustansiriyah University, Baghdad, Iraq.

Received 27/2/2018 Accepted 23/4/2018 Published 1/7/2019

Abstract: Experimental and numerical investigations to study the effect of add one slot to the impeller blades on the rotating stall and pressure fluctuations in a high speed centrifugal blower. The experimental test rig which includes a blower of centrifugal, transducer of pressure as well measurement instrumentations which constructed for this study. A data acquisition system (hardware) and its (software) have been developed to transfer the signal from transducer of pressure to the computer then analysis with time. The experimental work has been implemented through measuring the static pressure variation as well fluctuating of pressure for two cases of impeller (without slot and with one slot). Static pressure has been taken in different points prepared on the front-wall of the volute casing along one track for two cases of the impeller. This track is angular track about the impeller. The results of experimental show that the fluctuations of pressure increase with the mass flow rate decrease. Also the results indicate that fluctuations of pressure decrease with adding slots into the impeller blades. The simulation of numerical has been carried out on blower of centrifugal into analysis both field of flow and fluctuations of pressure through using ANSYS (FLUENT 15). The simulation of numerical has been completed through solve the continuity and momentum equations with moving reference framework technique inside the blower. The numerical simulation results show a good agreement with the results of experimental.

Keywords: *Centrifugal Blower, Rotating Stall, Surge, Impeller and Slots.*

تحقيق لتأثير الشق في الريش على الانهيار الدوار في منفاخ ذو السرعة العالية

الخلاصة: تم انجاز دراسة عملية وعددية في هذا العمل وذلك لدراسة تأثير اضافة شق واحد الى ريش الدافعة على الانهيار الدوار وتذبذب الضغط في منفاخ الهواء ذو السرعة العالية. فتمت الدراسة باستخدام متحسس الضغط ومعدات قياس الجريان التي صممت لتناسب متطلبات هذا البحث، كما تم تصميم وتصنيع معالج الاشارة لنقل الاشارة من متحسس الضغط الى جهاز الكمبيوتر ليتم تحليلها بدلالة الزمن. وقد نفذت الدراسة التجريبية بقياس التغير الحاصل في الضغط وتذبذب الضغط لحالتين اثنتين للدافعة (بدون شق ومع شق واحد). وقد تمت قراءة قيم الضغط لعدة نقاط والتي اعدت خصيصا على الجدار الامامي للمجمع ولمسار واحد هو مسار محيطي حول الدافعة. وقد اظهرت النتائج العملية ان تذبذب الضغط يرتفع عند نقصان معدلات الجريان. وأيضا تم استنتاج بان معدلات تذبذب الضغط تقل عند اضافه الشقوق الى الدافعة. وقد أنجزت محاكاة عددية لمنفاخ الهواء لتحليل جريان المائع وتذبذب الضغط باستخدام برنامج ANSYS (FLUENT 15). وتم حل المحاكاة العددية باستخدام معادلات الاستمرارية والزخم مع تقنية الهيكل الدوار للدافعة. وان نتائج المحاكاة العددية للجريان قد اعطت نتائج متوافقة وبصورة جيدة مع نتائج التجارب العملية.

*Corresponding Author alaa.alhjame1996@yahoo.com

1. Introduction

A turbo machine is a device at which energy transferring happens between flowing fluid and a rotary element due into dynamic action resultant at an alteration at pressure as well momentum of the fluid. Mechanical energy transferring happens into or out of turbo machine, generally at flow of steady [1].

The beneficial domain of operation of systems of centrifugal is limited, through chocking in high mass flows when sonic speed is reached at some components, and in low mass flows through the beginning of two instabilities recognized as (rotating stall as well surge). The rotating stall is a three dimensional instability characterized through the existence of one or more cells of stall fluid rotating slower from the rotor. Indeed, the cells induce a partial blockage of the machine and important damages, lead to drop at the mass flow as well ratio of pressure [2]. Surge is phenomenon of system is not only dependent on the system of centrifugal, however onto each components of the operation; such as, valves, piping, impeller, pressure vessels, volute, etc. Surge is defined as operating condition in which whole flow reversal happens; i.e., flow proceed backward by the system of centrifugal section as well comes out the inlet [3]. According to literature survey, slots are made in the separation points in order to attenuate the layer of boundary through boundary layer buildup can be controlled at the impeller. These slots had been designing at the literature survey at such a way that a jet of fluid effluxes by the converge slot than the side of pressure to the side of suction.

A large deal of work has been implemented onto flow conduct investigation at different portions of the turbo machines, such as inlet duct, outlet duct, impeller, vane less and vane diffuser and volute, etc. Guleren and Pinarbasi [4] analyzed the simulation of numerical of centrifugal pump through solve (Navier – Stokes equations), to side the (standard k–e) model of turbulence. The pump contains the impeller have (5) curved blades with (9) vanes of diffuser. The shaft rotates in (890rpm). Techniques of numerical analysis are implemented onto a commercial (FLUENT) package program assume flow of incompressible, steady conditions and decreasing rate of flow. Being below design conditions, there is a constant high-velocity leakage flow at the gap between the diffuser and the impeller than the exit side of the diffuser into the onset of volute. Separation of this flow of leakage than the vane of diffuser reasons the beginning of stall. As the rate of flow decreases both the magnitude of the leakage within the vane less portion of the pump and reverse flow within a stalled increase of diffuser passage. As this happens, the stall cell size extends than one to two passages of diffuser. Comparisons are made with results of experimental as well showed good agreement. Stefan et al. [5] studied the instability of flow as well rotating stall at a high energy centrifugal pump phase. The sensors of pressure were piezo-resisstive small transducers of pressure allowing the measuring of together, static pressure and unsteady utilized to the sensors surface. Sensors of pressure had been installed at the diffuser as well the impeller. The measurements at impeller trailing edge allowed the identity of stall at different diffuser channels, which were showed as stationary or rotating discontinuities at the patterns of periodic pressure fluctuation. The rotational velocity of the stall cells was less than 1.0 % of the impeller rotational velocity. Sivagnensundaram et al. [6] studied the enhancement of compressor diagram width by shroud bleed slot with different slot geometries. Three various slot geometries were gained through modify the originality slot width. 3D steady condition (CFD) simulations were implemented for single passage of the turbocharger centrifugal compressor phase into study the enhancement of the compressor diagram width. He showed that rising the slot width has large effect onto the compressor performing at the conditions of moving the point of surge to lower flow rate. At this analysis the broader slot width (0.4cm) reduced the surge flow through around (15%) comparison to the baseline slot (0.3cm). Kassim et al**.** [7] investigated to study the influence blade number onto the rotating stall as well fluctuations of pressure at centrifugal blower. The experimental test rig which includes a blower of centrifugal, a transducer of pressure as well measurement instrumentations which constructed for this study. A data logging system and its (software) have been developed to transferring the signal than pressure transducer to the computer. The experimental work has been implemented through measuring the variation of static pressure as well pressure fluctuating for three cases of the impeller which they various at blades number (5, 9, 10) as well outlet blade angle. The results of experimental show that the fluctuations of pressure increase with decrease values of mass flow rates. Also, the results show that the impeller with nine number of blades show high fluctuation of pressure comparison with other cases. The numerical simulation results gave good accord with the results of experimental.

At this study, experimental as well numerical investigations have been utilized so as to obtain the fluctuations of pressure in the impeller-volute of centrifugal blower. At present study, we shall discussing the effect of add slots into the impeller blades on fluctuation of pressure as well rotating stall inside the centrifugal blower. The results of numerical have been compared with results of experimental obtain than transducer of pressure which placed on the blower casing.

2. Description of the Blower and Experimental Procedures

The blower utilized at this work is single phase device with unshroud impeller as well volute. The blower is directly driven through 0.6 kW AC motor which has a constant speed of rotational which have maximum value (16000 rpm). The unshroud impellers test have (10) backward- curved blades and outlet diameter of (110mm) as shown at Figure (1). Table (1) shows the fundamental dimensions of the impeller as well Figure (2) show two cases of impeller (without slot and with one slot). Also, Figure (3) shows the schematic graph for slot which it made onto the impeller blades and Figure (4) shows the photograph of the impeller with slot in centrifugal blower. The test rig, as shown at Figure (5) and Figure (6), has been designed as well constructed to be suitable for the purpose of the present as well future study. The test rig has been constructed to than open loop system. The loop involved constant velocity electrical blower, a control valve, piping, as well metering orifice. The air is gathered through a volute (scroll) for circular cross sectional area. The outlet pipe for the blower is linked into orifice plate air flow meter through (100mm) length flexible pipe. The control valve installation onto the discharging side for the piping, and allows precise as well fine control for the mass flow rate.

The pressure of static measurement is implemented, through using transducer of pressure. The transducer is manufactured through Thornton (EMI) Company. The transducer is developed in order to show a pressure of differential. The signals of output than the transducer are suitably contingent as well digitized utilizing the data acquisition system. The signals of output are as well linked to observer the beginning for any disturbance at the flow. This observation will give continuously display of the difference as well measurement problems in order to avert it.

The measurements for the present work are implemented onto the front wall of the casing. Numerous measuring taps are placed on this wall, as shown at Figure (7). The numbers of taps are (11) and placed in each (30) interval about the front side for the impeller, as well in location of (5mm) than the impeller outlet.

The fluctuations of the flow in centrifugal blower are important components of the dynamic behavior of centrifugal system. Measurements of the behavior of the flow can be accomplished by employing high response transducers. With special data transmission system, it is possible to study the pressure fluctuations of the flow.

Experiments related to the fluctuating flow behavior usually tend to generate large quantities of data and they require high frequency response data transmission and recording system. Figure (8) and Figure (9) show the data acquisition system which it used in the present study.

High speed digital processing of data is essential in obtaining the desired information, this can be achieved by using data acquisition (or data logger) system. The high response transducer output signal is processed by using personal computer. The transducer signal is initially passes to the amplifier (LM741) to amplify the output signal and then digitized and prepared by 10-bit analogue to digital converter (ADC) which is based on 16F877A. The data logging process is controlled by additional processes in the software.

Table (1) Main Characteristics of the Tested Impeller

Impeller parameters	Case
Impeller exit diameter	D = 110 mm
Impeller inlet diameter	d = 28 mm
Number of impeller blades	Z = 10
Maximum speed	Ω = 16000 rpm
Inlet blade angle	$\beta_1 = 54^\circ$ from tangential direction
Outlet blade angle	$\beta_2 = 42^\circ$ from tangential direction
Blade thickness	t = 3 mm
Discharge width	b = 25 mm
The gap between the tongue and the impeller	B = 12 mm

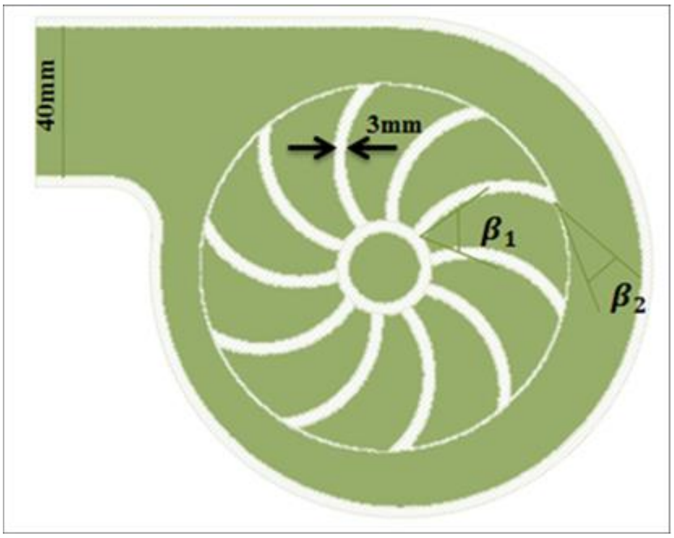
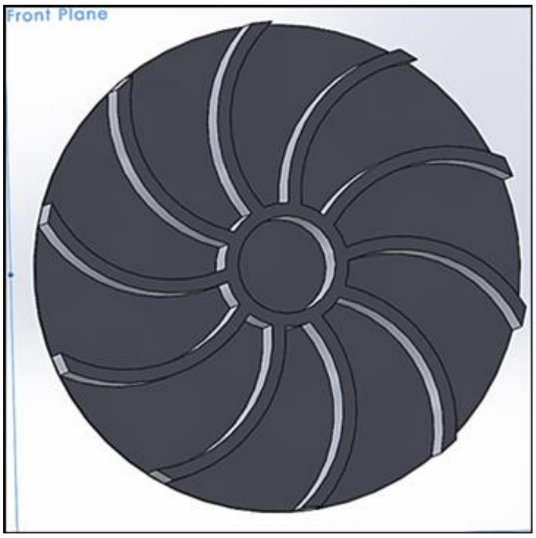
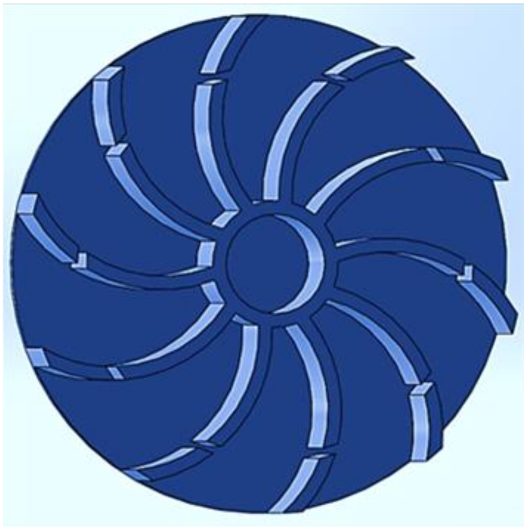


Figure (1): The Centrifugal Blower Dimensions.





Without slot With one slot (2mm)

Figure (2): Impeller Shapes.

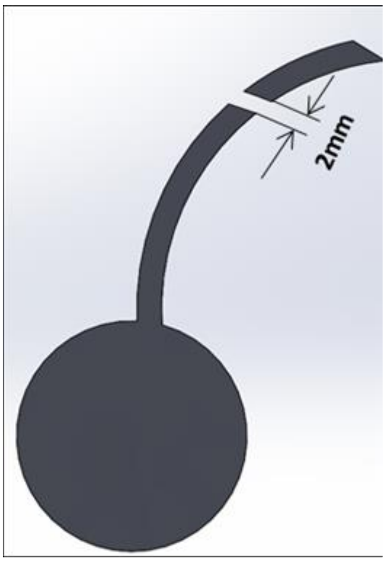


Figure (3): Schematic Diagram for Slot of Impeller.

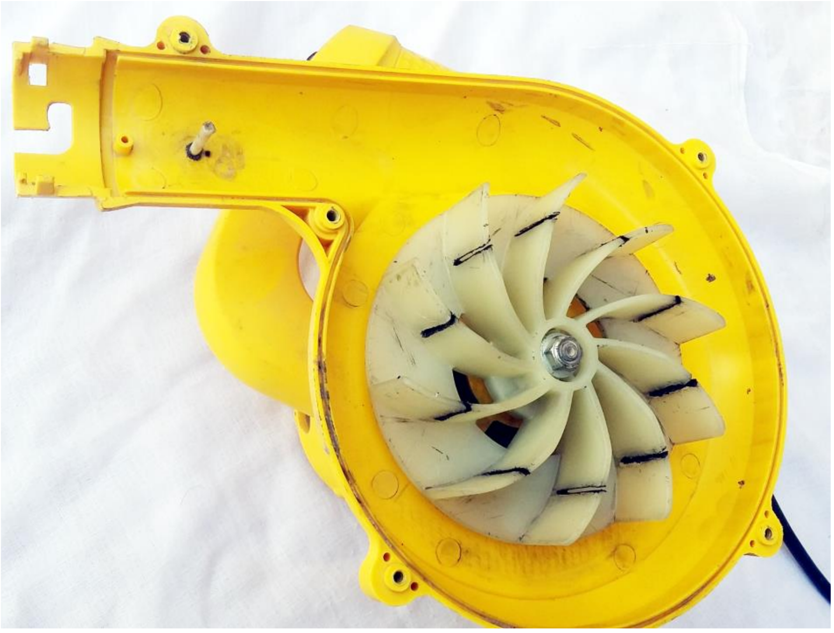


Figure (4): Photograph for the Impeller with Slot in Centrifugal Blower.

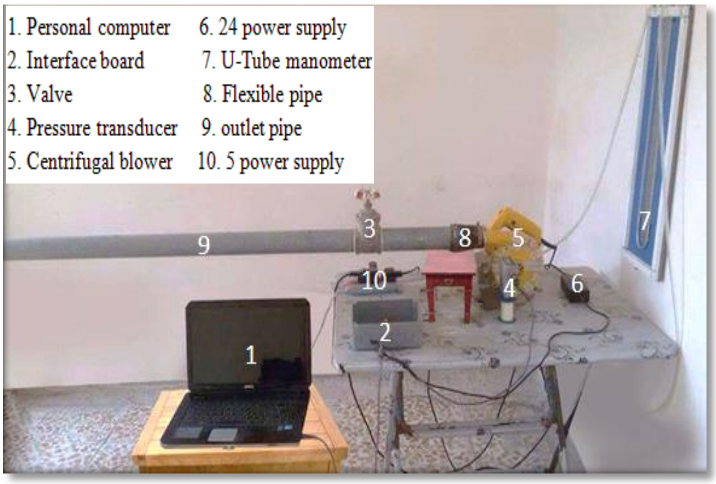


Figure (5): Experimental Test Rig and Devices of Measurements.

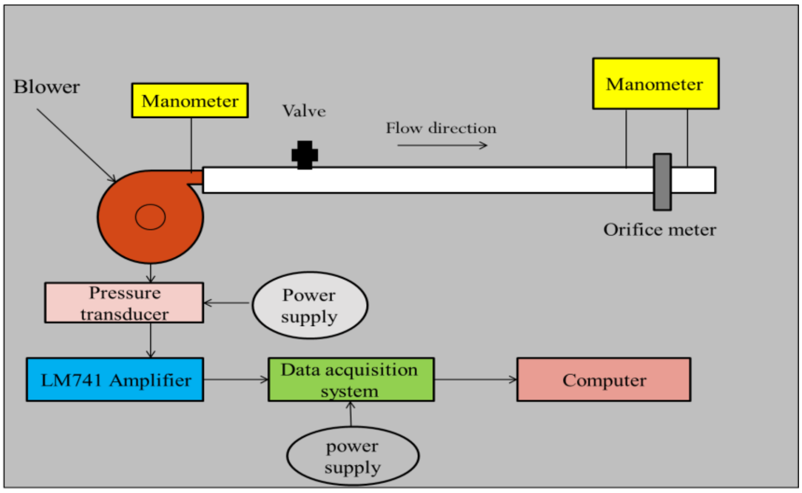


Figure (6): Schematic Diagram for Test Rig.

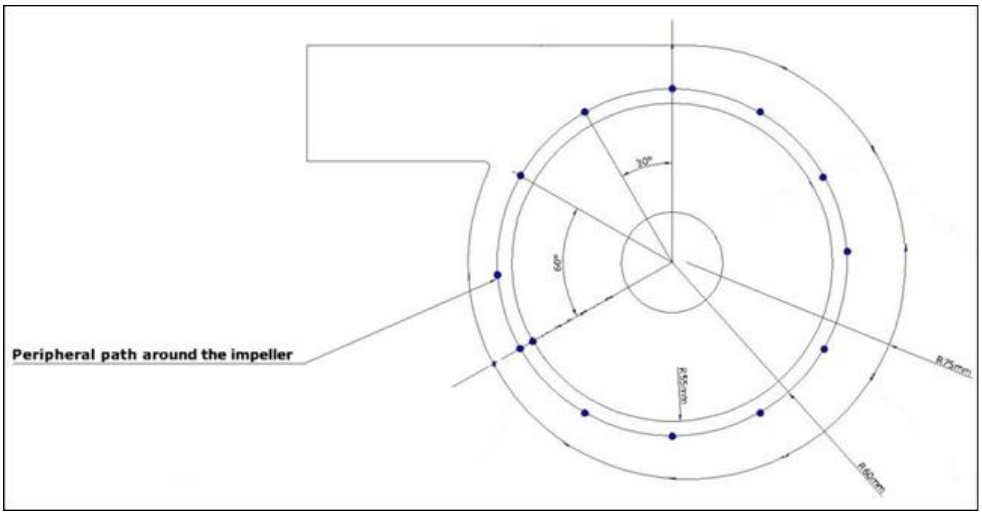
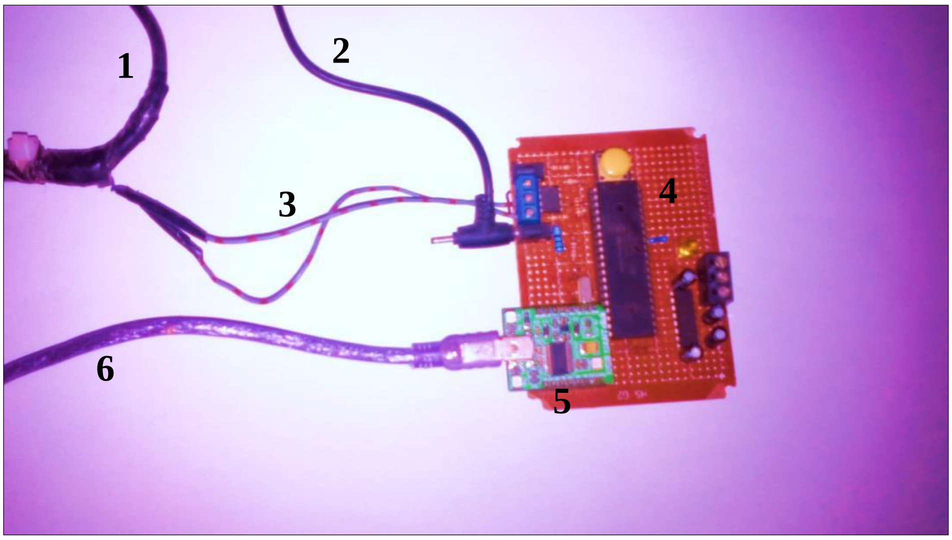


Figure (7) Locations of Measuring Points.

Figure (8): Data Acquisition System.



1- Power supply to the transducer 2- Power supply to the ADC
3- Signal input 4- Interface board 5- USB (UART) 6- Output signal
Figure (9): Photograph of Data Acquisition System.

3. Numerical Flow Simulation

Simulations of numerical for the unsteady flow at the blower of centrifugal described above were implemented. In order to confirm the capability of the model of numerical to characterize the flow advantages inside the blower, 3-D numerical simulation of the unsteady flow has been implemented. Each the calculations have been implemented with commercial (software) package FLUENT 15.

In this study, the following assumptions were taken for simulation:

1. The friction coefficients for each surfaces between the walls as well fluid were neglected.
2. The walls of the casing were presumed to be smooth therefore any turbulences at flow due to roughness of the surface were neglected.
3. Steady state conditions.
4. Incompressible flow (ρ=constant)
5. Newtonian fluid.
6. Flow of turbulent.
7. 3-D flow simulations.

The conservation equation for continuity as well momentum equations can be written as follows [8]:

1- Continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \quad (1)$$

2- Momentum equation in x-direction:

$$\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(\lambda\nabla.\vec{V} + 2\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\rho f_x \quad (2)$$

3- Momentum equation in y-direction:

$$\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)\right]+\frac{\partial}{\partial y}\left(\lambda\nabla.\vec{V} + 2\mu\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right]+\rho f_y \quad (3)$$

4- Momentum equation in z-direction:

$$\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right]+\frac{\partial}{\partial z}\left(\lambda\nabla.\vec{V} + 2\mu\frac{\partial w}{\partial z}\right)+\rho f_z \quad (4)$$

Geometric discretization of the blower of centrifugal is made for the numeric treatment, and computational mesh is produced by (Fluent) preprocessor (Gambit). There are basically two types of approaches at meshing of volume, unstructured as well structured meshing. At the unstructured approaches, the integral method for governing equations is discretized, either finite-element or finite-volume schema is utilized. Grids of unstructured are generally successful for complicated geometries, therefore it was utilized at present study.

At grid of structured, the equations of governing are transformed into the curvilinear coordination system aligned with a surface. So, it becomes highly inefficiently as well consumes time for complicated geometries.

Therefore, it has been excluded at this study. At this study tetrahedral mesh type was used one of types of unstructured grid because it is superior at the complicated geometry. Figure (10), demonstrates the mesh of centrifugal blower.

The point of final at a good grid is the cells total number produced. It is necessary to have enough cells number for good decision however memory requirements increase with the cells number increase. The cells average number at this study are (2.6) millions.

A control - volume based technicality that involves the following stages is utilized for solve [9]:

1) For pressure, speed as well conserved scalars, algebraically sets of equations are constructed through the incorporation of the governing equations onto all control volume.
2) Discretized equations are solved iteratively as well linearized.

Any solve for a collection of PDE's. needs a collection of conditions of boundary for closing. From physical perspective one requires to specifying conditions of boundary of flow variables in each boundary regions of the flow field.

The conditions of modeled boundary are those considered with more physical meaning of turbomachinery flow simulations, namely, total pressure in the inlet of domain as well static pressure proportionate to the kinetic energy at the outlet of domain [10].

The rate of flow is changed through modifying the pressure in the condition of outlet, which simulates the valve closing.

Turbulence is simulated with SST k-ω model. Air is used as working fluid. The semi implicit method pressure link equation (SIMPLE) algorithmic, second order, upwind discretization have been utilized into implemented the solving of flow inside the blower.

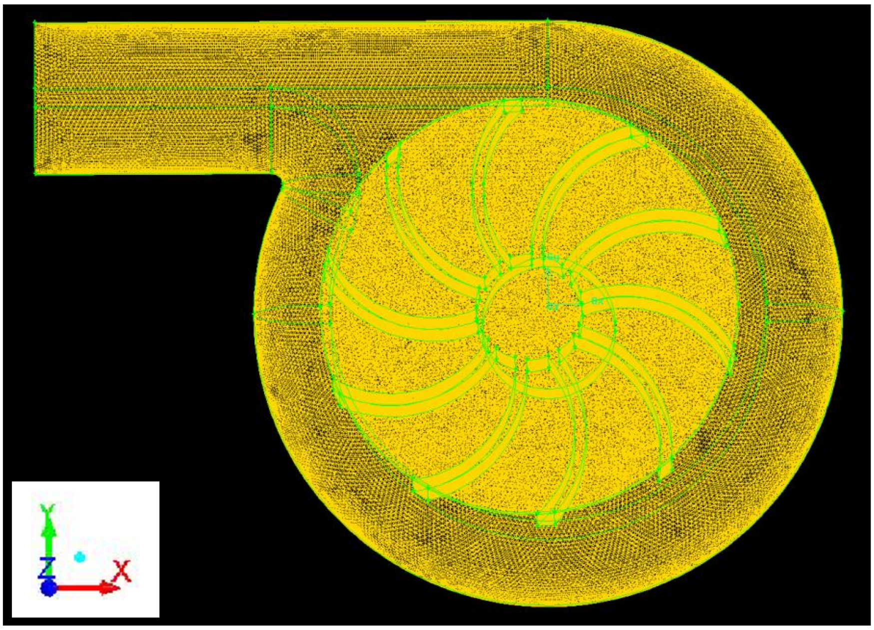


Figure (10): The Mesh of Centrifugal Blower Generated in the Gambit.

4. Results and Discussions

We shall discuss the results linked into the flow behavior at impeller - volute of centrifugal blower (without slot and with one slot). At this work detailed study as well analysis of the rotating stall at term of fluctuations of pressure at time domains has implemented. The experimental were implemented under constant speed of rotational (16000 rpm), for different mass flow rates. The fluctuations of pressure take in one point on the casing of centrifugal blower. This point in angular position (30°). Tables (2) and (3) show the specifics of the mass flow rates calculations. The values were selection arbitrarily however the maximum values of the mass flow were limited through the design of control valve as well speed of rotational.

Table (2) Details of Final Calculation for Mass Flow Rate

Case 1			
ΔP (Pa)	$Re_D * 10^4$	c	$\dot{m}$ (kg/s)
4620	2.9319	0.617103	0.02501
3465	2.5404	0.617329	0.021674
2310	2.0749	0.617562	0.017702
0	0	0	0

Table (3) Details of Final Calculation for Mass Flow Rate

Case 2			
ΔP (Pa)	$Re_D * 10^4$	c	$\dot{m}$ (kg/s)
4700	2.9577	0.617121	0.02523
3525	2.5626	0.617318	0.021861
2350	2.0926	0.617553	0.017855
0	0	0	0

4.1.Time Domain Analysis

Fluctuations of static pressure at time domain on a period of (3) second with different mass flow rates in angular location ($\emptyset = 30^{\circ}$) (beyond of the tongue) about the impeller for two cases are studied. The figures show that fluctuations of pressure for different mass flow rates are non-periodical at nature.

Figure (11) and Figure (12) demonstrate the fluctuations of pressure for different mass flow rates in speed of rotational (16000 rpm) for cases (1, 2). Figure (11), the impeller is without slot while Figure (12), the impeller is with one slot. It is obvious than these figures that the fluctuations of pressure increase with the mass flow rates decrease. The maximum pressure fluctuations amplitude happens in mass flow rate equal to approximately non-flow. The figures show also that the static pressure increases and fluctuations of pressure decreases through adding one slot into the impeller blades.

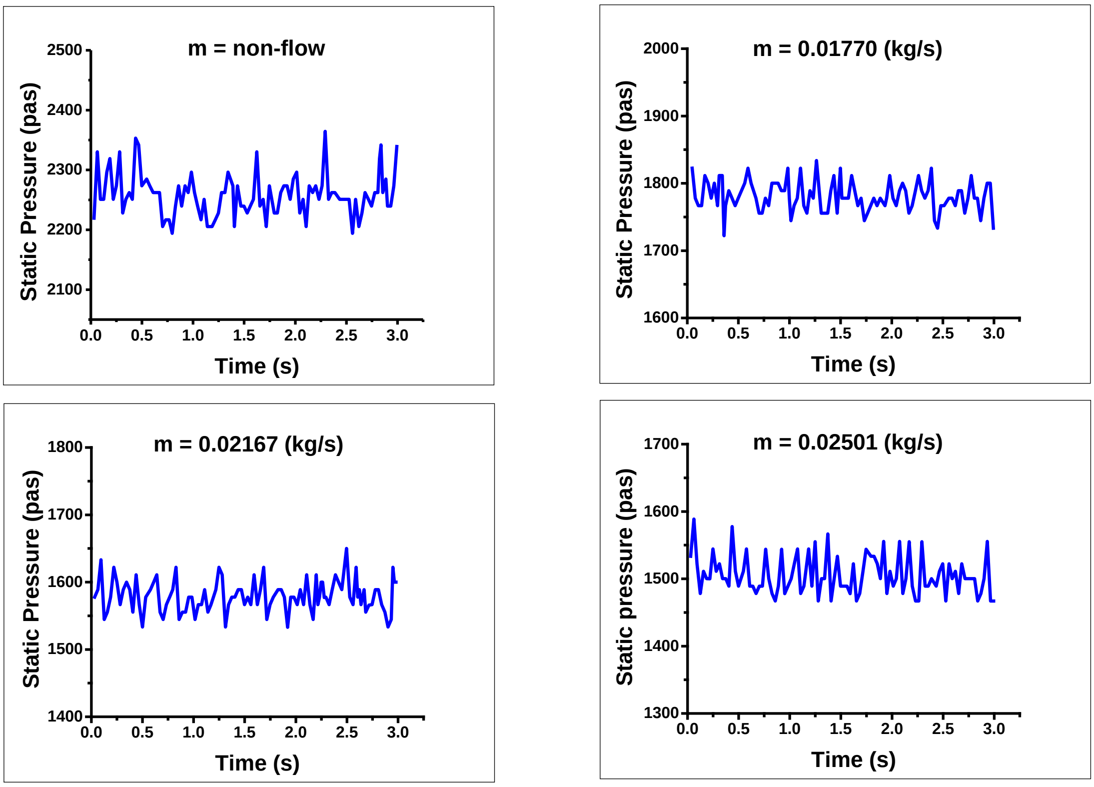
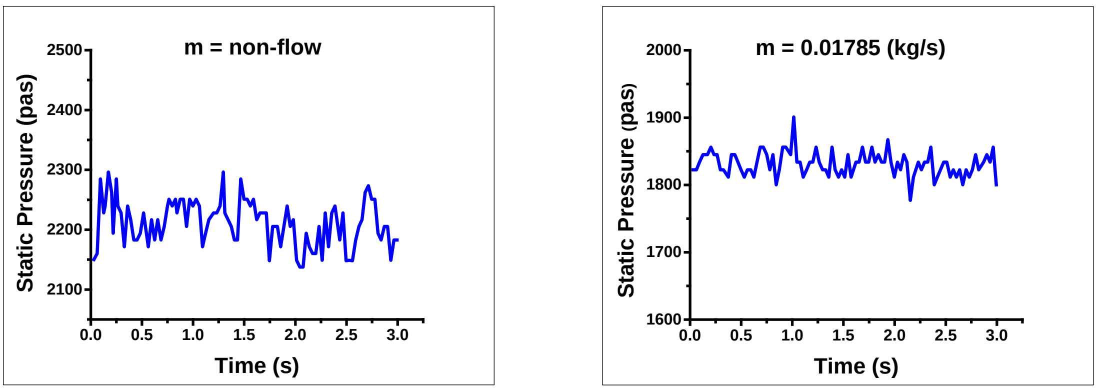


Figure (11): Pressure Fluctuations for Various Values of Mass Flow Rate (kg/s) and at Rotational Speed 16000 rpm (Case 1), Without Slot.



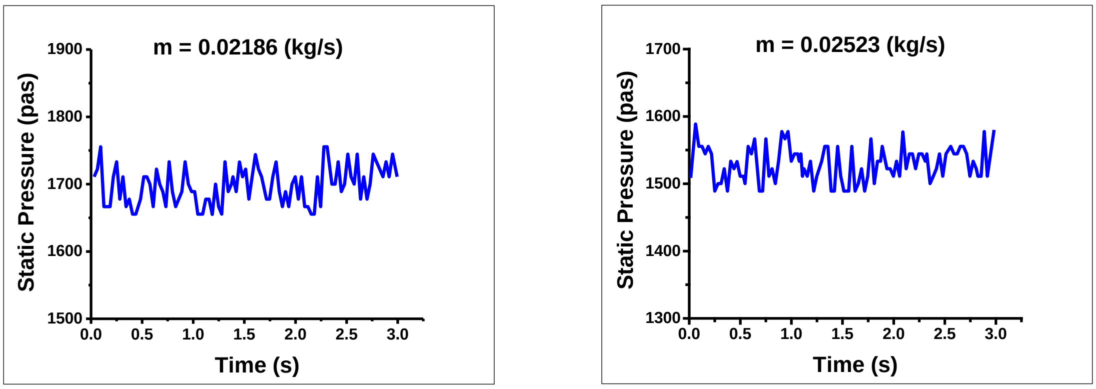


Figure (12): Pressure Fluctuations for Various Values of Mass Flow Rate (kg/s) and at Rotational Speed 16000 rpm (Case 2), With One Slot.

4.2. Static Pressure Distribution about the Impeller

Figure (13), demonstrates the distribution of static pressure along the angular location about the impeller for two cases (1, 2) for different flow rate values in speed of rotational (16000 rpm). From this figure, the minimum value for pressure of static in case (1) is showed in angular location (Ø=120) in all mass flow rates values. In case (2) it is showed that the minimum value for pressure of static in angular location (Ø= 60°) and (Ø= 120°). The figure, also shows that the pressure of static have maximum value at the vicinity of the tongue in (Ø= 0°).

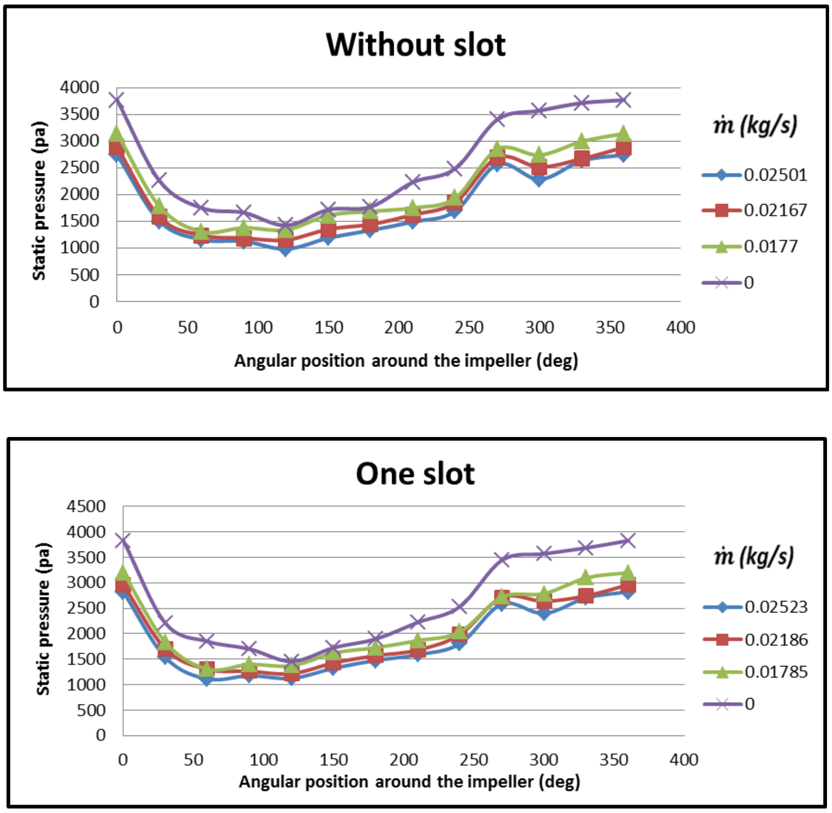


Figure (13): Circumferential Static Pressure Distribution around the Impeller for Several Values of the Flow Rate and No. of Revolution = 16000 rpm for Two Cases (1, 2).

4.3. Analysis of Numerical Results

The possibilities of the simulation of numerical at the flow study inside blower are broader from the experimental ones. Especially, results corresponding into the distributions of pressure inside the impeller, the flow at volute are presented, as well the unsteady calculation common with the moving reference framework technique has proved to be a good tool into study the volute-impeller interaction. The results of numerical simulation taking for the plane geometry at z = 18mm for cases (1, 2) from casing of blower.

Figures (14) to (17) demonstrate the distribution of static pressure for two cases of impeller (1, 2) in different mass flow rates and in speed of rotational (16000 rpm). The static pressure increase by the blower is clearly seen at these figures, as are the radial pressure gradients.

The static pressure has a minimum value in the eye of impeller, and about the impeller in angular location between ($\text{Ø} = 90^{\circ}$ to 180°). We as well showed the minimum static pressure in angular location ($\text{Ø} = 300^{\circ}$).

The static pressure increases between the angular location ($\text{Ø} = 0^{\circ}$ to 90°) about the impeller and also the static pressure increases between angular location ($\text{Ø} = 30^{\circ}$ to $\text{Ø} = 270^{\circ}$) about the volute. The flow recirculation showed than the distance between the tongue and the exit of impeller. This leading to increase the pressure of static between angular location

The results of numerical simulation were recorded for the same locations considered in the experiments. The static pressure distribution around the impeller for two cases (1, 2) for maximum mass flow rates at rotational speed 16000 rpm compared in Figure (18).

The agreement between the experimental and numerical data is good of fairly. Some differences have arisen at the comparison between the experimental and numerical static pressure at the impeller of tested centrifugal blower, particularly beyond the tongue region.

From the comparison it can be concluded that the increase in pressure of static beyond the tongue at the simulation of numerical is due to the flow recirculation at the region between the tongue as well exit of impeller, note that percentage of difference between the numerical and experimental is 4.8%.

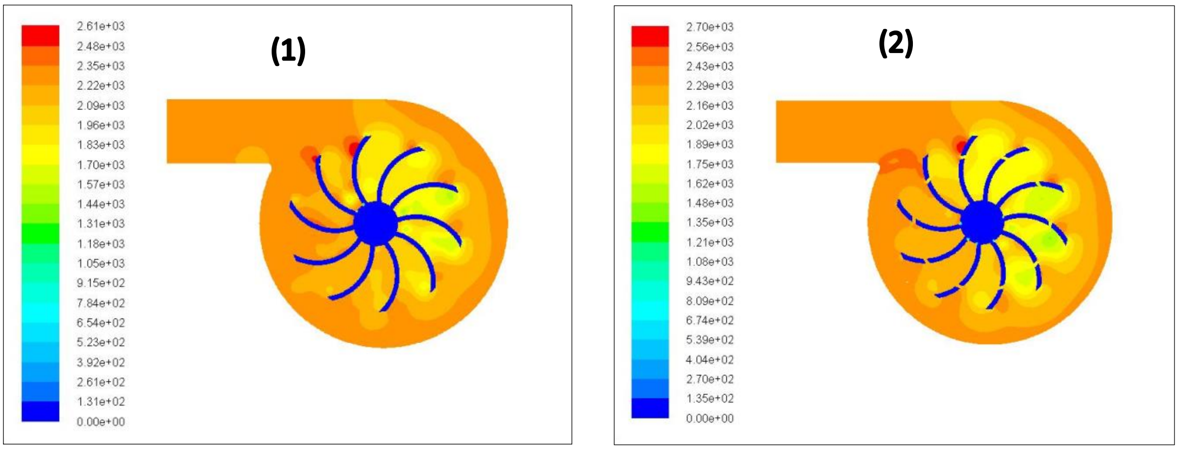


Figure (14): Contours of Static Pressure (pas) for Two Cases (1, 2) at Rotational Speed 16000 rpm and at ṁ = non-flow.

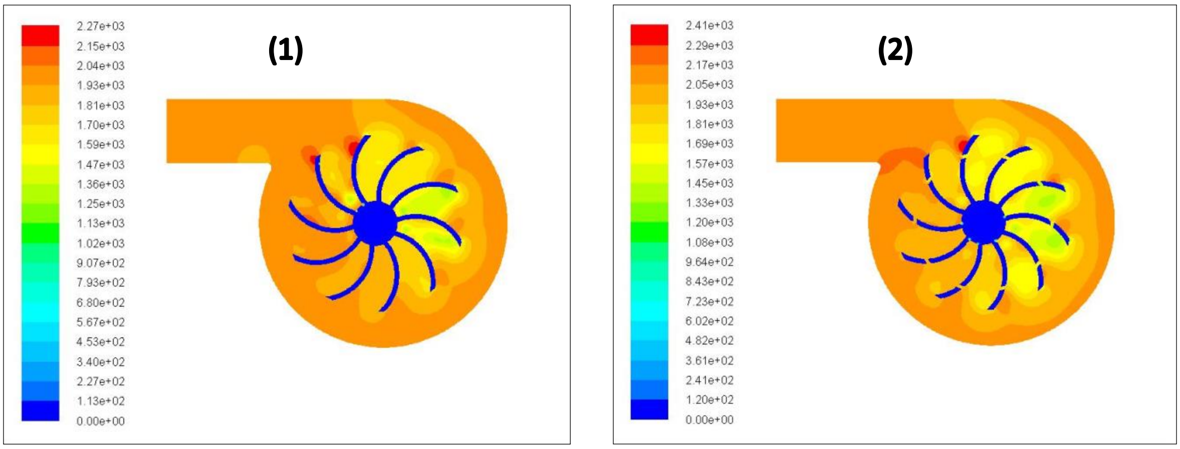


Figure (15): Contours of Static Pressure (pas) for Two Cases (1, 2) at Rotational Speed 16000 rpm and at ṁ = 0.01770, 0.01785 kg/s Respectively.

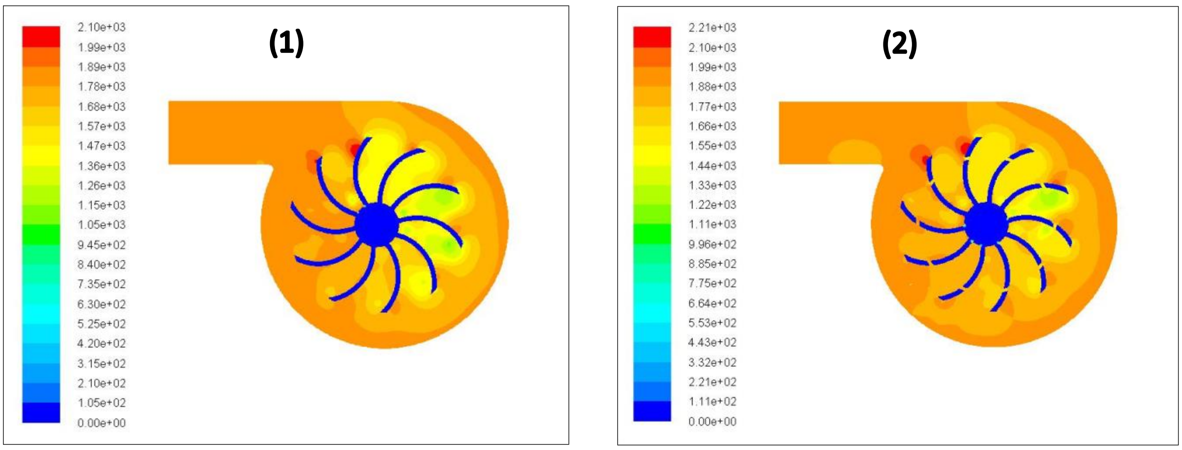


Figure (16): Contours of Static Pressure (pas) for Two Cases (1, 2) at Rotational Speed 16000 rpm and at ṁ = 0.02167, 0.02186 kg/s Respectively.

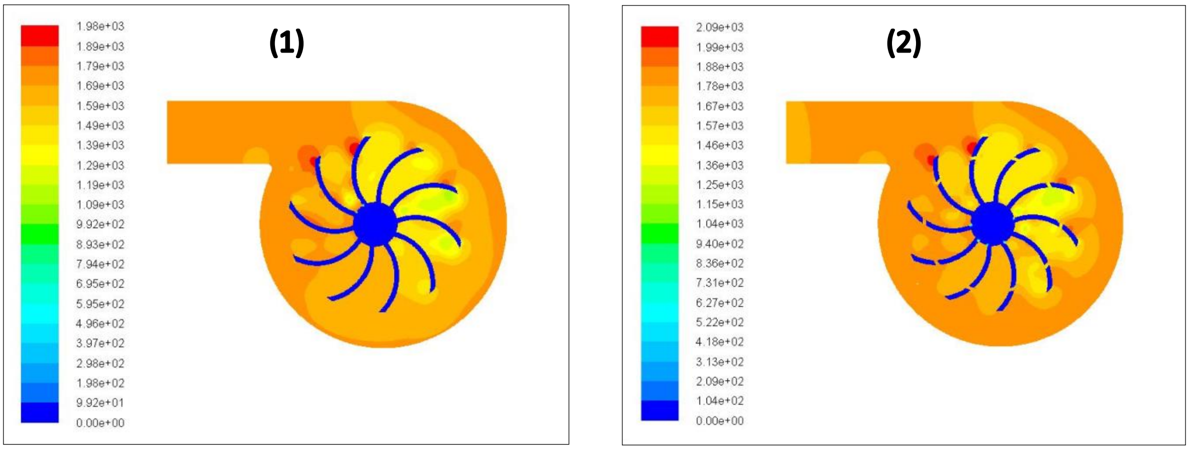


Figure (17): Contours of Static Pressure (pas) for Two Cases (1, 2) at Rotational Speed 16000 rpm and at $\dot{m}$ = 0.02501, 0.02523 kg/s Respectively.

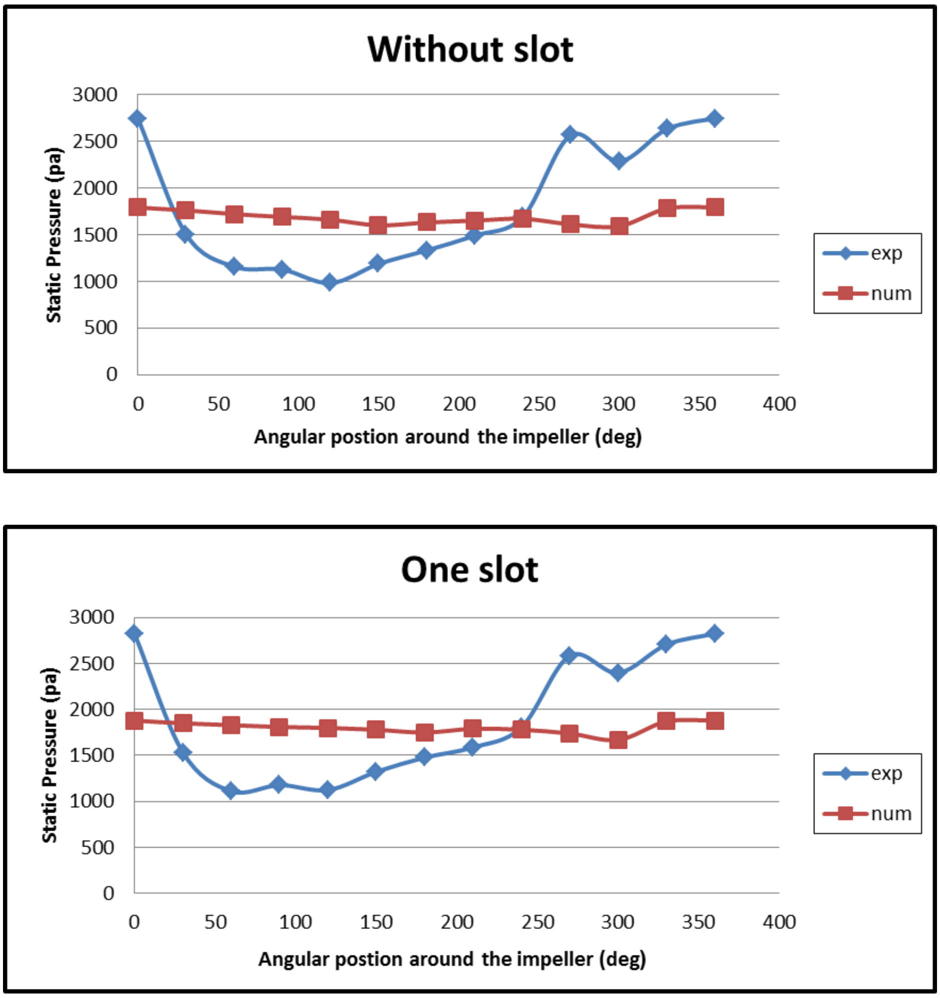


Figure (18): Comparison of Static Pressure around the Impeller for Two Cases (1, 2) at Maximum Mass Flow Rate and at Rotational Speed 16000 rpm.

5. Conclusions

The main task of the present study is to develop as well construct a research test rig and carry out tests of experimental onto it. The static pressures were measured in locations along circumferential tracks about the impeller. These results with flow simulation results leading to the following conclusions:

1. The minimum value for pressure of static about the impeller for case (1) showed in angular location ($\emptyset = 120^{\circ}$) in all mass flow rates values. In case (2) showed the minimum value for pressure of static in angular location ($\emptyset = 60^{\circ}$) and ($\emptyset = 120^{\circ}$).
2. The maximum value for pressure of static about the impeller for cases (1, 2) showed in angular location ($\emptyset = 0^{\circ}$) in all mass flow rates values.
3. The fluctuation of pressure for different mass flow rate is non-periodical nature, the pressure fluctuations increase with the mass flow rates decrease. The maximum pressure fluctuations amplitude happens in mass flow rate equal to approximately non-flow.
4. The pressure fluctuations amplitude decrease with adding slots into the impeller blades. Detailed analysis shows that the fluid than slots is injected into the passage of blade near the surface as well re-energizes the leakage flow.
5. The agreement between the experimental as well numerical static pressure data at the impeller is good of fairly, some differences have arisen, particularly beyond the tongue region and the average percentage between the experimental and numerical is 4.8%.

Nomenclature

D	Impeller exit diameter (m)
d	Impeller inlet diameter (m)
Z	Number of blade
β	Blade angle (deg)
ΔP	Pressure difference (pascal)
$\dot{m}$	Mass flow rate (kg/s)
C	Discharge coefficient
Re	Reynolds number
∅	Circumferential angle (deg)
SST	Shear stress transform
u, v, w	Velocity component (m/s)
ρ	Density (kg/m^3)
Ω	Impeller rotational speed (rpm)
μ	Dynamic viscosity (kg/m.s)
CFD	Computational Fluid Dynamic
PDE	Partial Differential Equation
P	Pressure (pascal)
ADC	Analogue to Digital Converter

6. References

1. Kadambi V. and Parasad M., (1974). "*Energy Conversion*", Vol. II, Energy Conversion Cycle, New Age International Publishers, New Delhi, PP.74-226.
2. Nicolas Courtiade, (2012). **"***Experimental Analysis of the Unsteady Flow and Instabilities in a High-Speed Multistage Compressor***"**, Thesis.
3. Saeid Niazi, (2000). "*Numerical Simulation of Rotating Stall and Surge Alleviation in Axial Compressors*", Ph. D., Thesis, Georgia Institute of Technology.
4. K. M. Guleren and A. Pinarbasi, (2004). "*Numerical Simulation of the Stalled Flow within a Vaned Centrifugal Pump*", Proc. Instn Mech. Engrs Vol. 218 Part C: J. Mechanical Engineering Science.
5. Stefan B., Philippe D., Laurent F., Maher K., Francois A. and Mohamed F., (2009). **"***Experimental Investigation of Flow Instabilities and Rotating Stall in a High-Energy Centrifugal Pump Stage*", Proceedings of ASME Fluid Engineering Division Summer Meeting, FEDSM.
6. S. Sivagnanasundaram, S. Spence and J. Early, (2010). "*Investigation of Compressor Map Width Enhancement and the Inducer Flow Field Using a Shroud Bleed Slot*", School of Mechanical and Aerospace Eng. Queen's University Belfast, UK.
7. Muna S. Kassim, Fouad A. Saleh and Mohammed A. Kadhum, (January 2015). "*Experimental and Numerical Investigation of Rotating Stall Phenomenon in Centrifugal Blower at Different Blade Number of the Impeller*", Journal of Engineering and Development, Vol. 19, No.1.
8. J.D. Anderson, Jr., (2009). "*Governing Equations of Fluid Dynamics*", National Air and Space Museum, Smithsonian Institution, Washington, DC, e-mail: AndersonJA @si.edu.
9. Versteeg H. K., Malalasekera W., (1995). "*An Introduction to Computational Fluid Dynamic, the Finite Volume*", Longman Group Limited.
10. Yonas Teshome, (2007). "*CFD Study on the Performance of Regenerative Flow Pump (RFP) with Aerodynamic Blade Geometry*", Thesis, Addis Ababa University.