

Experimental and Computational Investigation of Dynamics Effects in a High Speed Centrifugal Pump

Khudheyer S. Mushatet

College of Engineering

Thiqr University

Khudheyer2004@yahoo

Abdul Kareem A. Wahab

College of Engineering

Babylon University

kareemwahab@yahoo.com

Wissam H. Ajeel

lecturer

wissam_ajeel@yahoo.com

Abstract: - Experimental and numerical investigation of a high speed centrifugal pump has been carried out. In experimental work, a test rig which includes a centrifugal pump, fast response piezoelectric pressure transducers, Rota meter flow and measurement instrumentations is designed and constructed. Two impellers having different outlet diameters were examined for the same volute. A data acquisition system (hardware) and its software (Visual Basic language) have been developed to analyze the pressure transducer signal in order to obtain the pressure fluctuations. The governing partial differential equations (continuity, momentum) besides to turbulence model are discretized to algebraic equations by using finite volume method. The solution of these equations was done by using FLUENT 6.3 commercial code. This code has the ability to use a grid generation technique and transform the physical domain to a computational domain by using GAMBIT 2.2.3 software. The effect of turbulence was simulated by using a standard $k-\varepsilon$ model. The wall function laws are used to remedy the regions near the rotating and volute walls. The effect of angular position of impeller and impeller's diameter on the flow behavior is studied for different values of volume flow rate. The obtained results show that the pressure fluctuations registered in the volute were found to be very dependent on both angular position and volume flow rate. It is observed that the pressure fluctuations have maximum values at volume flow rate ($Q=0.5Q_N$) for the considered two impellers at angular position ($\varphi= 270^\circ$). These values increase as impeller diameter increases. The results also show that the higher static pressure values are observed upstream of the leading edge at the volute tongue (stagnation point) while the minimum was found at impellers eye. A comparison between experimental and numerical results is done and the agreement was acceptable.

Key words: Centrifugal pump, impeller volute interaction, shrouded impeller

المستخلص: تم اجراء دراسة عملية وعددية لسلوك الجريان والتغير غير المستقر للضغط في المجمع الحلزوني لمضخة الطرد المركزي ذات السرعة العالية. في الدراسة العملية تم انشاء وتصنيع منظومة اختبار تحتوي على متحسسات الضغط وادوات قياس معدل الجريان. استخدمت منظومة معالجة الاشارة لحساب اشارات تذبذب الضغط واجريت الاختبارات على دافعتين وبأقطار مختلفة $D_1=124\text{mm}$, $D_2=132\text{mm}$. تم معالجة الاشارات الكهربائية الخارجة من متحسسات الضغط عن طريق منظومة معالجة الاشارة (Data acquisition and logging interface) من اجل الحصول على اشارات تذبذب الضغط داخل المجمع الحلزوني لمضخة الطرد المركزي. في الدراسة تم تحويل المعادلات التفاضلية الحاكمة الى معادلات جبرية باستخدام طريقة الحجم المحددة وتم حل هذه المعادلات باستخدام برنامج (FLUENT 6.3) والذي لديه القابلية لاستخدام تقنية التحويل الشبكي وتحويل المجال الفيزيائي الى المجال الحسابي باستخدام برنامج GAMBIT 2.2.3. تم معالجة تأثير الاضطراب باستخدام نموذج الاضطراب $k-\epsilon$ بينما استخدمت قوانين دالة الجدار لمعالجة المناطق القريبة من جدران الدافعة والمجمع الحلزوني. تم دراسة تأثير البعد الزاوي وقطر الدافعة على سلوك الجريان ولقيم مختلفة من معدل الجريان. أوضحت النتائج التي تم الحصول عليها أن تذبذبات الضغط المسجلة تعتمد بصورة كبيرة على قطر الدافعة والموقع الزاوي للدافعة ومعدلات التدفق الحجمي للجريان حيث تصل الى اقصى قيمة عند معدل التدفق الحجمي ($Q=0.5Q_N$) والموقع الزاوي ($\varphi = 270^\circ$) علما ان هذه القيم تزداد مع زيادة قطر الدافعة. بينت النتائج ايضا ان هناك زيادة في قيم الضغط بعد مغادرة المائع منطقة القطع (stagnation point) وأن قيم الضغط الاستاتيكي تقل عند مركز الدافعة. أجريت مقارنة بين النتائج العملية والنتائج النظرية واعطت المقارنة توافقا مقبولا.

1. Introduction

The analysis of flow dynamics in a centrifugal pump is considered a very complex due to presence of system rotation, separation, turbulence and secondary flows. A pump is a machine which is called by this name due to a centrifugal force imposing on a liquid. The liquid attains its velocity and pressure when passing through impeller and this liquid is then decelerates at impeller. The centrifugal pump is widely separated in diverse technological applications. Many studies were done on investigating the flow dynamics and performance centrifugal pump. A particle Doppler velocimetry (PDV) technique was used by Dong et al.[1] to analyze the flow inside the volute of centrifugal pump. A particles $30\ \mu\text{m}$ used as seed. The obtained results showed that the entire flux pulsating within the value reached the maximum when the blade lines up with the tip of the tongue. Moreover, they observed that although most of the blade effects occur near the impeller tip, they are not limited to this region. The velocity distribution corresponding to design and off-design operation was studied by Elholm et al. [2]. LDV method was used to obtain the velocity distribution. They observed that the swirl has a forced vortex type velocity distribution and the location of its center was altered with mass flow rate.

Also it was found the circumferential curvature of the volute was responsible for a radial gradient of the flow velocity. Leakage flows in the tongue region was found to have a significant impact on position of the swirl center. Van Esch[2] performed an experimental study on the effect of a mixed-flow pump and hydrodynamic forces on the impeller. The obtained result show that the influence of the suction flow profile and blade interaction on pump performance and forces. Chu et al. [3] considered a particle image velocimetry complemented with noise and pressure measurements to measure the velocity distribution and compute the unsteady pressure field in the near-tongue region of the volute of a centrifugal pump. The concluded that the primary sources in noise generation were associated with the interaction of the non uniform out flux from the impeller (jet-wake phenomenon) with the tongue. Timushev and Ovsyannikov [4] performed the numerical techniques of pressure fluctuation amplitude estimation at blade frequencies in the volute casing passage of the centrifugal pump. Their results showed that the sources of hydrodynamic nature bring about considerable vibrations and noise of centrifugal pumps .A finite element technique was adopted by kaps [5] to study unsteady turbulent flow in centrifugal pumps. The conducted results showed that the strong influence of the casing tongue on the flow conditions was near the trailing edge and vice versa while the trailing edge was near to the tongue. Qin and Tsukamoto [6] studied the unsteady flow induced by the interaction between impeller blades and diffuser vanes volute casing in a diffuser pump. The unsteady flow in the diffuser vane passage, as well as the volute casing, was assumed to be induced by the five kinds of singularities—the bound vortices distributed on the impeller blades. They analysed the unsteady pressure downstream of the impeller for a variety of parameters by using the singularity method incorporated the effect of volute casing. The pressure fluctuations due to interaction between impeller and diffuser vanes was studied numerically by Wang and Tsukamoto [7].They found that the potential effect of the wake effect on the pressure fluctuations can be predicted satisfactorily by the present vortex method whereas the pressure fluctuations near the diffuser inlet were affected by the potential interaction, the pressure fluctuations near the outlet were predominantly affected by the wake effects due to wake transport. Shi and Tsukamoto [8] made a numerical study for the prediction of pressure fluctuations caused by impeller–diffuser interaction in a vaned centrifugal pump and they concluded that impeller–diffuser interaction was caused chiefly by potential interaction and wake impingement with the diffuser vanes. Dong et al. [9] used PDV technique to visualize the flow inside the volute of a centrifugal pump.

They verified that the entire flux pulsating within the volute reaches a maximum when the blade lines up with the tip of the tongue. Ozturk et al. [10] investigated the flow behavior in a non- traditional centrifugal pump, whose diffuser was subjected to different radial gaps using multi- purpose fluent code. They found that the Performance curve for centrifugal pumps shows a good approximation with experimental results under fluent solver. The three dimensional turbulent flow was investigated numerically by Silva et al. [11]. They found that an attempt to model an industrial centrifugal pump as a bench-mark test case. Many numerical studies such as Aarts a Jonker [12], Seldar [13], Sano et al. [14] and Sano et al. [15] were performed to predict the velocity distribution and pressure fluctuations through the centrifugal pump.

In the present work, an experimental and computational study was done to investigate the dynamics effects arise due to impeller-volute interaction in a shrouded logarithmic profile high speed centrifugal pump. The test rig was designed and constructed to get the pump performance and the mentioned dynamic effects while the computations were done by using a Fluent 6.2. Commercial code. Two impellers having different outlet diameters were examined for the same volute. Modern high response pressure transducers along with a data acquisition system were used to obtain the pressure fluctuations at multiple points on the centrifugal pump volute. The volume flow rate values were ranged from 0.5QN to 1.5QN. To the knowledge of the authors, there is no many studies has been cited on a high speed centrifugal pump of logarithmic profile. So the present work extends the related research area through documentation of experimental and numerical results to obtain a good understanding of the flow mechanism for this type of centrifugal pumps. The description of physical problem is shown in Fig.1. The objective of the present study is to cover the following:

1. The variation of static, dynamic pressure and unsteady flow effects inside a centrifugal pump due to impeller-volute interaction is to be experimentally and numerically investigated.
2. Constructing an experimental test rig by using modern measurement instrumentations.
3. Using data acquisition techniques to process the pressure fluctuating signals.
4. The Comparison between numerical and experimental results is to be conducted.

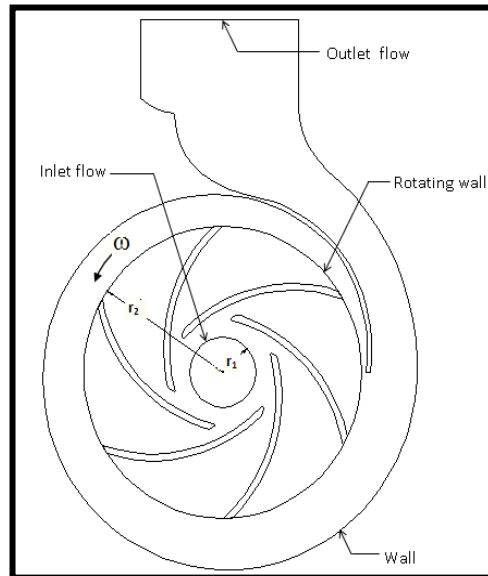


Fig.1 Schematic diagram of the considered problem.

2. Theoretical Work

2.1 Model description.

The physical model considered in this study is shown in Fig.1 which represents a centrifugal pump having a single axial suction and vane-less volute casing, equipped with an impeller of 132mm in outside diameter and six backwards curved blades. The liquid particles enter into the impeller via a cylindrical surface of radius r_1 with an absolute velocity V_1 and exit the impeller via a cylindrical surface of radius r_2 with an absolute velocity of V_2 and inclined at angle α_2 to the peripheral velocity U_2 . The flow patterns at inlet and exit of an impeller rotating with a constant angular velocity which is defined by the velocity triangle as shown in Fig.2. It is assumed that the flow at impeller is irrotational ($\alpha_1 = 90^\circ$).

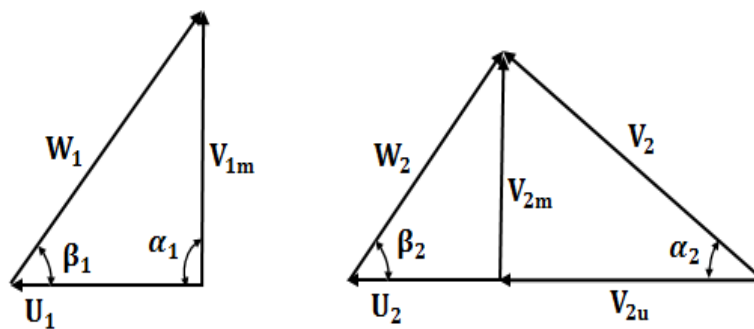


Fig.2. velocity triangles at the impeller inlet and outlet

2.2 Mathematical Model and Numerical simulation.

To simplify the proposed model, the following assumptions are conducted.

- 1- The fluid has constant properties.
- 2- Two dimensional flows.
- 3- Steady state flow.
- 4- Incompressible flow.
- 5- Non-slip flow is assumed.

The governing equations of fluid flow for a steadily rotating frame can be written as follows [14]:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \rho \vec{v}_r = 0 \quad \text{..... (3.1)}$$

Conservation of momentum:

$$\frac{\partial}{\partial t} (\rho \vec{v}_r) + \nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho (2\vec{\omega} \times \vec{v}_r + \vec{\omega} \times \vec{\omega} \times \vec{r}) = -\nabla p + \nabla \bar{\tau}_r + \vec{F} \quad \text{..... (3.2)}$$

$$\bar{\tau}_r = \mu \left[(\nabla \vec{v}_r + \nabla \vec{v}_r^T) - \frac{2}{3} \nabla \cdot \vec{v}_r I \right] \quad \text{..... (3.3)}$$

The effect of turbulence was modeled by using a k-ε model which describes two equations, one for turbulence energy and the other for the dissipation of turbulence energy.

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \epsilon \quad \text{..... (3.4)}$$

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} G_k - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \quad \text{..... (3.5)}$$

The turbulent (or eddy) viscosity, μ_t , is computed by combining k and ϵ as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon} \quad \text{..... (3.6)}$$

The model constant $C_{1\epsilon}$, $C_{2\epsilon}$, C_μ , σ_k and σ_ϵ have the following default values [40]:

$$C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\epsilon = 1.3$$

2.2.1 Boundary Conditions

The physical understanding of turbomachinery flow made assigning the design conditions for the present research. Steady uniform axial velocity is imposed at inlet while a constant static pressure is maintained at outlet. The flow rate was changed by modeling the static pressure at exit to kinetic energy by monitoring closing positions of the valve. The considered boundary conditions for the flow can be mentioned as follows:

For impeller A ($D_2=132$ mm):

- 1- For flow rate ($Q=0.5Q_N$), the static pressure at outlet ($P_{\text{outlet}}=200000$ pa).
- 2- For flow rate ($Q=0.7Q_N$), the static pressure at outlet ($P_{\text{outlet}}=180000$ pa).
- 3- For flow rate ($Q=Q_N$), the static pressure at outlet ($P_{\text{outlet}}=140000$ pa).
- 4- For flow rate ($Q=1.3Q_N$), the static pressure at outlet ($P_{\text{outlet}}=130000$ pa).
- 5- For flow rate ($Q=1.5Q_N$), the static pressure at outlet ($P_{\text{outlet}}=120000$ pa).

For impeller B ($D_2=124$ mm)

- 1- For flow rate ($Q=0.5Q_N$), the static pressure at outlet ($P_{\text{outlet}}=190000$ pa).
- 2- For flow rate ($Q=0.7Q_N$), the static pressure at outlet ($P_{\text{outlet}}=170000$ pa).
- 3- For flow rate ($Q=Q_N$), the static pressure at outlet ($P_{\text{outlet}}=130000$ pa).
- 4- For flow rate ($Q=1.3Q_N$), the static pressure at outlet ($P_{\text{outlet}}=120000$ pa).
- 5- For flow rate ($Q=1.5Q_N$), the static pressure at outlet ($P_{\text{outlet}}=110000$ pa).

Fluent 6.2 commercial codes were used to simulate the discretized governing equations. The discretization of the governing equations was done by using finite volume method with sliding mesh techniques. Creating the required computations mesh for the considered problem was done by using GAMBIT2.3. This facility gives different types of meshes such as hexahedral and tetrahedral and the mesh may be structured or non-structured. To ensure that the computed flow field is not affected by changing the mesh, different numbers of cells were used. It was found that any increase above 60940 will not affect the trend of the predicted mass flow rate at outlet as depicted in Fig.3.

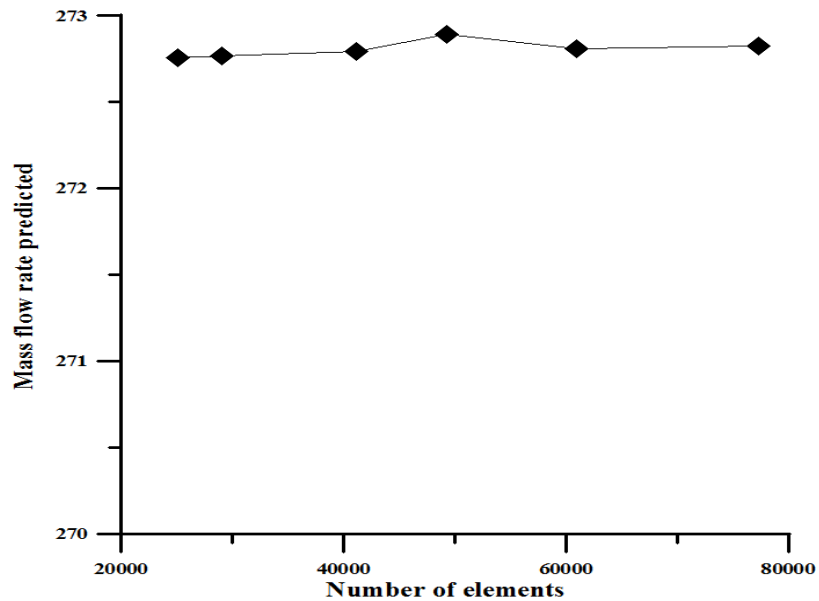


Fig. 3. Mass flow rate versus number of element

The computational mesh created by GAMBIT is shown in Fig.4. The calculations of Fluent were performed for convergence criteria less or equal 1×10^{-5} .

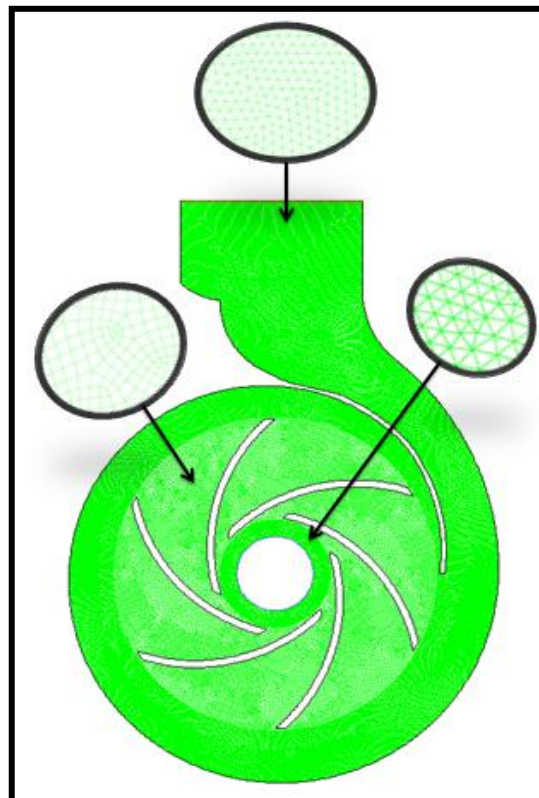


Fig. 4. Mesh used for centrifugal pump with 132 mm impeller

3. Experimental work

A test rig shown in Fig.2 has been designed and constructed to be suitable for the purpose of present and future research. The test pump is a single stage centrifugal pump with shrouded impeller and volute. The test rig consisted of a constant-speed electric pump, a throttle valve, a rotameter flow, and piping. The outlet pipe of the pump is connected to a water flow meter through 45 cm length pipe. The throttle valve (gate valve), fitted on the discharge side of the piping which allows an accurate and fine control of the volume flow rate. The pump is directly driven by a motor single phase 1.5 kW, 220 V AC which has a constant rotational speed of 2900 rpm. Two impeller (A and B) with different outlet diameters were considered. The blades of these impellers were of backward shape having a thickness of 3mm. The number of blades of impeller is six. The details of the impeller technical data are shown in table 1. The measurements for the present study are carried out on the volute of the pump (shrouded side). Four pressure transducers were located (one each 90 deg) on circumference with $R=71\text{mm}$. The location of the tongue of the volute is 270° after the first transducer as. Figure 5 shows the schematic diagram of the experimental test rig.

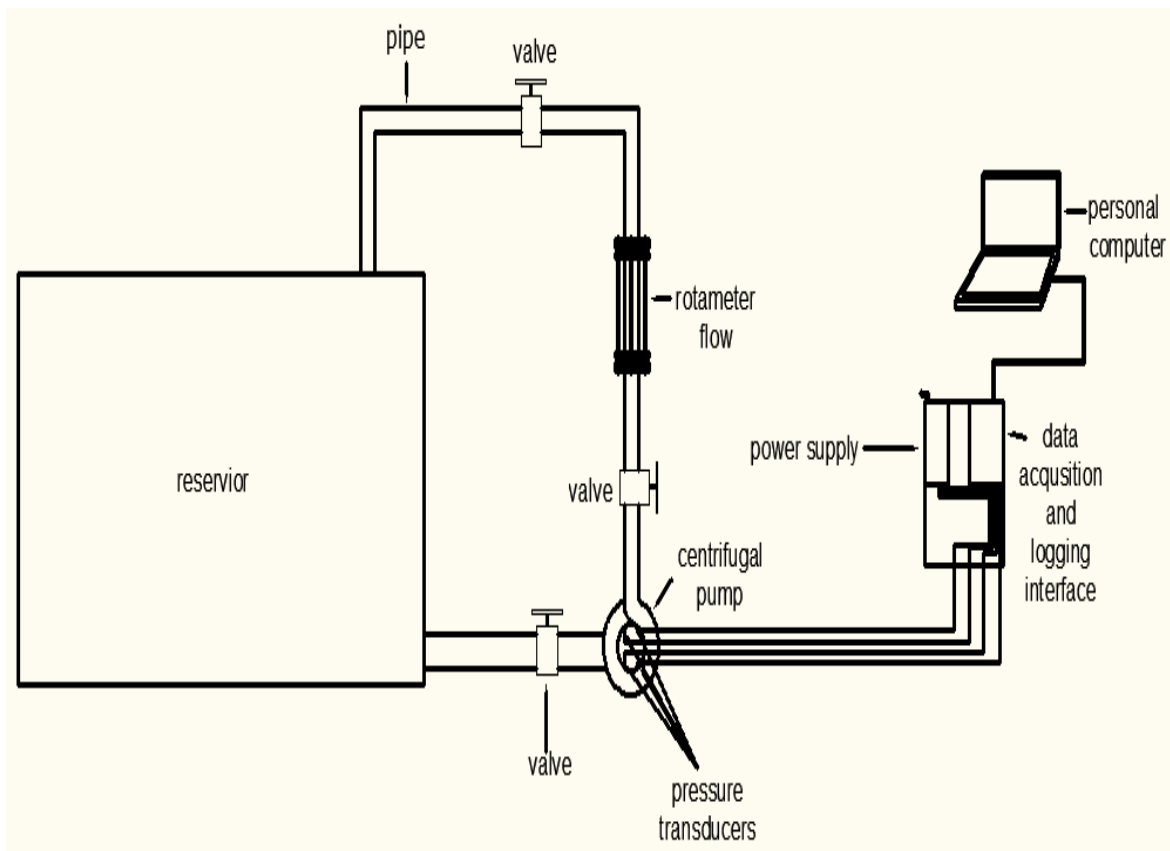


Fig. 5. Schematic diagram of the Experimental test rig

Table1. Pump technical data

Impeller exit diameter	Impeller A, $D_2=132\text{mm}$ Impeller B, $D_2=124\text{mm}$
Impeller inlet diameter	$D_1=48\text{mm}$
Number of impeller blades	$Z=6$
Speed	$N=2900\text{ rpm}$
Inlet blade angle	$\beta_1=15^\circ$
outlet blade angle	$\beta_2=20^\circ$
Blade thickness	$t=3\text{mm}$
Inlet blade width	$b_1=15\text{mm}$
Outlet blade width	$b_2=9\text{mm}$
The gap between the tongue and the impeller	Impeller A= 2mm Impeller B= 6mm

4. Results and Discussion.

Computational and experimental study for predicting the performance and the hydrodynamic parameters of high speed centrifugal pump has been performed. The discussion of the present results has been divided into two sections.

4.1 Analysis of Numerical Results

The obtained numerical results covered the distribution of static pressure, dynamic pressure, velocities, and turbulent kinetic energy. The results of dynamic pressure are not included here because of the limited available space.

Figs.6-7 show the contours of static pressure distribution for impeller A and impeller B at $Q=0.5Q_N$. It is observed that the static pressure has a minimum value at impeller eye and increases as angular position increases. The maximum values of pressure are recorded down stream the leading edge of volute tongue (stagnation point). The same trend of pressure was in the channels of the two impellers (A &B). At exit of the two impellers, there is obvious increase in the values of pressure distribution. As the figure show, when the impeller diameter increases, the values of pressure decreases.

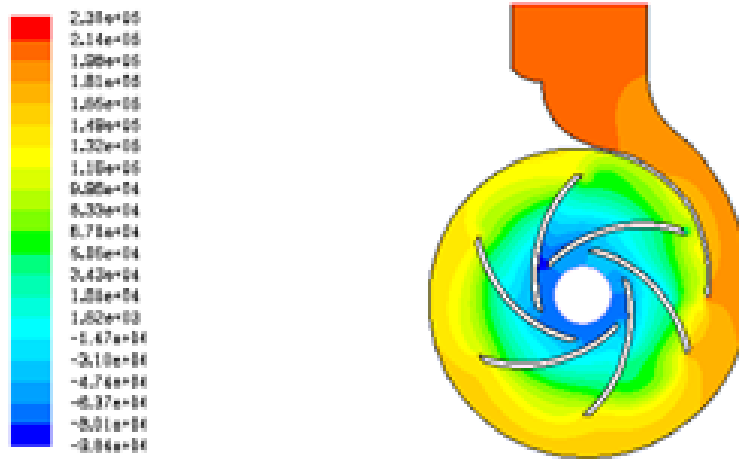


Fig.6 Contours of static pressure distribution (Pascal) for impeller A at $Q=0.5Q_N$.

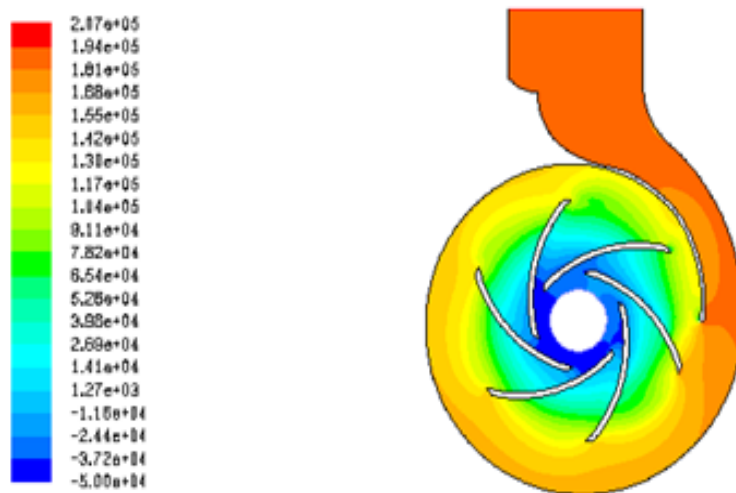


Fig.7 Contours of static pressure distribution (Pascal) for impeller B at $Q=0.5Q_N$.

The design operating condition ($Q=Q_N$) is demonstrated through Figs.8-9. For the two impellers (A&B). It can be observed that the pressure values increase gradually along the blade passage. It is expected that the pressure trend in the impeller is equal that in the volute. The pressure trend seems to be uniform for this design condition ($Q=Q_N$) but this behavior is changed as flow rates increase or decrease. The pressure trend is changed significantly when the flow rate increases to $Q= 1.5 Q_N$ as shown in Figs10-11. The large or minimum values of pressure distribution are shifted about the previous positions. A high pressure values were recorded in the tongue zone and these are larger in impeller A compared with impeller B.

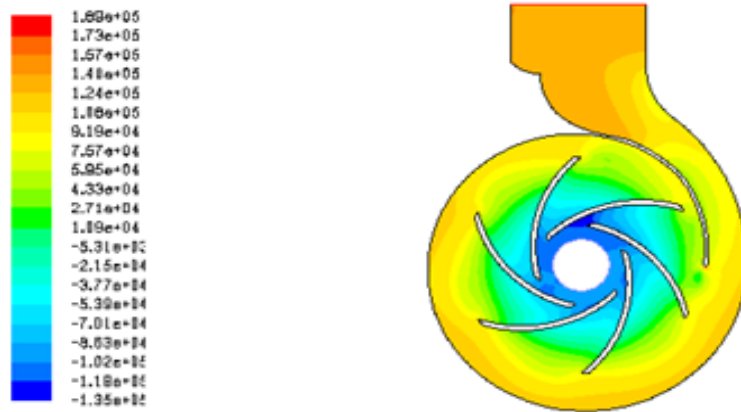


Fig.8 Contours of static pressure distribution (Pascal) for impeller A at $Q=Q_N$.

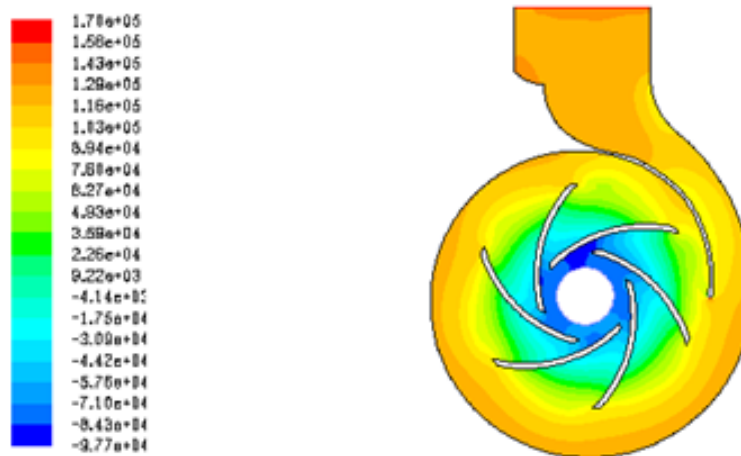


Fig. 9 Contours of static pressure distribution (Pascal) for impeller B at $Q=Q_N$.

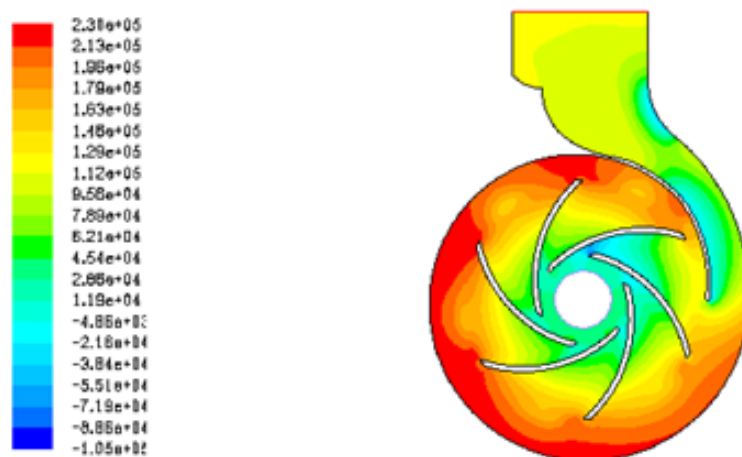


Fig.10 Contours of static pressure distribution (Pascal) for impeller A at $Q=1.5Q_N$.

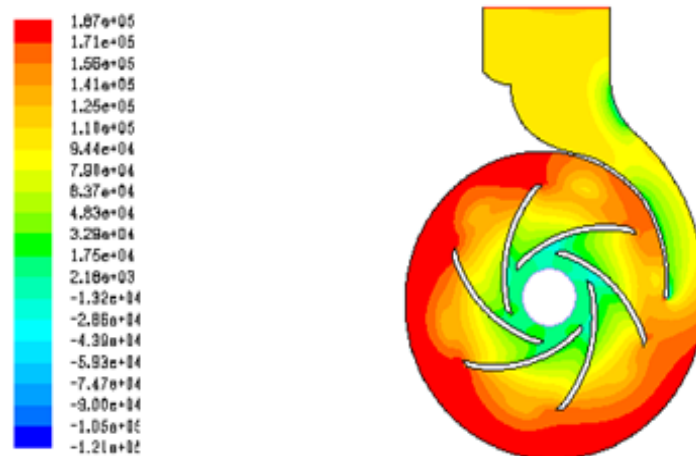


Fig.11 Contours of static pressure distribution (Pascal) for impeller B at $Q=1.5Q_N$.

The contours of absolute velocity for the considered two impellers and for different flow rates are shown in Figs. 12-13. It is observed that there is a recirculation region for all the studied volume flow rates. However the position of these regions is changed as volume flow rate changes. At $Q=0.5Q_N$, low values of velocity distribution is observed at discharge pipe and this increases as impeller diameter increases.

At $Q=Q_N$, the change in velocity becomes less and the velocity increases in the discharge pipe and the distribution of the recirculation regions and the maximum and minimum values of velocity seems to be matching in the impeller and volute. However this trend is reflected at high volume flow rates ($Q= 1.5 Q_N$) where the large amount of the flow is pushed towards the outlet duct. As a result, the stagnation point on the tongue is moved to impeller side and there is a wake on exit of the tongue.

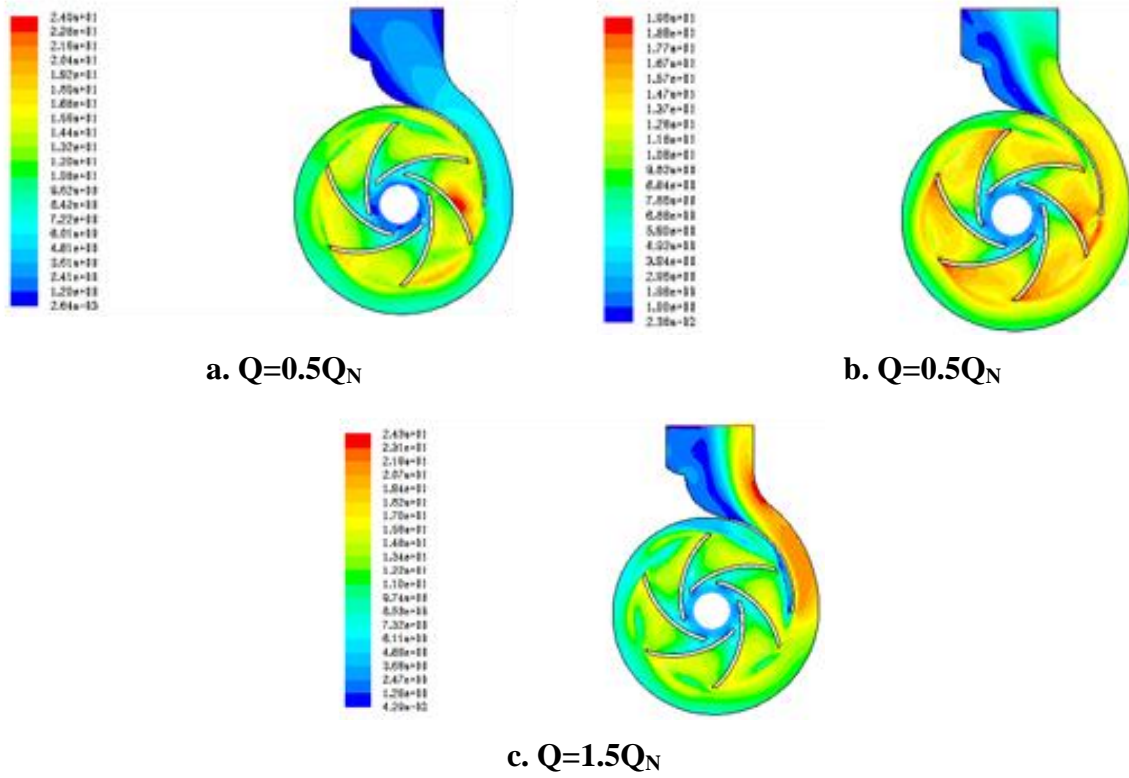


Fig.12 Contours of absolute velocity distribution (m/s) for impeller A.

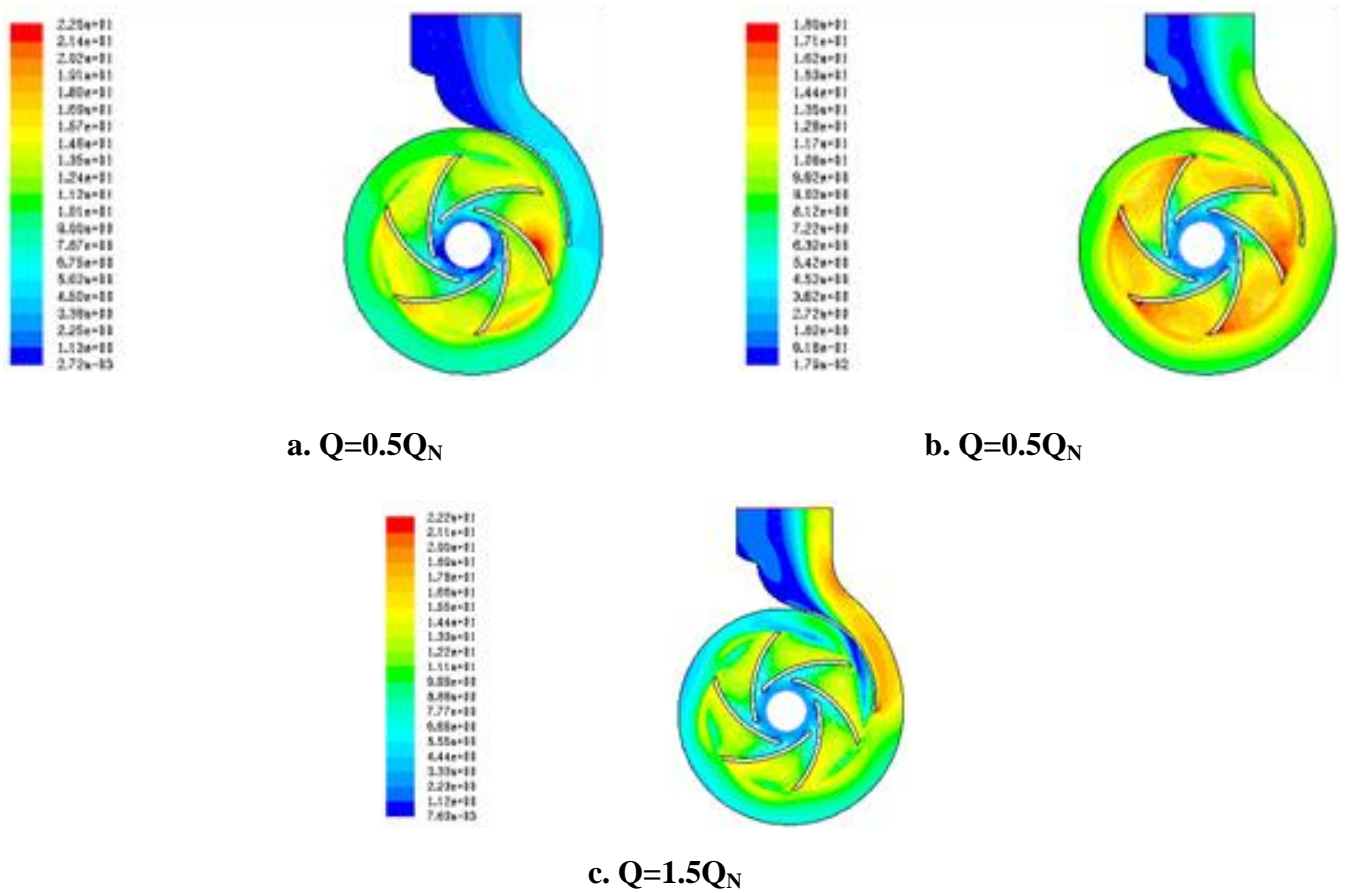


Fig.13 Contours of absolute velocity distribution (m/s) for impeller B.

Figs.14-15 shows the distribution of turbulent kinetic energy for the two considered impellers (A&B) and for different values of volume flow rates. It can be observed that the maximum values of turbulent kinetic energy are found at trailing edge of impeller blades for the mentioned cases. The values of turbulent kinetic energy are decreased as impeller diameter decreases. At nominal volume flow rates, the maximum values of this energy at trailing edge become little while it increases as the volume flow rate increase.

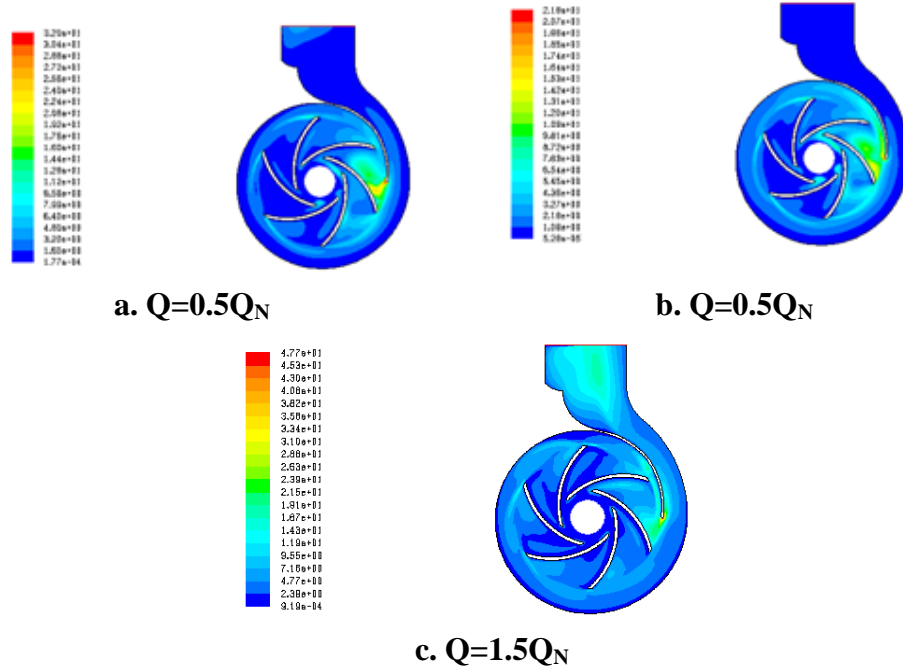


Fig.14 Contours of turbulent kinetic energy distribution (m/s) for impeller A.

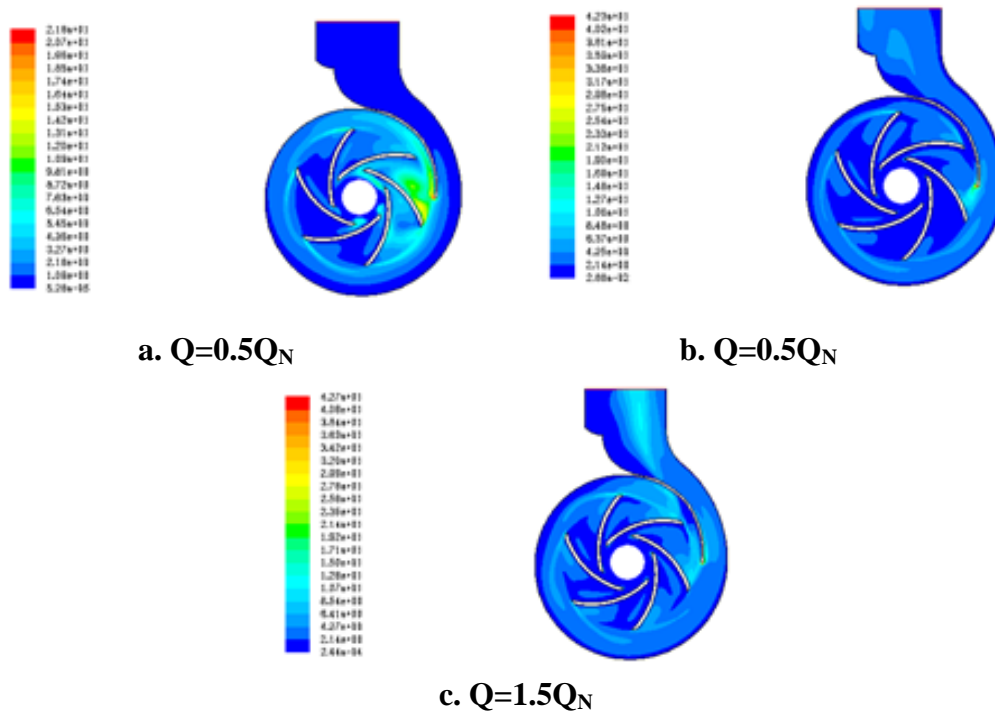


Fig.15 Contours of turbulent kinetic energy distribution (m/s) for impeller B.

4.2 Analysis of experimental results

Figures 16-17 show the distribution of the static pressure fluctuation around the impeller for two different values of diameters (impeller A & impeller B) at volume flow rates $Q=0.5Q_N$ & $Q=0.7Q_N$. It can be seen that the pressure fluctuations have a maximum value at position $\varphi=270^\circ$ and they have small values at other positions. This phenomenon occurs because of the switching in the flow trace between the recirculation duct and the discharge duct. It can be indicated that the pressure fluctuations have maximum values at volume flow rate ($Q=0.5Q_N$) for the two impellers (A & B). As the figures show, for a given flow rate, the maximum amplitudes of the pressure fluctuations increase as the impeller diameter increases i.e when reducing the blade to tongue gap. This is an expectable result since a smaller rotor-stator distance implies less space for the flow to adapt to the geometry changes, i.e., greater velocity gradients, greater stresses and, in summary, a greater fluid-dynamic interaction.

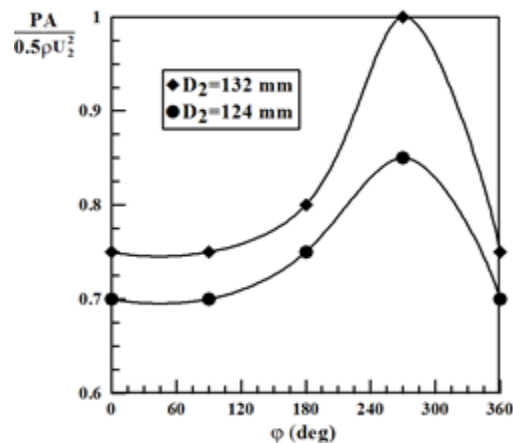


Fig. 16 Distribution of the pressure fluctuation around the angular position of the impeller at $Q=0.5Q_N$.

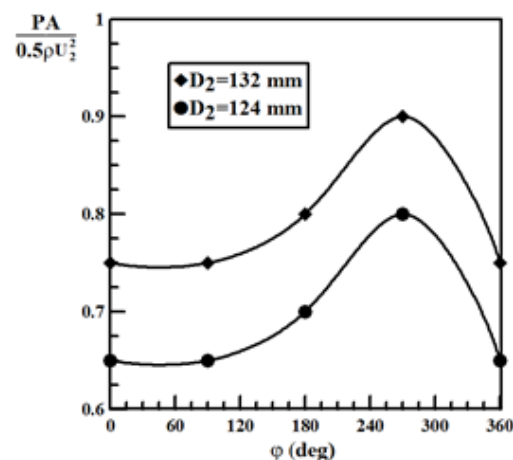


Fig. 17 Distribution of the pressure fluctuation around the angular position of the impeller at $Q=0.7Q_N$.

The variation of normalized static pressure distribution around the angular position of the two considered impellers (A & B) for nominal flow rate is shown in figure 18. It can be seen that the pressure fluctuations are not high in both impeller diameters at the tongue of the volute and there is a difference in the values of these fluctuations between impeller A and B. It can be noted that the pressure fluctuations have a maximum value at position $\varphi=100^\circ$ for impeller A and $\varphi=185^\circ$ for impeller B. Also the impeller A indicated the larger expected pressure fluctuations.

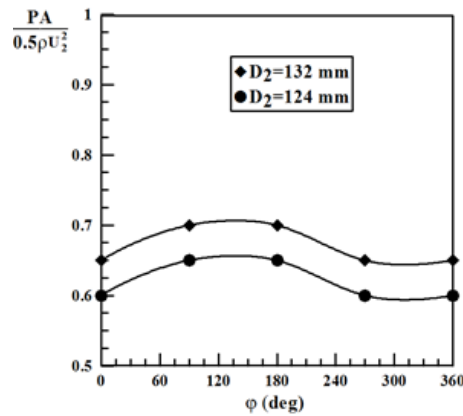


Fig. 18 Distribution of the pressure fluctuation around the angular position of the impeller at $Q=Q_N$.

Figures 19-20 show the distribution of the normalized pressure fluctuation around the angular position of the two considered impellers (A & B) for volume flow rate greater than nominal one. It can be noted that the trend of the pressure distribution is changed compared with the case of $Q < Q_N$. It is evident that the pressure fluctuation values are low and have the same value for the both impellers (A & B) at volute tongue. That means that the flow is accumulating at the tongue and because this accumulation, the pressure decrease at volute tongue. However, impeller A indicated the larger pressure distribution as expected.

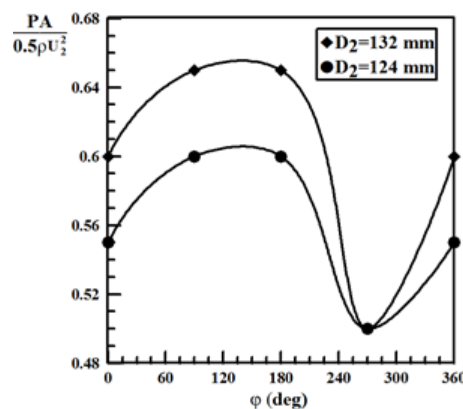


Fig. 19 Distribution of the pressure fluctuation around the angular position of the impeller A at $Q=1.3Q_N$.

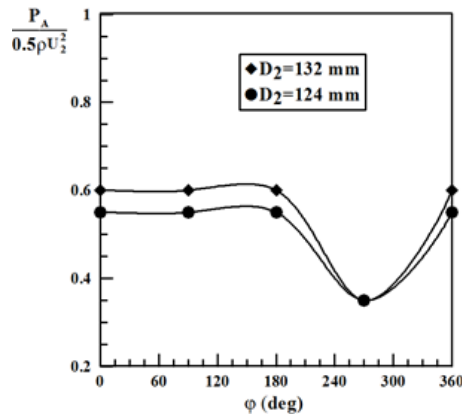


Fig.20 Distribution of the pressure fluctuation around the angular position of the impeller B at $Q=1.5Q_N$.

4.1 Validation

The results of experimental work is compared with the numerical results that obtained by using a commercial CFD Fluent 6.3 code as shown in figure 21-22. The numerical simulation were performed for the same geometry and locations considered in the experiments. It can be observed that an acceptable agreement is achieved between experimental and numerical result. However some discrepancies have arisen in the comparison between the numerical and experimental pressure fluctuation in the volute of the tested centrifugal pump, especially near tongue region. A part of these differences can be attributed to the leakage flow between the volute and the impeller inlet, which was not considered in the numerical model.

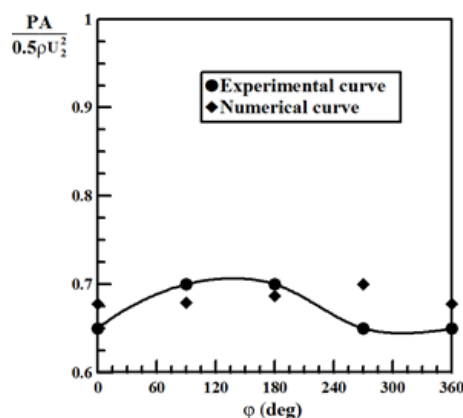


Fig. 21 Comparison between the present experimental and numerical results for impeller A at $Q=Q_N$.

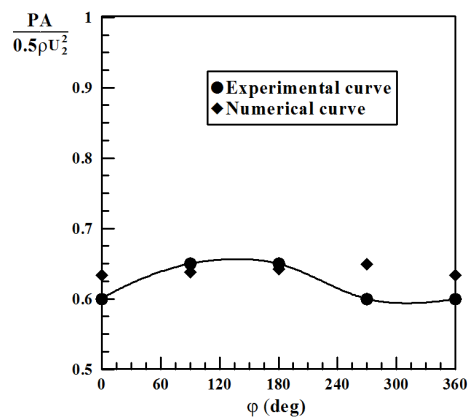


Fig.22 Comparison between the present experimental and numerical results for impeller B at $Q=Q_N$.

5. Conclusions

The following conclusions can be drawn from this study. The pressure fluctuations registered in the volute were found to be very dependent on both angular position and volume flow-rate, with maximum values corresponding to the tongue region for ($Q=0.5Q_N$) at angular position ($\phi=270^\circ$). Changing the impeller diameter from 124mm to 132mm results in an increase in maximum pressure fluctuation by a value approximated to 15%. The static pressure increases as the angular position (around the volute) increases and the higher pressure values can be observed upstream of the leading edge of the volute tongue (stagnation point). The static pressure has a minimum value at the impeller eye and around the impeller especially at angular position about ($\phi=185^\circ$) for impeller A and ($\phi=100^\circ$) for impeller B. Decreasing radial gap has a significant effect on the distribution of turbulent kinetic energy, where the level of the turbulence intensity is increased as the impeller diameter increases

Nomenclature

b_1	Inlet blade width	mm
b_2	Outlet blade width	mm
D_1	Impeller inlet diameter	mm
D_2	Impeller exit diameter	mm
g	Gravity acceleration	m/s^2
H_t	Theoretical head	m
N	Speed	rpm
P	Static pressure	N/m^2
P_A	Pressure Amplitude	N/m^2
P_{in}	Input power	Watt
Q	Volumetric flow rate	m^3/s
Q_N	Volumetric flow rate at nominal point	m^3/s
R	Radial position	mm
t	Blade thickness	mm
u	Velocity in x -direction	m/s
U	Tangential velocity of impeller	m/s
U_2	Peripheral velocity at impeller outlet	m/s

v	Velocity in y-direction	m/s
V	Absolute velocity	m/s
V_a	Axial velocity	m/s
V_m	Meridional velocity	m/s
V_r	Radial velocity	m/s
W	Relative velocity	m/s
x	Horizontal coordinate	m
y	Vertical coordinate	m
Z	Number of impeller blades	-
∇	Gradient	-

References

- [1] van Esch, B. P. M., Performance and Radial Loading of a Mixed-Flow Pump Under Non-Uniform Suction Flow, *ASME J. Fluid Eng.*, 131(2009), pp. 1-7.
- [2] Chu, S., Dong, R., and Katz, J., Relationship Between Unsteady Flow, Pressure Fluctuations, and Noise in a Centrifugal Pump; Part B: Effects of Blade-Tongue Interaction, *ASME J. Fluids Eng.*, 117(1995), pp. 30–35.
- [3] Elholm, T., Ayder, E., Van den Braembussche, R., Experimental Study of the Swirling Flow in the Volute of a Centrifugal Pump, *ASME J. Fluid Eng.*, 114(1992), pp. 366-372.
- [4] Timushev, S.E , and Ovsyannikov.,B.V., Pressure Fluctuation Numerical Simulation in a Centrifugal Pump Volute Casing, Colloque C1, supplement au Journal de Physique III, 2(1992), pp.619-622.
- [5] Kaps,A., Numerical Flow Analysis in a Centrifugal Pump for Optimization of the Casing with Regard to Hydraulic Efficiency and Bearing Forces, *Phenomena and Dynamic of Rotating Machinery*, 2(1996), pp.345-354.
- [6] Qin,W., and Tsukamoto,H., Theoretical Study of Pressure Fluctuations Downstream of a Diffuser Pump Impeller – Part 2: Effects of Volute , Flow Rate and Radial Gap, *ASME J. Fluid Eng.*, 119(1997),pp.
- [7] Wang, H., and Tsukamoto, H., Fundamental Analysis on Rotor-Stator Interaction in a Diffuser Pump by Vortex Method, *ASME Journal of Fluids Engineering*, 122(2001), pp. 737-747.
- [8] Shi, F., and Tsukamoto, H., Numerical Study of Pressure Fluctuations Caused by Impeller-Diffuser Interaction in a Diffuser Pump Stage, *ASME Journal of Fluids Engineering*, 123(2001), pp. 466-477.
- [9] Dong, R., Chu, S., and Katz, J., Quantitative visualization of the flow within the volute of a centrifugal pump. Part B: results and analysis. *Trans. of the ASME*, 114(1992), pp. 396-403.
- [10] Adnan Ozturk, Kadir Aydin, Besir Sahin and Ali Pinarbasi1, Effect of impeller-diffuser radial gap ratio in a centrifugal pump *Journal of Scientific & Industrial Research*, 68(2009), pp.203-213.
- [11] Cheah, K.W. , Lee, T. S., and Winoto, S. H., and Zhao, Z.M. , Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions, *International Journal of Rotating Machinery*, Article ID 83641, 2007, 8 pages.
- [12] Fabio J. Silva, Jose C. Pascoa , Joao S. Pinheiro , and Daniel J. Martins , Turbulent Flow structure Computation Inside a Pump-Pat Using an Industrial Benchmark Test Case, *European Conference Computational Fluid Dynamics*, Lisbon, Portugal, 2010.
- [13] Raul Barrio, Jorge Parrondo, and Eduardo Blanco, Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points” *International Journal Computers and Fluids*, 39(2010) pp. 895-870.
- [14] Houlin, LIU.,Yong, WANG., Shouqi, YUAN., Minggao, TAN., and Kai, WANG., “Effects of Blade Number on Characteristics of Centrifugal Pumps” *Chinese Journal Of Mechanical Engineering* , 23(2010).