AL-OBAIDI, A., PRADHAN, S., ASIM, T., MISHRA, R. and ZALA, K. 2014. Numerical studies of the velocity distribution within the volute of a centrifugal pump. Presented at *27th Condition monitoring and diagnostic engineering management international congress 2014 (COMADEM 2014), 16-18 September 2014, Brisbane, Australia*.

# Numerical studies of the velocity distribution within the volute of a centrifugal pump.

AL-OBAIDI, A., PRADHAN, S., ASIM, T., MISHRA, R. and ZALA, K.

2014

COMADEM 2014

16 – 18 September 2014 | Brisbane Convention and Exhibition Centre, Australia

27th International Congress of Condition Monitoring and Diagnostic Engineering

# Numerical studies of the velocity distribution within the volute of a centrifugal pump

Ahmed Al-Obaidi[a]*, Suman Pradhan[b], Taimoor Asim[b], Rakesh Mishra[b] and Karina Zala[b]

[a]*Al-Mustansiriya University, Baghdad, Iraq*
[b]*University of Huddersfield, Queensgate, Huddersfield, HD1 3DH UK*

ABSTRACT

Centrifugal pumps play an essential role in engineering systems since they are widely used in the process and power industries. The performance of a centrifugal pump needs to be maximised due to its importance and this depends on the flow structure within the pump. The flow structure within a pump is very complex due to the presence of a rotating impeller and its interaction with the volute casing. In this paper, a numerical investigation using CFD analysis has been carried out to determine the effect of volute geometry on the flow field within a centrifugal pump. The results obtained from the numerical investigation have been validated with the experimental data. Further analyses have been carried out to investigate the effect of volute cross-sectional area on the velocity distribution. The overall results indicate that the head increases as the volute cross-sectional area increases.

*Keywords:* *Centrifugal pump; CFD ; Volute cross-sectional area*

* Tel: +44 01484 471138; email:u1255330@hud.ac.uk

1. **Introduction**

Pumps are used to transfer liquids by increasing the energy of fluid within in a system. Currently, centrifugal pumps are widely used in industrial and residential applications, such as in irrigation, sewage, oil refineries, chemical plants, stream power plants, food-processing factories, hydraulic power service and mining industries [1]. A centrifugal pump consists of two main components, an impeller and a volute casing. The impeller is a rotating part which supplies mechanical energy to the fluid, whereas the volute is a stationary part. In a centrifugal pump, water enters through the eye of impeller then exits radially from the volute outlet. Since it is not practical to pursue extensive experimentation, computational fluid dynamics (CFD) is used as it is an effective tool to investigate performance characteristics of the pumps, enabling the geometry related effects in the pump to be analysed.

Many researchers have contributed significantly towards revealing the flow mechanisms of the high-performance centrifugal pump [1-7] and suggested ways through which improvements in performance can be achieved.

However, some of these studies are focused on the implications of impeller design on to the flow characteristics in the pump [2-4]. Furthermore, some [3-5] researchers have focused on the impact of volute casing design on the flow charcateristics, as the volute casing is another important part of the pump that plays a vital role in its overall performance.

Kelder et al. [3] investigated theoretically and experimentally the flow characteristics in the volute for a low specific speed centrifugal pump near the design point. The results have shown that near the design point of this pump, the core flow behaves like a potential flow, provided that no boundary layer separation occurs. Chan et al. [4] studied design considerations of volute geometry for centrifugal blood pump. The study was carried out using two methods, namely, the constant angular momentum (CAM) and constant mean velocity (CMV). The study has shown that the pressure distributions along the volute circumference is non-uniform, and pressure values are higher for the volute designed with CAM method as compared to the volute designed with CMV method.

Yang et al. [5] investigated the design of the centrifugal pump volute by using computational fluid dynamics (CFD). The results have shown that optimal radial gap for the studied parameter is 15 mm.

In the present study, CFD tools have been used to carry out extensive research on the effects of various geometric parameters, such as the cross-sectional area of the volute, on the performance of the centrifugal pumps. This aims to develop a simple design equation correlating various parameters. The effects of these parameters have been explicitly analysed in the current design process of the centrifugal pump making this study important for the development of design methodology.

## 2. Numerical Modelling

The geometry shown in figure 1 is a three-dimensional radial centrifugal pump. The modelled geometry consists of a backward type impeller with 5 backward curved impeller blades, and a volute casing. The dimensions of the pump are shown in table1, where the inlet diameter is 50mm, impeller diameter is 217. 5mm, rotational speed is 2900rpm, number of blades and their thickness is 5 and 4mm respectively. Above specifications are chosen from the commercially available pump F32/200HA from Pedrollo Company in order to compare the results with the published experimental data.

**Table 1.** Specifications of centrifugal pump impeller

| | |
|---|---|
| Inlet impeller diameter d1 | 50mm |
| Outlet impeller diameter d2 | 217. 5mm |
| Speed of the impeller N | 2900 rpm |
| No. of impeller blades | 5 |
| Blade thickness | 4mm |
| Type of impeller | Backward type |

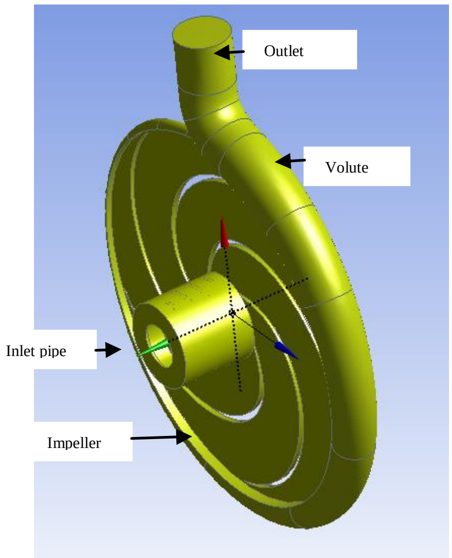



Figure 1 Geometry of the centrifugal pump

Three different volute flow areas have been used to analyse their effect on the flow characteristics. These geometries have been modified where the inner wall of the volute is maintained at a constant distance of 108.75mm from the centre of impeller whilst the outer wall has been offset inward and outward by 5% from the original cross-sectional area. The changes made at 5 different sections of volute are shown in figure 2.

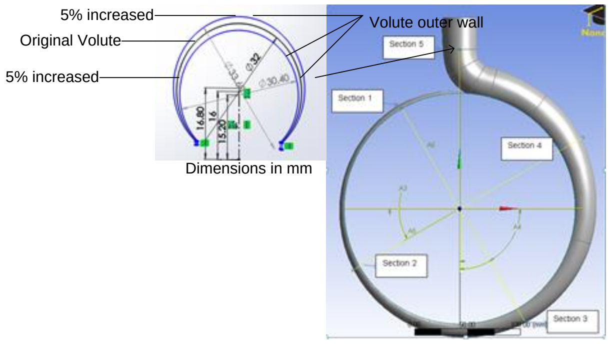



Figure 2 the location of cross-section within the volute

### 2.1. *Meshing the flow domain*

The model consists of three boundary regions inlet pipe, impeller and volute, as shown in figure 3. Tetrahedral mesh has been selected for the meshing flow domain. The element size of 1.2mm has been used in this study, resulting in the total number of nodes and elements of 475372 and 2008521 respectively.

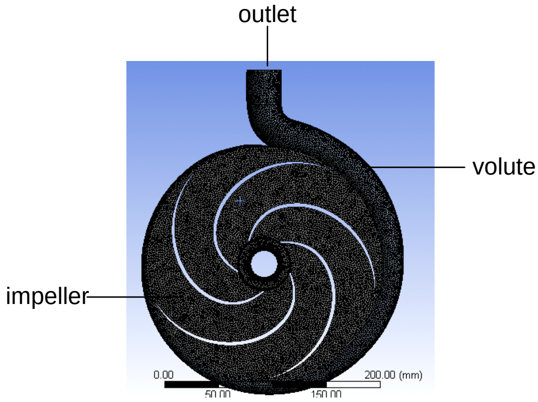



Figure 3 Geometry and the mesh of a five blades Pump impeller

For the numerical analysis, commercial ANSYS CFD – fluent code has been used. A Multiple Reference Frame (MRF) model with a rotational speed of 2900 rpm in the anti-clockwise direction has been used in present investigation. The working fluid through the pump is water. The Shear Stress Turbulence (SST) model is implemented, which can accurately analyses wall shear and provide accurate prediction of the onset and the size of separation zone [7]. Turbulence intensity of 5% has been considered.

## 3. Results

A numerical analysis has been carried out on the centrifugal pump with the cross-sectional area of volute similar to the pump specification F32/200HA of Pedrollo Company. Figure 4 and table 2 depict the comparison of the results between numerical analysis and the experimental data. From table 1, it can be clearly seen that the error between the numerical analysis and experimental data is around 4%, or less with standard 32 mm outlet diameter of volute. This indicates that the selection of the number of nodes, elements and boundary condition parameters chosen for this study is justifiable.

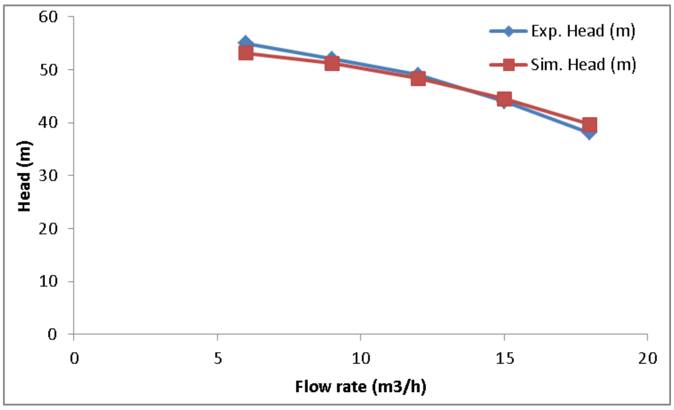



Figure 4 comparison performance curves between simulation and experimental analysis.

**Table 2.** Comparison of total head between experimental and simulation analysis

| Flow rate ($m^3/h$) | 6 | 9 | 12 | 15 | 18 |
|---|---|---|---|---|---|
| Experimental data (m) | 55 | 52 | 49 | 44 | 38 |
| Simulation (m) | 53.13 | 51.16 | 48.35 | 44.49 | 39.67 |
| Error (%) | 3.4 | 1.61 | 1.32 | 1.11 | 4.39 |

*3.1. Pressure Distribution for different flow rates*

Figure 5 depicts the variation in static pressure within the centrifugal pump. It can be seen that the flow distribution inside the centrifugal pump is asymmetric for the given flow condition. The static pressure increases gradually from the inlet of impeller to the outlet due to the dynamic head generated by the rotation of the pump impeller. The maximum static pressure can be noticed at volute tongue and outlet areas. The minimum static pressure can be observed at the leading edge of the blades on the suction side where the total pressure patterns are variable along the impeller.

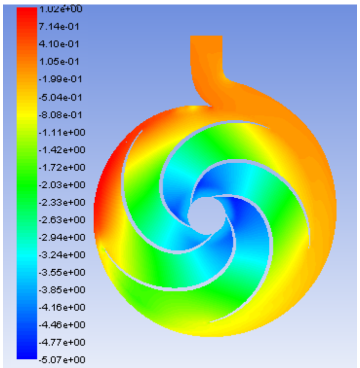
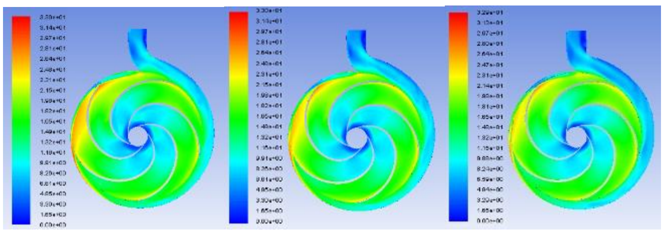


Figure 5 Pressure distributions for mass flow $15m^3/h$

*3.2. Velocity distribution for different flow rates*

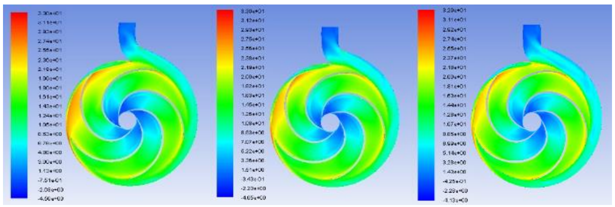
Figures 6 depict the distribution of velocity magnitude in centrifugal pump with different volute flow areas at flow rate of $15m^3/h$. It can be seen that the velocity increases from the inlet of the impeller to the outlet of the impeller. The velocity seems to decrease as it enters the volute, and is at its lowest velocity at the outlet of volute. Additionally, it can be seen that the velocity magnitude in the volute tends to decrease as the cross sectional area increases.



-5% origin +5%

Figure 6 the Velocity magnitude distribution for different cross-section area and mass flow $15m^3/h$

Figure 7 depicts tangential velocity in a pump for different volute cross-section area at $15m^3/h$ mass flow. It can be seen that the tangential velocity increases as flow enters the volute, decreases as the flow follows the path of volute and is higher for the small cross-sectional area of volute.



-5% origin +5%

Figure 7 Tangential velocities in a pump for different volute cross-section area at 15m$^3$/h mass flow

The tangential velocity at each section of the volute has been represented in figure 8. Figure 8 depicts that tangential velocity decreases as the cross-sectional area increases. Furthermore, it also indicates that the tangential velocity increase above 20m/s at section 2, which assumed to be due to the sudden change in the volute opening.

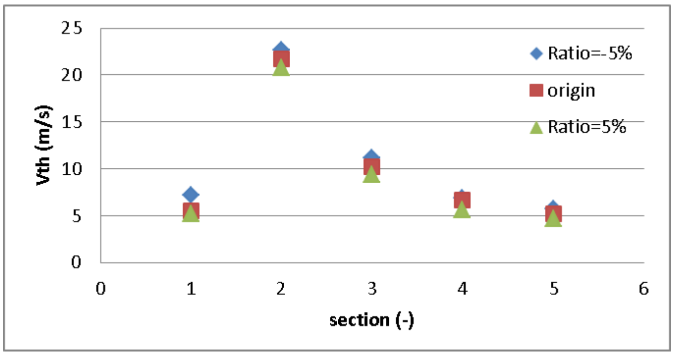
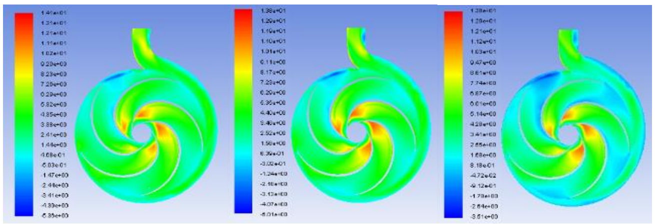



Figure 8 Tangential velocities at each section

Figure 9 depicts radial velocity in a pump for different volute cross-section area at 15m$^3$/h mass flow. This indicates that radial velocity is higher in the inlet of the impeller. Additionally, it shows that radial velocity is lower for the smaller cross-sectional area of the volute.



-5% origin +5%

Figure 9 Radial velocity distributions for different cross-section area and mass flow 15m$^3$/h

Figure 10 represents the radial velocity at each cross section of the volute. The results indicate that the radial velocity is comparatively lower than the tangential velocity. Furthermore, the results also indicate that the radial velocities at section 2 and 5 are relatively higher than the other sections in each case.

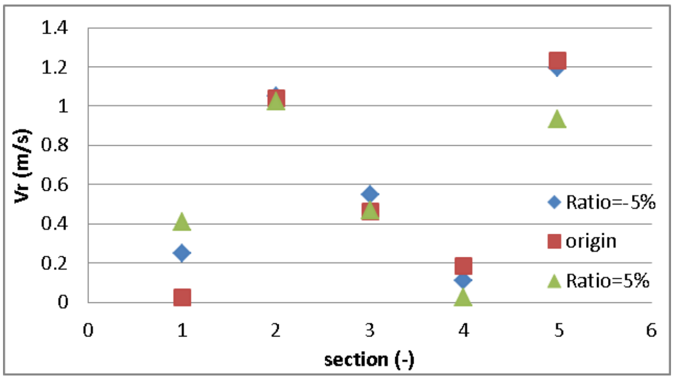
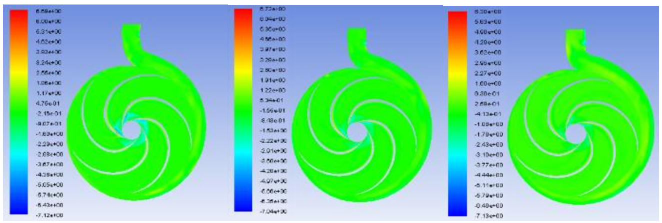



Figure 10 Radial velocities at each section

Figure 11 depicts axial velocity in a pump for different volute cross-section area at 15m$^3$/h flow rate. It can be seen that axial velocity in each case is almost same and much lower compared to the tangential velocity.



-5% origin +5%

Figure 11 axial Velocity distributions for different cross-section area and mass flow 15m$^3$/h

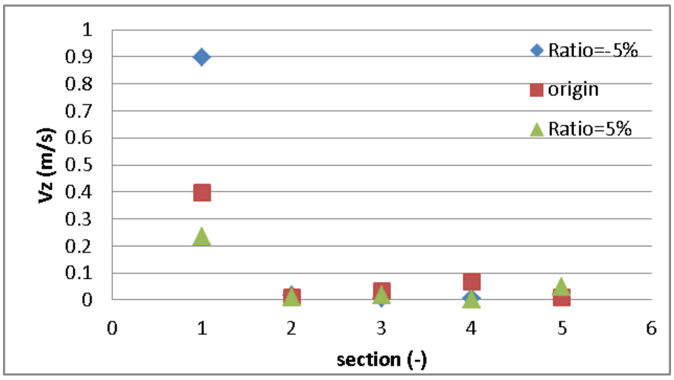



Figure 12 Axial velocities at each section

Figure 12 depicts axial velocity at each cross section of the volute. It can be seen that the axial velocity is much lower compared to the tangential and radial velocities. Additionally, the axial velocity in section 1 is higher than the other sections. Furthermore, the axial velocity is higher for the smaller cross sectional area of the volute in section 1.

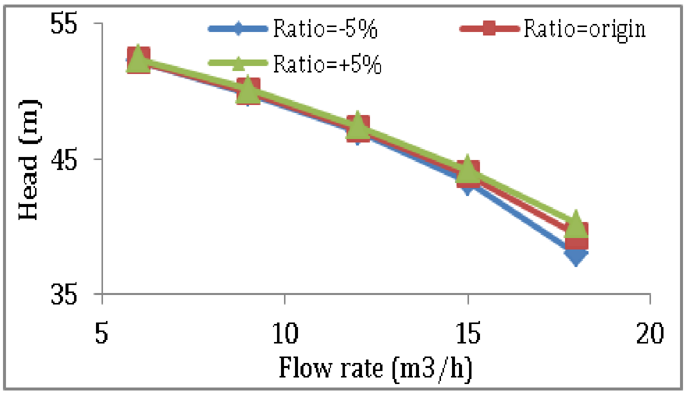



Figure 13 Total head at various flow rates with different cross section

Figure 13 and table 3 depict the variation in total head with various flow rates and different volute flow areas. The results show that total head decreases as the flow rate increases. Furthermore, the results also indicate that the total head increases with the volute flow area. However, increment in total head is minimal at lower flow rates compared to the higher flow rates.

**Table 3.** Comparison total head between various cross-sectional areas of volute

| Ratio | S/N | Flow rate ($m^3/s$) | Head (m) |
|---|---|---|---|
| -5% | 1 | 1.67E-03 | 52.30 |
| | 2 | 2.50E-03 | 49.87 |
| | 3 | 3.33E-03 | 47.09 |
| | 4 | 4.17E-03 | 43.35 |
| | 5 | 5.00E-03 | 38.04 |
| origin | 1 | 1.67E-03 | 52.28 |
| | 2 | 2.50E-03 | 50.00 |
| | 3 | 3.33E-03 | 47.26 |
| | 4 | 4.17E-03 | 43.87 |
| | 5 | 5.00E-03 | 39.37 |
| 5% | 1 | 1.67E-03 | 52.39 |
| | 2 | 2.50E-03 | 50.18 |
| | 3 | 3.33E-03 | 47.51 |
| | 4 | 4.17E-03 | 44.23 |
| | 5 | 5.00E-03 | 40.23 |

## 5. Conclusions

In this study, the effect of various volute flow areas on the centrifugal pump performance output has been numerically examined and analysed. Following are the main conclusions from this study.

- The pressure and the velocity fields in the centrifugal pump are highly unsymmetrical and non-uniform.
- The tangential velocity tends to decrease as the volute cross-sectional area increases.
- Radial velocity in the volute shows non-uniform behaviour in each section of the volute in question.
- Axial velocity in the volute is always minimal.
- Increase in volute cross-sectional area increases the performance of the centrifugal pump.

## 6. References


1. Thin, K. C., Khaing, M. M., & Aye, K. M. (2008). Design and Performance Analysis of Centrifugal Pump. World academy of science, engineering and technology, 46, 422-429.Goering, B. K. Ph.D. Dissertation, Cornell University, 1995.
2. Sidheswaare R. and Hebbal O. D. (2013).Validation Of Hydraulic Design Of A Metallic Volute Centrifugal Pump. International Journal of Engineering Research & Technology (IJERT) ISSN: 2278-0181 Vol. 2 Issue 7.Buchanan, J. G.; Sable, H. Z. In *Selective Organic Transformations*; Thyagarajan, B. S., Ed.; Wiley-Interscience: New York, 1972; Vol. 2, pp 1–95.
3. Kelder, J. D. H., Dijkers, R. J. H., Van Esch, B. P. M., & Kruyt, N. P. (2001). Experimental and theoretical study of the flow in the volute of a low specific-speed pump. Fluid Dynamics Research, 28(4), 267-280.
4. Chan, W. K., Wong, Y. W., & Hu, W. (2005). Design considerations of volute geometry of a centrifugal blood pump. Artificial organs, 29(12), 937-948.
5. Yang, S., Kong, F., & Chen, B. (2011). Research on pump volute design method using CFD. International Journal of Rotating Machinery, 2011.
6. Jin H. B., Kim M. J., Chung W. J.(2012). A study on the effect of variation of the cross-sectional area of spiral volute casing for centrifugal pump, World Academy of Science, Engineering and Technology 68.
7. ANSYS Release 13.0 (2010). Customer Training Material Lecture 6 Turbulence Modelling Introductions to ANSYS FLUENT.