ANALYSIS OF FLUID FLOW IN RADIAL CENTRIFUGAL PUMP

Jan ČERNÝ[1], Martin POLÁK[1]

[1]*Department of Mechanical Engineering, Faculty of Engineering, Czech University of Life Sciences Prague, Prague, Czech Republic*

Abstract

The paper presents validation of results of a numerical model of radial centrifugal pump flow using an experimental method PIV (Particle Image Velocimetry). For this purpose, a 3D model of the pump was created in Inventor, which was then used to design a numerical flow model in Ansys in the CFX module. The performance characteristics of the same pump were measured on an experimental test circuit, and vector maps of the flow in the suction pipe were obtained using the PIV method. The results of the experiment – vector fields of fluid velocity distribution in a suction pipe – were then compared with the outputs of the numerical Ansys model, namely the flow curves and pressure distribution. This comparison demonstrated that the numerical model achieves the best agreement with reality if the input variables are the pressure in front of the pump and the mass flow behind the pump. In this case, the model can calculate the pressure at the pump inlet with a deviation of 1% to 10% and create streamlines in the suction pipe corresponding to the results of PIV measurements.

Key words: *Particle image velocimetry (PIV), ANSYS, centrifugal pump, performance characteristic.*

INTRODUCTION

The more modern technologies evolve, the more the design and innovation of machine parts are transferred to the virtual environment. The programs Catia, Inventor, SolidEdge, SolidWorks, NX cad, CFturbo, and many others have been used recently for this purpose (*Chandrasekaran, Santhanam, and Venkateshwaran, 2021*). The programs Ansys, TCAE, FlexSim, AutoCAD CFD, and others are used to simulate fluid flow. They are used to verify the functionality or innovation of components (*Gülich, 2010*). For example, (*Sankar*, *2018*) used Ansys to study the impact of change in the number of impeller blades and the size of their outlet angle on the efficiency and head of the pump. However, the validation of numerical models, i.e., their experimental verification, remains an issue. For example, (*Hassan, Abdallah, and Abou El-Azm Aly, 2016*) compared the numerical model of a pump in the Ansys program with experimental measurements when innovating the impeller. In their work, (*Alemi et al., 2015*) verified the numerical model with available experimental data, and there was good agreement between the model's results and reality. The pressures and flow rates of the flowing fluid can be easily verified using pressure gauges and flowmeters. The measurement is described in the standard ISO 9906 (*International Organization for Standardization, 2012*). Other more sophisticated methods for capturing fluid flow can be PIV (Particle Image Velocimetry) methods (*Corpetti et al., 2006*). PIV methods are usually used successfully to verify Ansys simulations. For example, (*Furst et al., 2021*) reached an agreement on the streamline's shapes and the fluid velocity when verifying the mathematical model with the measured values in the test laboratory using PIV. On the other hand, (*Owida et al., 2010*) compared PIV and Ansys models and did not reach an agreement. The reason was the imperfect transparency of the pump during experimental measurements. To obtain reliable results from numerical models, the setting of the density of the computational mesh is one of the most important parameters. A coarse mesh is computationally simpler but can severely skew the results. In contrast, a finer mesh gives more accurate results, but the computation time increases, and convergence becomes complicated (*Gülich, 2010*).

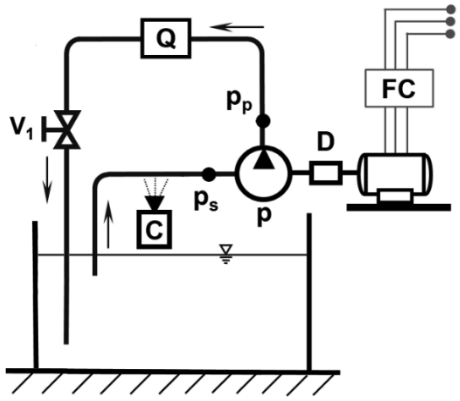
The aim of this research was to verify the reliability of the numerical Ansys model for predicting fluid behaviour when flowing through a radial centrifugal pump.

MATERIALS AND METHODS

Flow visualization in the suction pipe using the PIV method

Verification tests were conducted on an open hydraulic circuit in the laboratory of fluid mechanics at the Faculty of Engineering, Czech University of Life Sciences Prague. The circuit diagram is shown in Figure 1.

The test circuit consisted of a tested pump (P), a reservoir with pipes, and control and measuring devices. The motor with the momentum sensor (D) Magtrol TMB 307/41 (accuracy 0.1%) allowed for the continuous regulation of shaft speed via the frequency converter (FC) LSLV0055s100-4EOFNS. The water flow was measured using an electromagnetic flowmeter (Q) SITRANS FM MAG 5100 W (accuracy 0.5%). The pressures at pump inlet p_s (p-in) and pump outlet p_p (p-out) were measured by the pressure sensor HEIM 3340 (accuracy 0.5%), which was installed according to the first-class accuracy requirements.



Q – flowmeter
P – tested pump
V_1 – control valve
p_p, p_s – pressure sensors
D – dynamometer
FC – frequency converter
C – camcorder PIV

Fig. 1 Hydraulic circuit for pump testing (*Polák, 2019*)

A single-stage radial centrifugal cast iron pump with a spiral casing was used for the measurement. The diagram of the pump and its performance parameters guaranteed by the manufacturer is presented in Fig. 2. The evaluation tests were based on the CSN EN ISO 9906 standard providing the tests of hydrodynamic pumps. The measurements were performed at 1,450 rpm and 2,950 rpm, which were set using a frequency converter. The torque was measured by a torque sensor on the shaft between the pump and the electric motor. The pressures were monitored by pressure sensors on the suction (p_s) and discharge pipes (p_p). The flow through the pump was measured using a flowmeter located in front of the throttle valve. Gradual closing of the throttle valve increased the head of the pump. The power parameters of the pump were measured at constant speeds, and velocity field measurement was performed synchronously using the PIV method.

Guaranteed

Shaft speed:
n_P = 1,450 min^{-1}

Flowrate:
Q_P = 3.54 l/s

Total head:
H_P = 5.85 m

Power input:
P_P = 0.33 kW

Efficiency:
$\eta_č$ = 62.5 %.

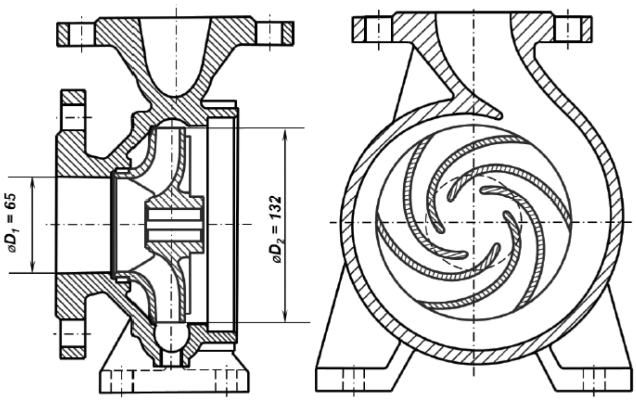


Fig. 2 Cast iron impeller in spiral casing and parameters of pump *(Polák, 2017*)

A 2D PIV set from the TSI company was used to measure the velocity fields in the pump suction pipe (see Fig. 3). The basis was a two-pulse Nd:YAG laser (YAG100-100-LIT) with a wavelength of 532 nm operating with an optical device Light Sheet Optics 610026 and a camera Powerview Cameras 630092. The set was completed by a synchronizer LaserPulse Model 610036 and COMPUTER for PIV 600054-64 with INSIGHT ™ 4G-2DTR Data Acquisition software. Fluorescent particles were dispersed in the flowing fluid –hollow glass spheres 100-SLVR with a diameter of 12 μm, silver-coated to increase the reflection of light on the surface.
A vertical plane in the axis of the transparent suction pipe at the pump inlet was selected to monitor the flow (see Fig. 3). A laser was placed above the pipe, repeatedly emitting two consecutive light pulses with a time delay of 50 μs. The optical system directed the emitting laser beam into a thin light sheet which illuminated the monitored area in the suction pipe. A high-speed camera positioned perpendicular to the plane scanned the area at the same frequency as the laser pulses. This was provided by the synchronizer. The images from the camera captured the positions of the fluorescent particles. The first image (t) displayed the initial positions of the particles and the second (t´) the final positions. The image processing was carried out by specialized software, which, by comparing the corresponding pairs, determined the directions and sizes of the velocity vectors of individual particles or flowing fluid. The Scilab program was used to visualize the measured data. The graphical form of the vector fields was created in this program.

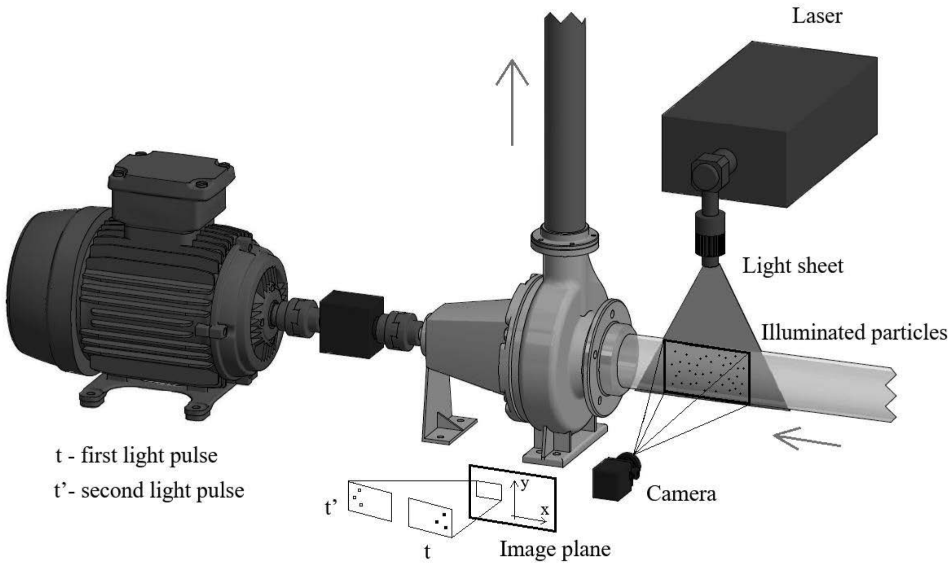


Fig. 3 Diagram of PIV method application during the experiment (*Černý and Sitte, 2020*)

Numerical model of flow in the suction pipe
The model of the pump (spiral casing, impeller, and suction pipe) was created in the Inventor 2022 program. For the purposes of numerical flow simulations in the Ansys program, the model was further modified to fill the flowing fluid space with a fine tetrahedral mesh. Two of the most important mesh quality parameters are Element Quality and Skewness. The value of Element Quality ranges from 0 to 1. A value of 1 indicates a perfect cube or square, while a value of 0 indicates that the element has a zero or negative volume. Its value is calculated according to equation (1), where parameter C corresponds to the element type. For tetrahedrons, C = 124.70765802. The frequency of mesh elements was highest in the interval of element quality values from 0.9 to 1, inclusive.

$$Element\ Quality = C\left[volume/\sqrt{\left(\sum(Edge\ length)^2\right)^3}\right] \quad (1)$$

Skewness is actually directly related to the quality of mesh structure, and it shows how close the mesh structure is to its ideal shape or form. When the Skewness decreases, it means a higher element quality.

Figure 4 presents the graphical dependence of the number of elements on the Skewness value and, at the same time, the Skewness quality spectrum (*Ansys, 2012*). The numerical model of the pump experimentally verified in this study had the majority of mesh elements in the range of Skewness value 0 – 0,25, which corresponds to excellent mesh quality.

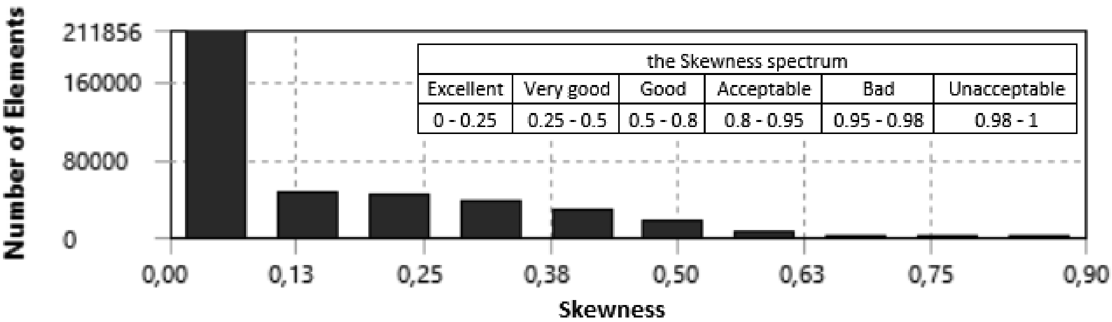


Fig. 4 Mesh quality according to skewness

To validate numerical models of flow using the PIV method, three sets of simulations were created in Ansys, which differ from each other by input parameters. The sets are marked with numbers 1 to 3, and their overview is presented in Tab. 1. For the first set of simulations, the measured pressure at the pump inlet and the mass flowrate at the pump outlet were used as input parameters. For the second set, the flowrate at the pump inlet and the pressure at the pump outlet were used. And for the third set of simulations, the velocity at the pump inlet obtained from the PIV measurement and the flowrate at the outlet were chosen. The calculated values of pressures and flowrates at the control points and graphically represented curves of the streamlines in the vertical plane in the axis of the suction pipe were the results of the numerical models. Simulations were performed at all points of the pump performance characteristics.

Tab. 1 Setting input parameters of numerical simulations

Setting No.	Input variables for numerical model
1	Pressure at the pump inlet, mass flowrates at the pump outlet
2	Mass flowrate at the pump inlet, pressure at the pump outlet
3	Velocity at the pump inlet from PIV, mass flowrate at the pump outlet

RESULTS AND DISCUSSION

The curves in the graph in Fig. 5 represent the pressure values (p-in/p-out), including the standard deviation, measured on the test circuit at 1,450 rpm. The measurement of the pump performance characteristic consisted of five partial measurements (P1 to P5). The points in the graph (Ansys out) indicate the calculated outlet pressures at setting 1 and are used for comparison with the measured outlet pressures (p-out). The points (Ansys in) are the calculated inlet pressures at setting 2 and are used for comparison with the measured values of inlet pressures (p-in).

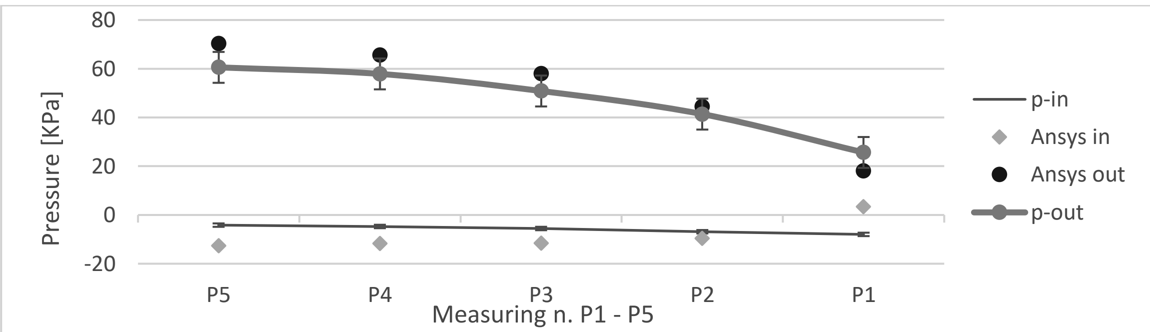
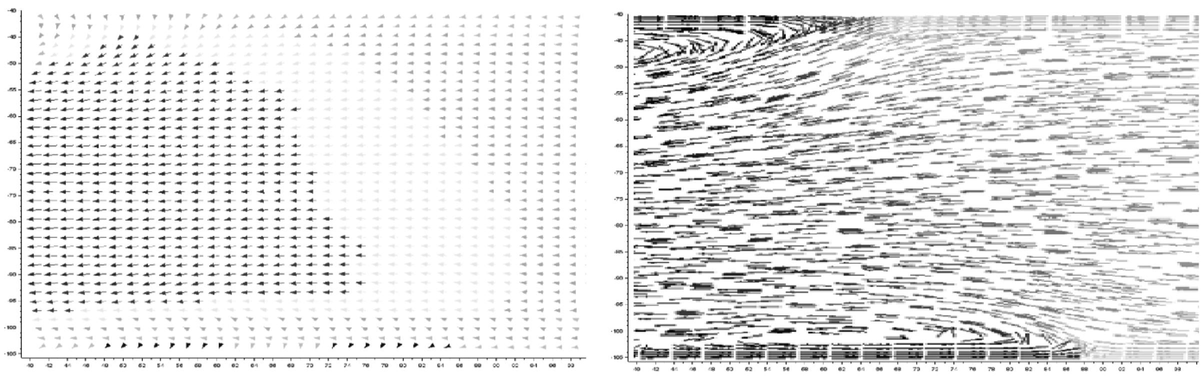


Fig. 5 Graph of measured and calculated pressures

At setting 1, the calculated pressure values are very close to the measured values – the largest deviation is 9.8%, the smallest less than 1%. At setting 2, the trend of the pressure course in Ansys was opposite to that of the measured values – i.e., the pressure at the pump inlet gradually decreased while the measured values increased. In most cases, the calculated pressure was significantly lower than the measured, even twice as much. For the calculations applied, the computation time of the numerical simulations increased with increasing head. In the first measurements (P1), the convergence occurred within 500 iterations; in the last (P5), the convergence occurred after 5000 iterations. This behaviour corresponds to the assumptions made by (*Gülich, 2010*) according to which the unsteady flow tends to converge more poorly. In setting 3, the calculation did not converge. Although this setting gave a good agreement with the shapes of the streamlines that corresponded to the PIV measurement, the results cannot be considered.

Another output of numerical models in Ansys were vector maps of fluid velocities in the suction pipe of the pump, which were compared with PIV measurements – see Fig. 6. For these purposes, vector maps were generated in Ansys in the vertical plane identical to PIV measurements, i.e., right in front of the impeller inlet. In this plane, velocity vectors were depicted only in the "tangential" direction (see Fig. 6, right). For comparison, the centre of the performance characteristic (P3) was selected, corresponding to the maximum efficiency and for which the manufacturer guarantees the parameters of the pump. The maps below indicate a steady fluid flow from the right side, which fully corresponds to the PIV measurement. The velocity increases as the fluid moves to the left towards the impeller. Both Ansys and PIV show this fact. In the lower part of the picture, PIV measurements show vectors pointing in different directions, indicating vortices' formation. A more detailed description of the PIV method results is provided by (*Černý and Sitte, 2020*). The vector field in Ansys illustrates this vortex more prominently. The vortex is not local but arises along the entire inner circumference of the suction pipe. However, a 3D PIV measurement would be needed to accurately describe it because the particles in this part of the pipe move generally in space and thus outside the monitored plane, not captured by the 2D PIV method.



PIV method Ansys calculation

Fig. 6 Measured and calculated vector maps at 1450 rpm at measuring point P3

CONCLUSIONS

This research aimed to determine with what reliability it is possible to use a numerical model in Ansys to predict the behaviour of a fluid flowing through a radial centrifugal pump. For this purpose, the model's numerical and graphical outputs were compared with the values of measured performance characteristics and velocity fields obtained from the PIV method.

A comparison of the three different setting methods of the model in Ansys proved the best agreement of the calculation with reality when the pump inlet pressure and the pump outlet mass flowrate were set as input variables (setting 1). The calculated and measured parameters differed from 1 to 9.8% in this case. If the mass flowrate at the pump inlet and the pressure at the pump outlet (setting 2) were set as input variables, the deviations from reality were much greater. When the velocity at the pump inlet and the mass flowrate at the pump outlet were set as input variables, the calculation did not converge at all.

From the point of view of the analysis of vector maps of fluid velocities in the pump´s suction pipe, the model presents comparable shapes of streamlines at settings 1 and 2. Vector maps from the Ansys program correspond well to the outputs from PIV measurements. Possible differences are caused by the fact that 2D PIV shows vectors projected only into the measured area. Therefore, particles moving in the direction from/to the monitored plane will not be displayed in the result. In contrast, the Ansys model can display not only vectors in the area, but also their projections into all streamlines, in any cross section of the pipe. The results of this research, together with the flow model in Ansys, will be used as a basis for a more detailed flow analysis regarding the anticipated innovations of the radial centrifugal pump.

ACKNOWLEDGMENT
This study was supported by the Internal grant agency 2022 of the Czech University of Life Sciences Prague, number: 2022:31130/1312/3102

REFERENCES

1. Ansys. "ANSYS Meshing User's Guide." Release 14.5 (2012): 124-137.
2. Alemi, Hamed, Seyyed Nourbakhsh, Mehrdad Raisee Dehkordi, and Amir Najafi. 2015. 'Effects of Volute Curvature on Performance of a Low Specific-Speed Centrifugal Pump at Design and Off-Design Conditions'. *Journal of Turbomachinery* 137. doi: 10.1115/1.4028766.
3. Černý, Jan, and David Sitte. 2020. 'Analysis of the Flow of Liquid in the Suction Pipe of the Radial Centrifugal Pump'. in *Proceeding of 22nd International Conference of Young Scientists 2020*. Vol. 22. Prague: Czech University of Life Sciences Prague.
4. Chandrasekaran, M., V. Santhanam, and N. Venkateshwaran. 2021. 'Impeller Design and CFD Analysis of Fluid Flow in Rotodynamic Pumps'. *MATERIALS TODAY-PROCEEDINGS* 37(2, SI):2153–57.
5. Corpetti, T., D. Heitz, G. Arroyo, E. Memin, and A. Santa-Cruz. 2006. 'Fluid Experimental Flow Estimation Based on an Optical-Flow Scheme'. *EXPERIMENTS IN FLUIDS* 40(1):80–97.
6. Furst, Jiri, Tomas Halada, Milan Sedlar, Tomas Kratky, Pavel Prochazka, and Martin Komarek. 2021. 'Numerical Analysis of Flow Phenomena in Discharge Object with Siphon Using Lattice Boltzmann Method and CFD'. *MATHEMATICS* 9(15).
7. Gülich, Johann Friedrich. 2010. *Centrifugal Pumps*. Vol. c2010. Berlin: Springer.
8. Hassan, Ahmed, Haïscam Abdallah, and A. Abou El-Azm Aly. 2016. 'Effect of Impeller Blade Slot on Centrifugal Pump Performance'. *Global Journal of Researches in Engineerin* 16:71–85. doi: 10.17406/GJREjVol16Is4pg71.
9. International Organization for Standardization. 2012. 'ISO - ISO 9906:2012 - Rotodynamic Pumps — Hydraulic Performance Acceptance Tests — Grades 1, 2 and 3'. 59.
10. Owida, Amal, Hung Do, William Yang, and Yos S. Morsi. 2010. 'PIV MEASUREMENTS AND NUMERICAL VALIDATION OF END-TO-SIDE ANASTOMOSIS'. *JOURNAL OF MECHANICS IN MEDICINE AND BIOLOGY* 10(1):123–38.
11. Polák, Martin. 2017. 'Experimental Evaluation of Hydraulic Design Modifications of Radial Centrifugal Pumps'. *Agronomy Research* 15(Special Issue 1):1189–97.
12. Polák, Martin. 2019. 'The Influence of Changing Hydropower Potential on Performance Parameters of Pumps in Turbine Mode'. *Energies* 12(11). doi: 10.3390/en12112103.
13. Sankar, Shanmugasundaram. 2018. 'Analysis of Centrifugal Pump Impeller Using ANSYS'. *International Journal of Innovative Research in Science, Engineering and Technology* 7:5021–26.

Corresponding author:
Ing. Jan Černý, Department of Mechanical Engineering, Faculty of Engineering, Czech University of Life Sciences Prague, Kamýcká 129, Praha 6, Prague, 16521, Czech Republic, email: cernyjan@tf.czu.cz