Radial Force Loads on the Impeller of a Spiral Casing Pump – Comparison of Experiment and CFD Simulation

David Bleho, Robert Olšiak, Branislav Knížat, and Peter Tóth

Abstract. Computational fluid dynamics (CFD) is becoming a widely used tool in the industrial fields. Experimental methods are expensive and time-consuming and CFD is a suitable substitute. In order to investigate the accuracy between the two methods, the radial force on a 6-bladed impeller was measured. The results reveal that the steady-steady analysis method fails, and higher accuracy is achieved using transient simulation. Fairly good accuracy is achieved when simulating the power parameters, where it is sufficient to use the steady-state method as well to save time. On the other hand, for force load detection, where the magnitude of the radial force varies with the position of the impeller, the transient method should be used. The distribution of the non-stationary radial force vectors is symmetric around the origin and lies mainly in 6 regions, which are the same as the number of impeller blades.

Keywords: Radial force; CFD; Experiment; Centrifugal pump

* Corresponding author: david.bleho@stuba.sk
1 Introduction

When designing the mechanical parts of the pump, it is necessary for the designer to determine the magnitude of the anticipated load on the shaft and bearings that may occur during operation. This load may be of different origin (mechanical or hydraulic) and therefore the resulting value is usually the sum of contributions from several sources. The hydrodynamic forces acting on the impeller contribute the most to the load on the rotor parts. These forces are the result of the interaction of rotor parts (impeller) and the stator parts (volute). Their vectors are usually decomposed into axial and radial directions, as implied by the principle of load trapping in the bearings. The magnitude of the hydrodynamic forces affects the sizing of the basic structural elements, which include the shaft (determination of its diameter) and the bearings (type and size). Therefore, it has an impact on the commercial parameters of the pump such as price and overall lifetime [6]. It is clear that the ability to determine these forces is a necessary part for the designer if he wants to optimize the design of the pump. The nature of the hydrodynamic forces is determined by the interaction of the impeller-volute. Since the flow at the impeller-volute interface is non-stationary, the resulting forces also have a pulsating character, as a result of which the spreading vibrations are accompanied by acoustic phenomena [10], [5]. Hydrodynamic pumps are machines that have undergone a rather intensive development over the last hundred years and exist in dozens of basic designs, differing in impeller type, distributor type, number of stages, number of blades, etc. This is because there is no uniform procedure for calculating radial and axial loads, but it is necessary to consider each pump separately. Dealing with this issue, we can find in the literature presentations of many results of either experimental or numerical research. Some of the fundamental works in the field of radial forces research are [1], [2], [12] and [13]. The researchers’ attention is mainly devoted to spiral hydrodynamic pumps. The spiral as a diffuser has an asymmetrical shape and causes asymmetries of pressure and velocity fields in non-optimal modes.

This creates radial components of hydrodynamic forces of considerable magnitude acting on the impeller. Csanady [2] gives a theoretical analysis of radial forces caused by a flow in the spiral. His approach is restricted to a spiral of logarithmic shape, and he proposes a procedure for calculating the magnitudes of these forces. The results of the calculations are compared with the experimental results of [13]. The article [13] by Iversen et al. is devoted to the experimental investigation of radial forces in a spiral pump. In addition to the results of force measurements, a model is developed to determine the pressure distribution at the inlet of the volute at non-optimal modes. The asymmetry of the pressure distribution outside the optimum mode is the cause of the radial force. A procedure for calculating the radial force is also given and compared with experimental data. In the article [1], Biheller presents experimental results for three different types of spirals. We can also find semi-empirical and empirical formulas approximating the measured data there. Adkins and Brennen [12] in their article deal with experimental and theoretical analysis of the causes of radial forces in a spiral pump. The authors compiled a mathematical model of flow in the impeller – spiral system, which is used to calculate the pressure fields behind the impeller and the associated radial forces. An experimental device made for the measurement of radial forces and pressure fields behind the impeller is presented. An extensive comparison of the results, measured and calculated pressure fields and radial forces, is presented. An example of reducing radial loads using a double spiral is presented in [8]. A pump with a long shaft used for fluid transport is described. The hydraulic load causes deflection of the shaft, which leads to a reduction in operational stability. By CFD methods, it is shown in this work that these undesirable phenomena can be eliminated by using a double spiral (by symmetrizing the flow fields). Similarly [11], describes the development of a multichannel spiral, which aims to significantly reduce the radial load on the impeller caused by hydrodynamic forces. The
authors reached the final solution using computer simulations of the flow in the pump. In
general, the work of determining the radial force on the impeller is either purely
computational (using e.g., CFD methods) [4] or a combination of experiment and calculation
(based on CFD methods or the flow hypothesis itself). In some publications, the experiment
is technologically simpler when the pressure force is calculated from the measured pressure
fields, or only the measured pressure fields themselves are presented [3], [7]. A more
technologically demanding experiment is represented by work where the radial force is
directly measured using strain gauge force or stress sensors. This group includes articles [9],
[12], [1], resp. [13]. In [4], the first method of experimental determination of force is referred
to as indirect and the second method is referred to as direct.

2 Radial force analysis

The determination of the hydrodynamic radial force is an essential step in the design of
spiral hydrodynamic pumps. The designer has several options available today – for example,
he can use empirical or semi-empirical equations of various authors (e.g., Stepanoff or
Biheller [1]) which are now considered classical in the literature. These relations have been
derived under certain general assumptions, that may not be satisfied. Their accuracy is
therefore questionable; they can only be used for preliminary analysis. Another possibility is
to determine the force for a particular impeller shape. In Fig. 1 [1] shows the silhouette of the
radial impeller. The radial force acting on an impeller of this shape is a combination of the
pressure action on its outer surface and the hydrodynamic effect of the flowing fluid due to
a change in its momentum. The components of the radial force $F_x$ and $F_y$ can be expressed by
relations (1), (2) respectively.

\[
F_x = -b_{2c} R_z \int_0^{2\pi} p_2(\varphi) \cos(\varphi) \, d\varphi \quad (1)
\]
\[
F_y = -b_{2c} R_z \int_0^{2\pi} p_2(\varphi) \sin(\varphi) \, d\varphi \quad (2)
\]

where $b_{2c}$, $R_z$ and $\varphi$ are geometric parameters evident from Fig. 1.; $p_2$ is the pressure (index
2 means the impeller output).
These fields need to be specified – i.e., by calculation or measurement. The designer is most satisfied if he has an efficient calculation procedure. In [13] a one-dimensional spiral flow model is proposed for this purpose. The flow is described by an ordinary differential equation whose solution gives the pressure distribution at the outlet of the impeller as a function of flow. The idea of flow is based on the fact that the pressure distribution is strongly influenced by the mixing process and frictional losses in the volute. However, this approach does not provide satisfactory results because the experiment and calculation do not match quantitatively low models of this type. Based on some hypothesis it will always be incomplete and will not sufficiently reflect the real phenomena taking place in the pump. Therefore, CFD methods have become indispensable in refining the calculation [10], [4], [3], [8], [9]. The advantage of the CFD approach is the possibility of obtaining such parameters, the measurement of which would be very technically demanding, such as shaft deflection (article [8]) or non-stationary pressures and radial forces [8], [4]. On the other hand, the CFD computation is dependent on the choice and setting of the mesh, turbulence model, boundary conditions, etc. Therefore, the results must be compared with the measurements at least for selected cases in order to generalize the procedure. Such a comparison is provided e.g., by article [9].

3 Experiment and measurement method

3.1 Model pump

The impeller selected for the experiment was developed and manufactured at the Department of Hydraulic Machines. The characteristic dimensions of the meridional section and the shape of the impeller are evident from Fig. 2 and the spiral tapered diffuser in Fig. 3. The operating parameters, along with the speed-acceleration, are evident from Tab. 1. The impeller is made of a combination of materials steel (rear disc), plexiglass (blades) and dentacryl (front disc). The spiral is made by wax core casting technology using dentacryl material.
Fig. 3. Hydraulic spiral design and overall view.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Sign</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow rate</td>
<td>Q (m³/h)</td>
<td>20.31</td>
</tr>
<tr>
<td>Head</td>
<td>H (m)</td>
<td>13.78</td>
</tr>
<tr>
<td>Rotational speed</td>
<td>n (r/min)</td>
<td>1200</td>
</tr>
<tr>
<td>Specific speed</td>
<td>n₀</td>
<td>0.072</td>
</tr>
<tr>
<td>Efficiency</td>
<td>η (%)</td>
<td>77.13</td>
</tr>
<tr>
<td>Blade number of guide vane</td>
<td>Zᵥ</td>
<td>6</td>
</tr>
</tbody>
</table>

**3.2 Radial force measurement method**

The chosen method of radial force measurement consists in releasing the shaft from its mechanical and hydraulic parts so that it can perform a spherical motion around the rear bearing and then transferring the applied force induced in the front bearing outside the pump body to the three strain gauge force transducers.

The bearing is housed in a sleeve that is released from the bearing housing with sufficient radial clearance. Three thrust rods are fixed in the sleeve at an angle of 120° to each other, which are in point contact with the tapered tip of the strain gauges by means of limiting screws. The transfer of radial force to the strain gauges is affected only by the pull of the prestressed tension rods without contact with the body of the bearing housing.
3.3 Experimental circuit

For the purpose of this experiment, a closed test circuit was used. The circuit starts in a tank of fluid whose level is above the shaft axis. The liquid from the tank flows through the pipe $Pt1$ to the slide on the suction $P1$, where it is fed through the pipe $Pt2$ to the pump inlet by means of a flexible hose $Phs$. A pressure tap is located in the $Pt2$ pipe for pressure measurement, which is led to a manometer for measuring the suction pressure ($p_s$). A $Pt3$ pipe is connected to the flange of the volute with a pressure tap connected to a pressure gauge for measuring the pump discharge pressure ($p_v$). The pump flow is controlled by means of a shut-off slide $P2$ attached to the $Pt3$ pipe. The speed control of the pump during its operation is carried out by changing the speed of the electric motor by means of the thyristor control $TR$. The circuit from the pump discharge continues through a flexible hose $Phv$ to three parallel apertures with pipe diameters $\phi 80$, $\phi 100$ and $\phi 150$. In each aperture, a standardized orifice for flow measurement with the corresponding diameter is incorporated in the pipe. By selecting an aperture with a suitable pipe diameter, it is possible to reduce losses in the system and thus ensure that the performance of any pump alternative is measurable beyond the optimum point. The circuit is closed by a $Pt4$ pipe which returns the liquid back to the collection tank.

Fig. 4. Drawing and fixing the sensors on the model pump.

Fig. 5. Experimental circuit.
Mass flow measurement
The mass flow rate was measured using the pressure differential upstream and downstream of the orifice on a given pipe. The pressure differential was determined from the deflection \((h_{C1}, h_{C2})\) of the mercury U-gauge and the corresponding orifice constant \((K_C)\).

\[
Q_m = K_C \cdot \sqrt{h_{C1} + h_{C2}} \quad \left[ \frac{m^3}{s} \right]
\]  

(3)

Specific energy measurement
The specific energy is defined by the pressure, kinetic and positional energy between the inlet and outlet of the pump.

\[
Y_m = \frac{p_v - p_s}{\rho} + \frac{v_v^2 - v_s^2}{2} + (h_v - h_s) \cdot g \quad \left[ \frac{J}{kg} \right]
\]

(4)

Power measurement
The transfer of torque to the pump shaft is carried out by means of an elastic coupling with a rubber strain member, which allows possible small misalignment of the pump shaft due to its loading by radial force.

\[
P_m = 2 \cdot \pi \cdot n \cdot M_k \quad [W]
\]

(5)

Determine the pump efficiency using the equation:

\[
\eta = \frac{\rho \cdot Q \cdot Y}{P} \quad [-]
\]

(6)

4 CFD prediction of radial force
Numerical analysis using CFD method was performed to find the radial hydraulic force on the impeller with six blades. The three-dimensional model for the fluid body of the centrifugal pump is built with CATIA V5. Ansys CFX commercial software was used to generate the mesh, perform the numerical calculations and create the computational model. First, a model of the impeller with the spiral was generated in CAD and then discretized into a mesh using Ansys Meshing. The straight section of the suction pipe must be long enough for the correct definition of the boundary conditions, because in some cases there are intense secondary phenomena on the suction side of the radial pump [18] and the fluid velocity at the inlet becomes stable if the length of the inlet pipe is more than twice the diameter of the inlet [19]. In this case, the length of the straight section of suction pipe was equal to five times the inlet impeller diameter.

Fig. 6. Assembly depiction.
The mesh was generated as unstructured and consisted of tetrahedral elements – impeller and volute, and hexahedral elements – suction and discharge pipes Fig.7. The total number of grid elements was 5.9 M (suction pipe: 0.4 M, impeller: 2.5 M, spiral with discharge pipe: 3 M), and the element size was set to achieve a balance between calculation accuracy and calculation time, because the rapid increase in the number of grid elements puts a burden on the computational device. As can be seen on the Fig. 7, areas close to the walls and in the gaps between the rotor blade tip and the pump body, the computational mesh was significantly densified.

The simulation model was divided into three domains: A – suction pipe, B – impeller and C – spiral casing with discharge pipe. By studying the strong interaction between the impeller blade and the spiral tongue [16], a non-stationary numerical analysis (transient) was performed. This eliminated the negative clocking effect [14] at the stator-rotor interface, especially at the narrowest point between the impeller and the spiral nose. The flow model was based on the solution of the RANS equations for the flow of an incompressible fluid. In this work, Menter's turbulent shear stress transport (SST) model was implemented. According to [15] Mentor's SST model appeared to be a suitable choice for flow in a multi-blade centrifugal pump. The interfaces between the rotor and stator parts of the computational domain were of the Transient Rotor Stator type.

The choice of the time step \( \Delta t \) for the transient simulation was set to \( 4.16667 \times 10^{-4} \) s. One time step corresponded to a 3° rotation of the impeller, thus 120 transient results are included to calculate one revolution of the impeller. At least 8 full impeller revolutions were required for each operating point to steady the flow, which represents 960 iterations. During the calculations, the radial forces in the \( x \) and \( y \) directions are monitored and recorded at each time step. The radial force \( F_{\text{rad}} \) value was calculated based on the procedures implemented in the CFX software or CFD Post applied in the CFD simulation. General Grid Interface (GGI) is used to interface the dynamic and static parts during CFX pre-processing.

A steady-state calculation with the same turbulent model and boundary conditions was also performed to compare the time-steady state with the time-unsteady state.

No-slip boundary conditions are enforced on all solid walls. The following boundary conditions were applied: mass flow rate and direction of the absolute velocity vector at the inlet to the computational domain, static pressure at the outlet of the computational domain equal to 1 atm (101 325 Pa), rotor speed 1200 rpm. The working medium was water.
5 Results and discussion

The characteristics of the head and efficiency of the centrifugal pump obtained from the experiment are shown in Fig. 8, and these were compared with those obtained from the unsteady numerical simulation. It can be observed in Fig. 8 that the simulated head and efficiency of the centrifugal pump were slightly higher than the experimental values, but both have the same trend. The experimental and simulation values of the specific energy and efficiency at the design flow rate are $Y_e = 133.913 \, \text{J/kg}$ and $\eta_e = 77.10 \, \%$ and $Y_s = 132.2 \, \text{J/kg}$ and $\eta_s = 80.60 \, \%$, respectively. The measured efficiency includes mechanical losses, which were not considered in the simulation. This accounts for some of the observed differences between experimental and simulation results.

![Fig. 8. Characteristics of the centrifugal pump.](image)

The radial force on the impeller is a vector with magnitude and direction and it changes with the rotation of impeller. In order to compare the radial force under different conditions, the radial force vector’s modulus at every time-step in the last calculation cycle is extracted, and the average values of all the radial force modulus are calculated based on these data. The average values of the radial force under different conditions are displayed in Fig. 9. It is obvious that an unsteady simulation has achieved more accuracy with experimental data, this can cause a large dependence of the impeller position to the stator position.

![Fig. 9. Radial force magnitude.](image)

Fig. 10 gives the distribution of radial force vectors in the last rotation cycle. Every point in the graph represents the radial force of the impeller at one instant. It can be seen that the direction and magnitude of the radial force vector change dramatically with time. The vector distribution of radial force under different flow rate in a cycle is regular and $(0,0)$ is the
symmetric centre of the vector distribution for all the cases. The radial force vectors are mainly in 6 regions, and the number is same as the number of blades of the impeller. This indicates that most radial forces are greater by non-operating conditions.

Fig. 10. Distribution of radial force vectors and time domain curves of radial force pulsation.

Fig. 11 shows the time history of instantaneous unsteady pressure contours at the boundary surface near the impeller outlet. The instantaneous non-stationary pressure contours for one revolution $\tau$ of one of the six pump impeller blades are compared. This rotation is divided into six steps to clarify changes in flow structure with time during one revolution of the impeller. In the reference design, high-pressure zones occur widely on the boundary surface near the impeller outlet, as shown in Fig. 11(e), and a high-pressure zone caused by impeller-volute interactions becomes gradually larger. Consequently, this results in the unbalancing phenomena, along with the fluid-induced vibrations caused by unsteady radial forces, throughout the annulus passage area of the pump.

Fig. 11. Unsteady pressure contours during one revolution of the impeller.

The procedure of analysis of measured data in the frequency domain is based on a group of mathematical algorithms of fast Fourier transformation (FFT). The frequency domains of radial force pulsation of the impeller are shown in Fig. 12. The amplitudes of the peaks at 20 l/s are less than at 25 l/s and also at 15 l/s. The amplitudes of the main peaks appear by
the first blade passing frequency (BPF) [20], which is 120 Hz. The pulsation frequency for the highest amplitudes, which indicates the main factor of the radial force fluctuations is the interaction between impeller and volute. This is mainly due to the small gap between rotary impeller and stationary volute, the gap is just 14 mm.

![Graph showing frequency domains of the impeller.](image)

**Fig. 12.** Frequency domains of the impeller.

### 6 Conclusion

In this article, the behaviour of the hydrodynamic force inside a centrifugal pump under different operating conditions was investigated. The investigation of the radial force of the pump was carried out experimentally with the support of CFD methods. The analysis of the results enables to draw the following conclusions.

1. Unfortunately, steady-state calculations cannot capture all the interaction effects with complete fidelity. Therefore, transient simulation was used for the calculations, the accuracy of which is satisfactory for the radial force investigation process, as the CFD results showed (Fig. 9) sufficient agreement with the experimental measurements. However, the calculated values are lower than the measured ones. The reason for this phenomenon may be due to the assumption that the simulation does not take into account all the processes occurring during real fluid flow in a multi-blade pump (e.g., backflow in the gap between the blade and the front disc). The pump performance parameters show good agreement with the measured curves (Fig. 8) but the efficiency curve is slightly higher. This may result in the CFD not considering mechanical losses. The measurement confirmed the fact that the maximum values of the pressure amplitude in the impeller are when the pump is operated outside the design conditions.

2. The radial force and vibration displacement of the impeller were lowest at the design flow rate and increased at off-design flow rates. The variation trend of the radial
displacements in the X- and Y-directions was consistent with that of the radial forces. The radial force vectors are in a centre of symmetric distribution, which mainly lie in 6 regions, corresponding to the number of blades of the impeller. The frequency for the largest amplitude is always in steps of 120 Hz, which is due to the periodic motion of pump impeller rotation.

References


This work was supported by the Slovak grant agency KEGA, project No 016STU-4/2022.