Test bench flow straightener design investigation and optimization with computational fluid dynamics methods

A Petrov1,2, N Isaev1 and M Kuleshova1
1Bauman Moscow State Technical University, 5 Second Baumanskaya Street, Moscow, 105005, Russian Federation
2E-mail: alex_i_petrov@mail.ru

Abstract: The article contains the application of computational fluid dynamics methods to optimization of flow straightener, which is a part of test bench for centrifugal pumps manufactured with additive technologies. Problems faced during testing of large-scale prototypes are shown, e.g. unsteady cavitation processes occurring in case there is no straightener, the ways to solve such problems are suggested. The design of the optimized flow straightener is described, and its effectiveness is proved by calculations carried in STAR-CCM+ software.

1. Introduction
For the last 10 years the E-10 department of BMSTU is actively working on optimization of centrifugal pumps wet parts using the methods of hydrodynamic modeling [1], as well as experimental verification of the results obtained based on testing of large-scale prototypes, manufactured with additive technologies. A 3D model and a photograph of such a test bench are shown in figure 1.

Figure 1. The test bench used to obtain characteristics of pump prototypes.

During testing several problems were faced due to relatively small size of the tank installed. In particular, when testing the fuel pump prototype with inlet flow of 400 m³/h, unsteady effects (pressure low-frequency pulsations) as a result of control valve cavitation were observed in the tank.
and suction line. These phenomena led to the impossibility of obtaining a stable experimental characteristic of the prototype, so it was important to study the flow in the test bench tank and optimize its design to eliminate the influence of cavitation phenomena on the pump characteristic.

In order to solve this problem methods of computational fluid dynamics in application to multiphase flows were used, as a result an optimized test bench and tank designs were obtained, and the tests were fully completed.

2. Methods

As it is known from years of experience [2–6], for correct testing of centrifugal pumps, it is necessary to achieve stabilization of the fluid flow at the pump inlet. This can be achieved in case the tank installed has a large size (it is recommended to use the tank with volume providing not less than 2 minutes of mass flow of the largest pump tested), several tanks of smaller volume installed consequently can also be used or flow straightener can be introduced to the tank. There are 3 main requirements for the flow straightener performance:

1. Full damping of the velocity head in the tank at the outlet of the discharge pipe;
2. Removal of air bubbles and cavitation pockets from the fluid, entrained in the flow from the discharged pipeline;
3. Creating minimal resistance of the flow in the tank.

It is quite difficult to develop a straightener capable of solving these problems in a tank of minimal volume purely empirically, therefore, there is a need to investigate and optimize the design of such a straightener with hydrodynamic modeling methods [7].

The difficulty in formulating the problem of hydrodynamic modeling lies in the fact that to correctly calculate the flow in the tank and the straightener, it is necessary to take into account both the two-phase nature of the working fluid (taking into account cavitation pockets and air cavities formed at the outlet of the control valve in the discharge pipeline) and the tank free surface of the fluid through which the exchange of matter between the fluid and the air pad in the tank happens.

As initial conditions for modeling, the initial gas phase concentration in the fluid at the outlet of the control valve (during the tests it can vary, but for convenience of comparative analysis of the calculation results a constant value of 5% volumetric gas concentration was taken), flow through the tank (speed at the outlet and the total pressure at the inlet) were set [8].

Criteria for assessing the quality of the straightener performance are based on the percentage of the gas phase at the outlet of the tank, i.e. at the inlet to the pump suction piping, and its ratio to the percentage of the gas phase at the tank inlet (estimated by the introduced value of the gas concentration reduction ratio calculated separately for the average and peak gas concentration values in the suction pipe), the uniform distribution of velocities in the suction pipe (described by the standard deviation of speed) and hydraulic losses from the entrance to the tank to the exit from the tank (described by the coefficient of the total hydraulic losses in the straightener).

The method of numerical simulation is based on solving discrete analogs of the basic hydrodynamic equations. [9–11]. In case of incompressible fluid ($\rho = \text{const}$):

Mass conservation equation (continuity equation)

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0,$$

where

$\bar{u}_j$ – averaged velocity value in projection on axis $j$ ($j = 1, 2, 3$).

Momentum equation (Reynolds averaged):

$$\rho \left[ \frac{\partial U_i}{\partial t} + U_i \frac{\partial U_j}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ T_{ij}^{(v)} - \rho \langle u_i u_j \rangle \right],$$
where $U, P$ — averaged velocity and pressure; 
\[ T_{ij}^{(v)} = 2\mu \bar{S}_{ij} \] — viscous stress tensor for incompressible fluid,
\[ \bar{S}_{ij} = \frac{1}{2} \left[ \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right] \] — strain velocity tensor;
\[ \rho \langle u_i u_j \rangle \] — Reynolds stresses.

The introduction of Reynolds averaged Navier—Stokes equations makes the system of equations nonclosed, since additional unknown Reynolds stresses appear. To solve this system, a semiempirical $k-\omega$ SST model of turbulence was used, which introduces the necessary additional equations: the transfer equations for the kinetic energy of turbulence and the relative dissipation rate of this energy:
\[ \frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{P_k}{\rho} - \beta k \omega + \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial + \sigma_{i3} \nu_f}{\partial x_i} \right) \frac{\partial k}{\partial x_j} \right] \]
\[ \frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha \cdot S^2 - \beta \cdot \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \frac{\partial + \sigma_{i3} \nu_f}{\partial x_i} \right) \frac{\partial \omega}{\partial x_j} \right] + 2 \cdot (1 - F_1) \cdot \sigma_{i3} \cdot \frac{1}{\omega} \cdot \frac{\partial k}{\partial x_j} \cdot \frac{\partial \omega}{\partial x_j} . \]

To simulate gas bubbles entering the tank, a two-phase model was used. An approach known as VOF (Volume of Fluid) [12] was adopted. In the VOF method, a multiphase medium is considered as a single fluid, which properties vary according to the volume fraction of each of the phases present:
\[ \alpha_i = \frac{V_i}{V} , \]
where $V_i$ — each phase volume fraction; $V$ — cell volume.

The equation for the mass concentration of each of the phases:
\[ \frac{\partial (\alpha_i \rho_i)}{\partial t} + \nabla \cdot (\alpha_i \rho_i V) = 0 . \]

The minimum bubble size and the minimum gas concentration in any calculation cell are strictly limited to very small, but non-zero, values. The time step was chosen based on the characteristic flow rates and cell size. The selected time step is 1e-3s. The number of internal iterations at each time step is 10. To obtain discrete analogues of the original continuous equations STAR-CCM + software uses the control volume method described in [13].

The wet part of the tank was discretized into a set of finite cells (figure 2), for each of which discrete analogs of continuous equations were composed. The combination of all discrete analogs forms a closed system of algebraic equations.

The number of cells is 580 000. Cell base size equals 12 mm.

Reduction ratio of the mean gas concentration:
\[ \bar{g} = \frac{\sigma_{out}}{\sigma_{in}} = \frac{\sum_{i=0}^{n} \sigma_i}{\sigma_{in}} , \]
where 
\[ \sigma_{in} \] — gas concentration at the tank inlet (a constant value of 5% volumetric gas concentration was taken);
\( \sigma \text{out} \) – gas concentration at the tank outlet;
\( \sigma_i \) – gas concentration at the i-time step;
\( n \) – number of time steps.

![Figure 2. The mesh of the tank without the straightener.](image)

Reduction ratio of the mean gas concentration:

\[
g_{\text{max}} = \frac{\sigma_{\text{max}}}{\sigma_{\text{in}}}
\]

where

\( \sigma_{\text{max}} \) – maximum gas concentration at the tank outlet.

Total hydraulic losses in the tank:

\[
\sum h_t = \frac{P_{\text{out}}}{\rho \cdot g} - \frac{P_{\text{in}}}{\rho \cdot g},
\]

where

\( P_{\text{in}}, P_{\text{out}} \) – pressure value at inlet and outlet of the tank respectively.

3. Results
Consequently, three configurations of tank and the straightener geometries were calculated – the tank without the straightener (initial design with worse test results), the tank with type 1 straightener and the tank with the optimized type 2 straightener design.

Figure 3 shows velocity distribution in the initial tank design, figure 4 – volume fraction of phase distributions in this tank.

Figure 5 describes the change of gas concentration in the suction line representing the unsteady behavior of 2-phase flow (gas concentration at the inlet of the tank is constant).
Figure 3. Velocity distribution in the tank without the straightener.

Figure 4. Volume fraction of phases in the tank.
According to the results shown above (figure 4), there is almost no gas separation obtained in the initial tank design (specific gas concentration at the outlet decreases on 25% maximum, table 1). At the same time as a result of flow separation on the turn there are pulsating cavity pockets at the inlet of suction line, which lead to unsteady pump operation during the tests. As practice of such tests in a number of enterprises has shown, such a picture is not an exception and is often found in the test benches with tanks of insufficient volume and without a flow straightener.

| No. | Straightener type       | Reduction ratio of the mean gas concentration/ Reduction ratio of the peak gas concentration | Total losses in the straightener, m |
|-----|-------------------------|-------------------------------------------------------------------------------------------------|------------------------------------|
| 1   | No straightener         | 0.74/2.4                                                                                       | 1.09                               |
| 2   | Straightener, type 1    | 0.09/0.31                                                                                      | 3.27                               |
| 3   | Straightener, type 2    | 0.14/0.42                                                                                      | 2.44                               |

Table 1. Results of calculations carried.

One of the possible ways to resolve such unsteady processes is tank pressurization that eliminates cavitation behind control valve. But the test bench design does not always allow to include such a pressurization, moreover in some cases it may lead to additional fluid aeration and gaseous cavitation [14–17].

Based on initial tank design the flow straightener geometry was developed, its 3D model is shown on figure 6.

However, simulation of the flow in such a straightener has shown that the flow, even in case of small velocities, leads to intensive fluctuations of the free surface (figure 7, 8). It does not affect gas concentration, apart from cases when there is an additional hydrostatic head in the tank (additional fluid aeration), but negatively affects the value of energy losses in the tank [18].

The values of mean gas concentration in the tank with a straightener has decreased 8.2 times, peak values – 7.9 times (table 1). Nevertheless, type 1 straightener is not appropriate in terms of values of losses in the tank and fluctuations of the free surface there [19–20].
Figure 6. The flow straightener.

Figure 7. Velocity distribution in the tank with the straightener, type 1.
Type 2 of straightener does not include upper straightener pipe ring in contrast to type 1 design. It led to decreased value of hydraulic losses and free surface fluctuations, but relative gas concentration at the suction line was increased insignificantly (from 0.09 to 0.14). Calculation results are show on figures 8 and 9 and in table 1.

**Figure 8.** Volume fraction of phases in the tank with the straightener, type 1.

**Figure 9.** Velocity distribution in the tank with the straightener, type 2.
Figure 10. Volume fraction of phases in the tank with the straightener, type 2.

The results of all simulations are presented in table 1.

4. Conclusions and discussion

1. Hydrodynamic simulations performed for the flow in the tank with/without the straightener allowed to identify the source of unsteady effects during pump prototypes tests;
2. Optimized design of the tank-straightener was developed, allowing to eliminate unsteady processes in the test bench without increasing tank volume;
3. The effectiveness of proposed designs was shown during pump prototypes tests on the modified test bench.

Published under licence in *Materials Science and Engineering* by IOP Publishing Ltd. Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

References

[1] Lomakin V O, Chaburko P S and Kuleshova M S 2017 Multi-criteria Optimization of the Flow of a Centrifugal Pump on Energy and Vibroacoustic Characteristics *Procedia Engineering* **176** pp 476-482

[2] Guskov A M, Lomakin V O, Banin E Pand Kuleshova M S 2017 Minimization of Hemolysis and Improvement of the Hydrodynamic Efficiency of a Circulatory Support Pump by Optimizing the Pump Flowpath *Biomedical Engineering* **4** pp 229-233

[3] Gouskov A M, Lomakin V O, Banin E P and Kuleshova M S 2016 Assessment of Hemolysis in a Ventricular Assist Axial Flow Blood Pump *Biomedical Engineering* **4** pp 12-15

[4] Lomakin V O, Kuleshova M S and Bozh’eva S M 2016 Numerical Modeling of Liquid Flow in a Pump Station *Power Technology and Engineering* **5** pp 324-327

[5] Lomakin V O 2015 *Proceedings of 2015 International Conference on Fluid Power and Mechatronics*
[6] Lomakin V O, Kuleshova M S and Kraeva E A 2015 Fluid Flow in the Throttle Channel in the Presence of Cavitation Procedia Engineering106 pp 27-35
[7] Shargatov V A, Gorkunov S V and Il'ichev A T 2019 Dynamics of front-like water evaporation phase transition interfaces Communications in Nonlinear Science and Numerical Simulation67 pp 223-236
[8] Kraposhin M V, Banholzer M, Pfitzner M and Marchevsky I K 2018 A hybrid pressure-based solver for nonideal single-phase fluid flows at all speeds International Journal for Numerical Methods in Fluids88 (2) pp 79-99
[9] Pelevin F V, Avraamov N I, Ir’yanov N Y, Orlin S A, Lozovetskii V V and Ponomarev A V 2018 Intensification of Heat Exchange in the Regenerative Cooling System of a Liquid-Propellant Rocket Engine Journal of Engineering Physics and Thermophysics91 (3) pp 601-610
[10] Bondarenko V L, Valyakina A V, Borisenko A V, Trotsenko A V and Valyakin V N 2018 Vapor-Liquid Equilibrium of the Ethylene–Butane Mixture Chemical and Petroleum Engineering53 (11-12) pp 778-787
[11] Arefyev K Y, Prokhorov A N and Saveliev A S 2018 Study of the breakup of liquid droplets in the vortex wake behind pylon at high airspeeds Thermophysics and Aeromechanics25 (1) pp 55-66
[12] Yakush S E, Iskhakov A S, Melikhov V I and Melikhov O I 2018 Pressure Waves due to Rapid Evaporation of Water Droplet in Liquid Lead Coolant Science and Technology of Nuclear Installations Article no 3087051
[13] Vladimirov I Y, Korchagin N N and Savin A S 2018 Effects of the Influence of a Wave on an Underwater Obstacle by a Stratified Flow of Finite Depth Rep. Earth Sciences478 (1) pp 132-136
[14] Krapivtsev V.G. 2017 Model Studies of Coolant Flow Hydrodynamics in VVER-1000 In-Reactor Pressure Channel Atomic Energy122 (5) pp 304-310
[15] Marchevskii I K and Puzikova V V 2017 Numerical simulation of the flow around two circular airfoils positioned across the stream using the LS-STAG method Journal of Machinery Manufacture and Reliability46 (2) pp 114-119
[16] Arefyev K Y, Voronetsky A V, Suchkov S A and Ilichev M A 2017 Computational and experimental study of the two-phase mixing in gas-dynamic ignition system Thermophysics and Aeromechanics24 (2) pp 225-237
[17] Leont’e v A I, Kuzma-Kichta Y A and Popov I A 2017 Heat and mass transfer and hydrodynamics in swirling flows (review) Thermal Engineering64 (2) pp 111-126
[18] Devistilov V A and Sharai E Y 2016 Numerical study of the flow structure in a hydrodynamic filter Theoretical Foundations of Chemical Engineering50 (2) pp 209-216
[19] Isaev S A, Schelchkov A V, Leontiev A I, Baranov P A and Gulcova M E 2016 Numerical simulation of the turbulent air flow in the narrow channel with a heated wall and a spherical dimple placed on it for vortex heat transfer enhancement depending on the dimple depth International Journal of Heat and Mass Transfer94 pp 426-448
[20] Leontiev A I, Milman O O and Fedorov V A 1992 Thermohydraulic vapor generation processes in vertical tubes of low-pressure natural convection systems Experimental Thermal and Fluid Science5 (1) pp 1-4