Impact of Interface Model on Simulation Results of a Radial Pump at Part Load and Shut-Off Conditions

E. F. Palamarchuk[1*], *A. A.* Zharkovsky[2] and *P. U.* Thamsen[3]

[1]Berlin Heart, Research and Development, 12247 Berlin, Germany
[2]Peter the Great St. Petersburg Polytechnic University, 195251 St.Petersburg, Russia
[3]Technische Universität Berlin, Chair of Fluid System Dynamics, 10623 Berlin, Germany

Abstract. Computational Fluid Dynamics (CFD) is a well-known tool for predicting and analyzing performance in a variety of engineering branches, including turbomachinery, allowing engineers to partially replace physical experiments with their virtual analog. Nevertheless, numerical analysis should be used carefully regarding possible deviation between simulated and experimental results due to multiple reasons (including but not limited to applied simplifications in the numerical model). These deviations usually have their minima close to the Best Efficiency Point (BEP). The paper deals with analyzing the outcome of steady-state simulations for a radial pump at strong part load and shut-off conditions by switching between three simulation types (steady-state with mixing plane, steady-state with frozen rotor, transient with sliding mesh). A comparison of velocity profiles on the interface surfaces is made, showing how the chosen interface model affects the structures being formed at part load conditions. These effects show particular impact on performance parameters (first of all, head production), which is discussed in the paper. The information provided could be helpful for adjusting the simulation parameters and finding an appropriate compromise between simulation reliability and demand for computational time thereby.

1 Introduction

Computational Fluid Dynamics (CFD) methods are widely used for predicting performance in rotordynamic machinery. Even though computational resources are getting more available in the last years, particular simplifications are still broadly used nowadays in order to achieve feasible results in a practically suitable time. For instance, modelling turbulence with RANS is generally used for general turbomachinery applications. Stationary simulations mostly provide feasible results for design conditions, while transient effects start playing a significant role by moving away from the Best Efficiency Point (BEP). Even more, the most basic approach is based on assuming the flow in the vaned rotor and/or stator elements to be periodic. It is thereby possible to limit the computational effort to simulating only one blade

[*] Corresponding author: evgenii.palamarchuk@berlinheart.de

channel, what introduces a significant speed-up proportional to the number of vanes in the element being investigated. Some issues start showing up by translating such results between rotating and stationary domains, which generally have different amount of vanes, making a direct transfer of data impossible. A simple approach would be to introduce a certain scaling (pitching) between rotational and stationary elements to make the data suit the geometrical dimensions of the zones. Feasible results could be achieved in the case of small pitch changes, which is usually the case for turbomachines with high amount of blades. Such method is hardly applicable for rotordynamic pumps, which are usually designed with 2 to 8 blades (depending on specific speed) for the impeller. Having a volute as diffusing element introduces additional limitation for scaling the data, making the pitch change even more significant.

Another method, most commonly known as "Mixing Plane" (or "Stage" in ANSYS CFX) interface, is used for data coupling between rotating and stationary zones. The method is based on introducing "circumferential bands", whose width usually represent mesh resolution on the interface surfaces. The solver is then constrained to balance fluxes of mass and momentum (and eventually other variables used in the simulation) on each side of the band. Circumferential non-uniformities are thereby evened out, making simulation of transient rotor-stator interaction (along with most other transient effects) impossible, but providing a possibility of scaling the results irrespective of the blade number thereby. Nonetheless, such method is usually not recommended for simulating off-design conditions, where secondary flow effects start playing a significant role. A more reliable approach would be to carry out a full transient simulation with sliding mesh interface, taking all the non-uniformities and relative rotor-stator motion into account, significantly increasing time needed for the simulation.

Depending on the aim of performing the simulation, using full transient simulation could be the most feasible type of analysis. Nonetheless, some simulations need to provide quick results. It may be relevant for preliminary simulations, providing a rough estimate of the performance (e.g. by developing complex devices with multiphysics simulations needed). It also may be related to optimization, where resolution of multidimensional search field would be directly related to simulation speed.

Current paper concentrates on providing comparison of simulation results using three types of simulations: transient simulation with sliding mesh interface and two types of stationary simulation: "Frozen Rotor" simulation, representing all six blades in a fixed position and "Stage" simulations with simulating only one of them. Conclusions on possible applicability are formulated at the end in the discussion section.

2 Methods

2.1 Investigated unit

A six-bladed radial pump ($n_q = 35$ min^{-1}) with volute casing, balancing holes for reducing axial thrust and sealing gaps on both sides of the impeller was used for current investigation (see table 1 and figure 1). Measurements of the head curve were performed on a closed-loop test rig shown in figure 2, including a magnetic inductive flow measurement device, frequency converter and two pressure measurement taps (before and after the pump).

Table 1. Main pump data (datasheet).

Parameter	Value
Head	47 m

Parameter	Value
Flow rate	650,7 m³/h
Rotational speed	1450 rpm
Motor power	160 kW
Impeller diameter	404 mm

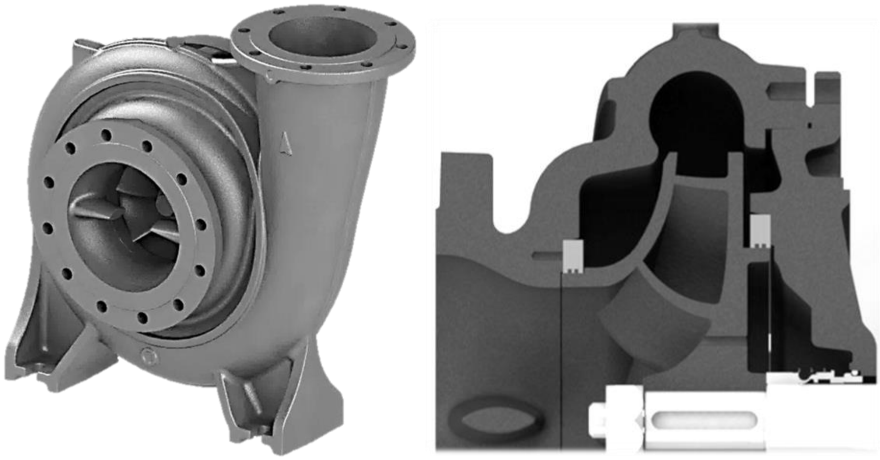

Fig. 1. Pump overview (CAD, rendered).

Inlet pressure measurement was made at a distance of $15D_0$ from the pump inlet to eliminate the influence of the pre-rotation. Head losses for this distance were estimated with around 0,5 % for the BEP (less for part load regimes) and were neglected. Measured data was normalized using affinity laws to get performance curves for constant rotation speed (1480 rpm).

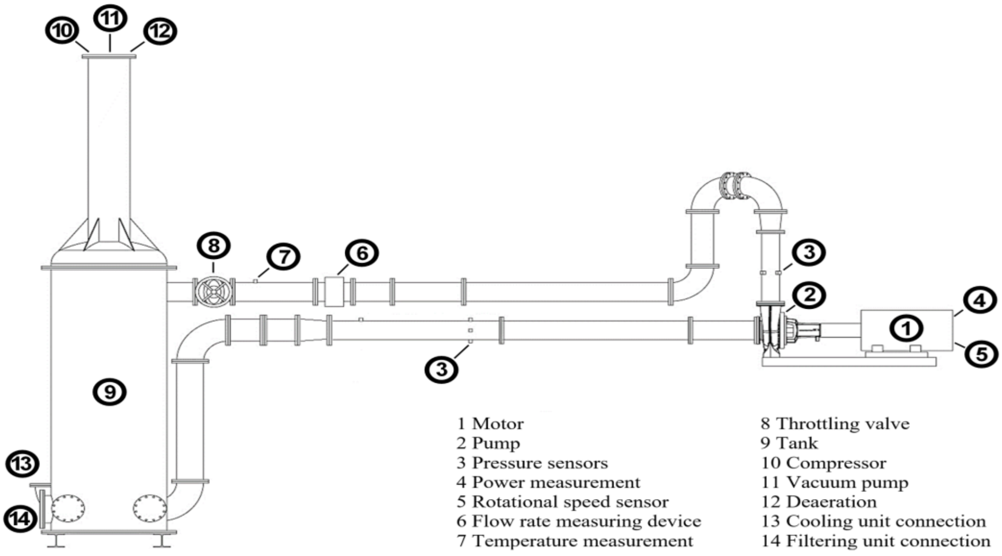

Fig. 2. Overview of the test rig.

2.2 CFD Setup

CFD Simulations were performed in ANSYS CFX 15.0. General setup for CFD simulations is shown in figure 3. A basic representation is shown, which was used for steady-state simulation with mixing plane interface. Position on inlet and outlet boundary conditions were chosen according to position of pressure measurement taps, used in the experimental measurements. Sidewall gaps were considered in the simulations and were included to the impeller domain. Seal gap zones were also considered (not shown in the figure 3), leakage flow was treated as axisymmetric by simulating a thin sector thereof (0,25°). Interface between impeller and volute casing was located roughly at the middle radius between impeller outer radius and radius of volute cutwater.

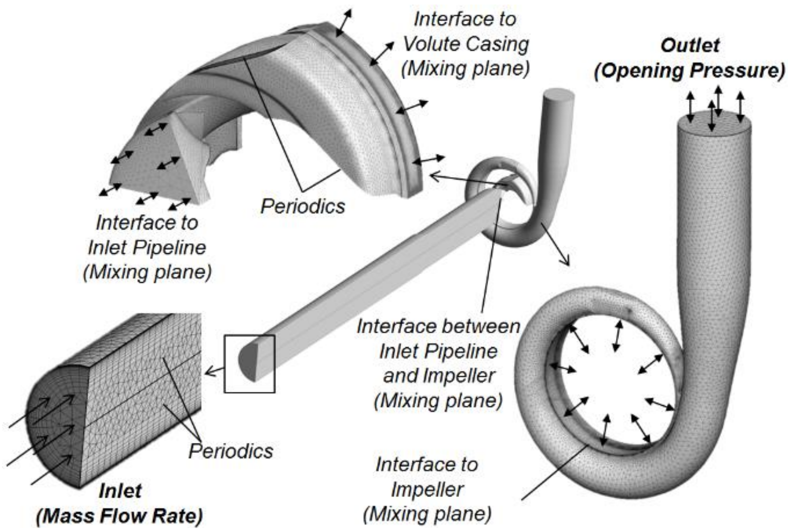

Fig. 3. CFD setup (used for mixing plane simulations).

Domains for impeller and inlet pipeline, shown above, were then multiplied to represent the whole 360° of the volume and used for Frozen Rotor and Transient simulations. Tetrahedral meshes with prism layers in the boundary layer were used (total number of elements: 4,5 mln for mixing plane simulations, 17 mln for Frozen Rotor and Transient simulations). Frozen Rotor simulations were performed for a particular rotor position, and the effects of rotational offset were not investigated (multiple Frozen Rotor simulations would remove most of the computational advantage of frozen rotor stationary simulations over one Transient simulation). Turbulence was modelled with RANS k-ω SST model; resulting y^+ values did not exceed $y^+_{max} = 3$. Time step for the transient simulations was corresponding to the rotation angle of 0,5°. Frozen Rotor simulations were used as initialization for Transient simulations.

3 Results

3.1 Performance curves

A comparison of integral parameters (e.g. head and power consumption) provides first insight on whether the applied model settings are plausible for the area of interest [1, 2, 4]. Current research was concentrated on part load regimes and therefore most of the data was collected only for the part load operation. Comparison of performance curves is shown on figure 4 and figure 5.

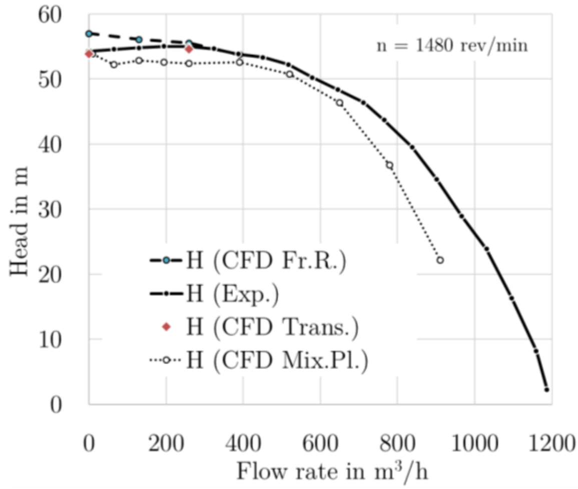

Fig. 4. Head curves (measured and simulated).

Mixing plane results provided relatively good convergence to measured values for design point (650 m³/h) and slightly in the part load (down to 400 m³/h). Direct power curves comparison was not possible due to measuring only the electric power (total), but the qualitative distribution is well represented for the whole part load range. The differences in predicted head start showing up between 0 and 400 m³/h. Mixing Plane simulations showed underprediction of head, while Frozen Rotor, on the opposite, overestimated head production. Transient simulations (performed for the operating points 0 and 260 m³/h), as expected, provided the best convergence with the experimentally measured head curves. Worth mentioning is better convergence of the Mixing Plane results to the measured curve for shut-off, while Frozen Rotor worked better for intermediate point at 260 m³/h.

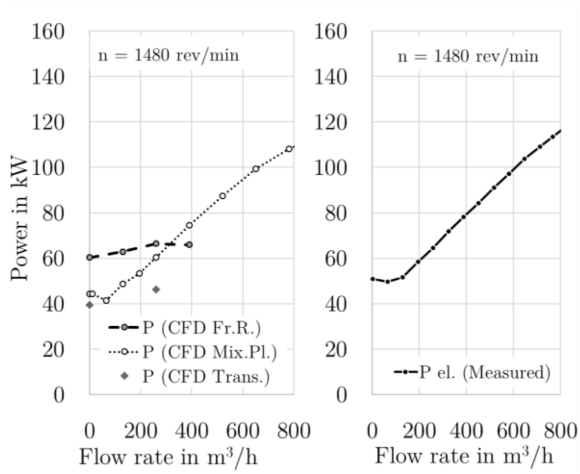

Fig. 5. Power curves (measured and simulated).

Further analysis will be concentrated on two operating points – 0 m³/h and 260 m³/h, using transient simulation results as reference for the two types of stationary simulation.

3.2 Local parameters

Possible explanation to mentioned differences could be found by analyzing local velocity distributions and comparing them between simulation types. It is pointed out in numerous sources, that head production at part load could be strongly influenced by recirculation, taking place at inlet and outlet of the impeller [4, 5, 6, and 7]. A more comprehensive investigation for influence of the pressure-side recirculation for this particular pump was published in [8].

By operating at strong part load, flow non-uniformities in the volute are maximized. Therefore, impeller channels encounter varying conditions depending on their position relative to the volute cutwater. Velocity distribution in volute is shown on figure 6, where velocities are plotted for a plane located in the middle of the impeller outlet width, based on the results of transient simulation.

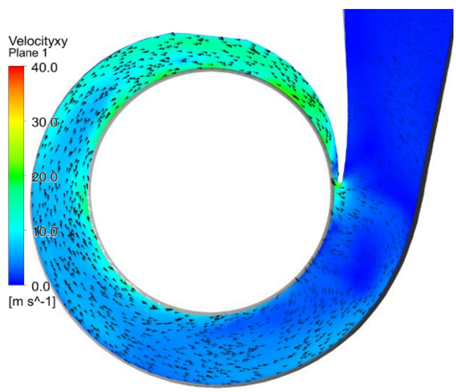

Fig. 6. Velocity contour at shut-off (CFD, transient).

As it can be seen, highest circumferential velocities are concentrated in the areas after passing the cutwater edge, being decelerated by moving towards the volute outlet, and forming a strong backflow zone right before passing the cutwater edge. Recirculating structures, formed at the impeller outlet, are thereby influenced by constantly changing conditions in the volute due to impeller rotation. This constant alternation of external conditions can be successfully captured by using sliding mesh, and that is assumed to be the reason for the improvement of transient simulations in comparison to both steady-state approaches.

A comparison between radial and circumferential velocity contours on interface surfaces is shown on figure 7 and figure 8 for shut-off regime. These contours help describing, respectively, the quantitative and energetic characteristics of fluid leaving and flowing back to impeller vanes. Zero angle position corresponds to the position of cutwater tongue. Rotor outlet is located in the range between −20 and 20 mm on the Z-axis (which can clearly be recognized by zones with increased radial and circumferential velocities).

Contours for transient simulations would be the most reliable representation of the flow features. Zones of intensified recirculation could be identified in areas approaching the cutwater tongue due to reciprocal interaction with the volute space (higher pressures in the volute provide conditions for intensified backflow to the impeller channel, leading to stronger recirculation and increased pressure build-up). Structures with even higher intensity were found in the results of Frozen Rotor simulation (based on both radial and circumferential velocities). This approach still represents the non-uniformities in the volute casing leading to uneven distribution between impeller channels. However, these differences to the transient simulation are believed to take place due to the lack of "virtual inertia" of fluid. Vanes located closer to the volute outlet are in this case "virtually" acting against constantly increased pressure, providing the conditions to build excessive recirculation thereby (only a fraction of this intensification could normally be observed by operating in changing pressure conditions, as it was represented in transient simulations).

While contours on both sides of the interface are identical for Frozen Rotor and Transient simulations (only impeller side is shown therefore), a transformation is performed in the case of stage simulation (circumferential "mixing"). Backflow zones are still represented for the shut-off regime, but they are evened out on the volute side in the simulation for 260 m³/h (see figure 9 and figure 10), leaving only a relatively small back flow zone right before the cutwater tongue.

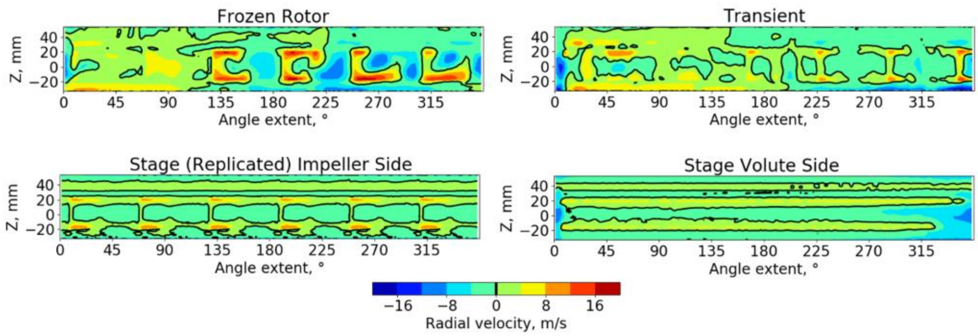

Fig. 7. Radial velocity contours on interface surfaces at shut-off (flattened).

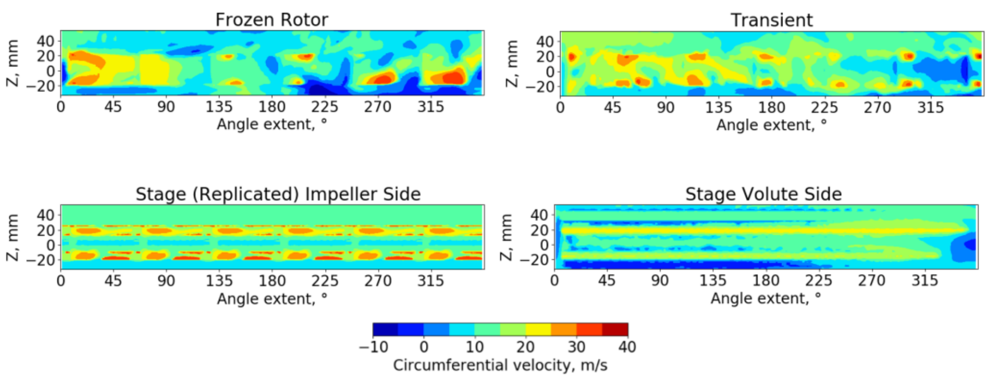

Fig. 8. Circumferential velocity contours on interface surfaces at shut-off (flattened).

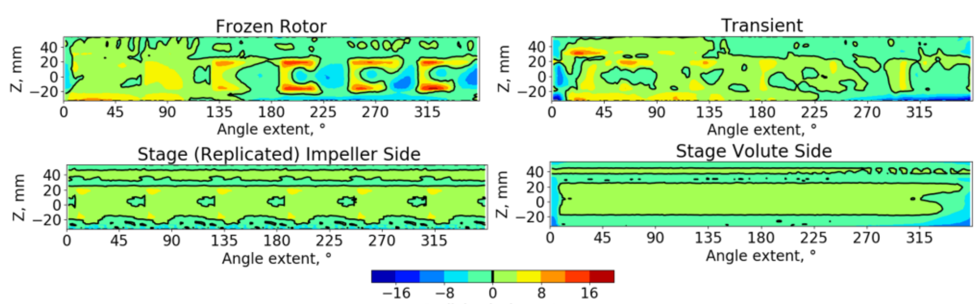

Fig. 9. Radial velocity contours on interface surfaces at 260 m³/h (flattened).

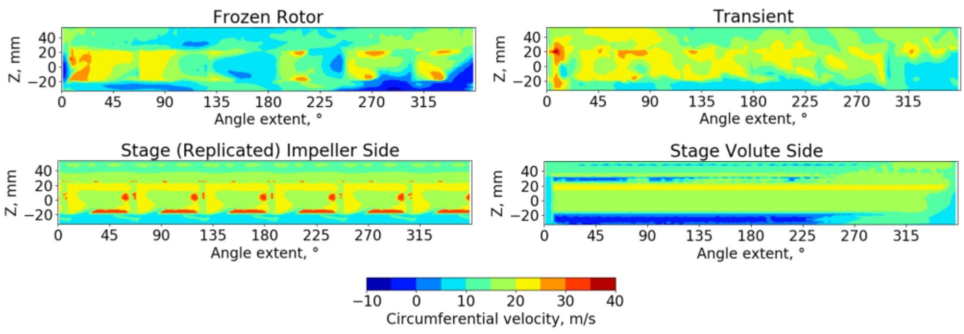

Fig. 10. Circumferential velocity contours on interface surfaces at 260 m³/h (flattened).

4 Discussion

A correlation between correct representation of impeller outlet recirculation and result correctness could be found for all described cases. Misrepresentation of the recirculating structures is thereby believed to be the driving factor for the deviations in head and power prediction.

Good results were provided by Mixing plane model at shut-off, having been able to reproduce a certain amount of recirculation. At the same time, equalizing the outflow from the impeller and virtually reducing recirculation intensity at 260 m³/h (which can be seen on both radial and circumferential velocity profiles on figure 9 and figure 10) was related to underpredicting the head.

5 Conclusion

It is thereby hard to make a solid conclusion regarding applicability of Frozen Rotor model for part load simulations. Considering the described results, it may deliver suitable results by certain conditions, but the reliability of such results would be low for strong part load (especially taking into account the uncertainty brought by its dependence on impeller rotational position in the CAD model).

Overprediction of flow non-uniformities in the case of Frozen Rotor simulation (shut-off) was related to overpredicted shut-off head and power. The simulation for 260 m³/h, though having zones with overpredicted radial velocities, had shown correct results in the view of head prediction due to correct representation of circumferential velocities.

Simulation model with Stage interface was shown being able to deliver a good estimation for shut-off performance (head and power), and it can therefore be considered as a reasonable trade-off between computational time and reliability of results not only for design point, but also for zero flow rate. However, transient simulations with sliding mesh are still recommended for the most comprehensive analysis of flow features and pump performance.

References

1. F. Pochylý, Haluza M., Fialová S., Dobšáková L, Volkov A. V., Parygin A. G., Naumov A v., Vikhlyantsev A A and Druzhinin A A, Application of heterogeneous blading systems is the way for improving efficiency of centrifugal energy pumps *Thermal Engineering* **64,** 2017, 794–801
2. A. V. Volkov, Parygin A. G., Vikhlyantsev A. A., Druzhinin A. A., Grigoriev S. V. and Sobolev G. V. 2018 Verification of approaches of optimal control theory for the

case study of low emission high head pump for petroleum and chemical industry *International Journal of Mechanical Engineering and Technology* **9** 1206–15

3. A. G. Parygin, Vikhlyantsev A. A., Volkov A. V., Druzhinin A. A. and Naumov A. V. 2018 An analytical method for predicting hydraulic head losses in the outlet of centrifugal pump *International Journal of Mechanical Engineering and Technology* **9** 1228–39
4. J. F. Gülich, 2020 *Centrifugal pumps* (Villeneuve, Switzerland: Springer International Publishing)
5. S. Yedidiah, Centrifugal Pump User's Guidebook (Springer US), 1996
6. I. J. Karassik, Messina J. P., Cooper P. and Heald C. C., Pump handbook, 3rd ed. (New York: McGraw-Hill), 2001
7. P. Hergt and Starke J, Flow Patterns Causing Instabilities in the Performance Curves of Centrifugal Pumps with Vaned Diffusers *Proceedings of the Second International Pump Symposium*, 1985, pp 67-75
8. E. Palamarchuk, Mengdehl T, Zharkovsky A and Thamsen P U 2019 Influence of the impeller outlet recirculation on the head curve instability *4th International Rotating Equipment Conference 2019*, Wiesbaden, Germany