Simulate the Reality to Improve the Pump Efficiency

Flow Structures and Loss Mechanisms in Side Channel Pumps

***B. Torner, F.-H. Wurm**, Institute of Turbomachinery, University of Rostock*

In Short

- Scale-resolving flow simulations in a side channel pump
- Detailed investigation of the unsteady flow structures and the loss mechanisms at different flow rates
- Based on the findings in this project: new design optimizations, which will increase the pump efficiency

Side channel pumps are an unique type of turbomachinery that can generate a high pressure build-up at a low flow rate. These pumps are self-priming and can handle multi-phase flow. A side channel pump consists typically of multiple stages. The design of a single stage and a typical path of a fluid particle can be seen in Figure 1.
Application fields of side channel pumps are the marine and the automotive industry, food processing and plants in the chemical industry. The use of a side channel pump is often without alternatives in many applications due to the mentioned advantages. Nevertheless, the strongest disadvantage of a side channel pump is an extremely low efficiency (often below 30%).

A reason for this low efficiency is that the loss mechanisms in the pump are not very well understood. The flow in the side channel pump is highly unsteady, three-dimensional and turbulent. The standard way to investigate such kind of flows and the connected loss mechanisms is the use of computational fluid dynamics (CFD). Nonetheless, the use of CFD methods for simulating side channel pumps is very challenging due to the very complex geometry and the strong time-dependency of the flow field, which makes an unsteady flow simulation with small time-steps necessary.
Although, there is some effort in the current literature to compute the side channel pump flow with scale-resolving simulations (in which all or a part of the turbulence spectrum is resolved), the understanding of the unsteady flow field and the associated loss mechanism have not been analyzed so far.
Therefore, the objective of this research project is to analyze the unsteady flow structures and the loss mechanisms in a side channel pump in detail. Flow simulations will be carried out to analyze a typical side channel pump at different flow rates. A scale-resolving Stress-Blended-Eddy-Simulation (SBES) model will be used to capture the unsteady (turbulent) flow structures in the pump. The SBES model needs a fine computational grid and small time-step length to resolve the turbulent flow structures in the pump. Therefore, the computational resources required for a simulation are very high, making it necessary to use the high-performance cluster HLRN.

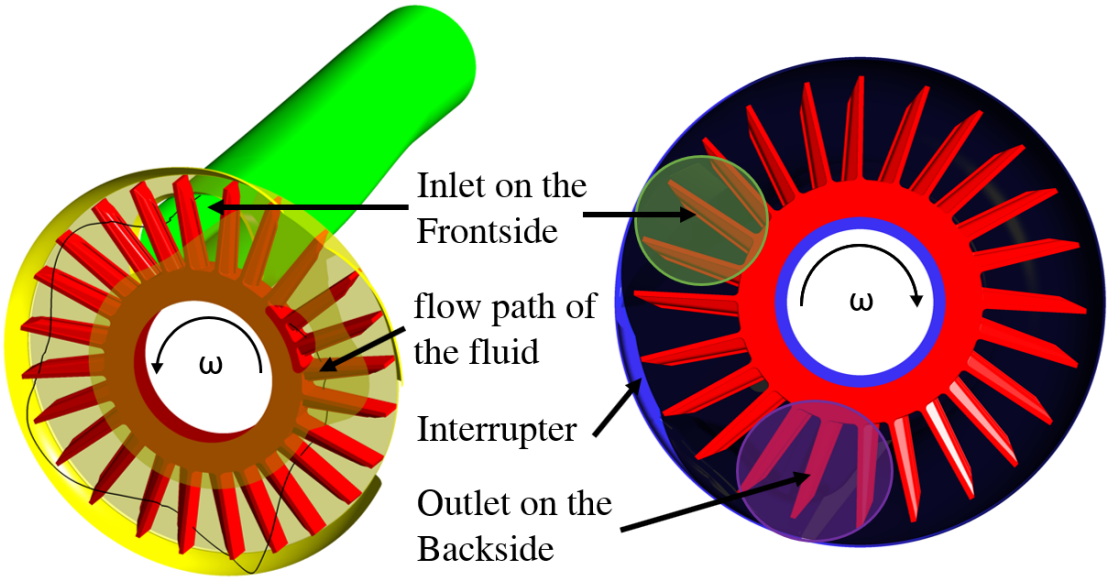

***Figure 1:** Flow path of a fluid particle through the impeller of the side channel pump (left). Axial sectional view of the impeller (right).*

The resulting flow resolution will be unique for simulations in side channel pumps, as such detailed grid and time-step resolution has not been achieved in the literature before.
Besides the typical evaluation methods, new procedures will be adapted for the evaluation of the side channel pump flow:

- Tracking of fluid particles is used for a comprehensive understanding of the pump operating principle to deduce how exactly the pressure build-up is realized.
- The Proper Orthogonal Decomposition (POD) method is used to identify periodic coherent vortex structures in the flow field to understand the turbulent pump flow more in detail.
- The Power Loss Analysis (PLA) with a phase-averaged flow field will show the exact regions of flow losses at different impeller positions.

With an increased understanding of unsteady flow and loss mechanisms in side channel pumps, optimization strategies can be developed and more efficient side channel pumps will be designed.

WWW

https://www.itu.uni-rostock.de

More Information

[1] A. Lass, F.-H. Wurm, *Proceed 3rd Int Rotat Equip Conf* (2019).

[2] M. Mosshammer, H. Benigi, H. Jaberg, J. Konrad, *Int J Turbomach Prop Power* **42(2)**, 6 (2019).

[3] Y. Wang, F. Zhang, S. Yuan, K. Chen, X. Wei, D. Appiah *ASME J. Fluids Eng.* **142**, 061503 (2020).

[4] R. Liu, F. Zhang, K. Chen, Y. Wang, S. Yuan, R. Xu *Processes* **10**, 1630 (2022).

[5] for information regarding POD see: M. Witte, B. Torner, F.-H. Wurm *ASME Turbo Expo 2018* (2018).

[6] for information regarding PLA see: B. Torner, L. Konnigk, N. Abroug, F.-H. Wurm *Int J Numer Method Biomed Eng* (2020).

DFG Subject Area

404-04