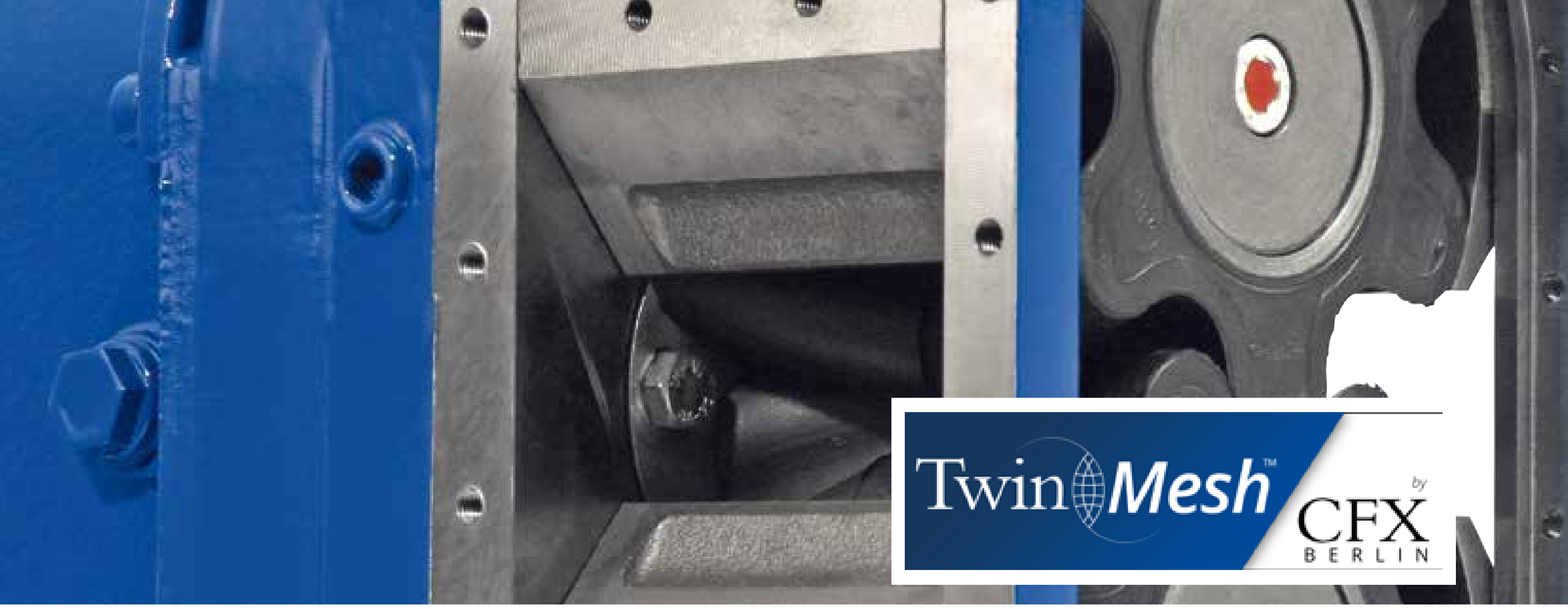

3D-Simulation Enhances Product Development for Positive Displacement Machines

Introduction

Positive displacement (PD) machines, as shown in Fig. 1, are essential devices in many industrial plants and processes: rotary-screw compressors, either oil-free or oil-injected, are used as gas compressors and superchargers. Scroll compressors are used for heat pumps and air conditioning systems. Moreover, both can also be used as expanders. Rotary vane pumps are used in low-pressure gas applications and even as vacuum pumps. Lobe pumps and eccentric screw pumps are used in the paper, chemical, food, waste-water, and pharmaceutical industries since they can handle liquids, solids, and slurries. Gear pumps, designed as internal gear pumps, external gear pumps, or gerotor pumps, are common for hydraulic power applications or to pump high-viscosity fluids.

Fig. 1 - Different PD machine types (from left to right): lobe pump, screw compressor, internal gear pump, roots blower. Photos: www.twinmesh.com

3D Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) simulations have the potential to become a key technology in the development process of positive displacement machines. CFD can deliver deep insights into the flow and thermodynamic behavior of PD machines. Although CFD is broadly accepted in academia as well as in industry, it is currently not used as much for PD machines as for other related applications such as axial-flow pumps, centrifugal pumps, or axial compressors. This is because the main challenge in simulating PD machines lies in the fact that the fluid region is subdivided into chambers that transport the fluid. These chambers change in position and shape due to the motion of the rotors. The transport of fluids in the chambers between the rotors, and between the rotors and the casing, requires complex meshes that change within a single computation. Additionally, the reliable identification of losses due to leakage flows, the heat transfer to the rotors, and multiphase effects like cavitation in pumps, or oil injection in compressors, need high spatial-resolution and the meshes need high numerical quality. Therefore, the crucial tasks for robust and reliable CFD simulations of PD machines are to accurately generate the grid to guarantee correct representation of the chamber volumes, and to create sufficient mesh resolution especially within the gaps and in their vicinity.

Meshing Techniques for CFD of PD machines

There are a few approaches that have been investigated over the past few years. They involve automated and hand-made generation of unstructured and structured grids. The most common approach is probably the use

Summary of the disadvantages of using the "common" approaches:

- Overlapping meshes: fast but insufficient wall treatment / gap resolution, large models, issues with compressible fluids / limited multiphase.
- Deform-and-remesh method: automatic mesh generation but produces large models, interpolation errors, and stability issues.
- Customized grid generation: best mesh and numerical quality but inefficient due to time-consuming manual mesh generation.

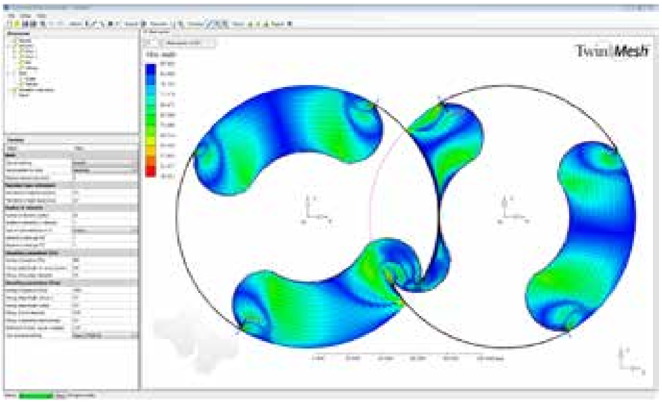

Fig. 2 - TwinMesh GUI showing the mesh quality (min. angle) of a rotary piston pump.

of overlapping meshes (immersed solid method and overset meshes). Other approaches are the deform-and-remesh method and customized grid generation. All these methods have more disadvantages than advantages, as has been summarized in the box to the right.

A Novel Meshing Approach

Consequently, CFX Berlin Software GmbH developed a meshing software (see Fig. 2) to automatically pre-generate high-quality hexahedral grids in a reasonably short time before running a simulation. Hexahedral grids allow the use of all the features and physical models offered by common general-purpose CFD solvers. Hexahedral meshes also permit the best compromise in terms of sufficient spatial-mesh resolution compared to the total number of nodes and elements. Furthermore, the use of block-structured hexa meshes allows engineers to share the same mesh topology for every computational time-step and, therefore, to avoid interpolation between meshes during a simulation. As a result, efficient CFD models with reasonable calculation times can now be generated for rotary PD machines and so CFD simulation can now be utilized on a day-to-day basis in industrial applications.

The overall workflow of the model setup is fairly automated. Grid generation starts with the import of the rotor's profile curves from the existing CAD geometry and ends with a ready-to-start simulation setup when exporting the 3D grids for all computational time-steps. The software allows the user to define the resolution of the boundary layer near walls, and the resolution of the hex grids of the working chambers and the clearances, with local

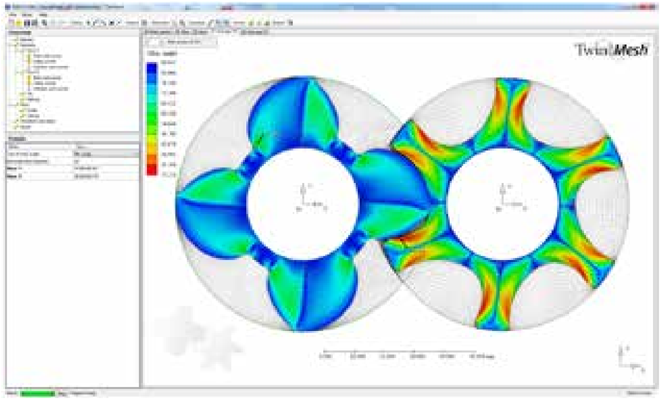

Fig. 3 - Axial gap mesh quality (min angle) of a screw compressor.

refinements. The quality of the elements, e.g. smallest element angle (min angle), aspect ratio of the edges, or volume ratio between connected elements can be visualized and checked, as shown in Fig. 3, before the simulation is started, which ensures both good convergence behavior of the CFD simulation and high-quality results. During run-time, the CFD solver reads the particular grid at the beginning of a time-step. Since all grids have the same topology no interpolation is necessary, and the grid deformation has already been taken into account in the discretization of the fluid system's partial differential equations. Radial and axial gaps are fully resolved so that leakage flows can also be simulated in full detail.

SE-51.2 Twin Screw Expander Example

To provide a real-world example, a brief discussion of Dortmund University's simulation of the screw expander SE-51.2 will be presented.

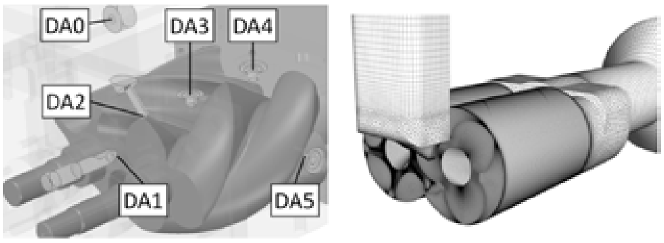

Fig. 4 - CAD model of the SE-51.2 screw expander with pressure measurement points (left) and assembly of rotary and stationary mesh regions (right).

The expander was simulated for several rotational speeds between 1,000 and 16,000 rpm to achieve a pressure ratio of 4:1 with an air-inflow temperature of 90°C. Some variations of the numerical model were performed to achieve the reference operating point of 4,000 rpm. To account for geometric variations, one simulation with a 25% increase in radial clearances (housing gap, intermesh clearance and blow hole) and one simulation with a 25% increase in axial clearances (front gaps) were carried out. A simulation with a decreased mesh-resolution was also performed, decreasing the total number of elements from about 920,000 to approximately 52,000 per rotor chamber. Rotor meshes were coarsened in the radial, axial, and circumferential areas, whereas the stator grids were only coarsened in the interface regions. The simulation results were compared to experimental data gathered from various pressure sensors, as shown in Fig. 4.

Leakage modelling requires high-quality meshes at high resolutions because coarse meshes tend to result in decreased accuracy in geometric approximation, particularly for the intermesh clearances of rotary PD machines. A comparison of the SE-51.2 twin screw expander's CFD simulation results with the experimental data indicated that the expander's working mechanism and its flow conditions were well captured by the simulation. The smaller pressure-drop measured in the experiment at DA4 was caused by the rotor bearing and represented an additional rotor chamber inflow, which was not modelled in the simulation. Apart from that, the examined flow quantities were in good agreement with the measurements overall, as can be seen in Fig. 5.

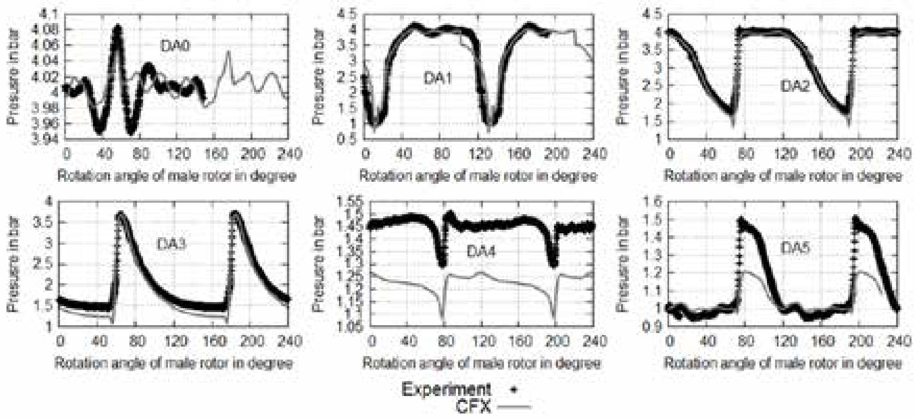

Fig. 5 - Measured and calculated pressure at measuring points DA0 to DA5 at 4,000 rpm.

The over-estimation of the mass flow (see Fig. 6) that was derived from the coarsened rotor meshes could have mainly been caused by an insufficient spatial resolution of the clearances. In order to account for effects like throttling, chamber refilling, or leakage flows, the computation, therefore, has to have a certain mesh quality. Once this is ensured, CFD easily allows tendencies like the variation of clearances to be analyzed, or the operation point to be changed.

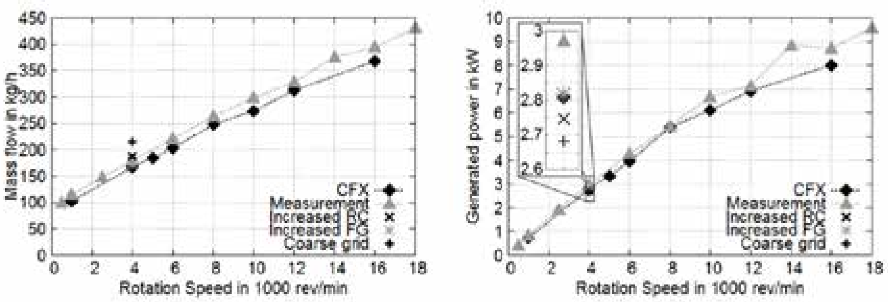

Fig. 6 - Comparison between CFD and measurements: Mass flow (left) and generated power (right) for different operation points with increased radial clearance (RC) and front gaps (FG) grids.

By offering a unique and efficient workflow and eliminating traditional numerical limits, the automated hexa-grid generation method drives real innovation in the field of PD machines, helping engineers to realize a faster time-to-market, achieve higher machine quality, and decrease development costs. Fluid dynamic and thermal analysis for PD machines has never been more intuitive and accurate.

Summary

Internationally-renowned manufacturers such as Gardner Denver, BOSCH, DESMI, Sullair, Danfoss, Nidec GPM, Eaton, Scherzinger, Hanbell, Fusheng, Vogelsang, Börger and others have already adopted CFD simulations for their product development.
Combining the right meshing approach with a reliable CFD solver offers engineers a unique, efficient, and fast workflow for transient, three-dimensional computational fluid and thermal analyses of PD machines with maximum reliability. By using the automated hexa-grid generation method in CFD for PD machines, any engineer can:

- Analyze the flow behavior inside the working chambers of: rotary screw compressors and expanders, internal and external gear pumps, gerotor pumps and orbital motors, lobe pumps and roots blowers, rotary piston pumps, scroll compressors and expanders, conical rotor pumps, progressive cavity pumps, vane pumps, and Wankel engines;
- Analyze complex phenomena such as compressible flow behavior, cavitation, viscous heating, acoustics, and pulsation behavior
- Predict machine performance (mass flow, efficiency, pressure ratio, torques etc.) with no need for physical prototyping

Further Reading and Information

Andres, R., Nikolov, A., Bruemmer, A.: CFD Simulation of a Twin Screw Expander Including Leakage Flows. In 23rd International Compressor Engineering Conference at Purdue, July 11-14, 2016.

Spille-Kohoff, A.; Hesse, J.; El Shorbagy, A.: CFD simulation of a screw compressor including leakage flows and rotor heating. In IOP Conference Series: Materials Science and Engineering (Vol. 90, No. 1, p. 012009).
www.cfd-online.com/Wiki/TwinMesh. www.twinmesh.com

Willie, J., Sachs, R.: Structural and torsional vibration analysis of a dry screw compressor. IOP Conference Series: IOP Conference Series: Materials Science and Engineering (Vol. 90, No. 1, p. 012005).