



Technical Paper

CFD Technology

Session 6-1

Computational Fluid Dynamics simulation of reciprocating pumps with respect to the fluid driven valve motion

Author:

Dipl.-Ing. Stefan Blendinger
Scientific employee
Institute for Process Technology and Machinery, University Erlangen-Nuremberg
91058 Erlangen, Germany

Co-Author 1:

Prof. Dr.-Ing. Eberhard Schlücker
Head of institute
Institute for Process Technology and Machinery, University Erlangen-Nuremberg
91058 Erlangen, Germany

Co-Author 2:

Dipl.-Ing. Oliver Schade
Scientific employee
Institute for Process Technology and Machinery, University Erlangen-Nuremberg
91058 Erlangen, Germany



Pump Users
International Forum 2008

Summary

To improve the future development and optimization of reciprocating pumps, one has to achieve a thorough understanding of the fluid-dynamic processes and interactions which take place in this sort of pump. The results from computational fluid dynamics (CFD) enable us to visualize and explain many phenomena that have either only been described in theory so far or were hitherto unknown.

The necessary steps to perform a CFD-analysis are sketched, including the basics of a fluid driven valve motion. Some results of a full CFD simulation of a reciprocating pump with respect to fluid driven valves are presented. Beside this, a simulation of static geometries (no moving parts) with variable flow boundaries constitutes a comparatively easy and faster approach on a geometric optimization of the working chamber and the valves. In an additional example a CFD based analysis and optimization of a standard plate valve is shown.