

R & D activity of pumps using CFD

Akira GOTO

Fluid Engineering Laboratory Center for Technology Development

Ebara Research Co., Ltd.

Abstract

The actual situation of application of Computational Fluid Dynamics (CFD) to pump R&D in industries will be briefly reviewed. The talk will include (1) CFD code assessment, (2) CFD application for complex geometry, and (3) CFD as a design tool. As regards CFD code assessment, three different approaches will be presented; one based on prediction of overall performance of diffuser pump stages, one based on multi-color oil-film flow visualization for wall surface streamlines, and one based on internal flow fields measurements. As regards CFD application for complex geometry, automatic grid generation based on CAD data will be presented for a multi-stage pump return channel and a volute casing. The effectiveness of solution adaptive technique in CFD will be also briefly described. And finally, as regarding CFD as a design tool, a new approach to optimizing hydraulic and aerodynamic design of turbomachines will be presented, based on a three-dimensional inverse design method and CFD. The design examples both for pumps and centrifugal compressors will be presented.