6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

Volute Shape Optimization using Evolutionary Algorithms

Martin Heinrich[1], Rüdiger Schwarze[2]

[1] Institute of Mechanics and Fluid Dynamics, TU Bergakademie Freiberg, Lampadiusstraße 4, 09596 Freiberg, Germany, martin.heinrich@imfd.tu-freiberg.de
[2] Institute of Mechanics and Fluid Dynamics, TU Bergakademie Freiberg, Lampadiusstraße 4, 09596 Freiberg, Germany, ruediger.schwarze@imfd.tu-freiberg.de

Key Words: *Optimization, Centrifugal Compressor, Computational Fluid Dynamics*

The volute is an integral part of a centrifugal fan or compressor. It is located downstream the compressor wheel and its purpose is to collect the flow. By doing so, kinetic energy is converted into potential energy and thus the static pressure is increased. The overall performance of a volute is determined by five geometric specifications according to Ayder et al. [1]: (1) area of volute cross-section, (2) shape of the cross-section, (3) radial position of the volute cross-section, (4) location of the volute inlet, and (5) tongue geometry.

In our research, we focus on the optimization of the volute housing using genetic algorithms. An in-house tool for the automated generation of the volute geometry is the key in this workflow. Twelve different variables parameterize the geometry, which can be defined by the user. In the second step, a hexahedral unstructured mesh generator is used to create the mesh for the centrifugal compressor. The flow is solved using the open-source CFD library OpenFOAM. The optimization software Dakota combines all the data and runs the single-objective evolutionary algorithm. Four different operating points are simulated for each geometry, whereby the average of the isentropic efficiency at each operating point is considered as the objective function.

The numerical model consists of a density-based solver for a wide range of Mach numbers using the Simple Low-dissipation AUSM (SLAU) scheme. Spatial discretization is second order and the SST k-ω turbulence model with rotation and curvature correction is used. The Multiple Reference Frame (MRF) approach is employed to model the rotation of the compressor wheel. The numerical model is validated using a centrifugal compressor of a turbocharger for commercial vehicles and passenger cars with a design speed of 100,000 rpm.

The results show, that a volute with an inlet eccentricity of 0.9 and a slightly lower radial volute channel offer the best compressor efficiency (see Figure 2). Moreover, the actual cross-sectional shape of the volute has only a minor influence onto the performance. The cross-sectional area of the volute as a function of the circumferential angle also has a large influence onto the isentropic efficiency. A slightly S-shaped area distribution offers the best results.

REFERENCES

[1] E. Ayder and R. Van den Braembussche, "Experimental Study of the Swirling Flow in the Internal Volute of a Centrifugal Compressor," in *Experimental Study of the Swirling Flow in the Internal Volute of a Centrifugal Compressor*, 1991.