Development of an automated process for turbine blade optimisation

B. Šojat[+], T. Uroić[*,+] and H. Jasak[+,†]

[+] Faculty of Mechanical Engineering and Naval Architecture (FSB), University of Zagreb
Ivana Lučića 5, 10000 Zagreb, Croatia
e-mail: borna.sojat@stud.fsb.hr, tessa.uroic@fsb.hr, hrvoje.jasak@fsb.hr, web page: https://www.fsb.unizg.hr/

[†]Wikki Ltd.,
Unit 459, Southbank House, Black Prince Road, London SE1 7SJ, United Kingdom
e-mail: h.jasak@wikki.co.uk

ABSTRACT

Computational fluid dynamics (CFD) is a critical part of the aerodynamic design and optimisation of turbomachinery components. Benefits of CFD range from faster design cycles to better performance and reduced costs and weight of the final product. There is a continued commercial pressure to produce components of highest possible quality and aerodynamic performance in the shortest possible time. The classical approach to design relies on creative designers who have developed intuition and physical understanding of the subject. CFD is especially useful for helping develop this understanding since many more blade designs, ranges of parameters, etc. can be studied than would ever be possible experimentally.

The blade optimisation process is iterative and consists of the following steps: geometry parametrisation using B-splines, morphing of the computational mesh, numerical calculation of compressible flow through the turbine stage, and obtaining a new set of control points describing the blade geometry. B-spline was used for parametrisation because of its favorable properties: each control point is associated with a unique basis function. The resulting curve does not fit the set of control points exactly, but rather approximates them as close as possible [1]. In this way, discontinuities on the blade surface are prevented. 30 parametrisation points are extracted between each pair of control points, and used for morphing of the mesh. The set of parametrisation points is compared to the set of points describing the starting blade profile and the mesh is then transformed using OpenFOAM dynamic mesh utility to match the new geometry. In this way, the uncertainty of the numerical error resulting from the computational mesh is localised [2]. Lift and drag coefficients are obtained from the numerical calculation using OpenFOAM solver for steady state compressible flow, and used as optimisation criteria. New set of control points is calculated using the evolutionary algorithm provided in Dakota optimisation software and the loop restarts with parametrisation of these points. Both single-objective and multi-objective approaches were examined and compared, as well as the CPU time of the optimisation cycle.

Developed automated optimisation process is robust and effective and has proven to give good results even for very irregular and distorted starting geometries.

REFERENCES

[1] Rogers, D.F. *An Introduction to NURBS With Historical Perspective.* Morgan Kaufmann Publishers (2001).

[2] Jasak, H. *Error analysis and estimation for the Finite Volume Method with applications to fluid flows.* PhD Thesis, Imperial College of Science, Technology and Medicine, London, 1996.