Optimisation of a Turbine Blade with Mesh Morphing

Tessa Uroić[1] and Hrvoje Jasak[1,2]

[1]Faculty of Mechanical Engineering and Naval Architecture, Croatia
[2]Wikki Ltd, United Kingdom
[1]`tessa.uroic@fsb.hr`, [2]`h.jasak@wikki.co.uk`, `hrvoje.jasak@fsb.hr`

Abstract

This work presents an automated optimisation work-flow for turbine blade applications. A large number of turbine blade shapes is considered, thus a large number of corresponding meshes has to be used for CFD calculations. A procedure of geometry variation in the CFD mesh is performed using a mesh morphing tool. Simulation results are presented for four examples of the optimisation with deformed meshes.

1. Introduction

It has been well documented that local mesh resolution and mesh quality is one of principal sources of discretisation error in CFD. For consistent results in optimisation studies it is essential to control the absolute value and distribution of mesh-induced errors, ideally by employing a family of self-similar meshes, *Ollivier-Gooch (2009)*. This is not easily achieved where the automated optimisation loop involves the use of an external or automatic mesh generator.
In this study, a procedure of geometry variation in the CFD mesh is performed using a mesh morphing procedure, where the existing mesh will be automatically deformed to accommodate boundary deformation, *Jasak and Tuković (2007)*.

2. Mesh Morphing Process

Mesh morphing work-flow:

1. New blade points are obtained from the optimisation algorithm;
2. Axial length of the new blade is compared to current mesh;
3. Mesh is extended or shortened in the axial direction in order to correspond to the new blade length;
4. The positions of the new blade points are compared to the current mesh in the tangential direction;
5. Mesh points are moved in the tangential direction in order to correspond to the current configuration.

Internal mesh points are moved using the `tetMotionSolver`, which is already incorporated in FOAM-Extend.

3. Process Automation

The optimisation process consists of the following steps:

- Parametrisation of the blade geometry is done using the proprietary tools agreed with the sponsors of the study;
- Using the mesh morphing utility, the existing mesh is adapted for the new geometry;
- Flow in the blade passage is then simulated using OpenFOAM. Simulation results are used to evaluate optimisation criteria.

4. Preliminary Results

Initial tests of the optimisation workflow were conducted on a standard NACA-63421 blade profile. Geometrical variations are shown in Fig. 1. The initial mesh was made for the original geometry (shown in blue). The mesh was then morphed to comply with new geometries, shown in red. The flow through the blade passage was simulated and the results are shown in Fig. 3. Optimisation criterion is the value of the pressure coefficient at the outlet of the blade passage and the comparison for different blades is shown in Fig. 4.

Figure 1: Blade profile geometry variation

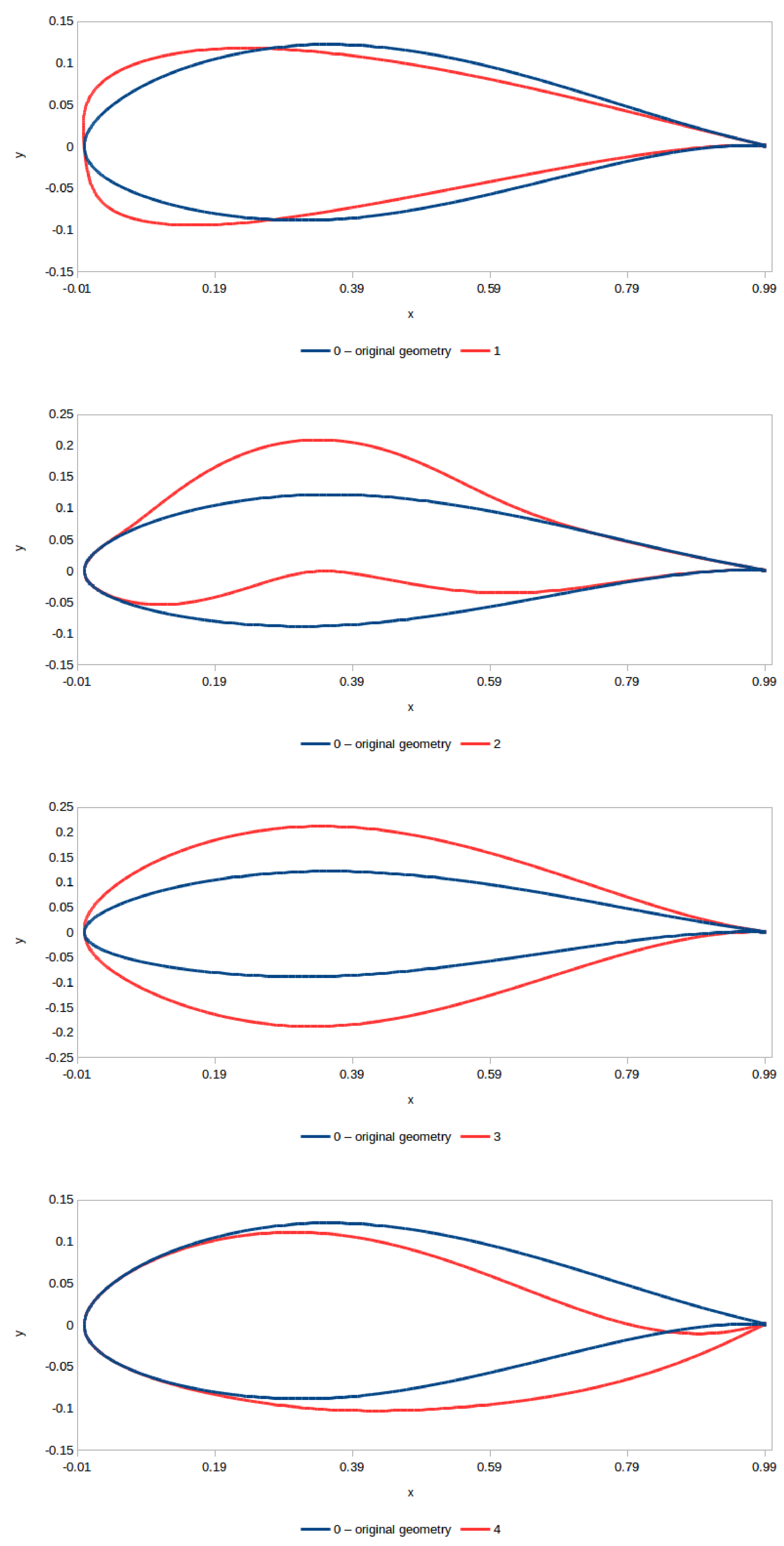


Figure 2: Original mesh and pressure distribution

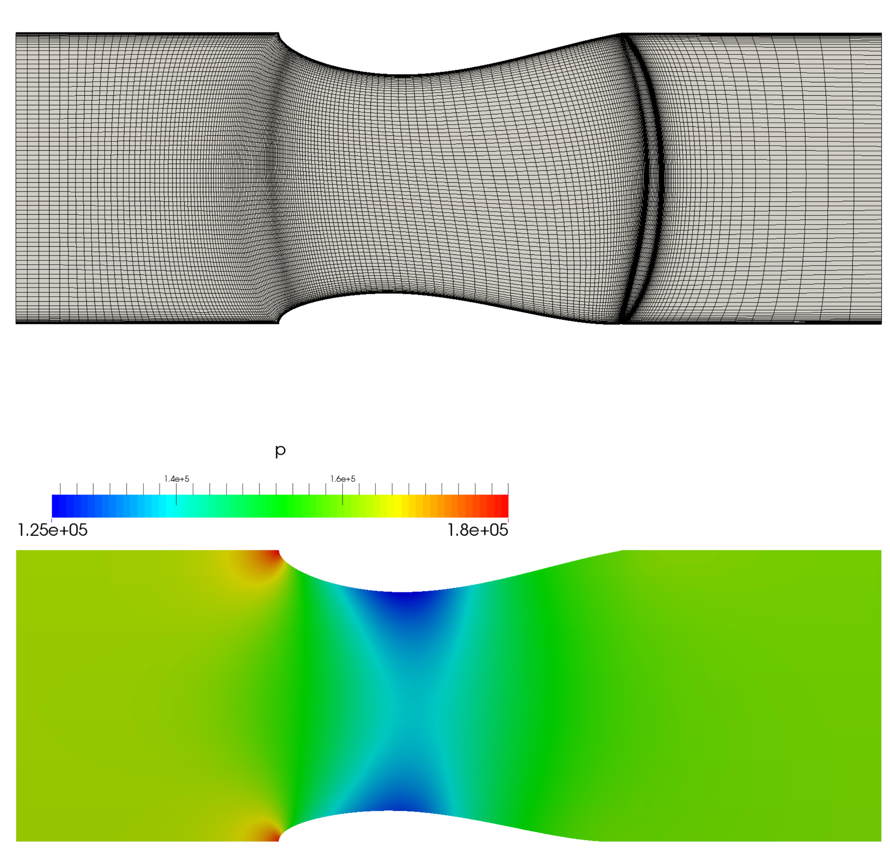


Figure 3: Pressure distribution for corresponding blade profile geometry variations

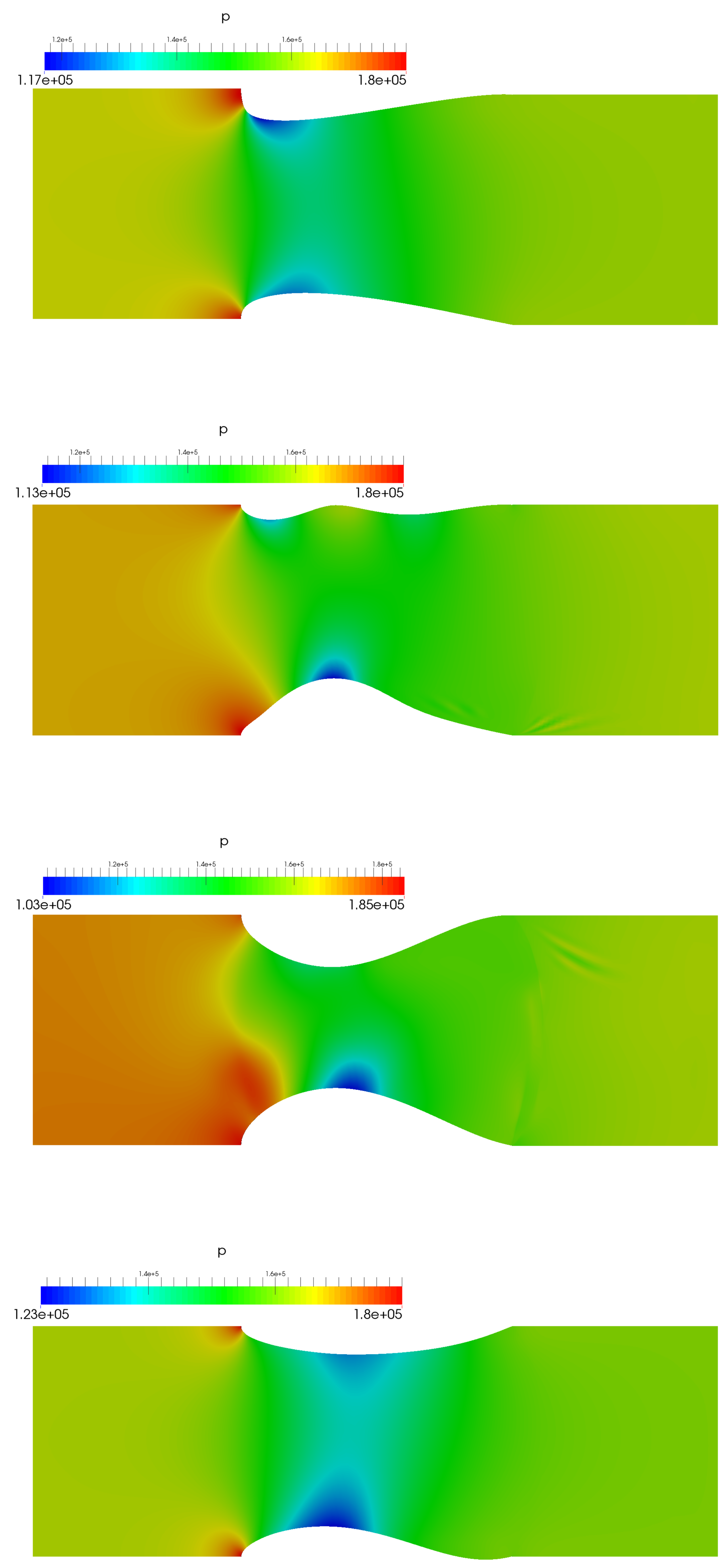
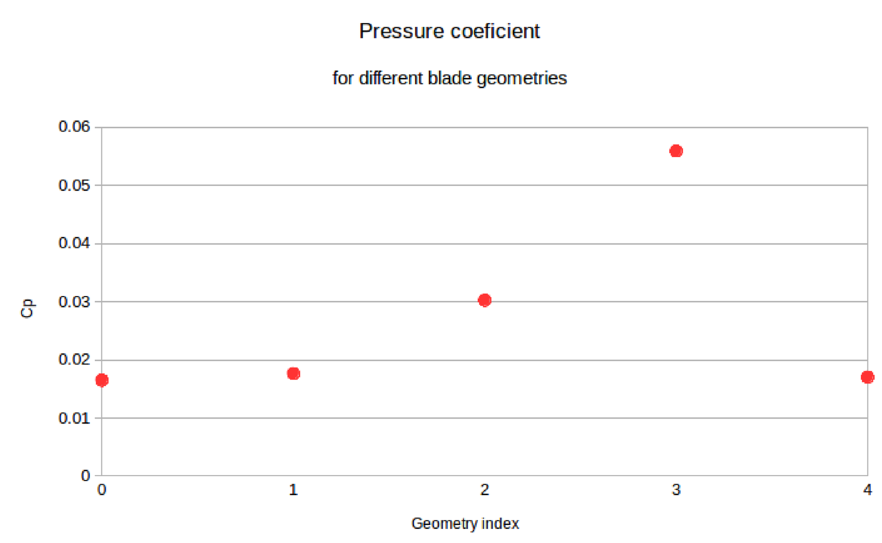


Figure 4: Pressure coefficients for corresponding blade profile geometry variations



5. Conclusion

The automated optimisation process uses mesh morphing which not only simplifies the optimisation loop but also produces a family of self-similar meshes, thus controlling the distribution and magnitude of the discretisation error. Mesh morphing procedure has proven robust and reliable even for large geometrical variations. Furthermore, in an automated process, convergence criteria of the CFD solver should be considered and solutions which have not converged should be eliminated from the optimisation process.

10th OpenFOAM Workshop, June 29–July 2, 2015, University of Michigan, Ann Arbor