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

ANALYSIS OF THE BOUNDARY LAYER STABILITY TO ASSESS FLOW SEPARATION CONTROL CAPABILITY IN LOW-PRESSURE TURBINES

Iván Padilla*[1], Jorge Saavedra[1, 2], Guillermo Paniagua[1, 2] and Fabio Pinna[1]

[1] Aeronautics and Aerospace department, von Karman Institute for Fluid Dynamics,
Chaussée de Waterloo 72, 1640 Rhode-Saint-Genèse, Belgium
ivan.padilla@vki.ac.be; fabio.pinna@vki.ac.be
[2] Zucrow Laboratories, Purdue University,
500 Allison Road, West Lafayette, IN, 47907
saavedra@purdue.edu; gpaniagua@purdue.edu

Key Words: *boundary layer, stability, transition, flow separation, low-pressure turbine, flow control.*

In low-pressure turbine airfoils, the occurrence of flow separation abates significantly the turbine efficiency, which in turn limits the lowest Reynolds number of operation. The use of effective flow separation control strategies can boost turbine performance at low Reynolds numbers [1][2]. Turbulent boundary layers are more resistant to detachment than those in laminar regime, therefore an accurate prediction of transition onset and control is a fundamental way of delaying flow separation. Nowadays, linear stability theory is progressively consolidating as a practical tool to predict transition in real-world applications [3], and can be applied to estimate the optimal flow control location based on a pure theoretical approach, without the need to rely on classical empirical correlations.

In this work we study the flow separation dynamics of a wall-mounted hump geometry like the one depicted in Figure 1. Our analysis examines the behaviour of the flow separation from three different perspectives: unsteady computational fluid dynamics, linear stability theory and wind tunnel experiments. This geometry induces a flow-field that mimics the behaviour of the suction side of a low-pressure turbine airfoil, where boundary layer detachment occurs at low Reynolds numbers while no separation takes place for high Reynolds numbers. The use of a smooth contoured hump, designed using a Bézier curve, does not impose a constraint in the location of the flow separation region and allows it to be fully governed by the flow similarity parameters, introducing a convenient flexibility to test different flow conditions both numerically and experimentally.

The numerical solution of the reference base flow needed for the stability analysis is carried out by means of unsteady RANS simulations using the commercial package CFD++ on a block-structured mesh. Different inflow conditions are tested that lead to freestream Mach numbers in the range of 0.1 to 0.3. The reference flow boundary conditions are represented in Figure 1. The freestream flow values are defined by the imposition of reservoir conditions (total pressure p_0 and total temperature T_0) at the left boundary and the outlet static pressure p at the right boundary. On the other hand, the top and bottom boundaries are treated as no-slip isothermal walls. Figure 1 also shows the streamwise velocity contours and streamlines obtained for a solution with $p_0 = 40600$ Pa, $T_0 = 500$ K, $p = 40000$ Pa and a wall temperature of $T_w = 300$

K. A large recirculation region can be observed behind the hump indicating flow separation.

The stability analysis is performed using the von Karman Institute's Extensible Stability and Transition Analysis (VESTA) toolkit [4] and based on a mean reference flow obtained from averaging the unsteady RANS solution along numerous time steps. The reference flow region prior to flow separation is analysed with one and two-dimensional linear stability theories (also commonly known respectively as LST and BiGlobal) and with linear parabolized stability equations (PSE), which also account for non-parallel effects in the reference flow field [3].

Regarding the experimental study, a test campaign will be performed in a linear blowdown wind tunnel at Purdue Experimental Turbine Aerothermal Laboratory (PETAL). This facility features a full visual access test section with a cross section of 230x170 mm and a length of 550 mm. The flow is discharged to a 283 m^3 vacuum tank that allows sub-atmospheric operation for low Reynolds number testing. It can operate at a wide range of Mach numbers from 0.05 up to 3, unit Reynolds numbers from 10^5 up to 4×10^7 m^{-1} and total temperatures from 280 K up to 700 K. Wall pressure, skin friction and particle image velocimetry (PIV) measurements will be carried out and compared against the theoretical predictions achieved with linear stability theory.

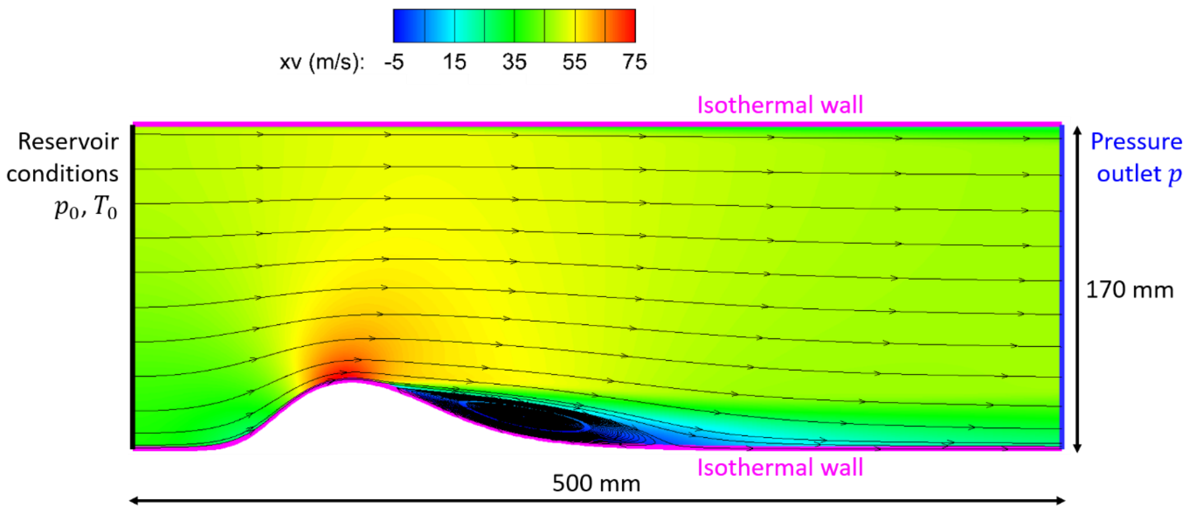

Figure 1: Wall-mounted hump geometry, computational domain and boundary conditions. The contours and streamlines represented correspond to the streamwise velocity field for a numerical solution with $p_0 = 40600$ Pa, $T_0 = 500$ K, $p = 40000$ Pa and a wall temperature of 300 K.

REFERENCES

[1] O. Sharma, "Impact of Reynolds number on low pressure turbine performance", CP-1998-206958 New York, USA. NASA, 1998: 65–70.
[2] H. S. Hura, J. Joseph and D. E. Halstead, "Reynolds number effects in a low pressure turbine", ASME GT2012-68501, 2012.
[3] V. Theofilis, "Global linear instability", Annual Review of Fluid Mechanics 2011, 43:319–352.
[4] F. Pinna, "VESTA toolkit: a software to compute transition and stability of boundary layers", 43rd Fluid Dynamics Conference, 24-27 June 2013, San Diego, California.