11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

NUMERICAL EXPERIMENT OF THE FLOW PAST A T106C TURBINE BLADE USING A DISCONTINUOUS GALERKIN METHOD

K. Hillewaert[1] and C. Carton de Wiart[1]

[1] Cenaero, Rue des frères Wright 29, 6041 Gosselies, Belgium.
corentin.carton@cenaero.be, http://www.cenaero.be

Key words: *Turbomachinery, Discontinuous Galerkin Method, Direct Numerical Simulation*

The T106 turbine blade has extensively been studied for transitional flows, in particular in the presence of passing wakes. Three configurations (A,B and C) have been used, which essentially differ in the pitch. The very high-lift configuration T106C has recently been extensively investigated during the European research project UTAT. Detailed measurements were undertaken at the von Karman Institute by [3] for Reynolds numbers ranging from 80.000 through 250.000 and near-transonic conditions (exit Mach number $M_{2,is} \sim 0.65$). The results were subsequently used to assess transition models by a number of authors on the basis of the distribution of the isentropic Mach number, the kinetic energy loss coefficient ξ, the mass averaged exit flow angle and the non-dimensionalised total pressure deficit in the wake. Although generally good predictions were obtained at higher Reynolds numbers, important discrepancies were still found at the lowest one $Re_{2,is} = 80.000$. At this regime, the separation zone and transitional flow structures in the wake are very large with respect to the geometry and are very intermittent. Therefore, it is quite a challenge to model this flow with RANS, even when complemented by transition models.

As RANS modelling fails to correctly represent this type of low Reynolds flow, even with transitional models, a scale resolving approach is required. For the past few years, an implicit time-integration compressible flow solver based on the discontinuous Galerkin method (DGM) has been developed ans successfully validated for DNS and LES of canonical and industrial flows [1, 2]. During those studies, the accuracy of the method for scale-resolving computations has been highlighted. Even though it can handle unstructured meshes and therefore complex geometries, the solver is extremely scalable and can be run on hundreds of thousand of processors due to the favourable structure of DGM. The T106C configuration at $Re_{2,is} = 80.000$ (Figure 1) is therefore considered as an interesting benchmark for the present method. Also, systematic difference found by several

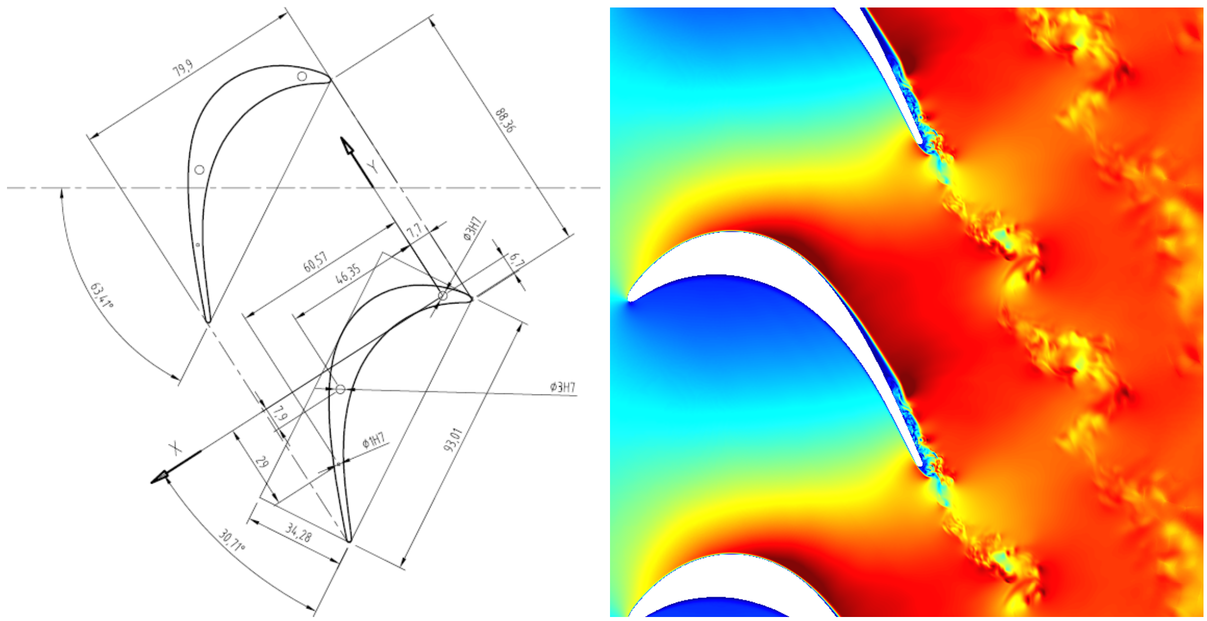

Figure 1: Simulation of the T106C cascade. Left: Cascade characteristics. Right: Snapshot of Mach number field obtained with DGM.

simulations with respect to the experiments can be found in the literature. This leads us to believe there is a mismatch between the experiment and the setup of the computation. The sources of potential errors (the absence of inlet turbulence, too high constraints on spanwise or pitchwise periodicity, etc.) are therefore analysed in this study to understand the reasons of the mismatch between the experiment and the simulations.

REFERENCES

[1] C. Carton de Wiart, K. Hillewaert, G. Winckelmans and L. Bricteux. *Assessment of a discontinuous Galerkin method for the simulation of vortical flows at high Reynolds number.* International Journal of Numerical Methods in Fluids (IJNMF), Early view, doi: 10.1002/fld.3859, 2013.

[2] C. Carton de Wiart and K. Hillewaert. *DNS and ILES of transitional flows around a SD7003 airfoil using a high order Discontinuous Galerkin Method*. Seventh International Conference on Computational Fluid Dynamics, ICCFD7-2012-3604, 2012.

[3] J. Michalek, M. Monaldi and T. Arts, *Aerodynamic performance of a very high lift low pressure turbine airfoil (T106C) at low reynolds and high mach number with effect of free stream turbulence intensity.* Journal of Turbomachinery, 134, 2012.