High-order DG Method for an Implicit LES of a Gas Turbine Cascade

High-order DG Method for an Implicit LES of a Gas Turbine Cascade

Michele Errante[1,a)], Andrea Ferrero[1,b)] and Francesco Larocca[1,c)]

[1]*Department of Mechanical and Aerospace Engineering (DIMEAS)*
Politecnico di Torino, Italy
Corso Duca degli Abruzzi, 24 - 10129 Torino, Italy

[a)]michele.errante@polito.it
[b)]andrea_ferrero@polito.it
[c)]francesco.larocca@polito.it

Abstract. Implicit Large Eddy Simulations (ILES) are performed assessing the performances and the reliability of the discontinuous Galerkin method (DGM) based code developed by the authors. The flow field in the T106C turbine cascade at low Reynolds number is simulated, comparing the results with experimental and numerical data in literature. Very good agreement with the reference data is shown, demonstrating a successfully code optimization.

INTRODUCTION

Although computational power has increased incredibly in recent years, solving turbulent and transitional flows is, and will remain for many years to come, an extremely complicated task. Direct Numerical Simulations (DNS) and Large Eddy Simulations (LES) are still prohibitive for parametric studies at high Reynolds numbers due to the low order of accuracy of state-of-the-art industrial CFD solvers, which involves an high mesh resolution. On the other hand, Reynolds Averaged Navier-Stokes (RANS), which are characterized by a computational cost almost two orders of magnitude lower than DNS and LES, reveal huge limits in presence of transition or separation phenomena.

In the last decade, unstructured finite element high-order methods such as the discontinuous Galerkin method (DGM) have been proven able to combine high accuracy to the flexibility offered by unstructured low quality meshes. The present work aims to validate the reliability of the research code, developed by the aerospace propulsion group of the Department of Mechanical and Aerospace Engineering (DIMEAS) at Politecnico di Torino, based on the DGM in a finite elements framework, using second-order schemes, as the first step in an optimization process leading to the development of higher-order schemes. Implicit Large Eddy Simulations (ILES) have been performed to solve the flow field around the T106C turbine cascade, comparing the results obtained with experimental and numerical data in literature, pointing out analogies and dissimilarities. The validation of such a computational tool will be a key building block for the capability to generate high fidelity data, exploitable in the framework of data-driven turbulence and transition model evolution.

TEST CASE DESCRIPTION

The aim of this study is to investigate, by means of implicit Large Eddy Simulations (ILES), the transitional and separated flow field around the T106C subsonic low pressure turbine (LPT) cascade. It is a well-known test-case for which experimental data have been measured at the Von Karman Institute (VKI) in the framework of the European research projects UTAT and TATMO. In agreement with the measurements undertaken by Michálek, Monaldi, and Arts [1], the isentropic exit Mach $M_{2,is}$ and the isentropic exit Reynolds number $Re_{2,is}$ were set equal to $M_{2,is} = 0.65$ and $Re_{2,is} = 80000$, respectively. The T106C cascade geometry is characterized by a pitch t to chord c ratio equal to $t/c = 0.95$ and an inlet flow angle β_1 equal to 32.7° with respect to the axial direction.

In the following x, y and z will indicate the axial, pitchwise and spanwise directions, respectively. The axial extension of the domain was set at about 4.8 times the chord length c and its spanwise dimension is equal to $0.1c$, where c represents the chord length. Periodic boundary conditions were imposed to connect the faces located at both the pitchwise and spanwise domain boundary.

NUMERICAL METHOD

The simulations were performed by means of a research code, developed by the aerospace propulsion group of the Department of Mechanical and Aerospace Engineering (DIMEAS) at Politecnico di Torino, based on the Discontinuous Galerkin method (DGM) in a finite elements framework. By solving the unsteady, three-dimensional, Navier-Stokes equations without any subgrid scale (SGS) model, the code performs implicit LES (ILES). Hence, the numerical dissipation acts as a SGS model, removing only high wavenumber turbulent structures. The DGM/ILES approach has been successfully validated on decaying homogeneous isotropic turbulence at very high Reynolds number and on the turbulent channel flow (de Wiart *et al.* [2]). The results showed excellent agreement with those of the references and even slightly better performance than state-of-the-art SGS model coupled with pseudo-spectral solvers.

Convective fluxes are discretized by means of an approximate Riemann problem solver implemented according to Pandolfi [3] and the Enhanced Stability Recovery (ESR) approach proposed by Ferrero, Larocca, and Puppo [4] is adopted for computing diffusive fluxes. This approach makes the scheme flexible and allows a robust implementation of p-adaptive algorithms. The basis used by the method has less terms than the original recovery basis (RDG1x) [5] and remains well conditioned also in the presence of higly distorted meshes. Furthermore, the ESR scheme, whose implementation is simplified from the original scheme, shows a larger stability limit than the RDG1x method.

The spatial domain was discretized by the open source curved grid generator Gmsh (Geuzaine and Remacle [6]). Two progressively finer meshes were generated for calculations: the first one, labeled in the following as *"coarse"*, consisting of about $20k$ hexahedra and a second one, labeled as *"baseline"*, composed by nearly $120k$ hexahedra. Since a second order accurate DG scheme was used for both grids, each element is characterized by 4 degrees of freedom (DOFs), which translates in $80k$ DOFs per equation for the *"coarse"* mesh and $480k$ DOFs per equation for the *baseline* mesh. For both cases, the explicit second order Runge-Kutta (RK2) method was employed in temporal discretization.

RESULTS

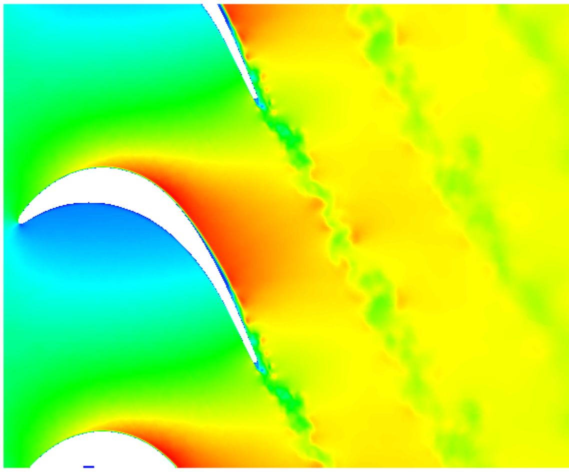
Figure 1 shows two visualizations of the instantaneous flow field for the *baseline* mesh. In particular, in Fig. 1a it is reported the typical instantaneous Mach number field on a plane perpendicular to the spanwise direction placed at $z = 0$, where it is clearly visible the separated turbulent flow over the suction side close to the blade trailing edge. Figure 1b diagrams the entropy field contours, which underline the turbulent nature of the aerodynamic structures present in the wake of the cascade, fully developed in each spatial direction.

Figure 2 reports the time-averaged isentropic Mach M_{is} number distribution over the blade, computed as follows:

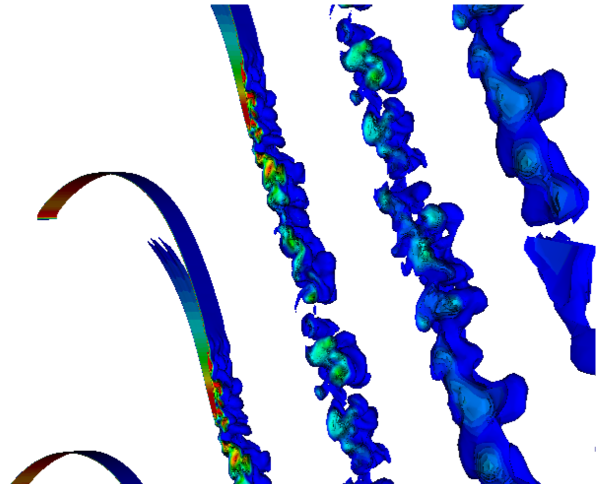
$$M_{is} = \sqrt{\left(\left(\frac{p_{in}^{\circ}}{p}\right)^{\frac{\gamma-1}{\gamma}} - 1\right) \cdot \frac{2}{\gamma - 1}}, \tag{1}$$

where p_{in}°, p and γ are the inlet total pressure, the static pressure and the ratio of specific heats, respectively. Results of *coarse* and *baseline* cases are compared with the experimental data measured at VKI and the numerical results presented by the Fernandez, Nguyen, and Peraire [7] at the 5th International Workshop on High-Order CFD Methods, obtained by performing ILES using the DIGASO solver [8], based on the Interior Embedded Disontinuous Galerkin (IEDG) method.

As can be seen from Fig. 2, there is a very good correspondence of the computed isentropic Mach number for both *coarse* and *baseline* cases with experimental and MIT [7] results. By first considering the *coarse* case, it is observed that the results, marked in red, offer a good estimation of the isentropic Mach despite the low number of DOFs. In particular, in the separation zone, the *coarse* results are completely consistent with those by MIT [7]. In contrast, for $x < 0.5\ c_x$, the computed *coarse* M_{is} slightly underestimates the reference data, while in the trailing edge proximity, it shows a small fluctuation. The *baseline* results, on the other hand, are perfectly congruent with the reference results for the entire axial extension of the blade. It can be seen that in the separation zone, the computed *baseline* M_{is} exactly



(a) Global overview of the Mach number field at $z = 0$.



(b) 3D entropy contours.

FIGURE 1. Instantaneous flow field for the *baseline* case. The passage has been duplicated for clarity.

matches with experimental results providing a better estimation than the numerical data in the literature. It should be noted that, precisely at the leading edge of the blade, the *baseline* and MIT [7] results, consistent with each other, under-estimate the experimental isentropic Mach, suggesting a discrepancy between experimental and computational setups.

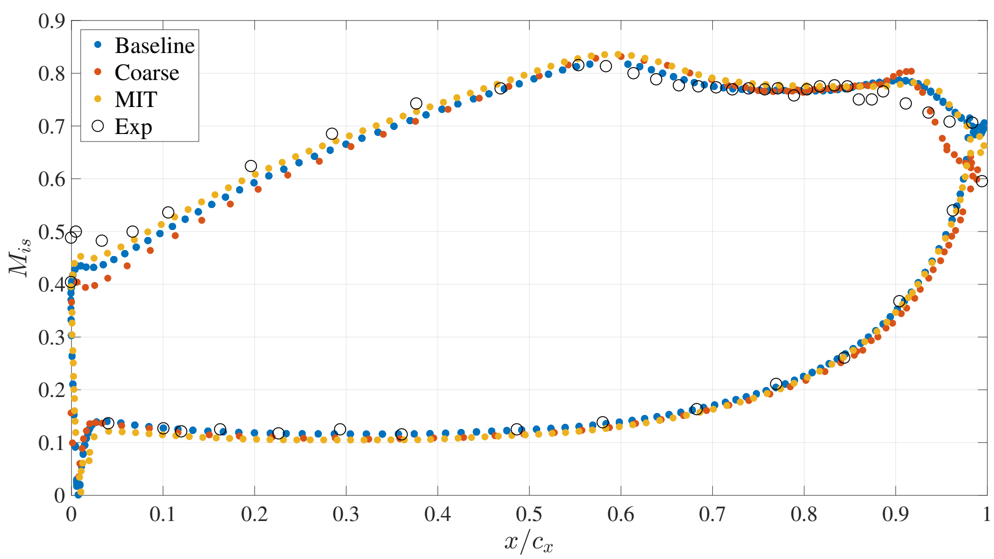


FIGURE 2. Time-averaged isentropic Mach number M_{is} around the blade. The results by Fernandez, Nguyen, and Peraire [7] are labeled as "MIT" and the Von Karman Institute experimental measurements labeled as "Exp" are presented with circles.

CONCLUSIONS

ILES of the transitional flow in the high-lift low pressure turbine cascade T106C have been performed in order to validate the reliability of the research code used, based on the Discontinuous Galerkin method (DGM) in a finite elements framework. A second order accurate scheme was selected for the space discretization in combination with two progressively finer 2^{nd} order curvilinear meshes. The results were compared with experiments conducted at the von Karman Institute and with numerical simulations carried out by Fernandez, Nguyen, and Peraire [7]. Perfect agreement with the literature reference data was found, particularly for the *baseline* case, characterized by a number of DOFs six times higher than the *coarse* one. This demonstrates that the optimization performed on the research code for the second order was successfully performed and that, as next step, it can be moved on to optimize higher orders.

REFERENCES

[1] J. Michálek, M. Monaldi, and T. Arts, Journal of Turbomachinery **134** (2012).

[2] C. C. de Wiart, K. Hillewaert, L. Bricteux, and G. Winckelmans, “Les using a discontinuous galerkin method: Isotropic turbulence, channel flow and periodic hill flow,” in *Direct and Large-Eddy Simulation IX* (2015).

[3] M. Pandolfi, AIAA journal **22**, 602–610 (1984).

[4] A. Ferrero, F. Larocca, and G. Puppo, International Journal for Numerical Methods in Fluids **77**, 63–91 (2015).

[5] M. Lo and B. van Leer, 19th AIAA Computational Fluid Dynamics (2009).

[6] C. Geuzaine and J.-F. Remacle, International Journal for Numerical Methods in Engineering **79**, 1309–1311 (2009).

[7] P. Fernandez, N. Nguyen, and J. Peraire, 5th International Workshop on High-Order CFD Methods (HiOCFD5), Gaylord Palms, Florida, USA 12 (2017).

[8] P. Fernandez, N. Nguyen, and J. Peraire, Journal of Computational Physics **336**, 308–329 (2017).