31st Congress of the International Council
of the Aeronautical Sciences

Belo Horizonte, Brazil; September 09-14, 2018

NUMERICAL ANALYSIS OF THE FLOW AROUND A BACKWARD-FACING RAMP

P.Catalano* , A.Mastellone* , M. Invigorito* , F. Capuano**
*C.I.R.A. - Italian Aerospace Research Center , **Univeristy of Napoli "Federico II"

Keywords: *Flow separation, Control, Ramp, URANS*

Abstract

This paper reports a numerical investigation of the incompressible flow over a 25° backward-facing ramp. Different approaches and flow solvers are applied. URANS simulations have been performed by the CIRA-developed UZEN and the Open-FOAM codes. The CIRA-developed flow solver Spark-LES together with OpenFOAM have been applied to resolve the turbulent structures on a simplified geometry and a reduced computational domain resembling the separated flow region over the ramp. The activities are being performed in the framework of the CIRA-funded project SHAFT. The final aim is to investigate the control of the separation phenomenon by means of zero-net mass flux actuators.

1 Introduction

An experimental test campaign to characterize the flow over a 25° backward-facing ramp is planned at University of Napoli "Federico II" in the framework of the CIRA funded project SHAFT. The control of the separation by means of zero-net-max flux actuators will be investigated.

The geometry, consisting of a NACA 0015 airfoil in the aft part followed by a ramp in the back part, is reported in figure 1. The main dimensions are: $a = 45mm.$, $b = 305mm.$, $c = 32mm.$, $d = 368mm.$. The Reynolds number (based on the height of the ramp) is $Re_h = 2.054 \times 10^4$. The free-stream velocity obtained in the wind tunnel is $U_\infty = 20$ m/sec.

2 Flow around the model

CFD activities are being performed to support the experiments and to provide data to be compared with the measurements in order to assess the numerical methods and models. The basic flow over the ramp is simulated by the URANS equations. The open-source Open-FOAM [1] code and the CIRA-developed flow solver UZEN [2] are applied. The UZEN code solves the compressible 3D steady and unsteady RANS equations on block-structured meshes and is designed to simulate flows around complex aerodynamic configurations. The spatial discretization adopted is a central finite volume formulation with explicit blended 2^{nd} and $4th$ order artificial dissipation. The dual-time stepping technique in employed for time-accurate simulations. The pseudo-time integration is carried out by an explicit hybrid multistage Runge-Kutta scheme. Classical convergence acceleration techniques, such as local time stepping and implicit residual smoothing, are available together with multigrid algorithms. Turbulence is modeled by either algebraic or transport equation models. The $\kappa-\omega$ SST turbulence model is applied and the the fully turbulent assumption is made for the flow.

It is expected that the flow presents some unsteadiness due to the bubble and to the thickness of the trailing edge. Therefore a time-accurate solution is searched. Actually, there is no much difference between an instantaneous and time-average flow field, as shown in figure 2

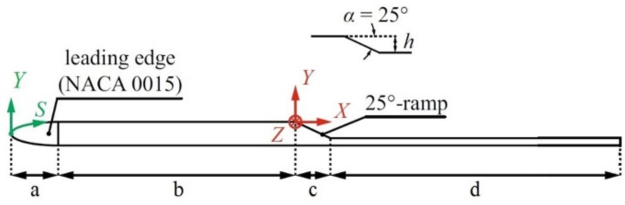


Fig. 1 Geomtery [3]

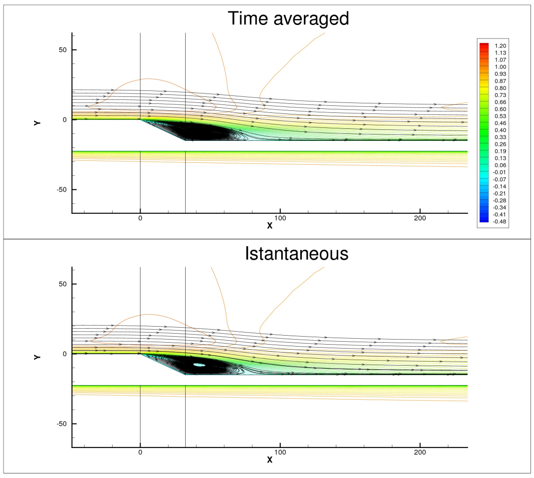
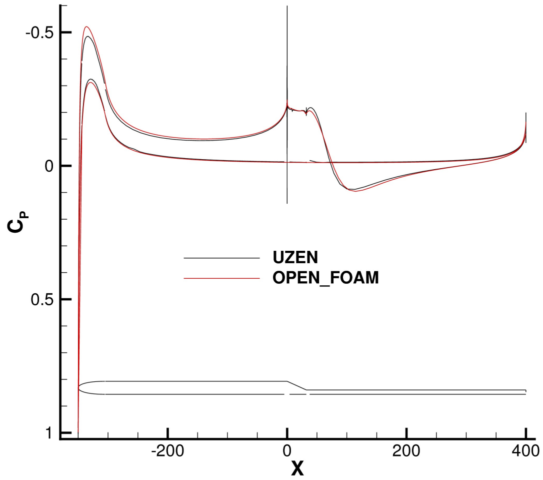


Fig. 2 Time-averaged bubble

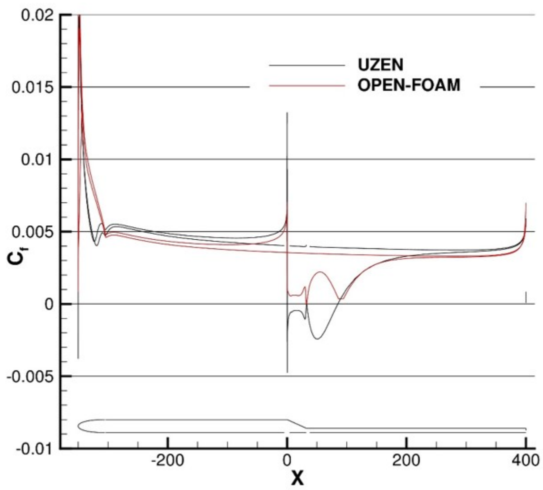
The two codes provide very similar results in line with literature data. A comparison of the time-averaged pressure and friction coefficients achieved by UZEN and OPEN-FOAM is showed in figure 3. The flow detaches at the beginning of the ramp. This means that that the angle of the ramp is sufficient for a pressure gradient able to separate the flow. The flow over the ramp reattaches at $x \approx 85$ mm. in good agreement with literature data [4].

The power spectral density (PSD) of lift and drag coefficients is reported in figure 4. A clear peak at a Strouhal number $Str = \frac{f*h}{U_\infty}$ of 0.33 is clearly visible.

Two probes have been located in the separated region (P_1) and at the trailing-edge of the ramp (P_2) as shown in figure 5. The power spectral density (PSD) of the pressure and of the velocity components are shown in figure 5. The data have been obtained by the $\kappa - \omega$ SST-LR, a low-Reynolds variation of the SST model [5]. The main peak is present in velocity and pressure at both the locations. At the point P_2, secondary



(a) C_P



(b) C_f

Fig. 3 Time-accurate pressure and skin-friction coefficients

frequencies with a relevant content of energy are also visible. It is worth noting that the SST $\kappa - \omega$ SST model returns a main peak at a little lower frequency with $Str = 0.28$.

3 Numerical characterization of the separated flow

The numerical activities performed in the framework of the project SHAFT have several goals. The first is to support the experimental test cam-

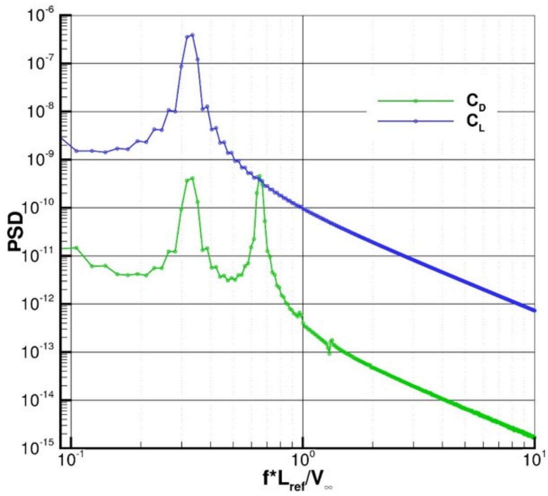


Fig. 4 Power spectral density of aerodynamic coefficients

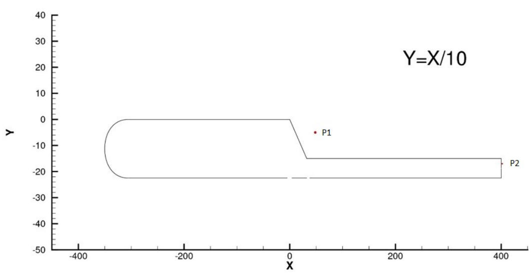
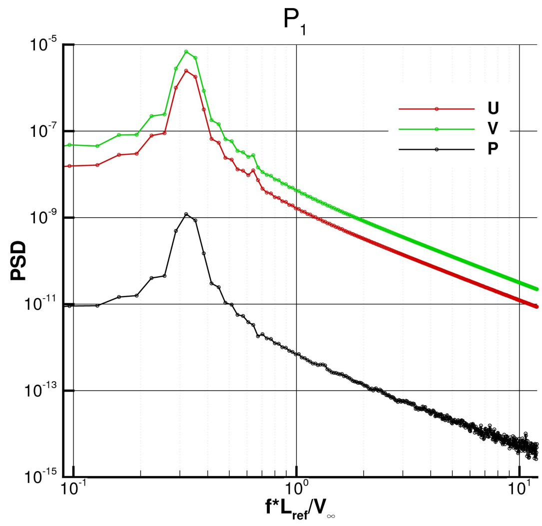


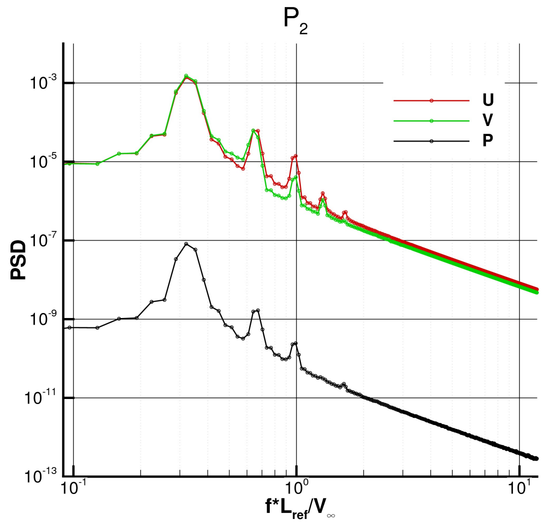
Fig. 5 Location of probes for PSD. Different scales for x and y axis.

paign providing information useful for choosing the parameters of the control device. The other aim is to further validate the in-house flow solver Spark-LES [6, 7, 8]. SPARK-LES is a code currently under development at CIRA in the framework of the HYPROB Program, funded by the Italian Ministry of Research. The solver features high-fidelity numerical methods and state-of-the-art modelling, and reproduces mixing and turbulence in general flows as the operation of a liquid-rocket thrust chamber at high pressures.

PIV measurements will be taken during the experiments and this will allow for a validation and calibration of the last features (sub-grid modeling, inflow turbulence) implemented in the code.



(a) P_1



(b) P_2

Fig. 6 Power spectral density of velocity components and pressure

The characterization of the separated flow is being performed by employing several codes and methods. The well-assessed in-house URANS code UZEN, the open-source solver OpenFOAM and also Spark-LES that needs to be used with some care because some features are still to be deeply verified.

3.1 Simplified Geometry

The parameters of the control device to be used in the experiments have been obtained by looking at the flow over a rounded ramp. [4]. Therefore the flow over this geometry has also been taken into consideration. The simulations by Dandois *et al.* [4] are taken as reference, because of the simplified geometry and of the employed control techniques. The difference are the Reynolds number and the fact is that the geometry has a rounded edge. However, despite the differences it is expected that the separation region and the length of the bubble are comparable.

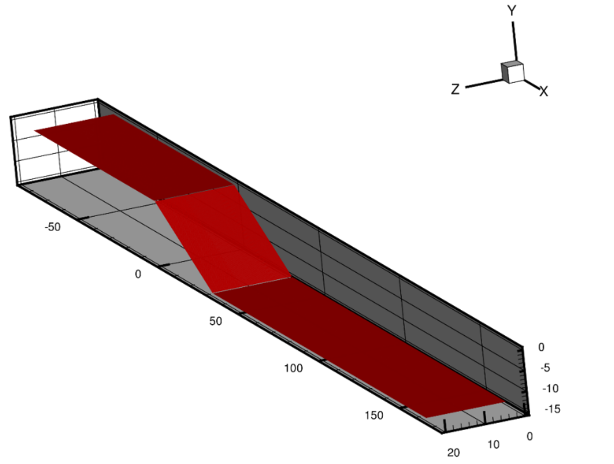
3.2 Eddy-resolving methods

Eddy-resolving methods are applied to the simplified geometry with a sharp edge and at the same Reynolds number as the experiments. The computational domain is reduced, as shown in figure 7 in order to save CPU time.

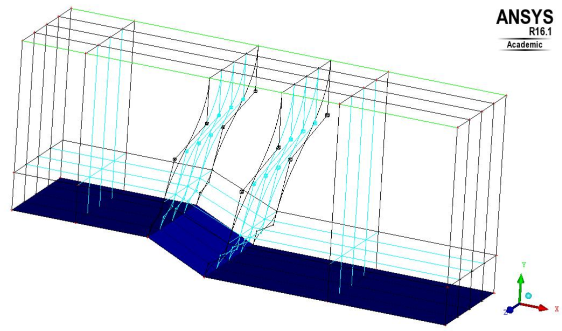
A grid made of $748 \times 96 \times 384 = 27.5 \times 10^6$ cells has been generated. A simulation without any turbulence/sub-grid model by the Open-FOAM code has been performed before applying the CIRA code Spark-LES. UZEN code by using the $\kappa - \omega$ SST model has been also employed on a much coarser mesh of 472×108 cells.

The OpenFOAM computations have been performed using the native solver pimpleFoam, which is based on a modified version of the classical PISO algorithm ("PIMPLE") to handle the pressure-velocity coupling. Second-order accurate central schemes in space and a second-order backward scheme in time have been employed, with maximum CFL number equal to 1. Boundary conditions consist of a uniform inlet profile without any superimposed perturbations, a pressure outlet, and slip boundaries for the top surfaces. The simulation has been advanced for about 20 through-flow-times and the results are time-averaged for about 10 through-flow-times. Spanwise averaging is not performed and the data in the mid-span plane are considered.

The bubbles returned by UZEN and Open-FOAM are very similar (figure 8). The flow re-



(a) Simplified geometry (dimensions in mm.)

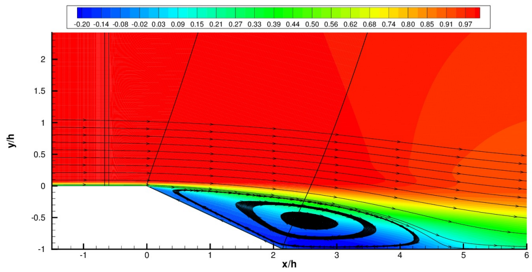


(b) Computational domain

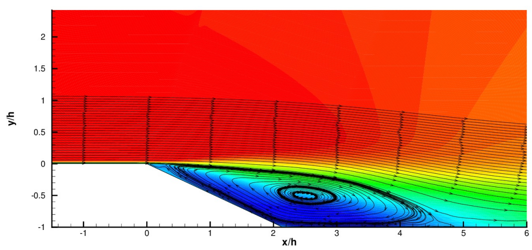
Fig. 7 Simplified geometry and computational domain

attaches at about $x/h \approx 4.9$. The reference DNS [4], instead, has a shorter bubble with a separation point at $x/h = 0.53$ and a re-attachment point at $x/h = 3.93$.

A comparison the the time-averaged streamwise velocity upstream of the edge at the stations $x/h = -1$ and $x/h = 0$ (edge of the ramp), is shown in figure 9. The boundary layer returned by Open-FOAM is completely different from the reference literature data. This was expected due to the differences in geometry and Reynolds number and because an uniform velocity profiles is employed in the Open-FOAM simulations, while *"a realistic turbulent inflow boundary condition based on an extraction/rescaling technique"* is used by Dandois *et al.* [4]. In fact,

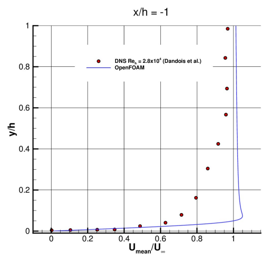


(a) UZEN

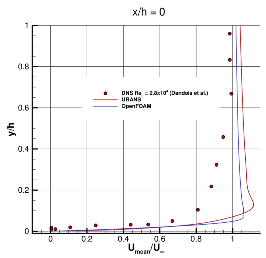


(b) OpenFOAM

Fig. 8 Time-averaged bubble on simplified geometry. contour of streamwise velocity

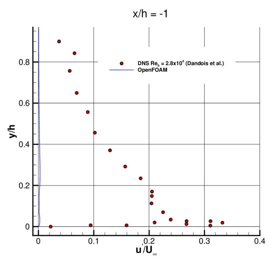


(a) $x/h = -1$

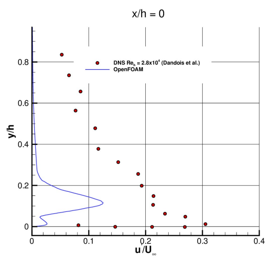


(b) $x/h = 0$

Fig. 9 Time-averaged stream-wise velocity upstream the edge



(a) $x/h = -1$

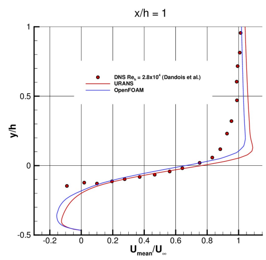


(b) $x/h = 0$

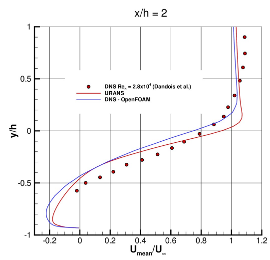
Fig. 10 Time-averaged stream-wise velocity fluctuations upstream the edge

the stream-wise velocity fluctuations by OpenFOAM are negligible at $x/h = -1$ and much lower at the edge of the ramp (figure 10). The "cheap" URANS results are in reasonable agreement with the OpenFOAM data.

The comparison in the zone of the ramp where the flow is separated is reported in figure 11. The agreement of mean velocities with the



(a) $x/h = 1$

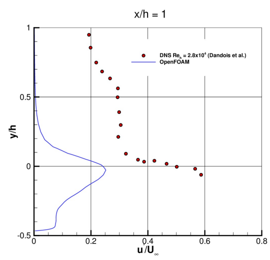


(b) $x/h = 2$

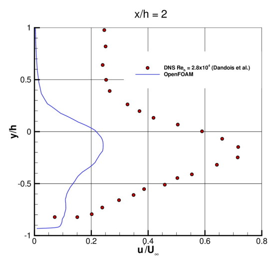
Fig. 11 Time-averaged stream-wise velocity over the ramp

DNS literature data is far better. Again the UZEN results are encouraging also in the view of using fast URANS simulations to support the experiments. It is worth noting that the reference points are taken from the published paper and therefore some inaccuracy is possible in the plotted data.

The velocity fluctuations are shown in figure 12. Fluctuations are present in OpenFOAM re-



(a) $x/h = 1$

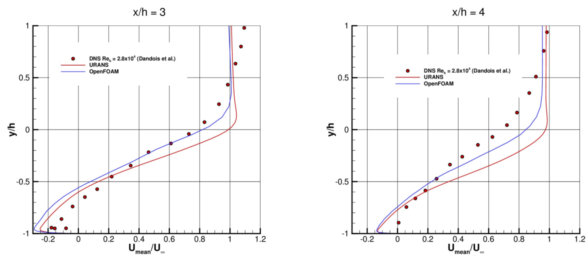


(b) $x/h = 2$

Fig. 12 Time-averaged stream-wise velocity fluctuations over the ramp

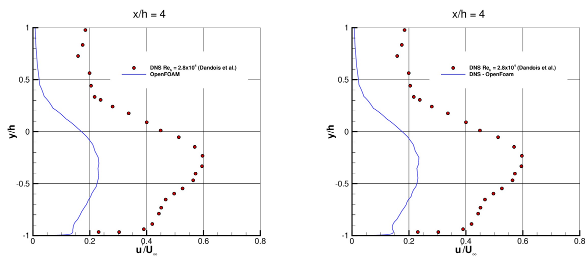
sults but are lower that DNS data. However, the peak is located at the same distance from the wall. It is important to note that the data are reported along the y axis and not along the normal to the wall.

Finally, the results in the recovery region are presented in the figures 13 and 14. Both OpenFOAM and UZEN data agree quite well with DNS at the station $x/h = 3$ where the flow is still separated. Some discrepancies are visible at $x/h = 4$ mainly due to the fact the flow is just re-attached in DNS while is still separated in the present computations. OpenFOAM underestimates the fluctuations with respect to the DNS. The inflow fluctuations of DNS still have some influence at theses stations while the u' provided by OpenFOAM goes to zero away from the wall.



(a) $x/h = 3$ (b) $x/h = 4$

Fig. 13 Time-averaged stream-wise velocity in the recovery region



(a) $x/h = 3$ (b) $x/h = 4$

Fig. 14 Time-averaged stream-wise velocity fluctuations in the recovery region

The simulations above discussed have provided useful information. The simplified geometry of the ramp can be used to reproduce the region of separated flow present in the model that will be tested in the wind tunnel experiments. URANS cannot reproduce the turbulent characteristics of the flow but are fast and can be employed to support the wind tunnel campaign and to give useful data for choosing the control parameters. The OpenFOAM computation has allowed to numerically set-up a direct numerical simulation.

The goal is to perform eddy-resolving simulations by the CIRA solver Spark-LES and OpenFOAM data will be useful for sake of comparison. A first attempt by Spark-LES has been performed on a coarser grid of $364 \times 72 \times 256$ cells without employing any sub-grid model. Fluctuations are added at the inflow of the computational domain. The turbulence develops as can be seen in figure 15 that shows a visualization of the turbulent structures by an iso-surface of $Q = \frac{1}{2}\left(\Omega_{i,j}\Omega_{i,j} - S_{i,j}S_{i,j}\right)$. The turbulent structures are forming upstream the ramp, then in the separated region, and in the boundary layer developing downstream the bubble.

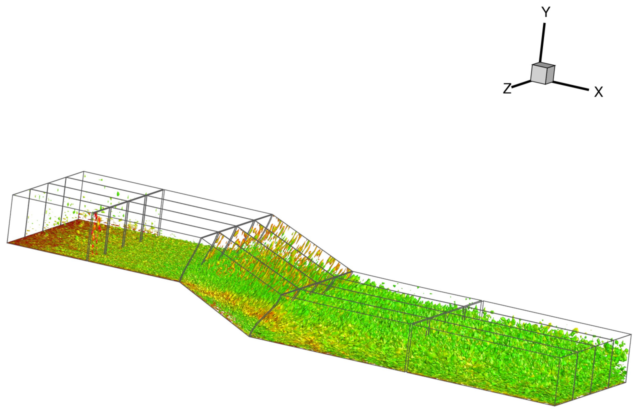


Fig. 15 Isosurface of $Q = 2 \times 10^6 m^2/s^2$ coloured by vorticity

3.3 Concluding Remarks

The numerical activities that are being performed in the framework of the project SHAFT, a project fully funded by CIRA whose aim is to investigate the control of the flow separation by zero-net mass flux actuators, have been presented in this paper. Different numerical methods and codes have been employed. The CIRA flow solver UZEN and the open-source code OpenFOAM have been employed for URANS simulations. The in-house developing Spark-LES and OpenFOAM are used as eddy-resolving numerical methods. A first simulation has been performed by OpenFOAM without any turbulene/subgrid model on a simplified geometry resembling the experimental model. Useful information have been obtained. The simplified geometry can be used to reproduce the region of separated flow that will be controlled in the experiments. URANS cannot reproduce the turbulent characteristics of the flow but are fast and can be employed to support the wind tunnel campaign and to give useful data for choosing the control parameters. The Spark-LES code has been used so-far on a coarse mesh and the simulation of the finer mesh is currently running.

3.4 Acknowledgments

The present work is conducted in the work plan of the CFD analyses task in the framework of the **S**ynt**H**etic jet **A**ctuators for **F**low con**T**rol (**SHAFT**) project funded by CIRA and in collaboration with the department of industrial Engineering of University of Napoli ”Federico II”.

References

[1] “OpenFOAM and the OpenFOAM Foundation,” https://openfoam.org/.

[2] Catalano, P. and Amato, M., “An Evaluation of RANS Turbulence Modelling for Aerodynamic Applications,” *Aerospace Science and Technology Journal*, Vol. 7, No. 7, 2003, pp. 493–509.

[3] Kourta, A., Thacker, A., and Joussot, A., “Analysis and characterization of ramp flow separation,” *Experiments in Fluids*, Vol. 56, 2005, pp. 104–118.

[4] Dandois, J., Granier, E., and Sagaut, P., “Numerical simulation of active separation control by a sinthetic jet,” *Journal of Fluid Mechanics*, Vol. 574, 2007, pp. 25–58.

[5] Catalano, P. and Tognaccini, R., “Turbulence modelling for low Reynolds number flows,” *AIAA Journal*, Vol. 48, 2010, pp. 1673–1685.

[6] Capuano, F., Mastellone, A., Di Benedetto, S., Cutrone, L., and Schettino, A., “Preliminary developments towards a high-order and efficient LES code for propulsion applications,” *ECCOMAS-IACM* 6^{th} *European Conference on Computational Fluid Dynamics (ECFD VI)*, Barcelona, Spain, July 2014.

[7] Mastellone, A., Cutrone, L., and Capuano, F., “DNS of the Taylor-Green vortex at Re=1600,” *ECCOMAS Congress 2016*, Crete Island, Greece, June 2016.

[8] Capuano, F., A. M. and De Angelis, E. M., “A conservative overlap method for multi-block parallelization of compact finite-volume schemes,” *Computers and Fluids*, Vol. 159, 2017, pp. 327–337.

Copyright Statement