

Feasibility of a Spinning Cylinder on the Leading and Trailing Edges of a Flap for High Lift

A. Francannet*, D. Di Pasquale†, A R. Proença‡ and S. A. Prince §

School of Aerospace, Transport and Manufacturing, Cranfield University, Cranfield, MK43 0AL, United Kingdom

This paper presents a numerical feasibility study of a spinning cylinder mounted on the leading edge and trailing edge of a flap to improve lift force generation. The baseline used in this study is a two-dimensional NLR 7301 airfoil with a single-slotted trailing-edge flap. For this configuration, experimental data is available in the literature. The Ansys computational fluid dynamics (CFD) simulation software has been used in this work to simulate the aerodynamic properties of several configurations. Firstly, a turbulence model dependency study over the existing geometry was conducted, providing lift, drag and pitching moment coefficients, as well as pressure distribution for three angles of attack. Data was then acquired using the Spalart-Allmaras turbulence model at Mach and (chord-based) Reynolds numbers 0.185 and 2.51×10^6 , respectively. Three configurations with a leading-edge spinning cylinder and one with a trailing-edge spinning cylinder were investigated. Results suggest that the introduction of a cylinder in the baseline geometry decreased lift and drag performance by inducing flow separation. Nevertheless, for the leading edge configurations, the rotation of the cylinder allowed recovery of the original flapped airfoil performance by re-energizing the flow around the flap and reattaching the boundary layer. For the trailing edge configuration, the spinning cylinder generated additional lift due to Magnus and suction effects at high rotation speed of the cylinder, and by modifying the flow circulation around the flap's trailing edge. Overall, no benefit over the lift-to-drag ratio could be derived from this preliminary study.

I. Nomenclature

α	=	angle of attack (°)
δ_{99}	=	total height of the boundary layer (m)
γ	=	ratio of specific heats
c	=	chord (m)
C_d	=	drag coefficient
C_l	=	lift coefficient
C_m	=	pitch coefficient
C_p	=	pressure coefficient
f	=	aerodynamic parameter
G	=	growth rate
M	=	Mach number
N	=	number of layers
U	=	free-stream velocity (m/s)
U_c	=	velocity of the cylinder (m/s)
r	=	cylinder radius (m)
Re_x	=	Reynolds number
y^+	=	non-dimensional wall distance
y_H	=	first cell height (m)

*Graduate MSc Student, Applied Aerodynamics Group

†Lecturer in Aerodynamics, Applied Aerodynamics Group, AIAA Member

‡Lecturer in Aerodynamics, AIAA Member

§Professor of Aerodynamics, Applied Aerodynamics Group, Senior AIAA Member

II. Introduction

At the age of the ecological transition starting for a greener future, the challenge of reducing the carbon emissions of air transport is paramount. In this context, the role of aerodynamicists is essential in order to keep aircraft ever more efficient and reduce their ecological footprint. Since the beginning of the 20th century, studies to improve the performance of the aircraft have been developed. It has been shown that the wing efficiency was directly linked to the lift-to-drag ratio, indicating that it was necessary to reduce drag while increasing the lift capabilities of the aircraft. Furthermore, it was demonstrated that for demanding manoeuvres such as take-off and landing, high lift was required. Based on all these findings, the aeronautical world has seen the development of high-lift systems that generate the required additional lift. In general, these systems have resulted in modifications of wing geometry through the introduction of flaps and slats to extend the stall limit at high angles of attack. It is also assumed that the loss in lift is due to the boundary layer separation.

In this context, the presented study proposes to investigate a method of effective flow control using moving surfaces on a two-element airfoil to generate lift more efficiently [1–5]. With this new technology, the aim is to reduce both weight and noise over existing systems by removing or modifying high-lift devices, such as flaps and slats. A significant drag benefit is also intended. Finally, as a reduction in weight and drag would reduce the energy consumption of the aircraft, this technology would also contribute towards zero carbon flight goals.

In this study, computational fluid dynamics (CFD) methods are used to numerically simulate the aerodynamic properties of several two-element airfoil configurations. A well-known airfoil, namely, the NLR 7301, has been selected as the baseline for this work. The starting point of the current investigation is then the experimental work conducted by B. Van den Berg in 1979 entitled: “Boundary Layer Measurements On A Two-Dimensional Wing With Flap” [6].

III. Methodology

A. Geometry

To study the effect of the rotating cylinder at the leading edge of the flap, four configurations were designed. An illustration of these configurations is displayed in Fig. 1. Configuration 0 represents the experimental airfoil without cylinder and was used to validate the CFD settings. In Configuration 1, a relatively small cylinder was designed in order to match perfectly the shape of the flap leading edge. Configuration 2 was developed by modifying the flap geometry and adding a cylinder at its maximum thickness position. Finally, Configuration 3 reused the Configuration 2 geometry, but the edge of the cylinder was moved forward in order to match the initial leading edge position of the flap. Concerning the rotating cylinder at the trailing edge of the flap, one model was built. To create the fourth configuration, the trailing edge of the flap was replaced by a cylinder of equivalent diameter to that of the first configuration. This final configuration is displayed in Fig. 2. The five configurations (0-4) were designed the same way using SolidWorks using the coordinates of the baseline airfoil with flap from B. Van den Berg’s report keeping the dimensions given in the report ($c=0.57m$) with a gap between the airfoil and the flap of $2.6\%c$ [6]. Concerning the flow domain, it was also drawn on SolidWorks in the shape of a half circle of radius equal to 10 times the chord continued by a rectangle of 15 times the chord by 20 times the chord. The flow domain was designed large enough to avoid the free-stream boundary conditions to interact with the flow around the airfoil.

B. Model Meshing

To begin with, four meshes for configurations 0, 1, 2, and 3, and five meshes for the trailing edge configuration (Config. 4) were created for the grid dependency study and the generation of the results of the study. The meshes were generated on Ansys Meshing and defined as hybrid in order to combine structured and unstructured meshes. The structured part of the mesh was represented by a prismatic layer near the wall while the unstructured part was represented by triangles around the complex geometry. The prismatic layer was calculated taking into account the total thickness of the boundary layer (in m) for a y^+ equal to 1 as follows:

$$\delta_{99} = \frac{0.37x}{Re_x^{1/5}} \quad (1)$$

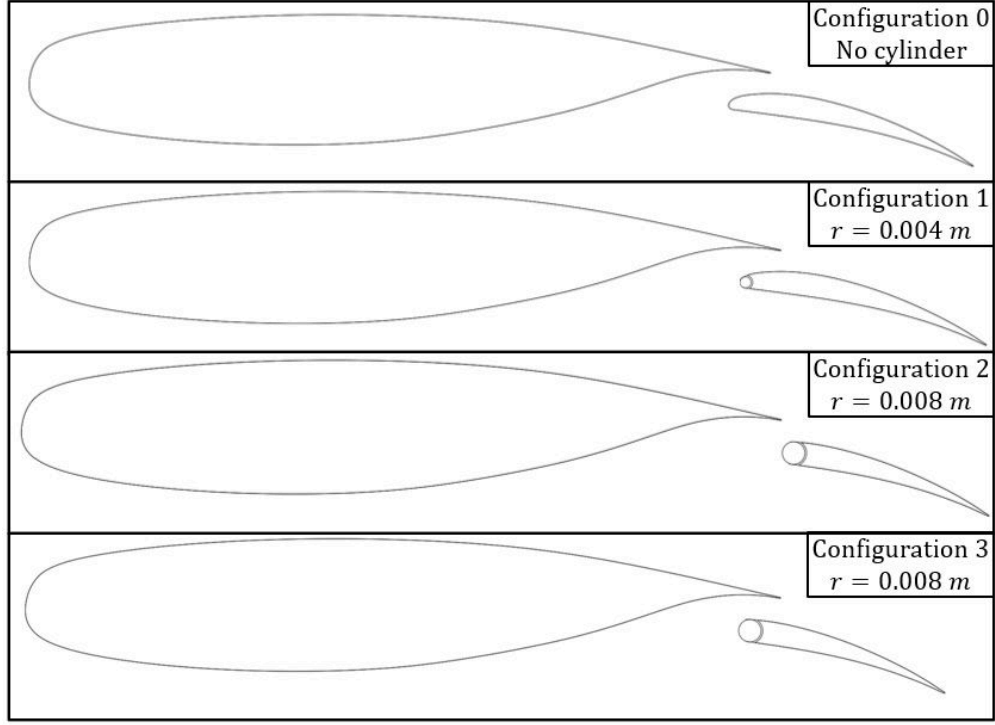


Fig. 1 Flap leading edge cylinder configurations

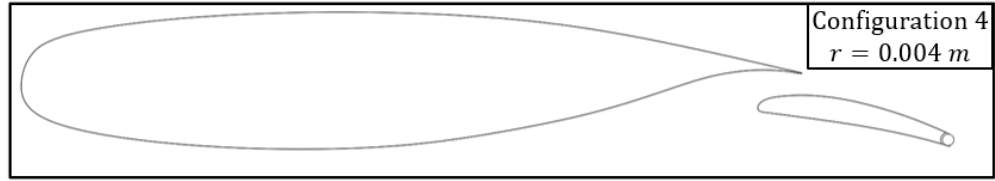


Fig. 2 Flap trailing edge cylinder configuration

From the knowledge of the thickness of the boundary layer and the height of the first cell, the growth rate and the number of layers were calculated using equation 2. The values computed for each of these properties are displayed in Table 1.

$$\delta_{99} = y_H \frac{1 - G^N}{1 - G} \quad (2)$$

Table 1 Prism layer

Main element					flap				
y_H	Re_x	δ_{99}	N	G	y_H	Re_x	δ_{99}	N	G
5.8e-6	2.51e6	1.1e-2	33	1.2	5.2e-6	7.4e5	4.36e-3	28	1.2

In order to assess the mesh convergence, lift, drag and pitching moment coefficients were obtained for all meshes and configurations at three angles of attack (i.e., 6° , 10.1° , 13.1° , matching the experimental data available). The first idea was to see if the results obtained were consistent with the experimental data, especially near the stall, by observing the values of each of the coefficients. Then, the parameter f defined by $C_l \times C_d \times C_m$ was compared against the number of elements to show the grid dependency over the aerodynamic parameters. To validate the results obtained with the mesh convergence method, it was decided to use Roache's method [7]. Indeed, in order to be sure that the results obtained with the selected mesh did not depend on this same grid, it was necessary to estimate the discretization errors by obtaining the grid convergence index (GCI).

The maximum GCI obtained for the mesh dependency study of the NLR 7301 with a flap and no cylinder was equal to 0.1951%. In the same way, the maximum GCI obtained for the mesh dependency study of the NLR 7301 with a flap and a leading edge cylinder was equal to 0.14%. Finally, the maximum GCI obtained for the mesh dependency study of the NLR 7301 with a flap and a trailing edge cylinder was equal to 0.2395%. If it was highlighted that the error related to the mesh of the configurations 0, 1, 2 and 3 was lower than the one related to the mesh of Config. 4, the study also showed that the selected meshes allowed to obtain a total independence of the results with respect to the grid. To summarize, it was selected the second mesh (fine) for all configurations in accordance with the GCI saying that these meshes were respectively fine enough for the study. A representation of the selected mesh for the first configuration is displayed in Fig. 3 and Fig. 4.

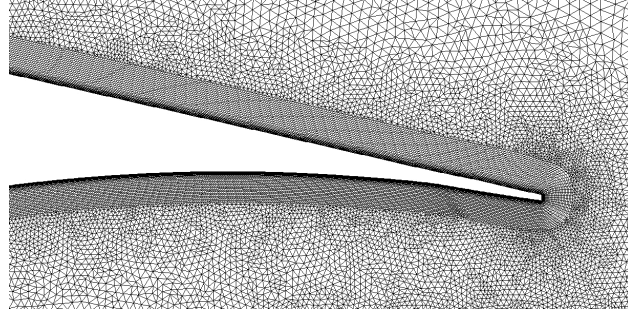


Fig. 3 Mesh around the trailing edge of the main element

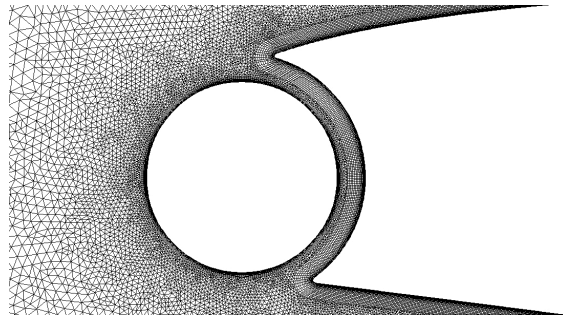


Fig. 4 Mesh around the cylinder and the leading edge of the flap

C. CFD Settings

Based on the literature, the choice was made to use a RANS turbulence model [8]. For the verification part, the choice was made to use the Spalart-Allmaras model considered as the most robust [9]. The validation was carried out on the three turbulence models, namely: Spalart-Allmaras, SST $k - \omega$, SST Transition. Steady state with a pressure based model and a coupled method as a low Mach number were selected in this study. The order of accuracy was set to 2nd order except for the modify turbulent viscosity equation which was set to 1st order. Then, the energy equation was used. Finally, the solutions were computed after 10,000 iterations by ensuring convergence of the residuals to 10^{-4} . Both the main element and the flap of the aerofoil were defined as stationary walls with no-slip condition and a

standard roughness model. For the simulations with the cylinder, it was set as a moving wall with no-slip condition and a relative to adjacent cell zone rotational motion. On a second hand, the boundary of the domain was defined as a pressure far-field with a gauge pressure of 1 atm, a temperature of 300 K and a Mach number equal to 0.185. The X and Y components of the flow direction were set for 17 angles of attack ranging from 0° to 16.1° . Finally, a pressure outlet condition was assigned to the right boundary with an absolute backflow reference frame of 1 atm and a backflow total temperature of 300 K.

The fluid was set as air. The density was calculated using ideal gas assumption, and Sutherland's law was used to calculate the dynamic viscosity (based on the experimental Reynolds number equals to 2.51×10^6). Finally, all walls were set with aluminium constants on Ansys Fluent.

At the end of the 10,000 iterations, all results were collected and then knowing the applied angle of attack, the lift and drag coefficients were calculated. The convergence was assessed using residuals, forces and moments values. When the coefficients did not fluctuate after the 3rd decimal place, they were considered as converged. For a more accurate result, all the coefficients were averaged over their last 1,000 iterations. The pressure distribution was obtained after convergence of the results using the "XY Plot" tool. Finally, velocity, pressure, density and turbulence contours were displayed allowing an analysis of the physics around the model.

D. Validation

Using the verified mesh for the cylinder-free case, the validation of the CFD model was performed. First, an investigation over the impact of the solver was conducted. Then, the influence of the discretization scheme on the result was studied. Finally, the study focused on the assessment of the best turbulence model available to match the experimental data.

1. Investigation of the impact of the solver

Initially, the assumption was made to use a pressure based model considering the low Mach number studied. In order to validate this assumption, two simulations were performed with the pressure and the density based models at an angle of attack of 13.1° considering that the viscous effects were more important near the stall. The results were contrasted with the experimental data in order to conclude.

Table 2 Solver comparison

	C_l	C_m	C_d
Pressure based solver	3.1062	-0.454	0.0748
Density based solver	Not converged	Not converged	Not converged
Experimental data	3.141	-0.440	0.0445

Results in Table 2 showed that with the density based model, the simulation had difficulty to converge while for the pressure based model, the results were converged after only a few thousand iterations (between 1,000 and 10,000). Indeed, if the pressure based model allowed to obtain results close to the experimental in terms of lift and pitch coefficients very quickly and accurately, the density based model was only oscillating around the solution translating the unstable state of the flow. For this study, the pressure based model was, therefore, preferred.

2. Investigation of the impact of the discretization scheme

The choice was made to use a Coupled scheme as it was more robust and efficient than the SIMPLE scheme for steady-state flows. In order to validate the choice of this discretization scheme over the SIMPLE scheme, two simulations were done with both scheme at 6° and 13.1° . The validation study was performed over an angle of attack in the middle of the lift curve (6°) and an angle of attack near the stall (13.1°) in accordance with the experimental data.

From Table 3, it was noted that no significant differences were observed. Indeed, the maximum relative difference observed between the results obtained with the Coupled scheme and those obtained with the SIMPLE scheme was

equal to 1.32% and considered as negligible. Furthermore, the convergence with the SIMPLE scheme took over 50,000 iterations while the results were converged after only a few thousand iterations (between 1,000 and 10,000). Therefore, the choice to use the Coupled scheme for the study was confirmed.

Table 3 Discretization scheme comparison

	$\alpha = 6^\circ$			$\alpha = 13.1^\circ$		
	C_l	C_m	C_d	C_l	C_m	C_d
Coupled	2.3947	-0.4760	0.0380	3.1062	-0.454	0.0748
SIMPLE	2.3893	-0.4750	0.0385	3.1131	-0.4559	0.0747
Relative difference (%)	0.23	0.21	1.32	0.22	0.42	0.13

3. Investigation of the impact of the turbulence model

In order to find the accurate turbulence model for the study, lift and pitch coefficients were computed for Spalart-Allmaras, SST $k - \omega$, and SST Transition models and compared against experimental data. In addition, an analysis of the pressure distribution was carried out to confirm the conclusions obtained with the study of the aerodynamic coefficients.

The lift coefficient study (Fig. 5) showed that both the Spalart Allmaras and the SST $k - \omega$ models underpredicted the results over the entire range of angle of attack. However, the SA model perfectly captured the stall angle, whilst the SST $k - \omega$ underpredicted it. Finally, the SST Transition underpredicted the results from 0° to 11.1° and perfectly matching them from 12.1° to 14.1° . Nevertheless, it was over-predicting the stall angle.

Concerning the pitch coefficient, it was shown through Fig. 6 that while the curvature of the curve was followed by all the models, none of them was able to match the experimental data from 0° to 8° . Then, from 8° to 14.1° the SST $k - \omega$ performed better than others. Finally, only the SA model was able to capture the stall angle.

From the pressure distribution at 6° (Fig. 7), it was shown that the three turbulence models studied were performing well while matching the experimental results. Indeed, it was noticed that the three curves obtained for each turbulence model were almost superimposed and passed through all the experimental points. This aspect was recognized for all three angles of attack presented in the experimental report studying the NLR7301 aerofoil with a flap [6].

By zooming into the areas of interest (Fig. 8) defined in the experimental report through the flow visualization [6], it was shown that the SST Transition model perfectly captured the separation bubble at the leading edge of the airfoil ($0.017\text{m} < c < 0.023\text{m}$). Concerning the transition region on the upper surface of the flap, it was noticed that the SST Transition model gave good results by displaying a transition zone at a chord position comprise between 0.62m and 0.64m. The measurements in the wind tunnel had shown a transition zone at a chord position comprise between 0.622m and 0.633m. Finally, the study of the pressure distribution has shown that the SST Transition model was able to determine precisely the transition regions around the airfoil as well as the separations bubbles according to flow visualizations.

In conclusion, it was stated that the results confirmed the choice for the Spalart-Allmaras model since the study was interested in the improvement of the aerodynamic capacities of the wing with a major interest on the lift. Thus, the choice of a model correctly predicting the stall angle as well as the lift coefficient proved to be the most judicious. It is also important to note that regarding the drag, it was shown that all the models gave good results and the choice of the Spalart-Allmaras model was obvious as considered as the most robust.

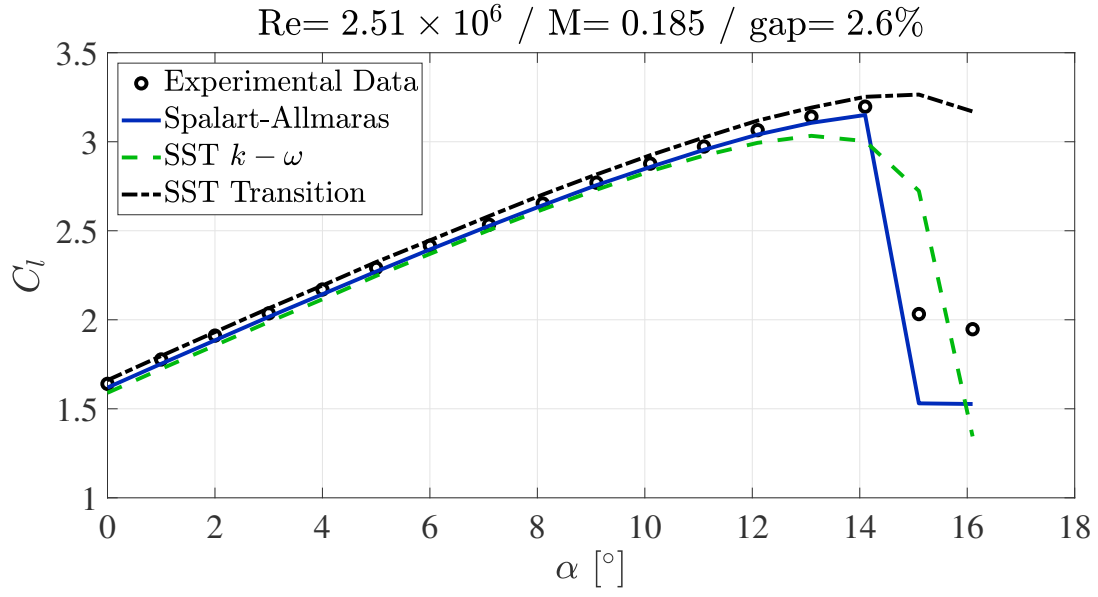


Fig. 5 Lift coefficient against angle of attack / Turbulence model study

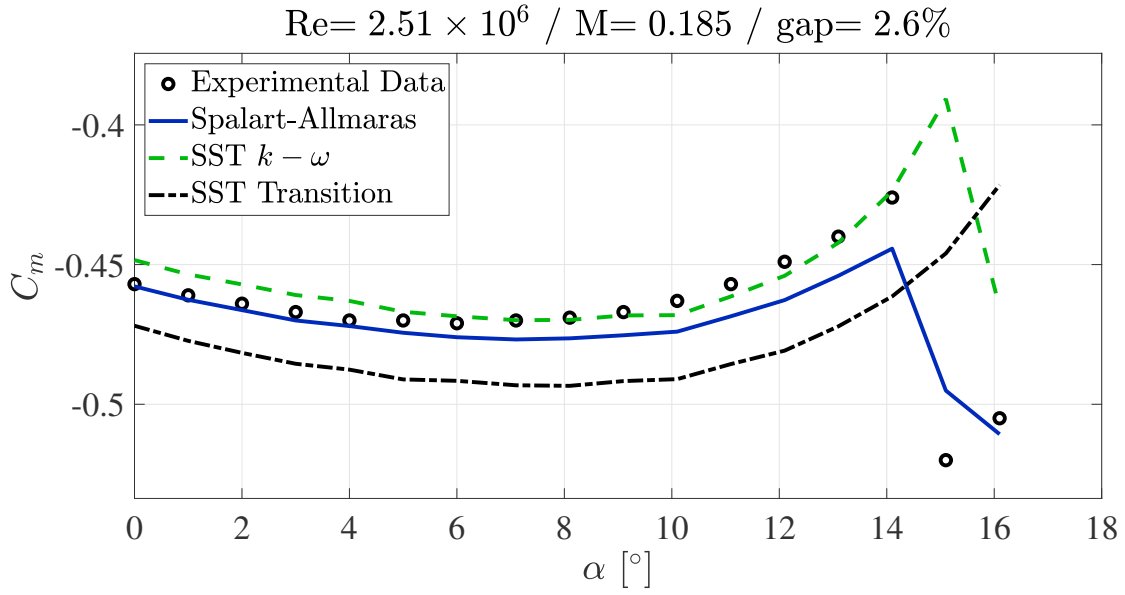


Fig. 6 Pitch coefficient against angle of attack / Turbulence model study

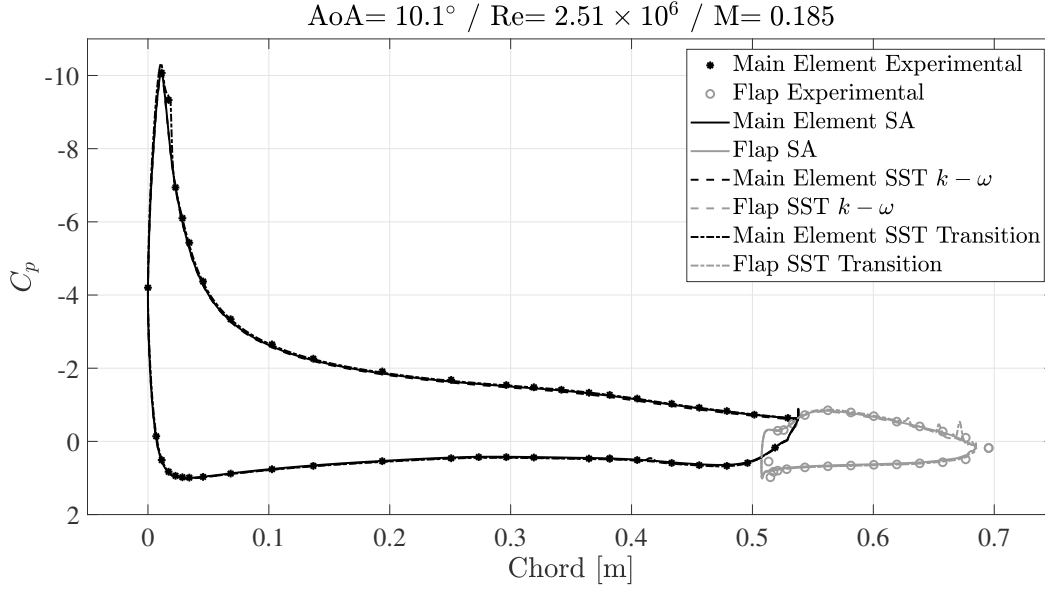


Fig. 7 Surface pressure distribution around an NLR 7301 aerofoil / CFD models vs Experimental results

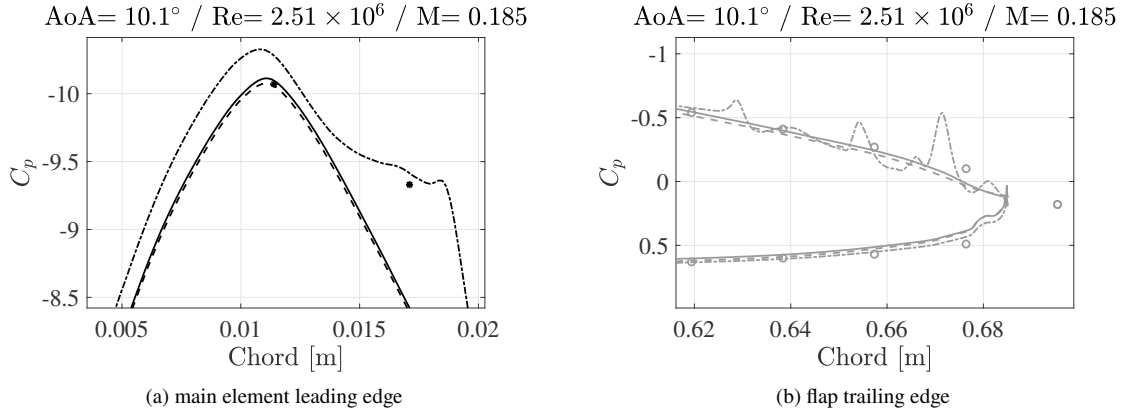


Fig. 8 Surface pressure distribution around an NLR 7301 aerofoil / CFD models vs Experimental results / Zoom

IV. Results and Discussion

A. NLR 7301 with flap and leading edge cylinder

From the CFD results, it is possible to display the flow over an object and study its characteristics. Firstly, flow over the four leading edge configurations with no rotation of the cylinder were studied. Fig. 9 shows that with the introduction of a cylinder at the leading edge of the flap, a discontinuity was introduced in the geometry generating a separation bubble (Configs. 1 and 3). Additionally, a large separation was observed with the flap positioned backward (Config 2). It then appeared that when the flap was moved back to its original leading edge position (Config. 3), the separation was reduced but still larger than in configurations 0 and 1. Thus, the introduction of cylinders of different sizes and at different positions generated detrimental effects to the flow development along the flap and, more generally, to the aerodynamic properties of the system.

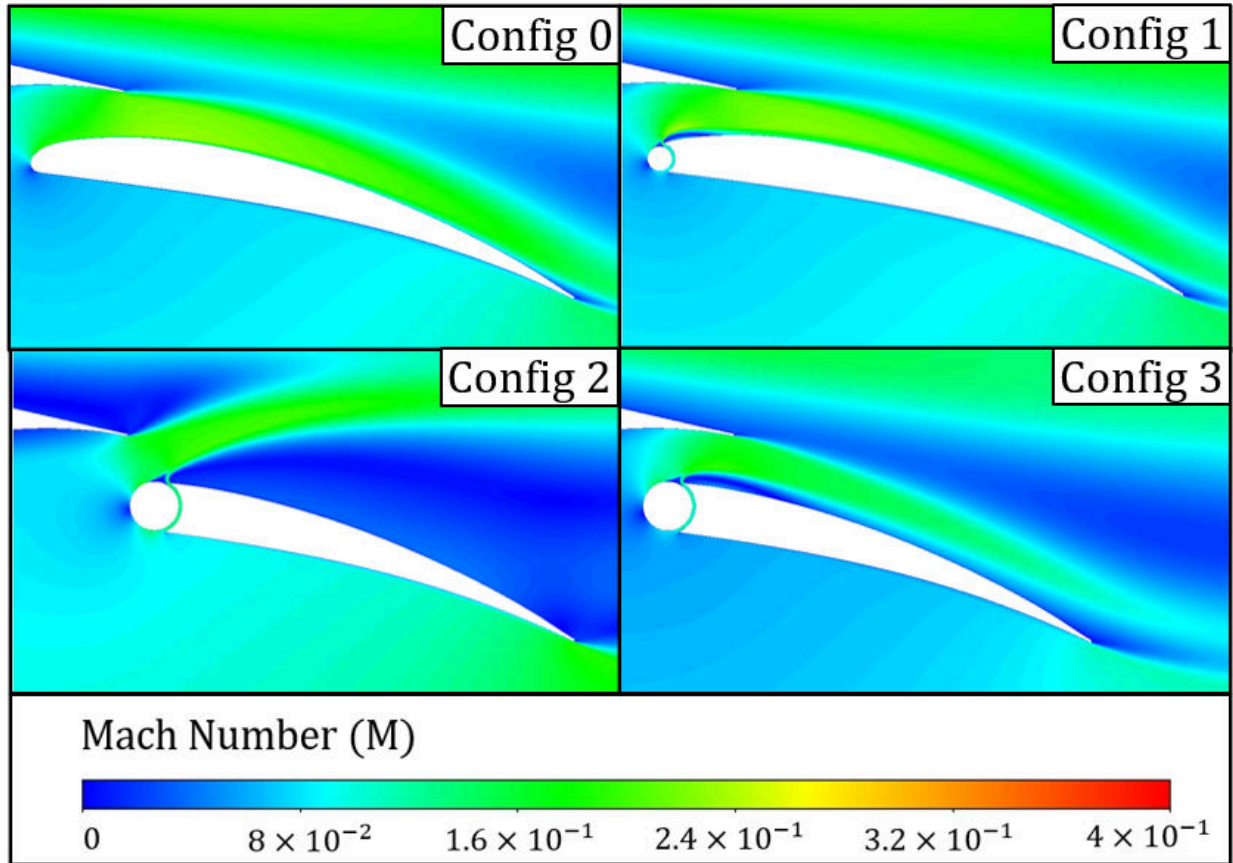


Fig. 9 Contours of the four configurations without rotation of the cylinder

1. Configuration 1

By focusing on the contours of the first configuration (Fig.10), it was shown that the rotation of the cylinder allowed to suppress the separation bubble from a ratio U_c/U equal to 1. However, no improvement was noticeable when the cylinder speed was increased up to U_c/U equal to 4.

In all velocity ratios studied, Fig. 11 suggests that no benefit in lift was obtained compared to the baseline experimental data. Furthermore, if the lift obtained with the ratios 0 and 0.5 was clearly lower than that of the experiment because of the separation bubble, ratios 1 to 4 made it possible to return to the level of the experiment and the curves were practically superimposed. Thus, beyond a ratio of 1, the effect of the rotation speed of the cylinder was negligible. Also, no delay in the stall was observed regardless of the ratio used.

It was observed that regardless of the cylinder speed for U_c/U ratios from 0 to 4, the drag was in the same order of magnitude. Therefore, as the drag and the lift obtained with or without the cylinder remained similar, no improvement over the lift-to-drag ratio was observed (Fig.12).

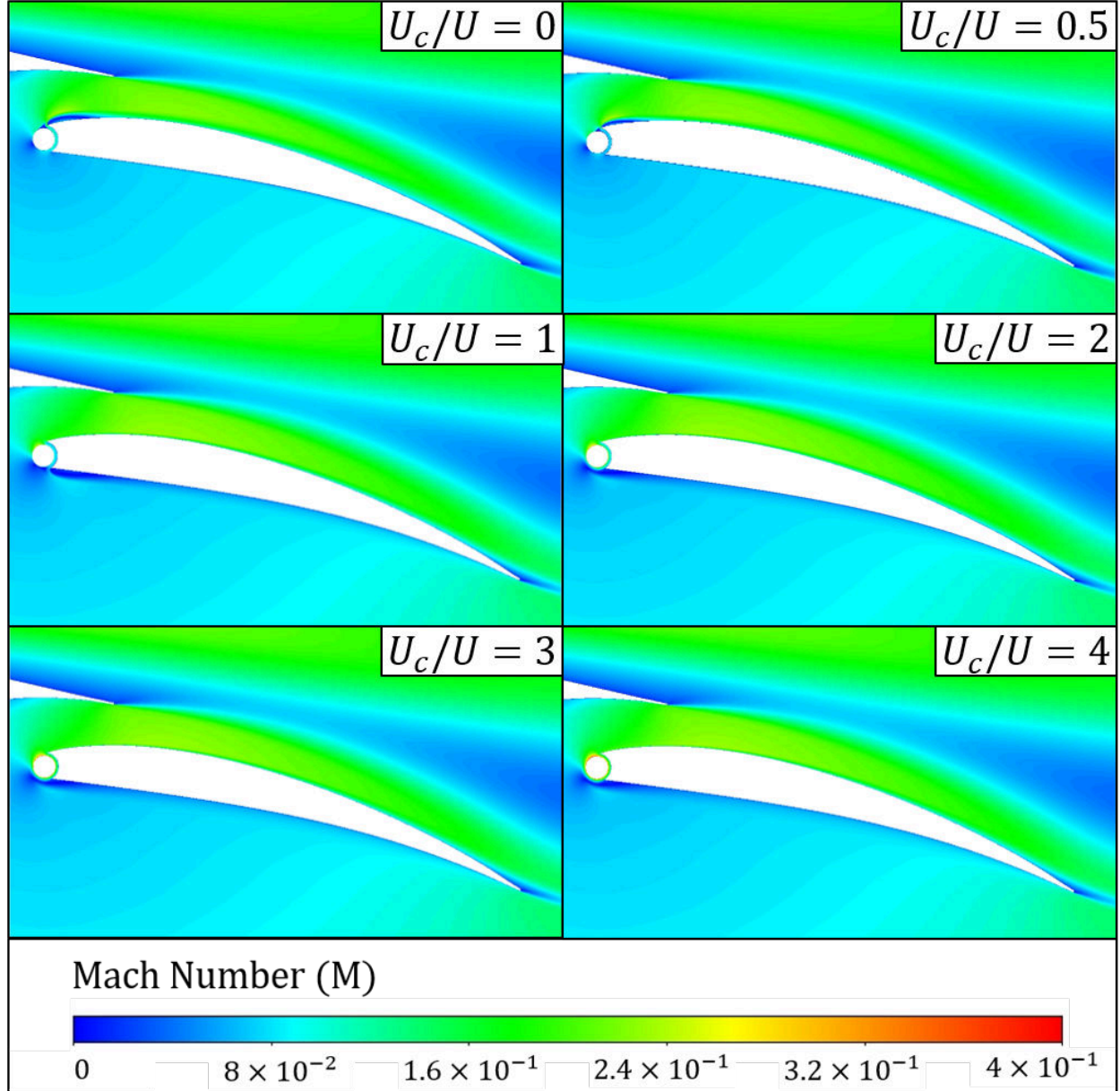


Fig. 10 Configuration 1 contours / $\alpha=13.1^\circ$

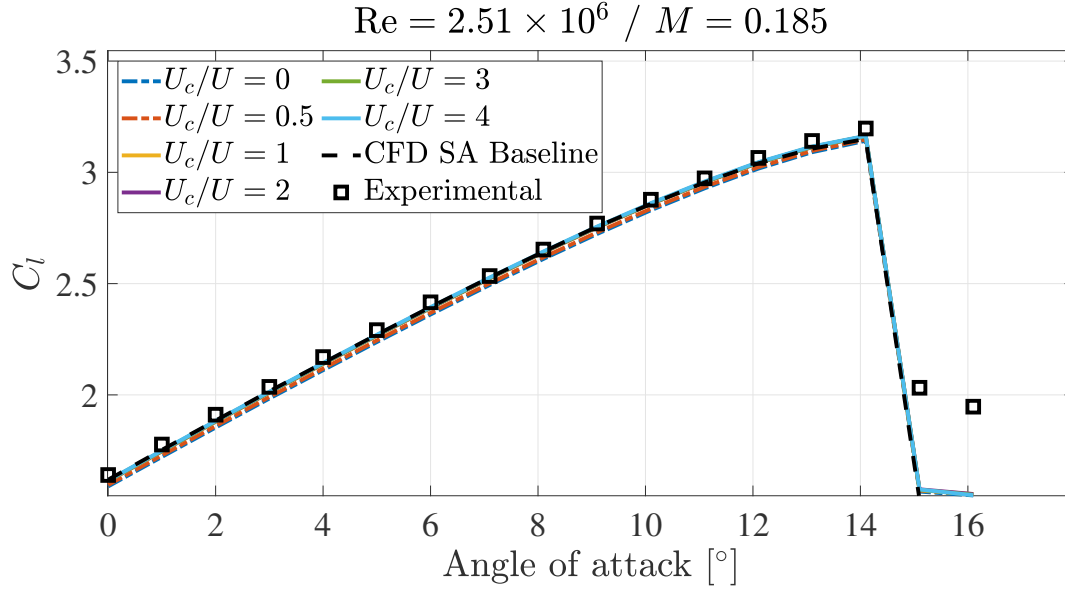


Fig. 11 Configuration 1 / C_l vs α

Note: The yellow ($U_c/U = 1$), purple ($U_c/U = 2$), green ($U_c/U = 3$) and light blue ($U_c/U = 4$) curves are merged into a single curve represented by the light blue curve. The same is true for the two dotted curves ($U_c/U = 0$ and $U_c/U = 0.5$), which are also merged into a single line.

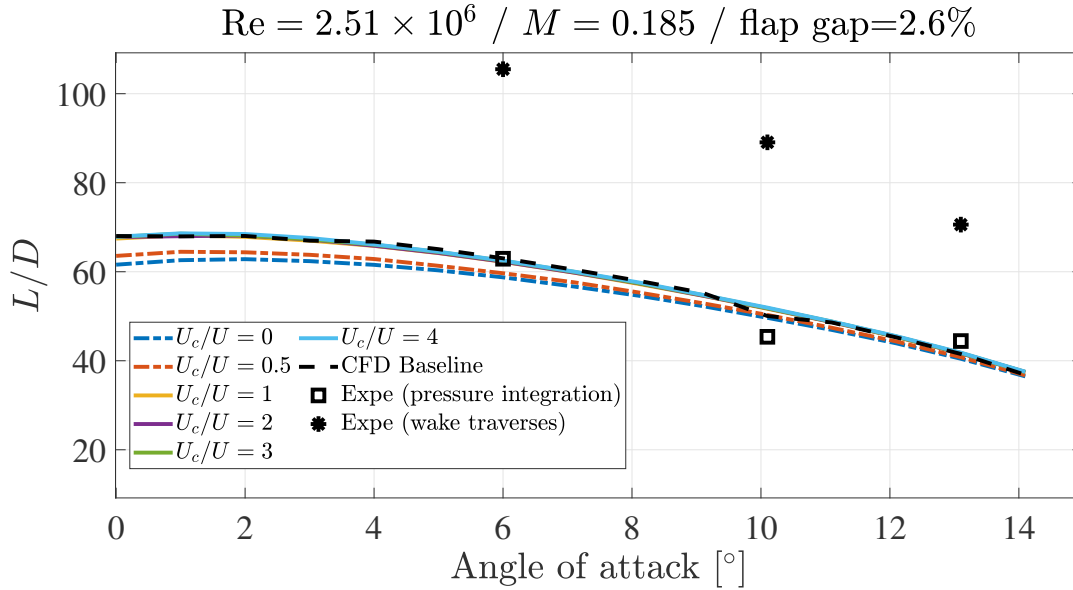


Fig. 12 Configuration 1 / L/D vs α

Note: The yellow ($U_c/U = 1$), purple ($U_c/U = 2$), green ($U_c/U = 3$) and light blue ($U_c/U = 4$) curves are merged into a single curve represented by the light blue curve and fitting perfectly the CFD baseline.

2. Configuration 2

Contour results over the second configuration (Fig.13) showed a large separation on the upper surface of the flap, due to its position and its geometry. In this configuration, it was shown that the rotation of the cylinder allowed to control the separation by reattaching the boundary layer at high cylinder speed ($U_c/U = 2$ & 3) and reducing it at low cylinder speed ($U_c/U = 1$). This aspect was recognized as the re-energizing of the boundary layer. Additionally, the velocity of the flow on the upper surface of the cylinder increased with the speed of the cylinder. This resulted in a low pressure region translating into lift generation due to Magnus effect. Finally, it was found that the rotation of the cylinder generated a small re-circulation bubble and a tiny region of high pressure (low velocity) on the lower surface of the flap translating an over-pressure beneficial to the lift. At all angles of attack, an average gain in lift of 30 % was perceived with the rotation of the cylinder compared to the static case. However, through Fig. 14 it was shown that no gain in lift over the experimental results and the baseline (without cylinder) was achieved regardless of the chosen cylinder speed. Indeed, it was stated that the large separation induced by the cylinder affected the performance of the flap, so that even with the cylinder in rotation, the results obtained were worse than those of the initial configuration with losses ranging from 24.04% to 47.86% over the experimental values. Nevertheless, as it was found for previous contour studies that the rotation of the cylinder helped to reattach the flow, the baseline was nearly recovered with ratios U_c/U equal to 2 and 3. In regards to the stall, it was found that the discontinuity introduced with this configuration deteriorated the lift capabilities of the aerofoil at high angles of attack. Indeed, whatever the cylinder speed studied, it was shown that the stall occurred earlier with the cylinder (Config. 2) than without (Config. 0 / baseline).

Finally, with this configuration, minor gain in performance over the drag was obtained but without any improvement over the lift-to-drag ratio (Fig. 15)

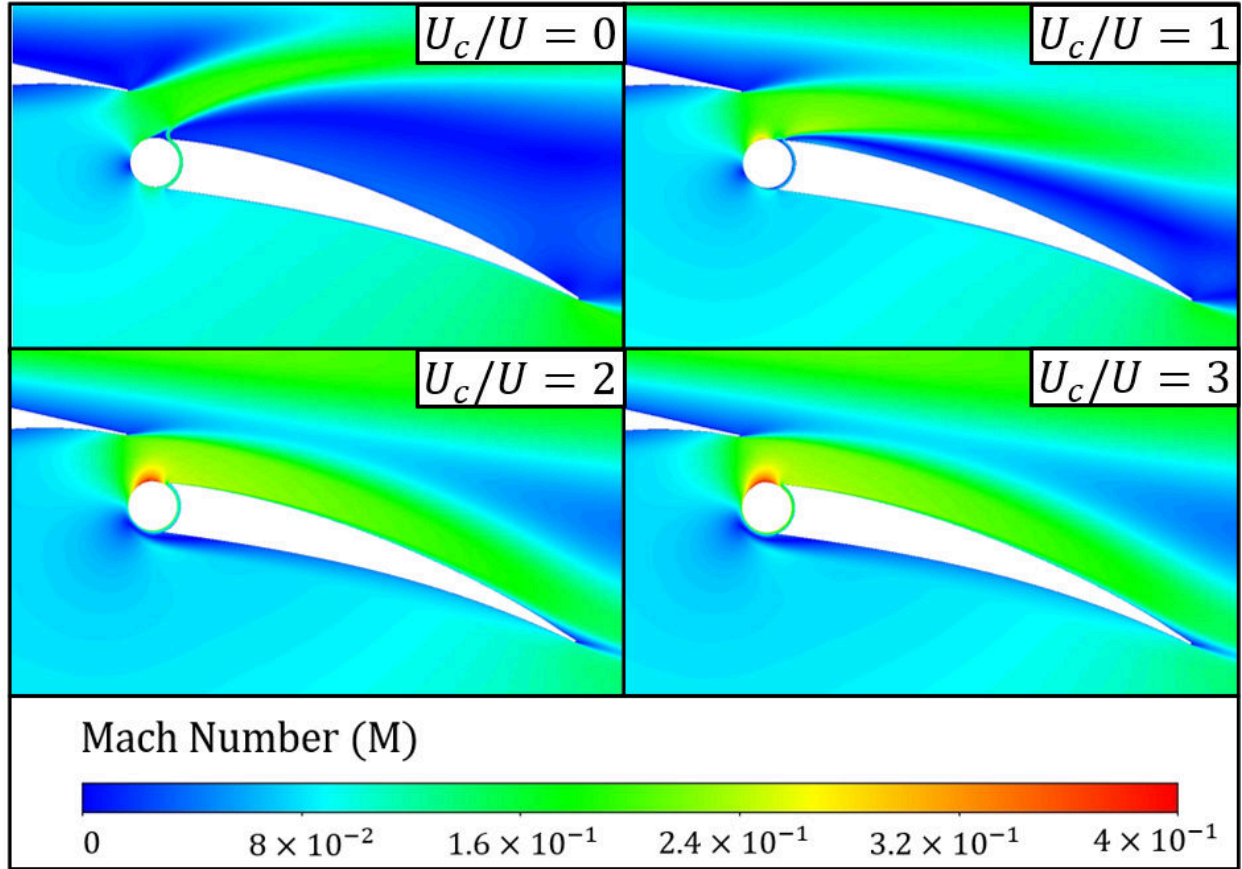


Fig. 13 Configuration 2 contours / $\alpha=13.1^\circ$

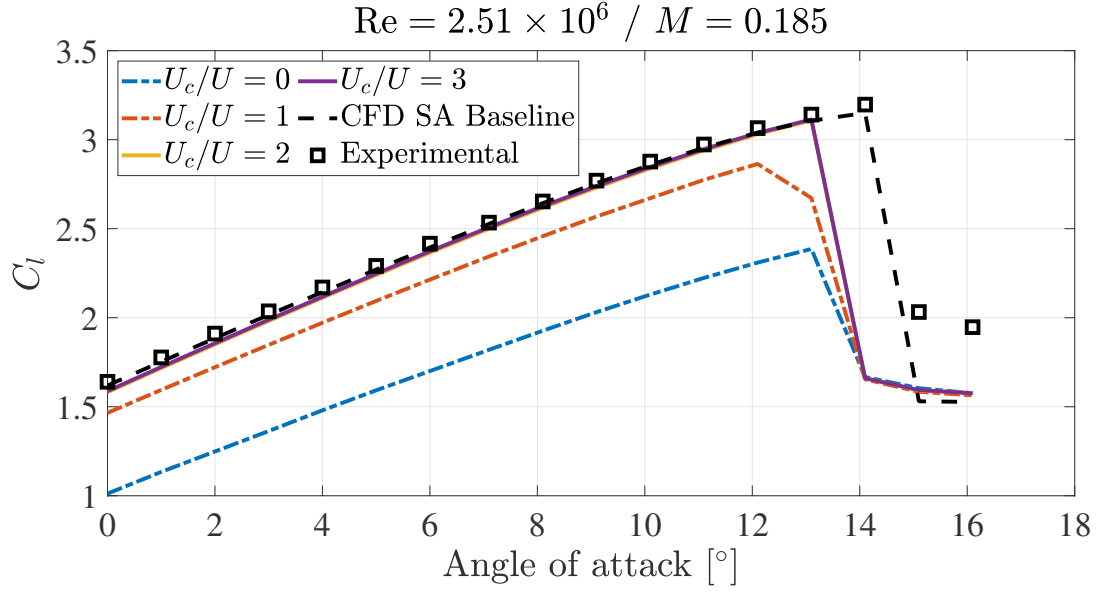


Fig. 14 Configuration 2 / C_l vs α

Note: The yellow ($U_c/U = 2$) and purple ($U_c/U = 3$) curves are merged into a single curve represented by the purple curve.

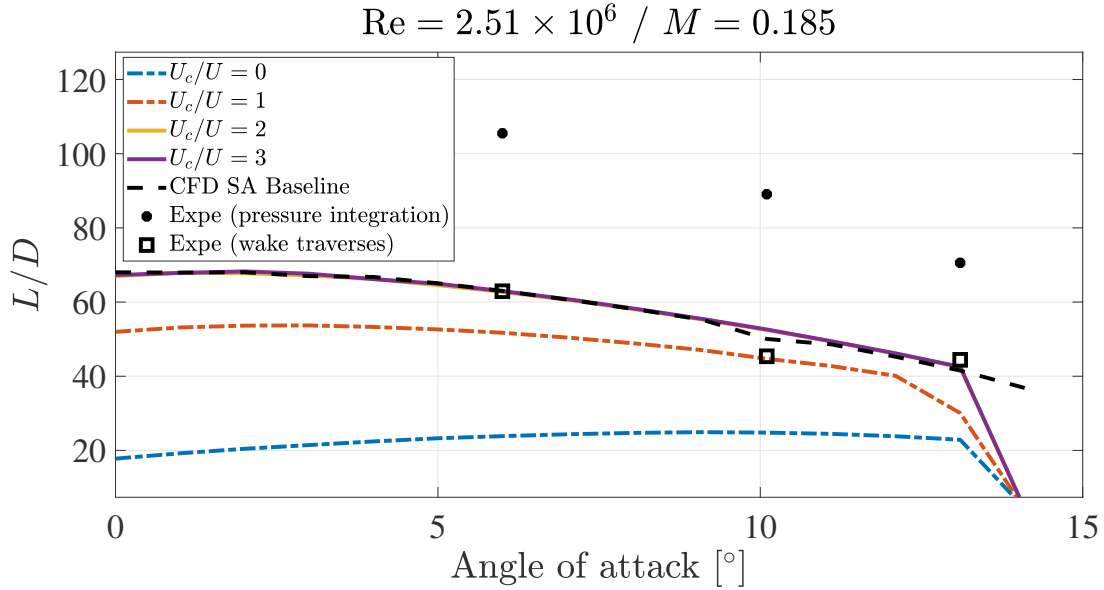


Fig. 15 Configuration 2 / L/D vs α

Note: The yellow ($U_c/U = 1$) and purple ($U_c/U = 2$) curves are merged into a single curve represented by the purple curve.

3. Configuration 3

It was shown through the contours of the third configuration that, as in the first configuration, the position of the flap allowed to avoid flow separation. Thus, it has been suggested by results presented in Fig.16 that only a separation bubble due to the new geometry of the flap was visible when the cylinder was not rotating impacting the smooth development of the boundary layer on the upper surface of the flap. Nevertheless, this bubble was totally suppressed by the effect of the rotating cylinder when reaching a ratio U_c/U equal to 2. As in Config. 2, with the cylinder rotating at high speed ($U_c/U = 2$ & 3), a region of high velocity was generated on the upper surface of the cylinder. This was translating an effect of suction due to low pressure and therefore, more lift was generated. This effect was recognized as the Magnus effect on a rotating cylinder. Finally, by looking at the trailing edge, it was seen that the spinning cylinder was reducing the separation at this point leaving only the wake generated by the sharp edge of the flap. From the Fig.17, it was found that none of the velocity ratios tested succeeded to recover the baseline and the experimental results. As a consequence, no additional lift over the original configuration was created from this one. However, it was noted that the reattachment of the boundary layer from the case with no rotation of the cylinder and the cases known as ratios 2 and 3 allowed to create some additional lift even if the baseline was not recovered. At its maximum, this benefit was equal to 3.82%. Finally, concerning the stall, no change between the experimental results, the baseline and the rotational cases was observed.

Finally, for all velocity ratio U_c/U studied, drag results are in agreement with the CFD baseline. Therefore, as no gain in lift was observed, the lift-to-drag ratio of Config.3 was worse than the baseline regardless of the rotational speed of the cylinder (Fig.18).

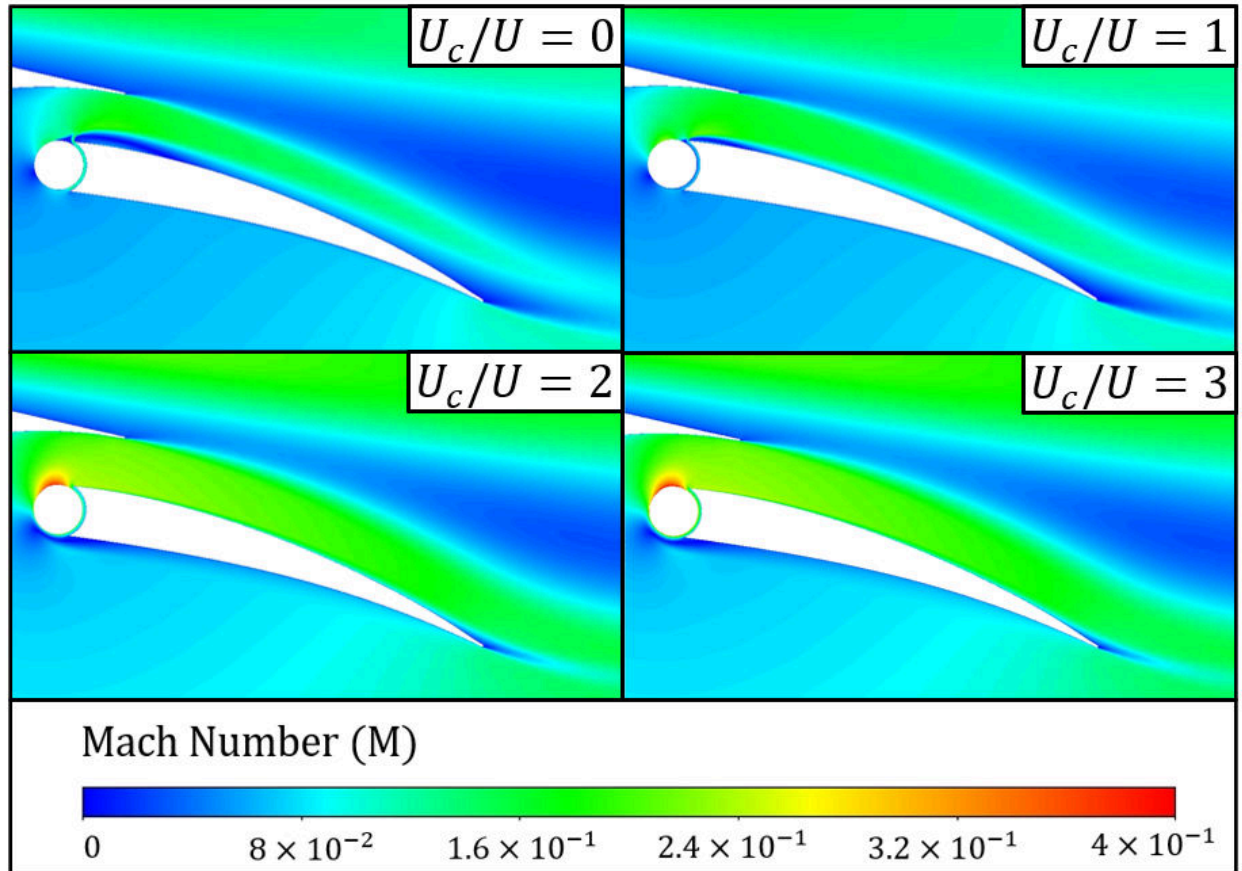


Fig. 16 Configuration 3 contours / $\alpha=13.1^\circ$

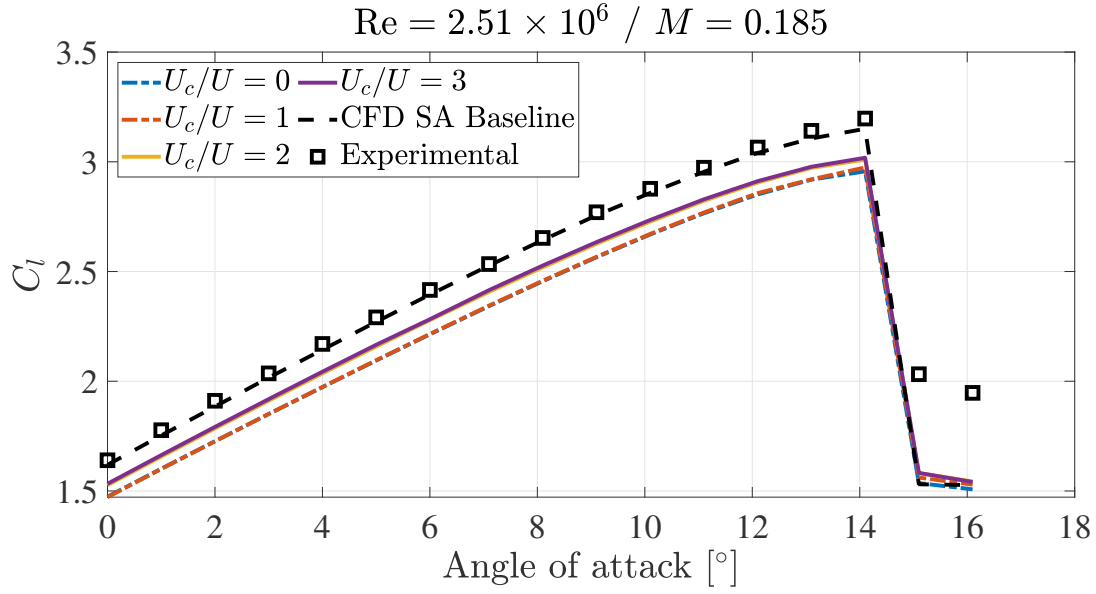


Fig. 17 Configuration 3 / C_l vs α

Note: The yellow ($U_c/U = 2$) and purple ($U_c/U = 3$) curves are merged into a single curve represented by the purple curve.

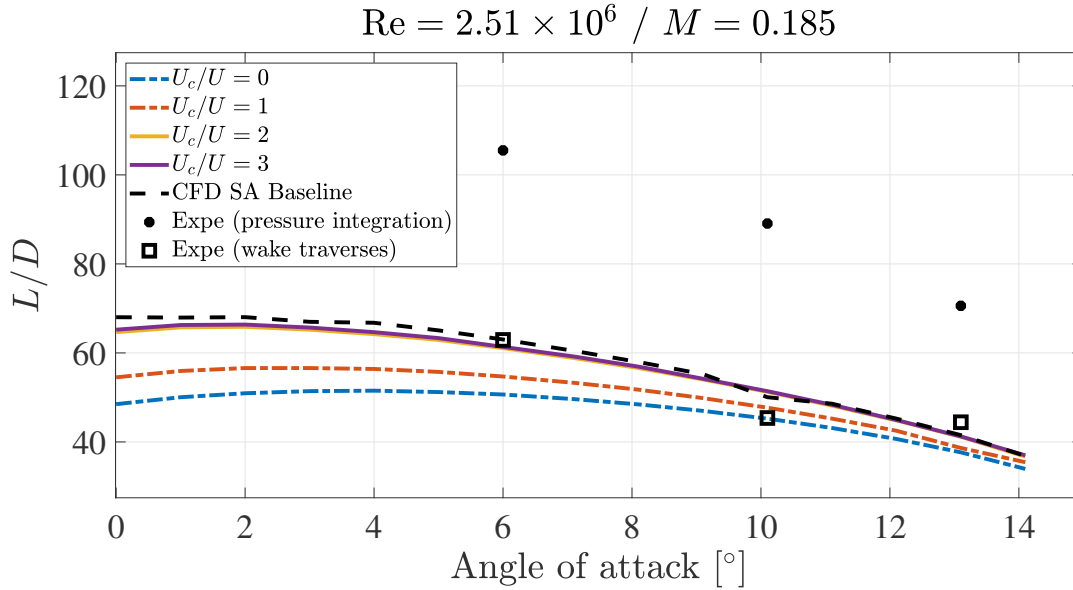


Fig. 18 Configuration 3 / L/D vs α

Note: The yellow ($U_c/U = 1$) and purple ($U_c/U = 2$) curves are merged into a single curve represented by the purple curve.

B. NLR 7301 with flap and trailing edge cylinder

By comparing the contours of the initial configuration without flap and the contours of Config. 4 with no rotation of the flap trailing edge cylinder (Fig. 19), it can be seen that a discontinuity was introduced in the geometry due the cylinder. Indeed, with this new geometry, the lower and upper surfaces of the flap were not merging on a single point at the trailing edge. Therefore, the circulation around the flap was modified and a large re-circulation region was produced behind it. The flow was not affected by the cylinder at the leading edge of the flap.

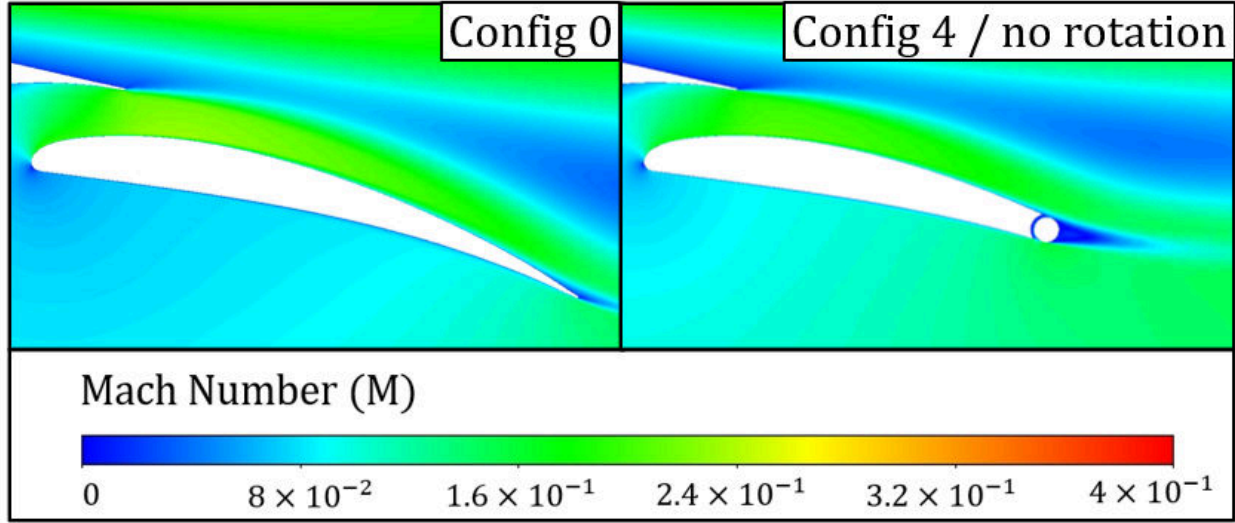


Fig. 19 Contours of the initial configuration and Configuration 4 with no rotation of the cylinder / $\alpha=13.1^\circ$

On the other hand, Fig.20 illustrates that the rotation of the cylinder allows flow deflection downwards in a clockwise direction, whilst also creating a high velocity region on the top of the cylinder. It can be observed that on the lower surface of the flap, a re-circulation region was generated due to counter rotating vortices producing a stagnation point downstream of the trailing-edge and becoming larger and larger with the cylinder speed. In this case it appeared that the Kutta condition was altered at the trailing edge resulting in a probable gain in lift. Additionally, by changing the circulation around the trailing edge of the flap it was found that some additional lift could be created due to the generation of suction on the upper surface of the cylinder and a region of over-pressure on its lower surface. It was recognized that this suction effect began to be present at high cylinder speeds (starting from $U_c/U = 4$ to $U_c/U = 6$). Finally, it was seen that no major effect appeared on the leading edge of the cylinder. No separation was seen on the flap with the initial configuration and, therefore, the introduction of a device at its trailing edge did not modify the behaviour of the flow around the leading edge of the flap.

Fig.21 suggests that a ratio U_c/U of 4 was required to recover the baseline. Starting from this cylinder velocity, it was possible to obtain a gain in lift over the experimental data. Indeed, it was shown that for a ratio U_c/U equal to 5, a gain up to 10.68% was obtained before the stall. In the same way, for a ratio of 6, a gain in lift comprises up to 18.64% was found. However, the lift generated with the velocity ratios of 0 to 3 was much lower than the one generated with the initial configuration. Concerning the stalling incidence, an increase of 1° was found when the cylinder was not rotating or at low speed ($U_c/U = 1$). Nevertheless, for the others cylinder velocity tested, the stall angle remained the same as for the baseline and the experimental data.

Concerning the drag, it was shown that for ratios U_c/U lower than 4 (when it recovered the baseline), the drag was much lower than the baseline (up to 28.72%). However, additional drag was generated with the cylinder spinning at high velocity ($U_c/U = 5$ and 6). What was interesting to note with the study of the drag was that when the speed of the cylinder allowed to obtain a higher lift than the one obtained for the baseline it also generated more drag and conversely, when the speed of the cylinder was too low to generate much lift, little drag was obtained with no benefit regarding the lift-to-drag ratio (Fig. 22).

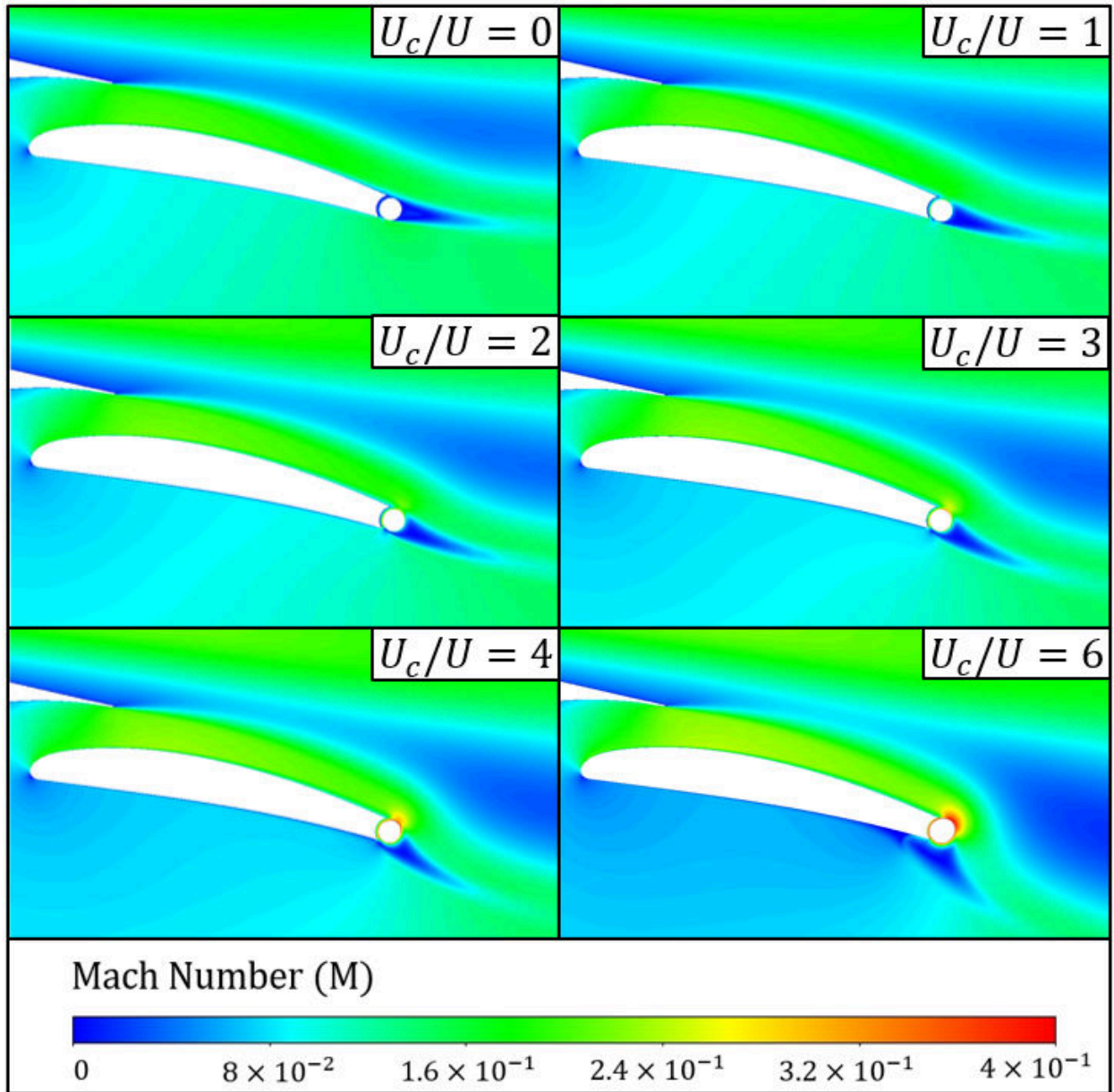


Fig. 20 Configuration 4 contours / $\alpha=13.1^\circ$

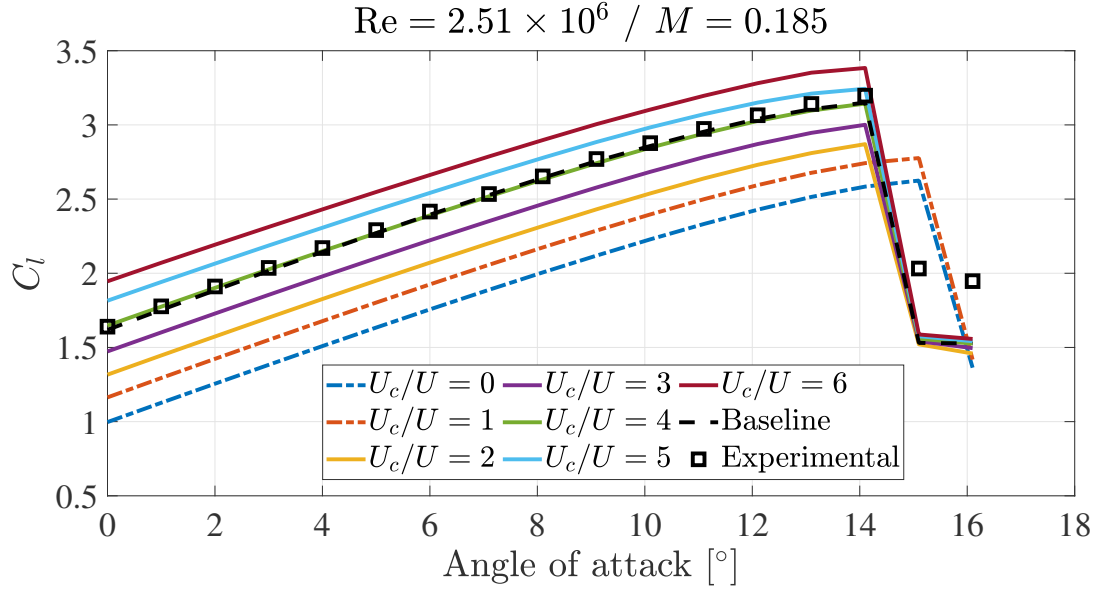


Fig. 21 Configuration 4 / C_l vs α

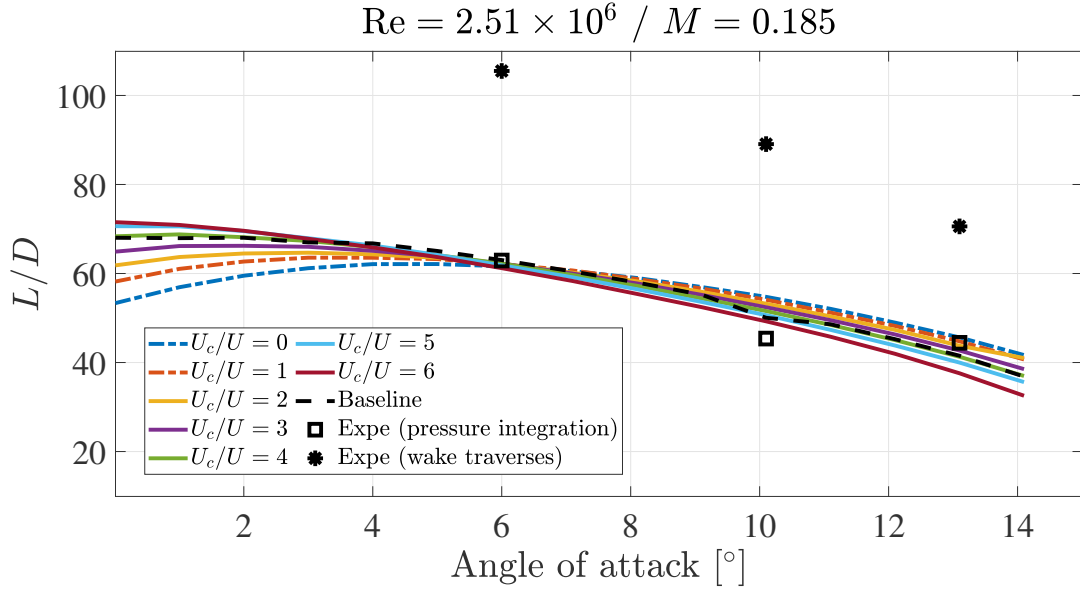


Fig. 22 Configuration 4 / L/D vs α

The figures presented above highlighted the fact that with the trailing edge spinning cylinder system, a cylinder velocity of at least four times the free-stream fluid speed was required to recover the baseline and fit the experimental data. Thus, it seemed relevant to evaluate the minimum power required to obtain a gain in lift with respect to the experimental data and the baseline. Neglecting all friction, mechanical stresses and thermal stresses, it was shown using Fig.23 that an average power (over all angles of attack) of 536 W was required for the cylinder to reach a rotational speed four times the fluid speed in vacuum. After the stall, a maximum power of 605 W was required. Thus, this last study allowed to conclude that such a system was not feasible. Indeed, the power required to obtain a gain in lift was too high compared to the benefits that could be produced.

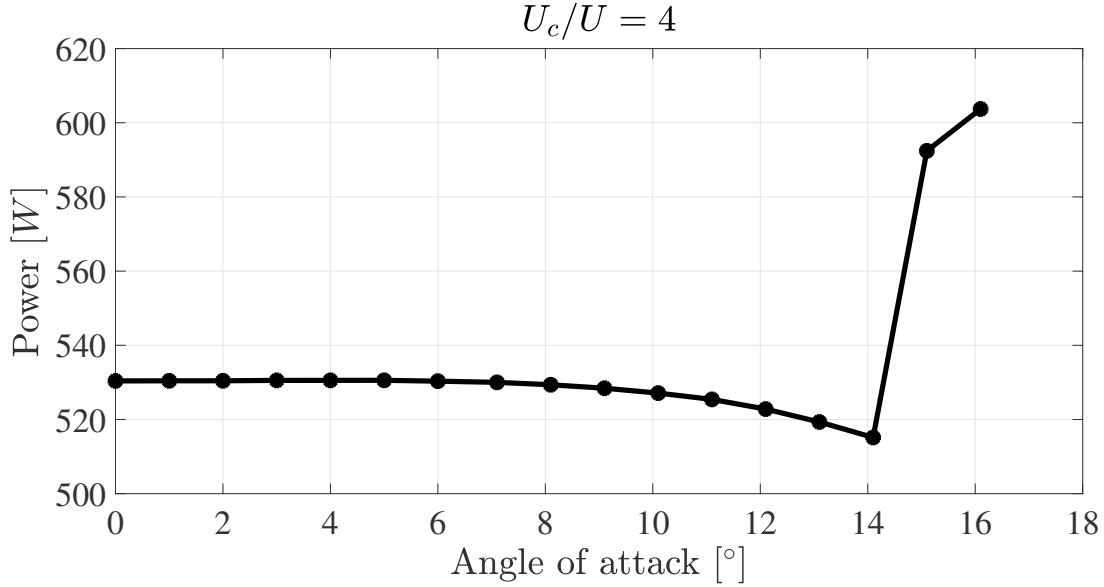


Fig. 23 Power estimation to recover the baseline / $U_c/U = 4$

V. Conclusion

The introduction of a cylinder, whatever its size and position, at the leading and trailing edges of the flap destroyed the lift capabilities of it by creating separation. Nevertheless, it was shown that the cylinder has the effect of re-energizing the flow around the flap allowing to reach again the lift levels of the baseline by reattaching the flow on the surface of the flap.

Concerning the leading edge devices, it has been shown that with a tiny cylinder, no gain in lift regarding to the Magnus and Coanda effects was obtained. However, by doubling the size of the cylinder, it was shown that some additional lift was generated by suction making use of the Magnus effect. Therefore, it was recognized that the size of the cylinder was a relevant parameter in lift generation. In every leading edge cylinder configurations, it was shown that no delay in the stall was obtained. Thus, the theory concerning the slat effect was not confirmed with this CFD study. In regards to the drag evaluation, it was shown that no improvement of the lift-to-drag ratio was obtained despite the different rotation speeds of cylinder tested. Indeed, it was not possible to recover the performance of the baseline and the experimental data. To conclude on this first part of the project, it was stated that the use of the cylinder can be considered when the flap is subject to large separation. However, for flaps like the one displayed on the NLR7301 where the separation is low, even negligible whatever the angle of attack, the leading edge cylinder is useless. Indeed, this kind of device does not allow to produce enough lift when using the Magnus effect and is only useful to reattach the boundary layer when flow separation occurs.

The second part of this study concerned the effect of a rotating cylinder on the trailing edge of the flap with the objective of proving the possibility of translating the lift curve upwards using this technology considering the flap effect. Through this study it was shown that the trailing edge cylinder allowed to change the flow circulation around the flap and to enhance the aerodynamic performance of the system. Indeed, it was found that with the trailing edge

cylinder configuration, it was possible to obtain a gain in lift up to 20% with the velocity ratios tested. However, this was only achievable when the cylinder was rotating at high velocity and the power required to perform at this speed was excessively high (around 600 W in vacuum). In addition, no benefit in the lift over drag ratio was observed saying that the system was not performing successfully.

Finally, to answer the research question over the feasibility of a spinning cylinder positioned at the leading and trailing edges of a flap for high-lift, it was demonstrated that such a system was barely feasible. Indeed, for the leading edge device, no major gain could be observed while the gain in lift obtained with the trailing edge device was too low compared to the resources required to implement such a system on a real wing.

References

- [1] Francannet, A., “Feasibility of a flap leading and trailing edges spinning cylinders for high lift,” Master’s thesis, Cranfield University, SATM, 2022.
- [2] Johnson, W., Tennant, J., and Stamps, R., “Leading-edge rotating cylinder for boundary-layer control on lifting surfaces,” *Journal of Hydronautics*, Vol. 9, No. 2, 1975, pp. 76–78. <https://doi.org/10.2514/3.48139>, publisher: American Institute of Aeronautics and Astronautics _eprint: <https://doi.org/10.2514/3.48139>.
- [3] Alvarez-Calderon, A., Arnold, F., Research, U. S. O. o. N., and Army, U. S., *Study of the Aerodynamic Characteristics of a High Lift Device Based on a Rotating Cylinder and Flap*, Technical report, Stanford University, Thermosciences Division Department of Mechanical Engineering, 1961.
- [4] Modi, V., “Moving Surface Boundary-Layer Control: a Review,” *Journal of Fluids and Structures*, Vol. 11, No. 6, 1997, pp. 627–663. <https://doi.org/https://doi.org/10.1006/jfls.1997.0098>.
- [5] Modi, V., and Triplett, B., “Moving surface boundary-layer control for aircraft operation at high angles of attack,” *41st Aerospace Sciences Meeting and Exhibit*, 2003, p. 221.
- [6] Van den Berg, B., and Oskam, B., “Boundary layer measurements on a two-dimensional wing with flap and a comparison with calculations,” *NLR MP 79034 U*, 1979. Publisher: Nationaal Lucht- en Ruimtevaartlaboratorium.
- [7] Roache, P. J., “Quantification of Uncertainty in Computational Fluid Dynamics,” *Annual Review of Fluid Mechanics*, Vol. 29, No. 1, 1997, pp. 123–160. <https://doi.org/10.1146/annurev.fluid.29.1.123>.
- [8] Haase, W., and al, e., *Euroval - an European Initiative on Validation of Cfd Codes: Results of the Ec/Brite-euram Project Euroval, 1990-1992*, softcover reprint of the original 1st 1993 ed. édition ed., Friedrich Vieweg & Sohn Verlagsgesellschaft mbH, Braunschweig, 1993.
- [9] Spalart, P., and Allmaras, S., “A one-equation turbulence model for aerodynamic flows,” *30th Aerospace Sciences Meeting and Exhibit*, American Institute of Aeronautics and Astronautics, Reno,NV,U.S.A., 1992. <https://doi.org/10.2514/6.1992-439>, URL <https://arc.aiaa.org/doi/10.2514/6.1992-439>.



DSSPACE

<https://dspace.org/>

Feasibility of a spinning cylinder on the leading and trailing edges of a flap for high lift

Francannet, Antoine; Prince, Simon A.; Di Pasquale, Davide; Proenca, Anderson R.

2023-01-19

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2023-0053>

Francannet A, Prince S, Di Pasquale D, Proenca A. (2023) Feasibility of a spinning cylinder on the leading and trailing edges of a flap for high lift. In: AIAA SciTech Forum 2023, 23-27 January 2023, National Harbor, Maryland, USA. Paper number AIAA 2023-0053