Annals of the Academy of Romanian Scientists
Series on Engineering Sciences
ISSN 2066-8570 Volume **15**, Number **1**/**2023**

WINGLETS EFFECT ON THE AEORDYNAMICS OF AIRCRAFT WING; COMPUTATIONAL STUDIES

Marcel ILIE[1], Augustin SEMENESCU[2]

Rezumat. *Reducerea fortelor de frecare ale avionului cu aerul a fost de interes major pentru industria aeronautica de mai mult timp. De aceea diferite metode de control ale fluidelor au fost folosite cum ar fi metode pasive si active de control ale fluidelor.Aripioarele folosite la capatul aripilor au fost studiate experimental si computational. Studiile experimentale sunt costisitoare si pretind echipamente specializate. De aceea metodele computationale sunt folosite in studiul prezent folosind metoda de simulare a turbioanelor mari.Studiul prezent arata ca folosirea aripioarelor reduce separarea aerului si fortei de drag, in timp ce contribuie la cresterea fortei de lift.*

Abstract. *Aircraft drag reduction has been of interest to aerospace industry for long time. Therefore, in the past decade different flow control techniques have been used such as active and passive flow control techniques. Winglet design has been explored in the past both experimentally and computationally. Experimental studies pose significant challenges due to the wing-span and costly equipment and facilities. Therefore, this study concerns the computational studies of winglet design using the large-eddy simulation approach. The results show that the winglet design reduces the flow separation and drag force, while the pressure and lift force increases.*

Keywords: Aircraft, Winglet, Wing-tip Vortex, Large-Eddy Simulation

DOI https://doi.org/10.56082/annalsarscieng.2023.1.138

1. Introduction

Aircraft drag reduction has been of interest for decades to the aerospace industry, particularly related to the aerodynamic performance and cost of operations. The pressure difference between the upper and lower surfaces of the wing generate the tip-vortex as shown in Figure 1. The wing tip-vortex shedding generates high drag

---

[1] PhD, Assistant Professor: Dept. of Mechanical Engineering, Georgia Southern University, Statesboro, GA 30458, USA, e-mail: milie@georgiasouthern.edu

[2] PhD, Professor, Dept. of Material Sciences, University Politehnica Bucharest, Bucharest, Romania, augustin.semenescu@upb.ro; Corresponding Member of Academy of Romanian Scientists

forces which affects the aerodynamic performance of the aircraft. The wing tip-vortex shedding also generates aeroelastic effects of the aircraft wing which can affect the structural integrity of the wing, leading to irreversible damage. Therefore, various approaches and methods have been studied including passive and active flow control techniques to minimize the effect of the wing-tip vortex shedding.

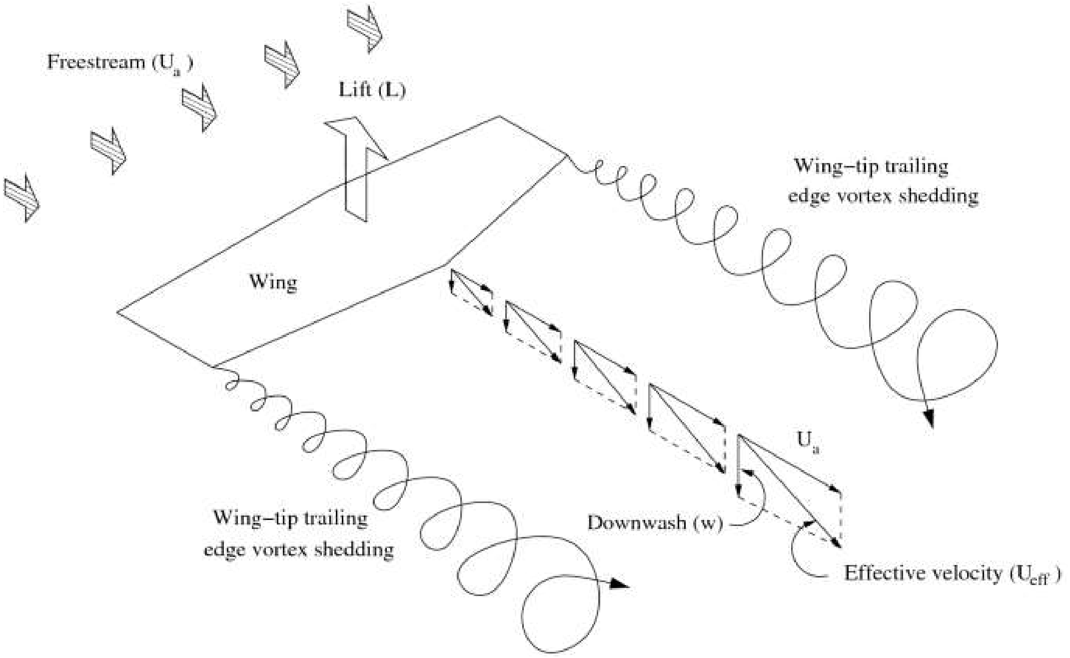


Fig. 1. Wing-tip vortex [6]

One of the methods investigated and implemented is the use of winglets in fixed-wing aircraft. Winglet are used for the purpose of reducing the aircraft drag and hence increasing the aerodynamic performance. Figure 2 presents the schematic of two different aircrafts, one using standard wing configuration while the other one using the winglet design. The use of winglet design in fixed-wing aircraft leads to the reduction of the tip vortex formation and its impact on the aircraft drag.

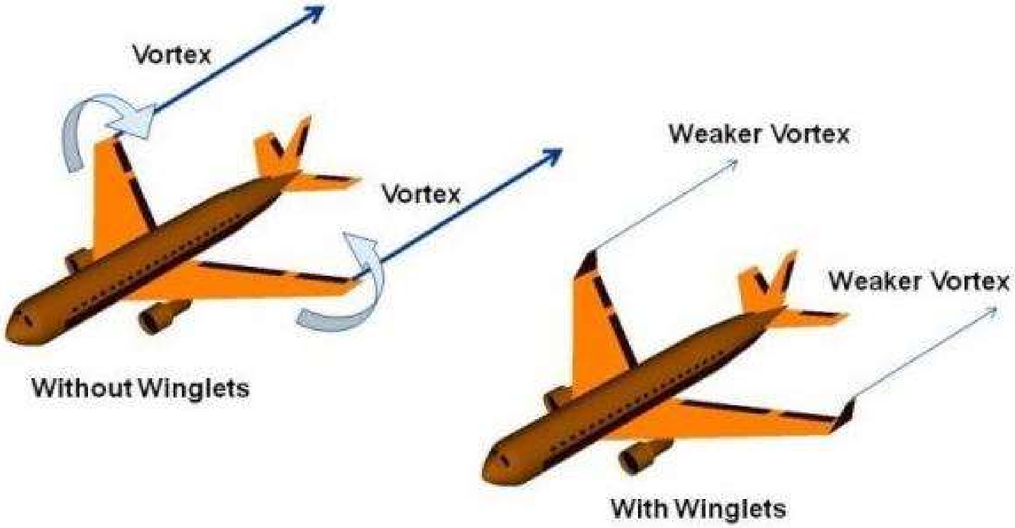


Fig. 2. Fuel injection scheme for scramjet combustion [7]

2. Background

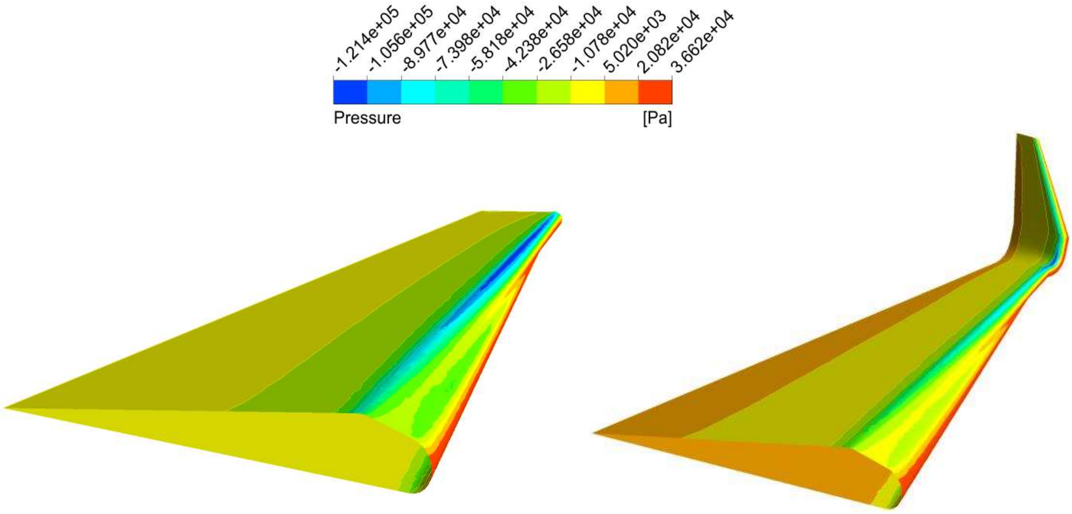
Computational methods were employed in the past in the computation of fluid-structure interaction phenomenon. The first methods employed the vortex panel methods [3]. However, these methods assume that the flow is inviscid and therefore, it cannot capture the flow separation and predict the drag coefficient. Therefore, viscous methods must be employed in the computation of high Reynolds number flows [2-20]. Studies showed that the numerical approach based on the Reynolds-averaged Navier-Stokes (RANS) are not suitable for time-dependent flow dynamics such as the unsteady aerodynamics of compressor/turbine stage [15]. Direct numerical simulations (DNS) also pose computational challenges due to the high-resolution grid size required by the DNS. Generally, DNS approach is not suitable for high Reynolds number flows and thus, alternative time-dependent solutions must be sought. Therefore, large-eddy simulation (LES) approach is a promising approach for the numerical computations of high-Reynolds number flows in a rotating-frame of reference [15].

3. Modeling

In the present research the computational studies are carried out using the large-eddy simulation (LES) approach. For more details on the computational modeling using LES, the reader is referred to [4].

4. Results and discussion

Figure 3 presents the pressure field at the wing surface for two wing design namely the standard-wing and winglet design. The analysis shows that the winglet design exhibits an increase of the pressure at the wing surface and thus, it is expected that this would lead to an increase of the lift coefficient. However, the pressure exhibits highest pressure for both designs at the leading-edge of the wing. The lower surface of the wing presents higher pressure for the winglet design compared with the standard wing and this would also lead to an increase of the lift force.

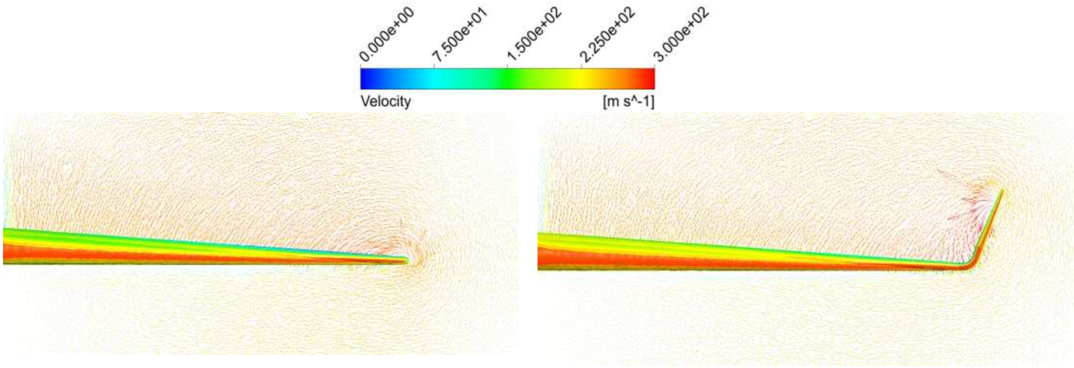


a. No-winglet b. Winglet

Fig. 3. Pressure distrubution at the wing surface

Figure 4 presents the velocity field for both wing design configurations. The analysis shows the presence of the tip-vortex at tip of both wings configurations.

However, the winglet configuration minimizes the strength of the tip-vortex and thus, it is expected that will reduce the drag force.



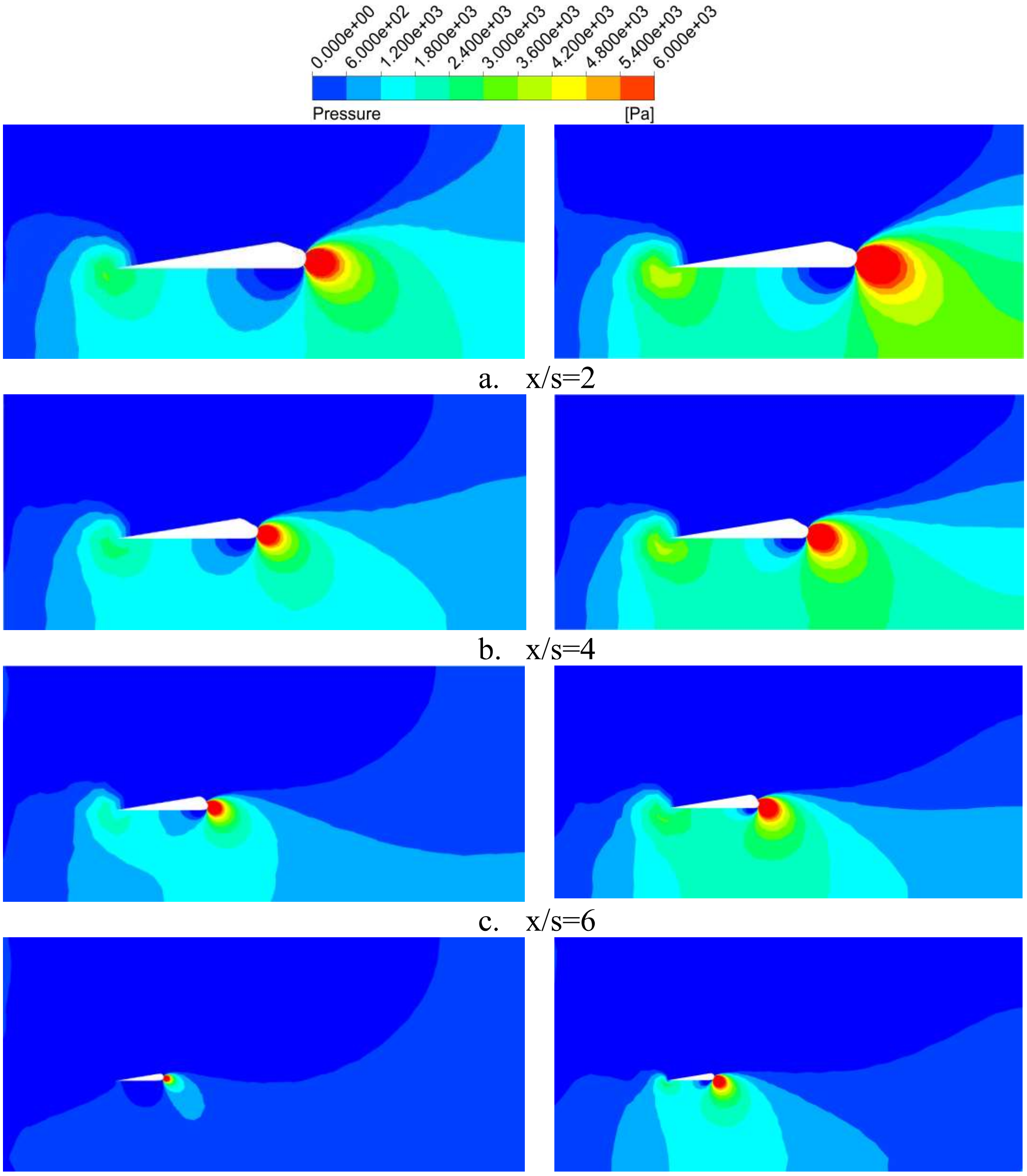
a. No-winglet b. Winglet

Fig. 4. Velocity vector field

The analysis of the velocity vector field also reveals that due to the higher pressure at the lower surface the airflow runs towards the tip of the blade. The pressure imbalance at the lower and upper surfaces of the wing cause the tip-vortex as shown in Figure 4.

It is expected that pressure imbalance to be reflected onto the near-field pressure levels and thus, this is presented in the following. Figure 5 presents the pressure field near the wing of the airplane. The pressure field is presented at four different spans of the wing based on the ratio x/s, where s represents the wing-span. Figure 5 presents the comparison of the pressure field for standard-wing and winglet design. The analysis shows that the pressure exhibits largest values, at the leading-edge of the wing, for both wing designs. However, the winglet design exhibits largest value compared with the standard-wing design and thus, it is expected that the lift aerodynamic coefficient to be higher for the winglet design.

It is expected that the velocity field would also affect the turbulence kinetic energy (TKE) and thus, this is presented in Figure 6. The analysis of the turbulence kinetic energy in Figure 6 shows that the winglet configuration also exhibits largest values of turbulence kinetic energy. The turbulence kinetic energy also contributes to the aerodynamic coefficients. The large values of the turbulent kinetic energy are observed in hub region of the wing.



a. x/s=2

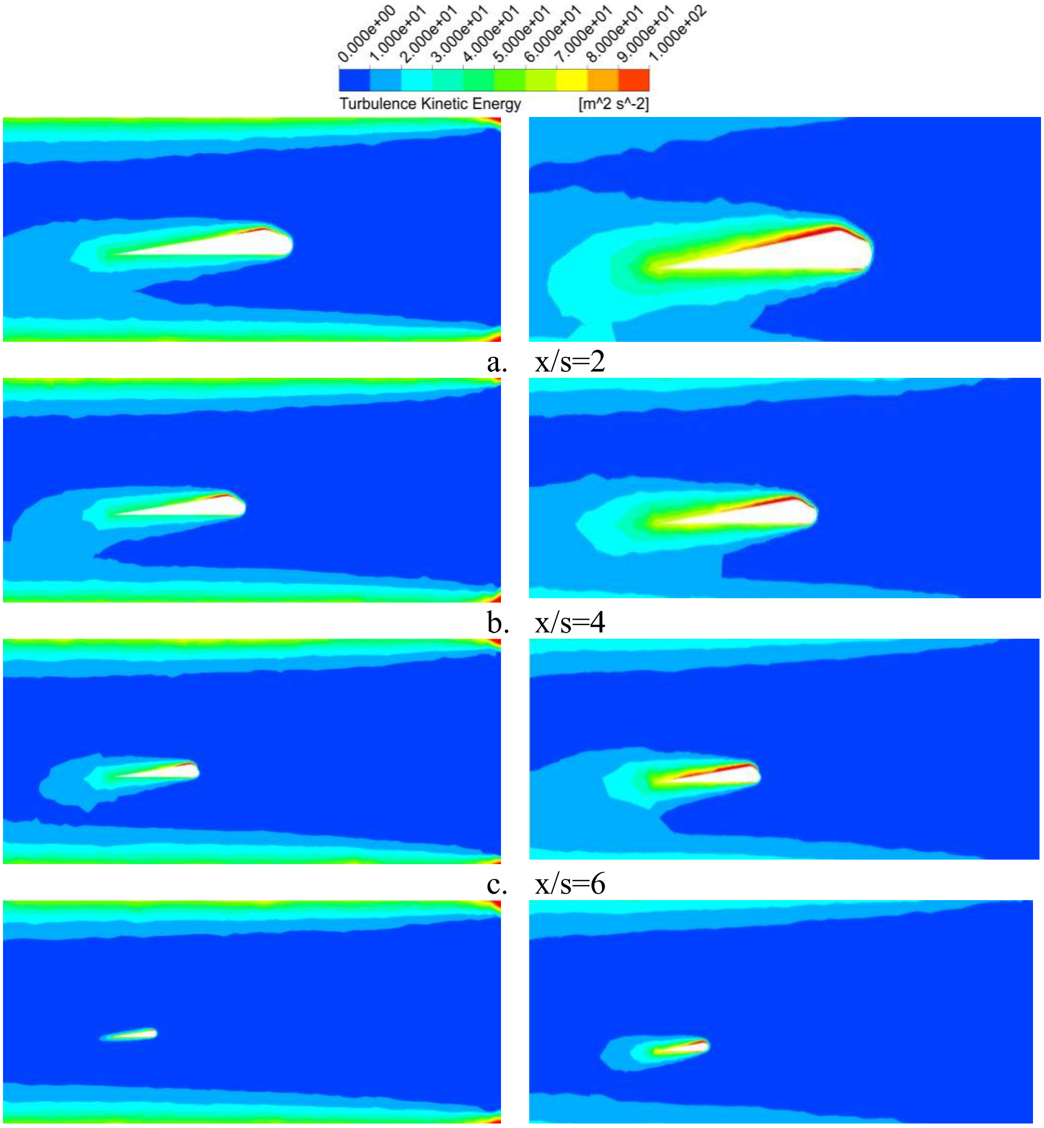
b. x/s=4

c. x/s=6

d. x/s=8

a. No-winglet b. Winglet

Fig.5. Pressure field distribution



a. x/s=2

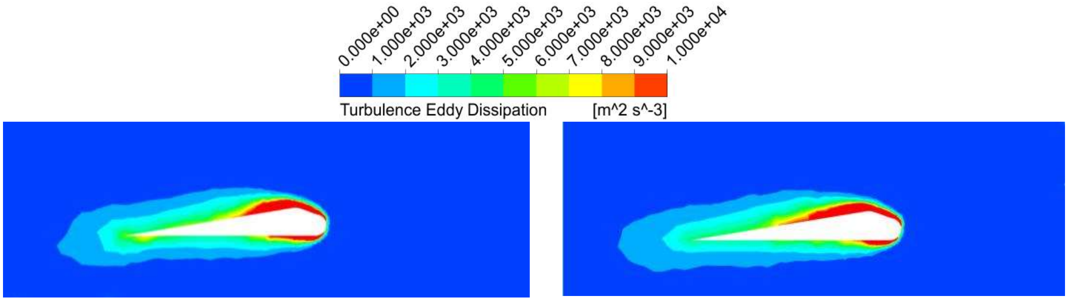
b. x/s=4

c. x/s=6

d. x/s=8

a. No-winglet b. Winglet

Fig. 6. Turbulence kinetic energy (TKE)



a. x/s=2

b. x/s=2

c. x/s=2

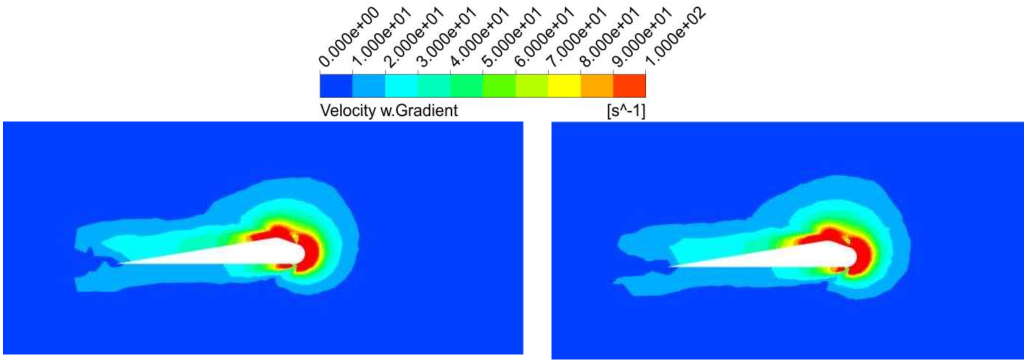
d. x/s=2

a. No-winglet b. Winglet

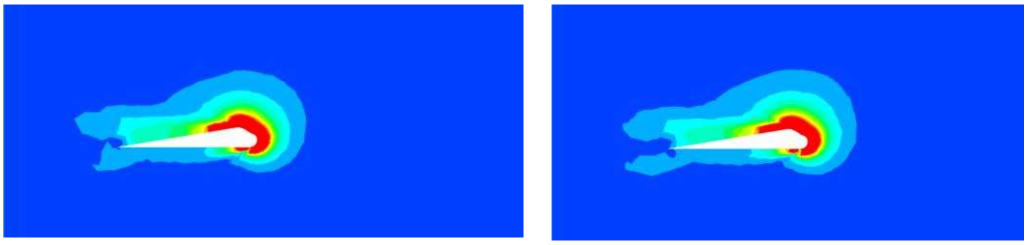
Fig.7. Turbulence eddy disipation

The turbulence eddy dissipation is an indicator of the small-scale of turbulence and thus, the largest values of the turbulent eddy dissipation are associated with the smallest turbulent eddies. The small-scale eddies are more prone to dissipation and thus, the largest values of turbulence eddy dissipation indicate the presence of the small-scale eddies.

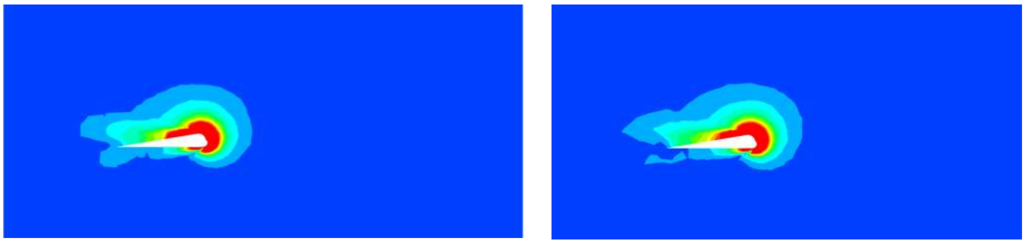
Figure 8 presents the velocity gradients in the direction of the flow. The analysis shows that the largest velocity gradients are encountered at the leading-edge of the wing and this is due to the accelerating flow. The upper surface of the wing is cambered and therefore, the flow is accelerating.



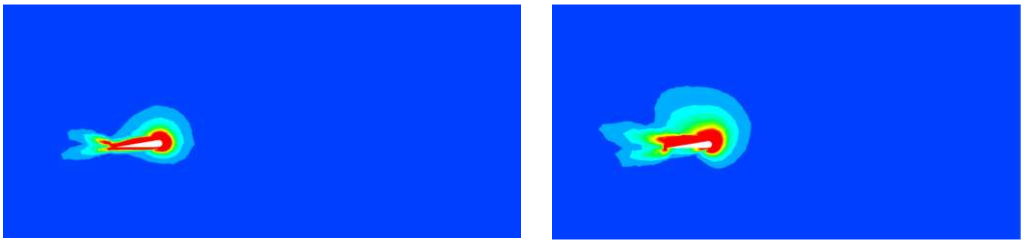
a. x/s=2



b. x/s=2



c. x/s=2



d. x/s=2

a. No-winglet b. Winglet

Fig. 8. Velocity gradients

Skin-fraction coefficient is a good aerodynamic parameter to assess the efficiency of the winglet design over the standard wing design. Therefore, the negative values of skin-friction coefficient indicates the flow separation while positive values of skin-friction coefficient represent flow attachment.

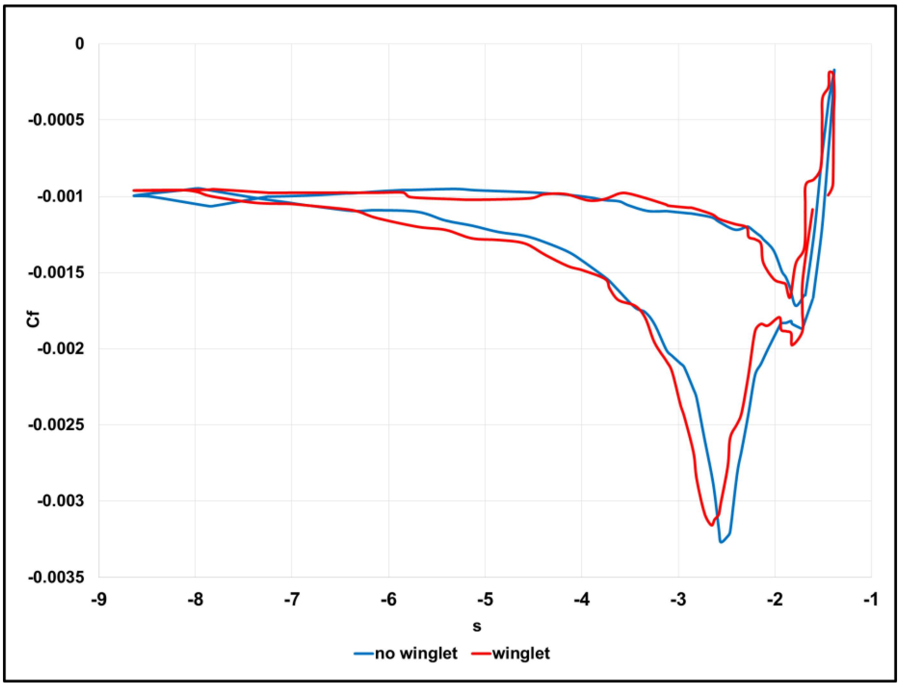


Fig. 9. Skin-friction coefficient at x/s=2

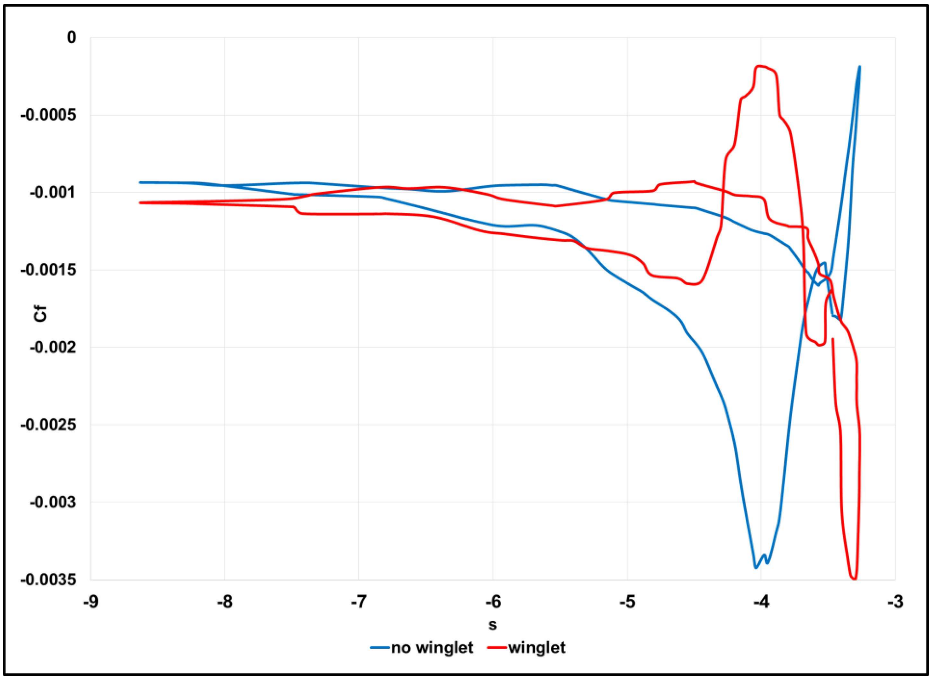


Fig. 10. Skin-frction coefficient at x/s=4

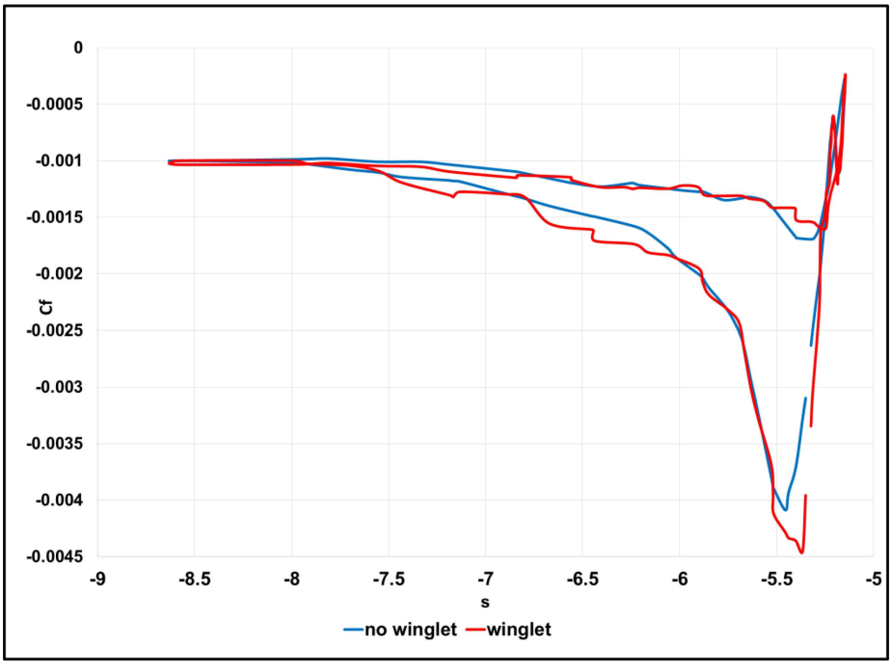


Fig. 11. Skin-frction coefficient at x/s=6

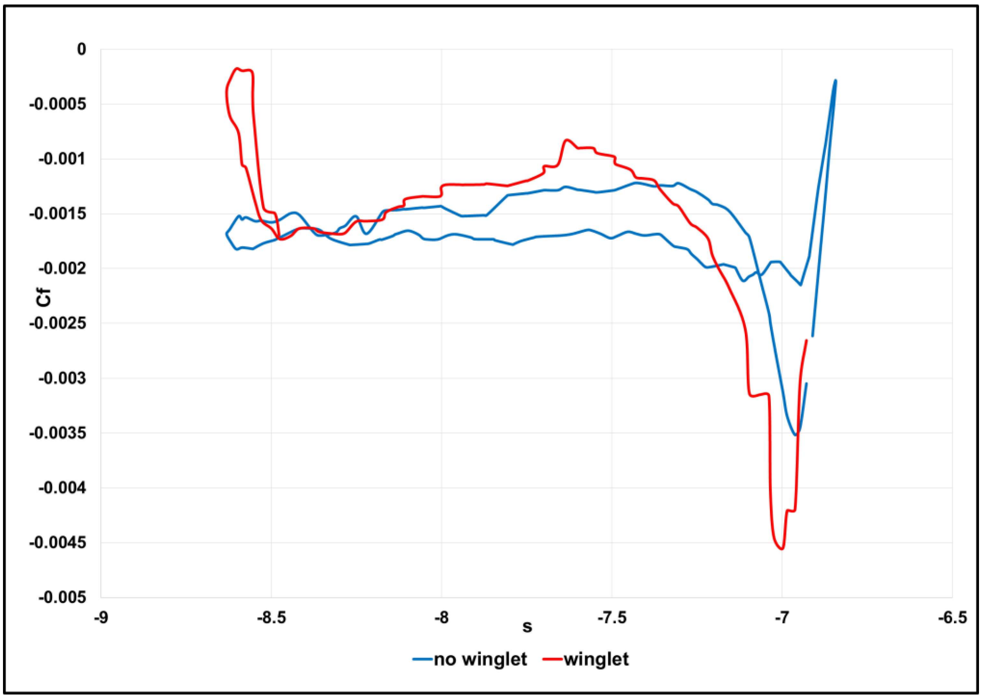


Fig. 12. Skin-frction coefficient at x/s=8

Figure 9 through 12 presents the skin-friction coefficient at four different locations along the wing-span. The study shows that overall, the winglet design reduces the flow separation and thus, it exhibits better aerodynamic performance compared with the standard-wing design.

Conclusions

An efficient computational approach, for the fluid-structure interaction, is developed for the computations of large Reynolds number flows. The developed model is employed for the computation of aeroelasticity effects in highly turbulent flows. The analysis of the flutter phenomenon shows that the pressure fluctuation increase the flutter phenomenon and it increases with the Reynolds number. The study also shows that there is a strong interaction between the vortices developed at the trailing-edge of the stator blade and the leading-edge of the rotor blade row. The frequency of the vortex shedding governs the blade-vortex interactions. Thus, the higher the vortex shedding frequency, the higher the vibrations of the rotor blades.

REFERENCES

[1] Urzay, J., Supersonic Combustion in Air-Breathing Propulsion Systems for Hypersonic Flight, Annual Review of Fluid Mechanics, 50:1, 593-627, 2018

[2] Galeazzo, F. C. C., Donnert, G., Habisreuther, P., Zarzalis, N., Valdes, R. J., and Krebs, W., Measurement and Simulation of Turbulent Mixing in a Jet in Crossflow, ASME J. Eng. Gas Turbines Power, 133(6), p. 061504, 2011

[3] M. Ilie, A. Semenescu, G. Liliana Stroe, S. Berbente, Numerical Computations of the cavity flow using the potential flow theory, Annals of the Academy of Romanian Scientists Series on Engineering Sciences, Volume 13, Number 2/2021, ISSN 2066-8570

[4] M. Ilie, Augustin Semenescu, Matthew Chan. "Computational studies of turbine- stage, with variable inlet temperature; comparison between LES and IDDES", AIAA AVIATION 2021 FORUM, 2021

[5] M. Ilie, Matthew Chan, Jackson Asiatico. "Combustion instabilities in backward-step premixed reacting flows; computational studies using LES", AIAA SCITECH 2022 Forum, 2022

[6] Kaushik, M. (2019). Finite Wing Theory. In: Theoretical and Experimental Aerodynamics. Springer, Singapore. https://doi.org/10.1007/978-981-13-1678-4_6

[7] https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/winglets/