ON AIRCRAFT PERFORMANCE IMPROVEMENT BY USING WINGLETS

B. Rasuo, V. Parezanovic and M. Adzic
Aeronautical Department, Faculty of Mechanical Engineering,
University of Belgrade, Belgrade, SERBIA

Keywords: *Winglet, LES, CFD, Fluent*

Winglets are all but common on today's airplanes. Being of various shapes and sizes, of conventional design or otherwise, the resulting benefits to the performance of a modern airplane are ever present. However, by using CFD techniques, we can also analyze the impact of introducing winglets as a low cost modification to older airplanes, so as to improve performance and extend their useful service life.

1 Introduction

During the course of work on this paper, we will simulate flows around a wing with different winglet designs and compare the resulting aerodynamic coefficients with results from a simulated flow around an unmodified wing. Our objectives are, therefore, to investigate the influence of winglets on performance of the aircraft, and to select an optimal winglet design from a number of possible solutions [1].

Fig 1. One of the winglet designs investigated

The wing analyzed is designed for a medium size, twin engine, propeller driven, transport aircraft, it has a span of 16.45 *m*, a 3.59 *m* root chord, and 1.79 *m* tip chord. The wing is of a trapezoidal shape with a centroplane section. The wing root airfoil is NACA66(2)215, and the wingtip airfoil is NACA66(2)212. The winglet shown in Figure 1. represents the first investigated solution. The flow around the wing is simulated at various angles of attack, while conditions correspond to a cruising flight speed of 127 *m/s*, at the altitude of 3000 *m*. The calculated Reynolds number of the flow is $Re=15x10^6$.

2 Numerical setup and results

The simulations are performed using Fluent, a commercial CFD solver package. Model meshes are done in Gambit, the meshing tool for Fluent. The mesh for both models is unstructured, and consists of around 50.000 tetrahedral elements, and is shown in Figures 2. and 3. This mesh proved adequate for the preliminary calculations presented here.

Fluent's segregated, implicit solver mode is used, and the case is set as a steady state simulation. Since the Reynolds number shows that the flow is turbulent, we are using the Spalart-Almaras viscosity model for our initial calculations. This is a single equation turbulent viscosity model, developed for aeronautical applications. The first order upwind discretization method is used in preliminary calculations [2-4].

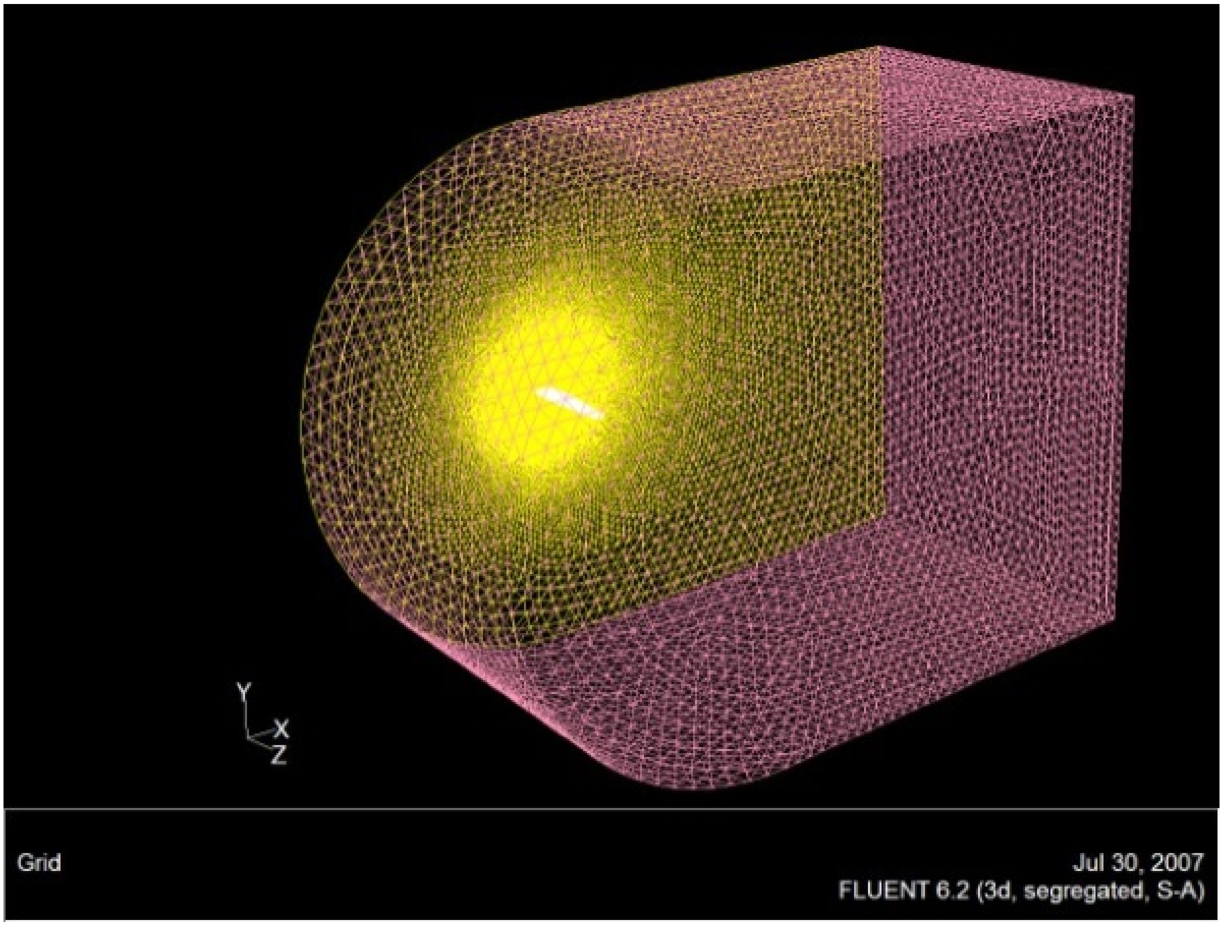

Fig 2. Mesh of the fluid domain

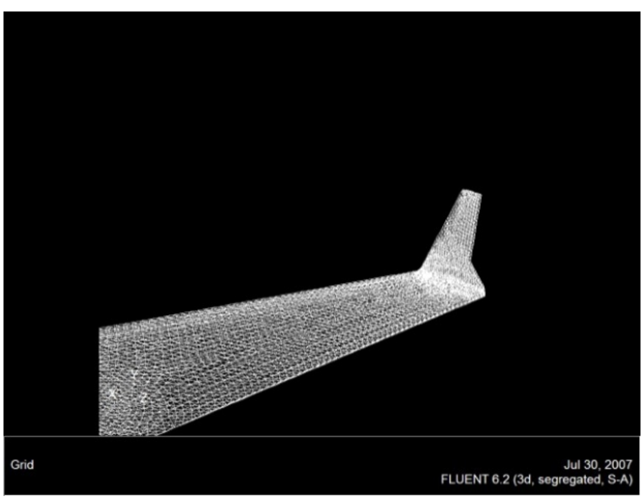

Fig 3. Mesh of the wing surface

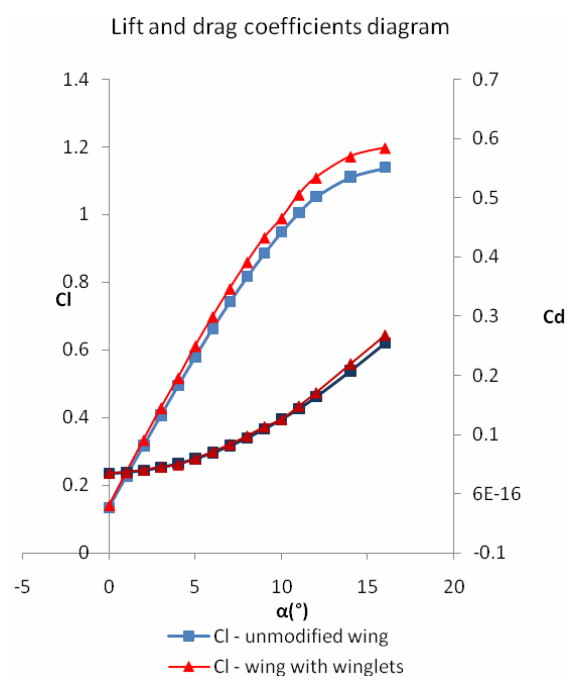

Fig 4. Lift and drag coefficients

As can be seen in Figure 4. the results indicate benefits from the use of winglets. Figure 4. shows lift and drag coefficients for both cases.

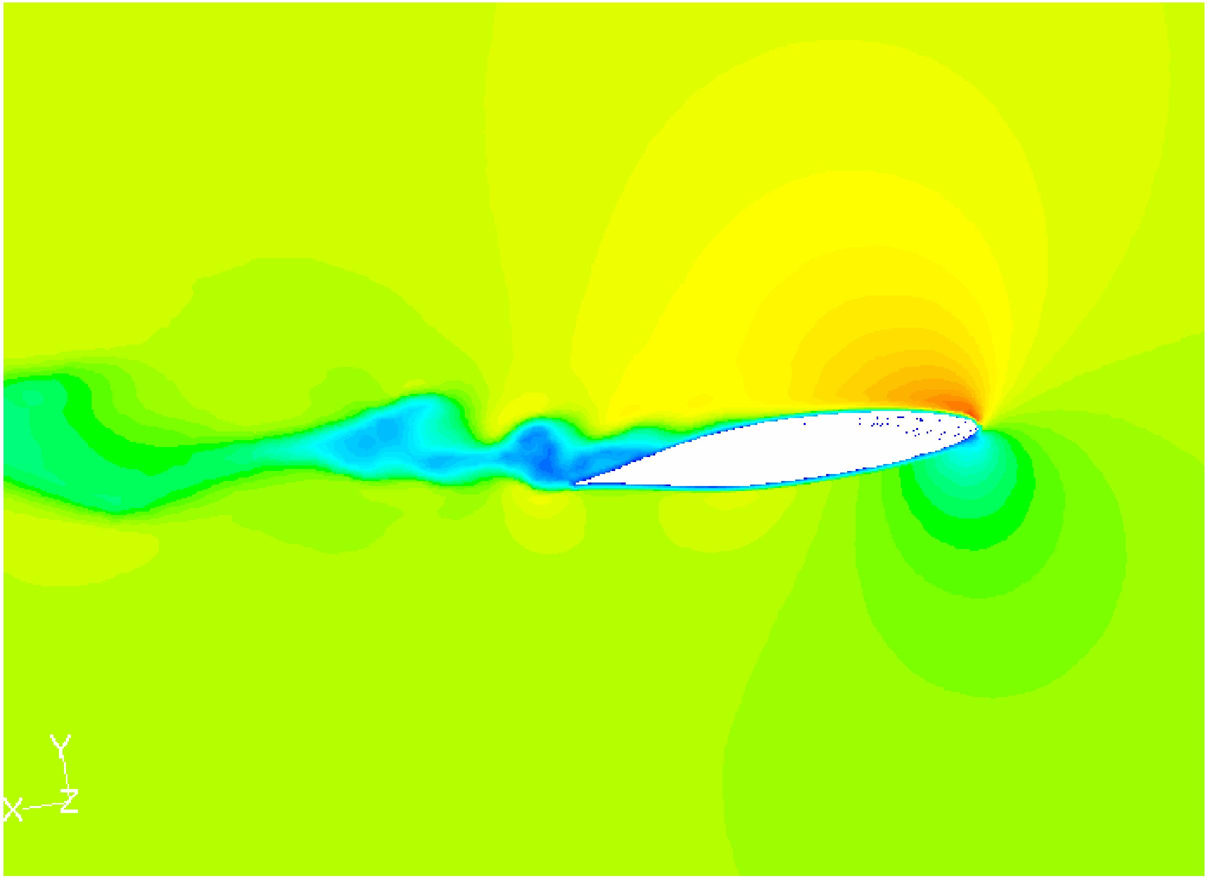

Fig 5. Turbulent wake behind the wingroot at AoA=10°

The preliminary results enable us to do a comparative analysis of aerodynamic coefficients for both cases. However, with regard to aircraft performance we can use this data, even this early in the project, to get a rough estimate of the aircraft's range.

The classic equation for calculating the range of a propeller driven aircraft flying at a constant altitude and angle of attack, shows that, with some approximation, the only value that changes with the introduction of the winglets is the ratio of lift and drag coefficients. Based on the data from our preliminary calculations, we can see that the most improvement in aircraft's finesse is at the angle of attack of 4deg. Therefore, we can deduce that the increase of range, while flying at a constant angle of attack of 4 degrees is around 7.5%.

This is a good indication of a positive impact of winglets on aircraft performance.

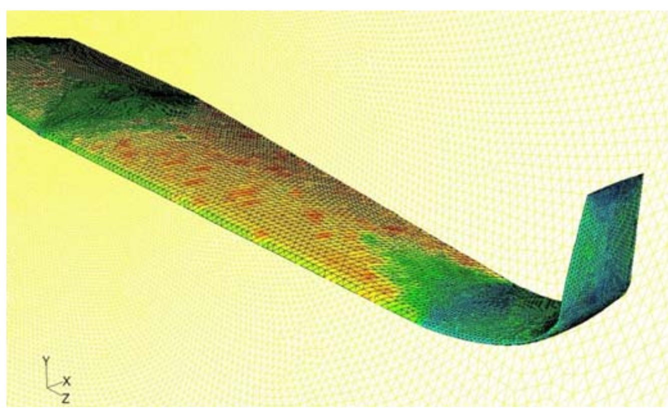

Fig 6. Wall y+ function values of the wing

3 Continued work

Further development of this investigation has led us, inevitably, to the use of finer mesh,

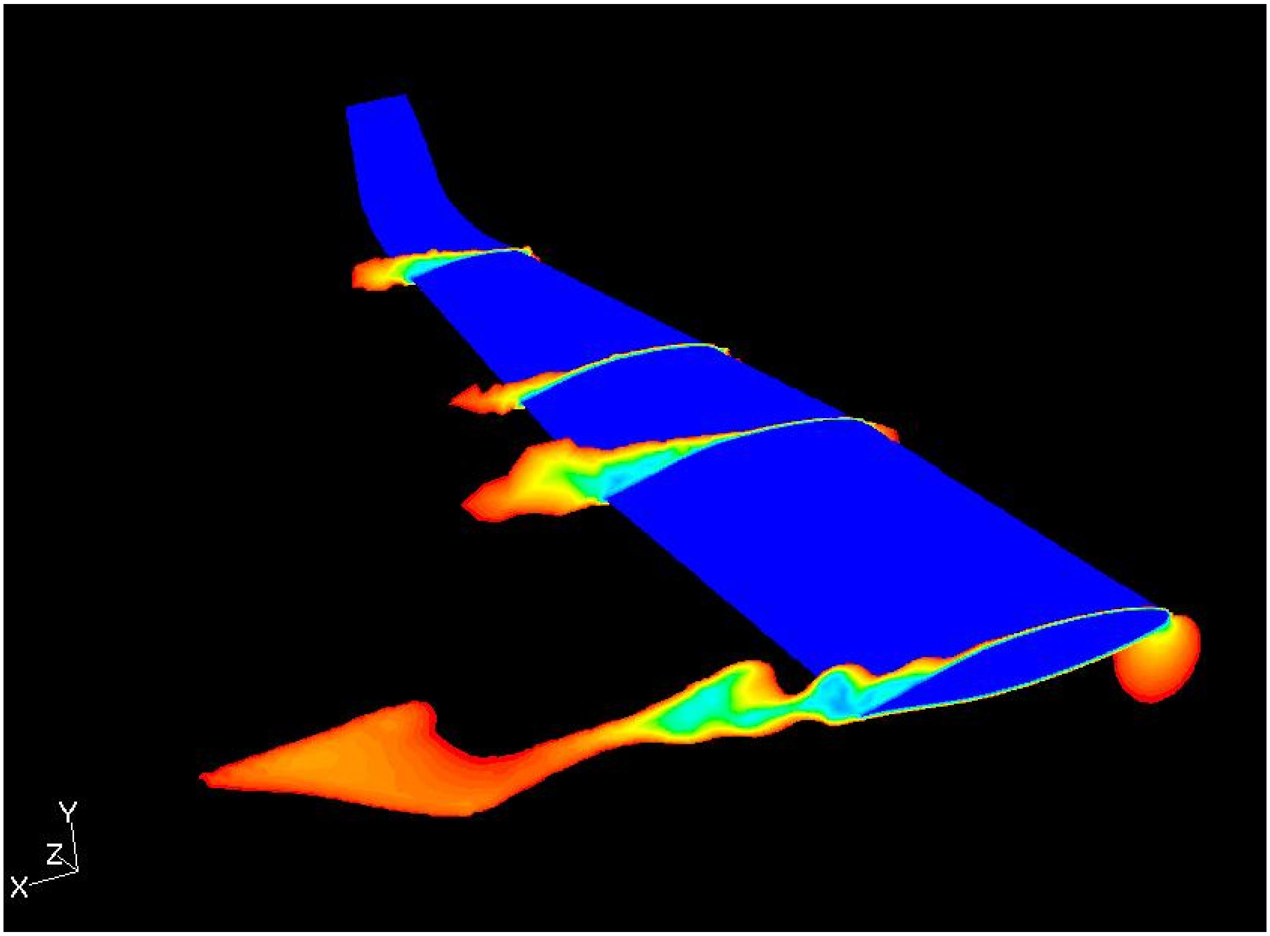

Fig. 7. Flow detachment at various positions of the wingspan

and more advanced turbulence models, such as the Large Eddy Simulation model (LES). In the Figure 5. we can see a time dependant flow around the root of our wing at time t=91s. This solution was obtained using the density based, implicit solver. Air is computed as the ideal gas, the angle of attack of the flow is 10deg and the Mach number is 0.386.

Currently, work is continuing on refining the computation mesh; which consists at present of 76.0000 cells. We have achieved good precision near the root of the wing; wall y+ function values of around 100 are quite sufficient to allow sub-grid models to resolve the boundary layers. However, there is still room for improvement, so, the next step will be to refine the mesh around all wall faces, which should allow much greater accuracy for computing total lift and drag of the wing.

Figure 6. shows the values of the wall y+ function values. The red patches here, are still problematic areas, in which the y+ values are beyond the values established as sufficient to allow the boundary layers to be solved.

Even with this mesh, the LES simulation is showing promising results.

4 Conclusion

Our main goal now is to improve the mesh around the winglet, so that we can finally begin in earnest with the investigation of various winglet designs.

With these improvements, we expect to have a fully resolved turbulent flow around winglets, well defined wingtip vortices, and an

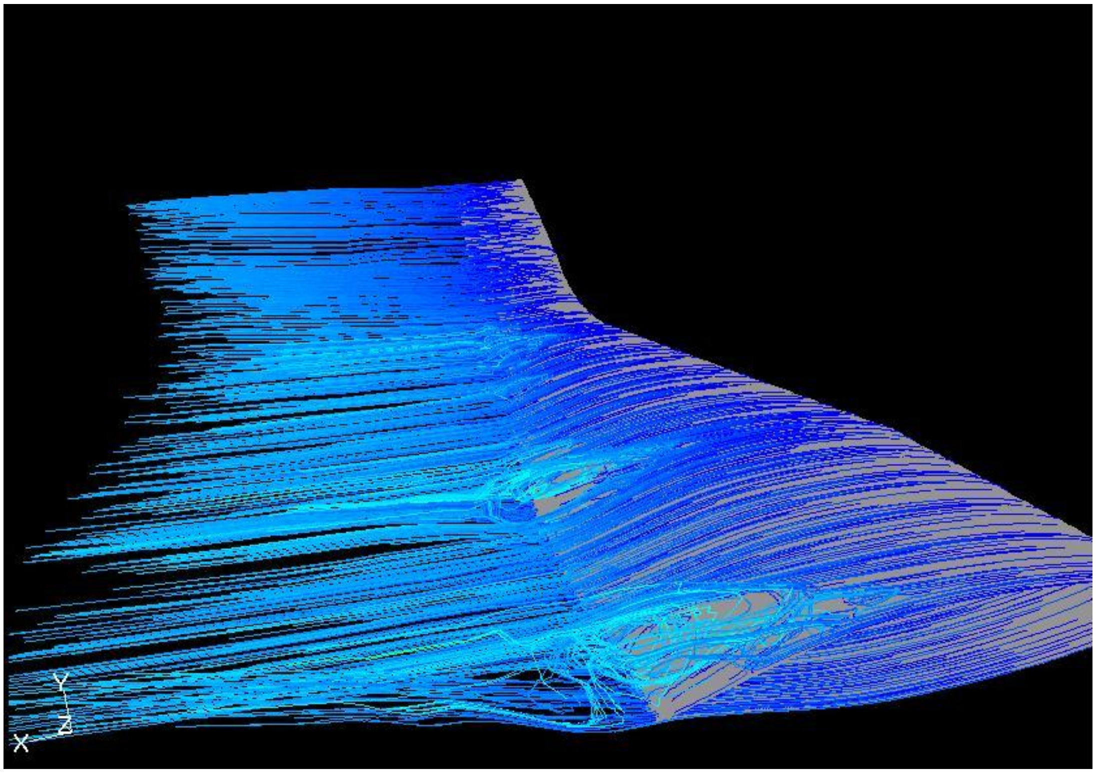

Fig. 8. Pathlines of detached turbulent flow

accurate picture of critical areas of detached flow.

References

[1] Dees, P. and Stowell, M., 737-800 Winglet Integration, *Aerospace Congress & Exhibition and 2001 World Aviation Congress*, Seattle, Washington, USA, September 10-14, 2001, Paper SAE 2001-01-2989, Society of Automotive Engineers, Inc. 2001, CD-ROM.

[2] Breitsamter, C.; Laschka, B.; Heinemann, H.-J.; Hilbig, R. (Eds.), *New Results in Numerical and Experimental Fluid Mechanics IV*: Contributions to the 13th Stab/Dglr Symposium, Munchen, Germany 2002, Series: Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), Vol. 87, Springer, 2004, pp. 1-9, 50-57.

[3] Krause, E. and Gersten, K. (Eds.), *Iutam Symposium on Dynamics of Slender Vortices*: Proceedings of the Iutam Symposium Held in Aachen, Germany, 31 August - 3 Sep. 1997, Germany Iutam Symposium on Dynamics of Slender Vortices 1997 Aachen, Kluwer Academic Pub., 1998, pp. 415-434.

[4] Ballmann, J. (Editor), *Flow Modulation and Fluid-Structure Interaction at Airplane Wings*: Research Results of the Collaborative Research Center SFB 401 at Rwth Aachen, University of Technology, Aachen, Germany, Springer, 2003, pp. 1-56.

Copyright Statement