All Sciences Proceedings
http://as-proceeding.com/

5th International Conference on Applied Engineering and Natural Sciences

July 10-12, 2023 : Konya, Turkey

https://www.icaens.com/

RANS-DES Comparison on the Numerical Modelling of a Downforce Generating Wing Moving over a Solid Surface

Yavuz Hakan Ozdemir[1], Taner Cosgun[2*]

[1] *Department of Motor Vehicles and Transportation Technologies, Canakkale Onsekiz Mart University, Turkey*
[2]*Department of Naval Architecture and Marine Engineering, Yildiz Technical University, Turkey*

(tcosgun@yildiz.edu.tr)

Abstract – Present paper covers the numerical modelling of a 3D wing operating near a solid surface using Reynolds Averaged Navier Stokes (RANS) and Detached Eddy Simulation (DES). The characteristics of the flow over a wing near a surface is significantly different from the free flight. Capturing the changing aerodynamics of an object moving close to a surface have importance for flight and vehicle aerodynamics. This study aims to investigate the reliability of RANS and DES turbulence modelling approaches on the numerical solution of the challenging ground effect flow physics of the downforce generating (or inverted) wings. For this purpose, a 3D symmetrical wing moving in the close proximity of a solid surface is numerically modelled in fully turbulent flow conditions. The results show that, DES approach predicts the lift force slightly better than RANS. On the other hand, tip leakage flow capturing and the separating vortex field modelling of the DES approach is clearly superior. Comparison of velocity, vorticity and the vortex visualization results are presented for the better understanding of the performance of the two modelling approaches.

Keywords – RANS, DES, Wing, CFD, Aerodynamics

I. Introduction

When an object moves close to the ground, flow characteristics around the body considerably change due to the interactions with the surface. Air cushion beneath the body provides a lift enhancement and several other benefits for many flow configurations. This phenomenon is called ground effect. Many research have been concluded to study the aerodynamics of wings in ground effect. Ahmed and Sharma [1] performed wind tunnel measurements to investigate the aerodynamics of a wing in ground proximity. Jung et al. [2] conducted an experimental study to examine the performance of the NACA6409 wing section in ground effect for varying ground clearances and AoAs. Suh et al. [3] utilized both experimental and numerical tools to study the effect of ground proximity on the DHMTU wing. Rozhdestvensky [4] revealed the features of the extreme ground effect scenarios. Kinaci [5] investigated the performance of the Boundary Element Method on the numerical modelling of a wing in ground effect. Diasinos et al. [6] studied the influence of the geometrical parameters on the ground effect aerodynamics of a wing. Lee and Lin [7] presented a review of the experimental investigations considering the wings is ground effect.

On the other hand, if the object has negative angle of attack (AoA) or geometrically inverted, the ground proximity cause the object to generate negative lift. The inverted wing has an important role for the vehicle and racing car design because the negative lift produced by the wing is a useful and simple tool to keep the vehicle attached to the ground in high speeds. Ground effect aerodynamics of the Tyrrell wing has been studied by Zerihan and Zhang [8]. Vogt and Barber [9] performed 2D computations to investigate the ground effect on the lift and downforce generating airfoils. Jacuzzi and Granlund [10] examined the ground effect aerodynamics of an inverted wing subjected to a sinusoidal heaving motion. Doig and Barber [11] have numerically tested five different wings to study the influence of the geometrical configurations on the obtained results.

The present paper aims to investigate the reliability of two different turbulence modelling approaches, Reynolds Averaged Navier Stokes (RANS) and Detached Eddy Simulation (DES), on the numerical modelling of a downforce generating symmetrical wing. NACA0012 wing section was utilized in computations. The results of the Computational Fluid Dynamics (CFD) predictions are presented in terms of lift force, velocity and vorticity fields and the tip vortex visualizations.

II. MATERIALS AND METHOD

Numerical modelling of a 3D symmetrical wing with a NACA0012 section performed via computational fluid dynamics. The geometry of the problem can be seen in fig.1. the wing has a chord length, c, of 0.317 m. and aspect ratio, AR, of 3.02. The rectangular-shaped solution domain of the problem was created. The inflow boundary was placed at 10c, and outlet boundary is placed at 20c away from the wig. The distance of the sidewalls are 3.3c. the geometry of the problem was configured similar to the experimental work of Moore et al. [12] for validation purposes. The clearance between the wing and the ground is determined as h/c=0.1 and the angle of attack is -5 degree.

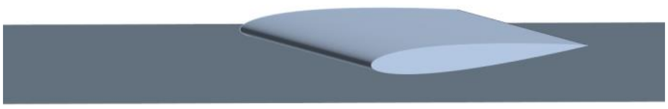

Fig. 1 Geometry of the problem

A commercial CFD tool, Simcenter StarCCM+ was used in computations. The solver implements finite volume method to discretize the governing equations. A uniform velocity profile at Re=8x10^5 ($\mathrm{Re} = uc/\nu$, u velocity, ν kinematic viscosity) was imposed at the inflow boundary. The outlet boundary is treated with pressure outlet BC. No-slip condition was applied to the rest of the boundaries. The moving wall condition with the same speed with the wing was applied to the bottom wall of the solution domain. Detached eddy simulation (DES) and the Reynolds averaged Navier-Stokes (RANS) approach with k-w turbulence model was used to model the turbulence field. Details of the models can be found in solvers manual [13].

The structure of the grid topology around the wing is presented in fig.2. Unstructured hexahedral grids were used to create the solution domain. Several grid refinements were created around the wing, wake and the wing tips using volumetric volumes. The grid structure between the wing and the ground was also refined to capture the high velocity gradients in this region. the surface mesh was created to keep the y+ ($y+ = u_\tau\, y/\nu$, u_τ friction velocity, y the height of the first grid on the wall, υ kinematic viscosity) below 1. The total mesh count is around 11M. the same mesh structure was used for both DES and RANS calculations for proper comparisons.

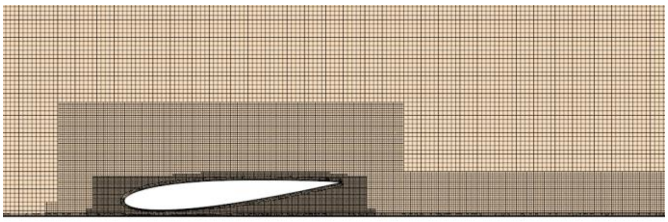

Fig. 2 the mesh structure around the wing

III. RESULTS

The performance of DES and RANS turbulence modelling approaches on the numerical modelling of a downforce generating wing near the ground was investigated via computational fluid dynamics. Fig. 1. shows the lift coefficient predictions of both methods and the comparison with experimental data of Moore et al.[12].

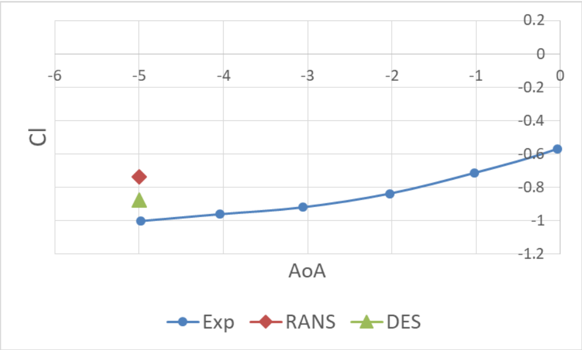

Fig. 3 The Lift coefficient results

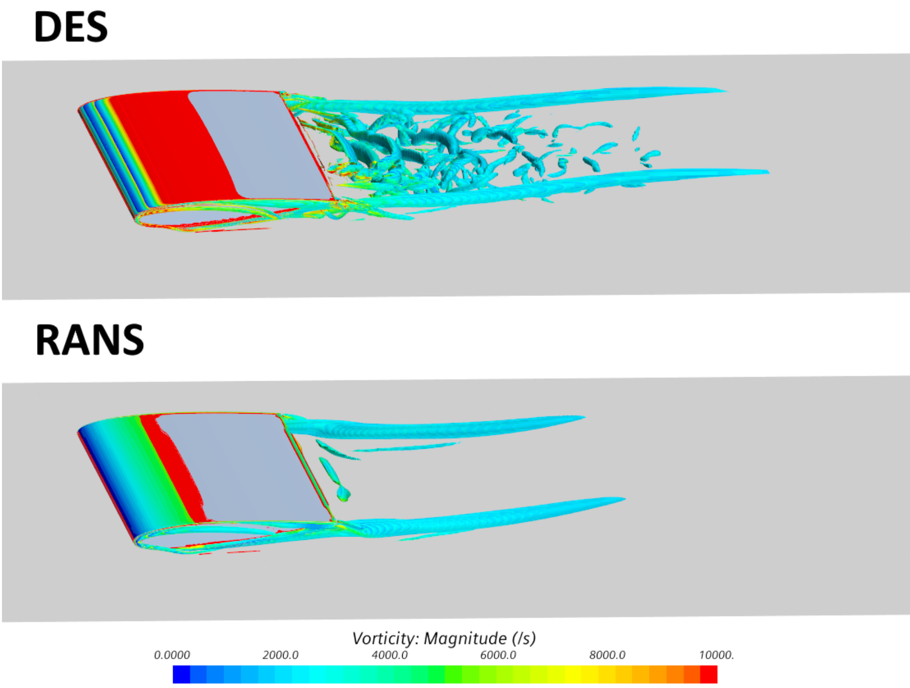

Fig. 4 Tip vortex visualization with DES and RANS computations ($Q_{crit}=1x10^6$)

Both DES and RANS approach produce satisfactory results. The lift coefficient obtained with DES is slightly closer to the experimental data.

Fig.4 show the visualization of the tip vortex in DES and RANS computations. Des is apparently superior on the modelling of the tip vortex. The wing produce a shorter tip vortex in RANS calculations. Furthermore, DES also successfully models the flow separation in the wake. On the other hand, RANS predicts nearly no separating vortex behind the wing.

The velocity field around the wing is presented in fig.5. In the RANS computations, wake behind the wing is shorter and the velocity distribution seems smoother. But the wake in the DES predictions extends through a larger distance. Furthermore, the effect of flow separation on the velocity field is apparent in DES results.

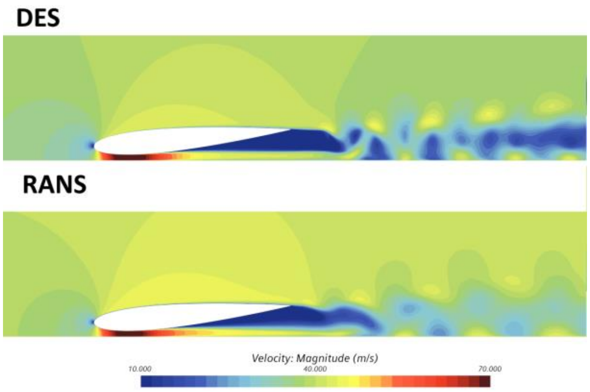

Fig. 5 The velocity field around the wing

Fig. 6 shows the vorticity distribution on the midsection plane of the wing. The results of the both method are in parallel with the vorticity visualization in fig.4. DES models the flow separations in the wake of the wing with success. Also, the interactions of the wake with ground can e seen in DES results. However, the vortices in the wake smears out in a short distance in RANS computations.

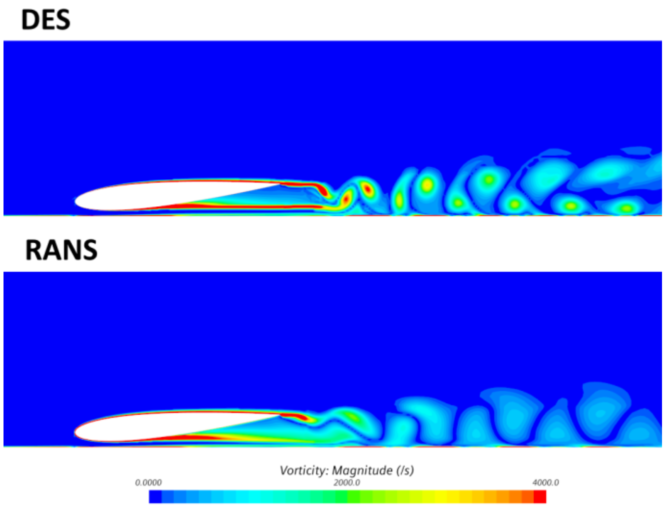

Fig. 6 The velocity field around the wing

IV. CONCLUSION

The paper presents the numerical modelling of a downforce generating symmetrical wing in ground effect using computational fluid dynamics. The reliability of two turbulence modelling approach, Detached eddy simulation and the Reynolds averaged Navier-Stokes, on the modelling of the ground effect aerodynamics were investigated. A 3D wing with NACA0012 cross section with a -5 degree of AoA was used in calculations. The wing height is kept constant at h/c=0.1

The results show that, both DES and RANS predicts the lift force with an acceptable amount of error. The lift results of DES is slightly closer to the experimental data. On the other hand, DES performs clearly better on the modelling of the tip vortex and the wake. The flow separations and the vortex field behind the wing seems to exhibit more realistic flow physics, while vortex field and the wake in RANS computations vanishes in a short distance.

REFERENCES

[1] M. R. Ahmed and S. D. Sharma, "An investigation on the aerodynamics of a symmetrical airfoil in ground effect," *Exp. Therm. Fluid Sci.*, vol. 29, no. 6, pp. 633–647, 2005.

[2] K. H. Jung, H. H. Chun, and H. J. Kim, "Experimental investigation of wing-in-ground effect with a NACA6409 section," *J. Mar. Sci. Technol.*, vol. 13, no. 4, pp. 317–327, 2008.

[3] S.-B. Suh, K.-H. Jung, and H.-H. Chun, "Numerical And Experimental Studies On Wing In Ground Effect," *Int. J. Ocean Syst. Eng.*, vol. 1, no. 2, pp. 110–119, 2011.

[4] K. V. Rozhdestvensky, *Aerodynamics of a Lifting System in Extreme Ground Effect*, Springer,. Springer Berlin Heidelberg, 2000.

[5] O. K. Kinaci, "An iterative boundary element method for a wing-in-ground effect," *Int. J. Nav. Archit. Ocean Eng.*, vol. 6, no. 2, pp. 282–296, 2014.

[6] S. Diasinos, T. J. Barber, and G. Doig, "Influence of wing span on the aerodynamics of wings in ground effect:," *Proc. Inst. Mech. Eng. Part G J. Aerosp. Eng.*, vol. 227, no. 3, pp. 569–573, Jan. 2012.

[7] T. Lee and G. Lin, "Review of experimental investigations of wings in ground effect at low Reynolds numbers," *Front. Aerosp. Eng.*, vol. 1, no. November, 2022.

[8] J. Zerihan and X. Zhang, "Aerodynamics of a single element wing in ground effect," *J. Aircr.*, vol. 37, no. 6, pp. 1058–1064, 2000.

[9] J. W. Vogt and T. J. Barber, "Ground effect phenomena about lift and downforce generating cambered aerofoils," *Int. J. Numer. Methods Heat Fluid Flow*, vol. 22, no. 2, pp. 153–174, 2012.

[10] E. Jacuzzi and K. Granlund, "Heaving inverted wing in extreme ground effect," *J. Fluids Eng. Trans. ASME*, vol. 142, no. 11, Nov. 2020.

[11] G. Doig and T. J. Barber, "Considerations for

Numerical Modeling of Inverted Wings in Ground Effect," *AIAA J.*, vol. 49, no. 10, pp. 2330–2333, Apr. 2012.

[12] N. Moore, P. A. Wilson, and A. J. Peters, "An investigation into wing in ground effect aerofoil geometry," in *RTO-MP-095; NATO RTO.11; NATO RTO*, Washington DC, USA, 2002, p. 20.

[13] Siemens, "Star-CCM+ User Guide version 14.02." 2019.