ID 093

Design of an Active Flow Control Method for a 3-D Wing

KNS Perera[1#], MM Basnayake[1] and SLMD Rangajeeva[1]

[1]Faculty of Engineering, General Sir John Kotelawala Defence University, Ratmalana, Sri Lanka

[#]jamesnipun23@gmail.com

Abstract

This research performs a numerical proof for enhancement of aerodynamic characteristic of a finite wing (Wing of Nanchang CJ-6 Aircraft) which is included with a conceptual design of an active flow control method. The active flow control design is a combination of continuous suction and blowing of air profile over the surface of wing. The effectiveness of active flow control model was tested by changing the number of slots and locations of suction and blowing. The numerical analysis was consisted with Reynolds-averaged Navier-Stokes (RANS) equations which were used in combination with a k-ω SST turbulence model. The optimum results were obtained for locating the blowing slots within the range of 0.3-0.47 in the chord length and suction slots within the range of 0.6-0.77 in the chord length. The Computational Fluid Dynamics (CFD) analysis was carried out at freestream conditions with a Mach number of 0.238, a Reynolds number of 6.166×10^6 and Angles of Attack (AOA) from $0°$ to $15°$. The delaying of point of flow separation at higher AOA was clearly observed and an increment of 30% and 24% in lift to drag ratio was obtained at an AOA of $0°$ and $12°$ respectively. The CFD simulations were performed using openFOAM open-source software by giving the custom boundary conditions for the slot surfaces.

Keywords: *Active flow control, Aerodynamic characteristics, Flow separation*