CFD Prediction of Cavity Drag at Transonic and Low Supersonic Speeds

김희동 · 구병수 · 우선훈
(School of Mechanical Engineering, Andong National University)

(E-mail : kimhd@andong.ac.kr)

In the high lift devices specifications for surface smoothness requirements, as manufacturing tolerances, arise out of aerodynamic consideration to minimise drag. True optimisation of tolerances is a multi-disciplinary problem involving fluid mechanics, device performance, manufacturing philosophy and life cycle costing. One of the reasons for degradation of wetted surface is discrete roughness as a consequence of manufacturing defects, collectively termed as one of the excrescences effect. Usually, excrecsence drag arising out of discrete roughness is of considerable lower order of magnitude as compared to the total drag of the flight bodies. Nor was there adequate predicting tool to account for the extent of drag degradation. Estimation of excrescence drag remained as a state-of-the art based on experimental results.

A considerable amount of experimental work of excrescence drag has to date been published on flat plate in turbulent boundary layers at subsonic speeds, but not in transonic and low supersonic flow regimes. Experimental results and semi-empirical relationship to estimate excrescence drag of surface irregularities on flat plates are incorporated in ESDU data sheets but are not valid where there is large scale separation, a phenomenon which can become quite serious if shock wave/boundary layer interaction is present.

In particular there is considerable interest in the excrescence drag of cavities where the local Mach number is of transonic or low supersonic speeds. The boundary layer is likely to be turbulent and it is estimated that the depth of the cavities or steps formed by manufacturing limits will be comparable with the local boundary layer thickness. With this small scale there are limits on the measurement of pressure distributions in the cavities.

This paper describes a computational investigation of the transonic and low supersonic flows around and over small cavities. Various types of cavities were tested to investigate the effect of the cavity configuration on the cavity drag and on the flow filed around the cavity. Two-dimensional compressible Navier-Stokes equations were numberically solved using a fully implicit finite volume differencing scheme. Temporal derivatives were integrated using a multi-step Runge Kutta Scheme, and serveral types of two-equation turbulence models were applied to the governing equations. Computations were carried out based on a fine structured grid system of 300×150 node points. Free stream Mach umber was changed in a range between 0.8 and 2.5. The current CFD results were in good agreement with the previous wind tunnel data.