Simulation of flow-induced cavity resonance with turbulence models

K. S. Jang [1], S. O. Park [2]
1.KAIST Aerospace Engineering, bampi@sop1.kaist.ac.kr
2.KAIST Aerospace Engineering, sopark@sop1.kaist.ac.kr

Abstract

A numerical simulation of an incompressible cavity flow is conducted using turbulence models. Cavity geometry and flow conditions are based on Cattafesta's experiment. Baldwin-Lomax model and $k-\varpi$ model are employed. While simulation with Baldwin-Lomax model predicts the oscillatory features of the flow, the use of $k-\varpi$ model in its original form makes the simulation converge to steady flow. To acquire oscillatory flow solution, Kato-Launder form and Time scale bound are adopted in production term of $k-\varpi$ model. The strouhal number of the flow oscillations from the simulation results corresponds to 1 st mode in simulation but 2 nd mode in experiments. However mean velocity profile is in good agreement with the experimental data and the fluctuation profile follows the tendency of Cattafesta's results.

***Keyword:** cavity, turbulence model, limiter*

1. Introduction

The procedure of cavity flow oscillation is such that the vortices at the upstream edge are generated and run on mouth of cavity and are dashed against the downstream edge. When this happens, the complex feedback phenomenon exists with turbulence regions. To simulate numerically this flow, turbulence modeling , DNS or LES method are used widely. In turbulence modeling of cavity flow, zero-equation model like Baldwin-Lomax model and the two-equation models($k-\varepsilon$, $k-\varpi$ etc) are often adopted[1]. These models predict well the time averaged mean values but often are not capable of predicting the frequency of flow oscillation and fluctuation values. Sinha[1]'s results show that the simulation with turbulence modeling converges to steady state

For the case of impinging jet flow and flow around airfoil, the excessive turbulent kinetic energy and viscosity are predicted in the region of high strain rate or near the leading edge when two equation models are employed. Medic and Durbin[2] call this "stagnation point anomaly".

Several methods are presented to overcome this anomaly in two-equation turbulence models. Kato and Launder[3] proposed the modification of kinetic energy production term by representing this term as the strain rate times the vorticity. This equates the production to zero in irrotational flow. However, this approach is known to violate energy conservation. Time scale bound is proposed by Medic and Durbin[2].

In the present work, the numerical simulations are carried out for the cavity flow experimentally studied by Cattafesta[4]. The purpose of the work is to examine the capability of simulation based on the incompressible flow formulation by adopting the turbulence models. Baldwin-Lomax model and $k-\varpi$ models with modifications are employed.

2. Simulation Results

2.1 Frequency of oscillation

The simulation results for the case of $k-\varpi$ original model converged to the steady solution as stated in Sinha et al.[1]. The use of Kato-Launder form and the time scale bound of Durbin made the flow oscillatory. The Strouhal number for both cases were 0.4. The use of Baldwin-Lomax model also resulted in the oscillatory flow with the Strouhal number of 0.4.

However, the Strouhal number of the first peak was 0.87 in Cattafesta's experiments, which is close to the value 0.78 obtained by the formula proposed by East[8]. Thus, the present sumulation is not in accord with the experiment.

Rockwell and Naudascher [5] predicted the frequency of cavity flow oscillation mathematically based on the linear stability theory and validated his results against the experimental data of Ethembabaoglu[9]. The results are plotted in Fig. 2. The filled points represent frequencies with largest amplitudes. The Strouhal number of the Cattafesta's experiments agrees with the second mode of Fig. 2 and our result corresponds to the first mode. It is likely that the over-dissipative nature of turbulence model damped out the second mode of oscillation, which is subject to further study .

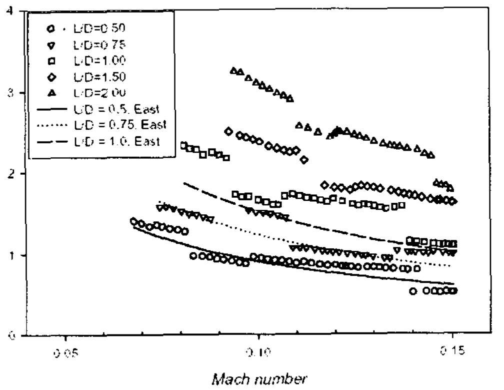

Fig. 1 Strouhal number of Cattafesta[4] Exp.

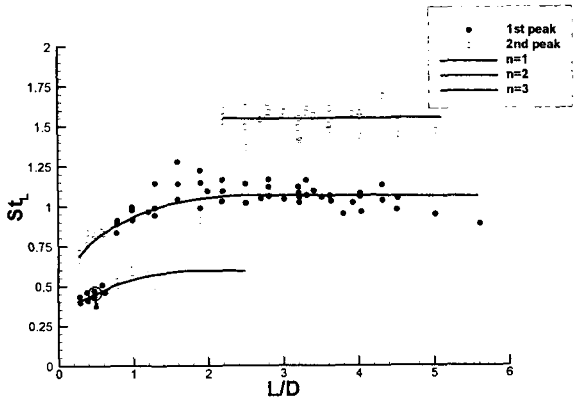

Fig. 2 Strouhal number of Rockwell[5] and Our simulation

2.2 Vorticity

Fig. 3 show the visualization of smoke front the upstream edge in Cattafesta[4]'s experiment. Two large vortical structures are seen over the cavity mouth, and this is consistent with the fact that the oscillation frequency is the second mode of Fig. 2. The vorticity contour plots over one period in the present simulation with Kato-Launder form are given in Fig. 4. The vortex separates from the upstream edge and collides against the downstream edge to breaks out. The chronological pattern of Fig. 4 reveals that only one vortical structure exists over the cavity mouth. This results prove the first mode oscillation in our simulation.

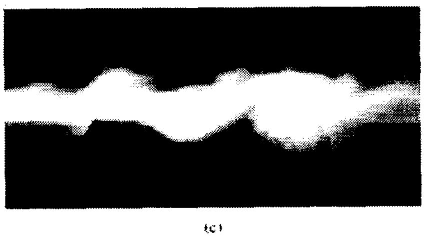

Fig. 3 Vorticity visualization of Cattafesta[4]

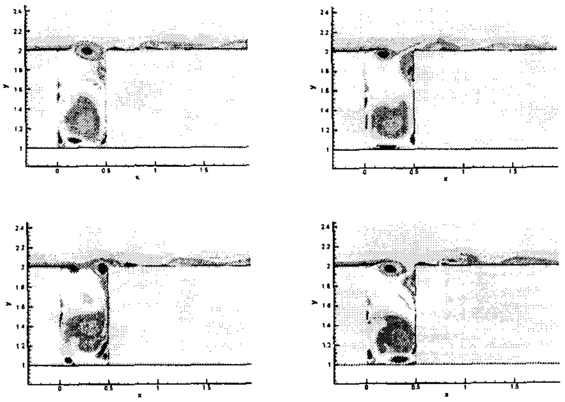

Fig. 4 Vorticity contour plots over one period

2.3 Velocity profile at two positions

Cattafesta[4] measured the streamwise mean velocity using hot-wire at the positions of x/L equal to 0.02 and 0.78. The predicted velocity profile are compared with experimental data in Fig. 5. It is seen that all of the mean velocity profiles are in good agreement with the experimental data.

In Fig. 6 the fluctuating streamwise velocity profiles are plotted together with the experimental data. It can be seen that the predictions follow the tendency of the fluctuating velocity data of the experiment. The magnitude of the fluctuation increases with the downstream distance.

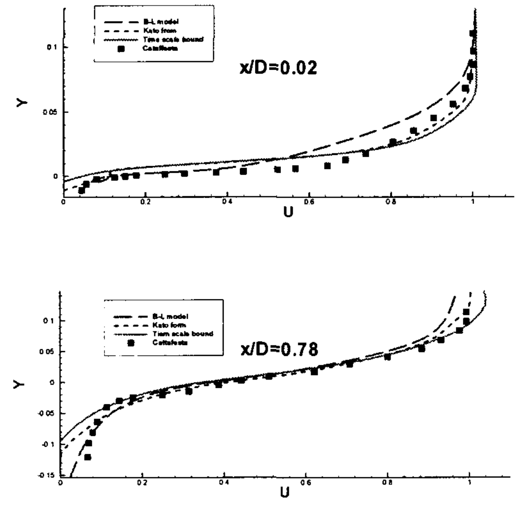

Fig. 5 Streamwise mean velocity profile

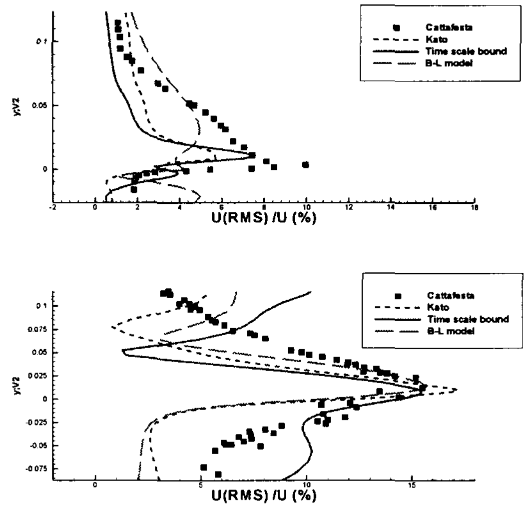

Fig. 6 Mean velocity fluctuation profile

3. Conclusions

The numerical simulations of low speed cavity flow are conducted by adopting turbulence models of Baldwin-Lomax model and $k-\varpi$ model. When the $k-\varpi$ model in its standard form is employed, the simulation converges to a steady flow. To capture the oscillatory phenomenon, Kato-Launder form and the time scale bound have to be adopted in the $k-\varpi$ model. The mean velocity profiles of the simulation are in good agreement with the experimental data. However, the dominant frequency of the velocity fluctuation is not in accord with the experimental data. This discrepancy is subjected to further investigation.

References

[1] Sinha N., Dash S. M. and Chidambaram N., "A Perspective on the simulation of cavity aeroacoustics", *AIAA paper*, 98-0286

[2] Medic G., Durbin P. A., "Toward improved prediction of heat transfer on turbine blades", *Journal of Turbomachinery.*, Vol. 124, April, (2002), pp 187-192.

[3] Kato M., and Launder B. E., "Modeling flow induced oscillation in turbulent flow around a square cylinder", *ASME FED,* Vol. 157, pp 189-199

[4] Cattafesta III L. N., "Active control of flow-induced cavity resonance", *AIAA paper*, 97-1804.

[5] Rockwell D., and Naudascher, E., "Review: Self-sustaining Oscillations of flow past cavities", *Trans. ASME,J. Fluids Eng.* Vol. 100, (1978), pp 152-165.

[6] Rodi W., and Scheuerer G., "Calculation of heat transfer to convection-cooled gas turbin blades", *Journal of Engineeing for Gas Turbine and Power.*, Vol. 107, July, (1985), pp 620-627.

[7] Musker A. J., "Explicit expression for the smooth wall velocity distribution in a turbulent boundary layer", *AIAA J.*, Vol. 17, 6, (1979), pp 655-657.

[8] East L. F., "Aerodynamically induced resonance in rectangular cavities", *Journal of sound vibration.*, Vol. 3, 3, (1966), pp 277-287.

[9] Ethembabaoglu S., "On the fluctuating flow characteristics in the vicinity of gate slots", Division of Hydraulic Engineering, University of Trondheim, June, (1973).