S.A. ISAEV[1], V.L.ZHDANOV[2]
[1] *Academy of Civil Aviation, St.Petersburg, Russia*
[2] *Luikov Heat and Mass Transfer Institute, Minsk, Belarus, (15, P.Brovka str., 220072 Minsk, Belarus; e-mail: Zhdanov@hmti.ac.by)*

CONTROL OF CYLINDER DRAG AND LIFT FORCE AMPLITUDE IN TURBULENT CROSSFLOW

Abstract. The numerical results on turbulent cross flow over a circular cylinder are presented. The cylinder has inner ducts used to bypass some amount of the oncoming flow to the rear cylinder part. The influence of the angular position of bleeding jets and bypass flow rate on the cylinder drag coefficient and lift force amplitude are investigated.

1. INTRODUCTION

The sensitivity of the aerodynamics characteristics of bodies in crossflow to bleeding flow rate as well as to positions of jets ejected into the near flow wake has been shown in experimental and numerical investigations [1-4]. Using energy of the oncoming flow with some of its amount being passed through the inner body ducts from leading to back surfaces is quite an attractive method to reduce not only body drag but also vortex-induced vibration of the body in crossflow. In some cases the reduction of lift force amplitude is a more urgent problem than body drag reduction due to unsteady vortex shedding. To clear up the influence of angle positions of bleeding jets as well as jet flow rate on the aerodynamics characteristics of circular cylinder the present investigation has been done.

2. NUMERICAL METHOD

The known mathematical models based on the system of the Reynolds equations are used for numerical solution of two-dimensional unsteady turbulent flow around a circular cylinder with base bleed. Digitizing the governing differential equations is realized in the framework of splitting of physical phenomena into their characteristic parameters with the use of the implicit finite-volume method. It is applied to the equations written down for an increment of dependent variables in the so-called E-factor formulation (E=2.5). The time derivative of a dependent variable is rearranged to the explicit part of equations and due to this the computational algorithm of the unsteady-state problem becomes a generalized solution of its steady analogue. The solution is determined at global iterations. The essential feature of the developed algorithm is the application of different-scale grids for various flow regions. A special iteration procedure for the parameters of different-scale crossing grids was developed [5]. Due to the algorithm, such flow regions as a boundary layer, a shear layer, and large vortices in the wake were reproduced with a quite accuracy.

3. RESULTS

An object of investigation was a circular cylinder with inner ducts. Two ducts started from the window coincident with the flow front stagnation point on the cylinder surface and bent symmetrically round the central part of the cylinder. They were used to bypass some amount of the oncoming flow from the forward stagnation point (high-static pressure zone) to the rear part of the body (low-static pressure zone). The ducts had equal lengths and configuration. Five angular positions of these duct exits were examined: 1) α_1 = 110°, α_2 = 250° 2) α_1 = 130°, α_2 = 230°; 3) α_1 = 140°, α_2 = 220°; 4) α_1 = 160°, α_2 = 200°; 5) α_1 = α_2 = 180°. As for the last configuration both exits coincided with the flow rear stagnation point and the width of the formed window was equal to the entrance one. The heights of these ducts were $h_1 = h_2 = 0.1D$. The width of the entrance window around the cylinder periphery was $L = 0.1D$, $0.2D$ and $0.3D$. The widths of the exit windows were twice as narrow ($l_1 = l_2 = 0.05$, 0.1 and $0.15D$, respectively). The flow wake control by discharging fluid jets was considered when the cylinder was placed into the flat channel $7.3D$ high. The flow at the channel entrance was uniform with a turbulence level of 1.5% at $\mathrm{Re} = 1.45 \cdot 10^4$. The turbulent boundary layer thickness at the channel entrance was set as $0.15D$.

The mean cylinder drag varies with angular positions of bleeding jets and also with bleeding flow rate. The latter increases as the width of the entrance window grows (Fig. 1). Coefficient C_x decreases about 40% within the angular interval α=130 -180°.

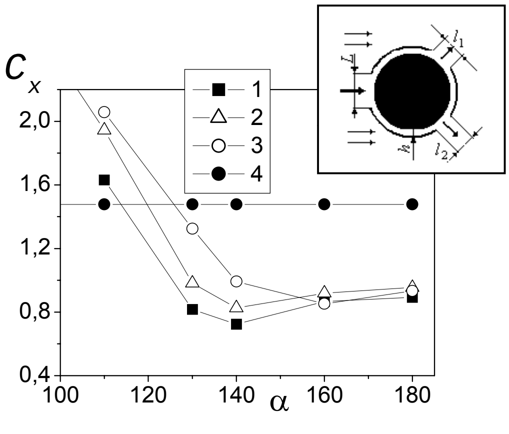

Figure 1. Mean cylinder drag variation vs. angle positions of bleeding jets: 1 – L=0.3; 2- 0.2; 3 –0.2; 4 – without bleeding.

The data presented show that bleeding through two symmetrical narrow slots more strongly reduces cylinder drag as compared with the central jet (α=180°) at all considered jet flow rates (at all widths of the forward window). The cylinder drag was above the value corresponding to the cylinder without inner ducts at angles $\alpha > 120°$. It was a result of a profile drag growth due to expanding wake width under the action of jets.

The strongest reduction of the cylinder drag was fixed at jet angles α_1=160°, α_2=200° for all jets flow rates. It is interesting to note that bleeding through the narrowest slots ($l_1=l_2=0.05$) results in the lowest cylinder drag.

As mentioned elsewhere in [4] two symmetrical jets localised wake buffeting and reduced lift force amplitude. When influenced by the jets, the amplitude reduction is the strongest, the higher is the jet flow rate within $110° < \alpha < 160°$ (Fig.2). A maximal reduction takes place at the angle interval α_1=140°, α_2=220°; α_1=160°, α_2=200° and the strongest lift force amplitude suppression corresponds the case of the jet that was ejected through the smallest slots $l_1=l_2==0.05$.

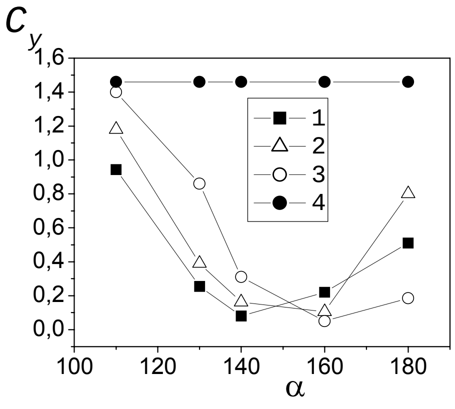

Figure 2. Lift force amplitude variation vs. angle positions of bleeding jets: 1 – L=0.3; 2- 0.2; 3 –0.2; 4 – without bleeding.

Bleeding through the central exit window is accompanied by shifting wake relative to the channel symmetry line (as if there was an attack angle of the oncoming flow) and by increasing buffeting amplitude.

The computed distributions of the dynamics wake parameters (pressure, mean and pulsations velocity, vorticity) showed that steady vortex regions were formed as a result of jets bleeding behind the cylinder.

4. CONCLUSIONS

The present investigation showed that cylinder drag and fluctuation of amplitude unsteady lift force could be significantly reduced when a small amount of the oncoming flow was bled through the narrow windows placed on the rear cylinder surface. The drag coefficient was reduced by a factor of about 40% and amplitude, by more than order. The angle coordinates of these windows were very important. The maximal variations of the cylinder aerodynamics coefficients took place when jets were ejected at α_1=160°, α_2=200°. These variations were a sequence of near wake rearrangement. Two steady symmetrical vortex regions were formed behind the cylinder under the jet bleeding action.

5. REFERENCES

1. Tamotsu Igarachi. Flow Characteristics Around a Circular Cylinder with Slit. Bull. JSME, 1978, Vol.21, No. 154, pp. 656-664.
2. Suryanarayana G.K., Meier G.E.A. Effect of Ventilation on the Flow Field Around Sphere. Experiments in Fluids, 1995, Vol.19, pp.78-88.
3. Lu M., Grosche F.-R., Meier G.E.A. Drag Reduction of a Sphere at Supercritical Reynolds Numbers Using Passive Ventilation. Proc. IUTAM-Symposium on Mechanics of Passive and Active Flow Control, 7-11 Sept. 1998 (eds. G.E.A. Meier and P.R. Viswanath) Kluwer Academic Publishers, Dordrecht / Boston / London, 1999, pp.127-133.
4. Isaev S.A., Zhdanov V.L., Niemann H-J. Numerical study of the bleeding effect on the aerodynamics coefficients of circular cylinder. J.Wind Eng. Ind. Aerodyn. 2002, Vol.90, Issue 11, p. 1217.
5. Isaev S.A., Guvernuk S.V., Zubin M.A., Prigorodov Yu.S. Numerical and physical modeling of a low-velocity air flow in a channel with a circular vortex cell. J. Eng. Phys Thermophys. 2000, Vol.73, No.2. pp. 337-344.